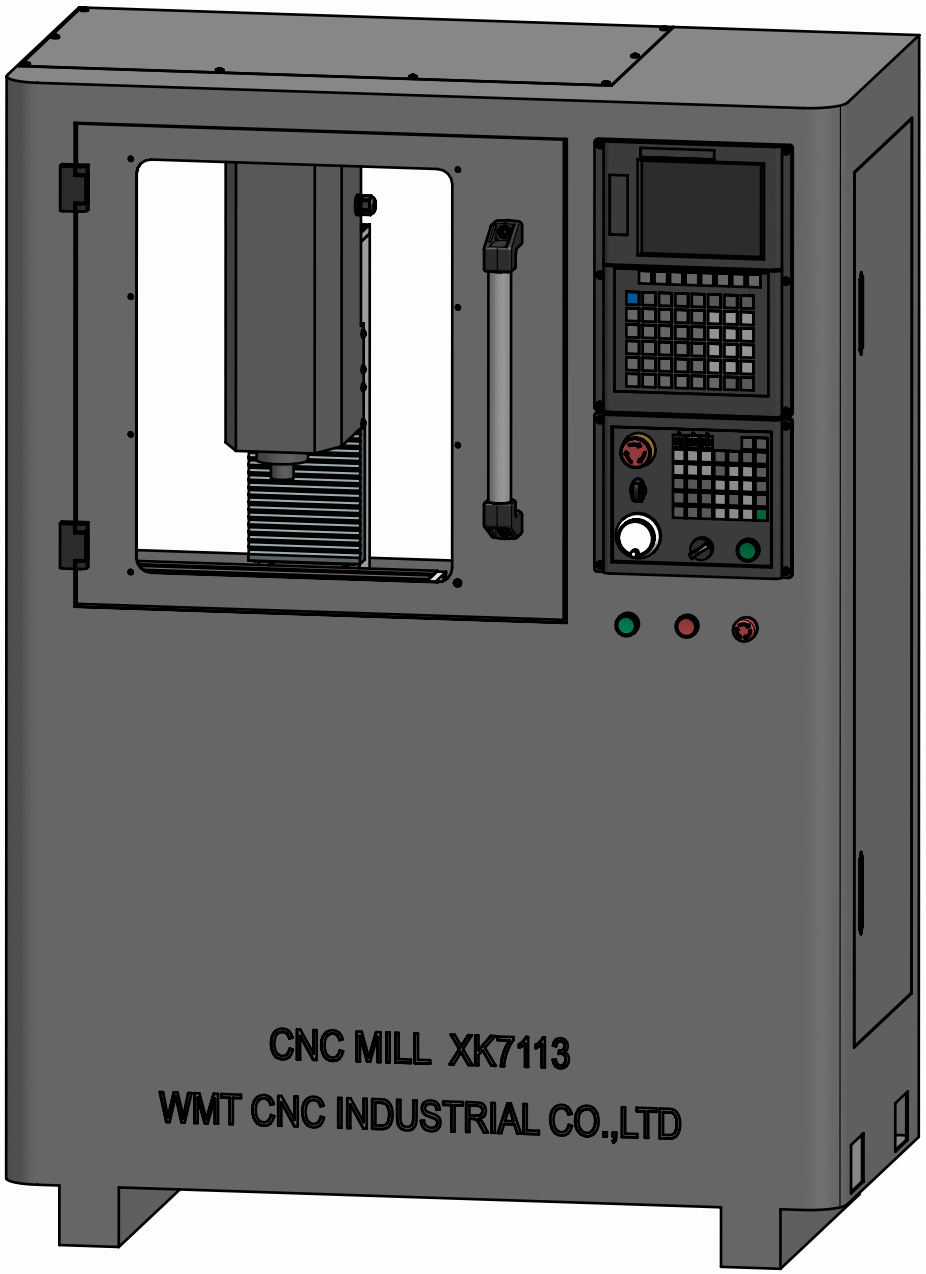
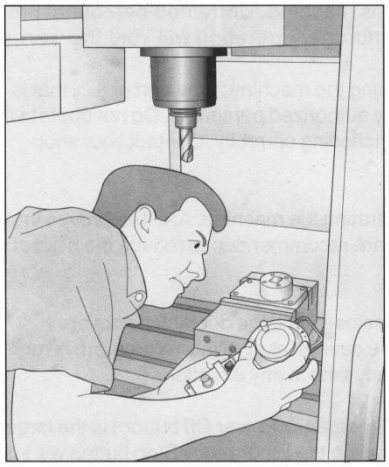
XK7113



Safety operation



Don not get caught up in your work!

All milling machines contain hazards from rotating parts, belts and pulleys, high voltage electricity, noise, and compressed air. When using CNC machines and their components, basic safety precautions must always be followed to reduce the risk of personal injury and mechanical damage.

Important - This machine to be operated only by trained personnel in accordance with the Operator' s manual and instructions for safe operation of machine.

Read Before Operating This Machine:

No staff, unless authorized, can operate this machine. There may be injury or damage caused to persons or the machine, when it is operated by those staff who have not been trained. Those troubles resulted from improper operation are beyond the range of maintenance under warranty.

When lifting and handling the machine, persons should keep far away from it, so as to prevent any accident from occurring unexpectedly.

Put on suitable eye and ear protection articles, when operating this machine, so as to prevent any injury caused by chips.

Be sure to have the door closed, when operating this machine. The rotating tools may cause injury to persons, as, when the program is running, the work table and the spindle head may shift very fast in any direction at any time.

As this machine is automatically controlled, it may start to run at any time.

When the machine is working, fingers should be kept far away from the workpiece and the cutting tool.

When the machine is running, the workpiece that is clamped improperly may be ejected out and shot through the safety door caused by the high feed at a high speed. Machining a workpiece too large in size or clamping at the edge of the platform is also not safe.

When the machine is running, do not make any adjustment or move of the machine. Have the moving parts running in the machine stopped before changing the rotation speed of the spindle.

The emergency stop button is a red button on the control panel. With it pressed down, all motion of the motors, tool change and cooling pump in the machine are stopped. Only in case of emergency to prevent any collision in the machine, can this emergency stop button be used. The machine should work within its performance range and workpieces should be machined at

proper cutting speeds and with proper feed rate. To obtain the detailed regulations on cutting speeds, maintenance & service and operation, refer to such documents as 《 Mechanical Handbook》or its similar manuals.

Do not machine toxic or flammable materials, which will give out fatal smoke and fume, when machining. Consult your material supplier to obtain safe materials before machining.

tools before operation. All the damaged fitting parts and the tools should be repaired and replaced by professional staff. If there is anything abnormal found, do not operate and have it recovered before operation.

Do not carry out any maintenance & service, when the machine is power-on. Check the control elements and the operation elements regularly, so as to ensure the machine working normally.

Qualified parts should be used for replacement, when repairing the machine.

No refitting or alteration should be made presumptuously on the equipment, which, if necessary, should be solved by this company. Any personal injury or mechanical damage resulted from such refitting or alteration made on the CNC Milling Machine should not be within the responsibility of this company.

Be sure that all the staff related should be instructed on the relevant items and safety before actual operation and installation, so as to ensure production and personal safety in subsequent applications.

Keep the work environment tidy and clean.

Keep children far away from the machine.

Before shutdown of the machine, the tool used for the last machining should be sent back to the tool magazine before the spindle is cool down. Be sure not to fall into a bad habit to keep the tool in the spindle for a long time, or otherwise 10% of the tool shanks would be difficult to be drawn out as a result of rustiness in the tool shanks because of expansion on heating and contraction on cooling and corrosion by the cutting fluid for a long time.

Some or all of these warnings may be on your machine.

Be aware of the possible dangers.

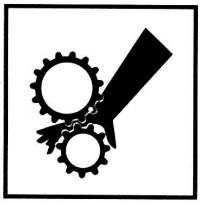
If a sticker is missing from the machine, or an extra one is needed to remind employees of the need to work safely contact the Haas factory

Automatic Equipment

As this machine equipment is programmable it may start unexpectedly. Operators and other shop personnel need to be aware of this. Keep the doors closed at all times other than loading
or unloading parts and tools.

Machine Guarding

Some guarding and access panels may close unexpectedly due to machine operation. Ensure all guarding and panels are losed and if possible, secured. Personal injury will result if the guarding falls or the access panels swing closed.



Rotating Parts

If it is necessary to work on the machine, ensure the power is disconnected before servicing the machine. Rotating parts within the machine can cause serious or fatal injuries.

Dangerous Voltages

Potential lethal voltages and currents are present within the system.Service should be done only by qualified personnel.

Job Set-up

Incorrect tooling, machining practices, or improperly clamped workpieces or fixtures will create devastating results. Double check
your set-up before beginning any machining operations.

Hot Areas

This warning is on areas of the machine that have a lot of heat associated with it. Serious burns will result if personnel come in contact with these areas.

Machine Awareness

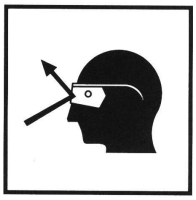
Most areas of the machine are not designed to support the weight of the operator.

Proper Attire

Watch for loose clothing and other personal belongings. These may become entangled in the machine and cause personal injury.

Tooling

Do not slow, or try to stop the rotating tools. These are extremely sharp and will cut or cause extensive bodily harm.



Eye-Safety

Always wear safety glasses or a face mask during machine operation.Flying hot chips will cause injury.

Electrical Service

Electrical shock is possible with any machine. Only authorized certifiedpersonnel should service the machine. Do not open the control cabinet or any other electrical guarding.

Spills

Clean up spills immediately. Liquids, such as machine coolant will be slippery and will cause a hazardous work environment.

Safety Windows

Workpieces clamped improperly may be caused to fly out of the safety window, during machining, thus leading to serious injury to personal body. In the course of all operations, prudential and proper machining technologies should be adopted.

Points of Attention

2.1 Instruction for Safety Warnings and Note Markings

There are safety precautions included in this manual for the purpose of protection of the users and prevention of the machine tool from any damage, which
are classified into warnings and cautions according to the nature of safety and the supplementary information is described in notes. Read these warnings, cautions and notes carefully before operation of the machine.

Warning		Noteare not observed, the users may get injured orharmed or the equipment damaged.
Caution		If these operation methods or steps specified are not observed, the equipment may get damaged.
Note		Notes are used to provide supplementaryinformation other than warnings and cautions.

2.2 Statement on Product Instructions

This company will go on to devote its effort in the perfection of machines, so as to make the machines better in performances and simpler in operation. There may be some small adjustment in the machines, which are not described specially in this edition.

2.3 Serious Warning! 

The max rotation speed of this machine is 250 r/min. The rotation speed permissible for the tools self-provided by the customers should be no less than this max rotation speed, or otherwise the rotation speed of the machine should be limited, so as to ensure the personal and equipment safety.

2.4 General Requirements 

Operating temperature range: 5°C to 40°C (41°F to 104°F)

Storage temperature range: −25°C～+55°C

Optimal environment temperature for this machine: 20°C

The place where this machine is installed should be free of any obvious impact or vibration.

There should be no interference sources in the surroundings and the machine should be kept from direct sunshine.

2.5 Power Requirement 

Unless otherwise stated, the power supply for the machine is single-phase AC115V,60HZ Its fluctuation should be within ±10% of the line voltage.

If the voltage inputted is not stable, going beyond the limit permitted, it is impossible to reach the rated horsepower of the machine. The machine may run normally but can not transmit the power required.

The reliable earthing is a guarantee for personal safety and the safe running of the machine. When connecting the power wires to the machine, it should be first to connect the protective earth wire, which should not be connected to the cooling pipes or the ground bar at the side of the machine. The specification of the earth wire should be no less than that of the power conductors.

The power inputted into the machine should be earthed. Customers are requested to check the machine frequently to make sure that its earth wire is connected reliably.

2.6 Operation Requirement

The operatior must read the manual carefully before he starts the machine.

Performances of Machine

3.1 Main Performances, Applications and Features of Machine

This machine comes with CNC control. It can machine flat surface, hole. The machine is very suitable for mechanism and training.

The main transmision adopts DC brushless motor and spindle various speed. X、Y、Z axises are connected by step motor and ball screw. Then all axises achieve feeding. The handwheel can controls the X\Y\Z axis.

3.2 Main specification

Items	S.A.E.	METRIC
CAPACITIES:		
Max. Drilling Dia.	25/32"	20 mm
Max. Milling Dia.	25/32"	20 mm
Max. Tapping Dia.	M10	M10
TRAVELS:		
X-Axis	8-21/32"	220 mm
Y-Axis	4-23/32"	120 mm
Z-Axis	7-7/8"	200 mm
Spindle Centerline to Column	7-41/64"	194 mm
Spindle Nose to Table	8-15/32"	215 mm
SPINDLE:		
Spindle Power	1.34HP	1KW
Maximum Speed	3000 RPM	
Transmission	Direct drive	
Spindle Taper	ER25	
FEED RATES:		
Rapids on X Axis	158 IPM	4 m/min
Rapids on Y Axis	158 IPM	4 m/min
Rapids on Z Axis	158 IPM	4 m/min

AXIS MOTORS:		
Max. Torque of X Axis	1.5 ft·lbf	2N·m
Max. Torque of Y Axis	1.5 ft·lbf	2N·m
Max. Torque of Z Axis	2.2 ft·lbf	3N·m
Work Table:		
Table Size (L × W)	15-3/4" × 5-1/2"	400 × 140 mm
Table Slots	3 @ 1-37/64" centers, 15/32" width	3 @ 40mm centers, 12mm width
Max Weight on Table (evenly distributed)	33 lbs.	15 kg
ACCURACY:		
Positioning Accuracy X/Y/Z	0.004"	0.1mm
Machine Repeatability X/Y/Z	0.008"	0.2mm
SPECIFICATIONS:		
CNC System	After-CNC 1500MDC / GSK980	
Coolant Pump	0.05hp	40W
Power	Single-Phase 115V, 60Hz	
Rated Amps	20A	
Package size (L × W × H)	43-5/16" × 23-5/8" × 54-5/16"	1100 × 600 × 1380mm
N.W.	728 lbs.	330 kg
G.W.	772 lbs.	350 kg

Machine struture

4.1 Main body

4.1.1 Base

This machine is mainly composed of a base, a vertical column, a work table, a headstock, a tool magazine, an electric cabinet, a control panel and button station, a cooling tank and a lubrication system, with the electric cabinet installed on the back side of the machine, the overall layout of which is compact in structure, convenient in operation and small in area occupied.

4.1.2 Worktable

The work table is a key component to put the workpiece to be machined into feed motion along the X and Y direction to fulfill the cutting processing. The feed motion along the X direction is realized by the stepping motor installed on the right side, in which the stepping motor drives, through a coupling, the ball screw pair into rotation and the ball screw pair drives further the work table into working.

The ball screw shaft and the screw pair have been applied with pre-tightening force to increase the axial rigidity and the transmission accuracy of the screw.

4.2 Column

The vertical column is also one of the base parts of the machine and on it there are the guideway, along which the headstock moves upward and downward, and the ball screw pair for the Z axis installed.

The feed motion of the headstock in the Z direction is fulfilled by a stepping motor, in which the stepping motor drives, through the coupling, the ball screw pair into rotation.

The ball screw shaft and the screw pair have been applied with pre-tightening force to increase the axial rigidity and the transmission accuracy of the screw.

The guideway is also in a rectangular structure and is ground precisely after supersonic frequency hardening treatment, so as to increase its rigidity and wear resistance.

4.3 Headstock

The spindle is driven by the main motor into rotation through the synchronous belt and the belt pulleys.

The tool unclamping in the machine is realized by the ER collet installed on the top of the spindle.

Coolant system (optional)

5.1 Brief Description of Cooling Pump

The cooling fluid tank used in this machine is of an external and mobile type, which is placed on the lower part at the rear of the machine, when it is used (see the figure below). The cooling fluid is sent through a conveying pipe to a hose nozzle, the position and the cooling fluid flow rate of which can be adjusted at will. The cooling fluid used flows back through filtration to the cooling tank for recycling use.

5.2 Brieft introduction

A single-phase vertical coolingpump for machine tools is adopted in this machine and the cooling fluid is usually saponification liquid, emulsion liquid, soda water, light mineral oil and other fluids without serious corrosion.

Rated voltage： 115V

Rated current：

Rated frequency： 60Hz

Rated lift： 118 1/8 inch

Flow rate guaranteed： 12 L/Min

5.3 Maintenance & Service and Points of Attention

Before the machine is started, check the cooling fluid level and check the deposition amount of chips in the chips collection pan and remove them timely.

Change the cooling fluid and clean the cooling tank thoroughly once every 6 months.

The mineral cutting oil can do damage to rubber parts in the machine.

When machining casting parts, sands on them and abrasive component in cast aluminum and iron will shorten the service life of the pump. ⚠

Porcelain and similar product machining is not included in the range of the statement of guarantee for wearing and its risk should be born by customers themselves. For the abrasive cutting, a maintenance plan should be prepared additionally. The cooling fluid should be changed and settlings at the bottom of the cooling tank should be cleared off more frequently. It is recommended to use a larger cooling tank. In an environment full of abrasive dust, that the service life of the pump is thus shortened is not within the range of guaranteed maintenance & repair under warranty.

Handling and Installation

6.1 Unpacking

When unpacking, first remove the top cover plate, then dismount the plates around and finally remove the bottom plate, when it is moved to its installation position. After unpacking, first check the external conditions of the machine and then check the accessories and tools attached against the packing list of the machine. If there is any question, contact this company immediately.

6.2 Handling of Machine

Before handling, bring down the headstock to its middle or lower position of its travel, pad the spindle with square pad wood under it (the supporting square wood has been provided in packing before delivery) and move the saddle and the working table to the middle position of their travel respectively, so as to prevent any impairment to the accuracy of the machine due to vibration during transportation.

When handling, it is recommended to use a fork lifter. Put steel bars through the lifting holes of the machine body, use a fork lifter to lift the machine up slowly and convey it to its installation position.

The machine, when handled with a travelling crane or a hoisting machine, should be lifted according to the marks. It should be lifted and put down smoothly without too much inclination. There should be no impact or too much vibration against the bottom and the sides, so as to prevent any impairment to the accuracy of the machine.

6.3 Installation

Whether the working accuracy of a machine is kept stable is decided by the installation accuracy of the machine. If the machine is installed securely and accurately, its working accuracy should be guaranteed. Keep this in your mind that the milling machine has been adjusted and cutting-tested comprehensively before delivery and, if installed improperly, its

accuracy and performances will be impaired.

To ensure the machine working stably, when installing, enough depth of foundation should first be guaranteed, usually no less than 470 mm in depth.

For the foundation screw bolts to fix the machine securely, customers are required to prepare them according to the drawing below by themselves.

Land Occupation Drawing for Milling Machine Installation：

The permissible error in both the longitudinal and the transverse direction is 0.03/1000 mm. If it is checked to meet the requirement, pour cement to fix the foundation bolts (Cement No.40 should be used for the machine foundation). From now on, the machine and the foundation should not be vibrated seriously. Wait till the cement is completely solidified and tighten up the nuts of the foundation bolts securely. Then use the level instrument to check it again to ensure that the installation requirement is satisfied.

When the machine has been installed in the foundation and adjusted properly, wipe off the anti-rust agent on it (For prevention of getting rusty, those locations of the machine liable to be eroded have been coated with anti-rust coating in the packing before delivery).

Use a piece of soft cloth soaked in kerosene or gasoline to clear off the anti-rust coating.

It is forbidden to dismount the components important for machine accuracy to clean.

Be sure not to use solution damageable to painted surfaces, so as to prevent their peeling off.

With the cleaning completed, apply a thin coating of engine oil on the exposed surfaces to prevent them from getting rusty.

Machine Electricity

The power supply in the machine place must be full and steady. Otherwise, the fluction may damage the machine and its electronics. What's worse, it may pose a threat to operator's safety. The total power is 2KVA.

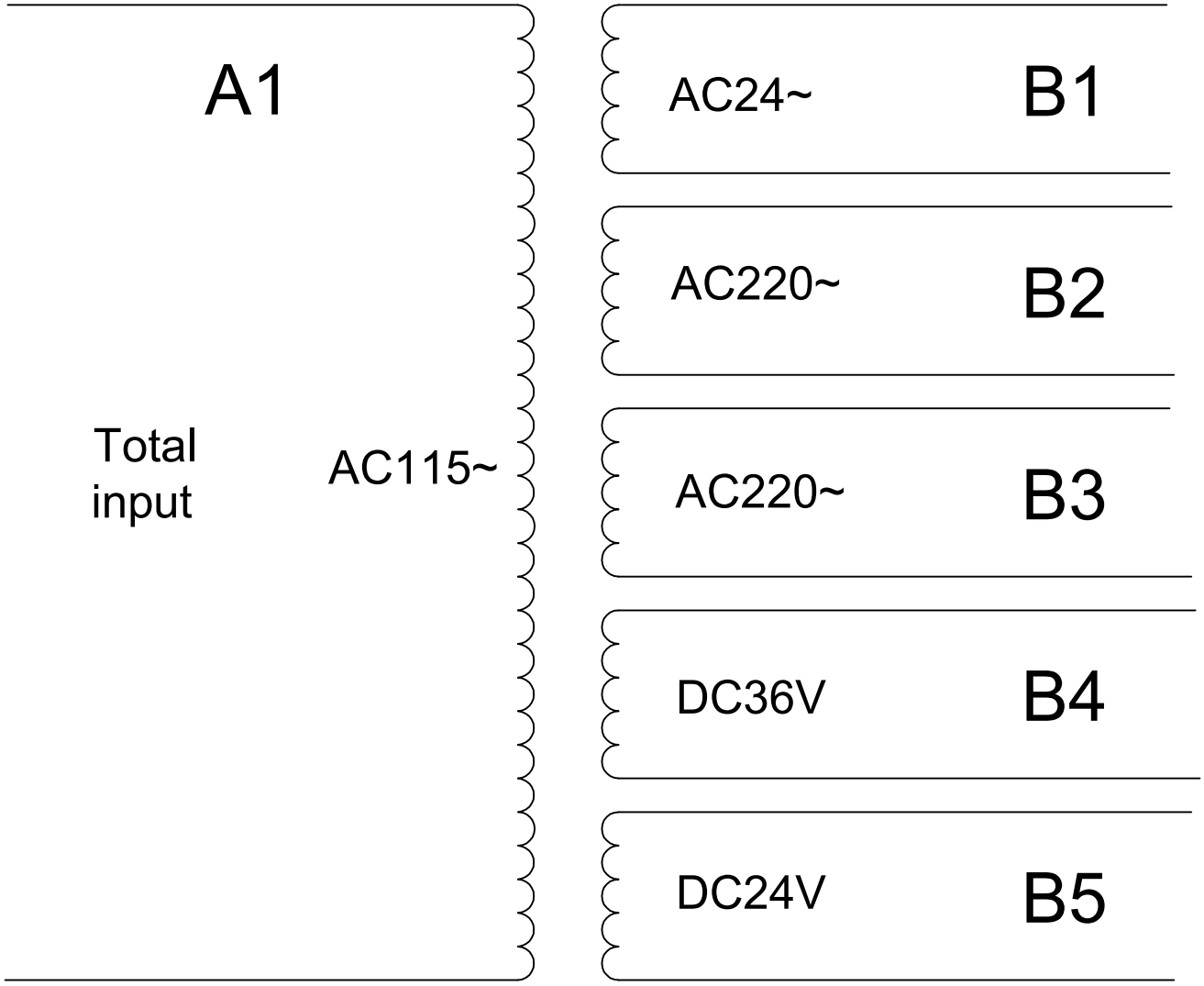
7.1 Check the power supply

The total power supply must be consistent with the machine specification. Otherwise, the machine works abnormally and the fuse blown, and it may even cause safety accidents.

! ATTENTION!

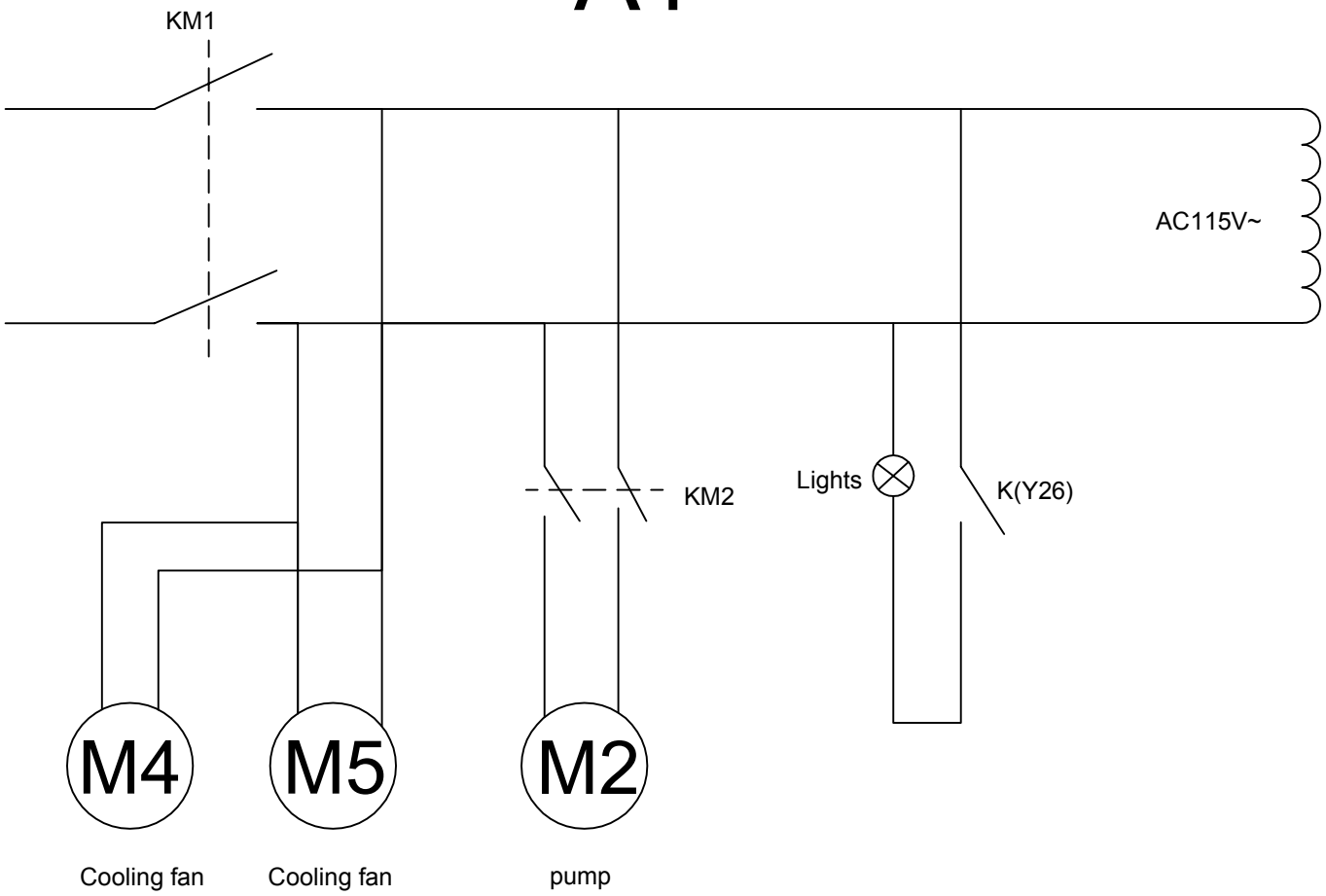
1. If the voltage is not steady, please install stabilizer.
2. Keep dry in the electric cabinet.
3. All swithes must be turned off when the machine connects the power supply.

Main circuit diagram

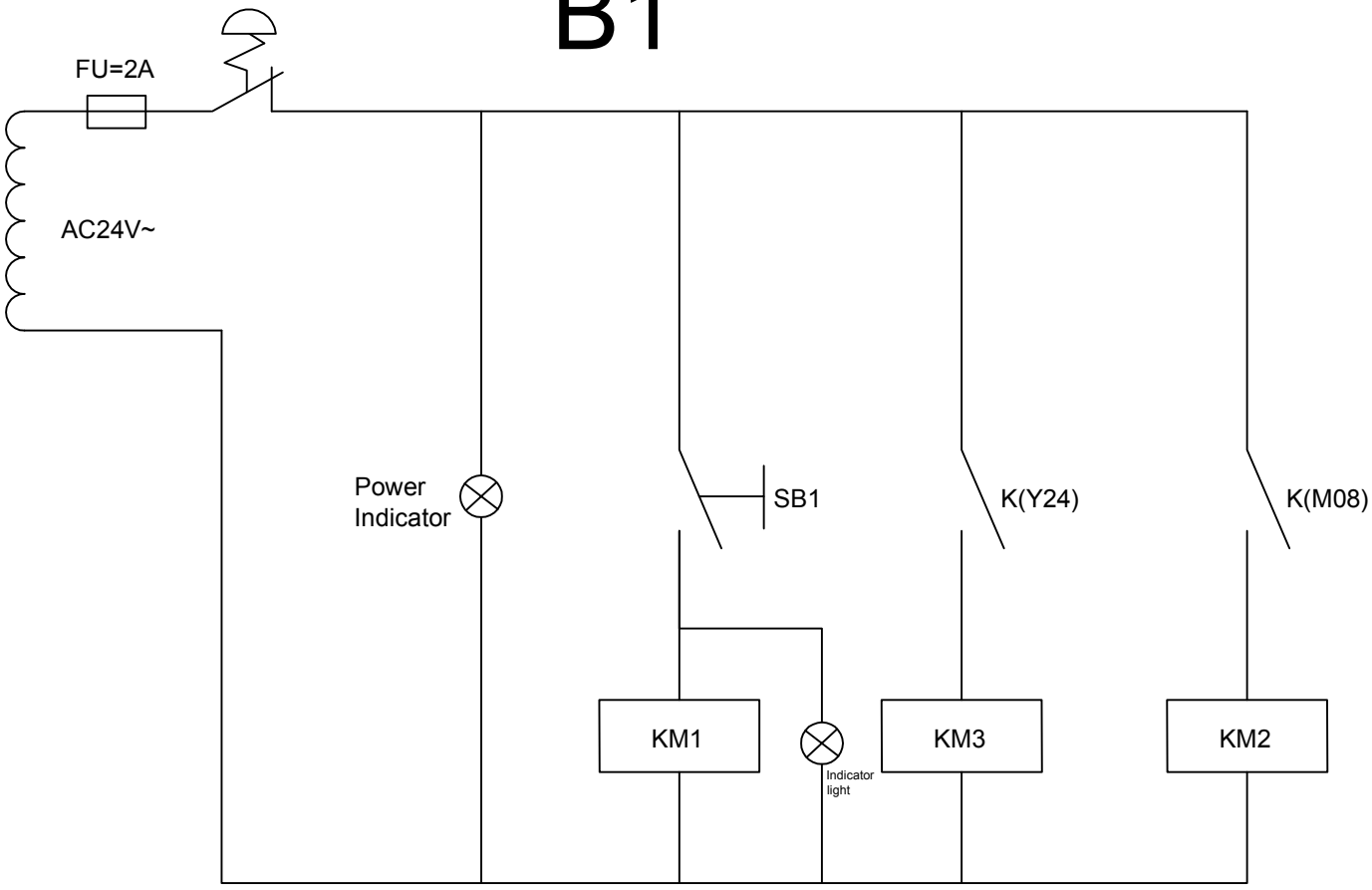

A1
Total input
AC115~
AC24~
B1
AC220~
B2
AC220~
B3
DC36V
B4
DC24V
B5

General map

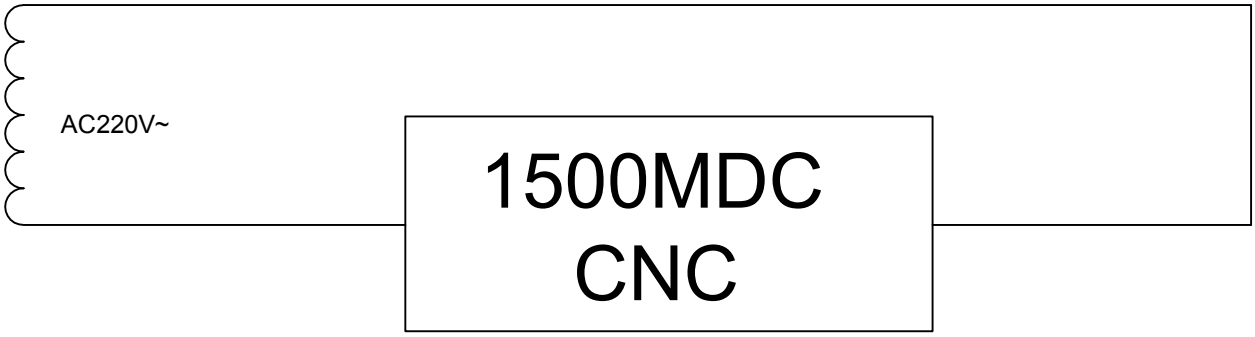
A1


KM1
AC115V~
KM2
Lights
K(Y26)
M4
M5
M2
Cooling fan
Cooling fan
pump

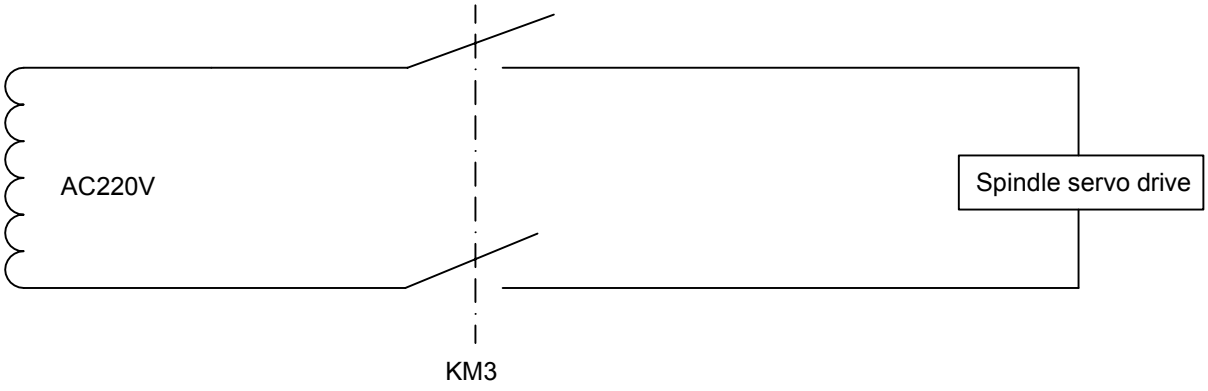
B1


FU=2A
AC24V~
Power
Indicator
SB1
K(Y24)
K(M08)
KM1
KM3
KM2
Indicator
light

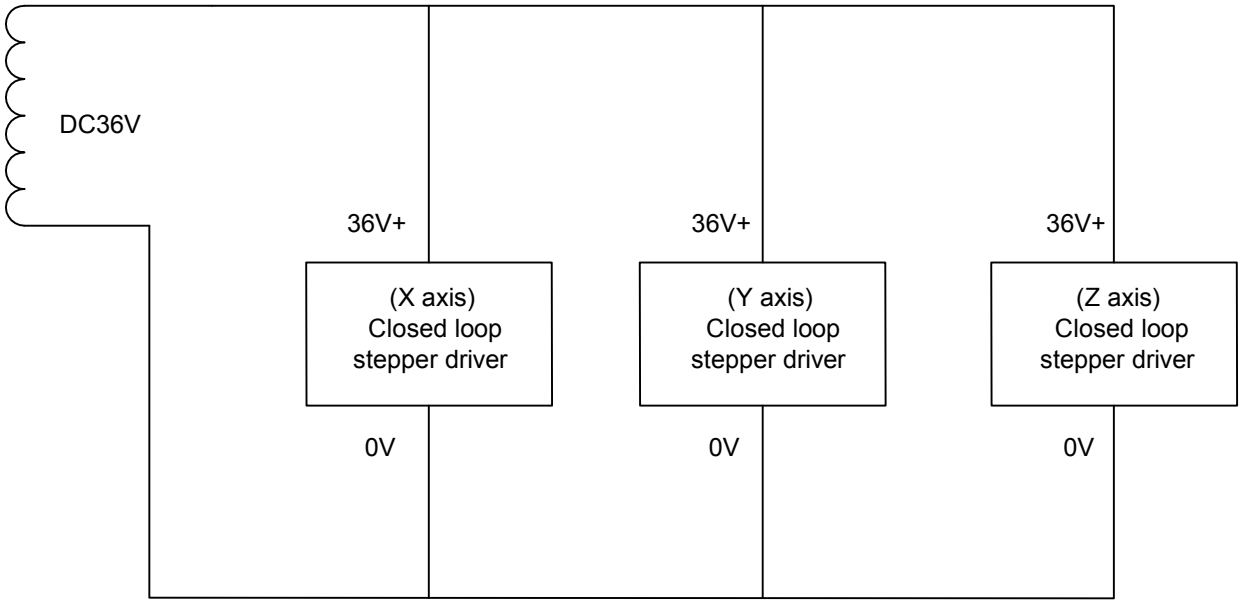
B2



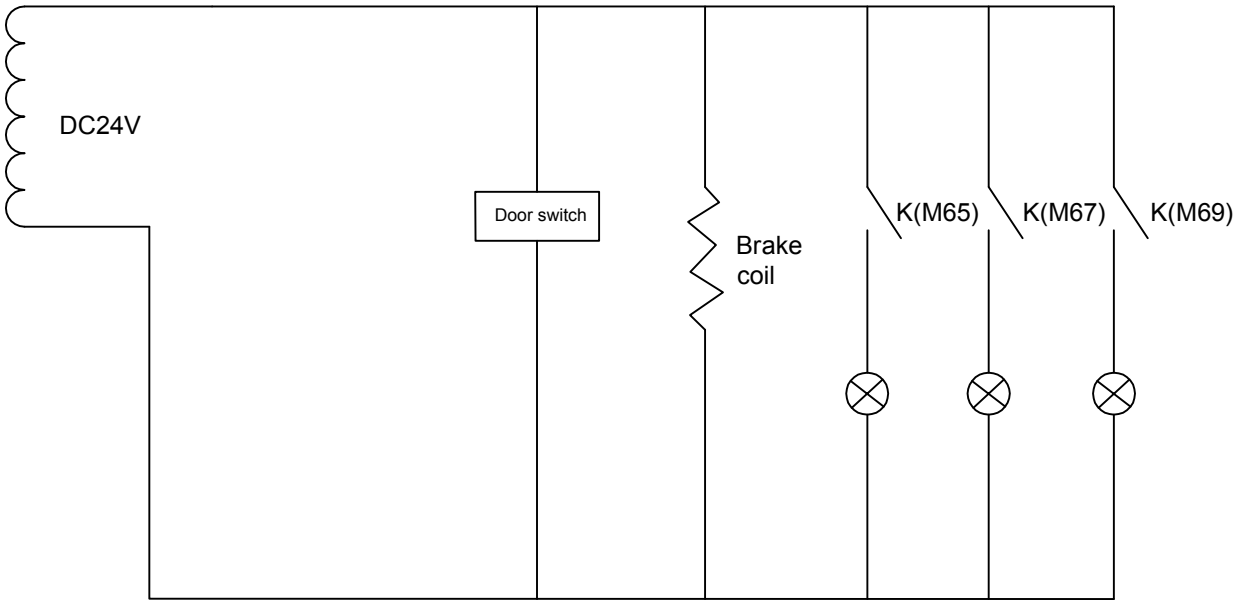
B3

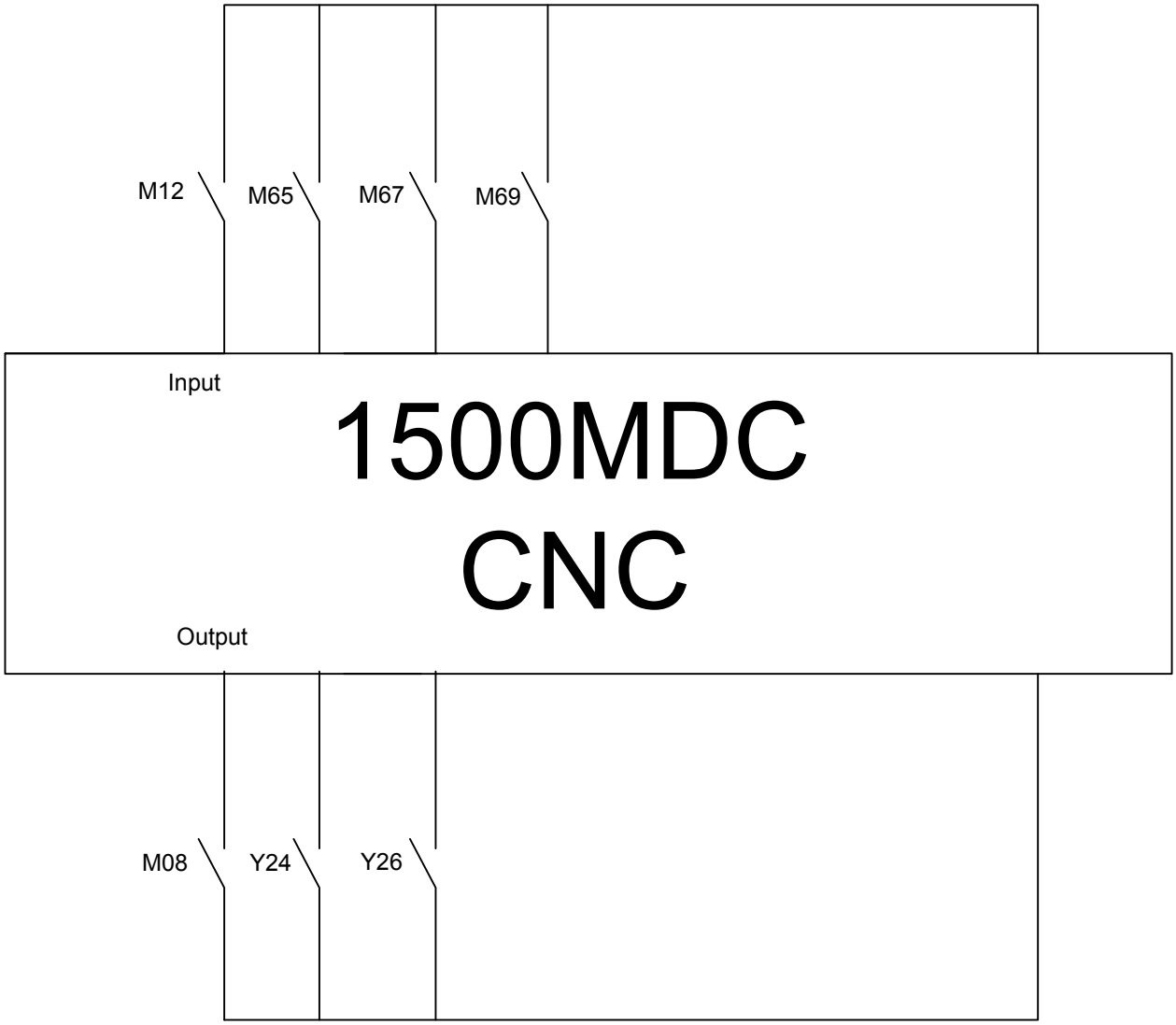


B4


DC36V
36V+
36V+
36V+
(X axis)
Closed loop
stepper driver
(Y axis)
Closed loop
stepper driver
(Z axis)
Closed loop
stepper driver
0V
0V
0V

B5


DC24V
Door switch
Brake
coil
K(M65)
K(M67)
K(M69)


M12
M65
M67
M69
Input
1500MDC
CNC
Output
M08
Y24
Y26

Machine operation

Although the machine has some safety devices and a warning label is attached to each part of the machine, the operator may, after operating the machine, have some unpredictable hazards, so after reading these warnings and instruction manuals, In order to operate this machine.

8.1 Machine Start-up Safety Precautions

8.1.1 Confirmation before starting up

Make sure the cable is not damaged, in order to avoid short circuit, leakage and electric shock. Make sure the power supply is adequate.

Make sure the cable is properly used and the wire diameter can be reliably loaded.

Make sure that the power cord is connected to the mains and that the ground wire is connected. Make sure that the operation button is confirmed for operation. Make sure the oil has been added to the normal level. To determine the machine tool parts and work area clean and tidy environment. Make sure the machine is not operating under overload. Determine the machine package, all fixed parts removed.

Make sure that you have sufficient knowledge of the machine or have read the instruction manual of the machine.

8.1.2 HOT! Warning!

1. In the test machine process, not only the speed of G00 to finish the whole process, G00 is usually used in the return to the origin and quickly reach the designated point, do not just use G00 back and forth movement, this will make machine life damage, The company is not responsible for the warranty.

2. Operation

The first time every day to start the heat engine, manual reference point in the shaft back, with AUTO or MDI side

The spindle is operated for one minute or more in S250, S500, S1000 and S2000, and the axis is continuously operated for 3 to 5 minutes at the speed of G01 F1000.

8.2 Running

The operator should read the instruction manual (mechanical and electrical parts), the machine tool numerical control system operation manual, the servo unit, the instruction manual of the servomotor and so on, and then follow the relevant operating method and sequence first,Power is connected, you must open empty check the parts of the operation, in order to work.

Machine maintenance

9.1 Maintenance beforehand

Cleaning is very important in the maintenance before hand. Dust, moisture and fume intermittent and worktable rust.All maintenance method should be consisten with the effective plan. Even though the maintenance plan conflicts with the production plan, the operator must insist the maintenance plan to achieve longer life of the machine. The operator must take notes for the parts, especiall y those in the controller.

9.2 Daily maintenance

1. Remove the chips on the wortable, base and saddle.
2. Clean the oil and coolant liquid and other material on the machine surface.
3. Clean the cover on the guideway.
4. Clean the spindle hole by clean rag and apply thin oil.
5. No tools can be placed on the cover.
6. The coolant tube must be smooth to allow adequate liquid.

9.3 Monthly maintenance

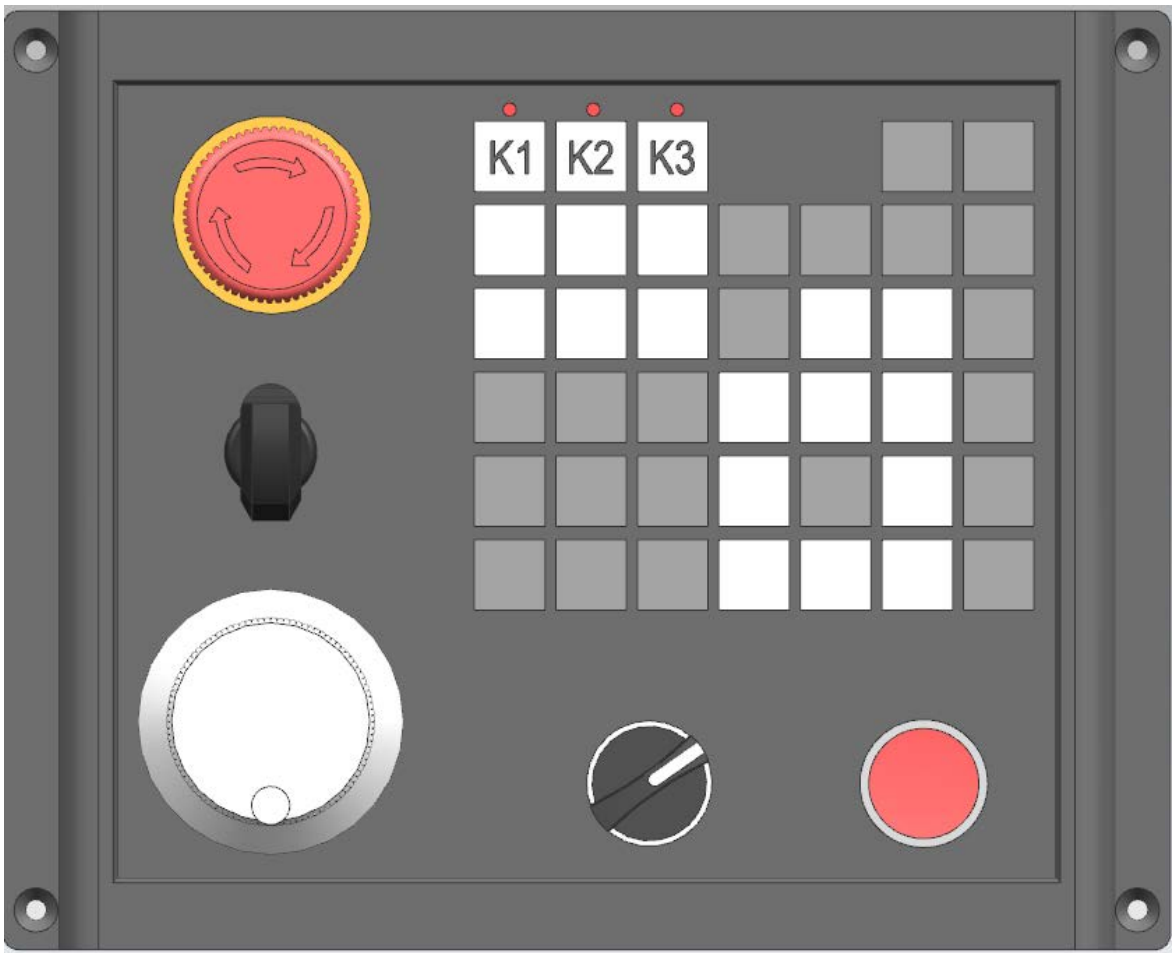
1. Deal with the daily maintenance record.
2. Clean the electric cabinet and machine.
3. Check the table and base.
4. Check the end of the wiring.
5. Romove the liquid in the. Clean the tank and add new liquid.

9.4 Manitenance twice a year

1. Deal with the monthly maintenance record.
2. Clean thoroughly on the electric cabinet and machine.
3. Clean all the motors on the machine.
5. Check the connection between motor and machine. Repair and replace them in time.

Note：

10.1 The machine does not have zero-return switch. Do not press zero-return button.
10.2 The soft-collision limit is set well on the machine before delivery. For first operation, please use handwheel X10 magnification to run the axis. If alarm, restart the machine. Reset the soft-collision limit.
10.3 The machine has G53 coordinate system and workpiece coordinate system. Please check all the being-programmed coordinate system. We advise G54 system. You can press F8 back to the main control screen. F7 to set the value.
10.4 K1 button on the control pad is the spindle driving start and stop. You must make K1 is open before changing tool. At this time, the machine will go alarm. It is normal. Press K1 button after installing tool. The alarm will disappear.
10.5 K3 button on the control pad is for light control system.
10.6 The machine has rigid tapping function. The machine will automatically shift to interpolation mode after getting the signal of G74/G84.
10.7 The contents of this manual and the performance of this product are subject to change without notice. Please pay attention to our website
10.8 Please refer to the operation manual to operate the machine.



CONTENTS

Chapter 1 Preface

This system is a universal milling machine, drilling machine, boring machine CNC system which is developed by our company. The control circuit is using the latest industrial high-speed ARM processor, large-scale field and programmable FPGA technology, multilayer PCB, the machine adopts the high integrated chip and surface mount components, the structure is more compact and reasonable so that make sure the reliability and stability of the system.

Real-time control of high speed (the highest speed shift speeds of up to 240 m / min, the highest feed interpolation speed can reach 30 m / min), high precision; the use of 800x600 dot matrix TFT LCD adaptive brightness LCD display, LED backlight brightness uniformity and long service life, and overcomes the display brightness by environmental temperature changes the shortcoming. Full screen English menu display, operation is simple and convenient.

This system is based on the lathe as the representative of the three, four, five, six, seven and eight axis linkage, This system is based on the lathe as the representative of the two or three, four axis linkage, closed-loop control universal fully digital control system, powerful function and many instructions, programming code accords with ISO international code standard. Direct control of AC servo, choosing the dual channel AC servo driver which is high price ratio.

This manual details the programming and using method of lathe system.

Important Notice:

1. Function of A, B ,C ,Xs ,Ys axis is effective when selecting the corresponding system.

2. When using this system for the first time, please read carefully all the details of each chapter so as to make it work more efficiently.

3. All the absolute function must choose the absolute system which can be effective, all the bus function must choose the bus system which can be effective.

4. The "Run" button on the panel of system can be used when debugging (No.9 parameter in other parameter to set "Effective" "Invalid"), must plus an external "Run" button when fitting system, otherwise may cause accident because of the life of button!!!So the system prohibits using the button for many times, otherwise the consequences has nothing to do with our company

Chapter 2 Programming

CNC lathe is highly effective automatical equipment according to programmed program to process workpiece. Programming is using the CNC system control language according to the requirement and drawing of the workpiece to describe the processing trajectory and the assistant action. Ideal system not only could promise process qualified workpiece, but also make the functions of lathe reasonable application and fully use, so it is very important to programming, this chapter will introduce many kinds of instructions and usage of CNC program, please read carefully.

2.1 Basic concepts

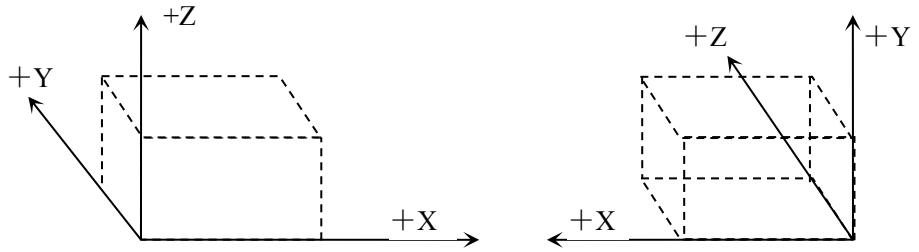
Program Segment：It is a complete command line consisted of instruction segment and data segment.

