

User Manual

SZGH-CNC990MDb/MDc Milling Control System

Shenzhen Guan hong Automation CO.,LTD

Website: www.szghauto.com

Add:Room 503 Anxin Building, No 536 Shenhui Road, Liuyue community, Henggang Street ,Longgang District, Shenzhen City,Guangdong Province, ChinaProvince, China

Post code: 518115

CONTENTS

Chapter 1 Preface.....	1
Chapter 2 System technical features.....	2
2.1 System constructions.....	2
2.2 System technical parameter.....	2
2.3 System function.....	2
2.4 System operation condition.....	3
Chapter 3 Programming.....	4
3.1 Basic concepts.....	4
3.2 General description of program.....	5
3.3 Program instruction.....	5
3.4 Programme instruction and usage.....	11
3.4.1 Programme convention.....	11
3.4.2 Instruction of G function.....	11
(1) Choose instruction of coordinate system.(G53/G54/G55/G56/G57/G58/G59).....	11
(2)Local coordinate system instruction (G52)	12
(3)Programming method(G90/G91).....	12
(4)Plane Selection(G17/G18/G19).....	12
(5)Rapid Positioning(G00).....	12
(6) Linear Interpolation(G01).....	13
(7) Circular Interpolation(G02/G03).....	13
(8) Helical Interpolation (G02/G03).....	14
(9) Dwell(G04).....	15
(10) Mirror image instruction(G11/G12).....	15
(11) Scaling instruction(G36/G37).....	16
(12) Rotate Coordinate Instruction (G68/G69).....	17
(13)Return Zero point(G28/G281/G282/G283/G284/G301/G302/G303/G304).....	18
(14)Tool length compensation instruction(G43/G44/G49).....	18
(15)Offset of tool radius instruction(G45/G46/G47/G48).....	19
(16) Tool radius compensate instruction(G40/G41/G42).....	20
(17) Program circulation instruction (G22--G800).....	21
(18) Accurate localization/Continual way processing (G60/G64).....	22
(19) Circle instruction (G73、 G74、 G80~G89)	22
(20) High speed drill deep hole(G73).....	23
(21) Drilling cycle, point drilling cycle(G81).....	24
(22) Drill cycle(G82).....	25
(23) Intermission drill cycle (G83).....	26
(24) Boring cycle(G85).....	26
(26) Boring cycle(G89).....	28
(27) Left tap cycle(G74).....	28
(28) Right Tap Cycle(G84).....	29
(29) Cancel cycle instruction (G80).....	30

(30) Pole coordinate instruction(G15/G16).....	30
(31) Switch millimeter and inch(G20/G21).....	31
(32) Go back the starting point of program(G26/G261/G262/G263/G264/G265)	32
(33) Remember the current point(G25).....	32
(34) Return to the memorial point(G61/G611/G612/G613/G614).....	32
(35) Check skip(G31、 G311).....	32
(36)Automatical beveling (I) and smoothing(R).....	32
(37) The calling program(M97 M98 M99).....	33
(38) Feeding speed F function.....	35
(39) T/H/D function.....	36
(40) Spindle speed S, SS function.....	36
(41) Macroprogram instruction.....	36
(42)User-defined macro instruction(G120-G160,M880-M889).....	39
3.5 Radius compensation of tool C.....	41
3.6 Comprehensive examples for programming.....	45
3.7 Usage for automatic tool setting gauge.....	47
3.8 Usage for automatical dividing center.....	48
Chapter 4 Operation explanation.....	49
4.1 Summary.....	49
4.2 Operation panel.....	49
4.3 Keyboard description.....	49
4.4 Manual operation.....	51
4.5 Auto operation.....	55
4.6 Operate safety, prompt alarm.....	58
4.7 Parameter operation.....	59
4.7.1 User parameter (processing).....	60
4.7.2 Speed parameter.....	60
4.7.3 Axis parameter.....	65
4.7.4 Tool parameter.....	71
4.7.5 Other parameter.....	71
4.7.6 Coordinate system.....	76
4.7.7 Password.....	78
4.8 Set parameter of tool redeem.....	78
4.9 Screw compensation.....	80
4.10 Diagnosis.....	83
4.11 Operation of program.....	85
Chapter 5 System installation and connection.....	89
5.1 System installation and connection.....	90
5.2 System Installation Dimension (400x245x120).....	90
5.3 System Rear View.....	90
5.4 Interface connection graph.....	91
5.5 The Installation of Milling Machine.....	99
5.6 Electrical appliance plate of Milling.....	105

Chapter 1 Preface

This system is the newest universal CNC system for milling machine, drilling machine, boring machine, 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 with high speed (the highest moving speed is up to 30 m / min, the highest feeding interpolation speed can reach to 15 m / min) and high precision; Adopt 800x600 dot matrix TFT LCD adaptive brightness LCD displayer, 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 milling as the representative of three, four axis linkage, closed-loop control universal fully digital control system, powerful function and many instructions, programming codes accords with ISO international code standard. Directly control of AC servo system. Choosing . If with choose dual servo driver (SZGH-302), this is a more economical solution.

This manual introduce the programming and using method of milling system detailly.

Special Attention:

- 1. All the functions of A axis are effective when configure with 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 efficient and safer.**
- 3. The “Run” button on the panel of system can be used when debugging (P9 in Other parameter to set “Valid” or “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 “Run” button of panel for long times, otherwise the consequences has nothing to do with my company.**
- 4. Don't plug & insert connector when system is on power.**
- 5. P -- Parameter, P1 stands for the No.1 parameter .**

Chapter 2 System technical features

2.1 System constructions

32 bits high performance, industrial grade ARM(32bits)+DSP+FPGA.
64MB memory.
32Mb user store room.
640x480 8 inch real colour LCD displayer.
Touch screen main and sub panel.
High anti-jamming switch power.
USB movable U disc copy interface.
RS232 interface.
Spindle servo speed control/spindle frequency conversion speed control.
Manual pulse generator.

2.2 System technical parameter

Controllable axes: X, Y, Z, A four axes.
Linkage axes: Arc X/Y/Z : any 2 axes; liner : X/Y/Z/A : 4 axes.
Pulse equivalent: 0.001mm.
Max speed: X,Y,Z,A: 30000mm/min.
Processing speed: 0.01-15000mm/min.
Min input unit: 0.001mm.
Program size range: ± 99999.999 mm.
Program code: ISO-840 international standard codes.
Program coordinate system definition: ISO-841.
Chassis protection complies with regulation of IP43.
MTBF(Mean Time Between Failures): Over 6000hours.

2.3 System function

2.3.1 Auto-diagnosis function

All around diagnosis of CPU, storer, LCD, I/O interfaces, parameter status, coordinates, machining program etc. shall execute when the system starts or resets. In operation, it makes real time diagnosis of power, spindle, limit and all I/O interface.

2.3.2 Compensation function

Automatic backlash compensation.
Tool radius automatic compensation.
Tool radius automatic offset and sharp angle transition.
Leading screw pitch error automatic compensation.

2.3.3 Abundant instruction system

scaling up/down instruction.
mirror machining instruction.
mutiple tool offset instruction.
program cycle, jump, call and different program ending.

multiple positioning instruction: starting point, setting fixed point,etc.

linera, circular, spiral line interpolation instruction.

program management instructions: program cycle, call, transfer and different program ending method, etc.

6 workpeices coordinates system .

2.3.4 Chineses/English menu, full screen edition

Easy operation, conveyent viewing.

2.3.5 Abundant debugging functions

it can point out clearly what errors of operation are and guide to correct them.

2.3.6 Progam changing between CNC system and IBM/PC series compatible computer

it can conduct CAD/CAM/CAPP auxiliary programming by using PC series compatible computer's abundant software resources , then transfer the CNC program into the system to machining through(USB movable U disc copy port,RS232 port). Likewise it also can transfer the program from system to PC through communication port.

2.3.7 PLC function

The System is with 24*24 I/Os , please check the part of PLC function.

2.4 System operation condition

2.4.1 Power supplying

AC 220V(+10%/-15%), Frequency 50Hz±2%. Power: ≤ 200W.

