华兴数控
WASHING

Milling machine control system

USER′ MANUAL

Suitable for:WA710MI/710MD/

720M/730M/740M

NANJING WASHING CNC Technology CO.,LTD

Safety handling explanation

To make sure of proper using the system, please read this manual throughout and carefully before operating the machine tool.

General warning and matters needing attention

(1)When using a new programme to carry on actual workpiece processing, please do not directly carry on the processing, but to use single segment stage to execute test run or confirm the machine tool's mechnical movements are correct without installing cutter and workpiece. It is possible to result in unforeseeing movement that may endanger the cutter, machine tool, workpiece and people if the programme is not approved correct during test run.

(2)Operation should only be carried out after full comfirmation all date input proved correct, in case of improper data input, potential damagies may be caused to the cutter, machine tool, workpiece and people.

(3)Setup proper feeding speed and main axle rotary speed. Each set of mahine tool has a maximum feeding speed limit and the setup feeding speed can not exceed the limit that the machine tool can bear. Moreover, different processing objects have different optimum feeding speed and main axle rotary speed, please defer to machine tool manual. Improper feeding speed and main axle setup may endanger the cutter, machine tool, workpiece and people.

(4)When using cutter compensation function, compensation direction and compensation amount should be fully confirmed correct otherwise wrong data may endanger the cutter, machine tool, workpiece and people.

(5)System parameter should be set to proper value. When adjustment is needed, adjustment can only be carried out based on fully understanding the meaning of the parameter value, in case improper parameter setup, it may endanger the cutter, machine tool, workpiece and people.

(6)Configured parameter should be backup in case restoring is needed.

Programming related warning explanation

When programming you must be familiar with and fully understanding the operation maual, the following issues are needed to pay attention related to safety handling.

(1) Reference frame setup

In case coordinate system setup improperly, even if the programme movement command is correct, the anticipated movement can not be achieved and may endanger the cutter, machine tool, workpiece and people.

(2) Non-linear motion interpolation positioning

When non-linear interpolation positioning, (it is non-linear motion mode from start point to end point, such as G02, G03) it is necessary to correctly confirm the path before programming, otherwise it may endanger the cutter, machine tool, workpiece and people.

(3) Axis of revolution movement function

When axis of rovolution moves, the workpiece installation mode, centrifugal force, rotary speed should be taken into consideration before programming, improper programme may cause axis of revolution overspeed that may throw off the workpiece and endanger the cutter, machine tool, workpiece and people.

(4) End surface constant linear velocity control.

When controlling constant linear velocity, maximum rotary speed should be correctly assigned because if workpiece radius on the constant linear control axle is close to zero, the main axle will be over speed, so improper command may endanger the cutter, machine tool, workpiece and people.

Operation related warning explanation

(1) Manual operation

When manually operating the mahine tool movement, you must watch the cutter and work piece's position, affirm that the move axle, move direction and feeding speed parameter choice are correct. Mishandling may endanger the cutter, machine tool, workpiece and people.

(2) Manual return to mahine tool zero point

To the mahine tool that needs manual return to zero point,it is a must to return to the zero point after power on otherwise the machine tool may excute unexpected movement that may endanger the cutter, machine tool, workpiece and people.

(3) Hand wheel feeding

When using manual handle feeding, special attention is needed if choosing the 100 times ratio operation because the speed of cutter, worktable movement will be much faster to endanger the cutter, machine tool, workpiece and people.

(4) Invalid ratio

When threading, the ratio adjustment may cause threading error, so the manual ratio adjustment is invalid.

Directory

Ⅰ System Introduction

WASHING CNC milling machines and machining centers CNC system CNC Technology Co., Ltd. Nanjing WASHING set ten years experience in the development of numerical control system, based on the original multi-generational mature products on the introduction of a new generation of high-performance cars milling CNC machining centers. System uses ultra-large-scale programmable logic solutions, with higher quality and process control system upgrade space. System can control digital AC servo drives and three-phase stepper drives; electronic gear function of the system so that the system can be used with any pitch screw Direct; pitch compensation so that the machine's accuracy test greatly simplified; display with 8.4 "and 10.4 "LCD color display, real-time tracking with graphical display of machined parts and coordinate character display function, interface design, richer humanity; graphic simulation more powerful, functional three-dimensional and two-dimensional plane simulated analog display. System architecture using integrated plastic die parts, handsome in appearance; convenient double U disk interface, the program can be achieved between the U disk and the system of mutual survival, and system software upgrades in the field.

1.1 WASHING CNC milling and CNC System Features

1.Control equivalent：	0.001mm
2.Programming ranges：	±99999.999mm
3.Fast forward speed：	60000mm/min（0.001mm Equivalent）
4.Processing speed max：	15000mm/min（0.001mm Equivalent）
5.Control/compounding axis number：	3/6
6.Program capacity：	2000k-8000k ， The maximum number of 600 stores
7.Program Management：	Full-screen editing (ISO code), rename, delete, serial input / output, USB input / output
8.Operating functions：	Auto, Manual, jog, handwheel, MDI, back to the machine zero, single-stage, pause, coordinate and tweens memory, any segment start
9.Communication：	Two USB ports
10.Display Resolution：	Color TFT LCD 800 × 600
11.Electronic gear：	65536/65536
12.Parameter：	Tool parameters, backlash compensation, system parameters, bit parameters, pitch error compensation
13. Provide processing timing piecework	
14. Support Open PLC ladder programming, macro programming, achieve a variety of functions and conditional transfer	

15. Bidirectional pitch error compensation, the soft limit and backlash compensation
16. Nose radius compensation (C-type tool compensation)
17. Block pre-reading and pre-reading fast synchronization
18. Tool change program allows users to redefine 、a fixed cycle or M function macro call
19. Free to customize the startup screen
20. Tracking real-time graphical display and simulation
21. The new optimization algorithm, to achieve high-speed high-precision control
22. Support Chinese and English operation interface, perfect information helpful tips, operating more concise
23. U disk support DNC, any data storage and U disk U disk expansion and CNC systems for mutual survival

Above WASHING CNC milling machines and machining centers CNC system common characteristics, because each system type and configuration differences, there are different functions. The following table:

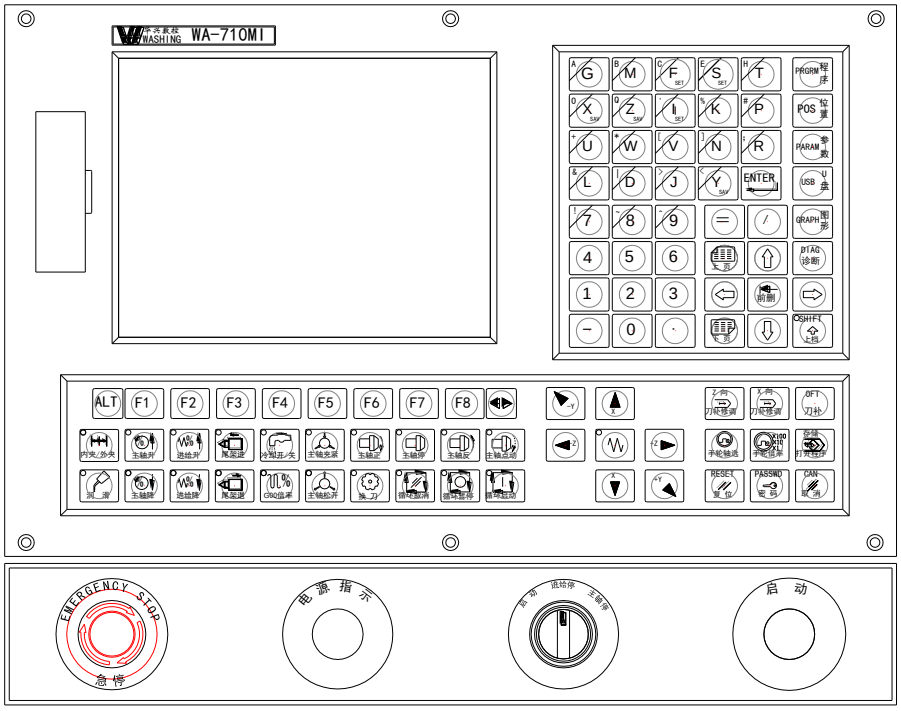
Model / Function	WA710MI	WA710MD	WA720M-4	WA740M-5	WA730M-6
Controlled axes	3	4	4	5	6
PLC execution time	4μ / Step	3μ / Step	3μ / Step	3μ / Step	3μ / Step
Display	8.4 inch	8.4 inch	8.4 inch	8.4 inch	10.4 inch
Output port	17 way	29 way	29 way	31 way	33 way
Input port	37 way	47 way	47 way	51 way	55 way
Program capacity	2000K	8000K	8000K	8000K	8000K
Serial Communications Interface	RS232	RS485	RS485	RS485	RS485
Remote Expansion I/O	Not supported	Supported	Supported	Supported	Supported

1.2 7-series milling machines and machining centers CNC systems division

Due to differences system functions and configuration, the 7-series milling machines and machining centers CNC system specific division:

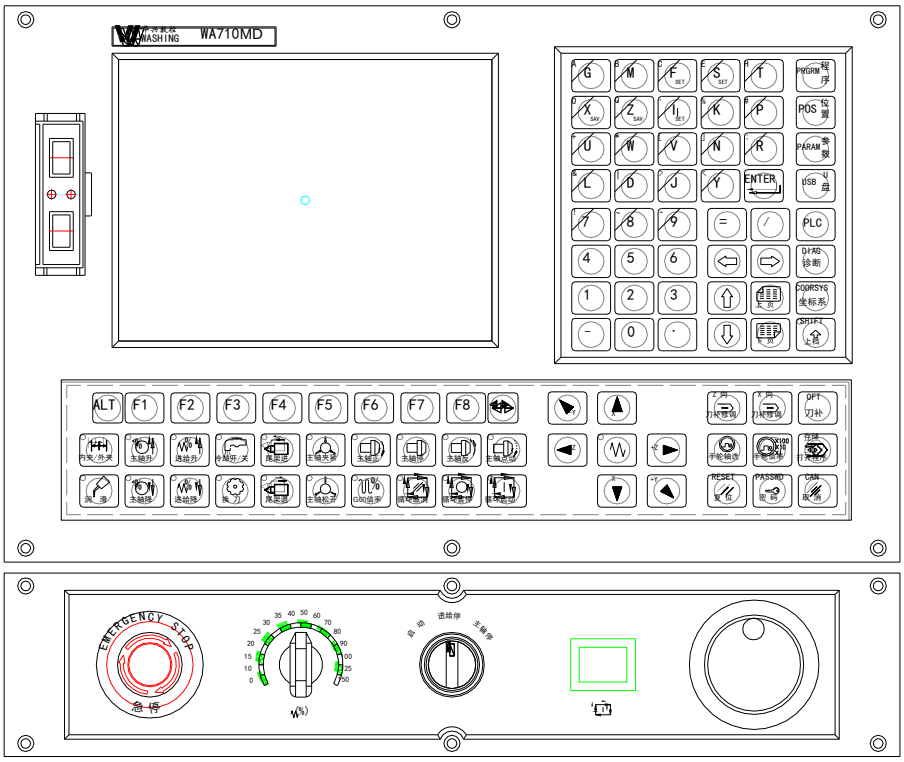
WA710MI	Fit 3-axis milling machine, horizontal panel structure
WA710MD	Fit 3-axis milling machine and 4-axis machining centers, horizontal panel structure
WA720M-4	Fit 3-axis milling machine and 4-axis machining centers, horizontal panel structure
WA740M-5	Fit 3-axis milling machine and 5-axis machining centers, Vertical panel structure
WA730M-6	Fit 3-axis milling machine and 6-axis machining centers, horizontal panel structure

1.WA710MI，fit 3-axis milling machine, horizontal panel structure



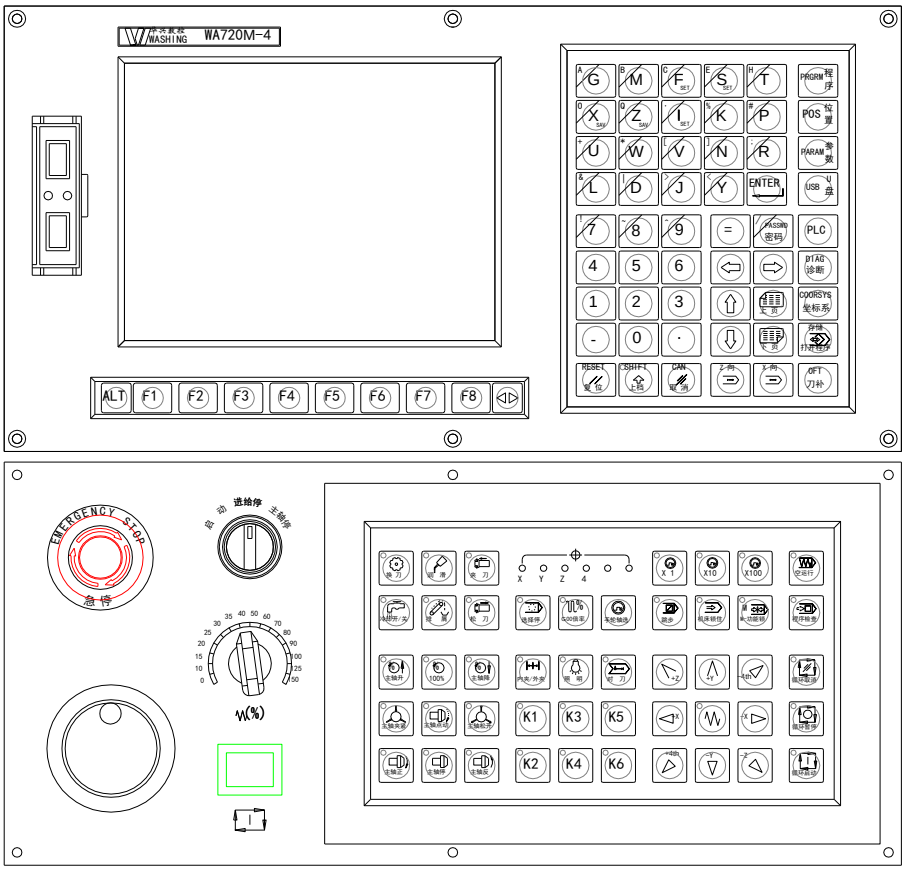
F1.1 WA710MI System Panel

2.WA710MD，fit 3-axis milling machine and 4-axis machining centers, horizontal panel structure



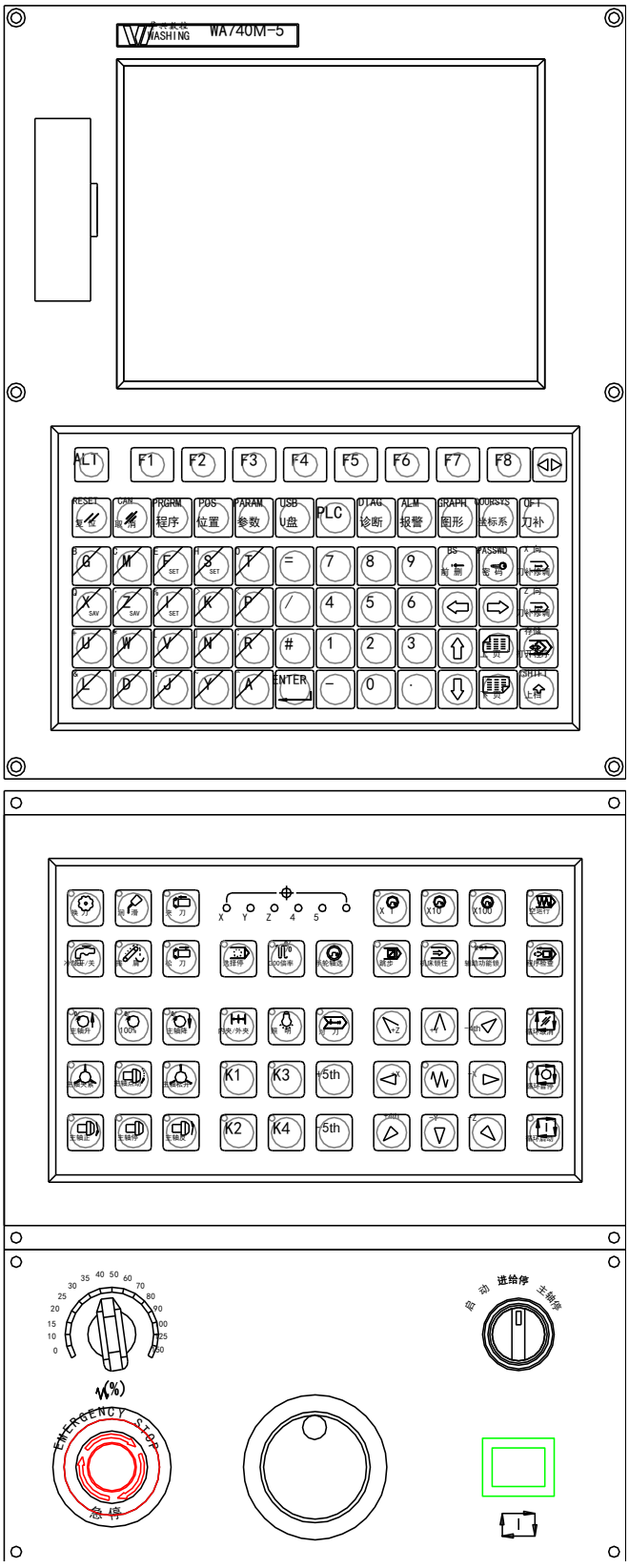
F1.2 WA710MD System Panel

3. WA720M-4， fit 3-axis milling machine and 4-axis machining centers, horizontal panel structure



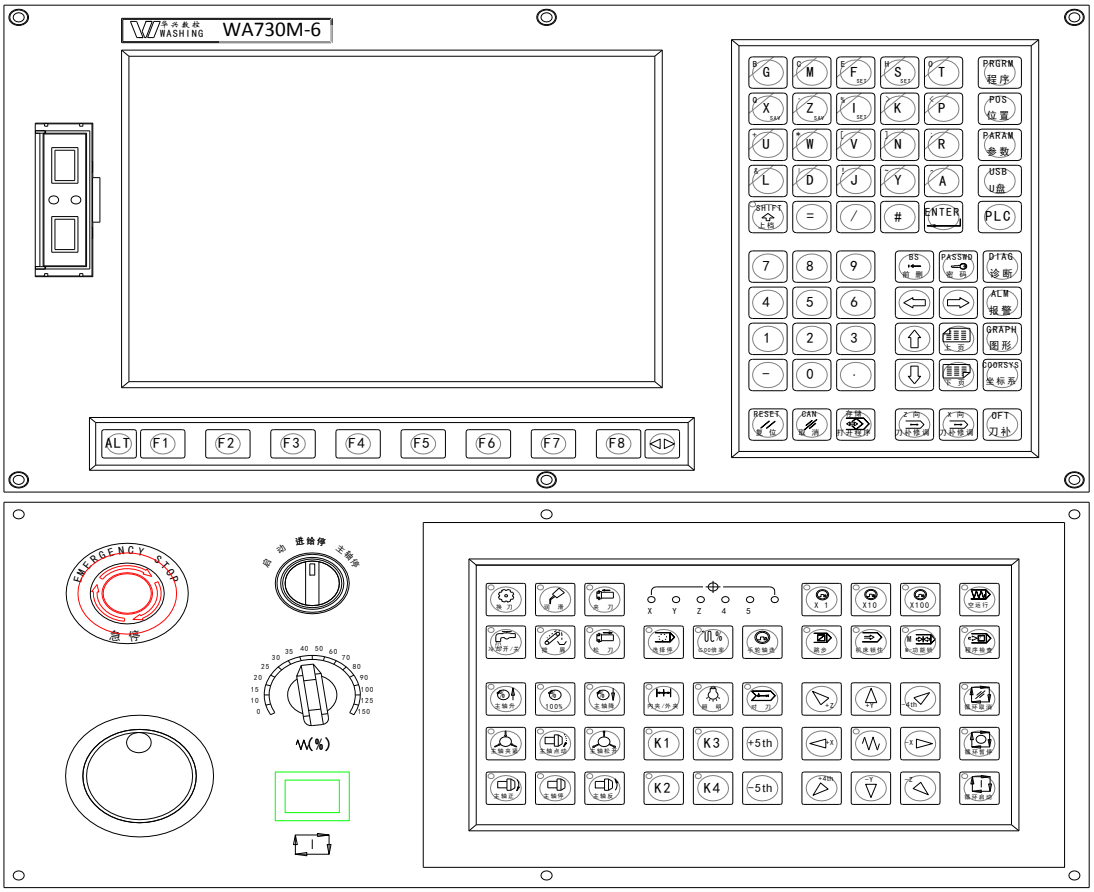
F1.3 WA720M-4 System Panel

4. WA740M-5，fit 3-axis milling machine and 5-axis machining centers, Vertical panel structure



F1.4 WA740M-5 System Panel

5. WA730M-6，fit 3-axis milling machine and 6-axis machining centers, horizontal panel structure



F1.5 WA730M-6 System Panel

1.3 PLC Introduction

Open PLC debug software, factory built-in standard PLC program, compatible with 31/99 Series CNC system.

There are 14 kinds of basic instruction as well as dozens of commonly used features instruction, self-defined functions M, S function and T functions to meet the machine tool plant secondary development requirements. Real-time PLC ladder diagnostic capabilities, to facilitate on-site commissioning.

1.4 Macro Program

Support access to local variables,global variables and system variables,users can use the arithmetic operations, bit operations, logic operations, and relational operator operator operator precedence computing, in the longer expression, can also beuse parentheses to change the order of operations. You can also use the conditional, conditional branch, unconditional transfer of structured programming statements.

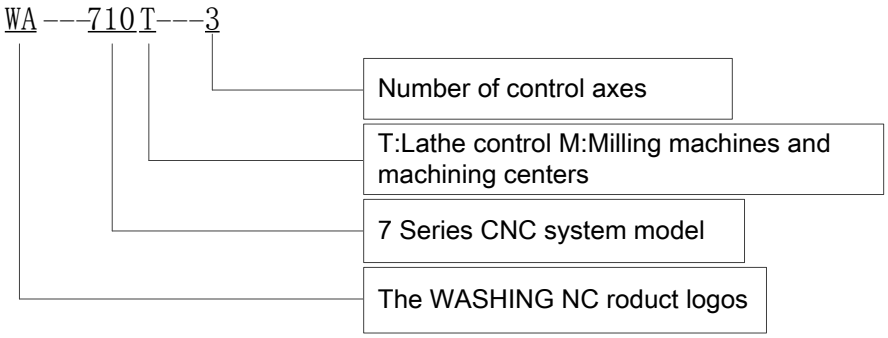
System with various function operations, such as: trigonometric functions, inverse trigonometric functions, square, square root, exponential and logarithmic functions, and so on. System data type uses 64-bit double-precision floating-point representation, to achieve maximum operational precision.

Support user-defined macro call, open the NC parameters, access to data in the PLC.

Powerful macro capabilities, preparation processing procedure more convenient and easy.

1.5 Selection and meaning

For example:



F1.6 Selection

II Programming

1 Summary

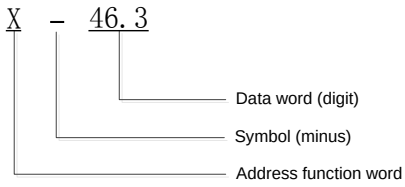
The so-called program segment format refers program segment writing rules, which include CNC machine to perform the functions and parameters of the functions required to perform a part program is composed of several program segment, each program segment has a different function from the word composition, the system commonly used functions words as follows:

Table2.1

FUNCTION	ADDRESS	RANGE	NOTE
PROGRAM NUMBER	p、n	0～9999	DEFINE PROGRAM NUMBER,SEGMENT NUMBER
SEGMENT NUMBER	N	0000～9999	DEFINE SEGMENT NUMBER
PREPARATORY FUNCTION	G	00～99	INSTRUCTION MOTION MODE
COORDINCATE CHARACTER	X、Z、I、K、R、L	±0.001～±99999.999	MOTION INSTRUCTION COORDINATES, THECENTER OF ARC COORDINATES, PITCH,RADIUS,CIRCULATION TIMES.
FEEDING SPEED	F	1～6000mm/min	FEEDING SPEEDING INSTRUCTION
SPINDLE FUNCTION	S	0～5000RPM	MAIN AXLE ROTARY SPEED INSTRUCTION
TOOL FUNCTION	T	1～8	CUTTER INSTRUCTION
AUXILIARY FUNCTION	M	0～99	AUXILIARY INSTRUCTION

The system does not require each segment to include the above listed instructions, but the instruction should be arranged in certain rules in each segment. Each function character may stand different meaning in different segment, see details in specified instruction.

The system uses variable segment instruction format, which means the segment length varies according to the character number and character length. The program segment is built up by one or more instruction characters, instruction character normally is built up by address characters and the data and symbols behind address characters, for example:



F2.1

This kind of program segment format to address functional character-led, followed by a string of digits, a number of words to form a program segment. Those words in a program segment has stated changes do not occur in the program segment is still valid and can not be rewritten. Dimensions words, you can write only valid number, does not require a fixed number of bits per word to be filled. Only negative was to add "-" sign, a positive number usually omit the "+" sign.

On the prepare function and auxiliary function, if the instruction code is less than 10, you can write only one value. G00 can be written as G0,G09 can be written as G9; M01 you can write to M1,M08 to write to the M8. Similarly, program number, P0001, P001, P01 is equivalent to P1.

For example： N0420 G03 X70 Z-40 I0 K-20 F100

n the previous segment of program:

N、G、X、Z、I、K、F	；all address function character
N	；segment number
G03	；Preparative function, which can be written as G3
X Z I K	；coordinate address
F	；feeding amount
" = " " - "	；symbols
03、70、-40、0、-20、100	；numeric character

In program segment, the address function English characters can be divided into dimensional address and non-dimensional address.

Dimensional address are indicated by following characters：X、Z、I、K、R

Non-dimensional address are indicated by following characters：N、S、T、G、F、M、P、L

A complete program is built up of program name, program segment number and relevant symbols. See sample program below:

```
N0010  G00  Z2
N0020  S1200  M03
N0030  G01  Z-1  F300
N0040  G91  X20  Y20
N0050  X30  Y10
N0060  X30
N0070  G03  X15  Y15  I0  J15
N0080  G02  X15  Y15  I15  J0
N0090  G01  Y10
N0100  X-5
N0110  G02  X-30  Y0  I-15  J0
```

```
N0120  G01  G90  X20  Y20
N0130  X0   Y0
N0140  G01  Z5
N0150  M02
```

In general situation, a segment is an operation in workpiece processing,numeric control program is a segments sequence that is stored in the memory. When processing work piece, these sentences are whole numerated from the memory and one-off compiled to executive data form then implement.

Program segment number is to mark out each segment that compose a program, it is headed by letter N and followed by numbers (0000-9999), the segment number should be in the head position of each segment and can be generated by segment number automatic generator.(see program edit function) In a program, a segment may use any number between 0000 and 9999, but generally, the segment number is in increment way according to the execution sequence in a program. To insert new segment in case needs, we suggest not assign the segment number in continuous way when programming, if programming on CNC panel, we suggest the segment number counted by 10 so different segment number can be assigned to newly inserted programme.

2 Preparatory function（G function）

The preparative function is programmed with G following two digits, G function is also called preparation function command, which is used to define geometry and CNC operating status of track. The functions of any numerical control device all include fundamental functions and optional functions. The fundamental functions are requisite for system, and the optional functions are for user to select according to machine tool features and applications. It's recommended to program after understanding the machine specifications first. The machine tool can configure control function according to numerical control system, namely the machine tool may not implement all functions of numerical control system.

The whole G functions of system are as follows:

Table2.2

Type	G Code	Function	Remark
Modal	G00	Fast locating	
Modal	G01	Line interpolation	
Modal	G02	Clockwise circle interpolation	
Modal	G03	Counterclockwise circle interpolation	
	G04	Time delay	
	G09	Servo accurate positioning pause	
	G10	Cancel Segment mirror image	
	G11	Segment mirror image to Y axes	
	G12	Segment mirror image to X axes	
	G13	Segment mirror image processing to zero point	
Modal	G17	Choose XOY plane	Default when Power on

Type	G Code	Function	Remark
Modal	G18	Choose XOZ plane	
Modal	G19	Choose YOZ plane	
	G20	Independent sub-program call	
	G22	Independent sub-program definition	
	G24	Return call program after sub-program ends	
	G25	Jump processing	
	G26	Transfer processing (sub-program call in the program)	
	G27	Endless cycle	
Modal	G30	Magnification and minification cancel	Default when Power on
Modal	G31	Magnification or minification	
	G35	Skip function	
Modal	G40	Cancel cutting tool radius compensation	Default when Power on
Modal	G41	Cutting tool (tool nose) radius left compensation	
Modal	G42	Cutting tool (tool nose) radius right compensation	
Modal	G43	Cancel cutting tool length compensation	Default when Power on
Modal	G44	Cutting tool length compensation	
Modal	G47	Short straight line speed Auto-transitions	
Modal	G48	Cancel a short straight line speed automatic transition	Default when Power on
Modal	G54～G59	Workpiece coordinate system selection	
Modal	G61	Follow-up procedure quick cleaning corners	
Modal	G62	Current program quick cleaning corners	
Modal	G64	Cancel the fast corner cleaning function	Default when Power on
	G65	Custom macro Program Calling	
Modal	G66	Cancel G67	Default when Power on
Modal	G67	F to the longest axis when line interpolation	
	G73	High speed deep hole processing circulation	Macro
	G74	Return to reference point (mechanical origin)	
	G75	Return processing start point based on machine tool coordinate	
	G76	Return processing start point based on workpiece coordinate	
	G78	Precision boring circulation	Macro

Type	G Code	Function	Remark
	G81	Central hole drilling circulation	Macro
	G82	Central hole drilling circulation with pause	Macro
	G83	Deep hole drilling circulation	Macro
	G84	Metric rigidity threading cycle	
	G85	English system rigidity threading cycle	
	G86	Hole boring circulation(automatic return)	Macro
	G87	Counter hole boring circulation	Macro
	G89	Hole boring circulation with suspension	Macro
Modal	G90	Absolute value mode programming	Default when Power on
Modal	G91	Programming with incremental mode	
	G92	Modify coordinate zero point position of workpiece coordinate system (change workpiece coordinate value of tool nose)	
Modal	G96	Effective for constant linear speed cutting	
Modal	G97	Cancel constant linear speed cutting	Default when Power on
Modal	G98	Cancel feed of every rotation	Default when Power on
Modal	G99	Set feed of every rotation	

P=variable parameter

Note: the so-called modal, when after the g-function is programmed, it remains valid until being replaced by another modal function of the same nature. For macro instruction, this description of function is system factory Shi scheduled meaning macro program body of function, user can write own of macro program body to alternative macro instruction of scheduled meaning function (the scheduled meaning macro program body see 5.4 system macro instruction), such as user without this system for deep hole processing cycle, but wanted to with macro instruction G83 implementation in circle Shang punch of function, is reference 5.3.5 macro program using for example, can operation following: to 5.3.5 examples in of N10 child program edit deposited system in the, to its take p beginning of any name, such as P30; 183th modify system parameters, as given above for the names of 30 of digits (the macro instructions G73, the corresponding system parameters for the 173th, macro, G78 for, and so on. If returns to modify the parameters of a value of 0, the macro instruction is implemented is factory predefined function). At this time the two G20N10 in the example are 5.3.5 into G83, using modified procedures of processing results and 5.3.5 are the same.

The following, the above g features explained.

2.1 G00——Fast positioning

Format：G00 X_ Y_ Z_

Description：

(1) All programme assigned axles will move independently at the velocity defined in No.06~ No.09 system parameters, the axle will stop once finishing the programmed value

and the other axle will continue.

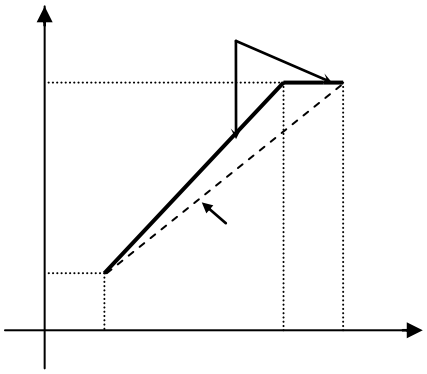
(2) The actionless coordinates do not need to programme.

(3) We can use absolute value or incremental value to describe the target point coordinate value, 6 digits number(including symbol digit) before decimal point and 3 digits number after decimal point are permitted, in case positive number, the symbol "+" can be omitted.(This rule applies to all coordinate system programming).

For example： in Figure 2.2, the program is：

In absolute value format： G00 X170 Y150

In incremental value format：G91 G00 X160 Y140



F2.2 G00 application instructions

During the execution of G00 command, the trajectory of the tool is not necessarily a straight line, as shown in F2.2, if the system parameter setting X-axis and Y-axis rapid traverse rate the same, the first X-and Y-cutter while quickly moved to point A , then X to be fast to point B. Therefore, when using the G00 programming, pay attention to whether the tool and the workpiece or fixture interference occurs, in order to avoid a collision. Common practice is to move to a safe height Z-axis, add carefully executed G00 command.

2.2 G01——Linear interpolation

Format：G01 X_ Y_ Z_ F_

G01 X_ Y_ F_

G01 Y_ Z_ F_

Description：

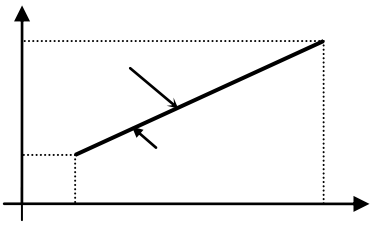
(1) Processing began at a time, in G01 State automatically.

(2) Do not sport coordinates can be omitted.

(3) Coordinates of the destination point can written with absolute or incremental values.

(4) G01 when processing, its f feed rate according to the given value run, F:1~60000mm/min.

For example： the program follows the trajectory shown in Figure 2.3：



F2.3 G01 application instructions

In absolute value format：

G01 X210 Y120 F150

In incremental value format：

G91 G01 X194 Y104 F150

2.3 G02——Clockwise interpolation

Format：G02 X_ Y_ I_ J_ F_

G02 X_ Y_ R_ F_

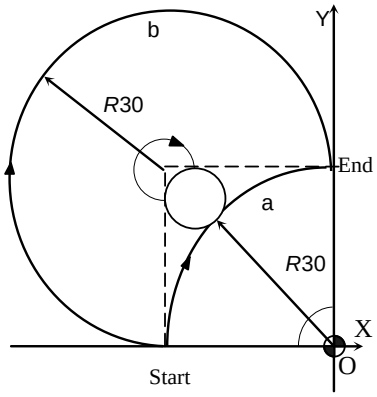
Description：

(1) When X and Y are at G90, the circular arc end point coordinates are absolute coordinates relative to programming zero point. When in G91 form, the circular arc end point coordinates are incremental value to the circular arc start point. In both G90 or G 91 form, I and J are arc center coordinates relative to circular arc start point, I stands for X axes direction while J stands for Y axes direction. The arc center coordinates can not be omitted when processing arc interpolation unless programme use R(arc radius).

(2) G02 instruction can directly programme trans-quadrant circle, full circle, etc,(R programme can not be applied in entire circle)

Note: The system can automatically adjust gap compensation to the circle if it crosses quadrants. It will cause cutting mark on the workpiece if the parameter section does not contain the gap compensation parameter or the disparity value between gap compensation parameter and machine tool reverse direction actual gap value is too big.

(3) R is the workpiece unilateral r the radius of the arc. R is the number sign, "+" represents the arc angle less than 180 °; "-" Represents the arc angle greater than 180 °.



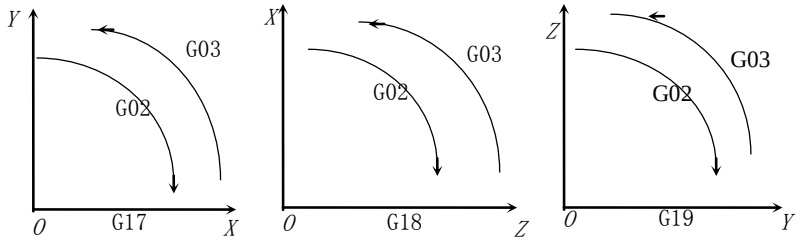
Clockwise circular

Circular Instruction

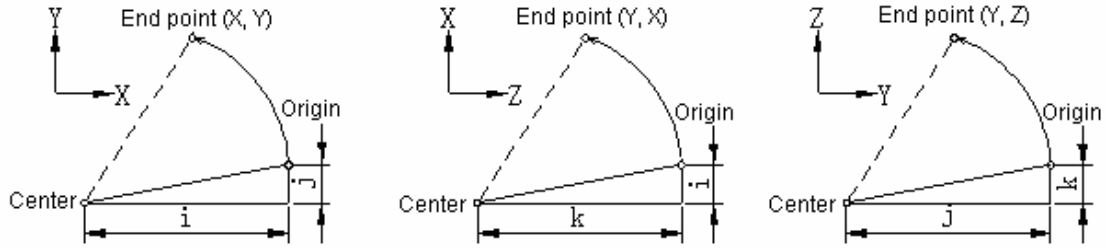
(i) Arc a
G91 G02 X30 Y30 R30 F300
G91 G02 X30 Y30 I30 J0 F300
G90 G02 X0 Y30 R30 F300
G90 G02 X0 Y30 I30 K0 F300
(ii) Arc b
G91 G02 X30 Y30 R–30 F300
G91 G02 X30 Y30 I0 J30 F300
G90 G02 X0 Y30 R–30 F300
G90 G02 X0 Y30 I0 J30 F300

F2.4

(4) XOY, ZOX, arc YOZ plane without defining interpolation plane (G17, G18, G19).

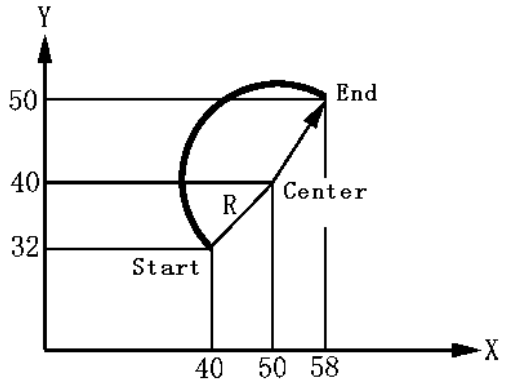


F2.5 G02 and G03 to select different planes



F2.6 The use of I、J or K in deferity planes

Example 1：processing as in Figure 2.7 AB arc programs are as follows：



F2.7

Absolute value：

G90 G02 X58 Y50 I10 J8 F150 (Center coordinates programming)

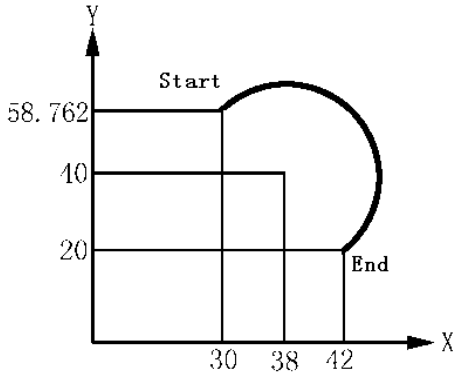
G90 G02 X58 Y50 R12.81 F150 (radius programming)

Incremental modal：

G91 G02 X18 Y18 I10 J8 F150 (Center coordinates programming)

G91 G02 X18 Y18 R12.81 F150 (radius programming)

Example 2：processing as in Figure 2.8 AB arc programs are as follows(>180º) ：



F2.8

Absolute value：

G02 X42 Y20 I8 J－18.76 F50 (Center coordinates programming)

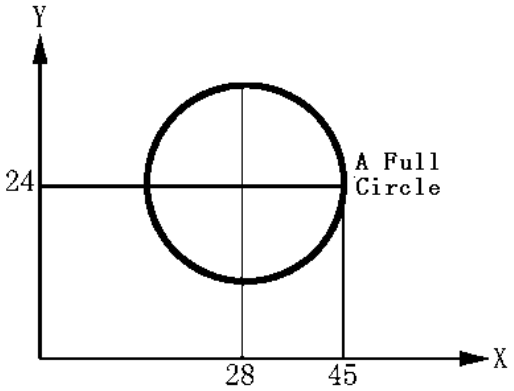
G02 X42 Y20 R－20.40 F50 (radius programming)

Incremental modal：

G91 G02 X12 Y－38.76 I8 J－18.76 F50 (Center coordinates programming)

G91 G02 X12 Y－38.76 R－20.40 F50 (radius programming)

Example 3：processing as in Figure 2.9 as follows(radius programming is forbid with a full circle) ：



F2.9

Absolute value：

G02 X45 Y24 I－17 J 0 F50 (Center coordinates programming)

Incremental

G91 G02 X0 Y0 I－17 J0 F50 (Center coordinates programming)

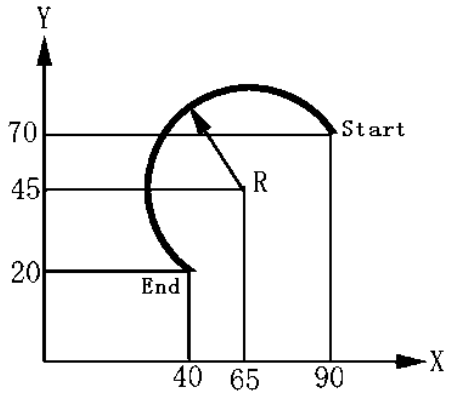
2.4 G03——Counterclockwise interpolation

Format：G03 X_ Y_ I_ J_ F_

G03 X_ Y_ R_ F_

Description：G03 instructions in programming, apart from arcs in the opposite direction, remaining with G02 instructions the same.

Example 1：processing as in Figure 2.10 AB arc programs are as follows：



F2.10

Absolute value：

G90 G03 X40 Y20 I-25 J-25 F100 (Center coordinates programming)

G90 G03 X40 Y20 R35.36 F100 (radius programming)

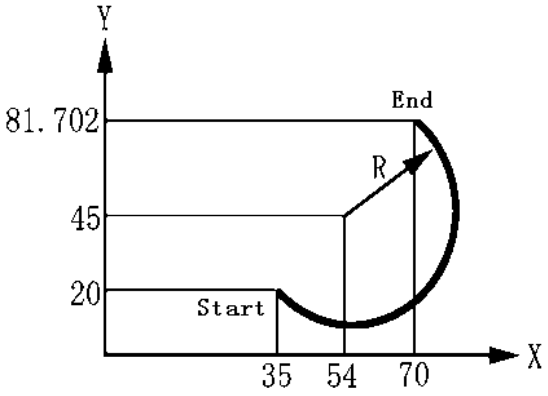
incremental：

G91 G03 X-50 Y-50 I-25 J-25 F100 (Center coordinates programming)

G91 G03 X-50 Y-50 R35.36 F100 (radius programming)

G03 U-50 V-50 R35.36 F100

Example 2：processing as in Figure 2.11 AB arc programs are as follows：



F2.11

Absolute value：

G03 X70 Y81.70 I19 J30 F50 (Center coordinates programming)

G03 X70 Y81.70 R-35.51 F50 (radius programming)

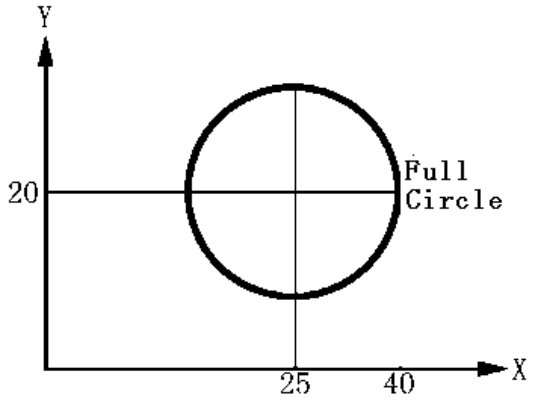
incremental：

G91 G03 X35 Y61.70 I19 J30 F50 (Center coordinates programming)

G91 G03 X35 Y61.70 R-35.51 F50 (radius programming)

G03 U35 V61.7 R-35.51 F100

Example 3：processing as in Figure 2.12 AB arc programs are as follows：



F2.12

Absolute value：

G03 X40 Y20 I-15 J0 F50 (Center coordinates programming)

incremental

G91 G03 X0 Y0 I-15 J0 F50 (Center coordinates programming)

Helical Interpolation：

Description：

It is used to move the tool to a position specified from current position in a helical path.

Only two axes of circular interpolation can be linked for controlling tool movement along with the arc on the selected plane in any time. If the 3rd axis is specified simultaneously in linear interpolation mode, it will be linked by linear interpolation type to constitute helical interpolation. G02 movement path is CW from start to end points. G03 movement path is CCW from start to end points

Arc interpolation in XY plane, Z axis linear interpolation linkage

Format 1：G17 G02 X_ Y_ I_ J_ F_ Z_ K_ ;center CW

Or G17 G02 X_ Y_ R_ F_ Z_ K_ ;radius CW

Format 2：G17 G03 X_ Y_ I_ J_ F_ Z_ K_ ;center CCW

Or G17 G03 X_ Y_ R_ F_ Z_ K_ ;radius CCW

Arc interpolation in ZX plane, Y axis linear interpolation linkage

Format 1：G18 G02 Z_ X_ K_ I_ F_ Y_ J_ ;center CW

Or G18 G02 Z_ X_ R_ F_ Y_ J_ ;radius CW

Format 2：G18 G03 Z_ X_ K_ I_ F_ Y_ J_ ;center CCW

Or G18 G03 Z_ X_ R_ F_ Y_ J_ ;radius CCW

Arc interpolation in YZ plane, X axis linear interpolation linkage

Format 1：G18 G02 Y_ Z_ J_ K_ F_ X_ I_ ;center CW

Or G18 G02 Y_ Z_ R_ F_ X_ I_ ;radius CW

Format 2：G18 G03 Y_ Z_ J_ K_ F_ X_ I_ ;center CCW

Or G18 G03 Y_ Z_ R_ F_ X_ I_ ;radius CCW

The fellow explain is based on plane XOY.

Note：

(1) Those of X、Y、I、J、R、F is like circular interpolation instruction G02 and G03.

(2) Z is the length of Helical in Z axle，K is helical pitch length.

(3) Z=K*（n+length of arc/length of circumference）……n must be integral number

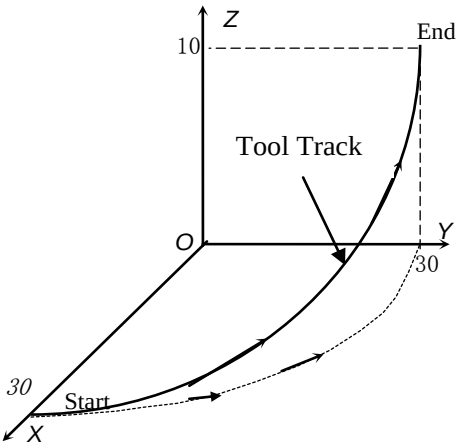
and >=0.

The F value is circular interpolation speed. An interpolation of linear and circular arc.

Helical interpolation is XOY plane circular interpolation, Z-axis linear motion. Thread cutting machining or entity large radius of the cam can move the tool to achieve the spiral line.

Command speed is the speed at the arc tangent direction. Thus, the linear speed of the Z axis using the following equation:

f=F*helical pitch length/length of circumference



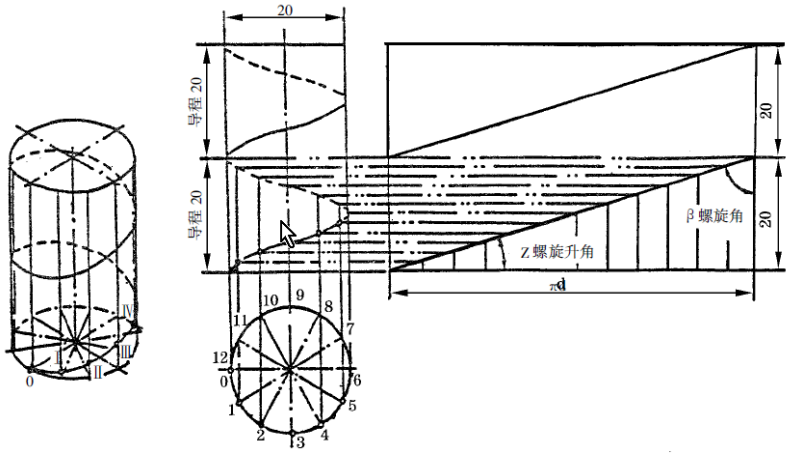
At G91 status:
G91 G17 F300
G03 X-30 Y30 R30 Z10 K40

At G90 status:
G90 G17 F300
G03 X0 Y30 R30 Z10 K40

F2.13 Helix Programming

Example: Fig length L as helix diameter 20mm, helix 40mm, pitch 20mm. Procedures are as follows:

G03 X0 Y0 I-10 J0 Z40 K20 F200



F2.14

2.5 G04——Delay

Format: G04 K××.××

Description:

(1) Procedures delayed programming behind the k value (in seconds), continue to run down, delay range 0.01~65.5 seconds.

(2) G04 instructions cannot have other g feature in the program.

2.6 G09——Angle-specified stop of feeding

Format：G09

Description：

(1) G09 are only valid for this program, at the end of this program CNC waiting for accurate positioning of servo signal, and then under start a program, to avoid sharp corners processed into fillets.

(2) G09 wait time is determined by system parameters, 72th, timeout error is 57.

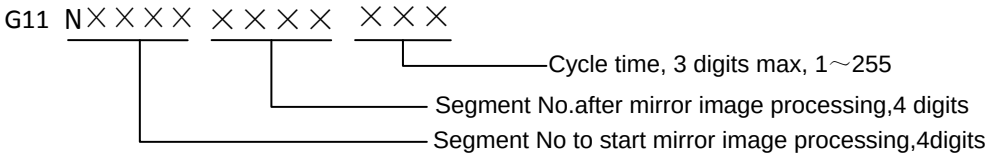
2.7 G10——Cancel Symmetrical mirror image

Format：G10

Description：To cancel the effect of G11、G12 and G13.

2.8 G11——Symmetrical mirror image to Y axes

Format：



F2.15

Description：

(1) The G11 instruction processes the programme segment between the segment numbers in reverse X axes direction and circulates certain times defined in the segment.

(2) The mirror image processing start segment number and conclusion segment number are separated by decimal point, the start segment number must ahead of the conclusion segment number.

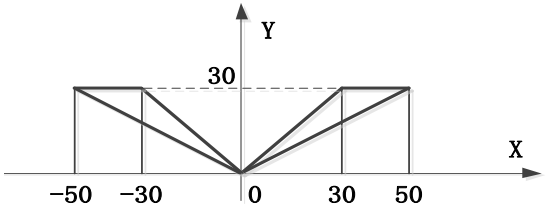
(3) Circulation time is defined by the 3 digits integer after the second decimal point.It will circulate one time if the circulation time is omitted.

(4) When the mirror image processing ends, the next processing segment is the one after G11 segment.

(5) G11 can not take as end segment. If the G11 segment take as the end segment,it must followed by another segment such as MO2.

(6) G11 usage is explained in the following example,（the serial number in the chart is the cutter center motion path sequence）.

(7) Note：Any other processing transition instruction can not be included in the G11 defined mirror image segment, such as subprogramme transition and so on.



F2.16 G11 application instructions

Example：

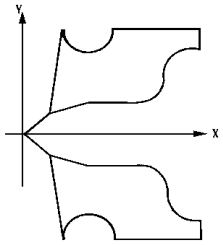
```
N0010  G01  Z-2  M03  S1000  F100
N0020  G91  G01  X30  Y30
N0030  X20
N0040  G01  G90  X0  Y0
N0050  G11  N0020.0040
N0060  M02
```

2.9 G12——Symmetrical mirror image to X axes

Format： G12 N××××.××××.×××

Description： The G12 instruction the programme segment reverse processing along Y axes between the segment numbers while the other details are the same as G11 instruction.

G12 usage is explained in the following example：



F2.17 G12 application instructions

We can see the cutter center motion path sequence that is defined by the programme in the chart：

Sample programme segment as below：

```
N0010  G01  Z-1  F1000  S1000  M03
N0020  G91  G42  T01  X20  Y20
N0030  X30  Y10
N0040  X30
N0050  G03  X15  Y15  I0  J15
N0060  G02  X15  Y15  I15  J0
N0070  G01  Y10
N0080  X-50
N0090  G02  X-30  Y0  I-15  J0
N0100  G01  G90  X20  Y20
N0110  G40  X0  Y0
N0120  G12  N0020 . 0110
N0130  M02
```

2.10 G13——Symmetrical mirror image to zero point

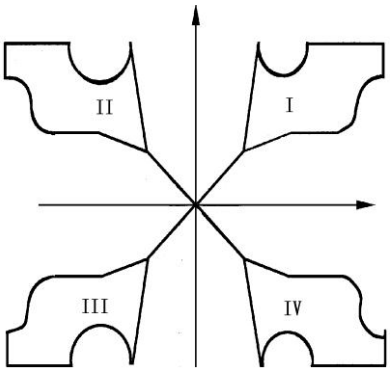
Format： G13 N××××. ××××. ×××

Description：

G13 instruction is the result when executing both G11 and G12 instructions,the other

explanations are the same as G11 and G12.

