CONTENTS

Chapter 1 Preface

This system is a universal lathe CNC system 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 two or three, four 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. 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.

2. All the functions of the third and fourth axis are effective when configuring the designated system.

3. All the function of absolute-value must choose the absolute-value system, all the bus function must choose the bus system can be effective.

4.The “Run” button on the panel of system can be used when debugging (No.9-D14 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 my 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 Program coordinate

Absolute programming: it is confirmed coordinates data programming mode based on established absolute coordinate system. X, C/Y, Z, A stand for it.

Example: G00 X200 Z300

When using C to programme, it will execute according to the shortcut if C axis is rotate axis.

Also could use G990 to specify absolute coordinate to programme

Relative programming (increment programming): is the distance and direction of operation end point, compared with starting point. U, V, W stand for it.

Example: G00 U100 W-200

Also could use G991 to specify the relative coordinate to programme

Mixed programming: is the programming of one segment program by both absolute programming and relative programming.

Example: G00 X200 W30

Diameter programming: all X axis coordinate are presented by diameter.

Minimum program unit: the minimum program unit of XZ is 0.001mm. Real movement of X axis in diameter program is 0.0005mm; real movement of Z axis is 0.001mm.

2.2 Program structure

Name of program: the name of machining program. Contains of letter and number, the name of program cannot be the same.

Program segment: Be composed of number of line, instruction and data.

Example: N0001 G00 X20 W-300

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 instruction is called mode origin. In the above mode group, M05 is such a mode origin.

2.3 Program instruction

2.3.1 Functional meaning of address symbol, data list.

Functions	Address symbol	Meaning	Data range
File Name		Program file name	0-9,A-Z
Program segment No	N	Number of program segment	0000-9999
Preparation function	G	Content and mode of designate instruction operation	00-99
Auxiliary function	M	Auxiliary operation instruction	00-99
Tool function	T	Tool changing instruction	Tool 0000-9999
Spindle function	S	The first spindle speed	0-99999
Spindle function	SS	The second spindle speed	0-99999
Cutting speed	F	Speed per minute, per rotation	1-15000mm/min, 0.001-200mm/r
Coordinates character	XU/CV /ZW/A	The coordinates value of X C Z A axis	±99999.9999mm
Lead of screw thread	F(I)	F for metric, I for imperial	0.1-1000mm, 1-99teeth/inch
Core coordinates	I K	X Z axes coordinate increment value	±99999.9999mm
Arc radius	R	Arc radius value, tool radius value	0.001-99999.999mm
Delay time	P	Delay time of designate delay	0.001-65s
Program entrance	P	Entrance of calling program segment	0000-99999
Repeat times	L	Times of cycle or subprogram calling L can be used as numbers of multiply screw thread	1-99999 1-99

Program skip	/	There is "/"Before N ,this line does not run.	No.9 parameter in other parameter could avoid this function

2.3.2 Program instruction table

Group	Nature	Code	Functions	Origin	Mode	remarks
1	Main control functions	G00	Rapid point positioning	√	√	
		G01	Linear interpolation		√	
		G02	Circular—clockwise		√	
		G03	Circular—counterclockwise		√	
		G32	spiral interpolation		√	
		G332	G02 spiral			
		G333	G03 spiral			
		G31	Check jump No alarm			
		G311	Check jump alarm			
	Loop command	G70	Finish machining loop		√	
		G71	Cylindrical face thick loop		√	
		G72	end face thick loop		√	
		G73	Enclosed cutting loop		√	
		G74	end face deep hole loop		√	
		G75	slot loop		√	
		G76	complex screw thread loop		√	
		G90	Circular loop		√	
		G92	screw thread loop		√	
		G93	tap loop		√	
		G94	end face loop		√	
		G22	loop end		√	
		G800	cancel loop		√	
	Go to start	G26	All go to start point		√	
		G261	X go to start point			
		G262	Y go to start point			
		G263	Z go to start point			
		G264	A go to start point			

	Go to G25	G61 G611 G612 G613 G614	All go to G25 point X go to G25 point Y go to G25 point Z go to G25 point A go to G25 point		√	
	Save	G25	Save current coordinate		√	
	Go to Zero	G28 G281 G282 G283 G284 M800 M881 M882	go to zero X go to zero Y/C go to zero Z go to zero A go to zero C go to spindle encoder zero C axis stop, output M61, check M22 A axis stop, output M63,check M24		√	
	set	G50	Setup coordinate system and max spindle speed of constant line speed		√	
		G52	Setup part coor system		√	
		G184	Setup current Tool coor			
		G185	Setup all Tool coor			
2	Constant line speed	G96	Constant line speed cutting		√	
		G97	cancel	√	√	
	feed	G98	Set feed per minute	√	√	
		G99	Set feed per revolution		√	
	Program mode	G15	Cancel polar coor programme	√	√	
		G16	polar coor programme		√	
		C990	Absolute programme	√	√	
		C991	Relative prorgamme		√	
		G21	metric system program	√	√	
		G20	British system program		√	
3	delay	G04	Delay time			

4	Running mode	G60 G64	Orientation stop Continuum part	 √	√ √	
5	compensation	G40 G41 G42	Cancel tool compensation Tool in the left of workpiece Tool in the right of workpiece	√	√ √ √	
6	Workpiece coordinate	G53 G54 G55 G56 G57 G58 G59	Machine coordinate Work coordinate 1 Work coordinate 2 Work coordinate 3 Work coordinate 4 Work coordinate 5 Work coordinate 6	√	√ √ √ √ √ √ √	
7	Spindle	M03 M04 M05 M203 M204 M205 M881 M882	Spindle on CW Spindle on CCW Spindle brake M7053/M7054 Pxxxx: Spindle stop after rotate CW/CCW a while, the time is specified by P, unit:ms Example: M7053 P2000; means spindle rotate CW 2s and then stop C axis stop, output M61, check M22 A axis stop, output M63, check M24	√	√ √ √	M203 M204 M205 The second spindle rotate CW, CCW, stop
8	Cooling	M08 M09	Coolant on Coolant off	√	√ √	
9	Chuck	M10 M11	Chuck tightens Chuck looses	√	√ √	

10	Tailstock	M79 M78	Tighten Loose	√	√ √	Control M79
11	Lubrication	M32 M33	Lubrication on Lubrication off	√	√ √	Control M32
12	Huff	M59 M58	Huff on Huff off	√	√ √	Control M59
13	User-defined output	M61 M60	user-define 1 open user-define 1 close		√ √	Control M61
		M63 M62	user-define 2 open user-define 2 close		√ √	Control M63
		M65 M64	user-define 3 open user-define 3 close		√ √	Control M65
		M67 M66	user-define 4 open user-define 4 close		√ √	Control M67
		M69 M68	user-define 5 open user-define 5 close		√ √	Control M69
		M71 M70	user-define 6 open user-define 6 close		√ √	Control M71
		M73 M72	user-define 7 open user-define 7 close		√ √	Control M73
		M75 M74	The spindle servo select the position controlling mode when C axis is output pulse signal, close M75 signal when system starting M03/M04		√ √	Control M75
14	user-defined input	M12 M13	Check M12 valid Check M12 invalid		√ √	Conditional skip, example: M20 P120 means if M12 is valid, program skips to 120th line. M1810 means
		M14 M15	Check M14 valid Check M14 invalid		√ √	
		M16 M17	Check M16 valid Check M16 invalid		√ √	
		M18 M19	Check M18 valid Check M18 invalid		√ √	

		M28 M29	Check M28 valid Check M28 invalid		√ √	check if X10 valid
		M22 M23	Check M22 valid Check M22 invalid		√ √	
		M24 M25	Check M24 valid Check M24 invalid		√ √	
15	Spindle Gear	M41 M42 M43 M44	first gear second gear third gear fourth gear			To skip when conditions are tenable Example: M12 P120 Program skips to 120th line to carry out.
16	Subprogram	M97 M98 M99	jump call return			L=1-99 P is the line number of transferring program
17	Program	M87 M188 M187 M00 M01 M02 M30 M20	Part number plus 1 Keep feeding and not back home Keep feeding and back home pause M22 avail pause Program end M05、M09 end Loop go start			When other parameter P10=0 is set not to auotomatical plus 1, instruction M87 to make workpiece number plus 1
18	Spindle	S SS	Speed of the first spindle Speed of the second spindle		√ √	S=0-65000 SS=0-65000
19	Tool	Tab	a means change into a tool b means execute redeem of b		√	a.b=00-12 a=0 means not change the tool b=0 means show lathe coordinate

20	Read the position of absolute motor	M500 M501 M502 M503 M504	M500: read absolute motor position of all the feeding axis and reset the current lathe coordinate. M501: read absolute motor position of X axis and reset the current lathe coordinate. M502: read absolute motor position of Y axis and reset the current lathe coordinate. M503: read absolute motor position of Z axis and reset the current lathe coordinate. M504: read absolute motor position of the fourth axis and reset the current lathe coordinate.			
21	Clear workpiece coordinate	M315 M317 M318 M319 M320	M315:Clear workpiece coordinate of A axis M317:Clear workpiece coordinate of X axis M318:Clear workpiece coordinate of Y axis M319:Clear workpiece coordinate of Z axis M320:Clear workpiece coordinate of all axis			
22	Clear lathe coordinate	M415 M417 M418 M419 M420	M415:Clear lathe coordinate of A axis M417:Clear lathe coordinate of X axis M418:Clear lathe coordinate of Y axis M419:Clear lathe coordinate of Z axis M420:Clear lathe coordinate of all axis			

	Instruction of bus driver	M133 M135	M133: Driver rotate according to the specified speed M135: Enable to specify driver			Example: M133 X300; M135 Z0;Close the Z axis driver M135 X1;Enable X axis driver

2.4 Programming instruction and usage

2.4.1 Programming stipulation

1. Multiply instruction exist in a segment simultaneously: one program line allows multiply instructions in order to reduce the lines, but the same group of instruction can not share one segment.
2. Within a program segment, instruction and parameters can be arranged optionally. Such as: G01 U10 W-30 can be written: U10 G01 W-30
3. No repeat of instruction within a program segment.
4. It can't be irrelative parameters and operation in a segment.
5. "0" before a instruction is allowed to delete, such as: G01 G03 can be written as G1 G3.
6. The command of optional point, line start or that after tool changing instruction must be programmed by absolute coordinates.
7. Non mode command only in the in specified program line is effective,such as: G61.
8. Mode instruction is always effective before appearing the same instruction.

For example:
N0000 G01 X300 F100; G01 instruction
N0001 X260; G01 instruction
N0002 G00 Z200; G00 instruction, G01 is invalid

2.4.2 Instructions

The basic G mode include: G00, G01, G02, G03, G04, G28, G32, G50

（1）Rapid location(G00)

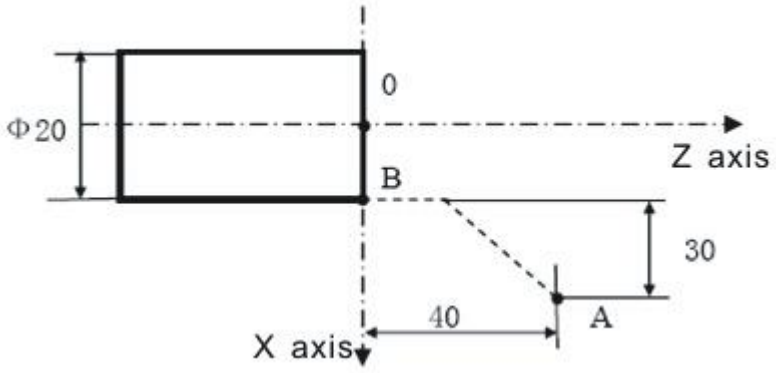
Tool move to instructive position according to G00 speed in parameter。
As absolute method, use section end point coordinate to program；
As increase method, use motion distance to program。

Format：G00 X(U)_ Y/C(V)_ Z(W)_ A_

Note：X，Y，Z，A 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。

Moving speed is determined by parameter。



Example：from A to B.

Absolute program: G00 X20 Z0；

Relative program: G00 U－60 W－40；

Pay attention: The nearest calculation when using absolute coordinate of rotating axis to programme, programme calculation when using relative coordinate;

G00 of every axis is set by parameter,the specified feeding speed with F is invalid. The speed of G00 can be divided into 5%～100%, totall six gears, it can be selected by the key on panel.

G00 is mode instruction, when the next instruction is G00 too, it can be omitted.G00 can be written G0.

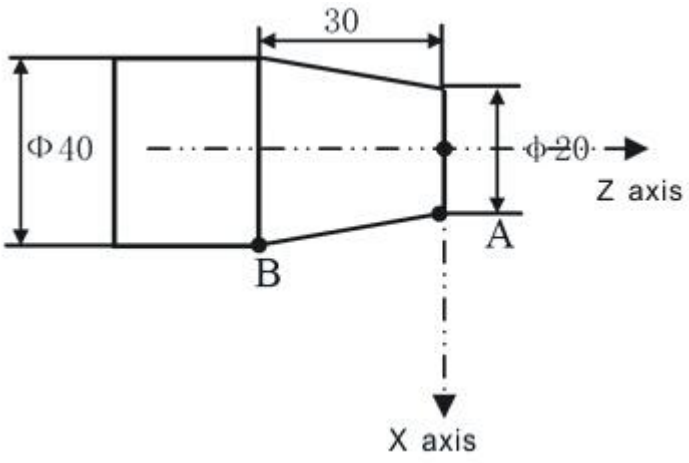
Pay more attention to whether the position of tool is in a safe area or not when X and Z axis both rapid moving, crashing tool in case.

（2）Line interpolation(G01)

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

Format：G01 X(U)_ Z(W)_ Y/C(V)_ A_ F_

Note：X，Y，Z，A means motion 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.it will valid till next specifying, so it is unnecessary to specify feed speed in each line.



Linear interpolation

Example：from A to B with F speed.

Absolute program: G01 X40 Z-30 F100;

Relative program: G01 U20 W-30 F100;

G01 also can specify the movement of X axis and Z axis all alone

F feeding speed of G01 could be modified by the feeding rate through the panel, the range is 0%~150%

G01 also can be written G1.

（3）Arc interpolation(G02/G03)

G02 stands for Circular clockwise interpolation, and G03 for Circular counterclockwise interpolation. I is the 2 times of increment of X axis which starting point to center of circle (needless to double it when X axis is radius programming), K is the increment of Z axis which starting point to center of circle, and X Z are the terminal coordinates. It can be also programmed by R not I K.

Format：

G02 X（U）__ Z（W）__ I__ K__ F__;

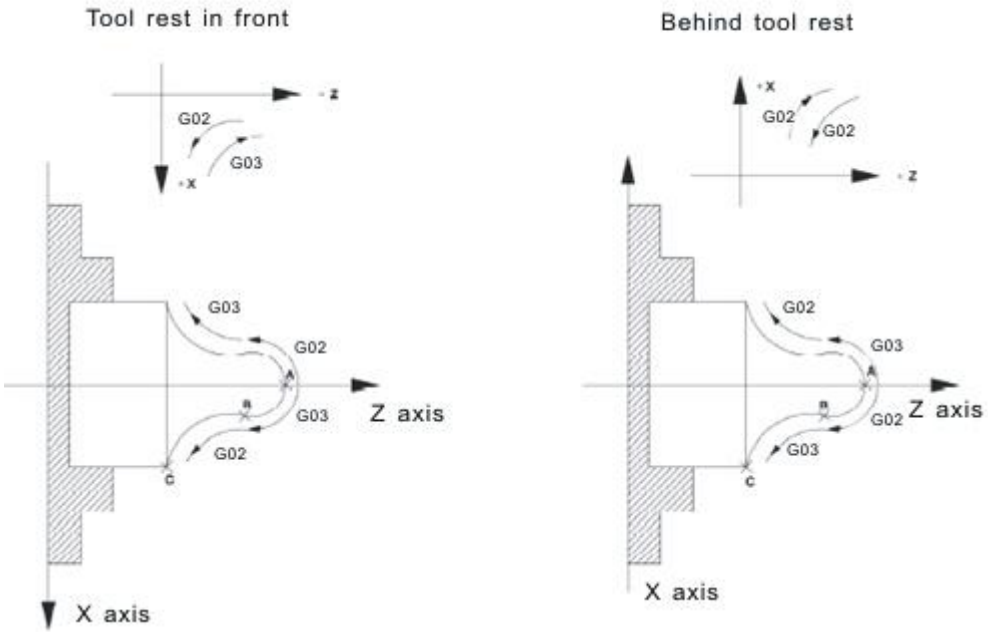
G03 X（U）__ Z（W）__ I__ K__ F__;

G02 X（U）__ Z（W）__ R__ F__;

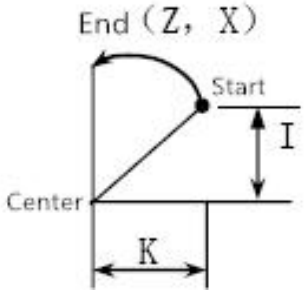
G03 X（U）__ Z（W）__ R__ F__;

Item	Specification		instruction	意义
1	Arc direction		G02	Rotate CW
			G03	Rotate CCW
2	End position	absolute	X、Z	Workpiece end position in corrdinate system
		relative	U、W	Distance from beginning to end
3	Distance between start point and center		I、K	I（radius）means X，K means Z axis direction
	Arc radius		R	Arc radius（specified radius）
4	Feeding speed		F	Speed along arc tangent of arc

Range of I,K,R: -99999.999～+99999.999



Use X,Z or U,W to specify the endpoint of arc with incremental or absolute value. Incremental value is the distance from start point to endpoint of arc. Center of arc is defined by I,K, corresponding to X,Z. value behind I,K is the vector component from start point(speed parameter=+4 means from arc endpoint) to center, and is incremental value(I is distance, not radius), I and K contains symbol according to position of center.



Arc center also can be specified by radius R, but if arc is over 180° , R should be minus.

Speed of arc interpolation is Fx speed fixing, and is speed of tool along arc tangent direction.

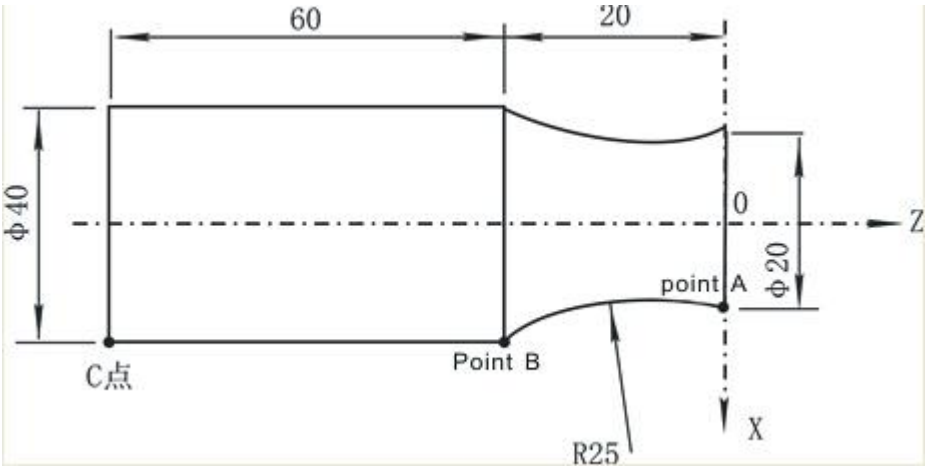
Example：from A to B.

Absolute program: G02 X40 Z-20 I25 K0 ;

Relative program: G02 U20 W-20 I25 K0 ;

R program: G02 X40 Z-20 R25 ;

G02 U20 W-20 R25 。



I0,K0 can be omitted; it I.K,R existed at the same time, R is valid, and I,K are invalid.

Pay attention:

1, Processing arc workpiece usually use ball tool(arc tool) in the actual process, it must use function of tool radius compensation in programming, that's G41 G42 instruction.

2, F is the speed of the tool along the arc cutting direction.

（4）Screw thread (G32)/Polygonal cutting(G809)

The system can process straight thread, taper thread, continuous straight thread, end surface thread, thread of metric or British system, male and female thread, trapezoidal thread.

Straight thread: just specify Z axis direction and length;

Taper thread: must specify X and Z direction and length;

End surface thread: just specify X axis direction and length;

Continuous thread: several pieces of G32;

Trapezoidal thread: offset angle of spindle in the second loop.

Variable pitch thread: successively input G32, specifying a thread pitch in each segment to vary screw pitch K, it will not detect synchronizing signal of encoder at the beginning of the second loop.

Format: G32 Z(W)- X(U)- F(I)- SP- P-

G32 is mode instruction,

X(U) and Z(W) are absolute or relative coordinate of thread end point. If X(U) is omitted, it is straight thread cutting. If Z(W) is omitted, it is end surface thread cutting. If both X(U) and Z(W) are included, it will be taper thread cutting.

SP(Q): is tool angle at the start point, unit is degree, range: 0-360° ; unit of Q is 0.001° . SP defaults as 0.

P: side quantity of polygonal cutting, range:1-12, cutter motor connect with A axis pulse, and set A axis parameter.

F is metric pitch, range is 0.1-1300mm;max pitch=spindle encoder line number/50mm(in the controller with position feedback, namely displacement of long

axis when spindle rotate one revolution. max pitch= spindle encoder line number/10mm)

I is tooth number in British system, range is 1-99 teeth/inch; namely revolution of spindle when long axis move by 1 inch(25.4mm).

During the thread cutting, it will cut on the same route several times from rough cutting to finish cutting. Thread cutting starts from one revolution detection of spindle encoder, so even process cutting for many times, the cutting point on circumference of workpiece is the same one, and route is also the same. Then the speed of spindle is fixed from rough cutting to finishing cutting. If spindle speed varies, it will lead to offset of thread. Besides, at the beginning and end of thread cutting, acceleration and deceleration will cause incorrect section of lead. So instructional thread length should be longer the actual thread length.

Sample1: Straight thread

Thread lead: 4mm

&1=3mm(acceleration section, &1≧3mm).

&2=1.5mm(deceleration section, &2≧1.5mm)

Program according to above data(by twice)

.....

G00 U-62.0；

G32 W-74.5 F4.0；

G00 U62；

W74.5；

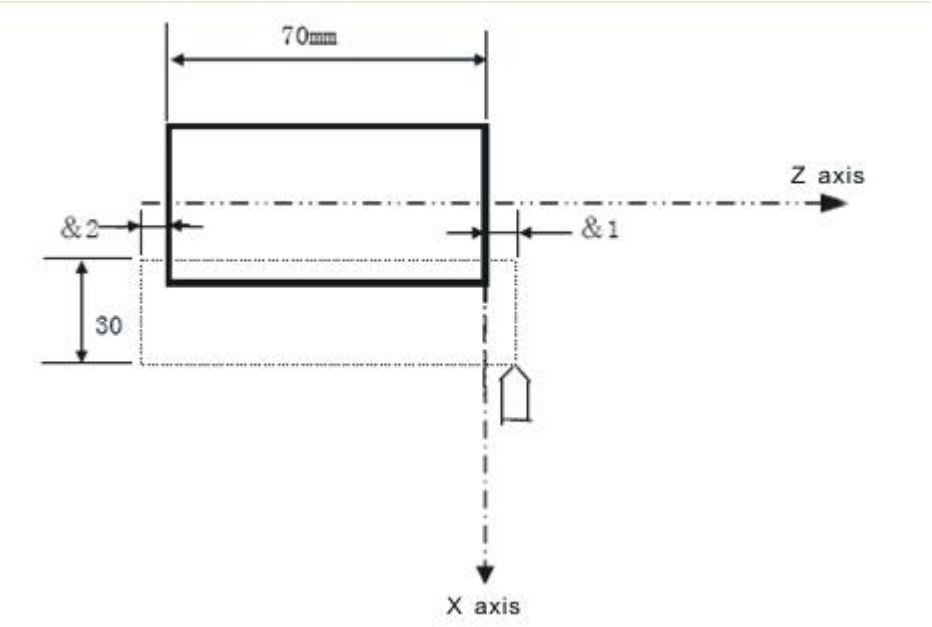
U-64； (the second time cutter further by 1mm)

G32 W-74.5；

G00 U64.0；

W74.5；

.....



Sample 2: taper thread cutting

Taper thread cutting: ad following picture

Thread lead: 3.5mm

&1=3mm, &2=1.5mm

Program according to above data(by twice)

.....

G00 X12 Z3.0；

G32 X41.0 Z-41.5 F3.5；

G00 X50；

Z3；

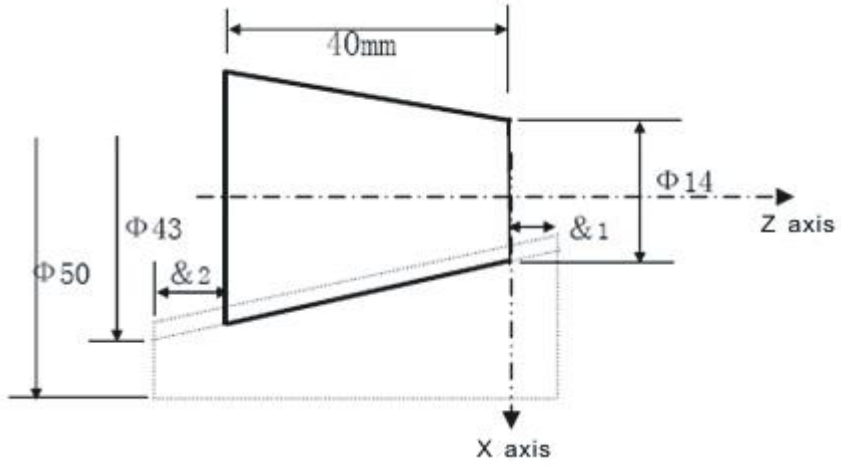
X10；

G32 X39 Z-41.5；

G00 X50；

Z3；

……



If a program segment is for thread cutting, and next segment is so, then it will not detect one revolution signal at the beginning of cutting, and move immediately.

For example:

G32 W-20 F3; detect one-revolution signal at the beginning of cutting;

G32 W-30 F2; not detect one-revolution signal at the beginning of cutting

Pay attention:

1. When the transmission of spindle and encoder not as 1:1, please modify the Axis parameter No.412 No.413;

No. 412 means the number of teeth (requirements: less than or equal to the encoder tooth number, when it is greater than the encoder tooth number, it must match switching board of our company);

No. 413, the encoder tooth number;

2. In the process of cutting screw, F and rate is invalid.

3. In the process of cutting screw, spindle will not stop whatever you do, if the user want to operate suspend,the system will suspend after processing this segment.

Polygonal cutting loop instruction G809(controller must activate A axis and C axis);

Format: G809 X(U)_ Z(W)_ I_ J_ F_ P_
X(U)_ Z(W)_

X(U) and Z(W) refer to distance of tool route; I refers to sides quantity; J refers to C axis speed(r/min); F refers to Z axis moving speed(mm/min); P1 means C axis back to zero but A axis does not back to zero; P2 means A axis back to zero but C axis does not back to zero; P3 means both of A and C axis back to zero; if P is omitted, it means neither A nor C axis don not back to zero.

Attention:

1)X and Z is relative to status of G90/G91.

2)If I is positive value, then A axis rotate CW, otherwise A axis rotate CCW.

3)If J is positive value, then C axis rotate CW, otherwise C axis rotate CCW.

4)C axis and A axis needs to enable electronic gear, the higher rotary speed, the bigger electronic gear(numerator is divided by denominator). Or excessive frequency of pulse from controller will lead to step missing in driver.

5)G809 is mode instruction, so in the next sections of fixed polygonal cutting loop, G809, and I,J,F,P can be omitted, till the mode status of G809 is ended by G80/G00/G01/G02/G03/G32.

6)If C axis is required to home before execution of G809, then execute G800. if A axis is required to home, then execute GOTO instruction.

Steps are as following:

1)according to P instruction: if C axis did not back to zero, then C axis back home firstly(the same as execution of M800). if back home then rapidly locate at zero point;

2)according to P instruction: A axis rapidly locates at zero point.

3)long axis of X and Z axis move in the speed of F, while C axis and A axis move in the specified speed proportionally.

4)Z axis move to coordinate Z(W), C axis and A axis also stop running.

5)Machine coordinate of C axis and A axis execute reduction by 360 degrees.

Polygonal cutting sample:

```
N1 T0101
N2 G00 X200 Z0
N3 X20
N4 M881    ; C axis back home(M61 connects with spindle orientation, M22 connects with orientation in position)
N5 G150    ; A axis back home(M63 connects with A axis EN, M65 connects with INTH, M28 connects with COIN)
N6 G809 Z-100 I4 J400 F200 ;    cutting four faces
N7 Z-105 X30     ; G809 is mode instruction, polygonal chamfering.
N8 G00 X200 Z0 ; cancel G809 mode instruction.
N9 M02
```

（5） Circularity screw thread(G332、G333)

Format：G332/G333 Z(W)_ X(U)_ R_ F(I)_ SP_

Use method refer to G02、G03、G32instruction.

（6）Arc in three-dimensional space(G06)

