SHN990MDC
Milling machine system

USER MANUAL

CONTENTS

Chapter 1   Preface

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

   Real-time control of high speed (the highest speed shift speeds of up to 30 m / min, the highest feed interpolation speed can reach 15 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. All the functions of A axis are effective when configure fourth axis system.

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

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

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

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

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

As no work coordinate, make machine coordinate as work coordinate.

<div align="center">

Machine Coordinate & direction

sketch map

</div>

Vertical milling、 drill machine   Horizontal milling
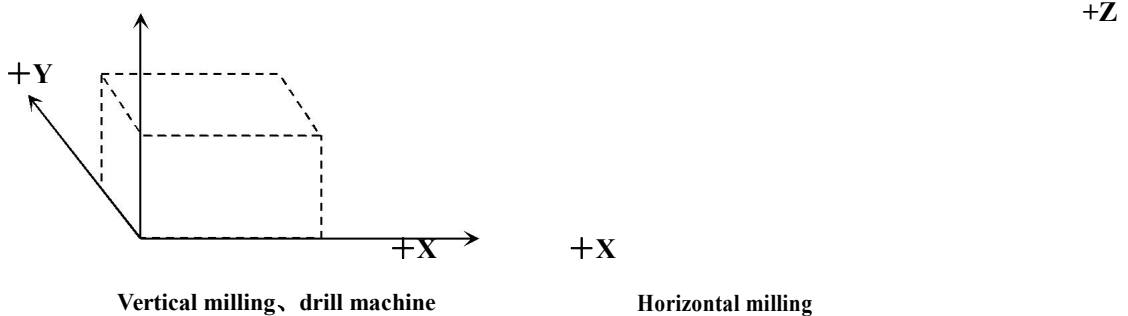
Working Coordinate System: Work piece processing uses the coordinate system is called as the work piece coordinate system, it is set by CNC. The work piece coordinate system could change to move its zero point.

Set the work piece coordinates:

1.Use G54 ~ G59: Use operating parameter set coordinate system may set 6 work piece coordinate system.

2.G52 instruction: Set a value in front of G52 instruction to set the workpiece coordinate system in program.

With absolute value instruction ,it must use the above method to establish the work piece coordinate system

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

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

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

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

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

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

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

2.2 General description of program

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

Note 1:"-"means this data can be use.

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

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

2.3 Program instruction

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

2.3.1 Functional meaning of address symbol,data list

Functions	Address symbol	meaning	Data range

Document No.	%	Name of machining workpiece	0-9、A-Z
Program segment No.	N	No. of program segment	0000-9999
Preparation function	G	Content and mode of designated instruction operation	00-99
Auxiliary function	M	Auxiliary operation instruction	00-99
Tool chosen	T	No.of Tool.	01-99
Tool compensation	H D	The length compensates No.of the radius compensates of the parameter	1- 4
Spindle function	S SP	The spindle speed; spindle localization	00-99999
Cutting speed	F	Speed per minute	1-3000mm/min
Coordinates character	X Y Z A(B/C/ U/V/W)	The coordinates value of X Z and 4th axes.	9999.999mm
Core coordinates	I J K	X Z axes and Z axes core coordinate increment value	9999.999mm
Step length	R	Circular arc radius	0.001-999.999mm
Delay time	P.X	Delay time of designated delay	0.001-99.999s
Program entrance	P	Entrance of calling program name	0000-9999
Repeat times	L	Times of cycle or subprogram calling	1-9999
Line skip	/	Not to carry out when "/" is in front of program line	No.12 parameter in other parameter can shield the function

2.3.2 Program instruction

Table 1 G Instruction-code and functions

G code	function
G00	Fast decides
G01	The straight line inserts makes up

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

G16	Polar coordinate instruction: The polar coordinate (radius and angle), the angle to is chooses the plane the first axis to anti-clockwise changes, but the negative direction is clockwise changes . Form : 　　G** G## G16； 　　　　G00　　IP； 　G** Expresses the plane chosen 　　　G## means G90 （Work piece coordinate system original point ）or G91（Current position ） Assigns the polar coordinate and zero point	
G17	Choose the X Y plane	X：X axis or its parallel axis
G18	Choose the Z X plane	Y：Y axis or its parallel axis
G19	Choose the Y Z plane	Z：Z axis or its parallel axis
G20	Inch　input	
G21	Millimeter input	
G28/G281/G282/G283/G284	Go to first reference point	
M882 M883	Set drop instruction of Z axis tool setting automatically	
M884 M885	Automatical dividing center of X Y axis	
M880	Adjust tool automatically	
G30/G301/G302/G303/G304	Go to 2,3,4 reference point	
G26	ZXY axis go to program original point	
G261	X axis go to program original point	

G262	Y axis go to program original point
G263	Z axis go to program original point
G264	A axis go to program original point
G25	Memory current coordinate of X Y Z A
G61/G611/G612/G613/G614	Back to the memory point
G40	Cancel tool radius compensate
G41	tool radius compensate，left
G42	tool radius compensate，right
G43	Tool lengthen positive compensate
G44	Tool lengthen negative compensate
G45	Tool adding offset
G46	Tool subtract offset
G47	Tool adding two multiple offset
G48	Tool subtract two multiple offset
G49	Cancel tool lengthen compensate
G37	Cancel scale zoom
G36	Enable scale zoom：format：G36 X_Y_Z_R_
G12	Cancel programmer mirror
G11	Enable programmer mirror
G52	Set local coordinate
G53	Choice coordinate
G54	Choice work coordinate 1	
G55	Choice work coordinate 2	Note：These six work coordinate save in CNC，user may choice any one。
G56	Choice work coordinate 3	
G57	Choice work coordinate 4	

G58	Choice work coordinate 5	
G59	Choice work coordinate 6	
G60	exactitude stop	
G64	Continue path work。	
G68	rotate coordinate。format： G17⎫ G18⎬ G68 a-b- R-; R:Angle G19⎭	
G69	Cancel rotate coordinate	
G73	Drill deep hole cycle：format： G73 X-Y-Z-R-Q-F- L - Z：distance from R to hole bottom R：distance from original to R Q：feed depth every time F：feed speed L：repeat time	
G74	Left Tap cycle：。format： G74X-Y-Z-R-P-F- L -	
G80	Cancel cycle mode	
G81	Drill cycle：format： G81 X-Y-Z-R-F- L -	
G82	Drill cycle。format： G82 X-Y-Z-R-P-F- L -	
G83	Drill cycle。format： G83 X-Y-Z-R-Q-F- L -P-	
G84	Right Tap cycle：format：G84 X-Y-Z-R-P-F- L -	
G85	Drill cycle：。format： G85 X-Y-Z-R-F- L -	
G86	Drill cycle：。format： G86 X-Y-Z-R-F- L -	
G89	Drill cycle：： G89 X-Y-Z-R-P-F-L-	
G90	Absolute program	
G91	Increase program	

G98	Go back to original point
G99	Go back to R point
G22	Program cycle order
G800	Cancel Program cycle order

Table 2 M code and function

Spindle	M03 M04 M05	Spindle CW Spindle CCW Stop Spindle	M7053/M7054 P xxxx; It will delay time after spindle CW or CCW, the time is determined by "P". Unit: Millisecond. Example: M7053 P2000; Means it will delay 2 seconds to stop spindle.
Cooling	M08	Turn on cool	
	M09	Turn off cool	
Chuck	M10	Tighten tool	
	M11	Loosen tool	
Huff	M58	Turn off huff	Controls M59 signal
	M59	Turn on huff	
Lubrication	M32	Turn on lubricate	Controls M32 signal
	M33	Turn off lubricate	
User-defined	M79	User self-defined1 output turn on	Double signal output
	M78	User self-defined1 output turn off	
User-defined output	M61	User self-defined2 output turn on	Controls M61 signal
	M60	User self-defined2 output turn off	
	M63	User self-defined3 output turn on	Controls M63 signal
	M62	User self-defined3 output turn off	
	M65	User self-defined4 output turn on	Controls M65 signal
	M64	User self-defined4 output turn off	
	M67	User self-defined5 output turn on	Controls M67 signal
	M66	User self-defined5 output turn off	
	M69	User self-defined6 output turn on	Controls M69 signal

	M68	User self-defined6 output turn off	
	M71	User self-defined7 output turn on	Controls M71 signal
	M70	User self-defined7 output turn off	
	M75	User self-defined8 output turn on	Controls M75 signal
	M74	User self-defined8 output turn off	
Spindle shifting	M41	SP Speed first gear	
	M42	SP Speed second gear	
	M43	SP Speed third gear	
	M44	SP Speed fourth gear	
User-define d input	M12	Check M12 input valid	To skip when conditions are tenable
	M13	Check M12 input invalidate	Example: M12 P120 Program skips to 120th line to carry out.
	M14	Check M14 input valid	
	M15	Check M14 input invalidate	
	M16	Check M16 input valid	
	M17	Check M16 input invalidate	
	M18	Check M18 input valid	
	M19	Check M18 input invalidate	
	M28	Check M28 input valid	
	M29	Check M28 input invalidate	
	M22	Check M22 input valid	
	M23	Check M22 input invalidate	
	M24	Check M24 input valid	
	M25	Check M24 input invalidate	
Subprogram	M97	Program skip	L=1-99 P is the line number of transferring program
	M98	Subprogram is on tap	
	M99	Back to subprogram is on tap	

Program controlling	M87 M00 M01 M02 M30 M20	**Number of workpiece plus 1** **Suspend program** **Suspend program,input M22 effective suspend** **Program is over** **M05,M09 program is over** **Program is over, automatical repeat run the program according to the parameter which set the running times, be used for the debugging**	**When other parameter P10=0 is set not to auotomatical plus 1, instruction M87 to make workpiece number plus 1**
Speed of spindle	S SS	**Set speed of the first spindle** **Set speed of the second spindle**	**S=0-99999** **SS=0-99999**
Read the position of absolute motor	M500 M501 M502 M503 M504	**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.**	

2.4 Programme instruction and usage

2.4.1 Programme convention

(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 Instruction of G function

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

Format: G53 （G54/G55/G56/G57/G58/G59） Mode after
setting G53 machine coordinate
G54 work coordinate 1
G55 work coordinate 2
G56 work coordinate 3
G57 work coordinate 4
G58 work coordinate 5
G59 work coordinate 6

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

G54/G55/G56/G57/G58/G59 work coordinate have offset relative to machine coordinate which can be set in parameter.
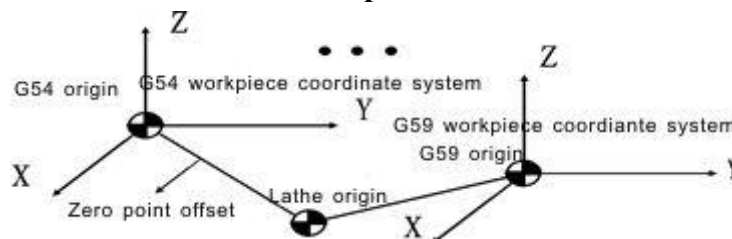
Example: G00 G54 X50 Y60 Z70

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

(2)Local coordinate system instruction

Set local coordinate system when programming in workpiece coordinate system is to programme easily.

Format:G52 X_ Y_ Z_ ; Set local coordinate system (Mode)

G52 X0 Y0 Z0;   Cancel local coordinate system

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

Local coordinate system setting does not change the workpiece coordinate system and machine tool coordinate system. When using the G52 instruction to set workpiece coordinate system, if it is not the instruction of all the axis coordinate value, the local coordinate system of non-specified coordinate value will not cancel and remain unchangeable .Use the absolute mode to specify motion instruction after the G52 program segment .
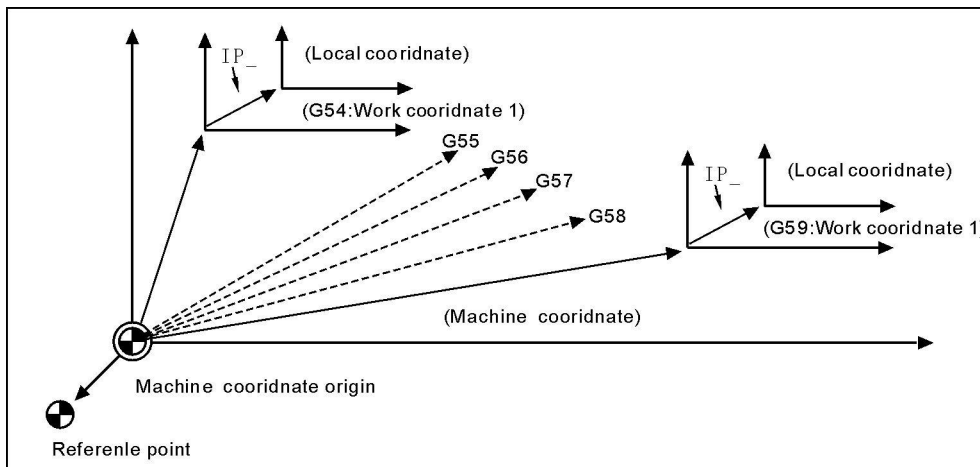
(3)Program method(G90/G91)

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

Format: G90 （Mode,initial）;Absolute

        G91   （Mode）   ;Increment

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

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

Format：G17 (Mode，Original)   ;Set XY Plane

        G18 (Mode)   ;Set ZX Plane

        G19 (Mode)   ;Set YZ Plane

Used to point out arc interpolation plane.

Note：this instruction does not produce motion.

(5)Rapid motion(G00)

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

Format：G00 X- Y- Z- A- (Mode，original)

 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.
(6) Line interpolation(G01)
   Used for single axis motion or 2,3,4 axis interpolation motion.

      Format：G01 X_ Y_ Z_ A_ F_ (Mode)

   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.
   The feeding speed of F in G01 can be adjusted by feeding override on the panel, the range is 0%~150%.
   G01 instruction is also can be written G1.
(7) Arc interpolation(G02/G03)
   In the program plane，these instruction is to execute G02 clockwise and
G03 counter-clockwise arc interpolation.
Note：Arc interpolation must point out interpolation plane，the X、Y、Z word point out the arc end coordinate value，I、J、K separate is X、Y、Z increase value from original point or center point.In another words, Make the original point as zero point，As center point locate to positive direction of original point the value will be positive，As center point locate to negative direction of original point the value will be negative。I J K function is describe center point coordinate。On the side, We can use R program，the R is negative when arc angle largen 180 degree。
   The arc track as follow:
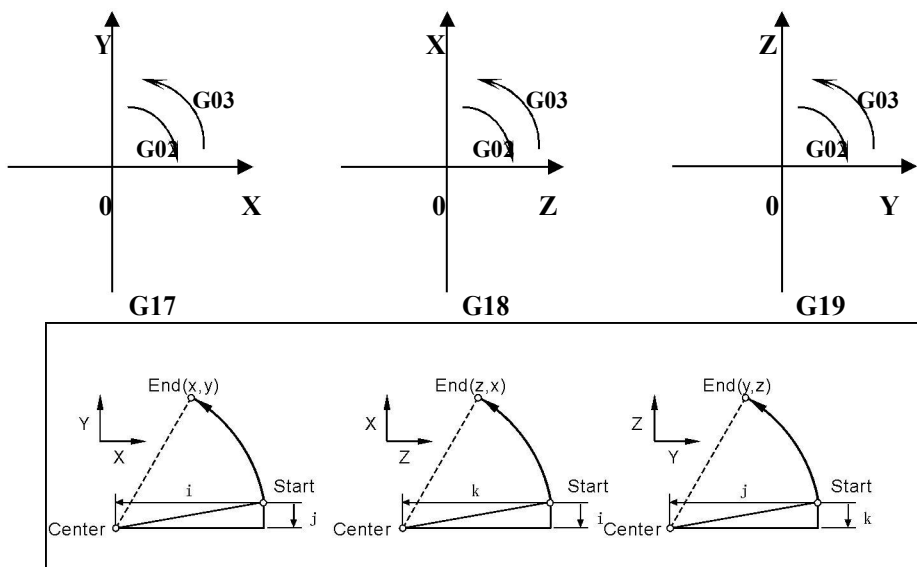
   The arc interpolation speed is determined by F word.
   Attention: I, J, K and R are the non-modality instruction.

Demonstration:

1) absolute programming

N0000 G92 X200 Y40 Z0;

N0010 G90 G03 X160 Y40 I-20 J0;

N0020 G02 X120 Y40 R20;

N0030 G02 X120 Y40 R20;

N0040 G26 M02;

2) Relative programming

N0000 G91 G17 G03 X-40 Y0 R20

F300; N0010 G02 X-40 Y0 R20;

N0020 G02 X0 Y0 R20;

N0030 G26 M02;

Two methods have the same result.

(8) Spiral interpolation (G02/G03)

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

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

Format：

$$G17\left\{\begin{matrix}G02\\G03\end{matrix}\right\}X\_Y\_\left\{\begin{matrix}I\_J\_\\R\_\end{matrix}\right\}F\_\quad;\text{XY plane(Mode)}$$

$$G18\left\{\begin{matrix}G02\\G03\end{matrix}\right\}X\_Z\_\left\{\begin{matrix}I\_K\_\\R\_\end{matrix}\right\}F\_\quad;\text{ZX plane(Mode)}$$

$$G19\left\{\begin{matrix}G02\\G03\end{matrix}\right\}Y\_Z\_\left\{\begin{matrix}J\_K\_\\R\_\end{matrix}\right\}F\_\quad;\text{YZ plane(Mode)}$$

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

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

(9) Delay instruction(G04)

Format：G04 P_ ;or

G04 X_ ;or

G04 U_ ;

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

Instruction introduction:

a.The unit of P delay time is ms(Millisecond)/ b.The unit of X and U delay time are S. c.Example:

G04 X1；　　delay 1s.

G04 P1000; delay 1s.
G04 U1;   delay 1s.

  d.Special application:G04 can be accurate stop instruction, such as processing corner kinds of workpiece, it appears over cutting sometimes, if use G04 instruction around the corner, it will clear the over cutting.
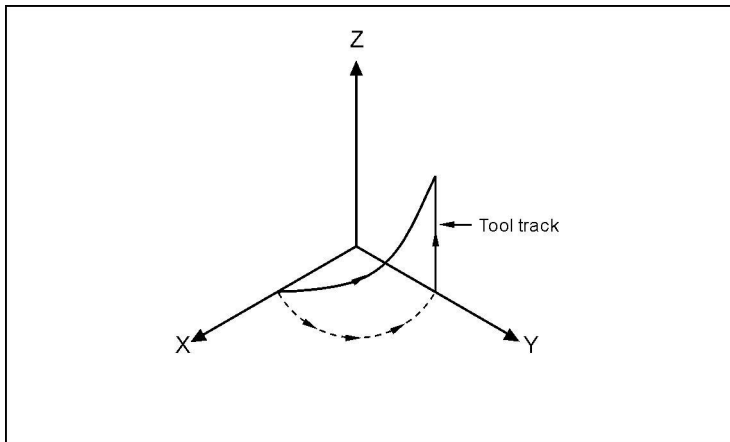Example as follows:
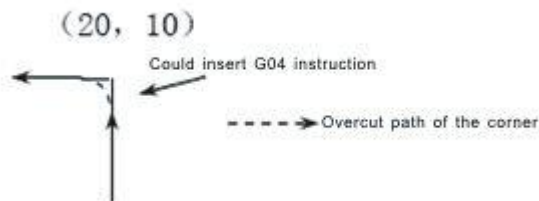
  Program: … …
N150 G01 X20 Z10 F100;
N160 G04 P150;（Clear the over cutting）
N170 G01 W-10;
……

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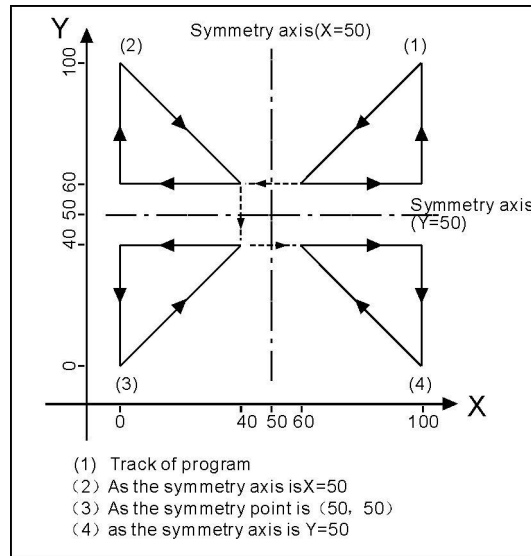 N0002 G01 X20 Y0 Z0 F600
  N0003 G03 X0 Y20 R20 Z15 F180

  ……
Pay attention: Set No.21 parameter in "N"(processing) parameter to clear the over cutting.
(10) Mirror instruction(G11/G12)
    In order to decrease program codes, be used for machining symmetry workpiece.