G13 instruction programming method is explained in the following example：



F2.18 G13 application instructions

```
N0010  G01  Z-1 F100 S1000 M03
N0020  G91  G42  X20  Y20
N0030  X30  Y10
N0040  X30
N0050  G03  X15 Y15 I0 J15
N0060  G02  X15 Y15 I15 J0
N0070  G01  Y10
N0080  X-50
N0090  G02  X-30 Y0 I-15 J0
N0100  G01  G90  X20  Y20
N0110  G40  X0  Y0
N0120  G11  N0020.0110
N0130  G12  N0020.0110
N0140  G13  N0020.0110
N0150  G00  Z5
N0160  M02
```

To process the above shapes in the chart, the programme use segments of N0020-N0110 to process the shape I, then use G11 instruction call segments of N0020-N0110 to process the shape II, use G12 instruction call segments of N0020-N0110 to process the shape IV and use G13 instruction call segment of N0020-N0110 to process the shape III.

2.11 G17、G18、G19——Interpolation plane selection

Format：

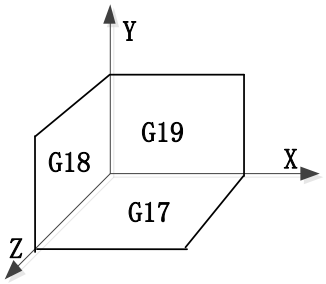
G17（OR G18，OR G19）

G17 Selection of XOY plane interpolation

G18 Selection of XOZ plane interpolation

G19 Selection of YOZ plane interpolation

G17, G18, G19 plane interpolation diagram as follows：



F2.19 Interpolation plane selection

Description：

(1) The plane definition only applies in the following situations：

1、To define cutter radius compensation plane.

2、To define spiral arc plane when executing spiral arc interpolation .

(2) The defined plane can not be changed when executing G41,G42,G43 and G44 cutter compensation.

(3) It is not necessary to define a plane in the general path interpolation system that can automatically differentiate interpolation plane.

(4) It is not necessary to define a plane in three-axis simultaneous motion interpolation situation.

(5) When system is powered on, it is automatically set to G17 status.

2.12 G20/G65——Sub-program call

2.12.1 No parameters type subroutine calls G20

Format： G20 Nxx.xxx

Description：

(1) N after the first 2 digits of the subroutine to be called the name of the program, allowing 2-digit, 3 digit after the decimal point of the loop that calls the number, available from 1~255.

(2) The program shall not appear content other than described above.

(3) Different repeatable nested subroutine call 10 times, but not call itself.

The main program： P10

N0010 S1000

N0020 G20 N05. 2

N0040 M02

Subprogram： N05

N1000 G22 N05

N1005 G01 G90 X0 Z0 F100

N1010 X－47.65 F200

N1050 G24

2.12.2 G65 macro variable type subroutine call

Format： G65 Pxx.xxx A×××× B×××× …… Z××××

Description：

(1) N after the first 2 digits of the subroutine to be called the name of the program, allowing 2-digit, 3 digit after the decimal point of the loop that calls the number, available

from 1~255. Those of A×××× B×××× …… Z××××, A~Z as a variable, the actual values passed to the subroutine named macro parameters.

Table2.3 Macro parameters and local variables CrossRef

A	1		H	8		O	15		V	22
B	2		I	9		P	16		W	23
C	3		J	10		Q	17		X	24
D	4		K	11		R	18		Y	25
E	5		L	12		S	19		Z	26
F	6		M	13		T	20			
G	7		N	14		U	21			

Warning：The letter with gray background should be used fewer.

(2) If macro parameter is not defined when subprogram is called, then the value of macro parameter will be NULL variable.

(3) Child participation in the program variables in G20 calls, must be given specific values.

(4) The program shall not appear content other than described above.

(5) Different repeatable nested subroutine call 10 times, but not call itself.

To achieve the diameter of the rod 200, the material out of one end of the car hemisphere, use G65 can be programmed as follows：

```
N0010 G92 X200 Z120 F300          ；Positioning to the starting position
N0020 M3 S1800
N0030 G65 P8000 A100 B10          ；A subroutine call to the bar P8000 radius circle
                                    B is cutting the amount of each car
N0040 M02

;P8000 sub-program
N0010 #3=#2                       ；# 3 as the current radius
N0020 G1 U[-2*[#3]]               ；move to arc start point
N0030 G3 U[2*[#3]]W[-#3]R[#3]     ；Inverse circular interpolation
N0040 G0 W[#3]                    ；back to arc start point
N0050 #3=#3+#2                    ；increase radius
N0060 IF[#3<=#1]GOTO20            ；if done over?
N0070 G24
```

2.13 G22——Sub-program definition

Format： G22 N××

Description：

(1) Subroutine name begins with N, N-digit after the subroutine name.

(2) Code G22 N × × block, do not have other instructions appear.

(3) G22 and G24 in pairs to form a complete subroutine body.

(4) Internal subroutine parameter data have two kinds of formats：

1) A constant, the data for a given programming constants, i.e., 0 to 9.

2) Parametric format, program function number and parameters are available variable represents the digital section, and specific values of the variables passed by

the calling subroutine block. The system can handle 10 variable parameters： P0 P1 ..... P9.

(5) Subroutines can not have internal transfer process, mirroring processing.

2.14 G24——Completion and returning of sub-program

Format： G24

Description：

(1) G24 indicates completion of program and returns to the next segment of program calling this sub-program.

(2) G24 and G22 appear in pair.

(3) The segment of G24 disallows other instructions to appear.

Note: If you call a subroutine P parameter is not defined, then the value of the subroutine parameter P is uncertain.

2.15 G25——Jump processing

Format：G25 N××××. ××××. ×××

Description：

(1) The cycle body which is defined by this format is the defined block (including these two segments) between two block numbers following N, and the digit defines the call number of this block, from 1 to 255, and 1 will be considered without compiling.

(2) The next segment processing program after G25 instruction completes is the one of jump processing block.

(3) Other instructions are prohibited in the G25 block.

Example：

```
N0010  G92    X50    Y100
N0020  G25    N0040.0060.02
N0030  G00    X10    Y20
N0040  G01    X40    Y80    F300
N0050  Y60
N0060  G00    X50    Y100
N0070  G04    K3
N0080  M02
```

The processing sequence of program above is：

N0010→N0020→N0040→N0050→N0060→N0040→N0050→N0060→N0070→N0080

2.16 G26——Transition processing

Format：G26 N××××. ××××. ×××

Description：The transition processing instruction completes, the next processing section is the next one of G26 N××××. ××××. ×××, which is the difference from G25, and the rest are same as G25.

Example：

```
N0005  S800  M03
```

```
N0010  G26  N0050.0080.02
N0020  G4   K2
N0030  G01  X2     F20
N0040  G00  X0     Z0
N0050  G92  G90  X0   Z0
N0060  G01  Z-20  X20  F300
N0070  M00
N0080  Z-40
N0090  Z-60  X0
N0100  M02
```

The processing sequence of program above is:

N0005→N0010→N0050→N0060→N0070→N0080→N0050→N0060→N0070→N0080→N0020→N0030→N0040→N0050→N0060→N0070→N0080→N0090→N0100

2.17 G27——Endless cycle

Format 1：G27 N××××. ××××

Format 2：G27 N××××. ××××. ×××

Description：

(1) The block between the first and second block numbers after N is the interval of endless cycle, once entering G27 status, the system will infinitely repeat carrying out the operating track which is defined by this block.

(2) In order to guaranty that the coordinate doesn't offset when every cycle starts, the block is required to be the sealed track, else the start point will shift at every start, and finally exceeds working table.

(3) In Format 2,the cycle body only run a few time define as the third number.

2.18 G30——Magnification and minification cancel

Format：G30

Description：When the G31 magnification and minification is implemented, G30 cancels effect of G31.

2.19 G31——Magnification or minification

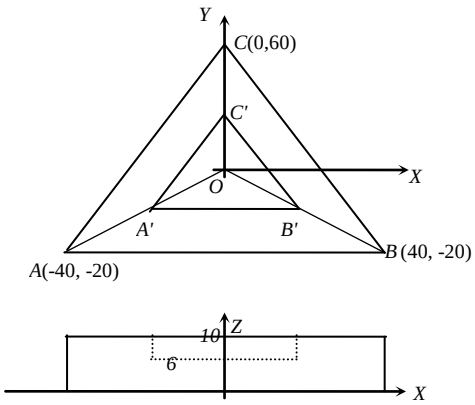
Format：G31 Kx××. ××

Description：

(1) The scope of multiply factor is 0.001-65.5, namely K0.001-K65.5.

(2) The effect of multiply factor is to magnify or minify K times for the various segments' size of processing track evenly.

(3) The multiply factor has no effect on the cutting tool.



F2.20 ⊿ABC Zoom indicate

N02 G92 X0 Y0 Z20
N04 M03 S600 F300
N06 G00 Z10
N08 G00 X-40 Y-20
N10 G01 X40
N12 X0 Y60
N14 X-40 Y-20
N16 G00 X0 Y0
N18 Z6
N20 G31 K0.5
N22 G26 N08.16
N24 G30
N26 G0 Z20
N28 M2

2.20 G35——Skip function

Format：G35 X_ Y_ Z_ A_ B_ C_

Description：After the G35 command, like G01 Linear interpolation can be commanded. When the instruction is executed, if the input signal external to jump, then interrupt the execution of the instruction, the next block is executed instead. This function can be used to measure the size of the workpiece.

Skip signal is turned on when the coordinate values can be used for user macro program, because these coordinate values are stored # 5060 to # 5065 in custom macro system variables.

2.21 Tool radius compensation G40/G41/G42

Format：

G40 G01 X_ Y_
G41 G01 X_ Y_
G42 G01 X_ Y_

Description：

(1) G41 specified on the left to compensate tool movement direction. G41 cutting

direction in FIG. 2.21 (a).

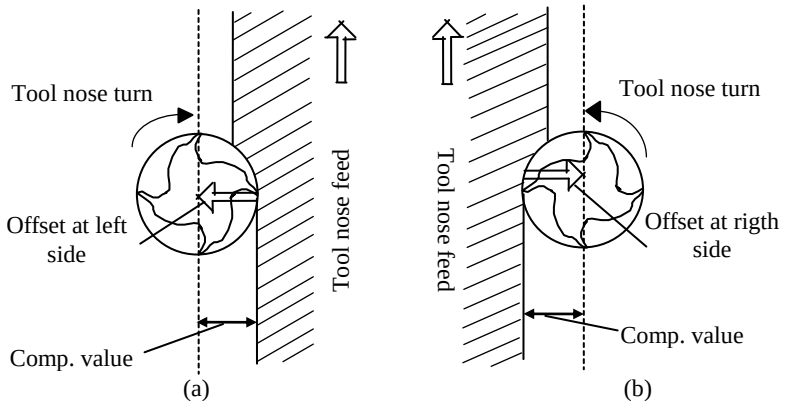
(2) G42 specifies the moving direction of the tool to the right compensation. G42 is shown in the cutting direction 2.21 (b).

(3) G40 cancel tool radius compensation with G41 or G42 must be used in pairs.

(4) D tool radius compensation amount specified code.

Tool radius compensation tool radius compensation is mainly used to influence the actual contour, thus only the actual contour programming, no programming is required to tool center, it brings another benefit is that you can simply change the radius in the tool table, you can change cutting margin to avoid double counting due to changes in the balance of programming work.

Note：G01 cutter compensation established line segment segment segment and tool compensation revocation procedures used must be programmed in the same block with G40, G41 or G42, and subsequently wrote the coordinates.



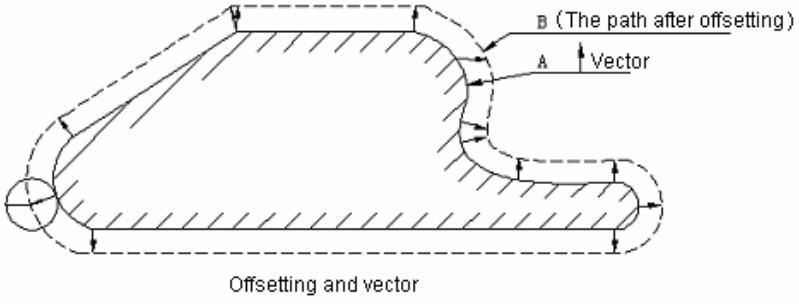
F2.21 Tool compensation

(a)G41 (b)G42

Note：

(1) Tool radius compensation

As shown below, with the radius R of the cutting tool to the workpiece A, the tool center path diagram B, the path from A to B R. A tool offset workpiece radius distance is called compensation.



F2.22 Compensation and Vector

Therefore, programmer may use the tool offset mode to program the outline of a workpiece.

Furthermore, if tool radius (offset) is measured and set in the NC during machining, the tool path will be offset (path B) regardless the programmed path.

(2) Offset

At most 24 amounts of offset may be set in the offset storage (including 24 amounts of offset for tool length compensation and tool position offsetting). The offset depends upon the T instructed in program and the number of digits is set with MDI/LCD.

The offset corresponding to No. 00 or T0.0 is always 0.So the offset corresponding to T0.0 need not be set.

(3) Plane selection and vector

The calculation of offset is performed in the plane established by G17, G18 and G19, which is called offset plane. For instance, (X, Y) or (I, J) is used to calculate the offset as well as vector when XY plane is selected. The coordinates of the axes beyond the offset plane are not under the influence of offset but follows instructed value in the instruction.

In 3-axis gang control, the tool path projected on the offset plane is compensated.Plane selection switching shall be performed in the mode of offset cancellation.

Table2.4

G instruction	compensation plane
G17	X O Y
G18	Z O X
G19	Y O Z

(4) G40、G41 and G42

G40, G41 and G42 are used to specify the cancellation and generation of a tool radius compensation vector. To determine the direction of an offset vector and the moving direction of the tool, G40, G41 and G42 may be instructed concurrently with G01, G02 or G03.

Table 2.5

G instruction	Function
G40	To cancel tool compensation
G41	Tool compensation, left
G42	Tool compensation, right

2.21.1 Tool radius compensation (G40)

Description：

From the starting point toward the end of the old linear motion vector. Using this code, tool compensation from the state of the system into the tool offset cancel state.

2.21.2 Tool radius compensation left (G41)

Description：

(1) Compensation for the tool on the left side of the workpiece contour cutting exercise.

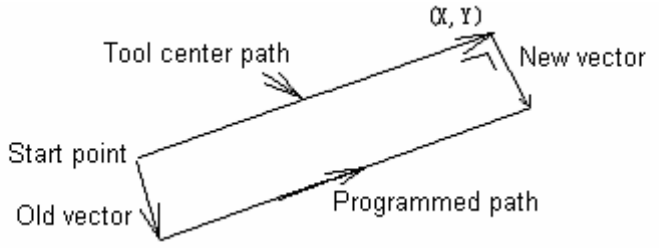
(2) Occurs prior to G41, the tool parameters must be set to complete in the main function of the tool parameter PARAM.

(3) G41 segment, you must have a G01 function and the corresponding coordinates to be effective to build up a knife.

Compensation straight trajectory when G01

Instruction block end point is formed with a (X, Y) perpendicular to the direction of the

new vector, the tool tip is moved from the starting point to the tip of the old vector new vector.



F2.23

The old vector is zero, use the code, the tool from the tool offset cancel state into the tool radius compensation state.

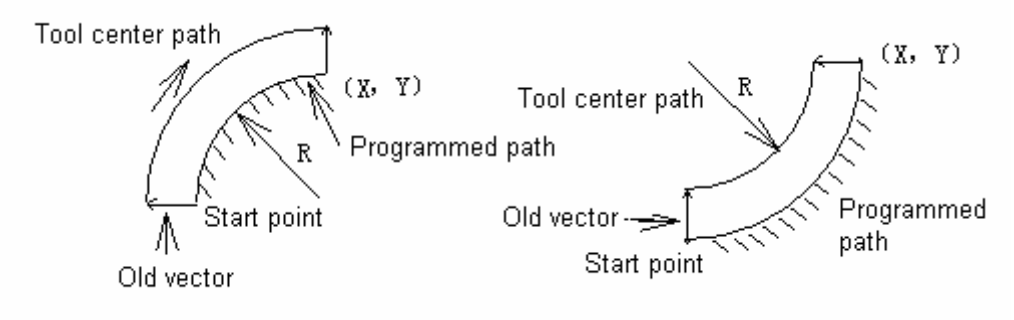
If the program path is arc. the offset vector points to or is apart from the circle center from the start point or the end point.

G41……； ……

……

G02 /G03 X__ Y__ R__ ；

By program above, the new vector that is located on the line between the circle center and the end point can be made out. From the arc advancing direction, it points to the left (right). The tool center moves along an arc from the old vector tip to the new vector tip with the precondition that the old vector is has been made out. The offset vector points to or is apart from the circle center from the start point or the end point.



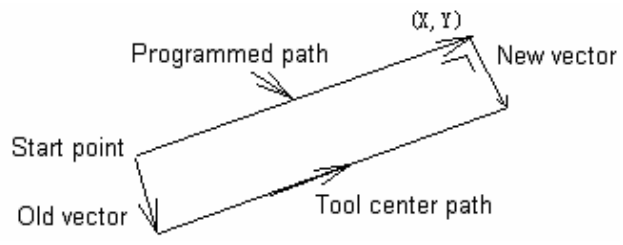
F2.24

2.21.3 Right tool radius compensation (G42)

Description：

By contrast to G41, G42 specifies the tool to deviate at the right side of the workpiece along the tool advancing direction. I.e. the vector direction got in G42 is reverse to the vector direction got in G41. Besides the direction, the deviation of G42 is identical with that of G41.

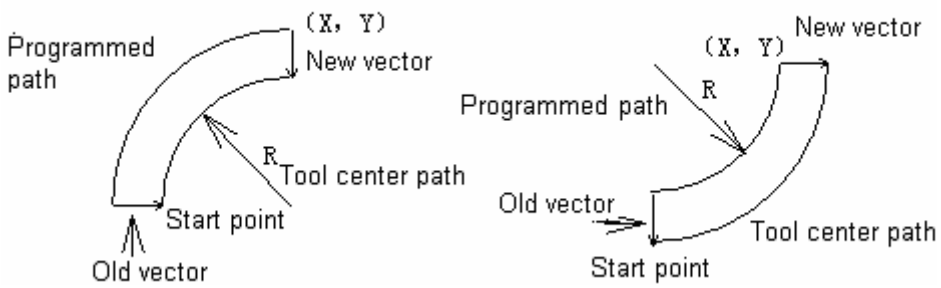
At G01 mode



F2.25

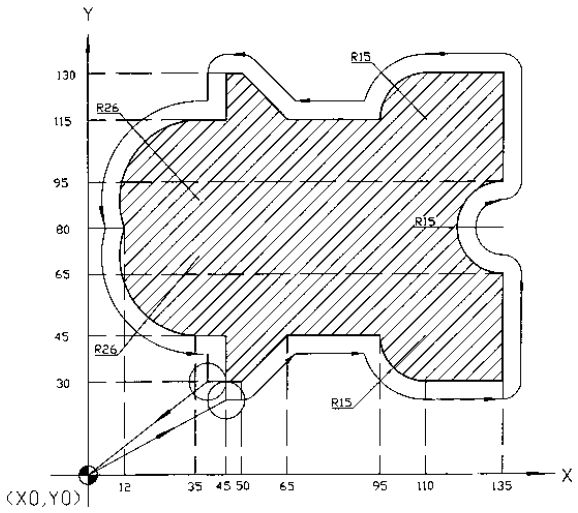
When the old vector is zero, by this instruction the tool is switched to Right side tool radius compensation mode from tool offset cancel mode.

At G02、 G03 mode



F2.26

An example of right side radius compensation：



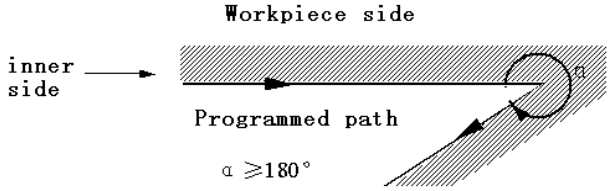
F2.27

N0010 G0 X0 Y0
N0020 T1（select T code 1，Comp. value =12mm）
N0030 G42 G01 X45 Y30 F500
N0040 G01 X50 Y30
N0050 G01 X65 Y45
N0060 G01 X95 Y45
N0070 G03 X110 Y30 R15
N0080 G01 X135 Y30
N0090 G01 X135 Y65
N0100 G02 X135 Y95 R15
N0110 G01 X135 Y130
N0120 G01 X110 Y130
N0130 G03 X95 Y115 R15
N0140 G01 X65 Y115
N0150 G01 X50 Y130
N0160 G01 X45 Y130
N0170 G01 X45 Y115
N0180 G01 X35 Y115

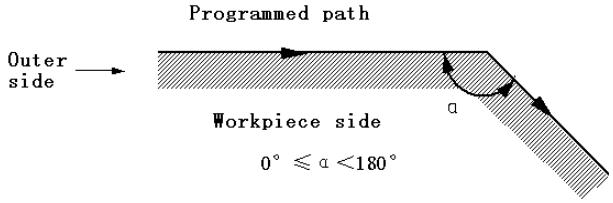
```
N0190  G03  X12  Y80  R26
N0200  G03  X35  Y45  R26
N0210  G01  X45  Y45
N0220  G01  X45  Y30
N0230  G40  G01  X0  Y0
N0240  M02
```

Details about tool radius compensation：

Conception for inner side or outer side："Inner side" and "outer side" will be employed in the following explanations. When an angle of intersection created by tool paths specified by move commands for two blocks is over or equal to 180°, it is referred to as "inner side". When the angle is between 0° and 180°, it is referred to as "outer side".



F2.28

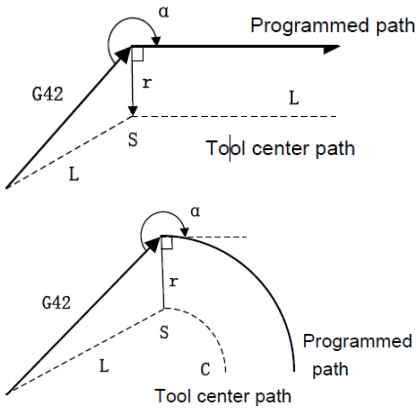


F2.29

A）Tool movement in start-up

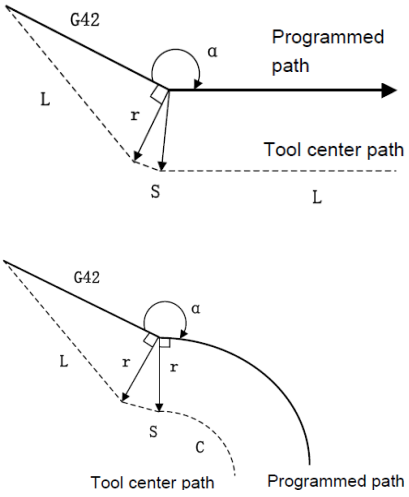
The tool movement performed from offset cancellation mode to G41 or G42 command establishment is called tool compensation establishment (also called start-up)：

a）Tool movement along an inner side of a corner（α≥180°）



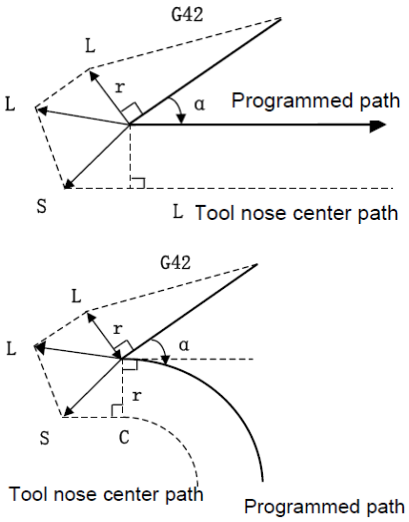
F2.30

b) Tool movement along the outside of a corner at an obtuse angle（180°＞α≥90°）



F2.31

c) Tool movement along the outer side of a corner at an acute angle（α＜90°）

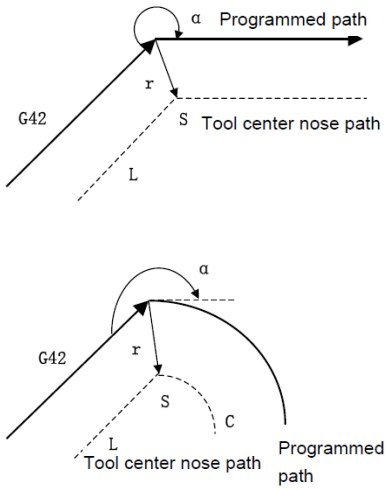


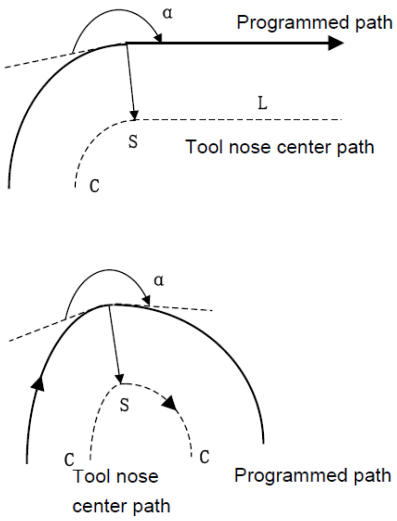
F2.32

B）Tool movement in offset mode

The mode after setting the cutter radius compensation and before cancelling the cutter radius compensation is called offset mode.

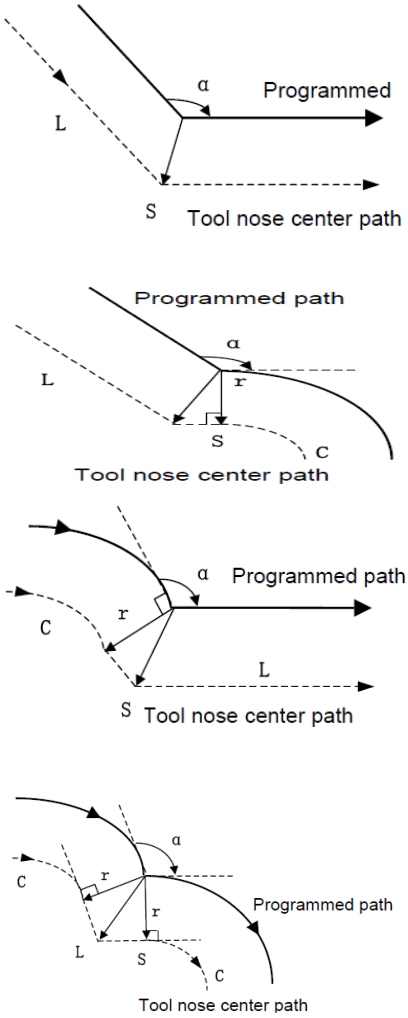
a) Tool movement along the inside of a corner（α≥180°）：





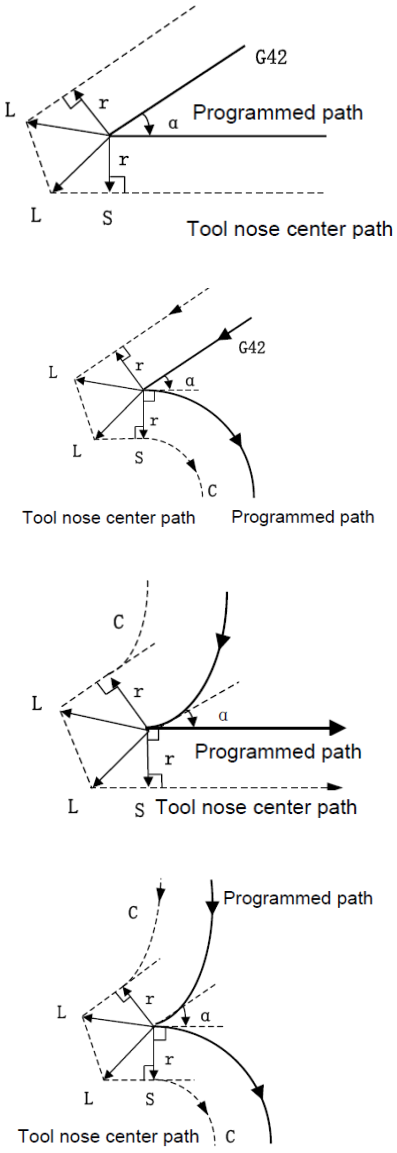
F2.33

b) Tool movement along the outside of a corner at an obtuse angle（180°＞α≥90°）



F2.34

c) Tool movement along the outside of a corner at an acute angle（α＜90）



F2.35

C）Offset cancellation

When the G40 command is employed in block in compensation mode, the CNC enters the compensation cancellation mode. This is called compensation cancellation.

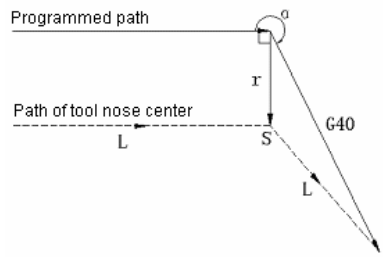
When a block is read in offset cancellation mode, two blocks including that stored in the buffer (not display) for tool compensation will be executed. In single block mode, a block is read in（G41 mode）and executed before stop. The next block is read in and executed by pressing the START button.

After the control system enters into cancellation mode, the next executed block will be normally saved in the buffer register rather than typed in the tool compensation buffer register.

(G02) and (G03) cannot be instructed when offset cancellation is performed.

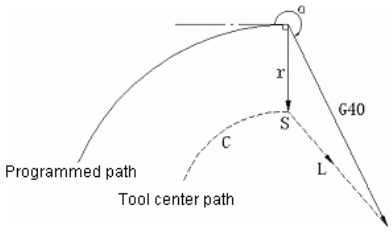
a) To machine an interior angle (α≥180°)

Straight line →Straight line



F2.36

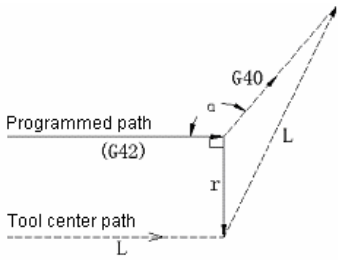
Arc→ Straight line



F2.37

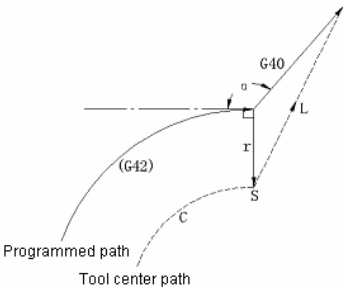
b) To machine an exterior angle (90°≤α＜180°, obtuse angle)

Straight line →Straight line



F2.38

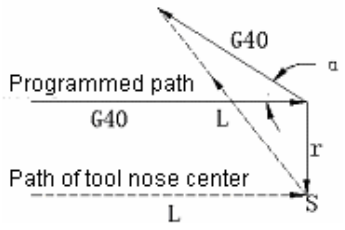
Arc→ Straight line



F2.39

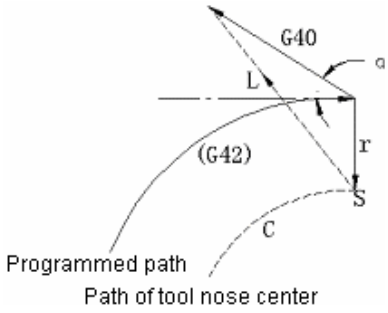
c) To machine the exterior angle of an acute angle (α＜90°)

Straight line →Straight line



F2.40

Arc→ Straight line



F2.41

D）The blocks without tool movement

The following blocks are free from tool movement. The tool does not move even tool radius compensation is valid in these blocks.

M05; ……M code input

S21; ……S code input

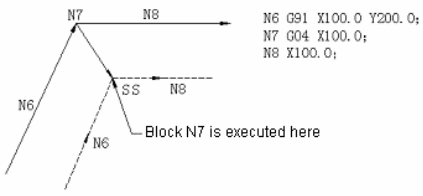
G04 K1.0; ……D well time

(G17) Z200; ……The movement beyond offset plane

G90; ……Only G codes

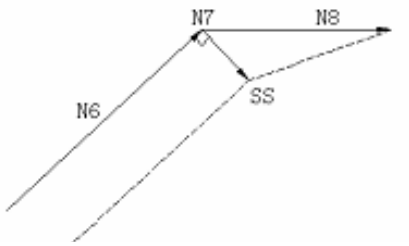
G91 X0; ……...Offset is 0

a) When a single block without tool movement is instructed in offset mode, its vector and tool center path are the same as when the block is not instructed. The block is executed at the stop point of the single block.



F2.42

b) When a block without tool movement and offset cancellation are specified at one time, its vector whose length is equal to the offset is generated in the direction normal to the tool movement of the previous block. The vector will be cancelled in the next move instruction.

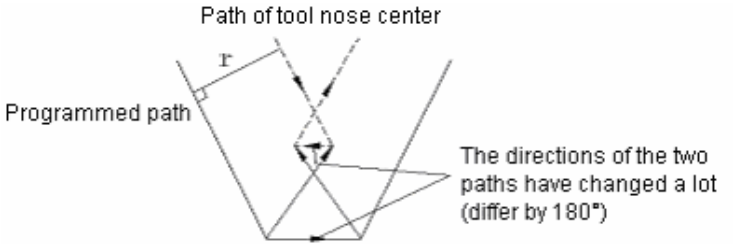


F2.43

E）Interference verification

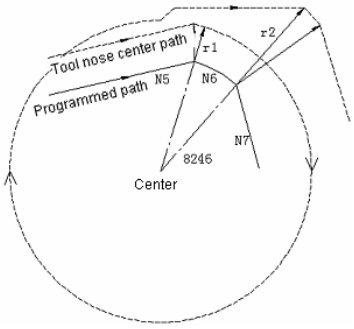
Tool overcutting is called "interference".The benchmark for overcutting verification：

In tool compensation, the traveling direction of the tool center path differs from that of sine path (differ by 90°～270°).



F2.44

During arc machining, the angle difference between the starting point and end point of the tool center path extremely varies from that between those of programmed path except the above state a (more than 180°).



F2.45

F）General cautions about compensation

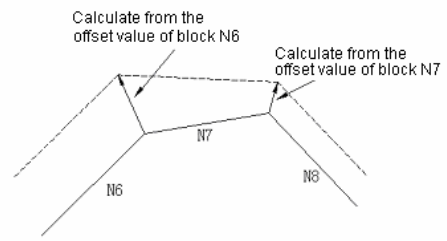
(a) To specify offset

The T codes that are specified with an offset number instructs offset.Once they are specified, the T codes will remain valid until another T code is specified or they are cleared.

Besides that T codes are used for the offset of tool radius compensation, they are also intended for specifying the offset of tool length offset.So must be set correct in the "Tool Parameter Table".

(b)To change offset

In general, if offset is changed during tool change in offset cancellation mode, the vector of the end point of the block is fit for the new offset.

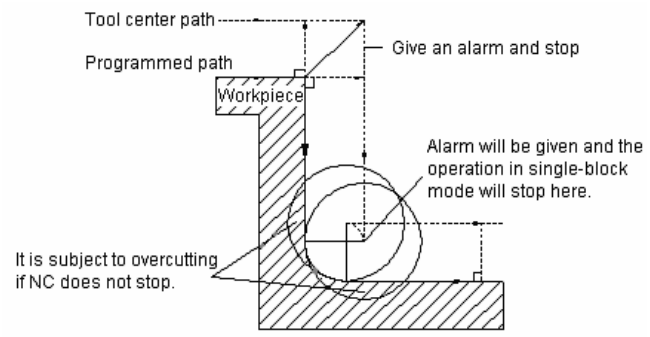


F2.46

(c) Overcutting resulting from tool compensation

a) To machine the inner side of an arc whose radius is less than that of a tool

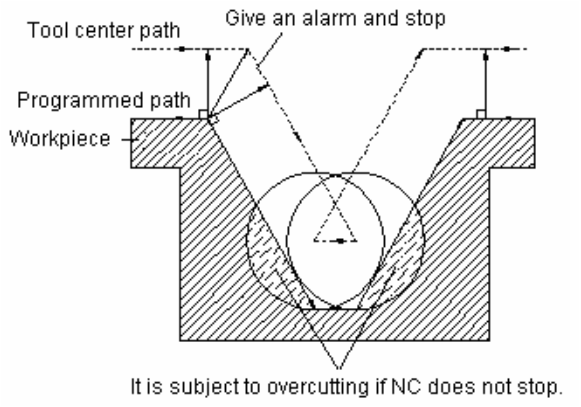
When the instructed arc radius is less than that of the tool, the system will give an alarm.



F2.47

b) To machine a groove smaller than tool diameter

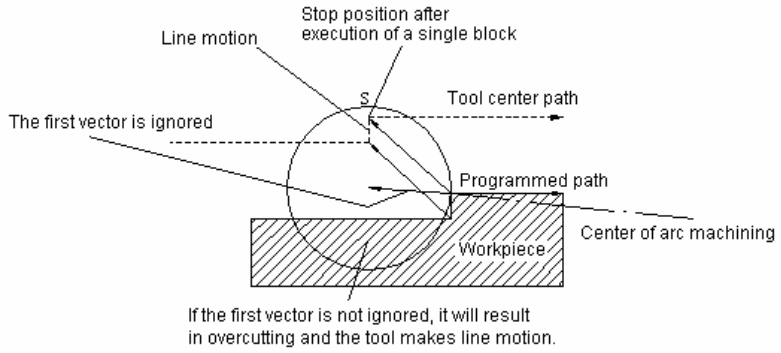
Since that tool compensation forces the tool center path to move in the direction contrary to the programmed direction may cause overcutting, No. 41 alarm will be given and NC will stop at the origin of the block.



F2.48

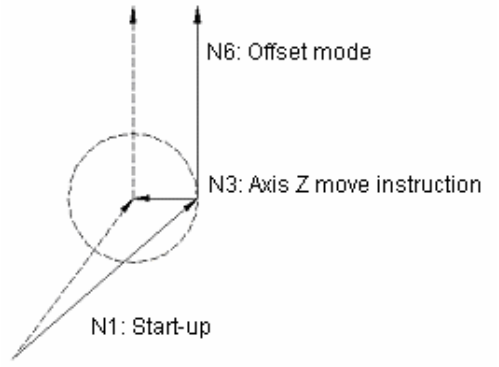
c) To machine a step less than the tool radius

When the program includes a step smaller than the tool radius and it is machined under circular cutting instruction, the tool center path using normal offset 6.3.6 (3) will become the reverse programmed direction. Now the first vector will be ignored and the tool moves the position of the second vector. The tool stops at this point. The program continues to execute if it is not machining in single block mode.



F2.49

The startup associated with tool compensation C and the movement in the direction of axis Z.



F2.50

At the beginning of cutting, tool compensation (generally XY plane) shall be preset at a distance from the workpiece and then the tool feed by means of movement along axis Z.

Now take note of the following issues if the rapid traverse (positioning) of axis Z shall not be independent of the cutting feed.

Refer to the following program：

N1 G91 G00 X500 Y500

N3 Z-250

N5 G01 Z-50 F100

N6 Y100 F200

N1 G91 G0 X500 Y500

N3 G01 Z-300 F100

N6 Y100 F200

In the above example, N3 and N6 will also be read in buffer register during execution of block N1 and the relationship between them will be properly compensated as shown in the right figure.

（G）An example of offsetting course will be illustrated in the following figure

Start-up block ①: Offset cancellation mode becomes offset mode (G41) in this block. The tool center is offset at the end point (P_1) of the block by the radius perpendicular to the lower section of program path (from P_1~P_2). The tool compensating value is specified by T0.2, i.e.offset No.1. G41 means that the tool offsets to the left.

After workpiece shape $P_1 \rightarrow P_2 \longrightarrow P_8 \rightarrow P_9 \rightarrow P_1$ programming and starting, the system automatically performs tool compensation.

In block ①, the tool returns to the point of origin (offset cancellation) through G40. The tool center moves (from $P_9 \sim P_1$) normal to the programmed path at the end point of the 10^{th} block.

G40 instruction (program cancellation) shall be instructed at the end point of the program..

N010 G92 X0 Y0

N020 T0.2 ； (The offset is preset in T0.2 by MDI)

(1) N030 G90 G17 G41 G1 X250 Y550 F1500

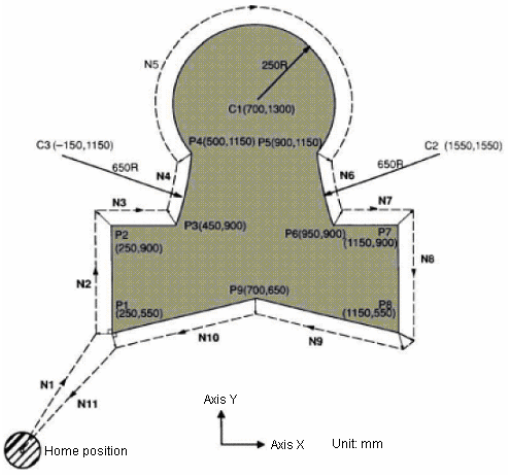
(2) N040 G1 Y900

(3) N050 X450

(4) N060 G3 X500 Y1150 R650

(5) N070 G2 X900 Y1150 R-250

(6) N080 G3 X950 Y900 R650
(7) N090 G1 X1150
(8) N100 Y550
(9) N110 X700 Y650
(10) N120 X250 Y550
(11) N130 G40 X0 Y0
(12) N140 M30



F2.51

2.22 G43/G44——Establishment of tool length compensation

Format：G43 H_
Format：G44 H_
Description：

(1) Tool length compensation , as the name suggests is used to compensate for the difference in tool length . When the actual tool length is inconsistent with the programming tool length , tool length compensation can be achieved by the difference between the tool length compensation.

(2) As long as the tool length compensation is usually the difference between the actual tool length as a programming tool length offset value can be stored in the tool offset parameter table .

(3) H code selection length compensation number, ranging in 0 ~ 24, H0 to cancel length compensation . Another way to cancel the tool length compensation is to use the command G49. NC to execute commands or H00 G49 , immediately cancel tool length compensation (G49 that the system is in a state , and the length compensation number is reset to 0) , and to the Z-axis movement without compensation value to the commanded position .

(4) When the Z -axis executing the operating system is back to zero length compensation cancel mode (ie G49 mode) .

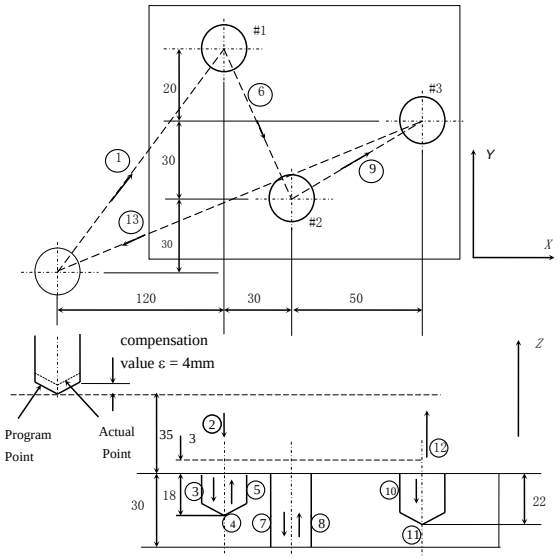
(5) B031 parameter controls whether the system after power weight retention length compensation mode (ie G43/G44/G49 in a) and length compensation number. If retained

after power is re-set to G49 states, take the knife length compensation No. current value.

(6) G43 to establish a positive tool length compensation, that the actual position of the Z axis to reach the command value and compensation for the added value of the position; G44 to establish a negative tool length compensation, that the actual position of the Z-axis command value is reached subtracting a position compensation value.

(7) If the system is in state G49, you must follow the instructions H programming G43 or G44 (can be on the same line) is valid; If the system is already in state G43 or G44, you can not only be programmed with G43 or G44 H to change the length compensation number.

Example：Consider the tool length compensation, prepared as shown in Figure 2.52 parts machining program: requires the establishment of the workpiece coordinate system as shown, according to the path indicated by the arrows for processing.



F2.52 Tool length compensation processing

```
G92 X0 Y0 Z0
G91 G00 X120 Y80 M03 S600
G43 H01      ; select compensation number
Z−32
G01 Z−21 F300
G04 K2
G00 Z21
X30 Y-50
G01 Z−41
G00 Z41
X50 Y30
G01 Z−25
G04 K2
G00 Z57
G49
X−200 Y−60 M05 M30
```

2.23 G47——Short straight line speed Auto-transitions

Format：G47

Description：When processing non-round curve surface, the cutter approaches the surface with very short straight lines while certain precision is preconditioned in general CAD/CAM software. In the condition of invariable cutting linear speed, the CNC controls each coordinate axes carrying out velocity adjustment between two section of straight lines to transit from one straight line to the the other straight line. This kind of transitaion can reduce the machine tool vibration, enhance actual cutting speed and improve finish quality when the linear velocity variation is avoided and the variation is caused by acceleration and/or deceleration in each section of short straight line.

G47 when in effect, two lines must meet the following conditions in order to automatically transition:

(1) line length <32.5MM

(2) two straight angle <20 °

2.24 G48——Cancel G47

Format：G48

Description：

(1) G47 is invalid when cutter radius and length compensation functions are valid.

(2) The G47 functions automatically invalid when executing DNC processing, except G48 is used.

2.25 G49——Revocation of tool length compensation

Format：G49

Description：G49 function is to withdraw the tool length compensation, the tool offset memory length in the Z-axis offset value does not work.

2.26 G52——Setting the local coordinate system

Format：G52 X_ Y_ Z_ A_ B_ C_

Description：

(1) G52 can create a local coordinate system, the local coordinate system is equivalent to the sub-coordinate system G54 ~ G59 coordinate system.

(2) G52 instruction will set before the revocation by the G92 workpiece coordinate system.

(3) The directive, X_ Y_ parameters gives an offset relative to the current G54 ~ G59 coordinate system, that is, given the position coordinates of the origin of the local coordinate system in the current G54 ~ G59 coordinate system, even the G52 command before execution has been reviewed by a G52 command to create a local coordinate system. Cancel the local coordinate system approach is very simple to use G52 X0 Y0 Z0 A0 B0 C0 can.

2.27 G53——To run to the end of the machine coordinate system

Format：G53 X_ Y_ Z_ A_ B_ C_

Description：Run to the machine coordinate position programmed values. G73 behind the X, Y, Z, A, B, C can be programmed into each, the whole can also be programmed.

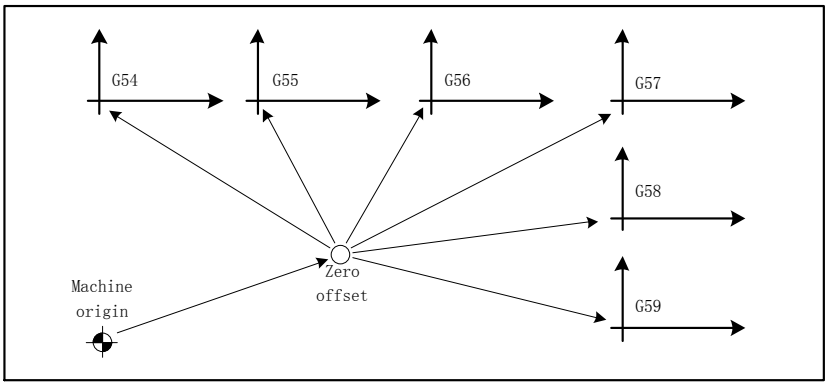
2.28 G54～G59——Workpiece coordinate system

Format：G54(The G54 for example)

Description：

(1) G54 ~ G59 are six pre-set coordinates, when programming or MDI can choose any one of them. Once the coordinate system is selected, the screen on the selected coordinate system to the new workpiece coordinate display execution, the program is also based on this coordinate system, with the processing code, or until a mode change until the MDI coordinates.

(2) G54 ~ G59 is modal function can be canceled each other, G54 is the default.



F2.53 Workpiece coordinate system selection (G54～G59)

2.29 G61——Current segment and subsequent processing segment continuous clearance angle

Format：G61

Description：Both this segment and subsequent processing trace adopt sharp corner connection until they are cancelled by G64.

2.30 G62——Rapid clearance angle command current section

Format：G62

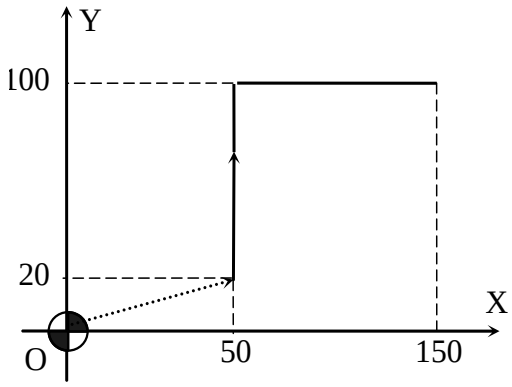
Description：Sharp corner interim is between this segment of trace and the next segment .

2.31 G64——Cancel clearance angle transition

Format：G64

G61～G64 Description：

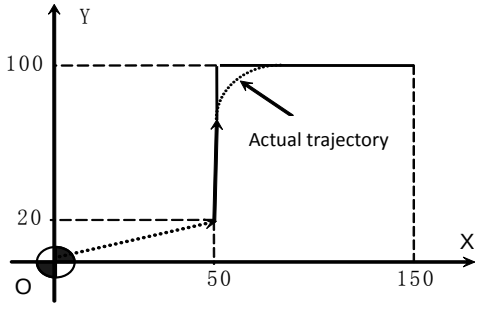
The cutting speed of system control between two sections of processing blocks in 4.2.1 isn't changed, the interim radian will occur between two rails, in the occasion having requirement for workpiece molding surface, these radian must be eliminated such as order shaft and various end surfaces. If G61 or G62 are adopted, the next program can be started after the cutter runs this program completely, to ensure the shape between two mold surfaces matches the programming shape, called "Angle clear" function.




```
G92 X0 Y0
G00 X50 Y20
G01 G61 Y100 F300
X100
…
```

F2.54 G61 Program

When the cutting is performed, because the feed speed is low, about below F300, when the cutting time constant (39＃) is smaller (<100ms), this interim radian is also small, and the influence to workpiece is also small, under the precondition of meeting processing requirement, the machining efficiency can be enhanced and the shock vibration can be reduced without adopting angle clear function, and it's favorable to enhance smoothness degree, in the occasion having higher requirement or it must be sharp corner, G61 (G62) function shall be adopted. G62 only works to current program, namely this section of program realizes angle clear, and the following program still adopts interim radian mode.G61 is valid to current section and follow-up program until G64 cancels, the interim processing is only suitable for continual G01, G02 and G03, once the follow-up program isn't be above track, system will cancel the interim function automatically.




```
G92 X0 Y0
G00 X50 Y20
G01 G64 Y100 F300
X100
…
```

F2.55 G64 Program

Fixed cycle

CNC machining, some machining cycle has stylized action. For example, the action drilling, boring a hole plane positioning, rapid introduction of work feeding back fast, so a

series of typical processing action has been pre-programmed (5.4 system macros), stored in memory, available is called a G-code block is called a fixed cycle, thus simplifying programming.

Hole machining canned cycle instruction has G73, G81 ~ G89, usually consists of the following five actions (see fixed cycle operation):

(1) X, Y-axis positioning;

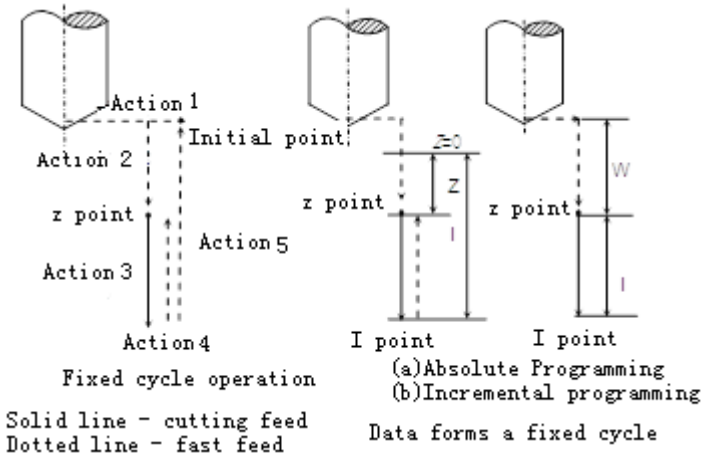
(2) Positioning to point Z;

(3) Hole Machining;

(4) In the bottom of the hole Machining;

(5) Return to the Z point (reference point) or initial point.

Data are expressed in the form of a fixed cycle can be absolute coordinates and relative coordinates, as shown in the form of fixed-cycle data, which Figure (a) is a representation of absolute programming mode, Figure (b) is a representation of incremental programming mode.



F2.56

2.32 G66——Cancel G67

Format：G66

Description：Cancel G67, when the rate of synthesis of each axis linear interpolation by programming speed sport.

2.33 G67——When the long axis linear interpolation by programming speed sport

Format：G67

Description：The long axis linear interpolation by programming speed sport, G67 G66 remains valid until revoked execute G66.

2.34 G68——G02/G03 adopted before landing speed

Format：G68

Description：

G02 and G03 circular interpolation using the lifting speed before interpolation, G68 G69 command remains in effect until revoked using G68. Instruction usually use when high accuracy is required, the use of interpolation before landing speed, the processing efficiency will be reduced.

2.35 G69——Cancel G68

Format：G69

Description：The cancellation G68, G02 and G03 circular interpolation processing after lifting speed can improve processing efficiency.

2.36 G73——High-speed deep hole machining cycle

Intermittent feed for Z-axis, reducing the amount of retraction can be highly efficient processing.