Format:G06 X_ Y_ Z_ I_ 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
```

（7）delay Instruction(G04)

Require of work process，delays some time before execute other motion。

Format：G04 P_ or X_ or U_

1. P _ unit ms，means delay time。
2. X _ unit S，means delay time, U_ unit S，means delay time。

For example：

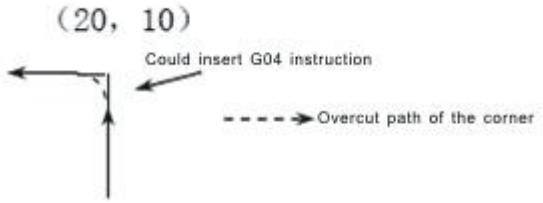
G04 X1； delay 1s.

G04 P1000； delay 1s.

G04 U1； delay 1s.

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；

（8）Return Reference(G28)

Return Reference instruction means tool go to reference point according to appointed axis. If there is the mechanical zero point, it will be reference point of machine; if there is no mechanical zero point, the float zero point can be regarded as reference point of machine. It can back to reference point in manual mode, or by G28 instruction.

format： G28 X(U)_ Z(W)_ Y(V)_; ZXYA return to reference across middle point.

G281 ;only X return to reference point

G282 ;only Y return to reference point

G283 ;only Z return to reference point

G284 ;only A return to reference point

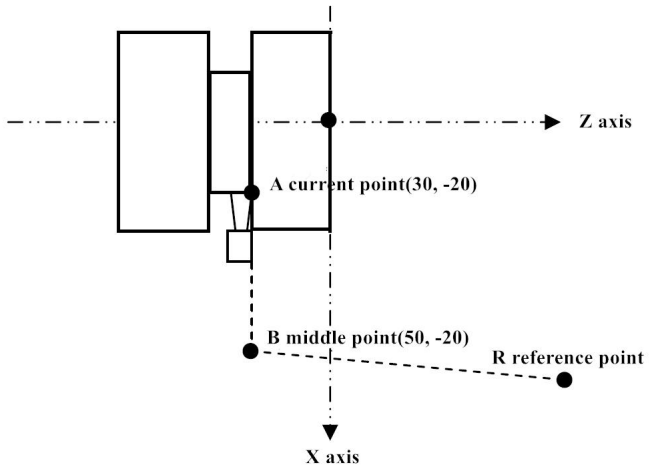
M800 C back to the reference point of encoder, output M75 signal to select controlling position mode when C axis is rotation axis, close M75 signal when system starting M03/M04.

X Z is the middle point of backing to the reference point, use absolute or incremental instruction.

The process:

(1) rapidly locate at middle point from the current point(A→B)

(2) rapidly locate at reference point from the middle point.(A→B)



Example: return from current point to reference point

G28 X50 Z－20；or

G28 U20 W0；

Attention:

1. if power on, never return reference point manually, then under G28, the time back from middle point to reference point will be the same as the time back to reference point manually.

2. when G28 commands to back reference point,if only one axis's middle point is specified, then only the axis back, another one will not.

（9）Setup workpiece coordinate system, max spindle speed of constant linear speed(G50)

Format： G50 X（x） Z（z）； To set workpiece coordinate

Establish a processing coordinate system according to this instruction, the position of current tool in the processing coordinate system is (X,Z), and this coordinate system is defined as workpiece coordinate system. Once coordinate system is established, the position of following absolute instructional coordinate will be express in this coordinate system. The details refers to operation section.

Pay attention:

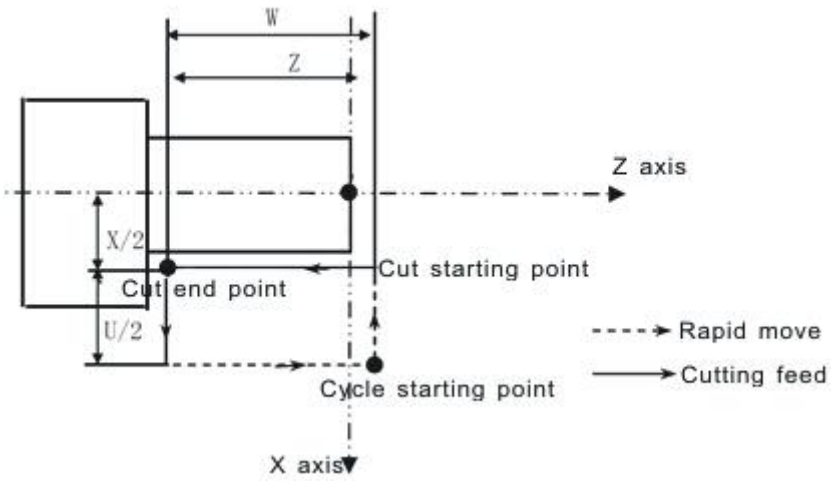
In the status of compensation, if use G50 to set the coordinate, then use G50 to set the position in process coordinate system. Usually cancel the tool compensation first before starting program.Automatically cancel the tool compensation after the system backing to the datum point.

G50 S; Set the maximum speed of spindle when cutting constant linear velocity.

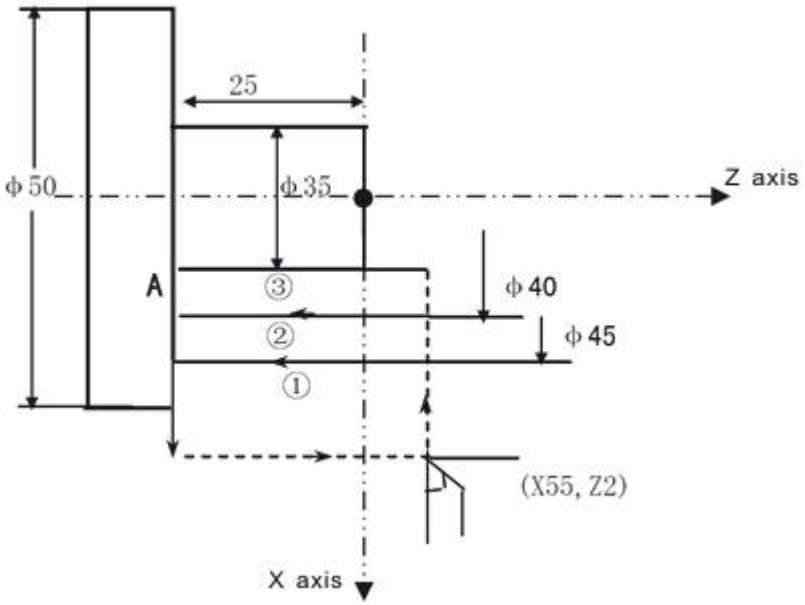
According to this instruction, set the maximum spindle speed of instruction G96 constant line speed in cutting.

（10）Column or taper loop(G90)

Column loop format: G90 X(U)__ Z（W）__ F__；



Column loop cutting



For example:
N10 T0101；
N20 G00 X55 Z4 M03；
N30 G01 Z2 F100 M08；
N40 G90 X45 Z-25；
N50 X40；
N60 X35；
N70 G00 X100 Z100；
N80 T0100 M09；
N90 M05；
N100 M30；

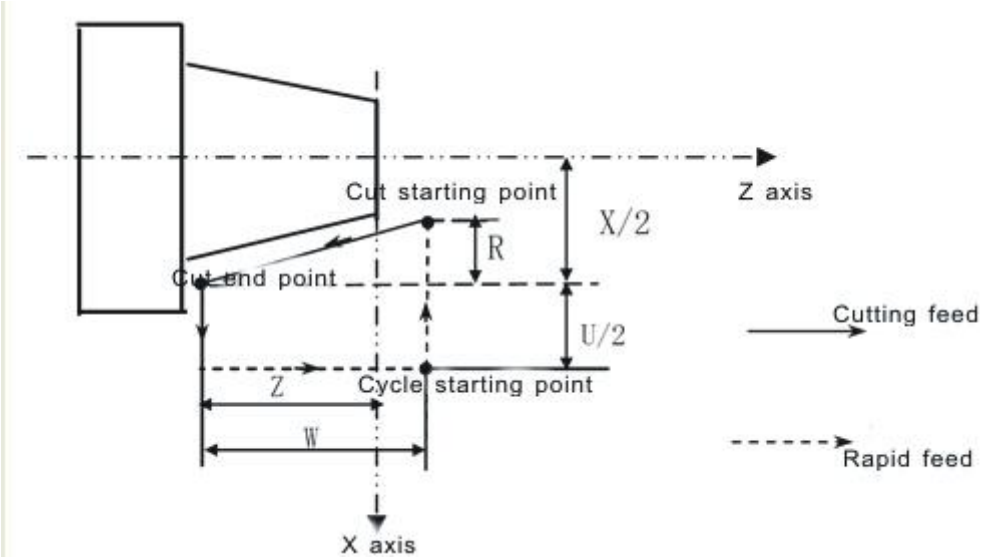
Every cycle is backing to the starting point at the above program, so cause the situation of cutting endface A agin,modify the cycle part program as follow in order to improve efficiency:

N50 G90 X45 Z-25 F100；
N60 G00 X47；
N70 G90 X40 Z-25；
N80 G00 X42；
N90 G90 X35 Z-25；
N100 G00 ；

Taper loop format:

G90 X(U)__ Z（W）__ R__ F__；

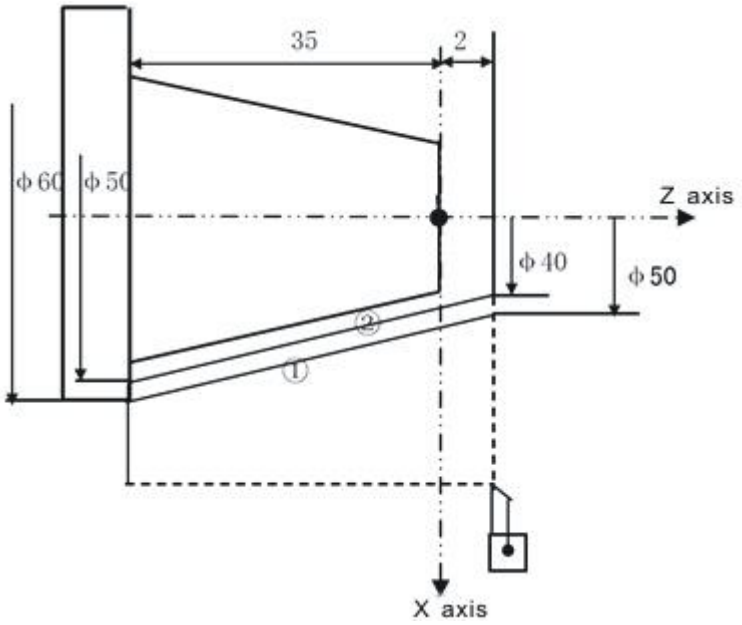
R is radius balance between start point and endpoint of taper. If the coordinate value of start point is less than that of endpoint in the direction of X axis, it will be forward taper with negative R, otherwise it will be backward taper with positive R.



Taper loop cutting

For example:

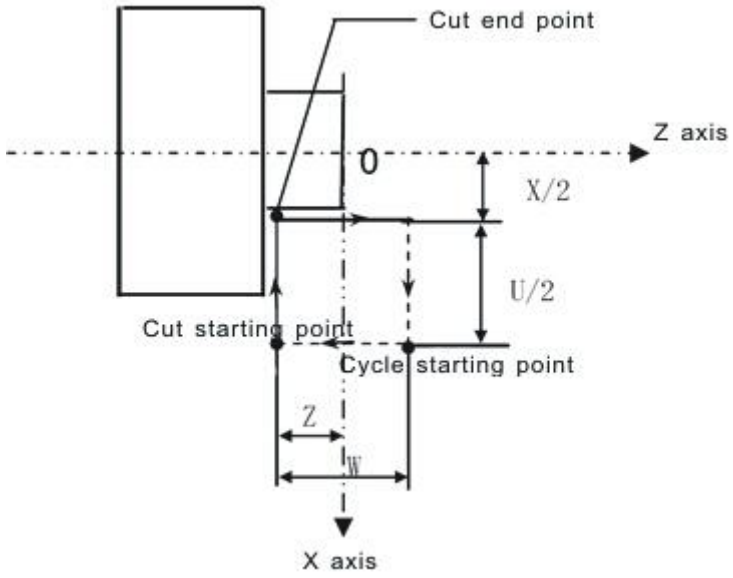
N10 M03 S1000；
N20 T0101；
N30 G00 X65 Z5；
N50 G96 S120；
N60 G99 G01 Z2 F1 M08；
N70 G90 X60 Z-35 R-5 F0.2；
N80 X50；
N90 G00 G98 X100 Z100 M09；
N100 G97 S1000 T0100；
N110 M05；
N120 M30；



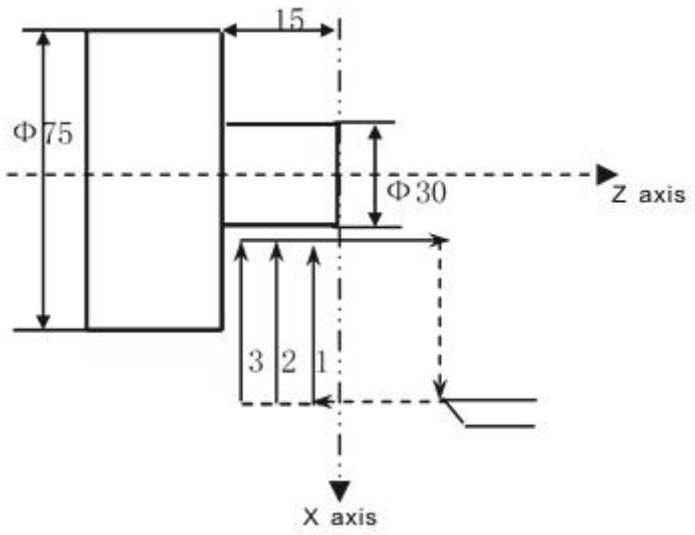
（11）End face loop(G94)

Format：G94 X(U)__ Z（W）__ F__;

The loop is as following picture, X,Z is coordinate value of cutting end point, U,W is coordinate incremental value of endpoint relative to start pint.



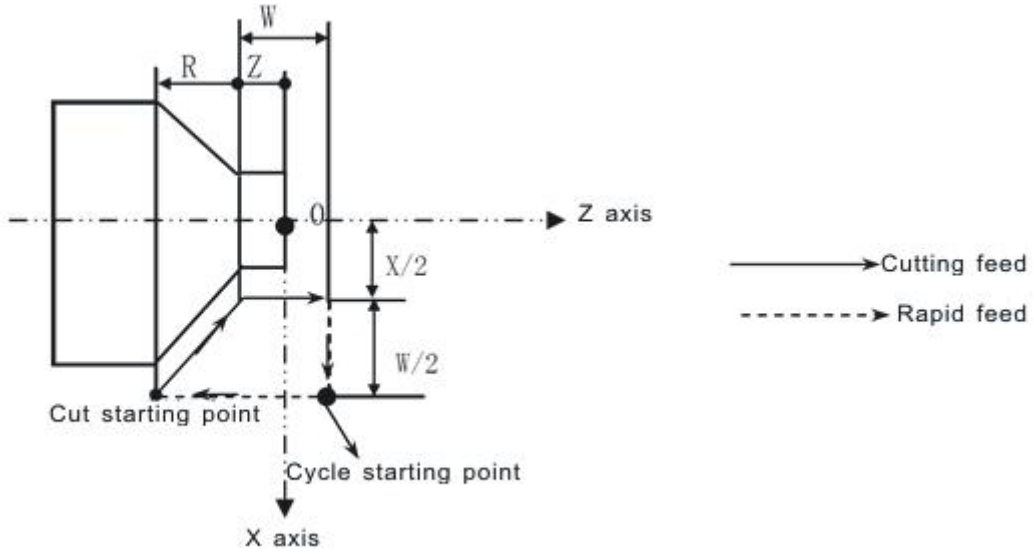
End face loop cutting



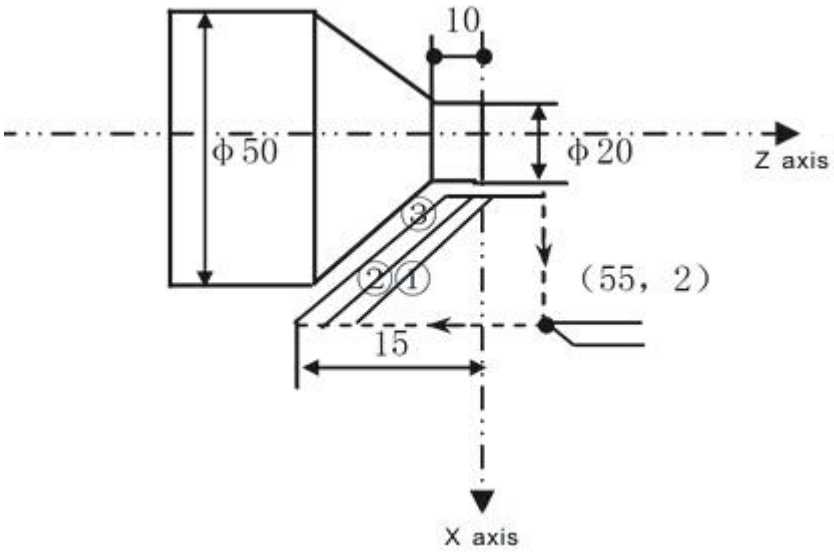
For example：
N10 M03 S1000；
N20 T0101；
N30 G00 X85 Z10 M08；
N40 G01 Z5 F200；
N50 G94 X30 Z-5 F100；
N60 Z－10；
N70 Z－15；
N80 G00 X100 Z60 M09；
N90 T0100 M05；
N100 M30；

Every cycle needs back to starting point at the above program, make the external diameter parts be cut again, it wastes time, so change the cycle part of program as follow in order to improve efficiency:
N50 G94 X30 Z-5 F100；
N60 G00 Z-3；
N70 G94 X30 Z-10；
N80 G00 Z-8；
N90 G94 X30 Z-15；
N100 G00 X100 Z60；

Taper end face loop format: G94 X(U)__ Z（W）__ R__ F__；



Taper end face cutting cycle



For example：

……

N40 G01 X55 Z2 F200；

N50 G94 X20 Z0 R-5 F100；

N60 Z-5；

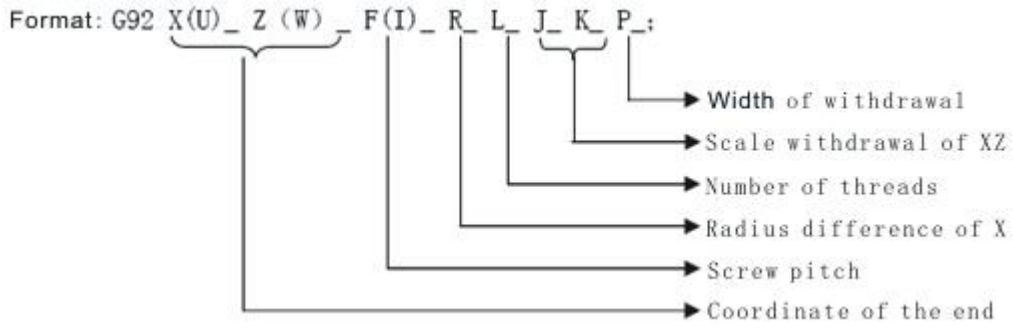
N70 Z-10；

N80 G00 X Z；

……

R-5 in N50 program: R-5=-15-(-10)=-5mm

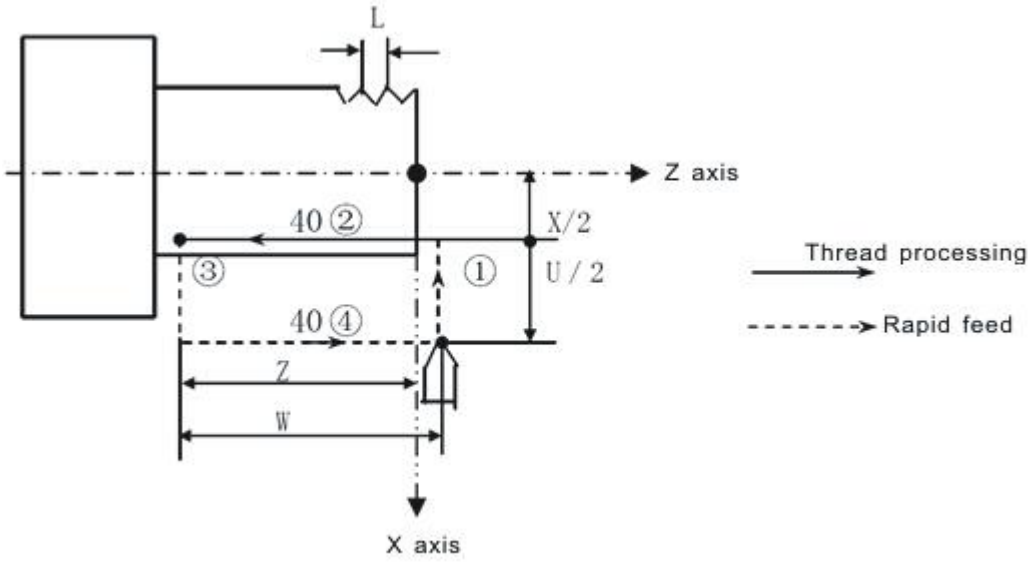
（12）Screw thread loop (G92)



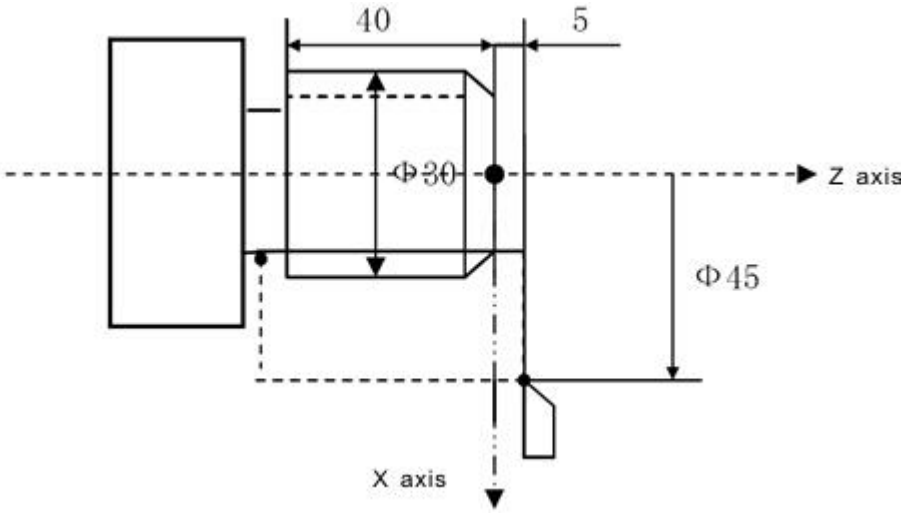
1）Straight screw thread loop format：

G92 X(U)__ Z（W）__ F/I__;

The process id as following picture. X,Z is the coordinate value of screw end pint, U,W is coordinate incremental value of end point relative to start point, dividing into positive and negative. F/I define screw lead L, defined with G32.



Straight screw thread loop

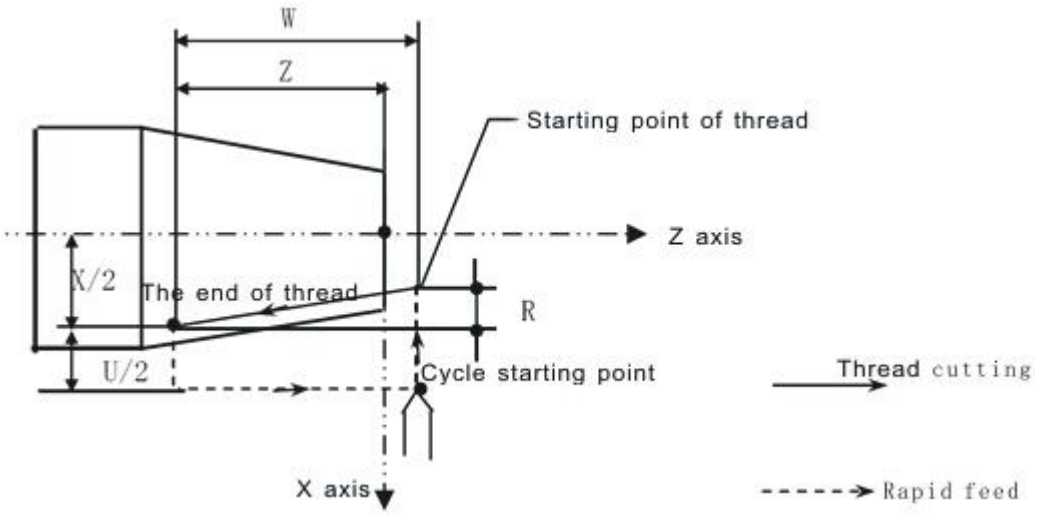


Format:
N10 M03 S××；
N20 T0101；
N30 G00 X45 Z5；
N40 G92 X29.2 Z-45 F1.5；
N50 X28.6；
N60 X28.2；
N70 X28.04；
N80 G00 X100 Z50；
N90 T0100 M05；
N100 M30；

2）Taper screw thread loop format：

G92 X(U)__ Z（W）__ R__ F/I__；

R is radius balance between start point and endpoint of screw cutting in the direction of X axis. If the coordinate value of start point is less than that of endpoint in the direction of X axis, R will be negative, otherwise it will be positive.



Taper screw thread loop

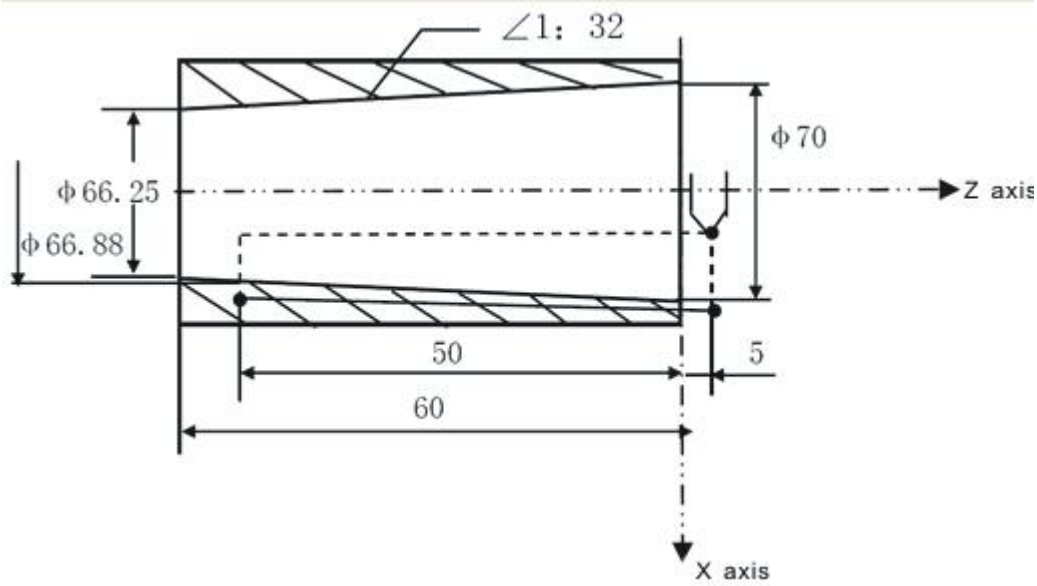
For example:
N10 M03 S××；
N20 T0101；
N30 G00 X55 Z10；
N40 G01 X60 Z5 F100；
N50 G90 X66.25 Z-60 R1.875；
N60 G92 X66.88 Z-50 R1.4 I11；
N70 X66.9 I11；
N80 X67 I11；
N90 X67.4 I11；
N100 X67.6 I11；
N110 X67.8 I11；

N120 G00 X100 Z50；
N130 T0100 M05；
N140 M30；
Pay attention:

1. When the transmission of spindle and encoder not as 1:1, please modify the axis parameter No.412 No.413;
412 means the number of teeth (requirements: less than or equal to the encoder tooth number, when it is greater than the encoder tooth number, it must match keysets of our company);
413, the encoder tooth number;

2.When processing inch thread, pitch I is non-mode,just be effective in one sentence,so every sentence should plus I in thread cycle.

3.Spindle speed S is the speed of the processing thread of pitch Fx, system requires this value should be less than 3000mm/min; The back tail speed of X axis is FxSx which is set by speed parameter No.24, system requires this value is less than 5000mm/min, such as: When processing F2, S1200,this value should be less than 20.



3) Deceleration or acceleration control in cutting thread cycle:

At the end of thread, because of the index of deceleration control,cause the distance of pitch is inhomogeneous,the higher speed of spindle the longer of inhomogeneous pitch. To reduce the error,should reduce the index of deceleration or acceleration time, but it will cause the motor stuck if match the step motor. In order to solve this problem:

● could choose Z axis according to linear acceleration or deceleration speed constant;

● could choose the X axis with the rapid speed G00 to back tail.

The relevant parameter is as follows (see the chapter of parameter):
The speed parameter

No.22 the acceleration or deceleration constant of Z axis in thread processing
No.23 the acceleration or deceleration constant of Y axis in thread processing
No.24 The backing tail speed rate of servo motor in thread cycle
No.25 The starting speed of servo motor in thread cycle
No.26 The maximum backing tail speed of servo motor in thread cycle

4) Multi thread processing function, processing function of offset angle

Using L to specify thread number directly. The SP offset angle processing.
Format: G92 X_ Z_ F_ L_ [or SP];
Note:

● Repeat L times of G92 cycle: L thread number. L is mode, after specifying,it will be effective.

● The range of L:1 ～100. Otherwise, according to the L1 to process (single thread).

● The SP (Q) specifies the cutting angle of start, unit of SP is degree, range: 0-360 °, the unit of Q is 0.001 degree. Could not use SP to specify when processing multi thread.

Such as: L03, 3 thread, continuous executing 3 times G92.The first time, begin processing at once when the spindle start rotating, the second time,after 120 degrees offset, begin cutting thread, the third time, after 240 degrees offset, begin cutting thread,

For example:
G92 X50.Z-100 F5 L5; at X50, process 5 thread.
X48.5; at X48.5, processing 5 thread.
X45; at X45, processing 5 thread.
G00 X100 Z100;
.....

5)Thread backs tail when fixed cycle in cutting thread

Program format:
G92 X Z F/I P;
P: volume of backing tail: the default value of P could be set by No.20 parameter in process parameter(Default when powering on).
Set unit: P1 means 0.1 pitch ; P10 means 1 pitch .
Scope: 1--225, when the set value beyond to the range is invalid.

6) Back tail at any angle function

When cutting thread without backing fuller, the system must have the function of automatical backing tail in thread processing to produce a qualified section of thread. Including the program format of backing tail in thread:
G92 X_ Z_ F_ J_ K_P_;

● J, K set the ratio of back tail X, Z. When J2 K1, X is twice faster than Z.

● P: back tail volume. Setting: 0.1 pitch. Set range: 1~255 (beyond to this range is invalid). The default value can be set by No.20 parameter in process parameter (Default when powering on).

● J, K, P are mode value.

● When executing J0 or K0 in G92, cancel any angle specify, fixed 45 degrees.The

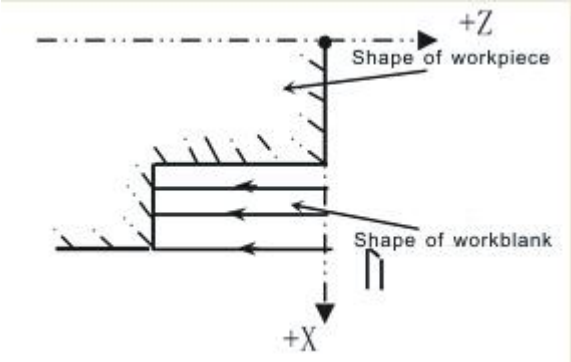
default value is 45 degrees when powering on.

● When J K are set to be negative number, or beyond to 65535,it's invalid setting. The range: 1~65535.

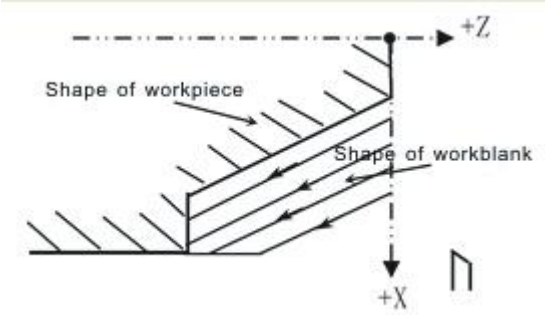
(13)Usage for fixed cycle

Could according to the shape of the workpiece and workblank to choose the suitable fixed cycle.As follows:

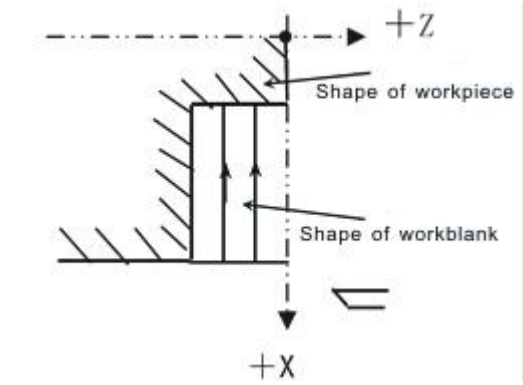
1) Cutting cylindrical cycle G90:



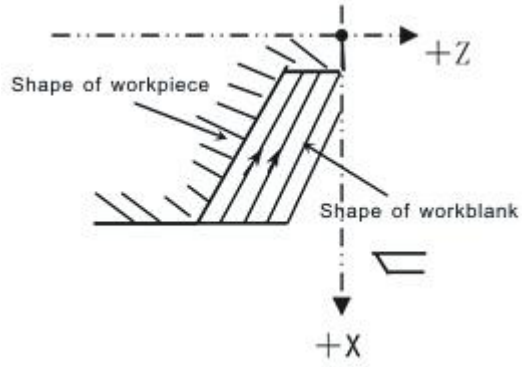
2) Taper cutting cycle G90:



3) Cutting cycle of the end face G94:



4）Taper cutting cycle of the end face G94:



Attention 1: The data X(U)、Z（W）、R are all mode value in fixed cycle of G90 G92 G94, the front data is always effective when there is not new specified X(U)、Z（W）、R, except the screw pitch I in inch thread process. Program instruction G90, G92, G94 has been always effective, could use the G code G00, G01 instruction to cancel.

Attention 2: The following three kinds of situation;

①When there is just no moving instruction program behind the fixed cycle,so repeat the fixed cycle.

②It just only use start button to run the program when using MDI.

③In the fixed cycle, if there is M、S、T ,then use function of fixed cycle to process together. If cancel the fixed cycle after M、S、T(Because of G00 G01), please fixed cycle again.

Example:

N10 T0101;

……

N50 G90 X20 Z10 F100;

N60 G00 T0202;

N70 G90 X20.5 Z10;

……

(14)Fixed cycle of tapping (G93).

G93 Z (W) F/I;

Tapping has two kinds of method: Tracking the spindle encoder (P411=0, spindle must match encoder), spindle servo interpolation of Z axis and spindle (P405=0, P410=92, P411=4). Through the No.404-No.411 parameter in axis parameter. The execution is as follows:

● The first is the same as G32,Z axis feed negative according to the thread cutting.

● Move to the coordinate which is specified by program, stop spindle automatically, after spindle stop completely, the spindle rotates negative automatically according to specification, Z axis returns to the starting position.

● Spindle stop rotating, recover to the direction of spindle rotating in front of the program.

Note:

● The cycle is the same as G90, G93 is a mode code. So G93 should specify G01 or G00 etc..

G93 Z-100 F5; tapping cycle to Z-100;
Z-101; tapping cycle to Z-101;
G00 X50; G00
Tapping method in X axis:
1), No.41 parameter in speed parameter "mode of arc interpolation (0 means A; 8 means B)" is set to 2;
2), Enter G19 into the program segment of X tapping,enter G17 into the program segment of Z tapping, enter G18 into the next program segment after finish tapping.
For example: G93 G19 X-100 F2
G93 G17 Z-100 F2
G18 G0 X30
Pay attention:

● If execute G93 after Z moving positive, the system should execute interpolation negative first because of negative. Should set the parameter of interpolation. If the step motor is stuck , should set the smaller frequency value of interpolate compensation. Or enter the instruction to move the Z axis negative first before executing G93.

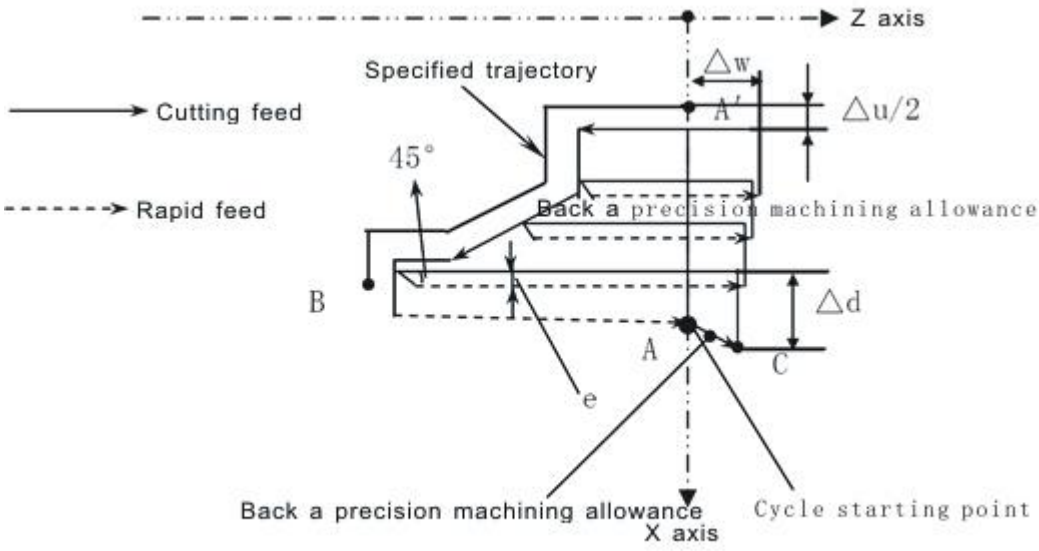
● The parameter of the spindle breaking time is affect the start rotating time after stop. Please pay attention to the setting:

Note 1: Z must move negative.
Note 2: Must start spindle rotating before executing G93.
Note 3: Require the spindle breaking time of lathe is short.
Note 4: Require the rotating speed of spindle not to high.
Note 5: For specifying inch thread when specifying I is the same as G32 G92.
Note 6: When choosing the acceleration and deceleration control, if the spindle speed change, there is some delay when making the thread change. So choose the non-speed up or down if require the accuracy. However, with a step motor, the spindle speed can not be too high, otherwise cause the stuck.

（15）Column roghf cutting loop (G71)

When use G90, G92, G94, the program is simplified, and there is also some code is defined as compound loop code(G70-G76), simplifying program further. When use these compound loop code, just need to specify the shape of processing, it will perform the full process from rough to finishing.

When command the following shape in the picture, and route A→A'→B, depthΔd, controller will do multicutting parallel with Z axis, finally do finishing cutting according to working allowance Δw and Δu/2.



Column thick loop

Format：

G71 U（△d） R(e);

G71 P（ns） Q（nf） U（△u） W（△w） F（f） S（s） T（t）;

△d: feed thickness，no sign; User parameter P1. mode value;

e：backward distance; mode value, User parameter P2.

I: 1 means start from end face of workpiece. 2 means start from the middle of workpiece.

ns：first N line.

nf：end N line.

△u：finishing process allowance in the direction of X axis; User parameter P4. input negative value if inner hole.

△w：finishing process allowance in the direction of Z axis; User parameter P5.

Note 1: G71 in the use of rough machining cycle, only F S T in the program G71 function is effectively, but F S T in program of ns→nf is effectively to fine machining, invalid in rough machining cycle.

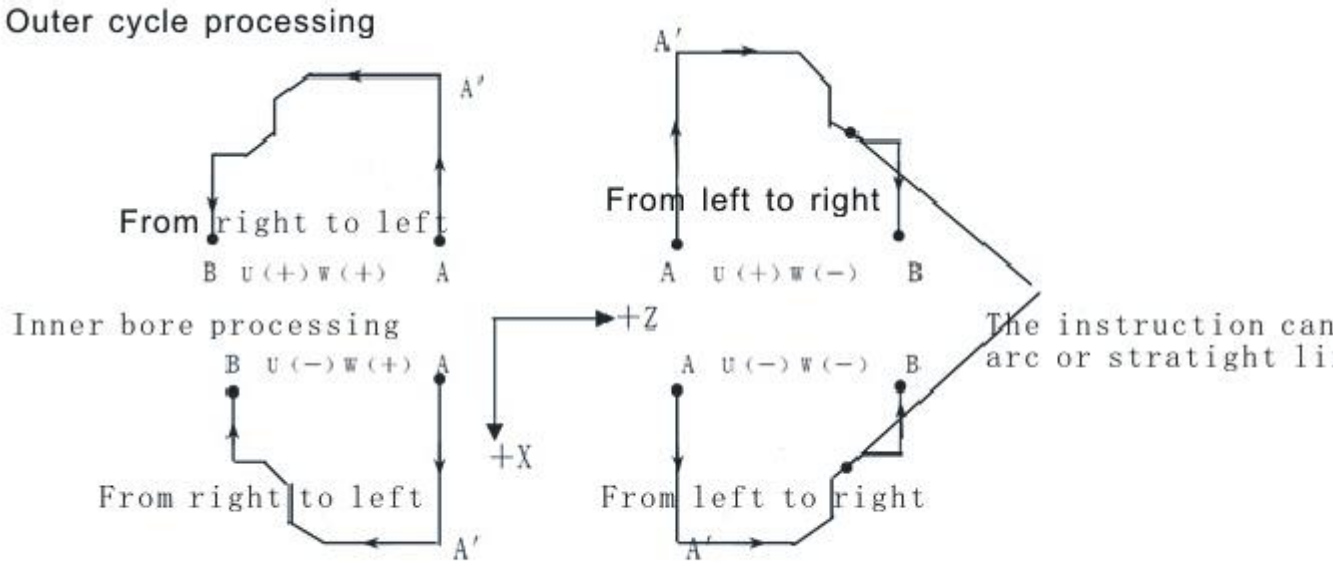
Note 2: A—B must conform the increase or decrease together mode of X and Z axis.

Note 3: G97 G96 is invalid for rough machining cycle when the program ns→nf includes choose function of constant linear velocity, G96 G97 in the G71 or front program segment is effective for rough machining cycle.

Note 4: from A to A ', maybe G00 G01 is in the NS program segment, but not containing Z axis movement instruction.

Note 5: In program segment of NS to NF, can't call subprogram.

Note 6: Use G71 to cut shape, four method as follows, the four mode is cutting according to the tool which is parallel Z axis, △ u, △ w finishing machining allowance, as shown below:



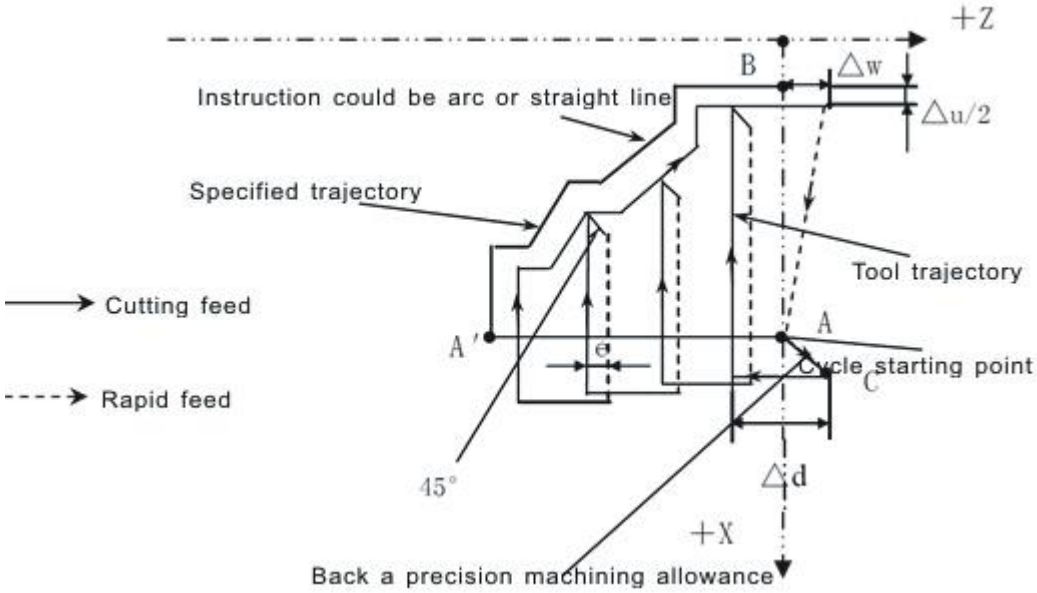
(16) End face thick loop(G72)

G72 and G71 are rough machining cycle instruction, but G72 is along the direction which is parallel X axis to process cutting cycle,format:

G72 W (△ d) R (e);

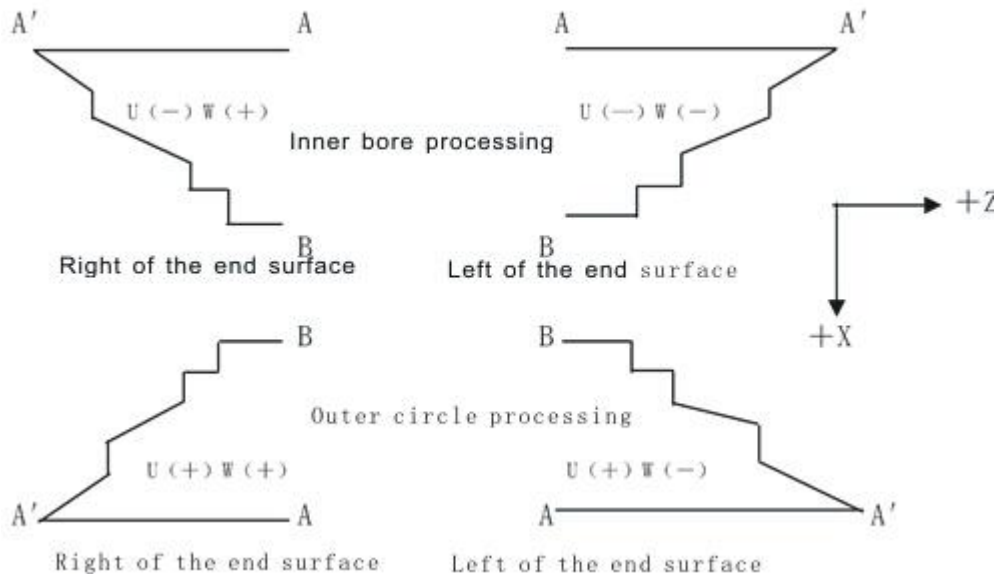
G72 P（ns） Q（ns） U（△u） W（△w） F（f） S（s） T（t）;

The parameters △d、e、ns、nf、△u、△w、f、s、t and G71 have the same meaning



Rough machining cycle of end surface

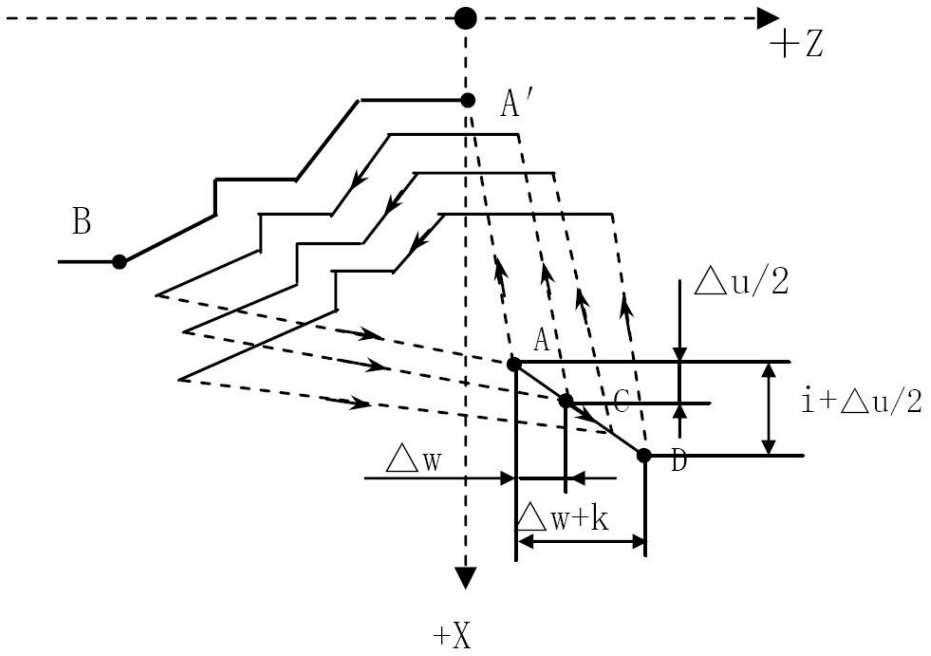
Use G72 to cut the shape, there are four situation. No matter what kind of is the tool parallel the X axis to cut again. △ u, △ w symbols are as follow



Between A and A ', the program segment ns could contain G00 or G01, but can not contain X axis instruction, between A ' to B, the X and Z axis must be all increase or decrease graph, that's a direction to increase or decrease.

（17）Close cutting cycle(G73)

The closed cutting cycle according to the cutting shape to be close to final shape. This way is high effective to cast or cut workblank. G73 cycle is shown as follow:



Format:

G73 U（i）__ W（k）__ R（d）__;

G73 P（ns）__ Q（nf）__ U（△u）__ W（△w）__ F（f）__ S（s）__ T（t）__;

N（ns）……; --\
……; |
. > route of A→A′→B finishing machining, segment from ns to nf to instruct.
. |
. |
. |
N（nf）……; --/

i：distance and direction of backing tool in the direction of X axis; mode instruction; User parameter P7.

k：distance and direction of backing tool in the direction of Z axis; mode instruction; User parameter P8.

d：cutting times; mode instruction; User parameter P6.

Other parameters are same as G71. difference of G73 from G71, G72 are F,S,T only effective in rough machining.

Note 1: The cycle is according to the program which is between P and Q in G73.The tool backs to A point automatically after finish cycle.

Note 2: Increase or decrease X or Z axis is invalid when using G73.

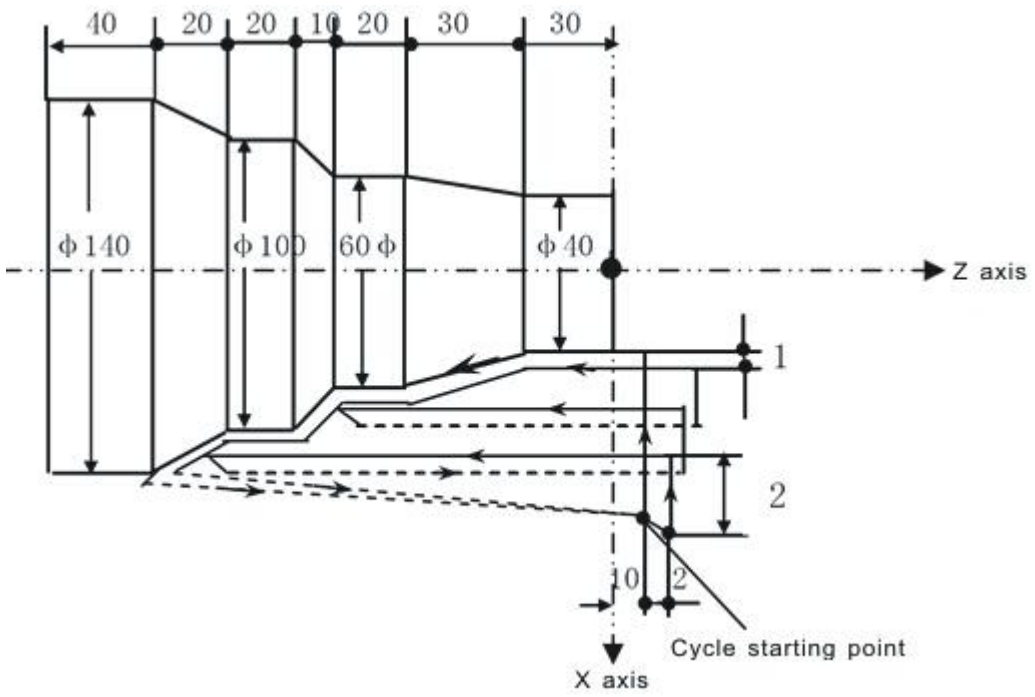
（18）Finishing machining cycle (G70)

Could use G70 to fine machining after using G71 G72 G73 to finish rough machining.

Program format:

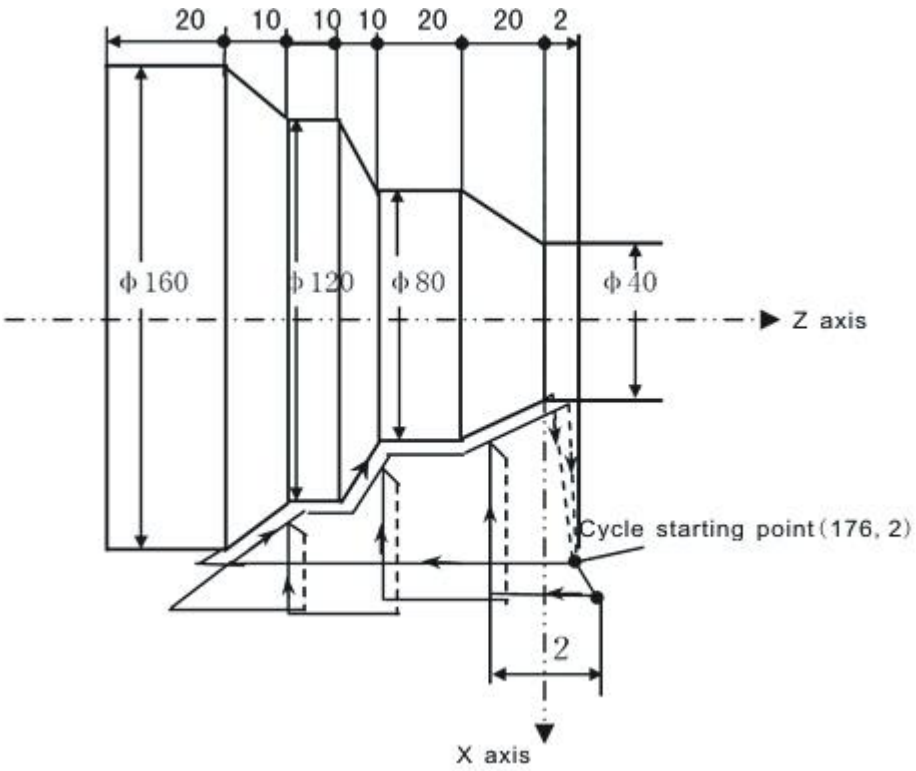
G70 P (ns) Q (nf) ns and nf are the same as the above.

F、S、T are all invalid in G71 G72 G73 when fine machining, F、S、T are effective only in ns→nf.Tool use rapid feeding to back to starting point when G70 is over and start read the next program segment of G70 cycle.

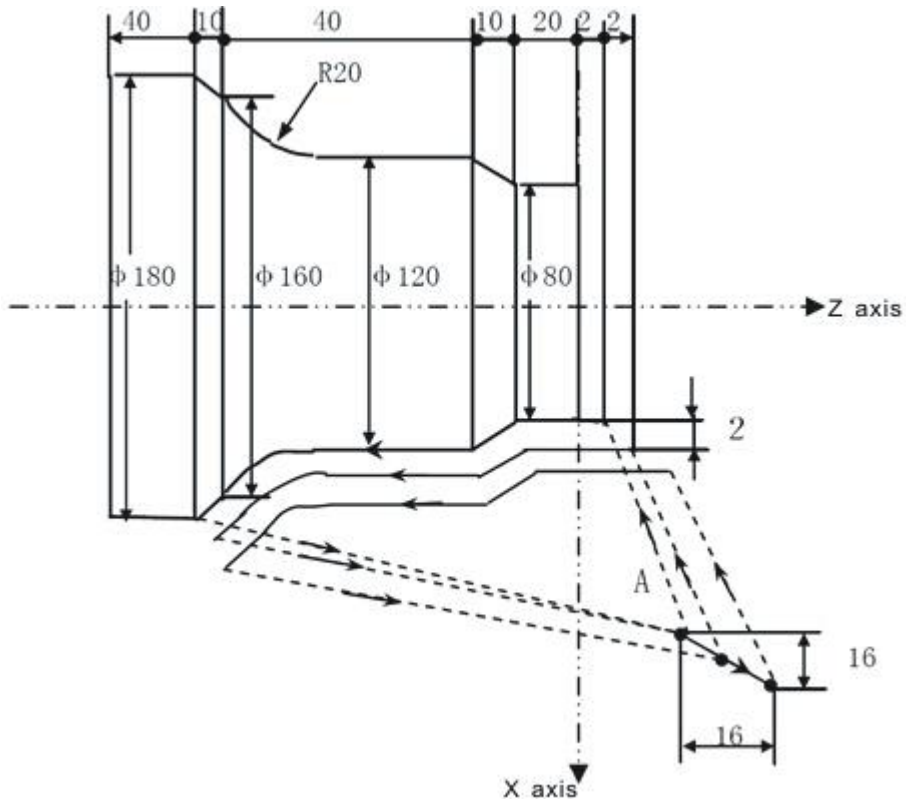


N10 M03 S1500；
N20 T0101；
N30 G00 X160 Z10；
N40 G71 U2 R1； in rough machining loop, cut 2mm and back 1mm each time single side in the direction of X axis.
N50 G71 P60 Q120 U2 W1 F100 S2000; the final route of rough machining is from N60-N120. and leave finishing machining allowance X axis 2mm, Z axis 1mm.
N60 G00 X40;
N70 G01 Z-30 F80；
N80 X60 W-30；
N90 W-20；
N100 X100 W-10；
N110 W-20；
N120 X140 W-20；
N130 G70 P60 Q120； specify route of finishing cutting.
N140 G00 X200 Z50；
N150 T0100 M05；
N160 M30；

For example2：G72 G70



N10 M03 S2000；
N20 T0202；
N30 G00 X176 Z2；
N40 G72 W2 R1； in rough machining loop, cut 2mm and back 1mm each time
N50 G72 P60 Q120 U2 W1 F100 ； the final route of rough machining is from N60-N120. and leave finishing machining allowance X axis 2mm, Z axis 1mm.
N60 G00 Z-72；
N70 G01 X160 Z-70 F80；
N80 X120 W10；
N90 W10；
N100 X80 W10；
N110 W20；
N120 X36 W22.08；
N130 G70 P60 Q120；
N140 G00 X200 Z50； specify route of finishing cutting.
N150 T0200 M05；
N160 M30；
For example3: G73 G70

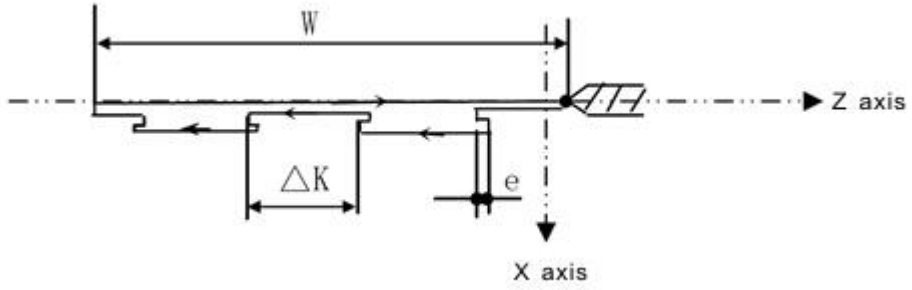


N10 M03 S3000;
N20 T0303;
N30 G00 X220 Z40;
N40 G73 U14 W14 R0.010; rough cutting allowance: X axis 14mm, Z axis 14mm, 10 times.
N50 G73 P60 Q110 U 4 W2 F100; specify rough cutting route according to N60-N110, leaving finishing cutting allowance: X axis 4mm(diameter), Z axis 2mm;
N60 G00 X80 Z2;
N70 G01 Z-20 F80;
N80 X120 W-10;
N90 W-20;
N100 G02 X160 W-20 R20;
N110 G01 X180 W-10;
N120 G70 P60 Q110; specify route of finishing cutting.
N130 G00 X250 Z50;
N140 T0300 M05;
N150 M30;

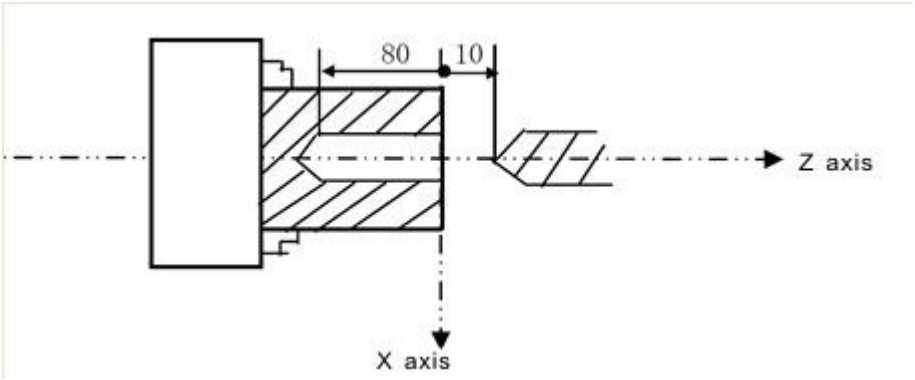
（19）End face deep hole loop(G74)

Format:
G74 R（e）;
G74 X（u）P（△i）Z（w） Q（△k） F（f）;

e: backward distance;User parameter P10.
Z（w） Z depth;
X（u） X end-point coordinate;
Δk： Z feed thickness;User parameter P9.
Δi： X feed thickness.

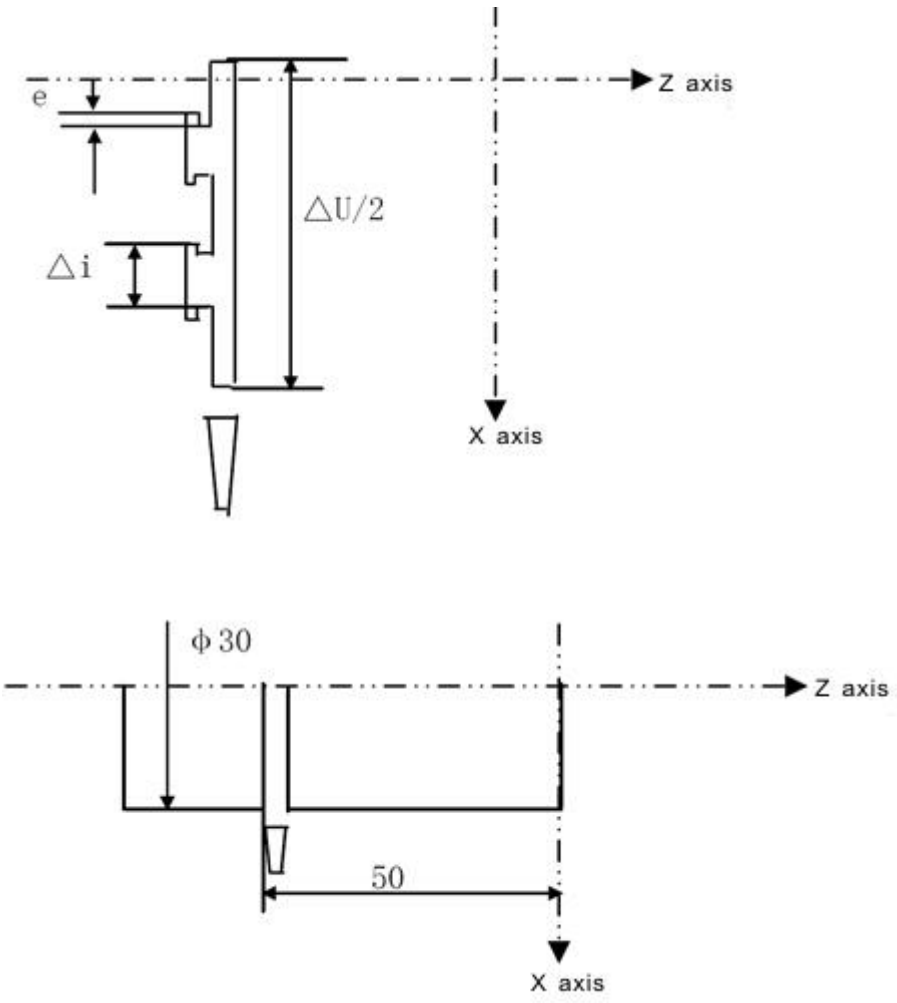


For example:
N10 G00 X0 Z10；
N20 G74 R2；
N30 G74 Z-80 Q10000 F800；
N40 G00 X50 Z50；
N50 M30；



（20）Slot cutting loop(G75)

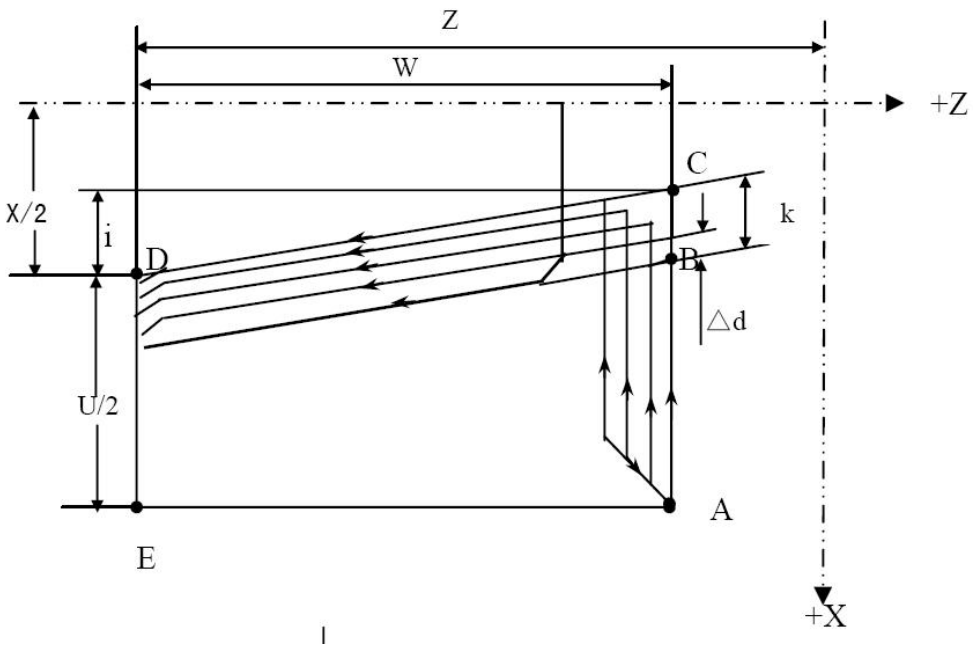
Format：
G75 R(e)__;
G75 X(U)__ P(Δi)__ Z（w）__ Q（△k）__F(f)__;
e: backward distance;User parameter P10.
X（u） X depth;
Z（w） Z end-point coordinate;
Δi： X feed thickness;User parameter P9.
Δk： Z feed thickness.



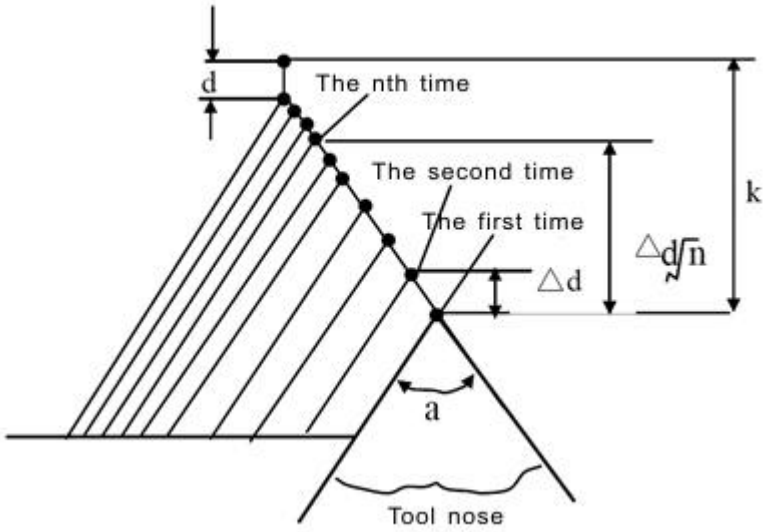
For example:
N10 M03 S1000;
N20 T0101;
N30 G00 X35 Z-50;
N40 G75 R1;
N50 G75 X-1 P5000 F60;
N60 G00 X100 Z50 M09;
N70 M05;
N80 T0100;
N90 M30;

（21）Complex screw thread cycle(G76)

According to following program, the screw cutting is as picture：



Cutting method:



Format:

G76 P(b)(c)(m)(r)(a) Q(Δd min) R(d) ;

G76 X(U) Z(W) R(i) P(k) Q(Δd) F(f) L(L)[or SP];

b：0———degression feed；

1———equidistance feed；

2———If the fist feed is too long in degression feed,so divide into two feed.

c：0———right enter；

1———left enter；

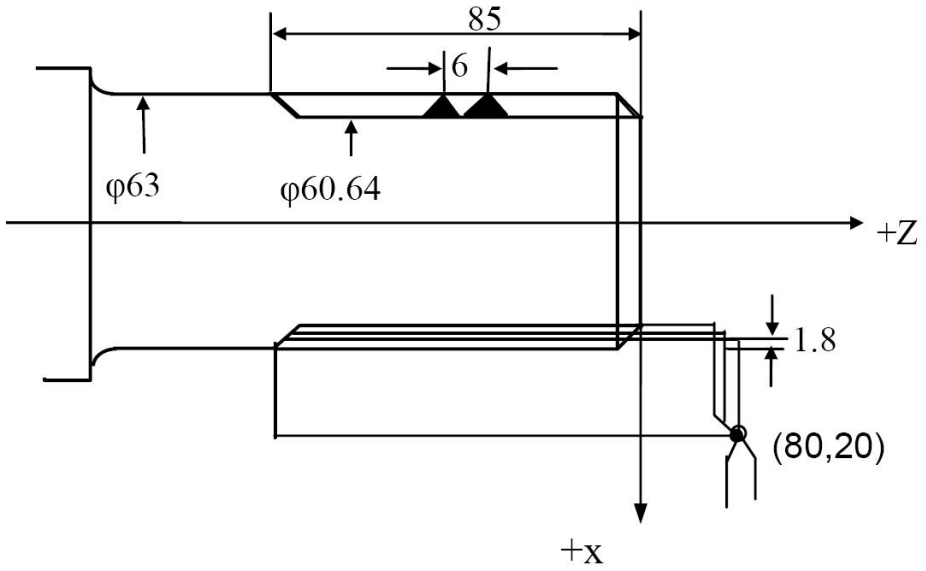
2———middle enter

3———right and left enter, the first feed is middle.

m: finish turn times,User parameter P11.

r: quit length,User parameter P12.

a: thread tooth angle(degree),User parameter P13.
Δd min: minimal cutting depth,User parameter P14.
d: finish turn remaining,User parameter P15.
I：X taper screw thread feed measure.
f: metric lead.
L: multiple thread head numbers.
SP: start angle：0-360°
Compound fixed loop G76:




```
N10 M03 S1000;
N20 T0101;
N30 G00 X80 Z20;
N40 G76 P00011060 Q100 R0.1 ;
N50 G76 X60.64 Z-85 P3680 Q1800 F6.0 ;
N60 G00 X100 Z50;
N70 T0100;
N80 M05;
N90 M30;
```

Note 1:pay attention to cut thread, use G32 to cut is the same as using G92.
Note 2: specify the chamfering amount of thread, it's also effective to the G92 thread cutting circle.
Note 3: when matching step motor, because of the acceleration or deceleration the thread in tail will be inhomogeneous. So should choose the linear acceleration or deceleration to control X axis with G00 to back tail fast.
Pay attention to compound type of fixed cycle (G70 ~ G76):

(a) In the specified compound type of fixed cycle program segment, P, Q, X, Z, U, W, R and others necessary parameter must be specify directly in each program segment.

(b) In the program segment of G71, G72, G73 , if there is a P specify the sequence number, then the program segment corresponding to the sequence number must be the G code of G00 or G01.

(c) In the MDI mode, can not execute G70, G71, G72, G73 instruction. G74, G75, G76 can be executed.

(d) in the program segment of G70, G71, G72, G73 and the program segment between P and Q,could not specify M98/M99.

(e) In the program segment of G70, G71, G72, G73, can not have the following instruction within the program segment which is specified by P and Q.

★ except the G04 (pause) one-time code
★ except the G00, G01, G02, G03 code
★ M98/M99

(f) in the implementation of compound fixed cycle (G70 ~ G76, can make the action to stop inserting manual movement, but to start the execution of compound fixed cycle again, must return to insert a manual before motion position. If you do not return again, the mobile manual is not in the absolute value, the action of dislocation, and its value is equal to the quantity of mobile manual.

(g) G70, G71, G72, G73, P, sequence number specified in Q, not be coincident in this program.

(h) In G70, G71, G72, G73, use P Q to specify the program segment of fine machining shape, the last motion instruction could not be chamfer or transition round.

(22) Program loop(G22、G800)

G22 is a cycle instruction of program, G800 is the end of the cycle instruction. The two must be used together, be used for the situation of repeat process. L is the cycle times, the range is 1-99999. The cycle instruction can be nested.

Format：G22 L-

.
. Loop segment;
.

G800 ;End

For example:

```
N0000  M03 M08
N0001  G0  X200 Z200
N0002  G01 W-100 F300
N0003  G22  L6            ；loop 6 times
N0004  G01  U-22  F100
N0005  W－11  U6
N0006  W-30
N0007  W-10 U5
N0008  G0  U10
N0009  W51
N0010  G800               ；loop end
N0011  G26
N0012  M30
```

(23) Part coordinate setup(G52),Setup or offset tool coordinate

（G184、G185）

When programming in the workpiece coordinate system, to set sub coordinate system in order to programme.

Format:

G52 X_Z_; absolute coordinate setup

G52 U_W_ ;relative coordinate setup

G52 X0Z0 ; cancel

G184 X_Z_P_L_;setup current&P&L tool absolute coordinate

G184 U_W_P_L_;setup current&P&L tool relative coordinate

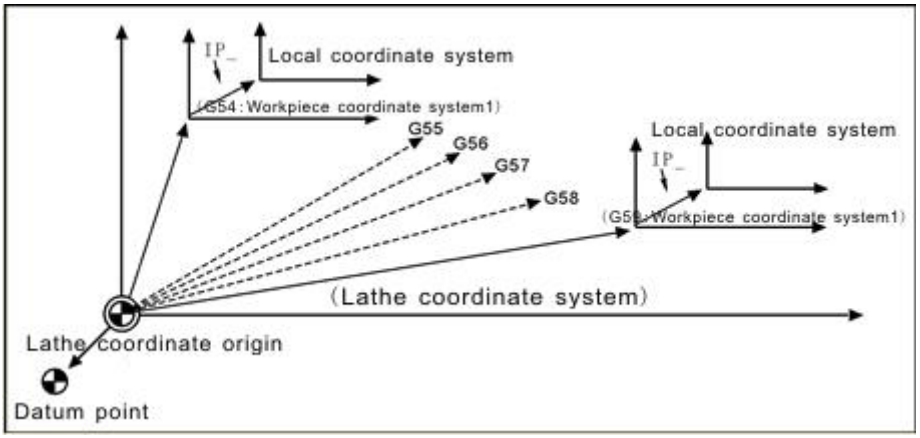
G185 X_Z_; setup all tool absolute coordinate

G185 U_W_ ; setup all tool relative coordinate

Use G52 to set sub coordinate system in lathe coordinate system and workpiece coordinate system. Set the origin of sub coordinate system in lathe coordinate system or use X- Z- to specify position in workpiece coordinate system.

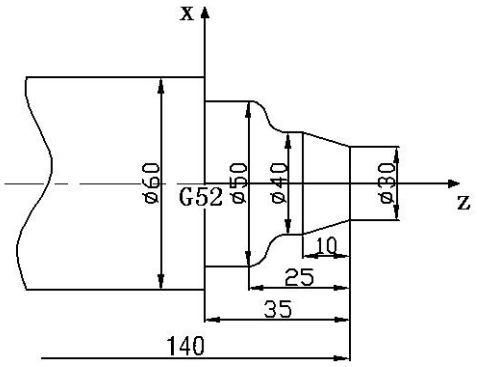
When setting local coordinate system, the motion with absolute mode is the coordinate value of local coordinate system. Use G52 to specify a new zero point which could change the position of local coordinate system. In order to cancel the local coordinate system and the specified coordinates in the workpiece coordinate system, should make the local coordinate system

Local coordinate system setting does not change the workpiece coordinate system and lathe coordinate system. When setting the workpiece coordinate system, if not specify all axis coordinate value, not to specify the coordinate value of the axis of the local coordinate system will not cancel but be remain unchanged. After the G52 program segment should specify motion instruction with absolute mode.



Part coordinate setup

For example：



N1 G00 X60 Z20
N2 G52 X0 Z-236 set part coordinate system;
N3 T0101 select tool number;
N4 M03 S800 M08 spindle rotate CW by 800rpm and coolant.
N5 G01 Z35 F100 move to end face Z35;
N6 X-1 cutting end face;
N7 X70 back tool;
N8 G71 U2 R1
N8 G71 P10 Q15 U0.5 W0.5 F150; excircle compound rough cutting loop;
N10 G01 X30
N11 X40 Z25
N12 Z20
N13 G02 X50 Z15 R5
N14 G03 X60 Z10 R5
N15 G01 Z0
N16 G00 X70 back tool rapidly;
N17 G52 X0 Z0 cancel part coordinate system;
N18 M05 stop spindle;
N19 M30 end program.

Example: processing five time on one workpiece.

G22 L5
X50 Z60 ； process workpiece;
.....
G52 W-50 ； Z axis offset by 50mm each time;
G800
G52 X0 Z0； cancel part coordinate system;
G0 X100 Z100
M02

（24）Back to starting point(G26、G261、G262、G263、G264)

Format：
G26、G261、G262、G263、G264; XZ、X、Y、Z、A
Example:
N0000 G00 X120 Z300 ; move to workpiece face rapidly;
N0001 G01 X150 Z50 F160 ； feed to X150, Z50;

N0002 G26 ; X and Z back to start point simultaneously
N0003 M2 ; program end.

(25) Back to G25 point(G25、G61、G611、G612、G613、G614)

G25 remember the current X Z coordinate, G61 returns X Z coordinate point of G25, G611 returns X, G612 returns Y, G613 returns Z, G614 returns A, G25 is the same as G26 if not set.

Format:
G25 ;Save current coordinate
G61、G611、G612、G613、G614; XZ, X,Y,Z,A
For example:
N0000 G0 X20 Z80 ; rapidly move to X20, Z80.
N0001 G01 U5 W－16 F200 ; G01 interpolation, speed is 200.
N0002 W－100 ; cutting excircle, Z is 100.
N0003 G00 U10 ; X tool back by 10, to X35.
N0004 Z80 ; Z back to 80.
N0005 G25 ; memory coordinate X35, Z80.
N0006 G01 U10 W-30 ; processing to X45, Z50.
N0007 G0 X100 Z200 ; rapidly move to X100 Z200.
N0008 G61 ; back to X35, Z80.
N0009 M2 ; program end.

（26）Continue feed cutting(G60/G64)

Format：G60 ; cancel (mode)
G64 ; continue feed(mode, initial)

(27）Constant speed cutting(G96/G97)

Format：G96 S_ Constant speed cutting
G97 cancel

In the status of constant linear speed G96, S_ address word means linear speed (M / min)

After the cancellation of constant linear speed G97, S_ address word means spindle speed (R/M)

Press "absolute coordinate" to change the spindle speed in the status of constant linear speed G96.

Note:

G96: constant linear speed effective

G97: cancel the function of constant linear speed

G50 S: limit the maximum speed of spindle.

S:S value behind the G96 is the cutting constant linear speed, unit is m/min; S value behind the G97 is specifying spindle speed after cancel the constant linear speed; such as the default, is for the implementation of G96 Directive before spindle speed.

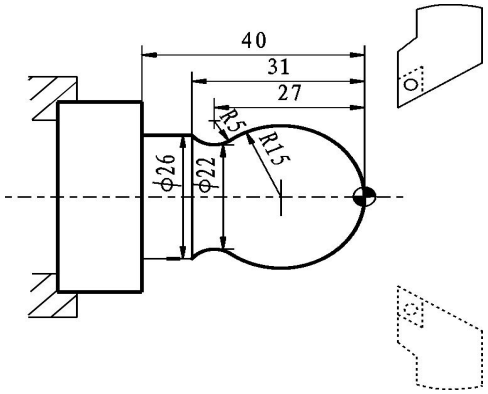
Note 1, use constant linear speed, the spindle must be able to change speed automatically. (such as: servo spindle, variable frequency spindle) set maximum

limit speed in the system parameter.

Note 2, under control of the constant linear speed, the lower limit value of spindle speed in G96 can be set by No.35 parameter in speed parameter.

Note 3, spindle override couldn't work when processing in constant linear speed.

For example:



N1 T0102 X40 Z5

N2 M03 S400

N3 G96 S80

N4 G00 X0

N5 G01 Z0 F60

N6 G03 U24 W-24 R15

N7 G02 X26 Z-31 R5

N8 G01 Z-40

N9 X40 Z5

N10 G97 S300

N11 M30

（28）Feed mode(G98、G99)

Format:

G98 feed method per minute instruction of G01, G02, G03. range: 0.01-15000mm/min;

G99 feed method per revolution instructiono f G01, G02, G03. range: 0.01-15000mm/r;

When use G99, the spindle need to be equipped with encoder, or it will be standby state. And if spindle speed is slow(10r/min), processing may be not balanced.

（29）Check skip(G31、G311)

Format：G31 X_ Z_ F_ P_ ;No alarm

G311 X_ Z_ 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. If X21 is

valid, then keeping alarm.

Note: input points X00-X39 can be checked in diagnosis interface.

(30）Work coordinate(G53/G54/G55/G56/G57/G58/G59)

Format：G53(G54/G55/G56/G57/G58/G59)

G53 machine coordinate

G54 workpiece coordinate1

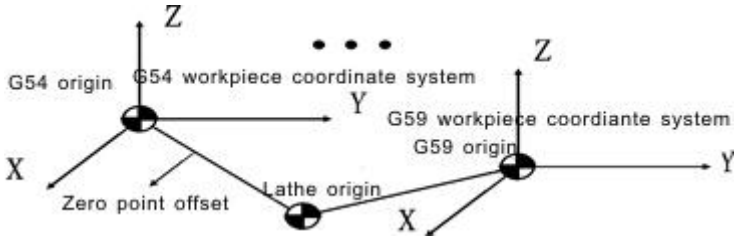
G55 workpiece coordinate 2

G56 workpiece coordinate 3

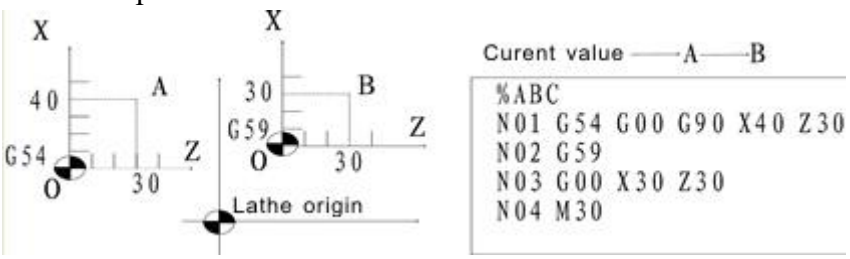
G57 workpiece coordinate 4

G58 workpiece coordinate 5

G59 workpiece coordinate 6



For example:



Note: 1. it is necessary to set original point of coordinate system in machine coordinate system before use these instructions.

2. it need to back reference point before use these instruction.

(31）Tool radius compensate G40、G41、G42

Format :G40 ; tool radius compensation cancellation, (mode,initial)

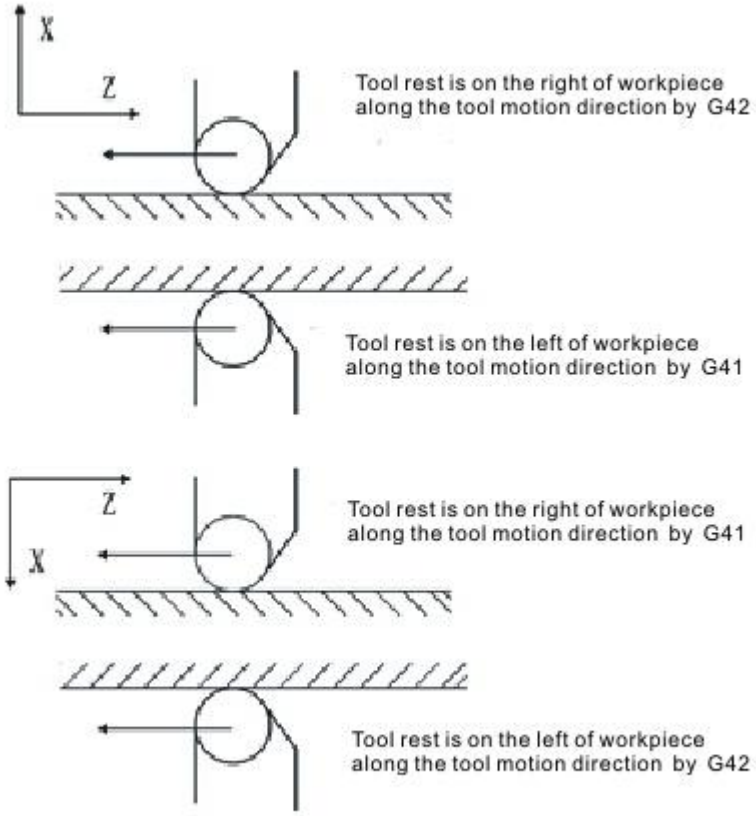
G41 ; tool radius compensation of cutter in the left of workpiece offset.(mode)

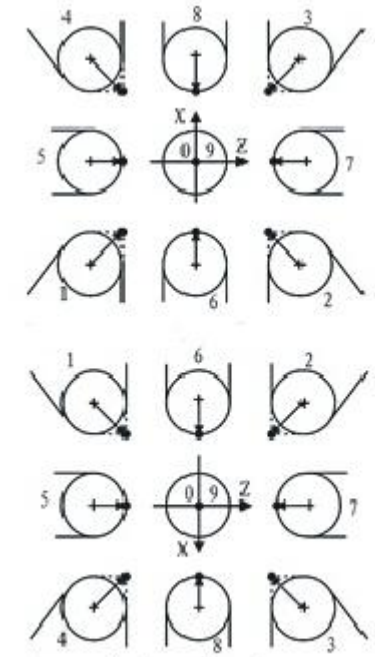
G42 ; tool radius compensation of cutter in the right of workpiece offset.(mode)

The left and right in G41, G42 are viewed from direction of cutting, tool lies in the left or right of workpiece. Tool radius is designated by R. executing offset begins at the program line of G41, G42. in the closed angle, system generates automatically transiting arc, allowing tool radius offset vector of last program segment transmit to that of the next program segment. Tool offset vector describes value and direction method of tool offset, its radius vector is tool radius. For arc, its direction is in radius direction. For line, its direction is vertical to the line direction.

When it transits at the closed angle arc, it will cause errors when the angle is less

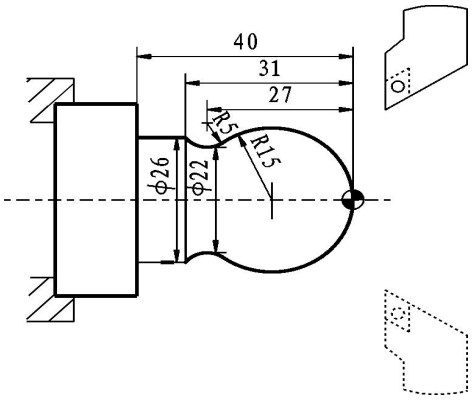
than 180 degree, because the transition becomes the inner closed angle transition. This system only transits outer closed angle, and remains valid only for G01, G02, G03, that is, the outer closed angle between strait line and strait line, arc and arc, strait line and arc can generate transiting arc.





Left and right compensate Tool parameter

For example：