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

For example:

(1) Track of program
(2) As the symmetry axis isX=50
(3) As the symmetry point is（50，50）
(4) as the symmetry axis is Y=50

The mirror procedure gives an
example Sub program
KG11
N10 G00 G90 X60.0 Y60.0；
N20 G01 X100.0 F100；
N30 G01 Y100.0；
N40 G01 X60.0 Y60.0；
N50 M99； Main
program N10 G00
G90; N20 M98
P%9000; N30 G11
X50.0 N40 M98
P%9000;
N50 G11 X50.0 Y50.0
N60 M98 P%9000;
N70 G11 Y50.0
N80 M98 P%9000;
N90 G12；
N100 M12;

(11) Proportions scale instruction(G36/G37)

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

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

Note：the scale coefficient is behind
R. For example:

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

P0：Scale center
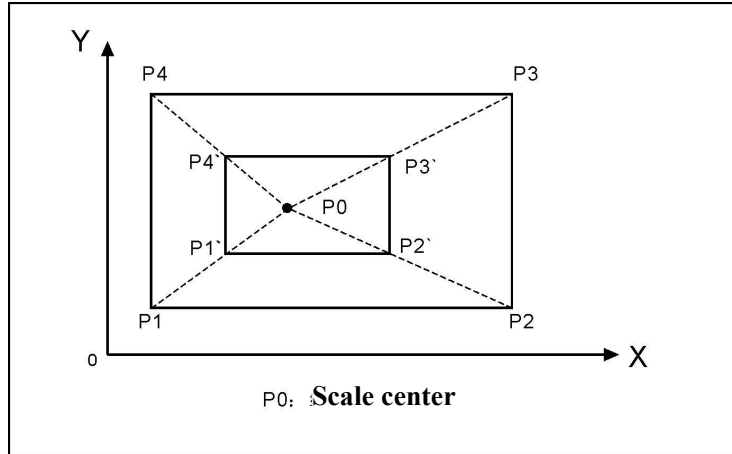
In the proportions scale section, cannot use tool length and radius compensation:

Original track

Track after scale
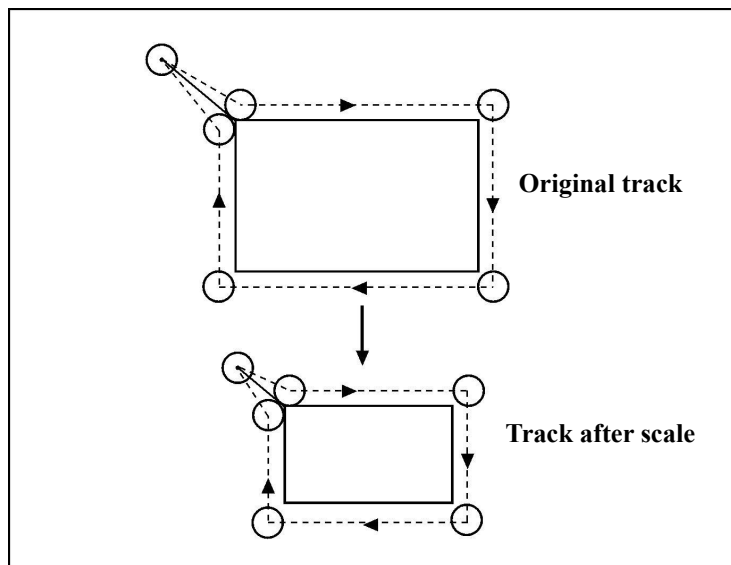
(12) Coordinate rotate(G68/G69)

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

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

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

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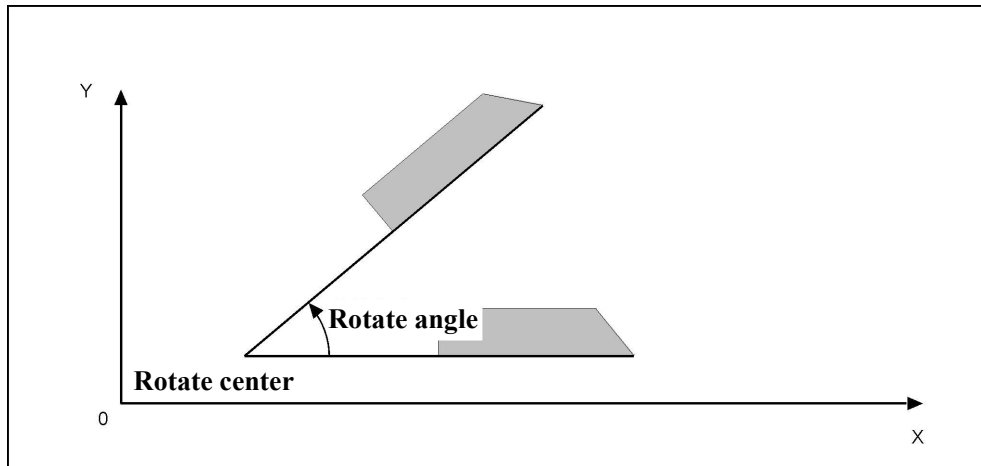
To specify plane and select code(G17 G18 or G19) before the G68 code program segment, select the code cannot be specified in the mode of rotating coordinate system. Cancel the mode G69 code must occupy a program segment separately. After rotating the coordinate system to carry out tool radius compensation,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

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

Original program tool track

Rotate center
（7000,3000）

(0, 0)

61° 朵

（-500，-500） Tool track after rotate
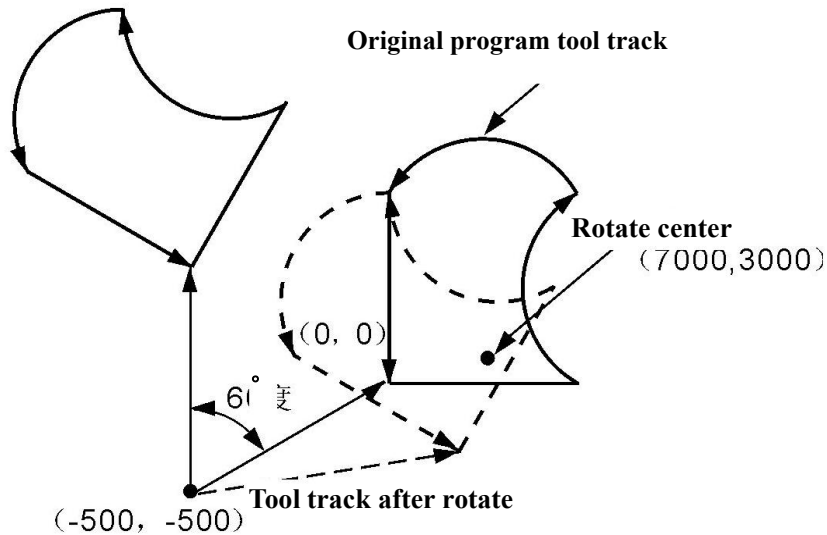
N1 G92 X-500Y-500F20000 G17;
N2 G68 X700Y300R60;
N3 G90 G01 X0 Y0 F20000;
N4 G91 X100
N5 G02 Y100 R100
N6 G03X-100I-50J-50;
N7 G01Y-100
N8 G69
N9 G90 X-500Y-500
N10 M02;

(13)Return reference point(Zero point)(G28/G281/G282/G283/G284/G301/G302/G303/G304)

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

Format： G28 X(U)_ Y(V)_ Z(W)_ A_ B_ ;ZXYAB return to reference		
G281	;only X return to reference	
G282	;only Y return to reference	
G283	;only Z return to reference	
G284	;only A return to reference	
G285	;only B return to reference	
G301	;X axis return to zero	
G302	;Y axis return to zero	
G303	;Z axis return to zero	
G304	;A axis return to zero	

Pay attention: Should clear tool radius compensation and tool length

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

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

Format：

G43 H_ ;Add tool length compensate。

G44 H_ ;subtract tool length compensate。

G49 or H0 ;cancel tool length compensate.

Program tool   Real tool

Make this distance as
tool lengthen after。
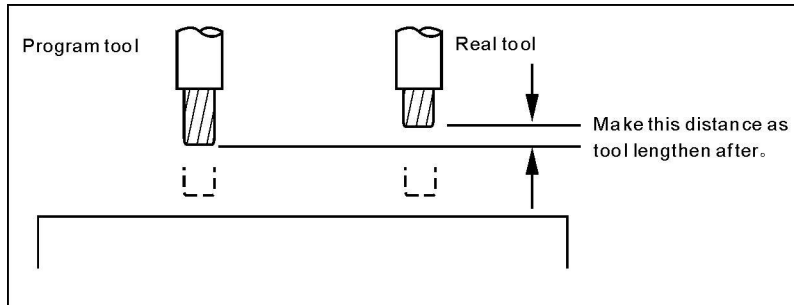
Example: N0000 G43 H2 X10   (H2 value is 5)

N0010 G44 H3 X20   (H3 value is 10)

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

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

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

Format： G45 T_   ;Add one radius。

G46 T_   ;subtract one radius。

G47 T_   ;Add two radius。

G48 T_   ;subtract two radius.

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

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

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

G45/G46/G47/G48 are non-modal instructions.
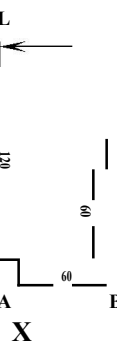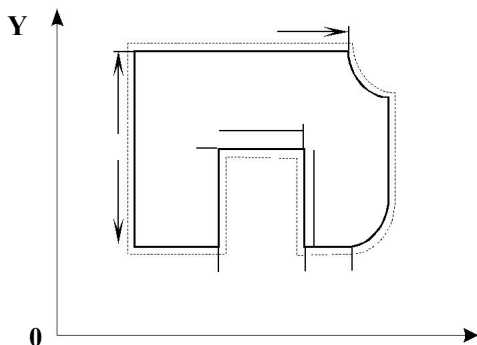
Program as follows:

N0000 G01 Z-20 F400 G91	; Z point starts
N0010 G46 T01 X55 Y55	; Point location to A point, X and Y axis both decrease one tool radius(T01)
N0020 G47 G01 X60 F200	; A-B X axis increase two tools radius
N0030 Y60	; B-C length not change
N0040 G48 X60	; C-D X axis decrease two tools radius
N0050 Y-60	; D-E length not change
N0060 G45 X30	; E-F X axis increase one tools radius
N0070 G45 G03 X30 Y30 R30	; F-G CCW radius increase one tool radius
N0080 G45 G01 Y60	; G-H Y axis increase one tool radius
N0090 G46 X0	; H-I Move for one tool radius along the negative direction of X axis
N0100 G46 G02 X-30 Y30 R30	; I-J CW radius decrease one tool radius
N0110 G45 G01 Y0	; J-K Move for one tool radius along the positive direction of Y axis
N0120 G47 X-150	; K-L X axis increase two tools radius
N0130 G47 Y-120	; L-M Y axis increase two tools radius
N0140 G46 X-55 Y-55	; M-N X and Y axis both decrease one tool radius
N0150 G26	; Program backs to the starting point to end
N0151 M02	

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

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


workpiece

Cancel tool
radius offset

Start
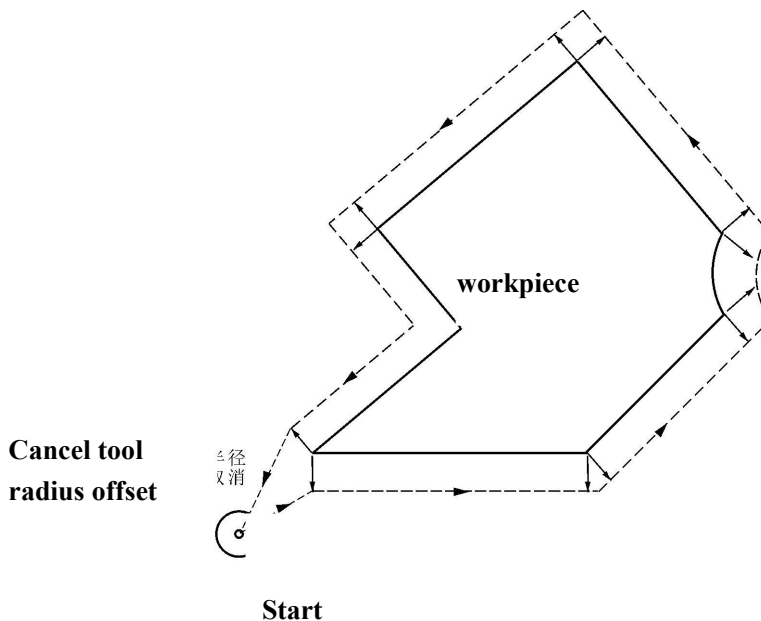
> Format：G40　(mode，original)　;Cancel compensation。
>
> G41 T－　　(mode)　　　　;tool locate to Left offset of workpiece　　　　　　　　　　　　　　。
>
> G42 T－　　(mode)　　　　; tool locate to Right offset of workpiece。

Note：

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

　　For example:
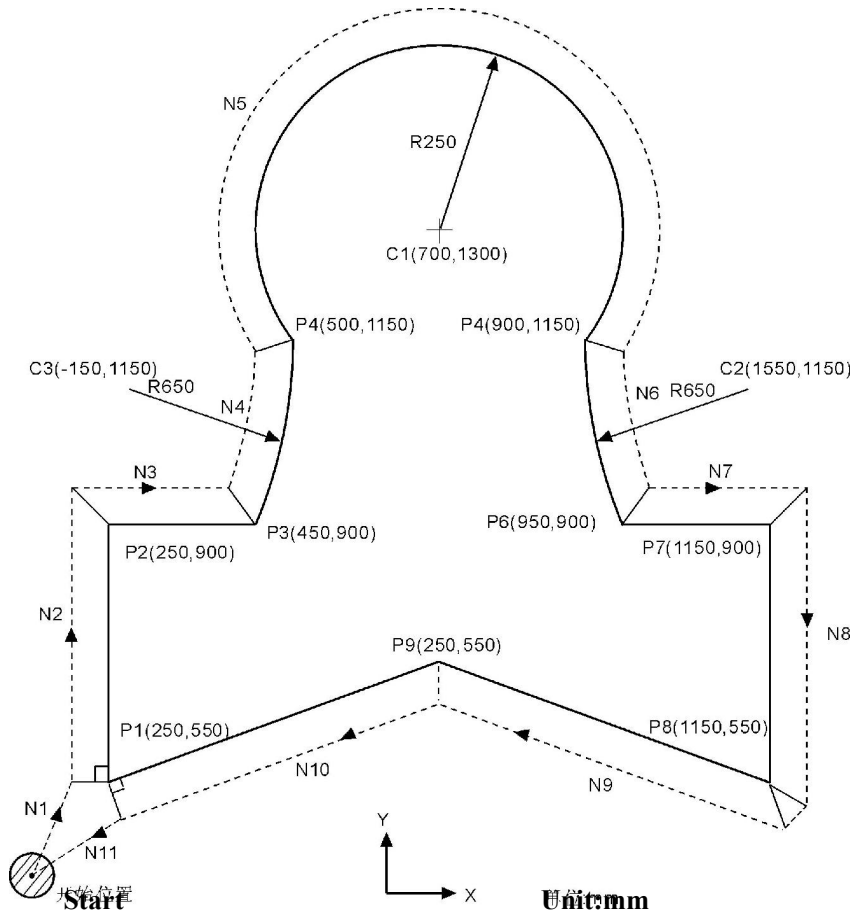
G54 X0 Y0 Z0;……………………………

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

N2 G01 Y900.0 F150; ……………………… from P1 to P2

　N3 X450.0; …………………………………from P2 to P3

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

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

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

N7 G01 X1150.0; …………………………from P6 to P7

N8 Y550.0; ………………………………from P7 to P8

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

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

N11 G00 G40 X0 Y0; ………………………   cancel compensation

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

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

```
Format：  G22 L2              ;begin
          ：        ⎫
          ：        ⎬        ;circulating
          ：        ⎭
          G800               ;end
```

For example
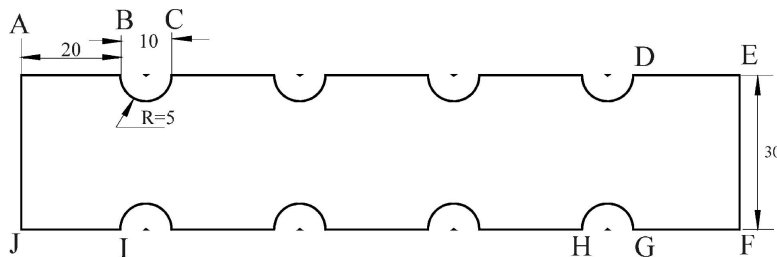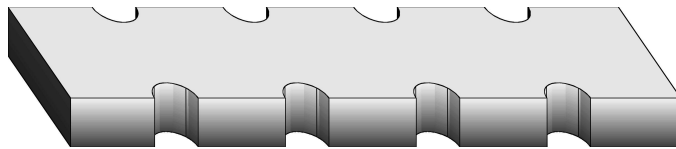
Program as follows:

N0000 G17 G90 X0 Y0 F250 M03   ;

N0001 G91 G01 Z-10

N0010 G22 L4   ;

N0020 G01 X20   ;

N0030 G03 X10 I5 J0 Y0   ;

N0040 G800   ;

N0050 G01 X20   ;D-E line

N0060 Y-30   ;E-F line

N0070 G11 X140 Y-30   ;set mirror

N0080 G22 L4   ;circulation begin

N0090 G01 X20   ;F-G line

N0100 G03 X10 I5   ;G-H arc

N0110 G800   ;circulation end

N0120 G01 X20   ;I-J line

N0130 G01 Y-30   ;cancel mirror

N0140 G12   ;J-A line

N0150 G26   ;go back to program begin point

N0160 M02   ;over

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

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

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

(19) Circle instruction（G73、G74、G80～G89）

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

Circle instruction table

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

Cycle instruction is consist of six motions

Motion 1 location of X and Y axis

Motion 2 rapid move to R point

Motion 3 machining hole

Motion 4 action in the bottom of hole

Motion 5 withdraw to R point

Motion 6 rapid move to original point
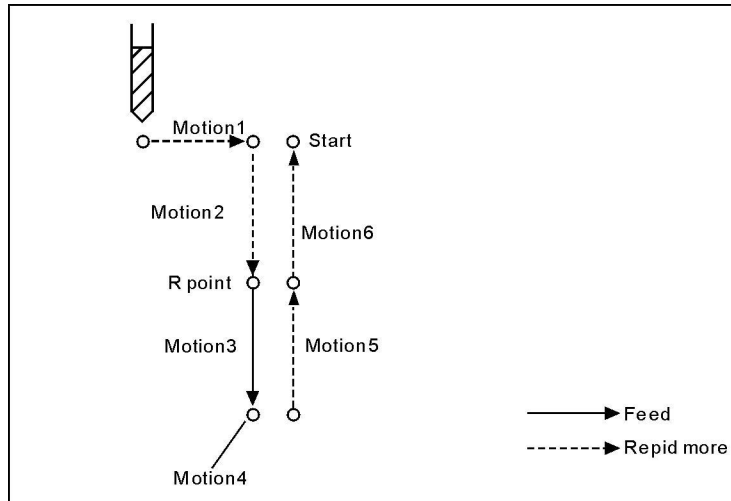
The difference of G90 and G91 as follow:
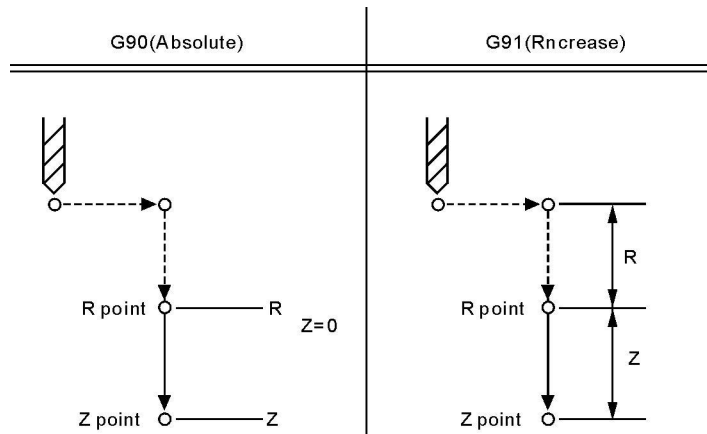
The difference of G98 and G99 as follow:
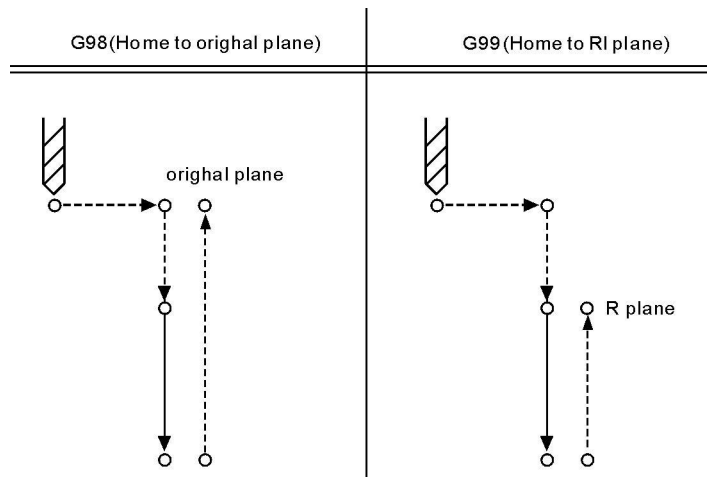
Use the L word to set cycle time, the maximum value is 9999, the default value is 1;

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

(20)High speed drill deep hole(G73)

This cycle execute high speed drilling deep hole until reaching to bottom, at

the same time, remove the cutting trifling from hole.

> **Format:** G73 X_Y_Z_R_Q_F_L_ ;
>   X_Y_:hole position data
>   Z_:the distance(G91) or coordinate(G90) from
> R point to hole bottom
>   R_: the distance(G91) or coordinate(G90) from
> original point to R point
>   Q_:cutting depth every
>   time F_:cutting speed
>   L_:repeated times
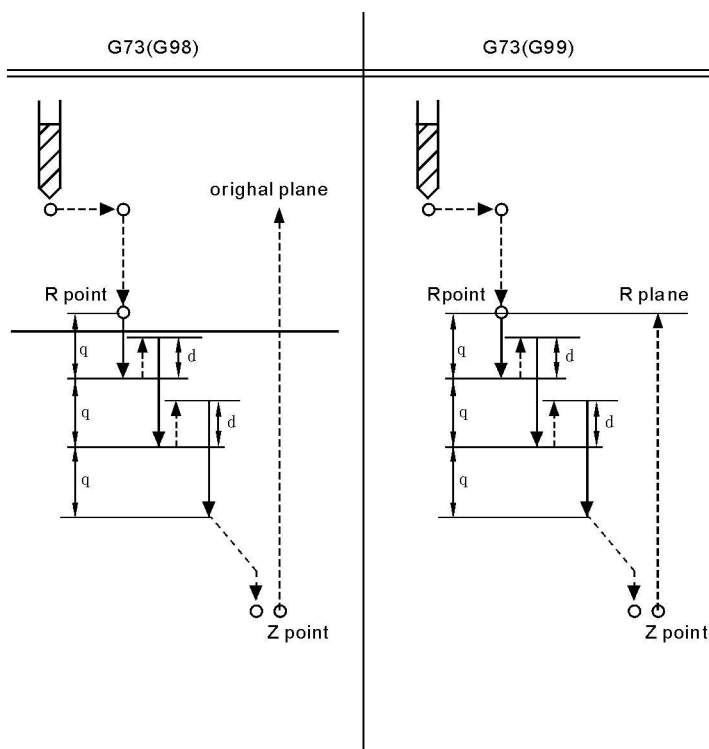
Note:High speed drilling cycle along the Z axis to carry out intermittent feeding, when using this cycle, the swarf can be easily discharged from the hole and can set the smaller value when backing. This allows to carry out drilling effectively. No.1 parameter in "N"(Processing) parameter is to set the value of the tool withdrawal(d).