Format：G73 Z_ I_ J_ K_ R_ F_

Description：

Z：The hole top enterance coordinates

W：The distance from top to bottom

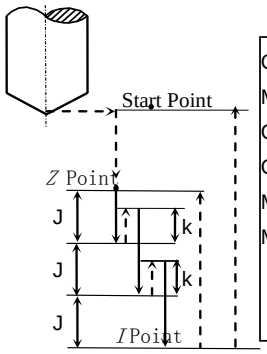
I：The hole bottom coordinates (in absolute value model) or distance to hole top enterance (in incremental model)

J：Feeding distance of each time

K：Cutter retract distance of each time

R：Delay time

F：Feeding speed




```
G92 Z60
M03 S600
G73 Z40 J10 K5 I0 R1 F200
G00 Z80
M05
M30
```

```
G92 Z60
M03 S600
G73 W-20 J10 K5 I-40 R1 F200
G00 Z80
M05
M30
```

F2.57 G73 function motion circulation in the chart

2.37 G74——Returning to reference point (mechanical origin)

Format：G74 X_ Y_ Z_

Description：

(1) Other contents in this segment are prohibited.

(2) The coordinates following G74 will return to zero in turn with Z, X, Y sequence.

(3) It is must be confirmed that the reference point switch to be installed on the machine tool before using G74.

2.38 G75——Return to the center point

Format：G75 X Y

Description：Used to move to the center point set by the user, the coordinate origin is set to the center. G75 behind the X, Y can be programmed separately, but also the whole series.

The center of the set：

(1) In the position of the interface, as in "hand wheel", "jog" or "Manual" one mode, press the ◄► button;

(2) Users can choose the "rectangle points" or "pie" function;

(3) If the "rectangle Points" under the function, operation X-axis to the left of the right position, press the "Save X1", operating to the right the right position, press the "Save X2", empathy select two points up and down the Y axis and save to Y1, Y2, the system will calculate the midpoint and the midpoint of the upper and lower right and left to remember where Xm and Ym;

(4) If the "pie" on the next function, operation shaft to the first point, press the "Save P1", and then the operation to the second, three points, respectively, according to the "Save P2" and "Save P3", then the system will calculate the three-point circle of dots bound to remember where Xo and Yo.

2.39 G76——Zero return to the program from its current location

Format：G76 X_Y_Z_

Description：

(1) Other contents in this segment are prohibited.

(2) The coordinate relative to entry point on the machine tool is displayed with big coordinate, the cutting tool position coordinate of processing start can be memorized in reference point parameter at COORSYS -> PAGEDOWN，This function can return to this position from machine tool optional position, and the speed is same as G00.

2.40 G77——Based on set on the center of the workpiece coordinate

Format：G77 X_ Y_

Description：Used to move to the center position after the user settings, set the center point of the workpiece coordinate system coordinates for the programmed value. G77 behind the X, Y can be programmed separately, but also the whole series.

2.41 G78——Precision boring circulation

When processing precision boring,the main axle reverse moves to the cutter nose direction and the cutter fast retractings after the main axle directional stops at the hole bottom to protect the finished surface from scratch.

Format 1：G78 Z_ R_ U_ K_ F_

Description：reverse move reference X axis.Z can be replace by W, meanings the distance from current point to top.

Format 2：G78 Z_ R_ V_ K_ F_

Description：reverse move reference Y axis.Z can be replace by W, mean the distance from current point to top.

Explanation：

Z：The hole top enterance coordinates

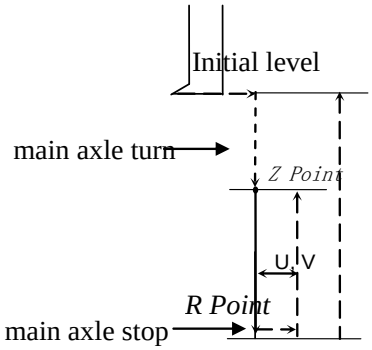
R：The hole-bottom coordinates,If W be used, R means the distance from top to bottom

U：Reverse displacement distance of cutter nose in X axes direction

V：Reverse displacement distance of cutter nose in Y axes direction

K：Delay time

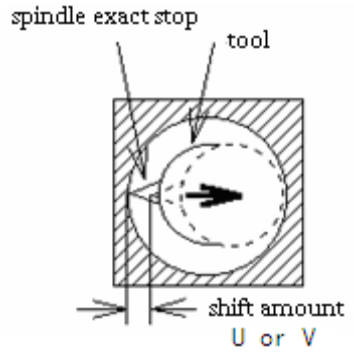
F：Feeding speed




```
G92 Z50
M03 S600
G78 U-6 Z-10 R-40 K1 F200
Z40
M05
M30
```

F2.58 G78 Instruction action and programming examples

G78 function motion circulation in the chart.



F2.59 G76 Reverse let knife schematic

Caution：before use this instruction,you must confirm that your machine have already install a spindle with fixed stay function and order a special PLC.

2.42 G81——Center hole drilling cycle

Format 1：G81 Z_ I_ F_ ；Machining a hole

Format 2：G81 U_ V_ Z_ I_ L_ F_ ；Processing a row of holes in a straight line

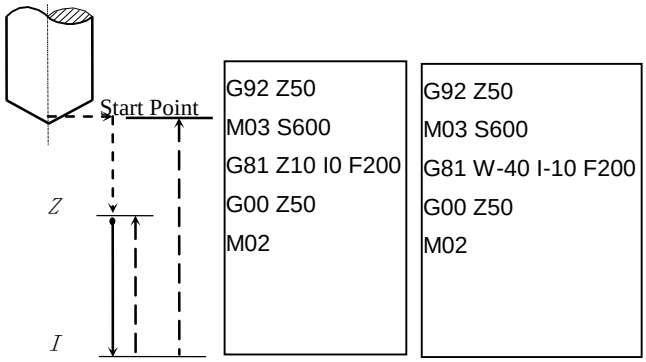
Description：

Z：Hole top entrance coordinate, expressed relative to the initial point of the Z-axis incremental value for incremental programming using W

I：Hole bottom coordinate, If you are using W for incremental programming the bottom of the hole I said incremental value relative to the top of the hole

L：Number of hole

F：Feed rate



F2.60 See G81 function motion circulation in the chart

2.43 G82——Central hole bore drill circulation with pause

Format 1：G82 Z_ I_ R_ F_ ； Machining a hole

Format 2：G82 U_ V_ Z_ I_ R_ L_ F_ ；Processing a row of holes in a straight line

Explanation：

Z：Hole top entrance coordinate, expressed relative to the initial point of the Z-axis incremental value for incremental programming using W

I：Hole bottom coordinate, If you are using W for incremental programming the bottom of the hole I said incremental value relative to the top of the hole

R：Delay time

L：Number of hole

F：Feed rate

G82 mainly used for processing the blind hole, in order to improve the accuracy of the hole depth.

Example：G92 Z80

M03

G82 Z60 I－10 R1.6 F500

or G82 W－20 I－70 R1.6 F500

G82 Instruction motion circulation is the same as that of G81.beside G82 do delay at bottom.

2.44 G83——Deep hole processing cycle

It's used for interval feed of Z axis. Rapidly retract to the top position of hole after each feed cutting and pause the second set by R parameter (make it easier to discharge chips), later move to the position to the previous hole bottom with K distance. Perform second cutting feed cycle until cutting to the hole bottom represented by I. Pause the second specified by R, finally, rapidly retract to the top position of hole, and G83

instruction segment is completed.

Format 1:G83 Z_ I_ J_ K_ (Q_) R_ F_

Format 2:G83 Z_ I_ D_ J_ K_ (Q_) R_ F_

The difference from format 1 is that the D parameter is used to appoint the cut depth of the first cutter (greater than J)

Z：Coordinate of hole top

I：Coordinate of hole bottom

J：Feed depth every time (unsigned number)

K：The distance from previous hole bottom (unsigned number) from fast feed to work feed when performing second feeding after retracting every time

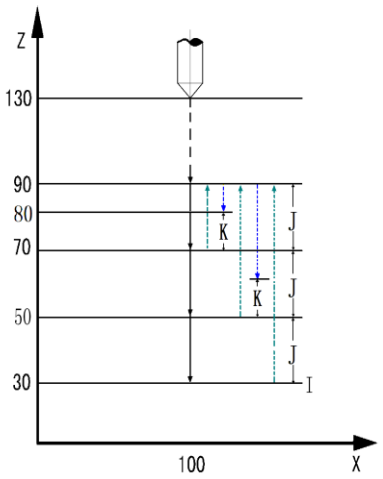
Q：Discharge chip delay，if Q did not appear,the default delay time is 0.1s

R：Hole bottom delay time

D：Cut depth of first cutting (unsigned number)

F：Feed speed

Example 1：



F2.61 G83 command operation diagram

G92 Z130

M03

G83 Z90 I30 J20 K10 R1 F600

or G83 W-40 I-60 J20 K10 R1 F600

Example 2：

G92 Z2

M03

G83 Z-5 I-35 D7 J5 K2 Q0.3 R1.2 F600

or G83 W-7 I-30 D7 J5 K2 Q0.3 R1.2 F600

2.45 G84——Metric rigid threading cycle

Format： G84 Z__ K__ L__ N

Description：

(1) G84 (G85) can be only used in case of installing the spindle encoder.

(2) Z is the threading terminal coordinate, K is the pitch.

L：Material compensation amount, scope is from 0 to 15; 0 is for general material (L

value isn't programmed), L can be increased for fragile material to enhance the threading rotation speed.5 to 10 can be properly increased for sticky material to reduce possibility of broken threading. If L isn't programmed, it'll be determined by P87#.

N：After threading feed reaches Z value, the system will give spindle stop signal; when the spindle reduces to the rotation speed set by N, the system will give reverse signal, thus reducing the reversing time; when N isn't programmed, the system will give spindle reverse signal after checking that the rotation speed of spindle is reduced to 0.

When the spindle is under frequency control, it'll be useless to program N owing to characteristic of frequency converter.

(3) Select rotation speed of spindle in rigid threading.

When rigid threading is performed, one thread pitch will be fed axially along the spindle every time the spindle rotates for one circle, and this relationship should be strictly maintained when the rotation speed of spindle is increased and decreased. The spindle multiplying factor and feed multiplying factor are prohibited in threading.

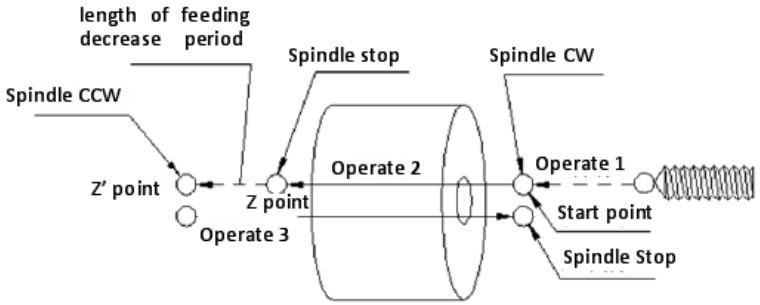
When rigid threading is performed, the Z-direction feeding is synchronized with the spindle; when threading feed reaches Z value, the system will give spindle stop signal; within the deceleration time for spindle from the set rotation speed to zero speed, Z direction is still followed up (the longer the time of spindle deceleration time is, the bigger the follow-up length is); in order to reduce the follow-up length within the deceleration time, the rotation speed increase and decrease time should be reduced as possible.

The proportional relation between the threading feed speed and the spindle speed is as follows:

F=S×K

In the formula:

F-Threading feed speed; S-Spindle speed; K-Tap thread pitch：



F2.62

Operation：

(1) Rapidly position to the start point of threading and the spindle rotates positively.Operation.

(2) Threading feeds to Z point and spindle stops.Operation.

(3) Spindle rotates negatively, tap returns to the start point and spindle stops.

2.46 G85——English system rigid threading cycle

Format：Same as G84.

Description：Thread pitch is K tooth/inch.

2.47 G86——Hole boring circulation(automatic return)

Format 1：G86 Z_ I_ F_

Format 2：G86 U_ V_ Z_ I_ L_ F_

Description:

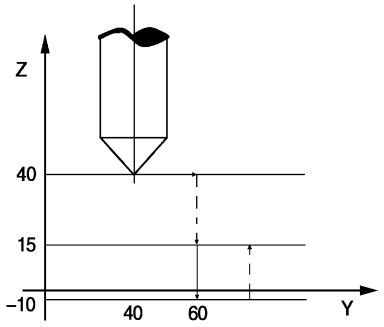
Z：Hole top entrance coordinates

I：Hole bottom coordinates

L：Number of hole

U/V：The distance to next hole in X axle and Y axle

F：Feeding speed



F2.63

Compare to execute G81 instruction，The main axle stops at the hole bottm when excute the G86 instruction.

Example：

G92 Z40

M03

G86 Z15 I－10 F500

Or：G86 W－25 I－25 F500

See G86 instruction motion circulation in the chart.

2.48 G87——Counter Precision boring circulation

Format 1：G87 Z_ R_ U_ K_ F_ ；X-axis is parallel to the flank

Format 2：G87 Z_ R_ V_ K_ F_ ；Y-axis is parallel to the flank

Description：

Z：Hole top entrance coordinate, expressed relative to the initial point of the Z-axis incremental value for incremental programming using W

R：Bottom of the hole coordinates, if used for incremental programming the R W represents relative to the bore hole bottom top of the incremental value

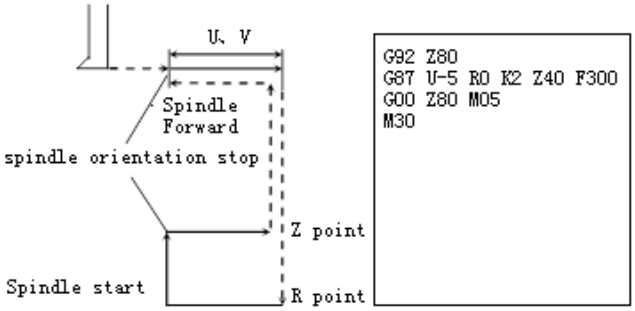
U：The tool tip along the X-axis displacement of the reverse

V：Reverse tip displacement amount along the Y axis

K：Delay time

F：Feed rate

Basically identical to G78, from the bottom of the hole just beginning knife processing.



F2.64 G87 Instruction action and programming

The G87 instruction cycle of motion is described as follows:

(1) Oriented spindle stop;

(2) U or V values move in the opposite direction of the tip of the X or Y direction;

(3) Positioning to point R (bottom of hole);

(4) U or V values to move the tip in the direction of the X or Y direction;

(5) Spindle forward;

(6) Working on the Z-axis direction to point Z;

(7) Spindle orientation stop;

(8) U or V respectively move the tip in the direction opposite to the X or Y direction;

(9) To return to the initial point;

(10) U or V values to move the tip in the direction of the X or Y direction;

(11) Spindle forward.

Note: this command must have a function in the machine tool spindle orientation under the condition of the CNC System Download exclusive PLC program before use.

2.49 G89——Hole boring circulation with pause

Format 1：G89 Z_ I_ R_ F_ ；Machining a hole

Format 2：G89 U_ W_ Z_ I_ R_ L_ F_ ；Processing a row of holes in a straight line

Description：

Z：Hole top coordinates, using W as the incremental programming representation with respect to the Z axis incremental current location value

I：The hole bottom coordinates, if the use of W for incremental programming I said hole bottom with holes on the top of the value increment

R：Delay time

L：The number of holes

F：Feed speed

Example：

G92 Z40

M03

G89 Z15 I－10 R1.6 F500 或 G89 W－25 I－25 R1.6 F500

Compared with G86, G89 directive pause time delay in the bottom of the hole.

2.50 G90——Absolute programming

Format：G90

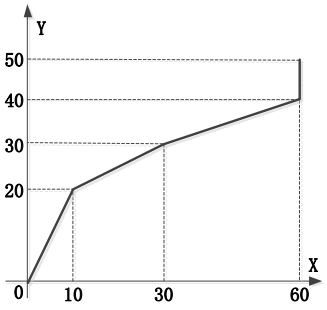
Description：

(1) When G90 is compiled into program, the programming zero will be regarded as benchmark for all the compiled coordinate values afterwards.

(2) After the system is powered on, the machine tool is in the G90 status.

Example：

N0010 G90 G01 X10 Y20 F200
N0020 X30 Y30
N0030 X60 Y40
N0040 Y50
N0050 M02



F2.65

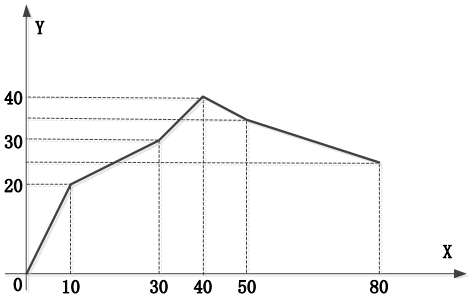
2.51 G91——Incremental programming

Format: G91

Description: G91 programmed, after all the coordinate values before a position as a starting point to compute the motion of a programming value.

In the following coordinate system, always had a point as the starting point to programming.

Example：



F2.66

N0010 G91 G01 X10 Y20 F200
N0020 X20 Y20
N0030 X10 Y20
N0040 X10 Y-10
N0050 X30 Y-20

N0060 M02

2.52 G92——Setting workpiece coordinate system

Format: G92 X_ Y_ Z_ A_

Description:

(1) G92 only changes the current displayed coordinate value of system, and the coordinate axes require no shift to set coordinate origin.

(2) The effect of G92 is to change the displayed tool nose coordinate into the set value.

(3) The X and Y following G92 can be compiled respectively, and can also be compiled wholly.

3 Auxiliary function(m functions)

M function also known as auxiliary functions, input and output ports for the state of CNC control and the implementation of control procedures. Move instructions and M functions in the same block, the execution order as the following:

(1) At the same time perform the move command and m-command;

(2) Move instruction is executed before, after the m command;

The specific use of which the execution order, refer to the machine tool manufacturer specifications.

M function has the following two categories:

Standard PLC-defined m-code:

M00	Program suspended
M01	Select Stop
M02	End of program and stop
M03	Spindle Forward
M04	Spindle Reverse
M05	Spindle stop
M08	Coolant On
M09	Coolant Off
M10	Spindle clamping
M11	Spindle release
M28	Spindle position mode
M29	Cancel spindle position mode

The system used M function:

M46	Set relative coordinates
M47	Clear piece count
M48	Number of piece increase 1
M49	Number of piece cleared
M52	Reading Tool number and assign to local variables
M53	Set cutter compensation number and display to the screen
M54	Compile synchronous instruction
M57	Tool change instructions

M58 Display alarm
M64 Set Fxxx.x to the first level of PLC cycle
M65 PWM output
M66 Clear the designated bit F table
M67 Wait X table input effective
M68 Wait X table input signal revoked
M90～M99: System takes to be expanded
Note:

(1) M06 directly #3 local variable, the M57 by programming the k value determines the number of locals, mainly for the turret tool change. M function occupied non-system code can be defined by the PLC.

(2) In General, a program section, if there are two or more m-code, the last work.

(3) M functions often manufacturers and machine tool structure and models and with the provisions of the standard m functions is different, so m code see machine-specific instructions.

M features explained below:

3.1 M00——Program suspended

Format: M00

Description: M00 appears in the program, the end of the program running, the system is in a wait state, processing starts key is pressed, the program continues to run.

3.2 M01——Select Stop

Format: M01

Description: when a machine on the operator panel "Optional stop" switch is in the open State, after the M01, the system is in a wait state, cycle start key is pressed, the program continues to perform.

3.3 M02/M30——End of program and stop

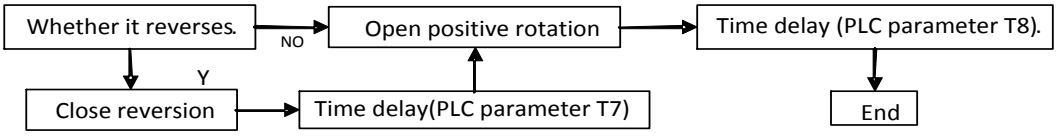
Format: M02

Description: represents the end of the main program, and return to the program head, closed cooling, the spindle can be defined by the machine tool manufacturer.

3.4 M03——End of program and stop

Format: M03

Description: write M03 instructions in the program, first take the spindle forward relay snap, then s-function output analog, controlled spindle clockwise. M03 action sequence:

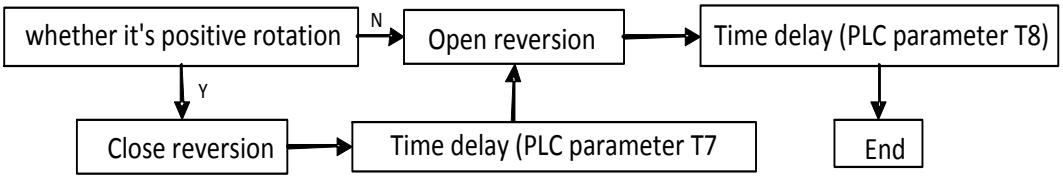


F2.67

3.5 M04——Spindle Reverse

Format: M04

Description: Start the spindle reversal. M04 action sequence:



F2.68

3.6 M05——Spindle stop

Format: M05

Description: spindle stop instruction, M05 instructions output pulse signal, turn off the spindle or reverse control relays, stop output analog, spindle stop. Output pulse width is determined by T0007 PLC parameter.

3.7 M08——Coolant On

Format: M08

Description: M08 function in the program execution at the beginning of the paragraph, connected liquid cooling control relay.

3.8 M09——Coolant Off

Format: M09

Description: M09 function in the paragraph after the program has finished running, turn off the coolant control relay.

3.9 M10——Workpiece clamping

Format: M10

Description: M10 feature enables control of clamping action relay closed.

3.10 M11——Loosen the workpiece

Format: M11

Description: Disconnect the clamp relay

3.11 M28——Spindle position mode

Format: M28

Description:The operating mode is set to the position of the spindle mode.

3.12 M29——Cancel spindle position mode

Format: M29

Description: The spindle work mode to speed mode.

Note: the above m functions specified by the PLC, features and usage please refer to the machine tool manufacturer specifications.

3.13 M46——Set relative coordinates

Format: M46 X_ Y_ Z_

Description: the relative coordinates of each axis is set to the programmed value after the M46 command.

3.14 M47——Clear piece count

Format: M47

Description: M47 features make the system total processing piece zero.

3.15 M48——Number of piece increase 1

Format: M48

Description: M48 function makes the system number of piece increase 1.

3.16 M49——Number of piece cleared

Format: M49

Description: M49 features enable system cumulative processing time zero.

3.17 M52——Reading Tool number and assign to local variables

Format: M52 KXX

Description: Read the tool number, and assigned to the local variable.

3.18 M53——Set cutter compensation number and display to the screen

Format: M53 KXX

Description: The tool offset number is displayed to the screen, and set the system tool.

3.19 M54——Compile synchronous instruction

Format: M54

Description: 7 series system using advanced compiler, in some cases, the need to compile and execute synchronization. As in reading the current coordinates or machine tool workpiece coordinate.

3.20 M55——Set the tool number

Format: M55 Kn

Description: The n into F029 to F026, the M function code, open the PLC MF strobe signal. The function details please refer to machine tool factory.

3.21 M56——Tool change to the target in place

Format: M56 Kn

Description: The n into F table in F029 to F026, the F192.0, and open the TF strobe signal, generally used as a tool change instruction. The function details please refer to machine tool factory.

3.22 M57——Tool change instructions

Format 1: M57Kn

Description: Place number n to F029~F026 in the F table, and open the TF gate signal.

Format 2: M57K[#n]

Description: Place n# local macro variable to F029~F026 in the F table, and open the TF gate signal.

3.23 M58——Display alarm

Format: M58 Knnn

Format: nnn is the alarm number.

3.24 M64——Set Fxxx.x to the first level of PLC cycle

Format: M64 Fxxx.x

Description: M64 feature set PLC F table correspond position to 1.

3.25 M65——PWM output

Format: M65 X_ Y_

M65

Note: M65 can specify any two shaft or a shaft to a certain speed, unit for rpm. Cancel the M65 can be RESET or specify M65 behind without any address word in a program. For example: M65 Y100 A200 said that the Y axis to 100 rpm, the A axis to 200 rpm rotation.

Application: M65 can be used for polygon turning, hobbing.

Example:

```
G00 X100 Z20
M65 Y100 A400 ； The Y axis, A axis ratio 1:4, used for turning square work
G04 K2 ； Two second delay, waiting for the speed stability
G01 Z0 F100 ； Square began turning
G00 X120 Z20 ； Retreat Cutter
```

M65 ；The Y axis, A axis rotation stops

3.26 M66——Clear the designated bit F table

Format: M66 Fxxx.x

Description: M66 feature allows the PLC F table corresponds to position 0.

3.27 M67——Wait X table input effective

Format: M67 Xxxx.x

Description: The program execution will stop and wait for instructions to M67 delayed 78 system parameters set, the corresponding bit is valid until the X table, the program was to continue downward. If the delay time to the 69 # error alarm and stop the program.

3.28 M68——Wait X table input signal revoked

Format: M68 Xxxx.x

Description: The program execution will stop and wait for instructions to M68 delayed 78 system parameters set until X table corresponding signal revoked, the program was to continue downwards. If the delay time to the 69 # error alarm and stop the program.

4 F、S、T function

F, S, T function is to feed function, spindle function, tool function for short.

4.1 F——Feed fucntion

The feed function is called F function in general, and F function can directly provide feed speeds of G01, G02 and G03. F function is signified by F and following digits, which the feed speed is mm/min. The feed speed of this system is between 1 mm/min to 15,000 mm/min, and user can randomly select according to actual cutting condition. Once F is set, it can be only modified by F value of posterior statements.

4.2 S——Spindle function

S0-S5(system saves S0-S5, which are gear change signal control of multi-speed motor), the spindle control is divided into the frequency control motor and spindle-use change speed motor (two-speed motor and three-speed motor).

4.2.1 The spindle with frequency conversion motor

The spindle control is implemented by the S××××, M03, M04, M05, 03# system parameter, 04# system parameter and 11#～14# system parameters which determines the analogue output controlled by the spindle (default is 0-10 V). In general, the machine tool spindle has one level manual gear shift to implement various rotating speed scope, which the bigger output torque still exist at low speed. The system can output high speed and low speed analog voltage, and the top limit of high and low gear is determined by 3# and 4# system parameters in the system parameters.

Fro instance:

03# system parameter = 5000，04# system parameter =1500,

When it's at the high gear (the 'H/L input' must be switched off with the '24V earth'):

$$V_S = (V_{ref} \times S) / 03\#\,\mathrm{systemparameter}$$

When it's at the low gear (the 'H/L input' must be switched on with the '24V earth'):

$$V_S = (V_{ref} \times S) / 04\#\,\mathrm{systemparameter}$$

In the formula: V_{ref} :Maximal analog voltage 10V;

V_s :The analog voltage actually output by CNC;

03# parameter: High gear top limit;

04# parameter: Low gear top limit;

4.2.2 The spindle operation controlled by digital quantity

Instructions for S1,S2,S3 (for multiple-speed motors), used with M03,M04,M05. M03, M04, M05 determines spindle forward, reverse, or stop.

Some machine tool has a manual gear shift, to achieve a range of different speeds, so that when the low speed is still large output torque, can be used outside of the input signal to control, for example:

Low-speed gear: 0-1500rpm:1500rpm 10V output voltage of D/A and the remaining speed linear change.

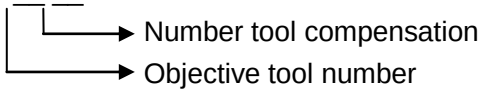
High speed gear: 0-5000rpm:5000rpm 10V output voltage of D/A and the remaining speed linear change.

Maximum speed can be adjusted according to user requirements.

High low-speed stall asked the external input signals a switch to inform CNC (see technical manuals). System can be controlled through the 8# parameter of S-MOT four spindle motor, when you S-MOT=1, controlled by the M41~M44-speed analog linear output.

4.3 T——Tool function

Format :T__.__



Description: Tool function, also known as t-functions, used for tool selection, back of the letter t for tool features and figures. When a mobile command and tool command appears in the same program, execution order has the following two ways:

1) Move instructions and tool command to execute simultaneously;

2) The tools command first doing,then move instruction is executed. Tool changing procedure specified by the PLC, the actual execution order refer to the machine tool manufacturer specifications.

Tool parameters address： this system a total of 24, from T01~T24, corresponding to the Cutter radius compensation, one by one. In PARAM State, press the F1 key, 24 tool parameters displayed on the screen, the operator can in this screen, type the actual tool

parameters. Tool parameters setting method as described in the section III-6.2.

3) The actual execution order of tool change procedure can custom as below:

a) Modify the system parameters P190, the value of the system parameters P005 +8000, P190 parameter is assumed to be 8001, then the tool change macro program for the P8001;

b) Editing tool change macro P8001, which can be redefined T command function. ATC macro is called, the system passes the following parameters:

1: The current tool number;

2: The current edge number;

3: Target tool number;

4: Target tool compensation number.

An example of tool change function：

```
N0010 #5=#[4000+#2]-#[4000+#4]+#5000
N0020 #7=#[4200+#2]-#[4200+#4]+#5002
N0030 M57K[#3]          ；Inform PLC to firing cutter frame and stop at target position
N0040 M54               ；Waiting until PLC to tool change finish
N0050 G92X[#5]Z[#7]     ；Adjust workpiece coordinate
N0060 M53K[#4]          ；Set Offset number to target one
N0070 M52K8             ；Get current position to local variable 8#
N0080 IF[#8==#3]G24     ；If current equal to target position?
N0090 M58K6             ；Alarm 06# error
N0100 G24N0110
```

5 Custom macro

Custom macro allows use of variable arithmetic and logic operations and library calls, making the procedures for the preparation of the same processing operations easier and more convenient.

5.1 Macro Variables

Ordinary user processes directly specified with the value g code, distance, and feed rate, for example, G01, and X100.0, when you use macro program, values can be specified with macro variables, macro variables the value specified by the program, such as:

```
N0010#1=1
N0020#2=100
N0030#3=500
N0040G[#1] X[#2*SIN[20]] F[#3]
```

5.1.1 Variable representation

The macro variable is specified by a symbol # and number.

For example: #1 And also can be expressed in an expression, variables, for example: ##1,#[#1+#2+2].

Note: we propose that the expression enclosed in parentheses to avoid ambiguity and error. For example: ##1, expressed as #[#1], meaning is taken with the value of the

variable #1 as a variable number of variable value, assuming the value of #1 to 100, then #[#1] is equal to #100.

5.1.2 The types of Macro variables

1. Null variable

We will not define the status of the value of the variable is called the "null", it has no value, or not equal to 0, it cannot be written, but it can read.

（a）Refer to Null variable

When referencing a variable is not defined, addresses itself also be ignored. For example:

G90 X100 Y[#0] be equa G90 X100

G90 X100 Y [# 0] is equivalent to G90 X100

（b）Assign value, addition, multiplication

To assign values to local variables or global variables directly as "null", its results as "null". Use "null" when the operation, the variable value is treated as a 0lai. For example:

1 = # 0; # 1 is null variables

2 = # 0 +1; # 2 to 1

3 = # 0 * 3; # 3 to 0

（c）Comparison operation

When operator for == or! =, "null" and 0 were found to be a different value.

When operators are >=, >, and <=, and <, the "null" and 0 were found to be the same value. For example:

When # 1 is null variable:

1 == # 0; established

1 == 0; not established

1! = 0; established

1> = 0; established

2. Local variables (# 1 ~ # 33)

Data stored in a local variable can only be used in a program, subroutines and other programs in the value and significance may be different. Local variables can be used to transmit the arguments. Have not been transferred to the local variable, in the initial state is "null", the user is free to use.

3. Global variables (# 100 ~ # 199)

The values and significance of the global variables in the different subroutine is the same ,When power on,the #100~#199 is initialized to null.

4 . System variables

PLC signal

System variables	function
#1000-#1031（Read-only）	Used to read from the PLC (G0054, G0055, G0056, G0057) to the user macro signals.
#1032（Read-only）	Read from the PLC for a 16-bit signal
#1033（Read-only）	Read from the PLC for a 32-bit signal
#1100-#1131（Read- write）	The signal sent from the user macro to PLC (F0054,

	F0055, F0056, F0057).
#1132（Read-write）	# 1132 for a write to the PLC 16-bit signal
#1133（Read-write）	# 1132 for a write to the PLC 32-bit signal

(1) System parameters (# 2000 - # 2199 read-only)

Format can be expressed as: # 2XXX, XXX is the system parameters, ranging from 0 to 199, for example: #2010, said the 10^{th} system parameters.

(2) Bit parameter (# 3000 - # 3398 read-only)

Format can be expressed as: # 3AAB, AA is the parameter number, value range is 0 ~ 39, B is the position number in the parameters of the AA bit number, ranging from 0 to 8.

When you 0<=B<=7, #3AAB read the AA bit parameter, b-bit value.

When B is 8, # 3AA8 read the value of the parameter in position AA;

(3) Tool parameter (#4000 read-only)

Offset number	X-axis compensation	Y-axis compensation	Z-axis compensation	Radius compensation	Tool tip position
1 : 99	#4001 : #4099	#4101 : #4199	#4201 : #4299	#4301 : #4399	#4401 : #4499

(4) Position message(#5000-read-only)

System variables	Function
#5000-#5005	Is respectively the previous segment end point of the workpiece coordinate values of XYZABC axis.
#5020-#5025	Is respectively the current coordinate system of machine tool value of XYZABC axis.
#5220-#5225	Is respectively G54 workpiece zero offset value of XYZABC axis
#5240-#5245	Is respectively G55 workpiece zero offset value of XYZABC axis
#5260-#5265	Is respectively G56 workpiece zero offset value of XYZABC axis
#5280-#5285	Is respectively G57 workpiece zero offset value of XYZABC axis
#5300-#5305	Is respectively G58 workpiece zero offset value of XYZABC axis
#5320-#5325	Is respectively G59 workpiece zero offset value of XYZABC axis
#5340-#5345	Is respectively zero offset value of XYZABC axis
#5360-#5365	Is respectively Tool point from the absolute coordinates value of XYZABC axis

5.1.3 Arithmetic and logical operations

Operators listed in the table can be executed in a variable or constant, operator can be a constant, variable, or on both sides of a function or expression operators, that is, variables #j and #k can be a constant, variable, or expression. I can be a constant, variable, or expression.

Function	Full format	Abbreviated format	Remark
Assignment	#i=#j	None	
Addition Subtraction	#i=#j+#k #i=#j-#k	None	

Product Division	#i=#j*#k #i=#j/#k		
Sine Cosine Tangent Arcsine Arccosine Arctangent	#i=SIN[#j] #i=COS[#j] #i=TAN[#j] #i=ASIN[#j] #i=ACOS[#j] #i=ATAN[#j][#k]	#i=SI[#j] #i=CO[#j] #i=TA[#j] #i=AS[#j] #i=AC[#j] #i=AT[#j][#k]	In degrees, 90°30' of 90.5 degrees
Square root Absolute value Round Integer Round down Round up Natural logarithm Exponential Power	#i=SQRT[#j] #i=ABS[#j] #i=ROUND[#j] #i=INT[#j] #i=FIX[#j] #i=FUP[#j] #i=LN[#j] #i=EXP[#j] #i=POW[#j][#k]	#i=SQ[#j] #i=AB[#j] #i=RO[#j] #i=IN[#j] #i=FI[#j] #i=FU[#j] #i=LN[#j] #i=EX[#j] #i=PO[#j][#k]	ROUND keep three decimal, rounded to the fourth INT rounded rounded
Logic and Logic or Logic not	#i=#j && #k #i=#j || #k #i=！#j	none	Logical operations, mainly for conditional
or XOR and not	#i=#j | #k #i=#j ^ #k #i=#j & #k #i=~#j	none	Bitwise operators can be used to determine the position parameters
Equal Not equal Great Great or equal less less or equal	#i=#j == # k #i=#j != # k #i=#j > # k #i=#j >= # k #i=#j < # k #i=#j <= # k	none	Relational operators are used for conditional
Pi	PI	none	Pi constantπ

instruction	
#i=ASIN[#j]	-1<＃j<1 -90°<#i<90°
#i=ACOS[#j]	-1<＃j<1 180°<#i<0°
ROUND function	Keep the 3 digits after the decimal point, the fourth rounding, rounding operations arising after the number whose absolute value is greater than or equal to the original number, such as: Prepared drilling program according to the value of the variable #1, and #2 cutting and then returned to its original

	location, assuming that the minimum set 1/1000mm variable #1 is the unit is five variable #2 1.2345 G00 G91 X[-#1] moves 1.235mm G01X[-#2]F300 moves 2.346mm Because the 1.2345+2.3456=3.5801 moved to 3.580 tool does not return to the initial position, the error comes from the rounded G00X[ROUND[#1]+ROUND[#2]] must be specified before or after rounding adding]] to make the tool returns to its original location.
Round up,round down an round to integer	If the operand of the fractional part is not 0, caused by rounding up operations on integer part of the integer number of absolute value is greater than the original, if it is less than the integer part of the original number is round down, after rounding to integer operation produces an integer whose absolute value is greater than or equal to the integer part of the original number, be careful handling of negative numbers. For example, And assume that # 1 = # 2 = -1.2 1.2 When you do # 3 = FUP [# 1], the 2.0 assigned to # 3 When you do # 3 = FUP [# 2], the -2.0 assigned to # 3 When you do # 3 = FIX [# 1], the 1.0 assigned to # 3 When you do # 3 = FIX [# 2], the -1.0 assigned to # 3 # 1 = 1.5 and assume that # 2 = -1.5 When you do # 3 = INT [# 1], the 2.0 assigned to # 3 When you do # 3 = INT [# 2], the -2.0 assigned to # 3
Operator precedence	(1) multiplication and division operations (*,/) (2) Add and subtract (+,-) (3) relational operators (==,!=,>, etc.) (4) logical (||,&&, etc.) (5) bit operations (|,^,&)朗读
The use of Brackets []	In the macro expression, Brackets can be used to change the order of operations For example: # 1 = 3 * # 1 value of 20-10 is 50 # 1 = 3 * [20-10] the value of 30 # 1 The system will automatically change the operator precedence order of operations For example: # 1 = 10 +2 * 10 # 1 = 30 When the macro expression in parentheses to change operator precedence or too many times (including the less than 10 times), the system will prompt an error.

5.1.4 Macro variables and expressions in the block of CNC

In the CNC program variables by using a macro or a macro expression needs to be added when "[" and "]", the formats are as follows:

(1)The format the use of macro variables: [#variable number].

(2) Using the macro expression format: [expression]. Connect with operator constants, macros, variables form the expression.

For example:

N0010G01X[#1+#2]F[#3]

N0010G01X[100*COS[50]＋20]

Minimum setting of the referenced address according to the value of a variable unit automatically rounded for example

When G00X[#1] to 1/1000mm units executing, CNC 12.3455 effective instruction value for the assignment to the variable #1 G00X12.346.

Change the reference value of the variable symbol you want to put a minus "-" placed in front of the sign such as N0010G00X[-#1]

Note: using the macro variable has not been assigned a value, it is assumed that is illegal, and prompt an error.

When you use a macro expression, note of operator precedence, you can use parentheses to change the order of operations when necessary.

5.2 Assignment statement

With a constant, or expression value is used to specify variable value is called assignment.

Format:

variable number = constant

= # variable number variable number

variable number = expression

For example:

N0010 #1 = 60; #1 value to 60

N0020 #2 = COS[#1]; #2 value = 0.5

N0030 #3 = 175 * #2; #3's value is changed to 87.5

N0040 #4 = #3; #4 becomes the value of 87.5

5.3 Branch and cycle

The use of GOTO statements in the program and the IF statement can change the control flow.

5.3.1 Unconditional branch

Format: GOTO n

Description:value for n is 0~9999, can be a number or expression.When the system when a GOTO statement is executed, the system from the file header to find the row number and n are the same, for example, n is equal to 200, the system finds N200 or N0200.

For example:

N0010GOTO200
N0010GOTO#2
N0010GOTO[#1+100]

5.3.2 Conditional branch

Format: IF [conditional expression] GOTO n

Description:When the condition value of the expression after IF is not 0, then the jump to the order number n of the block execution. If the condition expression evaluates to 0, then the excecute the next block.

For example:

N0010G0X0
N0020#1=0
N0030#1=#1+20
N0040G0X#1
N0050IF[#1 < 100]GOTO30
N0060

Conditional expression can be a macro variable or operational expression, the system will be calculated according to operator precedence, for example:

IF[#100]
IF[#100+20 > 100] That IF[[#100+20] > 100]
IF[#100 == 1 && #101 == 20] That IF[[#100 == 1] && [#101 == 20]]

Note: When expression calculation is more complex, we recommend the use of parentheses for ease of understanding.

5.3.3 Conditional execution

Format:

IF [conditional expression] # n = expression
IF [conditional expression] CNC block

IF conditional expression when established, the system execute the statement after "]", otherwise the next block.

For example:

N0010#1=1
N0020#2=20
N0030IF[#1] #2=10
N0040IF[#1] G0X[#2]

5.3.4 Loop

By assignment statements, IF and GOTO statements can be combined to achieve cycle control. The general format for recycling:

Format 1:

N0010 # 1 = 0; initialize loop counter, where # 1 is the loop counter
N0020 # 1 = # 1 +1; loop start, loop counter by 1
...
...
N0090IF [# 1 <10] GOTO20; cycle determine the statement, 10 for the number of cycles

N0100 ...; outside of the loop

Format 2:
N0010 # 1 = 0; initialize loop counter
N0020IF [# 1> = 10] GOTO100; cycle to determine if # 1> = 10 out of the loop
N0030 # 1 = # 1 +1; loop counter by 1
...; inside of the loop
...
N0090GOTO20; Skip to determine the statement cycle
N0100 ...; outside of the loop
For example:
N0010G01Z20X100
N0020 # 1 = 0
N0030 # 2 = 80
N0040 # 1 = # 1 +1; cycle started, the counter is increased by 1
N0050G01X [# 2]; feed
N0060G0X [# 2 +4]; return the tool
N0070 # 2 = # 2-2; set the target position
N0080IF [# 1 <20] GOTO40; Cycle 20
N0090 ...
...

5.3.5 The example of using macro

To achieve the circumference of a radius of 100 to play 20 holes in a radius of 200 to play 40 holes on the circumference, the use of macro programming is as follows:

N0010G0X0Y0Z20; positioning to the starting position
N0020 # 100 = 100; radius
N0030 # 101 = 20; round hole number of the week
N0040G20N10; N10 subroutine call
N0050 # 100 = 200; radius
N0060 # 101 = 40; round hole number of the week
N0070G20N10; N10 subroutine call
N0080M02

; N10 subroutine
N0005G22N10
N0010 # 102 = 0; # 102 as a loop counter
N0020 # 103 = 360 / # 101; hole angle between
N0030 # 104 = # 102 * # 103; current hole angle (loop starting position)
N0040G00X [# 100 * COS [# 104]] Y [# 100 * SIN [# 104]]; positioning to a specified point on the circle
N0050G01Z0F100; punch
N0060G0Z20; carrying tool
N0070 # 102 = # 102 +1; counter + 1
N0080IF [# 102 <# 101] GOTO30; judge whether the finish holes

N0090G24

5.4 System macro

Conclusion:

（1）Fast feeding is executed by G00 function and the velocity is defined by parameter P. We recommend to modify theP27.P28,P29 to proper small value before run fixed circulation.

（2）Before using fixed circulation instruction, you must use M03 or M04 instruction to slew the main axle.

（3）The hole drilling position and direction are decided by the relation between hole top entrance coordinate with hole bottom coordinate I.

（4）In fixed circulation programme segment, the coordinate Z of hole top entrance coordinate is the cutter retract stop point after circulation(except.

（5）13# error report will happen if the data within fixed circulation programme segment.is wrong.

5.4.1 Central hole bore drill circulation G81

N8100IF[#23==0]GOTO8106
N8102#26=#23+#5002
N8104#9=#9+#26
N8106#9=#9-#26
N8108#26=#26-#5002
N8110IF[#9>0&&#26<0]#0=0
N8112IF[#9<0&&#26>0]#0=0
N8114IF[#3170!=0&&#26==0]#0=0
N8116IF[#12>1]GOTO8128
N8118G0W[#26]
N8120G1W[#9]
N8122G0W[-#9]
N8124IF[#3170!=0]G0W[-#26]
N8126G24
N8128IF[[#21==#00]&&[#22==#00]]#0=0
N8130IF[[#21==#00]&&[#22==0]]#0=0
N8132IF[[#22==#00]&&[#21==0]]#0=0
N8134IF[[#21==0]&&[#22==0]]#0=0
N8136IF[#21==#00]#21=0
N8138IF[#22==#00]#22=0
N8140G0W[#26]
N8142G1W[#9]
N8144G0W[-#9]
N8146#12=#12-1
N8148IF[#12==0]GOTO8154
N8150G0U[#21]V[#22]
N8152GOTO8142

```
N8154IF[#3170!=0]G0W[-#26]
N8156G24
```

5.4.2 Central hole bore drill circulation with pause G82

```
N8200IF[#23==0]GOTO8206
N8202#26=#23+#5002
N8204#9=#9+#26
N8206#9=#9-#26
N8208#26=#26-#5002
N8210IF[#9>0&&#26<0]#0=0
N8212IF[#9<0&&#26>0]#0=0
N8214IF[#18==#00]#0=0
N8216IF[#18==0]#0=0
N8218IF[#3170!=0&&#26==0]#0=0
N8220IF[#12>1]GOTO8234
N8222G0W[#26]
N8224G1W[#9]
N8226G4K[#18]
N8228G0W[-#9]
N8230IF[#3170!=0]G0W[-#26]
N8232G24
N8234IF[[#21==#00]&&[#22==#00]]#0=0
N8236IF[[#21==#00]&&[#22==0]]#0=0
N8238IF[[#22==#00]&&[#21==0]]#0=0
N8240IF[[#21==0]&&[#22==0]]#0=0
N8242IF[#21==#00]#21=0
N8244IF[#22==#00]#22=0
N8246G0W[#26]
N8248G1W[#9]
N8250G4K[#18]
N8252G0W[-#9]
N8254#12=#12-1
N8256IF[#12==0]GOTO8262
N8258G0U[#21]V[#22]
N8260GOTO8248
N8262IF[#3170!=0]G0W[-#26]
N8264G24
```

5.4.3 Hole boring circulation(automatic return) G86

```
N8600IF[#23==0]GOTO8606
N8602#26=#23+#5002
N8604#9=#9+#26
N8606#9=#9-#26
N8608#26=#26-#5002
N8610IF[#9>0&&#26<0]#0=0
```

```
N8612IF[#9<0&&#26>0]#0=0
N8614IF[#3170!=0&&#26==0]#0=0
N8616IF[#12>1]GOTO8632
N8618G0W[#26]
N8620G1W[#9]
N8622M05
N8624G0W[-#9]
N8626IF[#3170!=0]G0W[-#26]
N8628M03
N8630G24
N8632IF[[#21==#00]&&[#22==#00]]#0=0
N8634IF[[#21==#00]&&[#22==0]]#0=0
N8636IF[[#22==#00]&&[#21==0]]#0=0
N8638IF[[#21==0]&&[#22==0]]#0=0
N8640IF[#21==#00]#21=0
N8642IF[#22==#00]#22=0
N8644G0W[#26]
N8646G1W[#9]
N8648M05
N8650G0W[-#9]
N8652#12=#12-1
N8654IF[#12==0]GOTO8662
N8656M03
N8658G0U[#21]V[#22]
N8660GOTO8646
N8662IF[#3170!=0]G0W[-#26]
N8664M03
N8666G24
```

5.4.4 Hole boring circulation with pause G89

```
N8900IF[#23==0]GOTO8906
N8902#26=#23+#5002                ;if at incremental mode，adjust W and I
N8904#9=#9+#26
N8906#9=#9-#26
N8908#26=#26-#5002
N8910IF[#9>0&&#26<0]#0=0
N8912IF[#9<0&&#26>0]#0=0
N8914IF[#18==#00]#0=0
N8916IF[#18==0]#0=0
N8918IF[#3170!=0&&#26==0]#0=0
N8920IF[#12>1]GOTO8938            ;if Lgreat than 1,do a row process
N8922G0W[#26]
N8924G1W[#9]
N8926G4K[#18]
N8928M05
```

```
N8930G0W[-#9]
N8932IF[#3170!=0]G0W[-#26]          ;tool nose park at start point or top of hole
N8934M03
N8936G24
N8938IF[[#21==#00]&&[#22==#00]]#0=0;nome of U and V appear,error
N8940IF[[#21==#00]&&[#22==0]]#0=0    ;if only V,and with a value of 0,error!
N8942IF[[#22==#00]&&[#21==0]]#0=0    ; if only U,and with a value of 0,error!
N8944IF[[#21==0]&&[#22==0]]#0=0      ;if both of U and V appear,but all is
0,error!
N8946IF[#21==#00]#21=0
N8948IF[#22==#00]#22=0
N8950G0W[#26]
N8952G1W[#9]
N8954G4K[#18]
N8956M05
N8958G0W[-#9]
N8960#12=#12-1
N8962IF[#12==0]GOTO8970
N8964M03
N8966G0U[#21]V[#22]                  ;move to next hole position
N8968GOTO8952
N8970IF[#3170!=0]G0W[-#26]
N8972M03
N8974G24
```

5.4.5 High speed deep hole processing circulation G73

```
N7300IF[#23==0]GOTO7306
N7302#26=#23+#5002
N7304#9=#9+#26
N7306#9=#9-#26
N7308#26=#26-#5002
N7310G00W[#26]
N7312#10=ABS[#10]
N7314#11=ABS[#11]
N7316IF[#9>0]GOTO7322
N7318#10=-#10
N7320GOTO7324
N7322#11=-#11
N7324#30=-#9
N7326IF[ABS[#9]<=ABS[#10]]GOTO7346
N7328G01W[#10]
N7330G00W[#11]
N7332#9=#9-#10-#11
N7334#10=#10-#11
N7336IF[ABS[#9]<=ABS[#10]]GOTO7346
```

```
N7338G01W[#10]
N7340G00W[#11]
N7342#9=#9-#10-#11
N7344GOTO7336
N7346G1W[#9]
N7348G4K[#18]
N7350G0W[#30]
N7352G24
```

5.4.6 Deep hole processing cycle G83

```
N8300IF[#23==0]GOTO8306
N8302#26=#23+#5002
N8304#9=#9+#26
N8306#9=#9-#26
N8308#26=#26-#5002
N8310IF[#09==0]#0=0
N8312IF[[#26>0]&&[#09<0]]#0=0
N8314IF[[#26<0]&&[#09>0]]#0=0
N8316#02=#09
N8318#10=ABS[#10]
N8320#11=ABS[#11]
N8322IF[#04!=#00]GOTO8328
N8324#4=#10
N8326GOTO8332
N8328#04=ABS[#04]
N8330IF[#04<#10]#0=0
N8332IF[#10<=#11]#0=0
N8334IF[ABS[#02]<=[#04]]#0=0
N8336IF[#02>0]GOTO8344
N8338#04=-#04
N8340#10=-#10
N8342#11=-#11
N8344#03=#17
N8346IF[#03<0.1]#03=0.1
N8348G00W[#26]
N8350G01W[#04]
N8352G00W[-#04]
N8354G4K[#03]
N8356#02=#02-#04
N8358IF[ABS[#02]<=ABS[#10]]GOTO8374
N8360#02=#02-#10
N8362G00W[[#04]-[#11]]
N8364G01W[[#10]+[#11]]
N8366#04=#04+#10
N8368G00W[-#04]
```

N8370G4K[#03]
N8372GOTO8358
N8374#10=#02
N8376#02=0
N8378G00W[[#04]-[#11]]
N8380G01W[[#10]+[#11]]
N8382#04=#04+#10
N8384G4K[#18]
N8386G00W[-#04]
N8388G24

5.4.7 Precision boring circulation G78

N7800IF[#23==0]GOTO7806
N7802#26=#23+#5002
N7804#18=#18+#26
N7806IF[[#21==#00]&&[#22==#00]]#0=0
N7808IF[[#21!=#00]&&[#22!=#00]]#0=0
N7810IF[[#21==#00]&&[#22==0]]#0=0
N7812IF[[#22==#00]&&[#21==0]]#0=0
N7814#18=#18-#26
N7816#26=#26-#5002
N7818G0W[#26]
N7820G1W[#18]
N7822G4K[#11]
N7824M5
/N7826M19
N7828IF[#21==#00]G0V[#22]
N7830IF[#22==#00]G0U[#21]
N7832G0W[-#18]
N7834IF[#3170!=0]G0W[-#26]
N7836IF[#21==#00]G0V[-#22]
N7838IF[#22==#00]G0U[-#21]
N7840M3
N7842G24

5.4.8 Counter Precision boring circulation G87

N8700IF[#23==0]GOTO8706
N8702#26=#23+#5002
N8704#18=#18+#26
N8706IF[[#21==#00]&&[#22==#00]]#0=0
N8708IF[[#21!=#00]&&[#22!=#00]]#0=0
N8710IF[[#21==#00]&&[#22==0]]#0=0
N8712IF[[#22==#00]&&[#21==0]]#0=0
N8714#18=#18-#26
N8716#26=#26-#5002

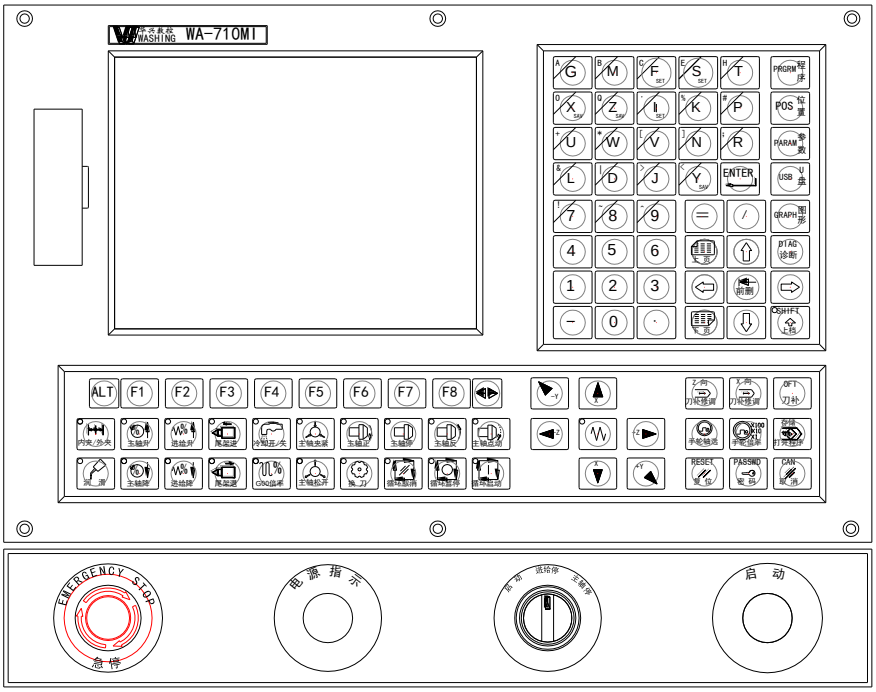
```
N8718IF[#26==0]#0=0
N8720M5
/N8722M19
N8724IF[#21==#00]G0V[#22]
N8726IF[#22==#00]G0U[#21]
N8728G0W[#26]
N8730G0W[#18]
N8732M3
N8734IF[#21==#00]G0V[-#22]
N8736IF[#22==#00]G0U[-#21]
N8738G1W[-#18]
N8740G4K[#11]
N8742M5
/N8744M19
N8746IF[#21==#00]G0V[#22]
N8748IF[#22==#00]G0U[#21]
N8750G0W[-#26]
N8752IF[#21==#00]G0V[-#22]
N8754IF[#22==#00]G0U[-#21]
N8756M3
N8758G24
```

III Operation

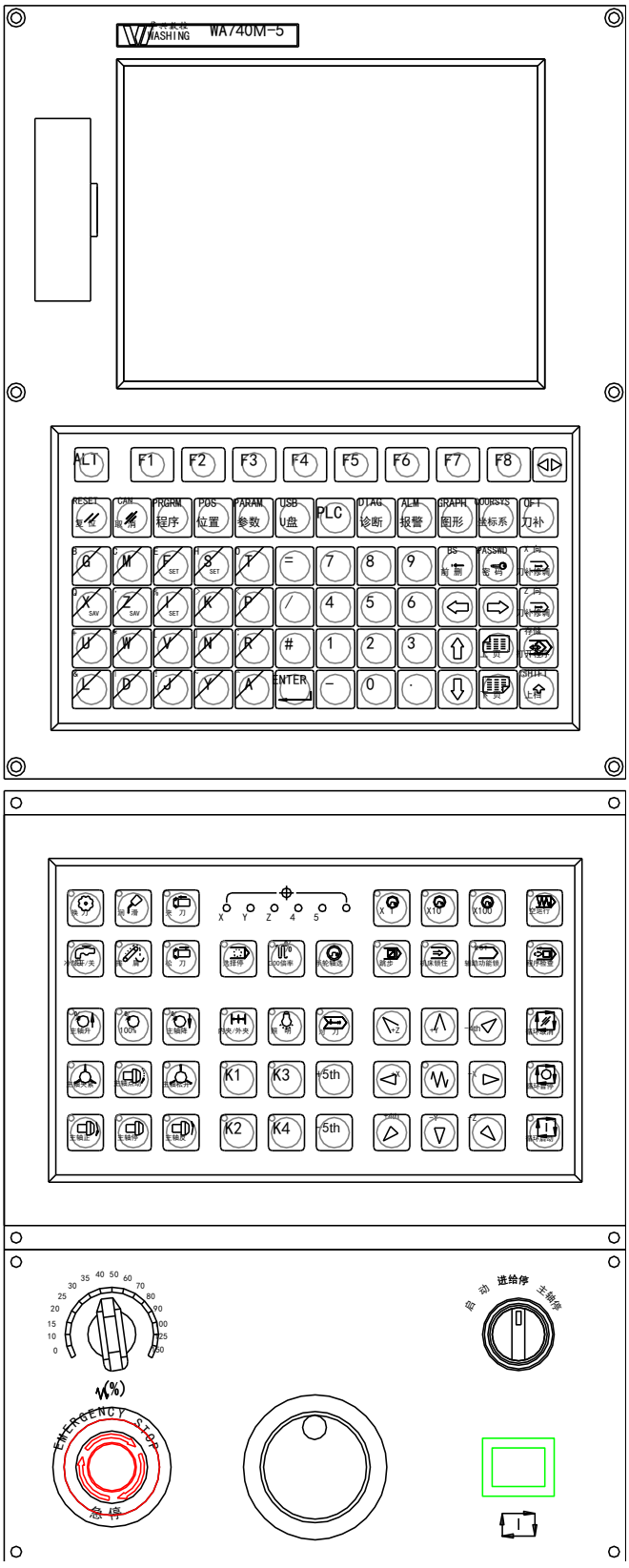
1 LCD (liquid crystal) display

1 Operation panel

WA-7XXT lathes numerical control system with integrated and split two categories in the operator panel, shown in F3.1, and in F3.2. Operator panel is divided into LCD (LCD) display area, the main function keys area, f keys, edit keyboard areas and machine tool control key areas, such as several major areas.