Program： is a congregation of program segment by machining logic structure in order to complete the machining of workpiece.

Machine Coordinate System： The establishment of coordinate is based on machine's zero point。The milling machine coordinate axis and its direction should follow to "ISO841" standard。The method as follow: Through right hand rule we can make the program coordinate, The Z axis is parallel as spindle, The X axis is horizontal, The Y axis is determined by right hand rule. The A, B, C are rotated axis or assistant axis which parallel as X,Y,Z axis. Furthermore, The coordinate axis direction is the increasing workpiece dimension direction.

As no work coordinate, maordke machine coordinate as work coinate.

Machine Coordinate & direction sketch map



Vertical milling、drill machine Horizontal milling

Working Coordinate System: Work piece processing uses the coordinate system is called as the work piece coordinate system, it is set by CNC. The work piece

coordinate system could change to move its zero point.

Two way to set the work piece coordinate:

1. G50/G52

Specify a value behind G52 in program to set workpiece coordinate.

2. G54/G54.1-G54.48/G55/G56/G57/G58/G59

Using the coordinate in parameter to set six workpiece coordinate

Must use this way to set when using absolute instruction

Local coordinate system: Set local coordinate system of workpiece coordinate system in order to programme easily when programming in workpiece coordinate system.

Absolute Programming： It is confirmed coordinates data programming mode based on established absolute coordinate system.。It is settlements by “G90”。

Relative Programming(increment programming)： It is distance and direction of operation end point ,compared with starting point。It is settlements by “G90”。

The maximum programming unit: 0.001mm

Mode Instruction：The instruction which can remain the function in the program.It works both in this program and program in the future.

In the same operation, there may be several mode instruction, such as M03(spindle clockwise),M04(spindle counter clockwise),M05(spindle stop).They are all mode used to control spindle.The mode of same kind are categorized into one mode group.At any time it must be one of them,and there is only one of them.The original chosen mode is called mode origin.In the above mode group,M05 is such a mode origin 。

Suspending Mode（destroying mode)： It is instruction which can turn mode instruction into mode origin or destroy the mode.Such as M20(program ending instruction),meaning the end of operation and returning to original status.

None Mode instruction： It is the instruction which has no function to store,and only works in the segment of program.

2.2 General description of program

G02, T02, H02, D02, M02, S04, F04, X-043, Y-043, Z-043, A-043, B-043, C-043, Xs-043, Ys-043, I-043, J-043, K-043, L04, P4, R043。

Note 1:“-”means this data can be use.

Note 2:In front of the numeral is 0, indicated this data only write the effective data。

Note 3:The digital presentation is a figure, when is two, top digit expression integer figure biggest figure, after low position expresses decimal point most imperial throne.

2.3 Program instruction

Introduce all the functions and using method of instruction code in this system.

2.3.1 Functional meaning of address symbol,data list

Functions	Address symbol	meaning	Data range
Document No.		Name of machining workpiece	0-9、A-Z
Program segment No.	N	No. of program segment	0000-9999
Preparation function	G	Content and mode of designated instruction operation	00-99
Auxiliary function	M	Auxiliary operation instruction	00-99
Tool chosen	THD	Specify the No. of tool	00-99
Spindle function	S	The speed of the first spindle	0-99999
	SS	The speed of the second spindle	0-99999
Cutting speed	F	Cutting speed in each minute	0.01-15000mm/min
Coordinates character	X Y Z A B C Xs Ys	The coordinates value of X Y Z A B C Xs and Ys axis.	±99999.999mm
Core coordinates	I J K	Incremental value of circle center of X Y Z axis	±99999.999mm
Radius of arc	R	Circular arc radius	0.001-99999.999mm
Delay time	P.X	Delay time of G04	0.001-65s
Program entrance	P	Entrance of calling program name	0000-9999
Repeat times	L	Times of cycle or subprogram calling L also can be number of thread	1-9999 1-99

Line skip	/	Not to carry out when "/" is in front of program line	No.9 parameter in other parameter can shield the function when D12=1

2.3.2 G、M Function instruction data list

Table 1 G Instruction-code and function

G code	function
G00	Fast locate
G01	The straight line inserts makes up
G31	No alarm when checking skip
G311	Alarm when checking skip
G02	Inserts along the circle makes up/the spiral line to insert makes up CW: The spiral motion spiral line inserts makes up the 2 circular arcs insert makes up the axis synchronization migration other axes. The instruction method only is simply adds on is not the circular arc inserts makes up the axis the shifting shaft
G03	The counter circle inserts makes up/the spiral line to insert makes up CCW
G01(G00)X I	Beveling automatically
G01(G00)Y I	Beveling automatically
G01(G00)Z I	Beveling automatically
G01(G00)X R	Smoothing automatically
G01(G00)Y R	Smoothing automatically
G01(G00)Z R	Smoothing automatically
G04	Delay
G15	Polar coordinate instruction cancellation

<table>
<tr><td>G16</td><td colspan="2">Polar coordinate instruction: The polar coordinate (radius and angle), the positive angle is counterclockwise direction of the first axis in the plane which be chosen, but the negative direction is clockwise changes.</td></tr>
<tr><td>G17</td><td>Choose the X Y plane</td><td rowspan="3">X：X axis or its parallel axis

Y：Y axis or its parallel axis

Z：Z axis or its parallel axis</td></tr>
<tr><td>G18</td><td>Choose the Z X plane</td></tr>
<tr><td>G19</td><td>Choose the Y Z plane</td></tr>
<tr><td>G20</td><td colspan="2">Inch input</td></tr>
<tr><td>G21</td><td colspan="2">Millimeter input</td></tr>
<tr><td>G28/G281-G288</td><td colspan="2">Go to the reference(zero) point of lathe</td></tr>
<tr><td>M881</td><td colspan="2">Spindle of machining center stop Output M61 check M22</td></tr>
<tr><td>M882 M883</td><td colspan="2">Set the height of Z axis tool automatically</td></tr>
<tr><td>M884 M885</td><td colspan="2">Automatical dividing center of X Y axis</td></tr>
<tr><td>M880</td><td colspan="2">Adjust tool automatically</td></tr>
<tr><td>G30/G301-G308</td><td colspan="2">Go to the reference(zero) point of coordinate</td></tr>
<tr><td>G26/G261-G265</td><td colspan="2">Axis go back to the original point of the program</td></tr>
<tr><td>G25</td><td colspan="2">Remember the current coordinate of X Y Z A B C Xs Ys</td></tr>
<tr><td>G61/G611~G615</td><td colspan="2">Back to the memory point of G25</td></tr>
<tr><td>G40</td><td colspan="2">Cancel the tool radius compensate</td></tr>
<tr><td>G41</td><td colspan="2">tool radius compensate，left</td></tr>
<tr><td>G42</td><td colspan="2">tool radius compensate，right</td></tr>
</table>

G43	Tool length positive compensate	
G44	Tool length negative compensate	
G49	Cancel tool lengthen compensate	
G45	Tool adding offset	
G46	Tool subtract offset	
G47	Tool adding two multiple offset	
G48	Tool subtract two multiple offset	
G37	Cancel scale zoom	
G36	Enable scale zoom	
G12	Cancel programmer mirror	
G11	Enable programmer mirror	
G50/G52	Set coordinate: to set sub coordinate in order to programme in workpiece coordinate	
G53	Chose coordinate	
G54.1-G54.48	Chose work coordinate 1	Note：These six workpiece coordinate is saved in CNC，user may choice any one。
G55	Chose work coordinate 2	
G56	Chose work coordinate 3	
G57	Chose work coordinate 4	
G58	Chose work coordinate 5	
G59	Chose work coordinate 6	
G60	exactitude stop	
G64	Continue path work。	
G68	rotate coordinate	

G69	Cancel rotate coordinate
G65	Non-Mode macro program
G66	Mode macro program
G67	Cancel Mode macro program
G73	Drill deep hole cycle: intermittent feed
G80	Cancel cycle mode
G81	Drill cycle: Cutting feed, stop at the bottom of the hole and back rapidly
G82	Drill cycle with chip removal: Cutting feed, stop at the bottom of the hole and back rapidly
G83	Drill small and deep hole cycle: intermittent feed
G85	Boring cycle:Cutting feed , back with cutting feed
G86	Boring cycle:Cutting feed, stop at the bottom of the hole and back rapidly
G89	Boring cycle:Stop at the bottom of the hole and back rapidly
G74	Left Tap cycle
G84	Right Tap cycle
G76	Boring cycle accuracy
G90	Absolute program
G91	Increase program
G98	Go back to the original point: used for the last drilling
G99	Go back to the R point: used for the first drilling
G22	Program cycle

G800	Cancel Program cycle

Table 2 M code and function

Spindle	M03 M04 M05 M203 M204 M205	Spindle CW Spindle CCW Stop Spindle M7053/M7054 P xxxx; It will delay time after spindle CW or CCW, the time is determined by "P". Unit: Millisecond. Example: M7053 P2000; Means it will delay 2 seconds to stop spindle.	M203 M204 M205 The second spindle of bus system rotate CW ,CCW and stop
Cooling	M08	Turn on cool	
	M09	Turn off cool	
Chuck	M10	Tighten tool	
	M11	Loosen tool	
Huff	M58	Turn off huff	Controls M59 signal
	M59	Turn on huff	
Lubrication	M32	Turn on lubricate	Controls M32 signal
	M33	Turn off lubricate	
User-defined	M79	User self-defined1 output turn on	Double signal output
	M78	User self-defined1 output turn off	
User-defined output	M61 M60	User self-defined2 output turn on User self-defined2 output turn off	Controls M61 signal
	M63 M62	User self-defined3 output turn on User self-defined3 output turn off	Controls M63 signal

	M65 M64	User self-defined4 output turn on User self-defined4 output turn off	Controls M65 signal
	M67 M66	User self-defined5 output turn on User self-defined5 output turn off	Controls M67 signal
	M69 M68	User self-defined6 output turn on User self-defined6 output turn off	Controls M69 signal
	M71 M70	User self-defined7 output turn on User self-defined7 output turn off	Controls M71 signal
	M75 M74	User self-defined8 output turn on User self-defined8 output turn off	Controls M75 signal
Spindle shifting	M41	SP Speed first gear	
	M42	SP Speed second gear	
	M43	SP Speed third gear	
	M44	SP Speed fourth gear	
User-defined input	M12 M13	Check M12 input valid Check M12 input invalidate	To skip when conditions are tenable Example: M12 P120 Program skips to 120th line to carry out when M12 is effective.
	M14 M15	Check M14 input valid Check M14 input invalidate	
	M16 M17	Check M16 input valid Check M16 input invalidate	

	M18 M19	Check M18 input valid Check M18 input invalidate	
	M28 M29	Check M28 input valid Check M28 input invalidate	
	M22 M23	Check M22 input valid Check M22 input invalidate	
	M18xx M28xx	Check xx input valid Check xx input invalidate	
Subprogram	M97 M98 M99	Program skip Subprogram is on tap Back to subprogram is on tap	L=1-99 P is the line number of transferring program
Program controlling	M87 M00 M01 M02 M30 M20	Number of workpiece plus 1 Suspend program Suspend program,input M22 effective suspend Program is over M05,M09 program is over Program is over, automatical repeat run the program according to the parameter which set the running times, be used for the debugging	When other parameter P10=0 is set not to auotomatical plus 1, instruction M87 to make workpiece number plus 1

Speed of spindle	S SS	Set speed of the first spindle Set speed of the second spindle	S=0-99999 SS=0-99999
Read the position of absolute motor	M500 M501 M502 M503 M504 M505 M506 M507 M508	M500:Load absolute motor position of all the feeding axis and reset the current lathe coordinate. M501-M508:Load the absolute motor position and reset the current lathe coordinate of X Y Z A B C Xs Ys axis	

Reset the workpiece coordinate	M312 M313 M314 M315 M316 M317 M318 M319 M320	M312:Reset the workpiece coordinate of C axis M313:Reset the workpiece coordinate of Xs axis M314:Reset the workpiece coordinate of Ys axis M315:Reset the workpiece coordinate of A axis M316:Reset the workpiece coordinate of B axis M317:Reset the workpiece coordinate of X axis M318:Reset the workpiece coordinate of Y axis M319:Reset the workpiece coordinate of Z axis M320:Reset all the workpiece coordinate	

Reset the lathe coordinate	M412 M413 M414 M415 M416 M417 M418 M419 M420	M412:Reset the lathe coordinate of C axis M413:Reset the lathe coordinate of Xs axis M414:Reset the lathe coordinate of Ys axis M415:Reset the lathe coordinate of A axis M416:Reset the lathe coordinate of B axis M417:Reset the lathe coordinate of X axis M418:Reset the lathe coordinate of Y axis M419:Reset the lathe coordinate of Z axis M420:Reset all the lathe coordinate	
Bus driver instruction	M133 M135	M133:Specify the rotate speed of driver(must be in multiples of 30) M135:Enable the driver work or not	Example:M133 X300; M135 Z0;Closet the driver of Z axis M135 Z1;Open the driver of Z axis

2.4 Programming instruction and using method

2.4.1 Programming conventions

(1) Multiple instruction in a line: It can be many instructions in the same line to increase the number of line, but the same instruction can't be together.

(2) The instructions and parameter can be random placement in a same line.

Example: M03 G01 X20 Y-30 can be written G01 Y-30 X20 M03

(3) The instruction can't be repeated emergence in the same program line.

(4) There can't be appear independent parameter and operation with instruction in the same program line.

(5) "0" in front of the instruction code couldn't write, example: G01 M03 can be

written G1 M3.

(6) The instruction behind the starting any point, any line or change tool must use absolute program.

(7) Non-mode instruction must be used in specified program line, example: G61.

(8) Mode instruction is always effective before appearing other similar instruction.

Example: N0000 G01 X300 F100 ; G01 instruction
N0001 X260 ; G01 instruction
N0002 G00 Y200 ; G00 instruction, G01 is invalid.

2.4.2 G function instruction

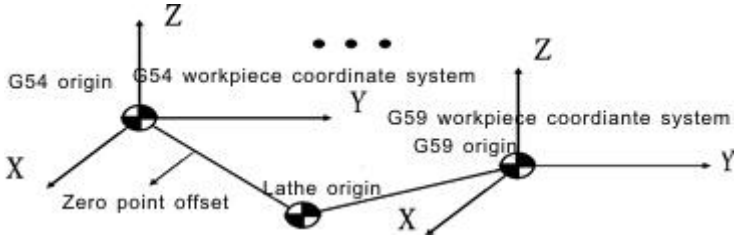
(1) Choose instruction of coordinate system.(G53/G54/G55/G56/G57/G58/G59)

Format: G53（G54/G55/G56/G57/G58/G59） Mode after setting

G53 machine coordinate
G54.1/G54.48 work coordinate G54.1/G54.48
G55 work coordinate 2
G56 work coordinate 3
G57 work coordinate 4
G58 work coordinate 5
G59 work coordinate 6

G53 machine coordinate is decided by machine reference point。The default coordinate is G53. Suggest not to adjust the value of G53, all the workpiece coordinate will have offset.

G54/G55/G56/G57/G58/G59 work coordinate have offset relative to machine coordinate which can be set in parameter.



Example: G00 G54 X50 Y60 Z70

Move to X50 Y60 Z70 of G54 with speed of G00.

(2)Coordinate system (G50/G52)

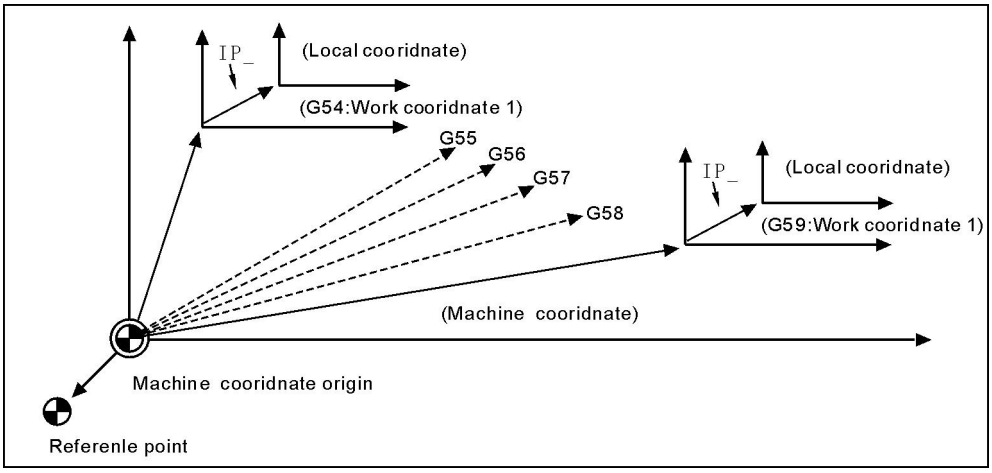
Set sub coordinate system when programming in workpiece coordinate system is to programme easily.G50 is to set workpiece coordinate, G52 is to set local coordinate.

Format:G52 X_ Y_ Z_ ; Set local coordinate system (Mode)
G52 X0 Y0 Z0; Cancel local coordinate system

Use G52 to set local coordinate system in lathe coordinate system(G53) or workpiece coordinate system(G54~G59). The origin of local coordinate system is set by X_ Y_ Z_.

Local coordinate system setting does not change the workpiece coordinate system

and machine tool coordinate system. When using the G52 instruction to set workpiece coordinate system, if it is not the instruction of all the axis coordinate value, the local coordinate system of non-specified coordinate value will not cancel and remain unchangeable .Use the absolute mode to specify motion instruction after the G52 program segment .



(3)Program method(G90/G91)

Two methods to move tools: Absolute value instruction and incremental value instruction. Programme coordinate value of end point with absolute value; Programme incremental value of moving distance with incremental value. G90 and G91 corresponding to point out the coordinate use absolute value or incremental value.

Format: G90 （Mode,initial） ;Absolute
G91 （Mode） ;Increment

Pay attention:The rotating axis use nearest calculation to carry out when using absolute coordinate to programme, use programming calculation to carry out when using relative coordinate to programme.

(4)Select Plane(G17/G18/G19)

Format：G17 (Mode，Original) ;Set XY Plane
G18 (Mode) ;Set ZX Plane
G19 (Mode) ;Set YZ Plane

Used to point out arc interpolation plane.

Note：this instruction does not produce motion.

(5)Rapid motion(G00)

Tool move to instructive position according to G00 speed in parameter.

As absolute method, use section end point coordinate to program；

As increase method, use motion distance to program.

Format：G00 X_ Y_ Z_ A_ B-_C_ Xs_ Ys_(Mode，original)

Note：X，Y，Z，A , B means motion axis.The data point out motion distance and direction by absolute or increase method.

G00 move to aim point according to line way. Also could use linkage mode: set D6=0 of No.9 parameter in other parameter to linkage mode.

Moving speed is determined by parameter.

(6) Line interpolation(G01)

Used for single axis motion or 2,3,4 axis interpolation motion.

Format：G01 X_ Y_ Z_ A_ B_ C_ Xs_ Ys_ F_ (Mode)

Note： X, Y, Z, A, B, C, Xs, Ys mean motional axis.The data point out motion distance and direction by absolute or increase method.Motion speed is determined by F word. The F instruction is mode.

The feeding speed of F in G01 can be adjusted by feeding override on the panel, the range is 0%~150%.

G01 instruction is also can be written G1.

(7) Arc interpolation(G02/G03)

In the program plane，these instruction is to execute G02 clockwise and G03 counter-clockwise arc interpolation。

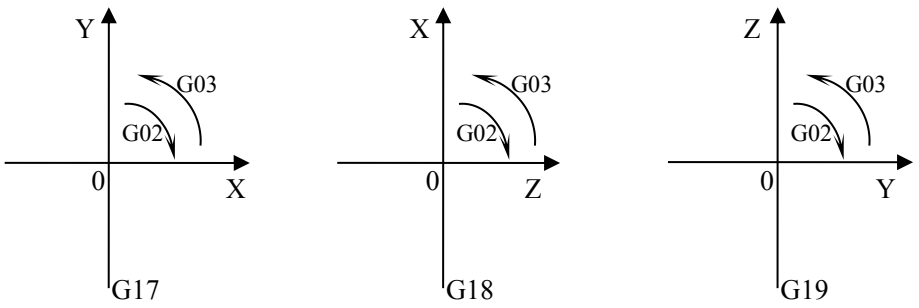
Format：$G17\begin{Bmatrix}G02\\G03\end{Bmatrix}X\_Y\_\begin{Bmatrix}I\_J\_\\R\_\end{Bmatrix}F\_$;XY plane(Mode)

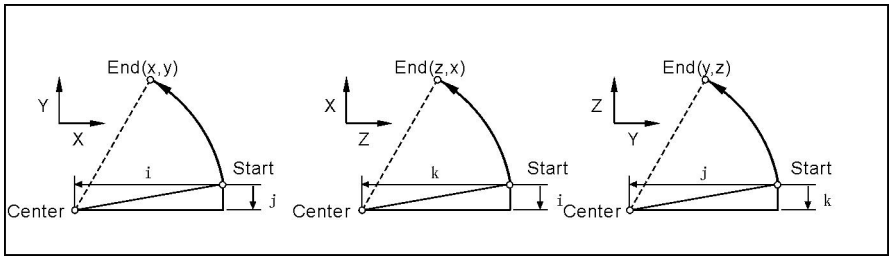
$G18\begin{Bmatrix}G02\\G03\end{Bmatrix}X\_Z\_\begin{Bmatrix}I\_K\_\\R\_\end{Bmatrix}F\_$;ZX plane(Mode)

$G19\begin{Bmatrix}G02\\G03\end{Bmatrix}Y\_Z\_\begin{Bmatrix}J\_K\_\\R\_\end{Bmatrix}F\_$;YZ plane(Mode)

Note：Arc interpolation must point out interpolation plane，the X、Y、Z word point out the arc end coordinate value，I、J、K separate is X、Y、Z increase value from original point or center point。In another words, Make the original point as zero point，As center point locate to positive direction of original point the value will be positive，As center point locate to negative direction of original point the value will be negative。I J K function is describe center point coordinate。On the side, We can use R program，the R is negative when arc angle largen 180 degree。

The arc track as follow:





The arc interpolation speed is determined by F word.

Attention: I, J, K and R are the non- modality instruction.

Demonstration:

1) absolute programming

N0000 G92 X200 Y40 Z0;

N0010 G90 G03 X160 Y40 I-20 J0;

N0020 G02 X120 Y40 R20;

N0030 G02 X120 Y40 R20;

N0040 G26 M02;

2) relative programming

N0000 G91 G17 G03 X-40 Y0 R20 F300;

N0010 G02 X-40 Y0 R20;

N0020 G02 X0 Y0 R20;

N0030 G26 M02;

Two methods have the same result.

(8) Spiral interpolation (G02/G03)

Spiral interpolation means arc interpolation adding another axis line interpolation,F instruction defines arc interpolation speed.therefore, the feed speed of line interpolation axis is as follow:

$$F \times \frac{\text{Lengthen of line axis}}{\text{Lengthen of arc}}$$

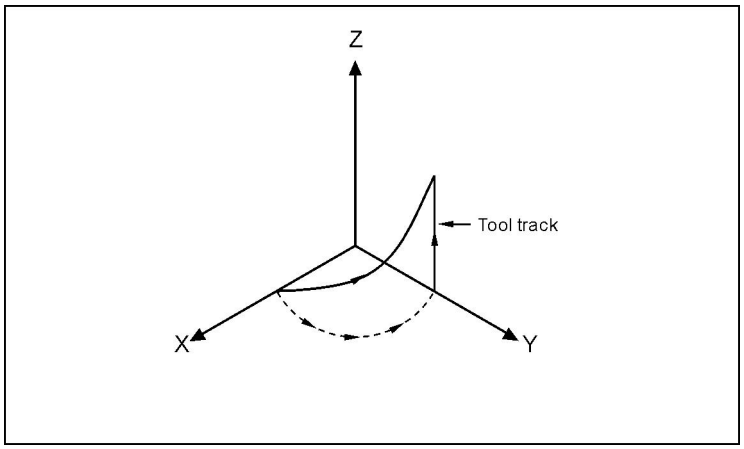
Format ： $G17\begin{Bmatrix}G02\\G03\end{Bmatrix}X\_Y\_\begin{Bmatrix}I\_J\_\\R\_\end{Bmatrix}F\_$;XY plane(Mode)

$G18\begin{Bmatrix}G02\\G03\end{Bmatrix}X\_Z\_\begin{Bmatrix}I\_K\_\\R\_\end{Bmatrix}F\_$;ZX plane(Mode)

$G19\begin{Bmatrix}G02\\G03\end{Bmatrix}Y\_Z\_\begin{Bmatrix}J\_K\_\\R\_\end{Bmatrix}F\_$;YZ plane(Mode)

The cutting tool radius compensates only carries on to the circular arc,Inserts in the segment in the instruction spiral line which makes up not to be able the instruction cutting tool bias and the cutting tool length compensates.

In the spiral interpolation section, cannot use tool length and radius compensation.



N0001 G90 G17 G54
N0002 G01 X20 Y0 Z0 F600
N0003 G03 X0 Y20 R20 Z15 F180
……

(9) Arc instruction in three dimensional space G06

Format: G06 X_ Y_ Z_ J_ K_ F_

Function:If you don't know the center and radius of the arc in three dimensional space.But you already knew the coordination of three point on arc,so you can use G06, the middle point between the starting point and end point to make sure the direction of the arc.

Instruction:G06 is a mode G code;

I:The relative coordination value of the starting point the arc pass through relative to the starting point(X)(radius value, direction);

J:The relative coordination value of the starting point the arc pass through relative to the starting point(Y)(direction);

K:The relative coordination value of the starting point the arc pass through relative to the starting point(Z)(direction).

Pay attention:

1).Middle point:Any point except the start point and end point on the arc

2).When the three points are collinear, the system alarm.

3).I=0 when omitting I,K=0 when omitting K,J=0 when omitting J;the system alarm

when omitting I,J,K the same time.

4).The meaning of I,J,K is similar to the displacement value I,J,K of the center coordinate relative to the starting point coordinate in G02/G03

5).G06 can't process the whole circle.

6).This instruction contains large amount of calculation which only can be used with bus system,maybe not smooth with other system;

Example:

```
G0 X10 Y28 Z10
G06 X30 Y98 Z10 I5 J-6 K-5 F100
X130 Y198 Z120 I55 J-86 K-65
G0 X0 Z0
M02
```

(10) Delay instruction(G04)

Format：G04 P_ ;or
G04 X_ ;or
G04 U_ ;

Function:Every axis is stop and mode instruction is still working when carry out this instruction, after delaying the specified time to carry out the next program segment.

Instruction introduction:

a.The unit of P delay time is ms(Millisecond)/

b.The unit of X and U delay time are S.

c.Example:

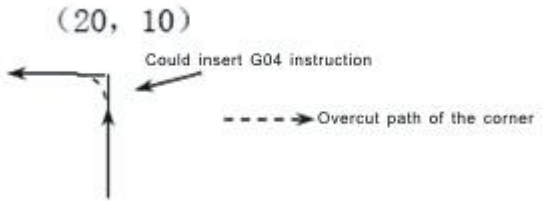
G04 X1； delay 1s.

G04 P1000； delay 1s.

G04 U1； delay 1s.

d.Special application:G04 can be accurate stop instruction, such as processing corner kinds of workpiece, it appears over cutting sometimes, if use G04 instruction around the corner, it will clear the over cutting.

Example as follows:



Program: … …

N150 G01 X20 Z10 F100；

N160 G04 P150； （Clear the over cutting）

N170 G01 W-10；

……

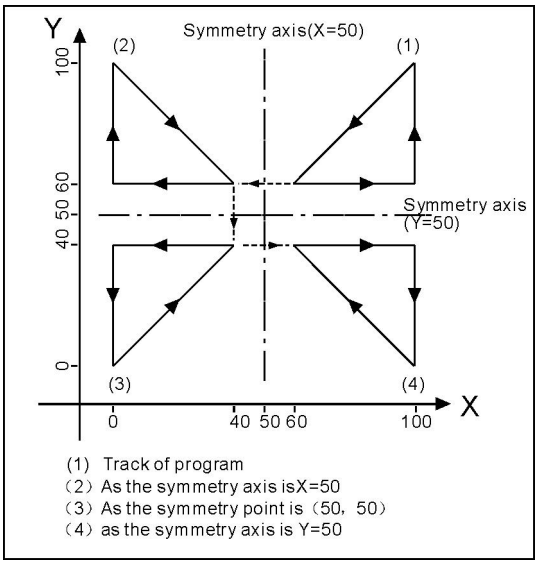
Pay attention: Set No.21 parameter in "N"(processing) parameter to clear the over cutting.

(11) Mirror instruction(G11/G12)

In order to decrease program codes, be used for machining symmetry workpiece.

Format：G11 X_ Y_ (Z_ X_) (Y_ Z_)(mode)
according to XYZ symmetry axis
G12 (mode，original) ;Cancel Mirror。

For example:



The mirror procedure gives an example

Sub program

KG11

N10 G00 G90 X60.0 Y60.0；

N20 G01 X100.0 F100；

N30 G01 Y100.0；

N40 G01 X60.0 Y60.0；

N50 M99；

Main program

N10 G00 G90;

N20 M98 PKG11;

N30 G11 X50.0

N40 M98 PKG11;

N50 G11 X50.0 Y50.0

N60 M98 PKG11;

N70 G11 Y50.0

N80 M98 PKG11;

N90 G12;
N100 M12;

(12) Proportions scale instruction(G36/G37)

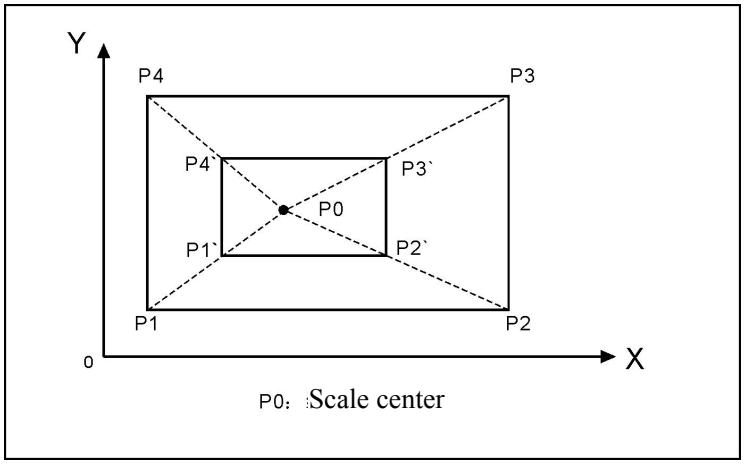
Used for not change the processing program, process according to the proportion of reduced or enlarged workpiece. Programming shape is magnified and reduced (scaling), use X_, Y_ and Z_ to specify scaling center. If an axis is not specified, the axis not to carry out scaling. The number behind the X_ Y_ Z_ is the coordinate of current workpiece coordinate system.

Format：G36 X_Y_Z_R_ (mode) ;Carry out G37 (mode，original) ;Cancel

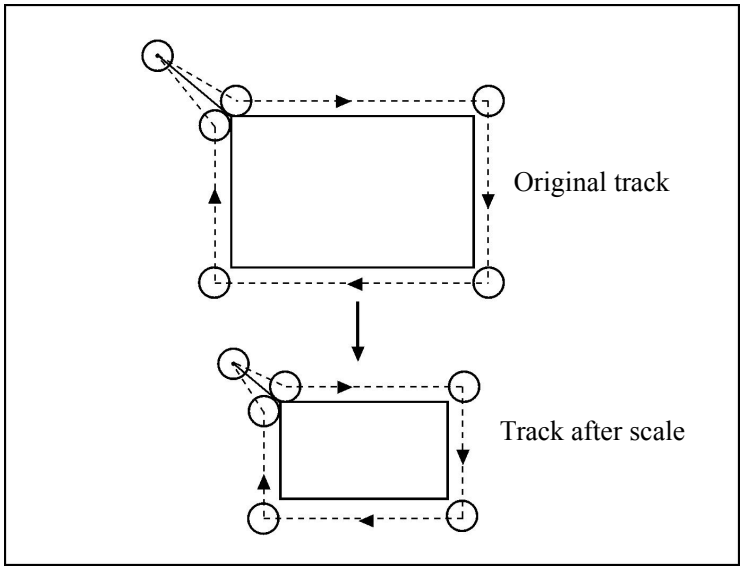
Note：the scale coefficient is behind R.

For example:

P1` P2` P3` P4` magnify to P1 P2 P3 P4,R=P0P4`/P0P4. When P1 P2 P3 P4 reduce to P1` P2` P3` P4`, R=P0P4/P0P4`. So: R<1 when magnifying, R>1 when reducing,R=1 can be default.



In the proportions scale section, cannot use tool length and radius compensation:



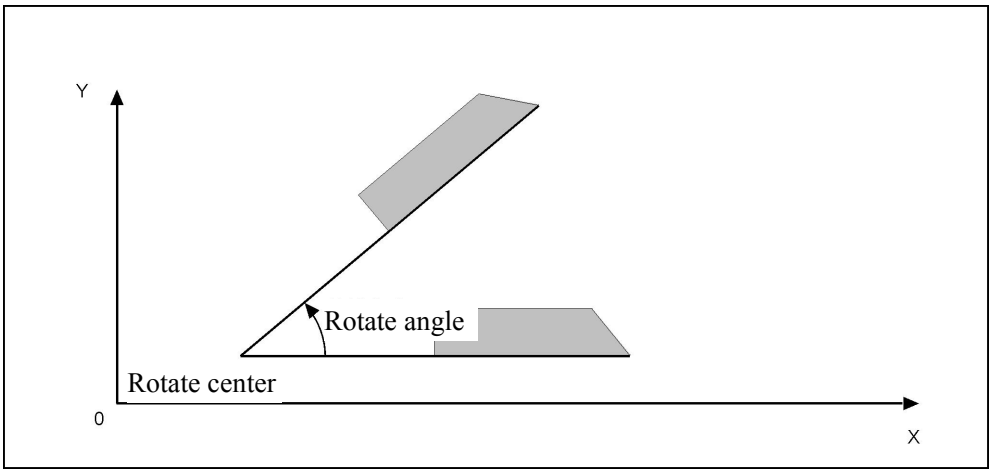
(13) Coordinate rotate(G68/G69)

Programming shape can rotate, the rotating instruction can make workpiece rotate for specified angle. Anyway, if the shape of workpiece is comprised of many same graphics, the graphics unit can be compiled to subprogram, then the subprogram is on tap for rotation instruction of main program. This can simplify the programming, saving storage space.

Format： G68 X- Y- R- (mode) ; enable
G68 Z- X- R- (mode) ; enable
G68 Y- Z- R- (mode) ; enable
G69 (mode，original) ; disable

Note： The (G17)X-Y- or (G18)Z-X- or (G19) Y-Z- behind G68 is used for pointing out rotate center.

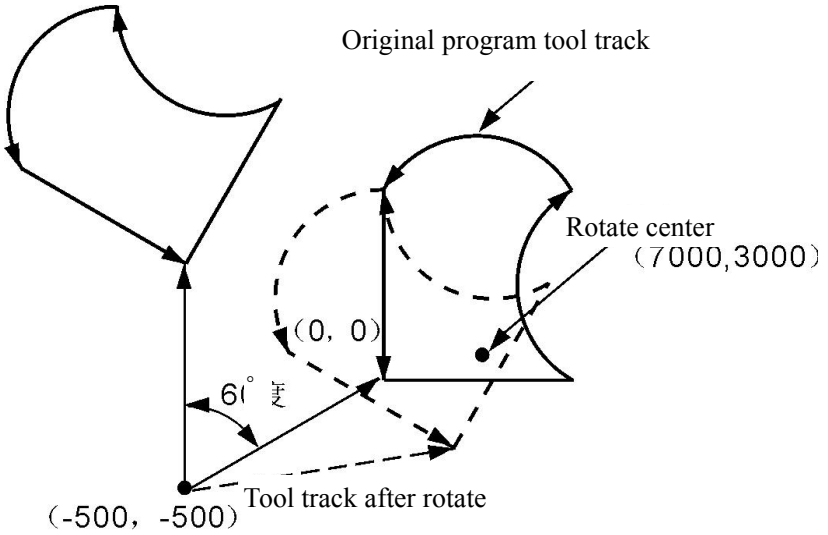
R word is used for pointing out rotate angle。



To specify plane and select code(G17 G18 or G19) before the G68 code program segment, select the code cannot be specified in the mode of rotating coordinate system. Cancel the mode G69 code must occupy a program segment separately. After rotating the coordinate system to carry out tool radius compensation, tool length compensation, tool offset and other compensation.

In the mode of rotating coordinate system, the G code (G28 G26 G31 G30 etc) which is related with backing to the datum point and those related with the the G code (G52, G54/G59) of coordinate system cannot be specified, if need these G codes, must specify after canceling coordinate rotation mode. The first moving instruction must use absolute value instruction after the coordinate system rotation cancel G69, if use incremental instruction, it will not to carry out correct motion.

Example:



N1 G01 X-500 Y-500 F20000 G17;
N2 G68 X700 Y300 R60;
N3 G90 G01 X0 Y0 F20000;

```
N4 G91 X1000
N5 G02 Y1000 R1000
N6 G03 X-1000 I-500 J-500;
N7 G01 Y-1000
N8 G69
N9 G90 X-500 Y-500
N10 M02;
```

(14)Return Reference(G28-G288G301-G308)

Return Reference instruction means tool go to reference point according to appointed axis.

Format：	G28 X(U)_ Y(V)_ Z(W)_ A_ B_	;ZXYAB return to reference
	G281	;only X return to reference
	G282	;only Y return to reference
	G283	;only Z return to reference
	G284	;only A return to reference
	G285	;only B return to reference
	G286	;only C return to reference
	G287	;only Xs return to reference
	G288	;only Ys return to reference
	G301	;X axis return to zero
	G302	;Y axis return to zero
	G303	;Z axis return to zero
	G304	;A axis return to zero
	G305	;B axis return to zero
	G306	;C axis return to zero
	G307	;Xs axis return to zero
	G308	;Ys axis return to zero

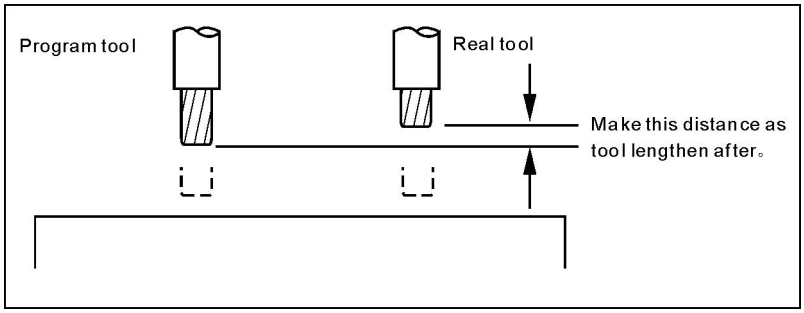
Pay attention: Should clear tool radius compensation and tool length compensation before the execution of the instruction

(15) Tool length compensation instruction(G43/G44/G49)

The difference between programming tool length and actual using tool length can be compensated with this function so that not to adjust the program. Use G43 or G44 to specify the direction of offset, import the corresponding H code address to select tool length offset value from the offset table.

Format：

G43 H_ ;Add tool length compensate。
G44 H_ ;subtract tool length compensate。
G49 or H0 ;cancel tool length compensate.



Example: N0000 G43 H2 X10 (H2 value is 5)
N0010 G44 H3 X20 (H3 value is 10)

Executing first section,tool length add 5. Executing second section,tool length subtract 10(real running is 10+5=15).

(16) Offset of tool radius instruction(G45/G46/G47/G48)

Used for processing groove of the workpiece, programming according to the size of workpiece drawing, the instruction line with the one of this set of instruction, then it can work out correct production in different tool radius.

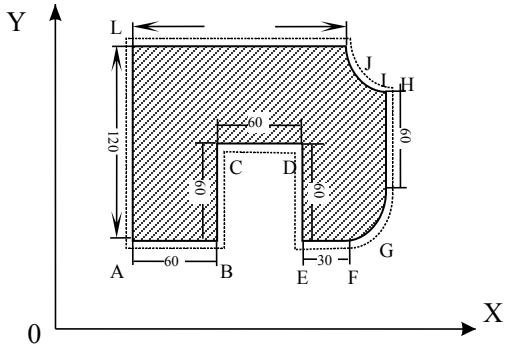
Format： G45 T_ D_ ;Add one radius。
G46 T_ D_ ;subtract one radius。
G47 T_ D_ ;Add two radius。
G48 T_ D_ ;subtract two radius.

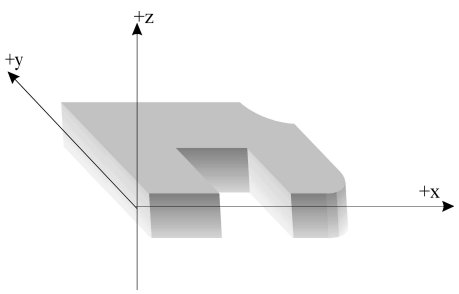
Note:The instruction of increase or decrease the tool radius is increasing or decreasing one or two tool radius parameter values with T to number in the execution. And be used with XY (G17) plane of the G00, G01, G02, G03 instruction. Can not be used with tool radius compensation instruction (G41,G42)

Increase or decrease one or two tool radius in the direction of axis for G00 and G01.

For G02 and G03 is increasing or decreasing one or two tool radius in the direction of arc radius.

G45/G46/G47/G48 are non-modal instructions.





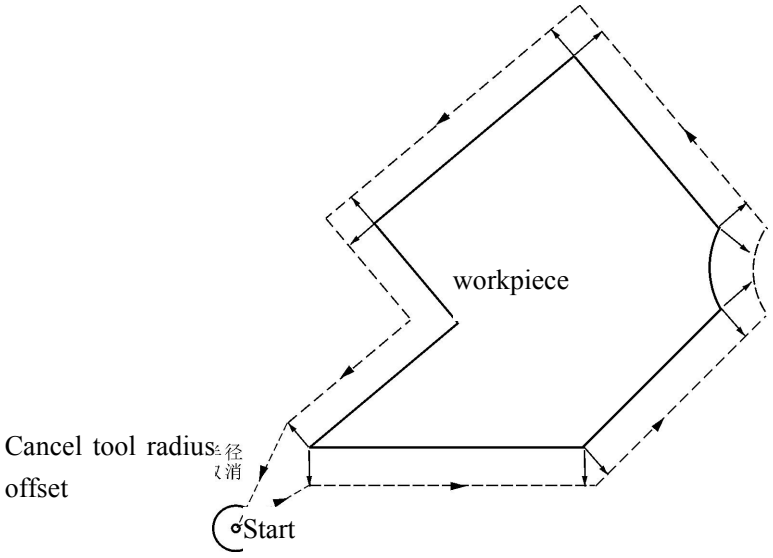
Program as follows:

```
N0000 G01 Z-20 F400 G91          ; Z point starts
N0010 G46 T01 X55 Y55            ; Point location to A point, X and Y axis both
decrease one tool radius(T01)
N0020 G47 G01 X60 F200           ; A-B X axis increase two tools radius
N0030 Y60                        ; B-C length not change
N0040 G48 X60                    ; C-D X axis decrease two tools radius
N0050 Y-60                       ; D-E length not change
N0060 G45 X30                    ; E-F X axis increase one tools radius
N0070 G45 G03 X30 Y30 R30        ; F-G CCW radius increase one tool radius
N0080 G45 G01 Y60                ; G-H Y axis increase one tool radius
N0090 G46 X0                     ; H-I Move for one tool radius along the
negative direction of X axis
N0100 G46 G02 X-30 Y30 R30       ; I-J CW radius decrease one tool radius
N0110 G45 G01 Y0                 ; J-K Move for one tool radius along the
positive direction of Y axis

N0120 G47 X-150                  ; K-L X axis increase two tools radius
N0130 G47 Y-120                  ; L-M Y axis increase two tools radius
N0140 G46 X-55 Y-55              ; M-N X and Y axis both decrease one tool
radius
N0150 G26                        ; Program backs to the starting point to end
N0151 M02
```

(17) Tool radius compensate instruction(G40/G41/G42)

When the tool is moving,tool track can offset a radius。In order to offset a radius，CNC establish offset vector whose length equal tool radius。Offset vector is vertical to tool track。Completed machining，it needs to cancel tool radius compensation。

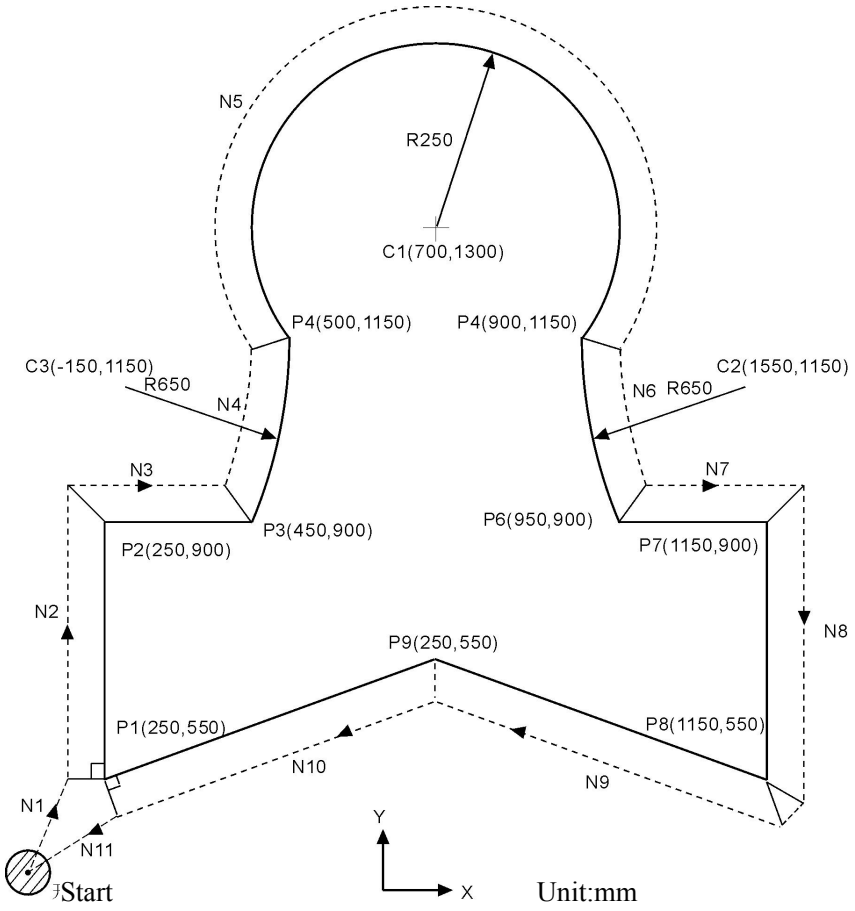


Format：G40 (mode，original) ;Cancel compensation。 G41 T_ D_ (mode) ;tool locate to Left offset of workpiece 。 G42 T_ D_ (mode) ; tool locate to Right offset of workpiece。

Note：

Tool radius compensation establish and cancel have two type: A type and B type， which can set in other parameter。 Furthermore, Tool radius compensation establish and cancel must be executed in line section。

For example:



G54 X0 Y0 Z0;……

N1 G90 G17 G00 G41 T15 D2 X250.0 Y550.0; establish compensation

N2 G01 Y900.0 F150; from P1 to P2

N3 X450.0; from P2 to P3

N4 G03 X500.0 Y1150.0 R650.0; from P3 to P4

N5 G02 X900.0 R-250.0; from P4 to P5

N6 G03 X950.0 Y900.0 R650.0; from P5 to P6

N7 G01 X1150.0; from P6 to P7

N8 Y550.0; from P7 to P8

N9 X700.0 Y650.0; from P8 to P9

N10 X250.0 Y550.0; from P9 to P1

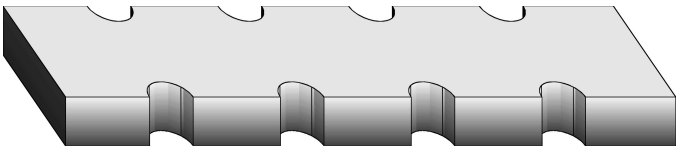
N11 G00 G40 X0 Y0; cancel compensation

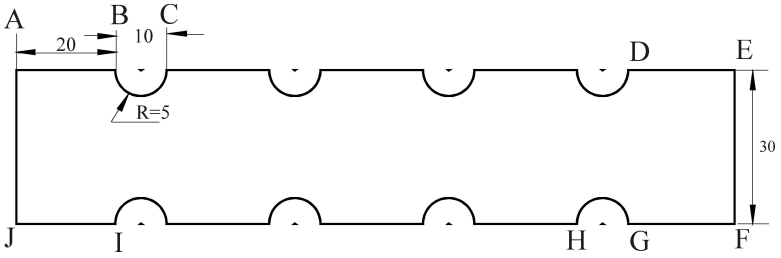
(18) Program circulation instruction (G22--G800)

G22 is program circulation instruction, G800 is an instruction to end circulate. But G22 must be used with G800 for repeated processing. L means circulation times, the range is 1-99999.The circulation instruction can nest.

Format：	G22 L2	;begin
	:	
	:	;circulating
	:	
	G800	;end

For example





Program as follows:

```
 N0000 G17 G90 X0 Y0 F250 M03     ;
N0001 G91 G01 Z-10
N0010 G22 L4                      ;
N0020 G01 X20                     ;
N0030 G03 X10 I5 J0 Y0          ;
N0040 G800                        ;
N0050 G01 X20                     ;D-E line
N0060 Y-30                       ;E-F line
N0080 G22 L4                     ;circulation begin
N0090 G01 X-20                    ;F-G line
N0100 G03 X-10 I-5                ;G-H counter arc
N0110 G800                       ;circulation end
N0120 G01 X-20                    ;I-J line
N0130 G01 Y30                    ;J-A line
N0140 G26                        ;go back to program begin point
N0150 M02                         ;over
```

(19) Accurate localization/Continual way processing (G60/G64)

According to requirement of processing， we can set program section connection way by the G60/G64 instruction.

Format：G60 ; accurate stop (mode)
G64 ; continue section（mode，original）

(20) Circle instruction（G73、G80～G89）

Using Circle instruction, we can shorten the program length,make the program more simple.

Circle instruction table

G code	Feed method	Motion in the bottom of hole	withdraw	application
G73	Intermission feed	No	Rapid move	High speed drill deep hole
G80	Cutting feed	No	No	Cancel fixed cycle
G81	Cutting feed	No	Rapid move	Drill cycle
G82	Cutting feed	Stop	Rapid move	Drill cycle
G83	Intermission feed	No	Rapid move	Drill deep hole cycle
G85	Cutting feed	No	Cutting feed	Bore hole cycle
G86	Cutting feed	Spindle stop	Rapid move	Bore hole cycle
G87	Cutting feed	Spindle stop, orient	Rapid move	Bore hole cycle
G76	Cutting feed	Spindle stop, orient	Rapid move	Bore hole cycle
G89	Cutting feed	Spindle stop	Cutting feed	Bore hole cycle
G74	Cutting feed	Stop-Spindle rotate CW	Cutting feed	Left tap cycle
G84	Cutting feed	Stop-Spindle rotate CCW	Cutting feed	Right tap cycle

Cycle instruction is consist of six motions

Motion 1 location of X and Y axis

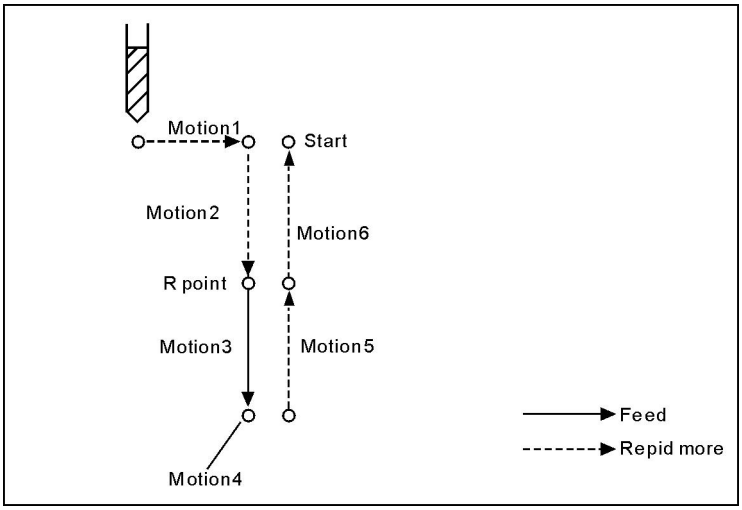
Motion 2 rapid move to R point

Motion 3 machining hole

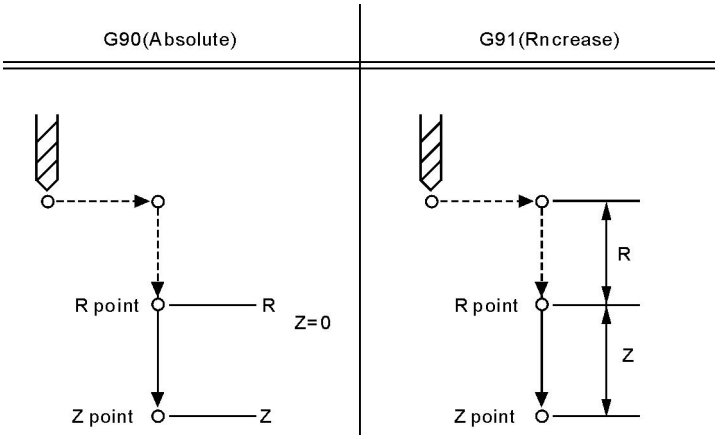
Motion 4 action in the bottom of hole

Motion 5 withdraw to R point

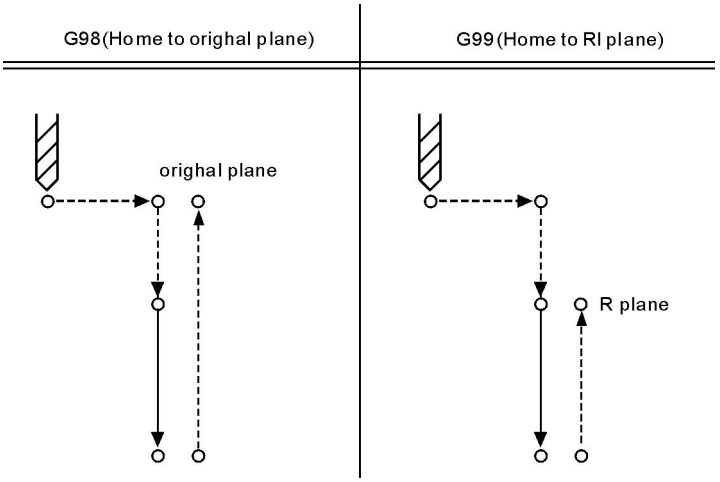
Motion 6 rapid move to original point



The difference of G90 and G91 as follow:



The difference of G98 and G99 as follow:



Use the L word to set cycle time, the maximum value is 9999, the default value

is 1;

Orientation plane is determined by G17(XY)/G18(ZX)/G19(YZ).