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

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

For example:

M3 S2000   Spindle starts rotating

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

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

27

G98   Y-750.;   **Drill the third hole and return to R point**
G80
M30;   **The end**

(21) Drilling cycle, point drilling cycle(G81)
The cycle is used for normal drilling, execute the feeding cut to hole bottom. Then, the tool moves fast return form hole bottom.

> **Format：G81 X-Y-Z-R-F-L- ；X-Y-**
> **：Data of hole position**
> **Z-：Distance form R point to hole bottom（G91）or coordinate value（G90）**
> **R-：Distance form initial point to R point（G91）or coordinate value（G90）**
> **F-：Cutting speed**
> **L-：Repeated times**
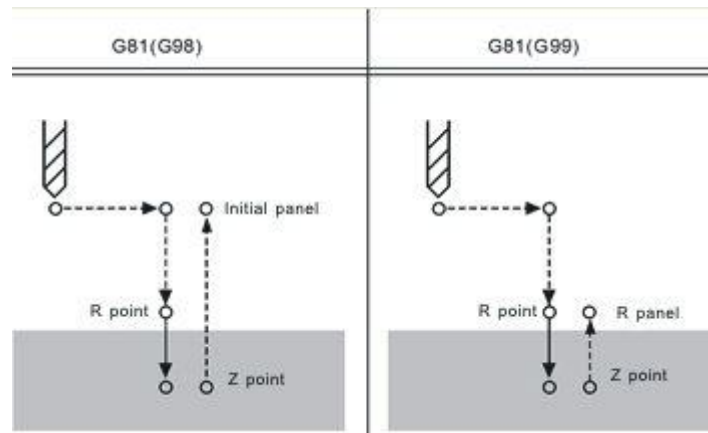
Note: Move fast to R point along the X and Y axis location, execute drilling process from R point to Z point, then tool returns with rapid move.
Use auxiliary function M code to rotate spindle before specifying G81
Locate to R point plus the offset when specifying the offset of tool length(G43 G44 or G49) in fixed cycle, the offset of tool radius is ignored.
Example:
M3 S2000; spindle begins to rotate
G90 G99 G81 X300 Y-250 Z-150 R-100 F120; drill the first hole, and then return to R point
Y-550; drill the second hole, and then return to R point
G98 Y-750; drill the third hole, and then return to panel of the initial position G80
M30; end
(22) Drill cycle(G82)

Format：G82 X-Y-Z-R-Ｐ-F-L- ; X-

         Y-: hole position data

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

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

         P-:pause time F-

         :cutting speed

         L-:repeat time
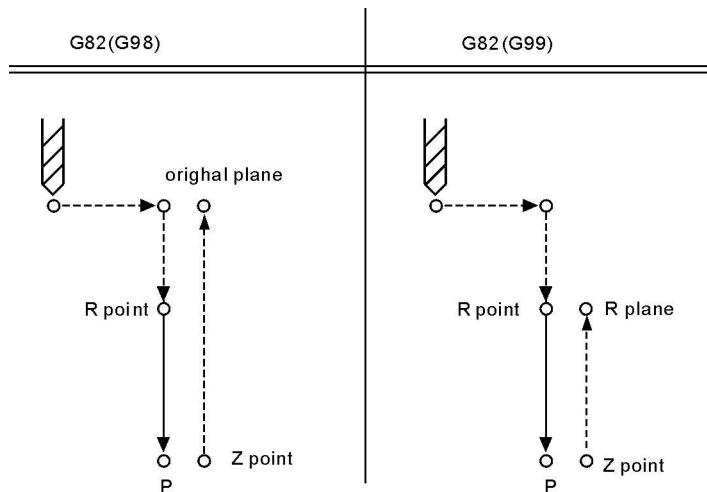
Note:

For example：

  M3 S2000

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

  F120. Y-550.;

  G98 Y-750.;

  G80

  M30;

(23) **Intermission drill cycle (G83)**

Format ：G83 X_ Y_ Z_ R_ Q_ F_ L_ ;

         X_ Y_ : hole position data

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

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

         Q_ :cutting depth every time F_ :cutting speed

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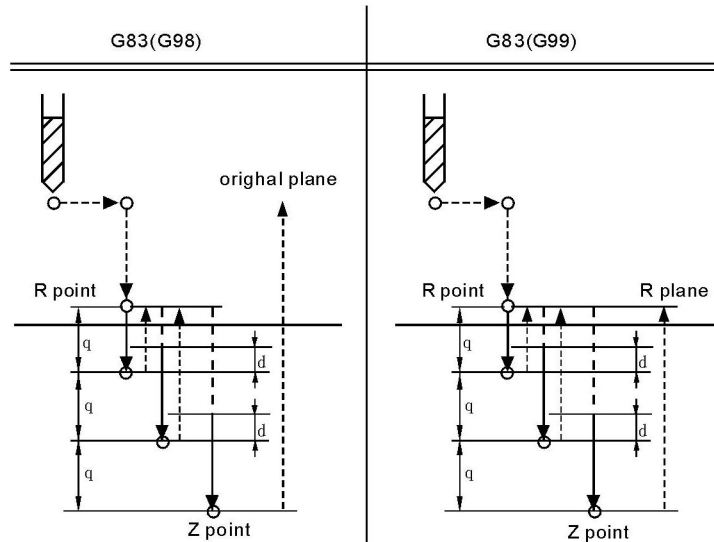
Note: Q means the cutting depth of each cutting feed, it must be specified in the incremental value. Cutting feed in the second and later must be executed quickly to move to d point to execute again before the end of last drilling, No.2 parameter in processing parameter set the cutting feed d, must specify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

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

     M3 S2000

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

     F120. Y-550.;

     Y-750.;

     G98 Y-

     750.; G80

     M30;

(24)Boring cycle(G85)

Format：G85 X_ Y_ Z_ R_ F_ L_ ; X_

         Y_: Hole position data

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

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

         F_ :cutting speed

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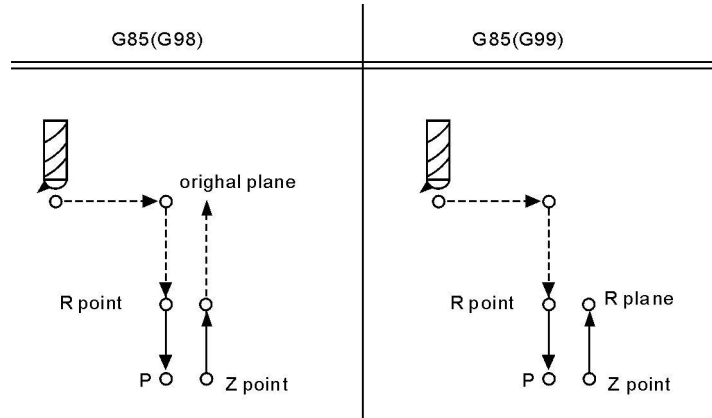
Note:Along the X and Y axis to locate, move fast to R point and carry out boring from R point to Z point, carry out cutting feed and return to R point when arriving the bottom of hole.

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

(25) Boring cycle(G86)

> Format：G86 X_ Y_ Z_ R_ F_ L_ ; X_
> Y_ : hole position data
> Z_ : the distance(G91) or coordinate(G90) from R point to hole bottom
> R_ : the distance(G91) or coordinate(G90) from original point to R point
> F_ :cutting speed
> L_ :repeated times
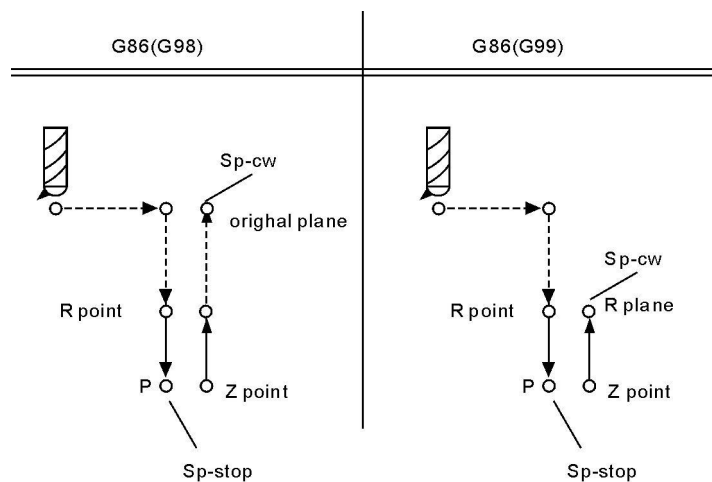
Note:Along the X and Y axis to locate, move fast to R point and carry out boring

from R point to Z point. The tool will return quickly when spindle stopping at the bottom of hole.

For example：

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

(26) Boring cycle(G89)

> **Format：** G89 X_ Y_ Z_ R_ P_ F_ L_ ;
>   X_ Y_ : hole position data
>   Z_ : the distance(G91) or coordinate(G90) from R point to hole bottom
>   R_ : the distance(G91) or coordinate(G90) from original point to R point
>   P_ :pause time
>   F_ :cutting speed
>   L_ :repeated times
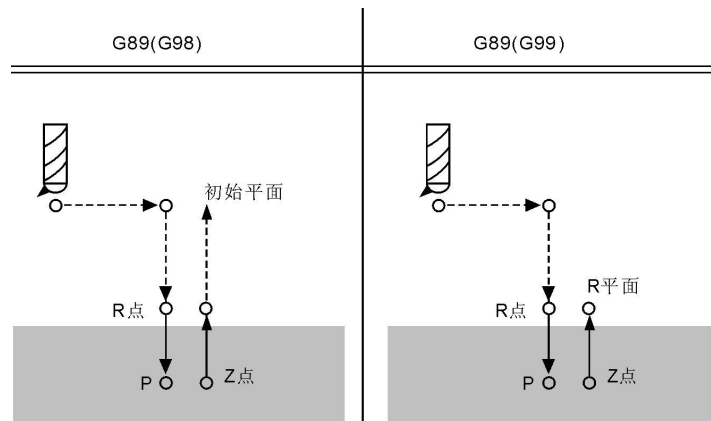
Note:It is the same as G85 unless suspending at the bottom of hole. For example:

    M3 S100
    G90 G99 G89 X300. Y-250. Z-150. R-120. P1000
    F120. Y-550.;
    Y-750.;
    G98 Y-
    750.; G80
    M30;

(27) Left tap cycle(G74)

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

Pay attention:

When the spindle and encoder drives not as 1:1, please modify the No.412 No.413 parameter in axis parameter;

412, the number of spindle teeth (requirement: less than or equal to the number of encoder teeth, must match our keysets when greater than the number of encoder teeth);

413, the number of encoder teeth

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

Format：G74 X_ Y_ Z_ R_ P_ K_ S_ L_ ;

X_ Y_ : hole position data

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

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

P_ : pause time   unit:s

K_ : screw pitch

S_ : spindle rotate speed

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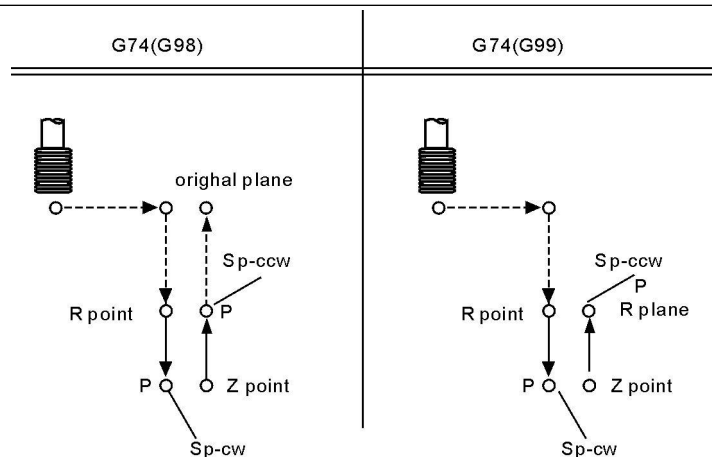
Note: The speed of coordinate axis is determined by speed of spindle and screw pitch when processing thread,it is not a matter with speed F. The system will limit the speed within maximum feeding speed.

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

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

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

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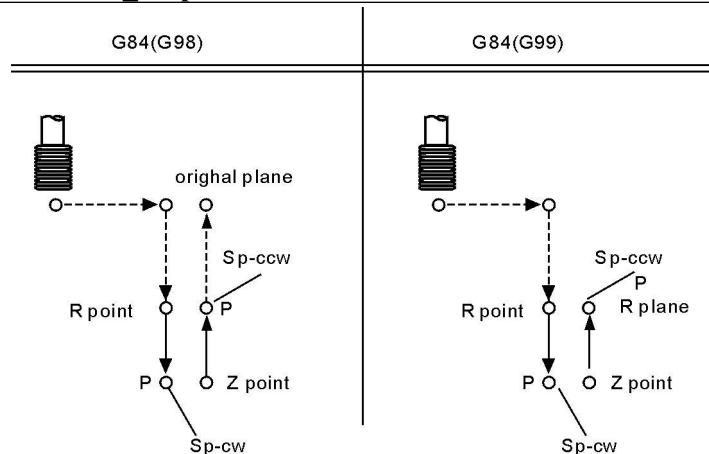
For example：

N1 M4 S100

N2 G90 G99 G74 X300. Y-250. Z-150. R-100. K5

S100 N3 G98 Y-550. K5;

N4 G80;

N5 M30;

(28) Right tap cycle(G84)

　　The same as G74 unless direction.

| Format：G84 X_ Y_ Z_ R_ P _ K_ L_ ; |
| X_ Y_ : hole position data |
| Z_ : the distance(G91) or coordinate(G90) from R point to hole bottom |
| R_ : the distance(G91) or coordinate(G90) from original point to R point |
| P_ :pause time |
| K_ :screw parameter |
| S_ :spindle rotate speed |
| L_ :repeated times |

　　For example：

　　M3 S100

　　G90 G99 G84 X300 Y-250 Z-150 R-120 P300 K5 S100

　　Y-550. K5;

　　G98 Y-750. K5;

　　G80

　　M30;

(29) Cancel cycle instruction (G80)

　　Cancel cycle instruction.

| Format：G80   ; |

　　Note: Cancel all cycle instruction and execute normal operation.

(30) Pole coordinate instruction(G15/G16)

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

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

Note:

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

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

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

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

First axis: Radius value of pole coordinate.

Second axis: Value of pole angle.

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

Use programming instruction of absolute value to specify the radius(The distance between zero point and programming point).When using local coordinate system(G52), the origin of local coordinate system changes into the center of pole coordinate system, the angle uses absolute value. As the follow shown:
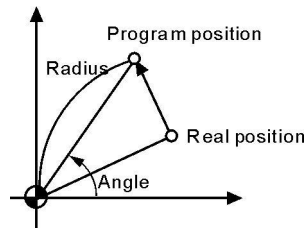
6.G91 set the current position as the origin of pole coordinate system:

Using programming instruction of incremental value to specify the radius(The distance between the current position and programming point).The angle use absolute value.As the follow shown:
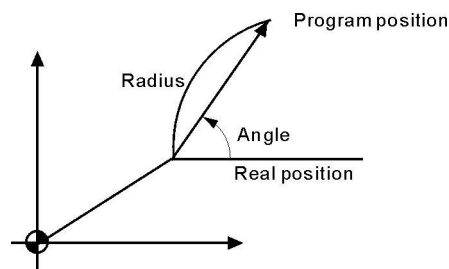
For example:
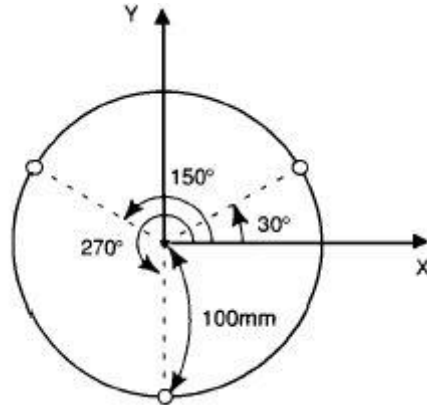
G17 G90 G16 X0 Y0;
G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0;
Y150.0;
Y270.0;
G15 G80;

(31) Switch millimeter and inch(G20/G21)

Format：
        G20 ; inch;
        G21 ; millimeter;

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:

0   **The feeding speed is specified by F code**

0   **Position instruction**

0   **Offset value of workpiece zero point**

0   **Compensation value of tool**

0   **The unit of manual pulse generator**

0   **The distance in incremental feeding**

(32) Go back the starting point of program(G26/G261/G262/G263/G264/G265）

Format ：	G26	; ZXY all go back.
	G261	; X go back.
	G262	; Y go back.
	G263	; Z go back.
	G264	; A go back.

Note：G26 motion is according to linkage type.

(33) Remember the current point(G25)

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

(34) Return to the memorial point(G61/G611/G612/G613/G614)

Format：G61   ;   Return to X Y Z of memorial point
G611   ;   Return to X of memorial point
G612 ; Return to Y of memorial point
G613   ;   Return to Z of memorial point
G614   ;   Return to A of memorial point

Note: The backing mode is G00.

(35) Check skip(G31、G311)

Format：G31 X_ Y_ Z_ A_ F_ P_   ;No alarm

        G311 X_ Y_ Z_ A_ 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.

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

The acquiescent panel of milling is G17

> Format for G17：
>
> G01(G00) X I automatical beveling, the coordinate in the next program segment must be G01(G00) Y.
>
> G01(G00) Y I automatical beveling, the coordinate in the next program segment must be G01(G00) X。
>
> G01(G00) X R automatical smoothing, the coordinate in the next program segment must be G01(G00) Y。
>
> G01(G00) Y R automatical smoothing, the coordinate in the next program segment must be G01(G00) X。
>
> Format for G18：
>
> G01(G00) X I automatical beveling, the coordinate in the next program segment must be G01(G00) Z。
>
> G01(G00) Z I automatical beveling, the coordinate in the next program segment must be G01(G00) X。
>
> G01(G00) X R automatical smoothing, the coordinate in the next program segment must be G01(G00) Z。
>
> G01(G00) Z R automatical smoothing, the coordinate in the next program segment must be G01(G00) X。
>
> Format for G19：
>
> G01(G00) Y I automatical beveling, the coordinate in the next program segment must be G01(G00) Z。
>
> G01(G00) Z I automatical beveling, the coordinate in the next program segment must be G01(G00) Y。
>
> G01(G00) Y R automatical smoothing, the coordinate in the next program segment must be G01(G00) Z。
>
> G01(G00) Z R automatical smoothing, the coordinate in the next program segment must be G01(G00) Y。

Pay attention:

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

2. The two adjacent lines must be 90
degrees. For example:

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

Beveling4x4 SmoothingR6 SmoothingR5 Beveling3x3

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

(37) The calling program(M97 M98 M99)

Unconditional jump

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

Subroutine call

    In this system the subroutine should be an independent program.

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

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

For example:

    P sub/1390 means subroutine is

tmp/NC/sub/1390 Note:

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

following 2.The subroutine must be a independent program.

    3.Method of the main program in USB calls the subroutine in USB:

P[or P]. For example:

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

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

    PP[FFDE means calling the subroutine FFDE in USB;

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

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

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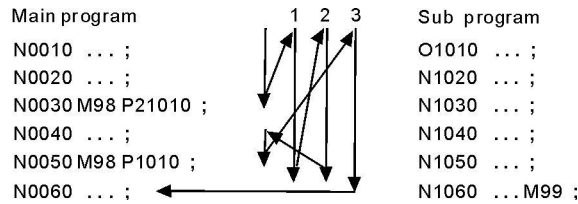
Pay attention:

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

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

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

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

The Sub-program can embedded call as follow :

Main program	Sub program	Sub program	Sub program	Sub program
O0001 ;	O1000 ;	O2000 ;	O3000 ;	O4000 ;
M98P1000;	M98P2000;	M98P3000;	M98P4000;	
M30 ;	M99 ;	M99 ;	M99 ;	M99 ;
	(First embed)	(Second embed)	(Third embed)	(Fourth embed)

For example:

Main program	1 2 3	Sub program
N0010 ... ;		O1010 ... ;
N0020 ... ;		N1020 ... ;
N0030 M98 P21010 ;		N1030 ... ;
N0040 ... ;		N1040 ... ;
N0050 M98 P1010 ;		N1050 ... ;
N0060 ... ;		N1060 ... M99 ;
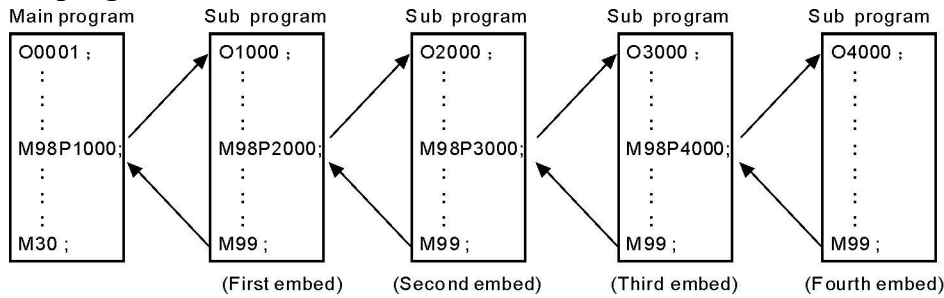
The calling instruction can be used for 9999 times in the most.
Conditional wait, jump instruction
The system of M code is used for detecting the external input signal as the condition, as follows:
Conditions wait
M12 M13 instruction are used to detect the input signal M12, M12 in program line is to detect M12 input signal is effective to execute the next program line , M13 means to detect M12 input signal is invalid to execute the next program line.The instruction is in an independent line.