F3.1 WA710MI the system Panel (integrated)



F3.2 WA740M-5 system Panel (split)

1.1 LCD (liquid crystal) display

The system's display with a resolution of 800x600 color/8.4-inch LCD monitor,

provides a wealth of machine information is displayed for the user.

1.2 The main function keypad

The main function key region includes 9 keys of:

Which are used for the 9 primary functions of the system. Their detailed functions and meanings are shown as table 3.1.

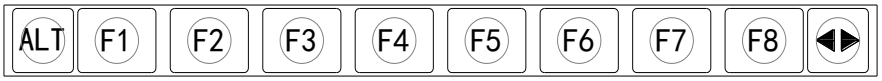
Table 3.1 Main function key description

Key	Symbol	Description
	[PRGRM]	Press this key to display the program screen.
	[POS]	Press this key to display the position screen.
	[PARAM]	Press this key to display the parameter screen.
	[USB]	Press this key to display the usb disk screen.
	[PLC]	Press this key to display the PLC screen.
	[DIAG]	Press this key to display the diagnose screen.
	[COORSYS]	Press this key to display the coordinate system screen.
	[ALM]	Press this key to display the alarm screen.(valid in WA-730M)
	[GRAPH]	Press this key to display the graph screen.(valid in WA-730M)

1.3 F function keyboard region

The F function keyboard region is located below the LCD screen, and is used for the operation of selecting corresponding sub-functions under every primary function interface. The sub function keyboard region is also called the F function keyboard region. For unified statements, we take the F function keyboard region as the substitute of the sub function keyboard region, and take F function keys as the substitutes of sub function keys, for expression. When exceeding 8 keys, pressing "◄►" key to switch to soft function menu of the next page.

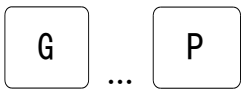
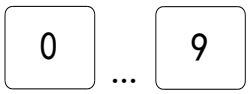
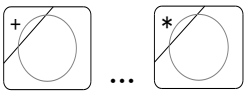
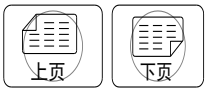
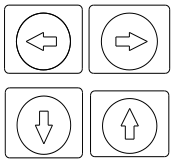
Table 3.2 Key-press description for sub-function keyboard region

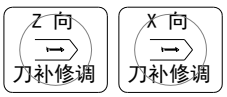
Function Keys	
Symbol	【ALT】【F1】【F2】【F3】【F4】【F5】【F6】【F7】【F8】【◂▸】

1.4 Editing keyboard region

The editing keyboard region provides the key-presses needed in editing the processing codes and setting up and modifying every kind of parameter value. The editing function keys are shown as table 3.3:

Table 3.3 Editing key description

Key	Symbol	Description
	【G】…【P】	Letter key. With G M F S T X Z I K P L D J N U W R…
	【0】…【9】	Number key. With 0 1 2 3 4 5 6 7 8 9
	【+】…【*】	Macro operator keys,effective when SHIFT is on.
	【ENTER】	ENTER key.Creating new lines under the file editing interface, and indicating the end of the current input in other status.
	[PAGEUP] [PAGEDOWN]	Page turning key. Showing the content of the previous page or the next page for the interfaces such as the program name list, program content, parameters, etc.
	【←】、【→】、【↑】、【↓】	Direction keys of up, down, left and right.

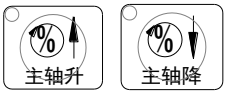
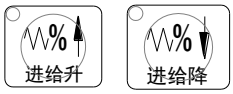
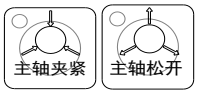
	【Fset】	F Set, is used for manually setting up the feed speed in position screen.
	【Iset】	I set, is used to set the length of increase move.
	【Sset】	S Set, is used for manually setting up the rotation speed of the spindle.
	【Gmdi】	G MDI, is used for indicating the input display of 'G' character under the statuses of file editing or character inputting, and indicating the entering to the input status of MDI mode under the operation and processing primary function interface.
	【Xsav】	X Save, is used for saving the numeral value of the X axis in the current programming coordinate system, as the measuring cutting tool compensation value parameter.
	【Zsav】	Z Save, is used for saving the numeral value of the Z axis in the current programming coordinate system, as the measuring cutting tool compensation value parameter.
	[PASSWD]	Password input key.
	[SAVE/OPEN]	OPEN key is valid in Position screen ,and SAVE key is valid in other screen.
	[OFFSET]	Tool offset setting key.
	[MODIFY]	Tool offset modify key.
	【CAN】	Press this key to cancel the input.
	【SHIFT】	Some keys have two characters on their keytop. Pressing the [SHIFT] key switches the characters.
	【RESET】	Press this key to reset the CNC, to cancel an alarm, etc.

Note: Edit the key to some of the keys are multiplexed with alternate function key, the system according to the current state. For example, in the program edit mode, press the "G" key is a "G" character at the cursor in the editing area, and in the position of the main screen, press the "G" key is to enter the MDI mode.

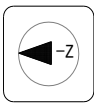
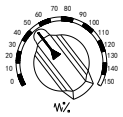
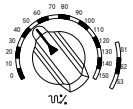
1.5 Operate keyboard region

Operate keyboard region is to do machine tool automatic, manual operation, switch control, and alarm actions such as emergency stop.

Table3.4 Operate key description

Key	Symbol	Description
	[Spindle CW]	Spindle clockwise rotation (implement M03)
	[Spindle CCW]	Spindle counterclockwise rotation (implement M04)
	[Spindle stop]	Shutting down the spindle, and stopping rotation (implement M05)
	[Spindle inc-move]	Spindle pulse control. The spindle rotates clockwise when this key is pressed, and the stops rotating when this key is released.
	[Spindle up] [Spindle down]	Spindle increase/decrease: dynamically adjusting the spindle multiply factor (only valid for main motor speed control by frequency variation).
	[Feed up] [Feed down]	Feed up/down: dynamically adjusting the feed multiply factor
	[Cool on/off]	Coolant on/off
	[Tool change]	Single-step tool-change: The tool will be changed to the next in turn by the system for each pressing.
	[Lub on/off]	Lubrication on/off key.
	[Spndl clamp] [spndl release]	Spindle clamp/release

	[Clamp in/out]	Select the inward/outward clamping chuck key.
	[Tailstock in] [Tailstock out]	Tailstock into/exit key.
	[Axis Select]	Hand pulse axis selected keys.
	[X1] [X10] [X100]	Hand pulse ratio setting key.
	[Cycle start]	Cycle start. It starts the processing and implementation of the current programs. Under the OPERT primary function interface, if the programs, which have been selected by the user and waiting for being processed, are used, the processing of the current program can be started to run after pressing this key under the 'automatic' sub-function.
	[Cycle pause]	Cycle pause (feed maintenance). The system pauses the implementation of the current program, until the implementation of the follow-up program segments is continued by the user pressing the Cycle start key.
	[Cycle cancel]	Cycle cancel. It terminates the automatic cycle processing operation of the current program, and this key-press is only valid when the system is in the cycle pause (feed maintenance) status.
	[G00 ovrd]	Rapid location rate switching keys
	[+X]	Manual feed in the X positive direction.
	[-X]	Manual feed in the X negative direction
	[+Z]	Manual feed in the Z positive direction

	[-Z]	Manual feed in the Z negative direction
	[Rapid move]	Rapid feed manually select keys.
	[Skip]	Program skip step key.
	[Light]	Machine tool light.
	[Dry run]	Dry run key.
	[Prgrm chk]	Program check key.
	[Feed ovrd]	Feed rate dial switch.
	[Spindle ovrd]	Spindle rate dial switch

Note 1: The machine control keypad buttons and some are optional, so not every series milling machines or machining centers have all the key functions described above.

Note 2: Operation chapter in the "parameter P012" said the 12th of system parameters (P indicates the system parameters); "parameter B047" said No. 7 on the 4th bit parameter (B indicates bit parameter), the other and so on.

2 System power up, shutdown and safety operations

2.1 System power up

Before the system is powered on, verify that:

1. The appearance of the CNC machine tool is normal;
2. The power supply meets the requirements;
3. Connection correctly, securely.

After Power up,and system self-test is normal, when the initialization is complete, displays the splash screen. P017 can set splash through the system.

2.2 System shutdown

Before the shutdown, you should confirm:

1. Movement of CNC axis is in the stopped state;

2. Auxiliary functions (such as spindles, water pumps, etc) are shutdown;

When you cut off the power, should make the following checks:

1. Check the LED on the operator panel indicates that the cycle should start in a stopped state;

2. Check for CNC machine tools of all moving parts are in a stopped state;

3. Cutting off power shutdown.

Note: emergency power off:

In the process of running immediately power down the machine in case of emergency, to prevent accidents. But it should be noted that, after cutting off the power system coordinates with the actual locations may be biased, must return to reference position again, set tool, and so on.

Note: on power down the machine in action, see Manual of machine tool in machine-tool factory.

2.3 Safety operations

Under normal circumstances, driven by stepping motor open loop due to their own principles, overtravel stall occurred does not have a significant impact on machinery, and systems for AC servo motor as actuator, overload capacity in AC servo, sharply increase in output torque, mechanical damage and even accidents can occur. Therefore, security for AC servo unit of machine tool driven machine was especially important. Through various aspects of the system to limit the possibility of mistakes.

2.3.1 Emergency Stop

In case of emergency, press emergency stop button, the system on receipt of the emergency stop signal, switch to manual mode, 55# alarm occurs, all axis motion stop immediately such as the spindle of machine tool rotation, coolants, also turned off. At the same time button had been blocking in the stop position. Button release varies with different machine tools factory, usually by pressing the button clockwise to release.

Note

1. This button is pressed, the electrical power is cut off.

2. Control unit in the reset State.

3. Troubleshooting before you release the button.

4. After the button is released, G74 instructions for manual operation or return reference point.

Emergency stop signal circuit connection are as follows: emergency stop button should have a normally open/normally closed contact, where the normally open contacts to the system, so that the emergency stop button is pressed when the system enters emergency status. Emergency stop button normally closed contact strongly recommended access strong cabinet to the primary circuit of the machine (spindle and servo) power control circuit, so that in emergency situations, with the highest reliability assurance of spindle and servo to go down.

2.3.2 Hard Limit

Machine tools that use AC servo as the implementation components, each axis

should be fitted with triple high reliable mechanical limit switch, when the system main circuit of soft limit does not work force cut off power supply control circuit (see emergency stop), General triple switch: two joins in the strong electric control circuit as limited input in two directions, third way as positioning signal in the early return of zero-point machine.

Because of the proximity switch actions not directly off the control loop, it is generally not recommended for inductive proximity switches as a servo axis limit switch, if it is necessary to adopt, should select the NPN-OC output closer to the door switch.

2.3.3 Soft limit

System provides real-time monitoring system for internal timing detection function of coordinates are crossed artificially set interval, once over, stop running, switch to manual mode, and a 40# or 42# alarm occurs, the process consists of a series of reference system.

1. Parameter B002 decision is to machine coordinates is to coordinate the work as a soft limit of coordinate reference. (B002=0, in machine tool coordinates; B002=1, in workpiece coordinates.)

2. Parameter B001=0: soft limit function is valid after returning the reference point, B001=1: soft limit function need not return reference point.

3. Soft limit parameters for each axis P060~P071 definitions, once the system choice of coordinates (machine tool or workpiece coordinates) over the axis interval, system alerts.

4. When the limit occurs when the axis movements down stops.

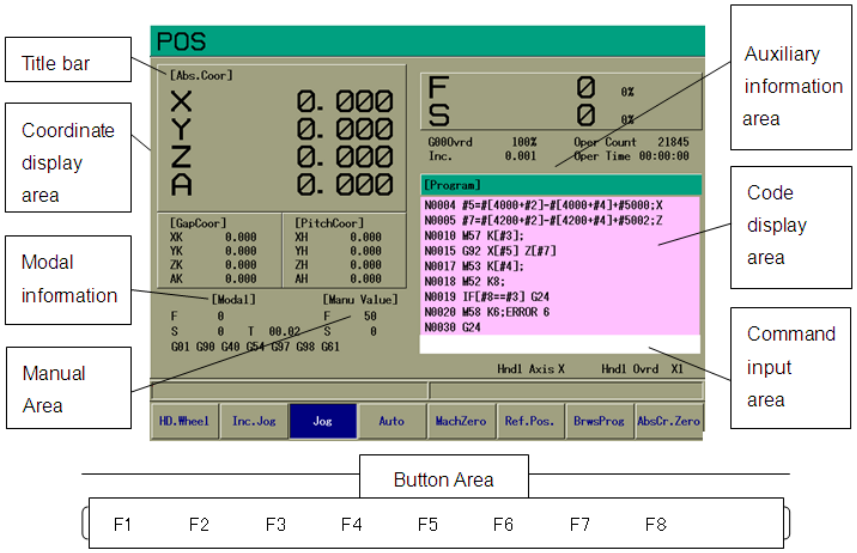
5. When the soft/hard limit slow down when stopping, its acceleration of negative time constant determined by the parameter P045, G00 maximum speed limits are used to calculate the acceleration at a rate of.

6. Soft limit slow down when stopping, will cause overshoot over the soft limit, its acceleration the less, over the interval is longer, can reduce the time constant (P045) methods to improve acceleration, reduce the bounds length. P045 shall be less than the time constant G00, G01.

3 Position screen functions

Press the key [POS] to display the postion screen, as shown in Figure 3.1. The position screen provides the operation and control for the system and the machine tool,And display the information to the user in real time, the main function including cycle start, MDI operation ,manual continuous feed, hand wheel operation, return reference point of machine tools, rapid compensation and trimming, graphic simulation and so on. Under this screen has the various sub functions select by F function keys F1~F8.

3.1 Introduction of position screen



F3.3 Position interface (absolute coordinates)

1. The title bar area, displays the current primary function, and displays the current procedure and the starting segment number.

2. Coordinate display, position screnn there are three coordinates displayed. Press [ALT] key, you can switch several display modes, the user can set in the parameter P091 coordinates displayed when you turn on this interface, default is to display an absolute coordinate.

3. Modal information area. Under the interface where dynamically displays the value of f, s, t and g mode code.

4. Manual, displays the feed rate and spindle speed in manual mode setting values.

5. Coordinate dynamic displays the actual feed rate and spindle speed.

6. Auxiliary information area, processing piece, cutting time: dynamic displays the number of machining, process time consuming; G00 rate, distance: G00 ratio value and displays the current state to move the distance value.

7. Code display, to display nine lines processing procedures, automatically highlight current operative paragraph in the recycling process.

8. Directive input area, MDI code input, f, s, I set value input.

9. Rate of spindle and feed rate displayed.

10. Button area, for the various machine f of functions under the location interface prompts. Display interface [HD.Wheel], [Inc.Jog], [Jog] and [Auto], [MachZero], [Ref.Pos.], [BrwsProg], [Graph] eight ways, corresponding F1~F8 function keys. By pressing the appropriate f key into the corresponding machine tool operation.

3.2 Automatic operation

Automatic cycle processing starts, the corresponding automatic operation of the machine.

3.2.1 Run Automatically

3.2.1.1 Open program

In the position screen, Press [F4](Auto) key to enter automatically State, then press [Open/Save] key, screen left below appears the input box, user can input the prepared program name in here, [←] key for modify program name when entered the wrong character; screen middle is the program list displaying information of all the user program including program name, and program size, and program property, ser can select the corresponding program. Enter the correct program name and press [ENTER] key, system is ready to run the program.

3.2.1.2 Start automatic cycle

Press [cycle start] button, and then the program begins. If before the cycle, press the [F6] (StartSeg) key, then system prompts the user to enter a line number, enter the line number and press [cycle start], then the system start execute the the program from the entering line number. For example, enter N0100, and then press the [cycle start] key, you find with N0100 as the line number of the first segment of the program. program segments in the front do not perform.If not found,then start from the first line, and prompt an error.

Under this system, the origin of the program, is the point where the coordinates display zero, all programmed tracks are based on the coordinates. Programming coordinate system coinciding with the workpiece coordinate system.

For instance：The programming is：

G90 G01 X10 Z30 F100

However, when cycle starts, the big coordinate of system is：

X —50.000

Z 3.500

Then the results of the execution above are: X coordinate moves 30 mm along the positive direction, and Y coordinate moves 26.5 mm along the negative direction, in addition, Z coordinate moves 26.5 mm along positive direction.

After pressing "Cycle start" key, the system firstly implements necessary internal processing such as examination, error detection etc. If error occurs, the note of error will appear; otherwise the program will be executed in order..

Note:

(1) The program name of auto-cycle processing will be called out by file open key.

(2) The figure simulation or track will be accessed under automatic processing, and see 3.5 section for specific contents.

3.2.1.3 Cycle stop

When the program running automatically, the system provides five methods to stop the automatic running:

1. Program stop (M00) directives or [Cycle pause] key

When M00 program execution, the program halts, all modal information have been saved. Press [cycle start] keys, the program continues to perform.

2. End of the process (M30) directives or [cycle to cancel] button

When M30 program execution, or in the pause state pressing [cycle cancel] button, the program stop running. If press [cycle start] button, then start to operate the program from the beginning.

3.2.2 MDI operation

In the position screen, press the letter key [G], [S],[T], The command input area appears the Caret, then typing a program segment, press [ENTER] key,and the system execute the commands. The program does not have to enter a segment number. In automatic mode the executable which can be performed in MDI mode.

Before pressing "ENTER" key, if an type error occur, press [←] key, delete the code individually; Before pressing the [ENTER]key, if you want to cancel the MDI operation, you can press any main function key or press [CAN] key to cancel the operation.

When MDI runs, the stop method is the same as automatically run, see automatically stop running.

3.2.3 DNC operation

In the automatic mode, you can use u disk DNC function.

3.2.3.1 DNC operation start

User can copy the program into a USB disk and insert it into a USB port on the system panel, press the [F3](USB DNC) key to display the screen for file selection.

You can press [up] key or [PAGE DOWN] key to browse USB disk file list of the current directory, if the cursor stays on the folder, you can also press "ENTER" key to enter the next level of sub-directories, "↑" (above) keys, "↓" (direction) key to rest the cursor on the file to be processed.

Press "ENTER" key and returns the position screen, and then "cycle start" button, start USB DNC system processing.

3.2.3.2 DNC operation pause and end

When system DNC processing, the user can pause or stop the process at any time.

Press [cycle pause] key to make the system into the processing pause state, the system temporarily suspended the execution of the program. Until you press [cycle start] key system will continue to run after, if press [cycle cancel] button, the DNC process is terminated.

3.2.4 Operation status

In the "Auto" sub-function of the position screen, there are two types of status:

1. Single segment: press [F5](SinglSeg) key to switch back and forth. When in effect, [SinglSeg] button is pressed, each perform a segment of the system will stop and wait for user input, each time you press [cycle start]button, the system executes a program segment down, if pressed [cycle cancel] key at the timer, then this cycle will be cancelled, and cannot be recovered.

2. Pause: corresponds to [cycle start], automatically cycle can be paused by pressing [cycle pause] key. When pausing, press the [cycle start] to continues to run, if press [cycle cancel] button, this automatic processing was cancelled.

3.2.5 Feedrate control in automatic mode

When running in automatic mode, the feed rate of the system can be changed by modifying the feed rate. Press [Feed up] button or [Feed down] key to change the feed rate, feed rate can achieve a 0%~150% 15 degree adjusting.

Note:

Feed rate is to adjust the programming speed.

Actual feed rate = set feed X feed rate ratio.

When automatically run, you can also press [G00 rate] key to select the speed of the rapid location, fast movement speed by P012 set manually, quick ratio F0,25%,50%,100% four regulation can be achieved.

3.2.6 Spindle speed control in Automatic operation

When running in automatic mode, the system can modify the spindle speed by changing the spindle speed rate. Press [spindle up] button or [spindle down] button to change the spindle rate,Spindle ratio 0%~150% a total of 15 level adjustment can be achieved.

Note:

Spindle speed ratio is to adjust the programming spindle speed .

Actual value x spindle spindle speed =S set magnification

3.3 Manual operation

Manual operation includes two modes of manual continuous feed and stepping feed, which are called manual mode and pulse control mode, and both refrigeration and spindle can be manually operated.

1. Manual mode: Manual mode can be accessed by pressing [Manual] key, and under the manual mode, ◄Z-、Z+►、X-▲、X+▼ signify operating keys which are used for various coordinate axes to move along their positive directions or negative directions. When pressing one of them, the corresponding coordinate axis will move long its direction. The feed speed can be manually set by pressing [Fset]. When pressing one of feed keys above and "  " simultaneously, the speed operation can be set by pressing parameter.

2. Incremental jog mode: It's same as the manual mode that presses [F2] key to access pulse control operating mode. The coordinate moves some given length along its corresponding direction by every coordinate feed key pressing. This length is set by [Iset].

3. Settings for manual operation parameter: They are only valid in manual and pulse control modes.

Pressing [Fset] key: Settings for coordinate movement speed in manual or pulse control modes (letter key region [Fset] key), and the cursor will appear after word 'Feed' with pressing [Fset], then the digits can be input, which signify millimeters for feed per minute. It'll be valid after pressing [ENTER].

The scope of this speed is 1～6000mm／min（when it's 0.001mm pulse equivalent), if input has error, the system will automatically set for 50.00mm ／min."F1" key can be used to modify the wrongly-typed digits.

Pressing "I set" key: Setting up pulse control stepping amount ("I set" key in the letter key region), it's only valid in the pulse control which is used to set up stepping amount.

Pressing "I set" and inputting stepping amount after cursor (0.001～65.5mm).

Pressing "S set" key: Setting up speed of spindle ("Sset" key in letter key region), and inputting digits in the cursor position which signifies speed of spindle. It'll valid after

pressing "ENTER". The system outputs the analogue which is correspondent to rotating speed. The top limit of this rotating speed is set with P13# parameter to P16# parameter.

(4) Spindle and refrigeration: In the modes of manual, hand wheel and pulse control, both spindle and refrigeration can be manually operated

3.4 Hand wheel operation

Towards to lathe control system, hand wheel can control machine tool's rectilinear movement in directions of X or Z. Hand wheel sets up three speed gears (multiply factors) for various requirement which are respectively X1, X10 and X100. Switch can be implements among speed gears, and the minimum control precision is identical with system control one, while the maximal control speed is 100 times than the pulse equivalent. Hand wheel is principally designed for fast rectilinear movement, presetting cutter etc. of machine tool.

Hand wheel operation is as follows:

1. Press [POS] primary function key in the main menu to access machine tool position screen.

2. Press [F1] to access hand wheel operating mode.

3. Press [axis select] key to select movement coordinate axis, simultaneously the selected coordinate axes are displayed on the screen.

4. Press [wheel ratio] key to select hand wheel multiply factor, simultaneously the selected multiply factor is displayed on the screen.

5. Turning hand wheel, and the machine tool makes response movement.

6. To quit hand wheel status, either "F" function key (F1—F5) can be pressed.

Note: If the hand wheel above 5 revolutions per second, the distance wheel rotation and machine tool moved difference will appear. Handwheel rotation not too fast.

3.5 Return to machine tool zero and reference point

3.5.1 Machine zero concept

Machine coordinate system is inherent in the machine coordinate system, origin of the coordinate system of the machine referred to as machine zero point (or zero point of machine tools), also known as point of reference in this manual, usually in the direction of x, y, z axis is at the maximum stroke. In the machine after the design, manufacture and adjustment, the origin is established, it is a fixed point. Did not know when machine tool CNC unit zero, usually to zero point of automatic or manual machine tools.

3.5.2 Machine zero operation

Machine back to zero can be manually return to zero or by g-code back to zero.

(1) Manual machine zero return

1.1）In the position screen, press F5 (MachZero) to enter zero returning mode.

1.2）Select axis which will return to machine zero,Press "-X", "+X", "-Y", "+Y", "-Z", "+Z" key to return machine zero,the direction set by the parameter B070~B075.

1.3）Machine tools move along the machine zero point, before the deceleration point, machine tool moved quickly (speed set by the parameter P086), met after the

deceleration switch to FL (set by the parameter P140~P151) moves to the machine at a rate of zero (or reference point). Back to the machine zero, and the axis stops moving, zero return light on.

(2) When you execute the G74XYZ... instruction, the axis of the system will move until it find the switch signal of the zero point, and machine coordinate systems such as x, y, z, automatically reset to 0, that represents the current point where the machine coordinate system origin locations, thus establishing the machine coordinate system.

Note 1: If the zero point of CNC machine tools machine tools are not installed, please do not use the machine back to zero operations;

Note 2: At the end of the machine zero points returning, the corresponding axis of the lamp is lit;

Note 3: Sfter returning to the zero point of machine tools, turn off the light in the following cases: when the operator when the corresponding axes from the zero-point machine;

Note 4: Machine zero point (or reference point), please refer to the manual of machine tool manufacturers of machine tools.

(3) Machine zero returning related Parameters

Bit parameter:

B035=1: =1:Need not return reference; =0:need return reference.

B034=1: zero out workpiece coordinate system after return to machine zero; B034=0: restoring workpiece coordinate system.

B033=1: soft limit is invalid when return to the reference point;

B030=1: motor keep moving when release the manual move key;

B016=1: zero out the relative coordinates when return to machine zero,and B035=0;

B015=1: restore the relative coordinates when return to machine zero,and B035=0;

B070~B075=0: looking for zero signal along the positive direction. = 1: find zero point signal along negative direction.

System parameter:

P086: speed of the axis returns the reference point, = 0: the speed can be defined separately.

P100~P105: the reference speed for each axis when P086=0;

P140~P151: The low speed when exit rough positioning to the reference point and the low speed of search for precision positioning.

3.5.3 Return to entry point

Machine back to the entry point of the system is easy to operate, the steps are as follows:

1. In the parameter interface, enter the coordinates offset interface, set the entry point;

2. In position screen, click [Ref.Pos.] button corresponds to the [F6] key, the operations become return machine tool feed point;

3. G76XYZ…will appear in the MDI input box, and the cursor flashing, press [ENTER] key back machine entry point.

3.6 Tool offset

In order to make easier and faster machine tool setting and trimming, in position screen, press [tool measure] keys to set the tool compensation values directly, press "x tool modify" or "z tool modify" keys directly modify the tool offset.

3.6.1 Tool offset measure

This system uses a test cutting method, before the establishment of tool compensation, must establish a work-piece coordinate system first, the workpiece coordinate system can be setup by G54~G59 (see section 6.6 coordinate offset), or could be established with the G92. For ease of operation, System x, z to separate save tool parameter or save x, z at the save time, the steps are as follows:

(一) Independently memorize presetting cutter in X & Z direction

a. Presetting cutter method in X direction

(1) Clamp one workpiece on the chuck;

(2) Enter manual operation mode;

(3) Turn the holder to select the cutter number m and cutter compensation number n required (see Section 2.4.3 for operation of setting cutter number and cutter compensation number);

(4) Set appropriate spindle rotation speed and manual feed speed, start spindle, move holder and use cutter selected to turn one section of excircle (or inner bore) on the workblank, press Xsav key and the status bar in the screen lower right corner will display "X direction cutter bias has been saved";

(5) Manually exit cutter, stop spindle and measure diameter Φ of excircle (inner bore) at cutting position, unit: mm, for instance: 19.785mm;

(6) Under operation & processing interface, press Cutter Bias key to enter cutter parameter interface, the hint column in the screen will display "Please input Tn X measured value", input the diameter Φ measured in the input procedure (5) on the keyboard; after ENTER key is pressed, press key to save the parameter after being modified to the electronic disk.

Note: If the cutter implements cutting at the other side of workpiece axial line (the cutter is in the negative direction of axial line), the diameter input will be negative value.

b. Z direction presetting cutter method

(1) Enter manual operation mode;

(2) Start spindle, move holder, utilize No. m cutter to turn one end surface on the workblank, then press Zsav key, the status bar in the lower right corner of the screen will display "Z direction cutter bias has been saved";

(3) Manually exit cutter, stop spindle and measure the length L from end surface of cutting position to chuck end surface (it can also be selected by the user; however, this position should be regarded as base every time), unit: mm;

(4) Under operation & processing interface, press Cutter Bias key to enter cutter parameter interface, the hint column in the screen will display "Please input Tn Z measured value", input the length L measured in the input procedure (3) on the keyboard; after ENTER key is pressed, press Open/Save key to save the parameter after being modified to the electronic disk. The relative position for this cutter in the workpiece

coordinate system has been confirmed.

Note: When each cutter implements presetting cutter, the base of measurement (such as chuck end surface) must be the same, else the cutter compensation value generated in Z direction will be incorrect.

（二）Simultaneously memorize presetting cutter in X & Z direction

(1) Clamp one workblank piece on the chuck;

(2) Enter manual operation mode;

(3) Turn holder, select the cutter number m and cutter compensation number n requiring presetting cutter;

(4) Set appropriate spindle rotation speed and manual feed speed, start spindle, move holder, utilize No. Tm cutter selected to turn one end surface on the workblank, press Zsav key to exit cutter to proper position along X-direction, press one section of ex-circle and press Xsav key;

(5) Exit cutter, stop spindle and measure the length L from diameter Φ of ex-circle cut and workpiece end surface to chuck end surface;

(6) Press Cutter Bias key on the subpanel to enter cutter parameter interface, the hint column in the screen will display "Please input X measured value of Tn", after the diameter Φ is input, press ENTER key for confirmation. The hint column in the screen will display "Please input Z measured value of Tn", after inputting the length L, press ENTER for confirmation, press Open/Save key to save the parameter after being modified to the electronic disk. After the cutter compensation is established, the relative position for this cutter in the workpiece coordinate system will be confirmed accordingly.

3.6.2 Tool offset modify

During the course of practical processing, if some cutter m is using cutter compensation n, and the size of workpiece processed is bigger or smaller, cutter compensation amendment function can be used to implement compensation for cutter compensation value.

(1) Pauses or single segment Effective, so that the axis motor stop;

(2) In the position screen, press the "x-to the tool offset trimming" keys or "z to the tool offset trimming" button, enter the tool offset trimming interface;

(3) Under the tool offset trimming interface, via the arrow keys, move the cursor to need trimming tool number;

(4) Input area to enter trimming of the cursor, then press 【ENTER】 key to confirm;

(5) Press any key or 【CAN】 key to exit the tool offset trimming interface.

Note: B045 can set the tool offset and tool offset of trimming method.

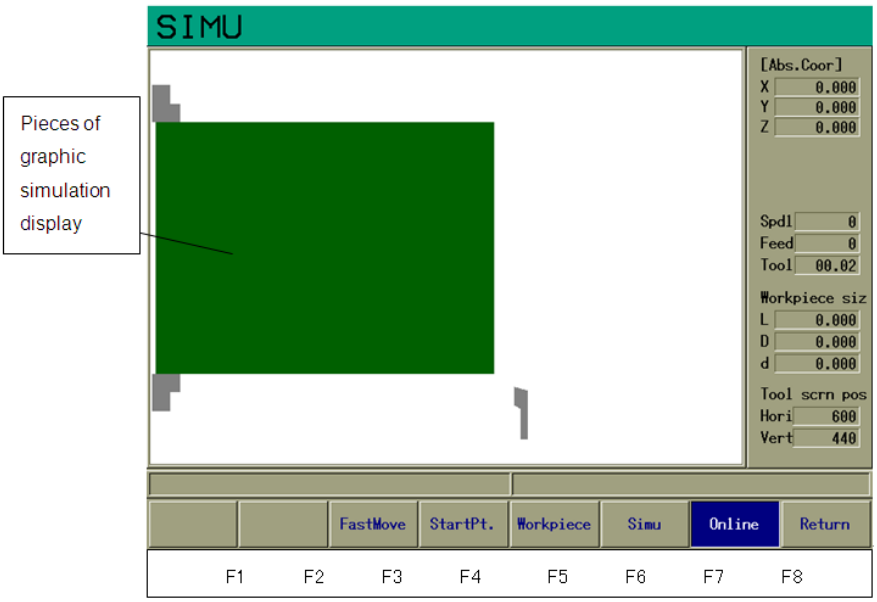
3.7 Graphics function

In the position screen, press the [F4] (Auto)key to enter the automatic mode, then press [F8] (graph) to display the graph screen(online or simulation). The ability to display the tool motion in a process under programmatic control, the track of the tool tip. This feature will allow users to directly observe the programming track running processes, combined with the coordinate values are displayed on the screen, can visually discover significant errors in the program. Graphics on the screen can be synchronized track

cutting movement of the tip, and displays the bar form factor, operators can be observed directly from the screen tip when formation process of Rotary. Simulation status, the cutter Center track is displayed on the screen, but the axis of the machine tool does not campaign and various machine tool electric apparatus control function is invalid, it is mainly for the user to the debugger, when the program is correct can be processed, avoid programming faults caused by the negligence or accident.

3.7.1 Enter into Graphic display screen

1. Press [POS] main function key to display position screen;
2. Press [F4] (Auto) key;
3. Press [save/open] key;
4. Enter a file name,for example P0750;
5. Press [ENTER] key;
6. Press [F8] (Graph) key to enter into graph screen.



F3.4 Graphic simulation

Graphical interface buttons features:

1. [F3]slow moving: changing the movement speed of the simulation tool, after the button is pressed, the button changes to quickly move. The button swiths the caption according to the state in the slow or fast moving.

2. [F4] screen orientation: simulation tool moves to the start position.

3. [F5] set workpiece: to set dimension of a workpiece. Press this key, and then move the cursor, you can set or modify the workpiece size. Enter the actual workpiece length after the l, d after the rough outer diameter, d after the inner diameter of the stock.

4. [F6] simulation: this key is pressed, when graphic simulation, the actual tool not exercising.

5. [F7] online: press this key, make the graphics simulation while, actual tool movement.

6. [F8] return: press this key to return to the position interface.

3.7.2 Graphic simulation steps

1. In the graphical display interface, press the "F5" (set workpiece) key.

2. Input workpiece length in "L", input outer diameter in "D", input the inner diameter in "d" (enter 0 when no hole)

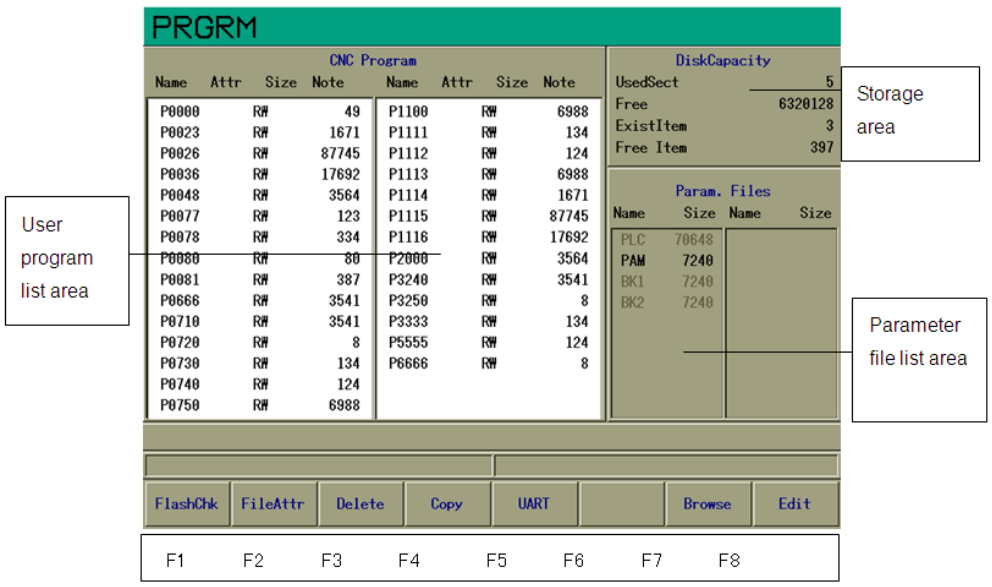
3. In graphics display area right below has a vertical bar, vertical bar of top this representative tool tip, user must move vertical bar to a suitable of location on the screen, the position where the vertical bar stopped,should be the position of current workpiece coordinates. Use "←", "→" or "↑", "↓" to move vertical bar, and press each key at a time,the distance of the movement can switch between 1 pixel/10 pixels.The move distance is selected by the "F3" key. You can move the tool to any postion on the screen by the direction keys..

4. Once seated bar,press [cycle start] button, the program will start. The tool motion indicated by a vertical bar in accordance with the user program. Machine tool motion. Depending on the drive is powered up and select simulate or online. Program is completed, according to the other main function key to exit the graphics track display. End of the simulation, you can use graphics tell if a program is correct. If there are errors, click [PRGRM] main function keys, to modify the program, and then simulation again.

4 Programme management

Press [PRGRM] primary function key to display the program interface. Program interfaces used to implement user program file operations such as create, modify, delete, and the system of electronic disk storage capacity, user files, display of parameter files and other information.

4.1 Interface introduction



F3.5 Program interface

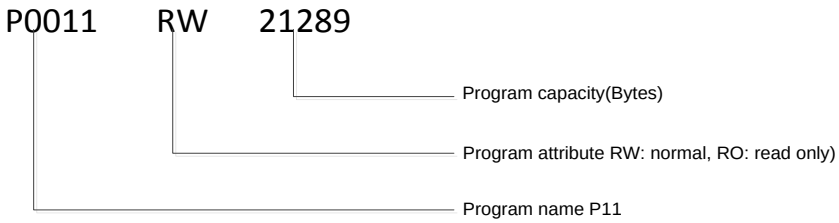
User program list area: to display the programme directory list in the user programme storage, When one page is full, [PAGE UP] and [PAGE DOWN]can be

pressed to switch program display page.

Parameter file list: to display system parameter file,including: current parameter PAM, backup-BK1, backup=BK2. bit parameter B01, cutter compensation parameter file T01, pitch compensation parameter file I01,, coordinate system offset parameter file C01. If the list is not showing the corresponding parameter files, it means the parameter file does not exit and the system can not in properly operation unless the correct parameter file is created.

Storage area: electronic disc capacity is 8M bytes, 500K system retained. Can store user programs the maximum number is 200, the maximum size of each program cannot exceed system Flash disk storage capacity.

4.2 The principle of the program name

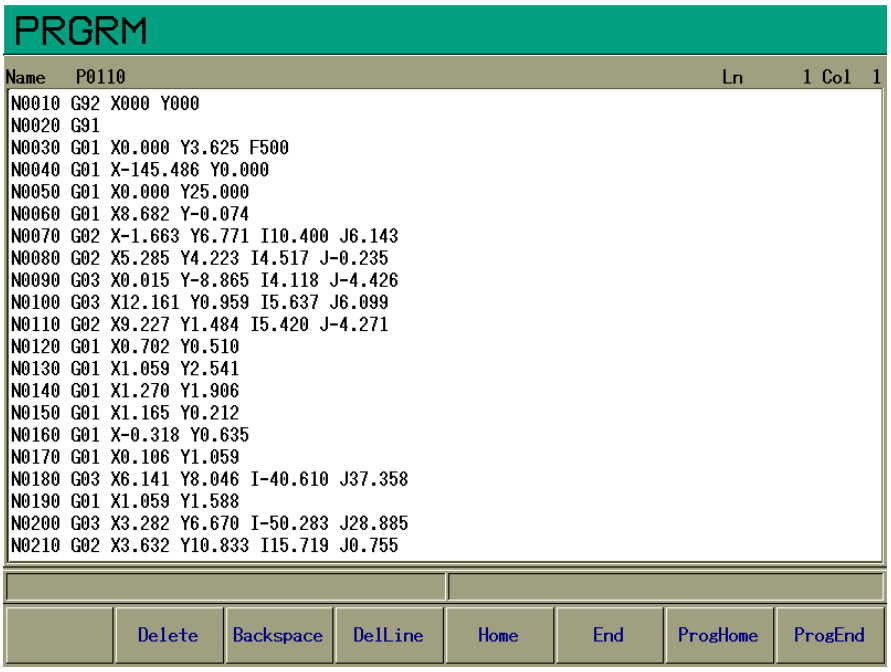


In the system, only the main program to process, master p or n is the first letter. Subroutine to n for the first letter, subroutines can only be called by the main program. Main program or subroutine, followed by two digits indicate the different programs, system requirements, between the main program can be P0000~P9999 or N0000~N9999 any one between subroutines can be N0000~N9999 any of them.

Enter the program name, the first p (or n), and then press 0~9 to enter four-digit number, press ENTER after the completed system to handle the enter the name of the program. Enter if the program name when p is less than 4 significant digits behind, leading zeros may be omitted. For example P0011 simply enter P11 three characters, after you press ENTER, the system automatically filled P0011.

4.3 Program edit

Under the program interface, press "F8" editing keys, corresponding to the F8 button being pressed on the screen, at the same time in the lower left of the screen the cursor "Please enter a file name" blinking after, the user can enter a main program P0000~P9999 or subroutine name N0000~N9999, after you press ENTER to enter the editing screen. Edit the picture in F3.6:



F3.6 Program edit

If the program already exists in the system, the program is displayed on the screen, if a new program, program on the screen without any program in the area, the middle of the screen space for the user to edit, Max can edit about the 8M of program, if the program you want to edit is read-only property, you cannot modify.

First line under the title bar displays the current program name, and the line number and column number of the cursor, edit the KeyTips feature at the end, for displays in the middle area, where users can edit program.

Number keys on the character: each character at a time, or number keys, cursor positions appear on the characters or numbers, while Fu Jun and beyond words one character after the cursor position, that is, the original cursor position into a character on (insert).

Editing function keys: f function keys and cursor keys, enter "ENTER" composition. Editing key role is to move the cursor to the appropriate location in order to add, delete a (or row) characters, they do not add a character directly in the program. Enter key role is to make the cursor reaches the beginning of the next line. If it is in the program in the middle of a row, in the line and insert a blank line between the next line, the row at the same time the original cursor position backward all the characters to an empty row, produced a new line.

For example: Edit the following two procedures:

N0010 G00 X100

N0020 G01 Z10

Key sequence：N—0—0—1—0—G—0—0—X—1—0—0—ENTER

N—0—0—2—0—G—0—1—Z—1—0—ENTER

Editing, the following editing function keys are valid:

Character numeric key: press the number keys on the character the cursor's current position on the corresponding character or number

Cursor move keys

"←": Move cursor left, cursor one character to the left from its current location, if the cursor is at the beginning of the current line, stop the left.

"→": The cursor to the right, from the current location of the cursor one character to the right, if the cursor is at the end of the current line, move the stop right.

The "↑": move the cursor, move the cursor from the current row, if the first row of the cursor is in the program, stopped up.

"↓": move cursor down, from the current row of the cursor down one line, if the cursor is at the end of program lines, stop down.

"PgeUp": text display that can be displayed at 22 lines, called a page, display one page forward from the beginning of the current page.

"PageDown": working backwards from the beginning of the current page to display a page.

F function keys:

The "F2": delete. Delete the characters on the location of the cursor, if the cursor is at the end of the next line of the content is moved to the end of the line the current line.

The "F3": deleted before. Deletes the character before the cursor position, if the cursor is at the beginning, you will move to the row on the end of the line the current line.

The "F4": delete rows. Delete the line where the cursor, while one line below the deleted row content.

"F5": the beginning. The cursor jumps to the beginning of the current line.

"F6": end of the line. The cursor jumps to the end of the line the current line.

"F7" program first. Displays the first page of the program content, and the top row of the cursor jump to the program beginning.

The "F8": end of the program. Show the program content on the last page, and the cursor jumps to the end of the program line line endings.

Note: no abnormal exit the Edit State, otherwise some or even all of the programs in the system will be destroyed, which a normal exit include:

(1) Press the hardware reset, shutdown, or when editing power system instantaneous power.

(2) If editor a great deal of the program, the system may have to wait a while to go to the Edit screen, occurs while waiting for the above acts can damage part or all of the program.

Above conditions occur, the system cannot guarantee the completeness of the program.

If you want to exit the Edit State, simply press a main function key, the system will switch to the other main function, before the exit, the system automatically saves the users after editing procedures for processing, and recorded in the program files directory, reflected in the program name in the table.

4.4 Copy, delete, and modify properties

Under the program interface, these three functions, mainly for the overall status of the program changes.

4.4.1 Copy files

This feature is to copy a program to another program.

In program interface, by "F4" key, screen below and F4 corresponds "copy" button was by Xia, while in screen left below "Please entered file name" in the appears cursor, in cursor Department type will was copy of program name, as P0067 (P0067 must is has in system in the of program), by enter Hou system prompted: "will program copy for", cursor in subsequently Flash, then entered copy of target program name, as P0068 (P0068 must is is not system in the of program), by "ENTER" Hou copy completed, new of program produced, while program name table in the also occurs corresponding of change.

4.4.2 Delete files

This is a useless programs removed from the system, can be the main program, or a subroutine, as long as it appears in the program name in the table. Addition of this feature, you can set the system program files delete all.

In the program interface, press the "F3" key that corresponds to the bottom of the screen and the F3 button "delete" is pressed, enter need to delete the name of the program (all deleted, enter "P."), press "ENTER" key and the program is deleted.

Note: once the program is deleted, will not be able to recover deleted, be sure to carefully operation.

4.4.3 Modify the properties

Every program of numerical control system can have two kinds of properties, namely: read and write b) read-only

For read-write property, carry out general editing, delete. For a read-only property, the program will only display on the screen available to watch, but not in which to increase or decrease the character. This feature prevents misuse destruction program.

Under the program interface, press "F2", corresponding to the F2 button at the bottom of the screen "Properties" is pressed, and enter the program name in the cursor. Press "ENTER" key, system prompts: "Please select a new property 0: reading and writing, 1: read-only" indicates that the program will be assigned properties. Enter 0, one of the 12 numbers, then the program properties will be reset, while in the program name in the table has changed meaning of these two figures: 0: read-write property 1: read-only property.

Under the program interface, press "F2", corresponding to the F2 button at the bottom of the screen "Properties" is pressed, and enter the program name in the cursor. Press "ENTER" key, system prompts: "Please select a new property 0: reading and writing, 1: read-only" indicates that the program will be assigned properties. Enter 0, one of the 12 numbers, then the program properties will be reset, while in the program name in the table has changed meaning of these two figures: 0: read-write property 1: read-only property.

Note: after a new program to edit, its property defaults to 0 (RW read-write properties).

4.5 program Browse

Browse feature to quickly browse CNC user program content in the store, which you can open files of any size, for viewing but cannot be modified.

In the program interface, press the "F7" and enter the file name, the system opens

the file, if carriage returns, the system opens the first file in the file list. Browsing interface shown in Figure 3.2.5, at this point, the function key F1~F8 be redefined.

"F1": Edit. To edit the currently displayed content of the program.

"F2": find. Find the string entered by the user, and moves the cursor to the first and the same location as the string.

"F3": find next. Can view and search function in the order in the input string of the same content.

"F4": program first. Move cursor to beginning of file.

"F5": end of the program. Move the cursor to the end of the file.

"F6": the previous program. Press "F6" display CNC user program area of the next program name and content. If the current program to the user program area of the final program, press this key to display the first program.

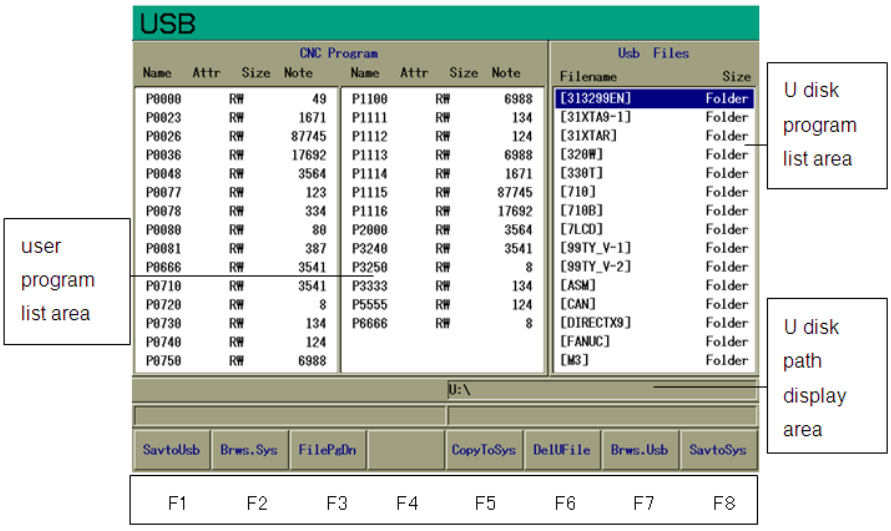
"F7": the next under. Press "F7" display CNC user program area of the previous program name and content. If the current program to the user program area of the first program, press "F7" post, showing the final program content and name.