(21) High speed drill deep hole(G73)

This cycle execute high speed drilling deep hole until reaching to bottom, at the same time, remove the cutting trifling from hole.

Format： G73 X_Y_Z_R_Q_F_L_ ;

X_Y_:hole position data

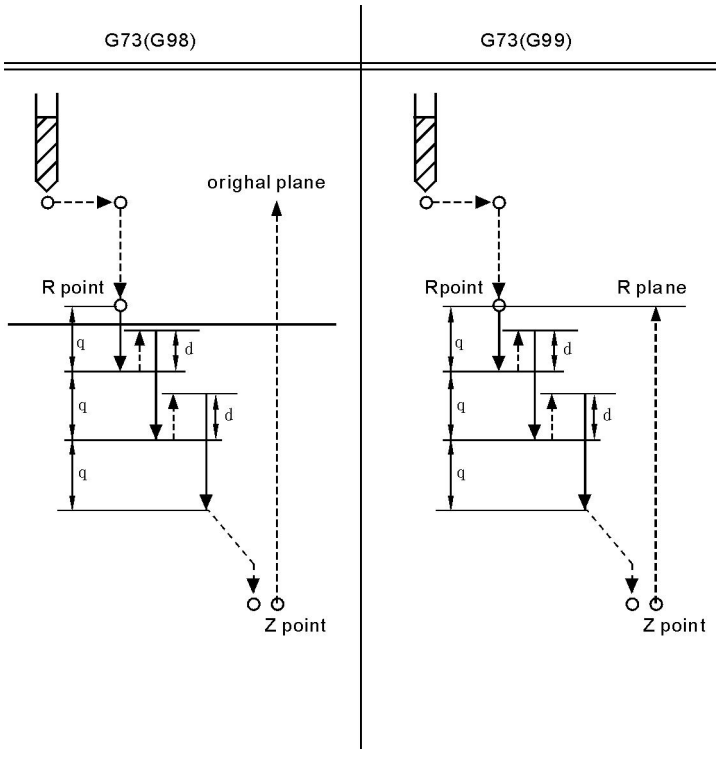
Z_:the distance(G91) or coordinate(G90) from R point to hole bottom

R_: the distance(G91) or coordinate(G90) from original point to R point

Q_:cutting depth every time

F_:cutting speed

L_:repeated times



Note:High speed drilling cycle along the Z axis to carry out intermittent feeding, when using this cycle, the swarf can be easily discharged from the hole and can set the

smaller value when backing. This allows to carry out drilling effectively. No.1 parameter in "N"(Processing) parameter is to set the value of the tool withdrawal(d).

Using auxiliary function to rotate spindle before specifying G73 (M code).

Set the offset of tool length(G43 G44 or G49) in changeless cycle which is located to R point to plus offset, the offset of tool radius is ignored.

For example:

M3 S2000 Spindle starts rotating

G90 G99 G73 X300. Y-250. Z-150. R-100. Q15. F120. Drill the fist hole and return to R point

Y-550.; Drill the second hole and return to R point

G98 Y-750.; Drill the third hole and return to R point

G80

M30; The end

(22) Drilling cycle, point drilling cycle(G81)

The cycle is used for normal drilling, execute the feeding cut to hole bottom. Then, the tool moves fast return form hole bottom.

Format：G81 X-Y-Z-R-F-L- ；

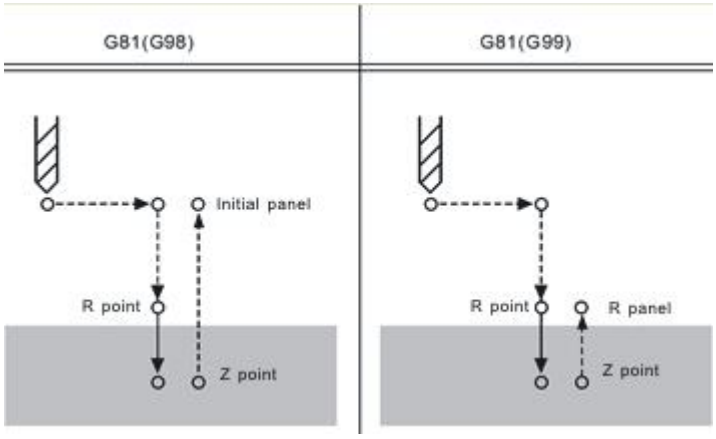
X-Y-：Data of hole position

Z-：Distance form R point to hole bottom（G91）or coordinate value（G90）

R-：Distance form initial point to R point（G91）or coordinate value（G90）

F-：Cutting speed

L-：Repeated times



Note: Move fast to R point along the X and Y axis location, execute drilling process from R point to Z point, then tool returns with rapid move.

Use auxiliary function M code to rotate spindle before specifying G81

Locate to R point plus the offset when specifying the offset of tool length(G43 G44 or G49) in fixed cycle, the offset of tool radius is ignored.

Example:

M3 S2000; spindle begins to rotate

G90 G99 G81 X300 Y-250 Z-150 R-100 F120; drill the first hole, and then return to R point

Y-550; drill the second hole, and then return to R point

G98 Y-750; drill the third hole, and then return to panel of the initial position

G80

M30; end

(23) Drill cycle, bore cycle(G82)

Format：G82 X-Y-Z-R-P-F-L- ;

X-Y-: hole position data

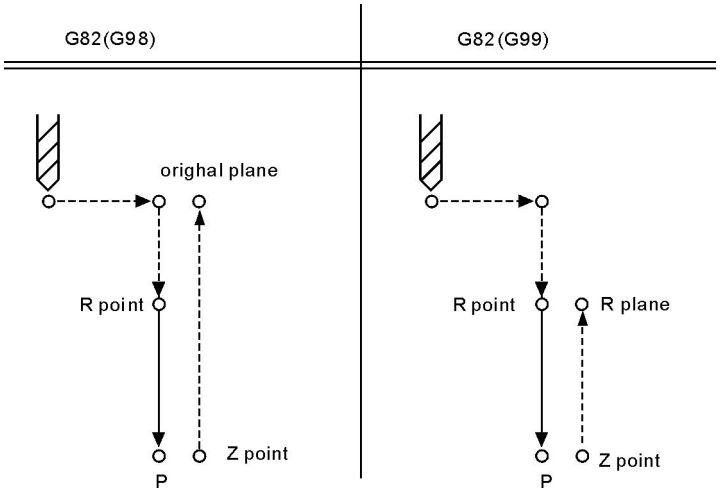
Z-: the distance(G91) or coordinate(G90) from R point to hole bottom

R-: the distance(G91) or coordinate(G90) from original point to R point

P-:pause time

F-:cutting speed

L-:repeat time



Note:

For example：

M3 S2000

G90 G99 G82 X300. Y-250. Z-150. R-100. P1000 F120.

Y-550.;

G98 Y-750.;

G80

M30;

(24) Intermission drill cycle (G83)

Format ： G83 X_ Y_ Z_ R_ Q_ F_ L_ ;

X_ Y_ : hole position data

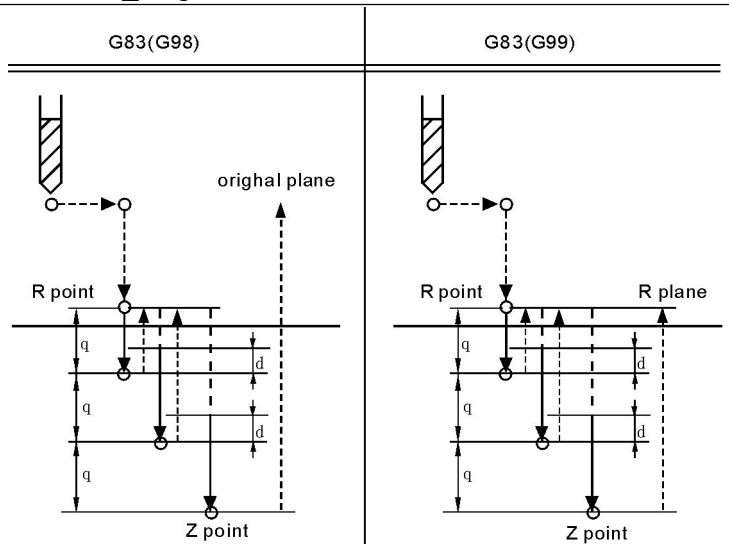
Z_ : the distance(G91) or coordinate(G90) from R point to hole bottom

R_ : the distance(G91) or coordinate(G90) from original point to R point

Q_ :cutting depth every time

F_ :cutting speed

L_ :repeated times



Note: Q means the cutting depth of each cutting feed, it must be specified in the incremental value. Cutting feed in the second and later must be executed quickly to move to d point to execute again before the end of last drilling, No.2 parameter in processing parameter set the cutting feed d, must specify the positive value in Q, negative is ignored.

Use the auxiliary functions M code to rotate spindle before specifying G83.

For example：

M3 S2000

G90 G99 G83 X300. Y-250. Z-150. R-100. Q15. F120.

Y-550.;

Y-750.;

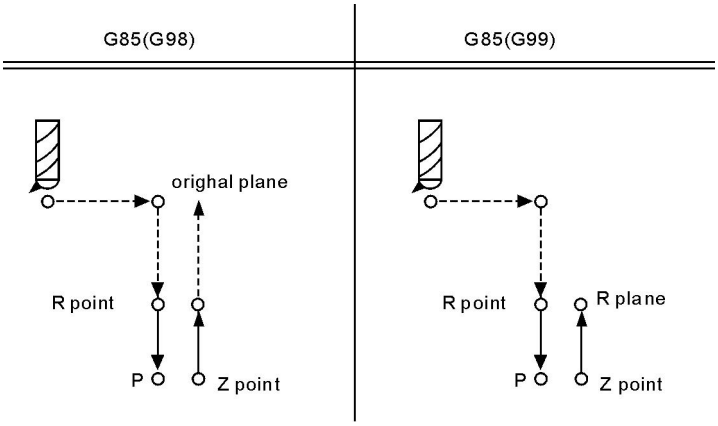
G98 Y-600.;

G80

M30;

(25) Boring cycle(G85)

Format：G85 X_ Y_ Z_ R_ F_ L_ ;
X_ Y_: Hole position data
Z_ : the distance(G91) or coordinate(G90) from R point to hole bottom
R_ : the distance(G91) or coordinate(G90) from original point to R point
F_ :cutting speed
L_ :repeated times



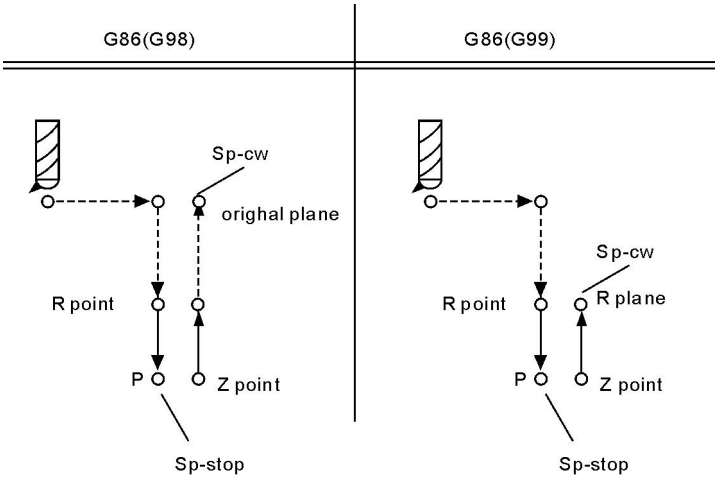
Note:Along the X and Y axis to locate, move fast to R point and carry out boring from R point to Z point, carry out cutting feed and return to R point when arriving the bottom of hole.

For example：

```
M3 S100
G90 G99 G85 X300. Y-250. Z-150. R-120. F120.
Y-550;
Y-750;
G98 Y-600;
G80
M30;
```

(26) Boring cycle(G86)

Format：G86 X_ Y_ Z_ R_ F_ L_ ;
X_ Y_ : hole position data
Z_ : the distance(G91) or coordinate(G90) from R point to hole bottom
R_ : the distance(G91) or coordinate(G90) from original point to R point
F_ :cutting speed
L_ :repeated times



Note:Along the X and Y axis to locate, move fast to R point and carry out boring from R point to Z point. The tool will return quickly when spindle stopping at the bottom of hole.

For example：

M3 S2000
G90 G99 G86 X300. Y-250. Z-150. R-100. F120.
Y-550.;
Y-750.;
G98 Y-750.;
G80
M30;

(27) Boring cycle, back boring cycle(G87)

This instruction only effect in machining center, because the spindle needs oriented stop function, so the spindle must with the locating function (output M61, check M22). Accuracy boring cycle to bore the precise hole. The spindle is stop and the cutting tool leaves the processing surface and return according to No.4 parameter in processing parameter when reaching the bottom of the hole.

Format: G87 X_ Y_ Z_ R_ Q_ P_ F_ L_

X_ Y_: The data of hole

Z_:the distance(G91) or coordinate(G90) from R point to hole bottom

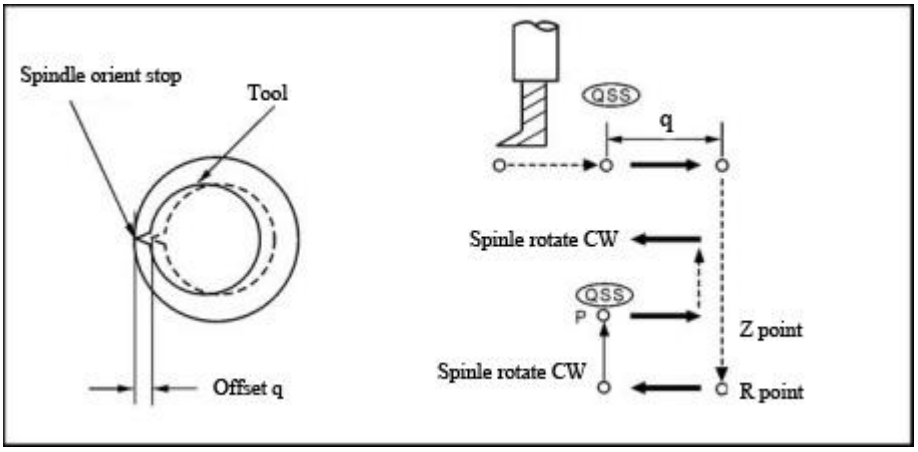
R_:the distance(G91) or coordinate(G90) from original point to R point

Q_:The offset of hole bottom, the direction is set through No.4 parameter in processing parameter

P_: Pause time, unit:s

F_:Cutting speed

L_:repeated times



Note: After locating along X and Y axis, the spindle stop at the fixed rotating position.The tool move rapidly according to the opposite direction of tool nose(Be set through No.4 No.5 parameter in processing parameter) and orient at the bottom of hole R point. Then, tool move along the direction of tool nose and spindle rotate CW. Bore hole until Z point along the positive direction of Z axis. The spindle stop again at the fixed rotating position, the tool move according to the opposite direction of tool nose(Be set through No.3 No.5 parameter in processing parameter). The tool backs to the original position, tool offset along the direction of tool nose, the spindle rotate CW. Processing the next program segment.

Use M code to rotate spindle before specifying G87.

Locating to the R point and add offset in the fixed cycle when specifying the tool length offset (G43,G44 or G49), the tool radius offset will be ignored.

Example:

N10 M3 S100; Spindle starts rotating

N20 G90 G99 G87 X300 Y-250 Z-150 R-100 Q5; Locate, bore 1 hole, and return to the R point; Orient at the hole bottom and move 5mm

N30 Y-350; Locate, bore 2 hole, and return to the R point

N40 Y-550; Locate, bore 3 hole, and return to the R point

N50 Y-750; Locate, bore 4 hole, and return to the R point

N60 G80;

N70 M30;

(28) Boring cycle(G89)

Format：G89 X_ Y_ Z_ R_ P_ F_ L_ ;

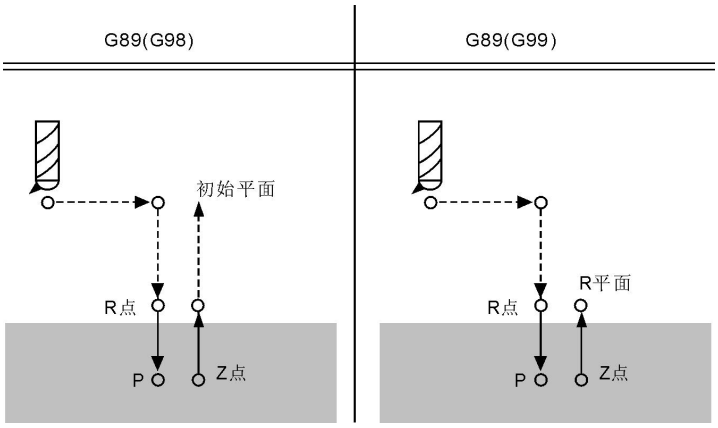
X_ Y_ : hole position data

Z_ : the distance(G91) or coordinate(G90) from R point to hole bottom

R_ : the distance(G91) or coordinate(G90) from original point to R point

P_ :pause time

F_ :cutting speed
L_ :repeated times



Note:It is the same as G85 unless suspending at the bottom of hole.

For example:

M3 S100; Spindle starts rotating

G90 G99 G89 X300 Y-250 Z-150 R-120 P1000 F120; Bore the first hole and return to the R point and suspend 1 second

Y-550; Bore the second hole and return to the R point

Y-750; Bore the third hole and return to the R point

G98 Y-750;

G80

M30; End

(29) Accuracy boring cycle(76)

This instruction only effect in machining center, because the spindle needs oriented stop function, so the spindle must with the locating function (output M61, check M22). Accuracy boring cycle to bore the precise hole. The cutting tool leaves the processing surface and return according to No.3 parameter in processing parameter.

Format: G76 X_ Y_ Z_ R_ Q_ P_ F_ L_ ;

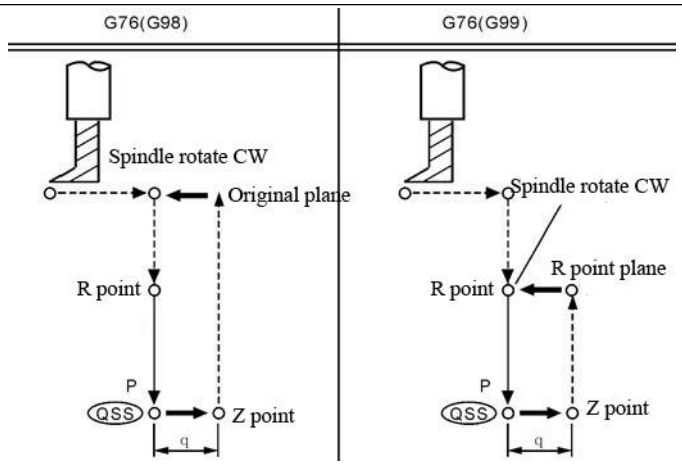
X_ Y_ : hole position data

Z_ : the distance(G91) or coordinate(G90) from R point to hole bottom

R_ : the distance(G91) or coordinate(G90) from original point to R point

Q_:The offset of hole bottom, the direction is set

through No.3 parameter in processing parameter
P_:pause time, unit: s
F_:cutting speed
L_:repeated times



Note: Spindle stop at the fixed rotating position when reaching at the bottom hole and the tool move back along the opposite direction of tool nose. To make sure the processing surface will not be destroyed, to realize accuracy and effective boring process. The oriented angle of spindle is set by No.5 parameter in processing parameter, use M code to rotate spindle before specifying G76.Locating to the R point and add offset in the fixed cycle when specifying the tool length offset (G43,G44 or G49), the tool radius offset will be ignored.

Example:

N10 M3 S100; Spindle starts rotating

N20 G90 G99 G76 X300 Y-250 R-100 Q5; Locating, bore the first hole and return to the R point; The hole bottom orient to move 5mm

N30 Y-350; Locating, bore the second hole and return to the R point

N40 Y-550; Locating, bore the third hole and return to the R point

N50 Y-750; Locating, bore the fourth hole and return to the R point

N60 G80

N70 M30

(30) Left tap cycle(G74)

Tapping has two kinds of methods: Tracking the spindle encoder (P411(No.211 in bus system)=2, spindle must assemble encoder) and interpolation of Z axis and spindle servo (P405(P205 in bus system)=0, P410(P210 in bus system)=95, P411(P211 in bus system)=3). No.404~No.413(P204~P213 in bus system) parameter in axis parameter to set.

Pay attention:

When the spindle and encoder drives not as 1:1, please modify the No.412 No.413(No.212 No.213 in bus system) parameter in axis parameter;

412(No.212 in bus system), the number of spindle teeth (requirement: less than or equal to the number of encoder teeth, must match our keysets when greater than the number of encoder teeth);

413(No.213 in bus system), the number of encoder teeth

The loop is executed with tapping left, in the left tapping cycle, the spindle rotates CW when arriving at the bottom of hole.

Format：G74 X_ Y_ Z_ R_ P_ K_ S_ L_ ;

X_ Y_ : hole position data

Z_ : the distance(G91) or coordinate(G90) from R point to hole bottom

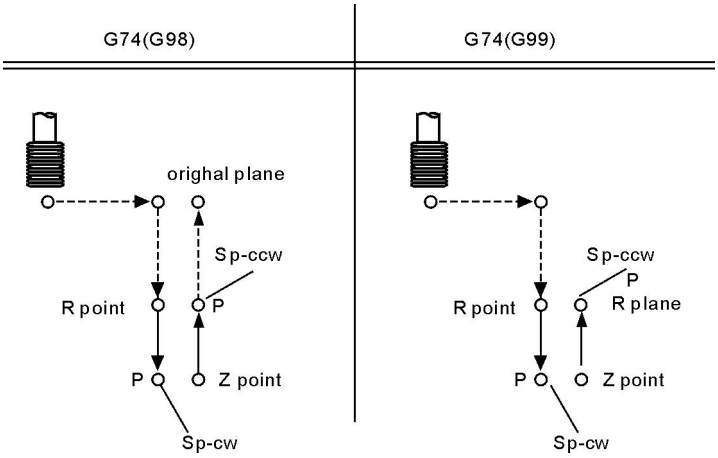
R_ : the distance(G91) or coordinate(G90) from original point to R point

P_ : pause time unit:s

K_ : screw pitch

S_ : spindle rotate speed

L_ : repeated times



Note: The speed of coordinate axis is determined by speed of spindle and screw pitch when processing thread,it is not a matter with speed F. The system will limit the speed within maximum feeding speed.

Spindle override switch and feeding axis override switch are invalid when processing thread.

Should specify the screw pitch K value in every processing program segment of thread, otherwise not through compile.

Use spindle CCW rotation to carry out tapping. Should process a negative thread in order to return spindle CW rotation when arriving at the bottom of hole. In the left tapping period, feeding suspension does not stop lathe until the return action

completed.

Use auxiliary function to rotate spindle CCW before specifying G74. When the spindle is pulse controlling mode without using auxiliary functions M code.

For example：

```
N1 M4 S100
N2 G90 G99 G74 X300. Y-250. Z-150. R-100. K5 S100
N3 Y-550 K5
N4 G98 Y-750. K5;
N5 G80;
N6 M30;
```

(31) Right tap cycle(G84)

The same as G74 unless direction.

Format：G84 X_ Y_ Z_ R_ P_ K_ L_ ;

X_ Y_ : hole position data

Z_ : the distance(G91) or coordinate(G90) from R point to hole bottom

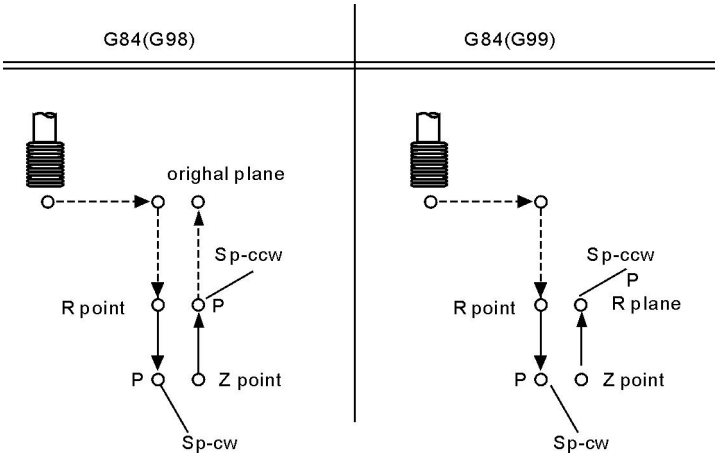
R_ : the distance(G91) or coordinate(G90) from original point to R point

P_ :pause time

K_ :screw parameter

S_ :spindle rotate speed

L_ :repeated times



For example：

```
M3 S100
G90 G99 G84 X300 Y-250 Z-150 R-120 P300 K5 S100
Y-550. K5;
G98 Y-750. K5;
G80
M30;
```

(32) Cancel cycle instruction (G80)

Cancel cycle instruction.

Format：G80 ;

Note: Cancel all cycle instruction and execute normal operation.

(33) Pole coordinate instruction(G15/G16)

Pole coordinate instruction inquire user provide radius and angle,Radius may use absolute and increase type(G90, G91),Angle only use absolute type(G90).

Format ： G15 Cancel Pole coordinate; (G17/G18/G19) (G90/G91) G16 IP_ ;establish

Note:

1.G17/G18/G19 specify the panel of pole coordinate instruction.

2.G90 specify the zero point of workpiece coordinate system as the origin of pole coordinate, measure radius from this point.

3.G91 specify current position as the origin of pole coordinate system, measure radius from this point.

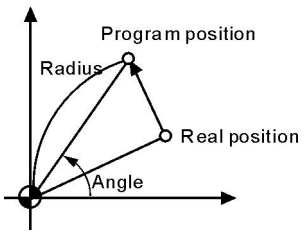
4.IP_ specify the panel axis address and value of pole coordinate system panel selection.

First axis: Radius value of pole coordinate.

Second axis: Value of pole angle.

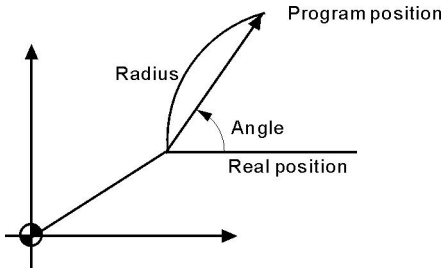
5.G90 set the zero point of workpiece coordinate system as origin of pole coordinate system:

Use programming instruction of absolute value to specify the radius(The distance between zero point and programming point).When using local coordinate system(G52), the origin of local coordinate system changes into the center of pole coordinate system, the angle uses absolute value. As the follow shown:

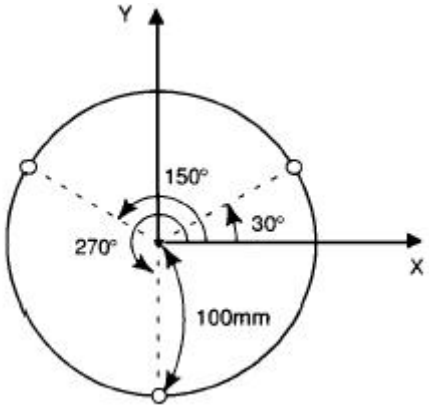


6.G91 set the current position as the origin of pole coordinate system:

Using programming instruction of incremental value to specify the radius(The distance between the current position and programming point).The angle use absolute value.As the follow shown:



For example:



G17 G90 G16 X0 Y0; Specify the pole coordinate and select XY plane to set the original point of pole coordinate and reference point of workpiece coordinate

G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0; specify 100mm distance and 30 degree angle

Y150.0; specify 100mm distance and 150 degree angle

Y270.0; specify 100mm distance and 270 degree angle

G15 G80; Cancel pole coordinate

(34) Switch millimeter and inch(G20/G21), feeding mode(G94/G95)

Format： G20 ; inch; G21 ; millimeter; G94 ; feeding each minute G95 ; feeding each rotation

Note:The G code must be compiled in the beginning of the program, using separate program segment to specify before setting the coordinate system.Switch the unit of input data into minimal inch or millimeter after G code of switching inch or millimeter specifying,the angle of data input unit keeps unchanged, change the units of value as follows after switching the inch or millimeter:

- The feeding speed is specified by F code

- Position instruction
- Offset value of workpiece zero point
- Compensation value of tool
- The unit of manual pulse generator
- The distance in incremental feeding

(35) Go back the starting point of program(G26/G261-G268）

Format ：	G26	; ZXY all go back.
	G261	; X go back.
	G262	; Y go back.
	G263	; Z go back.
	G264	; A go back.
	G265	; B go back.
	G266	; C go back.
	G267	; Xs go back.
	G268	; Ys go back.

Note：G26 motion is according to G00 mode.

(36) Remember the current point(G25)

Format：G25 ； To remember the coordinate of X Y Z A B C Xs Ys

(37) Return to the memorial point(G61/G611-G618)

Format:	G61	； Return to X Y Z of memorial point
	G611	； Return to X of memorial point
	G612	； Return to Y of memorial point
	G613	； Return to Z of memorial point
	G614	； Return to A of memorial point
	G615	； Return to B of memorial point
	G616	； Return to C of memorial point
	G617	； Return to Xs of memorial point
	G618	； Return to Ys of memorial point

Note: The backing mode is G00.

(38) Check skip(G31、G311)

Format：G31 X_ Y_ Z_ A_ B_ C_ Xs_ Ys_ F_ P_ ;No alarm

G311 X_ Y_ Z_ A_ B_ C_ Xs_ Ys_ F_ P_ ;alarm

P: N line+(X00/X39+1000 or 2000)， 1000 means availability skip,2000 mean invalidation skip.

For example：G31 X50 Z100 F100 P331022 ;if X22 availability then go to N33.

G311 X50 Z100 F100 P2021 ;if X21 invalidation then go to next line.

(39) Automatical beveling (I) and smoothing(R)

The acquiescent panel of milling is G17

Format for G17：

G01(G00) X I automatical beveling, the coordinate in the next program segment must be G01(G00) Y.

G01(G00) Y I automatical beveling, the coordinate in the next program segment must be G01(G00) X。

G01(G00) X R automatical smoothing, the coordinate in the next program segment must be G01(G00) Y。

G01(G00) Y R automatical smoothing, the coordinate in the next program segment must be G01(G00) X。

Format for G18：

G01(G00) X I automatical beveling, the coordinate in the next program segment must be G01(G00) Z。

G01(G00) Z I automatical beveling, the coordinate in the next program segment must be G01(G00) X。

G01(G00) X R automatical smoothing, the coordinate in the next program segment must be G01(G00) Z。

G01(G00) Z R automatical smoothing, the coordinate in the next program segment must be G01(G00) X。

Format for G19：

G01(G00) Y I automatical beveling, the coordinate in the next program segment must be G01(G00) Z。

G01(G00) Z I automatical beveling, the coordinate in the next program segment must be G01(G00) Y。

G01(G00) Y R automatical smoothing, the coordinate in the next program segment must be G01(G00) Z。

G01(G00) Z R automatical smoothing, the coordinate in the next program segment must be G01(G00) Y。

Pay attention:

1.The address of I and R are specified with radius model. The running distance of this line and the next line must be greater than the length of beveling or radius of smoothing, otherwise the system will decrease the length of beveling or radius of smoothing to minimal running distance of this line and the next line automatically.

2. The two adjacent lines must be 90 degrees.

For example:

```
0 G54 G0 X-50 Y-50 Z20
N1 M03 S500
N2 G01 G42 D01 X0 Y0 F200
N3 G01 Z-5
N4 X100 I4                    ; Beveling4x4
N5 Y40 R6                     ; SmoothingR6
N6 X47 R5                     ; SmoothingR5
N7 Y70 I3                     ; Beveling3x3
N8 X15

N9 X0 Y40
N10 Y0
N11 G0 X-50 Y-50 G40
N12 Z50
N13 M30
```

(40) The calling program(M97 M98 M99)

Unconditional jump

The line running unconditional jump to the line which is specified by P; P4 stands for using four field of digital specify the program to the entrance line of the calling main program (mark line).

Subroutine call

In this system the subroutine should be an independent program.

M98 P L unconditional call subroutine instruction. P is to specify the name and path of subroutine call, L refers to the calling times address of subroutine.

The M98 instruction can be omitted without writing, format: PP file name, the file name can be hidden files, the first character of hidden files must be "HIDEFILE" at the beginning. Such as the file "HIDEFILE01", this program in the program area is not displayed, can use the instruction M98 PHIDEFILE01 or M98 P*01 or PP*01 or PPHIDEFILE01 when calling.

For example:

P sub/1390 means subroutine is tmp/NC/sub/1390

Note:

1.tmp/NC/ is the system's default path, sub is a folder for the following

2.The subroutine must be a independent program.

3.Method of the main program in USB calls the subroutine in USB: P[or P].
For example:

M98 P[A1234 means calling the subroutine A1234 in USB;

M98 P]SS12 means calling the subroutine SS12 in USB;

PP[FFDE means calling the subroutine FFDE in USB;

It needs to write the path of file if call the subroutine in folder of USB.

There must be space in front of L(Subroutine calling times). Return to the next program segment of main program when subroutine running to the end.If the program contains a fixed sequence or repeated pattern, then the sequence or pattern can be compiled to subroutine to save in memory storage in order to programme easily, the subroutine can be called by main program which is also can be called by another subroutine.

M99 is an instruction of ending subroutine return, must have this instruction to end the subroutine.

Pay attention:

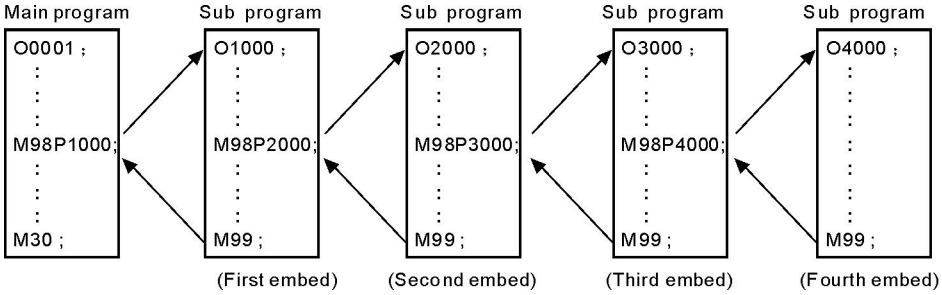
1) M99 in the main program is the same as M02;

2) M99 with P in the main program is the same as M97;

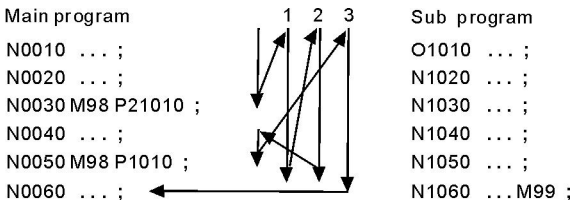
3) M99 returns to main program call in subroutine is in the next line;

4) M99 with P in the subroutine returns to P program line in main program;

The Sub-program can embedded call as follow :



For example:



The calling instruction can be used for 9999 times in the most.

Conditional wait, jump instruction

The system of M code is used for detecting the external input signal as the condition, as follows:

Conditions wait

M12 M13 instruction are used to detect the input signal M12, M12 in program line is to detect M12 input signal is effective to execute the next program line , M13

means to detect M12 input signal is invalid to execute the next program line.The instruction is in an independent line.

M14 M15 instruction are used to detect the input signal M14, M14 in program line is to detect M14 input signal is effective to execute the next program line, M15 means to detect M14 input signal is invalid to execute the next program line.The instruction is in an independent line.

M16 M17 instruction are used to detect the input signal M16, M16 in program line is to detect M16 input signal is effective to execute the next program line, M17 means to detect M16 input signal is invalid to execute the next program line.The instruction is in an independent line.

M18 M19 instruction are used to detect the input signal M18, M18 in program line is to detect M18 input signal is effective to execute the next program line, M19 means to detect M18 input signal is invalid to execute the next program line.The instruction is in an independent line.

M22 M23 instruction are used to detect the input signal M22, M22 in program line is to detect M22 input signal is effective to execute the next line program, M23 means to detect M22 input signal is invalid to execute the next program line.The instruction is in an independent line.

M28 M29 instruction are used to detect the input signal M28, M28 in program line is to detect M28 input signal is effective to execute the next line program, M29 means to detect M28 input signal is invalid to execute the next program line.The instruction is in an independent line.

Conditional jump

Plus Pxxxx (number of program line) in front of the M12 /M13 /M14 /M15 /M16 /M17 /M22 /M23 /M28 /M29 instruction. Shifting if the condition success, otherwise execute the next.

For example: M14 P0120

When the program running to this line and the system detecting the M14 input signal effectively, program will jump to the 120th line of program (the marking line),execute the next instruction if the M14 input signal is invalid.

(41) Feeding speed F function

It is the mode,actual running speed is the setting speed times the trimming rate of speed,

F is used for specify the processing speed of feeding instruction G01 G02 G03.

The range is 0.01-15000mm/min,feeding speed is Fx trimming speed, F has mode function.

Executing the F instruction at the first, and then execute the motion instruction when the F instruction and motion instruction are in the same line.

(42) T/H/D function

The T/H/D function is means that tool length and radius compensate,which is mode,used by code in program。

The tool code is from T01 to T99,every tool have four tool compensate value， which is length compensation from H1 to H4 and radius compensation from D1 to D4.

(43) Spindle speed S, SS function

The system offers two ways spindle controlling modes.

The first spindle speed is specified by S, the first spindle has two kinds of gear controlling mode:

(1) The first is four gear spindle speed electrical control, output four bits code of step speed change, M41-M44 instruction control corresponds to S01-S04 output code, step speed change. Use No.50 No.51 No.52 No.53 and No.54(No.59 No.60 No.61 No.62 No.63 parameter in bus system) parameter in axis parameter to set the mode of shifting.

(2) The second uses four gears + step-less speed, M41-M44 instruction control, correspond the output S01-S04 code. Use No.42 No.43 No.44 No.45(No.48 No.49 No.50 No.51 in bus system) parameter in speed parameter to set the maximum speed of corresponding gear, use No.50 No.51 No.52 No.53 No.54(No.59 No.60 No.61 No.62 No.63 parameter in bus system) parameter in axis parameter to set the mode of shifting.

Stepless speed,the range is 0-99999, output 0-10V variable-frequency voltage. The output voltage trims x10V of maximum speed of specified spindle.

Second spindle speed is specified by SS, the highest speed is controlled by the No.46 parameter in speed parameter, output 0-10V variable-frequency voltage.

(44) Macroprogram instruction(G65, G66, G67)

1.Input instruction: WAT

Waiting for the input port X is valid or invalid instruction, waif for X, Y, M is valid or invalid.

Format: WAT+(-)X(Y/M)**+(-)X**+(-)X**+(-)X**+(-)X**+(-)X**

Note: "+" to means the input is effective;

"-" means the input is invalid;

Y or M must be only one in the most, X may exist a lots

"X" means the input port X00-X55; see the I/O diagnosis;

2.The output instruction: OUT

Set the output port Y is valid or invalid instruction

Format: OUT +(-)Y

Note: "+" means the output is effective;

"-" means the output is invalid;

"Y" means the output port Y00-Y31; see the I/O diagnosis;

3. Variable and assignment: =

1) #0--#20 local variable: local variables only can be used to store data in macro program, such as a result of operation, when power is off, the local variables are initialized to the empty. The argument assignment to the local variable when calling the macro program.

2) #21--#600 global variables: The meanings are the same in different macro program.

When power is off, the variable #21--#100 is initialized to zero, the variable #101--#600 data is saved not to loss even if the power is off.

3) #1000-- system variable: the system variables are used to change various data when reading the running CNC. For example, the current position and the compensation of tool.

Special note: macro variables #100--#155 and #190--#202 have been used by the system, users can not use.

4) The macro variables #1001--#1099 corresponds the X axis offset value of lathe T1--T99(Unit: micron)

The macro variables #1401--#1499 corresponds the Z axis offset value of lathe T1--T99(Unit: micron)

Could read the value, for example: #200=#1003; To read the X axis offset value of the third tool into macro variables #200.

Could modify the value, for example: #1003=23000; To modify the X axis offset value of the third tool to 23000 micron.

#1003=#1003+50; To increase the X axis offset value of the third tool 50 micron.

5)The I/O variables:

#1800: X00-X07（D0-D7）
#1801: X08-X15（D0-D7）
#1802: X16-X23（D0-D7）
#1802: X16-X23（D0-D7）
#1803: X24-X31（D0-D7）
#1804: X32-X39（D0-D7）
#1805: X40-X47（D0-D7）
#1806: X60-X67（D0-D7）
#1808: Y00-Y15（D0-D15）
#1809: Y16-Y31（D0-D15）

Format:#i=Expression

4.The arithmetic and logic operation

Table:

Function	Format	Note
Definition	#i = #j	
Addition	#i = #j + #k ;	

Subtraction Multiplication Division	#i = #j - #k ; #i = #j * #k ; #i = #j / #k ;	
Sin Asin Cos Acos Tan Atan	#i = SIN(#j) ; #i = ASIN(#j); #i = COS(#j) ; #i = ACOS(#j); #i = TAN(#j); #i = ATAN(#j);	90.5 degrees mean 90 degrees 30 minutes
Square root Absolute value Rounding off Round down Round up Natural logarithm Exponential function	#i = SQRT(#j); #i = ABS(#j) ; #i= ROUND(#j); #i = FIX(#j); #i = FUP(#j); #i = LN(#j); #i = EXP(#j);	
Or Xor And	#i = #j OR #k ; #i = #j XOR #k ; #i = #j AND #k ;	Executing with binary system

5. Unconditional transfer: GOTO N

Transfer to the program line with sequence number appears error when specifying beyond the 1-99999, could use expression to specify the sequence number.

For example: GOTO 5, GOTO#100

6.Conditional transfer: IF (Conditional expression) GOTO or THEN

If the conditional expression specified met, execute this segment; if the conditional expression specified does not meet, execute the next segment.

For example: IF (#100 EQ 2) THEN #100=5

IF (#101 GT 2) GOTO 6

IF (#101 GT 2) GOTO 6

Operation meaning:

EQ equal

NE not equal

GT greater than >

GE greater than or equal

LT less than <

LE less than or equal

7. Cycle: WHILE (conditional expression) DO 1, 2, 3

Specifies a conditional expression in front of WHILE. When the specified conditions are met, execute the program between DO and END. Otherwise, turn to the

program line after END. Cycle of the embed is 3 at the most.

For example:WHILE （#100 LT 3）DO 1

……

WHILE （#103 EQ 5）DO 2

……

WHILE （#200 GE 20）DO 3

……

END 3

……

END 2

……

END 1

8.Non-mode to call macro program:G65

Format: G65 P- L- <A-B-C-..... Argument passing data >

P is the name of macro program, L is the calling times,A B C are argument, the name of argument as follows:

#0->A、#1->B、#2->C、#3->D、#4->E、#5->F、#6->H、#7->I、#8->J、#9->K、#10->M、#11->Q、#12->R、#13->S、#14->T、#15->U、#16->V、#17->W、#18->X、#19->Y、#20->Z.

Special attention: The address G、L、N、Q、P can't be used in argument.

For example:

Main program:9000

G00 X0 Z0

G65 P8000 L1 A5 B6

G0 X0 Z0

M30

Macro program:8000

N1 #2=#0+#1

N2 IF （#2 EQ 10）GOTO 4

N3 GOO X#2

N4 G00 Z#1

N5 M99 ; Return

9.Mode to call macro program:G66 G67

G67 instruction is to cancel G66 instruction.The format is the same as G65.

For example:

Main program:9000

G00 X0 Z0

G66 P8000 L2 A5 B6

A8 B1

A9 B10

G67
M30
Macro program:8000
N1 #2=#0+#1
N2 IF （#2 EQ 10）GOTO 4
N3 GOO X#2
N4 G00 Z#1
N5 M99 ； Return

10.Message dialog

Format: MSG(parameter) or MSG[parameter]; parameter could be string, pause.

Pay attention: The instruction could be used in the normal NC program(non-macro program)

The system change into pause state after appearing the message dialog.

Format: STAF(parameter) or STAF[parameter];parameter could be string, not to be pause.

11.Generate the processing program automatically

1> Build the instruction of open file:

FILEON(parameter) or FILEON[parameter]

Example:

FILEON(AABBCC) or FILEON[AABBCC]

Means to open the file(AABBCC)

2>Instruction of close file:

FILECE means close the current file which is opened, if without the instruction, it will close the file automatically after running the program

3> Add a list of instruction:

FILEWD(parameter) or FILEWD[parameter]

Example:

FILEWD(G54 G0 X0 Z0) or FILEWD[G54 G0 X0 Z0]

Means add G54 G0 X0 Z0 into the file which is opened.

4> Add the current absolute coordinate of the current feeding axis into the file which is opened:

FILEWC

Example:

G0 X0 Z0
FILEON[AABBCC]
FILEWD[G54 G0 X0 Z0]
G1 X45 Z89
FILEWC
G1 X99 Z76
FILEWC

FILECE

Generate a file(AABBCC) under the program directory after processing the program, the content is:

G54 G0 X0 Z0

X45 Z89

X99 Z76

(45)User-defined macro instruction(G101-G170,M880-M889)

Every user-defined G code is corresponding to a macro program ProgramGxxx, the M code is corresponding to a macro program of ProgramUser0 --ProgramUser9, the user cannot programme the macro program in NC system, must edit the macro code in the computer, and then copy into the system.

For example, defines the G152 function: the arc model porous drilling cycle. (must copy the macro program ProgramG152 into system).

Format:G152 Xx Yy Zz Rr Ii Aa Bb Hh Ff;

X: The X coordinate with absolute value or incremental value of center to specify.

Y: The Y coordinate with absolute value or incremental value of center to specify.

Z: Hole depth

R: Approaching fast to the point coordinate

F: Cutting feed speed

I: Radius

A: The angle of the first hole

B: Incremental angle specify(CW when negative)

H: The number of hole

Macro program ProgramG152 as follows:

```
#80=#0
#81=#1
#82=#2
#83=#3
#84=#4
#85=#5
#86=#6
#87=#7
#88=#8
#89=#9
#90=#10
#91=#11
#92=#12
#93=#13
#94=#14
#95=#15
```

```
#96=#16
#97=#17
#98=#18
#99=#19
#100=#20
#30=#4003
#31=#4014
G90
IF[#30 EQ 90] GOTO 1
G53
#98=#5001+#98
#99=#5002+#99
N1 WHILE[#86 GT 0] DO 1
#35=#98+#87*COS[#80]
#36=#99+#87*SIN[#80]
G81X#35Y#36Z#100R#92F#85
#80=#80+#81
#86=#86-1
END 1
G#30 G#31 G80
M99
```

(46) Rough milling grooves in circle G110/G111

Format: G99 G110/G111 X_ Y_ R_ Z_ I_ W_ Q_ K_ V_ E_ D_ F_

Note: Only closed-loop system with absolute type could use this function, the same as G110--G139.

Function: Start from the center of circle, multiple circular arc interpolation with helical mode until processing the circle groove which is programmed.

Note: The relative instruction refers to table 3-2(fixed cycle). G110: Rough milling grooves CCW in circle; G111: rough milling grooves CW in circle;

1. I: Radius of groove in circle, I should greater than the radius of the current tool;

2. W: The first time cutting depth of Z axis direction, the distance from the R plane to downward ,should be greater than 0(if the first time cutting depth is more than the groove bottom, then processing according to the groove bottom);

3. Q: Every increment of cutting depth with direction of Z axis;

4. K: Every increment of cutting width in XY plane, should less than radius of the tool, greater than 0;

5. V: The back distance of the tool when tool finish feeding, should greater than 0;

6. E: The processing allowance in the cycle of rough milling grooves in circle.(greater than 0 or equal to 0, take absolute value if negative)

7. D: The number of tool radius, the range is 0~32, D0 default to be 0. Take out the

current tool radius according to the number which is specified.

Cyclic process: (1) Orient to the position of XY plane;

(2) Rapid move to R point plane;

(3) Cutting down W depth with cutting speed;

(4) According to the K value to increase to mill the I radius circle plane outward from the center.

(5) Z axis returns to the R plane;

(6) X Y axis orient to the center of the circle;

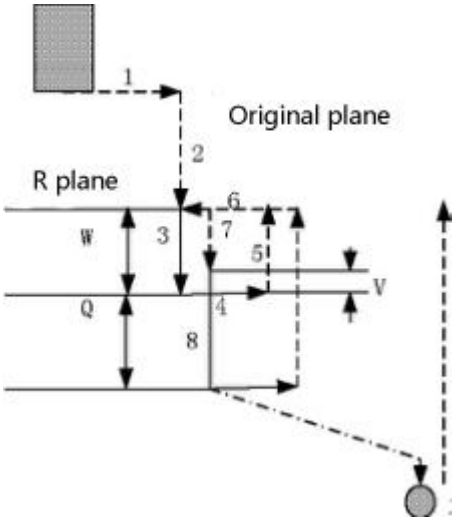
(7) Z axis down to the position which is V distance away from the processing plane.

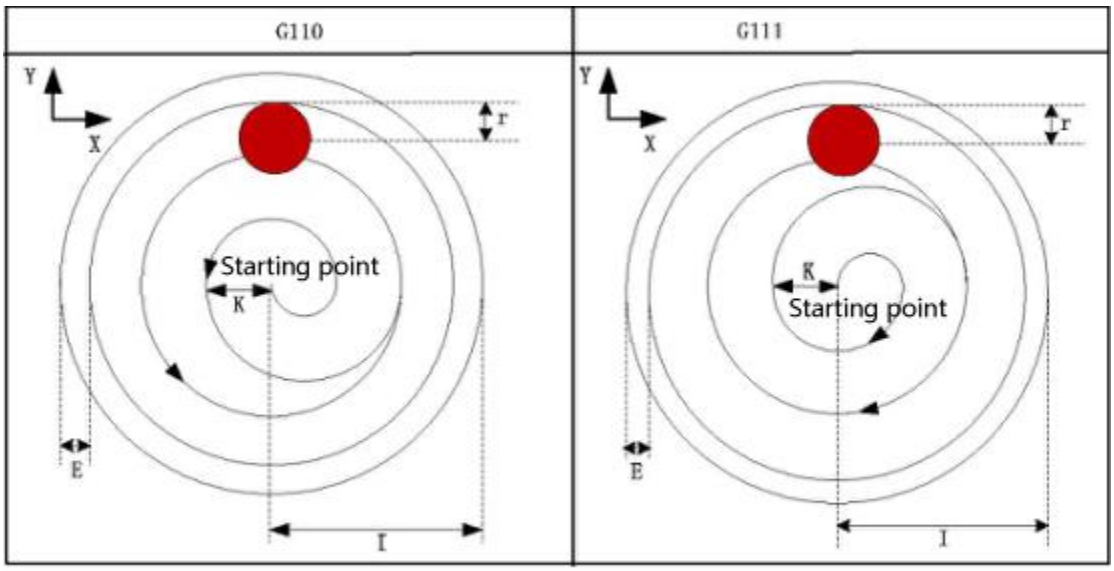
(8) Z axis cutting down with (Q+V) depth;

(9) Cycle (4) ~(8) until finish processing the circle;

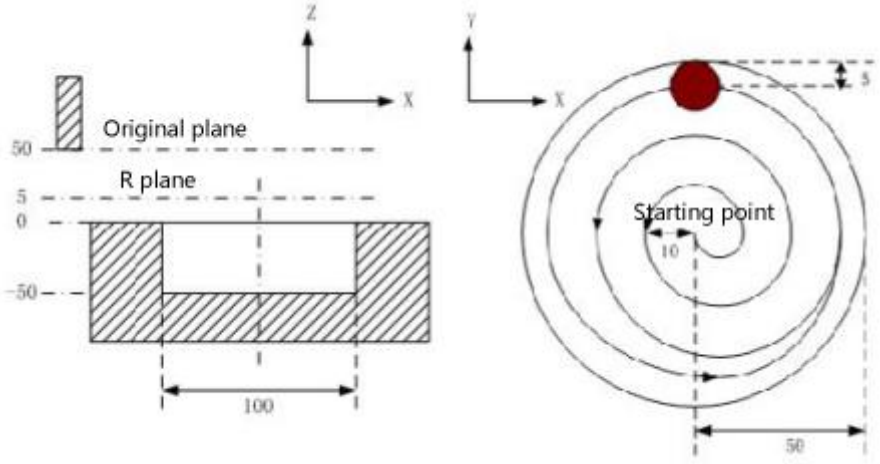
(10) Fixed return to R point.