```
N1 T0101
N2 M03 S400
N3 G00 X40 Z5
N4 G00 X0 G42
N5 G01 Z0 F60
N6 G03 U24 W-24 R15
N7 G02 X26 Z-31 R5
N8 G01 Z-40
N9 G00 X30
N10 G40 X40 Z5
N11 M30
```

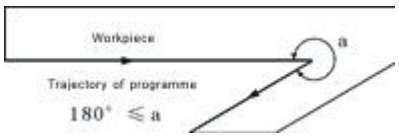
(32) Radius compensation of tool C

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

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：



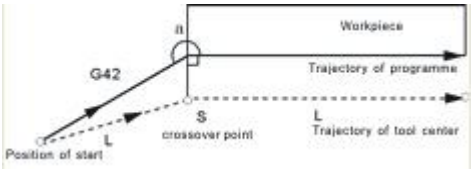
2) Tool motion when starting

The radius compensation without tool builds tool radius compensation

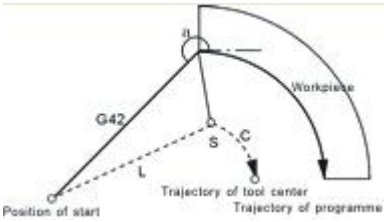
(a) Tool motion around the inside corner （α≥180）

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

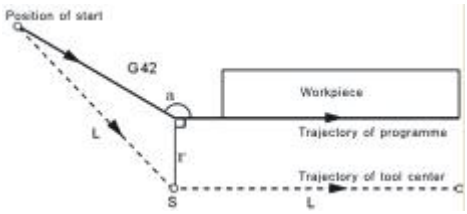


(b)The tool motion around the outside corner of obtuse angle（90≤α＜180）

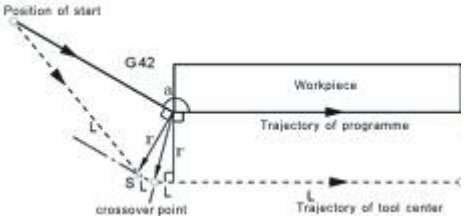
The tool center will move to the tool vector radius vertex of the end point in this program line.

Straight line->Straight line

A type

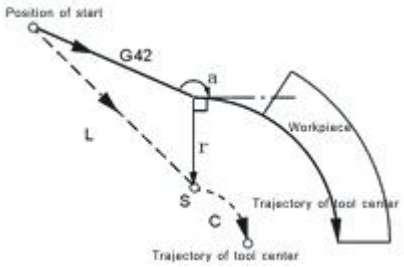


B type

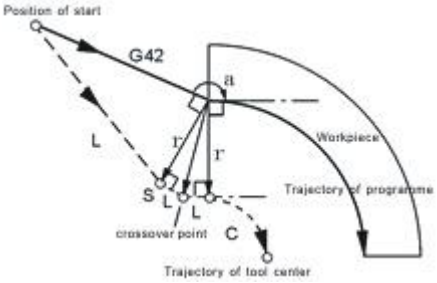


Straight line->Arc

A type



B type

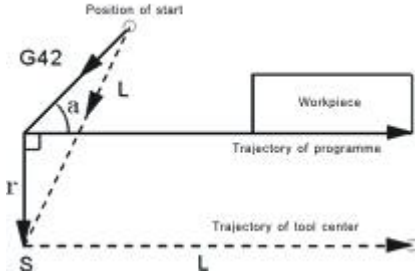


(c) The tool motion around the outside corner of acute angle（α＜90）

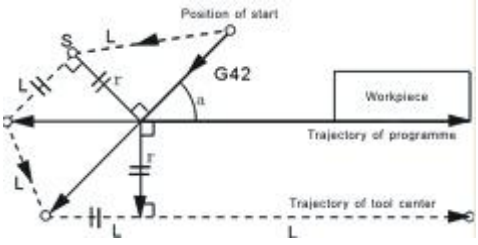
The tool center will move to the tool vector radius vertex of the end point in this program line.

Straight line->Straight line

A type

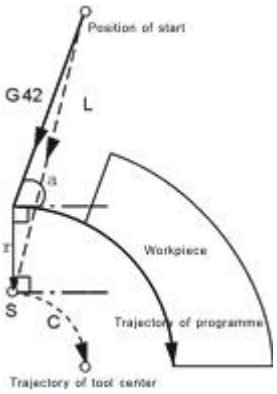


B type

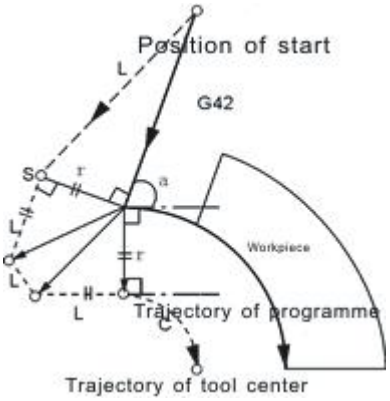


Straight line->Arc

A type



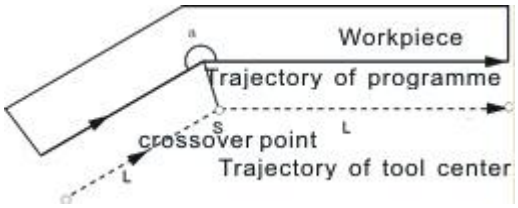
B type



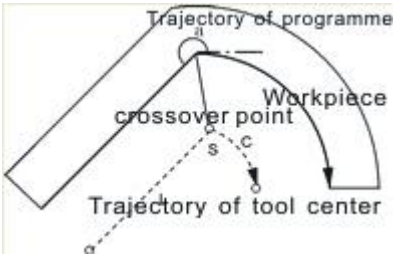
2) Tool motion in offset mode

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

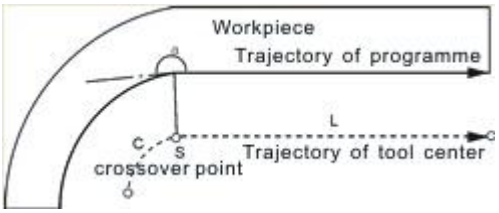
Straight line->Straight line



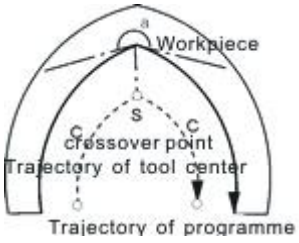
Straight line->Arc



Arc-> Straight line

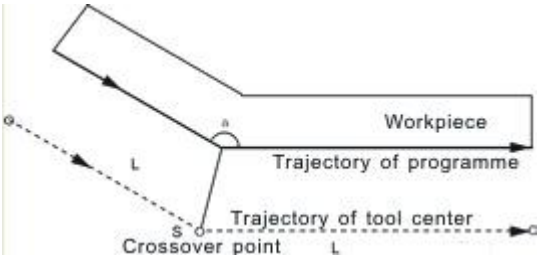


Arc->Arc

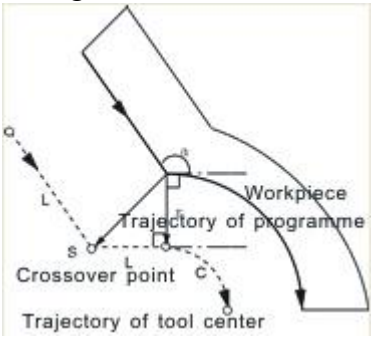


b) The tool motion around the outside corner of obtuse angle（90≤α＜180）

Straight line ->Straight line

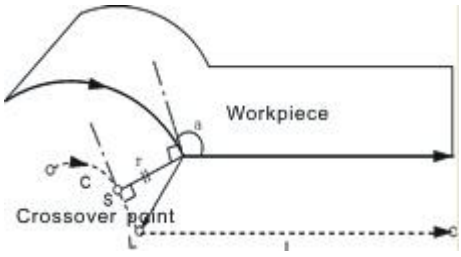


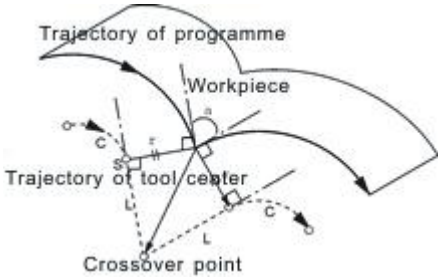
Straight line ->Arc



Arc-> Straight line

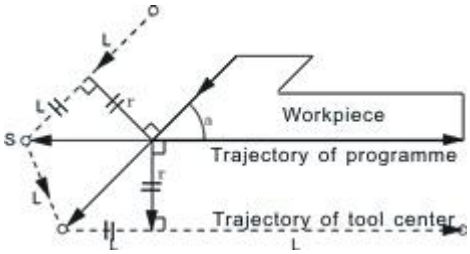
Arc->Arc



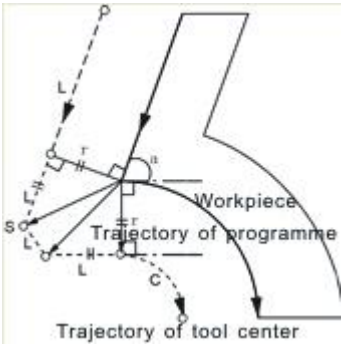


(c)The tool motion around the outside corner of acute angle（α＜90）

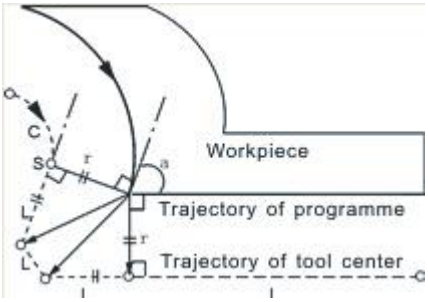
Straight line->Straight line



Straight line->Arc



Arc-> Straight line



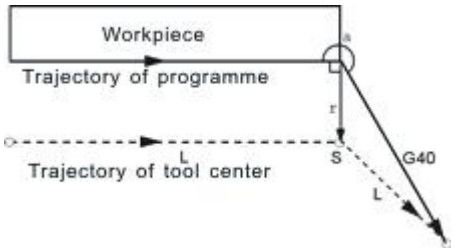
Arc->Arc

3) Tool motion in offset-cancel mode

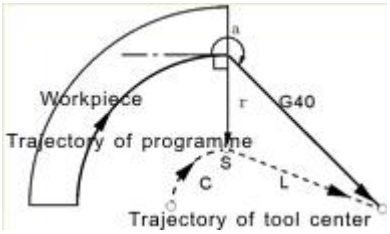
(a) Tool motion around the inside corner（180≤α）

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

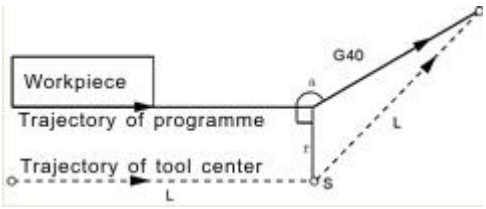


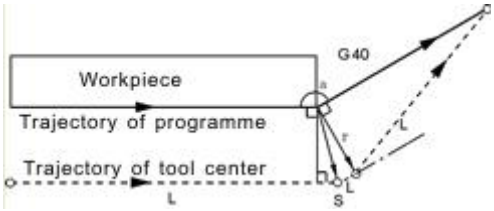
(b) The tool motion around the outside corner of obtuse angle（90≤α＜180）

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

Straight line－＞Straight line

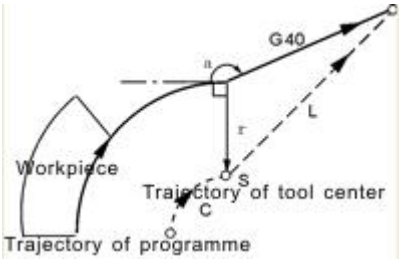
A type　　B type

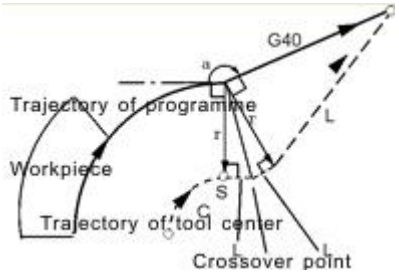




Arc－＞Straight line

A type　　B type



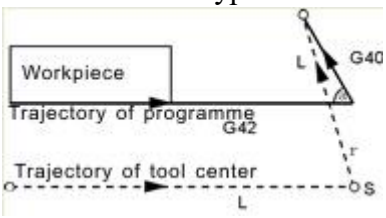


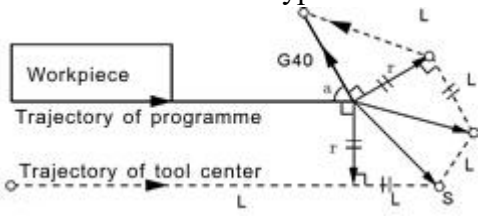
(c)The tool motion around the outside corner of acute angle（α＜90）

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

Straight line－＞Straight line

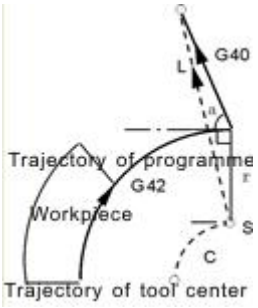
A type　　B type

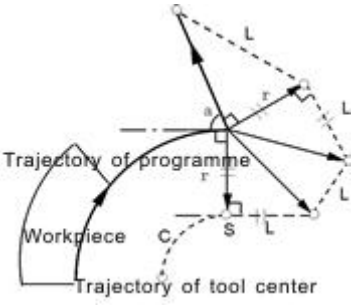




Arc－＞Straight line

A type　　B type





（33） automatic charmfering I and fillet R:

Format:

G01(G00) X(U) I: automatic charmfering, next segment including axis movements must be G01(G00) Z(W);

G01(G00) Z(W) I: automatic charmfering, next segment including axis movements must be G01(G00) X(U)。

G01(G00) X(U) R: automatic fillet, next segment including axis movements must be G01(G00) Z(W);

G01(G00) Z(W) R: automatic fillet, next segment including axis movements must be G01(G00) X(U).

Attention:

1. address I and R must be radius mode. The movement of the current line and next line must be over width I of charmfering or radius R of fillet, or system will automatically modify the width I or radius R into the minimum distance of current linear and next linear movement.

2. two adjacent lines must be 90°.

Sample:

```
N0 M03S500
N1 G0 X0 Z0
N2 G01Z-5 F100
N3 X10I4                    ; charmfering 4x4
N4 X20R5                    ; fillet R5
N5 Z-20
N6 G26
N7 M30
```

（34）Polar coordinate program(G15/G16)

Polar coordinate instructions can be input by polar coordinate radius and angle, positive direction of angle is counter clockwise of Z axis positive direction, and negative direction is clockwise. Radius specifies with absolute instruction (Z), angle is with absolute instruction (X).

Format：

```
G15              ; cancel
G16 IP-(XZ)  ; polar coordinate
```

Note:

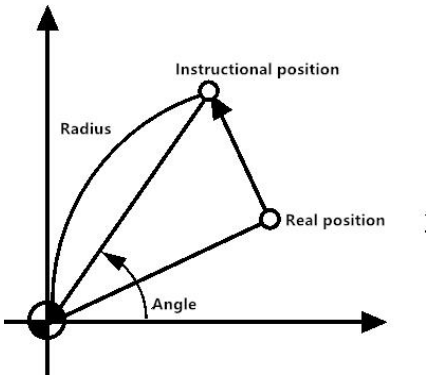
1. Z specifies the zero point of workpiece coordinate as the zero of polar coordinate system, from which measure the radius.

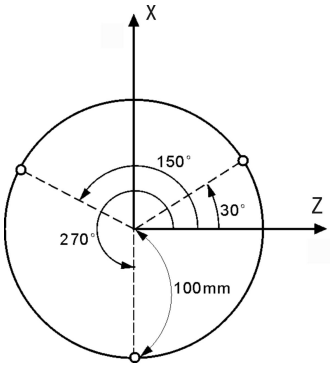
2. IP_ specifies the plane and the value of polar coordinate system by axis.

Z: radius of polar coordinate system;

X: angle of polar coordinate.

3. Z sets original of polar coordinate as original point of workpiec coordinate. As following picture.





For example：

N1 G16 X0 Z0 ；sets original of polar coordinate as original point of workpiec coordinate;

N2 G01 X30.0 Z100.0 F200.0; specify distance 100mm, angle 30°;

N3 X150.0 ； specify distance 100mm, angle 150°;

N4 X270.0 ； specify distance 100mm, angle 270°;

N5 G15 ； cancel polar coordinate instruction;

N6 M02

(35) Polar coordinate interpolation(G12.1/G13.1)

Converting shape controlling from instruction in rectangular coordinate into the movement of one linear axis(tool movement) and one rotary axis(workpiece rotary movement).

Format:
G12.1; start Polar coordinate interpolation；
G13.1; cancel Polar coordinate interpolation；

Note: G12.1, G13.1 are specified by single segment.

After enable the polar coordinate, you can instruct linear interpolation or arc interpolation in rectangular coordinate, containing of linear axis and rotary axis.

G12.1 starts polar coordinate system and choose a plane where polar coordinate interpolation is finished.

Process: polar coordinate interpolation based on X axis(linear) and C axis(rotary).

X axis is programmed by diameter or radius, C axis is programmed by radius(program unit: mm, display unit: angle)

Sample:

```
N10 T0202
…
N100 G00 X150 C0 Z0；
N110 G12.1；
N120 G01 X80 F200;
N130 C20;
N140 G03 X40 C40 R20;
N150 G01 X-40;
N160 G03 X-80 C20 R20;
N170 G01 C-20;
N180 G03 X-40 C-20 R20;
N190 G01 X40;
N200 G03 X80 C-20 R20;
N210 G01 C0;
N220 X150.0;
N230 G13.1;
N240 Z100.0 C0
…
N500 M30
```

Note 1: when controller power on or reset, polar coordinate interpolation will be canceled(G13.1); G12.1 and G13.1 are mode code.

Note 2: execution of G12.1 will be converted into G17 plane; execution of G13.1 will back into G18 plane;

Note 3: during polar coordinate interpolation, program is codded with rectangular coordinate instruction in polar coordinate system. No matter the linear axis in plane use diameter or radius, the rotary axis is programmed with radius.

Note 4: the available G codes in polar coordinate interpolation:

G01：linear interpolation；

G02、G03：arc interpolation；

G04：pause；

G40、G41、G42：tool endpoint compensation；

Note 5: during polar coordinate interpolation, speed of F code is the speed tangency speed with polar coordinate interpolation plane(rectangular coordinate system);

Note 6: the arc interpolation in polar interpolation plane will specify arc radius according to G17 interpolation plane, linear axis X use I, rotary axis C use J. also can use R code;

Note 7: in G12.1, coordinate system cannot be changed.

Note 8: in tool endpoint interpolation mode(G41 or G42), polar coordinate interpolation cannot be started or canceled; only if in G40 mode, G12.1 or G13.1 can work.

Note 9: in polar coordinate interpolation, when tool move near the center pint of workpiece, component feed speed of C axis will rise, once over the maximum cutting speed of C axis, controller will alarml

Note 10: during polar coordinate interpolation, program is rectangular code in polar coordinate plane. Axis address of rotary axis is address of the second axis(virtual axis) in plane. When instructing G12,1, tool position of polar coordinate interpolation starts from 0 degree. So before polar interpolation, spindle should be orientated as 0 degree. And before cancel the polar interpolation, C axis should be positioned at zero point of coordinate.

Note 11: in polar coordinate interpolation , the current position is real coordinate. But the balance distance in program segment will display according to polar coordinate interpolation plane(rectangular coordinate);

Note 12: during polar coordinate interpolation, please do not switch spindle gear.

（36）Switch metric system and British system (G20/G21), absolute program and relative program(G990/G991)

Format：

G990 ； XZYAC is absolute program；
G991 ； XZYAC is relative program；
G20 ； inch input；
G21 ； mm input;

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

Specification of unit under G20/G21(it is necessary to reset the unit in parameter):

1) The minimum unit of British system G20 is 0.0001inch, the minimum unit of Metric system is 0.001mm.
2) Under British system G20, unit of all instructions related to position is inch;
3) Under British system G20, unit of feeding speed F instruction is inch/min;
4) Under British system G20, unit of tool compensation and offset of worpiece coordinate system G54-G59 is inch;
5) Under British system G20, unit of parameters related to distance(such as soft limit and compensation) change from mm of Metric system into 0.1 inch or from um into 0.001inch;
6) Under British system G20, unit of parameters related to speed change from

mm/min into 0.1inch/min ;
7) Under British system G20, unit of parameters related to accelerated speed change from mm/min/s into 0.1inch/min/s;
8) Under British system G20, handwheel incremental value are 0.0001inch, 0.001inch, 0.01 inch;
9) Under British system G20,if pulse equivalent changes, system will adjust automatically, electronic gear will accords to metric system;
10) Under British system G20, coordinate displays four numbers behind the decimal point;
11) G20/G21 should be written in the beginning of program, instead of middle of program;
12) Under British system G20, rotary axis rotating 1 degree accords to 0.1 inch;
13) two systems can be converted by executing G20/G21 in MDI, and restating controller.
14) It is necessary to set tool after conversion between G20 and G21;
15) As to system prompt information, you just need to modify the parameter configuration file cncsystemen. For example, the parameter about speed: change mm/min into 0.1inch/min, parameter about position:change mm or degree into 0.1 inch, change um into 0.0001 inch.

(37) M instruction

Usage	instruction	Function	Initial	mode	Note
Spindle	M03 M04 M05 M203 M204 M205 M881 M882	Spindle on CW Spindle on CCW Spindle brake M7053/M7054 Pxxxx: Spindle stop after rotate CW/CCW a while, the time is specified by P, unit:ms Example: M7053 P2000; means spindle rotate CW 2s and then stop C axis stop, output M61, check M22 A axis stop, output M63, check M24	√	√ √ √	M203 M204 M205 The second spindle rotate CW, CCW, stop

Cooling	M08 M09	Coolant on Coolant off	 √	√ √	
Chuck	M10 M11	Chuck tightens Chuck looses	√	√ √	
Tailstock	M79 M78	Tighten Loose	 √	√ √	Control M79
Lubrication	M32 M33	Lubrication on Lubrication off	 √	√ √	Control M32
Huff	M59 M58	Huff on Huff off	 √	√ √	Control M59
User-defined output	M61 M60	user-define 1 open user-define 1 close		√ √	Control M61
	M63 M62	user-define 2 open user-define 2 close		√ √	Control M63
	M65 M64	user-define 3 open user-define 3 close		√ √	Control M65
	M67 M66	user-define 4 open user-define 4 close		√ √	Control M67
	M69 M68	user-define 5 open user-define 5 close		√ √	Control M69
	M71 M70	user-define 6 open user-define 6 close		√ √	Control M71
	M73 M72	user-define 7 open user-define 7 close		√ √	Control M73
	M75 M74	The spindle servo select the position controlling mode when C axis is output pulse signal, close M75 signal when system starting M03/M04		√ √	Control M75
user-defined input	M12 M13	Check M12 valid Check M12 invalid		√ √	Conditional skip, example:

	M14 M15	Check M14 valid Check M14 invalid		√ √	M20 P120 means if M12 is valid, program skips to 120th line.
	M16 M17	Check M16 valid Check M16 invalid		√ √	
	M18 M19	Check M18 valid Check M18 invalid		√ √	
	M28 M29	Check M28 valid Check M28 invalid		√ √	
	M22 M23	Check M22 valid Check M22 invalid		√ √	
	M24 M25	Check M24 valid Check M24 invalid		√ √	
Spindle Gear	M41 M42 M43 M44	first gear second gear third gear fourth gear			Input S01, S02, S03, S04 and modify the analog voltage of spindle.
Subprogram	M97 M98 M99	jump call return			L=1-99 P is the line number of transferring program
Program	M87 M188 M187 M00 M01 M02 M30 M20	Part number plus 1 Keep feeding and not back home Keep feeding and back home pause M22 avail pause Program end M05、M09 end Loop go start			When other parameter P10=0 is set not to auotomatical plus 1, instruction M87 to make workpiece number plus 1

Read the position of absolute motor	M500 M501 M502 M503 M504	M500: read absolute motor position of all the feeding axis and reset the current lathe coordinate. M501: read absolute motor position of X axis and reset the current lathe coordinate. M502: read absolute motor position of Y axis and reset the current lathe coordinate. M503: read absolute motor position of Z axis and reset the current lathe coordinate. M504: read absolute motor position of the fourth axis and reset the current lathe coordinate.			
Clear workpiece coordinate	M315 M317 M318 M319 M320	M315:Clear workpiece coordinate of A axis M317:Clear workpiece coordinate of X axis M318:Clear workpiece coordinate of Y axis M319:Clear workpiece coordinate of Z axis M320:Clear workpiece coordinate of all axis			
Clear lathe coordinate	M415 M417 M418 M419 M420	M415:Clear lathe coordinate of A axis M417:Clear lathe coordinate of X axis M418:Clear lathe coordinate of Y axis M419:Clear lathe coordinate of Z axis M420:Clear lathe coordinate of all axis			

All of M code input and output are low-level effective.

（38）Call program (M97/M98/M99)

Non-condition skip

M97 P: unconditional jump to P-word line;

Call sub-program:

In the controller, subprogram should be an independent program.

M98 P L: unconditional call subprogram. P specify route and name of subprogram, L is the address of calling times.

M98 can be omitted, format is PP file name, file name can be hidden file, but the head character must be “HIDEFILE”. for example: file “HIDEFILE01” will not be shown in program section, it can be called by M98/G65 PHIDEFILE01or M98/G65 P*01or PP*01or PPHIDEFILE01.

For example: Psub/1390 means subprogram is tmp/NC/sub/1390.

Attention:

1. tmp/NC/ is default route of controller program, sub is one of folders under it.
2. subprogram should be an independent program.
3. the method calling which happened between both main program and subprogram in USB disc. P [or P];

Sample: M98 P[A1234 means call subprogram A1234 in USB;
M98 P]SS12 means call subprogram SS12 in USB;
PP[FFDE means call subprogram FFDE in USB;

If call program under folder, it is necessary to edit route.

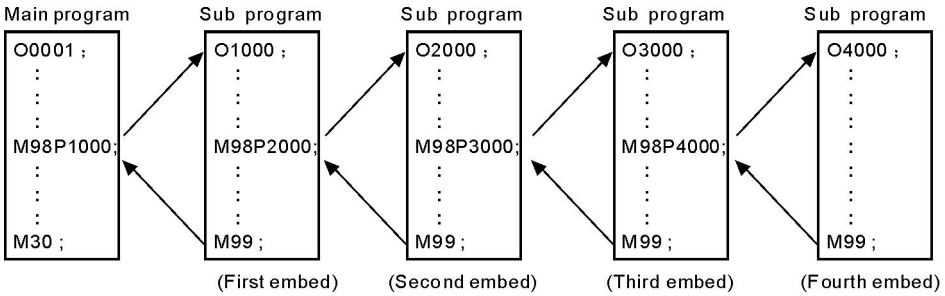
There must be a space before call times address L. the subprogram is finished, then back to next segment of main program. If program includes fixed sequence or repeated shape, then this kind of sequence or shape can be edited as a subprogram to simplify program. Main program can call subprogram, and subprogram can also call another subprogram.

M99: finish subprogram and back, subprogram finishing must use this.

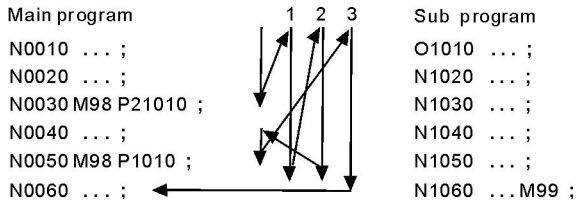
Attention:

1) M99 is identical with M02 in main program;
1) M99 with P is identical with M97 in main program;
1) M99 is in the next line of calling when back to main program;
1) M99 with P is in the P line when back to main program;

The Sub-program can embedded call under four levels as follows:



For example



The maximum call times is 9999.

（39）conditional wait or skip

N code can be sued to detect outer input signal as condition:

Conditional wait:

M12 M13 used to detect input status of signal M12, M12 in program means detect M12, if valid then execute next line. M13 means detect M12 signal, if invalid then execute next line. The instruction need to be edited in an isolated line.

M14 M15 used to detect input status of signal M14, M14 in program means detect M14, if valid then execute next line. M15 means detect M14 signal, if invalid then execute next line. The instruction need to be edited in an isolated line.

M16 M17 used to detect input status of signal M16, M16 in program means detect M16, if valid then execute next line. M17 means detect M16 signal, if invalid then execute next line. The instruction need to be edited in an isolated line.

M18 M19 used to detect input status of signal M18, M18 in program means detect M18, if valid then execute next line. M19 means detect M18 signal, if invalid then execute next line. The instruction need to be edited in an isolated line.

M22 M23 used to detect input status of signal M22, M12 in program means detect M22, if valid then execute next line. M23 means detect M22 signal, if invalid then execute next line. The instruction need to be edited in an isolated line.

M24 M25 used to detect input status of signal M24, M24 in program means detect M24, if valid then execute next line. M25 means detect M24 signal, if invalid then execute next line. The instruction need to be edited in an isolated line.

M28 M29 used to detect input status of signal M28, M28 in program means detect M28, if valid then execute next line. M29 means detect M28 signal, if invalid then execute next line. The instruction need to be edited in an isolated line.

Conditional skip:

M12/M13/M14/M15/M16/M17/M22/M23/M24/M25/M28/M29 add Pxxxx（line number of program), means if meet condition then skip to specified line, otherwise execute next line.

Example: M14 P0120

When program execute to this line, controller will detect M14 input signal, if M14 is valid, program will skip to 120th line(marked line), if M14 is invalid, then execute next line.

（40）T tool

Format	function
Tab	a: Tool number， b: compensate number a=0 means not change tool

B=0 means cancel tool compensation, and display machine coordinate.

a=0~99, b=0~99.

For example: T0204 ; change No.2 tool and No.4 tool compensation;

T0300 ; change No.3 tool and display machine coordinate;

T0004 ; not change tool and change No.4 tool compensation.

When install electronic turret, each tool position can equip different tools, then adjust different different tool compensation. For example, in four-tool turret, and install 2 tools in position 3(No.3 tool compensation, No.5 tool compensation).

Format:

```
N0000   T0101
N0001   G0  X30 Z500
N0002   T0303          ; change No.3 tool and No.3 tool compensation.
N0003   G00  X50
N0004   T0505          ; No.3 tool and No.5 tool compensation.
N0005   M02
```

Attention:

1. Txxxx, the first two number are tool number, last two number are tool compensation number. Range of tool number is 01-99, tool compensation can be called in any one.

2. when execute tool compensation, machine does not move, just calculate the difference and update coordinate. If want to display coordinate movement according to tool compensation, please programme with absolute instruction.

3. after power om, controller will execute the tool number before power off, and execute default compensation of default tool.

4. change tool in manual mode, the number of compensation will accords to tool number. For example, No.6 tool will execute No.6 compensation.

（41）S、SS Spindle speed

The first SP use “S” ,speed parameter P36 control the highest speed,output 0-10V frequency conversion voltage.

The second SP use “SS” ,speed parameter P40 control the highest speed,output 0-10V frequency conversion voltage.

（42）F feed speed

F use for G01、G02、G03 etc. feed speed.

G98: range F0.01-20000mm/min，G99: range 0.001-500mm/r.

Feed speed is Fx speed override, F is mode instruction.

(1) G98 feed speed per minutes. mm/min;

(2) G99 feed speed per turn. Mm/r.

If F instruction and movement instruction is in the same line, it will execute F instruction firstly.

Attention: G99 instructions requires spindle with encoder, or controller will be under waiting.

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

1.Input instruction: WAT

Waiting for the input port X valid or invalid instruction

Format: WAT+ (-) X

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

"-" means the input is invalid;

"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

6）save the current global macro variable with format of customer program FILEMS(AABBCC) or FILEMS[AABBCC]
Save #21-#999 into program AABBCC file；
If want to read these variable, just need to execute the file or call the file as a subprogram.

4.The arithmetic and logic operation

Table:

Function	Format	Note
Definition	#i = #j	
Addition Subtraction Multiplication Division	#i = #j + #k ; #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 Exclusive or 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.Information prompt dialog

Format: MSG(parameter) or MSG[parameter]; parameter is information character string, pause.

Note: the instruction also can work in normal NC program(none macro program).

Format: STAF(parameter) or STAF[parameter]; parameter is information character string, no pause.

11.Generate process program automatically

1>, the instruction of found and open file：

FILEON(parameter) or FILEON[parameter]

Example：

FILEON(AABBCC) or FILEON[AABBCC]

means found and open file AABBCC

2>,the instruction of close file：

FILECE means close the current file, if without the instruction, controller will close the file automatically after the program finish.

3>,write character string into the current opening file：

FILEWD (parameter) or FILEWD [parameter]

Example：

FILEWD (G54G0X0Z0) or FILEWD [G54G0X0Z0]

means edit character string into the current opening file G54G0X0Z0

4>, write absolute coordinate of current feed axis into the file：

FILEWC

Sample program：

G0X0Z0
FILEON[AABBCC]
FILEWD [G54G0X0Z0]
G1X45Z89
FILEWC
G1X99Z76
FILEWC

FILECE

After finish the program, it will generate a file AABBCC in program directory，contents is as following：

G54G0X0Z0

X45Z89

X99Z76

(44) User-defined macro instruction(G120-G160,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: quantity 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
```

(45) Composite function of turning and milling.