"F8": return. Return to the program's main interface.5 the U disk interface and function

According to the Panel on the "u" primary key into the USB interface. System provides a USB interface, support u disk access, USB interface, open the USB drive protection cover, insert the USB drive, the interface shown in F3.7. Realization of USB interface the main function can u disk files of user program memory and file storage.

5 U disk interface

5.1 U disk interface description



F3.7 USB interface

U disk program list area: U disk shows a list of files under the current directory.

The user program list area: displays a list of program files in the user program memory.

U disk path display area: Displays the current file path U disk, the maximum depth is

6.

F U disk function key interface description:

"F1": deposited in USB drive. Program files into the USB disk in the user program list the current directory.

"F2": browsing program. Browse programs in the user program list.

"F3": under the file page. User program list screen displays up to 30 user programs, when more than one screen when the program, you can loop page feature displays programs that are not currently shown in the user program list.

"F5": copy to the system. Select USB file copying to another program to the user program memory.

"F6": delete the u disk file. Delete the selected USB drive files.

"F7": Browse u file. Browse to select the USB drive files.

"F8": deposit system. Flash drive will be selected files into the user program memory.

U disk file list display operation key description:

"PageUp": the current USB drive file directory listing displays a page

"PageDown": backward through the current USB drive file directory listing displays a page

The "↑": move the cursor, the cursor move one line on the u disk file list area, and when the cursor moves to the u-top of the file list area, continue to press the "↑" button, the contents of the current file list as a whole down one line, while previous files in the directory does not appear in the list displayed at the top of the list. If the current cursor position is the USB disk file list on the first file or directory, move the stop up.

"↓": Move cursor down, u list area of the cursor down one line, when the cursor moves to the USB disk file list area at the bottom, continue to press the "↓" button, the contents of the current file list as a whole, one line, and will not display the next file in the directory list displayed at the end of the list. If the current cursor position is u disk files already on the list of last files or directories, stop down.

5.2 U Disk Management description

1. Supports USB1.1/USB2.0 protocols, USB memory, USB disk format file system formatted as FAT.

2. U can manage the maximum depth of six disk directory.

3. U disk file name display format to the 8.3 format, namely xxxxxxxx.xxx format, longer than the 8.3 format file names to be reduced to 8.3 format, support the Chinese directory name.

4. Support each other documents between the user program memory storage.

5. U disk file name automatically sorted.

5.3 U disk file selected

Because the USB disk in the file name may be a Chinese name or long file names, this brought inconvenience to USB input for file names, for ease of user operations on the USB disk file, the system cursor Select method to select the file you want to. System requirements: the u disk files before any action necessary to first select the file. So-called

selected, that is, move u the cursor in the file list, place them in the operations on the file name, means that the file is selected. Folder select the same. As shown in Figure 5.1, M20 is currently selected file.

5.4 U disk folder opens

System supports USB directory depth of level 6, to no more than folder under the directory depth can be opened. Users can sort processing to the appropriate folder, makes it easier to find and manage the program. To open a folder, first select the folder, and then press "ENTER" key, open the folder that, at the same time in the u disk file list displays the contents of the folder.

Assuming the u disk under the root directory to [processing] folder, to open the folder, for example shows how to open the folder actions:

Move the cursor of U disk, select [program] folder;

Press "ENTER" key, the contents of a folder that is in USB file displayed in the list.

5.5 Back to parent directory

By doing the following to return u disk directory for the current file:

1. Select the [Return];

2. Press the "ENTER" key to return to the parent directory, if it returns success, u disk path display shows the new path

3. If the current directory is the USB flash drive root directory, you cannot back up again.

5.6 U disk file Copied, or entered into the system

These two features for Flash drive file into the user program memory. By following examples to illustrate the specific operational processes.

5.6.1 U disk files Copied to the system

If P01 file already exists under the root directory of U disk, to store them in the user program memory.

In the U disk interface, select the P01 file;

Press "F5" (copied to the system) key, that is, start u disk files into the system;

If the operation is successful, P0001 file will appear in the user program list.

5.6.2 U disk file entered into the system

If test.nc file already exists under the root directory of U disk, to store them in the user program memory, and in user storage to P0003 named.

Under the USB interface, select the test.NC file;

Press "F8" (into the system) key, displays the prompt bar "to copy the program to" input box, enter the P3 (leading zeros when you enter can be omitted) then press "ENTER" key, begins to u disk files into the system;

If the operation is successful, P0003 file will appear in the user program list.

Note:

1. The name of the file into the system must comply with the user program naming,

that is, begin with a p or n, followed by the 1~4 digits.

2. Into system u cannot exceed system user procedures under maximum file size, about 8M, lathe system does not have a DNC functions, file is too large it cannot be processed.

3. The folder can not be entered into the system.

5.7 Program files stored in the user program memory

USB flash drive

Actions for example, assume the user already exists in the catalog list P0003 procedure, are paid into a USB root directory, in the u named P01.

U disk interface, press【F1】(into U disk) key, "Please enter the file name" input box, and a blinking cursor in the input box, enter the P3 and press【ENTER】 key, the prompt Column shows "to copy the program as " input box, and a blinking cursor in the input box, type P01 and press 【ENTER】 key, the user program memory began in the root directory of P03 into the operation of U;

If the operation was successful, after reordering, the P01 will appear in the USB disk in the file list.

5.8 U disk file browser

U disk interface in the browser can view the contents of the file U disk, making it easier for users to store files on the U disk or delete files before the operation to determine whether correct. Specific operational procedures are as follows:

Under the USB interface, select the files that need to browse;

Press "F7" (u file browsing) key, that is, switch to the USB disk file browsing interface;

In file browsing interface you can make the following buttons:

"F6" (program), viewing a file's Home.

"F7" (program), view the last file.

The "F8" (return), returns to the USB main interface.

"PageUp": the current u-file displays a page.

"PageDown": backward through the current u-file displays a page.

The " ↑ ": move the cursor, the cursor move one line on the u-file, and when the cursor moves to the u-top of the file list area, move the stop up.

"↓": Move cursor down, u list area of the cursor down one line, when the cursor moves to the USB disk file list area at the bottom, stop down.

To exit u disk file browsing, you can press one of the main function keys, exit the browsing interface, switch to the corresponding primary function interface;

To browse the next USB disk file, first exit u disk file browsing interface (see procedure 3 operation) returned to the u disk management interface, USB interface to browse, select the file, repeat process 2 to.

5.9 U disk file deletion

When the USB disk free space is low, you can by removing the USB disk file function

to remove unused USB disk file. Operational processes are as follows:

Under the USB interface, enter the directory where the file you want to delete (refer to 5.5), select the USB drive you want to delete files;

Press the "F6" (remove u file) key, you can remove the file;

If the removal was successful, is deleted disappears from the directory where the file name, directory list of file names to reorder the display, and in the status bar "deleted successfully" message appears.

5.10 Browse program file in the user program memory

This feature makes it easy for users to browse view user program memory programs before USB storage file is correct. P0003 programs to browse the list of users in, for example, the operations are as follows:

1. USB interface, press "F2" (Viewer), the pop-up "Please enter a file name" input box, and your cursor blinks behind;

2. Enter the file name to browse if P0003, press "ENTER" key, that switch to user program memory file browsing interface, and P0003 is displayed the contents of the file.

5.11 Cycle through the list of users

When a user exceeds the number of programs in the program list screen (30) when the scope is displayed, to view the current program name of the user is not listed, you can view this feature. The operations are as follows:

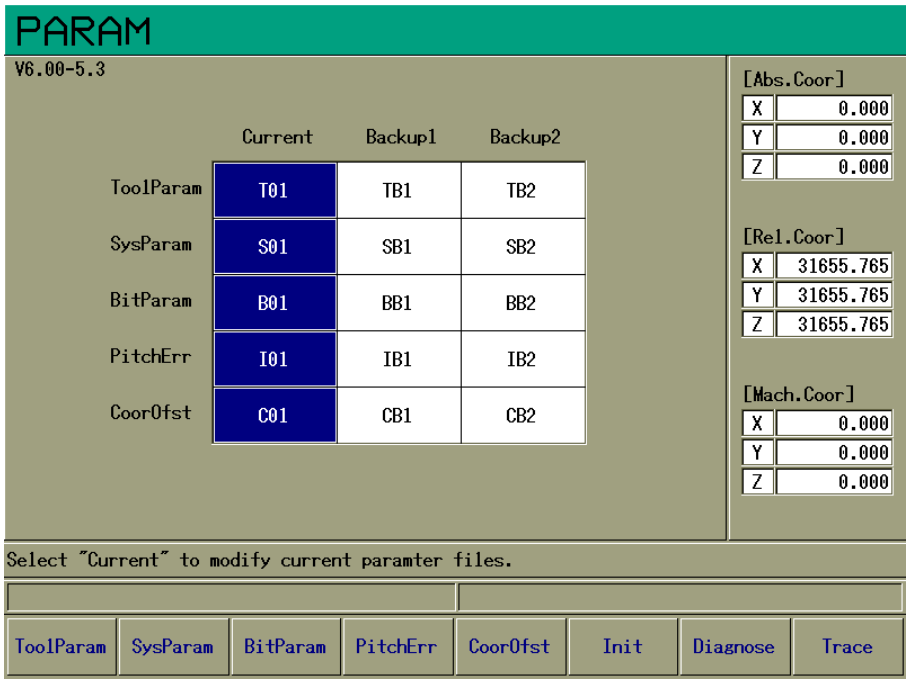
USB interface, press the "F3" (files next) keys in the program list box to display the next page in the name of the program, if it turned to the list last, again by the "F3" (files page) key to display the first page list content;

Under the USB interface, file page down features are only valid for the user program list.

6 Parameter managment

Press the [PARAM] primary function key to enter the parameter interface, as shown in Figure 3.8. Parameter interface of the main features include: tool parameter, system parameter, bit parameter, pitch compensation, initialization settings, diagnose and coordinate system and so on. These parameters are essential to the normal operation of machine tools and machining quality.

6.1 Introduce to the parameter screen



F3.8 Parameter interface

"Current parameter" column: It's the parameter that is being used by numerical control system.

"Backup No.1" column: It's the storage region for parameter file backup of user.

"Backup No.2" column: It's the storage region for parameter file backup of user (see Section 4.1.3 for difference between current parameter and backup parameter).

The parameter storage class cursor is used to select the parameter file storage type which will be viewed or modified, for instance, select "Current parameter" in the Figure, and if press F1 (cutter parameter), F2 (system parameter), F3 (bit parameter), F4 (pitch compensation) or F6 (coordinate system), the data of current parameter storage area will be displayed in the parameter data. The user can use cursor key ← and → to move parameter storage class cursor to select the storage class of parameter file which will be opened. The information of file at the position of parameter storage class cursor will be displayed in the hint column.

Version number: as Figure V10.00-2.2, V10.00 is the software version number and 2.2 is hardware version number.

6.1.1 Parameter system

The parameter system primarily includes the following parameter files:

1. Tool parameter: tool compensation of 24 cutting tools.

2. System parameter: Variables for use of 200 systems, including P000～P199

3. Bit parameter: Forty 8-bite parameters, 320 statuses which can be modified in total.

4. Thread pitch compensation: 160 point/axis, four axes of X-axis, Y-axis, Z-axis and Z-axis;

5. Coordinate system: contains two sets of functions: the first group, set the G54-G59 the corresponding coordinate values, the second group, coordinates the bias, when

system coordinates and tool does not match the actual location is through the functional adjustment system coordinates, bring it line with the actual location tool.

6.1.2 Initializing operation

Some operations related with parameter files are also included under the parameter management interface, and all of them are contained in the initializing interface; in this interface, the following operations can be set such as memory clearance, format, recover default value of parameter file, backup and recovery of parameter file, password set and time set and so on (see Section 6.7 for detailed information).

6.1.3 Difference between current parameter and backup parameter

According to difference of storage mode for various parameter files in the electronic disk, the system will divided the parameter file into three classes: current parameter, backup No. 1 and backup No. 2, the main differences among them are as follows:

1. The current parameter is the one that is being used by numerical control system, and its filename is displayed in the parameter display region under program management interface. It's stored in the user program storage area, and format of electronic disk will delete current parameter file; however, format won't impact backup No.1 and backup No.2 of parameter file.

2. The user can browse and edit current parameter; however, the backup parameter can only be browsed.

Before the user browses or edit the parameter file, it should be confirmed that "parameter storage class" will select the parameter file in current parameter or backup No. 1 & backup No.2; the backup file can't be edited, and the status bar will display that "this file is backup one and can't be edited" when the backup file is edited. The backup file is used to backup current parameter file, if it's necessary to edit or utilize some backup file, it's required being recovered to current file (see Section 4.6.4.2 for detailed operation of recovering backup file), then it can be edited or used.

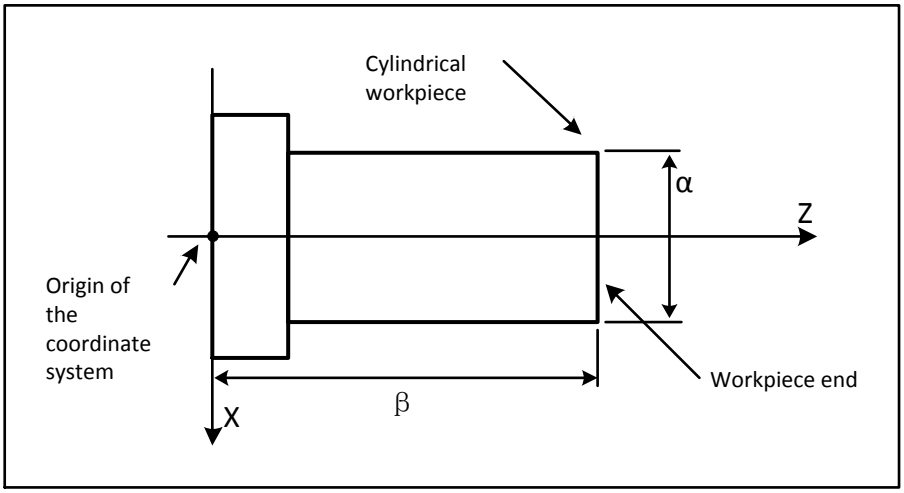
6.2 Tool parameter

Content of the tool parameters,is a tool offset table, the tool interface is to display and set the tool offset value.

6.2.1 Enter tool parameter screen

1. Press the [PARAM] main function key, enter the parameter interface.
2. Press the [F1] (tool parameter), enter the tool parameter interface.

6.2.2 Establishment of tool offset value



F3.9

1. Confirm that the workpiece coordinate system has been established;

2. Manual mode, start the spindle, try on the move tool in cylindrical workpiece cut a small section, click "Xsav" key, the bottom right of the screen displays "x to the knife has been saved";

3. Exit the tool, stop the spindle, measuring the diameter of the workpiece where test cutting, resulting in measurement value α;

4. Enter the tool parameter interface, with page key [PAGE UP], and [PAGE DOWN] and cursor key "←", and "↑", and "→", and "↓" move the cursor to the x column of the tool parameter.Press "F1" (Measure), input the measured valueα, Press "ENTER" key confirmed entered, then press [SAVE] key to save parameter;

5. Start the spindle, move tool to the surface on the workpiece end, try to cut a small section, press [Zsav] key, the bottom right of the screen displays "z to the knife has been saved";

6. Exit tool, stop the spindle, measurement of workpiece end interview-cut to the length of the Chuck end, measured values are β;

7. Enter the tool parameter interface, move the cursor to where you want to set the tool offset of the z column, press "F1" (tool), enter the measurements β, press "ENTER" key to confirm, then press "Save" to save parameter.

6.2.3 Tool offset value trimming

During the course of practical processing, if some cutter m is using cutter compensation n, and the size of workpiece processed is bigger or smaller, cutter compensation amendment function can be used to implement compensation for cutter compensation value.

1. Enter the tool parameter interface;

2. The value to move the cursor need trimming tool offset number, and then move the cursor to the appropriate on the axis;

3. Press [F2] (modify), enter the tone pitch, press "ENTER" key, then press "Save" to save parameter.

The parameters associated with the tool offset trimming:

1. B040 bit parameter (parameter No. 0, No. 04): when the B040=1, tool offset

trimming can be made in the process;

2. Parameter B157 (15th parameter 7th): when the B157=1, tool offset trimming, the original tool offset value added input values are reset to the new tool offset value.

Method to Confirme the trimming value as follows:

when the B157=1, such as processing large diameter of the workpiece, you enter a negative value when x to trimming, diameter is small then enter a positive value; such as machining of workpiece length too long, then z to enter negative values when trimming, length too short enter positive values.

When the B157=0, such as processing large diameter of the workpiece, enter positive values on x to trimming, diameter is small enter negative values; such as machining of workpiece length too long, then z to enter positive values when trimming, length too short enter negative values.

6.2.4 Directly input cutter compensation value

If it's necessary to directly input the cutter compensation value of some number cutter, the cutter compensation input function can be utilized. There are two methods to input cutter compensation value:

Action steps are as follows:

1. Enter the tool parameter interface;

2. Press the [F3] (input), enter the tool offset value, press [ENTER] key, then press [Save] to save parameter.

6.3 System parameter

Contains a numerical control system of many variables in the system parameters, for the user to set, 200 parameters in the system, including: G00 positioning speed, acceleration and deceleration time constant, ratio, pitch and backlash compensation, electronic gear, this parameter is important for normal operation of the CNC system users before making machine, be sure to know how these system parameters settings meet your needs. To prevent misoperation, before changes are required to enter a password.

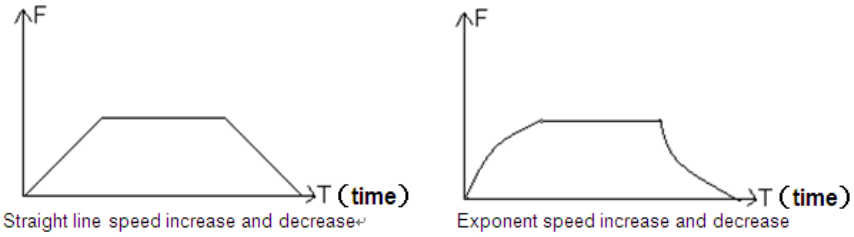
In this manual ,the index of the system parameter indicated by a letter P and a number, for example, P011 mease the 11th system parameters.

6.3.1 Basic conception

6.3.1.1 Time constant of speed increase and decrease

It's the time that the operating speed consumes from zero to maximum set one, or from maximum set speed to stop towards to the coordinate axis of machine tool.

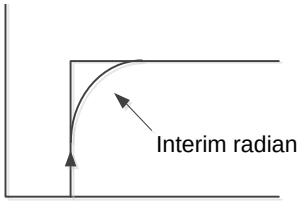
When the system positions thread cutting rapidly at G00 and performs manual feed, it'll adopt straight line speed increase and decrease, when machining is performed and hand-operated pulse generator is adopted, the exponent speed increase and decrease will be adopted.



Straight line speed increase and decrease Exponent speed increase and decrease

F3.10

The interim radian (as Figure) will occur between the cutting block by adopting the exponential type speed increase and decrease, this radian is related with the feed speed and exponent speed increase and decrease time constant (39#), it's favorable to reduce this radian through reducing 39#, if the interim radian isn't generated between two sections of traces, the fast angle clear command G61 and G62 can be adopted (refer to G61 and G62 description for details).



F3.11

6.3.1.2 Acceleration

Numerical control system adopts sampling control method and implements sampling control for one time to various coordinate axes. The speed increase and decrease of various axes adopt control method of even acceleration in every other 4.096 millisecond, namely the change of motor speed (if it operates) of each sampling interval (4.096ms) is constant value, which is the speed increase and decrease of the system. Its calculating formula is：

$$a = \frac{V_m \times Ts^2}{60 \times Tm}$$

Among it:

Ts: System sampling time (4.096 ms)

Tm: Time constant of speed increase and decrease (ms)

Vm: Maximal speed (mm/min)

a: Acceleration

The acceleration can be changed through changing maximum speed or changing speed increase and decrease.

Generally, the maximum speed is used to limit the one in this movement status, and the acceleration value can be determined with time constant of speed increase and decrease, while the actual operating speed of system should be less than or equal to the maximum speed.

The relationship among acceleration a, maximum speed Vm and time constant Tm: "—" signifies constant, while h and Vm change at the same time, it's specifically determined by the formula:

Vm	↑	↓	—	—
Tm	—	—	↓	↑
a	↑	↓	↑	↓

F3.12

6.3.1.3 Electronic gear ratio

Regulating system output which makes the coordinate operating value of system concord with the distance of table movement, and it's not necessary to regulate transmitting ratio of motor and screw for this.

The electronic gear is set by system parameters. Two parameters of each axis can be called multiply factor MLT and percentage DVT, and when the bearing ball screw transmitting working table operates:

$$\frac{MTL}{DVT} = \frac{Pmt \times Gf}{Pcn \times Gd}$$

Among it:

Pmt: Pulse number of each rotation for motor, and the stepping motor: Pulse of each rotation

Servo motor: Encoder line number X encoder multiple frequency number

Pcn：Screw thread pitch μm

Gf: Total of driven wheel teeth number in the transmitting between motor and screw connection

Gd: Total of driving wheel teeth number in the transmitting between motor and screw connection

Towards to direct connection，Gf＝Gd＝1

MLT and DVT must take the smallest positive integral value.

The scope of MLT and DVT is 0—65535, but the ratio must within 0.01~100.

6.3.2 The system parameters display and set

1. Press [PARAM] key to enter parameter screen;

2. Press [F2] (system parameter) to enter system parameter screen;

3. After [PASSWD](input password) is pressed, the right side of hint column will display the password input box; input correct password and press [ENTER]; if the input is wrong during the course of inputting password, the cursor key [←] can be pressed to delete the data input.

4. Utilize the page turn key [PGUP] & [PGDOWN] as well as cursor key [←], [↑], [→] and [↓]to move the cursor to the parameter to be set;

5. It can be input again through pressing numerical key, during the input process, if there is wrong, the cursor key [←] can be pressed to delete the data input, after input is completed, press "ENTER" key;

6. Press [Open/Save] key to save system parameter in the electronic disk, If other interface is switched directly without pressing Save key, system will save the modified parameter automatically.

Note: Refer to Appendix 2 for system parameters and range (System Parameters)

6.4 digit parameter

The digit parameter can be used for setting towards the many ones which have only two selective conditions and branches. The system has 40 digit parameters in total, each parameter has 8-bite, each digit only has two status of 0 and 1, parameter number is B001～B398. which can be as switch selection of certain status, 320 status can be determined. This part of parameter is significant towards normal operation of numerical control system, for instance: whether it's radius programming or diameter programming, whether the electronic gear ratio is opened as well as motor positive and negative rotation will be set in this parameter file. Before the user implements machine tool processing, please make sure whether the setting of these digit parameters meet the requirements. In order to prevent wrong operation, it's necessary to input password before modification.

The this manual,the bit parameters indicated by letter b and the number of bits, such as the B011 is No. 01-parameter 1th bit.

6.4.1 View and setting of digit parameter

1. Press the primary function key [PARAM] and press [F3] (digit parameter) to enter bit parameter interface;

2. Utilize the cursor key [←], [↑], [→] and [↓]as well as [F1] (digit left movement), [F2] (digit right movement) to move the cursor, select certain digit of the digit parameter to be set;

3. After [PASSWD](input password) is pressed, the right hint column will display password input box; input correct password and press [ENTER];

4. Press the numerical key [0] or [1] to modify the status of parameter digit selected;

5. After it's modified, then press [Open/Save] key to save the digit parameter after being modified to the electronic disk.

Note: Refer to Appendix 3 for parameter contents (Digit parameter)

6.5 Thread pitch compensation

The data of thread pitch compensation is used to compensate the error which is caused owing to difference of thread pitch, its content will be different with different machine tool, and final user don't require modifying it in principle. However, during the course of utilization, the screw rod thread pitch of certain axis will be changed owing to mechanical abrasion, at this time, it's necessary to modify the thread pitch compensation of this axis, which is used to reduce the error. Each axis of the system can input 240 dual-direction error compensation points. The thread pitch offset parameter has bigger influence towards processing quality, in order to prevent wrong operation, it's necessary to input correct password before the thread pitch compensation is set.

6.5.1 View and setting procedure of thread pitch compensation

1. Press [PARAM] and press [F3](thread pitch compensation) to enter thread pitch compensation interface;

2. After [PASSWD](input password) is pressed, the hint column will display "Please input password" and password input box, please input correct password in the password input box, then press [ENTER];

3. Press [F1] (X-axis), [F2] (Y-axis), [F3] (Z-axis) and [F4] (A-axis) to select the movement axis to be set;

4. Utilize the page turn [PGUP] & [PGDOWN] and cursor key [←], [↑], [→] and [↓] to move cursor to view or select the thread pitch offset value required being set.

5. Press the numerical key to directly input thread pitch offset value, then press [ENTER]; press [Open/Save] key to save data after it's set.

6.5.2 Pitch error compensation description

1. Set parameters such as the amount of compensation and compensation origin,compensation direction, compensation interval;

2. After returning to the mechanical zero point, make this reference point as the origin of compensation, compensation on the interval set in the axis, and compensation value as a parameter should be set;

3. Enter a value for error logging, which shall be entered in order to offset the error compensation values, the range of compensation for each point is-6.000~6.000, beyond the scope of compensation point system considered pitch error is 0;

4. Compensation axis: up to 6 axis;

5. Compensation points,: the axis bidirectional 240 points;

6. According to the axial screw length, precision, workpiece dimensions determine the axis screw pitch error compensation compensation point and interval length, between the two points considered change of pitch error is linear.

6.5.3 Pitch error compensation setting parameters

1. Parameter B060~B065 determines whether the axis of pitch error compensation;

2. Parameter P128~P133 set the axial pitch error number of the reference point;

3. Parameters pitch error compensation P110~P115 set the axis interval length (mm);

4. Parameter P116~P121 is set to the point of maximum pitch error compensation, the axis;

5. Parameter P122~P127 is set, the axis minimum negative pitch error compensation points;

6. Parameter P134~P139 set the axial pitch error compensation rate.

6.5.4 Examples of pitch error compensation

Example: x-screw effective length of 300mm, from zero to negative direction of machine tool compensation, when compensation interval is 3mm, the total compensation of 100 points.

You can set the x axis pitch error of reference point P128=0, point, the x axis is the minimum pitch error compensation point, to the point of maximum pitch error compensation, P122=0,X shaft P116=100, interval length P110=3.

With laser interference measured pitch errors: (first back zero points, makes machine tool coordinates for zero)

Along the-X to go to-3mm (systems), measured to the-2.974,1# forward pitch error-0.006；

Along the-X to go to-6mm (systems), measured to the-6.003,2# forward pitch error for +0.003；

Along the-X to go to-9mm (systems), measured to the-9.007,3# forward pitch error for

+0.007；

Along the-X to go to-12mm (systems), measured to the-11.990,4# forward pitch error-0.010；

.

.

.

Along the-X to go to-291mm (systems), measured to the-291.011,97# forward pitch error for +0.011；

Along the-X to go to-294mm (systems), measured to the-294.000,98# forward pitch error is 0；

Along the-X to go to-297mm (systems), measured to the-296.999,99# forward pitch error of-0.001；

Along the-X to go to-300mm (systems), measured to the-300.007,100# forward pitch error for +0.007；

When a forward pitch compensation compensation is done, and then by laser interferometry measuring reverse pitch error compensation: (x-coordinates machine tool for-300mm)；

Along the +X to go to-297mm (systems), measured to the-296.997,99# reverse pitch error for +0.003；

Along the +X to go to-294mm (systems), measured to the-294.002,98# reverse pitch error-0.002；

Along the +X to go to-291mm (systems), measured to the-291.000,97# reverse pitch error is 0；

Along the +X to go to-288mm (systems), measured to the-288.005,96# reverse pitch error-0.005；

.

.

.

Along the +X to go to-9mm (systems), measured to the-8.998,3# reverse pitch error-0.002；

Along the +X to go to-6mm (systems), measured to the-6.000,2# reverse pitch error is 0；

Along the +X to go to-3mm (systems), measured to the-3.002,1# reverse pitch error for +0.002；

Along the +X to go to 0mm (systems), measured to the-0.007,0# reverse pitch error for +0.007；

Final filed, automatic pitch compensation in NC system processing.

6.5.5 Import pitch error compensation through u disk

Using tools such as Laser Interferometer pitch error of the measured data can be output to the computer, and then follow the format of the system requirements to use USB-imported into the system. Instruments are integer error data are mostly in Micron units, in accordance with principles of convenience, requirements are as follows:

Pitch error the data file using Windows Notepad, each axis of pitch error of data

different from the file, respectively:

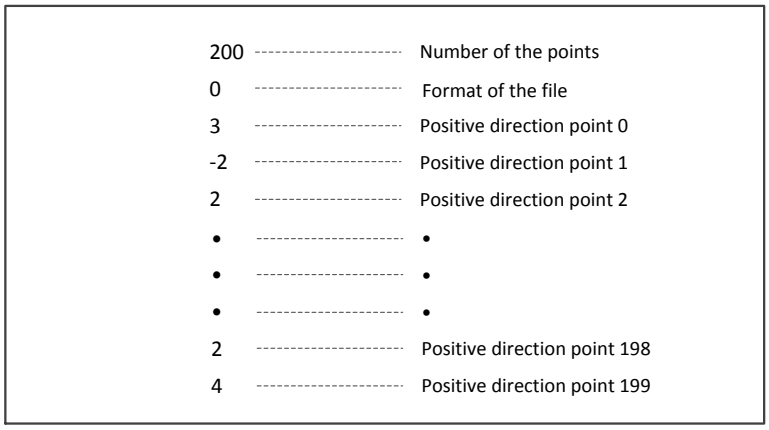
1. X axis is "I01X. TXT";
2. The y axis is "I01Y. TXT";
3. Z axis is "I01Z. TXT";
4. A axis to "I01A. TXT";
5. B axis is "I01B. TXT";
6. C axis is "I01C. TXT".

Pitch error the data file format is as follows:

1. Fill in the first row with the number of pitch compensation point (N), The range of the value N is 1 to 240, outside the range of values will result in data import failed;

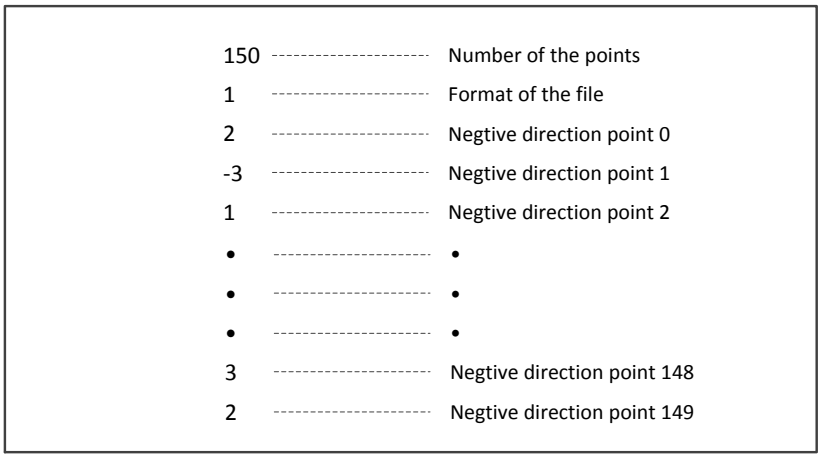
2. Second line fill in pitch compensation file format (F, the range of format F is 1 to 3, specific using method following:

When F=0, import are to pitch compensation, file 3rd line to N+2 line of data will was import to are to pitch compensation 0 to N-1 in the, negative to pitch compensation value was set for 0, file in the is greater than N+2 line of data was ignored, insufficient N+2 of data as 0 processing;



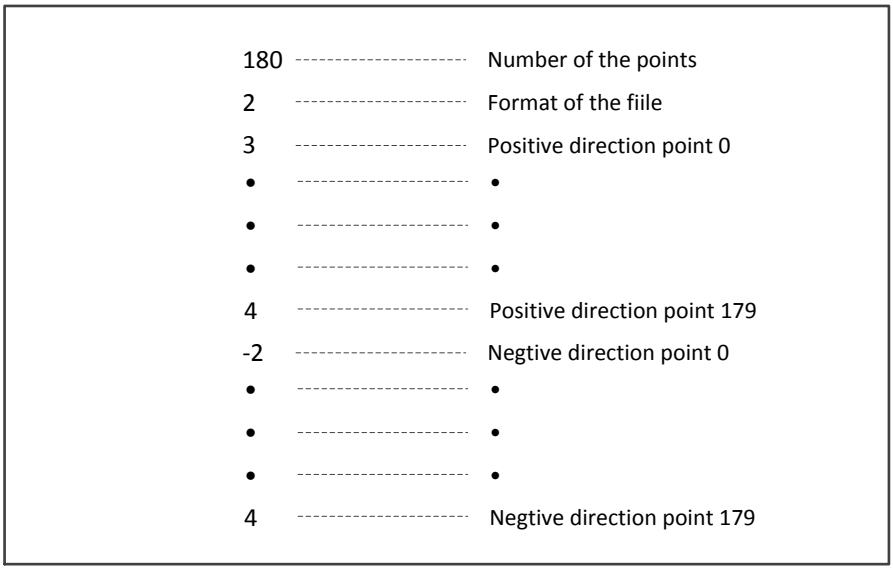
F3.13

When F=1, importing negative pitch compensation, 3rd line to the N+2 line of the file data will be imported to a negative pitch compensation from 0 to N-1, forward pitch compensation value is set to 0, the data is greater than the N+2 line in the file are ignored, under N+2 of data processed as 0;



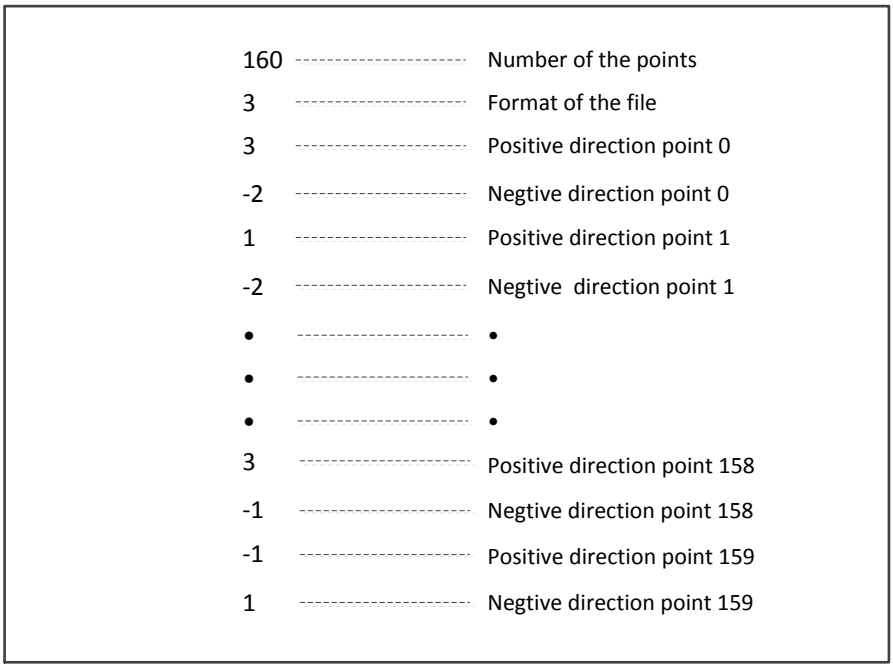
F3.14

When F=2, import two-way pitch compensation, data file N+2 of the 3rd line to line as a forward pitch compensation value, N+3 to the 2N+2 line of data as a negative pitch error compensation values is greater than the 2N+2 rows of data were ignored, under 2N+2 of data processed as 0;



F3.15

When F=3, import two-way pitch compensation, in the data file 2N+2 of the 3rd line to line, odd-numbered rows of data as positive pitch compensation value, an even number of rows of data as a negative pitch error compensation values is greater than the 2N+2 row data is ignored, under 2N+2 of data processed as 0;



F3.16

(3) The data from 3th line to n line is a pitch error compensation values, which is as a decimal integer (-6000~6000), when n is too large, the import operation, the excess data will be ignored by the system, n hours, lack of data processed as 0;

Process steps:

1. Handling of data files on the computer, named by naming rules, and then copy to the root directory in a USB disk;

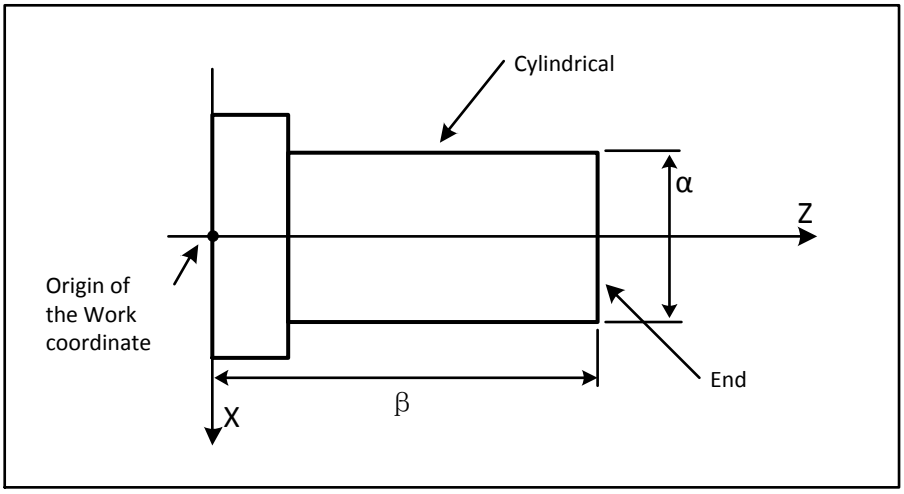
2. Under the pitch compensation interface, press F1~F6 to switch the interface to the interface you want to import, such as the x axis;

3. Press [F7] (import) button.

6.6 Coordinate offset

Bias in coordinate interface, you can G54~G59 the workpiece zero point, zero point of the coordinate system display and setting the bias as well as the entry point. If the installed machine tool zero, then G54~G59 the origin of the coordinate system in the workpiece under the machine coordinate system is fixed when the bit parameter B034 and B035 reset to 0, set before G54~G59 workpiece coordinate system back to machine zero (if power on the system back to zero after you do not need), and then use the procedure section 6.6.1 workpiece coordinate system. So in case of lost or coordinates when you drift, back machine zero operation, the system will automatically restore G54~G59 of workpiece coordinate system. If not installed machine tool zero-B034 and B035 reset to 1, G54~G59 feature can also be used at this time, but in case of lost or coordinates when you drift, need to knife way to restore the workpiece coordinate system.

6.6.1 Method of determining the offset of the workpiece coordinate origin and steps



F3.17

1. In the position screen, switch to manual mode, open the spindle, move cutting the workpiece outer circle;

2. Ensure that the case of the x axis without moving, the z axis is moved out, stop the spindle;

3. Measure the outer cutting diameter, resulting in measurement value α;

4. Press [COORSYS] primary keys directly into a coordinate offset interface or pressing the [PARAM] primary key after entering the parameter interface, then Press [F4] (Coorsys), enter the coordinates offset interface;

5. Press the cursor keys to select the required setting of workpiece coordinate

system;

6. Press the cursor keys to select the corresponding workpiece coordinate of x axis, press [F1] (measure), enter the measurement value α; press the [ENTER];

7. In the position screen, open the spindle, move the cutting workpiece surface;

8. Ensure that the z axis without moving case, the x axis is moved out, stop the spindle;

9. Measure end-cutting to want to set the length of the workpiece coordinate system origin of beta;

10. Press [COORSYS] primary keys directly into a coordinate offset interface or pressing the "parameter" primary key after entering the parameter interface, Press [F4] (Coorsys), enter the coordinates offset interface;

11. Press the cursor keys to select the required setting of workpiece coordinate system

12. Press the cursor keys, select the work-piece coordinate system the z axis, press "F1" (measure), enter the measurements beta; press the "ENTER";

13. Repeat steps 1~12, until all the work-piece coordinate system, and then "file/program" save the coordinate system.

14. If the coordinate system is set to the current coordinate system, the 1~13 operation is complete, the workpiece coordinate system is also established.

Note: when you set the current workpiece coordinate system, you must return the machine zero.

6.6.2 Trimming workpiece coordinate origin offset

This function is used to amend the workpiece coordinate system, a workpiece coordinate origin due to the measurement accuracy and programming a deviation between the coordinate system origin, the error causes the processing job too big or too small.

Modify the coordinate origin offset of the workpiece steps:

1. Press the "coordinate system" under the primary key or in the parameter interface press "F4" (coordinates bias), enter the coordinates offset interface;

2. Press the cursor keys to select the required setting and workpiece coordinate system axis;

3. Press "F2" (+ enter), through numeric keys to enter numbers, press the "ENTER", enter the data on the previous data will be added.

Method of determining the amount of trimming: determining bias trimming of input data, deviation of the artifacts that are mainly based on processing to determine if processing large workpieces of an axis direction, you enter a negative value when trimming, too small to enter a positive value.

6.6.3 Zero offset operation

In operation and the processing, if you need to G54~G59 all the coordinate system at the same time to a direction offset, zero offset function can be used. Zero offset value has an effect only after you return the machine zero, so every time when you modify the zero bias, return zero point of machine tools to reconnect the workpiece coordinate system.

Zero offset operation steps:

1. Press [COORSYS] main function key or in the parameter interface press [F4] (coordinates bias), enter the coordinates offset interface;

2. Press the cursor keys to select the required axis bias operation, press the number keys directly enter needs bias;

3. Press [ENTER] confirm, enter the complete press "files/programs" button saves the entered data.

6.6.4 Set the entry point

In operations and the processing, often need to return the entry point. For ease of use system increases G76 function to return the entry point, G76 need to first specify the entry point of the workpiece coordinate, specify the entry point of the workpiece coordinate steps are as follows:

1. Confirm the workpiece coordinate established;

2. In the position interface to move the cutter to the point of entry;

3. Press [coordinate system] primary key or in the parameter interface press [F4] (coordinates bias), enter the coordinates offset interface;

4. Press the cursor keys to select the entry point and need to set up operation axis, press [F7] (memory coordinates);

5. Click [Save] to save your data.

6.7 Initialization

The following functions are realized in the initialization interface: clear memory, format, modify password, recover default parameter, backup and recovery of parameter file, time set and serial number and so on.

Procedure of entering initialization interface:

1. Press main function key 【parameter】 to enter parameter interface;

2. Press【F5】(initialization) to enter the initialization interface.

6.7.1 Clear memory

If the abnormal condition such as system disorder, display disorder and data disorder due to due to external interference during the system operation, the memory shall be cleared. It'll set all the storage units for 0, including operation parameter of power-off protective zone and the program name table. Accordingly, the user shall implement this function to prevent the important parameter and program being lost.

Procedure of memory clear：

1. Enter the initialization interface and then press 【F1】(clear memory), and enter the memory clear interface

2. Input correct password and then press 【ENTER】

3. The system is reset automatically.

6.7.2 Format

When the user program has error, file or file directory is disordered and parameter file can't be saved, the electronic disk shall be formatted. Format will delete all user's program in the system and parameters except the backup parameter (time and password won't be influenced), user shall implement this function with care, to prevent important program and

current parameter file being lost. Before format is started, current parameter file shall be saved, refer to the parameter backup operation.

Format steps:

1. Enter initialization interface and then press 【F2】(format), and enter format interface

2. Input correct password and then press 【ENTER】

3. System starts format.

6.7.3 Modify password

System adopts password to prevent some important parameter files being damaged due to incorrect operation and causing system failure. The default system password is "**XZ0012**", user can modify password in the system password modify interface.

Steps of modifying password:

1. Enter initialization interface and then press 【F3】(modify password), and enter password modify interface;

2. Input the correct password in the input box of "Old password";

3. Input new password in the input box of "New password" and "Input password again", the passwords input two times shall be same;

4. After password is input, press 【F4】(confirm) (or the cursor is in the input box of confirming password, and 【ENTER】 can be also pressed) to save new password;

5. When the password is modified, system will indicate "Password has been modified, please keep new password well" in the status bar, if system indicates "Password is error", please input correct password in the old password input box, else password can't be modified, if system indicates "New password isn't same as the above one, please input again", it'll mean the two passwords input in the "New password" and "Confirm password" are different, please confirm the password and then input again.

6.7.4 Initialization

The default value interface primarily completes the following functions: recover the default value of parameter, backup parameter to backup data zone, recover parameter from backup data zone, import from U disk and export from U disk.

Enter the default value interface:

1. Press main function key 【parameter】 to enter parameter interface;

2. Press【◄►】key and press【F4】(initialization) to enter the initialization interface;

3. Press 【F4】(default value) to enter default value interface.

6.7.4.1 How to recover the default value of parameter

The function of recovering default value of parameter is to recover the parameter of appointed parameter system to the default parameter, this function is primarily used when the system can't operate normally due to numerical control system parameter disorder, the parameter default value can ensure normal operation of system, but it won't conform to specific demands of user, accordingly, user shall implement this function with care. In order to prevent incorrect operation, password shall be input before operation

Steps of recovering default value:

1. Enter the default value interface;

2. Press 【Password】 key, input correct password and then press 【ENTER】;

3. Using the cursor keys [←] and [→] move the cursor to "current parameters" column, then use the cursor keys [↑] and [↓] moves to the line on which the need to restore factory value;

4. Press 【F1】(default value) and system starts default value operation.

6.7.4.2 How to backup parameter

The main function of parameter backup is to prevent the parameter data being lost, the parameter backup file isn't impacted by format of electronic disk, accordingly, user shall keep a backup of parameter meeting its own operation parameter in the numerical control system, accordingly, the parameter can be recovered at any time when it has error. The numerical control system provides two storage spaces for each parameter, namely "Backup I" and "Backup II"

Steps of parameter backup:

1. Steps of parameter backup;

2. Press 【Password】 key, input correct password and then press 【ENTER】;

3. Using the cursor keys [←] and [→] move the cursor to "current parameters" column, then use the cursor keys [↑] and [↓] moves to the line on which the need to backup;

4. Press 【F2】(backup) and system starts backup operation.

6.7.4.3 How to recover backup parameter to current parameter file

The parameter recovery is to recover the data backup by user previously to current parameter for use of numerical control system, if user doesn't backup data, this operation can be omitted. The operation of parameter recovery will overwrite current parameter, if this parameter isn't backup, it can't be recovered, accordingly, user shall implement this function with care. Before the parameter is recovered, the user shall browse the backup data which will be recovered to current parameter

Steps of recovering parameter:

1. Enter the default value interface;

2. Press 【Password】 key, input correct password and then press 【ENTER】;

3. Using the cursor keys [←] and [→] move the cursor to "current parameters" column, then use the cursor keys [↑] and [↓] moves to the line on which the need to restore;

4. Press 【F3】(recover) and system start recover operation.

6.7.4.4 How to export current parameter to U disk

The current parameter of numerical control system is stored in the electronic disk in the form of file, including the following files: cutter parameter, system parameter, bit parameter, pitch compensation and coordinate bias, and the filename is respectively T01, S01, B01, I01 and C01.The export parameter is to export the current selected parameter of numerical control system to root directory of U disk, the filename is the one in numerical control system, the "Export file" operation is only for current parameter. User can export one set of parameter meeting requirement to U disk, and then save it in computer, the parameter can be recovered when system parameter is disordered

Steps of exporting file:

1. Enter the default value interface;

2. Press 【Password】 key, input correct password and then press 【ENTER】;

3. Using the cursor keys [←] and [→] move the cursor to "current parameters" column, then use the cursor keys [↑] and [↓] moves to the line on which the need to export;

4. Press【F2】(exported from U disk), after operation is completed, the corresponding parameter file will be copied to the root directory of U disk, if operation fails, the error will be prompted.

6.7.4.5 How to import parameter file from U disk

The U disk import parameter file operation is to find the file whose filename is same as the large cursor under root directory of U disk, after it's found, and use the file in the U disk to replace corresponding current parameter. The "Import file" operation can be only used for current parameter, and this operation can't be used for Backup I and Backup II of numerical control system. The user shall ensure the imported parameter is correct, else it'll cause disorder parameter of numerical control system. Before these imported parameters are processed, please confirm whether the parameter is correct, else it'll damage cutter, machine-tool, work piece and injure staff

Steps of importing file:

1. Enter the default value interface
2. Press 【Password】 key, input correct password and then press 【ENTER】
3. Using the cursor keys [←] and [→] move the cursor to "current parameters" column, then use the cursor keys [↑] and [↓] moves to the line on which the need to import;
4. Press【F4】(imported from U disk), after operation is completed, the corresponding root directory file of U disk will be copied to the system as current parameter to be used, if operation fails, the error will be prompted

6.7.5 Time set

In the time set interface, the user can view and set current date and time, and the data isn't impacted by the format, if the system belongs to demo version, it won't modify current time but it can be browsed

The method and procedure of time set:

1. Enter initialization interface, press 【F5】(time set), and enter time set interface
2. Move large cursor to the required position, press number key to input directly, please confirm whether the input data is correct, else the data can't be modified
3. After data is modified, press 【Save】 key to save the modified data

7 PLC

7 Series CNC system adopts open PLC, PLC system is equipped with standard system when factory, with standard lathes function. Machine tool works and the users can design their PLC to meet your specific needs, PLC parameters can be modified in the PLC interface, such as T,C,K,D, or you can manage operations, such as export PLC to u disk, from the u disk imported PLC, imported machine tool factory PLC, PLC and PLC backup and recovery of the import System.

We recommend that users use u export feature, and then between PLC and PC or USB drive will be backed up.

7.1 PLC Specifications

Programe language	PLC ladder

First level of program execution cycle	10ms
The average processing time of basic instructions	3us/step
Program capacity	10000steps
Instructions	Basic directions: 14 Function directions: 42
Intermediate relay (R) Data register (D) Counter (C) Timer (T) Information signal (A) Keep-relay (K) Jump labels (L) sub-program (P)	R0-R999，R9000-R9099 D0-D999 C0-C39 T0-T39 A0-A9 K0-K19 L1～L9999 P1～P9999
CNC→PLC(F) PLC→CNC(G) Machine→PLC(X) PLC→Machine(Y)	F0-F255 G0-G255 X0-X19 Y0-Y19

7.2 Introduction to basic instructions

Index	Instruction	Function
1	RD	Read status of the specified signal, and is set in ST0.
2	RD.NOT	Read the logical negation of the specified signal back into ST0.
3	WRT	Write results of logical operations (ST0) output to the given address.
4	WRT.NOT	On the results of logical operations (ST0) after taking non-output to the given address.
5	AND	Signal state and logic and with ST0, and written back to the ST0.
6	AND.NOT	Signal state logic negation and logic and with ST0 and,write back to the ST0.
7	OR	Signal status and ST0 logical or, and written back to the ST0.
8	OR.NOT	Signal status negated then logical or with ST0, and written back to the ST0.
9	RD.STK	Signal status of the specified address push into stack
10	RD.NOT.STK	Signal status the negation of the specified addresses and push into stack.
11	AND.STK	After the ST0 and ST1 logical and, the stack register stack, writes ST0.
12	OR.STK	After the ST0 and ST1 logic or, the stack register stack, writes ST0.

Index	Instruction	Function
13	SET	ST0 and after signaling logic or in the specified address, returns the results to the specified address.
14	RST	Negate the State of ST0, and logic and with signals of the specified address, returns the results to the specified address.

As described in the PLC programming manual

7.3 Introduction to PLC function instructions

Index	Instruction	Funtion
1	END1	End of first grade ladder diagram programs
2	END2	End of the second-level ladder diagram programs
3	TMR	Timer
4	TMRB	Fix timer
5	DECB	Binary decoder
6	CTR	Counter
7	CTRB	Fix counter
8	CTRC	Counter C
9	ROTB	Binary rotation control
10	CODB	Binary code conversion
11	MOVE	Logic and data transfer after
12	MOVOR	Logic or data transfer after
13	MOVB	Transmission of one byte
14	MOVW	Two-byte transfer
15	MOVN	Send N bytes
16	PARI	Parity
17	DCNVB	Extended data conversion
18	COMPB	Binary comparison
19	SFT	Shift register
20	DSCHB	Binary data search
21	XMOVB	Binary index data transfer
22	ADDB	Binary addition
23	SUBB	Binary subtraction
24	MULB	Binary multiplication
25	DIVB	Binary division
26	NUMEB	Binary constant definition
27	DIFU	Rising edge detection
28	DIFD	Falling edge detection
29	EORB	XOR
30	ANDB	Logic and
31	ORB	Logical or
32	NOTB	Logical not
33	COM	Common line control

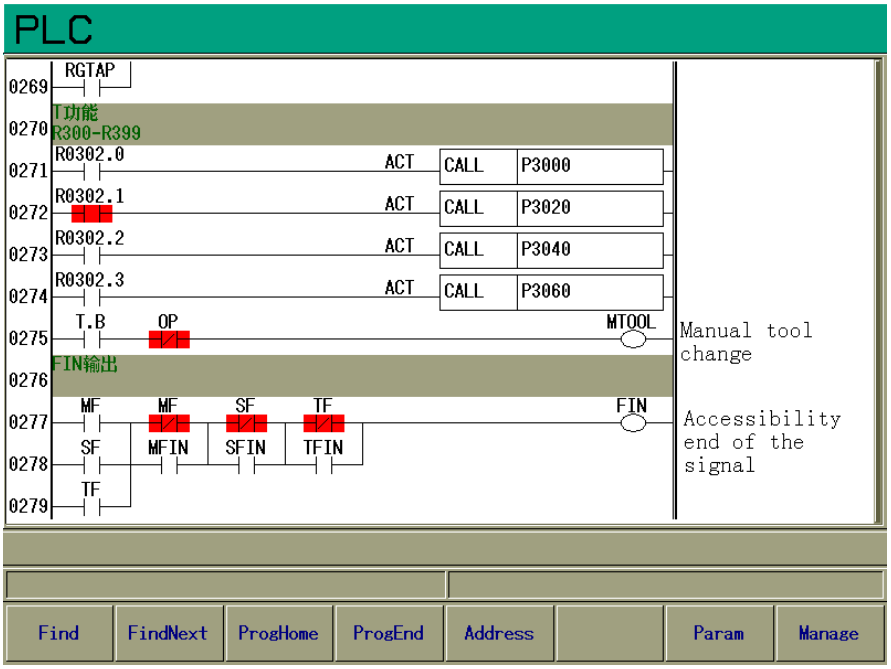
Index	Instruction	Funtion
34	COME	The results of the public line control
35	CALL	Condition subroutine call
36	JMPB	Jump to label
37	LBL	Lable
38	CALLU	Unconditional subroutine call
39	SP	Subroutine
40	SPE	End subroutine
41	SPLCNT	Spindle control

As described in the PLC programming manual.