(11) Return to the orient position(located position) of XY hole. The track:





Example: Use G110 to rough milling a groove of circle:



G90 G00 X50 Y50 Z50; G00 orient

G99 G110 X25 Y25 R5 Z-50 W20 Q10 K10 E0 V10 F800 D1; Cycle to process rough milling grooves in circle D1=5

G80 X50 Y50 Z50; Cancel fixed cycle, return from R point plane

M30;

(47) Accurate milling cycle in whole circle G112/G113

Format: G99 G112/G113 X_ Y_ R_ Z_ I_ J_ D_ F_

Function:The tool mill inside a whole circle with the direction and radius I which is specified, return after the accurate milling.

Note:The relative instruction according to the table 3-2. G112: accurate milling

cycle CCW. G113: accurate milling CW.

I: Circle radius of accurate milling, the range -99999999~99999999 times the minimum unit, take absolute value if negative.

J: The distance between the starting point of accurate milling and the center of accurate milling circle, the range J≦I- the radius value of tool, take absolute value if negative.

D: The number of tool, the range 1~32, D0 default is 0. Take out the radius of the current tool according the number.

Cycle process:(1) Orient to the position of XY plane

(2) Move down to the R point plane

(3) Cutting feed to the hole bottom

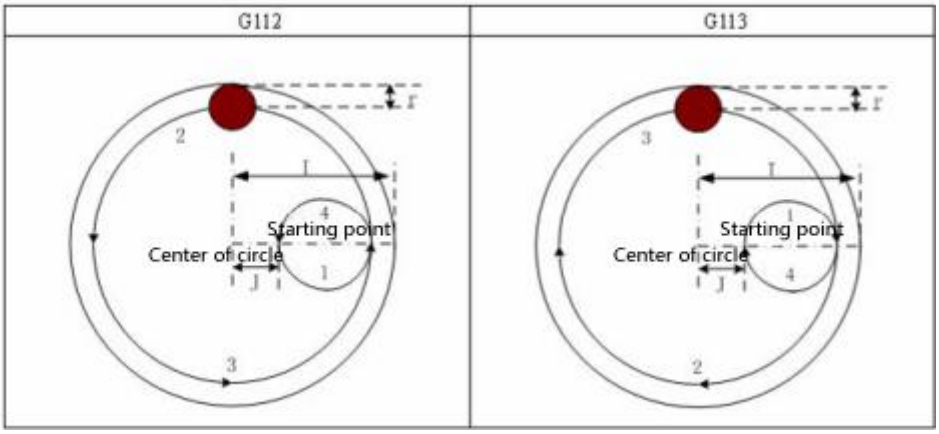
(4) Process the arc interpolation according to the track of transition arc 1

(5) Whole circle interpolation according to the arc 2, arc 3

(6) Arc interpolation to the starting point according to the track of transition arc 4

(7) Fixed to return R point

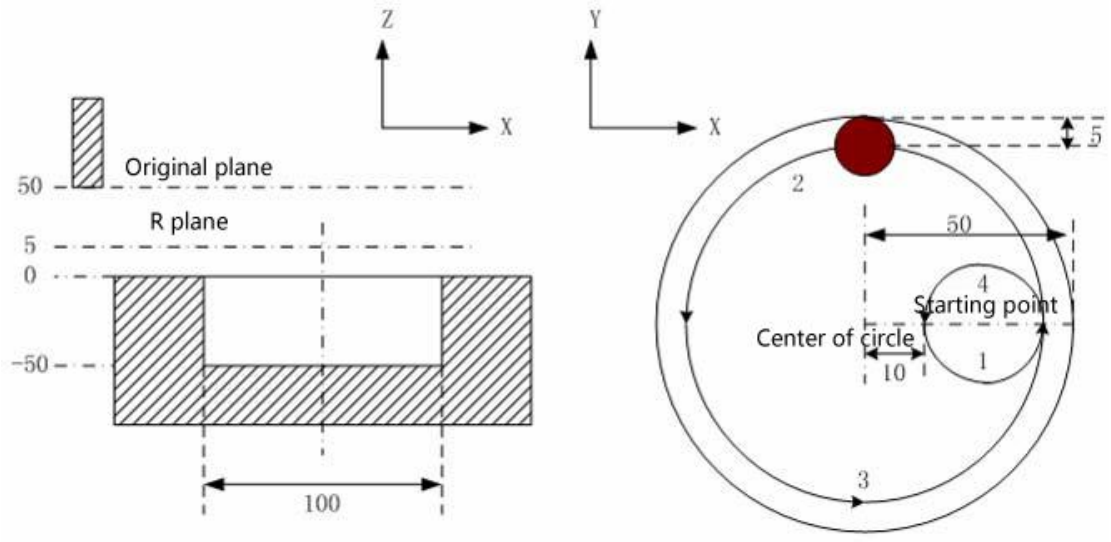
The track:



Note:

Q P L invalid under this cycle, but it will keep the value of Q P as the fixed mode cycle value to save.

Example: Use G112 to accurate milling the circle groove which has rough milled.



G90 G00 X50 Y50 Z50; (G00 orient with rapid move)

G99 G112 X25 Y25 R5 Z-50 I50 J10 F800 D1; (Start fixed cycle, move to the hole bottom and process accurate milling cycle in circle D1=5)

G80 X50 Y50 Z50; (Cancel the fixed cycle, return from R point plane)

M30;

(48) Accurate milling cycle excircle G114/G115

Format: G99 G114/G115 X_ Y_ R_ Z_ I_ J_ D_ F_

Function: The tool accurate milling a whole circle at excircle with the specified radius and direction, return after finishing.

Note: The relative instruction refers to the table 3-2. G114: accurate milling cycle excircle CCW. G115: accurate milling cycle excircle CW.

I:Circle radius of accurate milling, the range -99999999~99999999 times the minimum unit, take absolute value if negative.

J: The distance between the starting point of accurate milling and the center of accurate milling circle, the range J≥D(radius of tool), take absolute value if negative.

D: The number of tool, the range 1~32, D0 default is 0. Take out the radius of the current tool according the number.

Cycle process:(1) Orient to the position of XY plane

(2) Move down to the R point plane

(3) Cutting feed to the hole bottom

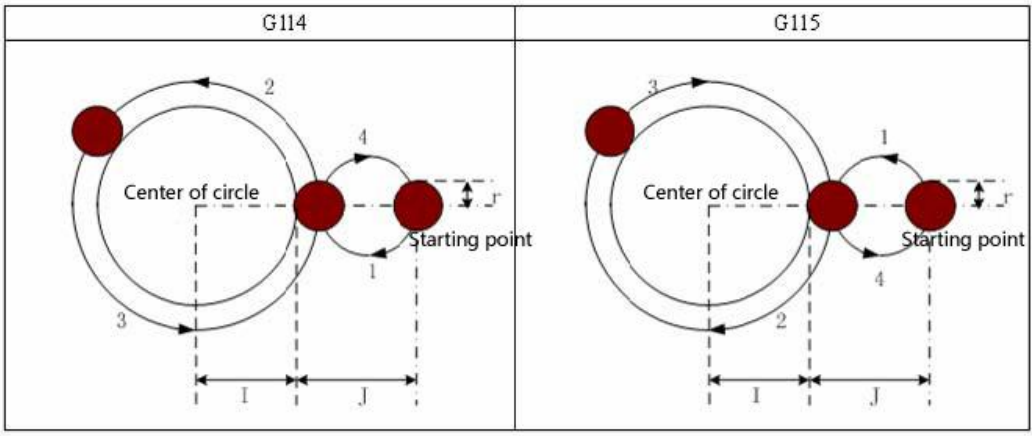
(4) Process the arc interpolation according to the track of transition arc 1

(5) Whole circle interpolation according to the arc 2, arc 3

(6) Arc interpolation to the starting point according to the track of transition arc 4

(7) Fixed to return R point

The track:

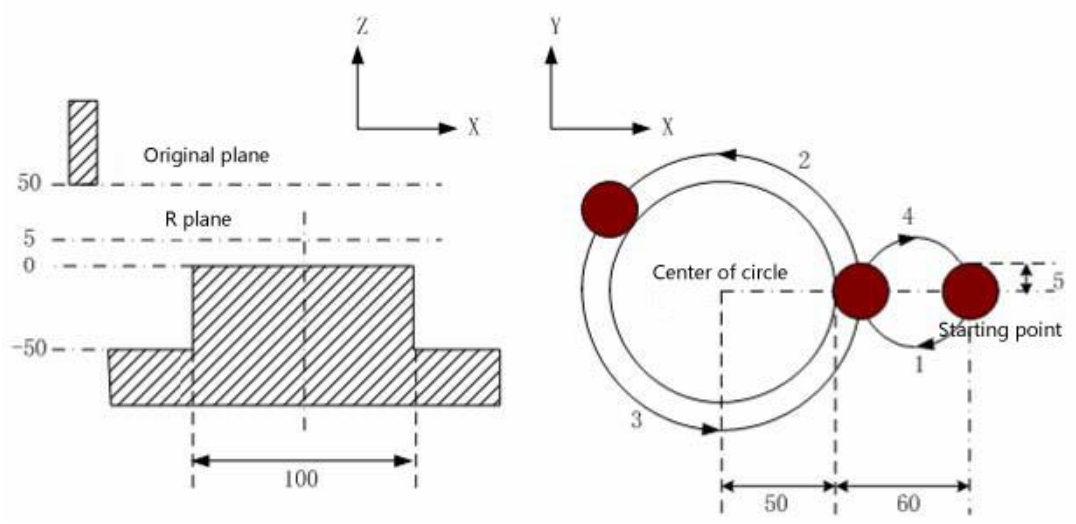


Note:

(1) The direction of transition arc and accurate milling arc is different when accurate milling excircle, the interpolation direction means the interpolation direction of accurate milling arc.

(2) Q P L invalid under this cycle, but it will keep the value of Q P as the fixed mode cycle value to save.

Example:



G90 G00 X50 Y50 Z50; (G00 orient)

G99 G114 X25 Y25 R5 Z-50 I50 J60 F800 D1; (Starts fixed cycle, move down to the hole bottom and process accurate milling cycle of excircle D1=5)

G80 X50 Y50 Z50; (Cancel the fixed cycle and return from R point plane)

M30;

(49) Rough milling cycle of excircle G116/G117

Format: G99 G116/G117 X_ Y_ Z_ R_ I_ J_ W_ Q_ K_ C_ E_ D_ F_

Function: Start from starting point, the tool process whole circle interpolation until process the size which is programmed.

Note: The relative instruction refers to the table 3-2. G116: Rough milling cycle of excircle CCW G117: Rough milling cycle of excircle CW

I: The radius of rough milling circle (should be greater than 0, take absolute value if negative)

J: The radius of workpiece (should be greater than 0, take absolute value if negative);

E: The allowance of rough milling excircle in XY plane (should be greater than 0 or equal to 0, take absolute value if negative);

W: The first time cutting depth with the direction of Z axis, the distance from the R plane move down, should be greater than0, take absolute value if negative(if the first time cutting is deeper than the groove bottom, so process from the groove bottom)

Q: Every incremental depth with direction of Z axis, take absolute value if negative

K: The cutting incremental width in XY plane(should be smaller than the tool radius, greater than 0, take absolute value, if not to specify K, the default is the tool radius D)

C: The first time feed cutting (C should be greater or equal to the tool radius +2.0, X axis move positive direction when greater than 0, the workpiece is located at the positive direction. When smaller than 0, X axis move negative direction, the workpiece is located at the negative direction)

D: The number of tool, the range 0~32, D0 the default is 0. Take out the current tool radius according to the number which is specified

Cycle process:

(1) Orient to the position of XY plane

(2) Move down to the R point plane

(3) Move down to W depth

(4) Process straight interpolation to feed tool with the first time feeding value C of X axis and track of straight 1.

(5) Process whole circle interpolation with track of arc 2

(6) Every time according to the K value to increase to mill the (I+E) radius circle from outside to the center

(7) Orient Z axis back to R plane

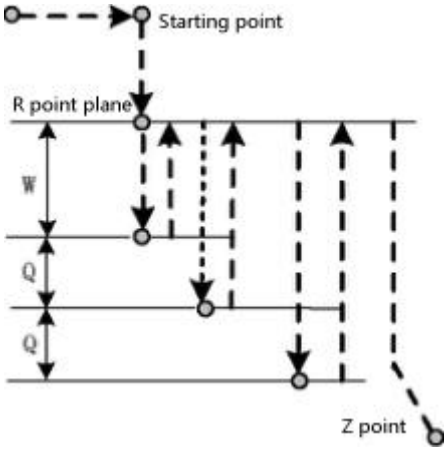
(8) Orient to starting position(XY plane)

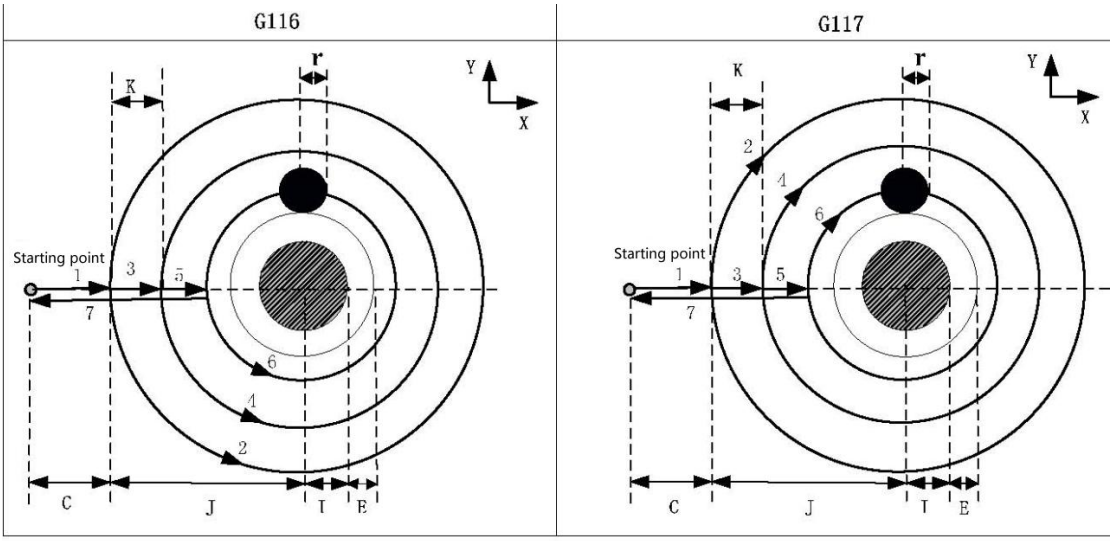
(9) Z axis rapid move to the position (unmachined plane + Q)

(10) Cycle 5~9 motion until finish all the cutting depth Z

(11) Fixed return to R point

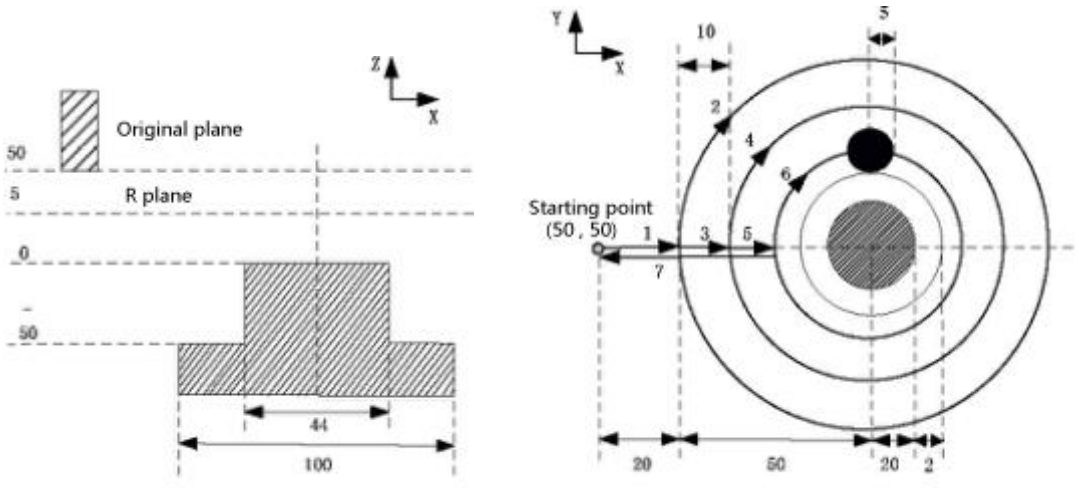
(12) Return to starting position of XY





Note: P L invalid under the cycle, but it will keep the value of P as the fixed mode cycle value to save.

Example: Use G117 to rough mill



G90 G00 X0 Y0 Z50; (G00 orient)

G99 G117 X50 Y50 R5 Z-50 I20 J50 W20 Q10 K10 C20 E2 F800 D1; (process rough milling excircle cycle D1=5)

G80 X50 Y50 Z50; (cancel the fixed cycle, return from R point plane)

M30;

(50) Rough milling outside rectangle G132/G133

Format:G99 G132/G133 X_ Y_ Z_ R_ I_ J_ A_ B_ W_ Q_ K_ C_ E_ D_ F_

Function: Start from starting point to cycle the straight cutting according to the parameter until process the size which is programmed.

Note: The relative instruction refers to the table 3-2. G132: Rough milling outside rectangle CCW G133: Rough milling outside rectangle CW XY: The coordinate position of processing point

I: The width of rough milling outside rectangle in direction of X axis (should be greater than 0. Take absolute value if negative)

J: The width of rough milling outside rectangle in direction of Y axis (should be greater than 0. Take absolute value if negative)

A: The width of blank in direction of X axis (should be greater than 0. Take absolute value if negative)

B: The width of blank in direction of Y axis (should be greater than 0. Take absolute value if negative)

E: The allowance of rough milling outside rectangle in XY plane (should be greater than or equal to 0. Take absolute value if negative)

W: The first time depth in direction of Z axis which is from R plane to downward position, should be greater than 0, take absolute value if negative (if the first time cutting is deeper than the groove bottom, so process from the groove bottom)

Q: Every time the incremental cutting depth in direction of Z axis, take absolute value if negative

K: The incremental cutting width in XY plane (should smaller than the tool radius, greater than 0, take absolute value if negative, it not specify K, the default is K=D tool radius)

C: The first time feed cutting in direction of X (C should be greater or equal to the tool radius +2.0, X axis move positive direction when greater than 0, the workpiece is located at the positive direction. When smaller than 0, X axis move negative direction, the workpiece is located at the negative direction)

D: The number of tool, the range 0~32, D0 the default is 0. Take out the current tool radius according to the number which is specified

Cycle process:

(1) Orient to the position of XY plane

(2) Move down to the R point plane

(3) Move down to W depth

(4) Process straight interpolation to feed tool with the first time feeding value C of X axis and track of straight 1.

(5) Process straight interpolation with track 2.

(6) Every time according to the K value to increase to mill the rectangle with (I+2E) long , (J+2E) width.

(7) Arc angle of milling rectangle

(8) Z axis returns to R point plane

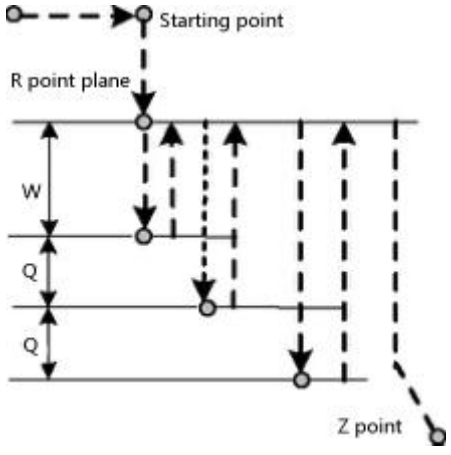
(9) X and Y axis orient to the starting point plane

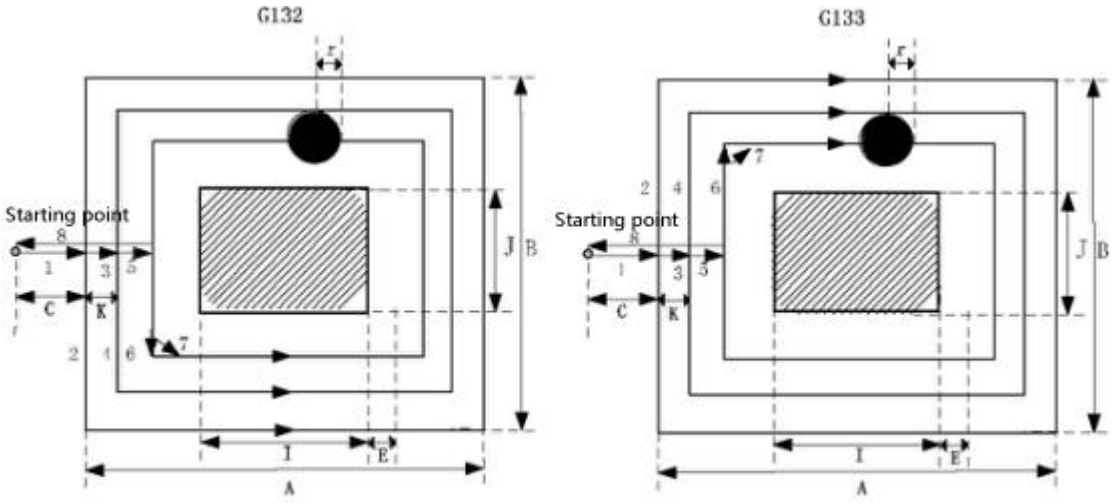
(10) Z axis rapid move to the position (unmachined plane + Q)

(11) Cycle 4~9 motion until finish all the cutting depth Z

(12) Fixed return to R point

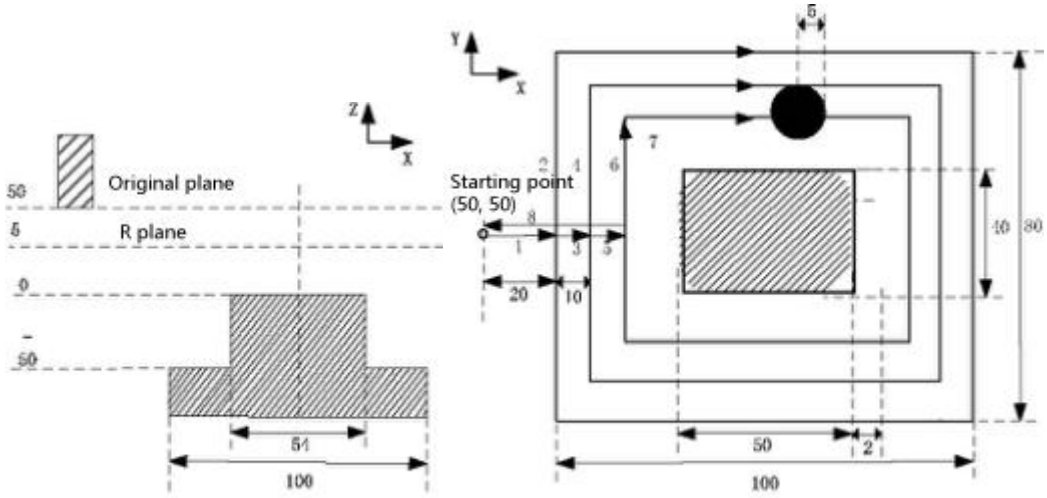
(13) Return to starting position of XY





Note: P L invalid under the cycle, but it will keep the value of P as the fixed mode cycle value to save.

Example: Use G133 to rough mill



G90 G00 X0 Y0 Z50; (G00 to orient)

G99 G133 X50 Y50 R5 Z-50 I50 J40 A100 B80 W20 Q10 K10 C20 E2 U5 F800 D1; (Process the rough milling groove cycle D1=5 in rectangle)

G80 X50 Y50 Z50;

M30;

(51) Rough milling rectangle groove G134/G135

Format: G99 G134/G135 X_ Y_ Z_ R_ I_ J_ K_ W_ Q_ E_ V_ D_ F_

Function: Start from the center of rectangle to straight cutting cycle with the specified parameter data until process the rectangle groove which is programmed.

Note: The relative instruction refers to the table 3-2. G134: Rough milling the rectangle groove CCW. G135: rough milling the rectangle groove CW.

I: The width of rectangle groove in direction of X axis.

J: The width of rectangle groove in direction of Y axis.

K: The cutting incremental width in X Y plane, should be less than the diameter of tool, greater than 0.

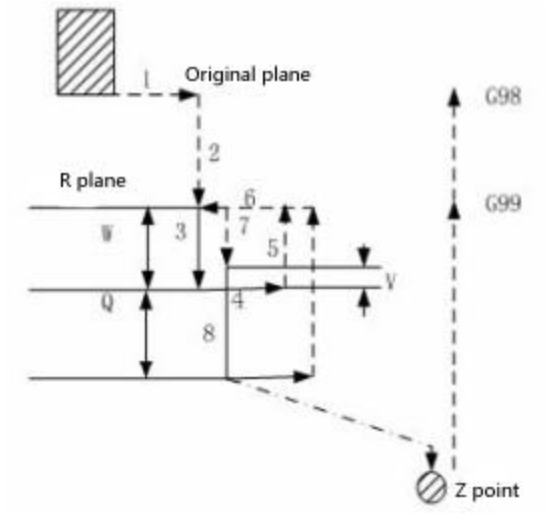
W: The first time depth in direction of Z axis which is from R plane to downward position, should be greater than 0, take absolute value if negative (if the first time cutting is deeper than the groove bottom, so process from the groove bottom)

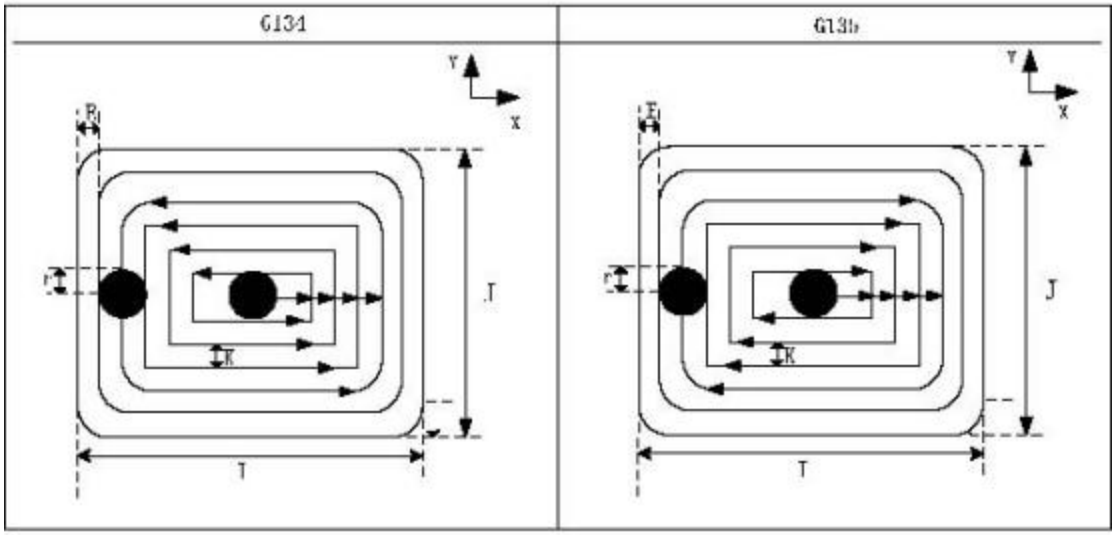
Q: Every time the incremental cutting depth in direction of Z axis, take absolute value if negative

V: The distance from the tool to the processing plane when tool starts moving, should be greater than 0

E:The allowance of rough milling rectangle groove in XY plane (should be greater than or equal to 0. Take absolute value if negative, take the parameter data when not to be specified)

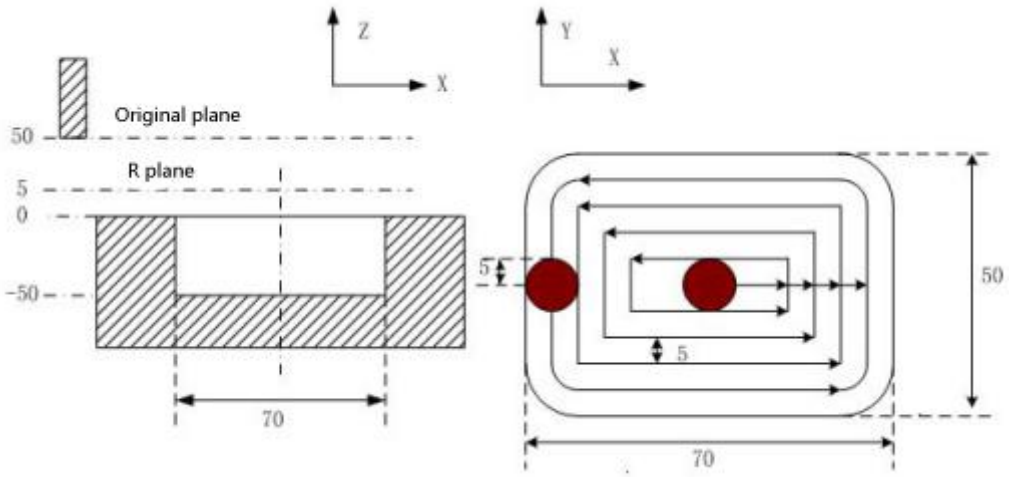
D: The number of tool, the range 0~32, D0 the default is 0. Take out the current tool radius according to the number which is specified





Note: P L invalid under the cycle, but it will keep the value of P as the fixed mode cycle value to save.

Example: Use the fixed cycle G134 to rough milling groove in rectangle



G90 G00 X50 Y50 Z50; (G00 to orient)

G99 G134 X25 Y25 R5 Z-50 I70 J50 W20 Q10 K5 E0 V10 U10 F800 D1;

(Process the rough milling groove in rectangle cycle D1=5)

G80 X50 Y50 Z5; (Cancel the fixed cycle, return from R point plane)

M30;

(52) Accurate milling cycle in rectangle groove G136/G137

Format:G99 G136/G137 X_ Y_ R_ Z_ I_ J_ D_ K_ U_ F_;

Function: The tool accurate milling in rectangle with specified width and direction, return after finishing.

Note: The relative instruction refers to the table 3-2. G136: Accurate milling cycle in rectangle groove CCW. G137:Accurate milling cycle in rectangle groove CW.

I: The width of rectangle in direction of X axis, the range: -99999999~99999999 times the minimum unit. Take absolute value when negative.

J: The width of rectangle in direction of Y axis, the range: -99999999~99999999 times the minimum unit. Take absolute value when negative. The value must greater than or equal to the tool radius D, otherwise it will alarm.

D: The number of tool, the range 0~32, D0 the default is 0. Take out the current tool radius according to the number which is specified

K: The distance from the starting point to the rectangle in direction of X axis, the range -99999999~99999999 times the minimum unit. Take absolute value when negative.

U: Corner radius of arc, omit means no corner arc to transition. When omitting U or U=0 and the tool radius greater than 0, it will alarm.

Cycle process:

(1) Orient to the position of X Y plane;

(2) Move to the R point plane;

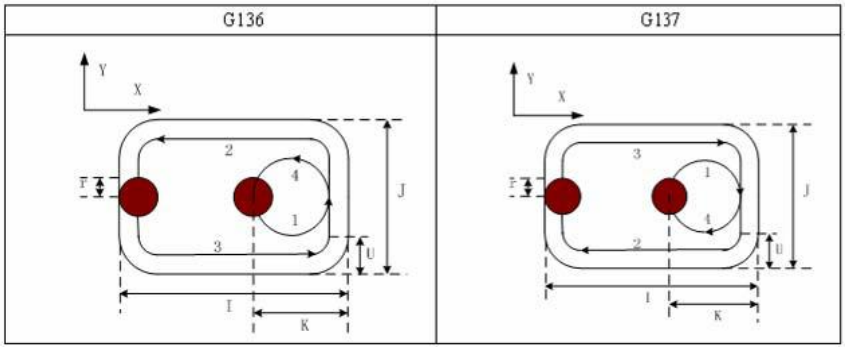
(3) Cutting feed to the hole bottom;

(4) Start from starting point to arc interpolation with transition arc 1;

(5) Process straight line and arc interpolation with 2-3;

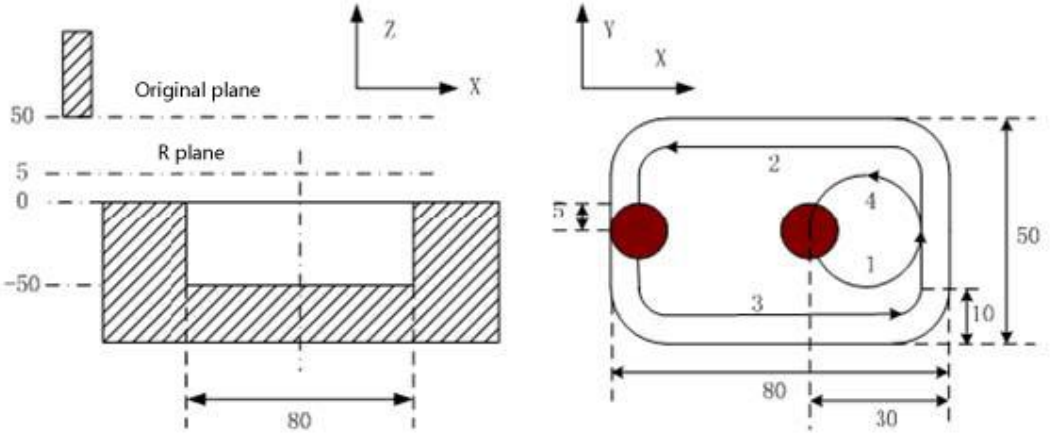
(6) Process arc interpolation to back to starting point with transition arc 4;

(7) Fixed back to R point



Note: Q P L invalid under the cycle, but it will keep the value of Q P as the fixed mode cycle value to save.

Example: Use fixed cycle G136 to accurate milling.



G90 G00 X50 Y50 Z50; (G00 to orient)

G136 X25 Y25 R5 Z-50 I80 J50 K30 U10 F800 D1; (Process accurate milling D1=5 at the bottom of hole under the fixed cycle)

G80 X50 Y50 Z50; (Cancel the fixed cycle and return from R point plane)

M30;

(53) Accurate milling cycle outside the rectangle G138/G139

Format: G99 G138/G139 X_ Y_ R_ Z_ I_ J_ D_ K_ U_ F_

Function: The tool accurate milling outside the rectangle with specified width and direction, return after finishing.

Note: The relative instruction refers to the table 3-2. G138: Accurate milling outside the rectangle CCW. G139: Accurate milling outside the rectangle CW.

I: The rectangle width in direction of X axis, the range -99999999~99999999 times the minimum unit. Take absolute value when negative.

J: The rectangle width in direction of Y axis, the range -99999999~99999999 times

the minimum unit. Take absolute value when negative.

D: The number of tool, the range 0~32, D0 the default is 0. Take out the current tool radius according to the number which is specified

K: The distance from the starting point to the rectangle in direction of X axis, the range -99999999~99999999 times the minimum unit. Take absolute value when negative.

U: Corner radius of arc, omit means no corner arc to transition.

Cycle process:

(1) Orient to the position of X Y plane;

(2) Move to the R point plane;

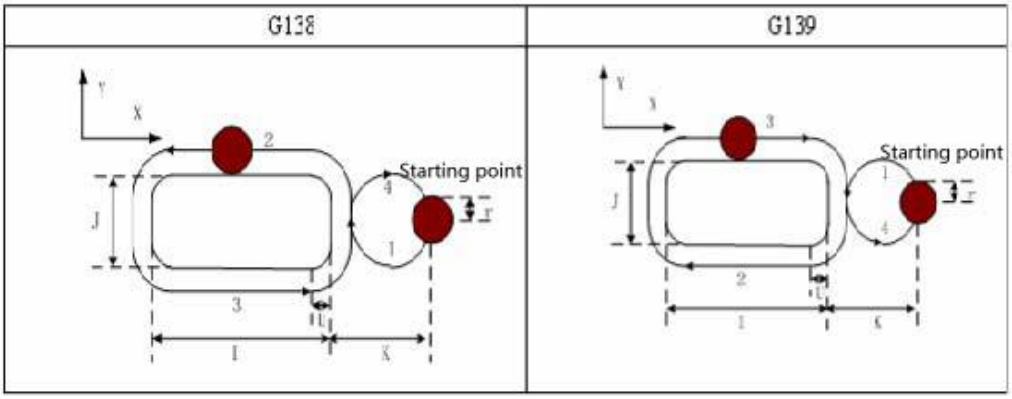
(3) Cutting feed to the hole bottom;

(4) Start from starting point to arc interpolation with transition arc 1;

(5) Process straight line and arc interpolation with 2-3;

(6) Process arc interpolation to back to starting point with transition arc 4;

(7) Fixed back to R point

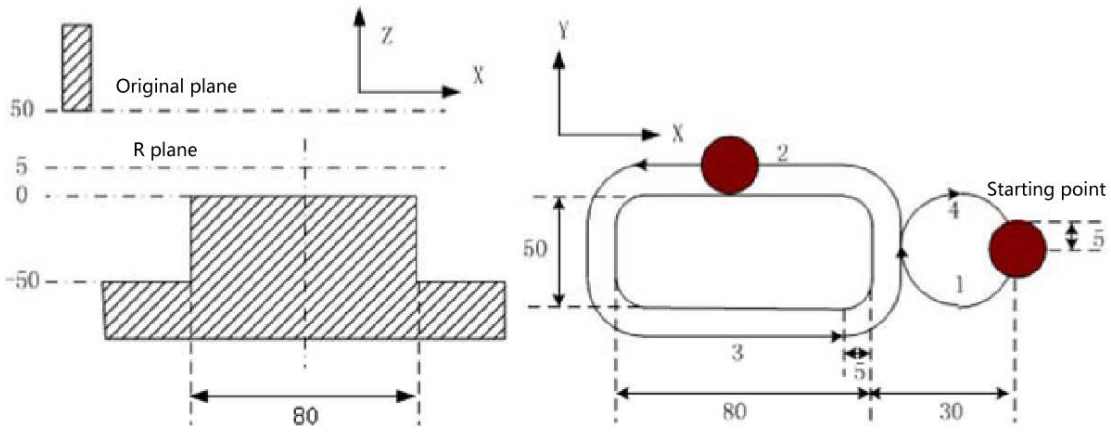


Note:

(1) The direction of transition arc and accurate milling arc is different when accurate milling outside the rectangle, the interpolation direction means the direction of accurate milling arc.

(2) Q P L invalid under the cycle, but it will keep the value of Q P as the fixed mode cycle value to save.

Example: Use the fixed cycle G138 to accurate milling



G90 G00 X50 Y50 Z50; (G00 to orient)
G99 G138 X25 Y25 R5 Z-50 I80 J50 K30 U5 F800 D1; (Process accurate milling outside the rectangle at the bottom of hole under the fixed cycle)
G80 X50 Y50 Z50; (Cancel the fixed cycle and return from R point plane)
M30;

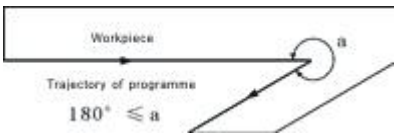
2.5 Radius compensation of tool C

C means the system calculates the tool trajectory of radius compensation according to the last program line and the next program line.

2.5.1 Inside and outside

It calls inside when the included angle of tool trajectory is over 180 degrees which is built by two program segments, it calls outside when the included angle is between 0 and 180 degrees. As the follows:

Inside：



Outside：

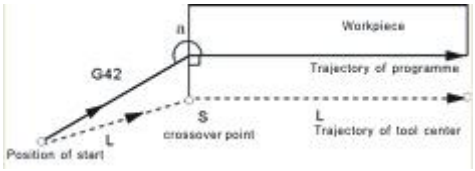
2.5.2 Tool motion when starting

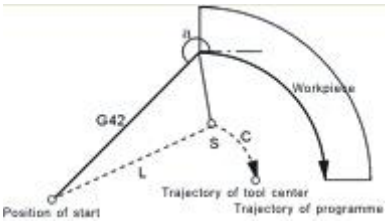
The radius compensation without tool builds tool radius compensation

(1) Tool motion around the inside corner （α≥180）

The tool center will move to the tool vector radius vertex of the starting point in next program line.

Straight line->Straight line Straight line->Arc



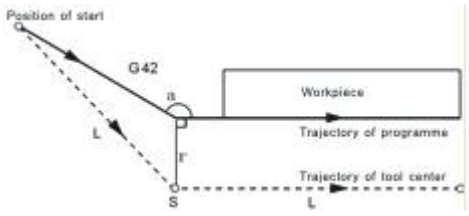


(2)The tool motion around the outside corner of obtuse angle（90≤α＜180）

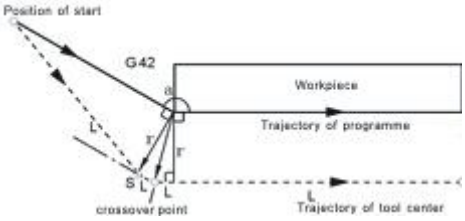
The tool center will move to the tool vector radius vertex of the end point in this program line.

Straight line−>Straight line

A type

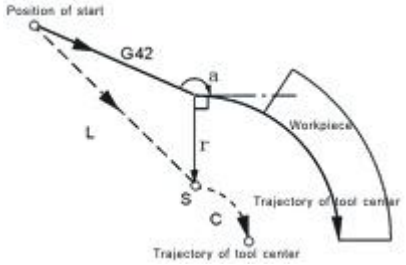


B type

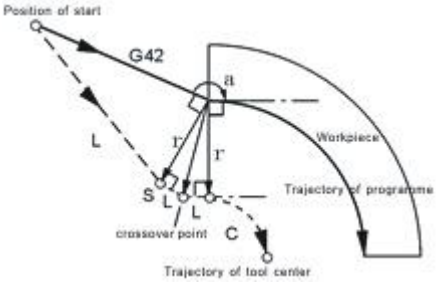


Straight line->Arc

A type



B type

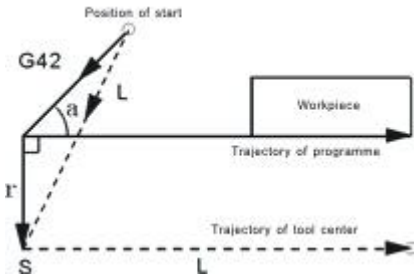


(3)The tool motion around the outside corner of acute angle（α＜90）

The tool center will move to the tool vector radius vertex of the end point in this program line.

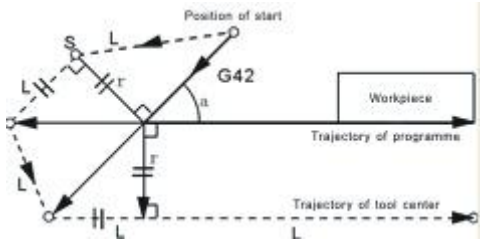
Straight line->Straight line

A type B type

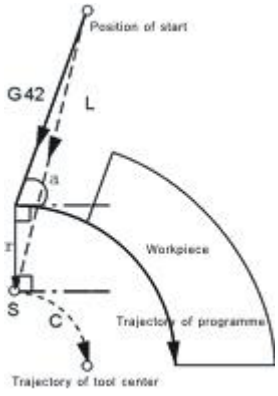


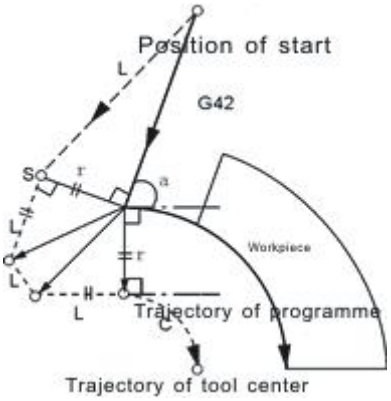
Straight line->Arc

A type



B type

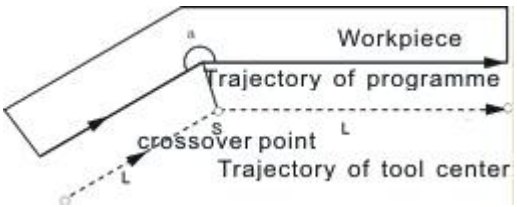




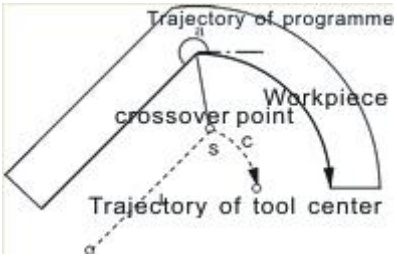
2.5.3 Tool motion in offset mode

(1)Tool motion around the inside corner（180≤α）

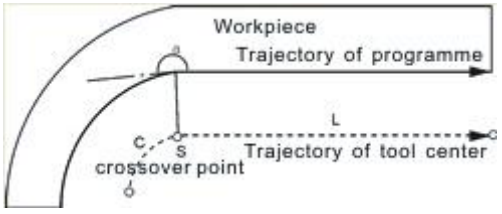
Straight line->Straight line



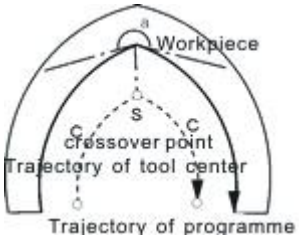
Straight line->Arc



Arc-> Straight line



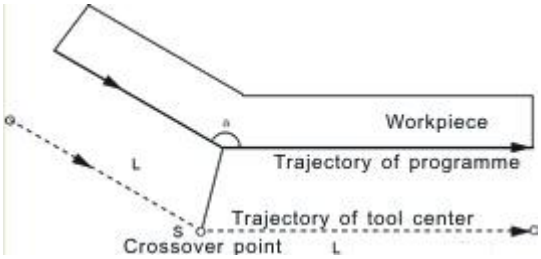
Arc->Arc



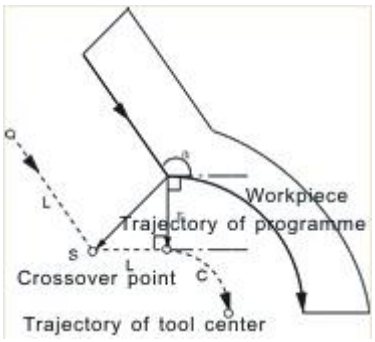
(2)The tool motion around the outside corner of obtuse angle（90≤α＜180）

Straight line ->Straight line

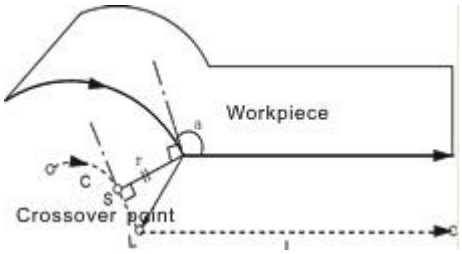
Straight line ->Arc

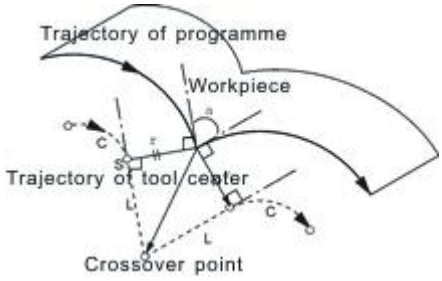


Arc-> Straight line



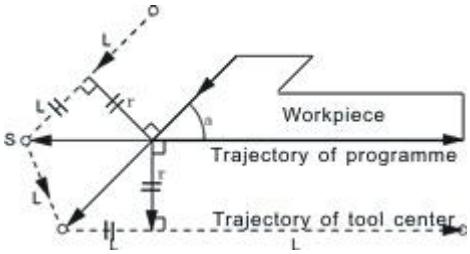
Arc->Arc



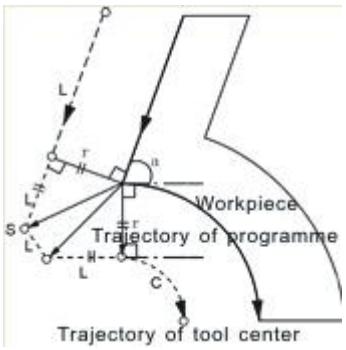


(3)The tool motion around the outside corner of acute angle（α＜90）

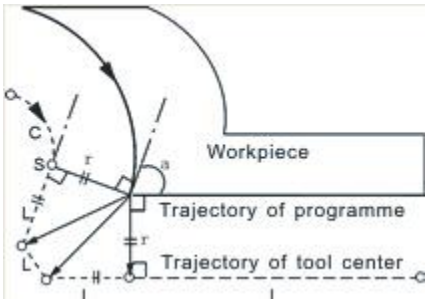
Straight line->Straight line



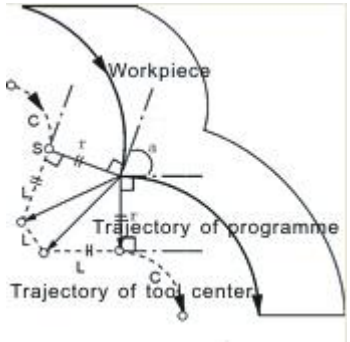
Straight line->Arc



Arc-> Straight line



Arc->Arc

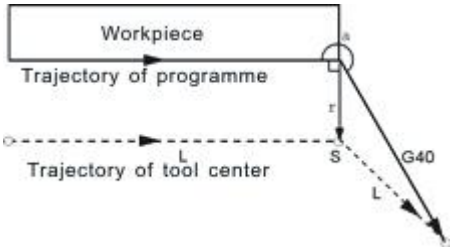


2.5.4 Tool motion in offset-cancel mode

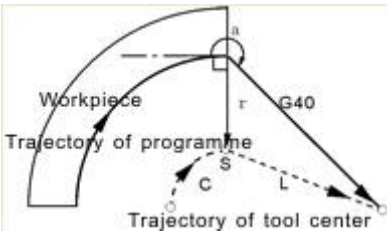
(1) Tool motion around the inside corner（180≤α）

The tool center will move to the tool vector radius vertex of the end point in this program line.

Straight line－＞Straight line



Arc－＞Straight line

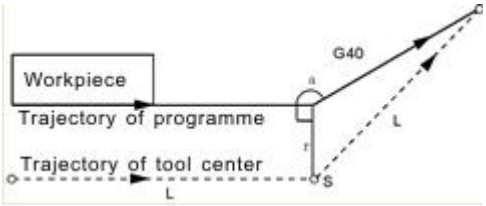


(2) The tool motion around the outside corner of obtuse angle（90≤α＜180）

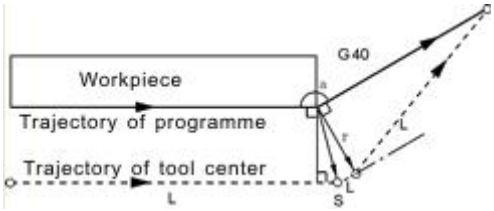
The tool center will move to the tool vector radius vertex of the starting point in next program line.

Straight line－＞Straight line

A type

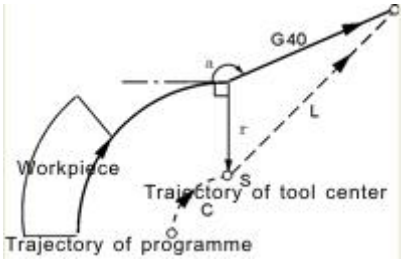


B type

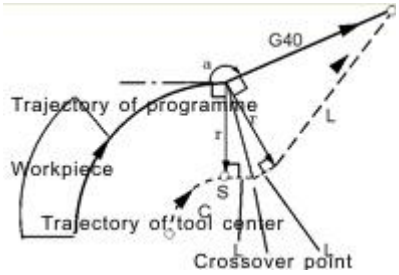


Arc－＞Straight line

A type



B type

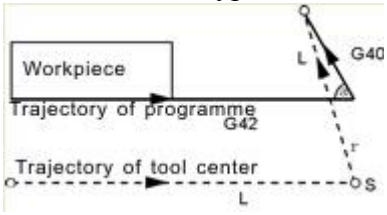


(3)The tool motion around the outside corner of acute angle（α＜90）

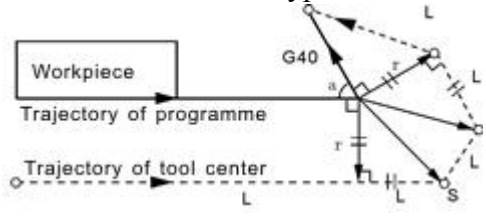
The tool center will move to the tool vector radius vertex of the starting point in next program line.