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

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

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

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

M24 M25 instruction are used to detect the input signal M24, M24 in program line is to detect M24 input signal is effective to execute the next line program, M25 means to detect M24 input signal is invalid to execute the next program line.The instruction is in an independent line.

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

Plus Pxxxx (number of program line) in front of the M12 /M13 /M14 /M15 /M16 /M17 /M22 /M23 /M28 /M29 instruction. Shifting if the condition success, otherwise execute the next.
For example: M14 P0120
When the program running to this line and the system detecting the M14 input signal effectively, program will jump to the 120th line of program (the marking line),execute the next instruction if the M14 input signal is invalid.

(38) Feeding speed F function
It is the mode,actual running speed is the setting speed times the trimming rate of speed,
F is used for specify the processing speed of feeding instruction G01 G02 G03. The range is 0.01-15000mm/min,feeding speed is Fx trimming speed, F has mode function.
Executing the F instruction at the first, and then execute the motion instruction when the F instruction and motion instruction are in the same line.
(39) T/H/D function
The T/H/D function is means that tool length and radius compensate,which is mode,used by code in program。
The tool code is from T01 to T99,every tool have four tool compensate value，which is length compensation from H1 to H4 and radius compensation from D1 to D4.
(40) Spindle speed S, SS function
The system offers two ways spindle controlling modes.
The first spindle speed is specified by S, the first spindle has two kinds of gear controlling mode:
(1) The first is four gear spindle speed electrical control, output four bits code of step speed change, M41-M44 instruction control corresponds to S01-S04 output code, step speed change. Use No.50 No.51 No.52 No.53 and No.54 parameter in axis parameter to set the mode of shifting.
(2) The second uses four gears + step-less speed, M41-M44 instruction control, correspond the output S01-S04 code. Use No.42 No.43 No.44 No.45 parameter in speed parameter to set the maximum speed of corresponding gear, use No.50 No.51 No.52 No.53 No.54 parameter in axis parameter to set the mode of shifting.
Stepless speed,the range is 0-99999, output 0-10V variable-frequency voltage. The output voltage trims x10V of maximum speed of specified spindle.
Second spindle speed is specified by SS, the highest speed is controlled by the No.46 parameter in speed parameter, output 0-10V variable-frequency voltage.
(41) Macroprogram instruction

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

4. The arithmetic and logic operation

Table:

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

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

(42)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)

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

2.5 Radius compensation of tool C

  C means the system calculates the tool trajectory of radius compensation
according to the last program line and the next program line.
2.5.1 Inside and outside
    It calls inside when the included angle of tool trajectory is over 180 degrees which
is built by two program segments, it calls outside when the included angle is between

  0 and 180 degrees. As the follows:
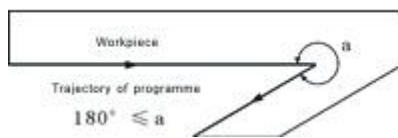
<div align="center">Inside：   Outside：</div>

2.5.2 Tool motion when starting
    The radius compensation without tool builds tool radius compensation
(1) Tool motion around the inside corner （α≥180）
    The tool center will move to the tool vector radius vertex of the starting point in
next program line.
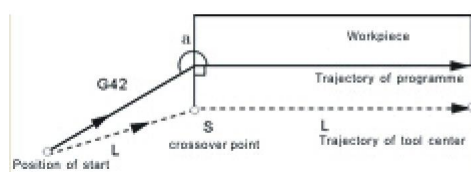
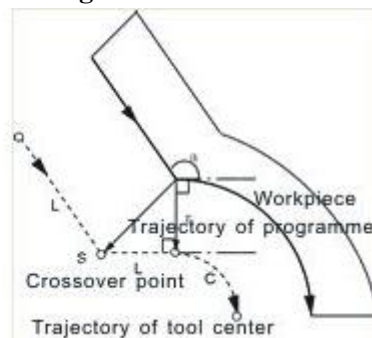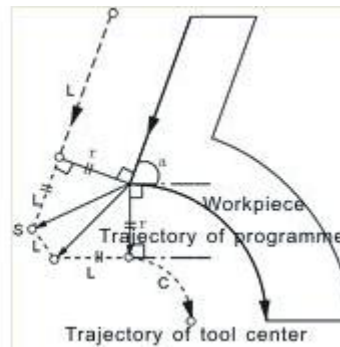<div align="center">Straight line->Straight line   Straight line->Arc</div>
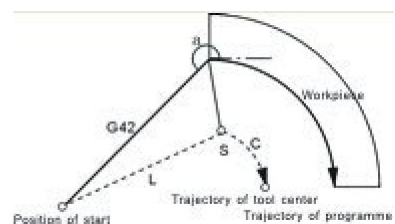
(2)The tool motion around the outside corner of obtuse angle（90≤α＜180）
   The tool center will move to the tool vector radius vertex of the end point in this program line.

  Straight line->Straight line
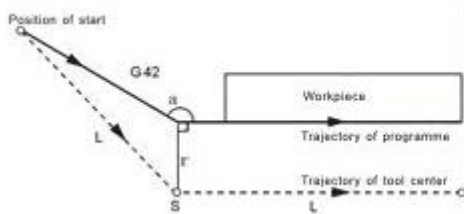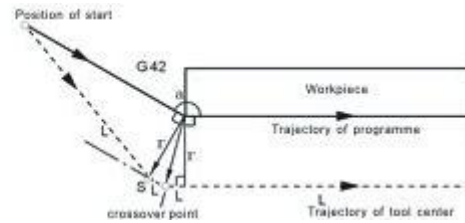
A type	B type

Straight line->Arc

A type	B type

(3)The tool motion around the outside corner of acute angle（α＜90）
   The tool center will move to the tool vector radius vertex of the end point in this program line.

     Straight line->Straight line

A type	B type

Straight line->Arc

A type	B type

47
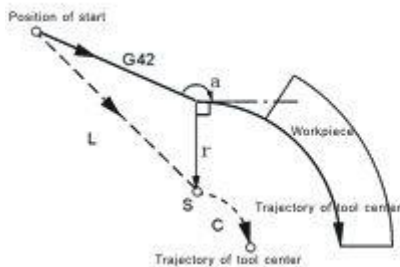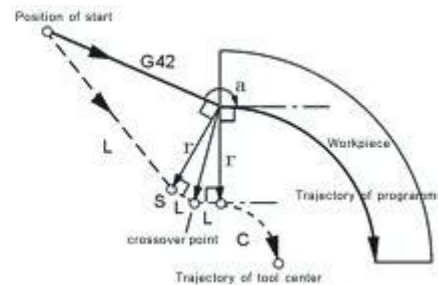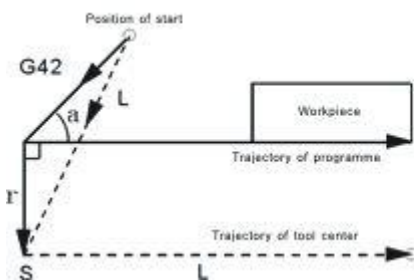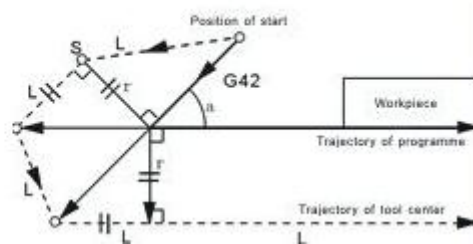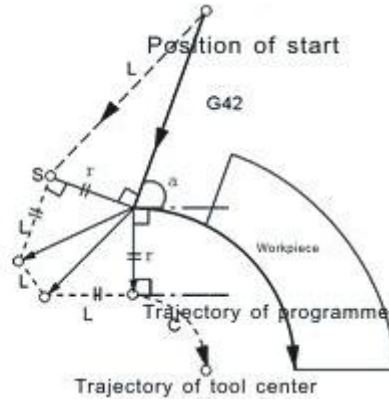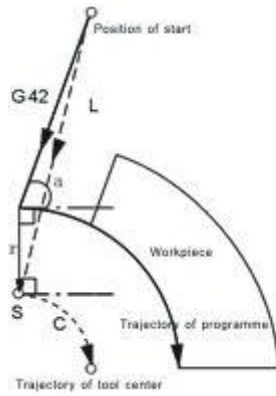
2.5.3 Tool motion in offset mode

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

Straight line->Straight line   **Straight line->Arc**
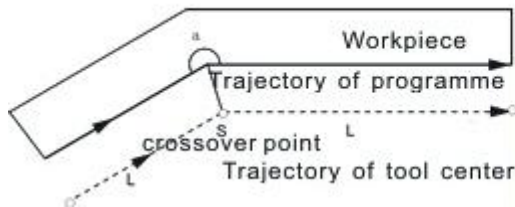
Arc-> Straight line   **Arc->Arc**

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

Straight line ->Straight line   **Straight line ->Arc**
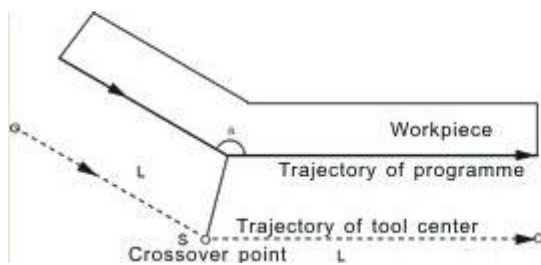
Arc-> Straight line   **Arc->Arc**

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

Straight line->Straight line   **Straight line->Arc**
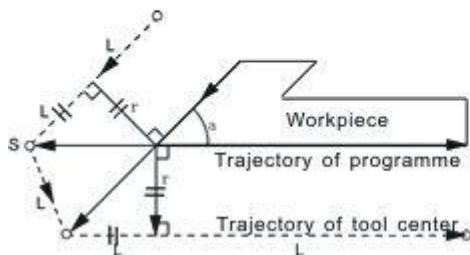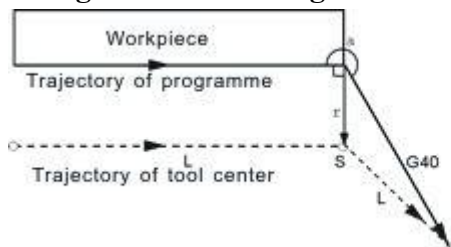

Arc-> Straight line   **Arc->Arc**
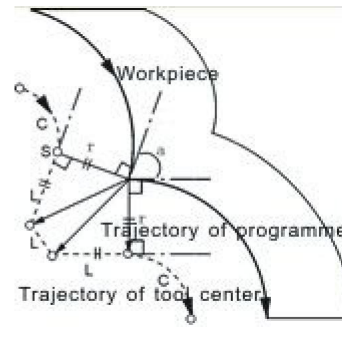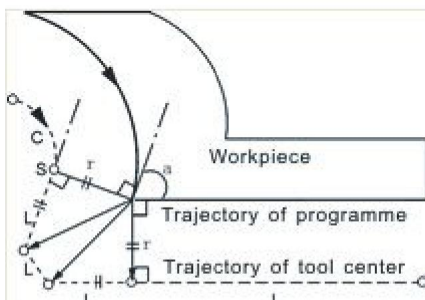
2.5.4 Tool motion in offset-cancel mode

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

The tool center will move to the tool vector radius vertex of the end point in this program line.
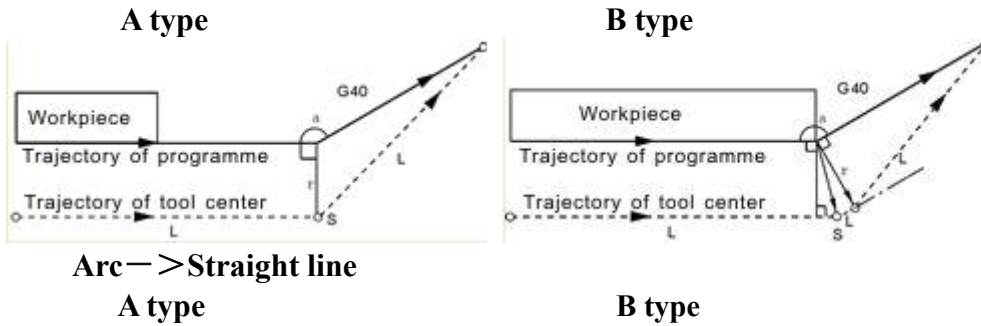
Straight line－＞Straight line   **Arc－＞Straight line**
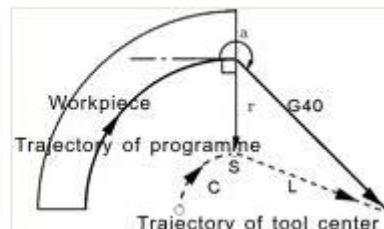
(2) The tool motion around the outside corner of obtuse angle（90≤α＜180）

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

Straight line－＞Straight line

49

A type   B type

Arc－＞Straight line

A type   B type

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

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

Straight line－＞Straight line

A type   B type

Arc－＞Straight line

A type   B type
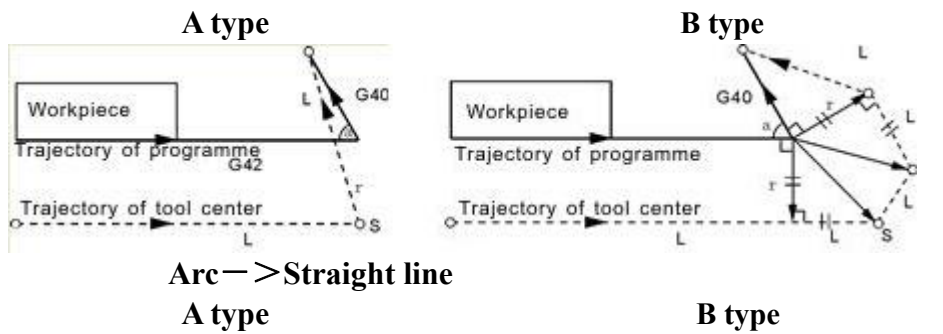
2.6 Comprehensive examples for programming

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

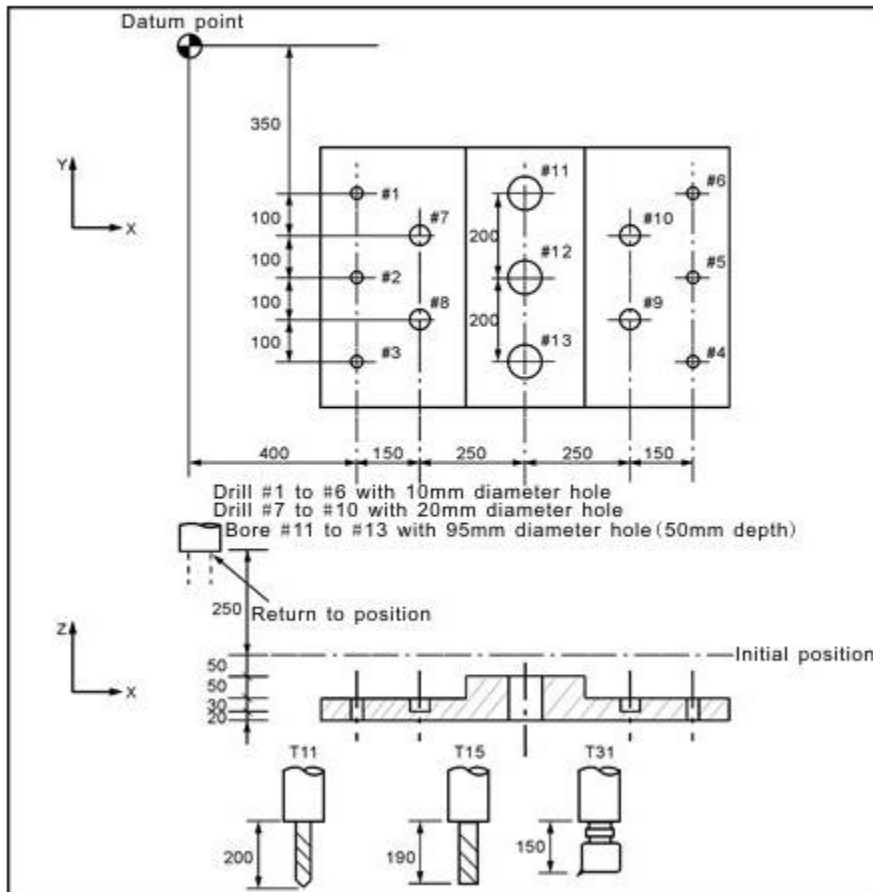   Example 1: Cut square and cut circle

50
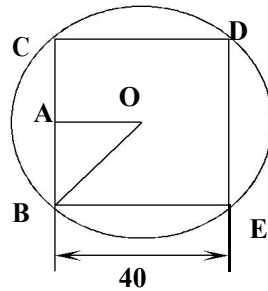
From the center to begin, the center coordinate is G54 X0 Y0

Z50 The tool radius in the D01 of the T01 parameter

N0 G54 G00 X0 Y0 Z50	Starting point for processing
N1 M03 S1000	Turn on spindle
N2 G00 Y-40 X－40 G90	Move outside of square
N3 Z10	
N4 G01 Z－10 F200	Start cutting from Z axis
N5 G01 G41 T01 D01 X-20 Y-20 F400	Move to B point
N6 Y20	Tool cuts BC line at the left of the workpiece
N7 X20	Closed angle transition in C point, cut CD line
N8 Y－20	Closed angle transition in D point, cut DE line
N9 X-25	Closed angle transition in E point,cut CB line is for smooth， move 5mm more
N10 G00 X-40 Y-40 G40	Move outside of circle
N11 G01 G41 X－20 Y－20 F500	Lengthen T01 value along A-B
N12 Z－20 F100	Start cutting from Z axis
N13 G91 G02 I20 J20 Y0 X0	Circle of contact
N14 G00 Z50	End cutting
N15 G40 G00 X0 Y0	Cancel tool compensation
N16 M05	Turn off spindle
N17 M02	Program is over

Example 2:

Drill #1 to #6 with 10mm diameter hole
Drill #7 to #10 with 20mm diameter hole
Bore #11 to #13 with 95mm diameter hole (50mm depth)

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

N1 G54 X0 Y0 Z0	Set workpiece coordinate at the datum point
N2 G90 G00 Z250.0 T11	Tool exchange
N3 G43 Z0 H1	The offset length in initial position
N4 S30 M3	Turn on spindle
N5 G99 G81 X400.0 Y-350.0 Z-153.0 R-97.0 F120	To drill #1 hole
N6 Y-550.0	To drill #2 hole and return to initial position
N7 G98 Y-750.0	To drill #3 hole and return to initial position
N8 G99 X1200.0	To drill #4 hole and return to initial position
N9 Y-550.0	To drill #5 hole and return to initial position
N10 G98 Y-350.0	To drill #6 hole and return to initial position
N11 G00 X0 Y0 M5	Return to the datum point and

	stop spindle
N12 G49 Z250.0 T15	Cancel the tool length offset to change tool
N13 G43 Z0 H1	Tool length offset in initial position
N14 S20 M3	Turn on spindle
N15 G99 G82 X550.0 Y-450.0 Z-130.0 R-97.0 F500	To drill #7 hole and return to initial position
N16 G98 Y-650.0	To drill #8 hole and return to initial position
N17 G99 X1050.0	To drill #9 hole and return to initial position
N18 G98 Y-450.0	To drill #10 hole and return to initial position
N19 G00 X0 Y0 M5	Return to the datum point and stop spindle
N20 G49 Z250.0 T31	Cancel the tool length offset to change tool
N21 G43 Z0 H1	Tool length offset in initial position
N22 S10 M3	Turn on spindle
N23 G85 G99 X800.0 Y-350.0 Z-153.0 R47.0 F150	To bore #11 hole and return to initial position
N24 G91 Y-200.0	To bore #12 hole and return to initial position
N25 Y-200.0	To bore #13 hole and return to initial position
N26 G90 G28 X0 Y0 M5	Return to the datum point and stop spindle
N27 G49 Z0 G80	Cancel the tool length offset
N28 M02	Program is over

2.7 Usage for automatic tool setting gauge

1. Note for parameter:
   Define macro variables of the automatical tool setting gauge function are as follows (corresponding to the other parameters P380 - P389):
   #380: The X axis lathe coordinate of initial position with automatical tool setting;(Unit:mm)
   #381: The Y axis lathe coordinate of initial position with automatical tool setting;(mm)
   #382: The Z axis lathe coordinate of initial position and returning point with automatical tool setting;(mm)

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

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

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

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

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

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

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

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

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

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

3. Automatic tool setting steps:

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

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

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

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

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

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

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

2.8 Usage for automatical dividing center

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

    1) Choose the current coordinate system such as G54;

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

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

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

    1) Choose the coordinate system such as G54;

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

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

Chapter 3   Operation explanation

3.1 Summary

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

3.2 Operation panel

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

3.3 Keyboard description

3.3.1 Rate increase or decrease

(1) Rapid override(G)

There are six gears in rapid override form 5% to 100%, by adjusting the key of rapid override is for the following instruction: G00,G26,G28,G611,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 Usage for intervention switch

(1) The left: normal processing.