7.4 PLC interface description

Press [PLC] main function key to enter the PLC interface, as shown in F3.18. In the PLC interface, you can explore and monitor the status of PLC, check the PLC to write logical and operation.

The interface to display real-time input, output and PLC status, can effectively help PLC writers to troubleshoot of the problem.



F3.18 PLC ladder diagram interface

PLC interface described in the following

"F1": Find. Enter the component number or symbol, to find the location of the component in the ladder.

"F2": Find next. Find in the ladder to the next location of the component.

"F3": Program first. Move the cursor to the first line of the PLC ladder diagram.

"F4": End of the program. The cursor moves to the last line of PLC ladder diagram.

"F5": Symbol/Address. PLC ladder diagram symbols in the symbol and switch between addresses.

"F7": Enter the PLC parameter interface.

"F8": Enter the PLC management interface.

PLC interface described in the following action key:

"PAGE UP": forward current PLC ladder diagram displays a page.

"PAGE DOWN": backward through the current PLC ladder diagram displays a page.

"↑": move the cursor, the cursor up one line in the current PLC ladder diagram, when the cursor moves to the top of the current PLC ladder diagram, continue to press

"↑" button, the current overall PLC ladder diagram down one line, while the PLC ladder diagram shown on the previous line that does not appear at the top of the ladder diagram. If the current cursor position is the current first line of the PLC ladder diagram, move the stop up.

"↓": Move the cursor, the cursor down one row in the current PLC ladder diagram.

Note: specific PLC ladder diagram programming and symbolic significance see PLC programming manual.

7.4.1 PLC parameter

PLC parameter can be set in PLC parameter interface, including t, c, k, d.

Enter the PLC parameter interface steps:

1. Press the "PLC" primary key, enter the PLC interface.

2. Press the "F7" (parameters) into the PLC parameter interface.

PLC parameter interface f function keys introduced:

"F1": timer t table. PLC timer setting preset values.

"F2": counter c table. Set PLC counter preset value.

"F3": keep the relay k. Set the PLC keep type relay preset values.

"F4": data table address d. Set the PLC data table address value.

"F7": restore parameter. You can use these parameters to the default value, the default value set in the PLC-editing software.

7.4.1.1 Timer T table view and set

1. Access to PLC parameter interface, and then press the "F1" (t) enter the timer t table interface;

2. Press the "password" button, the prompt bar displays "Please enter a password" and the password input box, please enter the correct password in the password input box, and then press "ENTER";

3. Press the key "PAGE UP", "PAGE DOWN " and cursor keys "←" or " ↑ ", "→", " ↓ " mobile t set cursor to view or select the desired preset;

4. Press the number keys to enter T preset values directly, and then press "ENTER", set after completion, the "file/program" to save the data.

7.4.1.2 Counters c table view and set

1. Access to PLC parameter interface, and then press "F2" (c) enter the counter c table interface;

2. Press the "password" button, the prompt bar displays "Please enter a password" and the password input box, please enter the correct password in the password input box, and then press "ENTER";

3. Press the key "PAGE UP", "PAGE DOWN " and cursor keys "←" or " ↑ ", "→", " ↓ " move the cursor to see c set or select the desired preset value;

4. Press the number keys directly into the c preset value, and then press "ENTER", set after completion, the "file/program" to save the data.

7.4.1.3 Keep type relay K table view and set

1. Access to PLC parameter interface, and then press the "F3" (k) enter the maintained type relay k table interface;

2. Press "password" button, the prompt bar displays "Please enter a password" and the password input box, please enter the correct password in the password input box, and then press "ENTER";

3. Press the key "PAGE UP", "PAGE DOWN " and cursor keys "←" or " ↑ ", "→", " ↓ " move the cursor to view or select the desired set of k-value;

4. Direct input k value, press the number key, then press "ENTER", set after completion, the "file/program" save data.

7.4.1.4 Data table address D viewing and setting

1. Access to PLC parameter interface, and then press "F4" (d) enter the data table address d table interface;

2. Press the "password" button, the prompt bar displays "Please enter a password" and the password input box, please enter the correct password in the password input box, and then press "ENTER";

3. Press the key "PAGE UP", "PAGE DOWN " and cursor keys "←" or " ↑ ", "→", " ↓ " d value for moving the cursor to view or select the desired setting;

4. Press the number keys to enter the d values directly, and then press "ENTER", set after completion, the "file/program" save data.

7.4.1.5 Restore PLC parameter operation

Enter the PLC parameter interface, press the "F7" (restoring parameters), will restore to PLC PLC parameter edit software settings, restore parameters include, T,C,D. Keep the relay k data unchanged.

7.4.2 PLC managment

PLC the PLC management can import and export, PLC backup and recovery operations. Huaxing NC 7 series systems using open PLC, machine tool works and users can be developed for the second time, end user use of PLC may be customized, so once lost will be difficult to restore. We recommend that users use export PLC to u disk function, PLC backup to PC or Flash drive to prevent memory corruption or other unpredictable happens, you can recover PLC.

7.4.2.1 PLC import

Imported PLC included: recovery PLC, PLC import, import from a USB drive manufacturers, import systems PLC. The features are as follows:

1. Restoring PLC: recover the current PLC to PLC status when a backup is performed for the last time;

2. Imported PLC manufacturers: import plcT.rom file from a USB drive, making it the current PLC;

3. Import system PLC: imported PLC of CNC system is factory, the PLC has standard lathes function, but may not be able to meet the needs of end users, please exercise caution. After the import operation is complete, the system needs to restart, the new PLC program to run.

Note: PLC is essential for normal operation of the system, please exercise caution.

7.4.2.2 PLC export

PLC PLC can be used to backup the export operation, if necessary, can recover PLC to protect CNC system is functioning correctly. PLC exported include: backup PLC, export, export user PLC, PLC manufacturers and export systems PLC. The features are as follows:

1. Backup PLC: corresponding to the restored PLC, PLC only executed backup operations to restore PLC;

2. Export user PLC: export the current PLC to u disk, generates plcT.rom files;

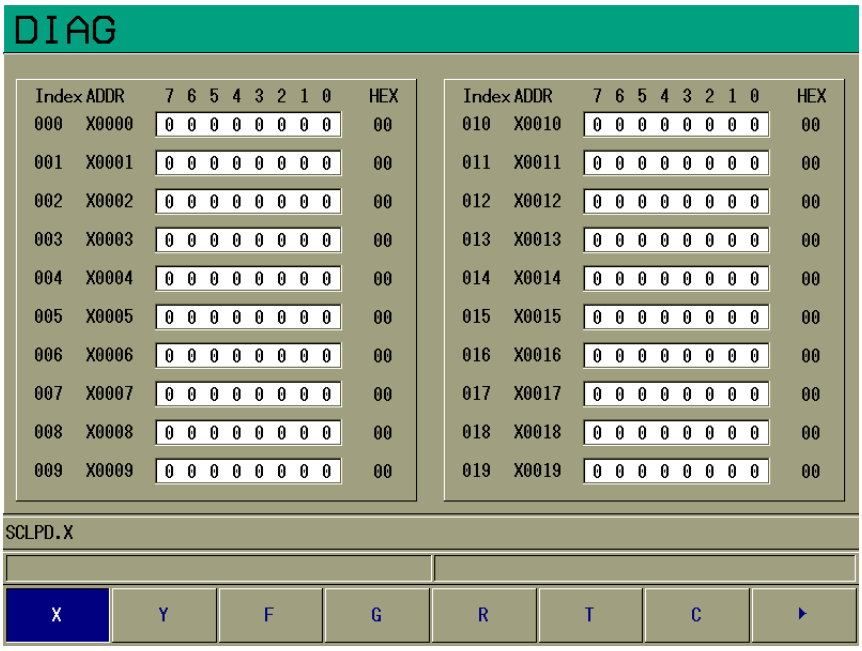
3. Export the system PLC: manufacturers factory PLC of CNC system export to USB disk, generates plcT.rom files;

Export files can be used for backup, PLC can also be used to edit the import functionality of the software, to modify the PLC program.

8 Diagnose screen

Press the "Diagnostics" primary key into diagnostic interfaces, as shown in F3.19. Diagnostic interface x table (machine tools → PLC), y table (PLC → machine) detection and monitoring capabilities, this feature in debugging and error test of machine tool, very useful.

8.1 Introduce to diagnose screen



F3.19 diagnostic interface

Diagnostic interface described in the following:

"F1" (x): input to the PLC signal diagnosis of machine tool.

"F2" (Y): PLC signal output to machine tool of diagnosis.

"F3" (f): NC output to the PLC signal diagnosis.

"F4" (G): diagnosis of PLC outputs between NC signal.

"F5" (r): PLC internal diagnosis of intermediate relay.

"F6" (t): diagnosis of PLC internal timer. "F7" (c): diagnosis of PLC internal counters.

"F8" (►): move to the right, the x, y, tables, and other interfaces as to the right a bit.

Press "◄ ►" key:

"F1" (a): PLC diagnosis of alarm signals.

"F2" (k): latching relay in the PLC signal diagnosis.

"F3" (spindle and handwheel): diagnosis of spindle and wheel.

"F4" (macro): diagnosis of macro variables.

"F5" (alarm list): alarm history.

Note: the interface corresponding to each diagnosis, the diagnosis of specific significance see PLC programming manual.

8.2 X table diagnose

1. Press the "diagnose" primary key, and then press the "F1" (x-table) into x-table diagnostic interface;

2. Press the key "PAGE UP" and "PAGE DOWN" cursor keys "←" or "↑", "→", "↓" and "F8" key (right), move the cursor to view each diagnosis x table;

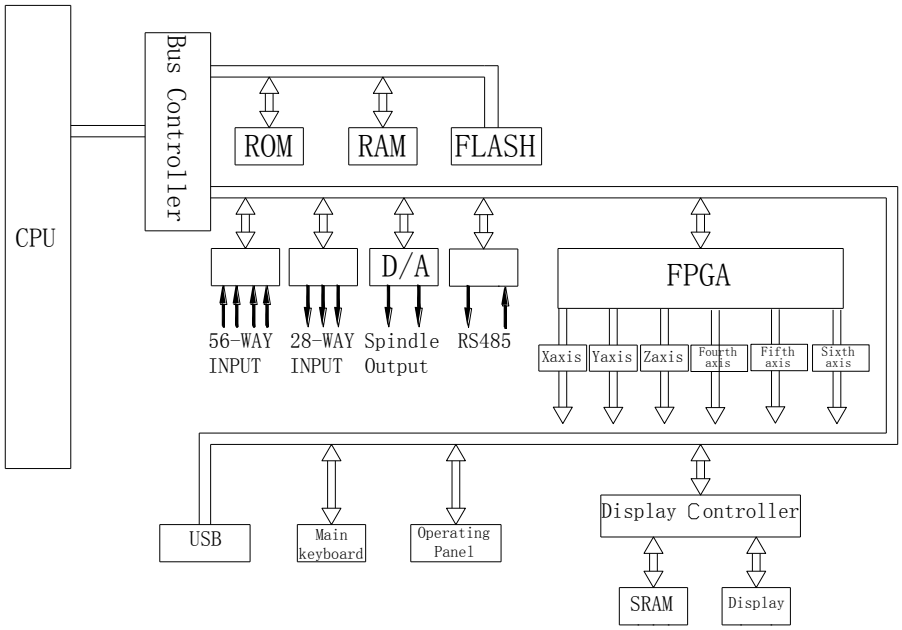
3. Prompting bar specific diagnosis, the cursor will be defined.

Note: Other diagnostic methods of table x table with diagnosis, no longer be addressed.

IV System connect

1 System composition

1.1 Numerical control system control unit block diagram

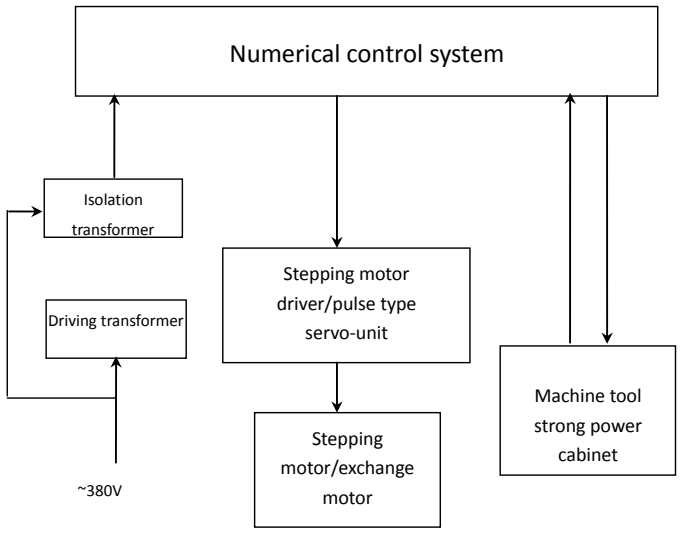


F4.1 WA710MI/WA710MD/WA720M-4/WA740M-5/WA730M-6 CNC control unit figure

Description: The picture shows an overall block diagram motherboard, a system model and configuration differences exist, so the actual provision of resources depending on the specific model.

1.2 A typical electrical machine program

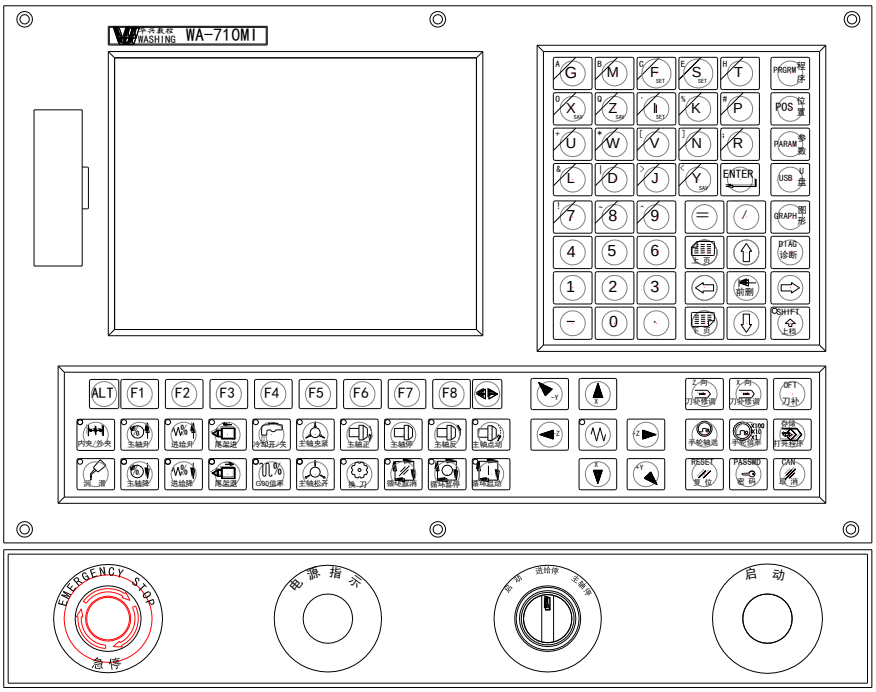
1. CNC control unit and accessories
2. Stepping motor drive driving power/pulse servo unit
3. Stepping motor/Servo motor
4. Machine tool power closet

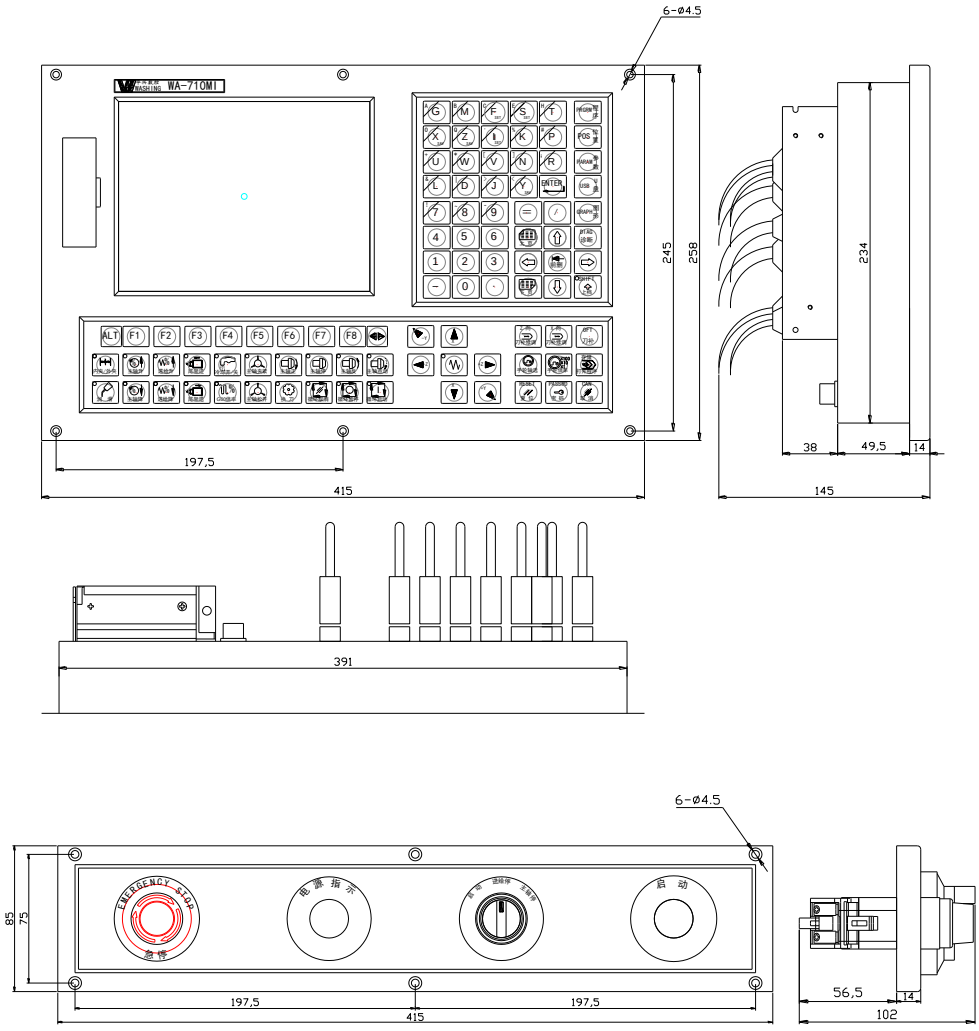


F4.2 A typical electrical machine program

1.3 System Installation Dimensions

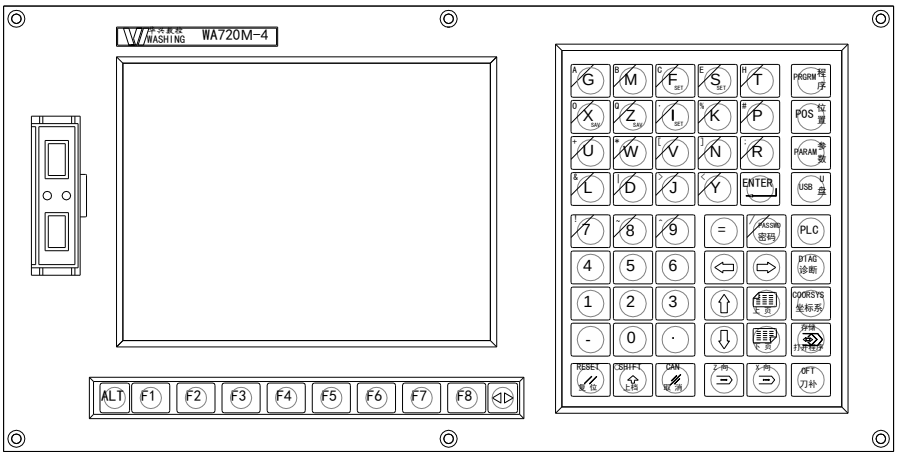
(1) WA710MI/WA710MD System Installation Dimensions

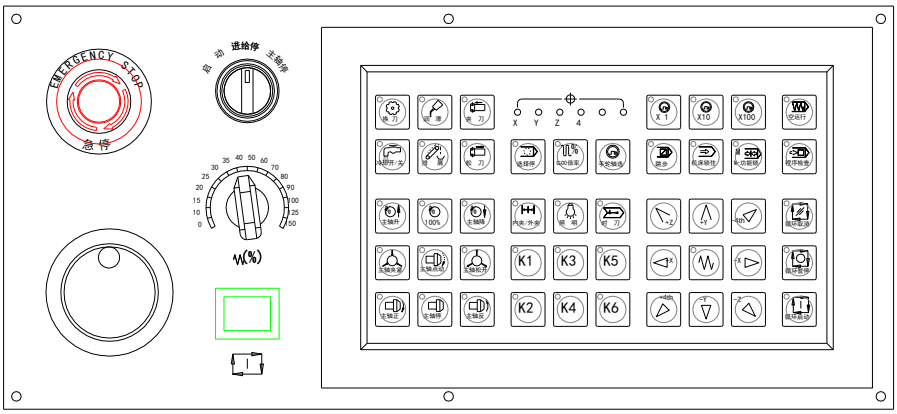


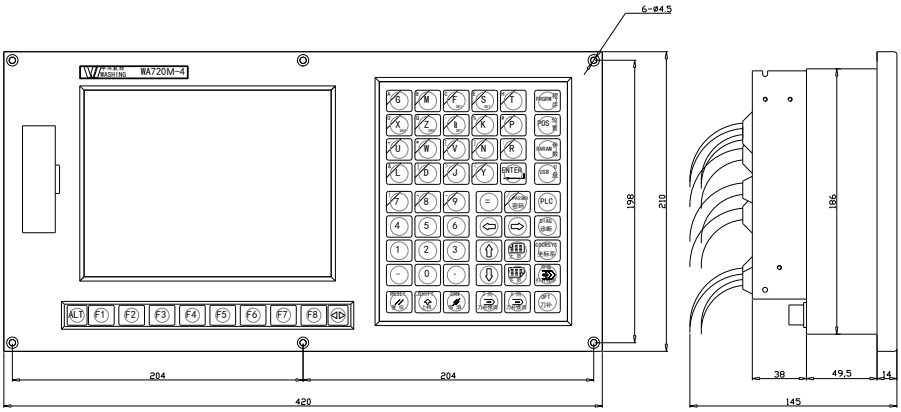


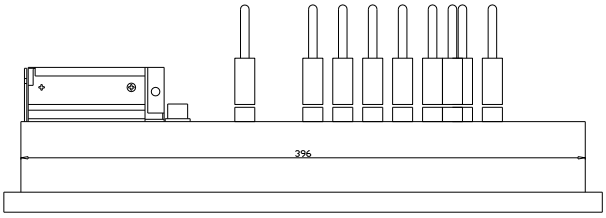
F4.3 WA710MI/WA710MD System Installation Dimensions

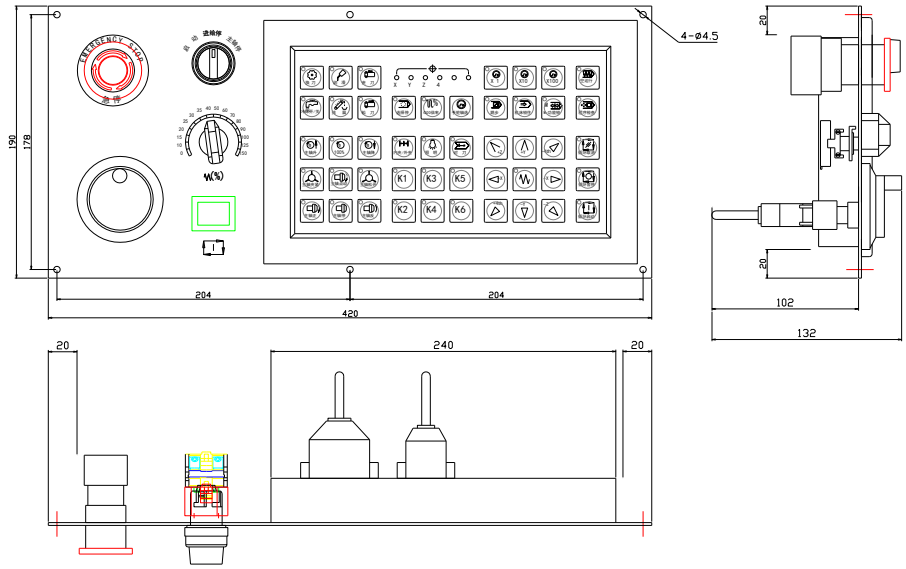
(2) WA720M-4 System Installation Dimensions


WASHING
WA720M-4
ALT
F1
F2
F3
F4
F5
F6
F7
F8


EMERGENCY STOP
急停

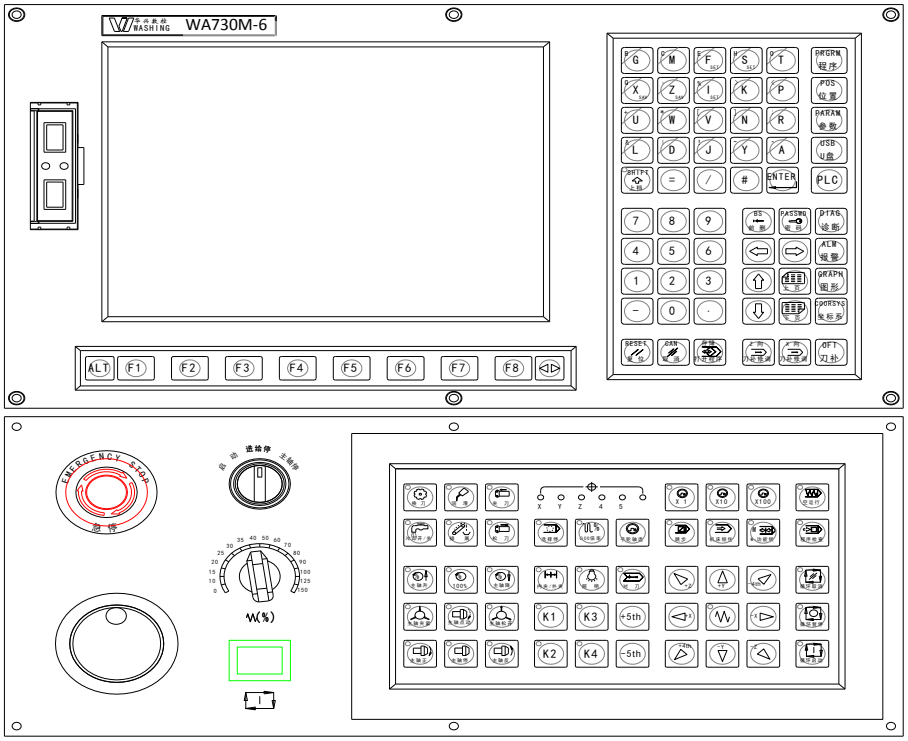

6-Ø4.5
WASHING
WA720M-4
198
210
186
204
204
420
38
49.5
14
145

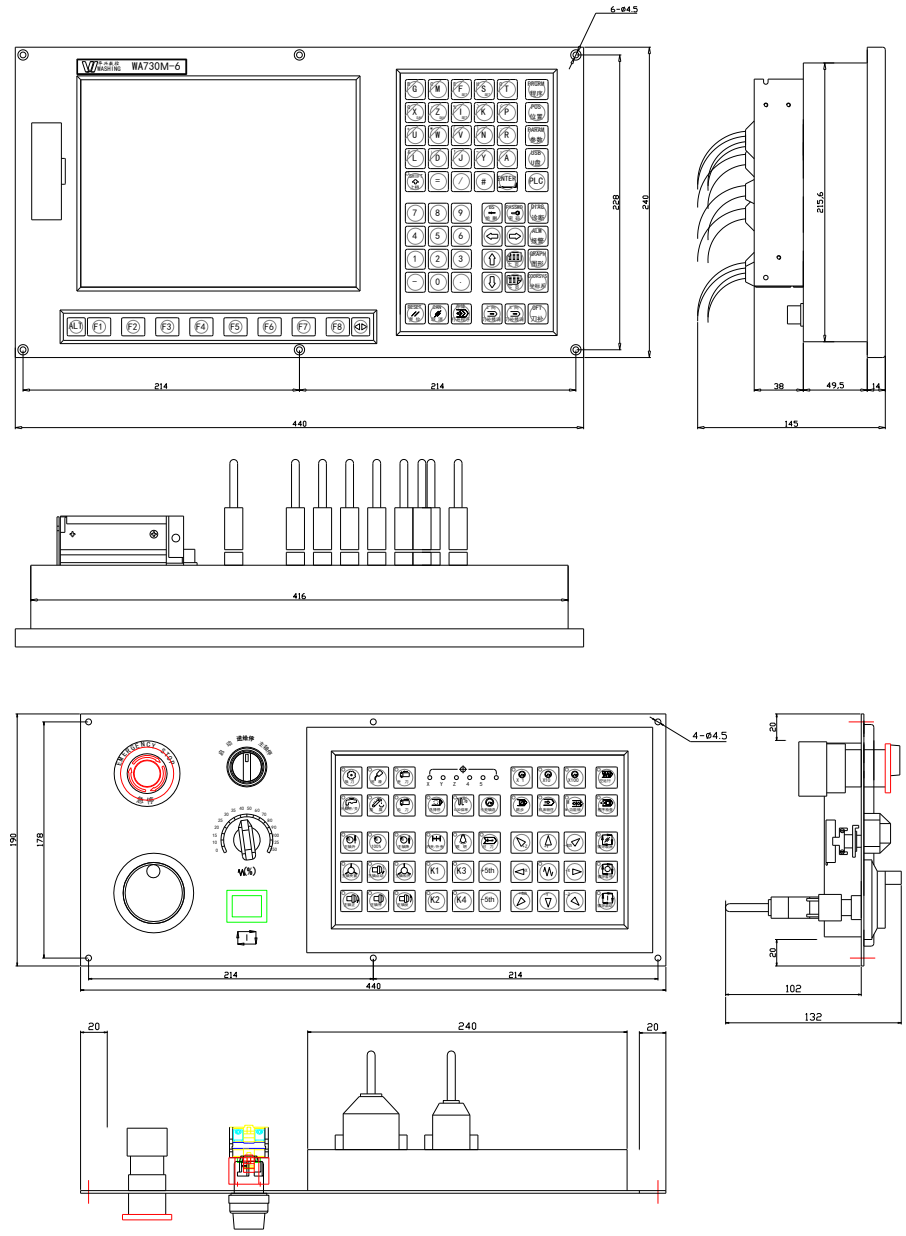

396



F4.4 WA720M-4 System Installation Dimensions

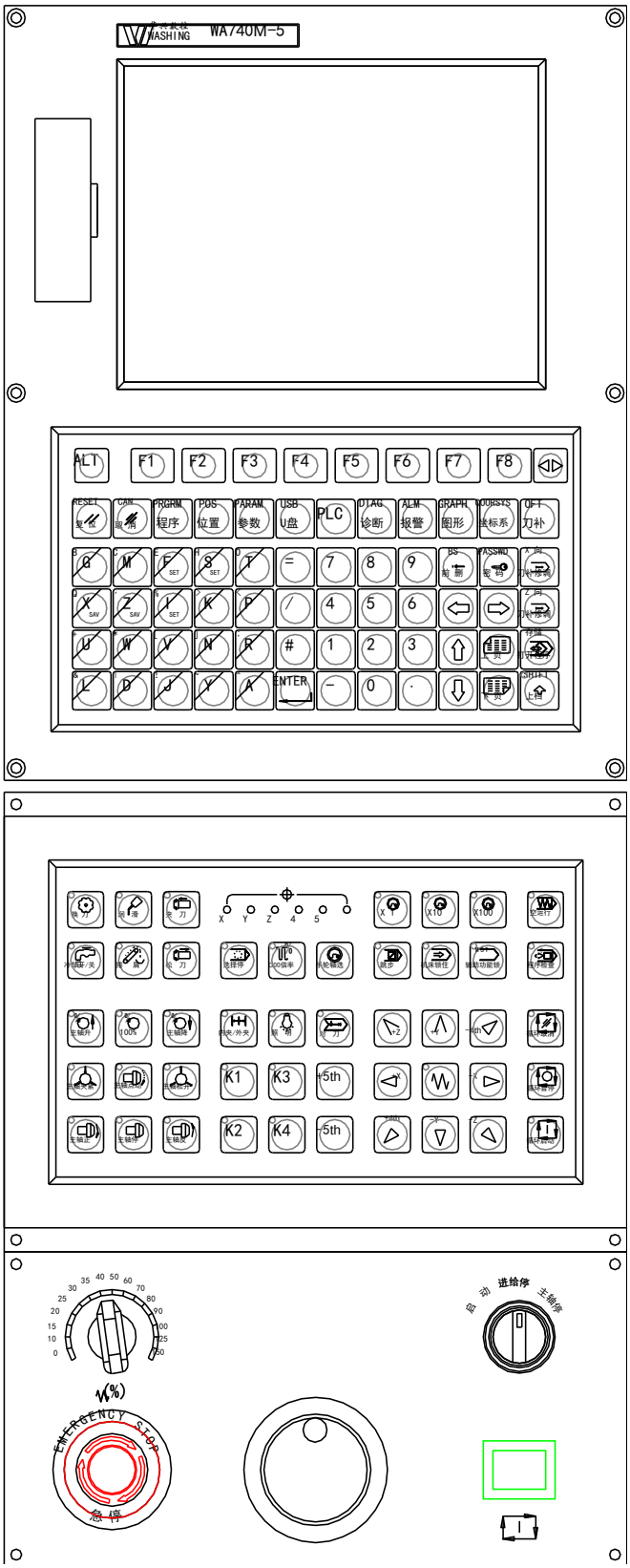
(3) WA730M-6 System Installation Dimensions

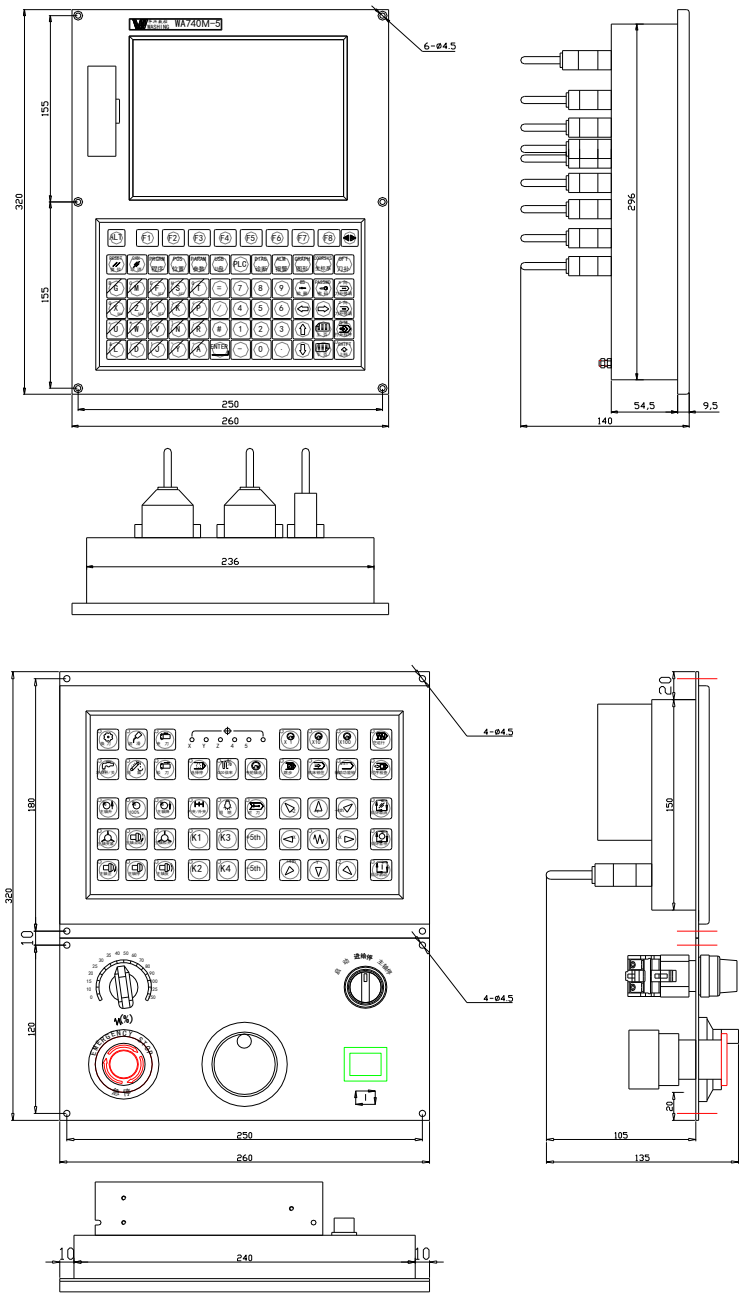




F4.5 WA730M-6 System Installation Dimensions

(4) WA740M-5 System Installation Dimensions

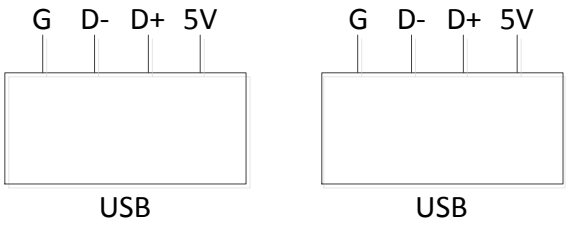

WASHING
WA740M-5
ALT
F1
F2
F3
F4
F5
F6
F7
F8
RESET
PRGRM
程序
POS
位置
PARAM
参数
USB
U盘
PLC
DIAG
诊断
ALM
报警
GRAPH
图形
OFT
刀补
ENTER
SHIFT
X
Y
Z
4
5
K1
K3
K2
K4
5th
-5th
EMERGENCY STOP
急停
进给停
主轴停



F4.6 WA740M-5 System Installation Dimensions

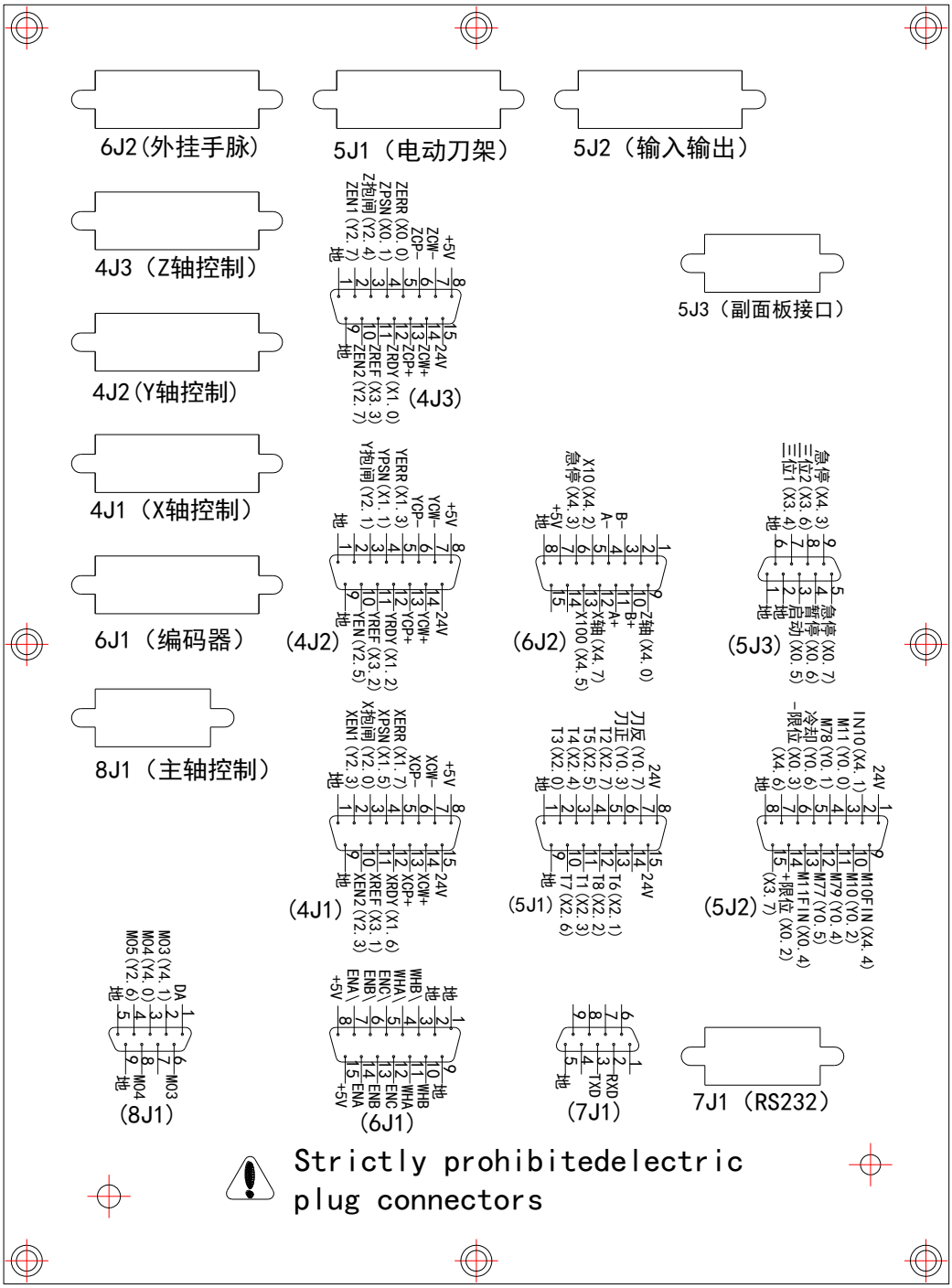
1.4 Port definition list

1.4.1 Front cover panel(in plastic box cover)definition：

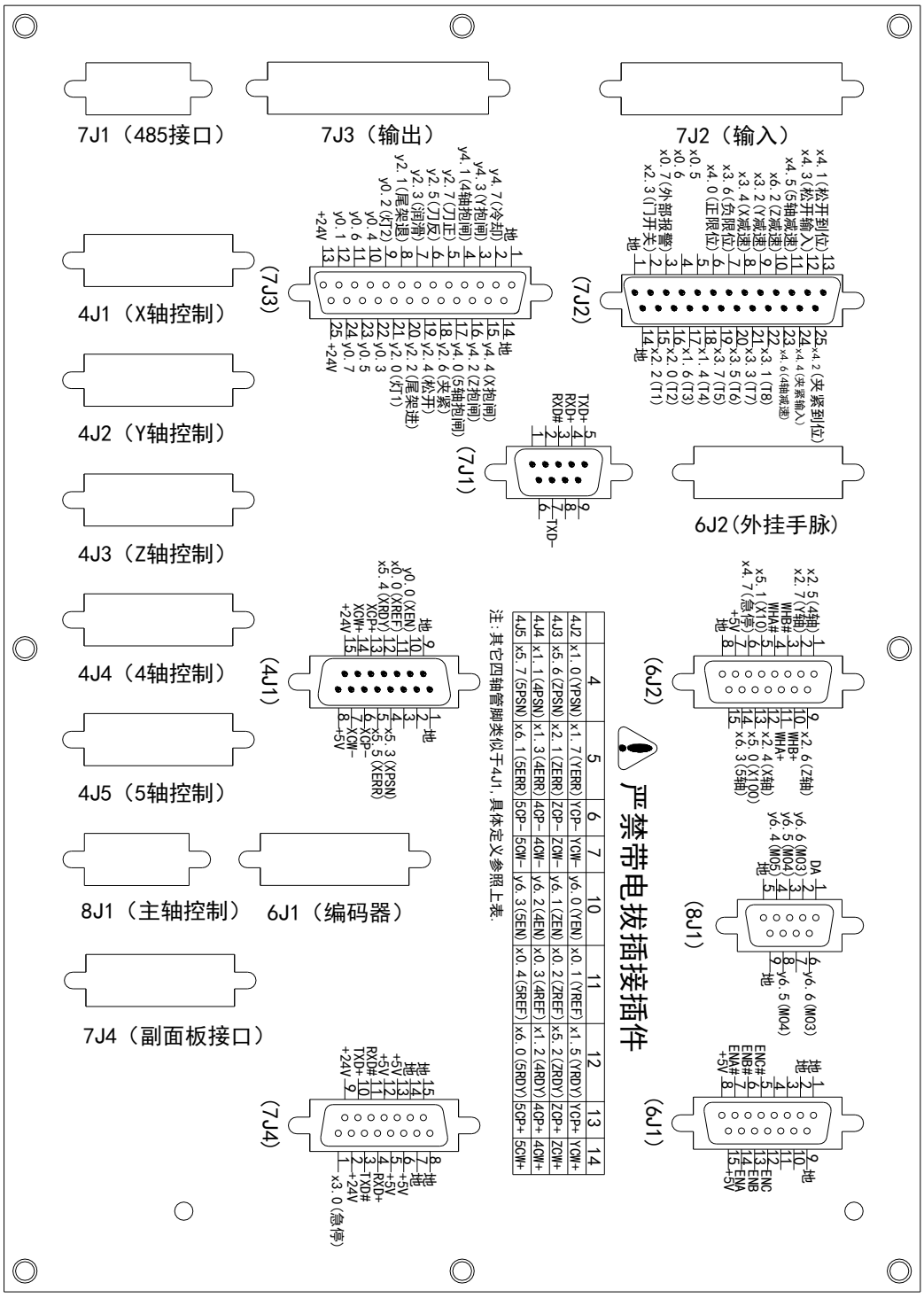


F4.7 Front panel interface definition

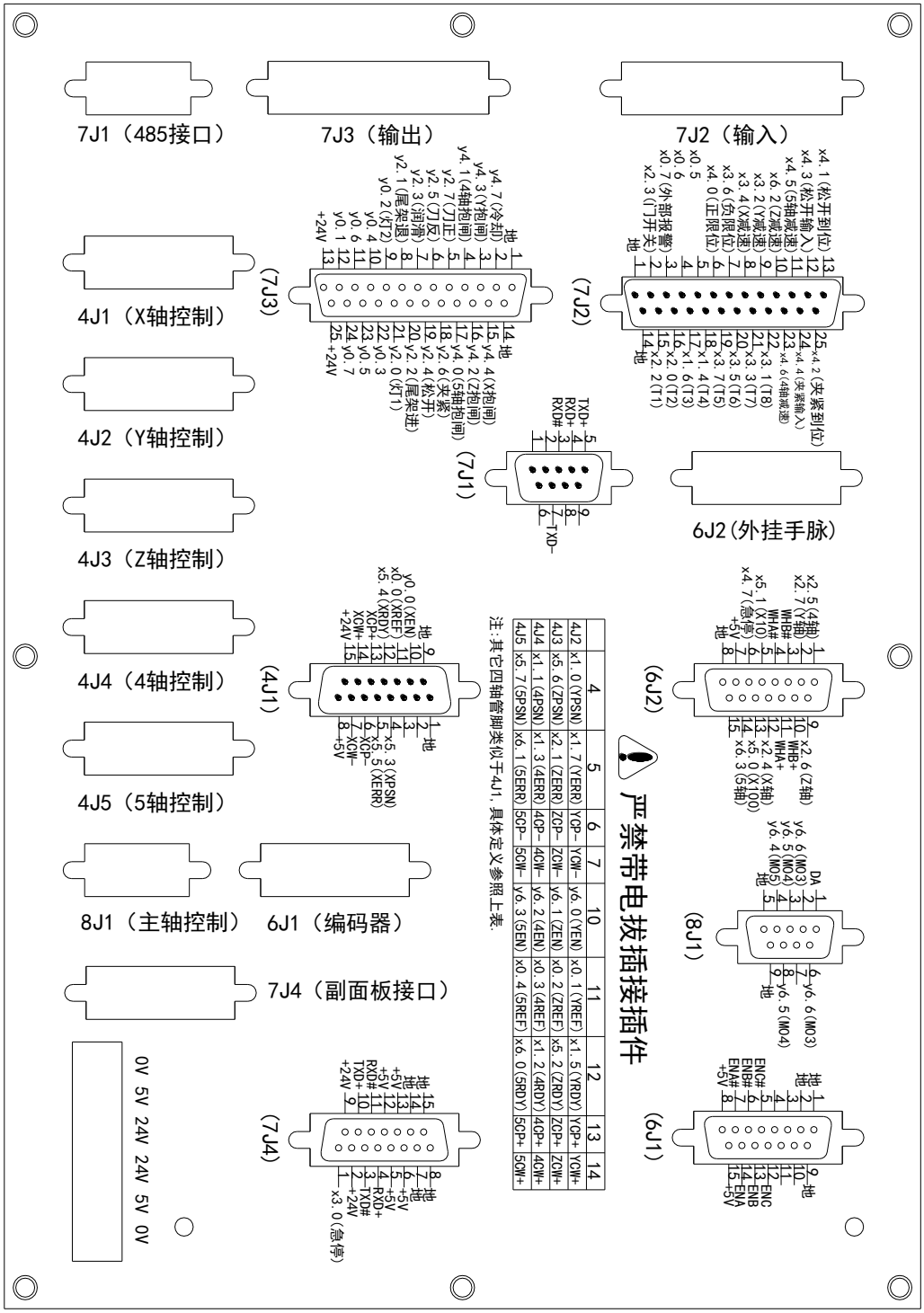
1.4.2 Back cover panel port definition



F4.8 710MI Back cover panel port definition



F4.9 WA710MD/WA720M-4/WA730M-6 Back cover panel port definition



F4.10 WA740M-5 Back cover panel port definition

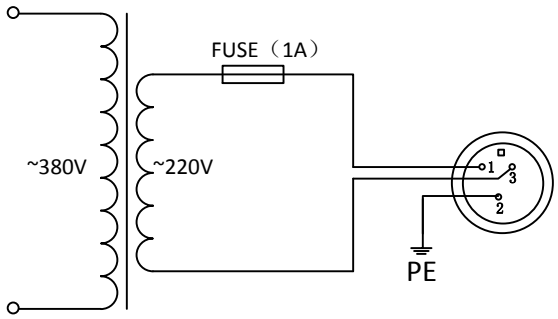
2 Heavy current power supply

2.1 Installation requirement

WASHING CNC system should work in good mechanical and electric environment, it should be properly installed in mechanical and electric consideration and the input/output ports should be properly connected. To CNC system, you should make a box to install the system, there are six-Twelve Φ4.5 through holes in the system panel to fix the system to the box with M4 screws. The box size should be big enough to take the length of plug behind the system and wires into consideration. The box should have good heat sink character.

2.2 Heavy current power supply

Washing CNC system requires the voltage of power supply variation stands within more/less 10% of standard nominal voltage. So we suggest to install a 150VA isolated transformer. See F4.11.



F4.11

Note: the output voltage of transformer mentioned in this manual are all no-load voltage, the capacity can not be lower than the specified value.

2.3 Earth

Earth is very important in electrical installation. Proper earth can make the CNC system operate more steady and reliably and avoid electricity leakage. Washing CNC system has external earth point and the point must be reliably connected to earth during application. Live up to:

1. Make sure the whole machine tool electrical system must be connected to a main earth point and properly earthed.

2. The signal earth of electronic device that has communication with CNC system must be connected to earth point and the earth point must be properly connected to the main machine tool earth point, the connection wire area no less than 2.5mm^2 .

3. The signal wire must have shield layer, for it can have better anti-jamming effect if to use the shield layer as power earth transfer.

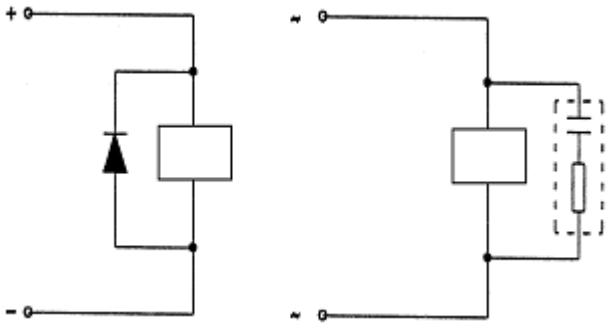
4. It is forbidden to use the A/C earth wire (middle line in the three phase currency) as

earth wire.

2.4 The issues need attention during H/C installation

CNC system must be connected to the machine part to strong electric control various actions throughout the machine. To ensure reliable operation of the system, strong power part of all machine should be installed with inductive load interrupter device. Recommendations are as follows (shown in Figure 4.12):

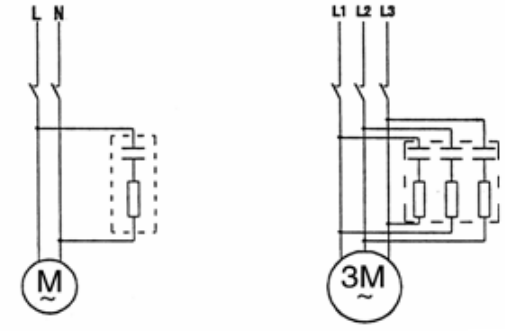
1. AC contactor coil, single-phase interrupter contacts in parallel across the coil;
2. DC relay coils in parallel freewheeling diodes.