Straight line－＞Straight line

A type

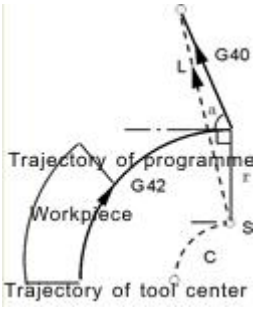


B type

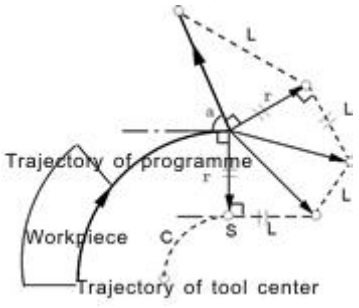


Arc－＞Straight line

A type



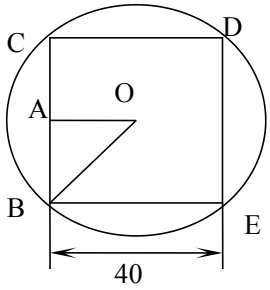
B type



2.6 Comprehensive examples for programming

In the actual programming, must according to the drawings and processing requirements to select the install method and suitable tool correctly, combined with the actual working performance of lathe to select the correct cutting allowance, for example:

Example 1: Cut square and cut circle



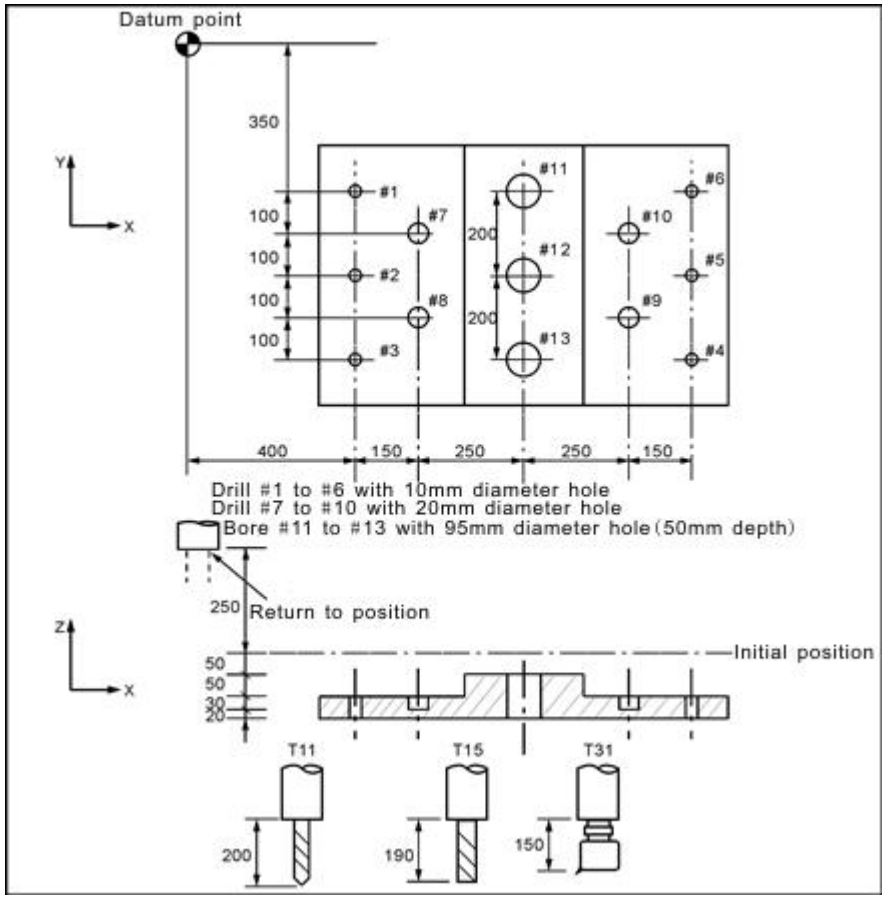
From the center to begin, the center coordinate is G54 X0 Y0 Z50

The tool radius in the D01 of the T01 parameter

N0 G54 G00 X0 Y0 Z50	Starting point for processing
N1 M03 S1000	Turn on spindle
N2 G00 Y-40 X－40 G90	Move outside of square
N3 Z10	
N4 G01 Z－10 F200	Start cutting from Z axis
N5 G01 G41 T01 D01 X-20 Y-20 F400	Move to B point
N6 Y20	Tool cuts BC line at the left of the workpiece
N7 X20	Closed angle transition in C point, cut CD line
N8 Y－20	Closed angle transition in D point, cut

	DE line
N9 X-25	Closed angle transition in E point,cut CB line is for smooth， move 5mm more
N10 G00 X-40 Y-40 G40	Move outside of circle
N11 G01 G41 X－20 Y－20 F500	Lengthen T01 value along A-B
N12 Z－20 F100	Start cutting from Z axis
N13 G91 G02 I20 J20 Y0 X0	Circle of contact
N14 G00 Z50	End cutting
N15 G40 G00 X0 Y0	Cancel tool compensation
N16 M05	Turn off spindle
N17 M02	Program is over

Example 2:



Assume to used 3 tools T11 T15 T31. The tool length compensation values were 200 (H1), 190 (H1), 150 (H1). Has entered into the tool parameter. According to the processing requirements of parts drawing, should write a program is as follows:

N1 G54 X0 Y0 Z0	Set workpiece coordinate at the datum point

N2 G90 G00 Z250.0 T11	Tool exchange
N3 G43 Z0 H1	The offset length in initial position
N4 S30 M3	Turn on spindle
N5 G99 G81 X400.0 Y-350.0 Z-153.0 R-97.0 F120	To drill #1 hole
N6 Y-550.0	To drill #2 hole and return to initial position
N7 G98 Y-750.0	To drill #3 hole and return to initial position
N8 G99 X1200.0	To drill #4 hole and return to initial position
N9 Y-550.0	To drill #5 hole and return to initial position
N10 G98 Y-350.0	To drill #6 hole and return to initial position
N11 G00 X0 Y0 M5	Return to the datum point and stop spindle
N12 G49 Z250.0 T15	Cancel the tool length offset to change tool
N13 G43 Z0 H1	Tool length offset in initial position
N14 S20 M3	Turn on spindle
N15 G99 G82 X550.0 Y-450.0 Z-130.0 R-97.0 F500	To drill #7 hole and return to initial position
N16 G98 Y-650.0	To drill #8 hole and return to initial position
N17 G99 X1050.0	To drill #9 hole and return to initial position
N18 G98 Y-450.0	To drill #10 hole and return to initial position
N19 G00 X0 Y0 M5	Return to the datum point and stop spindle
N20 G49 Z250.0T31	Cancel the tool length offset to change tool
N21 G43 Z0 H1	Tool length offset in initial position
N22 S10 M3	Turn on spindle
N23 G85 G99 X800.0 Y-350.0 Z-153.0 R47.0	To bore #11 hole and return to

F150	initial position
N24 G91 Y-200.0	To bore #12 hole and return to initial position
N25 Y-200.0	To bore #13 hole and return to initial position
N26 G90 G28 X0 Y0 M5	Return to the datum point and stop spindle
N27 G49 Z0 G80	Cancel the tool length offset
N28 M02	Program is over

2.7 Usage for automatic tool setting gauge

1. Note for parameter:

Define macro variables of the automatical tool setting gauge function are as follows (corresponding to the other parameters P380 - P389):

#380: The X axis lathe coordinate of initial position with automatical tool setting;(Unit:mm)

#381: The Y axis lathe coordinate of initial position with automatical tool setting;(mm)

#382: The Z axis lathe coordinate of initial position and returning point with automatical tool setting;(mm)

#383: The negative speed of automatical tool setting;(mm/min)

#384: The positive speed of automatical tool setting;(mm/min)

#385: The Z axis coordinate of workpiece surface in current workpice coordinate system after automatical tool setting;(mm)

#386: The speed which is rapid move to locating position with automatical tool setting;(mm/min)

#387: Automatic tool setting mode (1 means fixed point, 0 means floating point).

#388: The minimal lathe coordinate value of Z axis (mm);

#389: The gap value of Z axis [The height which is the gauge surface relative to the workpiece surface(mm)];

Fixed point gauge means putting the gauge in a fixed position, everytime the X Y Z axis are automatical running to the fixed point first in tool setting; But the floating point gauge search the tool setting gauge signal along negative of the Z axis.

The input point X25 is default to be the checking point of automatical tool setting gauge to input.

2.The instruction: M880 (corresponding to ProgramUser0) automatic tool setting instruction; M882 (corresponding to ProgramUser2), M883 (corresponding to ProgramUser3) set the gap of Z axis.

3. Automatic tool setting steps:

a)Set the No.380--No.388 parameter in other parameter;

b)Set the No.389 parameter in other parameter to set the gap of Z axis: this operation needs to be set only once.

A.Run M882 instruction in MDI to set the gap of Z axis;

B.Manual run Z axis to move the tool nose to the workpiece surface;

C.Run M883 instruction in MDI to automatical set the gap of Z axis No.389 parameter in other parameter;

c) MDI choose the workpiece coordinate system G54/G59;

d) Automatic tool setting: MDI running the M880 instruction, automatical set the Z axis offset of the current workpiece coordinate system.

2.8 Usage for automatical dividing center

1. The X axis is divided center: M884(Corresponding to ProgramUser4)

1) Choose the current coordinate system such as G54;

2) Manually moving the X to the negative terminal of workpiece; MDI running the M884 instruction;

3) Manually moving the Y to the positive terminal of workpiece; MDI running the M885 instruction, automatically divide the center of Y axis and set the middle point of workpiece as current coordinate system of Y axis origin, that's automatically setting the current coordinate system, such as the coordinate offset value of Y axis in G54.

2. The Y axis is divided center (Corresponding to ProgramUser5): M885

1) Choose the coordinate system such as G54;

2) Manually moving the Y to the negative terminal of workpiece; MDI running the M885 instruction;

3) Manually moving the Y to the positive terminal of workpiece; MDI running the M885 instruction, automatically divide the center of Y axis and set the center point of workpiece as the current coordinate system of Y axis origin,that's automatically setting the current coordinate system, such as the offset value of Y axis in G54.

2.9 Usage for function of follow-up coodinate

1. Method 1: The coodinate not follow-up with this method, not display the fourth axis when using. When the No.414 parameter in axis parameter is 7, it will follow the X axis, the same as X axis automatically when using manual; When set as 8, the fourth axis will follow the Y axis, the same as Y axis automatically when using manual; When set as 9, the fourth axis will follow the Z axis, the same as Z axis

automatically when using manual.

2. Method 2: Use instruction to follow the axis, not follow when using manual, follow the axis under mode of auto, the same follow as the coordinate.

M123: Y axis starts follow-up X axis in auto mode

M124: Y axis stop follow-up X axis

M125: A axis starts follow-up X axis in auto mode

M126: A axis stop follow-up X axis

Chapter 3 Operation explanation

3.1 Summary

When using this CNC system, as long as the master of the system parameters, the program edit, manual operation, automatically running, it can be very convenient to operate.

3.2 Operation panel

This system panel is composed of the main panel and side sub-panel. The main panel is used for parameter setting and program editing and the sub-panel is for tool setting and processing operations.

3.3 Keyboard description

3.3.1 Rate increase or decrease

(1) **Rapid override(G)** 

There are six gears in rapid override form 5% to 100%, by adjusting the key of rapid override is for the following instruction: G00,G26,G28,G611-G618, rapid feed fixed cycle, rapid manual feed.

(2) **Feed override(F)** 

There are sixteen gears in feed override from 0% to 150%, by adjusting the key of feed override is for the following instruction:G01,G02,G03, the feed override of the fixed cycle and manual run effectively.

(3) Spindle override(S) 

There are sixteen gears in spindle override from 5% to 150%, by adjusting the key of spindle override is for the speed of the first spindle.

3.3.2 Usage for intervention switch

(1) The left: normal processing.

(2) The middle: manual to stop feeding, suspend the automatical feeding. Return to the left to the normal status.

(3) The right: suspend automatic feeding. Return to the left to the normal status.

Spindle is out of control when running with non-coordinate axis.

3.3.3 Others keys

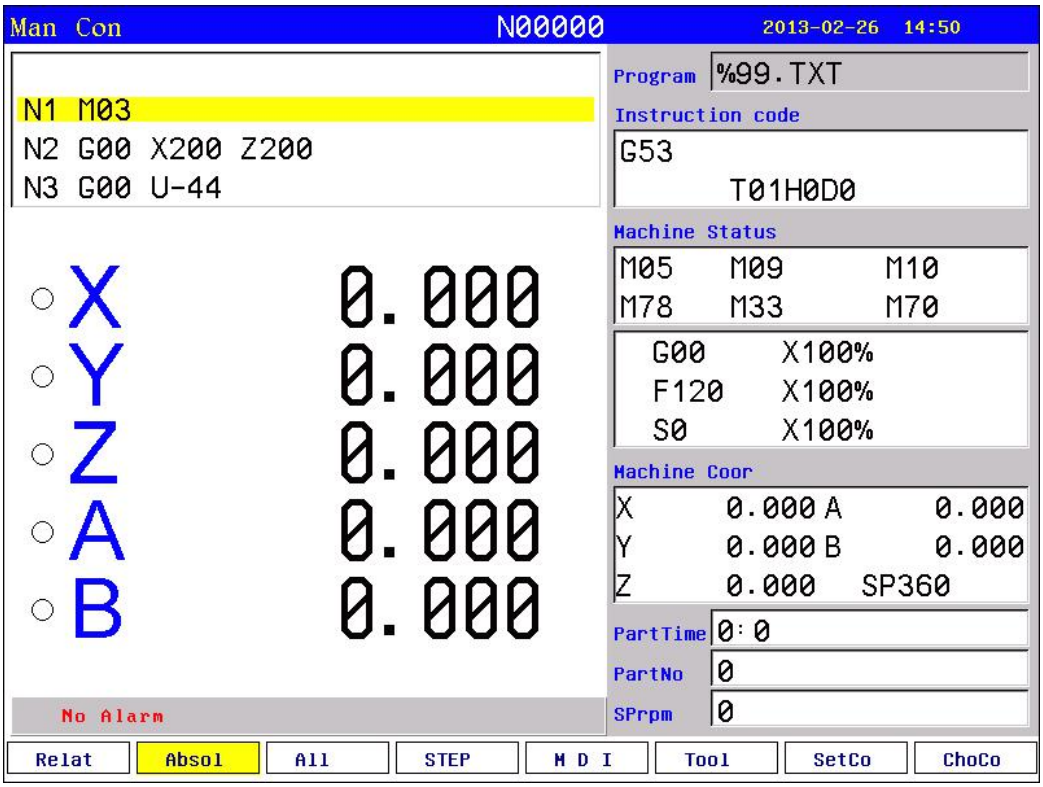
Keyboards	Functions
Letter key Number key	A B C S E F G H I J K L M N O P Q R S T U V W X Y Z 1 2 3 4 5 6 7 8 9 .-： for program instructions, parameters' edition; number keys are used for inputting data and selecting sub-menu.
Symbol key	#*[]()/=% are used for editing macroprogram
Edit key	"↑、↓、→、←、Del、PgUp、PgDn"for programming, direction keys can be used for selecting menu.
+X、-X、 +Y、-Y、 +Z、-Z、 +A、-A、 +B、-B	Use for feeding in manual; +B, -B are used for B/C/Xs/Ys axis to feed positive and negative, use "B" "C" "X" "Y" to select the current feeding axis;
Function key	"Esc" returning to upper level or stop a operation "Enter" selecting sub-menu and changing a newline "Del" delete program "program"entering program edition "parameter" entering parameter setting "diagnosis" entering diagnosis I/o function "manual" entering manual status "handwheel" for starting or stopping handwheel function "Tool" for confirming current tool 's position in machine too coordinates system. "Redeem" for amending tool change errors "Auto" entering automatic status "MDI" entering MDI function "  "selecting auto-coordinates/diagram machining "  "for single segment or constant work "  " for coordinates mode or diagram mode speedy simulating "  "for manual increment or manual continuous

Control key	“  ”spindle cw, ccw rotation “  ” coolant on/off “  ”for the shift between electric tool carrier and gang tool carrier “  ” for the shift between hand-driven continuous high speed and low speed. “  ” all axes return to datum point “  ” for spindle chuck on/off “  ” for thumbstall on/off “  ”handwheel gear selection “  ” adjusting spindle speed “  ”adjusting feed speed “  ”adjusting G00 speed
Control button	Emergency stop: Stop all action of lathe and wait for putting up the button; Intervention switch: Suspend the screw and spindle; Start: Execute processing program automatically;

3.4 Manual operation

The system adjusts one-level menu operation, intuitive, convenient, shortcut, prompt comprehensive information.

Powering the system is to enter the interface(Following shown)



3.4.1 The key of manual operation

(1) “F”: Taking mm/min as the unit to set the manual feed speed, the input range is from 1 to 30000mm/min. And the input method according to data input method in parameter.

(2) “ ”: Switching cycle from “manual continuous” to “manual increment”

(3)“S”: Set the speed of the first spindle. The range is from 0 to 99999, the max depends on the No.36 parameter in speed parameter.

(4)“I”： Modify the increment in manual increment

(5)“ ”： Press once to increase or decrease 10% feed speed when the No.1 axis parameter is 0, the range is from 0% to 150%,16 gears totally.

(6) “ ”: Press once to increase or decrease G00 or manual rapid override 20%.The range is from 5% to 100%,16 gears totally.

(7)“  ”: Press once to increase or decrease the spindle override 10% when the No.2 parameter in axis parameter is 0. The range is form 5% to 150%,16 gears totally.

(8)“  ”: To switch cycle “0.001” “0.01” “0.1”or “0.1” “0.01” “0.001” in the handwheel function.

(9)“Diagnosis”: Enter the diagnosis of input or output.

(10)“Setup”： To set a value(G54-G59) in workpiece coordinate(G54-G59);Use “MDI” to set G54-G59 in lathe coordinate(G53).

(11)“Auto”: Select automatic mode.

(12)“Manual”: Select manual mode.

(13)Spindle controlled: “  ”Controlling spindle on clockwise, counterclockwise, stop, correspond to instructions M03,M04,M05. When No.56 parameter in the axis parameter is “8”then press “spindle on counterclockwise” means counterclockwise inching turning.

(14)“Cooling”: Coolant on or off correspond to instructions M08,M09.

(15)“Chuck”: Chuck tightens or loose correspond to instructions M10,M11.

(16)“Manual speed controlled”: Press “1” “2” “3” “4” “5” “6” “7” “8” “9” to set feed override “F30” “F60” “F120” “F250” “F500” “F1000” “F1500” “F2000” “F2500” “F3000”.

(17) “Tailstock”: Tailstock tighten or loose correspond to instructions M79, M78.

(18) “Switch manual continuous or increment”: Press  to manual continuous or increment, it displays I=XXXX.XXX when it is manual increment.

(19) “Back to datum point”: Press  and X or Z, the X or Z axis goes back to the datum point automatically; Press “0”X axis firstly and then Z axis; Press “Esc” to cancel the construction. The speed controlled by No.30 No.32 No.34(No.38-No.47 in bus system) parameter in speed parameter, the direction is determined by No.40(No.47 in bus system) parameter in axis parameter.

(20) “Tool carrier controlled”: Press  to change next tool automatically if it is gang tool carrier; After changing next tool it will be stop if it is electric tool carrier; Which tool has changed is going to be redeem. Press “T” and number to change tool directly

(21) “Coordinates feed”: Press “↑ ↓ ← →”correspond to feed A axis and Z axis’s positive or negative direction.

(22) “Switch speed”: Press  to switch the speed to system speed which is changed by No.1 No.2 No.3 parameter in speed parameter when it is in coordinate feed, loosen it that will be the previous speed. If set the speed higher than the speed in parameter, it will be the set speed to feed.

(23) “Switch coordinates’ display”: Press “F1 relative coordinate” “F2 absolute coordinate” “F3 comprehensive coordinate”.

(24) “Workpiece number clear”: Press Del and Enter.

(25) User-defined key “K1”: Open/close Y24

(26)User-defined key “K1”: Open/close Y25

(27)User-defined key “K1”: Open/close Y26

(28)“Incremental coordinate”: Press “Setup” to fix or set 0 after select “relative” coordinate.

(29) Datum point of adjust tool: Press “T” to set the datum point(Z axis) of adjust tool automatically which be used for setting the datum of tool length compensation in redeem parameter.

(30) Adjust tool automatically: After specifying the datum point of adjust tool, install the tool and move tool to the adjust point, press “F5” input the tool number and the corresponding length compensation is confirmed.

Pay attention: Lathe coordinate clear: Press “E” in parameter and then press “Enter”.

3.4.2 Manual continuous

Continuous operation is to press the time as the basis, Press to feed, up to stop feeding. Making sure the axis and using “+X、-X、+Y、-Y、+Z、-Z、+A、-A、+B、-B” to feed, the speed of feed is determined by display on the interface(F) times the rate.

When continuous starting, press “  ” to switch the speed to No.1 No.2 No.3 parameter value in speed parameter. If set the speed higher than the speed in parameter, the feed speed will be No.1 No.2 No.3 parameter in speed parameter times rapid override.

In order to facilitate the user single axis cutting in the manual function, setting the manual speed in manual status. Press “F” and input the speed.

When the hard limit point beyond positive and negative feed running axis two direction at, stop the feed and prompt to feed reverse direction.(the same as hereinafter)

The manual maximum speed is limited by No.4 (No.6 in bus system) parameter in speed parameter, when setting the speed is higher than the value of parameter, then will be the No.4 parameter.

When No.38 other parameter is 8, “  ” is change into a switch, press once to turn on (no more to always press), press again to turn off.

3.4.3 Manual increment

This operation is to set the value of increment as the basis, press “↑ ↓ ← →” once to run a value of increment. It will prompts “I=0010.000” in manual increment represent for the value of increment is 10mm, press “I” to revise and Enter. But also press “handwheel” and “  ”to switch the value into 0.001mm 0.01mm 0.1mm.

The speed is the speed on display(F) times the rate.

3.4.4 Back to lathe’s datum point (reference point)

There are two ways to back to datum point in this system, not only the switch for datum point, but also can set floating point, the methods as follows:

Switch for datum point:

Back to datum point operation is to feed every axis to lathe’s datum point position in turn. When the parameter of feeding axis which back to datum point is 0, the axis of coordinate detects the datum point and return to the pulsing signal of “Zero”, the data of lathe’s coordinate will be 0 automatically.

Switch on the power supply of the system, release alarm and the button of emergency after the CNC is power off, the need to back to datum point to set lathe’s coordinate correctly.

Instruction:

1. The system requires for backing to the datum point every time when it is power on, the requirement can be set by No.38(No.45 in bus system) parameter in axis parameter, it can be prompt or force;
2. The way and type of detecting signal can be set by No.39(No.46 in bus system) parameter in axis parameter, so detect the switch of datum point is effective, also detect the Z pulsing signal of electrical motor after detecting the switch of datum point (precision higher), detect forward or reverse for Z pulsing signal of electrical motor.
3. The direction for backing to datum point can be set by No.40(No.47 in bus system) parameter in axis parameter, D2 D3 D4 correspond to X Y Z axis, 0 is forward, 1 is reverse.
4. The type of the switch for datum point can be set by No.41(No.48 in bus system) parameter in axis parameter, D0 D1 D2 correspond to X Y Z axis,0 is always on, 1 is always off.

5. The maximum length of detecting Z pulse of electrical motor can be set by No.37 No.38 No.39 (No.49-No.53 in bus system) parameter in axis parameter, the value must less than the pulse of electrical motor run a cycle.
6. The shifting distance after backing to datum point can be set by No.46 No.47 No.48 (No.54-No.58 in bus system)parameter in axis parameter, rapid move coordinate to the value of parameter after backing to datum point.

No switch for datum point:

To set floating point to make sure, turn on corresponding function of floating point by No.33(No.39 in bus system) parameter in axis parameter, setting No.34 No.35 No.36(No.40-No.44) parameter to make sure the floating point of X Y Z axis, the datum point of lathe.

The steps to set floating point as follows:
1. Setting the No.33(No.39 in bus system) parameter in axis parameter to set the axis which is starting up floating point. For example: Turn X axis on is “00001000”. (turn all of them on is 00111000.)
2. Moving X axis to designated position so that set floating point.
3. Press “Parameter”, “Axis parameter” and select No.34(No.40 in bus system) parameter, “Enter”, popup a dialog box of X axis’ floating point coordinate. Import the value of setting lathe coordinate.

If it is 0, the lathe coordinate of X axis now is the datum point of X axis. The lathe backs to this position every time when backing to the datum point.

If it is 15, the current lathe coordinate of X axis is 15.000, the distance to lathe’s datum point is 15mm.

The method to set floating point of Y Z axis is the same as the above to set X axis.

Operation for backing to the datum point:

At the manual condition, press “  ”and select X Y Z A B axis to back to the datum point in dialog box.Or import 0 to make the axis back in order, the cycle will turn to green in front when backing to the datum point successfully, defeat otherwise.

If stop in the process, press “Stop” or “Reset” to stop backing to the datum point.

Pay attention:

Every time to power up the system must back to the datum point to make sure the accuracy of lathe process. The system power off unusually or in an accident, it must back to the datum point, otherwise could cause trouble.

3.4.5 Handwheel

Two types: hand held and panel, No.1 parameter in other parameter to set.

Panel

Press “handwheel” and “X” “Y” “Z” “A” “B” to select an axis, “  ” to adjust the gear.

Hand held

Press “handwheel” and operate the switch of axis selection to select an axis, operate the axis and switch of handwheel override to adjust the gear.

Instruction

The handwheel is mainly used for “Tool”, the speed and the handwheel feed of one measure is related to rotate the handwheel fast or low. The speed is not too fast best when the system cooperate with stepping motor.

Handwheel’s pulse generator speed to be lower than 200r/min(The handwheel to 100 pulse a cycle), the handwheel’s acceleration is controlled by No.24(No.26 in bus system) parameter in speed parameter(the bigger the faster). The maximum speed is controlled by No.26(No.29 in bus system)(X axis) No.27(No.30 in bus system)(Y axis) No.28(No.31 in bus system)(Z axis).

Handwheel is of no effect in auto-coordinates diagram machining, it only works in working coordinates.

3.4.6 Workpiece coordinate system

1. Press "F8" to select coordinate or "F5 MDI" to choose cooresponding workpiece coordinate system(G54-G59);
2. Move the axis which is going to be set tool to specified position;
3. Press "F7" to set coordinate and import cooresponding coordinate of workpiece coordinate system.

3.5 Auto operation

Auto refers to processing the editing program of workpiece. This system can start at arbitrary point, and also can start at arbitrary line or with arbitrary tool. Starting arbitrary line or with arbitrary tool must use absolute coordinate to edit the program. Auto operation can’t move the manual coordinate.

Running program selection: In the program interface, press “↑ ↓” to move the cursor to a program which is going to be carry out, press “F7” to select the program to carry out automatically.

Switch display of coordinate: Press “F1 F2 F3” to switch the interface into “absolute coordinate” “relative coordinate” “comprehensive coordinate”.

3.5.1 Automatical process

“Single or continuous”: Press “  ” to switch cycle.

"Continuous": The program continue to execute every program segment(program line) to end or the instruction of stop to stop.

"Single ": The program just execute one program line and end, wait another operation or press "Run" again to execute one next program line.

"Coordinate or figure": Press "  " to switch cycle.

"Automatically coordinate": The axis of coordinate will display with value.

"Automatically figure": The axis of coordinate will display with a figure. There are two kinds of figure, horizontal lathe and slant-bed lathe, No.3 parameter in tool parameter to control.

"  ": The program is speedy simulate, the axis of coordinate can't move.

3.5.2 Processing at arbitrary program line or with arbitrary tool

A. Run at actual line

At the automatical process condition, press "—" to popup a dialog box, import a number of line, press "Enter" to confirm, the line will be the line to run.

Pay attention:1. The line is the actual line in the program, not the "N" stand for the line. The system process to the line you import with a speed which is set by No.6(No.8 in bus system) parameter in speed parameter(G01/G02/G03), then process the program normally.

2. The line of default is the line of suspend the program last time, to facilitate user's operations.

3. At the interface of coordinate to use "N" to search line and press "Reset" to back to the beginning of program.

B. Run at the marked line

The system has a function to run at the marking line. At automatical process condition, press "N" to popup a dialog box to import the marking line, press "Enter" to confirm. Press "Run" to process program at the line you import(mark).

Pay attention:

The line is not the actual line, is the "N" stand for the line. The system process to the line you import with a speed which is set by No.6(No.8 in bus system) parameter in speed parameter(G01/G02/G03), then process the program normally.

C. Some tool to run

The system has a function to run at some tool. At automatical process condition, press "G" and the number of tool to run(only the number of tool, not the number of compensation; Like: 0304, just import "03"), press "Enter" to confirm. Press "Run" to process program at the tool you import.

Pay attention: The system process to the line you import with a speed which is set by No.6(No.8 in bus system) parameter in speed parameter(G01/G02/G03), then process the program normally.

3.5.3 Start program

Press "Auto" to switch to automatical mode to process program, two methods as follows.

(1) Press "Run"

(2) Switch on the Run of external signal.

3.5.4 Stop processing program

Five methods as follows to stop:

(1) The instruction of program M00 M01 M02 M30 M20.

(2) Press "  " to run a current line and stop.

(3) Intervention switch in the middle or right.

(4) Switch on the Halt of external signal.

(5) Press "Reset" to stop all the actions of program.(Like spindle, tools and others)

3.5.5 Real-time control in automatical process

(1)"  ": Press once to increase or decrease 10% feed speed when the No.1 axis parameter is 0, the range is from 0% to 150%,16 gears totally; When the No.1 axis parameter is 1, external band switch takes in control, Adjust the speed of process arbitrarily in the process according to the different situation.

(2) "  ": Press once to increase or decrease G00 or manual rapid override 20%.The range is from 5% to 100%,16 gears totally. Adjust the rapid override arbitrarily according to the different situation.

(3) "  ": Press once to increase or decrease the spindle override 10% when the No.2 parameter in axis parameter is 0. The range is form 5% to 150%,16 gears totally. When the No.2 axis parameter is 1, external band switch takes in control, Adjust the speed of spindle arbitrarily in the process according to the different situation.

(4) Stop in the process: At the continuous mode in process condition, press "  " to stop running after executing a current program line, wait for operating.

(5) Suspend in the process: Turn the intervention switch right or middle and switch on external stop signal of Halt, the processing program will stop; Press “Reset” to exit automatical process mode and the program line is going to back to the first of the processing program.

(6) Keep feeding: When the process is suspending, press “Manual” to keep feeding automatically, also can adjust the coordinate, press “Auto” and “Run” to run to the point of suspend automatically to end.

(7) Exit process: Press “Reset” when processing, suspending or keep feeding.

3.5.6 The operation mode of MDI

At the manual or automatical coordinate conditions, press “M” to get into the processing mode of MDI. Processing a program line that you import in “MDI”, press “Esc” to give up and exit when importing, press “Run” to carry out the program line that you import.

3.5.7 The operation mode of Handwheel

Press “Handwheel” at automatical mode, the program of turn handwheel is processing automatically, the speed is related to the speed of “F”, feed override and turn handwheel fast or slow. This mode is for trying to process in running program usually.

Pay attention: The acceleration, deceleration and maximum speed of running handwheel are controlled by No.23-No.29(No.26-No.33, No.49 in bus system), use the acquiescent acceleration, deceleration and the speed of G00 when the parameter is set to be invalid.

3.5.8 The function of DNC

The storage space of user is 128Mbit(could be expanded to 32Gbit in bus system) in this system, use DNC to process when the processing program is greater than 128M or the remainder storage space. Switch on RS232 or USB to realize the function of DNC in this system.

A. Instruction for RS232-DNC

1. Use the dedicated communication line to connect the computer and the system to set the corresponding communication interface and speed by the system.
2. Use the dedicated communication software of this system by computer to set the corresponding communication interface and speed. Press “Send CNC program file”, select the program file to process linked, enter the status of sending program file.
3. To enter the interface of program file in NC system, press "L" to enter the status of linked process, now the upper right corner of the display interface is "RS232--DNC", press “Run” to running carry out linked process in the automatical status.

4. Turn “Intervention switch” to middle or right to stop the running system in the process of linked process, press “Stop” or “Reset” to exit the status of linked process.

Pay attention: 1. The baud rate is related to operational environment when using serial port to send files.

2. The communication cable can’t more than 10 meters length.

3. Only the dedicated communication software of this system can send program in user’s computer. To set the sending speed of PC as the NC, defeat otherwise.

B. Instruction of USB-DNC

USB-DNC is realized by U-disk, switch on U-disk and system, select program to execute in U-disk.

Press “B” to open U-disk in program interface, select corresponding program to press “C” to execute program, press “Auto” to get into automatical mode and press “Run” to process the program.

Pay attention: 1. Don’t unplug U-disk in the process of USB-DNC, defeat processing otherwise.

2. Back to the system program interface from U-disk interface after finish USB-DNC.

3. After selecting the program, it is best to press “P” to compile once to make sure the program is right before executing program of USB-DNC.

3.6 Operate safety, prompt alarm

3.6.1 Emergency stop

Press “  ” when emergency accidents happening, the system will stop all the actions of lathe and shows “Emergency stop” on the interface. Wait for the button up. M67 imports effective signal when No.29 parameter in other parameter to be set effectively.

Press “  ” in the process or running lathe, system coordinate and lathe’s position may change, make sure the system coordinate again before processing, it is

best to carry out operation of backing to the datum point to make coordinate same as the lathe's position.

The button can be external which is controlled by No.27 parameter in other parameter to set it always open or close.

3.6.2 Reset system

Press "Reset" to stop current operation in anytime when the system is running, especially stop all the actions of lathe(spindle, tools and so on) in automatical or manual mode, but the coordinate won't lose, so needn't to back to the datum point.

3.6.3 Alarm

The screen shows error information and twinkles when the lathe has alarm, the program is stop running, the coordinate stop moving, check the reason for alarm and clear troubles to run again. The signal M67 is effective when No.29 parameter in other parameter is "1".

(1) X Y Z A B axis are limited forcedly positive: X or Z axis is in the positive position which is limited forcedly.

(2) X Y Z A B axis are limited forcedly negative: X or Z axis is in the negative position which is limited forcedly.

(3) Spindle and inverter (frequency changer) alarm: The alarm signal of lathe's inverter is effective. (ALM1)

(4) No.0 alarm: The alarm signal of lathe's spindle is effective.(ALM2)

(5) X Y Z A B axis' driver alarm: The alarm signal of servo drivers is effective. (ALM). Press "B" to import INTH signal to reset the servo drivers in diagnosis mode.

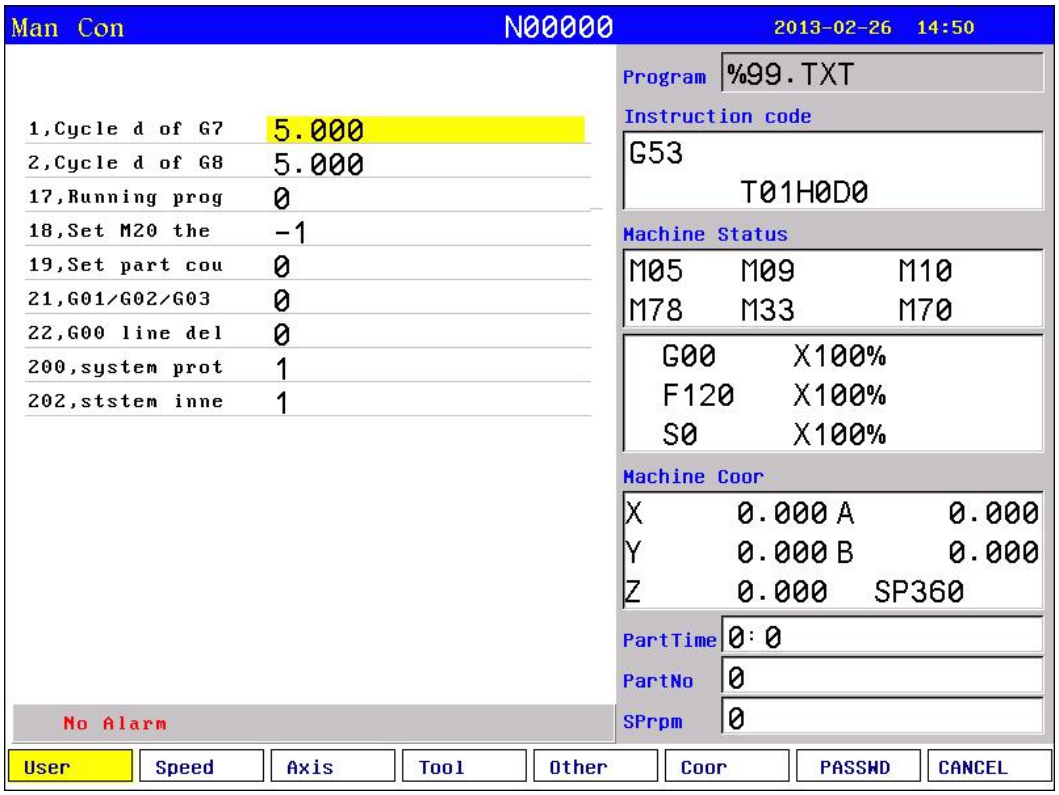
(6) No.5 alarm for door switch: The alarm signal of M12(door switch) is effective.

(7) +5V is under voltage: Supply voltage is low, +5V of the system is low.

(8) Emergency stop: Press the button of emergency stop.

3.7 Parameter operation

At any status conditions, press "parameter" to enter the status to set the parameter. Parameter in this system includes "processing parameter" "speed parameter" "axis parameter" "tool parameter" "other parameter" "coordinate" "password", 7 kinds totally.



Press F1-F8 correspondingly to corresponding interface after enter the parameter interface, press "↑ ↓" to select the number of parameter and press "Enter" to popup a dialog box to import data and press "Enter" again to fix parameter successfully.

Instructions for parameter as follows:

3.7.1 User parameter (processing)

Pay attention: More details for instructions refer to chapter 2.

1, Cycle d of G73 (mm)
2, Cycle d of G83 (mm)
3, Cycle Q direction of G76 G87(As G17 for example, 1 means +X, 2 means -X, 3 means +Y, 4 means -Y)
4, Cycle Q direction of G87(As G17 for example, 1 means +X, 2 means -X, 3 means +Y, 4 means -Y)
5, Cycle spindle angle of G76 G87 (0.1 degree)
6, Mode of G84 G74 (0 means high speed to d, 8 means to R)
7, Cycle d of G84 G74 (mm)
17, Running program need spindle run
[1 mean Yes, 0 mean No]
18, Set M20 the time of auto running

[Negative means immensity loop]
19, Set part count
21, G01/G02/G03 line delay (ms)[>100]
22, G00 line delay(ms) [>100]
23, Smooth acceleration/deceleration [50-100]
33, Whether M03/M04/S check the rotation speed is reached or not (0 means check M69 auxiliary relay, 8 means check feedback of encoder)
34, Check the permissible error of rotation speed is reached through the feedback of encoder(rpm)
200, System protect times[>=2minutes]
203, Whether strict check inspection G41/G42(34818 means no, 6326274 means yes)
210, Enable graphics display(8 means manual, 0 means automatically)
211, Show X axis negative end
212, Show X axis positive end
213, Show Y axis negative end
214, Show Y axis positive end
215, Show Z axis negative end
216, Show Z axis positive end
230, Select executive program through the input point (+4 +8 +16 +32 +64 +128 corresponding to X26-X31)
231, "Delete" (0 means backward deletion, 1 means forward delete)
232, Whether check the datum point of spindle before the tapping or not (18 means check, 0 means not)
233, Teaching function of G06 arc (0 means close, 1 means turn on)

Note:

21, This parameter could solve the program of over cut at the corner (G01)

230, Example: If set as +4+8=12, when X26 or X27 is effective, it will choose the program HIDEFILEX26 or X26 or HIDEFILEX27 or X27.

3.7.2 Speed parameter

1,X-axis's G00 speed(mm/min)
2,Y-axis's G00 speed(mm/min)
3.Z-axis's G00 speed(mm/min)
4.A-axis's G00 speed(mm/min)
5,B-axis's G00 speed(mm/min)
5a,C-axis's G00 speed(mm/min)
5b,Xs-axis's G00 speed(mm/min)
5c,Ys-axis's G00 speed(mm/min)
6.Manual maximum feed speed(mm/min)

7,Auto Maximum feed speed(mm/min)
8,G01/G02/G03 default speed(mm/min)
9,Null run speed(mm/min)
10,Feeding axis`s manual speed(mm/min)
11,Spindle`s manual speed(rpm)
12,Beginning feed speed(mm/min)
13,The maximum mutation of feeding axis speed(mm/min)
14,Limit G1G2G3 axis speed
[1 mean Yes,0 mean No]
15,X G1G2G3 max speed(mm/min)
16,Y G1G2G3 max speed(mm/min)
17.Z G1G2G3 max speed(mm/min)
18.A G1G2G3 max speed(mm/min)
19,B G1G2G3 max speed(mm/min)
19a,C G1G2G3 max speed(mm/min)
19b,Xs G1G2G3 max speed(mm/min)
19c,Ys G1G2G3 max speed(mm/min)
20,X acceleration
[1～99999]
21,Y acceleration
[1～99999]
22.Z acceleration
[1～99999]
23.A acceleration
[1～99999]
24,B acceleration
[1～99999]
24a,C acceleration
[1～99999]
24b,Xs acceleration
[1～99999]
24c,Ys acceleration
[1～99999]
25,Auto run acceleration
[1-500]
26,Handwheel acceleration
[500--30000]
27,Run program Handwheel acceleration
[>500]
28,Run program Handwheel G00 speed(mm/min)

[>10]
29,Handwheel X limit speed(mm/min)
30,Handwheel Y limit speed(mm/min)
31.Handwheel Z limit speed(mm/min)
32.Handwheel A limit speed(mm/min)
33,Handwheel B limit speed(mm/min)
33a,Handwheel C limit speed(mm/min)
33b,Handwheel Xs limit speed(mm/min)
33c,Handwheel Ys limit speed(mm/min)
33-1,Make thread Z acceleration
33-2,Make thread X acceleration
33-3,Servo motor screw thread X axis backing speed
34,Acceleration type
[0 mean line,8 mean curve]
35,Curve initial acceleration
[>=10]
36,Curve acceleration with second-order
[>=10]
37,Curve max acceleration
[>=500]
38,X go home positive speed(mm/min)
39,X go home negative speed(mm/min)
40,Y go home positive speed(mm/min)
41,Y go home negative speed(mm/min)
42.Z go home positive speed(mm/min)
43.Z go home negative speed(mm/min)
44.A go home positive speed(mm/min)
45.A go home negative speed(mm/min)
46,B go home positive speed(mm/min)
47,B go home negative speed(mm/min)
48,Spindle max speed in the first gear(rpm)
49,Spindle max speed in the second gear(rpm)
50.Spindle max speed in the third gear(rpm)
51,Spindle max speed in the fourth gear(rpm)
52,Second Spindle max speed(rpm)
53, The mode of arc gap compensation (0 means A: The lager gap the faster compensation speed,8 means B: compensation speed is set by the parameter, +4 means: Arc programme I J K are the coordinate which is from the end point to the center)
54, The speed of the B type of gap compensation (mm/min)

54-1, The beginning speed of the B type of gap compensation (mm/min)[>10]
54-2, The acceleration of the B type of gap compensation(mm/min)/s)[>10]
55, Activate the function of speed processing [1 means yes, 0 means no]
56, Forcedly limit drop speed critical(mm/min)
57, Handwheel stop speed(mm/min)[>18]
58, Follow the tapping knife when the spindle speed (rpm)[>1]
58-1, When tapping spindle backlash compensation(pulse)
58-2, Follow the tapping cutter withdrawal before reversal(um)[10-5000]
58-3, Tapping back speed(mm.min)[>=60]
60,G01/G02/G03 smooth running(1 means no, 60 means yes)
61,G01/G02/G03 running time normal[2-50]
101,X-beginning feed speed(mm/min)[>1]
102,Y-beginning feed speed(mm/min)[>1]
103,Z-beginning feed speed(mm/min)[>1]
104,The fourth axis beginning feed speed(mm/min)[>1]
105,The fifth axis beginning feed speed(mm/min)[>1]
106,The sixth axis beginning feed speed(mm/min)[>1]
107,The seventh axis beginning feed speed(mm/min)[>1]
108,The eighth axis beginning feed speed(mm/min)[>1]
111,X-jump speed at continuous(mm/min)[>1]
112,Y-jump speed at continuous(mm/min)[>1]
113,Z-jump speed at continuous(mm/min)[>1]
114,The fourth axis jump speed at continuous(mm/min)[>1]
115,The fifth axis speed at continuous(mm/min)[>1]
116,The sixth axis speed at continuous(mm/min)[>1]
117,The seventh axis speed at continuous(mm/min)[>1]
118,The eighth axis speed at continuous(mm/min)[>1]
121,The sixth axis go home positive speed
122,The seventh axis go home positive speed
123,The eighth axis go home positive speed
200,G00 continuous run is valid[1 means no, 16 means yes]
210,Thread is waiting for the spindle speed[0 means no, 1 means yes]
230,Spindle G00 speed in locating(0.1 rpm)
231,Spindle G01 mode in locating[+4 means F, +8 means G90/G91, +16 means pulse]
232,Spindle locating direction in interpolation mode[0 means positive, 1 means negative, others mean nearest]
233,Control mode of spindle back to zero(1 means pulse, 2 means driver control: output M61, check M22)

Note:

18-21: This value is all about the lathe configuration, it will cause accident or fault if set not right. In generally, the value couldn't greater than 15000 when match the step driver.

101-108: Used for setting the original speed of the raising speed when feeding axis is running, also is the end speed of reduced speed. There is no process when the running speed of feeding speed is lower than the this value, raising speed start from this value when the running speed higher than this value.

Pay attention: This value is all about the lathe configuration, it will cause accident or fault if set not right.

In generally, not greater than 100 if match the step driver, not greater than 500 if match the servo driver.

111-118: Used for increasing the coherence of continuous track interpolation with multi axis

Example: This value is 300, the speed of X axis start from F800 to F1600, the process is the speed F800 change to 1100 and then F1600.

3.7.3 Axis parameter

1,Feed axis band switch
[1 mean Yes,0 mean No]
2,Spindle band switch
[1 mean Yes,0 mean No]
3,X-axis`s negative scope(mm)
4,X-axis`s positive scope(mm)
5,Y-axis`s negative scope(mm)
6,Y-axis`s positive scope(mm)
7,Z-axis`s negative scope(mm)
8,Z-axis`s positive scope(mm)
9,A-axis`s negative scope(mm)
10,A-axis`s positive scope(mm)
11,B-axis`s negative scope(mm)
12,B-axis`s positive scope(mm)
12a,C-axis`s negative scope(mm)
12b,C-axis`s positive scope(mm)
12c,Xs-axis`s negative scope(mm)
12d,Xs-axis`s positive scope(mm)
12e,Ys-axis`s negative scope(mm)
12f,Ys-axis`s positive scope(mm)
13,Spindle stop time(10ms)
14,Spindle stop long signal
[0 mean No,1 mean Yes]
15,Soft limited invalid
[D2X;D3Y;D4Z;D5A;]
16,X-axis`s reverse compensation(um)

17,Y-axis`s reverse compensation(um)
18,Z-axis`s reverse compensation(um)
19,A-axis`s reverse compensation(um)
20,B-axis`s reverse compensation(um)
20a,C-axis`s reverse compensation(um)
20b,Xs-axis`s reverse compensation(um)
20c,Ys-axis`s reverse compensation(um)
21,X-axis's direction signal
[1 mean normal,0 mean reverse]
22,Y-axis's direction signal
[1 mean normal,0 mean reverse]
23,Z-axis's direction signal
[1 mean normal,0 mean reverse]
24,A-axis's direction signal
[1 mean normal,0 mean reverse]
25,B-axis's direction signal
[1 mean normal,0 mean reverse]
25a,C-axis's direction signal
[1 mean normal,0 mean reverse]
25b,Xs-axis's direction signal
[1 mean normal,0 mean reverse]
25c,Ys-axis's direction signal
[1 mean normal,0 mean reverse]
26,Close feed electron gear
[1 mean Yes,0 mean No]
27,X-axis's electron gear numerator(1-999999)
28,X-axis's electron gear denominator(1-999999)
29,Y-axis's electron gear numerator(1-999999)
30,Y-axis's electron gear denominator(1-999999)
31,Z-axis's electron gear numerator(1-999999)
32,Z-axis's electron gear denominator(1-999999)
33,A-axis's electron gear numerator(1-999999)
34,A-axis's electron gear denominator(1-999999)
37,XYZA positive limit
[0 open,1 close]
38,XYZA negative limit
[0 open,1 close]
39,float zero bit parameter
[D3X;D4Y;D5Z;D6A;0 machine Zero;1 float Zero]
40,X coordinate of floating zero point setting
41,Y coordinate of floating zero point setting
42,Z coordinate of floating zero point setting
43,A coordinate of floating zero point setting
45,Feed axis returns requirement
[1 mean No use, 0 mean clew, 8 compulsion , 9 must compulsion]
46,Feed axis returns mode
[0 reverse check,1 reverse No check ,2 No reverse check,3 No reverse No check]
47,Home reverse direction

[D2X;D3Y;D4Z;D5A; 0 Positive;1 Negative]
48,Home switch set
[D0X;D1Y;D2Z;D3A;1Close;0Open]
49,X check zero max length(100um)
50,Y check zero max length(100um)
51,Z check zero max length(100um)
52,A check zero max length(100um)
54,X Home offset(10um)
55,Y Home offset(10um)
56,Z Home offset(10um)
57,A Home offset(10um)
59,Have Spindle class control
[1 mean open,0 mean close]
60,Spindle class speed(1/100rpm)
61,Spindle class direction
[0 mean M03,1 mean M04]
62,Spindle class stop time(10ms)
63,Spindle class time(10ms)
64,Spindle stop time(10ms)
65,Check SP encode
[1 mean Yes,0 mean No]
68, Negative delay time of feeding axis(ms)
80,XZ axis coordinate plan
[D2Zwordpiece,D3Xwordpiece,D4Ztool,D5Xtool,D6Zcircumrotate,D7Xcircumrotate
]
81, Y axis
[0 mean rotating axis,1 mean line axis]
82,Y is rotating axis workpiece coordinate
[0 No;1 plan]
83,Y is rotating axis lathe coordinate
[0 No;1 plan]
100,Inner parameter
101,The fourth axis setting function
[0 mean rotating axis,1 mean line axis]
102,The fourth axis is rotating axis lathe coordinate
[0 No;1 plan]
200,SP encode pulse(4 times encode pulse)
201,Is the measurement of spindle position feedback (1 means yes, 0 means no)
202,Spindle orientation detection angle
203,Angle measurement error of spindle orientation
204,SP motor direction(0 means reverse, 1 means normal)
205,SP-axis's electronic gear(0 means yes, 1 means no)
206,SP-axis's electronic low gear numerator(1-999999)
207,SP-axis's electronic low gear denominator(1-999999)
208,SP-axis's electronic high gear numerator(1-999999)
209,SP-axis's electronic high gear denominator(1-999999)
210,Z-axis interpolation tap SP name[91 means X, 92 means Y, 93Z, 94A, 95B, 96C,
97Xs, 98Ys]

210a,Y-axis interpolation tap SP name[91 means X, 92 means Y, 93Z, 94A, 95B, 96C, 97Xs, 98Ys]
210b,X-axis interpolation tap SP name[91 means X, 92 means Y, 93Z, 94A, 95B, 96C, 97Xs, 98Ys]
211,Interpolation tap mode[2 means follow encode, 3 means interpolation to SP]
212,SP tooth number(<P213)
213,Encode number(>P212)
214,A-axis is moving by(7 means X, 8 means Y, 9 means Z)
220,First spindle full pulse control channel(88 means positive, 98 means reverse)
221,The first spindle full pulse control of the number of pulses per cycle
222,First spindle full pulse speed control(rpm/2ms)
223,Second spindle full pulse control channel(positive:82Y/84A/85B/86C, reverse:92Y/94A/95B/96C)
224,Second spindle full pulse control of the number of pulses per cycle
225,Second spindle full pulse speed control(rpm/2ms)
250,Ethernet bus servo motor encoder bus number
511,X-axis servo drive parameter P8
512,X-axis servo drive parameter P9
513,X-axis servo drive parameter P10
514,X-axis servo drive parameter P11
515,X-axis servo drive parameter P32
516,X-axis servo drive parameter P33
517,X-axis servo drive parameter P44
518,X-P30 M133 mode(0 means constant speed, 1 means oscillation, 2 means fixed length)
518-1,X-P40 M133 length low-16bits
518-2,X-P41 M133 length high-16bits
518-3,X-P46 M133 delay(ms)
521,Y-axis servo drive parameter P8
522,Y-axis servo drive parameter P9
523,Y-axis servo drive parameter P10
524,Y-axis servo drive parameter P11
525,Y-axis servo drive parameter P32
526,Y-axis servo drive parameter P33
527,Y-axis servo drive parameter P44
528,Y-P30 M133 mode(0 means constant speed, 1 means oscillation, 2 means fixed length)
528-1,Y-P40 M133 length low-16bits
528-2,Y-P41 M133 length high-16bits
528-3,Y-P46 M133 delay(ms)
531,Z-axis servo drive parameter P8
532,Z-axis servo drive parameter P9
533,Z-axis servo drive parameter P10
534,Z-axis servo drive parameter P11
535,Z-axis servo drive parameter P32
536,Z-axis servo drive parameter P33
537,Z-axis servo drive parameter P44
538,Z-P30 M133 mode(0 means constant speed, 1 means oscillation, 2 means fixed

length)
538-1,Z-P40 M133 length low-16bits
538-2,Z-P41 M133 length high-16bits
538-3,Z-P46 M133 delay(ms)
548,A-P30 M133 mode(0 means constant speed, 1 means oscillation, 2 means fixed length)
548-1,A-P40 M133 length low-16bits
548-2,A-P41 M133 length high-16bits
548-3,A-P46 M133 delay(ms)
558,B-P30 M133 mode(0 means constant speed, 1 means oscillation, 2 means fixed length)
558-1,B-P40 M133 length low-16bits
558-2,B-P41 M133 length high-16bits
558-3,B-P46 M133 delay(ms)
568,C-P30 M133 mode(0 means constant speed, 1 means oscillation, 2 means fixed length)
568-1,C-P40 M133 length low-16bits
568-2,C-P41 M133 length high-16bits
568-3,C-P46 M133 delay(ms)
578,C-P30 M133 mode(0 means constant speed, 1 means oscillation, 2 means fixed length)
578-1,Xs-P40 M133 length low-16bits
578-2,Xs-P41 M133 length high-16bits
578-3,Xs-P46 M133 delay(ms)
588,Xs-P30 M133 mode(0 means constant speed, 1 means oscillation, 2 means fixed length)
588-1,Ys-P40 M133 length low-16bits
588-2,Ys-P41 M133 length high-16bits
588-3,Ys-P46 M133 delay(ms)

3.7.4 Tool parameter

1,Radius C compensation's establish(0 mean A,1 mean B)
2,Radius C compensation's cancel(0 mean A,1 mean B)
32,Tool position signal/WAT signal filter
[+256+512+1024:2/4/8ms/+2048+4096+8192:2/4/8ms]

3.7.5 Other parameter

1,Set sub-panel type
[1 hand hold,0 panel]
3,use control switch
[1 Yes, 0 No]
4,Have auto lubricate(0 yes/1 no)
5,Auto lubricate time(0.01s)
6,Auto lubricate stop time(s)
7,Door switch checking M12(0 no,1 yes)

8,Door switch(0 open,1 close)
9,bit parameter
D0：Null;
D1：“1”Start CNC system clear part Number.;
D2：“1”Automatic space before letter when editing program;
D3：Null;
D4：Null;
D5：“1”Do not stopping SP and cooling when pressing “Reset”;
D6：“1”G00 X and Z’s peed by oneself;
D7：“1”Tool redeem by oneself;
D8：“1”Save SP chuck(M10/M11) state when power off;
D9：Tool redeem input Mode1 or Mode2;
D10：“1” Program edit automatic compositor Line;
D11：“1” First SP +10V output from second output port;
D12：“1” Shield skip function (“/”is invalidation);
D13：“1” Sheld go home function;
D14：“1” Sheld “run” key;
D15：“1” Tool redeem display relative,“0”absolute;
10,Auto count part
[1 mean Yes,0 mean No]
11,Program edit number increase
12,Inner parameter
13,Does lock for Spindle & chuck(0 mean no)
14,Is available keys of lub&cool as running(0 mean no)
17,servo ALM (0 open,1 close)
18,SP ALM1 (0 open,1 close)
19,Lathe ALM2 (0 open,1 close)
20,Chuck control signal(0 single,1 doubleM10/M71)
22,External chuck control(0 no,1 yes)
24,M10M11 short signal time(s)
26,Emerge Stop(0 open,1 close)
27,Emerge Stop2(0 open,1 close)
28,Run status output M69 STOP output M65(0 invalid,1 valid)
29,Alarm status output M67(0 invalid,1 valid)
30,Set language(1means Chinese, 0 means English)
31,Is enable I/O PLC program
32,Is enable High speed I/O PLC program
35,soft-limit without home as manual
[1 Yes,0 No]
36,Set system time

[year-month-day-hour-minute]
37,Velocity of RS232
[0=7200；1=9600；2=14400；3=19200；4=38400；5=57600；6=115200]
38,Manually rapid key to lock and save
[8 Yes]
39,Special parameter
40,Special parameter
41,Bake current parameter
42,Resume original parameter
50,Run from middle program ask going last line point[8 means yes, 0 means no]
120,Manual axis moving keying reverse
(4 means X, 8 means Y, 16 means Z, 32 means A)
200.X axis feed back the error of running alarm. (pulse)[>1]
201.Y axis feed back the error of running alarm. (pulse)[>1]
202.Z axis feed back the error of running alarm. (pulse)[>1]
203.Fourth axis feed back the error of running alarming. (pulse)[>1]
204.Fifth axis feed back the error of running alarming. (pulse)[>1]
205.X axis feed back the error of stop alarming. (pulse)[>1]
206.Y axis feed back the error of stop alarming. (pulse)[>1]
207.Z axis feed back the error of stop alarming. (pulse)[>1]
208.Fourth axis feed back the error of stop alarming. (pulse)[>1]
209.Fifth axis feed back the error of stop alarming. (pulse)[>1]
210.X axis feed back electrical gear molecules.
[Automatical calculate and import: crew pitch (um) and the number of encoder.]
211.Y axis feed back electrical gear molecules.
[Automatical calculate and import: crew pitch (um) and the number of encoder.]
212.Z axis feed back electrical gear molecules.
[Automatical calculate and import: crew pitch (um) and the number of encoder.]
213.Fourth axis feed back electrical gear molecules.
[Automatical calculate and import: crew pitch (um) and the number of encoder.]
214.Fifth axis feed back electrical gear molecules.
[Automatical calculate and import: crew pitch (um) and the number of encoder.]
215.X axis feed back electrical gear denominators.
[Automatical calculate and import: crew pitch (um) and the number of encoder.]
216.Y axis feed back electrical gear denominators.
[Automatical calculate and import: crew pitch (um) and the number of encoder.]
217.Z axis feed back electrical gear denominators.
[Automatical calculate and import: crew pitch (um) and the number of encoder.]
218.Fourth axis feed back electrical gear denominators.
[Automatical calculate and import: crew pitch (um) and the number of encoder.]