The method of performing milling in lathe controller:

1. parameter: enable the third axis, and set it as Y axis, the program method set as radius.

2. relative instruction: except following instruction, the other instructions refer to milling controller manual.

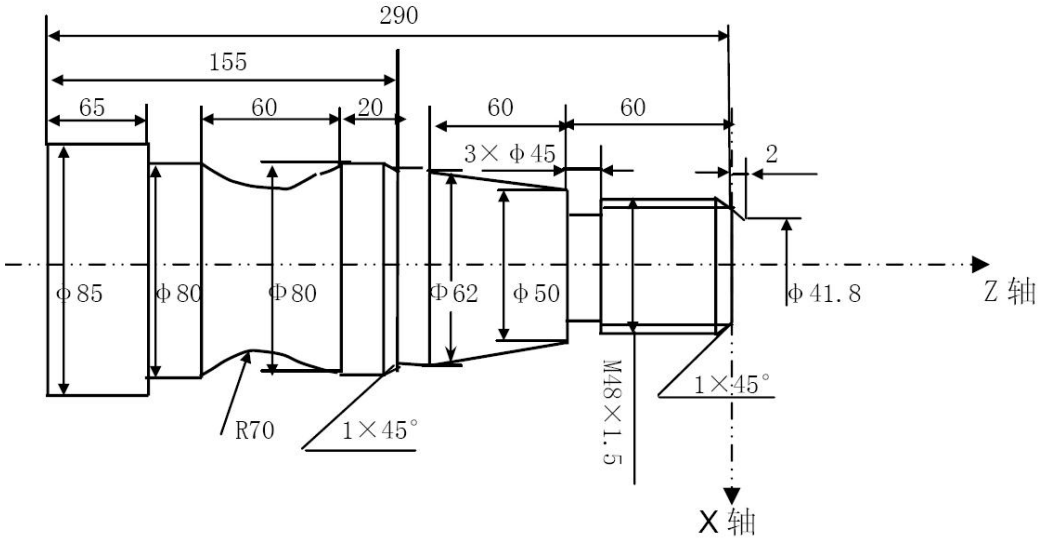
G990 accords to G90 in milling controller;
G991 accords to G91 in milling controller;
G994 accords to G94 in milling controller;
G995 accords to G95 in milling controller;
G998 accords to G98 in milling controller;
G999 accords to G99 in milling controller;
G973 accords to G73 in milling controller;
G974 accords to G74 in milling controller;
G976 accords to G76 in milling controller;
G981 accords to G81 in milling controller;
G982 accords to G82 in milling controller;
G983 accords to G83 in milling controller;
G984 accords to G84 in milling controller;
G985 accords to G85 in milling controller;
G986 accords to G86 in milling controller;
G987 accords to G87 in milling controller;
G989 accords to G89 in milling controller.

2.5 Synthetical instance for programming

In the actual programming, must according to the drawing and processing requirement to select the correct install folder mode and suitable tool, and combined with the actual working performance of lathe to select the right cutting allowance, for example:

Example 1：The tool is:

T01 cylindrical cutting tool; T02 cutting groove, tool width 3m; T03 thread tool with 60 degree angle



Program as follows:

N10 M03 S1000； Start spindle

N20 T0101； Choose the first tool and execute the first redeem

N30 G00 X41.8 Z2 M08； Move fast to the cutting point，cutting fluid is on

N40 G01 X48 Z-1 F100； Chamfer

N50 Z-60； Fine machining for thread

N60 X50； Tool is backing

N70 X62 W－60； Fine machining in cone

N80 W－15； Fine machining in Φ62mm excircle

N90 X78； Tool is backing

N100 X80 W-1； Chamfer

N110 W-19； Fine machining in Φ80mm excircle

N120 G02 X80 W-60 R70； Fine machining in arc（I63.25 K－30）

N130 G01 Z-225； Fine machining in Φ80mm excircle

N140 X85； Tool is backing

N150 Z-290； Fine machining in Φ85mm excircle

N160 X90 M09； Tool is backing，cutting fluid is off

N170 G00 X150 Z50； Move fast to the point of changing tool

N180 T0202； Change tool and set the No.2 redeem

N190 M03 S800； Change speed of spindle

N200 G00 X51 Z-60 M08； Move fast to the processing point，use the left point of

tool to redeem

N210 G01 X45 F90； Cutting Φ45mm groove
N220 G00 X51； Tool is backing
N230 X150 Z50 M09； Return to the point of backing tool，cutting fluid is off
N240 T0303； Change tool and set the redeem
N250 M03 S1500； Change speed of spindle
N260 G00 X62 Z6 M08； Move fast to the processing point， cutting fluid is on
N270 G92 X47.54 Z－58 F1.5； Cutting thread is cycle
N280 X46.94；
N290 X46.54；
N300 X46.38；
N310 G00 X150 Z50 M09； Return to the point of start cutting, cutting fluid is off
N320 T0300； Cancel redeem
N330 M05； Stop spindle
N340 M30； Program is over

2.6 Usage for grinder

Use for excircle grinder with active measure and control instrument to process

1、Face grinding method

T0101
M60 ； Use M60 to choose the mode of face measurement
M03 ； Start the spindle of emery cutter
M63 ；To measure the start to the measuring position,the system input M63 single when the lathe equipping this equipment
G04 X1 ； Delay 1 second
G00 X0 Z1 ； To the nearby face measurement
G31 W-3 F100 P1031 ； Rough machining in face enough M16
G31 W-1 F10 P1028 ； Fine machining in face enough M18
G0 Z1
M62 ； Start the equipment of face measurement
M02

2、Excircle grinding mode T0101

M61 ； Use M61to choose the mode of excirle measurement
M03 ； Start the spindle of emery cutter
G04 X1 ； Delay 1 second
G00 X10 Z1 ； To the nearby face measurement
G31 U-3 F100 P1027 ； Excircle rough grinding in face enough M22， make sure within 3mm
G31 U-2 F10 P1026 ； Excircle fine grinding in face enoughM24， make sure within 2mm
G31 U-1 F1 P1025 ；Excircle fine polish grinding in face enoughM28，make sure within 1mm
G0 X10

M02

3、Compensate the abrasion of emery cutter

T0202

G00 X100 Z100 ; Must be the fixed point of emery cutter

G22 L5

GO U-0.01 ; Fix 0.05mm

G01 W-20 F10

W20

G800

G185 X100

M02

4. setting for high-speed punching machine, there are two high-speed input point, Other parameter P32=28, P910=1;

T0101

G00 X0

G22 L50 ; loop times

WAT+M295 ; punch up in position, detect input point M18(X28)

OUT-M295

G0 U1

WAT+M297 ; punch down in position, detect input point M14(X30)

OUT-M297

G800

M02

2.7 Instruction of following axis

1.Method A: the coordinate of follow-up axis will not follow, while the fourth axis can be hidden in screen. When axis parameter P414 is 7, the fourth axis is following X axis in both automatic and manual mode. When P414 is 8, the fourth axis is following Y axis in both automatic and manual mode. When P414 is 9, the fourth axis is following Z axis in both automatic and manual mode.

When P414 is 17/18/19, the C axis is following X/Y/Z axis in both automatic and manual mode. When P414 is 27/28/29, the B axis is following X/Y/Z axis in both automatic and manual mode.

Besides, D5...D13=1 means A axis follows X/Y/Z, C axis follows X/Y/Z, B axis follows X/Y/Z, so three axis can follow at the same time.

2. Method B: it is applied to specify follow-up axis with instructions in automatic mode, but not follow in manual mode. In this way, is coordinate following when axis is following.

M123: turn on the Y axis following X axis, Y axis is following with radius, so Y axis should programme with radius, please note.

M124: turn off the Y axis following X axis;

M125: turn on the A axis following X axis, A axis is following with radius, so A axis should programme with radius, please note. ;

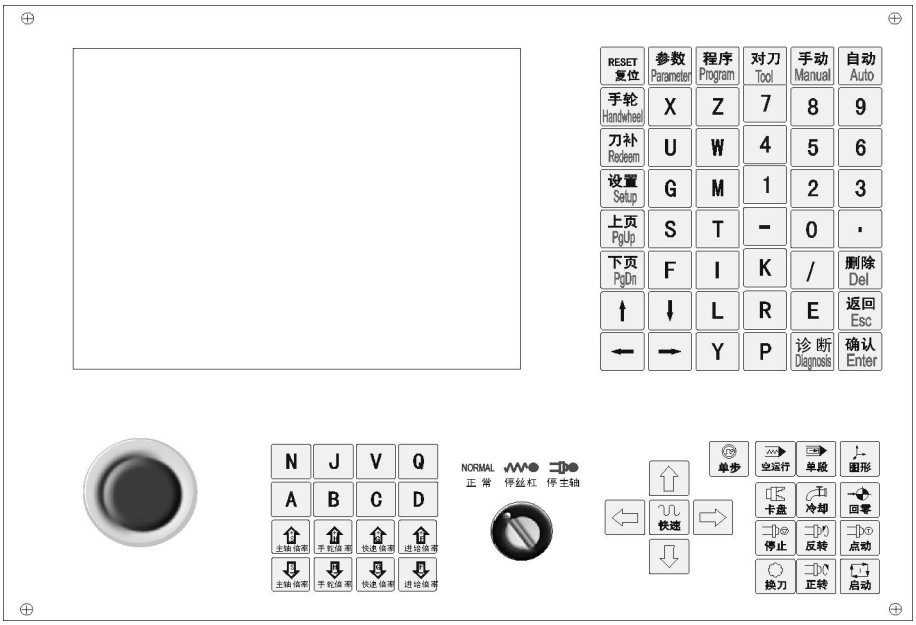
M126: turn off the A axis following X axis.

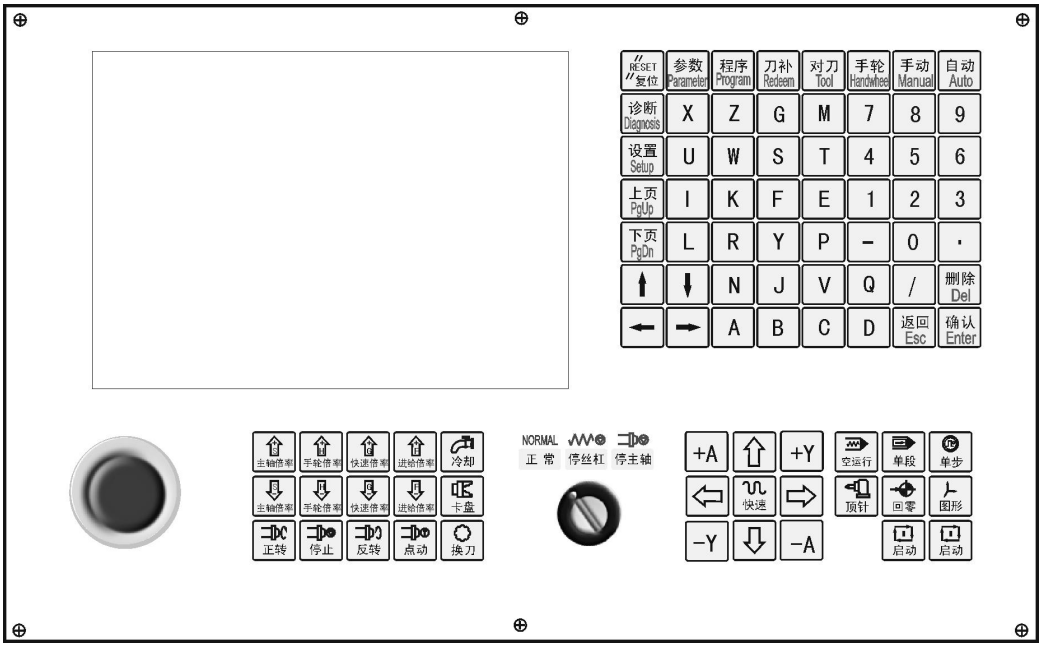
Chapter 3 Operation explanation

3.1 Summary

When using the CNC system, just master the parameter of system, edit program, manual operation, auto operation. Then you can operate easily. There are some details to instruct hereinafter.

3.2 Operation panel and switches(Following shown: two typies)





3.3 Keyboard instruction

3.3.1 Rate increase

(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,G613, 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 Switches

Switch	Functions
	Emergency stop Driver and motor stop immediately, turns off the spindle, coolant, waits for the rise of button, and initializes values
	Intervention switch: When program runs or in manual state, it can make a realtime adjustment of feed speed; Spindle percentage switch: In the process of spindle running, adjusts the speed accordingly.

3.3.3 Other keys

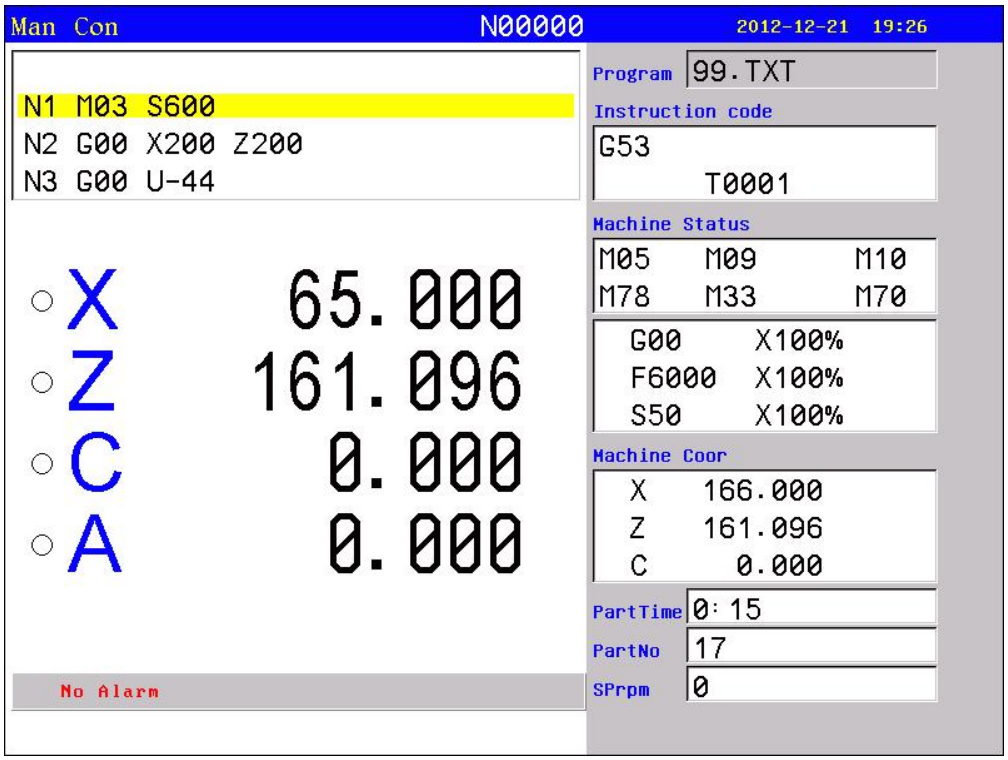
Keyboards	Functions
Letter key Number key	A B C S E F G H I J K L M N O P Q R S T U V W X Y Z 1 2 3 4 5 6 7 8 9 . - ： for program instructions, edition of parameter; number keys are used for inputting data and selecting sub-menu.
Edit key	“↑、↓、→、←、Del、PgUp、PgDn”for programming, direction keys can be used for selecting menu.

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 speed 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	
Feed key	+Y +A–Y -A	For X、Y、Z 、A axes direction feed

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)



The left part of the first line on the screen is the running mode of system(manual continuous, manual incremental, auto continuous, auto single block, auto idle running, handwheel), middle part is the current line of processing program, right part is date and time.

The left top area is program part, the second line is current program line;

The left button area is workpiecec coordinate display;

The button area is the information display, like alarm;

The right area the display of program name, instructions, machine status and machine coordinate etc;

Press “program” to enter into program management, then edit, modify, check, delete, copy program;

Press “parameter” to enter into parameter management, then check and modify parameter of controller;

Manual mode is used to modify machine coordinate and management of machine auxiliary performance, tool setting, specify start point of work piece and manual process. Manual mode cannot used to run program.

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)“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.

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

(4)“  ”：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.

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

(6)“  ”: 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.

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

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

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

(10)“Auto”: Select automatic mode.

(11)“Manual”: Select manual mode.

(12)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.

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

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

(15)“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”.

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

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

(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.31 No.33 parameter in speed parameter, the direction is determined by No.28 parameter in axis parameter.

(20) “change tool”: 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 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 “PgUp” or “PgDn” to switch the display which correspond to “relative” “absolute” “machine”.

(24) “Partno clear”: Press Del and Enter to clear quantity of processed work piece.

(25) “Back to G53 coordinate datum point”: Press “Q” and Enter.

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

(27) search center point: convert into G54-G59, press”K”. according to prompt information, controller will calculate value and save the offset of G43-G59.

PS: 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 tp stop feeding. Making sure the axis and using “↑ ↓ ← →” 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 parameter value in speed parameter. If set the speed higher than the speed in parameter, the feed speed will be No.1 No.2 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.3 parameter in speed parameter, when setting the speed is higher than the value of parameter, then will be the No.3 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.26 parameter in axis parameter, it can be prompt or force;
2. The way and type of detecting signal can be set by No.27 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.28 parameter in axis parameter, D2 D4 correspond to X Z axis, 0 is forward, 1 is reverse.
4. The sequence of X Z back to datum point can be set by No.28 parameter in axis parameter, X is first when D8 is 0, Z is first when D8 is 1.
5. The type of the switch for datum point can be set by No.29 parameter in axis parameter, D0 D2 correspond to X Z axis,0 is always on, 1 is always off.
6. The maximum length of detecting Z pulse of electrical motor can be set by No.30 No.31 parameter in axis parameter, the value must less than the pulse of electrical motor run a cycle.
7. The shifting distance after backing to datum point can be set by No.32 No.33 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.23 parameter in axis parameter, setting No.24 No.25 parameter to make sure X axis' and Z axis' floating point, the datum point of lathe.

The steps to set floating point as follows:

1. Setting the No.23 parameter in axis parameter to set the axis which is starting up floating point. For example: Turn X axis on is "00001000". (Z axis is "00100000" turn both of them on is 00101000.)
2. Moving X axis to designated position so that set floating point.
3. Press "Parameter", "Axis parameter" and select No.24 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 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 Z axis to back to the datum point in dialog box. Set the No.28 parameter in axis parameter to "1" to make Z axis bake to the datum point first. At the absolute and relative coordinate condition, 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 "Z" or "X" 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.17 parameter in speed parameter(the bigger the faster). The maximum speed is controlled by No.20(X axis) No.21(Z axis).

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

3.4.6 Set Tool

Because no tool is the same when using multiple tools machining so need to identify them previous the value of the redemption, that is to carry out redeeming. Tooling is actually move the tool to the workpiece surface at this point, the point's

actual measurement of values import directly into the system the system calculates the deviation and save to the corresponding tool offset register automatically.

Press “Tool” ,then choose “X” or “Z” and “Enter” to select axis. There are two methods:

Plan A(suggest)

(1) Clamped workpiece, select appropriate spindle speed and feed speed, start spindle.
(2)Select the tool to “Tool”, for example: T0202
(3)Using manual continuous to cut a bit of cylinder or bore on workpiece.
(4)Z axis exits(X axis can’t move), stop spindle.
(5) Measure the diameter of workpiece(cylinder or bore).
(6)Press “Tool”, then “X”, “Enter” and import the above value of measurement into dialog box, press “Enter” to confirm.
(7)Use the same method to cut end surface of workpiece.
(8)Measure the end surface of workpiece and spindle chuck (Z=0) into the distance.
(9)Press “Tool”, then “Z”, “Enter” and import the above value of measurement into dialog box, press “Enter” to confirm.
The second tool is already done(T02).Repeat (1) ——(9) to make others tools get done.

Plan B

(1) Clamped workpiece, select appropriate spindle speed and feed speed, start spindle.
(2) Select the tool to “Tool”, for example: T0202
(3) Using manual continuous to cut a bit of cylinder or bore on workpiece.
(4)Press “Tool”, the system will appear a dialog box.
(5)X axis and Z axis both exit, stop spindle.
(6) Measure the diameter of workpiece(cylinder or bore).
(7)Press “X” and import the above value of measurement into dialog box, press “Enter” to confirm.
(8) Use the same method to cut end surface of workpiece. Stop feeding.
(9) Press “Tool”, the system will appear a dialog box.
(10) X axis and Z axis both exit, stop spindle.
(11) Measure the end surface of workpiece and spindle chuck (Z=0) into the distance.
(12)Press “Z” and import the above value of measurement into dialog box, press “Enter” to confirm.
The second tool is already done(T02).Repeat (1) ——(12) to make others tools get done.
The difference between two methods:

Plan A(suggest)

(1)Make sure the “Tool” axis couldn’t exit.
(2)The tool must touch workpiece.
(3)Use “Z” axis’ direction to “Tool”

Plan B

(1) The axis can exit.
(2) The tool mustn’t touch workpiece.
(3) Use “X” axis’ direction to “Tool”.

In the above process, the import value's and lathe coordinate value's difference will be saved to the corresponding cutter compensation by system automatically. So the inaccuracy of setting tools can be modified correctly by corresponding cutter.

System to each tool independently of each other, each has its own coordinate system, so each tool can "Tool" anytime and the tool is destroyed in the process is only the tool.

Instruction

1. **When a group of tool to be used for two or more parts processing requires the working coordinate to achieve the overall shifting tool group. So, make sure the value of workpiece coordinate before setting tools. Methods of operation are as follows:**

(1)Select a tool.
(2) Press "MDI" to select corresponding coordinates (54-59).
(3) Using manual continuous to cut a bit of cylinder or bore on workpiece.
(4) Z axis exits(X axis can't move), stop spindle.
(5) Measure the diameter of workpiece(cylinder or bore).
(6) Press "Setup", "X" and "Enter", import the value of measure, press "Enter".
(7) Use the same method to cut end surface of workpiece.
(8) Measure the end surface of workpiece and spindle chuck (Z=0) into the distance.
(9) Press "Setup", "Z" and import the value of measure, press "Enter".

The import value's and Tool coordinate value's difference will be saved to the corresponding parameter by system automatically, corresponding workpiece coordinate is been set now. It's done after finishing setting tools as the above. It's done that the code in first line to execute selection of the corresponding coordinate.

2. **Just setting one of the tools after the lathe crashing or loss of step, every tool is ok.**

The method: In the G53 status, switch "Tool" into "Setup" as the above operations.

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 "C" to select the program to carry out automatically.

Switch the display of coordinate: Press "PgUp" or "PgDn" to switch the display which correspond to "relative" "absolute" "machine".

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. Import the line to run

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.5 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. Mark the line to run

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.5 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.5 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 automatic process

(1) Intervention switch: For suspend feed coordinate and stop spindle. Left, middle, right 3 gears totally.

Left: Feeding coordinate and spindle is not limited.

Middle: Stop feeding, spindle is not limited.

Right: Stop feeding and spindle.

Stop here means suspend, turn left the switch to continue process; Press “Reset” to exit automatical process and stop processing, the program line is going to back to the first of the processing program.

At the manual mode condition, spindle is not limited by intervention switch, only by the button.

(2)“  ”：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.

(3) “  ”: 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.

(4) “  ”: 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.1 axis parameter is 1, external band switch takes in control, Adjust the speed of spindle arbitrarily in the process according to the different situation.

(5) Stop in the process: At the continuous mode in process condition, press

“  ” to stop running after executing a current program line, wait for operating.

(6) 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.

(7) 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.

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

(9) enable inner macro program during processing. For example: if X7 is valid, start ProgramUser9, add PLC as following:

```
   X07    M122
- -|    |- - -|    |- ----------------(S)M300          ;if X7valid then set M300
   X07    M300 M122
- -|    |- - -|    |- - -|    |------------(S)M1014        ;if X7valid, reset current program;
M300    M122    X07
- -|    |- - -| / |- - -| / |---------(S)M1069        ; enter into manual mode;
  M300    M122    M01 X07
- - |    |- - -| / |- -|    |- - -| / |-----(S)M1079        ; execute ProgramUser9;
                                  |--(R)M300          ; reset M300;
```

M300: intermediary register，PLC cannot appear the same M300；
M1014: reset current program；
M1069: manual mode；
M1070-M1079: corresponding to ProgramUser0-9.

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.17 No.18 No.19 No.20 No.21 No.43 parameter in speed parameter and No.23 parameter in processing parameter, 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 in this system, use DNC to process when the processing program is greater than 32M 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 and Z axis are limited positive forcedly : X or Z axis is in the positive position which is limited forcedly.

(2) X and Z axis are limited negative forcedly: 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 and Z 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.

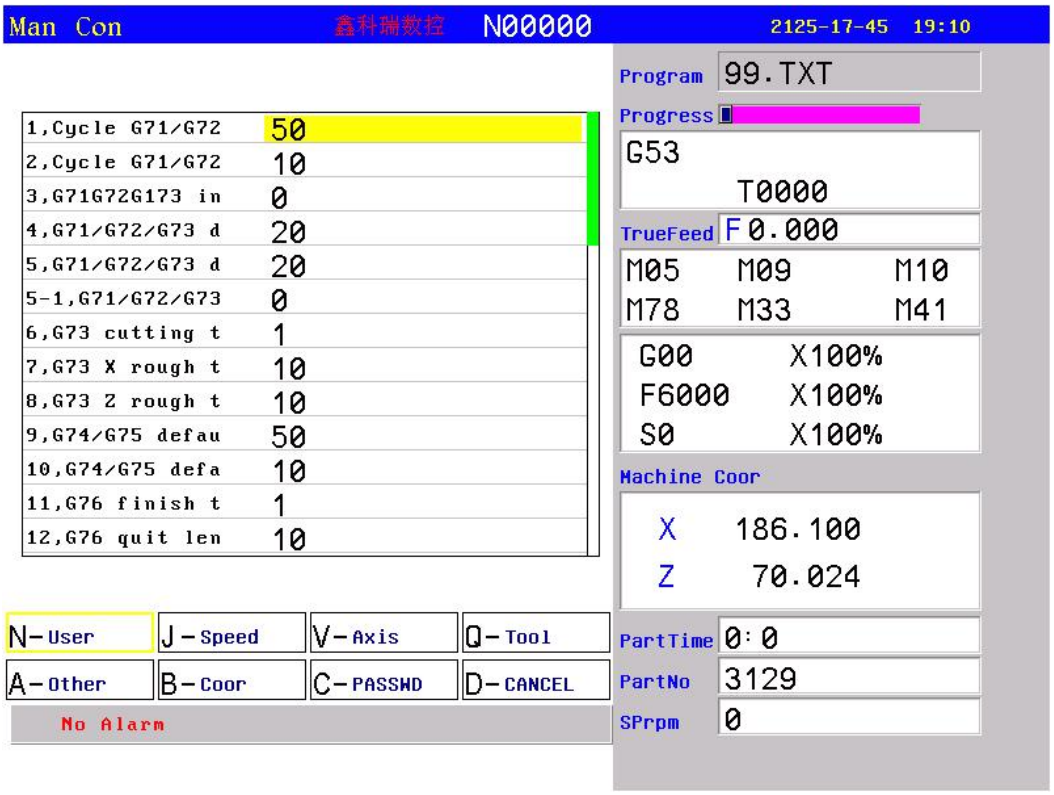
(6) No.5 alarm for door switch: The alarm signal of M12(door switch) is effective.

(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 N J V Q A B C correspondingly to enter corresponding interface after enter the parameter interface, press “↑ ↓” to select the number of parameter and press “Enter” to prompt 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 G71/G72 default feed thickness(10um)
[X axis radius]
Cycle G71 X axis feed thickness; Cycle G72 Z axis feed thickness;
2, Cycle G71/G72 default backward distance(10um)
[X axis radius]
Cycle G71 X axis backward thickness; Cycle G72 Z axis backward thickness.
3, G71 G72 G173 instruction

[1 mean Yes, 0 mean No]

"1" mean G71/G72/G73 instruction finish machining.

D2: auto machining, D3: G74 X fast speed,D4: Z fast speed, D5: auto direction, D6: auto point, D7: G70 no home, D8: tool radius.

4, G71/G72/G73 default X remain (10um)

5, G71/G72/G73 default Z remain (10um)

6, G73 cutting times

7, G73 X rough thickness (10um)

[X axis radius]

8, G73 Z rough thickness (10um)

9, G74/G75 default feed thickness (10um)

[X axis radius]

10, G74/G75 default backward distance (10um)

[X axis radius]

11, G76 finish turn times

12, G76 quit length (1/10 lead)

13, G76 thread tooth angle (degree)

[0～180°]

14, G76 minimal cutting depth (10um)

[X axis radius]

15, G76 finish turn remaining (10um)

16, X program mode

[1 mean Radius, 0 mean Diameter]

17, Running program need Sp run

[1 mean Yes, 0 mean No]

17-1,Running program need Sp M10

[1 mean Yes, 0 mean No]

18, Set M20 the time of auto-running

[Negative mean immensity loop]

20, G92 quit length (1/10 lead)

21, G01/G02/G03 line delay (ms)[>100]

22, G00 line delay (ms) [>100]

23, Handwheel acceleration [50-100]

24, G76 Q

[8 mean thick forward number]

33, M03/M04/S detect the spindle speed (0:M69 relay, 8:encoder feedback)

34,The error(RPM) of the spindle speed is detected by the encoder feedback

200, system screen protect times

[>=2minutes]

201, G92/G76 delay time (ms) [>100]

202,System inner parameter

203Enable pause key (6326274: No, 23103490: Yes)

210,Enable graphics display area (8: manual, 0: automatic)

211, X axis negative end in graphics

212, X axis positive end in graphics
213, Y axis negative end in graphics
214, Y axis positive end in graphics
215, Z axis negative end in graphics
216, Z axis positive end in graphics
230, Select executive program through input (+4.....+128, +256.....+32768: correspond to X26-X31, X16-X23)
231, “Delete” Mode [0:backward deletion, 1: Forward deletion]
232, whether detect the home of spindle before tapping
[18: Yes, 0: No]
233, G06 circle teaching function
[0:No, 1: Yes]
234, Program back function
[+8: enable by handwheel]
307,M18xx/M28xx/WAT alarm time(ms)[>=0]
400,Translate DXF file to G code[1:Sequencing, 4:Start point sequencing, 8:No sequencing]
401, Translate DXF file to G code z-axis coordinate
402, Translate DXF file to G code x-axis coordinate
500, G74 equal to programG No.[101-170(101-15Modeless, 151-170Mode)]
501, G81 equal to programG No.[101-170(101-15Modeless, 151-170Mode)]
502, G82 equal to programG No.[101-170(101-15Modeless, 151-170Mode)]
503, G83 equal to programG No.[101-170(101-15Modeless, 151-170Mode)]
504, G84 equal to programG No.[101-170(101-15Modeless, 151-170Mode)]

3.7.2 Speed parameter

1,X-axis's G00 speed(mm/min)
2,Z-axis's G00 speed(mm/min)
3,Manual maximum feed speed(mm/min)
4,Auto Maximum feed speed(mm/min)
5,G01/G02/G03 default speed(mm/min)
6,Null run speed(mm/min)
7,Feed axis`s manual speed(mm/min)
8,Spindle`s manual speed(rpm)
9,Beginning feed speed(mm/min)
10,Break variable of feed speed(mm/min)
11,Limit G1G2G3 axis speed
[1 mean Yes,0 mean No]
12,X G1G2G3 max speed(mm/min)
13,Z G1G2G3 max speed(mm/min)
14,X acceleration
[1～99999]
15,Z acceleration
[1～99999]

16,Auto run acceleration
[1-500]
17,Handwheel acceleration
[500--30000]
18,Run program Handwheel acceleration
[>500]
19,Run program Handwheel G00 speed(mm/min)
[>10]
19-1, Run program handwheel G00 start speed(mm/min)[>5]
20,Handwheel X limit speed(mm/min)
21,Handwheel Z limit speed(mm/min)
22,Make thread Z acceleration [Servo motor is 0]
23,Make thread X acceleration [Servo motor is 0]
24,Servo motor screw thread X axis Back speed
25,Step motor screw thread X axis Back rise speed
26,Step motor screw thread X axis Back Max speed
27,acceleration type [0 mean line,8 mean curve]
28,curve initial acceleration [>=10]
29,curve acceleration [>=10]
30,curve max acceleration [>=500]
31,X go home forward speed(mm/min)
32,X go home reverse speed(mm/min)
33,Z go home forward speed(mm/min)
34,Z go home reverse speed(mm/min)
35,G96 spindle min speed(rpm)
36,Spindle first max speed(rpm)
37,Spindle second max speed(rpm)
38,Spindle third max speed(rpm)
39,Spindle forth max speed(rpm)
40,Second Spindle max speed(rpm)
41,G02/G03 backlash compensation mode
0 means A mode: the bigger the backlash is, the faster the speed is;
+8 means B mode: parameter define the compensation speed;
+4 means Arc IJK is the coordinate from endpoint to center.
42,mode B reverse compensation speed(mm/min)
42-1,mode B reverse compensation Beginning feed speed(mm/min)[>10]
42-2,mode B reverse compensation acceleration(mm/min/s)[>10]
43,Handwheel stop speed(mm/min)[>100]
49,Speed Mode[1 Yes,0 No]
51,the spindle speed when following tapping retracting(rpm) [>1]
52, spindle backlash compensation when tapping(Pulse)
53,value of advanced retracting before following tapping spindle reversal(um) [10-5000]
54,Tapping back speed(mm/min)[>=60]

58,Hard limit drop speed critical(mm/min)
60,Smoothing function(+4: manual, +8: handwheel, +16: program)
61,manual smoothing time constant[2-50]
62,handwheel smoothing time constant[2-50]
63,program smoothing time constant[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]
111,X jump speed at continuous track (mm/min)[>1]
112,Y jump speed at continuous track (mm/min)[>1]
113,Z jump speed at continuous track (mm/min)[>1]
200,G00 continue run is valid[1 means No, 16 means Yes]
210,Thread is waiting for the spindle speed[0 means No. 1 means Yes]
230,SP G00 speed(0.1rpm)
231,SP G01 mode[+4:F, +8: G90/G91, +16: show value]
232,SP direction[0: Positive, 1:Negative]
233,SP back to zero mode(1:pulse; 2:output M61, check M22)
234,SP stop angle(0.1)
235,SP back to zero speed(0.1rpm)