(a) D/C relay loop (b) A/C relay loop

F4.12 Interrupter devices

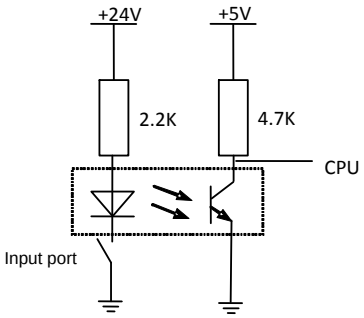
AC motor: according to the motor is a single-phase or three-phase selection of single-phase / three-phase interrupter, not with discrete resistors, capacitors homemade absorption circuit. RC shown in Figure 4.13 must be installed to switch or contactor load side, the typical connection:



F4.13

3 CNC internal connection

3.1 Input/output illustration

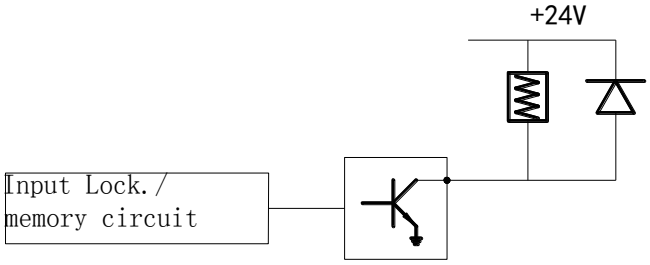


F4.14 Input port circuit

Note: All 7 Series CNC system 0V input signals are valid and can be directly connected with the use of an NPN Hall switch devices.

When knife or other external device, such as the use of PNP-type Hall switches, need to be equipped Washing NPN PNP switch signal conversion board.

3.2 Output port circuit illustration



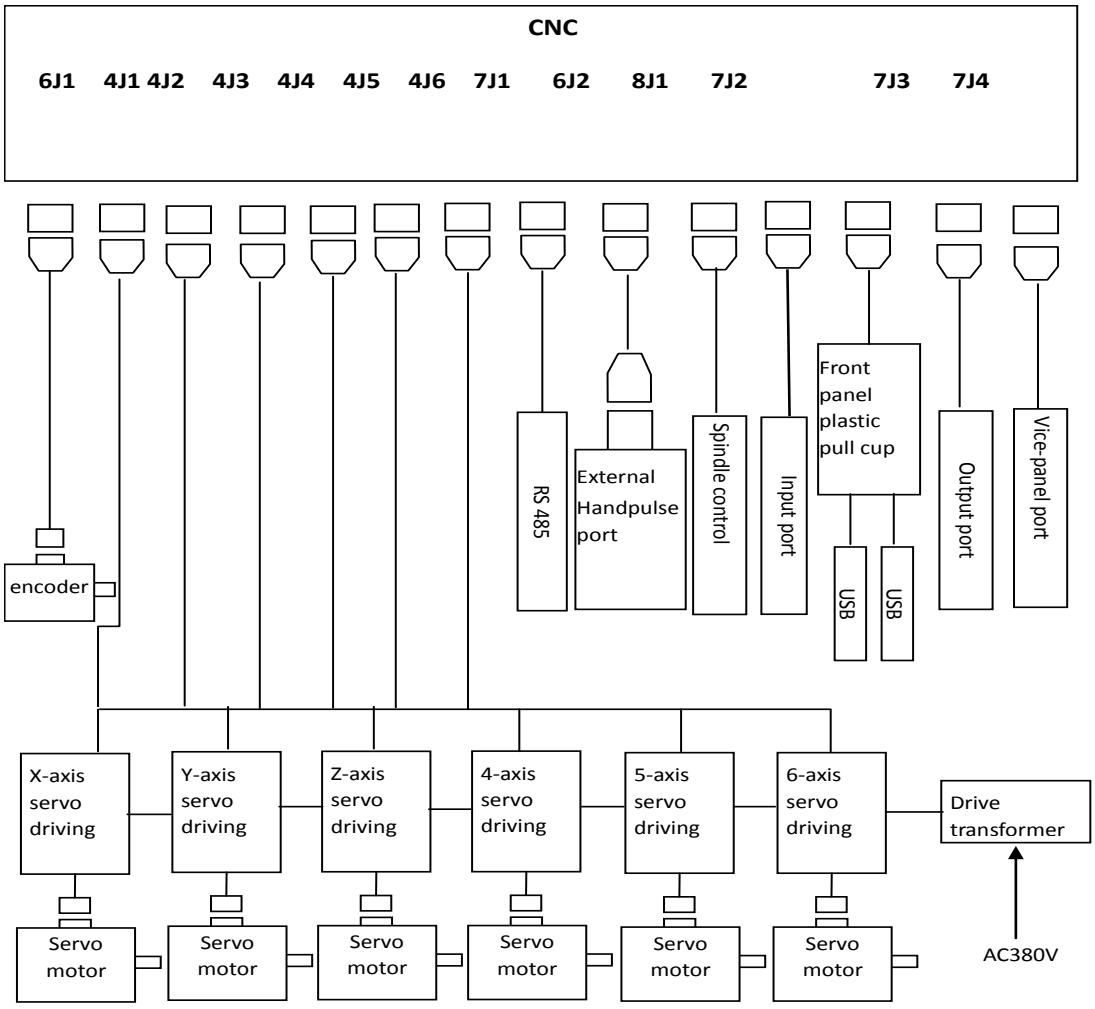
F4.15 Output circuit

Note :All output (besides M03,M04) are all collector output, limiting current of 70MA, limiting voltage: 30V, Users should consider its external loads. Beyond the load voltage and load current should be used when external auxiliary relay. M03,M04 for relay contact output, limiting current 0.5A, limiting voltage: 30V.

4 CNC signal interface definition

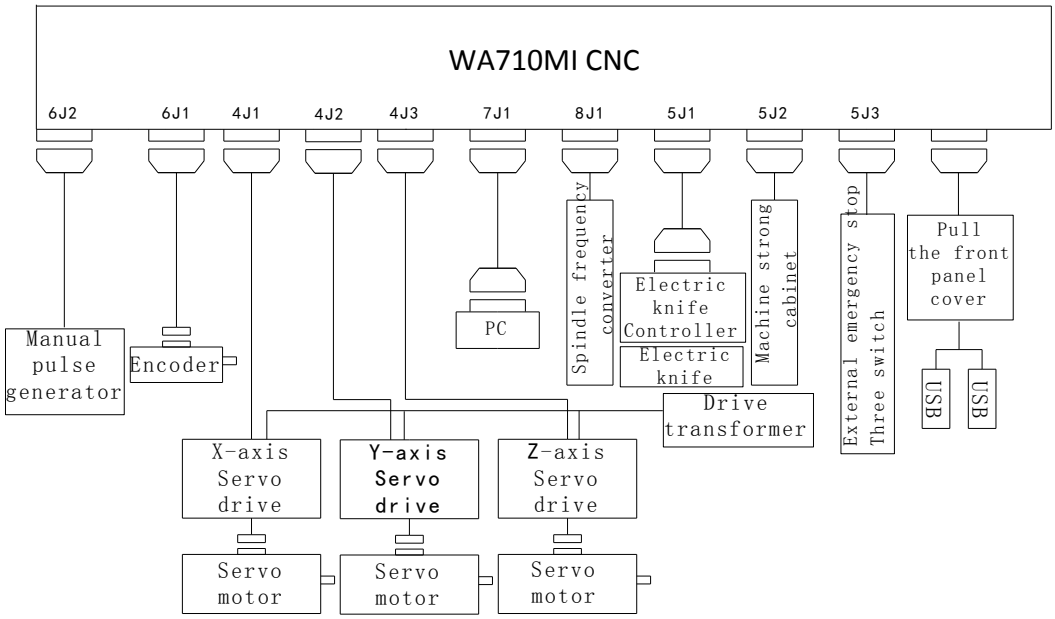
4.1 CNC external connection

The components related with this CNC control unit are connected as Figure 4.16:



F4.16 WA710MD/WA720M-4/WA740M-5/WA730M-6 external connection

Note: Not all systems contain all of the above interfaces.

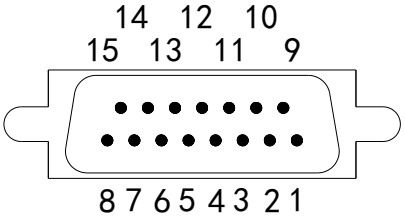


F4.17 WA710MI external connection

4.2 Motor drive interface

4.2.1 WA710MD/WA720M-4/WA740M-5/WA730M-6 Motor drive interface

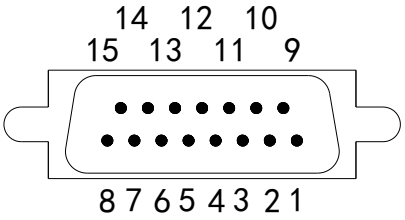
WA710MD/WA720M-4/WA740M-5/WA730M-6 CNC system motor driver interface4J1、4J2、4J3、4J4、4J5 and 4J6 output respectively the x, y, z, fourth-, v-axis, and the sixth axis motor drive signals. Port definition:



F4.18 Motor driver interface 4J1(DB15 pin)

Table 4.1 X-axis control 4J1

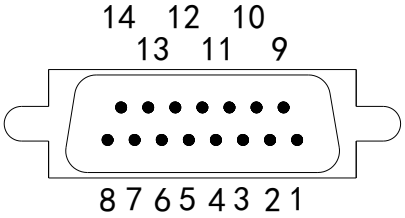
Pin	Function define	Pin	Function define
01	ground	09	ground
02		10	y0.0(XEN)
03		11	x0.0(XREF)
04	x5.3(XPSN)	12	x5.4(XRDY)
05	x5.5(XERR)	13	XCP+
06	XCP-	14	XCW+
07	XCW-	15	+24V
08	+5V		



F4.19 Motor driver interface 4J2(DB15 pin)

Table 4.2 Y-axis control 4J2

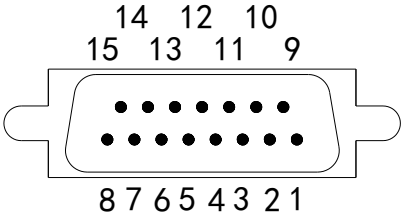
Pin	Function define	Pin	Function define
01	ground	09	ground
02		10	y6.0(YEN)
03		11	X0.1(YREF)
04	X1.0(YPSN)	12	x1.5(YRDY)
05	X1.7(YERR)	13	YCP+
06	YCP-	14	YCW+
07	YCW-	15	+24V
08	+5V		



F4.20 Motor driver interface 4J3(DB15 pin)

Table4.3 Z-axis control 4J3

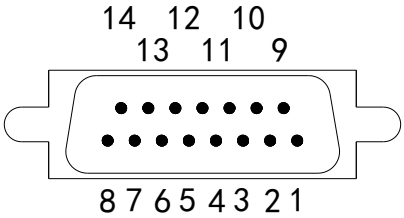
PIN	Function define	PIN	Function define
01	ground	09	ground
02		10	y6.1(ZEN)
03		11	x0.2(ZREF)
04	x5.6(ZPSN)	12	x5.2(ZRDY)
05	x2.1(ZERR)	13	ZCP+
06	ZCP-	14	ZCW+
07	ZCW-	15	+24V
08	+5V		



F4.21 Motor driver interface 4J4(DB15 pin)

Table4.4 Fourth-axis control 4J4

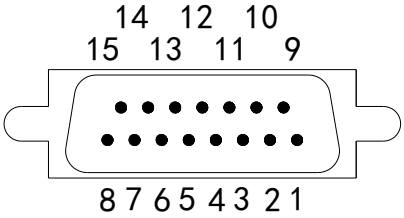
Pin	Function define	Pin	Function define
01	ground	09	ground
02		10	y6.2(AEN)
03		11	x0.3(AREF)
04	x1.1(APSN)	12	x1.2(ARDY)
05	x1.3(AERR)	13	4CP+
06	4CP-	14	4CW+
07	4CW-	15	+24V
08	+5V		



F4.22 Motor driver interface 4J5(DB15 pin)

Table4.5 Fifth-axis control 4J5

Pin	Function define	Pin	Function define
01	ground	09	ground
02		10	y6.3(BEN)
03		11	x0.4(BREF)
04	x5.7(BPSN)	12	x6.0(BRDY)
05	x6.1(BERR)	13	5CP+
06	5CP-	14	5CW+
07	5CW-	15	+24V
08	+5V		



F4.23 Motor driver interface 4J6(DB15 pin)

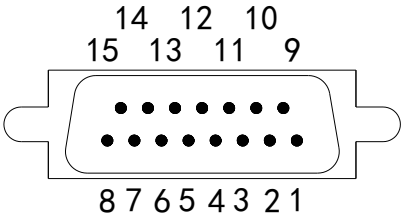
Table4.6 Sixth-axis control 4J6

Pin	Function define	Pin	Function define
01	ground	09	ground
02		10	y4.5(CEN)
03		11	x0.5(CREF)
04	x6.4(CPSN)	12	x6.6(CRDY)
05	x6.5(CERR)	13	6CP+

06	6CP-	14	6CW+
07	6CW-	15	+24V
08	+5V		

4.2.2 WA710MI CNC system motor driver interface

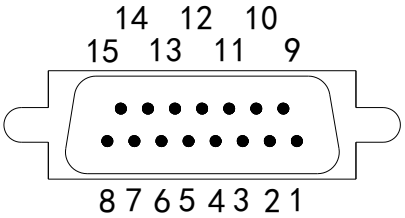
WA710MI CNC system motor driver interface 4J1、4J2、4J3 output respectively the x, y, z motor drive signals. Port definition:



F4.24 Motor driver interface 4J1(DB15 pin)

Table 4.7 X-axis control 4J1

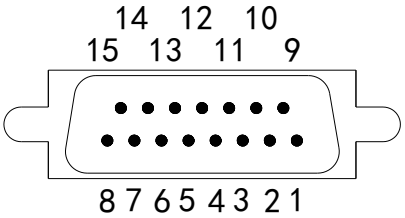
Pin	Function define	Pin	Function define
01	ground	09	ground
02	y2.3(XEN1)	10	y2.3(XEN2)
03	y2.0(X Brake)	11	x3.1(XREF)
04	x1.5(XPSN)	12	x1.6(XRDY)
05	x1.7(XERR)	13	XCP+
06	XCP-	14	XCW+
07	XCW-	15	+24V
08	+5V		



F4.25 Motor driver interface 4J2(DB15 pin)

Table 4.8 Y-axis control 4J2

Pin	Function define	Pin	Function define
01	ground	09	ground
02		10	y2.5(YEN)
03	y2.1（Y Brake）	11	x3.2(YREF)
04	x1.1(YPSN)	12	x1.2(YRDY)
05	x1.3(YERR)	13	YCP+
06	YCP-	14	YCW+
07	YCW-	15	+24V
08	+5V		



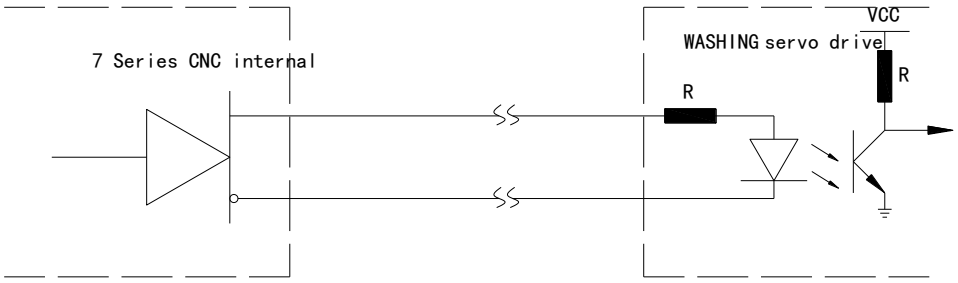
F4.26 Motor driver interface 4J3(DB15 pin)

Table 4.9 Z-axis control 4J3

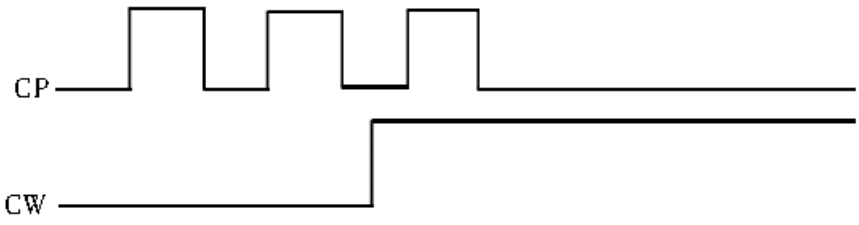
Pin	Function define	Pin	Function define
01	ground	09	ground
02	y2.7（ZEN1）	10	y2.7(ZEN2)
03	y2.4（Z Brake）	11	x3.3(ZREF)
04	x0.1(ZPSN)	12	x1.0(ZRDY)
05	x0.0(ZERR)	13	ZCP+
06	ZCP-	14	ZCW+
07	ZCW-	15	+24V
08	+5V		

The motor signal port only adopts hardware ring-sectioned stepping motor driver or pulse A/C servo unit. The output control signal mode：CP pulse and CW direction signal. CP outputs at positive pulse, and the corresponding motor moving forward one step according to one pulse. CW controls the motor to clockwise at high level, and counterclockwise at low level. The pulse signal is 1/2 duty，when the direction signal switch direction, the advance pulse is 1/4 duty.

The circuit diagram:



F4.27 Circuit diagram



F4.28

Description: when the shielding layer is used to connect +5V or 0V ground terminal at both sides, the core wire in the shielding layer can't be used to connect +5V or 0V ground and more.

Other signal related with motor control, take X-axis for instance:

P10:XEN2: servo enable signal, inform servo to operate with power on.

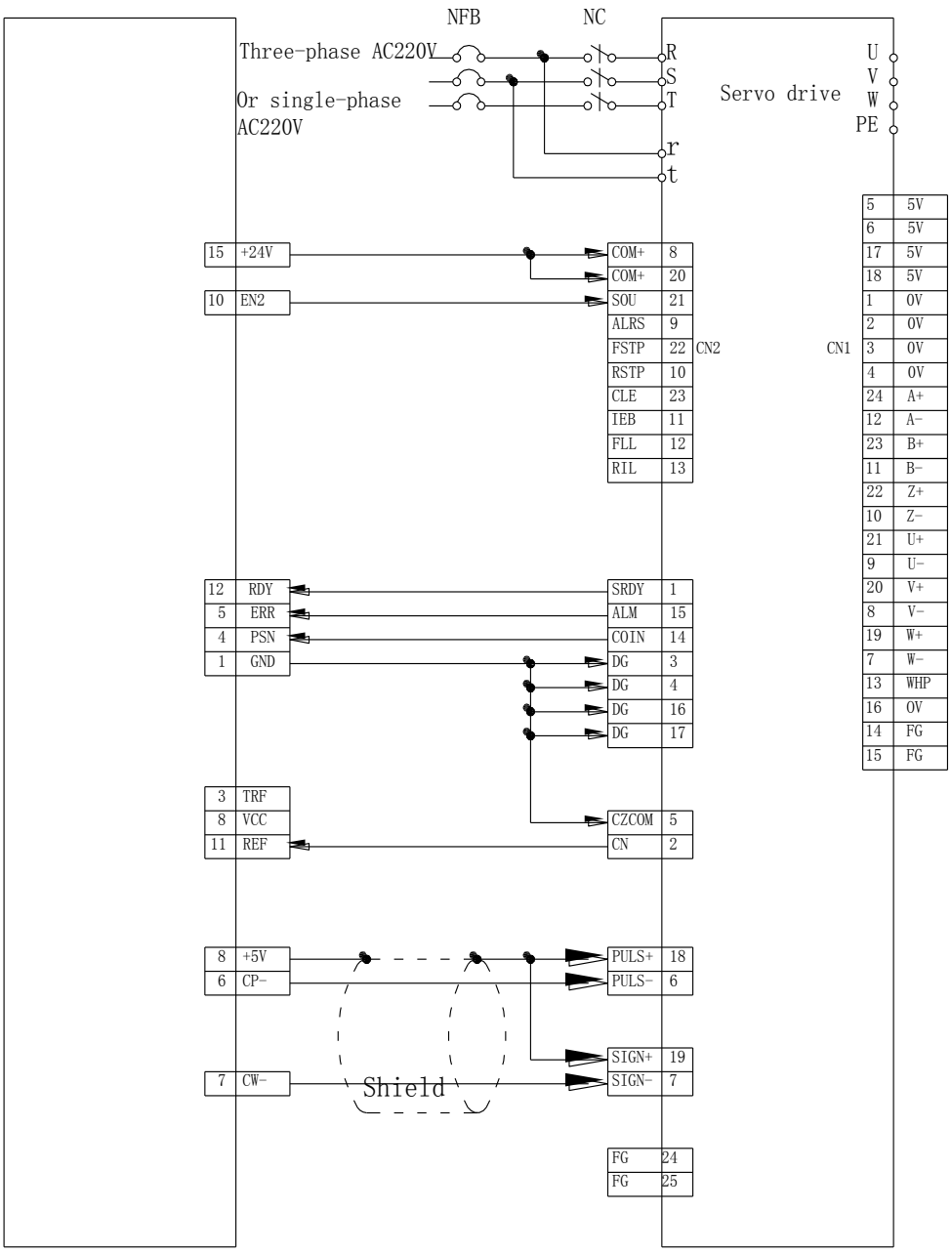
P12:XRDY: input, after the servo-unit accepts XEN1/2, self-check and lock with power on, send XRDY signal to CNC normally.

P4:XPSN: input, when CNC stops at Z pulse position or eliminate the following error to set value, the CNC XPSN signal will be answered.

P5:XERR: input, when the servo-unit has error owing to certain cause or can't operate, this signal will be fed back to CNC.

P11:XREF: the servo-unit zero-returning (machine tool returns to reference point) signal can also connect the Z signal of the electric machine encoder to the XREF terminal, CNC will directly detect the Z pulse of machine tool and the zero of machine tool will be determined; XTRF is invalid, it's suggested that the manufacturer should adopt this method to return reference point.

The motor signal port is connected with triple-phase composite driver of our company as Figure4.29.

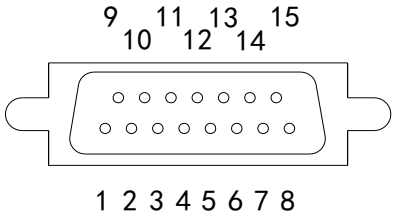


F4.29 Motor signal port and my company AC servo drive connection diagram

4.3 Spindle encoder interface

4.3.1 WA710MD/WA720M-4/WA740M-5/WA730M-6 CNC system Spindle encoder interface

WA710MD/WA720M-4/WA740M-5/WA730M-6 CNC system spindle encoder interface 6J1 is'DB15 hole' mode socket，the connection pin is'DB15 pin'.



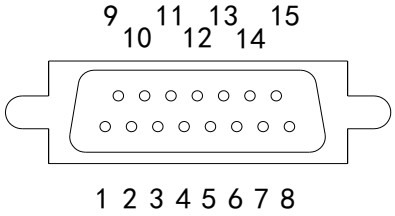
F4.30 Spindle encoder interface 6J1(DB15 hole)

Table 4.10 Spindle encoder interface 6J1

Pin	Function define	Pin	Function define
01	ground	09	ground
02	ground	10	
03		11	
04		12	ENC
05	ENC＃	13	ENB
06	ENB＃	14	ENA
07	ENA＃	15	+5V
08	+5V		

4.3.2 WA710MI spindle encoder interface

WA710MI spindle encoder interface 6J1 is'DB15 hole' mode socket，the connection pin is'DB15 pin'.



F4.31 Spindle encoder interface 6J1(DB15 hole)

Table 4.11 Spindle encoder interface 6J1

Pin	Function define	Pin	Function define
01	ground	09	ground
02	ground	10	WHB+
03	WHB-	11	WHA+
04	WHA-	12	ENC+
05	ENC-	13	ENB+
06	ENB-	14	ENA+
07	ENA-	15	+5V
08	+5V		

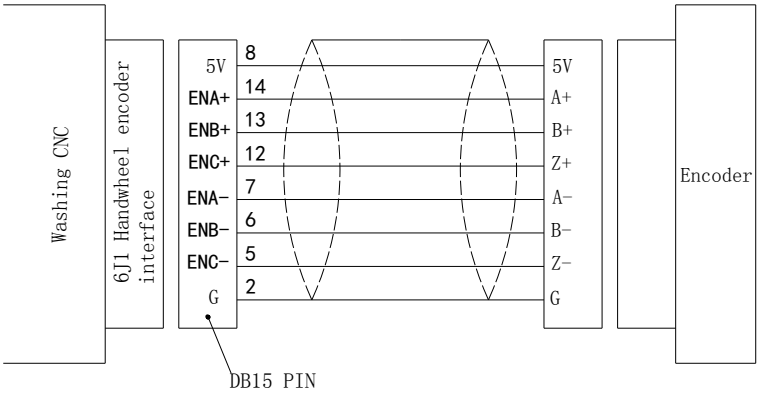
Only adapted to the spindle encoder which meets the following conditions :

1. Operating voltage：5V.
2. Pulse number per circle：700～2400.
3. Output signal：three ways of difference output，i.e. A＋、A－ ,B＋、B－and Z＋、

Z－.

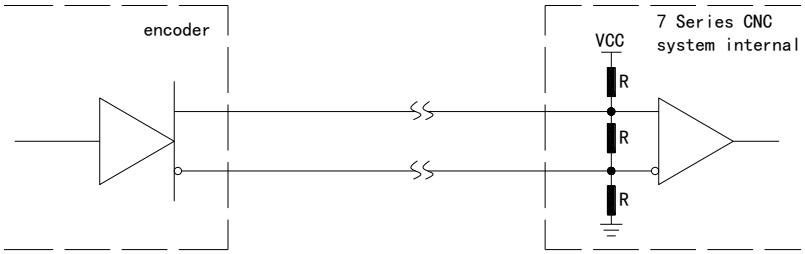
The coder connection must use shield wire, and dual cored shield wire is more preferred, two shield wires connect one way difference signal, the connection method in following diagram.

Explanation：Using the shield layer as ＋5V earth connection, and core wire is forbidden for ＋5V earth connection.



F4.32 Encoder connection diagram

The circuit diagram:

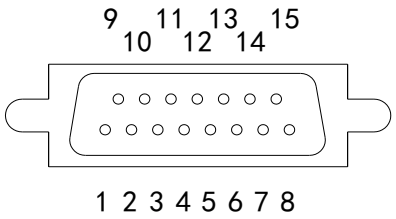


F4.33 The circuit diagram

4.4 External handwheel Interface

4.4.1 WA710MD/WA720M-4/WA740M-5/WA730M-6 external handwheel Interface

WA710MD/WA720M-4/WA740M-5/WA730M-6 external handwheel interfaces 6J2 is 'DB15 hole' type socket, and the connected plug is 'DB15 pin', as figure:



F4.34 External handwheel interfaces 6J2(DB15 hole)

Table 4.12 External handwheel interfaces 6J2

PIN	Function define	PIN	Function define
01	x2.5(fourth-axis)	09	x2.6(Z-axis)
02	x2.7(Y-axis)	10	WHB+

03	WHB#	11	WHA+
04	WHA#	12	x2.4(X-axis)
05	x5.1(X10)	13	x5.0(X100)
06	x4.7emergency stop	14	x6.3(fifth-axis)
07	+5V	15	
08	G		

System supports multiple hand-operated pulse generator access modes, and can freely choose current hand-operated pulse generator and parameters related with hand-operated pulse generator：

B100=1：External hand wheel Effective.

B103=1：External clock hands effectively, use the external override switch for axis selection.

B160=1：X-axis hand wheel to change direction.

B161=1：Y-axis hand wheel to change direction.

B162=1：Z-axis hand wheel to change direction

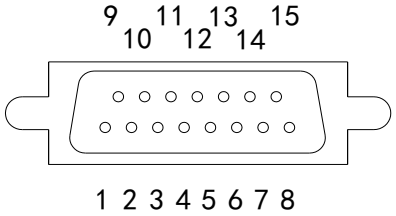
B163=1：Fourth-axis to change direction.

B164=1：Fifth-axis to change direction.

B165=1：Sixth-axis to change direction.

4.4.2 WA710MI external handwheel Interface

WA710MI external handwheel interfaces 6J2 is 'DB15 hole' type socket, and the connected plug is 'DB15 pin', as figure:



F4.35 External handwheel interfaces 6J2 (DB15 hole)

Table 4.13 External handwheel interfaces 6J2

PIN	Function define	PIN	Function define
01		09	x4.0(Z axis)
02		10	WHB+
03	WHB-	11	WHA+
04	WHA-	12	x4.7(X axis)
05	x4.2(X10)	13	x4.5(X100)
06	x4.3 emergency stop	14	
07	+5V	15	
08	GND		

System supports multiple hand-operated pulse generator access modes, and can freely choose current hand-operated pulse generator and parameters related with hand-operated pulse generator：

B100=1：External hand wheel Effective.

B103=1：External clock hands effectively, use the external override switch for axis

selection.

B160=1：X-axis hand wheel to change direction.

B161=1：Y-axis hand wheel to change direction.

B162=1：Z-axis hand wheel to change direction.

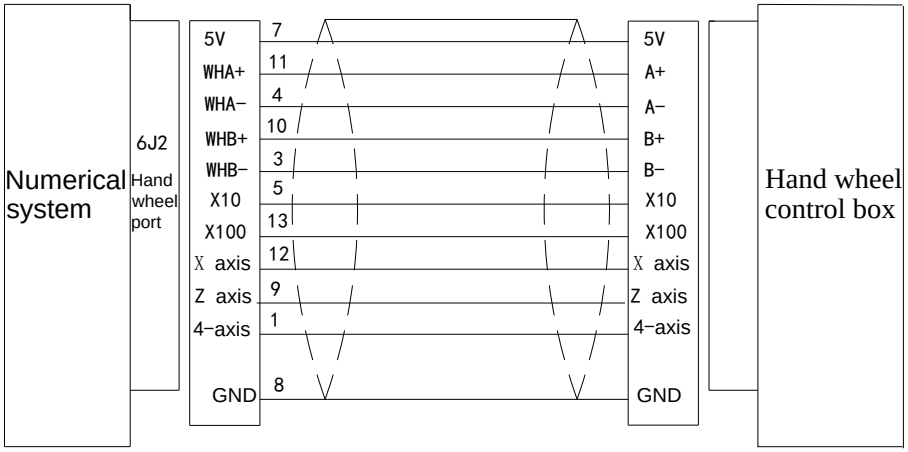
Following conditions are met only fit the hand wheel (also called hand pulse):

(1) Operating voltage: 5V.

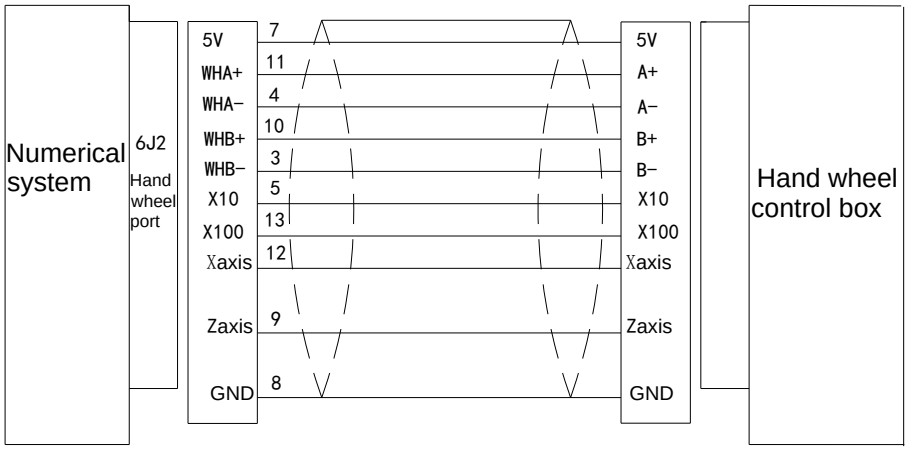
(2) The number of pulses per revolution: 100.

(3) Output signal: two differential outputs, namely A +, A-and B +, B-.

Handwheel connection must use shielded cable and shielded twisted-pair cable as possible, take along two twisted-pair differential signal, press the map production:



F4.36 WA710MD/WA720M-4/WA740M-5/WA730M-6 hand wheel connection

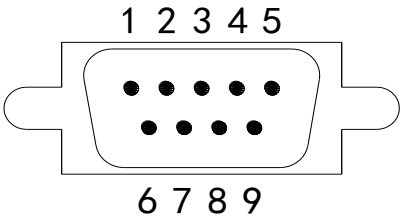


F4.37 WA710MI hand wheel connection

4.5 Serial communication interface

4.5.1 WA710MD/WA720M-4/WA740M-5/WA730M-6 Serial communication interface

WA710MD/WA720M-4/WA740M-5/WA730M-6 RS485 Port 7J1 is 'DB9 pin' socket, Full-duplex 485 interface, For PC and systems transmit programs,When the PC without full duplex RS485 interface, usually in PC machine and one USB full-duplex RS485 module.

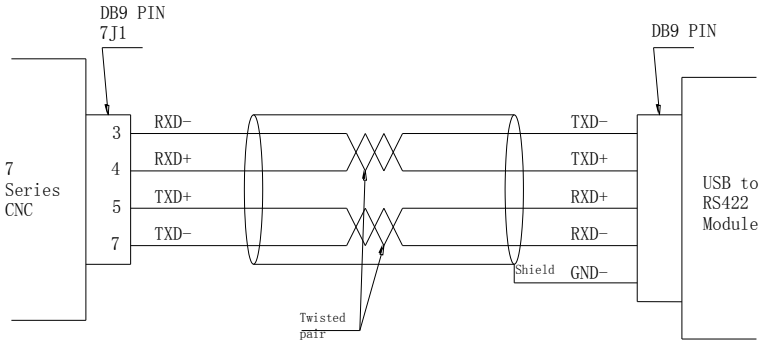


F4.38 7J1(DB9 PIN)

Table 4.14 RS485 7J1

PIN	Function define	PIN	Function define
01		06	
02		07	TXD-
03	RXD#	08	
04	RXD+	09	
05	TXD+		

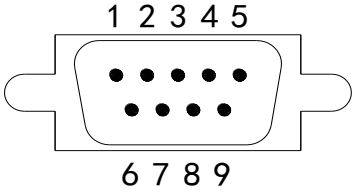
Description: long-distance transmission lines must use shielded twisted-pair cable as the signal transmission, TXD + and TXD-a pair of twisted-pair transmission, RXD + and RXD-with a pair of twisted-pair transmission.



F4.39 RS485 Interface Connection Diagram

4.5.2 WA710MI Serial communication interface

WA710MI Serial communication interface 7J1 is 'DB9 pin' socket, the corresponding plug for 'DB9 hole', Using RS232 level protocol, used between PC machine or system transmission procedures, interface definitions (not standard pin is empty):



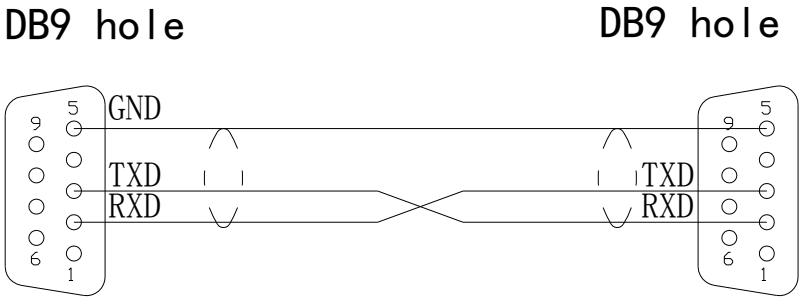
F4.40 7J1(DB9 PIN)

Table 4.15 Communication interface 7J1

PIN	Function define	PIN	Function define
01		06	
02	RXD	07	
03	TXD	08	

04		09	
05	GND		

With the two core shielded wire must be communication lines, and the shielding layer for wire length, ≤ 10M, according to figure 4.41 production:



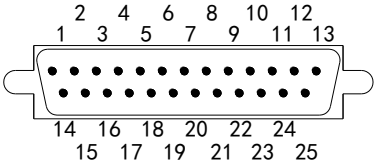
F4.41

4.6 Input and output interface

4.6.1 WA710MD/WA720M-4/WA740M-5/WA730M-6 Input and output interface

4.6.1.1 WA710MD/WA720M-4/WA740M-5/WA730M-6 Input and output interface

WA710MD/WA720M-4/WA740M-5/WA730M-6 Input Interface 7J2 'DB25 pin' socket，Corresponding to 'DB25 hole'. The interface has 23 input signals,Recommended external contact switch, proximity switches or Hall device, when you enter the proximity switch (or Hall), the requirement when it does not signal output device is terminated, when the signal output for low level and the output drive capability requirements for low level than 15mA, proposed to adopt power supplies DC10~30V range of devices.



F4.42 Input Interface 7J2（DB25 Pin）

Table 4.16 Input Interface 7J2

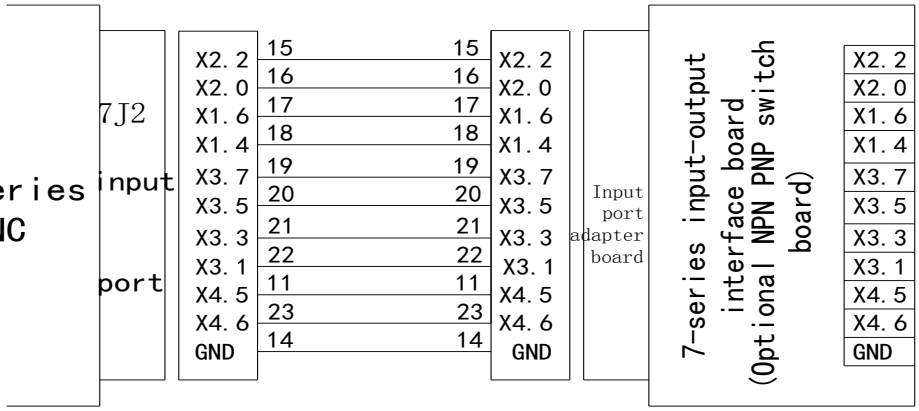
PIN	Function define	PIN	Function define
01	ground	14	ground
02	x2.3	15	x2.2（T1）
03	x0.7(Clamped in place)	16	x2.0（T2）
04	x0.6(Loosen the place)	17	x1.6（T3）
05	X6.7	18	x1.4（T4）
06	x4.0（Is limit）	19	x3.7（T5）
07	x3.6（Negative limit）	20	x3.5（T6）
08	x3.4（XSlow down）	21	x3.3（T7）
09	x3.2（YSlow down）	22	x3.1（T8）
10	x6.2（ZSlow down）	23	x4.6
11	x4.5	24	x4.4(Clamping input)

12	x4.3（Loosen input）	25	x4.2 (Clamped in place)
13	x4.1）（Loosen place）		

Description:

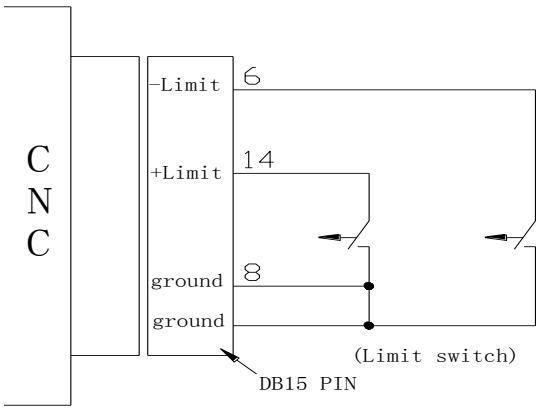
(1) All inputs are NPN type 0V effective, such as external devices for the PNP-24V with a valid need to add 7-series CNC system input and output interface board (PNP switch NPN type), to provide 10-way signaling NPN PNP signal (x2. 2, x2.0, x1.6, x1.4, x3.7, x3.5, x3.3, x3.1, x4.5, x4.6) Figure 4.43.

(2) the function in brackets in the table to the factory default, users can set their own PLC ladder programming through its pin functions.



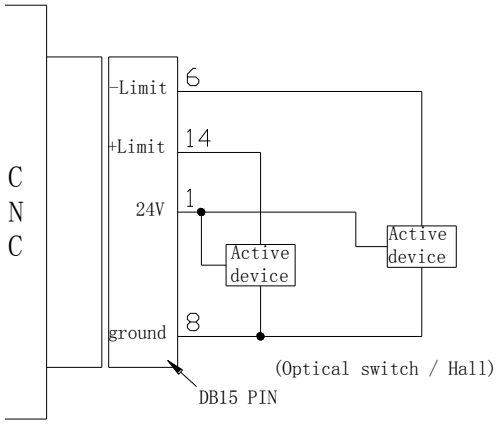
F4.43

Typical input signal connection is shown in Figure 4.44



(a) Mechanical contact switch with a typical input signal connection.

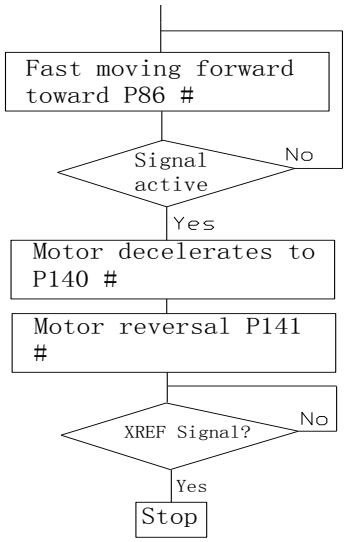
With active devices (such as proximity switches):



(b) With active devices (such as proximity switches) Typical input signal connection.

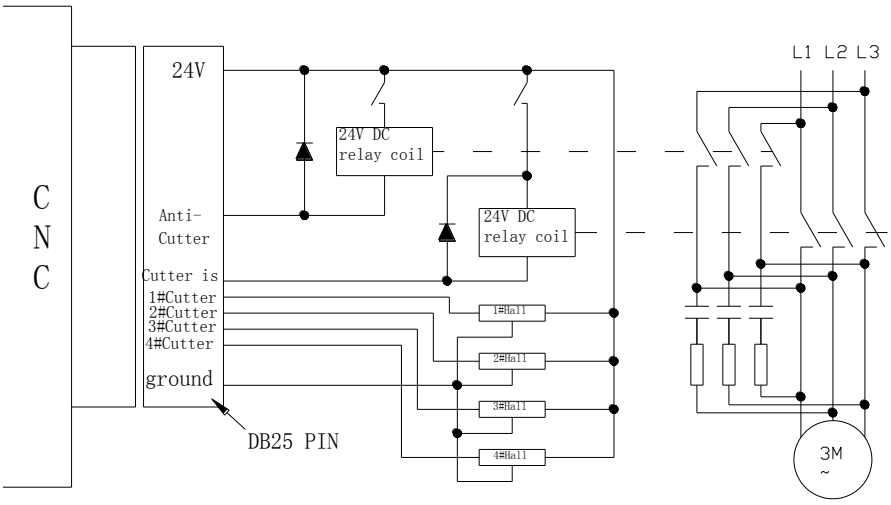
F4.44

Returns to machine origin (machine zero) action sequence:



F4.45

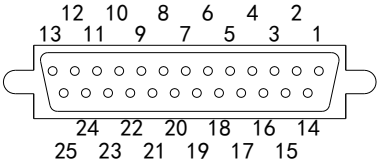
When the system power supply external wiring as shown in Figure 4-47 (with four-position, for example, six or eight-position, heartbeat, you need to set the remaining cutters access system.)



F4.46

4.6.1.2 WA710MD/WA720M-4/WA740M-5/WA730M-6 output interface

WA710MD/WA720M-4/WA740M-5/WA730M-6 output interface 7J3 is 'DB25 hole' sockets, plugs should be 'DB25 pin', the interface has 22 output signals, OC gate output current of 70mA or less restricted, limiting the voltage below 30V, the output load exceeds the current limit or restrict the voltage should be used to drive the load relay to avoid damage to the system.



F4.47 Output interface 7J3（DB25 hole）

Table 4.17 Output port 7J3

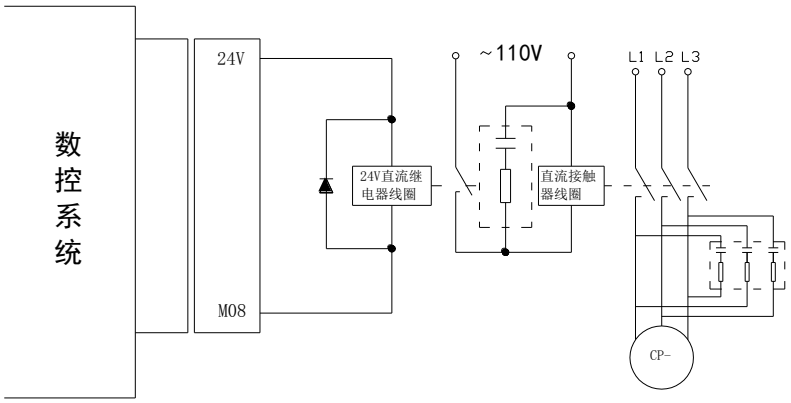
PIN	Function define	PIN	Function define
01	ground	14	ground
02	y4.7（Cooling）	15	y4.4（Brake 1）
03	y4.3（Brake 2）	16	y4.2
04	y4.1	17	y4.0
05	y2.7（Cutter is）	18	y2.6（Clamp）
06	y2.5（Anti-Cutter）	19	y2.4（Release）
07	y2.3（Lubricating）	20	y2.2（Tailstock into）
08	y2.1（Tailstockback）	21	y2.0（Light 1）
09	y0.2（Light 2）	22	y0.3
10	y0.4	23	y0.5
11	y0.6	24	y0.7
12	y0.1	25	+24V
13	＋24V		

Description:

(1) Output of a single power point output (OC Gate), limiting current of 70mA, external inductive load (such as DC relays, etc.) need additional freewheeling diode.

(2) The function in brackets in the table to the factory default, users can set their own PLC ladder programming through its pin functions.

Typical output signal connection is shown in Figure 4.48 (in M08, for example):

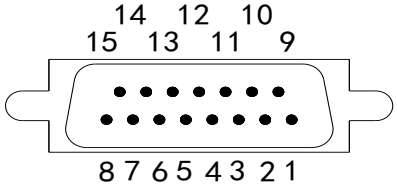


F4.48 Typical output signal connection

4.6.2 WA710MI Input and output interface

4.6.2.1 Tool interface

WA710MI tool interface 5J1 is' DB15 pin socket, plug 'corresponding' DB15 hole ', interface definition below (not standard pin is empty):



F4.49 5J1(DB15 PIN)

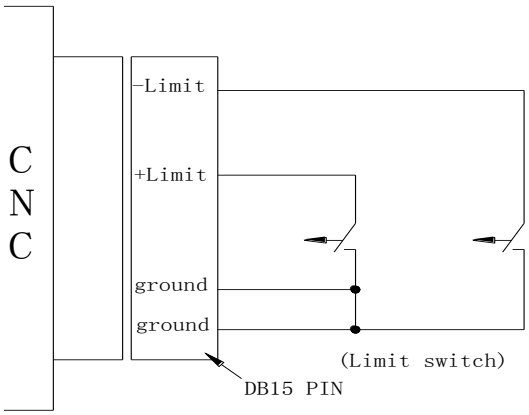
Table 4.18 Communication Interface 5J1

PIN	Function define	PIN	Function define
01	GND	09	GND
02	x2.0（3#cutter）	10	x2.6（7#cutter）
03	x2.4（4#cutter）	11	x2.3（1#cutter）
04	x2.5（5#cutter）	12	x2.2（8#cutter）
05	x2.7（2#cutter）	13	x2.1（6#cutter）
06	y0.3（Tool Forward）	14	
07	y0.7（Tool Reverse）	15	
08	+24V		

Note：Tool Forward，Tool Reverse is Single power point (OC gate) output current limit is 0.5A, external inductive loads (such as DC relay) is required plus freewheeling diode.

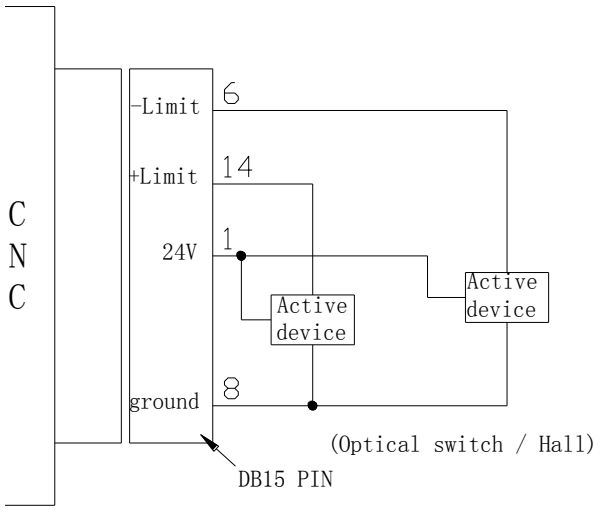
All input ports are NPN type 0V effective, such as an external device is a PNP 24V valid need extra NPN PNP switch signal conversion board.

Input signal typical connection as shown in F4.50:



(a) Mechanical contact switch with a typical input signal connection.

With active devices (such as proximity switches):

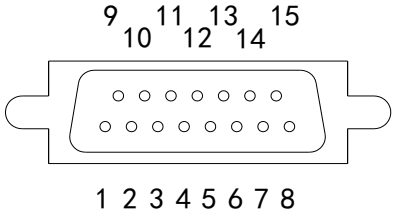


(b) With active devices (such as proximity switches) Typical input signal connection

F4.50 Typical input signal connection

4.6.2.2 WA710MI input / output interface 5J2

WA710MI input / output interface 5J2 is 'DB15 hole' sockets, plugs should be 'DB15 pin'. The interface has 6-way power driver output signal relay and five input signals. The input signal is recommended that the external use of contact switches, proximity switches or Hall device, when the input using the proximity switch (or Hall device), the request signal is not output when the device is high, the output signal is low level, its output low drive capability is greater than 15mA, suggested the use of the power range DC10 ~ 30V devices.



F4.51 5J2(DB15 hole)

Table 4.19 Input / output interface 5J2

PIN	Function define	PIN	Function define
01	+24V	09	x4.4(M10FIN)
02	x4.1（IM10）	10	y0.2（M10）
03	y0.0（M11）	11	y0.4（M79）
04	y0.1（M78）	12	y0.5（M77）
05	y0.6（cooling）	13	x0.4（M11FIN）
06	x0.3（-limit）	14	x0.2（+limit）
07	x4.6	15	x3.7
08	GND		

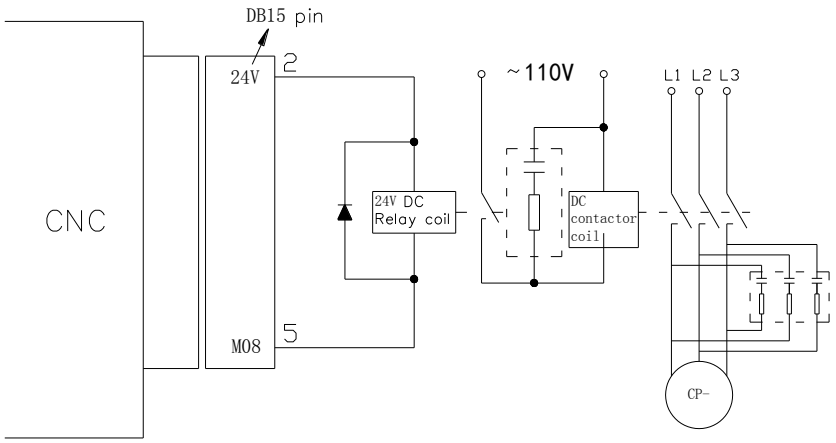
Description:

All output is a single power point output (OC door), limiting current of 70mA, external inductive loads (such as DC relays, etc.) need to add freewheeling diode.

Function in brackets in the table to the factory default feature, the user can set its

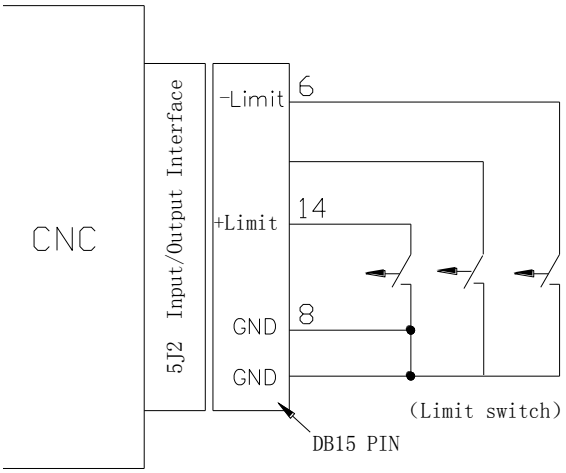
through PLC ladder programming pin functions.