219.Fifth axis feed back electrical gear denominators.
[Automatical calculate and import: crew pitch (um) and the number of encoder.]
300.Feed axis matches absolute encoder motor or not.
[X—D2, Y—D3,Z—D4,A—D5, B—D6, C—D7, Xs—D8, Ys—D9, "0" means not match, "1" means match.]
301.Absolute encoder address of lower 16bits multi-turn data.
302.Absolute encoder address of higher 16bits one-revolution data
303.Absolute encoder address of lower 16bits multi-turn data.
304.X axis of each cycle absolute encoder pulse number.
305.Y axis of each cycle absolute encoder pulse number.
306.Z axis of each cycle absolute encoder pulse number.
307.Fourth axis of each cycle absolute encoder pulse number.
308.Fifth axis of each cycle absolute encoder pulse number.
309.X axis of each cycle absolute encoder corresponding coordinate distance.[nm]
310.Y axis of each cycle absolute encoder corresponding coordinate distance.[nm]
311.Z axis of each cycle absolute encoder corresponding coordinate distance.[nm]
312.Fourth axis of each cycle absolute encoder corresponding coordinate distance.[nm]
313.Fifth axis of each cycle absolute encoder corresponding coordinate distance.[nm]
314.X axis of absolute encoder multi-turn data shifting.
[Import "E" to clear multi-turn data]
315.Y axis of absolute encoder multi-turn data shifting.
[Import "E" to clear multi-turn data]
316.Z axis of absolute encoder multi-turn data shifting.
[Import "E" to clear multi-turn data]
317.Fourth axis of absolute encoder multi-turn data shifting.
[Import "E" to clear multi-turn data]
318.Fifth axis of absolute encoder multi-turn data shifting.
[Import "E" to clear multi-turn data]
320,X-axis one-revolution coordinate denominator
321,Y-axis one-revolution coordinate denominator
322,Z-axis one-revolution coordinate denominator
380,Automatic tool X setting initial coordinate(mm)
381,Automatic tool Y setting initial coordinate(mm)
382,Automatic tool Z setting initial coordinate(mm)
383,Automatic tool forward speed(mm/min)
384,Automatic tool slow forward speed(mm/min)
385,Automatic tool Z axis coordinate value(mm)
386,Automatic tool speed of locating point(mm/min)

387,Automatic tool setting mode(1 means fixed tool, 0 means floating tool)
388,Automatic tool Z axis minimum machine tool coordinate value(mm)
389,Automatic tool Z axis difference value(mm)
500,Machine tool number
501,Interface display[1 means white, 8 means black]
601.One key to define step.
602.One key to define servo.
900,User-defined dialog box[1 means: invalid, 4 means valid]
901,Axis go home sequence(>9)[5 bits in front means X/Y/Z/A/B, last bit must is 0]
910,M18/M22/M24/M28 high speed input[1 means yes]
911,Whether to start M18 for the teaching, M28 for the recording function[1 means yes, 0 means no]
912,"Reset" to reset the output interface[1 means yes, 0 means no]

3.7.6 Coordinate system

The parameter has the function of multiple coordinate system, includes 6 workpiece coordinate system and a lathe coordinate system G53. A processing program can set a workpiece coordinate system, workpiece coordinate system could move its original to change,the value of coordinate system in parameter is coordinate value of the original point (zero point) in the lathe coordinate system.

Using G54 to G59 to set 6 workpiece coordinate system, in coordinate system interface could modify the coordinate value of original of 6 workpiece coordinate system in lathe coordinate system.

Parameter:

1-0,Current set of workpiece coordinate[G54-G59]
1-1,X of work coordinate G54-G59(mm)
1-2,Y of work coordinate G54-G59(mm)
1-3,Z of work coordinate G54-G59(mm)
1-4,A of work coordinate G54-G59(mm)
1-5,B of work coordinate G54-G59(mm)
1-6,C of work coordinate G54-G59(mm)
1-7,Xs of work coordinate G54-G59(mm)
1-8,Ys of work coordinate G54-G59(mm)
2-0,Current set of workpiece coordinate[G54.1-G54.48]
2-1,X of work coordinate G54.1-G54.48(mm)
2-2,Y of work coordinate G54.1-G54.48(mm)
2-3,Z of work coordinate G54.1-G54.48(mm)
2-4,A of work coordinate G54.1-G54.48(mm)
2-5,B of work coordinate G54.1-G54.48(mm)
2-6,C of work coordinate G54.1-G54.48(mm)
2-7,Xs of work coordinate G54.1-G54.48(mm)

2-8,Ys of work coordinate G54.1-G54.48(mm)
1. X offset of G54 mm[incremental modification, enter E to clear]
2. Y offset of G54 mm[incremental modification, enter E to clear]
3. Z offset of G54 mm[incremental modification, enter E to clear]
4. A offset of G54 mm[incremental modification, enter E to clear]
5,B offset of G54 mm[incremental modification, enter E to clear]
6,C offset of G54 mm[incremental modification, enter E to clear]
7,Xs offset of G54 mm[incremental modification, enter E to clear]
8,Ys offset of G54 mm[incremental modification, enter E to clear]
11. X offset of G55 mm[incremental modification, enter E to clear]
12 Y offset of G55 mm[incremental modification, enter E to clear]
13. Z offset of G55 mm[incremental modification, enter E to clear]
14. A offset of G55 mm[incremental modification, enter E to clear]
15,B offset of G55 mm[incremental modification, enter E to clear]
16,C offset of G55 mm[incremental modification, enter E to clear]
17,Xs offset of G55 mm[incremental modification, enter E to clear]
18,Ys offset of G55 mm[incremental modification, enter E to clear]
21. X offset of G56 mm[incremental modification, enter E to clear]
22. Y offset of G56 mm[incremental modification, enter E to clear]
23. Z offset of G56 mm[incremental modification, enter E to clear]
24. A offset of G56 mm[incremental modification, enter E to clear]
25,B offset of G56 mm[incremental modification, enter E to clear]
26,C offset of G56 mm[incremental modification, enter E to clear]
27,Xs offset of G56 mm[incremental modification, enter E to clear]
28,Ys offset of G56 mm[incremental modification, enter E to clear]
31. X offset of G57 mm[incremental modification, enter E to clear]
32. Y offset of G57 mm[incremental modification, enter E to clear]
33. Z offset of G57 mm[incremental modification, enter E to clear]
34. A offset of G57 mm[incremental modification, enter E to clear]
35,B offset of G57 mm[incremental modification, enter E to clear]
36,C offset of G57 mm[incremental modification, enter E to clear]
37,Xs offset of G57 mm[incremental modification, enter E to clear]
38,Ys offset of G57 mm[incremental modification, enter E to clear]
41. X offset of G58 mm[incremental modification, enter E to clear]
42. Y offset of G58 mm[incremental modification, enter E to clear]
43. Z offset of G58 mm[incremental modification, enter E to clear]
44. A offset of G58 mm[incremental modification, enter E to clear]
45,B offset of G58 mm[incremental modification, enter E to clear]
46,C offset of G58 mm[incremental modification, enter E to clear]
47,Xs offset of G58 mm[incremental modification, enter E to clear]

48,Ys offset of G58 mm[incremental modification, enter E to clear]
51. X offset of G59 mm[incremental modification, enter E to clear]
52. Y offset of G59 mm[incremental modification, enter E to clear]
53. Z offset of G59 mm[incremental modification, enter E to clear]
54. A offset of G59 mm[incremental modification, enter E to clear]
55,B offset of G59 mm[incremental modification, enter E to clear]
56,C offset of G59 mm[incremental modification, enter E to clear]
57,Xs offset of G59 mm[incremental modification, enter E to clear]
58,Ys offset of G59 mm[incremental modification, enter E to clear]
61,X of work coordinate G54.1(mm)
62,Y of work coordinate G54.1(mm)
63,Z of work coordinate G54.1(mm)
64,A of work coordinate G54.1(mm)
65,B of work coordinate G54.1(mm)
66,C of work coordinate G54.1(mm)
67,Xs of work coordinate G54.1(mm)
68,Ys of work coordinate G54.1(mm)
71,X of work coordinate G54.2(mm)
72,Y of work coordinate G54.2(mm)
73,Z of work coordinate G54.2(mm)
74,A of work coordinate G54.2(mm)
75,B of work coordinate G54.2(mm)
76,C of work coordinate G54.2(mm)
77,Xs of work coordinate G54.2(mm)
78,Ys of work coordinate G54.2(mm)
81,X of work coordinate G54.3(mm)
82,Y of work coordinate G54.3(mm)
83,Z of work coordinate G54.3(mm)
84,A of work coordinate G54.3(mm)
85,B of work coordinate G54.3(mm)
86,C of work coordinate G54.3(mm)
87,Xs of work coordinate G54.3(mm)
88,Ys of work coordinate G54.3(mm)
91,X of work coordinate G54.4(mm)
92,Y of work coordinate G54.4(mm)
93,Z of work coordinate G54.4(mm)
94,A of work coordinate G54.4(mm)
95,B of work coordinate G54.4(mm)
96,C of work coordinate G54.4(mm)
97,Xs of work coordinate G54.4(mm)

98,Ys of work coordinate G54.4(mm)
101,X of work coordinate G54.5(mm)
102,Y of work coordinate G54.5(mm)
103,Z of work coordinate G54.5(mm)
104,A of work coordinate G54.5(mm)
105,B of work coordinate G54.5(mm)
106,C of work coordinate G54.5(mm)
107,Xs of work coordinate G54.5(mm)
108,Ys of work coordinate G54.5(mm)
111,X of work coordinate G54.6(mm)
112,Y of work coordinate G54.6(mm)
113,Z of work coordinate G54.6(mm)
114,A of work coordinate G54.6(mm)
115,B of work coordinate G54.6(mm)
116,C of work coordinate G54.6(mm)
117,Xs of work coordinate G54.6(mm)
118,Ys of work coordinate G54.6(mm)
121,X of work coordinate G54.7(mm)
122,Y of work coordinate G54.7(mm)
123,Z of work coordinate G54.7(mm)
124,A of work coordinate G54.7(mm)
125,B of work coordinate G54.7(mm)
126,C of work coordinate G54.7(mm)
127,Xs of work coordinate G54.7(mm)
128,Ys of work coordinate G54.7(mm)
131,X of work coordinate G54.8(mm)
132,Y of work coordinate G54.8(mm)
133,Z of work coordinate G54.8(mm)
134,A of work coordinate G54.8(mm)
135,B of work coordinate G54.8(mm)
136,C of work coordinate G54.8(mm)
137,Xs of work coordinate G54.8(mm)
138,Ys of work coordinate G54.8(mm)
141,X of work coordinate G54.9(mm)
142,Y of work coordinate G54.9(mm)
143,Z of work coordinate G54.9(mm)
144,A of work coordinate G54.9(mm)
145,B of work coordinate G54.9(mm)
146,C of work coordinate G54.9(mm)
147,Xs of work coordinate G54.9(mm)

148,Ys of work coordinate G54.9(mm)
151,X of work coordinate G54.10(mm)
152,Y of work coordinate G54.10(mm)
153,Z of work coordinate G54.10(mm)
154,A of work coordinate G54.10(mm)
155,B of work coordinate G54.10(mm)
156,C of work coordinate G54.10(mm)
157,Xs of work coordinate G54.10(mm)
158,Ys of work coordinate G54.10(mm)

3.7.7 Password

In this system in order to prevent from the parameter modification in accident, make sure the lathe working, the system adopts the parameter setting of classify the authority. Divided into "CNC factory" and "Lathe factory", "User factory" three level authority. The "CNC factory" set for the function of the system,belong to internal parameter;"Lathe factory" set equipment configuration of lathe and mechanical index and some parameter about safety;"User factory" set processing technology, performance and the processing program.

The initial situation of three-level classification in this system: "CNC factory" is enabled,"Lathe factory"and "User factory" both are not enabled. If you want to enable the authority function, you must use the initial password to enable access function, then set the corresponding new access password to enable.Pay attention to the initial code is only be used once, the code will invalid after setting a new code, please remember the new code.

Pay attention: the code must be 6 digits, the code can be number and letter.

Parameter:

1. Whether to turn on the password protection of CNC manufacturer
2. Whether to turn on the password protection of lathe manufacturer
3. Whether to turn on the password protection of user manufacturer
4. Modify the password of CNC manufacturer
5. Modify the password of lathe manufacturer
6. Modify the password of user manufacturer
7. The using time of system
8. Software version

3.8 Set parameter of tool redeem

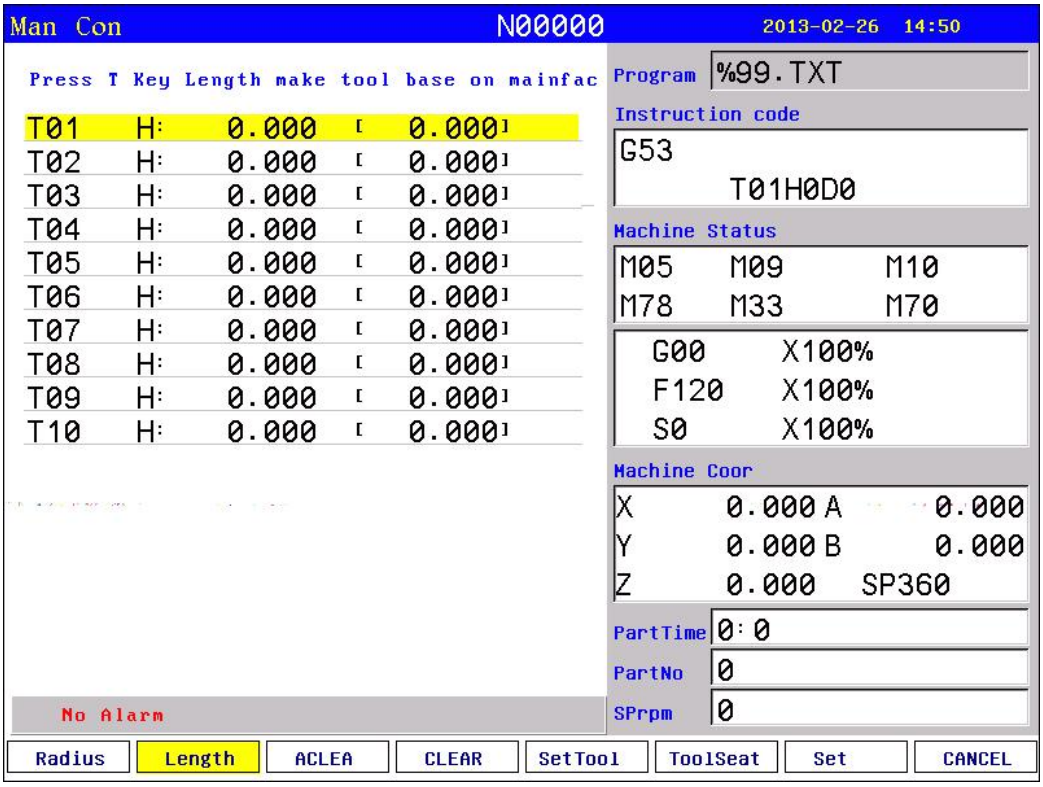
Press “Redeem” to enter interface of redeem in any interface, including “Radius compensation” “Length” “Clear all value” “Clear current value” “Set tool” “Posit tool” “Set”, total 7 functions, correspond to press “F1-F7” to enter corresponding

interface，press “Cancel” to back the primary menu interface.

3.8.1 Radius compensation

Press “F1” to enter radius compensation interface in redeem interface. The parameter is used to set adopt corner radius of the tool.

Setting method: Press “↑ ↓” to make cursor move to the corresponding tool and press “Enter” to popup a dialog box, import corresponding tool radius(Absolute value), press “Enter” at last.



3.8.2 Length of redeem

Press “F2” to enter length of redeem interface. The parameter is used to modify the length which is adopt or reset the length.

Method of modifying the length compensation:

Press “↑ ↓” to make cursor move to the corresponding tool number and press “Enter” to popup a dialog box, import the modifying axis into the dialog box and import the modifying value(import 0.05 to plus 0.05, import -0.05 to reduce 0.05), press “Enter” to confirm. The system calculates current value of redeem after finishing setting.

Method of set tool automatically:

Make lathe move to a position so that measure corresponding tool coordinate, press “↑ ↓” to make cursor move to corresponding tool number and press “F5” to popup a dialog box, import the reset axis into dialog box and import the value of measuring the workpiece of corresponding axis, press “Enter” to confirm. The length

compensation of corresponding axis has been reset. The system automatical refresh current value of redeem after finishing setting.

Method of initializing the length compensation value and radius of tool:

Press "F3(Clear all)" or "F4(Clear the current)" to initialize all the length compensation or current length compensation.

Pay attention: the length compensation can be positive or negative number, but the radius compensation only can be positive number.

3.8.3 Posit tool

Press "F6" to enter posit tool interface in redeem interface. The parameter is used to set the kind of tool when adopting radius compensation of tool.

Method of setting: Press "↑ ↓" to make cursor move to corresponding tool number and press "Enter" to popup a dialog box, import the code of corresponding tool kinds and press "Enter" to confirm.

Press "F1" to initialize all the kinds of tools to 0.

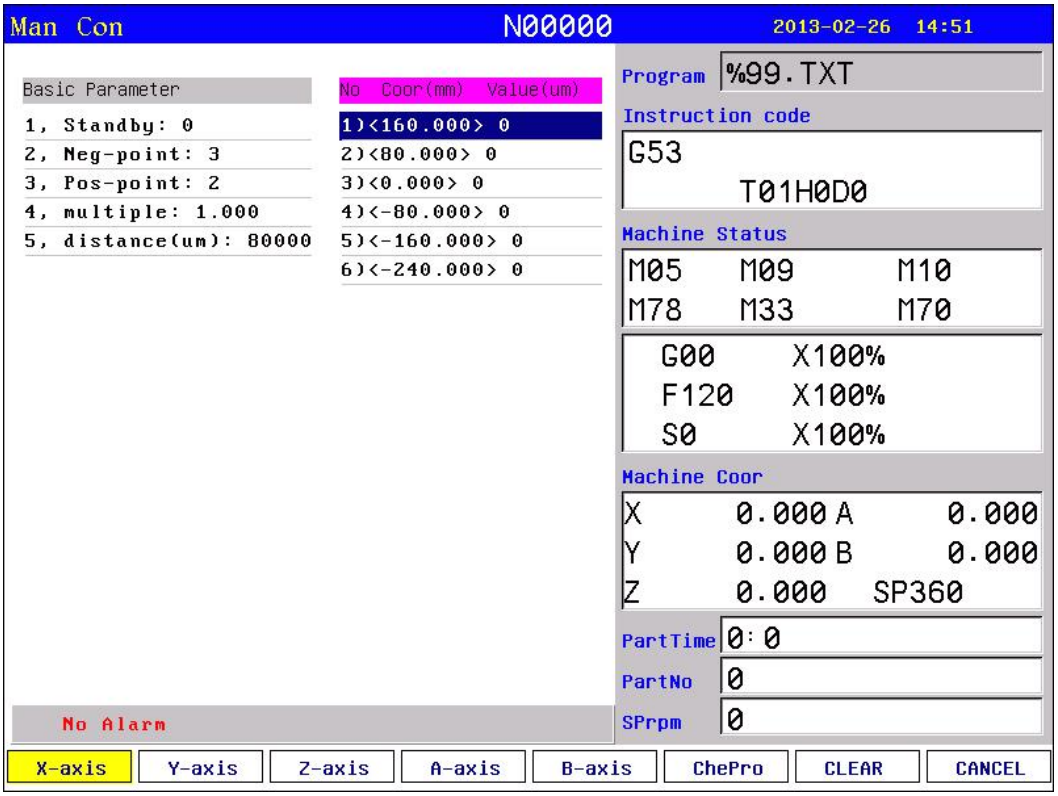
3.8.4 Set quantity

Press "F7" to popup a dialog box in the redeem interface to set and manage the total tools.The quantity of the tool in this system can be set 99.

3.9 Screw compensation

Press "Parameter" twice in parameter interface to enter screw compensation interface to set the screw compensation.

Screw compensation is used for automatical compensating the error of screw pitch, compensate the influence from the error of screw pitch to the prevision of operating lathe. The system adopts storage mode of screw compensation: Making the lathe's datum point as the starting point when debugging, measured the error curve of screw, studied out the correctional curve according to the error curve, import the value of correctional curve into the correctional parameter and system is going to compensate according to the parameter in automatical running.



Screw compensation by the axis as the unit to set storage, set X Y Z A B axis separately, by pressing "F1" "F2" "F3" "F4" "F5"to switch; Every axis of screw compensation interface has tow areas(basic parameter and set the compensation), by pressing "→ ←"to move the cursor to realize.

Basic parameter:

Press "↑ ↓" to select current basic parameter to set in basic parameter, press "Enter" to popup a dialog box to import the error compensation of every axis and import the basic information of screw compensation.

Set compensation value:

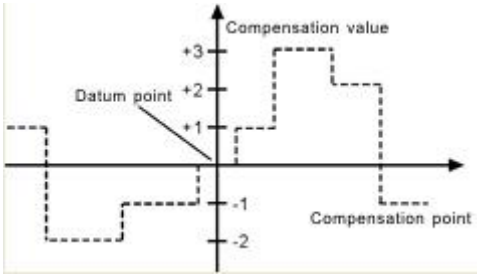
In the area of setting compensation, it will shows the value of compensation and every axis' error compensation point of screw pitch. Press "↑ ↓ PgDn PgUp" to select current compensation point and press "Enter" to popup a dialog box to import the value of compensation, import the value of current compensation point.

Test program generation automatically

Automatical generate a program of laser interferometer to check the screw compensation. Enter the screw pitch interface and set basic parameter, press checking program to popup a dialog box and press "Enter" to generate corresponding checking program of screw compensation.

The number of compensation points can be set freely, the maximum number of each axis is 300. The basic parameter of every axis' error compensation of screw pitch includes as follows:

1. Reserve.
2. Backward checking points.
3. Forward checking points.
4. Compensation override.
5. The spacing of compensation points (um).



The system calculates every axis' error compensation points' positions of screw pitch according to basic parameter automatically, every axis's error compensation points' spacing is uniform, user can import compensation value of each point (This system requires importing absolute value, relating the of datum point).

The compensation points are uniform, set the spacing into each axis.

For example:

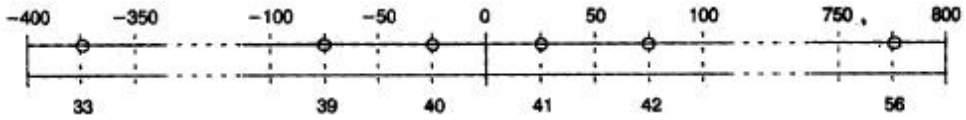
Example 1:Linear axis: when length of travel is -400mm～+800mm, spacing of points is 50mm,reference point compensation is No.40, it can figure out that Compensation point of farthest end in negative direction is:

Machine negative travel/point interval +1=40-400/50+1=33.

Compensation point of farthest end in positive direction is:

Machine positive travel/point interval +1=40+800/50=56.

The corresponding relationship between machine coordinate and compensation point is:



output compensation value in 0 position

parameters set as follows：

Compensation point No. of reference point: 40

Compensation point No. of farthest end in negative direction: 33

Compensation point No. of farthest end in positive direction: 56

Compensation override: 1

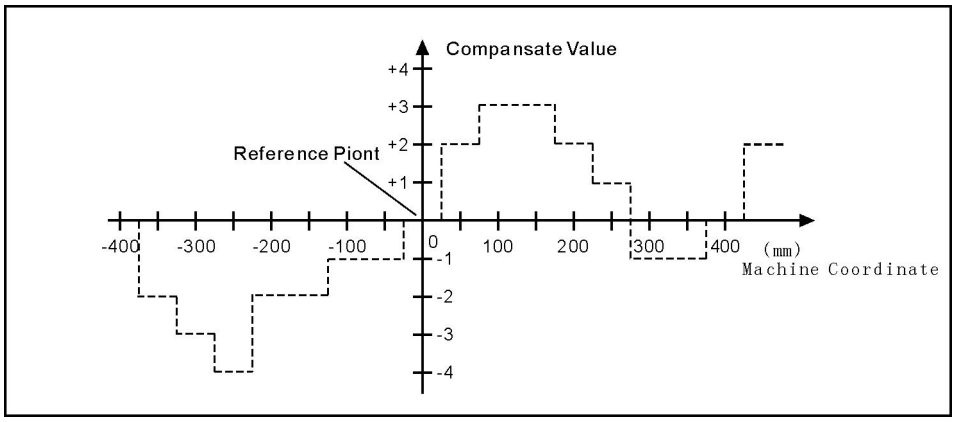
Compensation point interval: 50000

Corresponding compensation point and value:

The compensation value in corresponding compensation point:

No.	33	34	35	36	37	38	39	40	41	42	43	44	45	46	47	48	49		56
Value	+2	+1	+1	-2	0	-1	0	-1	+2	+1	0	-1	-1	-2	0	+1	+2		+1

The contrasted chart of compensation points and value as follows:

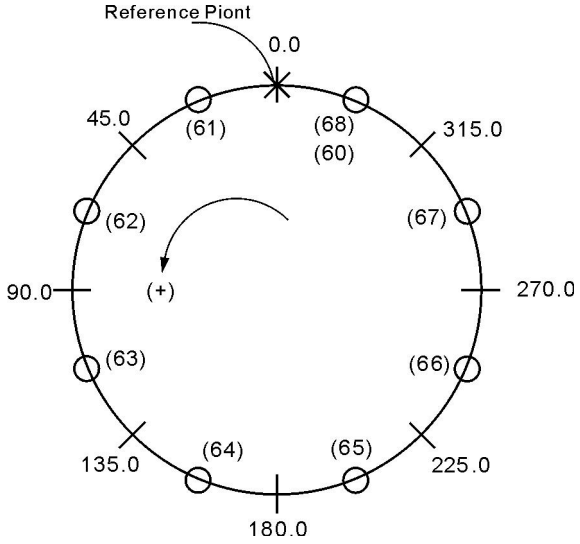


Example 2:rotor axis: when movement per revolution is 360°, interval of point is 45°,reference point compensation is NO. 60, Compensation point number of farthest end in negative direction is usually same as reference point compensation point number

Compensation point number of farthest end in positive direction is:

Reference compensation point number + movement per revolution/compensation point interval=60+360/45=68.

Machine coordinate and compensation point number correspondence is:

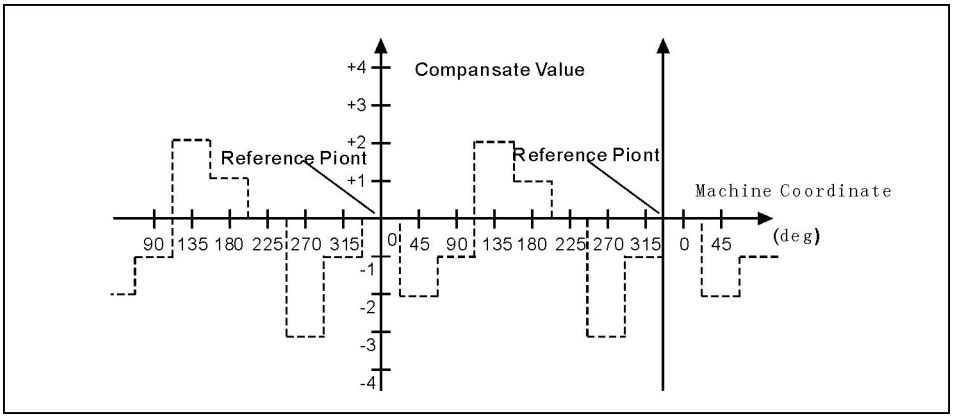


Note: input value in small circle. If the total amount from 61 to 68 doesn't equal 0, accumulated pitch error per revolution will deviate, so same value shall be put in 60 and 68.

Output compensation value at corresponding point:

NO.	60	61	62	63	64	65	66	67	68
VALUE	+1	-2	+1	+3	-1	-1	-3	+2	+1

Compensation point and value contrast:

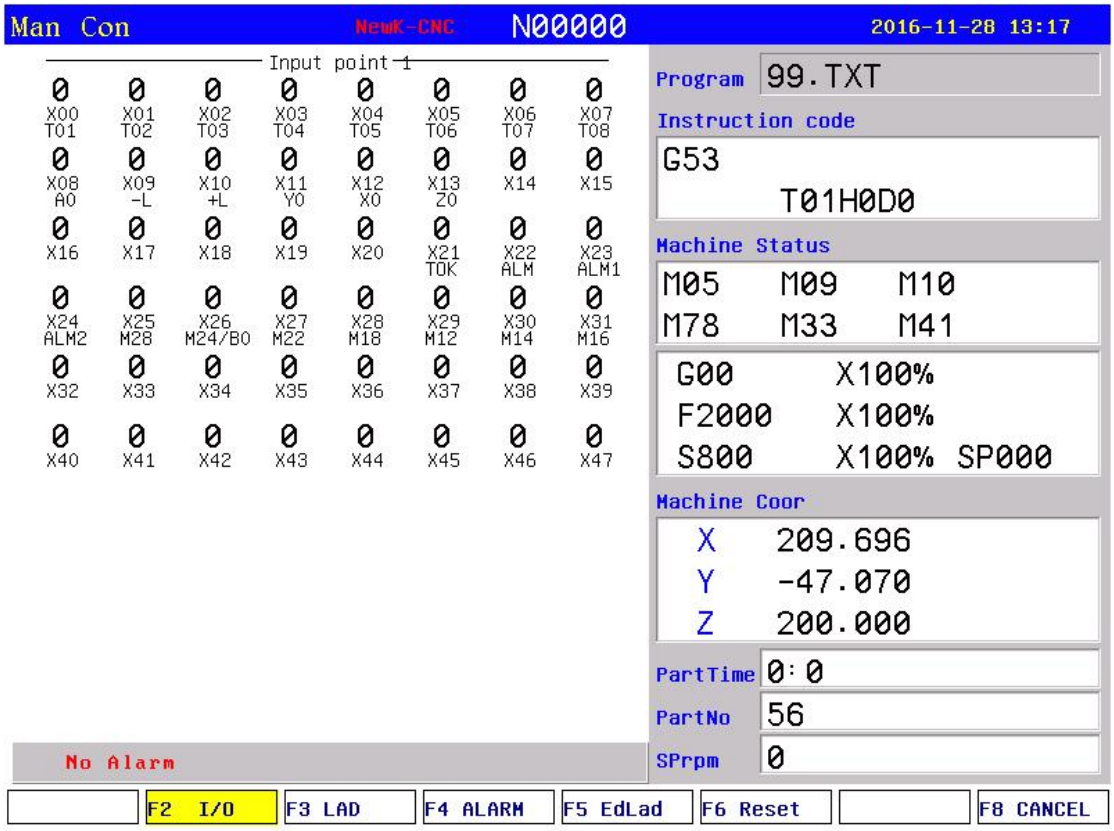


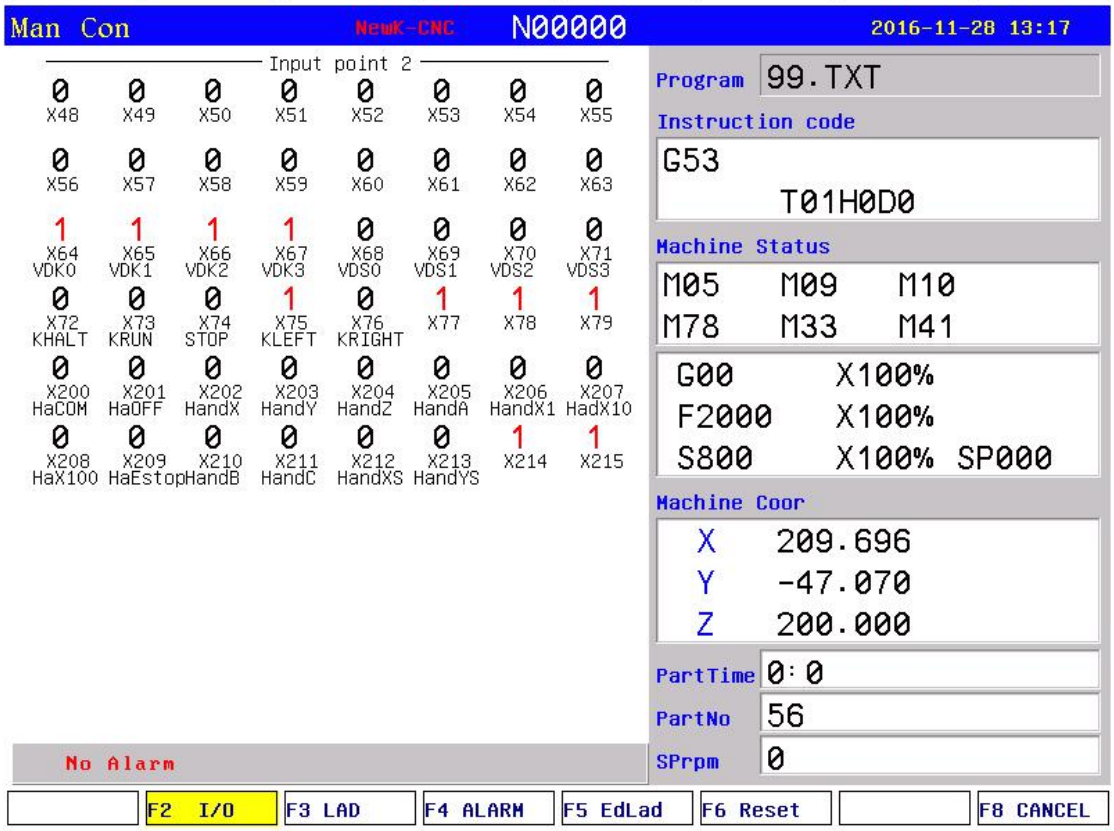
3.10 Diagnosis

Press “Diagnosis” to enter the diagnosis interface in parameter interface.

System diagnosis interface(Input signal)

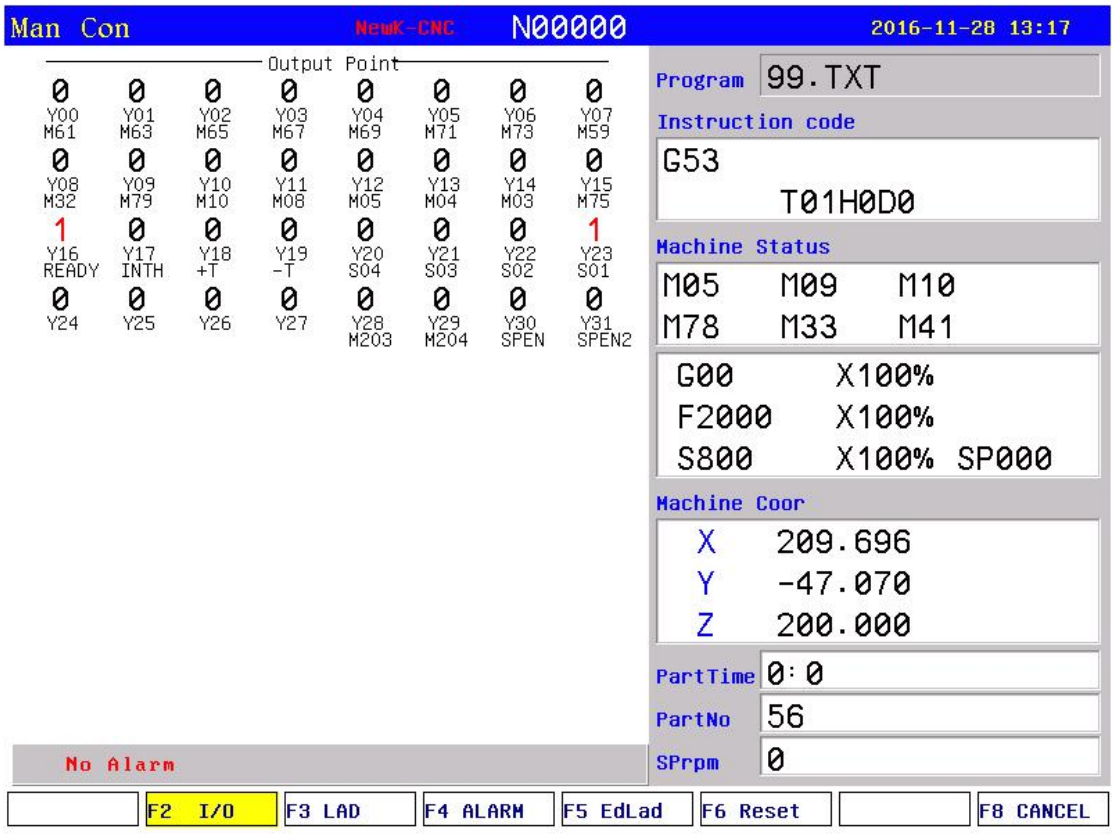
Press “F2 I/O” and “Pgup Pgdn” to check the status of input and output, press “F4” to check alarm information.


Man Con
NewK-CNC
N00000
2016-11-28 13:17
Input point 1
X00 T01
X01 T02
X02 T03
X03 T04
X04 T05
X05 T06
X06 T07
X07 T08
X08 A0
X09 -L
X10 +L
X11 Y0
X12 X0
X13 Z0
X14
X15
X16
X17
X18
X19
X20
X21 TOK
X22 ALM
X23 ALM1
X24 ALM2
X25 M28
X26 M24/B0
X27 M22
X28 M18
X29 M12
X30 M14
X31 M16
X32
X33
X34
X35
X36
X37
X38
X39
X40
X41
X42
X43
X44
X45
X46
X47
Program
99.TXT
Instruction code
G53
T01H0D0
Machine Status
M05 M09 M10
M78 M33 M41
G00 X100%
F2000 X100%
S800 X100% SP000
Machine Coor
X 209.696
Y -47.070
Z 200.000
PartTime
0:0
PartNo
56
SPrpm
0
No Alarm
F2 I/O
F3 LAD
F4 ALARM
F5 EdLad
F6 Reset
F8 CANCEL


Man Con
NewK-CNC
N00000
2016-11-28 13:17
Input point 2
X48
X49
X50
X51
X52
X53
X54
X55
X56
X57
X58
X59
X60
X61
X62
X63
X64 VDK0
X65 VDK1
X66 VDK2
X67 VDK3
X68 VDS0
X69 VDS1
X70 VDS2
X71 VDS3
X72 KHALT
X73 KRUN
X74 STOP
X75 KLEFT
X76 KRIGHT
X77
X78
X79
X200 HaCOM
X201 HaOFF
X202 HandX
X203 HandY
X204 HandZ
X205 HandA
X206 HandX1
X207 HadX10
X208 HaX100
X209 HaEstop
X210 HandB
X211 HandC
X212 HandXS
X213 HandYS
X214
X215
Program
99.TXT
Instruction code
G53
T01H0D0
Machine Status
M05 M09 M10
M78 M33 M41
G00 X100%
F2000 X100%
S800 X100% SP000
Machine Coor
X 209.696
Y -47.070
Z 200.000
PartTime
0:0
PartNo
56
SPrpm
0
No Alarm
F2 I/O
F3 LAD
F4 ALARM
F5 EdLad
F6 Reset
F8 CANCEL

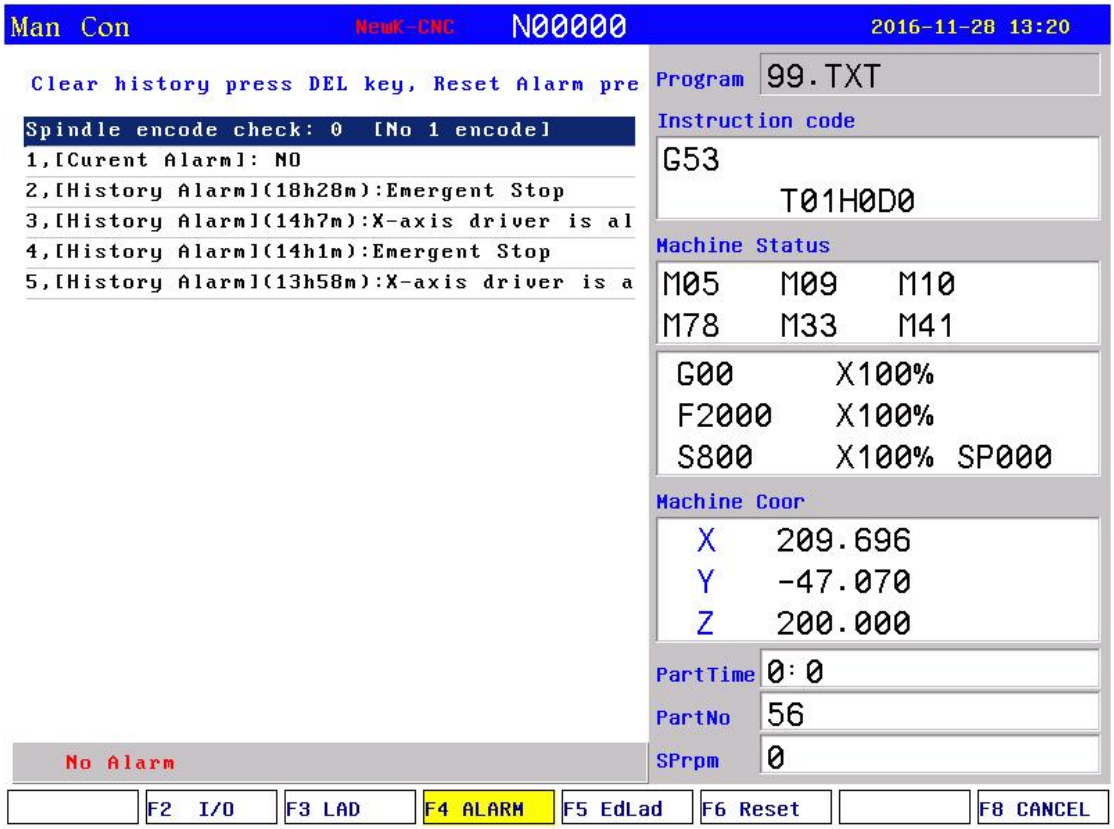
Checking interface of output signal

In the interface of output or input, No.0 or No.1 stands for status, 1 means effective, 0 means no effect.



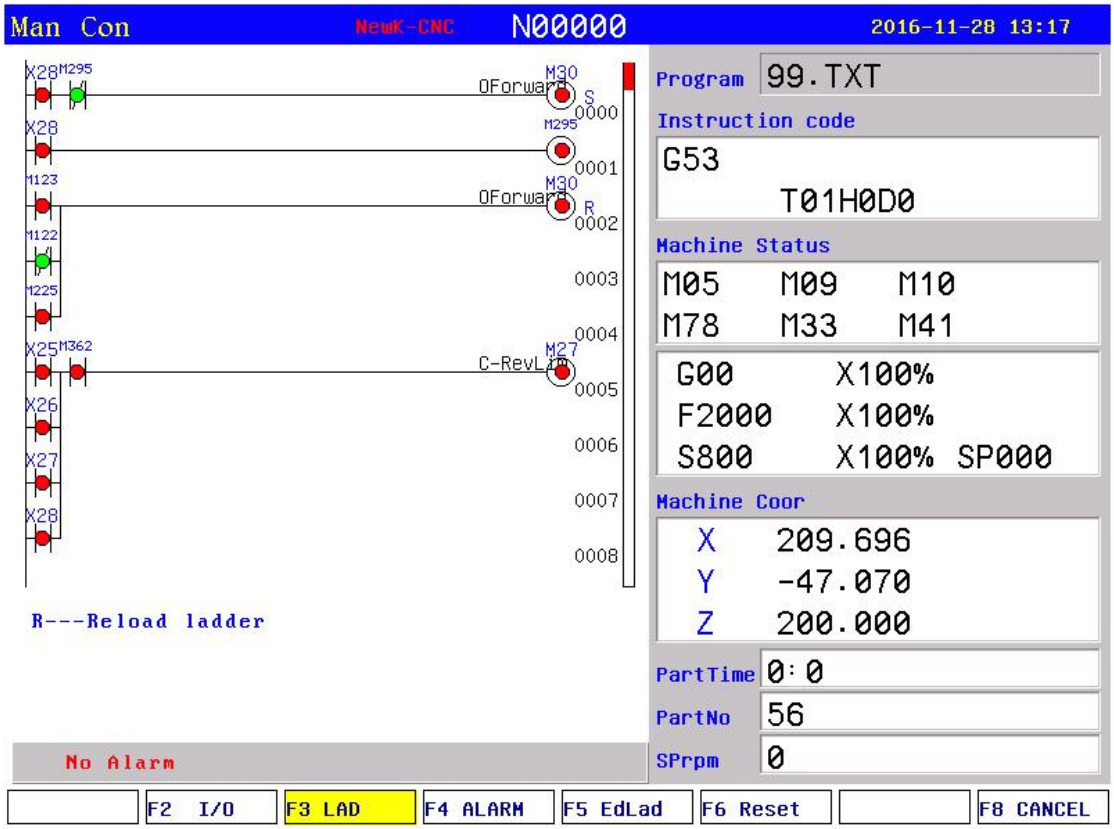
Alarm information interface

The first line in this interface shows the number of spindle encoder, the number of current and historical alarm information is record total 10, the superfluous part is clear automatically, only shows 10 alarm information recently.



Check and edit PLC of system

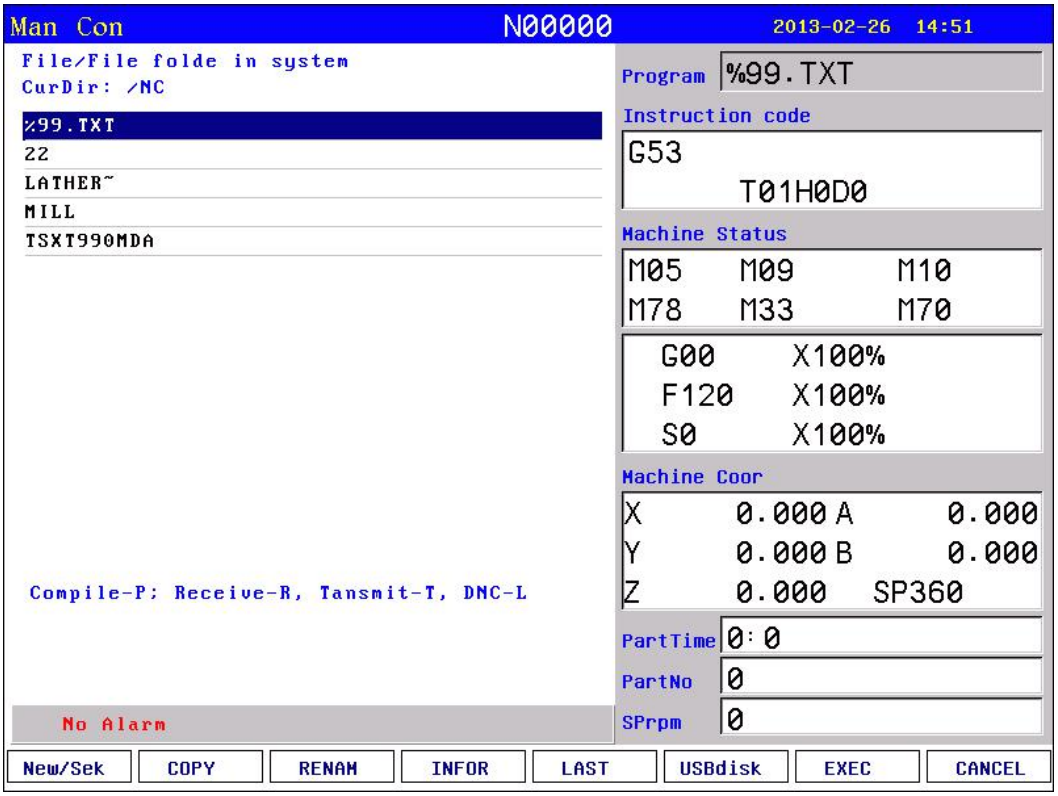
Press “F3” to check the ladder diagram of PLC, press “F5” to edit the PLC, press “S” to use function of search. Need restart the system after modifying the ladder diagram, also can press “R” to execute the new PLC without restart.