Note: it must use isolation transformator to supply power, first input:380V

2.4.2 Climate condition

opeation condition: temperature 0~45°C,relative moisture 40-80%.

storage & transportation condition: temperature -40~55°C , relative moisture <93%(40°C).

atmosphere pressure: 86-106kpa.

2.4.3 operation enviorment:

No excessive flour dust, no acid, no alkali gas and explosive gas,no strong electromagnetic interference.

Chapter 3 Programming

CNC milling System 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 milling 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.

3.1 Basic concepts

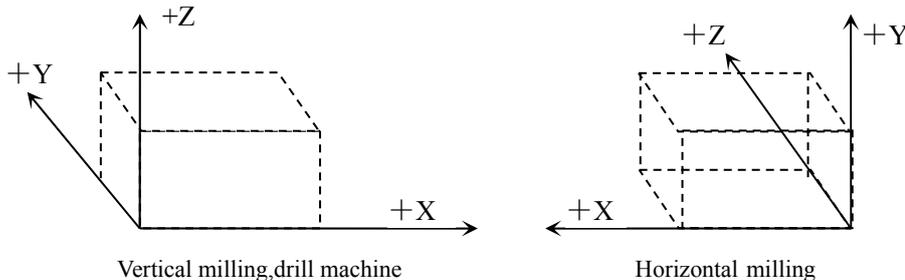
Program Segment:a complete command line consisted of instruction and data .

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.

When without set work coordinate, take machine coordinate as work coordinate.

Machine Coordinate & direction sketch map



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 workpiece coordinates:

- 1.Use G54 ~ G59: Use coordinate parameters to set 6 workpiece coordinate system.
- 2.G52: Set a value behind G52 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:coordinates data of programming mode based on established absolute coordinate system.Absolute coordinate value is corresponding to homing point of coordinate system. It is settlements by “G90 X_Y_Z_A_”.

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.

Non-Mode instruction: It is the instruction which has no function to store,and only works in the segment of program.

3.2 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.

Attention 1:“-”means this data can be use.

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

Attention 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.

3.3 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
Name of file		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 H D	No. /compensation of Tool	01-99
Spindle function	S	Set the speed of 1st spindle;	0-99999
	SS	Set the speed of 2nd spindle;	

Cutting speed	F	Speed per minute	0.01-15m/m
Coordinates character	X Y Z A	The coordinates value of X Y Z A axes.	+/-99999.999mm
Core coordinates	I J K	X Z axes and Z axes core coordinate increment value	+/-99999.999mm
Step length	R	Circular arc radius	0.001-99999.999 mm
Delay time	P,X	Designated delay time of G04	0.001-65s
Program entrance	P	Entrance of calling program name	0000-9999
Repeat times	L	Times of cycle or subprogram calling	1-9999
Line skip	/	No executive program when “/” is in front of program line	P9_D12 in Other parameter:shield the function

2.3.2 Program instruction

Table 1 G Instruction-code and functions

G code	function	
G00	Rapid positioning	
G01	Linear interpolation	
G31	No alarm when checking skip	
G311	Alarm when checking skip	
G02	Circular/Helical interpolation CW	
G03	Circular/Helical interpolation 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	Dwell	
G15	Polar coordinate command cancel	
G16	Polar coordinate command: Polar coordinate (Radius and Angle)	
G17	X Y plane selection	X: X axis or its parallel axis Y: Y axis or its parallel axis Z: Z axis or its parallel axis
G18	Z X plane selection	
G19	Y Z plane selection	
G20	Inch input	
G21	Millimeter input	
G28/G281/G282/G283/G284	Return to reference position of machine	
M882 M883	Set drop instruction of Z axis tool setting automatically	
M884 M885	Automatic dividing center of X Y axis	

M880	Adjust tool automatically	
G30/G301/G302/G303/G304	Return to reference point of coordinate system	
G26	Return to starting point of program	
G261	X-axis Return to starting point of program	
G262	Y-axis Return to starting point of program	
G263	Z-axis Return to starting point of program	
G264	A-axis Return to starting point of program	
G25	Save current coordinate value of X Y Z A axis	
G61/G611/G612/G613/G614	Back to the memory point(G25 specify the point)	
G40	Cutter compensation cancel	
G41	Cutter compensation left	
G42	Cutter compensation right	
G43	Tool lengthen compensation + direction	
G44	Tool lengthen compensation - direction	
G45	Tool adding offset	
G46	Tool subtract offset	
G47	Tool adding two multiple offset	
G48	Tool subtract two multiple offset	
G49	Tool length compensation cancel	
G37	Cancel scale zoom	
G36	Enable scale zoom	
G12	Cancel programmer mirror	
G11	Enable programmer mirror	
G52	Set local coordinate	
G53	Machine coordinate system selection	
G54	Workpiece coordinate system 1 selection	Attention:These 6 workpiece coordinate save in CNC, user may choice any one.
G55	Workpiece coordinate system 2 selection	
G56	Workpiece coordinate system 3 selection	
G57	Workpiece coordinate system 4 selection	
G58	Workpiece coordinate system 5 selection	
G59	Workpiece coordinate system 6 selection	
G60	exactitude stop	
G64	Continue path work.	
G68	Coordinate rotation	
G69	Coordinate rotation cancel	
G73	Peck drilling cycle	
G74	Left-spiral cutting circle	
G80	Cancel cycle mode	
G81	Drill cycle	
G82	Drill cycle	

G83	Drill cycle
G84	Right Tap cycle
G85	Right-spiral tapping cycle
G86	Boring cycle
G89	Boring cycle
G90	Absolute comand
G91	Increment command
G98	Return to initial point in canned cycle
G99	Return to R point in canned cycle
G22	Program cycle order
G800	Cancel Program cycle order

Table 2 M code and function

Spindle	M03	Spindle start CW	M7053/M7054 P xxxx; It will delay time after spindle rotate CW/CCW, the time is determined by "P". Unit: Millisecond. Example: M7053 P2000; Means delay 2000ms to stop spindle.
	M04	Spindle start CCW	
	M05	Spindle stop	
Coolant	M08	Coolant on	
	M09	Coolant off	
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 M13	Check M12 input valid Check M12 input invalidate	To skip when conditions are tenable Example: M12 P120 Program skips to 120th line to carry out.
	M14 M15	Check M14 input valid Check M14 input invalidate	
	M16 M17	Check M16 input valid Check M16 input invalidate	
	M18 M19	Check M18 input valid Check M18 input invalidate	
	M28 M29	Check M28 input valid Check M28 input invalidate	
	M22 M23	Check M22 input valid Check M22 input invalidate	
	M24 M25	Check M24 input valid Check M24 input invalidate	

Subprogram	M97 M98 M99	Program skip Subprogram is on tap Back to subprogram is on tap	L=1-99 P is the line number of transferring program
Program controlling	M87 M00 M01 M02 M30 M20	Number of workpiece plus 1 Suspend program Suspend program,input M22 effective suspend Program is over M05,M09 program is over Program is over, automatical repeat run the program according to the parameter which set the running times, be used for the debugging	When other parameter P10=0 is set not to autotomatical 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 milling coordinate. M501: read absolute motor position of X axis and reset the current milling coordinate. M502: read absolute motor position of Y axis and reset the current milling coordinate. M503: read absolute motor position of Z axis and reset the current milling coordinate. M504: read absolute motor position of the fourth axis and reset the current milling coordinate.	

3.4 Programme instruction and usage

3.4.1 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

3.4.2 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

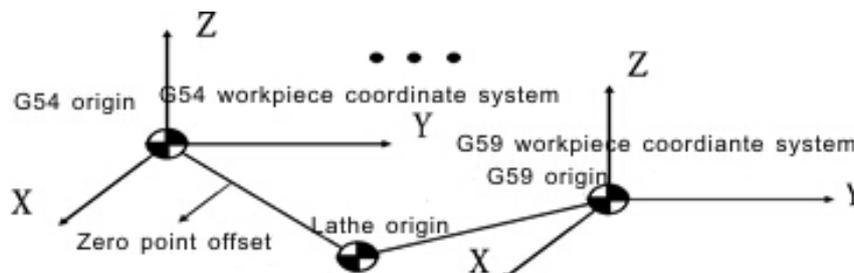
G54 work coordinate 1 G55 work coordinate 2

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.