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

(3) The right: suspend automatic feeding. Return to the left to the normal status. Spindle is out of control when running with non-coordinate axis.

3.3.3 Others keys

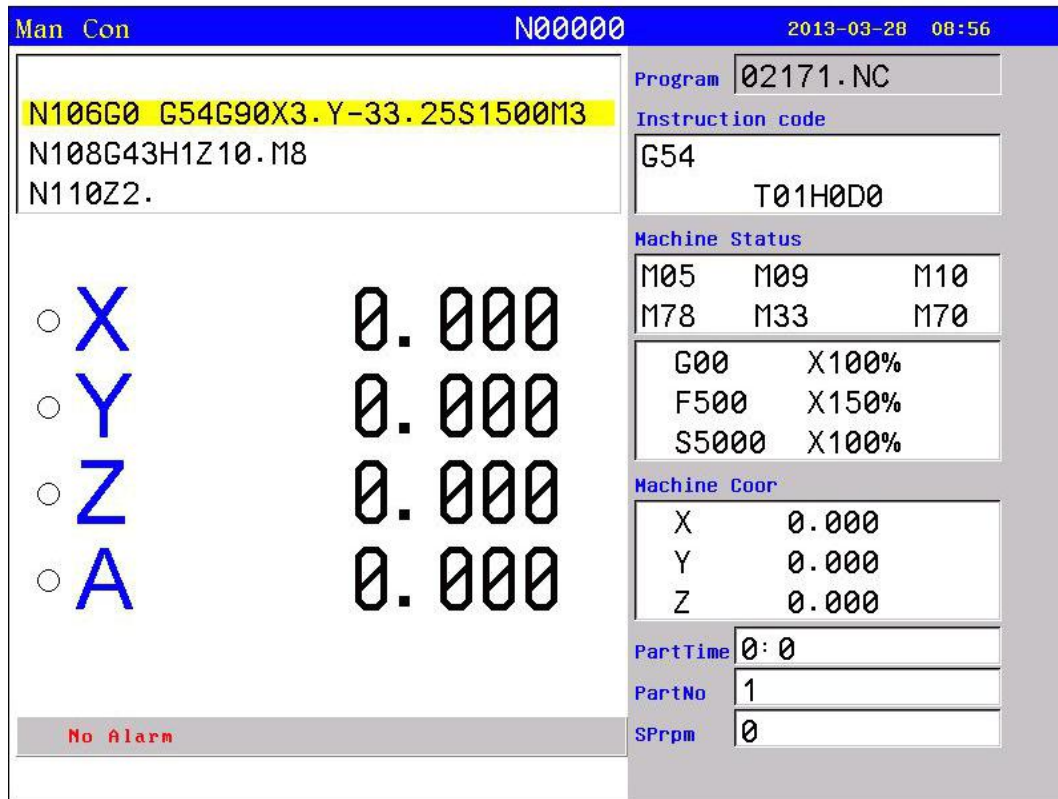| Keyboards	Functions

Letter key Number key	A B C S E F G H I J K L M N O P Q R S T U V W X Y Z 1 2 3 4 5 6 7 8 9 .- ： for program instructions, parameters' edition; number keys are used for inputting data and selecting sub-menu.
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 "M D I" entering MDI function " "selecting auto-coordinates/diagram machining " "for single segment or constant work " " for coordinates mode or diagram mode speedy simulating " "for manual increment or manual continuous

Control key	" ⬛ ‖ ⬛ ‖ ⬛ ‖ ⬛ "spindle cw, ccw rotation
	" 🚰 " coolant on/off
	" ⚙ "for the shift between electric tool carrier and gang tool carrier
	" 〰 " for the shift between hand-driven continuous high speed and low speed.
	" ⊕ " all axes return to datum point " 🔧 " for spindle chuck on/off
	" ⬛ " for thumbstall on/off
	" ⬆H ⬇H "handwheel gear selection
	" ⬆S ⬇S " adjusting spindle speed
	" ⬆F ⬇F "adjusting feed speed
	" ⬆G ⬇G "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 main interface

```
Man Con                    N00000        2013-03-28  08:56

N106G0 G54G90X3.Y-33.25S1500M3          Program 02171.NC
N108G43H1Z10.M8                         Instruction code
N110Z2.                                 G54
                                                T01H0D0

                                        Machine Status
  ○ X      0.000                        M05    M09        M10
                                        M78    M33        M70
  ○ Y      0.000                          G00      X100%
                                          F500     X150%
  ○ Z      0.000                          S5000    X100%
                                        Machine Coor
  ○ A      0.000                          X       0.000
                                          Y       0.000
                                          Z       0.000
                                        PartTime 0:0
                                        PartNo   1
        No Alarm                        SPrpm    0
```

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.
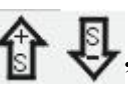
(2) " [icon] ": Switching cycle from "manual continuous" to "manual increment"

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

(4)"I"： Modify the increment in manual increment

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

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

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

(8)" [icon] [icon] ": To switch cycle "0.001" "0.01" "0.1"or "0.1" "0.01" "0.001" in the handwheel function.

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

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

(11)"Auto": Select automatic mode.

(12)"Manual": Select manual mode.

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

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

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

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

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

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

(19) "Back to datum point": Press and X or Z, the X or Z axis goes back to the datum point automatically; Press "0"X axis firstly and then Z axis; Press "Esc" to cancel the construction. The speed controlled by No.31 No.33 parameter in speed parameter, the direction is determined by No.28 parameter in axis parameter.

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

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

(22) "Switch speed": Press to switch the speed to system speed which is changed by No.1 No.2 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) "Workpiece number clear": Press Del and Enter.

(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.

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 to stop feeding. Making sure the axis and using "+X、-X、+Y、-Y、+Z、-Z、+A、-A" to feed, the speed of feed is determined by display on the interface(F) times the rate.

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

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

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

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

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

3.4.3 Manual increment

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

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

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

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

Switch for datum point:

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

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

Instruction:

1. The system requires for backing to the datum point every time when it is power on, the requirement can be set by No.38 parameter in axis parameter, it can be prompt or force;

2. The way and type of detecting signal can be set by No.39 parameter in axis parameter, so detect the switch of datum point is effective, also detect the Z pulsing signal of electrical motor after detecting the switch of datum point (precision higher), detect forward or reverse for Z pulsing signal of electrical motor.

3. The direction for backing to datum point can be set by No.40 parameter in axis parameter, D2 D3 D4 correspond to X Y Z axis, 0 is forward, 1 is reverse.

4. The type of the switch for datum point can be set by No.41 parameter in axis parameter, D0 D1 D2 correspond to X Y Z axis,0 is always on, 1 is always off.

5. The maximum length of detecting Z pulse of electrical motor can be set by No.37 No.38 No.39 parameter in axis parameter, the value must less than the pulse of electrical motor run a cycle.

6. The shifting distance after backing to datum point can be set by No.46 No.47 No.48 parameter in axis parameter, rapid move coordinate to the value of parameter after backing to datum point.

No switch for datum point:

To set floating point to make sure, turn on corresponding function of floating point by No.33 parameter in axis parameter, setting No.34 No.35 No.36 parameter to make sure the floating point of X Y Z axis, the datum point of lathe.

The steps to set floating point as follows:

1. Setting the No.33 parameter in axis parameter to set the axis which is starting up floating point. For example: Turn X axis on is "00001000". (turn all of them on is 00111000.)

2. Moving X axis to designated position so that set floating point.

3. Press "Parameter", "Axis parameter" and select No.34 parameter, "Enter", popup a dialog box of X axis' floating point coordinate. Import the value of setting lathe coordinate.

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

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

The method to set floating point of Y Z axis is the same as the above to set X axis. Operation for backing to the datum point:

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

If stop in the process, press "Stop" or "Reset" to stop backing to the datum point. Pay attention:

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

3.4.5 Handwheel

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

Panel

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

Hand held
Press "handwheel" and operate the switch of axis selection to select an axis, operate the axis and switch of handwheel override to adjust the gear.
Instruction
The handwheel is mainly used for "Tool", the speed and the handwheel feed of one measure is related to rotate the handwheel fast or low. The speed is not too fast best when the system cooperate with stepping motor.

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

Handwheel is of no effect in auto-coordinates diagram machining, it only works in working coordinates.
3.4.6 Workpiece coordinate system
1. Press "MDI" button to select the corresponding workpiece coordinate system (G54-G59);
2. Moving   the axis which is going to set tool to the specified location;
3. Press "Setup" to import the corresponding coordinate of workpiece coordinate system..

3.5 Auto operation

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

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

Press "Auto" to enter automatical mode in manual mode.

Switch display of coordinate: Press "Pgup" "Pgdn" to switch the interface into "absolute coordinate" "relative coordinate" "synthesize coordinate".
3.5.1 Automatical process

"Single or continuous": Press " " to switch cycle.

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

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

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

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

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

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

3.5.2 Processing at arbitrary program line or with arbitrary tool

A. Run at actual line

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

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

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

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

B. Run at the marked line

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

Pay attention:

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

3.5.3 Start program

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

(1) Press "Run"

(2) Switch on the Run of external signal.

3.5.4 Stop processing program

Five methods as follows to stop:

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

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

(3) Intervention switch in the middle or right.

(4) Switch on the Halt of external signal.

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

3.5.5 Real-time control in automatical process

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

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

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

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

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

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

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

3.5.6 The operation mode of MDI

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

3.5.7 The operation mode of Handwheel

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

Pay attention: The acceleration, deceleration and maximum speed of running handwheel are controlled by No.23-No.29 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 32Mbit 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、Y、Z、A axis are limited forcedly positive: X、Y、Z、A axis are in the positive position which is limited forcedly.

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

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

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

(5) X, Y, Z, A 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、T、R、Q、A、B、C correspondingly to corresponding interface after enter the parameter interface, press "↑ ↓" to select the number of parameter and press "Enter" to popup a dialog box to import data and press "Enter" again to fix parameter successfully.

Instruction for parameter as follows:

3.7.1 User parameter (processing)

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. 17, Running program need Sp run [1 mean Yes, 0 mean No]
18, Set M20 the time of auto-running [Negative mean immensity loop] 19, Set part count
21, G01/G02/G03 line delay (ms)[>100]
22, G00 line delay (ms) [>100]

23, Handwheel acceleration [50-100]

200, system screen protect
times [>=2minutes]

3.7.2 Speed parameter

1, X-axis's G00 speed(mm/min)

2, Y-axis's G00 speed(mm/min)

3. Z-axis's G00 speed(mm/min)

4. A-axis's G00 speed(mm/min)

5. Manual maximum feed speed(mm/min)

6, Auto Maximum feed speed(mm/min) 7,
G01/G02/G03 default speed(mm/min) 8,
Null run speed(mm/min)

9, Feeding axis`s manual speed(mm/min)

10, Spindle`s manual speed(rpm)

11, Beginning feed speed(mm/min)

12, The maximum mutation of feeding axis
speed(mm/min) 13, Limit G1G2G3 axis speed
[1 mean Yes,0 mean No]

14, X G1G2G3 max speed(mm/min)

15, Y G1G2G3 max speed(mm/min)

16. Z G1G2G3 max speed(mm/min)

17. A G1G2G3 max
speed(mm/min) 18, X acceleration
[1～99999]

19, Y acceleration
[1～99999]

20. Z acceleration
[1～99999]

21. A acceleration
[1～99999]

22, Auto run
acceleration [1-500]

23, Handwheel
acceleration [500--30000]

24, Run program Handwheel
acceleration [>500]

25, Run program Handwheel G00
speed(mm/min) [>10]

26, Handwheel X limit speed(mm/min)

27, Handwheel Y limit speed(mm/min)

28. Handwheel Z limit speed(mm/min)

29. Handwheel A limit
speed(mm/min) 30, Acceleration type
[0 mean line,8 mean curve]

31, Curve initial
acceleration [>=10]

32, Curve acceleration with second-
order [>=10]

33, Curve max
acceleration [>=500]

34, X go home positive speed(mm/min)

35, X go home negative speed(mm/min)

36, Y go home positive speed(mm/min)

37, Y go home negative speed(mm/min)

38. Z go home positive speed(mm/min)

39. Z go home negative speed(mm/min)

40. A go home positive speed(mm/min)

41. A go home negative speed(mm/min) 42,
Spindle max speed in the first gear(rpm)

43, Spindle max speed in the second gear(rpm)

44. Spindle max speed in the third gear(rpm)

45, Spindle max speed in the fourth gear(rpm)

46, Second Spindle max speed(rpm)

47, The mode of arc gap compensation (0 means A: The lager gap the faster compensation speed,8 means B: compensation speed is set by the parameter, +4 means: Arc programme I J K are the coordinate which is from the end point to the center)

48, The speed of the B type of gap compensation (mm/min)

48-1, The beginning speed of the B type of gap compensation (mm/min)[>10]

48-2, The acceleration of the B type of gap compensation(mm/min)/s)[>10]

49, Activate the function of speed processing [1 means yes, 0 means no]

50, The stop speed of handwheel (mm/min)

[>100] 58, The speed of hard limited(mm/min)

3.7.3 Axis parameter

1,Feed axis band switch
[1 mean Yes,0 mean No]

2,Spindle band switch
 [1 mean Yes,0 mean No]

 3,X-axis`s negative scope(mm)

 4,X-axis`s positive scope(mm)

 5,Y-axis`s negative scope(mm)

 6,Y-axis`s positive scope(mm)

 7,Z-axis`s negative scope(mm)

 8,Z-axis`s positive scope(mm)

 9,A-axis`s negative scope(mm)

 10,A-axis`s positive scope(mm)

 11,Spindle stop time(10ms)

 12,Spindle stop long
signal [0 mean No,1 mean
Yes]

13,Soft limited invalid

[D2X;D3Y;D4Z;D5A;]
14,X-axis`s reverse compensation(um)
 15,Y-axis`s reverse compensation(um)
 16,Z-axis`s reverse compensation(um)
 17,A-axis`s reverse compensation(um)
 18,X-axis's direction signal
[1 mean normal,0 mean reverse]
 19,Y-axis's direction signal
 [1 mean normal,0 mean reverse]
 20,Z-axis's direction signal
[1 mean normal,0 mean reverse]
21,A-axis's direction signal
 [1 mean normal,0 mean reverse]
 22,Close feed electron
gear [1 mean Yes,0 mean
No]
 23,X-axis's electron gear numerator(1-999999)
 24,X-axis's electron gear denominator(1-999999)
 25,Y-axis's electron gear numerator(1-999999)
 26,Y-axis's electron gear denominator(1-999999)
 27,Z-axis's electron gear numerator(1-999999)
 28,Z-axis's electron gear denominator(1-999999)
 29,A-axis's electron gear numerator(1-999999)
30,A-axis's electron gear denominator(1-
999999) 31,XYZA positive limit
 [0 open,1 close]
 32,XYZA negative
limit [0 open,1 close]
 33,float zero bit parameter
 [D3X;D4Y;D5Z;D6A;0 machine Zero;1 float Zero]
34,X coordinate of floating zero point setting
 35,Y coordinate of floating zero point setting
 36,Z coordinate of floating zero point setting
 37,A coordinate of floating zero point setting
 38,Feed axis returns requirement
[1 mean No use, 0 mean clew, 8 compulsion , 9 must
compulsion] 39,Feed axis returns mode
 [0 reverse check,1 reverse No check ,2 No reverse check,3 No reverse No
check]
 40,Home reverse direction
[D2X;D3Y;D4Z;D5A; 0 Positive;1 Negative]
 41,Home switch set
[D0X;D1Y;D2Z;D3A;1Close;0Open]
42,X check zero max length(100um)
 43,Y check zero max length(100um)
 44,Z check zero max length(100um)
 45,A check zero max length(100um)
 46,X Home offset(10um)
 47,Y Home offset(10um)
 48,Z Home offset(10um)
 49,A Home offset(10um)
50,Have Spindle class control
[1 mean open,0 mean close]
 51,Spindle class speed(1/100rpm)
 52,Spindle class direction

[0 mean M03,1 mean M04]

53,Spindle class stop time(10ms)

54,Spindle class time(10ms)

55,Spindle stop time(10ms)

56,Check SP encode

[1 mean Yes,0 mean No]

57,SP encode pulse

68, Negative delay time of feeding axis(ms) 80,XZ axis coordinate plan

[D2Zwordpiece,D3Xwordpiece,D4Ztool,D5Xtool,D6Zcircumrotate,D7Xcircumrotate

]

81, Y axis

[0 mean rotating axis,1 mean line axis]

82,Y is rotating axis workpiece coordinate [0 No;1 plan]

83,Y is rotating axis lathe coordinate

[0 No;1 plan]
100,Inner parameter

101,The fourth axis setting function

[0 mean rotating axis,1 mean line axis]

102,The fourth axis is rotating axis lathe coordinate

[0 No;1 plan]

3.7.4 Tool parameter

1,Radius C compensation's establish(0 mean A,1 mean B)

2,Radius C compensation's cancel(0 mean A,1 mean B)

3.7.5 Other parameter

1,Set   sub-panel type [1 hand hold,0 panel] 3,use control switch [1 Yes, 0 No]
4,Have auto lubricate(0 yes/1 no) 5,Auto lubricate time(0.01s)
6,Auto lubricate stop time(s)

7,Door switch checking M12(0 no,1 yes) 8,Door switch(0 open,1 close)

9,bit parameter

D0：Null;

D1："1"Start CNC system clear part Number.;

D2："1"Automatic space before letter when editing program;

D3：Null;

D4：Null;

D5："1"Do not stopping SP and cooling when pressing "Reset";

D6："1"G00 X and Z's peed by oneself;

D7："1"Tool redeem by oneself;

D8："1"Save SP chuck(M10/M11) state when power off;

D9：Tool redeem input Mode1 or Mode2;

D10："1" Program edit automatic compositor Line;

D11："1" First SP +10V output from second output port;

D12："1" Shield skip function ("/"is invalidation);

D13："1" Sheld go home function;

D14："1" Sheld "run" key;

D15："1" Tool redeem display

relative,"0"absolute; 10,Auto count part

[1 mean Yes,0 mean No]

11,Program edit number

increase 12,Inner parameter

13,Does lock for Spindle & chuck(0 mean no)

14,Is available keys of lub&cool as running(0 mean

no) 17,servo 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) 22,Outside chuck control(0 no,1

yes)

24,M10M11   short   signal

time(s) 26,Emerge Stop(0 open,1

close) 27,Emerge Stop2(0 open,1

close)

28,Run status output M69 STOP output M65(0 invalid,1

valid) 29,Alarm status output M67(0 invalid,1 valid)

30,Set language(1means Chinese, 0 means

English) 31,Is enable I/O PLC program

32,Is enable High speed I/O PLC program

35,soft-limit without home as manual

[1 Yes,0 No] 36,Set

system time

[year-month-day-hour-

minute] 37,Velocity of RS232

[0=7200；1=9600；2=14400；3=19200；4=38400；5=57600；6=115200]

38,Manually rapid key to lock and save

[8 Yes]

39,Special parameter

40,Special parameter 41,Bake

current parameter 42,Resume

original parameter

200.X axis feed back the error of running alarm. (pulse)[>1] 201.Y axis feed back the error of running alarm. (pulse)[>1] 202.Z axis feed back the error of running alarm. (pulse)[>1] 203.Fourth axis feed back the error of running alarming. (pulse)[>1] 205.X axis feed back the error of stop alarming. (pulse)[>1]

206.Y axis feed back the error of stop alarming. (pulse)[>1]

207.Z axis feed back the error of stop alarming. (pulse)[>1]

208.Fourth axis feed back the error of stop alarming.
(pulse)[>1] 210.X axis feed back electrical gear molecules.
   [Automatical calculate and import: crew pitch (um) and the number of encoder.] 211.Y axis feed back electrical gear molecules.
   [Automatical calculate and import: crew pitch (um) and the number of encoder.] 212.Z axis feed back electrical gear molecules.
   [Automatical calculate and import: crew pitch (um) and the number of encoder.] 213.Fourth axis feed back electrical gear molecules.
   [Automatical calculate and import: crew pitch (um) and the number of encoder.] 215.X axis feed back electrical gear denominators.
   [Automatical calculate and import: crew pitch (um) and the number of encoder.] 216.Y axis feed back electrical gear denominators.
   [Automatical calculate and import: crew pitch (um) and the number of encoder.] 217.Z axis feed back electrical gear denominators.
   [Automatical calculate and import: crew pitch (um) and the number of encoder.] 218.Fourth axis feed back electrical gear denominators.
   [Automatical calculate and import: crew pitch (um) and the number of encoder.] 300.Feed axis matches absolute encoder motor or not.
   [X—D2, Y—D3,Z—D4,A—D5, "0" means not match, "1" means match.] 301.Absolute encoder address of lower 16bits multi-turn data.
302.Absolute encoder address of higher 16bits one-revolution
data 303.Absolute encoder address of lower 16bits multi-turn
data. 304.X axis of each cycle absolute encoder pulse number.
305.Y axis of each cycle absolute encoder pulse number.
306.Z axis of each cycle absolute encoder pulse number.
307.Fourth axis of each cycle absolute encoder pulse number.
309.X axis of each cycle absolute encoder corresponding coordinate
distance.[nm] 310.Y axis of each cycle absolute encoder corresponding
coordinate distance.[nm] 311.Z axis of each cycle absolute encoder
corresponding coordinate distance.[nm] 312.Fourth axis of each cycle absolute
encoder corresponding coordinate distance.[nm]
314.X axis of absolute encoder multi-turn data
   shifting. [Import "E" to clear multi-turn data]
315.Y axis of absolute encoder multi-turn data
   shifting. [Import "E" to clear multi-turn data]
316.Z axis of absolute encoder multi-turn data
   shifting. [Import "E" to clear multi-turn data]
317.Fourth axis of absolute encoder multi-turn data
   shifting. [Import "E" to clear multi-turn data]
601.One key to define step.
602.One key to define servo.
3.7.6 Coordinate system
   The parameter has the function of multiple coordinate system, includes 6 workpiece
coordinate system and a lathe coordinate system G53. A processing program can set a

workpiece coordinate system, workpiece coordinate system could move its original to change,the value of coordinate system in parameter is coordinate value of the original point (zero point) in the lathe coordinate system.