3.11 Operation of program

Press “Program” in any menu to enter into status of programming.

Program management is the same as file management, the storage of the system is 32M bits to contain program and there is no limit for quantity of program. Programming adopts full screen operation.



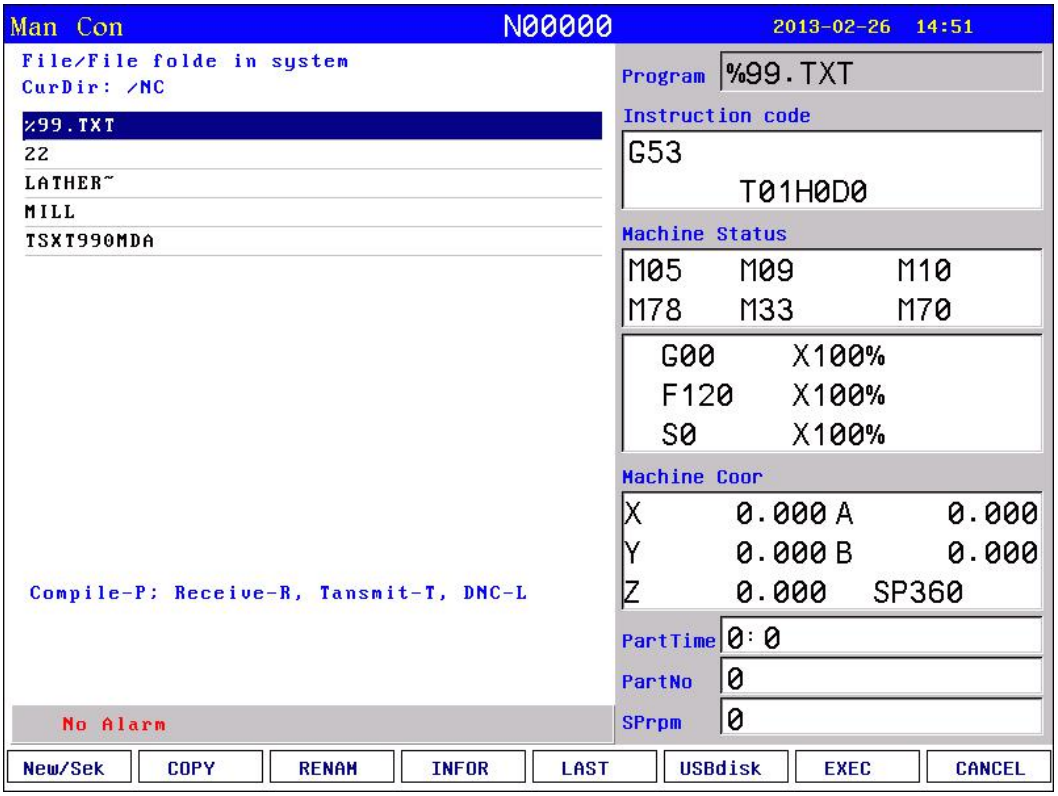
Center part of screen for program display， current program is showed by reverse display, press “PgUp”、 “PgDn” to choose program, and then press“Enter”to edit current program. Functional keys“F1、F2、F3、F4、F5、F6、F7、F8” include：“new file/search” 、“copy” 、“rename” 、“information”、“last grade”“USB disc”、“execute program”、 “cancel”.

3.11.1 Editing

Select “New file/search” to popup a dialog box to import the name of program, if the name is existent, the quondam program is called up; If the name is inexistent, the system will build a new file.

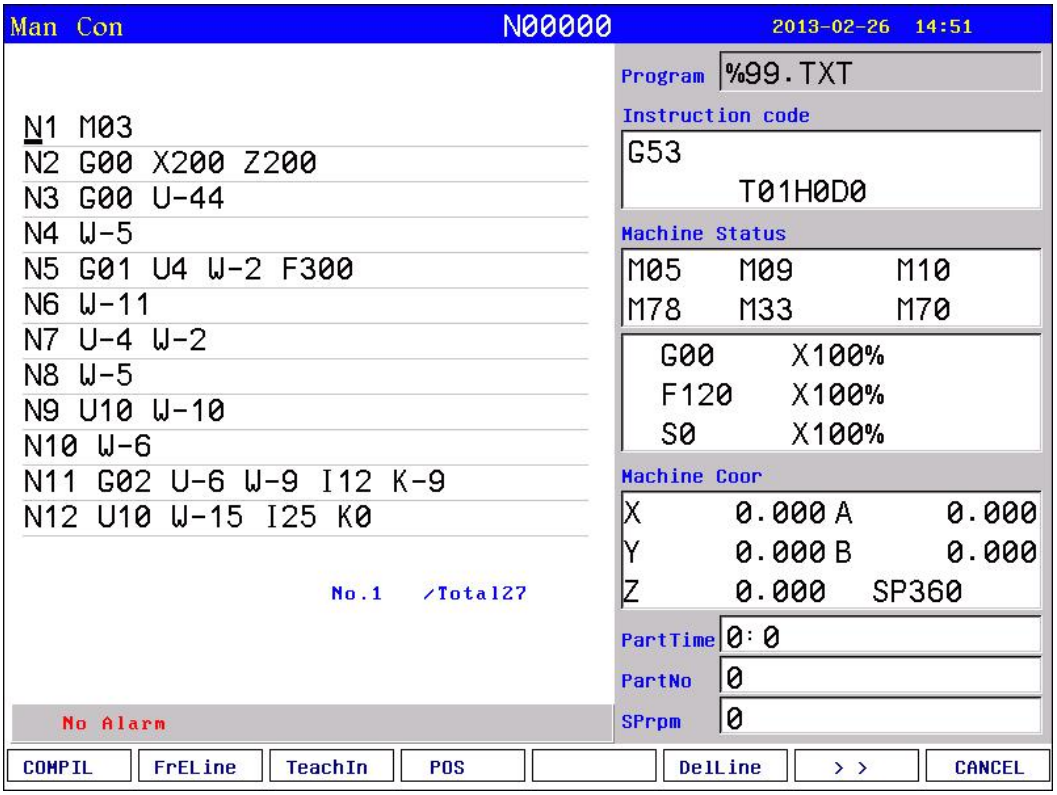
The name of program can be number, letter or mix, the length is 100 bits.

The system doesn’t allow the namesake, build a new program or select a program and press “Enter” to enter the editing interface.



The file name and the Chinese input of program content:

The directory interface of the "new / search", "copy", "rename", "to copy into the system", "to copy into the USB" can enter the Chinese characters; change the input method according to the prompt of system. Also can input Chinese characters in program editing interface according to the prompt of system.



The screen prompt the editing program name at the top left corner in the editing status; The left is the content, the right is the information for lathe status, the operation in the editing status as follows:

(1) The current cursor locate:

Press “↑ ↓ ←→” to move the cursor to any position of program content

Press "Pgup" to the last page.

Press "Pgdn" to the next page.

(2) Character modification: Delete the character at the position of the cursor, then enter the new character.

(3) The character insertion: Enter a new direct character at the cursor position. When the input is the letter,the letter in front of automatically generating space. If you want to enter a space, first enter a letter, and then delete this letter.

(4) The character deletion: Press "Del" directly at the cursor position

(5) Inset the line: Press "Enter" directly, inset a line in front of the current line if the cursor is at the first line, otherwise insert a line after the current line.

(6) "Fast" superposition key operation:

The first function:

A, "F1": compile the program.

B, "F2": to the fist line or last line of the program.

C, "F3": teaching function, enter the handwheel status;Press "F4" to read the tool coordinate in the current workpiece coordinate system according to the corresponding

axis X/Y/Z/A/B/C/ (all the axis).
D, "F4": located to the specified line.
E, "F5": no operation.
F: "F6": delete the current line.
G, "F7": the fist or second page selection.
H, "F8": Cancle.
The second function:
A, "F1": delete the program block.
B, "F2": copy the specified program block.
C, "F3": sort the program.
D, "F4": to find the specified character.
E, "F5": replacing the specified character.
F, "F6": all the content to are replaced by the specified character.
G, "F7": the first or second page selection.
H, "F8": Cancel

3.11.2 Copy

Press " ↑ ↓ " in program main interface to select program which need to copy and press "F2" to popup a dialog box to import a new name of program, to copy which is the same content but different name so that to modify, rename and back-up copy.

3.11.3 Delete

Press " ↑ ↓ " in program main interface to select program which need to delete and press "Del" to delete the program.

The operation of delete need to be careful, it can't be recovery after deleting.

3.11.4 Rename

Press " ↑ ↓ " in program main interface to select program which need to rename and press "F3" to popup a dialog box to import a new name.

3.11.5 Information

Press " ↑ ↓ " in program main interface to select program which need to check and press "F4" to popup a dialog box to check the size of program and the remainder space of the system.

3.11.6 Checking program

Press " ↑ ↓ " in program main interface to select the checking program and press "P", the system will check the form and grammar of program. Prompting when finding mistake.

3.11.7 Folder management

You can build a file in this system, Press "F1" in program main interface to import a file name and press "." to build a folder and it will prompt a "folder" after the name.

Move the cursor to the file name and press "Enter" to open to build a new file or folder in it.

Press “F5” go to the last folder.

Move the cursor to the file name and press “Del” to delete the folder.

3.11.8 Select automatical program to run

Press “ ↑ ↓ ” in program main interface to select a program and press “F7” to select the program and switch into the last interface.

3.11.9 Program communication

The system could adopt the RS232 serial port to deliver files.

Delivery (Transport)

Deliver the selected program in this system to another system or to PC to save. Press “ ↑ ↓ ” in program main interface to select program and press “T” to deliver, press “Reset” to interrupt in the deliver process.

Reception

Receive the selected program in another system or PC (Must be text file form). Press “R” to import a name of received program into the dialog box in program main interface, press “Reset” to interrupt in the receive process.

Pay attention: 1. Using the exclusive communication software to deliver program in User’s PC.

2. The speed of deliver of PC must be the same as the speed of receive, defeat otherwise.

3. The length of RS232 can’t greater than 10 meters.

4. The number of serial port must be the same as the system setting.

5. Editing program of PC must be text file form.

3.11.10 U-disk management

To exchange files of parameter or program with other system or PC by U-disk. It also can upgrade or back-up the software or parameter in system.

Pay attention: The name of folder can’t have space symbols.

Press “F6” to enter the U-disk management interface in program main interface when U-disk connects the USB port. Press “F6” again to back to the system interface.

A. The processing program management

Copy the files or folder of U-disk into system

After connecting the U-disk, press “F6” to enter the U-disk directory in program main interface. Press “ ↑ ↓ ” to move cursor to select file or folder to copy and press “F4” to popup a dialog box to import name, press “Enter” to confirm. If there is the same name of program in the system, it will popup a dialog box to ask if cover the file or folder or not.

Press “R” to copy all the program in USB into system.

Copy the files or folder of system into U-disk

Press “ ↑ ↓ ” to move cursor to select file or folder and press “F6”, press “F3” to popup a dialog box to import name in U-disk interface and press “Enter” to confirm.

If there is the same name of program in the system, it will popup a dialog box to ask if cover the file or folder or not.

Press “T” to copy all the program in system into USB.

Pay attention: Before unplugging the U-disk must return to the display system of program files directory interface. (Exit U-disk interface)

Otherwise the date which is copied just now will be loss.

The name of folder can’t have space symbol when using U-disk.

B. Using U-disk to manage parameter and system software

The system could use U-disk to deliver files or system software to upgrade and update, back-up files and parameter, the method of operation is as follows:

Using U-disk to copy parameter and system software into system(Upgrade, update).

First U-disk inserts the USB port and press “Program” to enter program main interface, press “F6” to show the files in U-disk. Press “ ↑ ↓ ” to move the cursor to select a folder which is going to be copied into system and press “ Enter” to open it, press “F2” to import code when appearing the files and press “Enter” to confirm, wait for seconds to copied the parameter successfully. Press “F6” to exit U-disk after copying successfully, restart the system, the system will reloads the new files to upgrade the parameter.

Pay attention: The parameter is better to be derived into a separate folder in U-disk to defend from the error operation to destroy the system files.

To derive or back-up parameter files by U-disk

First U-disk inserts the USB port and press “Program” to enter program main interface, press “F6” to show the files in U-disk. Press “F1” to import the code and press “Enter” to confirm, waif for seconds to derive successfully. The parameter in system is already derived into U-disk. Press “F6” to exit U-disk.

Pay attention: The U-disk is empty better to arrange the files (Parameter files is lots of about several dozens) so that derive parameter or create a folder on your computer first, open the folder before deriving to derive the parameters into the folder.

Chapter4 System installation and connection

4.1 System electrical specification

- 32bits high performance industrial grade ARM+DSP+FPGA
- 128M User's storage space(Could expanded to 32G in bus system)
- 800x600 TFT LCD adaptive brightness, LED backlight LCD
- Touch type key board to have excellent operational sensitivity.
- RS232 communication port
- USB port
- Highly anti-interference of switching power supply.
- Two-way spindle to variable frequency and speed governing
- Manual pulse generator
- The band switch trim the feeding speed and spindle speed

4.2 System technical index

- Controllable axis: X Y Z A B C Xs Ys eight axis
- Linkage axis: straight line X Y Z A B C Xs Ys eight axis, arc X Y Z any two axis
- Pulse equivalency: 0.001mm
- Maximum speed: 240m/min
- Processing speed: 0.01-30000min/min
- Minimal input unit: 0.001mm
- Programme size range: ±99999.999mm
- Programme coordinate system definition: IOS-841 international standard
- Programme code: IOS-840 international standard
- Mean Time Between Failure (MTBF): Greater than 6000 hours

4.3 Environment of operation

- Power supply: AC 220V (+10%,-15%), frequency 50Hz ±1%

- Power source≤100W
- Running temperature: 5~45℃, relative humidity: 40~80%
- Storage and transportation temperature: 0~55℃, relative humidity less than 90%(40℃)

4.4 System installation and connection

At first, users should check whether the hardware is complete, unwounded and compatible, such as: CNC system, driving power, servo motor, photoelectric encoder, electric tool carrier.

The installation of CNC system must be fastened tightly, with some spaces around to ensure the ventilation of air. Panel should be put in a place where it is not only convenient to operate and but also able to avoid hurt of heating by scrap iron.

Intense current, week current must be put separately; CNC system and driver should be possibly away from the machine intense current. In order to reduce interference, all signal cables should be kept away from AC contactor. Photoelectric encoder, limit, basic point signal are advisably not to be connected directly to CNC system through intense current box. All power cords must be grounding.

Fix all plugs with screw. Forbid to insert and extract all cables when power is on.

In installation of CNC system, panel should avoid hurting by hard and sharp materials. If the painting of other part of machine is needed, please take off CNC system to keep it clean.

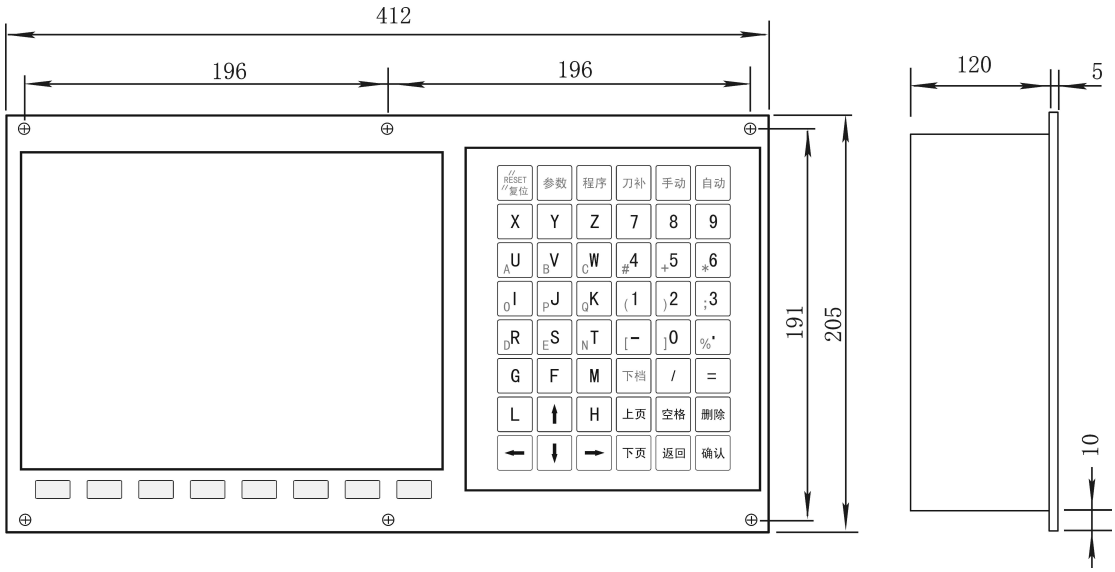
To ensure there is no strong magnet and current interference, keep away from inflammable, explosive and other danger materials.

Pay attention:

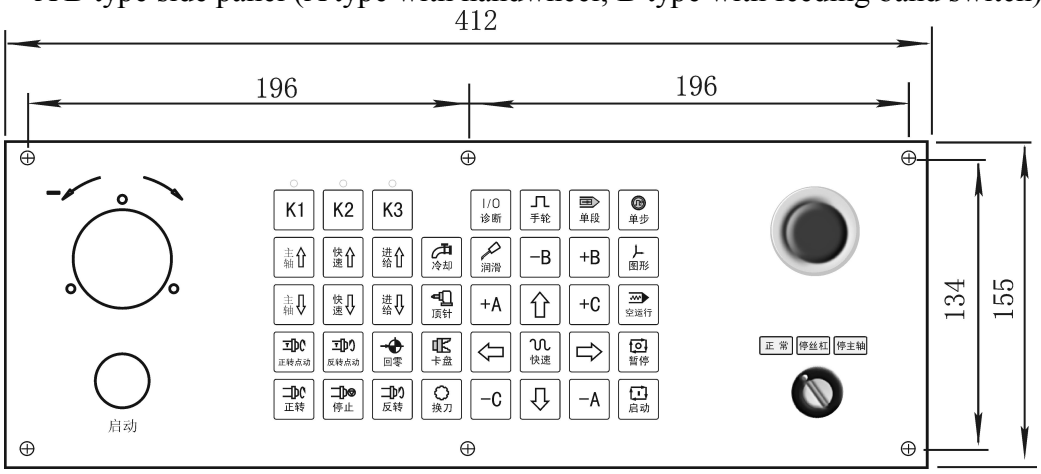
1. **Must install in an electricity cabinet which is good for protect from lightning.**
2. **Must install firmly to in order to avoid vibrating and loosing.**
3. **Don’t install on the inflammable things or nearby to keep away from fire.**

4.5 System installation dimension

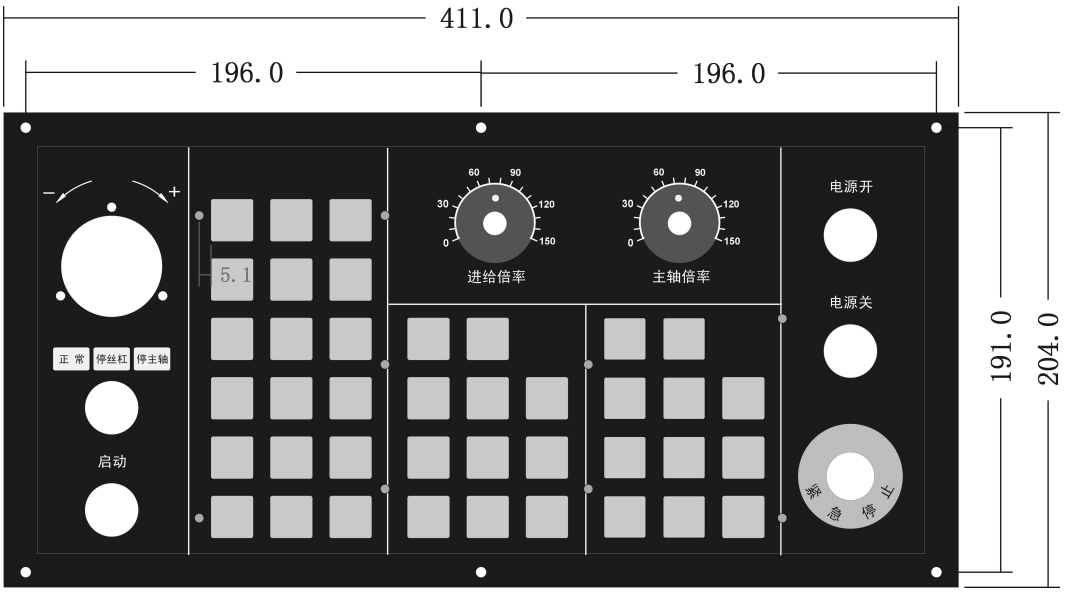
Main Panel:



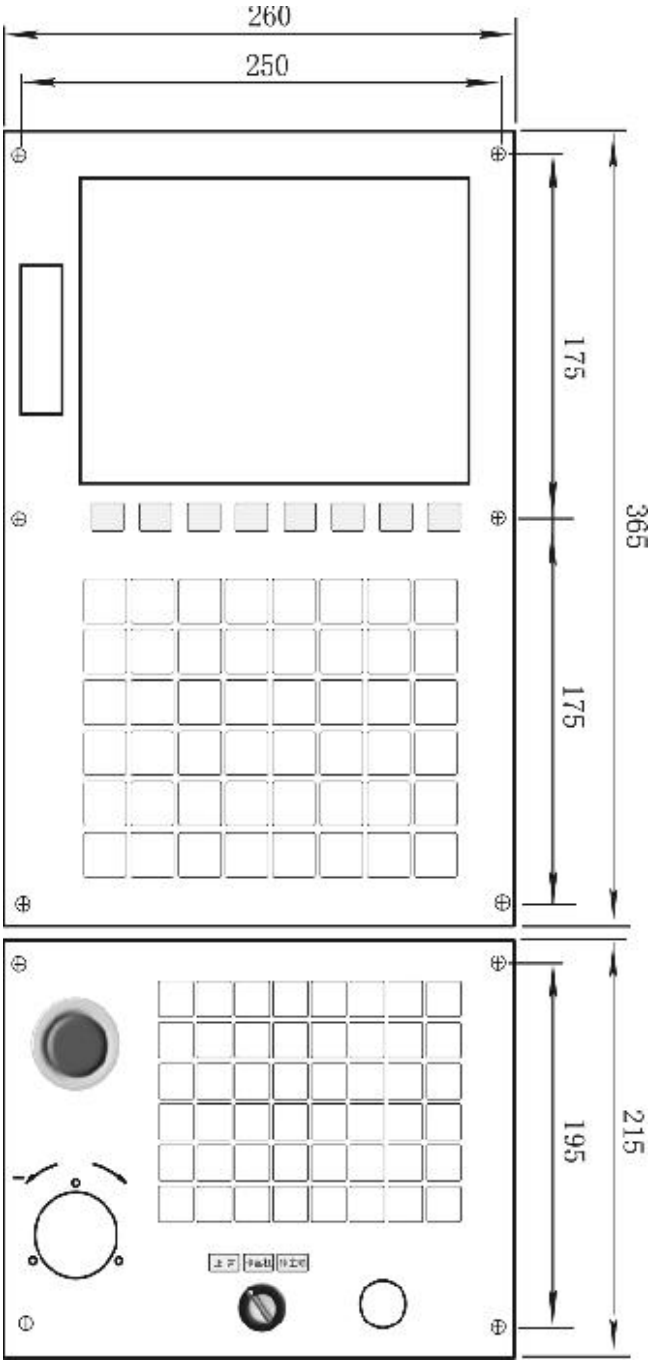
A B type side panel (A type with handwheel; B type with feeding band switch):



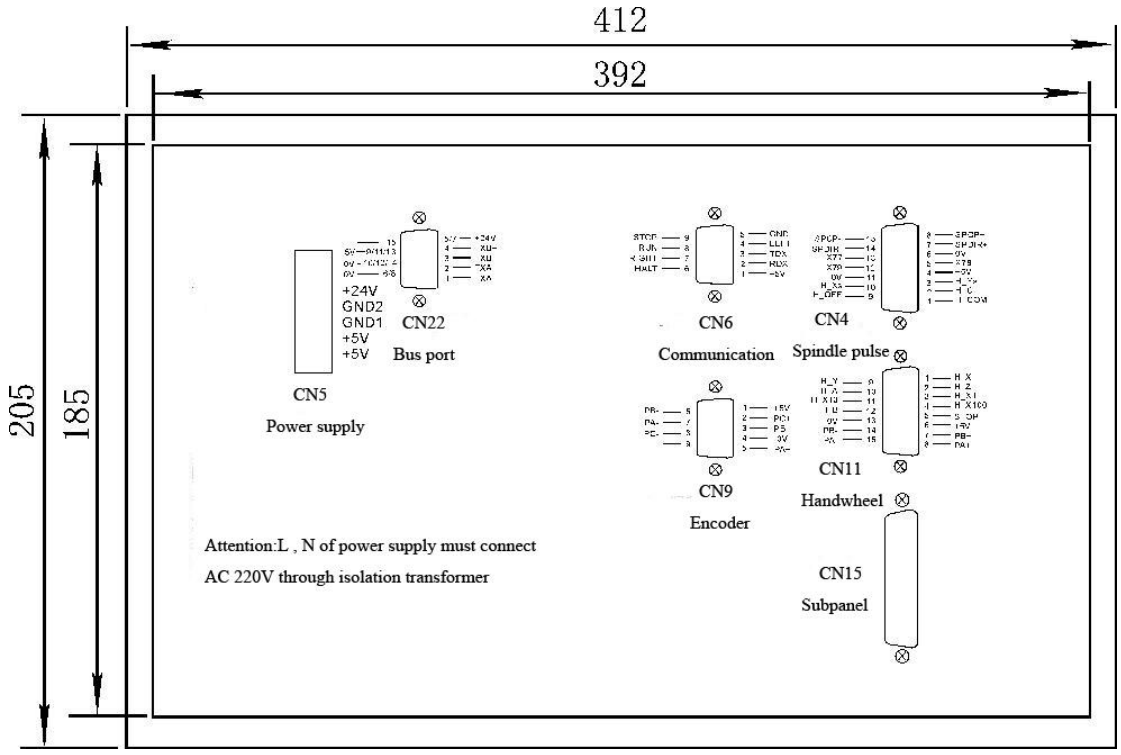
C type side panel:



Vertical mounting dimension:


260
250
175
175
365
195
215

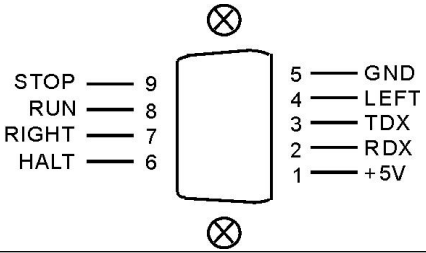
4.6 System rear view



Pay attention:Switching power supply L N must through isolation transformer and insert to AC 220V, current 0.5A.

4.7 Interface connection graph

4.7.1 Communication socket CN6 connect to the hole socket DB9



CN6 communication signal with the hole socket DB9				
Signal	Pin number	I/O	Function	Effective voltage
0V	5	OUT	The ground of signal	0V
RXD	2	IN	The received data signal	

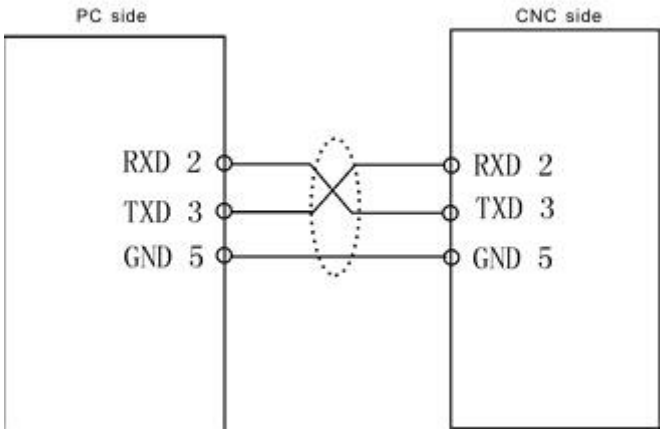
TXD	3	OUT	The transmission of data signal	
+5V	1	OUT	Power supply	+5V
LEFT	4	IN	Left of intervention switch	0V
HALT	6	IN	Suspend	0V
RIGHT	7	IN	Right of intervention switch	0V
RUN	8	IN	Start	0V
STOP	9	IN	Emergency stop	0V

Pay attention:

1. Connect to external PC with data communication, must be equipped with our special communication software.

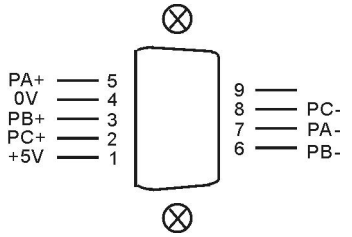
2. Communication line must adopt the shielded twisted pair cable, the length shall not exceed 10m.

The signal of CN6 connect to PC:



Pay attention: When PC programming, the files should be text files.

4.7.2 Spindle encoder socket CN9 connect to needle socket DB9



CN9 DB9(pin) spindle encoder				
signal	pin	I/O	function	availability
0V	4	OUT	0V	0V
+5V	1	OUT	+5V	+5V

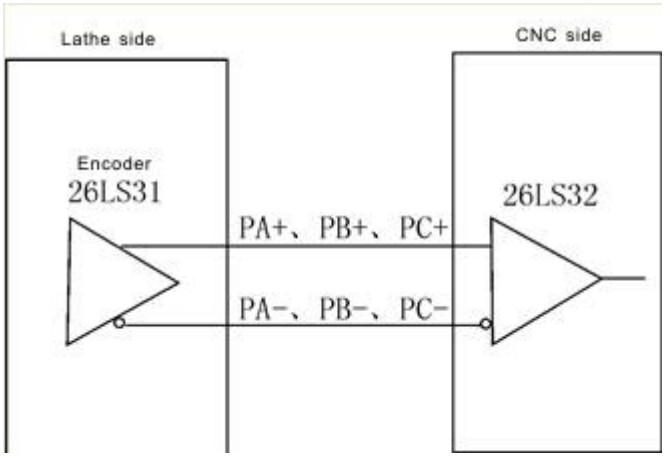
PA+	5	IN	+A signal	5V
PA-	7	IN	-A signal	
PB+	3	IN	+B signal	5V
PB-	6	IN	-B signal	
PC+	2	IN	+Z signal	5V
PC-	8	IN	-Z signal	

Pay attention:

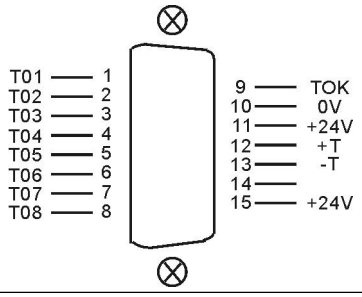
1. The output signal of encoder adopt the output way is line output, the power supply is +5V.

2. The signal line must adopt shielded twisted pair cable, the length is 20m at most.

The input signal of encoder PA PB PC:



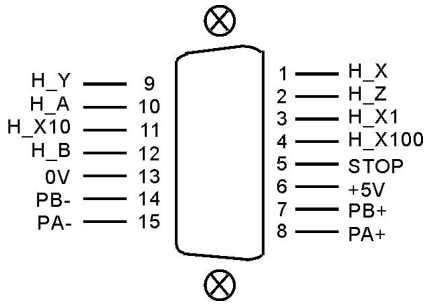
CN4 connect hole socket DB15



CN4 DB15 I/O4 signal				
Signal	Pin	I/O	Function	Effective voltage
0V	10	OUT	Ground of power supply	0V
+24V	11、15	OUT	24V power supply	+24V
Y18	12	OUT	+T Reserve	0V

			output	
Y19	13	OUT	-T Reserve output	0V
X00	1	IN	T1 Reserve input	0V
X01	2	IN	T2 Reserve input	0V
X02	3	IN	T3 Reserve input	0V
X03	4	IN	T4 Reserve input	0V
X04	5	IN	T5 Reserve input	0V
X05	6	IN	T6 Reserve input	0V
X06	7	IN	T7 Reserve input	0V
X07	8	IN	T8 Reserve input	0V
X21	9	IN	TOK Reserve input	0V

4.7.3 External switch, electrical handwheel socket CN11 connect to DB15 needle socket



CN11 DB15 I/O signal				
Signal	Pin	I/O	Function	Effective voltage
0V	13	OUT	Ground of power supply	0V
+5V	6	OUT	5V power supply	+5V
PA+	8	IN	Positive signal A	5V
PA-	15	IN	Negative signal A	
PB+	7	IN	Positive signal B	5V
PB-	14	IN	Negative signal B	
STOP	5	IN	External emergency signal	0V
H_B	12	IN	Turn off gear signal,selection signal of B axis	0V
H_X100	4	IN	One hundred gears signal	0V
H_X10	11	IN	Ten gears signal	0V
H_X1	3	IN	One gear signal	0V
H_A/HALT	10	IN	Selection signal of A axis/Stop	0V

			signal	
H_Z	2	IN	Selection signal of Z axis	0V
H_Y/RUN	9	IN	Selection signal of Y axis/Running signal	0V
H_X	1	IN	Selection signal of X axis	0V

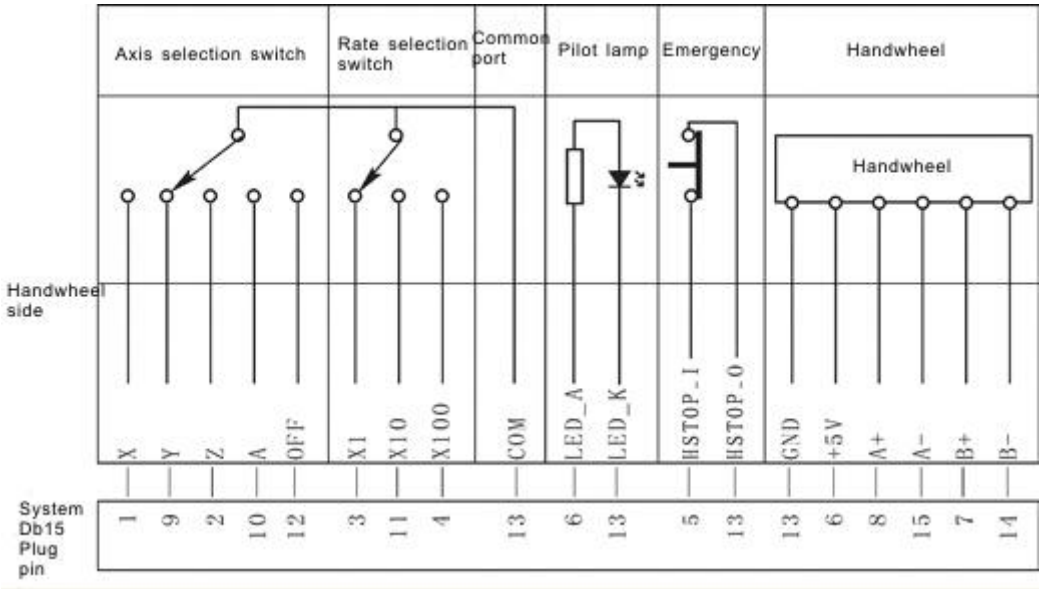
Pay attention: The inner power supply are all +5V of all signal, do not access voltage higher than 5V.

4.7.3.1 Usage for electrical handwheel(Manual pulse generator)

You can connect standard external handwheel when No.1 parameter in other parameter is 1 which could connect the standard suspended handwheel, A, X , Y, Z , X1, X10, X100 corresponding to select feeding axis and gear

PA+ PB- PA+ PA- corresponding input signal of handwheel pulse A B.

Handwheel contact diagrammatic as:



Pay attention:

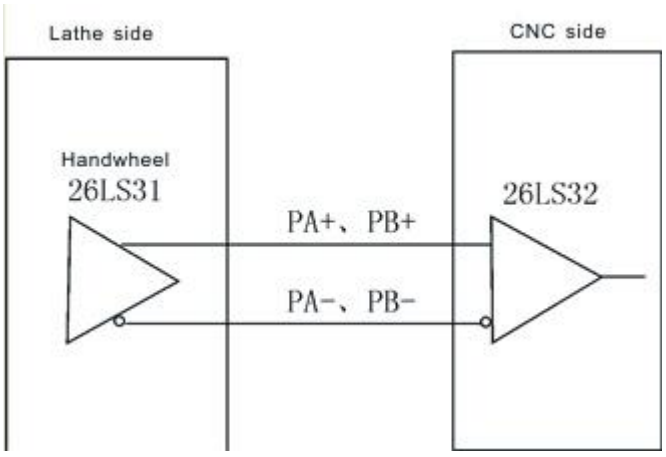
1. The output signal of handwheel adopts line output, the power supply is +5V.

2. Just connect PA+ PB+ if adopt voltage output.

3. Manual pulse generator needn't "Enter" button, if there is a "Enter" button, use the line to short the ends of switch.

4.The inner power supply are all +5V of all signal, do not access voltage higher than 5V.

The input signal of handwheel:



4.7.3.2 Usage for terminal band switch behind sub panel

Signal	CN15 Pin	I/O	Function	Effective voltage
0V	29、39	OUT	Ground of power supply	0V
+5V	11、26	OUT	5V power supply	+5V
PA+	41	IN	Positive signal A	5V
PA-	12	IN	Negative signal A	
PB+	27	IN	Positive signal B	5V
PB-	42	IN	Negative signal B	
STOP	44	IN	External emergency signal	0V
KRUN	14	IN	Running button	0V
KLEFT	15	IN	Left signal of intervention switch	0V
KRIGTH	30	IN	Right signal of intervention switch	0V
DS0	8	IN	1 spindle band	0V
DS1	38	IN	2 spindle band	0V
DS2	23	IN	3 spindle band	0V
DS3	37	IN	4 spindle band	0V
DK0	7	IN	1 band feeding axis	0V
DK1	22	IN	1 band feeding axis	0V
DK2	9	IN	1 band feeding axis	0V
DK3	24	IN	1 band feeding axis	0V
LRUN	43	OUT	Input signal of running program	0V
BDIR+	40	OUT	Positive direction	

			signal of B axis	
BCP+	36	OUT	Positive pulse signal of B axis	
BDIR-	28	OUT	Negative direction signal of B axis	
BCP-	10	OUT	Negative pulse signal of B axis	

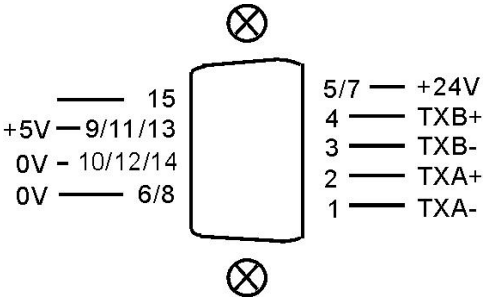
When the No.1 and No.2 parameter in "axis parameter" are set to "1", you can use the band switch. DS0, DS1, DS2, DS3 are for the spindle speed to trim input signal of gear, total 16 gears in four position controlling. DK0, DK1, DK2, DK3 are for the feeding speed to trim input signal of gear, total 16 gears in four position controlling.

Note: All the internal signal power is +5V, do not access more than 5V.

4.7.3.3 Using for external emergency button

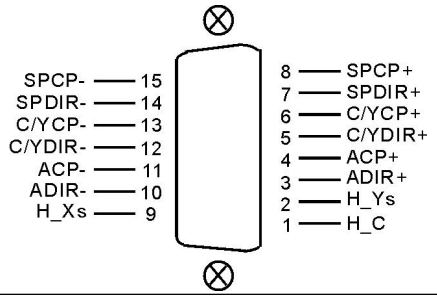
STOP signal is the input signal of external emergency button, No.27 parameter in other parameter controls the emergency button is always open or close.

4.7.4 Bus port CN22 connect DB15



CN22 Bus port				
Signal	Pin	I/O	Function	Effective voltage
TXA-	1		Negative A signal	
TXA+	2		Positive A signal	
TXB-	3		Negative B signal	
TXB+	4		Positive B signal	
+24V	5/7	OUT	24V Power supply	
0V	6/8/10/12/14	OUT	Ground of power supply	
+5V	9/11/13	OUT	5V Power supply	

4.7.5 Spindle pulse signal CN4 connect DB15



CN4 Spindle pulse signal				
Signal	Pin	I/O	Function	Effective voltage
H_C	1	IN	C axis signal of handwheel	0V
H_Ys	2	IN	Ys axis signal of handwheel	0V
ADIR+	3	OUT	Positive direction signal of A axis	
ACP+	4	OUT	Positive pulse signal of A axis	
C/YDIR+	5	OUT	Positive direction signal of Y axis	
C/YCP+	6	OUT	Positive pulse signal of Y axis	
SPDIR+	7	OUT	Positive direction signal of spindle	
SPCP+	8	OUT	Positive pulse signal of spindle	
H_Xs	9	IN	Xs axis signal of handwheel	0V
ADIR-	10	OUT	Negative direction signal of A axis	
ACP-	11	OUT	Negative pulse signal of A axis	
C/YDIR-	10	OUT	Negative direction signal of Y axis	
C/YCP-	11	OUT	Negative pulse signal of Y axis	
SPDIR-	14	OUT	Negative direction signal of Spindle	
SPCP-	15	OUT	Negative pulse	

			signal of Spindle	

4.7.6 Trimming method for the system matches the absolute bus type of motor

1. Turn on the power supply.
2. Set the axis parameter in system to trim motor mode of XYZAB axis.
3. Set the axis parameter in system to trim the electrical gear.

P11=00000001 soft limit effectively;

Lathe setting P39=11111011; Milling setting P39=11111011; Returning to zero with floating mode;

4. Set the other parameter in system

P300=11111100; the absolute value function;

P301=92; P302=91; P303=90;

P304/P305/P306/P307/P308=131072;

P309/P310/P311/P312/P313 set the coordinate values, such as when the screw pitch with direct of Z axis is 6mm,

P311=6000000;

Set P320/P324 when machine with reduction ratio, such as reduction ratio of Z axis is 3:2, thread is 6mm, electronic gear molecule is 5*3=15 (5 without reduction ratio), electronic gear denominator is 3*2=6(3 without reduction ratio) , so:

P322=electronic gear denominator=15, P311=10000000*electronic gear denominator=60000000

5. Set the coordinate axis which corresponds the No.56 parameter in driver parameter

P1=1, enter the password;

P56=1/2/3/4/5/6/7/8 corresponds to the X/Y/Z/A/B/C/Xs/Ys, such as the Z axis drive is set to 3;

Press "Enter" for a while to save parameter in EP-status;

6. MDI to run the M500 instruction, the small circle in front of the coordinate axis should be changed into "green", means read the absolute encoder data correctly;
7. Set the coordinate value and direction of lathe:

1) manual move coordinate and remember the lathe coordinate value;

2) MDI to run M500 instruction, reading whether the absolute value coordinate is the same as previous coordinate value or not, modifying the symbol of No.309 No.310 No.311 No.312 No.313 parameter in other parameter, such as the X coordinate is not right, the original is P309=4000000, to modify to P309=-4000000;

3) Repeat 1) 2) test for two times;

8. Setting the zero point of lathe: manually move to the position which is nearby the zero point , set the No.314 No.315 No.316 No.317 No.318 parameter in other parameter(Corresponds to X/Y/Z/A/B axis), press "Enter" and press "E" to clear the lathe coordinate;

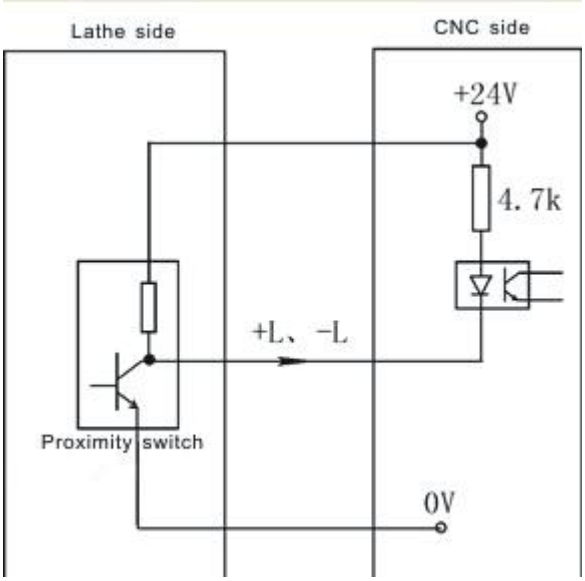
9. Manual move every axis to the limit position of lathe to set the soft limit value in axis parameter;

10. Set the No.41 parameter in other parameter and back up the current system parameter

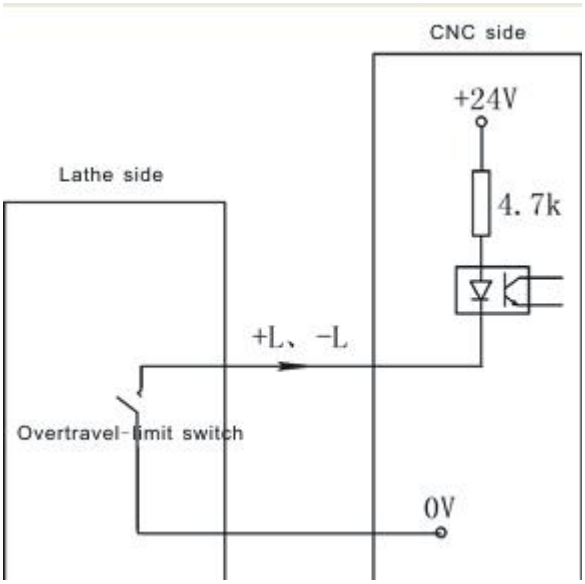
4.8 The machine installation of lathe.

4.8.1 Limited position: take X Y Z A B axis is limited position positively as example

Model 1: NPN approach switch



Mode 2: General switch



Axis parameter:

No.37 parameter is for setting the type of hard limit switch for +L positive, 0 means always open, 1 means always close.

No.38 parameter is for setting the type of hard limit switch for -L negative, 0 means always open, 1 means always close.

Pay attention:

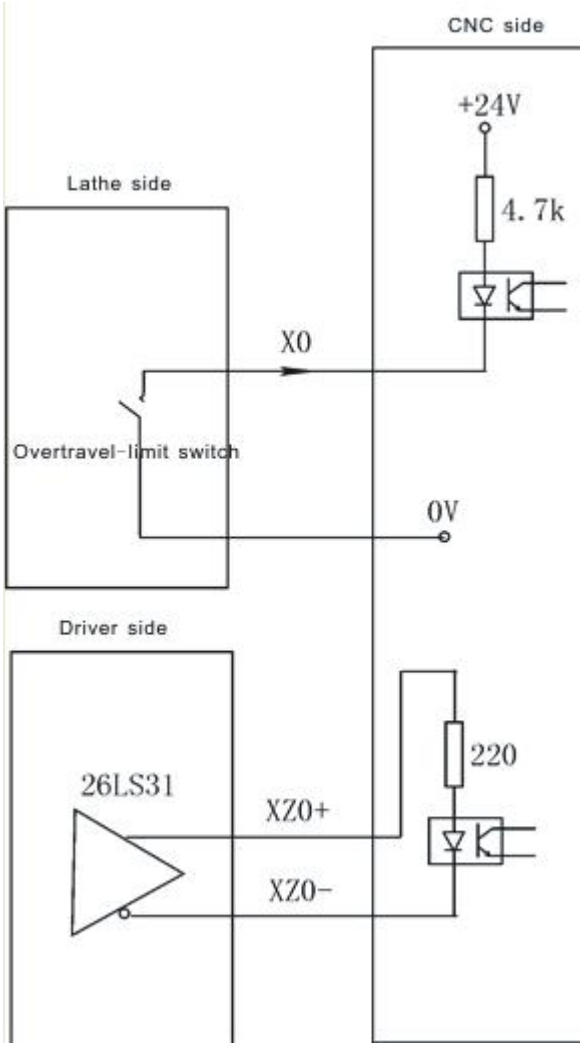
1. X Y Z A B axis limited shares a signal to always open or close together, positive limited and negative limited corresponding stand for +L and –L signal.

2. Could select our electrical appliance plate of lathe.

3. The system could define X0 Y0 Z0 A0 B0 to be limited input signal of X Y Z A B axis. X0 signal as the limited signal and datum point of X axis, the same switch to control.Y0 signal as the limited signal and datum point of Y axis, the same switch to control. Z0 signal as the limited signal and datum point of Z axis, the same switch to control. A0 signal as the limited signal and datum point of A axis, the same switch to control. B0 signal as the limited signal and datum point of B axis, the same switch to control.

The function must copy our exclusive PLC software.

4.8.2 Lathe's datum point(reference point or 0 point): take X axis as example (the same as Y Z A B axis)



Backing to the datum point

At the function of setting floating datum point is invalid conditions, backing to the datum point need to check approach switch signal and motor Z pulse signal. No.39 parameter in axis parameter is set to be “00000000”.

No.45 parameter in axis parameter set the function of backing to the datum points:

There are four ways for system to set backing to the datum point when turn on the system:

Needn’t when it is 1: system doesn’t prompt and no limits when turn on the system every time.

Prompting way when it is 0: system popup a dialog box to prompt user to process operation of backing to the datum point, it has no limits.

Forcing mode when it is 8: system popup a dialog box to prompt user to process operation of backing to the datum point and not to process before running automatically, the system will import “Feeding axis doesn’t back to the datum point”

and not to process program.

Super forcing mode when it is 9: moving the feeding axis when turn on system every time, the system will popup a dialog box to process operation of backing to the datum point, it will prompt “Feeding axis doesn’t back to the datum point” and not to process action if not to process operation of backing to the datum point.

No.46 parameter in axis parameter set the checking signal mode of backing to the datum point:

When it is 0: After hitting the datum point switch when backing to the datum point, run reverse to check the switch off and check 0 pulse signal of motor encoder.

When it is 1: After hitting the datum point switch when backing to the datum point, run reverse to check the switch off.

When it is 2: After hitting the datum point switch when backing to the datum point, continue running to check the switch off and check 0 pulse signal of motor encoder.

When it is residual value: After hitting the datum point switch when backing to the datum point, continue running to check the switch off.

The mode of backing the datum point should according to the circuit situation of equipment, at common conditions, it suggests to set to be 0 or 2, because if not to check 0 pulse signal of motor encoder, the accuracy can’t be promise.

No.47 parameter in axis parameter set the direction and sequence of backing to the datum point:

Every axis is to set separately. The parameter is positional parameter,D2 controls the processing direction of X axis, D4 is for direction of Z axis, 1 means negative, 0 means positive; D8 controls the sequence of X and Z, 1 means Z first, 0 means X first.

No.48 parameter in axis parameter set the type of datum point switch:

Every axis is to set separately. The parameter is positional parameter, D0 position controls X axis, D2 position controls Z axis, 1 means always close, 0 means always open.

No.49~No.53 parameter in axis parameter is set to check the processing length of motor Z pulse when backing to the datum point:

To set the scope of checking the motor encoder zero pulse signal after switch off when X(No.49) Y(No.50) Z(No.51) A(No.52) axis backing to the datum point. Unit: 0.1mm.

Pay attention: The parameter value must less than the distance of motor turns a round, otherwise could cause the wrong datum point situation.

No.38-No.43 parameter in speed parameter set the speed of reaching to zero point switch when backing to datum point (zero point):

The processing speed of reaching to datum point switch when X(No.38) Y(No.40) Z(No.42) A(No.44) axis backing to positive datum point. Unit: mm/min. Numerical range: Less than the G00 speed of X axis.

No.39-No.43 parameter in speed parameter is set to check the processing

speed of motor Z pulse when backing to the datum point:

When the X (P39) Y (P41) Z (P43) A (P45) axis backing to the datum point, the speed of checking the motor Z pulse signal after disengaging the switch for datum point. Numerical range: 20-500.

Pay attention: The parameter value influence with the accuracy of backing to the datum point, the smaller the value the higher the accuracy. This value has been set which not to change anymore, otherwise it will affect the reference point too.