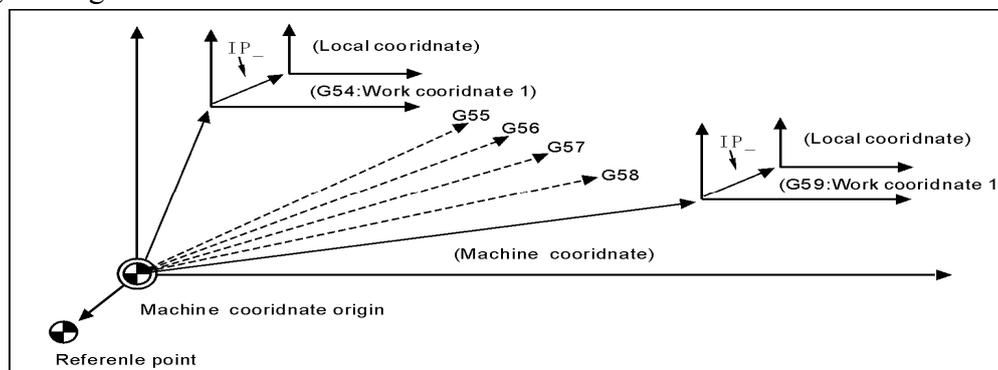
(2)Local coordinate system instruction (G52)

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 machine coordinate system(G53) or workpiece coordinate system(G54~G59).Home position 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)Programming method(G90/G91)

Two methods to move tools: Absolute programming and incremental programming. 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) ;Incremental

Attention:Rotating axis programming ,calculation is with nearest of in absolute coordinate system, calculation is with programming in incremental coordinate system.

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

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

Used to ensure circular interpolation plane.this instruction does not produce motion.

(5)Rapid Positioning(G00)

G00 is for rapid movements.Tool move to position with absolute/incremental command with the speed set by Speed parameter.

Format: G00 X_ Y_ Z_ A_ (Mode, original)

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

G00,each axis moves to position independtly,without interference.And also can moves with linkage mode,P9_D6 in Other parameter is set for this mode.

(6) Linear Interpolation(G01)

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

Format: G01 X_ Y_ Z_ A_ F_ (Mode)

X,Y,Z,A means motion axis.The data point out motion distance and direction by absolute or increamtal method.Interpolation speed is determined by F code.

The feeding speed of F in G01 can be adjusted by feeding rate selector on the panel, the range is 0%~150%. G01 instruction is also can be written G1.

(7) Circular Interpolation(G02/G03)

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

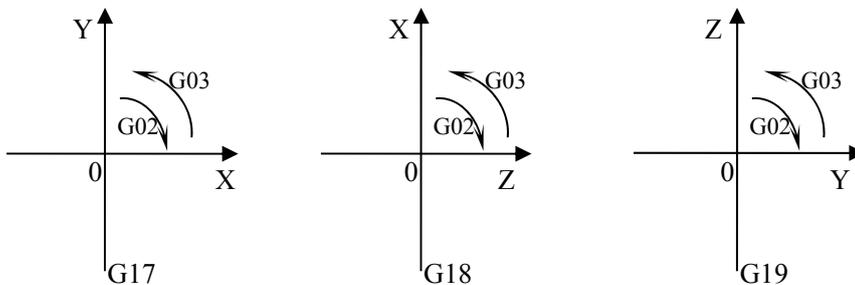
Format: G17 $\begin{Bmatrix} G02 \\ G03 \end{Bmatrix}$ $\begin{Bmatrix} X_Y_ \\ R_ \end{Bmatrix}$ $\begin{Bmatrix} I_J_ \\ R_ \end{Bmatrix}$ F_ ;**XY plane(Mode)**

G18 $\begin{Bmatrix} G02 \\ G03 \end{Bmatrix}$ $\begin{Bmatrix} X_Z_ \\ R_ \end{Bmatrix}$ $\begin{Bmatrix} I_K_ \\ R_ \end{Bmatrix}$ F_ ;**ZX plane(Mode)**

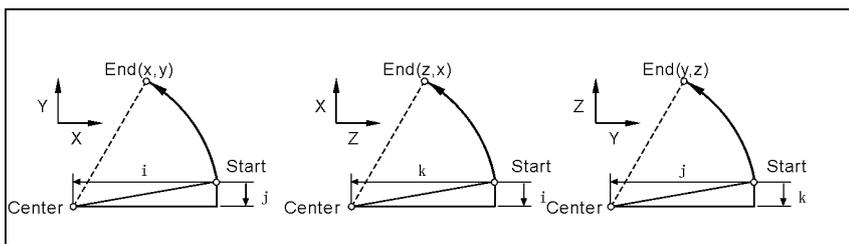
G19 $\begin{Bmatrix} G02 \\ G03 \end{Bmatrix}$ $\begin{Bmatrix} Y_Z_ \\ R_ \end{Bmatrix}$ $\begin{Bmatrix} J_K_ \\ R_ \end{Bmatrix}$ F_ ;**YZ plane(Mode)**

Circular interpolation must ensure the 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:



Use I J K ensure center of arc,the value behind of I J K is vector component from starting point to center of arc, '+/-' depends on direction of I J K.When starting point & end point is same and total 360° round whose center is specified with I J K, I J K can be omitted.



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) Helical Interpolation (G02/G03)

Helical interpolation means circular interpolation moving as another linear axis moving , F is set for speed circular interpolation.therefore, the feed speed of linear axis is as follow:

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

Format: G17 $\left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} X_Y_ \left\{ \begin{matrix} I_J_ \\ R_ \end{matrix} \right\} Z_F_$;XY plane(Mode)

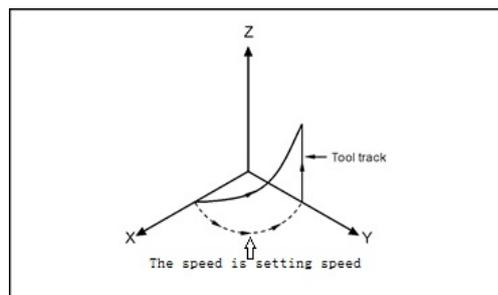
G18 $\left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} X_Z_ \left\{ \begin{matrix} I_J_ \\ R_ \end{matrix} \right\} Y_F_$;ZX plane(Mode)

G19 $\left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} Y_Z_ \left\{ \begin{matrix} I_J_ \\ R_ \end{matrix} \right\} X_F_$;YZ plane(Mode)

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

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

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



(9) Dwell(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:

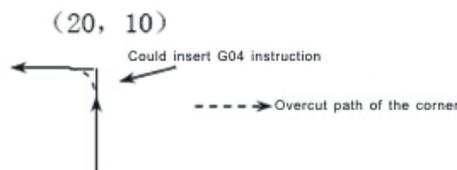
G04 X1; delay 1s.

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;

...

Attention: Set P21 in User parameter to clear the over cutting.

(10) Mirror image 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.**

Programmable mirror image finish processing of symmetric point in this axis by change the feeding direction of axis,which is specified by current interpolation plane G17(X_Y_)/G18(Z_X_)/G19(Y_Z_). The value behind of X_Y_Z_ is the coordinate value of current coordinate system.

Attention: In Mirror instruction,like these codes of return reference point ,G26,G28,... & these codes G53,G54~G59 , that can change coordinate system don't allow exist. If need to use any one of these,please cancel mirror mode.