Using G54 to G59 to set 6 workpiece coordinate system, in coordinate system interface could modify the coordinate value of original of 6 workpiece coordinate system in lathe coordinate system.

Parameter:

1. X offset of G54 mm[incremental modification, enter E to clear]
2. Y offset of G54 mm[incremental modification, enter E to clear]
3. Z offset of G54 mm[incremental modification, enter E to clear]
4. A offset of G54 mm[incremental modification, enter E to clear]
5. B offset of G54 mm[incremental modification, enter E to clear]
6. X offset of G55 mm[incremental modification, enter E to clear]
7. Y offset of G55 mm[incremental modification, enter E to clear]
8. Z offset of G55 mm[incremental modification, enter E to clear]
9. A offset of G55 mm[incremental modification, enter E to clear]
10. B offset of G55 mm[incremental modification, enter E to clear]
11. X offset of G56 mm[incremental modification, enter E to clear]
12. Y offset of G56 mm[incremental modification, enter E to clear]
13. Z offset of G56 mm[incremental modification, enter E to clear]
14. A offset of G56 mm[incremental modification, enter E to clear]
15. B offset of G56 mm[incremental modification, enter E to clear]
16. X offset of G57 mm[incremental modification, enter E to clear]
17. Y offset of G57 mm[incremental modification, enter E to clear]
18. Z offset of G57 mm[incremental modification, enter E to clear]
19. A offset of G57 mm[incremental modification, enter E to clear]
20. B offset of G57 mm[incremental modification, enter E to clear]
21. X offset of G58 mm[incremental modification, enter E to clear]
22. Y offset of G58 mm[incremental modification, enter E to clear]
23. Z offset of G58 mm[incremental modification, enter E to clear]
24. A offset of G58 mm[incremental modification, enter E to clear]
25. B offset of G58 mm[incremental modification, enter E to clear]
26. X offset of G59 mm[incremental modification, enter E to clear]
27. Y offset of G59 mm[incremental modification, enter E to clear]
28. Z offset of G59 mm[incremental modification, enter E to clear]
29. A offset of G59 mm[incremental modification, enter E to clear]
30. B offset of G59 mm[incremental modification, enter E to clear]

3.7.7 Password

In this system in order to prevent from the parameter modification in accident, make sure the lathe working, the system adopts the parameter setting of classify the authority. Divided into "CNC factory" and "Lathe factory", "User factory" three level authority. The "CNC factory" set for the function of the system,belong to internal parameter;"Lathe factory" set equipment configuration of lathe and mechanical index and some parameter about safety;"User factory" set processing technology,

performance and the processing program.

The initial situation of three-level classification in this system: "CNC factory" is enabled,"Lathe factory"and "User factory" both are not enabled. If you want to enable the authority function, you must use the initial password to enable access function, then set the corresponding new access password to enable.Pay attention to the initial code is only be used once, the code will invalid after setting a new code, please remember the new code.

Pay attention: the code must be 6 digits, the code can be number and letter.

Parameter:

1. Whether to turn on the password protection of CNC manufacturer

2. Whether to turn on the password protection of lathe manufacturer

3. Whether to turn on the password protection of user manufacturer

4. Modify the password of CNC manufacturer

5. Modify the password of lathe manufacturer

6. Modify the password of user manufacturer

7. The using time of system

8. Software version

3.8 Set parameter of tool redeem

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

3.8.1 Radius compensation

Press "N" to enter radius compensation interface in redeem interface. The parameter is used to set adopt corner radius of the tool.

Setting method: Press "↑ ↓" to make cursor move to the corresponding tool and press "Enter" to popup a dialog box, import corresponding tool radius(Absolute value), press "Enter" at last.
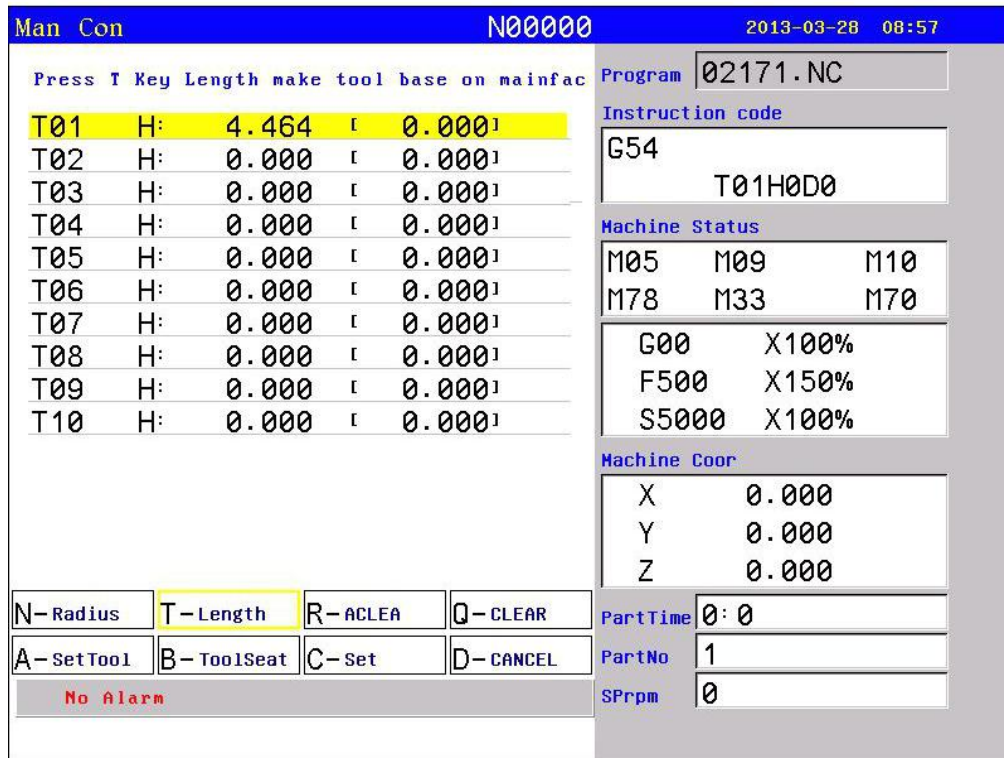
3.8.2 Length of redeem

Press "T" 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 set tool automatically:

Make lathe move to a position so that measure corresponding tool coordinate, press "↑ ↓" to make cursor move to corresponding tool number and press "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.It also can be set in manual status, press "H" in manual status to set tool automatically.

Method of initializing the length compensation value and radius of tool:

Press "R(Clear all)" or "Q(Clear the current)" to initialize all the length compensation or current length compensation.

Pay attention: the length compensation can be positive or negative number, but the radius compensation only can be positive number.

3.8.3 Posit tool

Press "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 tool.**

3.8.4 Set quantity

Press "C" to popup a dialog box in the redeem interface to set and manage the total tools.The quantity of the tool in this system can be set 99.

3.9 Screw compensation

Press "Parameter" twice in parameter interface to enter screw compensation interface to set the screw compensation.

Screw compensation is used for automatical compensating the error of screw pitch, compensate the influence from the error of screw pitch to the prevision of operating lathe. The system adopts storage mode of screw compensation: Making the lathe's datum point as the starting point when debugging, measured the error curve of screw, studied out the correctional curve according to the error curve, import the value of correctional curve into the correctional parameter and system is going to compensate according to the parameter in automatical running.
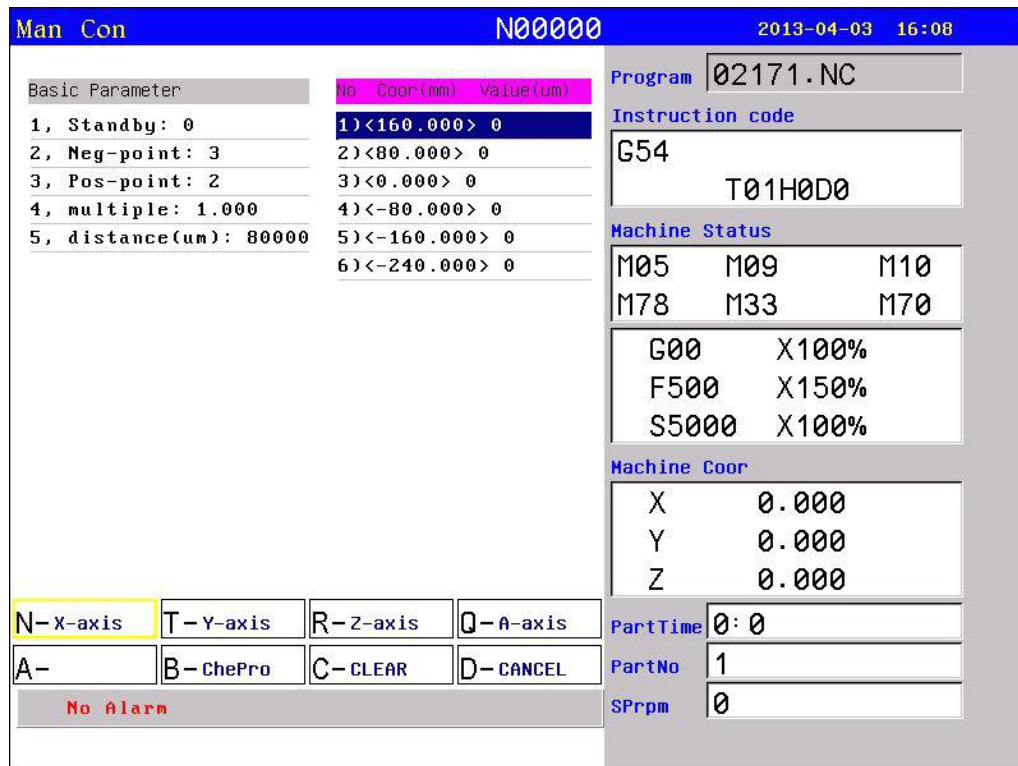
Screw compensation interface

  Screw compensation by the axis as the unit to set storage, set X Y Z axis separately, by pressing "N" "T" "R" to switch; Every axis of screw compensation interface has tow areas(basic parameter and set the compensation), by pressing "→ ←"to move the cursor to realize.

Basic parameter:

Press "↑ ↓" to select current basic parameter to set in basic parameter, press "Enter" to popup a dialog box to import the error compensation of every axis and import the basic information of screw compensation.

Set compensation value:

In the area of setting compensation, it will shows the value of compensation and every axis' error compensation point of screw pitch. Press "↑ ↓ PgDn PgUp" to select current compensation point and press "Enter" to popup a dialog box to import the value of compensation, import the value of current compensation point.

```
Man Con              鑫科瑞数控    N00000           2014-07-19  10:30

Basic Parameter      No  Coor(mm)  Value(um)    Program  G110
1, Standby: 0        1)<160.000> 0              Instruction code
2, Neg-point: 3      2)<80.000> 0               G53
3, Pos-point: 2      3)<0.000> 0                      T02H0D0
4, multiple: 1.000   4)<-80.000> 0
5, distance(um): 80000  5)<-160.000> 0          Machine Status
                     6)<-240.000> 0             M05    M09       M10
                                                M78    M33       M70

                                                  G00      X100%
   5, distance(um):                               F250     X150%
        p                                         S0       X100%

                                                Machine Coor
                                                  X      2.921
                                                  Y      0.000
                                                  Z    -14.930

N-X-axis   T-Y-axis   R-Z-axis   Q-            PartTime 0:3
A-         B-ChePro   C-CLEAR    D-CANCEL      PartNo   596
     No Alarm                                  SPrpm    0
```

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).
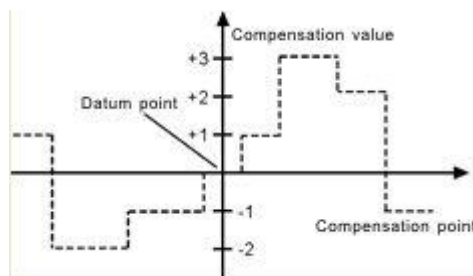
The system calculates every axis' error compensation points' positions of screw pitch according to basic parameter automatically, every axis's error compensation points' spacing is uniform, user can import compensation value of each point (This system requires importing absolute value, relating the of 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:
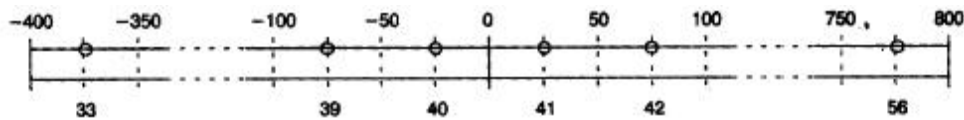
output compensation value in 0

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:
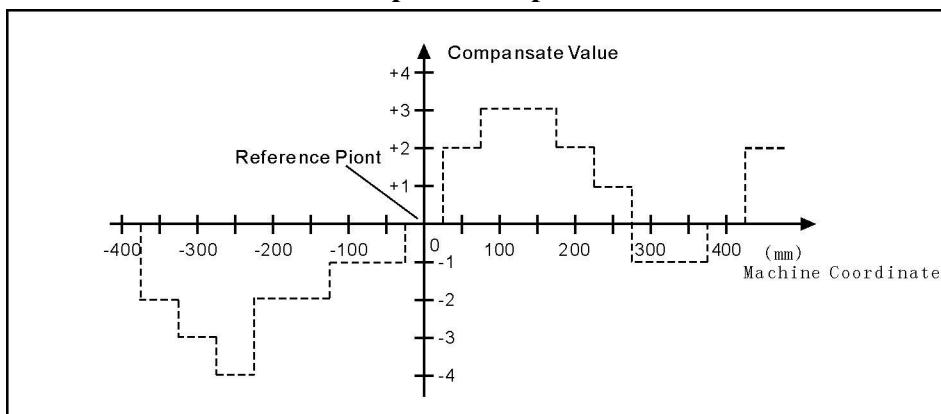
Example 2:rotor axis: when movement per revolution is 360°,   interval of points

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.
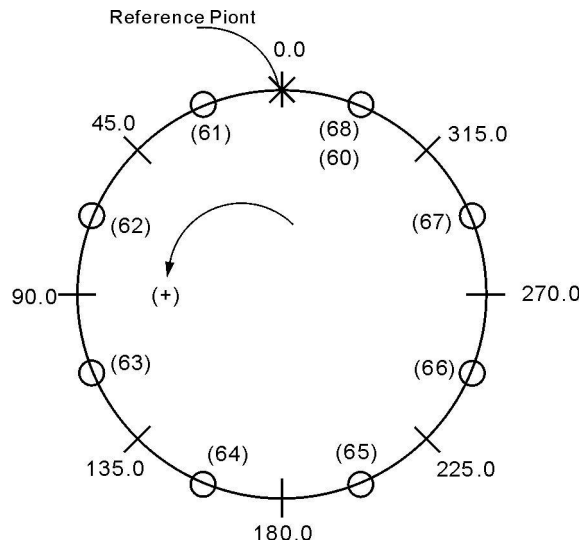
Machine coordinate and compensation point NO. correspondence is:

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.

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:
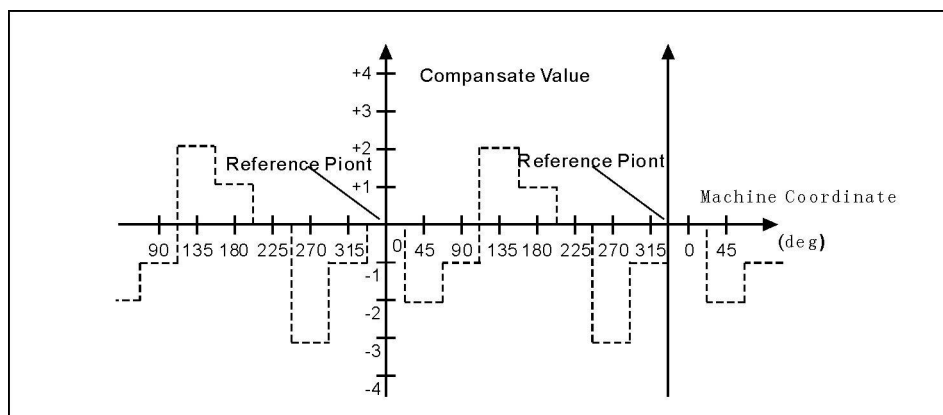
3.10 Diagnosis

Press "Diagnosis" to enter the diagnosis interface in parameter interface.

System diagnosis interface(Input signal)

Press "T" and "Pgup Pgdn" to check the status of input and output, press "Q" to check alarm information.

```
Man Con                    N00000        2013-03-28  08:57
───────── Input point ─────────     Program  02171.NC
 0    0    0    0    0    0    0    0    Instruction code
X00  X01  X02  X03  X04  X05  X06  X07
T01  T02  T03  T04  T05  T06  T07  T08   G54
 0    0    0    0    0    0    0    0           T01H0D0
X08  X09  X10  X11  X12  X13  X14  X15
M34/A0 -L  +L M36/Y0 X0  Z0  KRUN KHALT  Machine Status
 0    0    0    0    0    0    0    0    M05    M09      M10
X16  X17  X18  X19  X20  X21  X22  X23
XZ0  ZZ0 KLEFT KRIGHT STOP TOK ALM ALM1  M78    M33      M70
 0    0    0    0    0    0    0    0       G00    X100%
X24  X25  X26  X27  X28  X29  X30  X31      F500   X150%
ALM2 M28  M24  M22  M18  M12  M14  M16      S5000  X100%
 1    1    1    1    1    1    1    1
X32  X33  X34  X35  X36  X37  X38  X39   Machine Coor
HX/DS8 HY/DS4 HZ/DS2 HA/DS0 HX1/K8 HX10/K HX100/ HOFF/K  X      0.000
                                         Y      0.000
                                         Z      0.000
N-          T-I/O     R-        Q-ALARM  PartTime 0:0
A-          B-Reset   C-        D-CANCEL PartNo  1
    No Alarm                             SPrpm   0
```

```
Man Con                    N00000        2013-03-28  08:57
───────── Output Point ─────────    Program  02171.NC
 0    0    0    0    0    0    0    0    Instruction code
Y00  Y01  Y02  Y03  Y04  Y05  Y06  Y07
M61  M63  M65  M67  M69  M71  M73  M59   G54
 0    0    0    0    0    0    0    0           T01H0D0
Y08  Y09  Y10  Y11  Y12  Y13  Y14  Y15
M32  M79  M10  M08  M05  M04  M03  M75   Machine Status
 0    0    0    0    0    0    0    0    M05    M09      M10
Y16  Y17  Y18  Y19  Y20  Y21  Y22  Y23
LRUN INTH  +T   -T  S04  S03  S02  S01   M78    M33      M70
                                           G00    X100%
                                           F500   X150%
                                           S5000  X100%

                                         Machine Coor
                                         X      0.000
                                         Y      0.000
                                         Z      0.000
N-          T-I/O     R-        Q-ALARM  PartTime 0:0
A-          B-Reset   C-        D-CANCEL PartNo  1
    No Alarm                             SPrpm   0
```

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.

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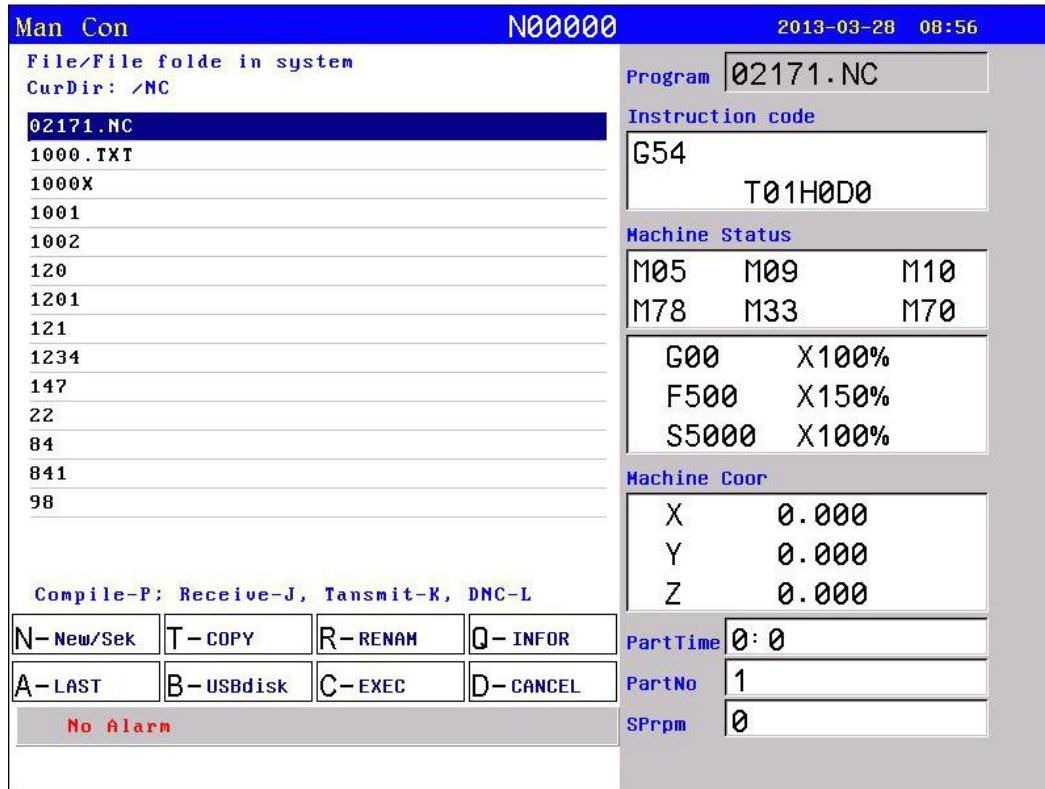
Center part of screen for program display ， current program is showed by reverse display, move PgUp、PgDn to choose program, and then press"Enter"to edit current program. Functional keys"N、T、R、Q、A、B、C、D" include： "new file/search" 、 "copy" 、"rename" 、"information"、"last grade""USB disc"、"execute program"、"cancel".