No.54~No.58 parameter in axis parameter set the offset of finishing back to the datum point:

Used to set how much distance to offset before returning to the reference point when the X(No.54) Y(No.55) Z(No.56) A(No.57) axis backing to zero and checking the zero pulse signal of servo motor. Unit: 0.01mm. Numerical range:-99999~+99999.

The parameter value is related to the install position of lathe's datum point and the lathe's coordinate.

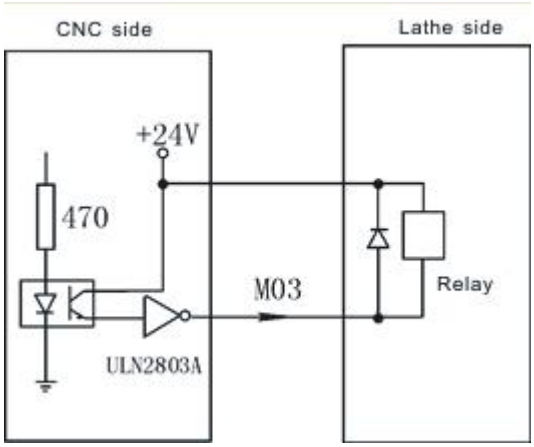
Pay attention: After backing to the datum point, the offset speed is determined by G00.

1. The speed reduce switch is also can use NPN switch.

2. Must consider the length of speed reduce when installing the speed reduce switch.Must be less than or equal to 25mm.

4.8.3 The controlling signal of switch: M03 as example (the same as M04 M05 M08 M10 M32 M79 M75 M59 M61 M63 M65 M67 M69 M71 M73 S01—S04 Y24/Y31)

M03 signal control:



As the picture say, it will form a return circuit with 24V when system outputting M03, The intermediate relay is working and a group of normally open contact form a circuit with spindle rotation AC contactor.

All the low level 0V of output signal is effective.

Pay attention:

1.When the relays and others load, must connected with the diode to absorb the reverse current so as not to damage the system, if use the electromagnetic contactor, then plus resistive and capacitive spark circuit.

2. Chip ULN2803A corresponds output ports:

1). U28 corresponds to M59 M61 M63 M65 M67 M69 M71 M73

2). U29 corresponds to M03 M04 M05 M08 M10 M79 M32 M75

3). U30 corresponds to +T –T S01 S02 S03 S04 LRUN INTH

4). U53 corresponds to Y24/Y31

3. When user-defined signal M71/M70 is used for input signal of spindle chuck and thumbstall, it can't be the other user-defined. No.20 No.21 parameter in other parameter to set.

4. When user-defined signal M65 M67 M69 is used for input signal of "Emergency", it can't be the other user-defined. M65 the program stops to output,M69 the program runs output,M67 the alarm output. No.28 No.29 parameter in other parameter to set.

5. Through the PLC could programme 32 points output: Y00-Y31, pay attention to the function can not be the same usage as original PLC system and the original system function.

4.8.3.1 System spindle control (M03/M04/M05)

System output two spindles (First spindle, second spindle) controlling signal, relative parameter as follows:

Axis parameter

No.13 parameter: Set the braking time of spindle, also the it's the output of the hold time, the time less the brake fast.Unit:10ms.

No.14 parameter: Set the braking signal is long signal 1 or short signal 0.

No.59 parameter: Whether turn on the spindle or not when spindle shifting [1 means on, 0 means off]

No.60 parameter: The turning speed of motor when spindle shifting (1/100rpm)

No.61 parameter: The turning direction when spindle shifting (0 means positive, 1means negative)

No.62 parameter: The stopping time when spindle shifting (10ms)

No.63 parameter: Turning time of low speed when spindle shifting (10ms)

No.64 parameter: Stopping delay time of spindle (10ms)

No.220 parameter: All pulse controlling channel in the first spindle(Positive 88,reversal 98)

No.221 parameter:All pulse controlling number of per cycle in the first spindle

No.222 parameter:All pulse controlling lifting speed in the first spindle(rpm/2ms)

No.223 parameter:All pulse controlling channel in the second spindle(Positive 82Y/84A/85B, reversal 92 Y/94A/95B)

No.224 parameter:All pulse controlling number of per cycle in the second spindle

No.225 parameter:All pulse controlling lifting speed in the second spindle(rpm/2ms)

SPEN(Y30) is the enabled signal if the spindle is all pulse controlling, set No.14 parameter as 1 means spindle is also enabled when spindle is stop, set as 1 to not to enable. SPEN2(Y31) as the enable signal of the second spindle servo.

Speed parameter

No.11 parameter: To set the speed of spindle setting value at manual condition. Unit: rpm.

No.48 parameter: To set the highest speed of spindle, that's the turning speed of corresponding 10V instruction voltage. Unit: r/min

No.49 parameter: To set the highest speed of spindle low gear(second gear) or the highest speed of second spindle, that's the turning speed of corresponding 10V instruction voltage. Unit: r/min

No.50 parameter: To set the highest speed of spindle (Third gear), that's the turning speed of corresponding 10V instruction voltage. Unit: r/min

No.51 parameter: To set the highest speed of spindle (Fourth gear), that's the turning speed of corresponding 10V instruction voltage. Unit: r/min

No.52 parameter: To set the highest speed of second spindle, that's the turning speed of corresponding 10V instruction voltage. Unit: r/min

Other parameter

No.13 parameter: Be used to set the tool and the spindle is interlock: set as 0 means there is no connection; Set as 1 means the spindle could rotate only in the tighten status.Setting parameter is related with the configuration of lathe and user's service condition, but consider for safety, suggest setting as 1.

No.20 parameter: To set system controls chuck only need one signal (one-way valve) or two signals (two-way valve), this parameter is related with equipment of lathe.

M10 is just a output signal to control tautness of chuck when set to 0, system carry out chuck to tighten when M10 is effective, loosen chuck when M10 is invalid;

M10 and M71 corresponding control loose and tight of chuck when set to 1, system carry out chuck to tighten when relay M10 is effective, M71 is invalid, loosen chuck when M10 is invalid and M71 is effective. Output M10 when M10, output M71 when M11.

No.22 parameter: To set external button to control loose and tight of chuck(or foot switch), the signal is reciprocating, it means loosen once and then tighten once, reciprocating mode. No external button when set to 0; There is an external button to control chuck when set to 1, the signal is M16.

No.24 parameter: To set the retention time when the output signal M10 M71 of chuck is short signal, set to 0 means the signal is long signal. Unit: S.

Pay attention: M16 is a multiple function signal, only choose one function to use.

4.8.3.2 System lubrication control (M32/M33)

No.4 parameter in other parameter controls the function of lubricate automatically. No.6 parameter set the spacing time of lubrication (Unit: S); No.5 parameter in other parameter set the lubrication time (Unit: ms).

Pay attention: The signal is controlled by M32 output.

4.8.4 System alarm signal: ALM, ALM1, ALM2, Door alarm M12 and Emergency

Other parameter:

No.7 parameter: To set the system whether to check the switching signal of protective door, no door switch when set to 0, there is a switch to control protective door when set to 1; Suggest to set 1 for safe.

No.8 parameter: To set the type of door switch, 0 means always open, 1means always close.

No.17 parameter: To set the type of system checking the servo alarm signal, 0 means always open, 1 means always close.

No.18 parameter: To set the type of system checking the spindle alarm signal of lathe, 0 means always open, 1 means always close.

No.19 parameter: To set the type of system checking the alarm signal of lathe, 0 means always open, 1 means always close.

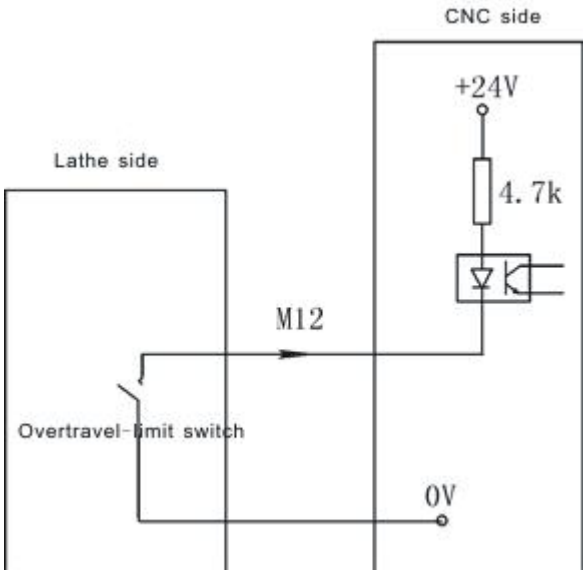
No.27 parameter: To set the Emergency always open or close of system CN11, suggest setting always close for safe.

Emergency STOP: Press "Emergency" when appearing emergent accident, the lathe will stop all actions and the screen of system shows "Emergency". Wait for pressing up the button. Output M67 signal is effective (output alarm) when No.29 parameter in other parameter is effective. This output signal can be used to protect the lathe (Cut off power supply).

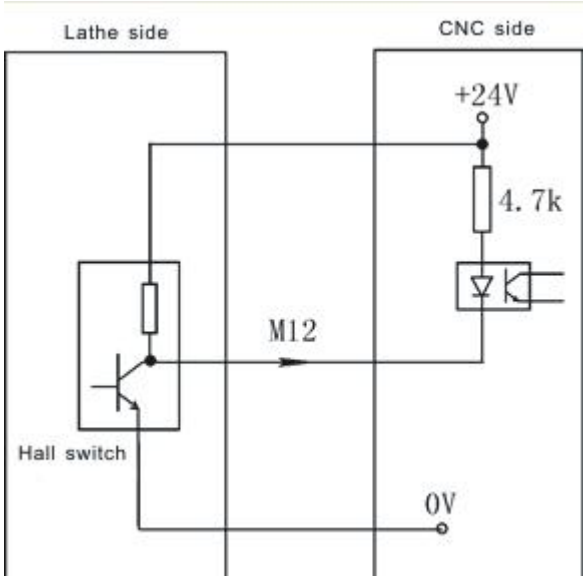
No.29 parameter in other parameter is effective when appearing alarm, the output signal M67 is effective.

4.8.5 User-defined output signal M12 (M14 M16 M18 M28 M22 M24,M24/B0、X40/X47, external "Run", external suspend "HALT", external "STOP" as the same)

M12 switch use overtravel-limit switch to wiring:



M12 switch also can use NPN as checking witch:



Pay attention:(I/O refers to the diagnosis interface)

1. M12 M14 M16 M18 and M28 are all multiple functions signal, only could choose one function to use.

2. All the low level 0V of output signal is effective.

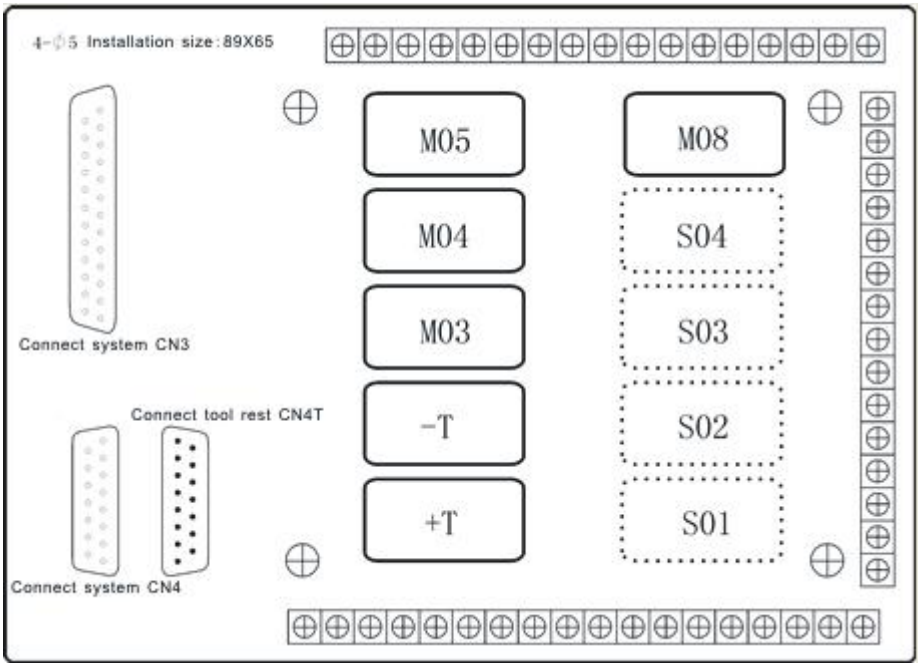
3.The PLC could programme 56 points input: X00-Y47, X60-X67. Note: the function can not be the same usage as the original PLC and the system function.

4. All the inputs X32-X39, X60-X67 whose internal power supply are +5V(internal is normally closed), not to access the voltage higher than +5V when using.

4.9 Electrical appliance plate of lathe

Our company produces the electrical appliance plate of lathe to choose as follows.

I/01 socket and CN3 are corresponding one-to-one with CN3 pins of system;



4.9.1 Spindle controlling

C1 is the common port of M03 and M04.

C2 is the common port of M05, M5B is always close.

4.9.2 Spindle gear controlling

C4 is the common port of S1 and S2, S1B and S2B are always close.

C5 is the common port of S3 and S4, S3B and S4B are always close.

4.10 I/O lathe electrical panel of Bus system

1, CN22 connect the CN22 bus socket of system

2, CN23 connect the bus driver

3, CN1 connect the electrical panel CN1, Pin CN1-1 to -11 control relay:
Y05,Y08,Y09,Y10,Y11,Y18,Y19,Y20,Y21,Y22,Y23

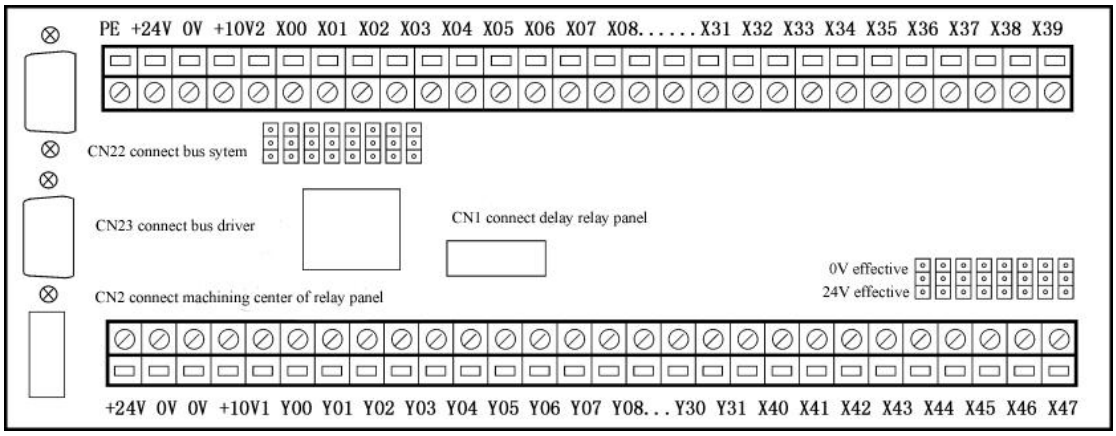
4,CN2 connect the relay panel of machining center, Pin CN2-1 to-11 control relay:
Y05,Y08,Y01,Y10,Y11,Y06,Y07,Y24,Y27,Y30,Y31

5, READY signal prepare the signal for system, the input is valid when power up the system.

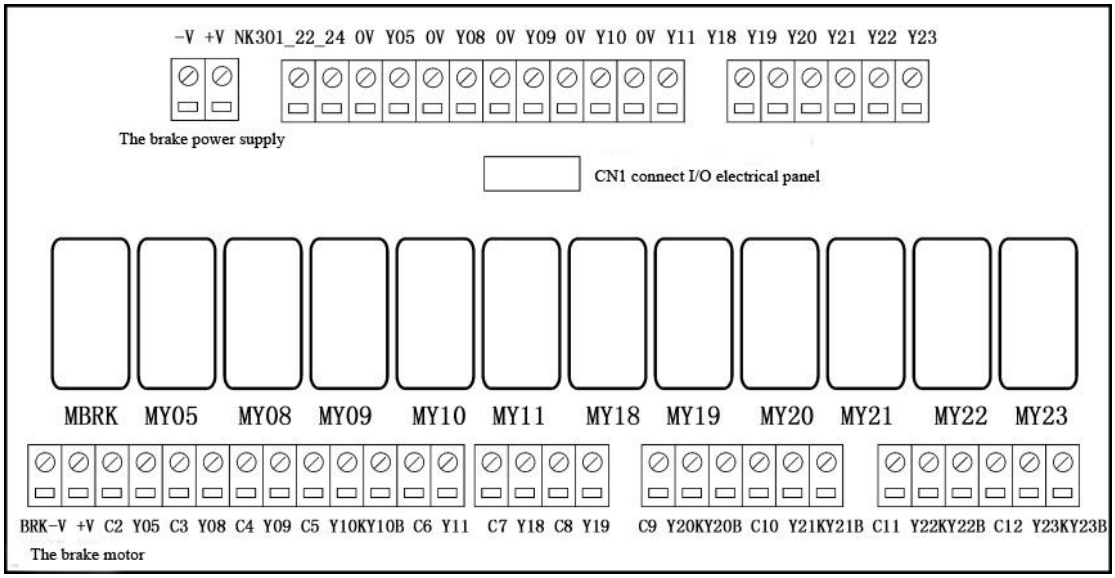
SPEN(Y30) signal is enable signal of spindle servo which is valid when spindle start, set it as invalid or valid by No.14 parameter in axis parameter when spindle is stop. SPEN(Y31) is the enable signal of the second spindle servo.

6, Input X00--X07, X40--X47 could select the high power +24V effective or low power 0V, through the skip stitch to select.

7, The terminal of PE must connect the ground of lathe to make sure without interference.



4.11 Relay panel of lathe with bus system



Our company produce the relay panel of lathe could be available

1. The pin of socket CN1 corresponding to the I/O electrical panel CN1, controls output:

Y05,Y08,Y09,Y10,Y11,Y18,Y19,Y20,Y21,Y22,Y23

2. Controls the other 11 output when socket CN1 connect the I/O electrical panel CN2:

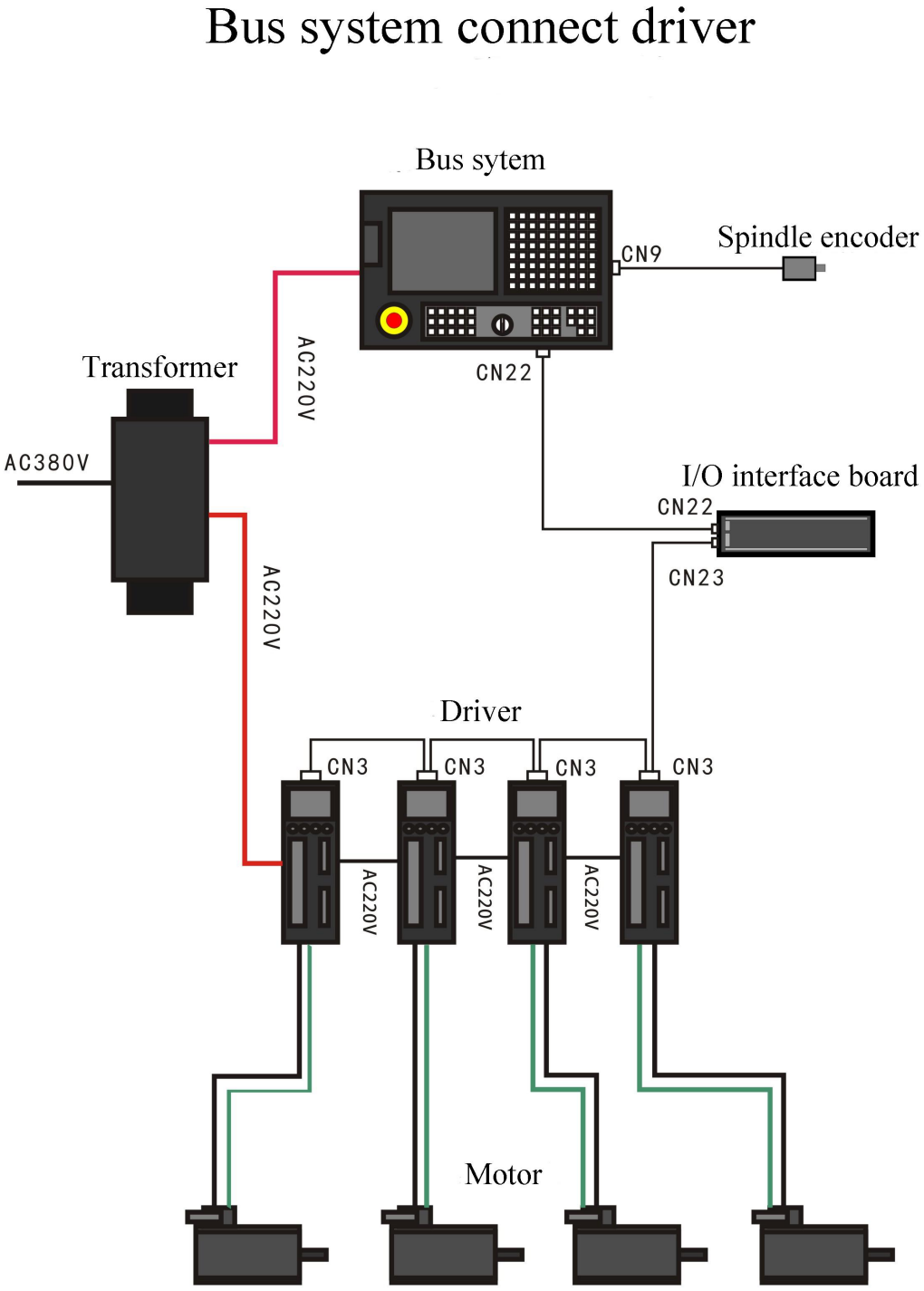
Y05,Y08,Y01,Y10,Y11,Y06,Y07,Y24,Y27,Y30,Y31

3. NK301_22 connect CN3-22 pin of driver, NK301_24 connect CN3-24 pin of driver.

4. -V, +V connect the brake power supply

5. BRK-V, +V connect the brake motor.

4.12 The circuit connection of bus system



Chapter 5 The CNC Machining Center

5.1 I/O ports

I/O	Signal	Bamboo hat type tool magazine	Mechanical hand tool magazine
X00	T01	Detection for the motor of tool magazine overload	Detection for the motor of tool magazine overload
X01	T02	Detection for loosing tool of spindle	Detection for loosing tool of spindle
X02	T03	Detection for lubricating oil level	Detection for lubricating oil level
X03	T04	Detection for lacking coolant	Detection for lacking coolant
X04	T05	Detection for tightening tool of spindle	Detection for tightening tool of spindle
X05	T06	Detection for tool magazine go forward	
X06	T07	Detection for tool magazine backward Requirement:Tool magazine must backward enough when turn on spindle	
X07	T08	Tool magazine count	Tool magazine count
X23	ALM1	Driver of spindle alarm	Driver of spindle alarm
X24	ALM2	Detection for cooling motor overload	Detection for cooling motor overload
X25	M28/C0	Backing(to the datum point) signal of C axis	Backing(to the datum point) signal of C axis
X26	M24/B0	Backing(to the datum point) signal of B axis	Backing(to the datum point) signal of B axis
X27	M22	Spindle finish locating/Detection for backing to zero position	Spindle finish locating/Detection for backing to zero position
X29	M12	Detection for lathe door switch Requirement: P7=1 in other parameter	Detection for lathe door switch Requirement: P7=1 in other parameter
X30	M14	Detection for pressure alarm of	Detection for pressure alarm of

		compressed air	compressed air
X31	M16	Button for spindle loosen tool remotely Requirement: P22=1 in other parameter	Button for spindle loosen tool remotely Requirement: P22=1 in other parameter
X40		Detection for motor of chip removal overload alarm	Detection for motor of chip removal overload alarm
X41			Detection for tool magazine locating
X42			Stop checking the mechanical hand tool magazine:(Mechanical hand tighten tool, catch tool, return tool)
X43			Signal of servo tool tighten or loosen to the position
X44			Detection for tool-case raising up Requirement: Raise up enough before changing tool
X45			Detection for tool-case falling down
X46		Stop instruction M01 input	Stop instruction M01 input
X47		Detection for fan of spindle overload	Detection for fan of spindle overload
Y00	M61	Backing to zero of spindle output, detection for backing to zero enough M22	Backing to zero of spindle output, detection for backing to zero enough M22
Y01	M63	Tool magazine rotate CW controlling output， Checking tool count of T08；	Tool magazine rotate CW controlling output， Checking tool count of T08；
Y02	M65	Yellow indicator lamp（Stop）	Yellow indicator lamp（Stop）
Y03	M67	Red indicator lamp（Fault）	Red indicator lamp（Fault）
Y04	M69	Green indicator lamp（Running）	Green indicator lamp（Running）
Y05	M71	Tool magazine rotate CCW controlling output， Checking tool count of T08；	Tool magazine rotate CCW controlling output， Checking tool count of T08；
Y06	M73	Tool magazine go forward	

		controlling output, checking going enough T06	
Y07	M59	M59 huff	M59 huff
Y08	M32	Lubrication controlling output	Lubrication controlling output
Y10	M10	The control output elastic tool: Spindle loose tool: output M10, detection for loosing enough T02; Spindle tighten tool: cancel M10, detection for tightening enough T05;	The control output elastic tool: Spindle loose tool: output M10, detection for loosing enough T02; Spindle tighten tool: cancel M10, detection for tightening enough T05;
Y11	M08	Cooling controlling output	Cooling controlling output
Y12	M05	M05	M05
Y13	M04	Spindle rotate CCW controlling output	Spindle rotate CCW controlling output
Y14	M03	Spindle rotate CW controlling output	Spindle rotate CW controlling output
Y15	M75	Switch controlling mode of spindle output	Switch controlling mode of spindle output
Y24		Tool magazine backward controlling output， checking backward enough T07	Motor controlling of mechanical hand tool magazine output(Mechanical hand tighten tool, catch tool, return tool)
Y25			
Y26			Tool case raise up controlling output， checking raise up enough X44
Y27			Tool case fall down controlling output， checking fall down enough X45，detection for tool magazine locating must be in place；
Y28	M203	The second spindle rotate positive	The second spindle rotate positive
Y29	M204	The second spindle rotate negative	The second spindle rotate negative
Y30	SPEN	Enable the servo driver of spindle	Special tool-case:The motor of mechanical hand controls output(Mechanical hand tighten

			tool, catch tool, return tool)
Y31	SPEN2	Enable servo driver of the second spindle	Enable servo driver of the second spindle

5.2 Parameter of the CNC machining center

(A) Axis parameter

P57, the number of encoder pulse per turn

P400, to check whether the spindle position feedback when tool is changing (1 means check, 0 means not to check)

P401, check angle when spindle orientation

P402, check error of angle when spindle orientation

P403, controlling signal of spindle pulse (0 means negative, 1 means normal)

P404, spindle whether or not to use electronic gear (0 means use, 1 means not to use)

P405, molecular of spindle low-gear electronic gear (1-32767)

P406, the denominator spindle low-gear electronic gear (1-32767)

P407, molecular of spindle high-gear electronic gear (1-32767)

P408, the denominator of spindle high-gear electronic gear (1-32767)

(B) Tool parameter

The total number of tool: according to "F7" set in the "Redeem"(tool compensation) status.

P10, whether to start the function of select tool or change tool [0 means no,1 means start]

P11, back to the point axis of tool changing[3 means Z X, 4 means Z Y,5 means Z X Y, others means Z]

P12, whether to check the spindle orientation and the tool changing point of feeding axis before changing tool[1,2,3,4 stand for corresponding datum point, others means not to run]

P13, change the knife before detection of spindle orientation and feed back to the point of exchange tool [0 means not to check, 1 means check]

P14, whether the spindle orientation when changing tool[0 means not orientate, 1 means orientate]

P15, the tool magazine type [0 means rotary table, 2 means bamboo hat type]

P16, special tool magazine [0: standard, 16: special bamboo hats, 64: special mechanical hand]

P17, The lifting height of Z axis when bamboo hat type of tool magazine changing tool (mm)

P18, The speed of Z axis when bamboo hat type of tool magazine changing tool (mm/min)

P19, Bamboo hat type return (tool case of rotary type down / tight tool) time delay (0.1 sec)
P20, delay time after loosing tool when changing tool (0.1 sec)
P21, delay time after tool case raising up (ms)
P22, whether the Z axis motion and main hand tool / bamboo hat type tool magazine are interlocking or not[1 means yes, 0 means no]
P23, whether the go forward instruction of tool magazine M71 check the position of Z axis or not [0 means yes, 1 means no]
P24, the maximum tool number of fixed tool position area [after setting the parameters, please initialize the tool case table]
P25, signal of tool magazine count tool [rise along the "1000+ number", down along the "2000+ number"]
P26, signal of tool magazine locating [1000+ number]
P27, output point of tool magazine rotate CW [1000+ number]
P28, output point of tool magazine rotate CCW [1000+ number]
P29, input point of mechanical hand brake [1000+ number]
P30, output point of mechanical hand rotating [1000+ number]
P32, The mode of choosing tool/count the number of tool
[D=0 means one-way, D0=1 means both way]
D0=1 means the way that machining center choose the tool is random select with both way;
D1=1 means the detection for counting tool signal along rotate reversal;
D2=1 means smoothing the counting tool signal of machining center;
D3=1 means smoothing the stopping signal of mechanical hand in machining center;
D4=1 means adapt 8ms smoothing way to signal of counting tool and stopping[otherwise adapt 4ms smoothing way]
D5=1 means the machining tool-case will let tool pouch down automatically, after processing M6 when the instruction is effective, the tool pouch will not lift up automatically until processing instruction T again;
D6=1 means not detect the tool pouch lifts up to the position when processing the changing tool;
D7=1 means not to wait the tool pouch lift up and process the next program;
D8=1 means smoothing 2ms to detection signal of tool position;
D9=1 means smoothing 4ms to detection signal of tool position;
D10=1 means smoothing 8ms to detection signal of tool position, D8-D10 could be set as 1 at the same time;
D11=1 means smoothing 2ms to input signal WAT;
D12=1 means smoothing ms to input signal WAT;
D13=1 means smoothing 8ms to input signal WAT, D8-D10 could be set as 1 at the

same time;

D14=1 means the mechanical hand of tool-case is the special mechanical hand which is production of TengZhouDaDi company

P70, Positive rotating speed of servo tool-case backing to the datum point (degree/min)

P71,Reversal rotating speed of servo tool-case backing to the datum point (degree/min)

P72, The offset distance of servo tool-case backing to the datum point(0.001 degree)

P73,Rotating speed when servo tool-case choosing tool(degree/min)

P74,Using coordinate axis of servo tool-case(1=A,2=B,3=C)

P75,Running acceleration of servo tool-case(degree/min/s)

P76,The maximum distance of detecting the datum point signal when servo tool-case backing to the datum point(0.001 degree)

P77,Whether detect the datum point signal of motor or not when servo tool-case backing to the datum point(0 means not to detect, others means detect, 367 means the motor of servo tool-case adapt absolute encoder)

P78,The servo tool-case rotate a tool position need the number of pulse that system send

P79,The current tool number after the servo tool-case backing to the datum point

P100, reference point1 X (mm)

P101, reference point1 Y (mm)

P102, reference point1 Z (mm)

P103, reference point1 A (mm)

P104, reference point2 X (mm)

P105, reference point2 Y (mm)

P106, reference point2 Z (mm)

P107, reference point2 A (mm)

P108, reference point3 X (mm)

P109, reference point3 Y (mm)

P110, reference point3 Z (mm)

P111, reference point3 A (mm)

P112, reference point4 X (mm)

P113, reference point4 Y (mm)

P114, reference point4 Z (mm)

P115, reference point4 A (mm)

(C) Speed parameter

P230,Locating speed G00 of SP spindle(0.1rpm)

P231,The locating controlling mode G01 of SP spindle[+4 means according to F, +8 means according to G90/G91] G90 means absolute value will be reduction, G91 means incremental value not to reduce and run according to handy direction.

P232,The locating direction of SP spindle in interpolation mode[0 means positive, 1 means negative, others mean hand direction] This parameter is working only not be controlled by G90/G91, P230/P231/P232 requires to set No.411 parameter in axis parameter the controlling mode in tapping[2 means tracking; 3 means interpolation] to be interpolation.

5.3 Debugging of CNC machining center

"Tab HxDx cdTef" meaning in the system status, "ab," means the tool number of spindle, "c d" means the current tool case position number in tool magazine, "e f" means the current tool number in tool magazine.

(A) The standard bamboo hat type tool magazine of CNC machine center

1, the manual button operation

K1 is spindle orientation (pilot lamp K1 will brighten after finishing spindle orientation): output M61, check M22 ;

K2 is the backing changing point of Z axis;

K3 is setting the tool case number of the current tool case;

"Tool magazine rotate CW": output M63, check T08 ;

"Tool magazine rotate CCW": output M71, check T08 ;

2, M instruction

M71 is tool magazine going forward, cancel Y24, output M73, tool parameter P23=1;

M73 is tool magazine backward, cancel M73, output Y24, tool parameter P23=1;

M881 is the same as "K1 function";

3, Tool changing operation

Txx: Move the tool of spindle back to the tool case and move the Txx to spindle.

M36 Txx: Move the Txx to spindle according to the step in single step mode. After the executing a step to be in a suspended state, continue to execute the next step after pressing button "Run", mainly use for debugging.

4, Set the offset of spindle orientation(spindle encoder signal must access into system)

1) P400=1, check spindle position feedback when changing tool, set to "0" not to check the position feedback;

2) set P401, check angle when spindle orientation, manual press K1 to import SP value in P401 after finishing spindle orientation;

3) set P402, check the error of angle when spindle orientation;

5, Tool changing process

1) Z axis to changing tool point (Z axis must return to zero first);

2) Spindle orientation: output M61, check the backing zero is in place M22;

3) Tool goes forward: output M73, check going forward in place T06;

4) Spindle tool loosen: output M10, check loosening tool in place of T02;

5) Z axis lifting;

6) Tool is rotating CW (or CCW): output M63 (or M71), check tool position counting T08;

7) spindle loose tool: output M10, detection of loosen tool in place of T02;

8) Z axis falls down to the tool changing point;

9) Spindle tighten tool: cancel M10, check tightening tool in place T05;

10) Tool backward: output Y24, check backward in place T07;

6, Tool parameter

P10=1, start the tool changing program;

P11=0, just Z back to the tool changing point;

P12=2, feeding axis back to the second datum point when changing tool;

P13=1, check spindle orientation and the tool changing point of feeding axis before changing tool,check M22 signal ;

P14=1, spindle orientation when changing tool,output signal M61;

P15=2, the tool magazine type is "bamboo hat type";

P16=0, the standard tool magazine;

P17=120, the lifting height of Z axis when changing tool(mm);

P18=2000, the raising speed of Z axis when tool magazine changing tool (mm/min);

P19=5, bamboo hat type to back tool delay (0.1 sec), tool magazine goes forward check the delay time after T06 is in the place and spindle tighten tool check the delay time after T05 is in the place;

P20=5, delay time(0.1 sec) after loosening tool when changing tool, loosen tool check the delay time after T02 is in the place;

P21=5, delay time after tool case lifting (ms);

P22=1, whether the Z axis motion and main hand tool / bamboo hat type tool magazine is interlocking [1 means yes, 0 means no], set to "1" means check the signal T07 of tool magazine backward in place;

P23=1, the tool goes forward instruction M71 is to check whether the position of Z axis[0 means yes, 1 means no], when setting to "0", to check the Z axis is at the tool changing position;

P24=0, the maximum tool number of fixed tool position area [after setting the parameter, please initialize the tool case table];

P25=1007 (X07), signal of tool magazine count [rising along "1000+ number", "down along "2000+ number"];

P26=1041 (X41), signal of tool magazine locating[1000+ number];

P27=1001 (Y01), output point of tool magazine rotating CW [1000+ number];

P28=1005 (Y05), output point of tool magazine rotating CCW [1000+ number];

P100=0, reference point1 X (mm) ;
P101=0, reference point1 Y (mm) ;
P102=0, reference point1 Z (mm) ;
P103=0, reference point1 A (mm) ;
P104=0, reference point2 X (mm) ;
P105=0, reference point2 Y (mm) ;
P106=50, reference point2 Z (mm) ;
P107=0, reference point2 A (mm) ;

Special attention:

1. When power off suddenly or emergency happening cause the tool case is in a mess in tool changing process, please rotate the tool case for some tool position in manual mode and use K3 to set the current tool case number;

2. Pay attention to check position of Z axis in case for accident when using instruction M71 to make tool case going forward;

3. No.T0 tool should not have tool, otherwise may be an accident;

4. The first time to install tool, spindle must have tool (if the display is T00, usually can initialize the tool case table, spindle should be No.T01 tool).

(B)The standard mechanical hand tool magazine machining center

1. The manual button operation

K1 is spindle orientation (pilot lamp K1 will brighten after finishing spindle orientation): output M61, check M22 ;

K2 is the backing changing point of Z axis;

K3 is setting the tool case number of the current tool case;

"Tool magazine rotate CW": output M63, check T08, the tool case must lift to position X44;

"Tool magazine rotate CCW": output M71, check T08, the tool case must lift to position X44;

2. M instruction

M71 is tool magazine going forward, cancel Y26, output M27, tool case locating check the X41 must in place;

M73 is tool magazine backward, cancel M27, output Y26;

M65 is unconditional rotation step of mechanical hand for debugging,input Y24;

M881 is the same as "K1 function";

3. Tool changing operation

M06: Move the tool of current tool case to spindle.

Txx: Move the Txx of instruction to the current tool changing position.

M06 Txx: Change the tool of current tool case into spindle and move the Txx of instruction into tool changing position for the next time tool changing(First choose tool and then change tool).

M106 Txx: Move the Txx of instruction into tool changing position and then

change the current tool of tool case into spindle.(First change tool and then choose tool)

M36: Move the Txx to spindle according to the step in single step mode. After the executing a step to be in a suspended state, continue to execute the next step after pressing button "Run", mainly use for debugging.

4. Set the offset of spindle orientation(spindle encoder signal must access into system)

1) P400=1, check spindle position feedback when changing tool, set to "0" not to check the position feedback;

2) set P401, check angle when spindle orientation, manual press K1 to import SP value in P401 after finishing spindle orientation;

3) set P402, check the error of angle when spindle orientation;

5. Tool changing process

1) Z axis to changing tool point (Z axis must return to zero first);

2) Spindle orientation: output M61, check the backing zero in place M22;

3) Fallen tool case: output Y27, check fallen in place X45, tool magazine locating check X41 must be in place;

4) Mechanical hand clasp tool: output Y24, check clasp tool in place X42;

5) Spindle loose tool: output M10, detection of loosen tool in place of T02;

6) Mechanical hand take tool and change tool: output Y24, check take tool in place X42;

7) Spindle tighten tool: cancel M10, check tightening tool in place T05;

8) Mechanical hand return: output Y24, check return in place X42;

9) Raise up tool case: output Y26, check raising in place X44, if not detect X44, it will alarm "Tool magazine fault";

Special attention:

1. When power off suddenly or emergency happening cause the tool case is in a mess in tool changing process, please rotate the tool case for some tool position in manual mode and use K3 to set the current tool case number;

2. For this kind of tool magazine, could set the No.24 parameter in tool parameter to set the maximum tool number of fixed tool position area, it's better to initialize the tool case table after setting the parameter. For example: Set to 8 stand for just could from 1 to 8 tool case table to put in the tool with corresponding number by one-one, that's not random, it's fixed to install. This function could be used for installing tool of large diameter cutter;

3. No.T0 should not have tool, otherwise may be an accident;

4. M65 doesn't check the condition, only stop in emergency, pay more attention when using, otherwise to cause the safe accident.

6. Tool parameter setting

P10=1, start the tool changing program;

P11=0, just Z back to the tool changing point;

P12=2, feeding axis back to the second datum point when changing tool;

P13=1, check spindle orientation and the tool changing point of feeding axis before changing tool,check M22 signal ;

P14=1, spindle orientation when changing tool,output signal M61;

P15=2, the tool magazine type is "bamboo hat type";

P16=0, the standard tool magazine;

P17=120, the lifting height of Z axis when changing tool(mm);

P18=2000, the raising speed of Z axis when tool magazine changing tool (mm/min);

P19=5, bamboo hat type to back tool delay (0.1 sec), tool magazine goes forward check the delay time after T06 is in the place and spindle tighten tool check the delay time after T05 is in the place;

P20=5, delay time(0.1 sec) after loosening tool when changing tool, loosen tool check the delay time after T02 is in the place;

P22=5, whether the Z axis motion and main hand tool / bamboo hat type tool magazine is interlocking [1 means yes, 0 means no], set to "1" means check the signal T07 of tool magazine backward in place;

P23=1, the tool goes forward instruction M71 is to check whether the position of Z axis[0 means yes, 1 means no], when setting to "0", to check the Z axis is at the tool changing position;

P24=0, the maximum tool number of fixed tool position area [after setting the parameter, please initialize the tool case table];

P25=1007 (X07), signal of tool magazine count [rising along "1000+ number", "down along "2000+ number"];

P26=1041 (X41), signal of tool magazine locating[1000+ number];

P27=1001 (Y01), output point of tool magazine rotating CW [1000+ number];

P28=1005 (Y05), output point of tool magazine rotating CCW [1000+ number];

P29=1042 (X42), braking output point of mechanical hand [1000+ number]

P30=1024 (Y24), rotating output point of mechanical hand [1000+ number]

P32, The mode of choosing tool/ signal of counting tool[0 means one-way, 1 means both way]

P100=0, reference point1 X (mm) ;

P101=0, reference point1 Y (mm) ;

P102=0, reference point1 Z (mm) ;

P103=0, reference point1 A (mm) ;

P104=0, reference point2 X (mm) ;

P105=0, reference point2 Y (mm) ;

P106=50, reference point2 Z (mm) ;

P107=0, reference point2 A (mm) ;

(C) Special turntable style of tool magazine CNC machining center

1. Parameter setting: tool parameter P15=2, P16=16, tool case is set to "special bamboo hat".

2. Editing tool changing program ProgramTool deliver into system. ProgramTool programming from the tool changing process of the third step begins, other operation is the same as standard bamboo hat type tool magazine.

(D) Special mechanical hand of tool magazine CNC machining center

1. Parameter setting: tool parameter P15=0, P16=64, tool magazine is set to "special mechanical hand"; tool parameter P29=0, P30=0.

2. Editing tool changing program ProgramM6 deliver into system. ProgramM6 programming from the tool change process of the third step begins, other operation is the same as standard mechanical hand tool magazine.

(E) Special gang tool magazine machining center

1. Parameter setting: tool parameter P15=1, P16=128, set tool magazine to be “gang feed tool”

2. Deliver changing tool program Program Tool into system. The whole process of changing tool is finished by Program Tool.

(F)Special servo tool magazine machining center

1. Parameter setting: tool parameter P15=0, P16=32 or P16=32+64=96
   Hat type of tool parameter P15=2, P16=32 or P16=32+16=48

2.When P15=0 P16=96, deliver Program Tool into system. ProgramTool programming from the tool changing process of the third step begins, others operation is the same as the standard machining hand tool magazine.

When P15=2 P16=48 deliver Program Tool into system. ProgramTool programming from the tool changing process of the third step begins,thers operation is the same as the standard hat type of machining hand tool magazine.

3. Servo tool magazine backing to the datum point: P74=1 to check A0 signal, P74=2 to check B0 signal, P74=3 to check C0 signal; Need to check the zero point signal of motor when P77=1; The motor of servo tool magazine is absolute motor when P77=367, need not to back to zero point.

4. Servo tool magazine tighten or loosen output signal Y25, to the position signal X43, must connect Y25 to X43 when there is no need tighten or loosen.

(G)Match disk type of machining hand with tool magazine machining center is made by TaiWan ShouLun machine company

1. Parameter setting: tool parameter P16=64; Set tool magazine to “Special machining hand”; tool parameter P29=0, P30=0

2.Deliver ProgramM6 into system. ProgramM6 programming from the tool changing process of the third step begins, others operation is the same as the standard machining hand tool magazine.

3. M instruction

M71 is tool case fallen down, cancel Y26, output M27, tool case locating check the X41 must in place;

M73 is tool case raising up, cancel M27, output Y26;

OUT+Y30 is an unconditional step for rotating of mechanical hand, be used for debugging, output Y30;

M881 is the same as "K1 function";

4. The signal connection table of tool magazine(input signal of tool magazine through 2803 negative):

I/O	Signal	Mechanical hand type of tool magazine	mechanical disk type of tool magazine
X00	T01	Detection for the motor of tool magazine overload	
X01	T02	Detection for loosing tool of spindle	
X02	T03	Detection for lubricating oil level	
X03	T04	Detection for lacking coolant	
X04	T05	Detection for tightening tool of spindle	
X05	T06		Confirm signal of clasping tool S6（Right）
X06	T07		Confirm signal of the original point S7（Left）
X07	T08	Tool magazine count	Tool magazine count S1
X23	ALM1	Driver of spindle alarm	
X24	ALM2	Detection for cooling motor overload	
X27	M22	Spindle finish locating/Detection for backing to zero position	
X29	M12	Detection for lathe door switch Requirement: P7=1 in other parameter	
X30	M14	Detection for pressure alarm of compressed air	
X31	M16	Button for spindle loosen tool remotely Requirement: P22=1 in other parameter	

X40		Detection for motor of chip removal overload alarm	
X41		Detection for tool magazine locating	Locating signal of tool magazine S2
X42		Stop checking the Mechanical hand tool magazine：	Confirm signal of braking S5（Middle）
X43			
X44		Detection for tool-case raising up Requirement: Raise up enough before changing tool	Locating signal of tool returning S4
X45		Detection for tool-case falling down	Locating signal of tool fallen S3
X46		Stop instruction M01 input	
X47		Detection for fan of spindle overload	
Y00	M61	Backing to zero of spindle output, detection for backing to zero enough M22	
Y01	M63	Tool magazine rotate CW controlling output， Checking tool count of T08；	Motor of tool changing motor rotate CW
Y02	M65	Yellow indicator lamp（Stop）	
Y03	M67	Red indicator lamp（Fault）	
Y04	M69	Green indicator lamp（Running）	
Y05	M71	Tool magazine rotate CCW controlling output， Checking tool count of T08；	Motor of tool changing motor rotate CCW
Y06	M73		
Y07	M59	M59 Huff	
Y08	M32	Lubrication controlling output	
Y10	M10	The control output elastic tool: Spindle loose tool: output M10, detection for loosing enough T02; Spindle tighten tool: cancel M10, detection for tightening enough T05;	
Y11	M08	Cooling controlling output	

Y12	M05	M05	
Y13	M04	Spindle rotate CCW controlling output	
Y14	M03	Spindle rotate CW controlling output	
Y15	M75	Switch controlling mode of spindle output	
Y24			
Y25			
Y26		Tool case raise up controlling output， checking raise up in place X44	Solenoid valve of tool returning S10
Y27		Tool case fall down controlling output， checking fall down in place X45， detection for tool magazine locating must be in place；	Solenoid valve of tool fallen S8
Y28		Feeding servo driver with energy	Feeding servo driver with energy
Y29		Servo driver of spindle with energy	Servo driver of spindle with energy
Y30		Motor controlling output of mechanical hand	Motor of the tool magazine structure
Y31			

(H) Example: Hat type of CNC machining center

If equip with spindle servo of the CNC of GuangZhou, pay attention as follows:

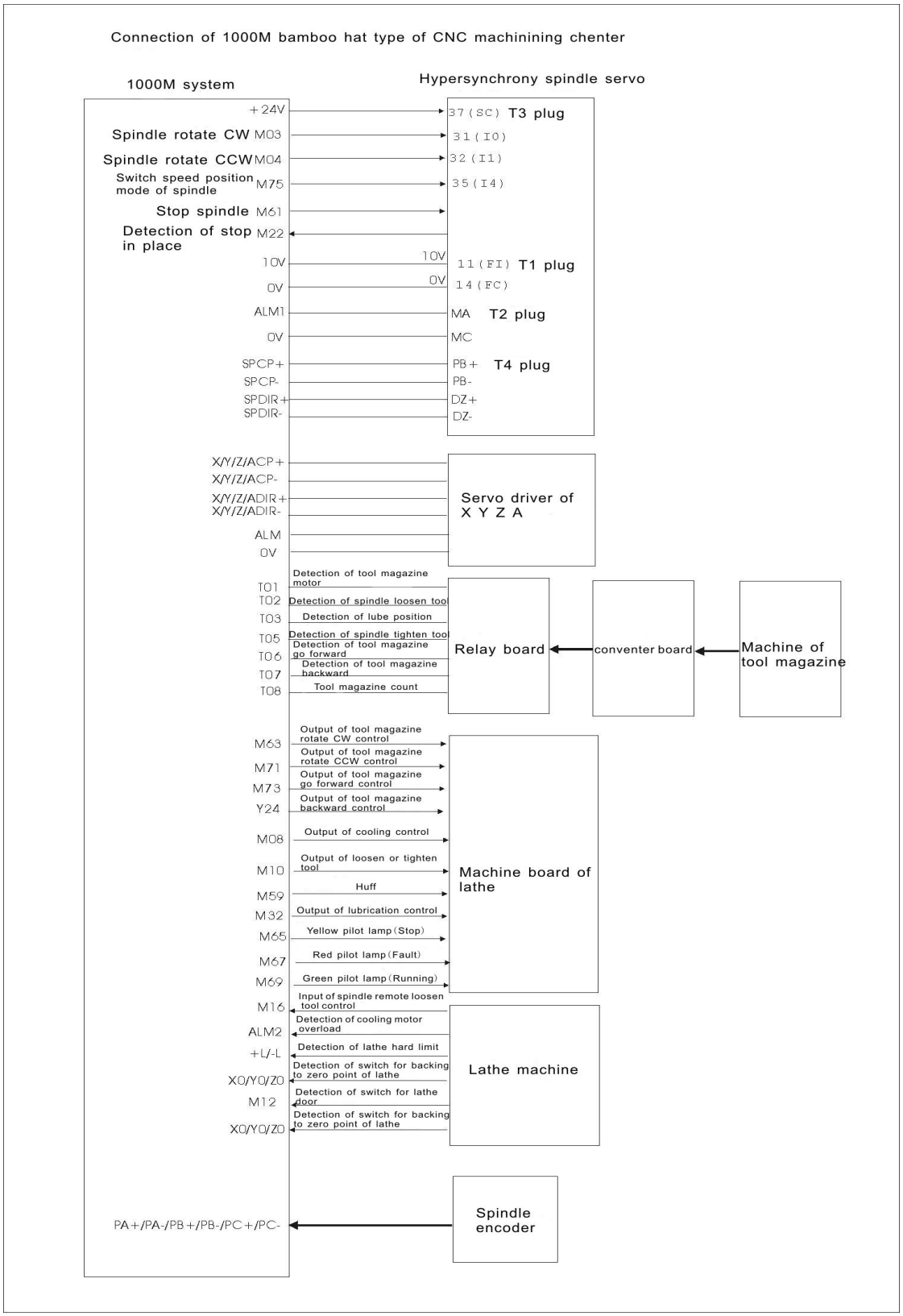
System M75 connects servo position of speed switching signal;

System M22 connects servo position of finish speed switching signal;

System M61 connects servo the starting orientation signal;

System M03/M04/M05/+10V/ALM1/+24V/0V connects servo drive corresponds to signal;

Servo drive parameter should be set to the position / speed controlling mode.


Connection of 1000M bamboo hat type of CNC machinining chenter
1000M system
Hypersynchrony spindle servo
+24V
Spindle rotate CW M03
Spindle rotate CCW M04
Switch speed position mode of spindle M75
Stop spindle M61
Detection of stop in place M22
10V
0V
ALM1
0V
SPCP+
SPCP-
SPDIR+
SPDIR-
37(SC) T3 plug
31(I0)
32(I1)
35(I4)
10V
0V
11(FI) T1 plug
14(FC)
MA T2 plug
MC
PB+ T4 plug
PB-
DZ+
DZ-
X/Y/Z/ACP+
X/Y/Z/ACP-
X/Y/Z/ADIR+
X/Y/Z/ADIR-
ALM
0V
Servo driver of X Y Z A
T01 Detection of tool magazine motor
T02 Detection of spindle loosen tool
T03 Detection of lube position
T05 Detection of spindle tighten tool
T06 Detection of tool magazine go forward
T07 Detection of tool magazine backward
T08 Tool magazine count
Relay board
conventer board
Machine of tool magazine
M63 Output of tool magazine rotate CW control
M71 Output of tool magazine rotate CCW control
M73 Output of tool magazine go forward control
Y24 Output of tool magazine backward control
M08 Output of cooling control
M10 Output of loosen or tighten tool
M59 Huff
M32 Output of lubrication control
M65 Yellow pilot lamp(Stop)
M67 Red pilot lamp(Fault)
M69 Green pilot lamp(Running)
Machine board of lathe
M16 Input of spindle remote loosen tool control
ALM2 Detection of cooling motor overload
+L/-L Detection of lathe hard limit
X0/Y0/Z0 Detection of switch for backing to zero point of lathe
M12 Detection of switch for lathe door
X0/Y0/Z0 Detection of switch for backing to zero point of lathe
Lathe machine
PA+/PA-/PB+/PB-/PC+/PC-
Spindle encoder