3.7.3 Axis parameter

1,Feed axis band switch
[1 mean Yes,0 mean No]
2,Spindle and G00 band switch
[1 mean Spindle,2 mean G00, 0 mean No]
3,X-axis`s negative scope(mm)
4,X-axis`s positive scope(mm)
5,Z-axis`s negative scope(mm)
6,Z-axis`s positive scope(mm)
7,Spindle stop time(10ms)
8,Spindle stop long signal
[0 mean No,1 mean Yes]
9,Check SP encode[1 mean Yes, 0 mean No]
10,SP encode pulse [4 times encode thread]
10-1,SP encode pulse alarm(diagnostics)[>10 mean valid]
10-2,SP encode pulse[4 times encode thread](valid when Encode:SP is not 1:1 and set integer >99)
11,Soft limit invalid
[D2X;D3C(Y);D4Z;D5A;1 invalidation,0 validation, D12=1 means MOVE unlimit,D14=1 means tool changing unlimit]
12,X-axis`s reverse compensation(um)[radius]
13,Z-axis`s reverse compensation(um)[radius]
14,X-axis's direction signal
[1 mean normal,0 mean reverse]
15,Z-axis's direction signal
[1 mean normal,0 mean reverse]
16,Close feed electron gear
[1 mean Yes,0 mean No]
17,X-axis's electron gear numerator(1-999999)

18,X-axis's electron gear denominator(1-999999)
19,Z-axis's electron gear numerator(1-999999)
20,Z-axis's electron gear denominator(1-999999)
21,XZ positive limit
[0 open,1 close]
22,XZ negative limit
[0 open,1 close]
23,float zero bit parameter
[D3X;D4C(Y);D5Z;D6A;0 machine Zero;1 float Zero]
24,X coor float zero set
25,Z coor float zero set
26,Feed axis home[1 mean No use, 0 mean clew, 8 mean compulsion, 9 mean must compulsion]
27,Feed axis home mode [0 mean reverse check, 1 mean reverse no check, 2 mean no reverse check, 3 mean no]
28,Home reverse direction[D2 X; D3 C(Y); D4 Z; D5 A; D8=1 first Z; 0 mean positive; 1 mean negative]
29,Home switch set [D0 X; D1 C(Y);D2 Z;D3 A;1 mean close, 0 mean open]
30,X check zero max length(100um)[radius]
31,Z check zero max length(100um)[radius]
32,X home offset(10um)
33,Z home offset(10um)
50,Whether spindle run when shifting [1 mean run, 0 mean not, 64 mean keep the status of spindle]
51Shifting Spindle speed(1/100rpm)
52,Shifting Spindle direction
[0 mean M03,1 mean M04]
53,Pause time when spindle shifting(10ms)
54,Shifting Spindle low-speed time(10ms)
55,Spindle stop time(10ms)
56,Spindle manual point M04[8 means M04]
68,XYZA reverse delay time(10ms)
80,XZ axis coordinate plan [D2 Zwordpiece, D3 Xwordpiece,D4 Ztool,D5 Xtool, D6 Zcircumrotate,D7 Xcircumrotate]
109,C(Y) G1G2G3 max speed(mm/min)
111, Handwheel C(Y) limit speed(mm/min)
117,C(Y) axis's negative scope(mm)
118,C(Y) axis's positive scope(mm)
119,C(Y) coor float zero set
200,system inner parameter
201,lathe A axis
[0 mean circumrotate axis,1 mean line axis]
202,lathe A is circumrotate axis
[0 null;1 absolute coordinate plan;2 tool coordinate plan;3 all]
203,A motor direction(0 reverse,1 normal)
204,A-axis's electron gear numerator(1-999999)
205,A-axis's electron gear denominator(1-999999)
206,A-axis`s reverse compensation(um)
207,A G00 speed (mm/min)
208,A G1G2G3 Max speed(mm/min)
209,A acceleration
210,Handwheel A limit speed(mm/min)
211,A go home forward speed(mm/min)

212,A go home reverse speed(mm/min)
213,A check zero max length(100um)
214,A Home offset(10um)
215,A-axis`s negative scope(mm)
216,A-axis`s positive scope(mm)
217,A coor float zero set
404,SP motor direction(0 mean reverse, 1 mean normal)
405,SP-axis's electronic gear(0 mean yes, 1 mean no)
406,SP-axis's electronic low gear numerator (1-999999)
407,SP-axis's electronic low gear denominator(1-999999)
408,SP-axis's electronic high gear numerator(1-999999)
409,SP-axis's electronic high gear denominator(1-999999)
410,Interpolation tap SP name(91 mean X, 92 mean Y/C, 93 mean Z, 94 mean A)
411,Interpolation tap mode[0 mean follow encoder; 4 mean interpolation to SP]
412,SP tooth number(<P413)
413,Encoder number(>P412)
414,A-axis is moving by(7/8/9,17/18/19, 27/28/29/D5..D13=1/A by x/y/z, C by X/Y/Z, B by X/Y/Z)

3.7.4 Tool parameter

1,Active tool function
[1 mean Yes ,0 mean No]
2,Active tool number +1
3,Lathe type[0 mean forward and horizontal, 1 mean behind horizontal, 8 mean forward vertical, 9 mean behind vertical]
4,Tool positive rotate max-time(s)
5,Delay time after tool positive rotate(ms)
6,Delay time after tool stop(ms)
7,Tighten time of tool reverse rotate(ms)
9,Have total signal TOK(1 mean yes, 0 mean no)
10,C Tool radius compensation's establish(0 mean A,1 mean B)
11,C Tool radius compensation's cancel(0 mean A,1 mean B)
20,Active tool mode
[1 mean normal,0 mean coding tool]
32,Tool position signal/WAT signal filter[+256+512+1024 correspond to 2/4/8ms of signal of tool position, +2048+4096+8192 correspond to WAT signal 2/4/8 ms]

3.7.5 Other parameter

1,Set sub-panel type
[1 hand hold,0 panel]
2,lathe outside chuck
[1 Extroversion,0 introversion]
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
D1："1" to clear.;
D2："1" blank;
D5："0"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;
D12："1" Shield skip function ("/"is invalidation);
D13："1" Shield go home function;
10,Auto count part
[1 mean Yes,0 mean No]
11,Program edit number increment
12,Inner parameter
13,Does interlock between Spindle&Chuck(0 mean no, 1 mean half lock, 8 mean all lock)
13-1,Does lock for Spindle & Thimble (0 mean unlock, 1 mean half lock, 8 mean all lock, 16 mean super all lock)
14,Is available keys of lub&cool as running(0 mean no, 1 mean yes)
15,Chuck clamp M10/loose M11 checking(1 mean need)
16,Thimble forward M79/backward M78 checking(1 mean need)
17,servo ALM1 (0 open,1 close)
18,SP ALM2 (0 open,1 close)
19,Tool ALM3 (0 open,1 close)
20,Chuck control signal(0 single,1 doubleM10/M71)
21,Thimble control signal(0 single,1 doubleM79/M73)
22,Outside chuck control(0 no,1 yes)
23,Outside thimble control(0 no,1 yes)
24,M10M11 short signal time(s)
24-1,M10 is long or short signal[0 mean short, 1 mean long]
24-2,M71 is long or short signal[0 mean short, 1 mean long]
24-3,Chuck M10 OR M11 boot[0 mean M10, 1 mean M11]
25,M79M78 short signal time(s)
25-1,M79 long signal[0 mean no, 1 mean yes]
25-2,M73 long signal[0 mean no, 1 mean yes]
26,Emerge Stop(0 open,1 close)
27,Emerge Stop2(0 open,1 close)
28,Run status output [M69:running, M65:stop](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[1:enable, 0: disable, 32:valid output point&plc when

diagnosis,64: valid output point& invalid plc when diagnosis]
32, High speed I/O PLC program[0 means no, 18 means high speed, 28 mean super high speed]
33,HY running signal
[1 mean Yes,0 mean No]
34,HA stop signal
[1 mean Yes,0 mean No]
35,Valid manual soft limit without home
[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,Lock Manual rapid function key
[8 Yes]
39,Special parameter
40,Special parameter
41,Bake current parameter
42,Resume original parameter
50,Locate point to the last segment before start from middle[8 mean yes, 0 mean no]
120,Manual axis keying reverse(4 mean X in vertical/Z in horizontal; 16 mean Z in vertical/X in horizontal; 32 mean the forth axis)
200.X axis feedback the alarm error. (pulse)[>1]
201.Y axis feedback the alarm error. (pulse)[>1]
202.Z axis feedback the alarm error. (pulse)[>1]
203.Fourth axis feedback the alarm error. (pulse)[>1]
205.X axis feedback stop alarm error. (pulse)[>1]
206.Y axis feedback stop alarm error. (pulse)[>1]
207.Z axis feedback stop alarm error. (pulse)[>1]
208.Fourth axis feedback stop alarm error. (pulse)[>1]
210.X axis feed back electrical gear numerator.
[Automatical calculate and import: L crew pitch (um) and M the number of encoder.]
211.Y axis feed back electrical gear numerator.
[Automatical calculate and import:L crew pitch(um) and M the number of encoder.]
212.Z axis feed back electrical gear numerator.
[Automatical calculate and import: L crew pitch (um) and the number of encoder.]
213.Fourth axis feed back electrical gear numerator.
[Automatical calculate and import: L crew pitch (um) and the number of encoder.]
215.X axis feed back electrical gear denominators.
[Automatical calculate and import:L crew pitch(um) and M the number of encoder.]
216.Y axis feed back electrical gear denominators.
[Automatical calculate and import:L crew pitch(um) and M the number of encoder.]
217.Z axis feed back electrical gear denominators.
[Automatical calculate and import:L crew pitch(um) and M the number of encoder.]

218.Fourth axis feed back electrical gear denominators.
[Automatical calculate and import: L crew pitch(um) and M the number of encoder.]
300.Whether feed axis matches absolute encoder
[X—D2, Y—D3,Z—D4,A—D5, “0” means No, “1” means Yes.]
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.pulse number of X axis absolute encoder per cycle.
305.pulse number of Z axis absolute encoder per cycle.
306.pulse number of Z axis absolute encoder per cycle.
307.pulse number of the fourth axis absolute encoder per cycle.
309.X-axis one-revolution coordinate(nm).[nm]
310.Y-axis one-revolution coordinate(nm).[nm]
311.Z-axis one-revolution coordinate(nm).[nm]
312.Fourth axis one-revolution coordinate.[nm]
314.X axis of absolute encoder multi-turn offset.
[Import “E” to clear multi-turn data]
315.Y axis of absolute encoder multi-turn offsetg.
[Import “E” to clear multi-turn data]
316.Z axis of absolute encoder multi-turn offset.
[Import “E” to clear multi-turn data]
317.Fourth axis of absolute encoder multi-turn offset.
[Import “E” to clear multi-turn data]
320. The denominator of X-axis one revolution corresponding coordinate distance.
321. The denominator of Y-axis one revolution corresponding coordinate distance.
322. The denominator of Z-axis one revolution corresponding coordinate distance.
323. The denominator of 4-axis one revolution corresponding coordinate distance.
500,Machine tool number.
501,Change the color of interface(1 is white,8 is black)
601.One key to define stepper.
602.One key to define servo.
900,The user-defined dialog of processing[1 mean invalid, 4 mean part of, 8 mean all]
901,The sequence of backing to zero(>9)[total 5 bits, first 4 bits 1/2/3/4 correspond to X/Y/Z/A, the last is 0]
910,M18/M22/M24/M28 is high-speed input[1 mean yes]
911,M18 is teaching, M24 is middle recording and M28 is end recording[1 mean yes, 0 mean no]
912,Press “Reset” to reset the output[1 mean yes, 0 mean no]
994,CNC Controller serial number
995,Machine tool serial number
996,user-define serial number

3.7.6 Coordinate

1-0,The 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)
2-0,Current set of workpiece coordinate[G54.1-G54.48]
2-1,X of workpiece coordinate[G54.1-G54.48](mm)
2-1,Y of workpiece coordinate[G54.1-G54.48](mm)
2-3,Z of workpiece coordinate[G54.1-G54.48](mm)
2-4,A of workpiece coordinate[G54.1-G54.48](mm)
1,X of work coordinates G54(mm)[Incremental input, E entry input means absolute]
2,C(Y) of work coordinates G54(mm)[Incremental input,E entry input means absolute]
3,Z of work coordinates G54(mm)[Incremental input, E entry input means absolute]
4,A of work coordinates G54(mm)[Incremental input, E entry input means absolute]
6,X of work coordinates G55(mm) [Incremental input, E entry input means absolute]
7,C(Y) of work coordinates G55(mm)[Incremental input,E entry input means absolute]
8,Z of work coordinates G55(mm) [Incremental input, E entry input means absolute]
9,A of work coordinates G54(mm)[Incremental input,E entry input means absolute]
11,X of work coordinates G56(mm) [Incremental input, E entry input means absolute]
12,C(Y) of work coordinates G56(mm)[Incremental input,E entry input means absolute]
13,Z of work coordinates G56(mm) [Incremental input, E entry input means absolute]
14,A of work coordinates G56(mm)[Incremental input,E entry input means absolute]
16,X of work coordinates G57(mm) [Incremental input, E entry input means absolute]
17,C(Y) of work coordinates G57(mm)[Incremental input,E entry input means absolute]
18,Z of work coordinates G57(mm)[Incremental input, E entry input means absolute]
19,A of work coordinates G57(mm)[Incremental input,E entry input means absolute]
21,X of work coordinates G58(mm) [Incremental input, E entry input means absolute]
22,C(Y) of work coordinates G58(mm)[Incremental input,E entry input means absolute]
23,Z of work coordinates G58(mm) [Incremental input, E entry input means absolute]
24,A of work coordinates G58(mm)[Incremental input,E entry input means absolute]
26,X of work coordinates G59(mm) [Incremental input, E entry input means absolute]
27,C(Y) of work coordinates G59(mm)[Incremental input,E entry input means absolute]
28,Z of work coordinates G59(mm) [Incremental input, E entry input means absolute]
29,A of work coordinates G59(mm)[Incremental input,E entry input means absolute]
41,X of work coordinates G54.1(mm)[Incremental input, E entry input means absolute]
42,C of work coordinates G54.1(mm)[Incremental input,E entry input means absolute]
43,Z of work coordinates G54.1(mm)[Incremental input, E entry input means absolute]
44,A of work coordinates G54.1(mm)[Incremental input, E entry input means absolute]
46,X of work coordinates G54.2(mm)[Incremental input, E entry input means absolute]
47,C of work coordinates G54.2(mm)[Incremental input,E entry input means absolute]
48,Z of work coordinates G54.2(mm)[Incremental input, E entry input means

absolute]
49,A of work coordinates G54.2(mm)[Incremental input, E entry input means absolute]
51,X of work coordinates G54.3(mm)[Incremental input, E entry input means absolute]
52,C of work coordinates G54.3(mm)[Incremental input,E entry input means absolute]
53,Z of work coordinates G54.3(mm)[Incremental input, E entry input means absolute]
54,A of work coordinates G54.3(mm)[Incremental input, E entry input means absolute]
56,X of work coordinates G54.4(mm)[Incremental input, E entry input means absolute]
57,C of work coordinates G54.4(mm)[Incremental input,E entry input means absolute]
58,Z of work coordinates G54.4(mm)[Incremental input, E entry input means absolute]
59,A of work coordinates G54.4(mm)[Incremental input, E entry input means absolute]
61,X of work coordinates G54.5(mm)[Incremental input, E entry input means absolute]
62,C of work coordinates G54.5(mm)[Incremental input,E entry input means absolute]
63,Z of work coordinates G54.5(mm)[Incremental input, E entry input means absolute]
64,A of work coordinates G54.5(mm)[Incremental input, E entry input means absolute]
66,X of work coordinates G54.6(mm)[Incremental input, E entry input means absolute]
67,C of work coordinates G54.6(mm)[Incremental input,E entry input means absolute]
68,Z of work coordinates G54.6(mm)[Incremental input, E entry input means absolute]
69,A of work coordinates G54.6(mm)[Incremental input, E entry input means absolute]
71,X of work coordinates G54.7(mm)[Incremental input, E entry input means absolute]
72,C of work coordinates G54.7(mm)[Incremental input,E entry input means absolute]
73,Z of work coordinates G54.7(mm)[Incremental input, E entry input means absolute]
74,A of work coordinates G54.7(mm)[Incremental input, E entry input means absolute]
76,X of work coordinates G54.8(mm)[Incremental input, E entry input means absolute]
77,C of work coordinates G54.8(mm)[Incremental input,E entry input means absolute]
78,Z of work coordinates G54.8(mm)[Incremental input, E entry input means absolute]
79,A of work coordinates G54.8(mm)[Incremental input, E entry input means absolute]
81,X of work coordinates G54.9(mm)[Incremental input, E entry input means absolute]

82,C of work coordinates G54.9(mm)[Incremental input,E entry input means absolute]
83,Z of work coordinates G54.9(mm)[Incremental input, E entry input means absolute]
84,A of work coordinates G54.9(mm)[Incremental input, E entry input means absolute]
86,X of work coordinates G54.10(mm)[Incremental input, E entry input means absolute]
87,C of work coordinates G54.10(mm)[Incremental input,E entry input means absolute]
88,Z of work coordinates G54.10(mm)[Incremental input, E entry input means absolute]
89,A of work coordinates G54.10(mm)[Incremental input, E entry input means absolute]

3.7.7 Password

password setting includes:
1,Is enable CNC Co.'s password ?(1 means No, 0 means Yes)
2,Is enable Machine Co.'s password ?(1 means No, 0 means Yes)
Original password is "NEWNEW".
3,Is enable User's password ?(1 means No, 0 means Yes)
Original password is "KERKER".
4,Modify CNC Co.'s password:
5,Modify Machine Co.'s password:
6,Modify User's password:
7,Curry Using time: (days)
8.Print screen function
Soft version.
Time

3.8 Set parameter of tool redeem

Press "Redeem" to enter interface of redeem in any interface, including "Radius compensation" "Redeem" "Clear all value" "Clear current value" "Measure tool" "Posit tool" "Set", total 7 functions, correspond to "N" "J" "V" "Q" "A" "B" "C" to enter corresponding interface，press "Esc" to back the primary menu interface.

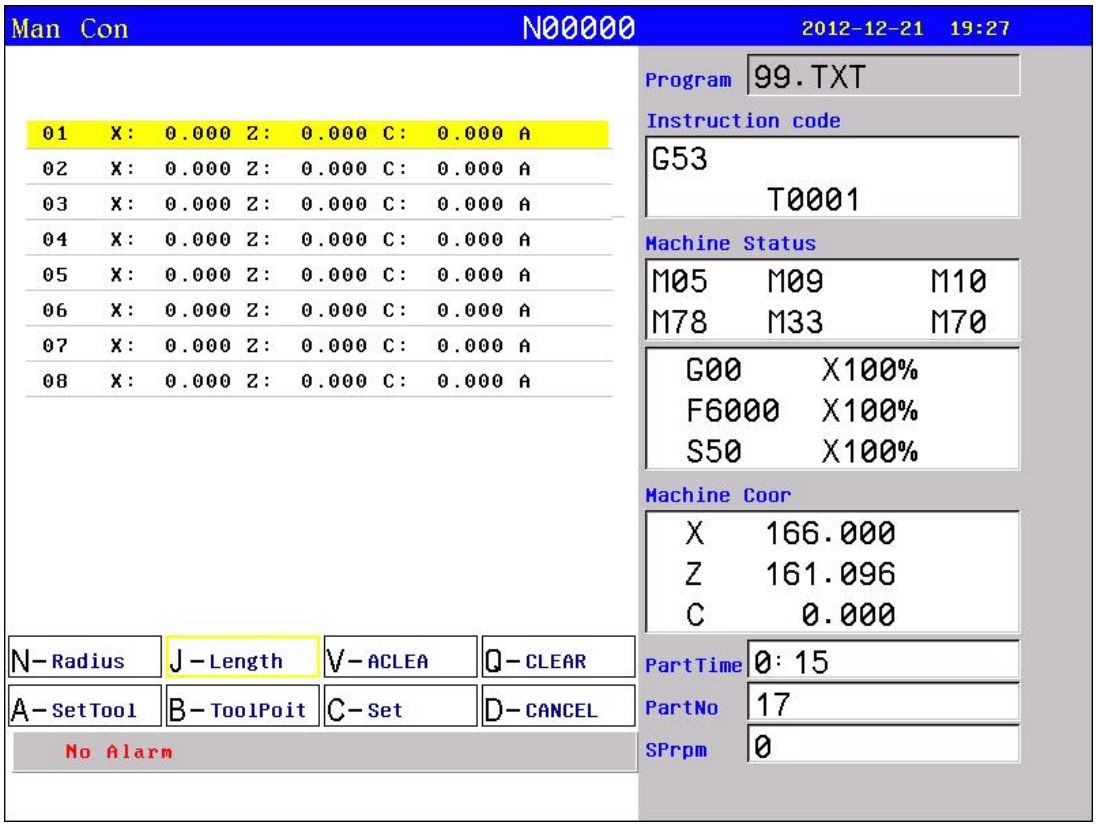
3.8.1 Radius compensation

Press "N" 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, press "Enter" at last.

Press "V" or "Q" to make all initial or current tools to be 0.

Pay attention: Redeem number couldn't correspond to tool, every tool could use any redeem number, radius compensation correspond to redeem number, so the number of redeem is the same as the number of radius compensation.



3.8.2 Length of redeem

Press “J” 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:

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 reset the length:

Make lathe move to a position so that measure corresponding tool coordinate, press “↑ ↓” to make cursor move to corresponding tool number and press “A” 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 of tool:

Press “V” or “Q” to initialize all the length compensation or current length compensation.

3.8.3 Posit tool

Press “B” to enter posit tool interface in redeem interface. The parameter is used to set the kind of tool when adopting radius compensation of tool.

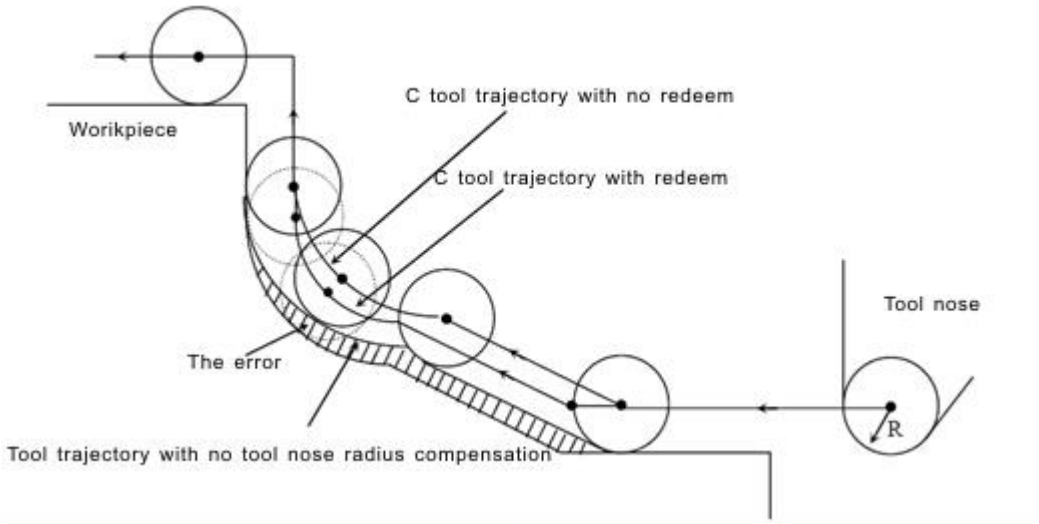
Method of setting: Press “↑ ↓” to make cursor move to corresponding tool number and press “Enter” to popup a dialog box, import the code of corresponding tool kinds

and press “Enter” to confirm.

Press “N” to initialize all the kinds of tools to 0.

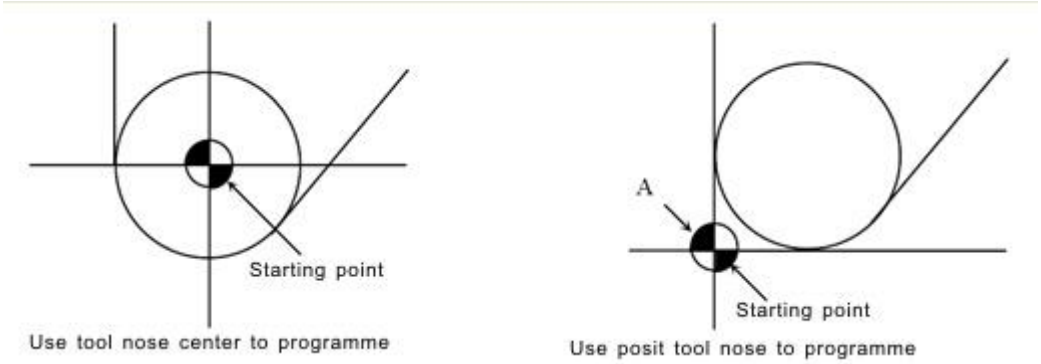
Introduction of posit tool

Only use the function of tool compensation offset when the tool is cycle to make a correct processing program is very difficult. The function of tool compensation could compensate automatically.



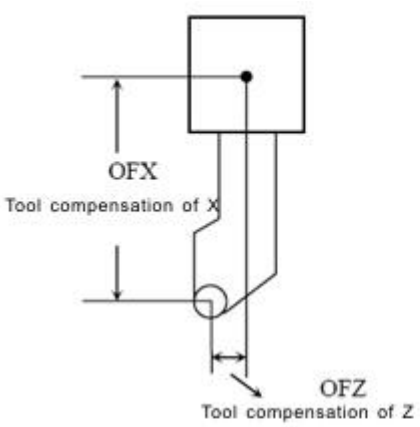
A. Posit tool

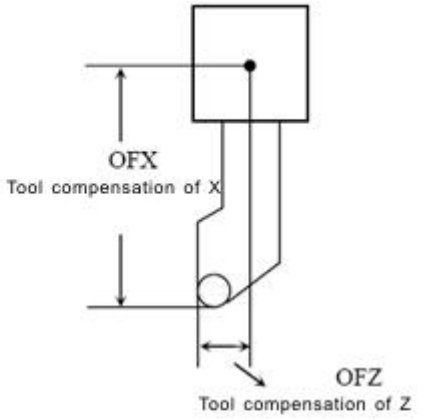
The nose of tool point(A) is non-existent actually as follow picture. Setting posit tool is easier than setting actual center of tool (Hereinafter). Using posit tool to programme do not need to consider the tool radius. The tool in the origin of the position relation as shown below.

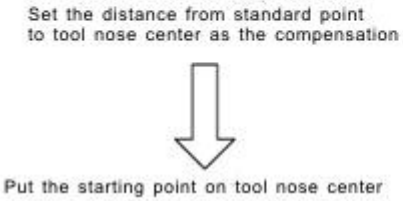


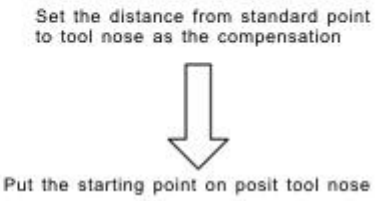
Pay attention: For some lathes with electrical datum point, a standard point(like center nose of tool) can be the starting point. The distance form the standard point to the radius center of tool rest or posit tool can be the offset of tool.

When tool rest is at the beginning point.

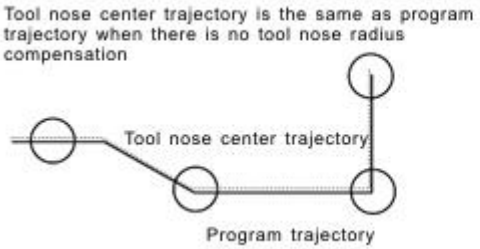


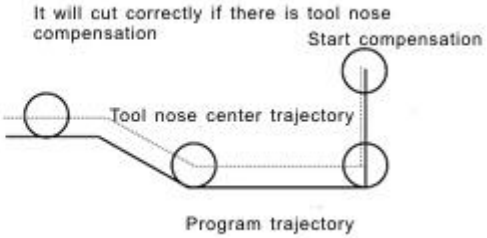




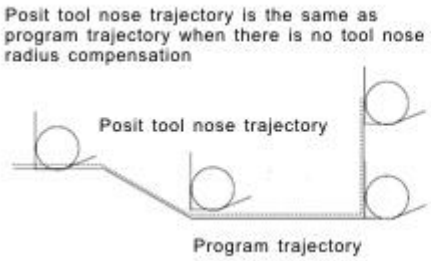


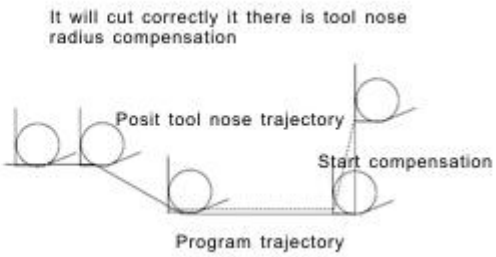
Using the central nose of nose to programme





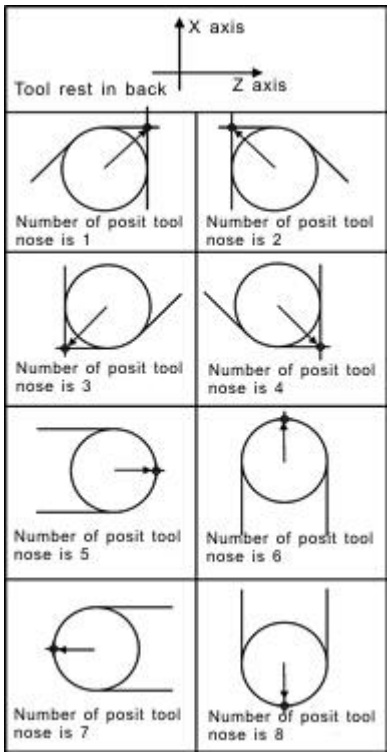
Using the posit tool to programme



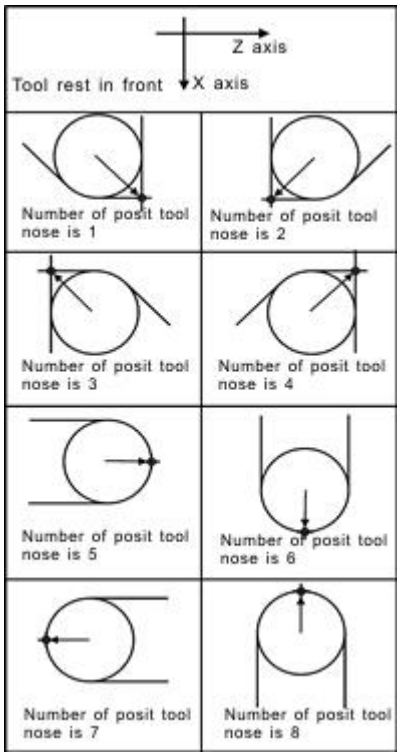


B. Direction of posit tool

From the central nose of tool to see the direction of posit tool is determined by the direction of cutting tool, so must set before as the same time with the value of compensation. The direction of posit tool can be selected by the 8 kinds of corresponding number as shown below.



The direction of posit tool of backward tool rest



The direction of posit tool of forward tool rest

Using No.0 posit tool or No.9 when the central nose of tool is coinciding with starting position. Set the number of posit tool into tool parameter in every number of offset.

3.8.4 Set quantity

Press “C” to popup a dialog box in the redeem interface to set and manage the total tools.

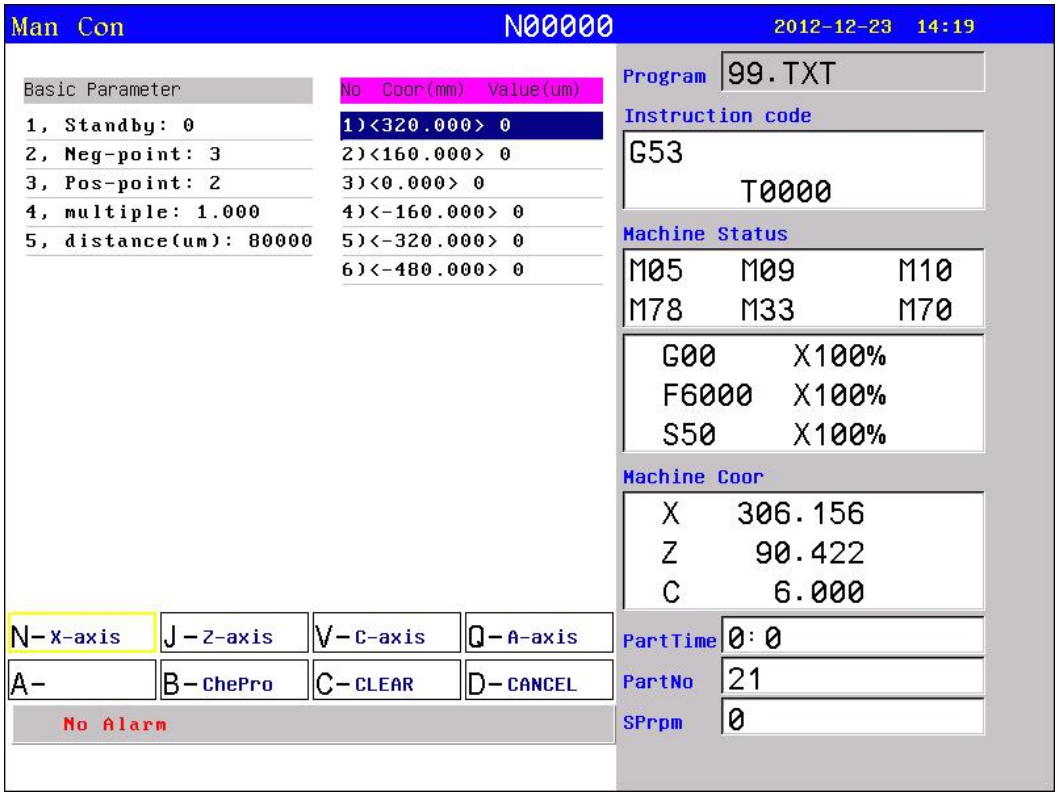
Including sum tools of electrical tool rest and toolpost.

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 interface

Screw compensation by the axis as the unit to set storage, set X Z axis separately, by pressing “N” “J” 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:

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.

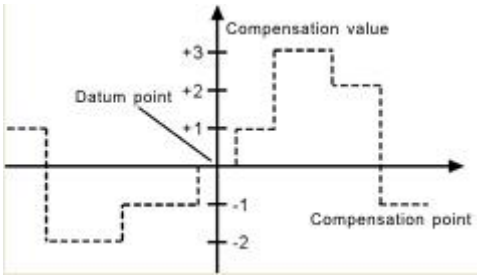
Test program generation automatically

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:

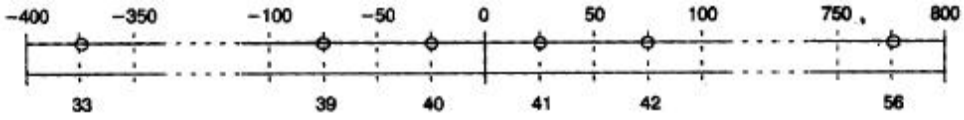
Example 1:Linear axis: when length of travel is -400mm～+800mm, spacing of points is 50mm,reference point compensation isNo.40, it can figure out that Compensation point of farthest end in negative direction is:

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: 30

Compensation point No. of farthest end in positive direction: 56

Compensation override: 1

Compensation point interval: 50000

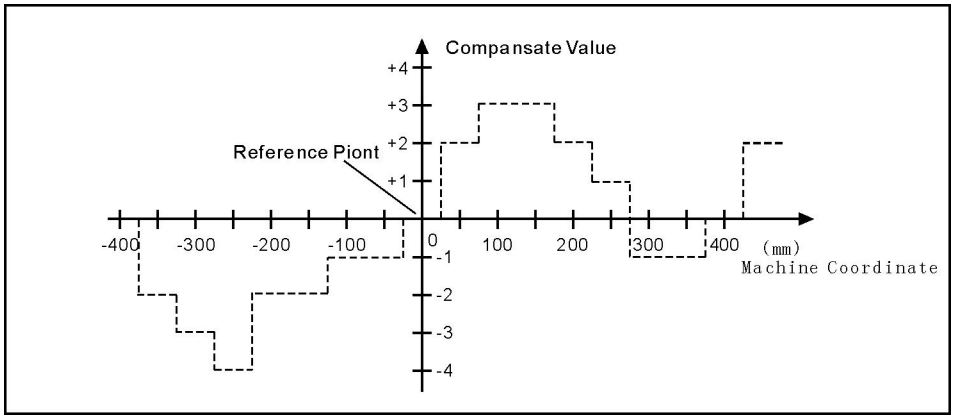
Corresponding compensation point and value:

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:

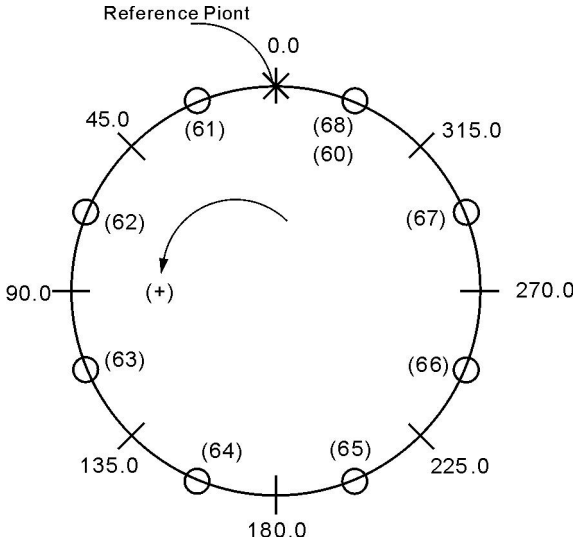


Example 2:rotor axis: when movement per revolution is 360°, interval of points 45°,reference point compensation NO. 60, Compensation point NO. of farthest end in negative direction is usually same as reference point compensation point NO.