3.11.1 Editing

Select "New file/search" to popup a dialog box to import the name of program, if the name is existent, the quondam program is called up; If the name is inexistent, the system will build a new file.

The name of program can be number, letter or mix, the length is 100 bits.

The system doesn't allow the namesake, build a new program or select a program and press "Enter" to enter the editing interface.

The file name and the Chinese input of program content:

The directory interface of the "new / search", "copy", "rename", "to copy into the system", "to copy into the USB" can enter the Chinese characters; change the input method according to the prompt of system. Also can input Chinese characters in program editing interface according to the prompt of system.
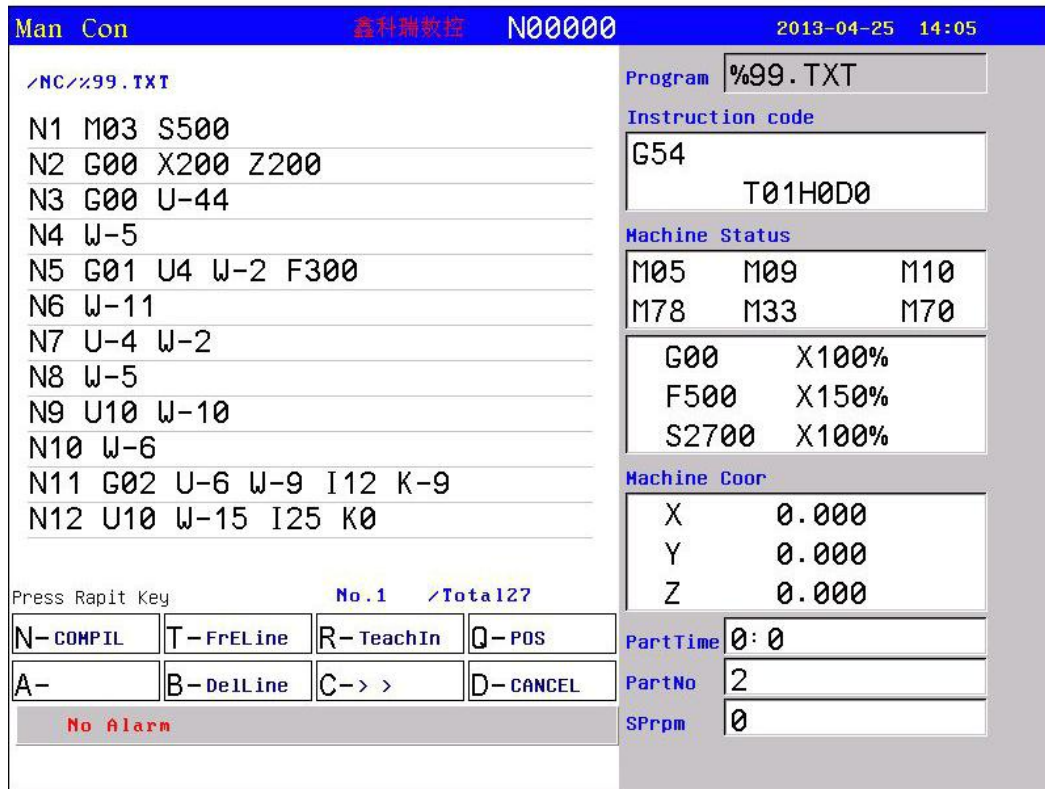
The screen prompt the editing program name at the top left corner in the editing status; The left is the content, the right is the information for lathe status, the operation in the editing status as follows:

   (1) The current cursor locate:

Press "↑ ↓ ← →" to move the cursor to any position of program content Press "Pgup" to the last page.

Press "Pgdn" to the next page.

(2) Character modification: Delete the character at the position of the cursor, then enter the new character.

   (3) The character insertion: Enter a new direct character at the cursor position. When the input is the letter,the letter in front of automatically generating space. If you want to enter a space, first enter a letter, and then delete this letter.

   (4) The character deletion: Press "Del" directly at the cursor position

   (5) Inset the line: Press "Enter" directly, inset a line in front of the current line if the cursor is at the first line, otherwise insert a line after the current line.

   (6) "Fast" superposition key operation:

The first function:

A, " "+"N": compile the program.

B, " "+"J": to the fist line or last line of the program.

C, " "+"V": teaching function, enter the handwheel status;Press "Q" to read the tool coordinate in the current workpiece coordinate system according to the corresponding axis X/Y/Z/A (all the axis).

86

D, " � "+"Q": located to the specified line.

E, " ⤳ "+"A": no operation.

F: " ⤳ "+"B": delete the current line.

G, " ⤳ "+"C": the fist or second page selection.

H, " ⤳ "+"D": Chinese characters and character input conversion. The second function:

A, " ⤳ "+"N": delete the program block.

B, " ⤳ "+"J": copy the specified program block.

C, " ⤳ "+"V": sort the program.

D, " ⤳ "+"Q": to find the specified character.

E, " ⤳ "+"A": replacing the specified character.

F, " ⤳ "+"B": all the content to are replaced by the specified character.

G," ⤳ "+ "C": the first or second page selection.

H, " ⤳ "+"D": Chinese characters and character input conversion.

3.11.2 Copy

Press "↑ ↓" in program main interface to select program which need to copy and press "T" 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 "R" 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 program to run

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 "J" 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 "K" 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.

Press "J" to copy all the program in USB into system.

    Copy the files or folder of system into U-disk

    Press " ↑   ↓ " to move cursor to select file or folder and press "B", press "R" to popup a dialog box to import name in U-disk interface and press "Enter" to confirm. If there is the same name of program in the system, it will popup a dialog box to ask if cover the file or folder or not.

    Press "K" to copy all the program in system into USB.

    Pay attention: Before unplugging the U-disk must return to the display system of program files directory interface. (Exit U-disk interface)

    Otherwise the date which is copied just now will be loss.

    The name of folder can't have space symbol when using U-disk.

B. Using U-disk to manage parameter and system software

    The system could use U-disk to deliver files or system software to upgrade and update, back-up files and parameter, the method of operation is as follows:

    Using U-disk to copy parameter and system software into system(Upgrade, update).

    First U-disk inserts the USB port and press "Program" to enter program main interface, press "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 "T" 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.

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**
- ☐ **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: straight line X Y Z A four axis, arc X Y Z any two axis**
- ☐ **Pulse equivalency: 0.001mm**
- ☐ **Maximum speed: 30000mm/min**
- ☐ **Processing speed: 0.01-15000min/min**
- ☐ **Minimal input unit: 0.001mm**
- ☐ **Programme size range: ±99999.999mm**
- ☐ **Programme coordinate system definition: IOS-841 international standard**
- ☐ **Programme code: IOS-840 international standard**
- ☐ **Mean Time Between Failure (MTBF): Greater than 6000 hours**

4.3 Environment of operation

- ☐ **Power supply: AC 220V (+10%,-15%), frequency 50Hz ±1%**

- ☐ **Power source≤100W**

- ☐ **Running temperature: 5~45℃, relative humidity: 40~80%**

- ☐ **Storage and transportation temperature: 0~55℃, relative humidity less than**

90%(40℃)

4.4 System installation and connection

At first, users should check whether the hardware is complete, unwounded and compatible, such as: CNC system, driving power, servo motor, photoelectric encoder, electric tool carrier.

The installation of CNC system must be fastened tightly, with some spaces around to ensure the ventilation of air. Panel should be put in a place where it is not only convenient to operate and but also able to avoid hurt of heating by scrap iron.

Intense current, week current must be put separately; CNC system and driver should be possibly away from the machine intense current. In order to reduce interference, all signal cables should be kept away from AC contactor. Photoelectric encoder, limit, basic point signal are advisably not to be connected directly to CNC system through intense current box. All power cords must be grounding.

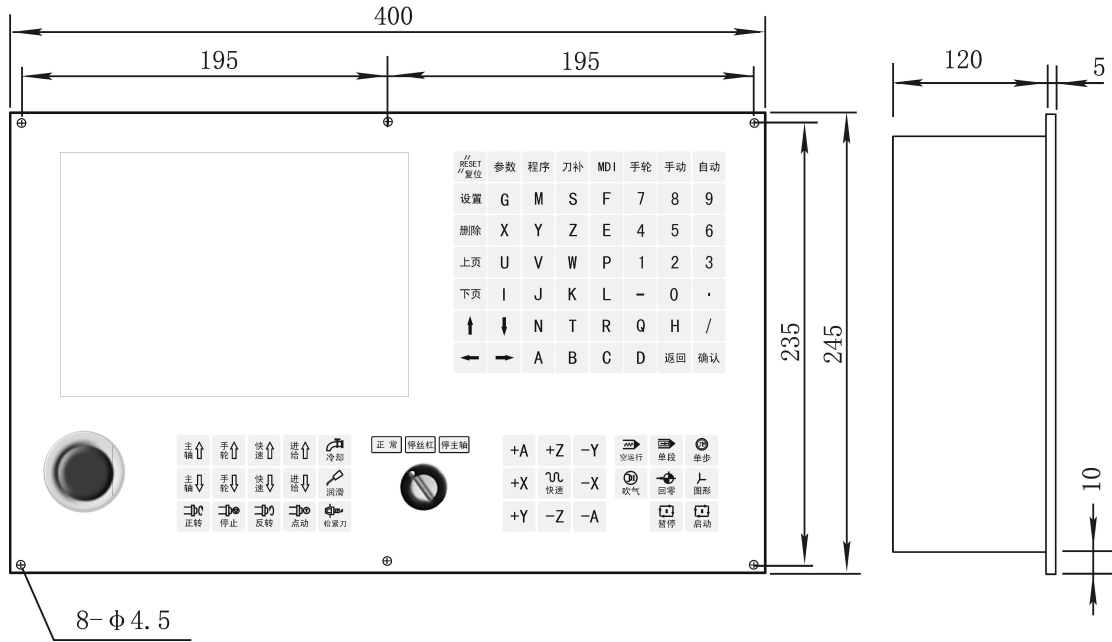Fix all plugs with screw. Forbid to insert and extract all cables when power is on.

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

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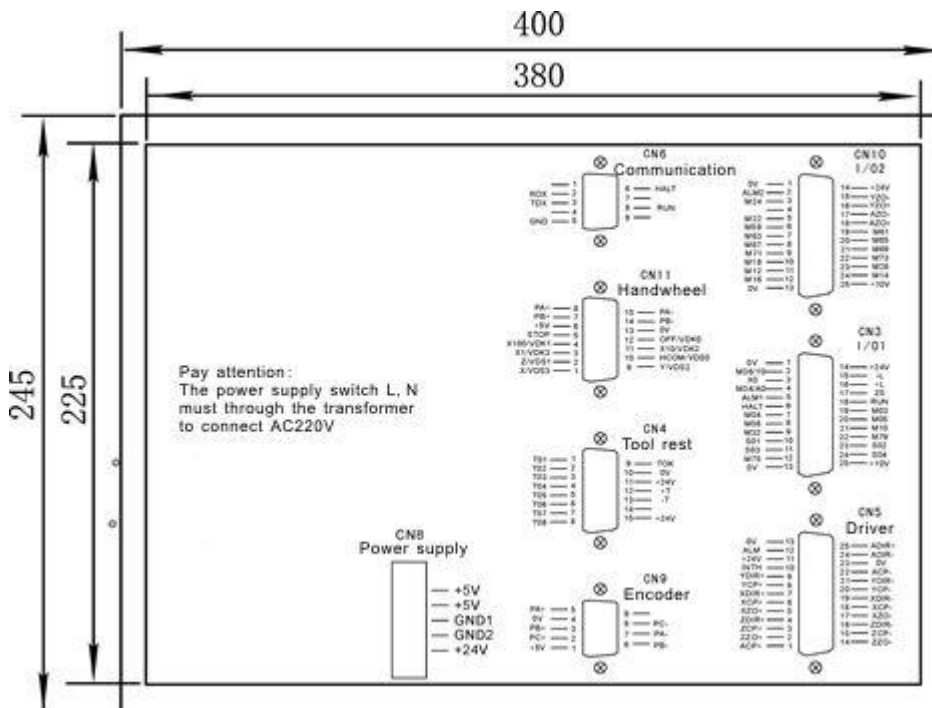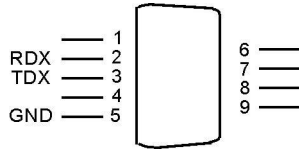
Pay attention:

1. Must install in an electricity cabinet which is good for protect from lightning.
2. Must install firmly to in order to avoid vibrating and loosing.
3. Don't install on the inflammable things or nearby to keep away from fire.

4.5 System installation dimension （400x245x120）

8-φ4.5

4.6 System rear view

Pay attention:
The power supply switch L, N must through the transformer to connect AC220V

Pay attention:Switching power supply L N must through isolation transformer and insert to AC 220V, current 0.5A.

4.7 Interface connection graph

4.7.1 Communication socket CN6 connect to the hole socket DB9

```
      ┌───┐
  ──── 1 │   │ 6 ────
RDX ── 2 │   │ 7 ────
TDX ── 3 │   │ 8 ────
  ──── 4 │   │ 9 ────
GND ── 5 └───┘
```

| CN6 communication signal with the hole socket DB9				
Signal	Pin number	I/O	Function	Effective voltage
0V	5	OUT	The ground of signal	0V
RXD	2	IN	The received data signal	
TXD	3	OUT	The transmission of data signal	
RUN	8	IN	Spindle of running	0V
HALT	6	IN	Spindle of stop	0V

Pay attention:

1. Connect to external PC with data communication, must be equipped with our special communication software.

2. Communication line must adopt the shielded twisted pair cable, the length shall not exceed 10m.

The signal of CN6 connect to PC:
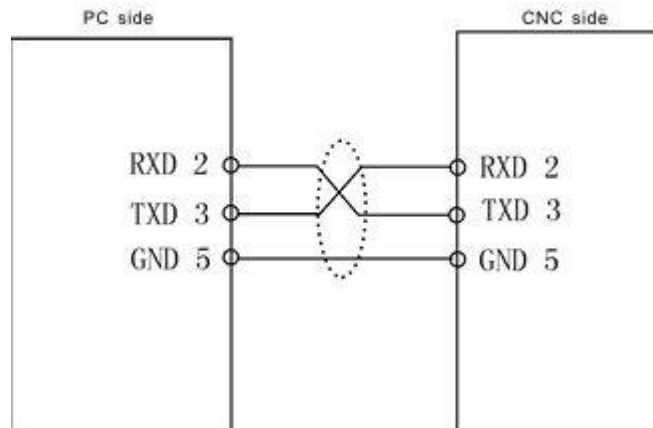
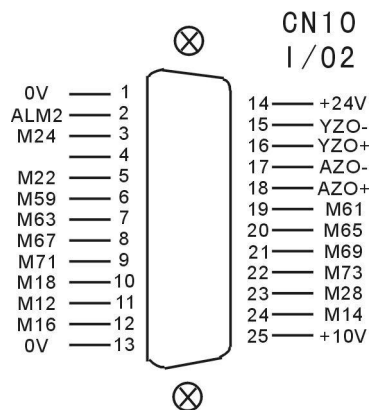Pay attention: When PC programming, the files should be text files.

4.7.2 The lathe machine CN3 connect to the hole socket DB25
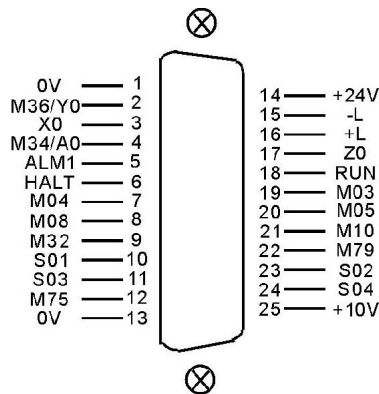
CN3 DB25(hole) I/O1 machine signal

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	Ground of converting	0V

4.7.3 Lathe machine CN10 connect to hole socket DB25

```
               ⊗   CN10
                   I/O2
0V   ── 1        14 ── +24V
ALM2 ── 2        15 ── YZO-
M24  ── 3        16 ── YZO+
     ── 4        17 ── AZO-
M22  ── 5        18 ── AZO+
M59  ── 6        19 ── M61
M63  ── 7        20 ── M65
M67  ── 8        21 ── M69
M71  ── 9        22 ── M73
M18  ──10        23 ── M28
M12  ──11        24 ── M14
M16  ──12        25 ── +10V
0V   ──13
               ⊗
```

CN10 DB25 I/O2 lathe signal				
Spindle	Pin	I/O	Function	Effective voltage
0V	1	OUT	Ground of the power	0V

			supply	
+24V	14	OUT	24V power supply	+24V
ALM2	2	IN	Lathe alarm2	0V
M24	3	IN	User-defined input7 or return to the zero point signal of B axis	0V
M22	5	IN	Stop M01 input	0V
M59	6	OUT	Huff output	0V
M61	19	OUT	User-defined output1	0V
M63	7	OUT	User-defined output2	0V
M65	20	OUT	User-defined output3	0V
M67	8	OUT	User-defined output4	0V
M69	21	OUT	User-defined output5	0V
M71	9	OUT	User-defined output6	0V
M73	22	OUT	User-defined output7	0V
M18	10	IN	User-defined input1	0V
M28	23	IN	User-defined input2	0V
M12	11	IN	User-defined input3	0V
M14	24	IN	User-defined input4	0V
M16	12	IN	User-defined input5	0V
YZO+	16	IN	Positive zero point of Y axis motor	5V
YZO-	15	IN	Negative zero point of Y axis motor	
AZO+	18	IN	Positive zero point of A axis motor	5V
AZO-	17	IN	Negative zero point of A axis motor	
+10V	25	OUT	The second spindle frequency conversion	0~10V
0V	13	OUT	Ground of frequency conversion	0V

**4.7.4 Driver signal CN5 socket connect needle socket DB25**
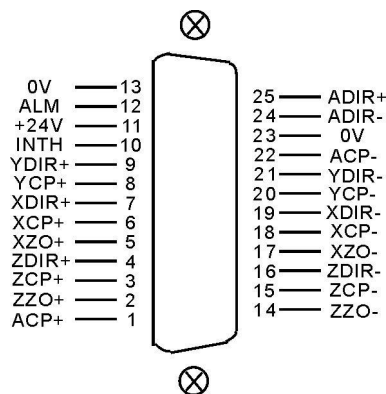
| CN5 DB25(pin) servo drive signal				
signal	pin	I/O	Function	Availability
XCP+	6	OUT	X pulse signal +	5V
XCP-	18	OUT	X pulse signal -	
XDIR+	7	OUT	X direction signal +	5V
XDIR-	19	OUT	X direction signal -	
YCP+	8	OUT	Y pulse signal +	5V
YCP-	20	OUT	Y pulse signal -	
YDIR+	9	OUT	Y direction signal +	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 signal line must adopt shielded twisted pair cable, the length is 20m at most.

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

Servo alarm signal:
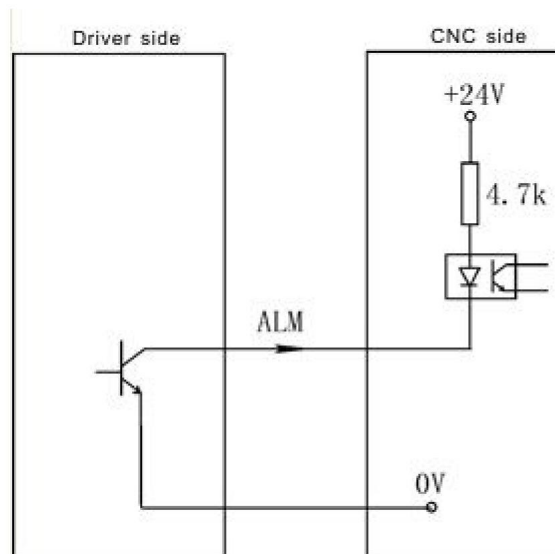
4.7.5 External switch, electrical handwheel socket CN11 connect to DB15 needle socket
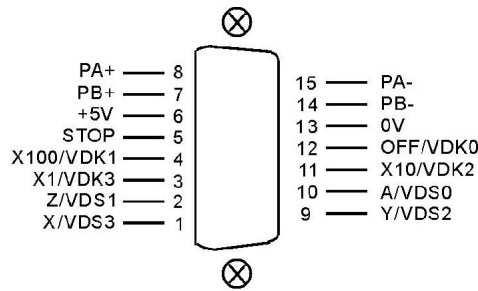
CN11 DB15 I/O signal				
Signal	**Pin**	**I/O**	**Function**	**Effective voltage**
0V	13	OUT	**Ground of power supply**	**0V**
+5V	6	OUT	**5V power supply**	**+5V**
PA+	8	IN	**Positive signal A**	**5V**
PA-	15	IN	**Negative signal A**	
PB+	7	IN	**Positive signal B**	**5V**
PB-	14	IN	**Negative signal B**	
STOP	5	IN	**External emergency signal**	**0V**
OFF/VDK0	12	IN	**Turn off gear signal,selection signal of B axis**	**0V**
X100/VDK1	4	IN	**One hundred gears signal**	**0V**
X10/VDK2	11	IN	**Ten gears signal**	**0V**
X1/VDK3	3	IN	**One gear signal**	**0V**
A/VDS0/HALT	10	IN	**Selection signal of A axis/Stop signal**	**0V**
Z/VDS1	2	IN	**Selection signal of Z axis**	**0V**
Y/VDS2/RUN	9	IN	**Selection signal of Y axis/Running signal**	**0V**
X/VDS3	1	IN	**Selection signal of X axis**	**0V**

Pay attention: The inner power supply are all +5V of all signal, do not access voltage higher than 5V.