For example:The mirror procedure gives an example

Sub program: KG11

N10 G00 G90 X60.0 Y60.0;

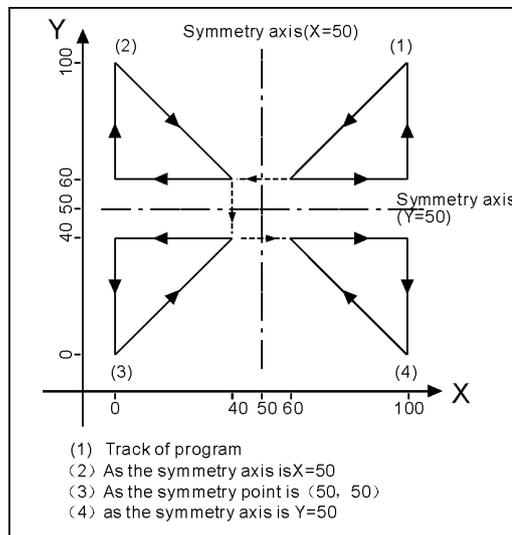
N20 G01 X100.0 F100;

N30 G01 Y100.0;

N40 G01 X60.0 Y60.0;
 N50 M99;

Main program

N10 G00 G90;
 N20 M98 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;

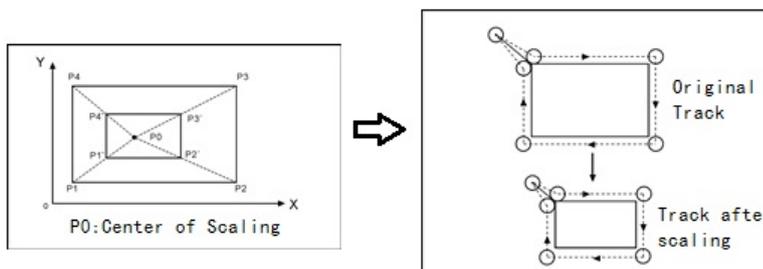


(11) Scaling instruction(G36/G37)

Used for not change the processing program, process workpiece according to scaling down/up. Programming shape is magnified and reduced (scaling), use X_, Y_ and Z_ to specify scaling center. If any one axis is not specified, the axis will not carry out scaling. The number behind the X_ Y_ Z_ is the coordinate value of current workpiece coordinate system.

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

- Attention: 1. the scaling coefficient is behind R.**
2.the scaling is invalid for value of compensation & offset.



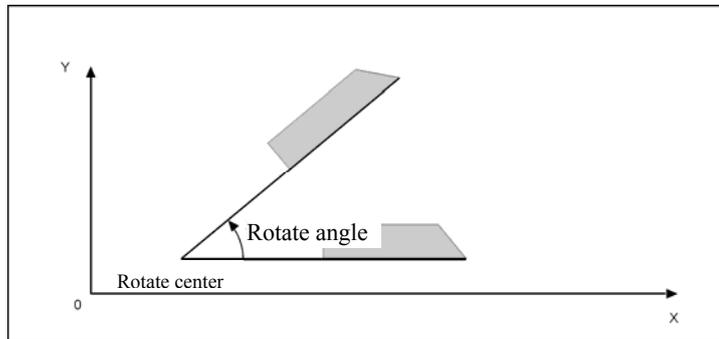
P1` P2` P3` P4` magnify to P1 P2 P3 P4,R=P0P4'/P0P4. When P1 P2 P3 P4 reduce to P1` P2` P3` P4`, R=P0P4/P0P4'. So: R<1 when magnifying, R>1 when reducing,R=1 can be default.

(12) Rotate Coordinate Instruction (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**

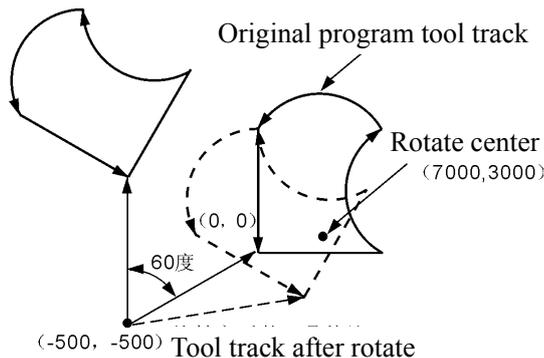
(G17)X_Y_/(G18)Z_X_/(G19)Y_Z_ behind G68 is for specifying center of rotate. R_ = rotating angle. When R is positive no. ,rotate CCW,Negative no. ,rotate CW.



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

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

Example:



N1 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 Zero point(G28/G281/G282/G283/G284/G301/G302/G303/G304)

Return zero point means tool go to reference point with appointed axis.

Format: G28 X(U)_ Y(V)_ Z(W)_ A_ ; XYZA firstly return to middle point and then return to zero point of machine .

- G281 ;only X return to reference point of machine**
- G282 ;only Y return to reference point of machine**
- G283 ;only Z return to reference point of machine**
- G284 ;only A return to reference point of machine**
- G301 ;X axis return to zero point of coordinate**
- G302 ;Y axis return to zero point of coordinate**
- G303 ;Z axis return to zero point of coordinate**
- G304 ;A axis return to zero point of coordinate**

Attention: Should clear radius&length compensation of tool before the code

(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.

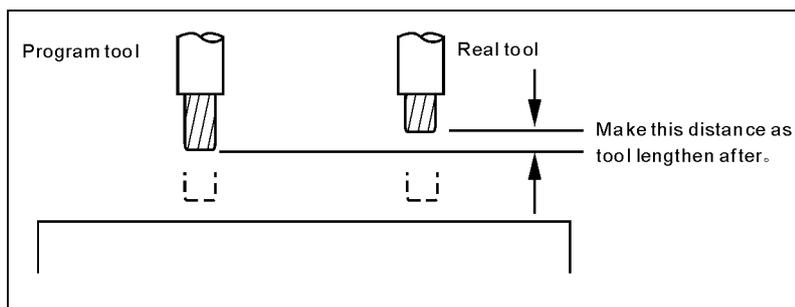
In the G17(XY) panel,length compensation puls to Z-axis,so tool in Z-axis best.

In the G18(ZX) panel,length compensation puls to Y-axis,so tool in Z-axis best.

In the G19(YZ) panel,length compensation puls to X-axis,so tool in Z-axis best.

- Format: G43 H_ ;Add tool length compensate.**
- G44 H_ ;subtract tool length compensate.**
- G49 or H0 ;cancel tool length compensate.**

Invoking no. is H1~H99,the starting point is at the un-compensating position.



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).G49 don't need parameters,return original position.

(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.

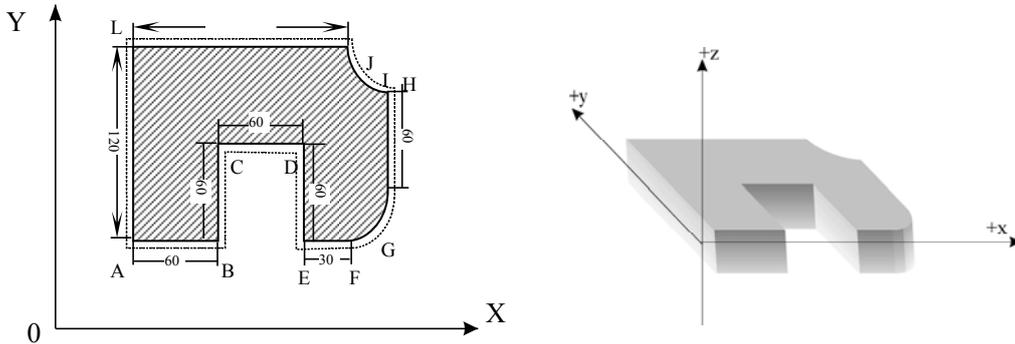
- Format: G45 T_ D_ ;Add one radius.**
- G46 T_ D_ ;subtract one radius.**
- G47 T_ D_ ;Add two radius.**
- G48 T_ D_ ;subtract two radius.**