Compensation point NO. of farthest end in positive direction is:

Reference compensation point NO.+ movement per revolution/compensation point interval=60+360/45=68.

Machine coordinate and compensation point NO. 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.

Parameter sets as follows:

Compensation point NO. of reference point:60

Compensation point NO. of farthest end in negative direction:60

Compensation point NO. of farthest end in positive direction:68

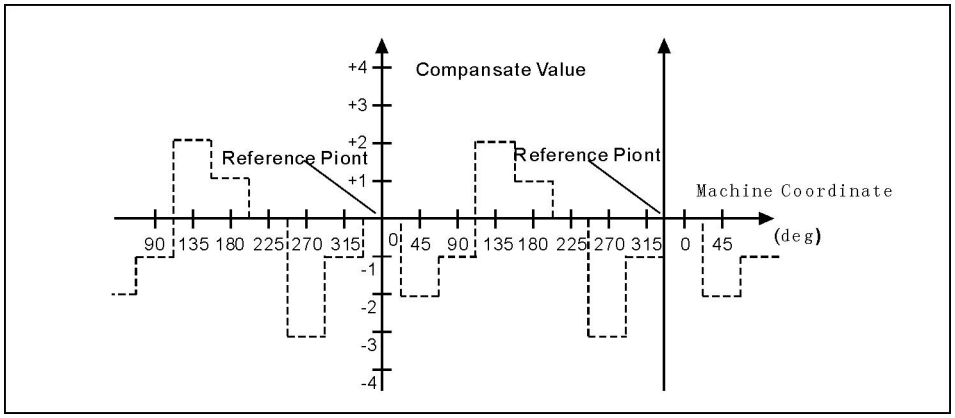
Compensation percentage: 1

Compensation point interval: 45000

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:

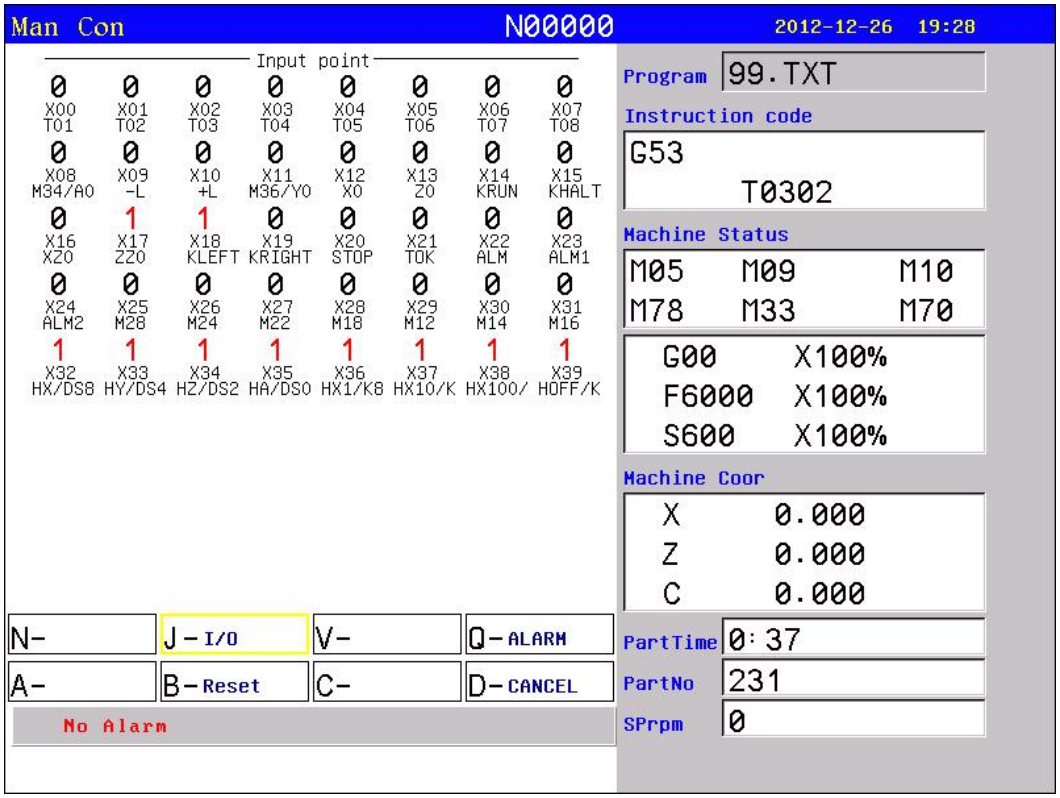


3.10 Diagnosis

Press “Diagnosis” to enter the diagnosis interface in parameter interface.

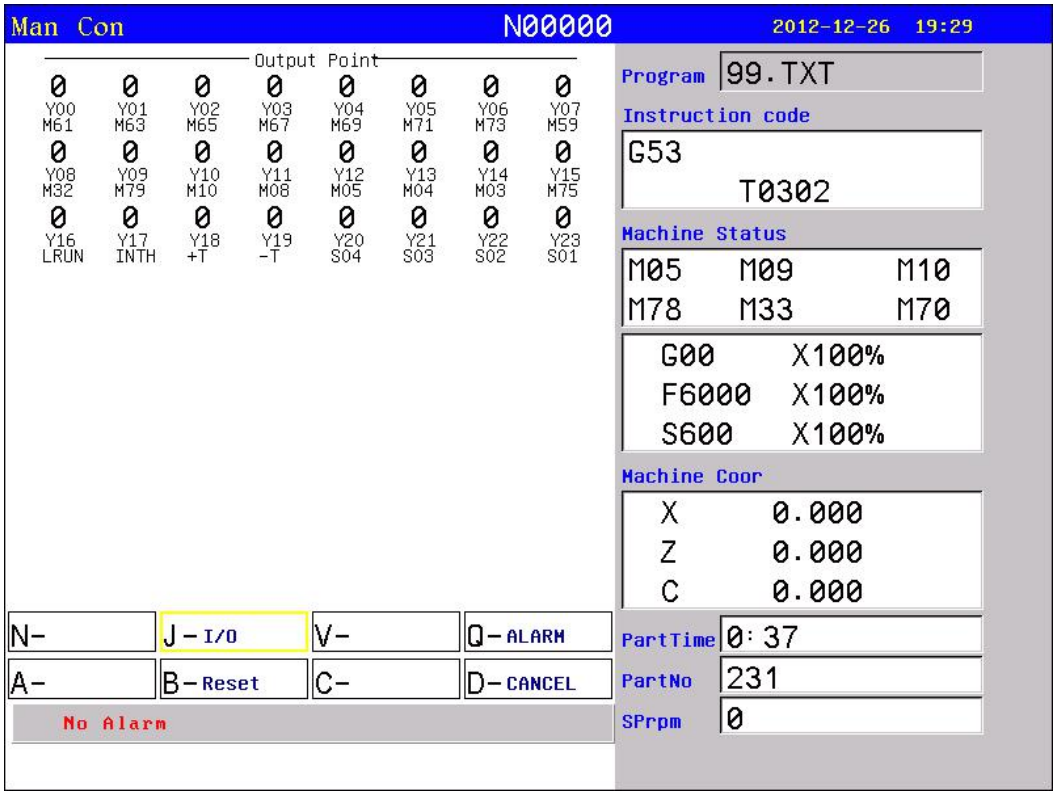
System diagnosis interface(Input signal)

Press “J” and “↑ ↓” to check the status of input and output, press “Q” to check alarm information.



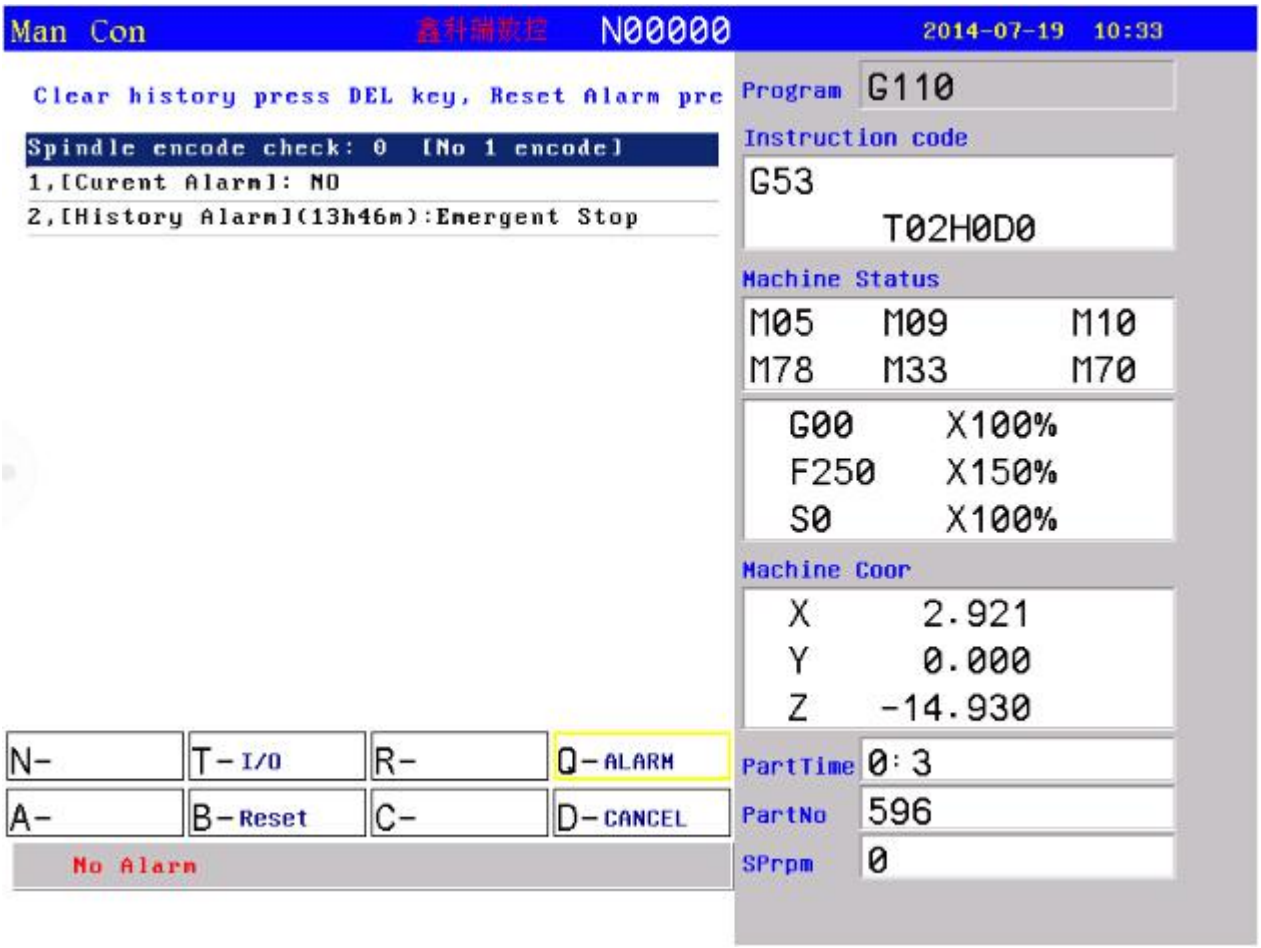
Checking interface of output signal

In the interface of output or input, No.0 or No.1 stands for status, 1 means effective, 0 means no effect.



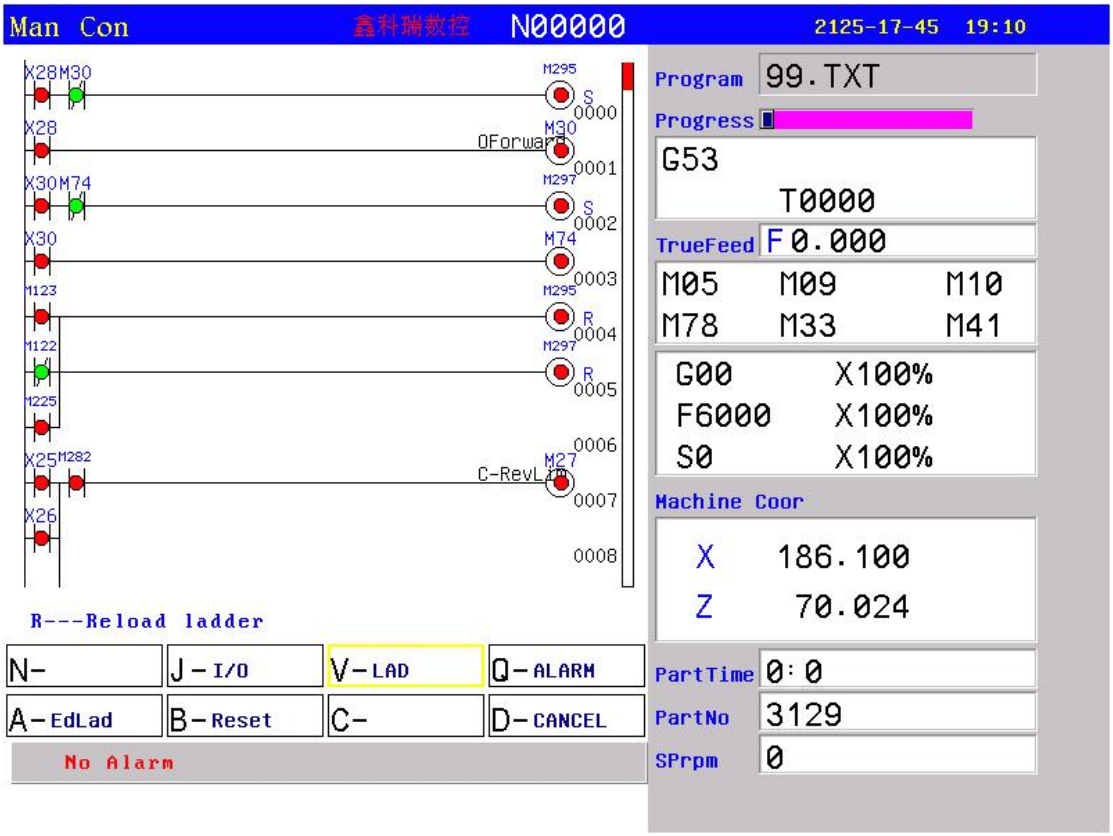
Alarm information interface

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.



PLC checking and editing:

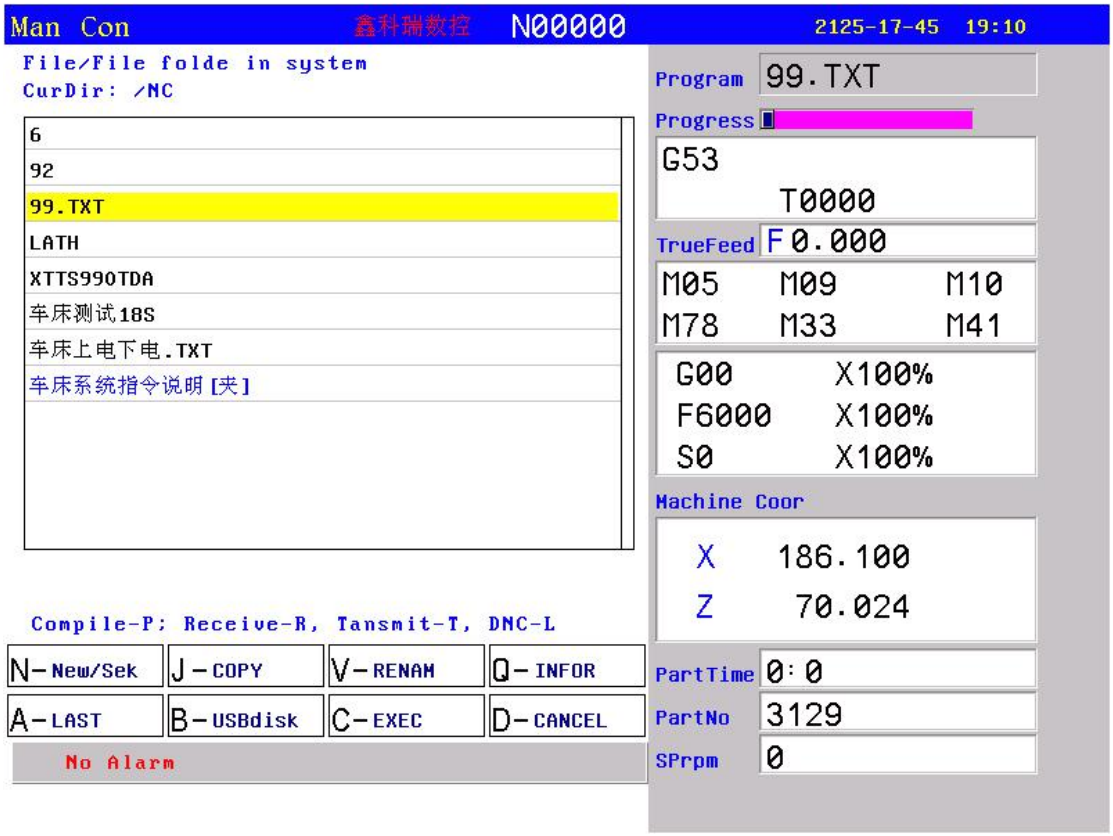
Press F3 to check PLC ladder in diagnosis interface, press F5 to edit PLC ladder. Press “S” to search during checking and diagnosis. After modifying PLC, it is to necessary to restart controller, then new PLC will work. You can also press “R” to validate new PLC without restarting.



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.



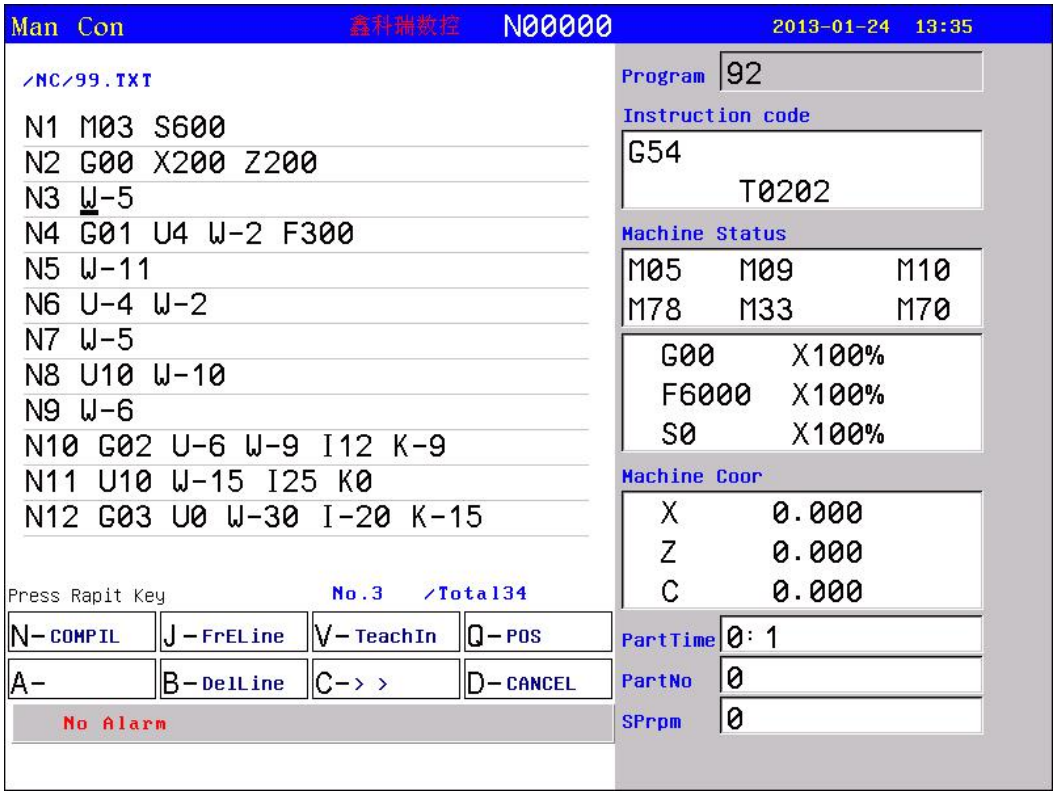
Center part of screen for program display， current program is showed by reverse display, press “PgUp、 PgDn” to choose program, and then press “Enter” to edit current program. Functional keys “N、J、V、Q、A、B、C、D” correspondes：“new file/search” 、“copy” 、“rename” 、“information”、“last grade” “USB disc”、“execute 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.



Prompting name of the program in top left corner of the screen; Left is content of the program; Right is the status information of lathe, to carry out operations as follows in editing status:

(1) Locate the current cursor:

Press " ↑ ↓ ← →" to move the cursor to any position in program content.

Press "PgUp" to move the cursor to last page.

Press "PgDn" to move the cursor to next page.

(2) Modify character: Delete the character in the position of cursor, and import new character directly.

(3) Inset character: Import new character directly in the position of cursor. When importing a letter, producing a space automatically in front. When importing a space, importing a letter first and delete the letter.

(4) Delete character: Press "Del" in the position of cursor directly.

(5) Inset a line between the lines: Press "Enter" directly. Inset a line in front of the current line when the cursor in the first line, otherwise inset a line behind the current line.

(6) "  " superimposing operation:

First page of function:

A. "  "+N: Compile the current program.

B. "  "+J: To the first line or last line of program.

C. "  "+V: The function of determination, enter the status of handwheel;

Press corresponding coordinate letters “X/Z/Y/C/A “  ” +Q(all coordinate axis)” to load coordinate of the current workpiece .

D. “  ” +Q : Locate to the appointed line.

E. “  ” +A: No operations.

F. “  ” +C: To select first page or second page of function.

G. “  ” +D: To switch Chinese and character.

Second page of function:

A. “  ” +N: Delete the appointed part of program.

B. “  ” +J: Copy the appointed part of program.

C. “  ”+V: Sorting the program.

D. “  ” +Q: Search for the appointed character.

E. “  ” +A: Replace the appointed character.

F. “  ” +B: All the content replace the appointed character.

G. “  ” +C: To select first page or second page of function.

H. “  ” +D: To switch Chinese and character.

3.11.2 Copy

Press “ ↑ ↓ ” in program main interface to select program which need to copy and press “J” 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 “V” 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 “Q” 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 "N" 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 "A" go to the last folder.

Move the cursor to the file name and press "Del" to delete the folder.

3.11.8 Select automatical to run program

Press " ↑ ↓ " in program main interface to select a program and press "C" 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.

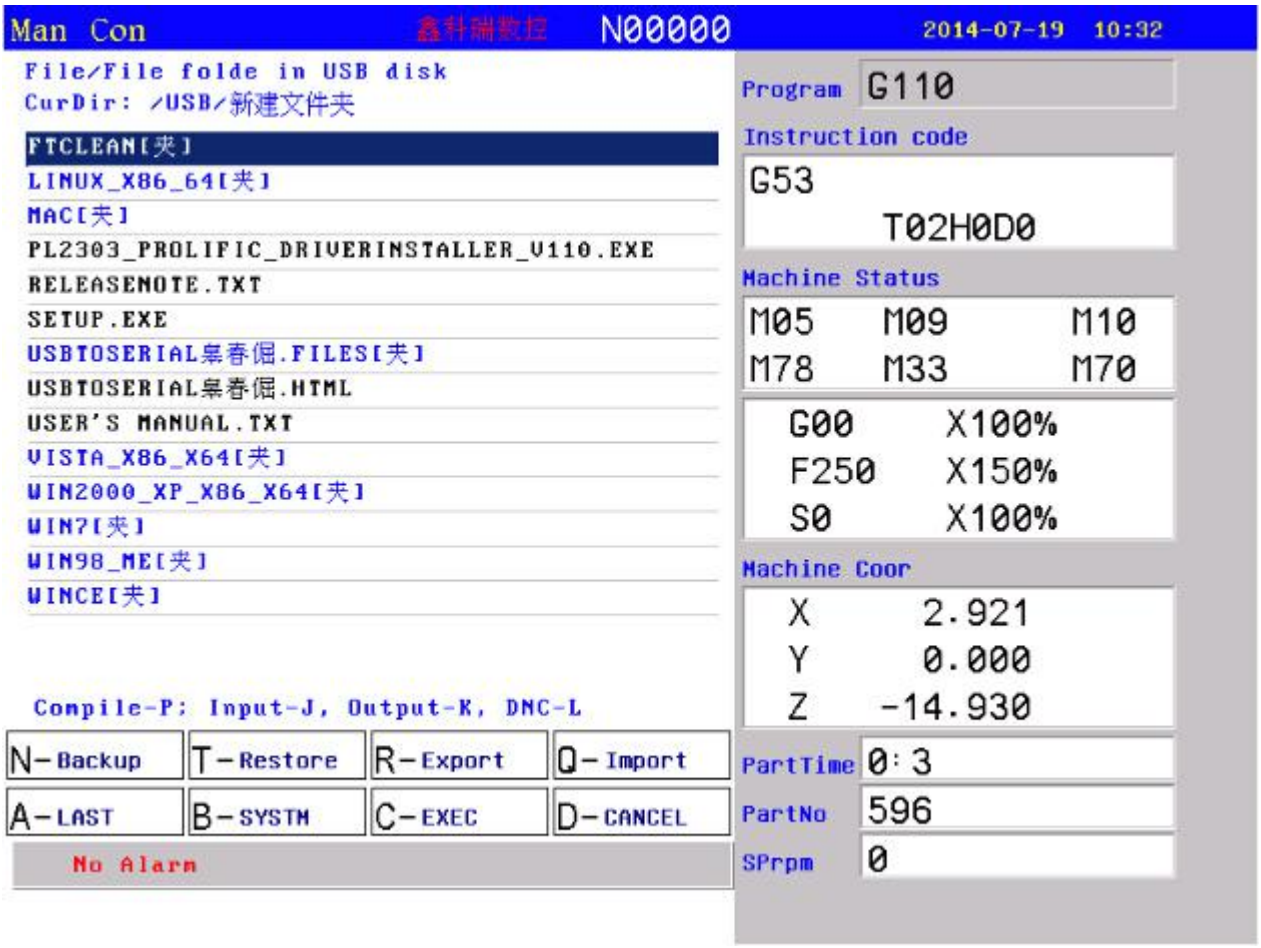
5. Editing program of PC must be text file form.

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 "B" to enter the U-disk management interface in program main interface when U-disk connects the USB port. Press "B" 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 "B" to enter the U-disk directory in program main interface. Press " ↑ ↓ " to move cursor to select file or folder to copy and press "Q" 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.

Copy the files or folder of system into U-disk

Press " ↑ ↓ " to move cursor to select file or folder and press "B", press "V" 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.

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 “B” 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 “J” to import code when appearing the files and press “Enter” to confirm, wait for seconds to copied the parameter successfully. Press “B” 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 “B” to show the files in U-disk. Press “N” 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 “B” 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.

3.11.11 Convert DXF files into G code.

In program directory, choose the DXF file then press”-”, it will be converted into G code file, whose name extension is .CNC. The format of the DXF file input into controller should be .DXF or .dxf. Parameter refers to process parameter No.400-402.

During generation of G code files, system will generate corresponding head and end code according to if there are HEADDXF.TXT and ENDDXF.TXT (or headdxf.txt and enddxf.txt) under current file directory.

Attention: head code file and end code file must be under current file directory.

3.12.12 The operation for folder of FTP server file

If the controller connect with Internet or Wi-Fi, could enter folder of FTP server.

1. Press “N” in program interface.
2. After entering folder of FTP server, press “N”(or F6) as the above operation to quit.

The advantage of FTP: You can see the contents of the PC folder on the controller , and you can choose the files you want to transfer freely, and it is more convenient to use it.

Chapter4 System installation and connection

4.1 System electrical specification

- 32bits high performance industrial grade ARM+DSP+FPGA
- 32M User's storage space
- 800x600 TFT LCD adaptive brightness, LED backlight LCD
- Touch type key board to have excellent operational sensitivity.
- RS232 communication port
- USB port
- Photoelectric encoder
- 4 and 8 position electrical tool rest or tool post.
- 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 four axis
- Linkage axis: X Y Z A four axis
- Pulse equivalency: X axis with 0.001mm, Z axis with 0.001mm
- Maximum speed: 30000mm/min
- Processing speed: 0.01-15000min/min
- Processing screw pitch: 0.1-1000mm in the metric system, 1-99 tooth/inch
- 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.

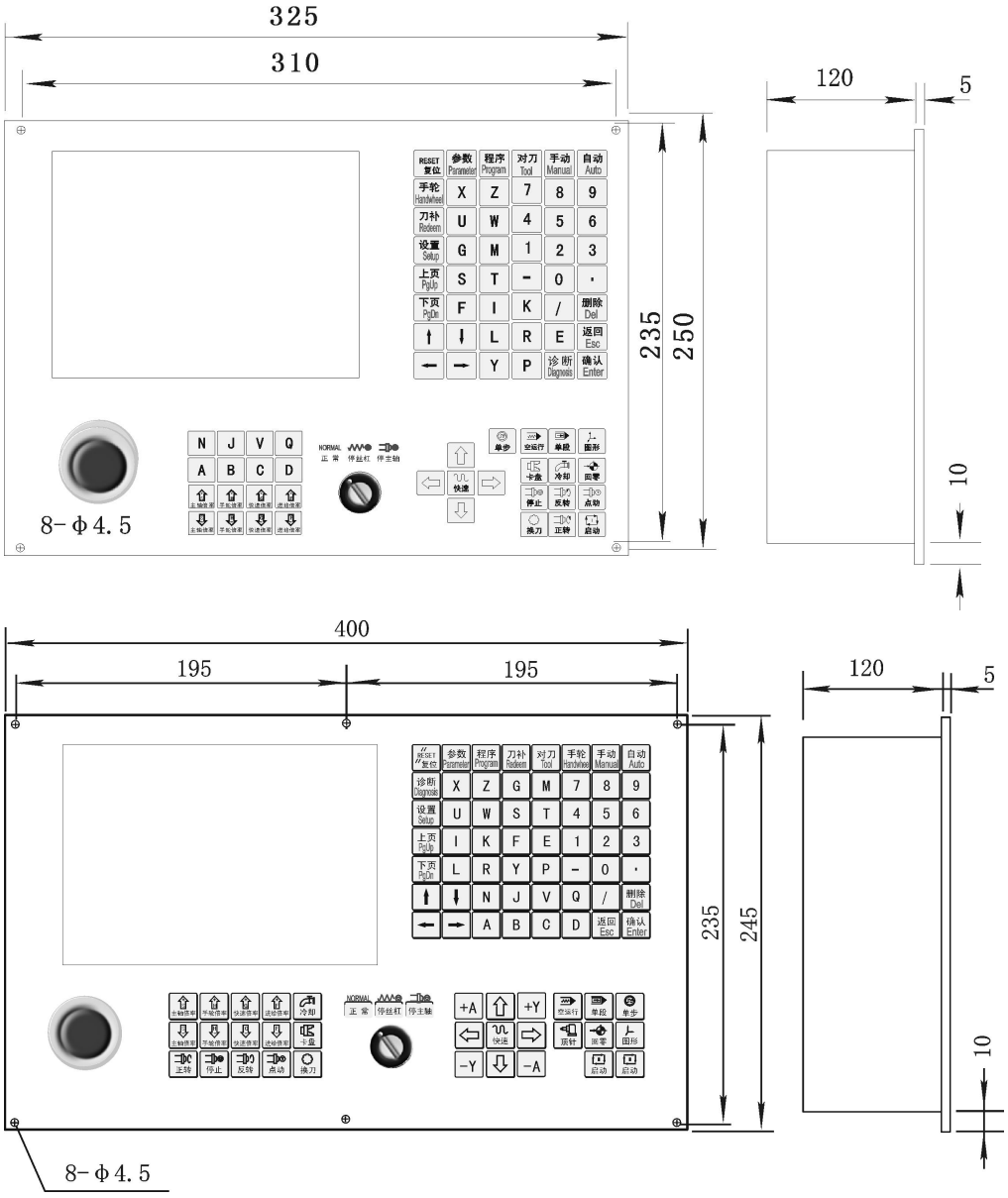
In installation of cnc system, panel should avoid hurting by hard and sharp materials. If the painting of other part of machine is needed, please take off cnc system to keep it clean.

To ensure there is no strong magnet and current interference, keep away from inflammable, explosive and other danger materials.

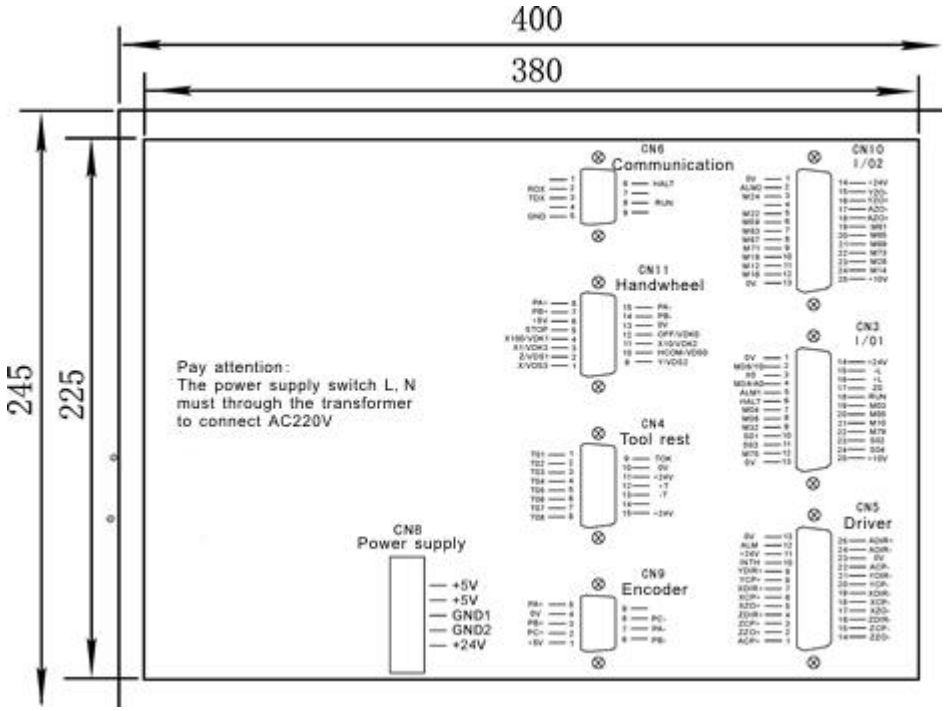
Pay attention:

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 (First type: 325x250x120; Second type: 400x245x120)


325
310
120
5
235
250
10
8-φ4.5
400
195
195
120
5
235
245
10
8-φ4.5

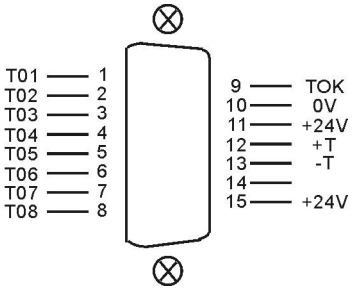
4.6 System rear view



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 CN4 and electric tool rest connection



CN4 DB15(hole) electric tool carrier				
signal	pin	I/O	function	availability
0V	10	OUT	Grounding	0V
+24V	11、15	OUT	+24V Power supply	+24V
+T	12	OUT	Positive rotate signal	0V
-T	13	OUT	Reverse rotate signal	0V

T1	1	IN	T1 signal	0V
T2	2	IN	T2 signal	0V
T3	3	IN	T3 signal	0V
T4	4	IN	T4 signal	0V
T5	5	IN	T5 signal	0V
T6	6	IN	T6 signal	0V
T7	7	IN	T7 signal	0V
T8	8	IN	T8 signal	0V
TOK	9	IN	Lock signal	0V

System can control 1-99 tools, the electrical tool is 8 max, default is 4. Press “C” to set the total tools in “redeem”, the starting tool is set in tool parameter.

No.1 parameter: Activate the function of electrical tool to select electrical tool or tool post. [1 means Yes, 0 means No]

No.2 parameter: Activate tool number when using electrical change into tool post, if the electrical tool is 4, set the number is 5 to switch to tool post.

No.4 parameter: The maximum time for tool positive rotate. Failed to find tool position beyond the time, system will stop rotating tool and alarm. (Unit: second)

No.5 parameter: Delay time after tool positive rotate to check the tool position signal. (ms)

No.6 parameter: Delay time after tool stop. (ms)

No.7 parameter: Tighten time of tool reverse rotate (ms).