4.7.5.1 Usage for 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"

PA+ PB- PA+ PA- corresponding input signal of handwheel pulse

A B. Handwheel contact diagrammatic as:
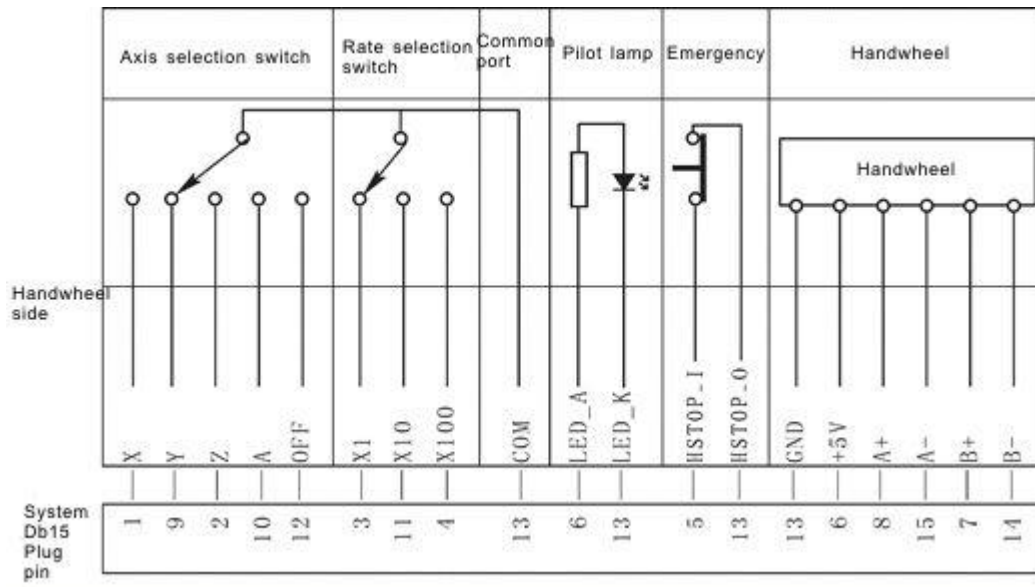
Pay attention:

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

2. Just connect PA+ PB+ if adopt voltage output.

3. Manual pulse generator needn't "Enter" button, if there is a "Enter" button, use the line to short the ends of switch.

4.The inner power supply are all +5V of all signal, do not access voltage higher than 5V.

The input signal of handwheel:
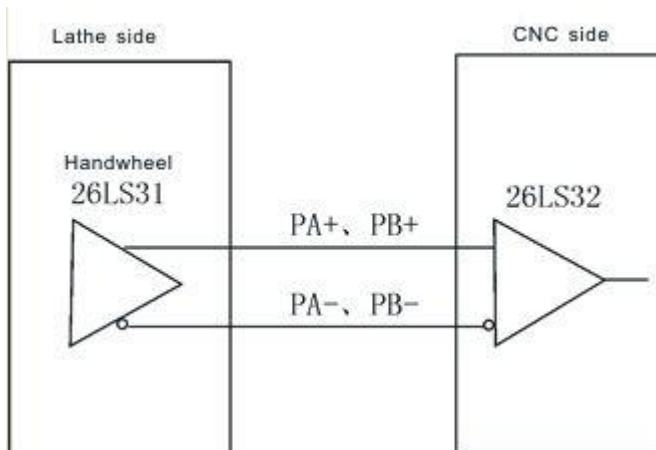
4.7.5.2 Usage for terminal band switch behind sub panel

Could use the band switch when the No.1 No.2 parameter in "axis parameter" are

set to 1,the No.1 parameter in "other parameter" should be set to 0, A、Z、Y、X、OFF、X100、X10 and X1 are corresponding to select the feeding axis and the gear of the spindle.

   VDS0 (A), VDS1 (Z), VDS2 (Y), VDS3 (X) are input signal which is for trimming gear of spindle speed, four position control,total 16 gears. VDK0 (OFF), VDK1 (X100), VDK2 (X10), VDK3 (X1) are input signal which is for trimming gear of feeding speed,four position control,total 16 gears.

4.7.5.3 Using for external emergency button

   STOP signal is the input signal of external emergency button, No.27 parameter in other parameter controls the emergency button is always open or close.

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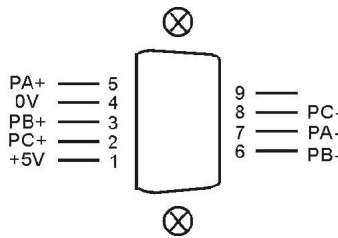
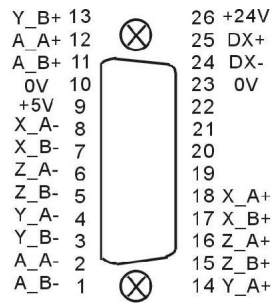| CN9 DB9(pin) spindle encoder				
signal	pin	I/O	function	availability
0V	4	OUT	0V	0V
+5V	1	OUT	+5V	+5V
PA+	5	IN	+A signal	5V
PA-	7	IN	-A signal	
PB+	3	IN	+B signal	5V
PB-	6	IN	-B signal	
PC+	2	IN	+Z signal	5V
PC-	8	IN	-Z signal	

 Pay attention:

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

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

   The input signal of encoder PA PB PC:
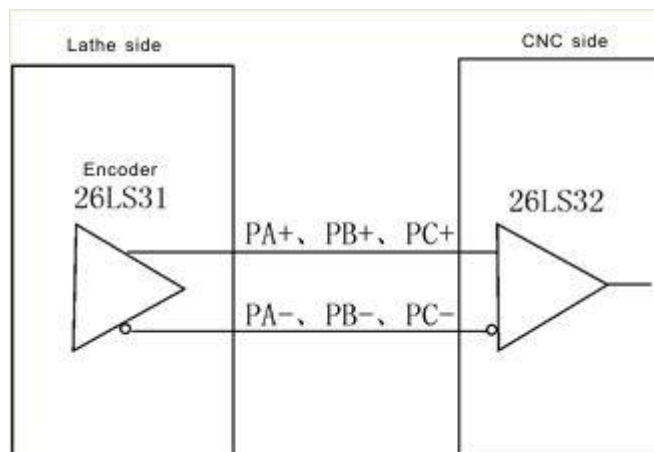
4.7.7 Feedback signal of coordinate axis position socket CN13 connect to needle socket DB26

```
Y_B+  13              26  +24V
A_A+  12      ⊗       25  DX+
A_B+  11              24  DX-
 0V   10              23  0V
+5V    9              22
X_A-   8              21
X_B-   7              20
Z_A-   6              19
Z_B-   5              18  X_A+
Y_A-   4              17  X_B+
Y_B-   3              16  Z_A+
A_A-   2              15  Z_B+
A_B-   1      ⊗       14  Y_A+
```

CN13 DB26 I/O signal				
Signal	Pin	I/O	Function	Effective voltage
0V	10,23	OUT	Ground of power supply	0V
+5V	9	OUT	5V power supply	+5V
+24V	26	OUT	24V power supply	+24V
DX+	25		RS485+	
DX-	24		RS485-	
XA+	18	IN	Positive signal A of X axis	5V
XA-	8	IN	Negative signal A of X axis	
XB+	17	IN	Positive signal B of X axis	5V
XB-	7	IN	Negative signal B of X axis	
YA+	14	IN	Positive signal A of Y axis	5V
YA-	4	IN	Negative signal A of Y axis	
YB+	13	IN	Positive signal B of Y axis	5V
YB-	3	IN	Negative signal B of Y axis	
ZA+	16	IN	Positive signal A of Z axis	5V
ZA-	6	IN	Negative signal A of Z axis	
ZB+	15	IN	Positive signal B of Z axis	5V
ZB-	5	IN	Negative signal B of Z axis	
AA+	12	IN	Positive signal A of	5V

|			A axis	
AA-	2	IN	Negative signal A of A axis	
AB+	11	IN	Positive signal B of A axis	5V
AB-	1	IN	Negative signal B of A axis	

4.7.8 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 No.318 parameter in other parameter(Corresponds to X/Y/Z/A/B axis), press "Enter" and press "E" to clear the lathe coordinate;

9. Manual move every axis to the limit position of lathe to set the soft limit value in axis parameter;

10. Set the No.41 parameter in other parameter and back up the current system parameter
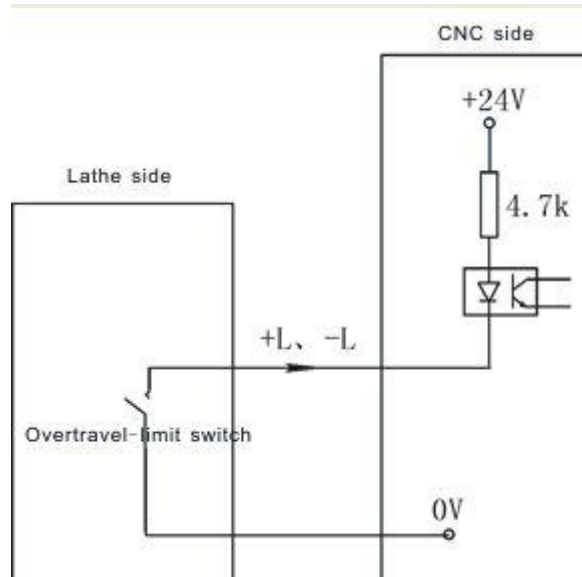
4.8 The machine installation of lathe.

   4.8.1 Limited position: take X Y Z A axis is limited position positively as example
      Model 1: NPN approach switch
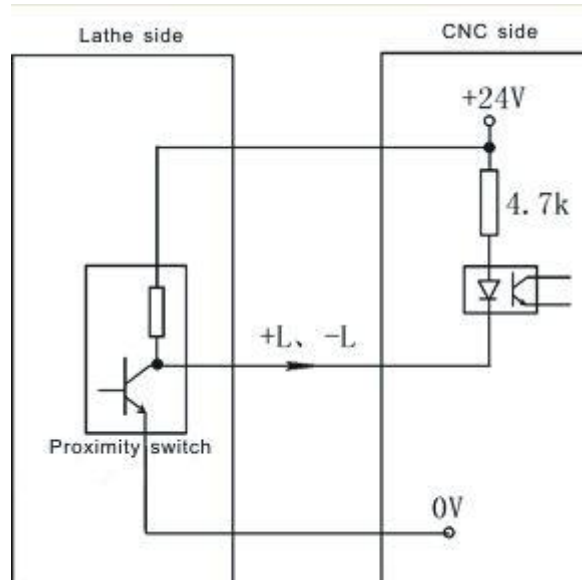
Mode 2: General switch

Axis parameter:
   No.27 parameter is for setting the type of hard limit switch for +L positive, 0 means always open, 1 means always close.
   No.28 parameter is for setting the type of hard limit switch for -L negative, 0 means always open, 1 means always close.
Pay attention:
   1. X Y Z A axis limited shares a signal to always open or close together, positive limited and negative limited corresponding stand for +L and –L signal.

2. Could select our electrical appliance plate of lathe.

3. The system could define X0 Y0 Z0 A0 to be limited input signal of X Y Z A axis. X0 signal as the limited signal and datum point of X axis, the same switch to control. Y0 signal as the limited signal and datum point of Y axis, the same switch to control. Z0 signal as the limited signal and datum point of Z axis, the same switch to control. A0 signal as the limited signal and datum point of A axis, the same switch to control.

The function must copy our exclusive PLC software.

4.8.2 Lathe's datum point(reference point or 0 point): take X axis as example (the same as Y Z A axis)
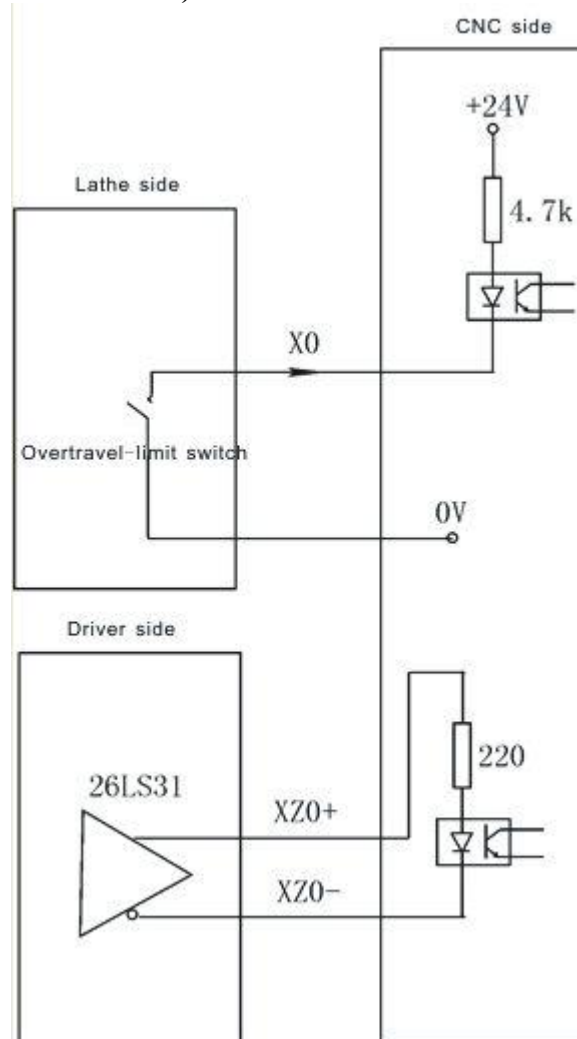
Backing to the datum point

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.33 parameter in axis parameter is set to be "00000000".

No.38 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.39 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.40 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, D3 controls the direction of backing to zero point in Y axis,D4 is for direction of Z axis, 1 means negative, 0 means positive;

No.41 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,D1 position controls Y axis,D2 position controls Z axis, 1 means always close, 0 means always open.

No.42~No.45 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.42) Y(No.43) Z(No.44) A(No.45) 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.34 No.36 No.38 No.40 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.34) Y(No.36) Z(No.38) A(No.40) axis backing to positive datum point. Unit: mm/min. Numerical

contactor, then plus resistive and capacitive spark

circuit. 2. Chip ULN2803A corresponds output ports:

1). U28 corresponds to M59 M61 M63 M65 M67 M69 M71 M73

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

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

3.When user-defined signal M71/M70 is used for input signal of spindle chuck and thumbstall, it can't be the other user-defined. No.20 No.21 parameter in other parameter to set.

4. When user-defined signal M65 M67 M69 is used for input signal of "Emergency", it can't be the other user-defined. M65 the program stops to output,M69 the program runs output,M67 the alarm output. No.28 No.29 parameter in other parameter to set.

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.9 parameter: Set the braking time of spindle, also the it's the output of the hold time, the time less the brake fast.Unit:10ms.

No.10 parameter: Set the braking signal is long signal 1 or short signal 0.

No.43 parameter: Whether turn on the spindle or not when spindle shifting [1 means on, 0 means off]

No.44 parameter: The turning speed of motor when spindle shifting (1/100rpm)

No.45 parameter: The turning direction when spindle shifting (0 means positive, 1means negative)

No.46 parameter: The stopping time when spindle shifting (10ms)

No.47 parameter: Turning time of low speed when spindle shifting (10ms)

No.48 parameter: Stopping delay time of spindle (10ms)

Speed parameter

No.9 parameter: To set the speed of spindle setting value at manual condition. Unit: rpm.

No.42 parameter: To set the highest speed of spindle, that's the turning speed of corresponding 10V instruction voltage.

No.43 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.44 parameter: To set the highest speed of spindle (Third gear), that's the turning speed of corresponding 10V instruction voltage. Unit: r/min

No.45 parameter: To set the highest speed of spindle (Fourth gear), that's the turning speed of corresponding 10V instruction voltage. Unit: r/min

No.46 parameter: To set the highest speed of second spindle, that's the turning speed of corresponding 10V instruction voltage. Unit: r/min

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).

Pay attention: The signal is controlled by M32 output.

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.

No.20 parameter: To set system controls chuck only need one signal (one-way valve) or two signals (two-way valve), this parameter is related with equipment of lathe.

M10 is just a output signal to control tautness of chuck when set to 0, system carry out chuck to tighten when M10 is effective, loosen chuck when M10 is invalid;

M10 and M71 corresponding control loose and tight of chuck when set to 1, system carry out chuck to tighten when relay M10 is effective, M71 is invalid, loosen chuck when M10 is invalid and M71 is effective. Output M10 when M10, output M71 when M11.

No.22 parameter: To set external button to control loose and tight of chuck(or foot switch), the signal is reciprocating, it means loosen once and then tighten once, reciprocating mode. No external button when set to 0; There is an external button to control chuck when set to 1, the signal is M16.

No.24 parameter: To set the retention time when the output signal M10 M71 of chuck is short signal, set to 0 means the signal is long signal. Unit: S.

Pay attention: M16 is a multiple function signal, only choose one function to use.

4.8.4 System alarm signal: ALM, ALM1, ALM2, Door alarm M12 and Emergency

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

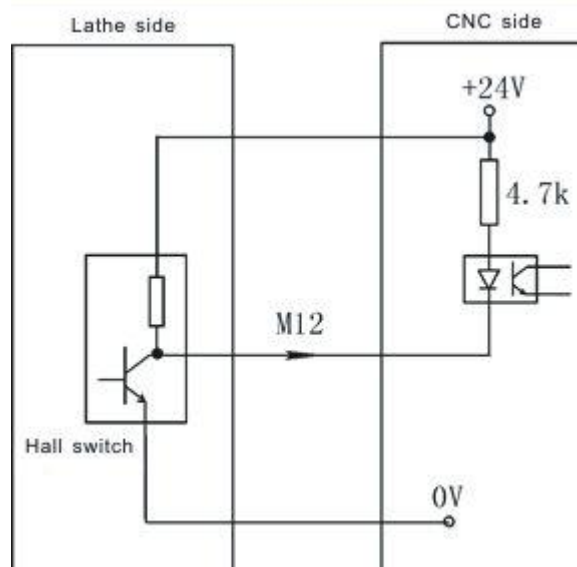to protect the lathe (Cut off power supply).

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,M24, external "Run", external suspend "HALT", external "STOP" as the same)

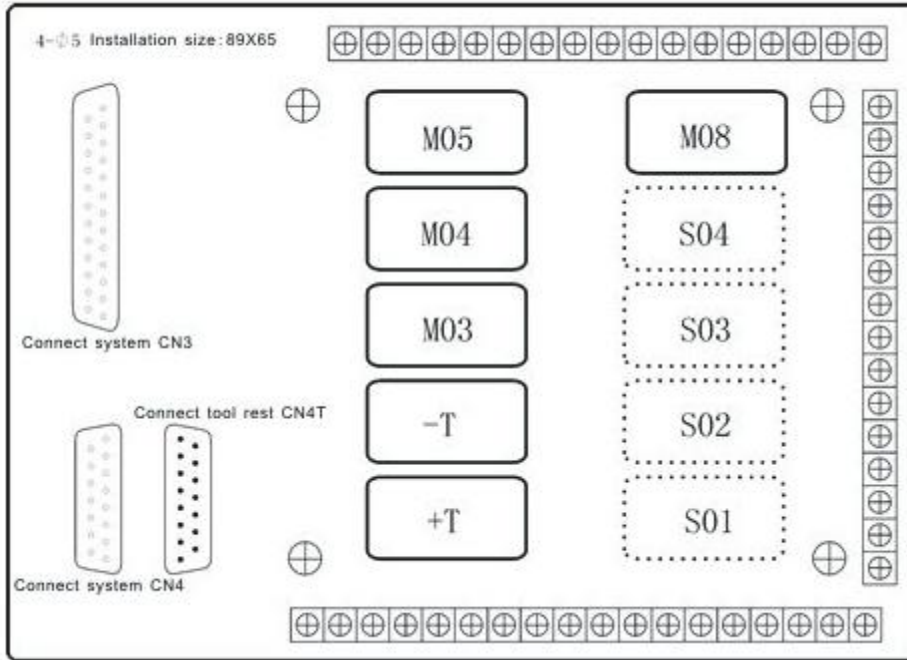M12 switch use overtravel-limit switch to wiring:
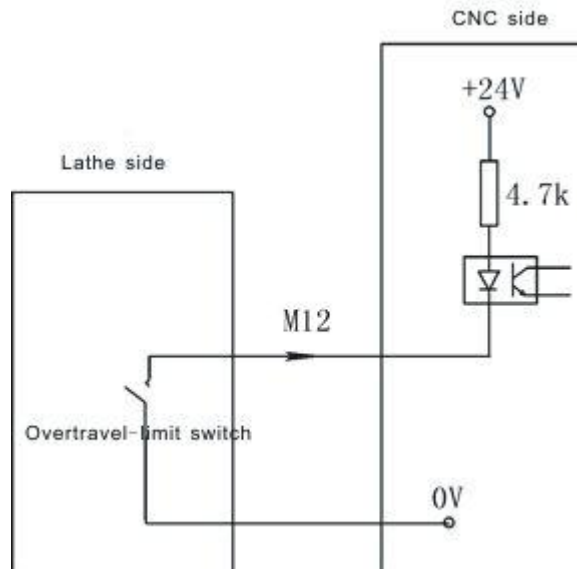
M12 switch also can use NPN as checking witch:

4.9 Electrical appliance plate of lathe

Our company produces the electrical appliance plate of lathe to choose as follows.

I/01 socket and CN3 are corresponding one-to-one with CN3 pins of system;

4.9.1 Spindle controlling

C1 is the common port of M03 and M04.

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

4.9.2 Spindle gear controlling

C4 is the common port of S1 and S2, S1B and S2B are always close.

C5 is the common port of S3 and S4, S3B and S4B are always close.