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:

```

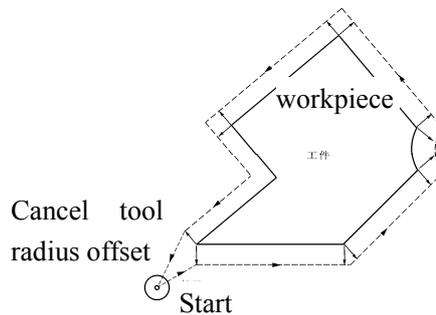
N0000 G01 Z-20 F400 G91 ; Z point starts
N0010 G46 T01 X55 Y55 D01 ; Point location to A point, X and Y axis both
decrease one tool radius(T01)
N0020 G47 G01 X60 F200 ; A-B X axis increase two tools radius
N0030 Y60 ; B-C length not change
N0040 G48 X60 ; C-D X axis decrease two tools radius
N0050 Y-60 ; 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)

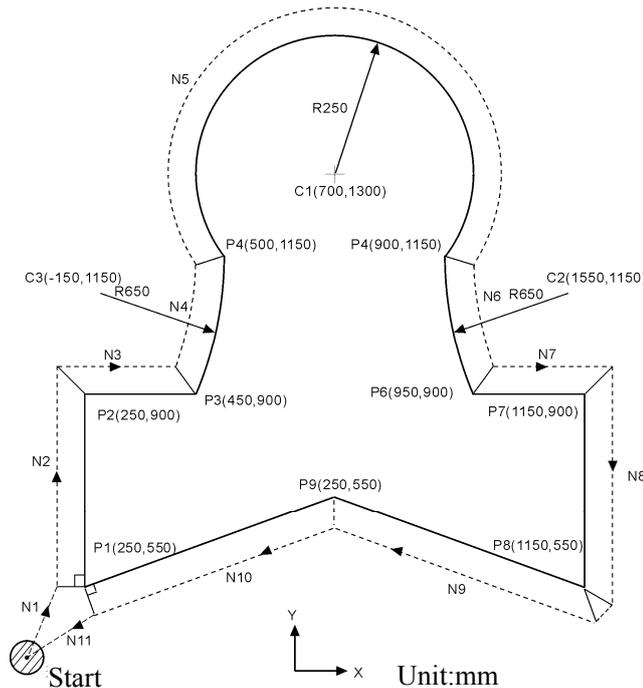
When the tool is moving,tool track can offset one radius.In order to offset one radius, CNC establish offset vector whose length equal tool radius.also starting point. Offset vector is vertical to tool track.Completed machining, it needs to cancel tool radius compensation.



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.

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:



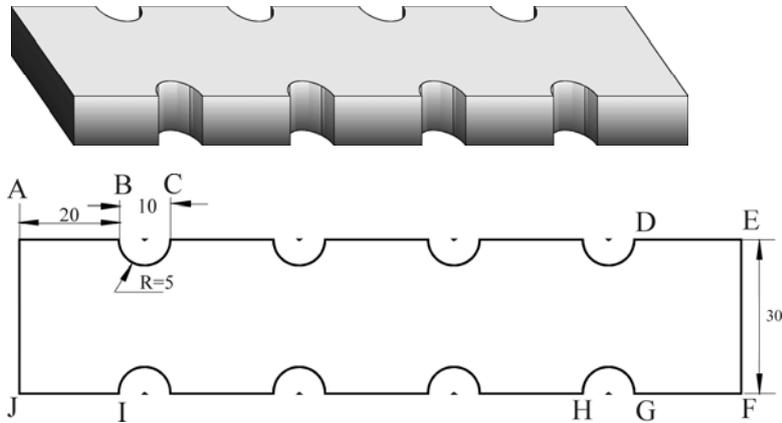
```
G54 X0 Y0 Z0 ; Starting Position: X0 Y0 Z0
N1 G90 G17 G00 G41 T15 D2 X250.0 Y550.0 ; establish compensation
N2 G01 Y900.0 F150 ; from P1 to P2
N3 X450.0 ;from P2 to P3
N4 G03 X500.0 Y1150.0 R650.0 ; from P3 to P4
N5 G02 X900.0 R-250.0 ; from P4 to P5
N6 G03 X950.0 Y900.0 R650.0 ; from P5 to P6
N7 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,and return to starting position
```

(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)

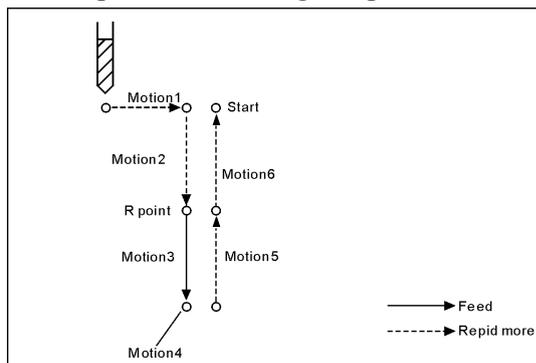
Using Circle code,we can shorten the program length,make the program more simple.

Circle instruction table

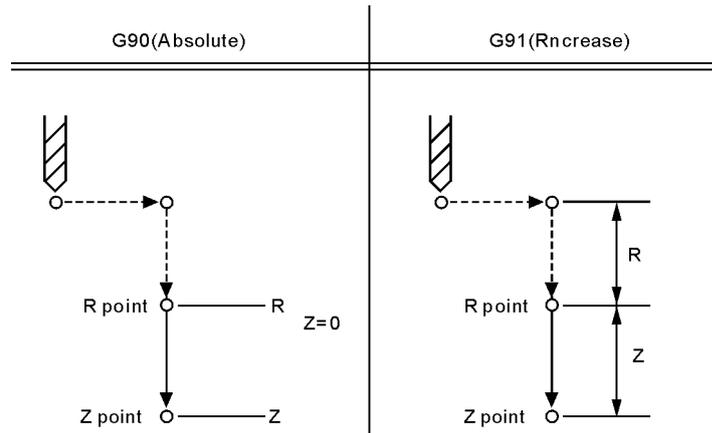
G code	Feed method	Motion in the bottom of hole	withdraw	application
G73	Intermission feed	-	Rapid move	High speed drill deep hole
G80	Continue feed	-	No	Cancel fixed cycle
G81	Continue feed	-	Rapid move	Drill cycle
G82	Continue feed	Stop	Rapid move	Drill cycle
G83	Intermission feed	-	Rapid move	Drill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

Cycle instruction is consist of six motions

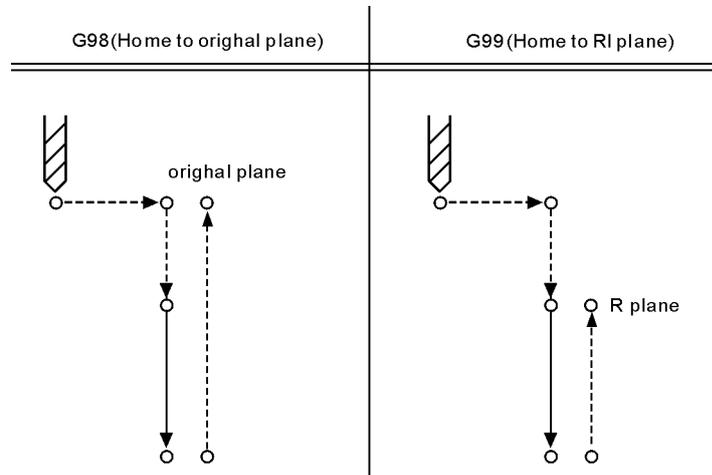
- Motion 1 Position of X and Y axis
- Motion 2 Rapid move to R point
- Motion 3 Machining hole
- Motion 4 Action in the bottom of hole
- Motion 5 Withdraw to R point
- Motion 6 Rapid move to original point



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

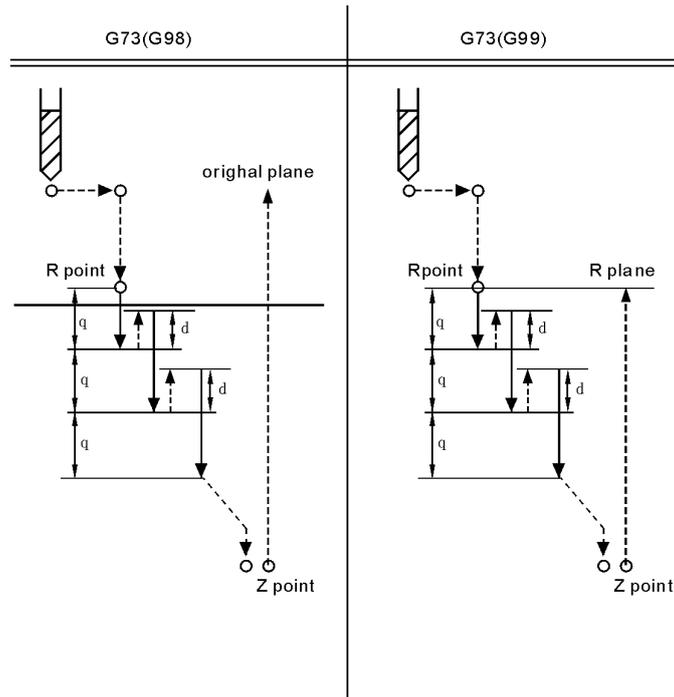
L_:repeated times

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. P1 in User parameter is to set the value of the tool withdrawal(d).

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

Set the offset of tool length(G43 G44 or G49) in changeless cycle which is located

to R point to plus offset, the offset of tool radius is ignored.



For example:

M3 S2000

Spindle starts rotating

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

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

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

G80

M30 ; The end

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

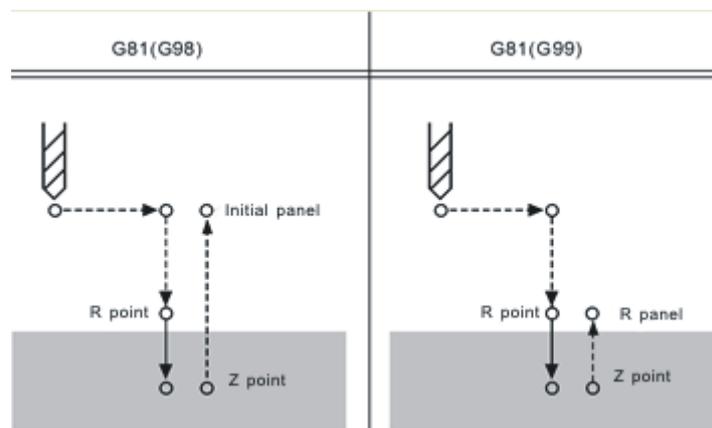
The cycle is used for normal drilling, execute the feeding cut to hole bottom. Then, the tool moves fast return form hole bottom.

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



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

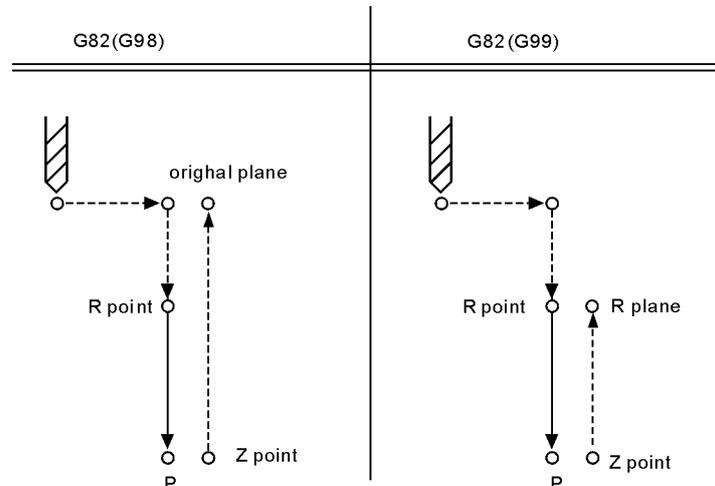
Z-: the distance(G91) or coordinate(G90) from R point to hole bottom

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

P-:pause time

F-:cutting speed

L-:repeat time



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

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

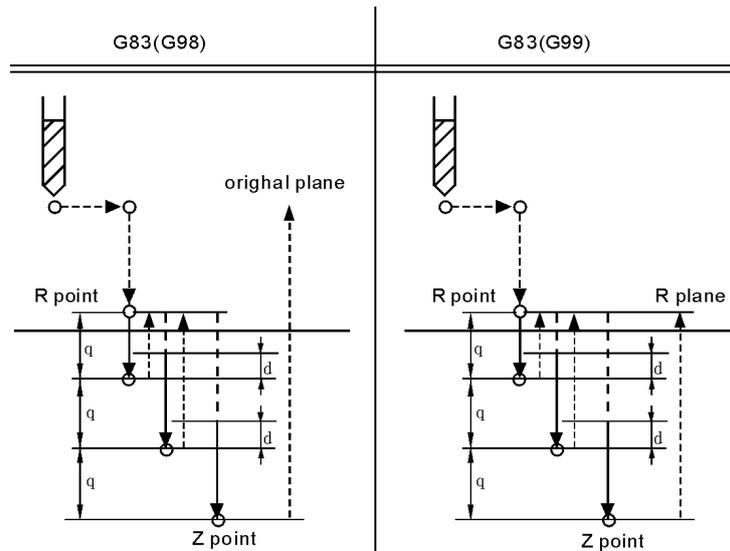
Z_ : the distance(G91) or coordinate(G90) from R point to hole bottom

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

Q_ :cutting depth every time

F_ :cutting speed

L_ :repeated times



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.

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

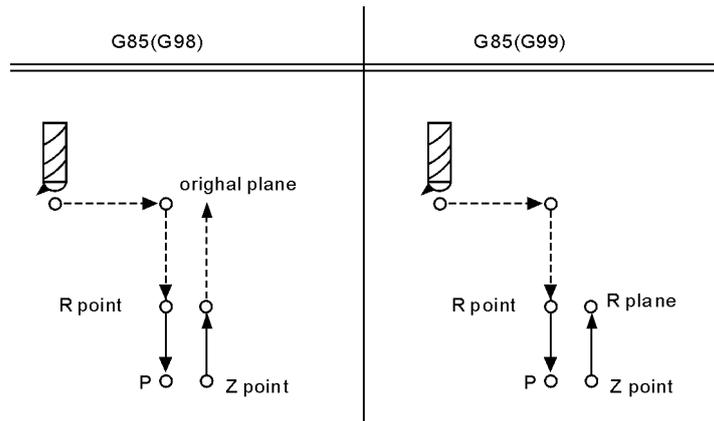
R_ : the distance(G91) or coordinate(G90) from original point to R point

F_ :cutting speed

L_ :repeated times

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.