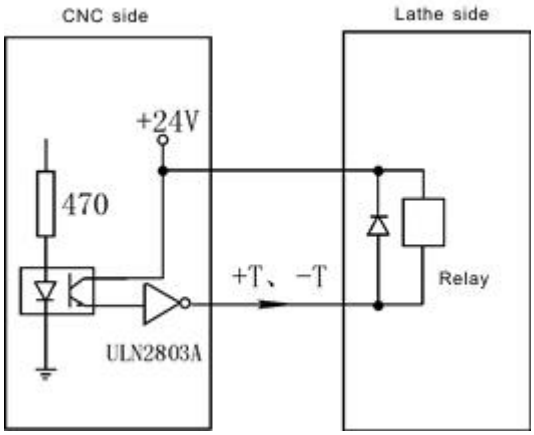
No.9 parameter: To set whether if it checks the key-locked switch after electrical tool reverse rotate to the position (total signal TOK), default is not to check.

Pay attention:

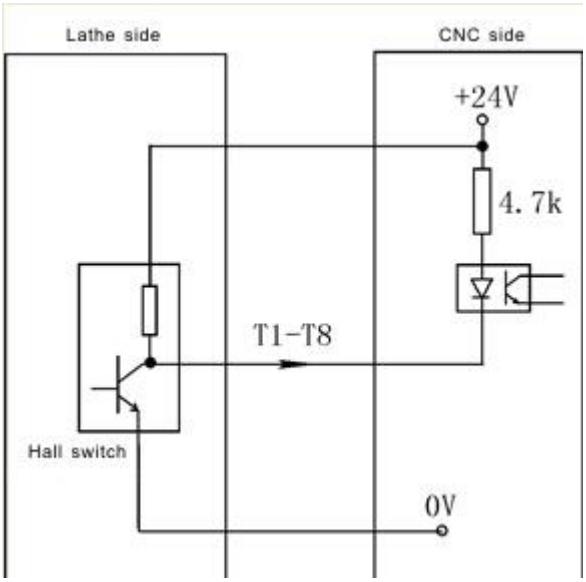
1. All the input or output is for system, input is from exterior signal to system, the output is from system signal to external.

2. When choosing the electrical appliance plate, +T and –T control single contact middle relay, user should install two AC contactors of +T and –T.

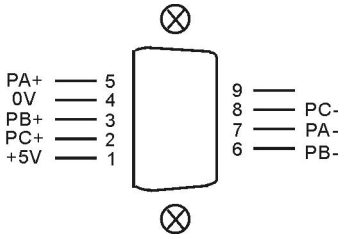
System output signal +T –T:



Tool input T1~T8 TOK:



4.7.2 CN9 and spindle encoder connection



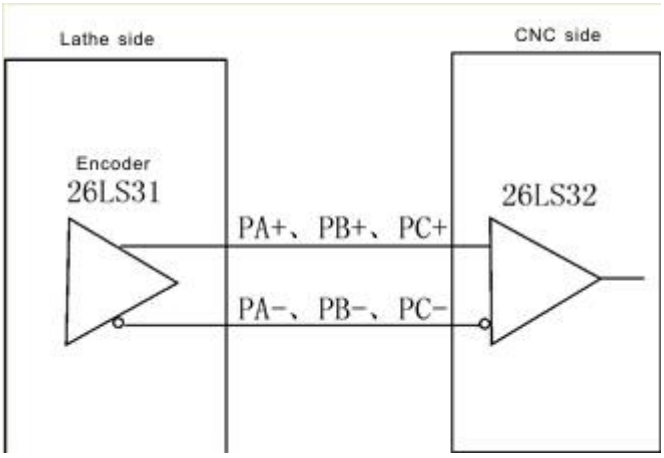
CN9 DB9(pin) spindle encoder				
signal	pin	I/O	function	availability
0V	4	OUT	0V	0V
+5V	1	OUT	+5V	+5V
PA+	5	IN	+A signal	5V
PA-	7	IN	-A signal	
PB+	3	IN	+B signal	5V
PB-	6	IN	-B signal	
PC+	2	IN	+Z signal	5V
PC-	8	IN	-Z signal	

Pay attention:

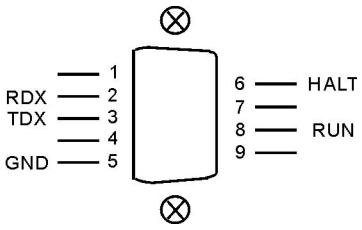
1. The output signal of encoder adopt the output way is line output, the power supply is +5V.

2. The signal line must adopt shielded twisted pair cable, the length is 20m at most.

The input signal of encoder PA PB PC:



4.7.3 CN6 and computer system connection



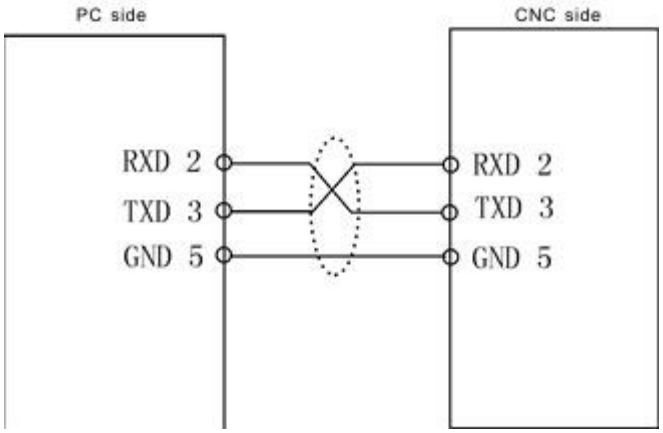
CN6 DB9(hole) RS232 communication				
signal	pin	I/O	function	availability
0V	5	OUT	0V	0V
RXD	2	IN	RXD	
TXD	3	OUT	TXD	
RUN	8	IN	run	0V
HALT	6	IN	pause	0V

Pay attention:

1. Must use the exclusive communication software to connect exterior PC to data communication.

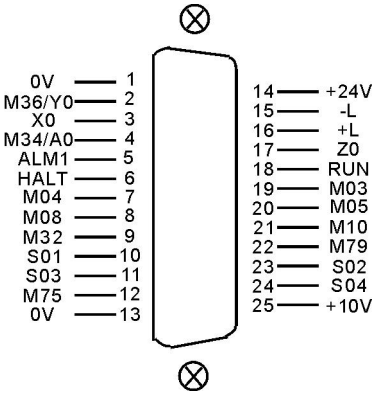
2. The signal line must adopt shielded twisted pair cable, the length is 10m at most.

The signal of CN6 connect to PC:



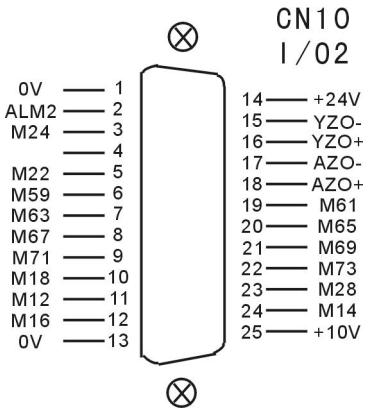
Pay attention: When PC programming, the files should be text files.

4.7.4 CN3 and machine electrical device I/O1 connection



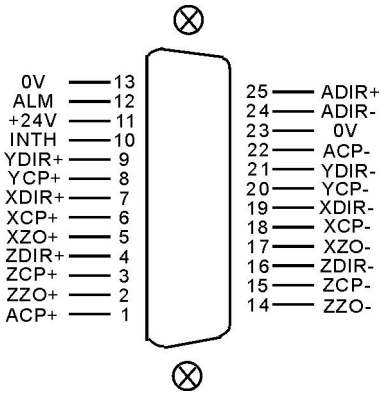
CN3 DB25(hole) I/O1 machine signal				
signal	pin	I/O	function	Availability
0V	1	OUT	0V	0V
+24V	14	OUT	+24V	+24V
M36/Y0	2	IN	M36/Y0	0V
X0	3	IN	X axis Zero	0V
Z0	17	IN	Z axis Zero	0V
-L	15	IN	Positive limit	0V
+L	16	IN	Negative limit	0V
M34/A0	4	IN	M34/A0	0V
ALM1	5	IN	Transducer alarm1	0V
HALT	6	IN	Pause	0V
RUN	18	IN	Run	0V
M03	19	OUT	spindle clockwise	0V
M04	7	OUT	SP counter clockwise	0V
M05	20	OUT	SP stop	0V
M08	8	OUT	coolant	0V
M10	21	OUT	spindle chuck	0V
M32	9	OUT	lubricating	0V
M79	22	OUT	spindle tailstock	0V
S01	10	OUT	spindle first gear	0V
S02	23	OUT	spindle second gear	0V
S03	11	OUT	spindle third gear	0V
S04	24	OUT	spindle fourth gear	0V
M75	12	OUT	C axis mode	0V
+10V	25	OUT	the first spindle converting	0～10V
0V	13	OUT	0V	0V

4.7.5 CN10 and machine electrical device I/O2 connection



CN10 DB25(hole) I/O2 machine signal				
signal	pin	I/O	function	availability
0V	1	OUT	Ground of supply	0V
+24V	14	OUT	+24V	+24V
ALM2	2	IN	Machine alarm2	0V
M24	3	IN	M24	0V
M22	5	IN	M01 input	0V
M59	6	OUT	Huff	0V
M61	19	OUT	User-defined output 1	0V
M63	7	OUT	User-defined output 2	0V
M65	20	OUT	User-defined output 3	0V
M67	8	OUT	User-defined output 4	0V
M69	21	OUT	User-defined output 5	0V
M71	9	OUT	User-defined output 6	0V
M73	22	OUT	User-defined output 7	0V
M18	10	IN	User-defined input 1	0V
M28	23	IN	User-defined input 2	0V
M12	11	IN	User-defined input 3	0V
M14	24	IN	User-defined input 4	0V
M16	12	IN	User-defined input 5	0V
YZO+	16	IN	+Y motor Zero signal	5V
YZO-	15	IN	-Y motor Zero signal	
AZO+	18	IN	+A motor Zero signal	5V
AZO-	17	IN	-A motor Zero signal	
+10V	25	OUT	the second spindle converting	0～10V
0V	13	OUT	0V	0V

4.7.6 CN5 and servo drive and motor connection



CN5 DB25(pin) servo drive signal				
signal	pin	I/O	Function	Availability
XCP+	6	OUT	X pulse signal +	5V
XCP-	18	OUT	X pulse signal -	
XDIR+	7	OUT	X direction signal +	5V
XDIR-	19	OUT	X direction signal -	
YCP+	8	OUT	Y pulse signal +	5V
YCP-	20	OUT	Y pulse signal -	
YDIR+	9	OUT	Y direction signal +	5V
YDIR-	21	OUT	Y direction signal -	
XZO+	5	IN	X motor Zero +	5V
XZO-	17	IN	X motor Zero -	
ZCP+	3	OUT	Z pulse signal +	5V
ZCP-	15	OUT	Z pulse signal -	
ZDIR+	4	OUT	Z direction signal +	5V
ZDIR-	16	OUT	Z direction signal -	
ZZO+	2	IN	Z motor Zero +	5V
ZZO-	14	IN	Z motor Zero -	
ACP+	1	OUT	A pulse signal +	5V
ACP-	22	OUT	A pulse signal -	

ADIR+	25	OUT	A direction signal +	5V
ADIR-	24	OUT	A direction signal -	
0V	13、23	OUT	0V	0V
ALM	12	IN	Servo alarm	0V
+24V	11	OUT	+24V	24V
INTH	10	OUT	Clear alarm	0V

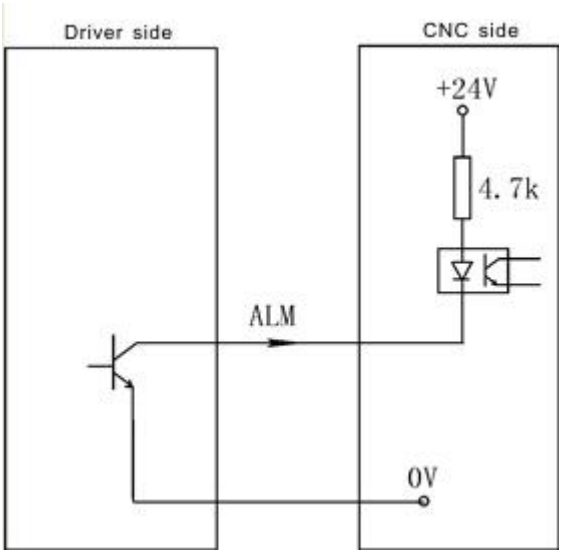
Pay attention:

1. The drive signal line must adopt shielded twisted pair cable, the length is 20m at most.

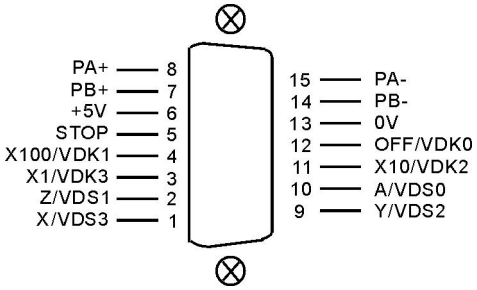
2. The alarm signal ALM is always open or close is set by No.17 parameter in other parameter.

3. When system choosing C axis to be rotate axis, M800 instruction is backing to datum point of encoder, M75 input signal is to choose position control mode of spindle servo, M03/M04 is to close M75 signal, spindle servo is choosing speed control mode.

Servo alarm signal:



4.7.7 CN11 and handwheel, band switch and external switch connection



CN11 DB15(pin) hand wheel, band switch connection				
signal	pin	I/O	function	Availability

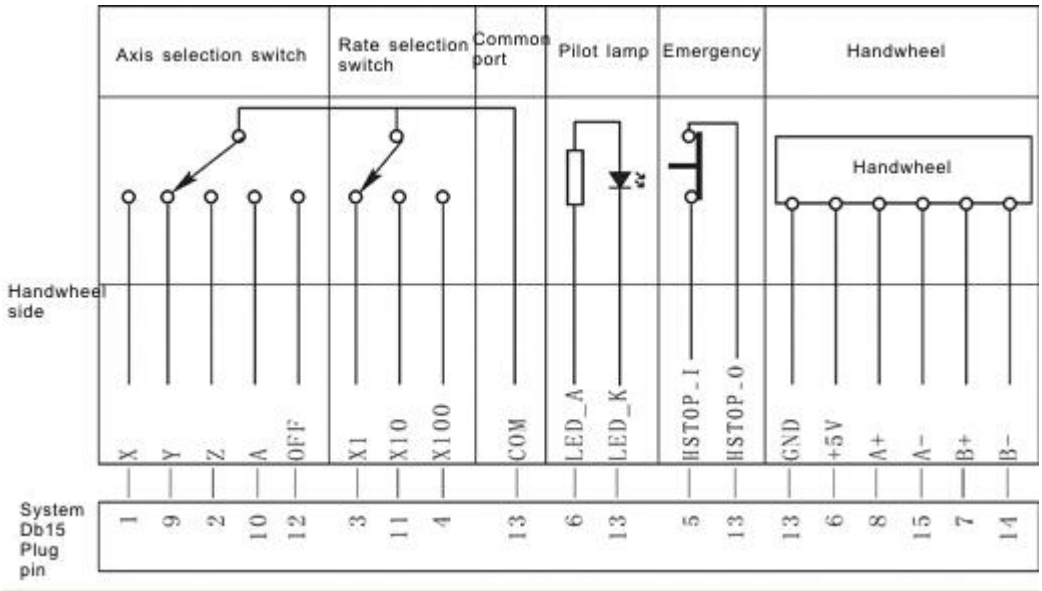
0V	13	OUT	0V	0V
+5V	6	OUT	+5V	+5V
PA+	8	IN	A signal +	5V
PA-	15	IN	A signal -	
PB+	7	IN	B signal +	5V
PB-	14	IN	B signal -	
STOP	5	IN	emergency stop	0V
OFF/VDK0	12	IN	Off/ feed amending 0	0V
X100/VDK1	4	IN	*100/ feed amending 1	0V
X10/VDK2	11	IN	*10/ feed amending 2	0V
X1/VDK3	3	IN	*1/ feed amending 3	0V
A/VDS0/HALT	10	IN	A/SP amending 0/halt stop	0V
Z/VDS1	2	IN	Z/SP amending 1	0V
Y/VDS2/RUN	9	IN	Y/SP amending 2/run	0V
X/VDS3	1	IN	X/SP amending 3	0V

4.7.7.1 Electrical handwheel (Manual pulse generator)

You can connect standard external handwheel when No.1 parameter in other parameter is 1 and can not use band switch to adjust spindle, feed and external stop running button, , so No.1 No.2 parameter in axis parameter only set to be “0”. A X Y Z X1 X10 X100 corresponding select feeding axis and shifting, No.33 No.34 parameter in other parameter only set to be 0.

PA+ PB- PA+ PA- corresponding input signal of handwheel pulse A B.

Handwheel contact diagrammatic as:



Pay attention:

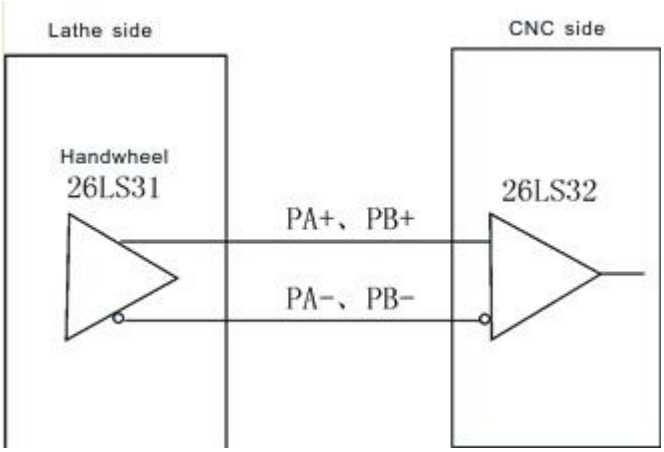
1. The output signal of handwheel adopts line output, the power supply is +5V.

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.

The input signal of handwheel:



4.7.7.2 Using for band switch

When No.1 No.2 parameter in axis parameter is "1", the band switch is on and the handwheel can't use external switch or external running button, No.1 parameter in other parameter is 0; No.33 No.34 parameter in other parameter is also only set to be 0. A Z Y X OFF X100 X10 X1 corresponding to choose feeding axis and shifting.

VDS0(A) VDS1(Z) VDS2(Y) VDS3(X) is the input signal of adjust spindle speed, total 16 gears. VDK0(OFF) VDK1(X100) VDK2(X10) VDK3(X1) is input signal of adjust feeding speed, total 16 gears.

4.7.7.3 External running, stop button

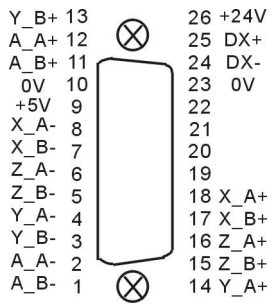
When No.33 parameter in other parameter is "1", the ninth hole of CN11 can be external running button to control program running automatically; When No.34 is set to be "1", the tenth hole of CN11 is set to be external stop button to control program suspending.

When the above function is effective, handwheel can not be used, so the No.1 parameter in "Other parameter" should be set to 0.

4.7.7.4 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.

4.7.8 Connection of the feedback signal of coordinate position CN13



CN13 Signal of feedback or grating				
signal	pin	I/O	function	Availability
0V	10,23	OUT	Ground of supply	0V

+5V	9	OUT	+5V	+5V
+24V	26	OUT	+24V	+24V
DX+	25		RS485+	
DX-	24		RS485-	
XA+	18	IN	Positive A signal of X axis	5V
XA-	8	IN	Negative A signal of X axis	
XB+	17	IN	Positive B signal of X axis	5V
XB-	7	IN	Negative B signal of X axis	
YA+	14	IN	Positive A signal of Y axis	5V
YA-	4	IN	Negative A signal of Y axis	
YB+	13	IN	Positive B signal of Y axis	5V
YB-	3	IN	Negative B signal of Y axis	
ZA+	16	IN	Positive A signal of Z axis	5V
ZA-	6	IN	Negative A signal of Z axis	
ZB+	15	IN	Positive B signal of Z axis	5V
ZB-	5	IN	Negative B signal of Z axis	
AA+	12	IN	Positive A signal of A axis	5V
AA-	2	IN	Negative A signal of A axis	
AB+	11	IN	Positive A signal of B axis	5V
AB-	1	IN	Negative A signal of B axis	

Other parameter No.200 -No.220 are used to set this function. If alarm, press "G" in diagnosis interface can clear instructional position and feedback position, then override alarm.

Pay attention:

1, The encoder or the grating output signal with long-line output mode, the power supply is +5V.

2, The signal line must adopt shielded twisted pair cable, the length shall not exceed 20m.

3, This function is for the matching.

4.7.9 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 XYZA axis.
3. Set the axis parameter in system to trim the electrical gear.

P11=00000001 soft limit effectively;

Lathe setting P23=11111011; Milling setting P33=11111011; Returning to zero with floating mode;

4. Set the other parameter in system

P300=01111100; the absolute value function;

P301=92; P302=91; P303=90;

P304/P305/P306/P307=131072;

P309/P310/P311/P312 set the coordinate values, such as when the screw pitch with direct of Z axis is 6mm,

P311=6000000;

5. Set the coordinate axis which corresponds the No.56 parameter in driver parameter

P1=1, enter the password;

P56=1/2/3/4 corresponds to the X/Y/Z/A, 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 M501/M502/M503/M504(corresponds to the X/Y/Z/A axis) 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 parameter in other parameter, such as the X coordinate is not right, the original is P309=4000000, to modify to P309=-4000000;

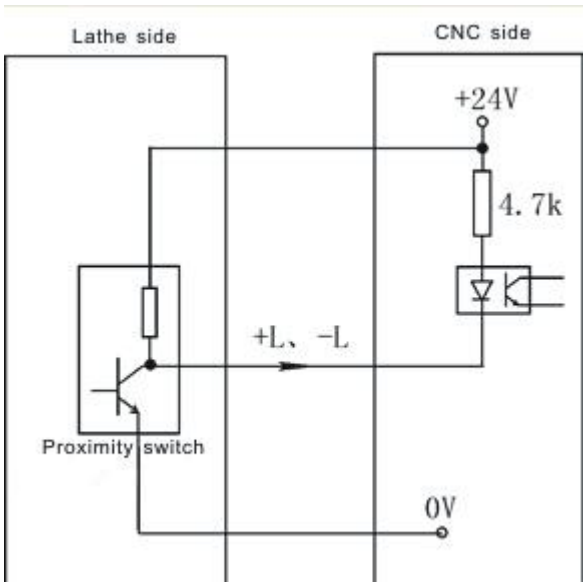
3) Repeat 1) 2) test for two times;

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 parameter in other parameter(Corresponds to X/Y/Z/A 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

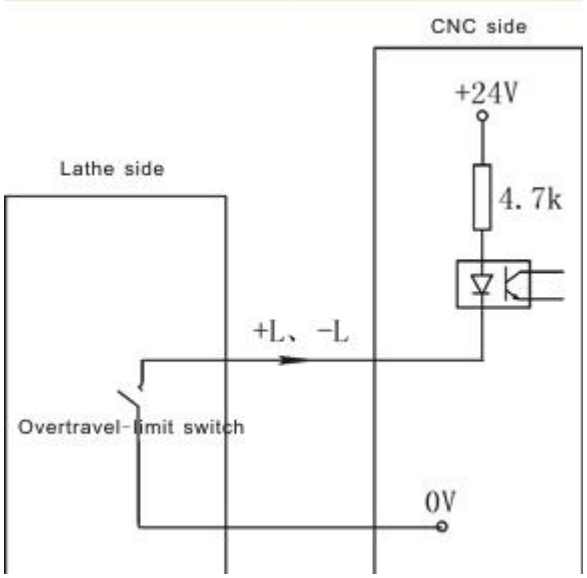
4.8 The machine installation of lathe.

4.8.1 Limited position: take X Z axis is limited position positively as example

Model 1: NPN approach switch



Mode 2: general switch



Axis parameter:

21, XZ positive limited
[0 open, 1 close]
22, XZ negative limited
[0 open, 1 close]

Pay attention:

1. X Z 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.

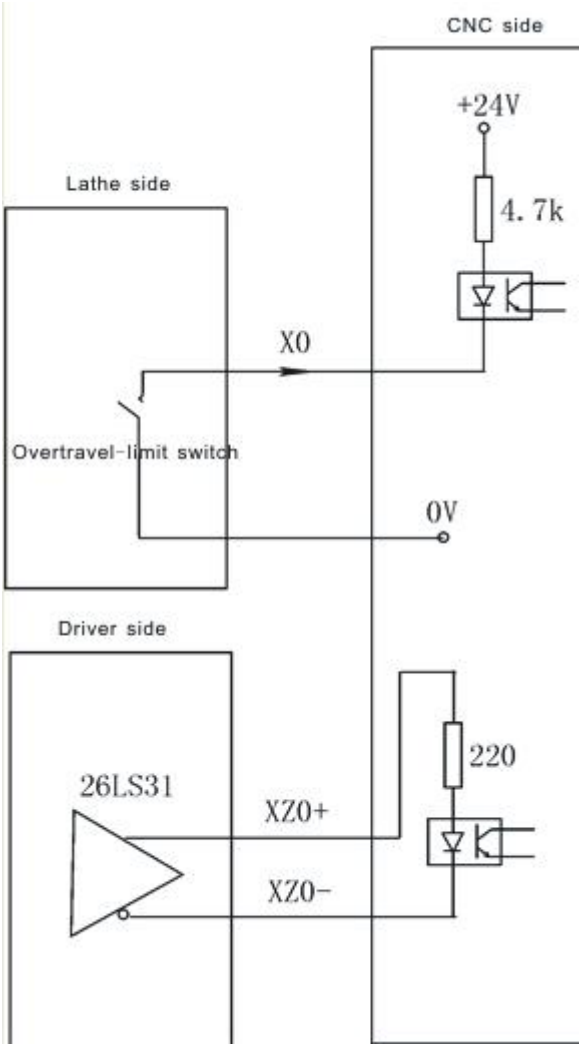
3. The system could define X0 Z0 to be limited input signal of X Z axis. X0 signal as the limited signal and datum point of X axis, the same switch to control.Z0 signal as the limited signal and datum point of Z axis, the same switch to control. No.21 No.22 function as follows:

No.21, X axis limited position [0 means always open, 1means always close]

No.22, Z axis limited position [0 means always open, 1 means always close]

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 Z 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.23 parameter in axis parameter is set to be "00000000".

No.26 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.27 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.28 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.29 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.30 No.31 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.30) Z(No.31) 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.31 No.33 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.31) Z(No.33) axis backing to positive datum point. Unit: mm/min. 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.32 No.33 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 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.

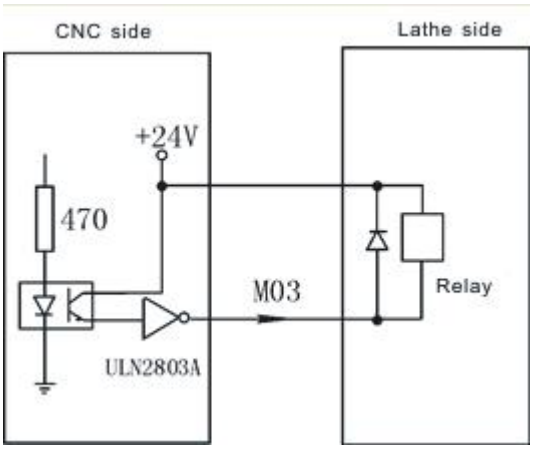
Pay attention: After backing to the datum point, the offset speed is determined by G00.

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.

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)

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 M59 M61 M63 M65 M67 M69 M71 M73

2). U29 corresponds M03 M04 M05 M08 M10 M79 M32 M75

3). U30 corresponds +T –T S01 S02 S03 S04 LRUN INTH

3. When user-defined signal M71/M70 M73/M72 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. No.28 No.29 parameter in other parameter to set.

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.7 parameter: Set the braking time of spindle, also the it's the output of the hold time, the time less the brake fast.

No.8 parameter: Set the braking signal is long signal 1 or short signal 0.

No.9 parameter: To set system whether checking spindle feedback signal of spindle position, the feedback signal is spindle encoder signal. To set the parameter value 1 means check; 0 means not to check.

No.10 parameter: To set feedback pulse numbers of spindle encoder turn a round, the value: Line number of encoder x 4.

No.50 parameter: Whether turn on the spindle or not when spindle shifting [1 means on, 0 means off]

No.51 parameter: The turning speed of motor when spindle shifting (1/100rpm)

No.52 parameter: The turning direction when spindle shifting (0 means positive, 1means negative)

No.53 parameter: The stopping time when spindle shifting (10ms)

No.54 parameter: Turning time of low speed when spindle shifting (10ms)

No.55 parameter: Stopping delay time of spindle (10ms)

Speed parameter:

No.8 parameter: To set the speed of spindle setting value at manual condition. Unit: rpm.

No.36 parameter: To set the highest speed of spindle, that's the turning speed of corresponding 10V instruction voltage.

No.37 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.38 parameter: To set the highest speed of spindle (Third gear), that's the turning speed of corresponding 10V instruction voltage. Unit: r/min

No.39 parameter: To set the highest speed of spindle (Fourth gear), that's the turning speed of corresponding 10V instruction voltage. Unit: r/min

No.40 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: To set whether spindle and chuck is interlocking or not: 0 means they are separately; 1 means the spindle only start turning when chuck on. The thumbstall can't be use when the spindle is turning.

Setting parameter is related with the configuration of lathe and user's service condition, but consider for safe, suggest setting 1.

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: S).

4.8.3.3 Chuck and thumbstall control (M10/M11 M79/M78)

Chuck and thumbstall of this system control is related with parameter as follows:

Other parameter:

No.2 parameter: To set the controlling type of chuck, setting the parameter to set controlling logic when the lathe has automatical controlling chuck. Could realize to control inner and outer chuck (Inner: Chuck to center when M10; Outer: Chuck opening outward when M10). 1 means outer, 0 means inner.

No.15 parameter: To check chuck loosen or tighten enough when automatical control. 1 means check, check M12 when M10, check M14 when M11; 0 means not to check.

No.16 parameter: To set checking whether thumbstall goes forward or back enough when automatical control. 1 means check, check M18 when M79, check M28 when M78; 0 means not to check.

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.21 parameter: To set system controls thumbstall only need one signal (one-way valve) or two signal (two-way valve), this parameter is related with equipment of lathe.

M79 is just a output signal to control go forward or back of thumbstall when set to 0, go forward when M79 is effective, go back when M79 is invalid.

M79 and M73 corresponding control go forward and back of thumbstall when set to 1, the system carry out going back when M73 is effective, M79 is invalid. Output M79 when M79, output M73 when M78.

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.23 parameter: To set external button to control thumbstall goes forward and back(or foot switch), the signal is reciprocating, it means going forward once and then going back once, reciprocating mode. No external button when set to 0; There is an external button to control thumbstall when set to 1, the signal is M14.

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.

No.25 parameter: To set the retention time when the output signal M79 M73 of chuck is short signal, set to 0 means the signal is long signal. Unit: S.

Pay attention: M12 M14 M16 M18 and M28 are multiple functions signal, only choose one function to use.

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 (twelfth pin of CN5 ALM), 0 means always open, 1 means always close.

No.18 parameter: To set the type of system checking the spindle alarm signal of lathe (fifth pin of CN3 ALM1), 0 means always open, 1 means always close.

No.19 parameter: To set the type of system checking the alarm signal of lathe (second pin of CN19 ALM2), 0 means always open, 1 means always close.

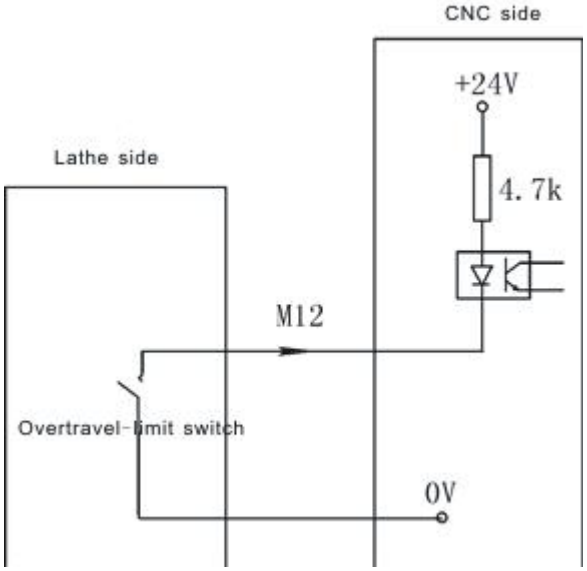
No.27 parameter: To set the Emergency always open or close of system CN11, suggest setting always close for safe.

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).

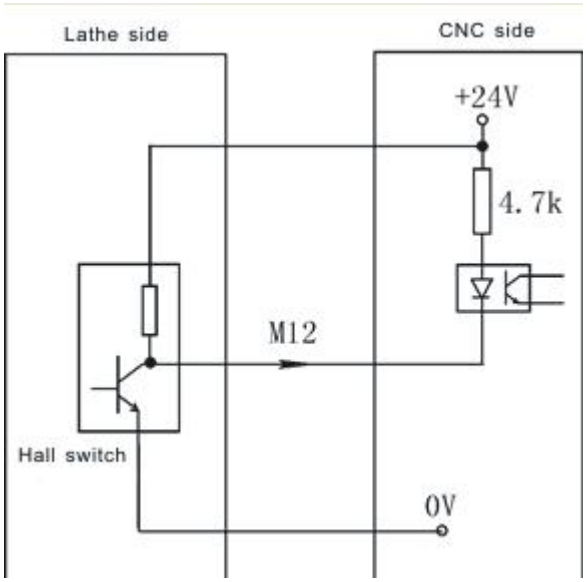
No.29 parameter in other parameter is effective when appearing alarm, the output signal M67 is effective.

4.8.5 User-defined output signal M12 (M14 M16 M18 M28 M22 M24, 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:

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.

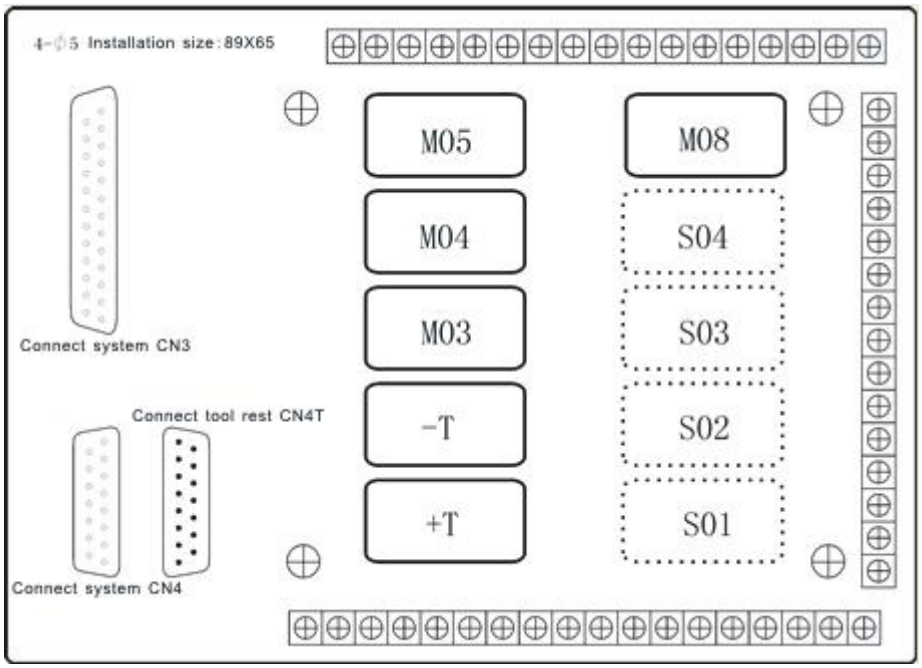
4.9 Electrical appliance plate of lathe

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;

CN4 is corresponding one-to-one with CN4 pins of system;

CN4T connect electrical tool rest: 1-T1, 2-T2, 3-T3, 4-T4, 5-T5, 6-T6, 7-T7, 8-T8, 9-0V, 10-power supply +12V, 14-T0K.



4.9.1 Tool rest controlling

+T, -T must controlled by external AC contactor.

C3 is common port of +T, -T and M08.

4.9.2 Spindle controlling

C1 is the common port of M03 and M04.

C2 is the common port of M05, M5B is always close.

4.9.3 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.

NEWKer数控
鑫科瑞数控

Made in china.

Adr. ChengLong Road 888, ChengDu
Tel：0086-28-84601148、84609226 Fax：0086-28-66627461
Email：NEWKER2008@163.COM WWW.NEWKER-CNC.CN
Sale mobile：0086-13981709850 Service mobile：0086-13550063039