A typical connection of the output signal as shown in Figure 4.52 (in M08 as an example):



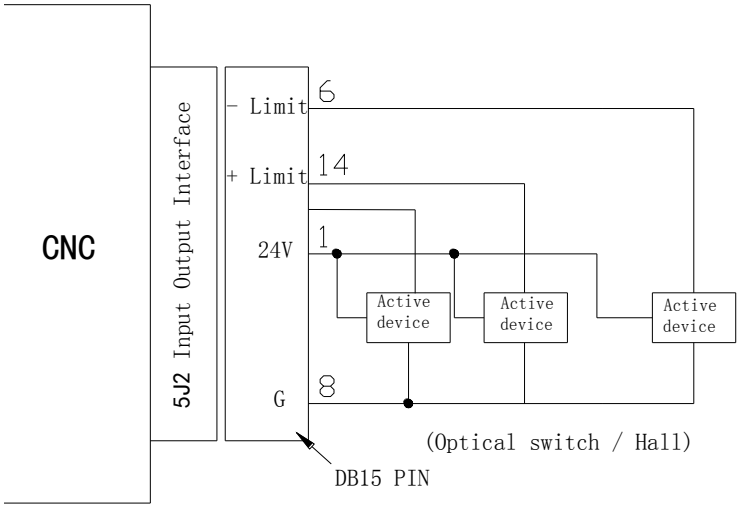
F4.52 A typical connection of the output signal

Typical input signal as shown in Figure 4.53 Connection:



(a) The input signal with a typical mechanical contact switch connection

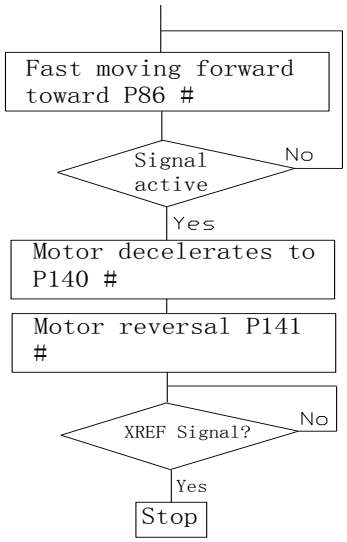
With active devices (such as proximity switches):



(b) With an active device (such as proximity switches) of a typical connection of the input signal

F4.53 Typical Input signal connection

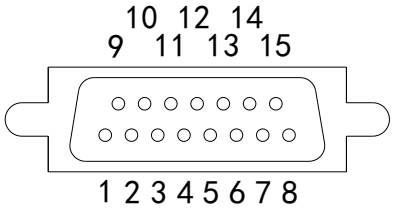
Returns to machine origin (machine zero) action sequence:



F4.54

4.7 Panel interfaces

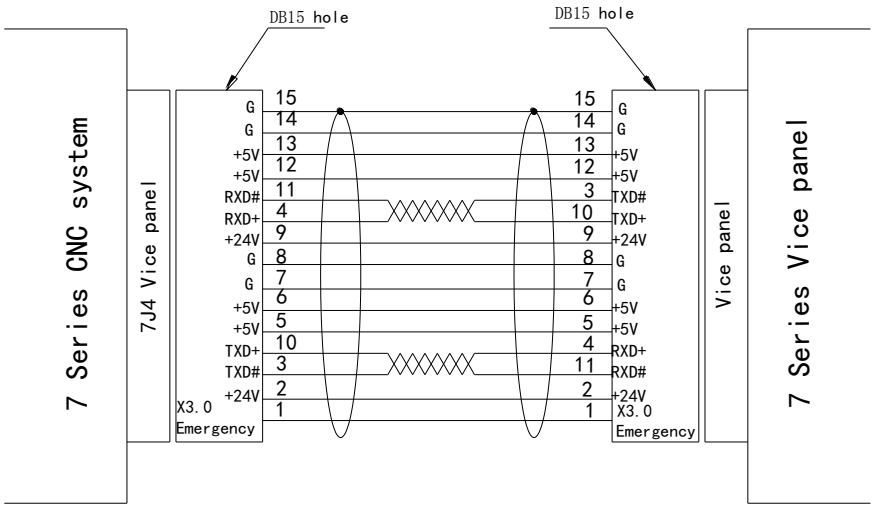
4.7.1 WA710MD/WA720M-4/WA740M-5/WA730M-6 panel interfaces 7J4



F4.55 Panel interfaces 7J4 (DB15 hole)

Table4.20 Panel interfaces 7J4

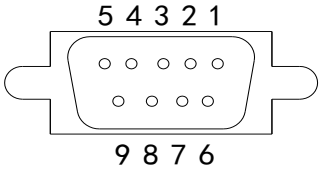
PIN	Function define	PIN	Function define
01	x3.0(Emergency Stop)	09	+24V
02	+24V	10	TXD+
03	TXD#	11	RXD#
04	RXD+	12	+5V
05	+5V	13	+5V
06	+5V	14	ground
07	ground	15	ground
08	ground		



F4.56

4.7.2 WA710MI Side panel switch interface 5J3

WA710MI Side panel switch interface 5J3 model for the "DB9 hole", connected to the plug should be "DB9 pin". Defined as follows:

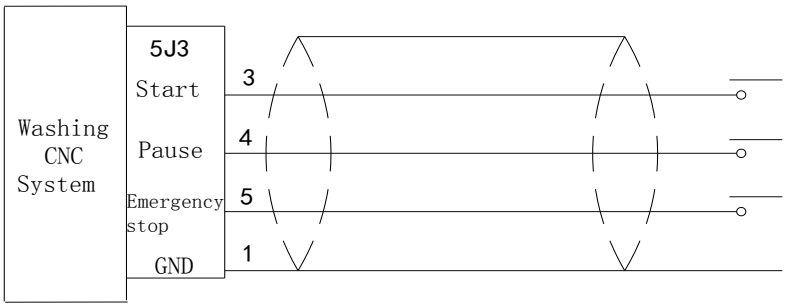


F4.57 Side panel switch interface 5J3（DB9 hole）

Table4.21 Side panel switch interface 5J3

PIN	Function define	PIN	Function define
01	GND	06	GND
02	GND	07	X3.4（three position switch 1）
03	X0.5（Start）	08	X3.6（three position switch 2）
04	X0.6（Pause）	09	X4.3（Emergency stop）
05	X0.7（Emergency stop）		

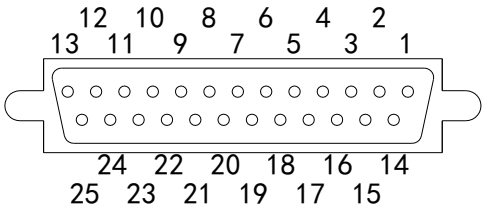
As shown below:



F4.58

4.7.3 WA710MD alternate vice-panel Spare Switch Interface

This alternate Vice-panel switch interface 'DB25 hole':



F4.59 WA710MD alternate Vice-panel switch interface（DB25 hole）

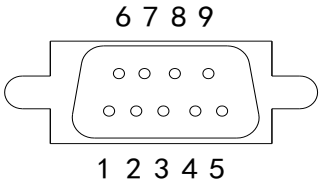
Table4.22 WA710MD alternate Vice-panel switch interface

PIN	Function define	PIN	Function define
01	+5V	14	+5V
02	+5V	15	+5V
03	Emergency ground	16	Ground
04	Ground	17	Ground
05	alternate input	18	Pause
06	Start	19	Three start
07	Three pause	20	Program Lock
08	Band switch G	21	Band switch F
09	Band switch E	22	Band switch C
10	Band switch B	23	Band switch A
11	Emergency stop	24	B- hand pulse
12	B+hand pulse	25	A+ hand pulse
13	A- hand pulse		

4.8 Spindle Interface

4.8.1 WA710MD/WA720M-4/WA740M-5/WA730M-6 spindle Interface

WA710MD/WA720M-4/WA740M-5/WA730M-6 interface model for the "DB9 hole", connected to the plug should be "DB9 pin". Defined as follows:



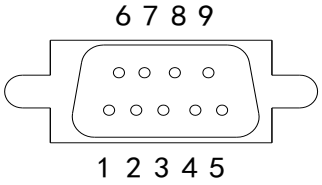
F4.60 Spindle Interface 8J1(DB9 hole)

Table4.23 Spindle Interface 8J1

PIN	Function define	PIN	Function define
01	DA	06	y6.6(M03)
02	y6.6(M03)	07	
03	y6.5(M04)	08	y6.5(M04)
04	y6.4(M05)	09	ground
05	ground		

4.8.2 WA710MI spindle Interface

WA710MI interface model for the "DB9 hole", connected to the plug should be "DB9 pin". Defined as follows:



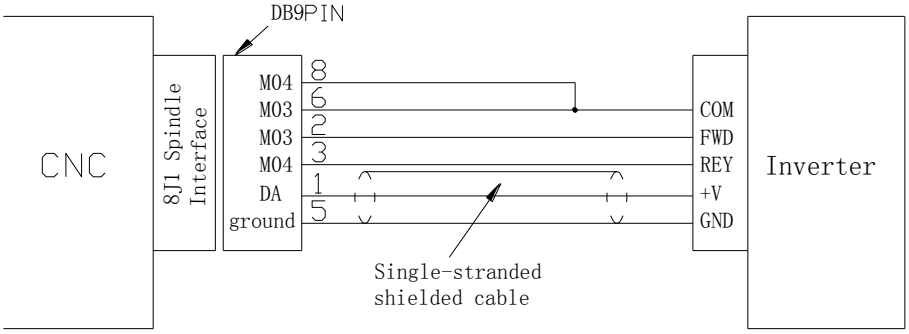
F4.61 spindle Interface 8J1(DB9 hole)

Table4.24 Spindle Interface 8J1

PIN	Function define	PIN	Function define
01	DA	06	y4.1(M03)
02	y4.1(M03)	07	
03	y4.0(M04)	08	y4.0(M04)
04	y2.6(M05)	09	GND
05	GND		

Description:M03, M04 as a relay contact output, correspond to the feet of No. M03: P2/P6, M04: P3/P8, the contacts are connected voltage ≤ 36V, current ≤ 500mA, M05 70mA current limit for power output below the limit voltage of 30V or less. Analog (DA) output 0 ~ 10V analog voltage, then the drive or the spindle servo drive. Analog signals in the system and interconnect. The line must be shielded with a single line, ground the shield. System factory set to 0 ~ +10 V, it can be controlled with contact frequency signal with forward, reverse and variable speed. Required peripherals (drives) to absorb current <5mA.

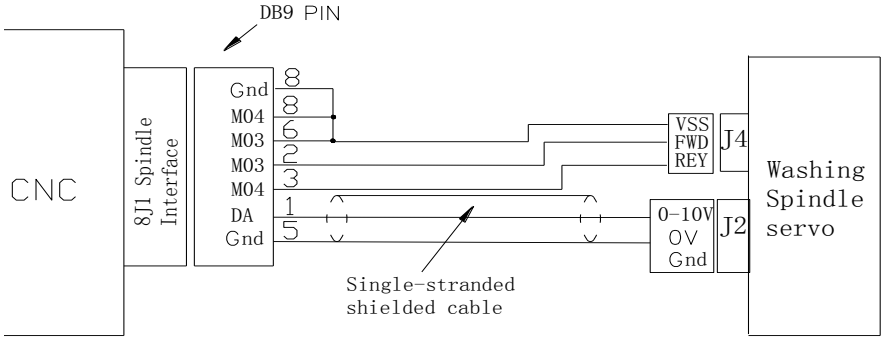
The interface connected to the drive as shown in Figure 4.62：



F4.62

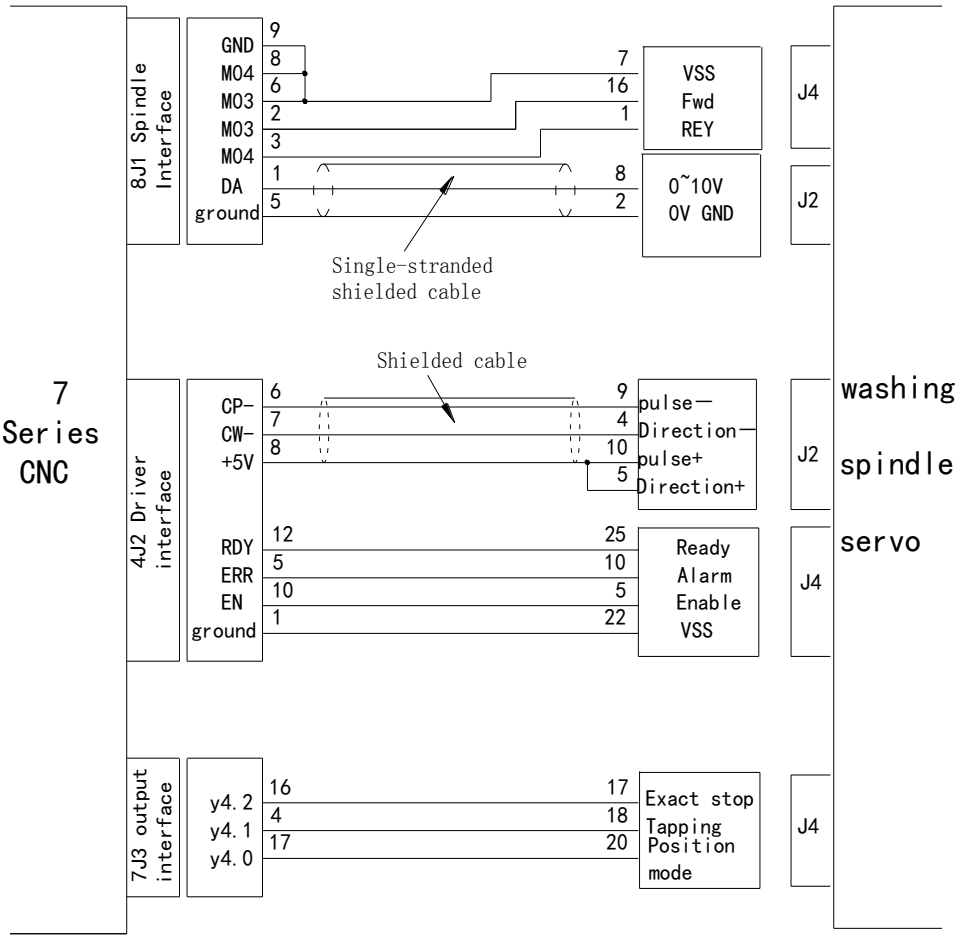
The interface with Washing spindle servo connections:

Work in speed mode is shown in Figure 4.63:



F4.63

Work in the position and velocity model shown in Figure 4.64:



F4.64

Appendix I Error number and error content

Error number and error content:

Error number	Error content.
01	G04 time error
02	Undefine K/L parameter
03	G24 subprogram return error
04	G31 zoom in overflow
05	Trim data error
06	Tool number or datum error
07	Illegal G,M function
08	Jump call or subprogram error
09	Helix data error
10	Charactor of line home error
11	Not G54 when return reference
12	Not storage workpiece Coor
13	Data format error
14	Can't find end of subprogram or XU/ZW
15	Boundary error of soft limitation
16	Tool offset error
17	Plane error of tool offset
18	Unable to identify feed direction for G35
19	No Use G01 when tool offset setup
20	M00,M02,M30 in the same segment
21	Called by G20 be illega
22	Error at Macro format or executing
23	Pitch too big or small
24	Segment number can't be found
25	CNC or servo driver not ready error
26	Jump call in the last line
27	Too few parameters in G02/G03
28	Undefined
29	The trace of tool offset is zero
30	No exec. when operating
31	RS232 error
32	File not found or file error
33	Undefined
34	Not enough memory

35	Undefined
36	Pitch error compensation distance=0
37	Undefined
38	Undefined
39	Parameter error of G,M function
40	Limit to feed
41	Servo driver error
42	General error
43	Pause needed at tool offset regulation,change B228=1
44	Time over to check tool
45	Illegal filename
46	Undefined
47	Too big tool offset regulation
48	File be destroyed
49	File name error
50	Arc radius not be congruence
51	Closed arc can't use R
52	Undefined
53	Not find segment number
54	Exact stop haven't done
55	Emergency stop
56	Zero feed
57	Syntax error for G54~G59
58	Not enough memory
59	Spindle start error
60	Punch signal error
61	Undefined
62	Undefined
63	Undefined
64	Undefined
65	Macro program not defined(190#=0)
66	Hasn't referrence
67	Tool or offset No. error when return reference
68	No signal of spindle clamp
69	Run time error
70	More null file lines than necessary
71	Circle start key pushed
72	Acceleration of thread is zero
73	Length of thread is too short
74	Spindle not even
75	Not detected encoder signal when thread
76	Undefined
77	PLC Alarm

78	Arc interpolation error
79	Tool exchange sub-program NOT found error
80	I/O undefined
81	The first time to use flash
82	Max workpiece met
83	Function undefined
84	Read/write file error
85	Error during writing flash disk
86	Read disk error
87	RS232 Error
88	RS232 Error when input
89	RS232 Error when input
90	Alarm still active, press RESET key to cancel
91	RESET
92	Read USB error
93	USB error
94	Servo error when enable
95	No use USB when operating
97	Program checked OK
98	C compensation error
99	Trial setting error

Appendix II System parameter

Parameter number	Scope	Factory set	Recommend scope
00	Accelerate and decelerate time of G00(ms)	150	50～1000
01	Reserved	0	0
02	Amount of tools	4	1～12
03	Maximum rotation rate of constant linear speed cutting	4000	100～8000
04	Minimum rotation rate of constant linear speed cutting	500	100～8000
05	=0 row tool;=1 electric tool;2-10 specific	1	0～10
06	X-axis clearance	0	0～32
07	Y-axis clearance	0	0～32
08	Z-axis clearance	0	0～32
09	A-axis clearance	0	0～32
10	B-axis clearance	0	0～32
11	C-axis clearance	0	0～32
12	Maximum speed of jog feed(mm/min)	5000	0～15000
13	Maximum rotation rate of spindle 1th rank	3000	100～8000
14	Maximum rotation rate of spindle 2th rank	1000	100～8000
15	Maximum rotation rate of spindle 3th rank	2000	100～8000
16	Maximum rotation rate of spindle 4th rank	2000	100～8000
17	Startup screen:=0 version;=1 custom;=2 operate screen	0	0～2
18	Maximum feedrate of cutting	6000	0～60000
19	Accelerate and decelerate time of cutting feed (ms)	150	50～1000
20	Line number of spindle encoder	1200	700～4800
21	Maximum feedrate of cutting screw in Z axial	6000	2000～4800
22	Accelerate and decelerate time of cutting screw in Z axial (ms)	150	50～1000
23	Maximum feedrate of compensate clearance	5000	1000-10000
24	Accelerate and decelerate time in handwheel feed mode	60	60～1000
25	Slice depth of smooth screw	0	0～65
26	Most percent of spindle rotation ripple rate	10	0～15
27	X-axis Maximum feedrate of G00(mm/min)	6000	0～60000
28	Y-axis Maximum feedrate of G00(mm/min)	6000	0～60000
29	Z-axis Maximum feedrate of G00(mm/min)	6000	0～60000

30	A-axis Maximum feedrate of G00(mm/min)	6000	0～60000
31	B-axis Maximum feedrate of G00(mm/min)	6000	0～60000
32	C-axis Maximum feedrate of G00(mm/min)	6000	0～60000
33	X-axis electronic gear multiplicator	1	0～1000
34	X-axis electronic gear divisor	2	0～1000
35	Y-axis electronic gear multiplicator	1	0～1000
36	Y-axis electronic gear divisor	1	0～1000
37	Z-axis electronic gear multiplicator	1	0～1000
38	Z-axis electronic gear divisor	1	0～1000
39	A-axis electronic gear multiplicator	1	0～1000
40	A-axis electronic gear divisor	1	0～1000
41	B-axis electronic gear multiplicator	1	0～1000
42	B-axis electronic gear divisor	1	0～1000
43	C-axis electronic gear multiplicator	1	0～1000
44	C-axis electronic gear divisor	1	0～1000
45	Accelerate and decelerate time while position limitation (ms)	150	50～1000
46	Number of detect input signal to eliminate dithering	2	3～30
47	Number of detect short input signal to eliminate dithering	1	3～30
48	Maximum feedrate in X direct of G00 screw	5000	1000～60000
49	Decelerate time of G00 screw tail in X axial	400	50～1000
50	Maximum feedrate in X direct of cutting screw	2000	0～60000
51	Maximal feedrate of cutting screw tail in X axial	6000	60～60000
52	Decelerate time of cutting screw tail in X axial	300	0～600
53	Reserved	0	100～300
54	Reserved	0	100～300
55	auto increment of line number when editing	10	0～100
56	Reserved	0	0～10
57	Reserved	0	0～10
58	Reserved	0	0～10
59	Motor encoder lines PWM output 0:2500; 1:5000 (715TA no such function)	0	0～60000
60	Soft limit position of X+ relative to workpiece	0	0～60000
61	Soft limit position of X- relative to workpiece	0	-60000～0
62	Soft limit position of Y+ relative to workpiece	0	0～60000
63	Soft limit position of Y- relative to workpiece	0	-60000～0
64	Soft limit position of Z+ relative to workpiece	0	0～60000
65	Soft limit position of Z- relative to workpiece	0	-60000～0
66	Soft limit position of A+ relative to workpiece	0	0～60000
67	Soft limit position of A- relative to workpiece	0	-60000～0
68	Soft limit position of B+ relative to workpiece	0	0～60000

69	Soft limit position of B- relative to workpiece	0	-60000～0
70	Soft limit position of C+ relative to workpiece	0	0～60000
71	Soft limit position of C- relative to workpiece	0	-60000～0
72	Input port number to locate X axis raw reference point (1.6=X001.6) （715TA Factory value 3.7）	3.4	0～40
73	Input port number to locate Y axis raw reference point（715TA Factory value 0）	3.2	0～40
74	Input port number to locate Z axis raw reference point（715TA Factory value 4.6）	6.2	0～40
75	Input port to locate A axis precision reference point	4.6	0～40
76	Input port to locate B axis precision reference point	4.5	0～40
77	Input port to locate C axis precision reference point	0.0	0～40
78	Max time for wait input	20	0～5000
79	G35 input X table location, such as X11.5: 79 # = 11.5	12.5	0～9999
80	Constant parameter KP(5-100) of servo position loop	0	0～50
81	Constant parameter KI(0-50) of servo position loop	80	5～100
82	Constant parameter KD(0-50) of servo position loop	0	0～50
83	Delay time of spindle reverse rotation when tapping	0	0～65
84	NO.1 angle of screw tooth by user defined	0	0～180
85	NO.2 angle of screw tooth by user defined	0	0～180
86	Referencing rate for all axes,see 130#~133# if 86#=0	4000	0～60000
87	Compensation length per millimeter when tapping(um)	0	0～15
88	Feedrate of cutting screw in X-axis	3000	1000～5000
89	Delay time of G09	20.000	0～65
90	Cutting speed limit	50	50～500
91	Coordinate display mode	0	0～3
92	Reserved	1	0～60000
93	Feedrate for dry run	200	0～60000
94	Reserved	0	0～60000
95	Reserved	0	0～60000
96	Motor's max RPM at pwm output mode	200	0～60000
97	Angle offset for M28 spindle positioning	0	0～60000

98	Reserved(715TA Hand pulse cutting feed rate is the percentage of acceleration, factory 30)	0	0～60000
99	Reserved	0	0～60000
100	X refrencing rate if 86#=0	4000	0～60000
101	Y refrencing rate if 86#=0	4000	0～60000
102	Z refrencing rate if 86#=0	4000	0～60000
103	A refrencing rate if 86#=0	4000	0～60000
104	B refrencing rate if 86#=0	4000	0～60000
105	C refrencing rate if 86#=0	4000	0～60000
106	Delay time of G83	0	0～60000
107	Max count of operation	0	1～10000
108	Reserved	0	0～60000
109	Max feed rate for PWM output	0	0～60000
110	Interval of screwpitch compensate in X axial	0	0～60000
111	Interval of screwpitch compensate in Y axial	0	0～60000
112	Interval of screwpitch compensate in Z axial	0	0～60000
113	Interval of screwpitch compensate in A axial	0	0～60000
114	Interval of screwpitch compensate in B axial	0	0～60000
115	Interval of screwpitch compensate in C axial	0	0～60000
116	Total number of compensate point in X axial	0	0～239
117	Total number of compensate point in Y axial	0	0～239
118	Total number of compensate point in Z axial	0	0～239
119	Total number of compensate point in A axial	0	0～239
120	Total number of compensate point in B axial	0	0～239
121	Total number of compensate point in C axial	0	0～239
122	Display mode in OPERT	0	0～239
123	Reserved	0	0～239
124	Reserved	0	0～239
125	Reserved	0	0～239
126	Reserved	0	0～239
127	Reserved	0	0～239
128	Reserved	0	0～239
129	Reserved	0	0～239
130	Reserved	0	0～239
131	Reserved	0	0～239
132	Reserved	0	0～239
133	Reserved	0	0～239
134	Reserved	1	1～100
135	Reserved	1	1～100
136	Reserved	1	1～100
137	Reserved	1	1～100
138	Reserved	1	1～100
139	Reserved	1	1～100

140	Feedrate of X-Axis after decelerate switch	200	0～60000
141	Feedrate of X-Axis when locate machine zero	50	0～60000
142	Feedrate of Y-Axis after decelerate switch	200	0～60000
143	Feedrate of Y-Axis when locate machine zero	50	0～60000
144	Feedrate of Z-Axis after decelerate switch	200	0～60000
145	Feedrate of Z-Axis when locate machine zero	50	0～60000
146	Feedrate of A-Axis after decelerate switch	200	0～60000
147	Feedrate of A-Axis when locate machine zero	50	0～60000
148	Feedrate of B-Axis after decelerate switch	200	0～60000
149	Feedrate of B-Axis when locate machine zero	50	0～60000
150	Feedrate of C-Axis after decelerate switch	200	0～60000
151	Feedrate of C-Axis when locate machine zero	50	0～60000
152	Reserved	0	0～60000
153	Reserved	0	0～60000
154	Reserved	0	0～60000
155	Reserved	0	0～60000
156	Reserved	0	0～60000
157	Reserved	0	0～60000
158	Reserved	0	0～60000
159	Reserved	0	0～60000
160	Reserved	0	0～60000
161	Reserved	0	0～60000
162	Reserved	0	0～60000
163	Reserved	0	0～60000
164	Reserved	0	0～60000
165	Reserved	0	0～60000
166	Reserved	0	0～60000
167	Reserved	0	0～60000
168	Reserved	0	0～60000
169	Reserved	0	0～60000
170	Program name called by M06	0	0～60000
171	Reserved	0	0～60000
172	Reserved	0	0～60000
173	Reserved	0	0～60000
174	Reserved	0	0～60000
175	Reserved	0	0～60000
176	Reserved	0	0～60000
177	Reserved	0	0～60000
178	Reserved	0	0～60000
179	Reserved	0	0～60000
180	Reserved	0	0～60000
181	Reserved	0	0～60000
182	Reserved	0	0～60000

183	Reserved	0	0～60000
184	Reserved	0	0～60000
185	Reserved	0	0～60000
186	Reserved	0	0～60000
187	Reserved	0	0～60000
188	Reserved	0	0～60000
189	Reserved	0	0～60000
190	Reserved	0	0～60000
191	Reserved	0	0～60000
192	Reserved	0	0～60000
193	Reserved	0	0～60000
194	Reserved	0	0～60000
195	Reserved	0	0～60000
196	Reserved	0	0～60000
197	Reserved	0	0～60000
198	Reserved	0	0～60000
199	Reserved	0	0～60000

Appendix III Bit parameter

00#

#7	#6	#5	#4	#3	#2	#1	#0
B007	B006	B005	B004	B003	B002	B001	B000

B000: =1: Enable soft position limit
B001: =1: The soft limit needs not return to home
B002: =1: Soft limit decides by machine coordinates; =0:Workp
B003: Reserved
B004: =1: Enable Y-axis on lathe system
B005: =1: Insert M02 at the end of the program when executing
B006: =1: Milling machine system; =0: Lathe system
B007: =1: Radius specification; =0: Diameter specification
00# factory settting:11000100

01#

#7	#6	#5	#4	#3	#2	#1	#0
B017	B016	B015	B014	B013	B012	B011	B010

B010: =1: Clear workpiece counter when power on
B011: =1: Clear workpiece timer when power on
B012: =1: Zeroing cutting time at each cycle start
B013: Reserved
B014: =1: Recover spindle speed when tapping finished; =0: Don't
B015: Reserved
B016: Reserved
B017: Reserved
01# factory setting:01000000

02#

#7	#6	#5	#4	#3	#2	#1	#0
B027	B026	B025	B024	B023	B022	B021	B020

B020: =1: Enable clearance compensation for X-Axis
B021: =1: Enable clearance compensation for Y-Axis
B022: =1: Enable clearance compensation for Z-Axis
B023: =1: Enable clearance compensation for A-Axis
B024: =1: Enable clearance compensation for B-Axis
B025: =1: Enable clearance compensation for C-Axis
B026: Reserved
B027: Reserved
02# factory setting: 00000000

03#

#7	#6	#5	#4	#3	#2	#1	#0
B037	B036	B035	B034	B033	B032	B031	B030

B030: =1: Enable auto return zero point function; =0:Disable
B031: Reserved
B032: Reserved
B033: =1: No soft limit at refrencing
B034: =1: Don't recover workpiece coordinates after home
B035: =1: Need not return reference; =0:Need return reference
B036: Reserved
B037: =1: CNC output ENABLE signal to servo driver(contact output)
03# factory setting:10110111

04#

#7	#6	#5	#4	#3	#2	#1	#0
B047	B046	B045	B044	B043	B042	B041	B040

B040: Reserved
B041: Reserved
B042: =1: Normal tool modify direction
B043: =1: Recover spindle axis D/A when G97 finish
B044: Reserved
B045: Reserved
B046: Reserved
B047: =1: Accelerate key mode output; =0:Not mode
04# factory setting: 00000010

05#

#7	#6	#5	#4	#3	#2	#1	#0
B057	B056	B055	B054	B053	B052	B051	B050

B050: =1: Enable the electric gear for X-Axis; =0:Disable
B051: =1: Enable the electric gear for Y-Axis; =0:Disable
B052: =1: Enable the electric gear for Z-Axis; =0:Disable
B053: =1: Enable the electric gear for A-Axis; =0:Disable
B054: =1: Enable the electric gear for B-Axis; =0:Disable
B055: =1: Enable the electric gear for C-Axis; =0:Disable
B056: Reserved
B057: Reserved
05# factory setting:00001111

06#

#7	#6	#5	#4	#3	#2	#1	#0
B067	B066	B065	B064	B063	B062	B061	B060

B060: =1: Enable pitch error compensation for X-Asix. =0: Disable
B061: =1: Enable pitch error compensation for Y-Asix. =0: Disable

B062: =1: Enable pitch error compensation for Z-Asix. =0:Disable
B063: =1: Enable pitch error compensation for A-Asix. =0:Disable
B064: =1: Enable pitch error compensation for B-Asix. =0:Disable
B065: =1: Enable pitch error compensation for C-Asix. =0:Disable
B066: Reserved
B067: Reserved
06# factory setting:00000000

07#

#7	#6	#5	#4	#3	#2	#1	#0
B077	B076	B075	B074	B073	B072	B071	B070

B070: =1: X-Axis return machine zero in Negative direction
B071: =1: Y-Axis return machine zero in Negative direction
B072: =1: Z-Axis return machine zero in Negative direction
B073: =1: A-Axis return machine zero in Negative direction
B074: =1: B-Axis return machine zero in Negative direction
B075: =1: C-Axis return machine zero in Negative direction
B076: Reserved
B077: Reserved
07# factory setting:00000000

08#

#7	#6	#5	#4	#3	#2	#1	#0
B087	B086	B085	B084	B083	B082	B081	B080

B080: =1: Motor of X-Axis move in the reverse direction
B081: =1: Motor of Y-Axis move in the reverse direction
B082: =1: Motor of Z-Axis move in the reverse direction
B083: =1: Motor of A-Axis move in the reverse direction
B084: =1: Motor of B-Axis move in the reverse direction
B085: =1: Motor of C-Axis move in the reverse direction
B086: Reserved
B087: Reserved
08# factory setting: 00000000

09#

#7	#6	#5	#4	#3	#2	#1	#0
B097	B096	B095	B094	B093	B092	B091	B090

B090: =1: Detect READY of servo of X-Axis when power-up
B091: =1: Detect READY of servo of Y-Axis when power-up
B092: =1: Detect READY of servo of Z-Axis when power-up
B093: =1: Detect READY of servo of A-Axis when power-up
B094: =1: Detect READY of servo of B-Axis when power-up
B095: =1: Detect READY of servo of C-Axis when power-up
B096: Reserved

B097: =1: Enable soft screw operate; =0: Disable

09# factory setting:11000000

10#

#7	#6	#5	#4	#3	#2	#1	#0
B107	B106	B105	B104	B103	B102	B101	B100

B100: =1: Handy manual handle effective

B101: =1: Display macro for tool exchange

B102: Reserved

B103: =1: External axis sel; =0: Axis sel. from pannel

B104: Reserved

B105: =1: -10V~+10V analog output; =0: 0~10V analog output

B106: =1: Speed down between; =0: Continue contour between tracks

B107: =1: High definition method when screw; =0: Low vibrating mode

10# factory setting: 10000001

11#

#7	#6	#5	#4	#3	#2	#1	#0
B117	B116	B115	B114	B113	B112	B111	B110

B110: =0: Index pulse as X-REF; =1: External switch as X-REF

B111: =0: Index pulse as Y-REF; =1: External switch as Y-REF

B112: =0: Index pulse as Z-REF; =1: External switch as Z-REF

B113: =0: Index pulse as A-REF; =1: External switch as A-REF

B114: =0: Index pulse as B-REF; =1: External switch as B-REF

B115: =0: Index pulse as C-REF; =1: External switch as C-REF

B116: =1: Set F1.0 to PLC at emergency stop; =0: NOT set

B117: =1: Spinning axis directly go to refrence

11# factory setting: 00000000

12#

#7	#6	#5	#4	#3	#2	#1	#0
B127	B126	B125	B124	B123	B122	B121	B120

B120: =1: Enable X driver error detection; =0: Disable

B121: =1: Enable Y driver error detection; =0: Disable

B122: =1: Enable Z driver error detection; =0: Disable

B123: =1: Enable A driver error detection; =0: Disable

B124: =1: Enable B driver error detection; =0: Disable

B125: =1: Enable C driver error detection; =0: Disable

B126: Reserved

B127: =1: Interface in English（715TA No function）

12# factory setting 为 00000111

13#

#7	#6	#5	#4	#3	#2	#1	#0
B137	B136	B135	B134	B133	B132	B131	B130

B130: Reserved
B131: Reserved
B132: Reserved
B133: Reserved
B134: Reserved
B135: Reserved
B136: Reserved
B137: B13.7:B13.6=10: M28 position B axis; =11: position C
13# factory setting 00000000

14#

#7	#6	#5	#4	#3	#2	#1	#0
B147	B146	B145	B144	B143	B142	B141	B140

B140: =1: X axis in spinning 360 degree
B141: =1: Y axis in spinning 360 degree
B142: =1: Z axis in spinning 360 degree
B143: =1: A axis in spinning 360 degree
B144: =1: B axis in spinning 360 degree
B145: =1: C axis in spinning 360 degree
B146: =1: Spinning run <360deg
B147: =1: Spinning Axis run in short cut
14# factory setting 00000000

15#

#7	#6	#5	#4	#3	#2	#1	#0
B157	B156	B155	B154	B153	B152	B151	B150

B150: Issue RESET key at cycle cancel
B151: =1: Spindle uses grade mode
B152: =1: No sync with compilation for canned cycle
B153: Reserved
B154: =0: Baud rate9600; =1: Baud rate19200
B155: =0: +-Ovrid mode; =1: Indipendent ovrid mode
B156: Reserved
B157: Reserved
15# factory setting 00000001

16#

#7	#6	#5	#4	#3	#2	#1	#0
B167	B166	B165	B164	B163	B162	B161	B160

B160: Reserved
B161: Reserved

B162: Reserved
B163: Reserved
B164: Reserved
B165: Reserved
B166: Reserved
B167: =1: G81 for helix roller machine
16# factory setting00000000

17#

#7	#6	#5	#4	#3	#2	#1	#0
B177	B176	B175	B174	B173	B172	B171	B170

B170: Reserved
B171: =0: For Jet,A coor. to degree<360 at G46
B172: =1: Arc speed down to finish
B173: Reserved
B174: H command inactive if X,Y,Z tool offset being used
B175: Reserved
B176: =1: Set Z axis as Y1 axis
B177: =1: Set B axis as Y1 axis
17# factory setting00000000

18#

#7	#6	#5	#4	#3	#2	#1	#0
B187	B186	B185	B184	B183	B182	B181	B180

B180: Reserved
B181: Reserved
B182: Reserved
B183: Reserved
B184: Reserved
B185: Reserved
B186: Reserved
B187: Reserved
18# factory setting00000000

19#

#7	#6	#5	#4	#3	#2	#1	#0
B197	B196	B195	B194	B193	B192	B191	B190

B190: =1: X movement for feed rate display
B191: =1: Y movement for feed rate display
B192: =1: Z movement for feed rate display
B193: =1: A movement for feed rate display
B194: =1: B movement for feed rate display
B195: =1: C movement for feed rate display
B196: Reserved

B197: Reserved

19# factory setting 00000111

29#

#7	#6	#5	#4	#3	#2	#1	#0
B297	B296	B295	B294	B293	B292	B291	B290

B290: Reserved

B291: Reserved

B292: Reserved

B293: Reserved

B294: Reserved

B295: Reserved

B296: Reserved

B297: Reserved

29# factory setting 00000000

710/720/730/740 Milling 30～39# Bit parameter definition：

30#

#7	#6	#5	#4	#3	#2	#1	#0
DRNEN	LAMPEN	MP1K	SCLPT	SCLPDT	SK	FK	ALUB

ALUB: =1: Auto lubrication

FK: =1: Feed lifting key effective

SK: =1: Spindle lifting key effective

SCLPDT: =1: Spindle clamping position signal detection

SCLPT: =1: When detected clamping spindle start

MP1K: =1: Handwheel override a key control

LAMPEN: =1: Lighting function effective

DRNEN: =1: Enable automatic processing time and space to run

31#

#7	#6	#5	#4	#3	#2	#1	#0
		6BRK1	5BRK1	4BRK1	ZBRK1	YBRK1	XBRK1

XBRK1: =1: X-axis brake 1 effective

YBRK1: =1: Y-axis brake 1 effective

ZBRK1: =1: Z-axis brake 1 effective

4BRK1: =1: 4-axis brake 1 effective

5BRK1: =1: 5-axis brake 1 effective

6BRK1: =1: 6-axis brake 1 effective

32#

#7	#6	#5	#4	#3	#2	#1	#0
		6BRK2	5BRK2	4BRK2	ZBRK2	YBRK2	XBRK2

XBRK2: =1: X-axis brake 2 effective

YBRK2: =1: Y-axis brake 2 effective

ZBRK2: =1: Z-axis brake 2 effective
4BRK2: =1: 4-axis brake 2 effective
5BRK2: =1: 5-axis brake 2 effective
6BRK2: =1: 6-axis brake 2 effective

33#

#7	#6	#5	#4	#3	#2	#1	#0
AIRBLW	XKR	CONVEN	SALH	SALEN	SSTCLP	SCLP1H	SCLP1P

SCLP1P: =1: Single-spindle clamp pulse signal release
SCLP1H: =1: Single-spindle clamp signal level release
SSTCLP: =1: Loosen clamping spindle rotation when the executable (use caution)
SALEN: =1: Spindle effective alarm
SALH: =1: Spindle is normally closed alarm signal
CONVEN: =1: Conveyor effective
XKR: =1: X-axis manual key reverse
AIRBLW: =1: Blowing function effective

34#

#7	#6	#5	#4	#3	#2	#1	#0
			EXTSEN	DOORH	DOORT	SRVPO	SCLPPO

SCLPPO: =1: M10,M11 Pulse output
SRVPO: =1: M03,M04 Pulse output
DOORT: =1: Detection door switch
DOORH: =1: Security door is normally closed signal
EXTSEN: =1: External tailstock effective

35#

#7	#6	#5	#4	#3	#2	#1	#0
						LIMH	SGEN

SGEN: =1: S1,S2,S3,S4 effective
LIMH: =1: Limit is normally closed signal

WA710MI 30～39# Bit parameter definition：

30#

#7	#6	#5	#4	#3	#2	#1	#0
		SCLPPO	SSTCLP	SCLP1H	SCLP1P	SCLPT	SCLPFT

SCLPFT: =1: Spindle clamping position signal detection
SCLPT: =1: When detected clamping spindle start
SCLP1P: =1: Single-spindle clamp pulse signal release
SCLP1H: =1: Single-spindle clamp signal level release
SSTCLP: =1: Loosen clamping spindle rotation when the executable (use caution)

SCLPPO: =1: Blowing function effective

31#

#7	#6	#5	#4	#3	#2	#1	#0
					SRVPO	SALH	SALEN

SALEN: =1: Spindle effective alarm
SALH: =1: Spindle is normally closed alarm signal
SRVPO: =1: M03,M04 Pulse output

32#

#7	#6	#5	#4	#3	#2	#1	#0
					ZBRK1	YBRK1	XBRK1

XBRK1: =1: X-axis brake 1 effective
YBRK1: =1: Y-axis brake 1 effective
ZBRK1: =1: Z-axis brake 1 effective

33#

#7	#6	#5	#4	#3	#2	#1	#0
					ZBRK2	YBRK2	XBRK2

XBRK2: =1: X-axis brake 2 effective
YBRK2: =1: Y-axis brake 2 effective
ZBRK2: =1: Z-axis brake 2 effective

34#

#7	#6	#5	#4	#3	#2	#1	#0
M22EN	LT3EN	TSEN	SGEN	LUBEN	ALUB	LIMH	XREV

XREV: =1: X +, and X-swap
LIMH: =1: Limit is normally closed signal
ALUB: =1: Auto lubrication
LUBEN: =1: Effective lubrication
SGEN: =1: S1-S4 effective
TSEN: =1: Tailstock function effective
LT3EN: =1: Three-color light function effective
M22EN: =1: Spindle brake (M22/M23) effective

35#

#7	#6	#5	#4	#3	#2	#1	#0
	XALMEN	M79EN	M78EN	M77EN	M18EN	M26EN	M19EN

M19EN: =1: Exact stop the spindle (M19) effective
M26EN: =1: Rigid tapping (M26/M27) effective, = 0: Y Active
M18EN: =1: Spindle position mode (M18) effective
M77EN: =1: M77 effective
M78EN: =1: M78 effective
M79EN: =1: M79 effective

XALMEN: =1: Enable external alarm

#30~#39 is defined by PLC, refer to the appropriate manual issued by the machine tool builder.

Appendix IV System upgrade

Washing CNC system with system upgrade through U disk function. At the user site, without opening the chassis can be upgraded, the upgrade of the existing parameters of the system files, the machine coordinates, knives and machine status information does not make an impact, that the upgrade is complete after the user does not need to reconfigure the parameter file, set up the cutter compensation and other operations, including the user interface is also unaffected boot.

1 When do I need to upgrade the system software

(1) Update software users to customize their own new features.
(2) System adds new features.
(3) System software update.

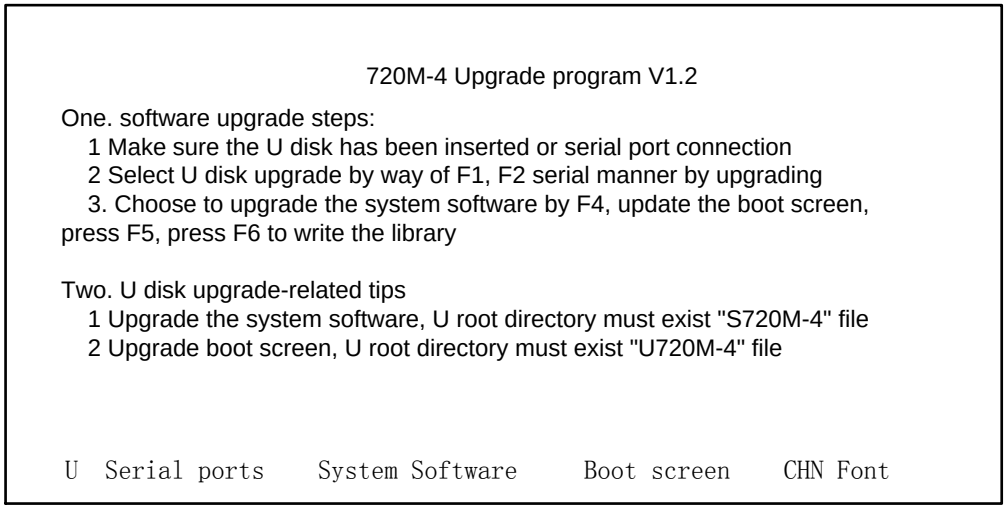
2 How to Get Upgrade Software

Get the upgraded software are the following ways:
(1) Provided directly by Huaxing service technician.
(2) By Email manner provided by Nanjing Huaxing NC company.
Note: The upgrade software file named:

WA710MI system：	S710MI
WA710MD system：	S710MD
WA720M-4 system：	S720M-4
WA730M-6 system：	S730M-6
WA740M-5 system：	S740M-5

3 How to enter the system upgrade user interface or download images interface

Press the hard reset button to reset the system (hard reset button is located next to the U disk jack, you need to open the protective cover to see U disk), according to the system before the hard reset button and hold the [ALT] key, or re-boot the system power-on power while holding down the [ALT] key until the pop-up "enter password" input box up before release [ALT] key. Input password in the input box (password GGG), the entry is complete press [ENTER], the system will pop up system upgrade interface.



Appendix 4.1 system upgrade interface

Upgrade Interface description:

U disk: press this button to select U disk upgrade

Serial: Press this button to select the upgrade through the serial port

System Software: press this button to select the upgrade system software

Boot screen: Press the button to select to update the user interface

CHN Font (Store): Press this button to select the write system Store, use this function in the factory, the end user is disabled.

4 How to upgrade the system through the U disk

Procedure is as follows:

(1) Acquisition system software upgrades, access methods, see Section 2 (how to get the software upgrade) description;

(2) To upgrade the software to get saved to the root directory of U;

(3) Into the system upgrade interface;

(4) Under the system upgrade interface, press [F1] (USB) key;

(5) Press the [F4] (system software) key is pressed two buttons to display, which began operating system upgrade, the system displays the "Start Upgrade", and dynamically display progress until the screen appears "Upgrade successful!" message appears, the system upgrade has been completed;

(6) Reboot the system to see whether the normal system startup, if you can not start properly, then the system upgrade was successful. Then you can operate the machine for simple tests, such as a manual feed operation, tool change, etc., if the tests are normal, you can try machining operations. Tests are normal, you can do the normal processing operation.

Appendix V Boot screen design

The system provides the user some boot space to store user interface, users can be edited by U disk images downloaded to the control system, display the boot interface both for your pictures updated after system startup. Picture maximum size supported by the system is 800 × 600 pixels, a color system uses 256 colors.

The system displays the boot splash screen, as shown in (to WA740M-5 as an example):

Appendix 5.1 System boot interface

1 How to edit the startup picture

Users can use the paint tools to edit a picture in the Windows operating system, you save the edited images to 256 colors, 800x600 pixel, palette must use the standard 256 colors or the system color that is displayed will be deviations.

2 How to update the user interface through USB

Press hard reset to reset the system (hard reset button is located next to the USB jack, need to open the USB drive protection cover to see), before the system hard reset key hold down the "ALT" key, or electricity system on again, in power at the same time press "ALT" key until the eject "enter password" input box so far before they can release the "ALT" key. In the input box, enter the password (password for GGG), enter the complete press the "ENTER", the system will eject the upgrade interface.

U disk boot screen upgrade requirements for file names:

WA710MI system： U710MI
WA710MD system： U710MD
WA720M-4 system： U720M-4
WA730M-6 system： U730M-6
WA740M-5 system： U740M-5

Processes are as follows:

1. Copy the upgrade files to USB drive root directory;

2. Enter the system upgrade interface (referential action article);

3. Press F1 in a system upgrade interface (USB) key, and then press F4 (user interface) keys, two key appears as pressed, system upgrade operation begins, and eject the download user pictures process interface until the interface appears on the "upgrade succeeded" message appears, indicating that the user picture to download the update has been completed;

4. Reboot your system.View the interface display is correct.

Appendix VI Serial DNC processing

1 How to get communications software

Washing provides serial communications software name for SingleComm.exe, version number 1.74 or update, get the software are the following ways:

（1）Washing service technician provided directly.

（2）Log Washing company website software download page to download.

（3）By Email solicits，Washing company technical personnel to provide

2 Serial cable connection

Turn off the PC and the system, both ends of the serial cable connected to the numerical control system with PC serial port.

3 Set communication parameters

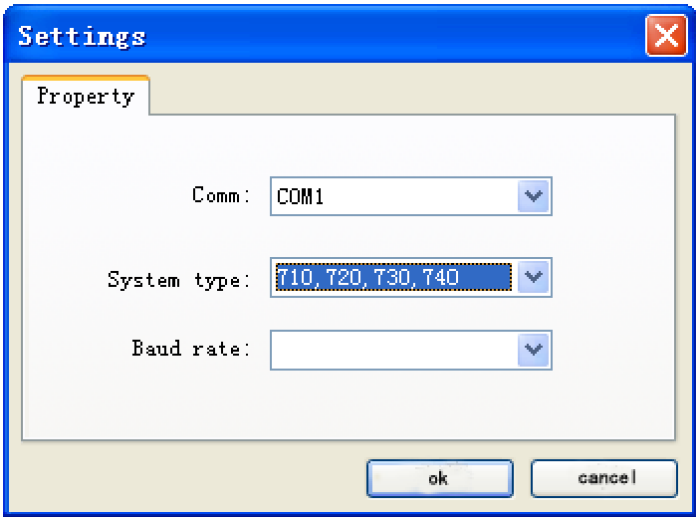
Start the PC, run SingleComm software, click SingleComm software "tool (T)" menu, select "System Settings" item, as shown in the pop-up "System Settings" dialog box, as shown in Appendix 6.1 settings below:

（1）Select the appropriate serial number (depending on the connection of the PC serial port number);

（2）System Type 710,720,730,740;

（3）According to the system bit baud rate parameter B154 to select 9600 or 19200;

（4）Press the OK button, which set up a PC-client communications software.

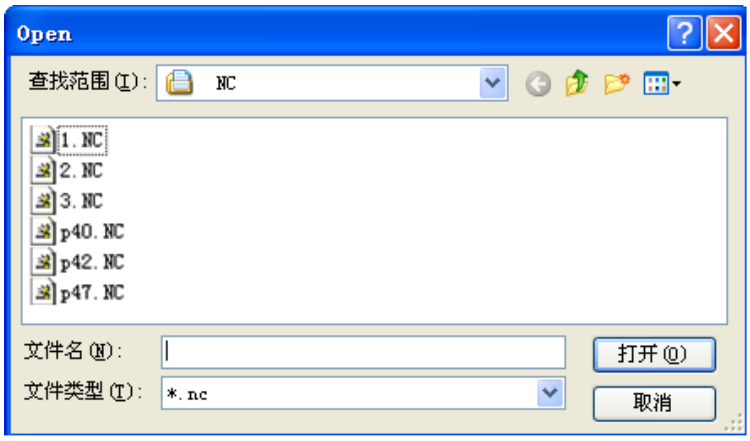


Appendix 6.1 Communication parameter settings

4 Select the machining program

Click SingleComm software "operation (O)" menu, select "Send File (S)" entry, as

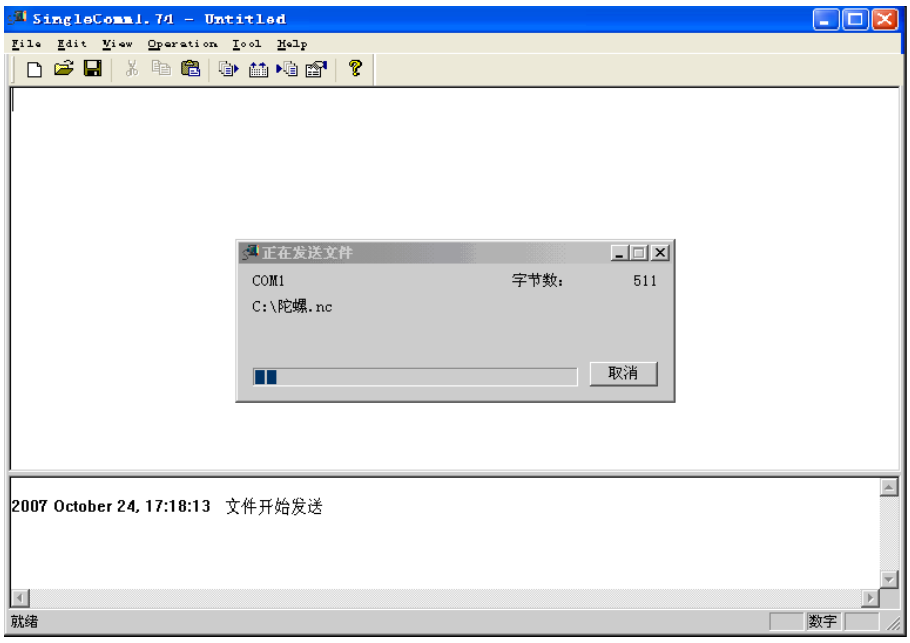
shown in Appendix 6.2, "select Send File" dialog box pops up as shown, select the NC program to be processed.



Appendix 6.2 Select NC machining program

5 Sending program

After selecting the NC program, the interface shown in Figure 6.3 in Appendix click "Open", PC sends the program to the CNC system, start CNC machining equipment to start DNC.



Appendix 6.3 Send DNC machining programs

6 System startup DNC processing or transfer files

Start CNC system, enter the serial DNC mode or serial input.

.

Nanjing Washing CNC technology CO.,LTD

Address: Centralized Industrial Zone, DongShan Bridge, JiangNing District, NanJing City, JiangSu Province

Telephone:+86-25 87170996 87170997 87170998

Hotline:4008606997

Fax: +86-2552627632

Post Code: 211153

Email:njwxcnc@163.com

Web Site:www.wxcnc.com