```

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_ ;

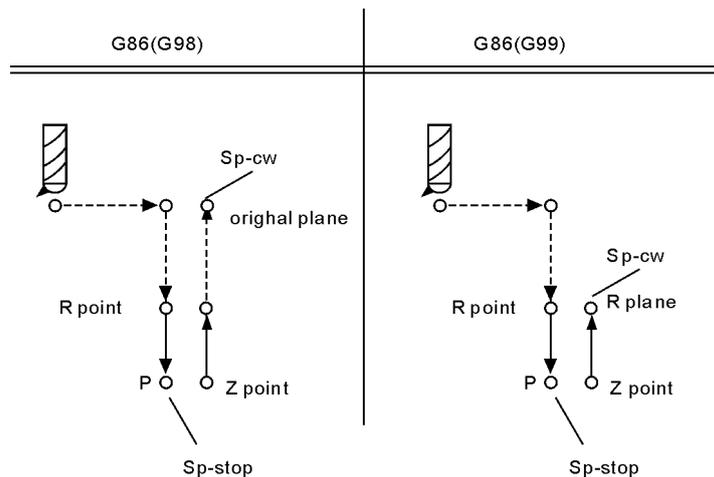
X_ Y_ : hole position data

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



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

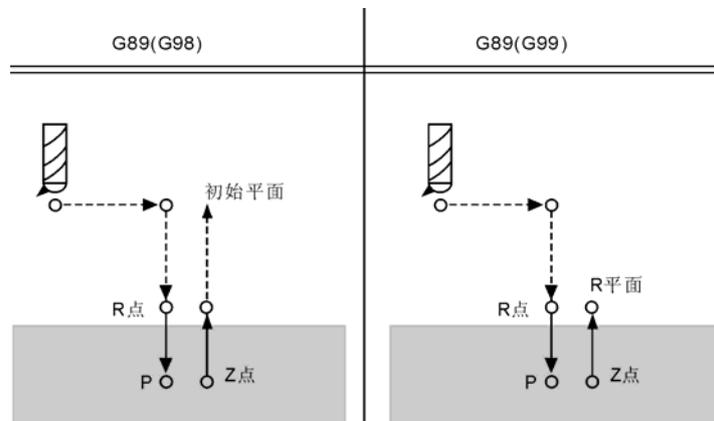
Z_ : the distance(G91) or coordinate(G90) from R point to hole bottom

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

P_ :pause time

F_ :cutting speed

L_ :repeated times



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

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). P404-P413 in Axis parameter to set.

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

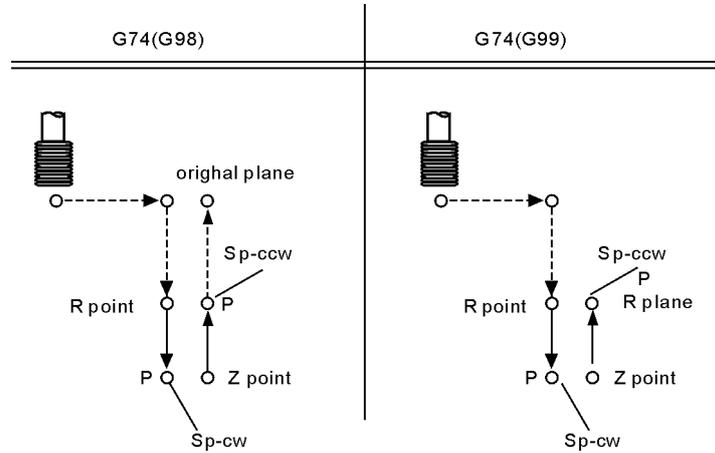
S_ : spindle rotate speed L_ : repeated times

Special Attention: When the spindle and encoder drives not as 1:1, please modifyP412,P413 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.



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 milling 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

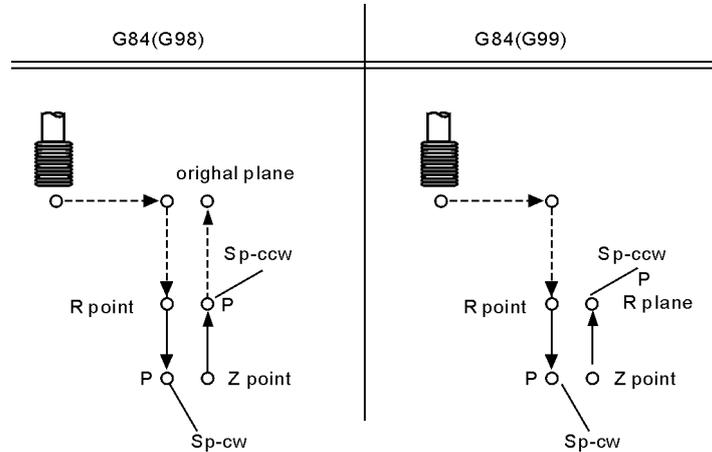
Special Attention: When the spindle and encoder drives not as 1:1, please modify P412, P413

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.



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 pole coordinate

- 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.

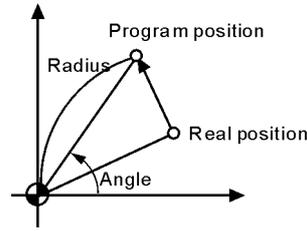
First axis: Radius value of pole coordinate.

Second axis: Value of pole angle.

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

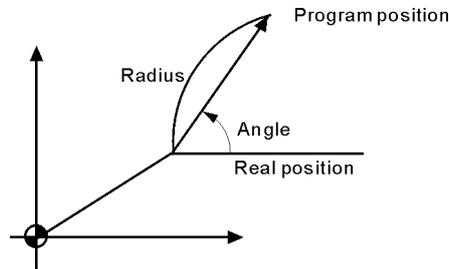
Use programming instruction of absolute value to specify the radius(The distance

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:

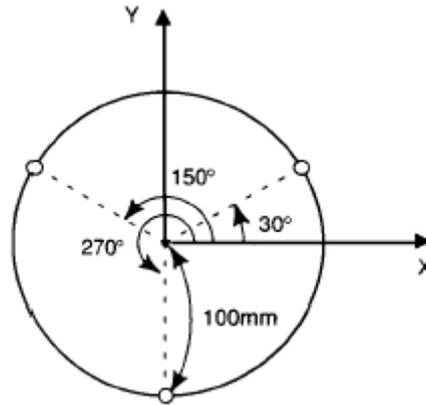


6.G91 set the current position as the origin of pole coordinate system:

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

The G code must be compiled in the beginning of the program, using separate program segment to specify before setting the coordinate system.Switch the unit of input data into minimal inch or millimeter after G code of switching inch or

millimeter specifying, the angle of data input unit keeps unchanged, change the units of value as follows after switching the inch or millimeter:

- The feeding speed is specified by F code
- Position instruction
- Offset value of workpiece zero point
- Compensation value of tool
- The unit of manual pulse generator
- The distance in incremental feeding

(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.

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)Automatic beveling (I) and smoothing(R)

The default panel of milling machine 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.

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:

N0 G54 G0 X-50 Y-50 Z20

N1 M03 S500

N2 G01 G42 D01 X0 Y0 F200

N3 G01 Z-5

N4 X100 I4 ; Beveling4x4

N5 Y40 R6 ; SmoothingR6

N6 X47 R5 ; SmoothingR5

N7 Y70 I3 ; Beveling3x3

N8 X15

N9 X0 Y40

N10 Y0

N11 G0 X-50 Y-50 G40

N12 Z50

N13 M30

(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

Attention: 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.

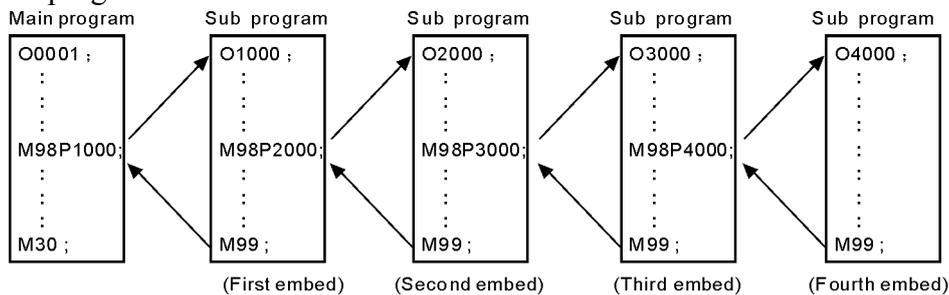
Attention: 1) M99 in the main program is the same as M02;

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

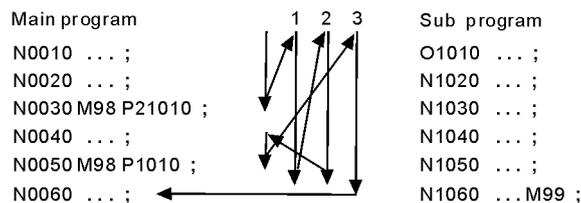
3) M99 returns to main program call in subroutine is in the next line;

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

The Sub-program can embedded call as follow :



For example:



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

Conditional wait, jump instruction

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

Conditions wait

M12 M13 instruction are used to detect the input signal M12, M12 in program line is to detect M12 input signal is effective to execute the next program line , M13 means to detect M12 input signal is invalid to execute the next program line.The instruction is in an independent line.

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

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

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

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

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 P50,P51,P52,P53,P54 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 P50,P51,P52,P53,P54 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 milling T1--T99(Unit: micron)

The macro variables #1401--#1499 corresponds the Z axis offset value of milling 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) ;	90.5 degrees mean 90 degrees 30 minutes
Asin	#i = ASIN(#j);	
Cos	#i = COS(#j) ;	
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
```

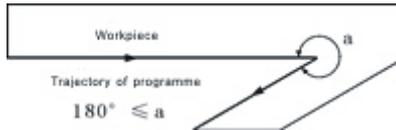
3.5 Radius compensation of tool C

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

3.5.1 Inside and outside

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

Inside:



Outside:



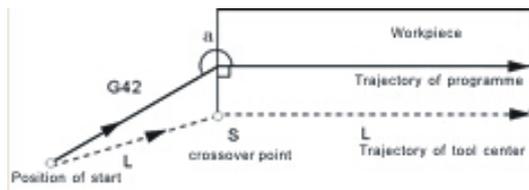
3.5.2 Tool motion when starting

The radius compensation without tool builds tool radius compensation

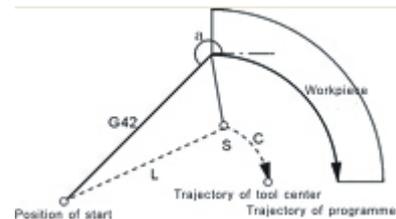
(1) Tool motion around the inside corner ($\alpha \geq 180$)

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

Straight line -> Straight line



Straight line -> Arc

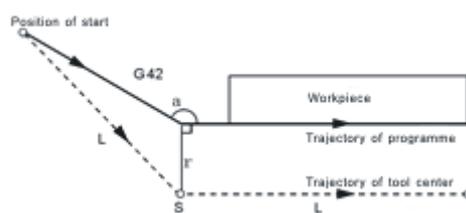


(2) The tool motion around the outside corner of obtuse angle ($90 \leq \alpha < 180$)

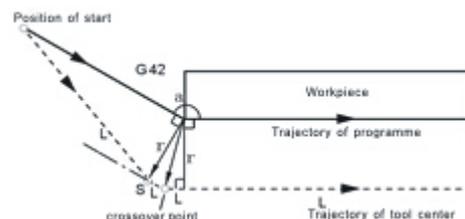
The tool center will move to the tool vector radius vertex of the end point in this program line.

Straight line -> Straight line

A type

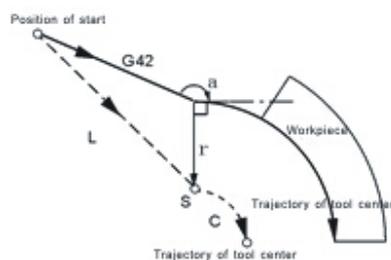


B type

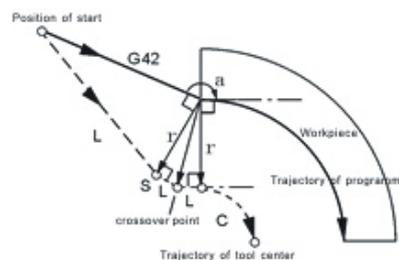


Straight line -> Arc

A type



B type

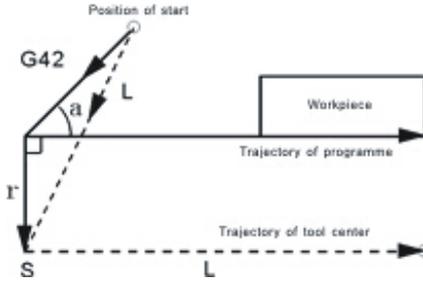


(3)The tool motion around the outside corner of acute angle ($\alpha < 90$)

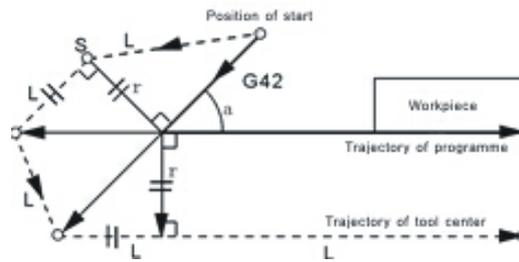
The tool center will move to the tool vector radius vertex of the end point in this program line.

Straight line->Straight line

A type

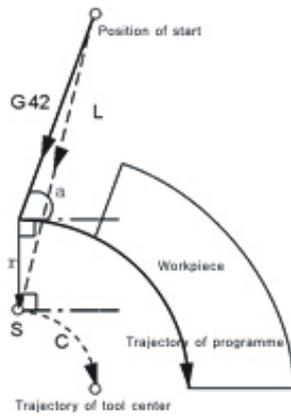


B type

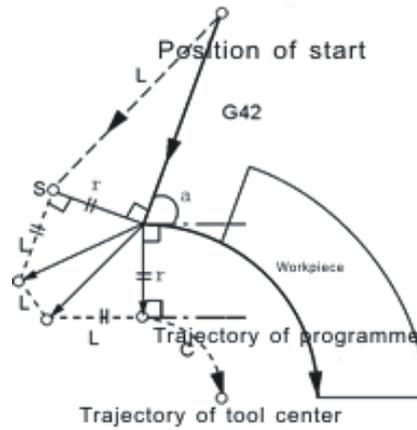


Straight line->Arc

A type



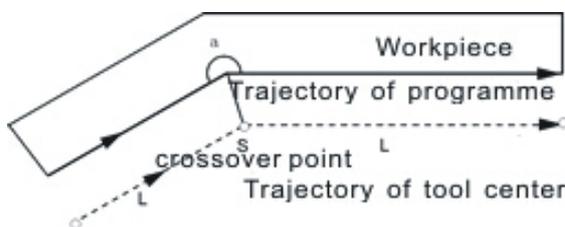
B type



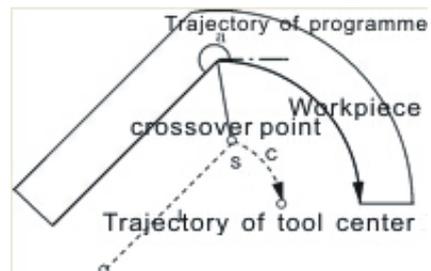
3.5.3 Tool motion in offset mode

(1)Tool motion around the inside corner ($180 \leq \alpha$)

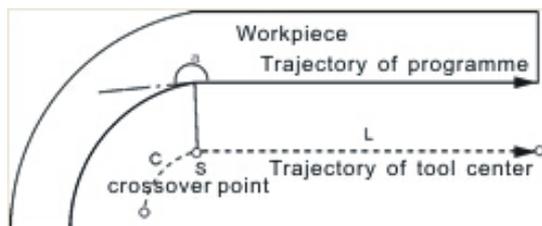
Straight line->Straight line



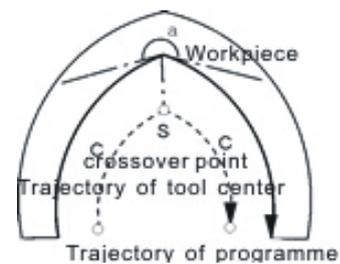
Straight line->Arc



Arc-> Straight line

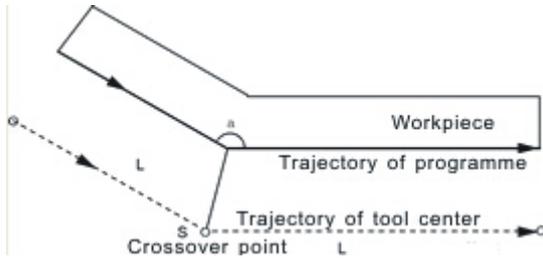


Arc->Arc

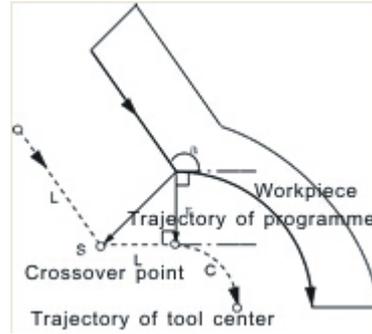


(2)The tool motion around the outside corner of obtuse angle ($90 \leq \alpha < 180$)

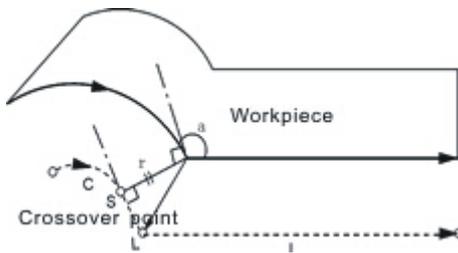
Straight line ->Straight line



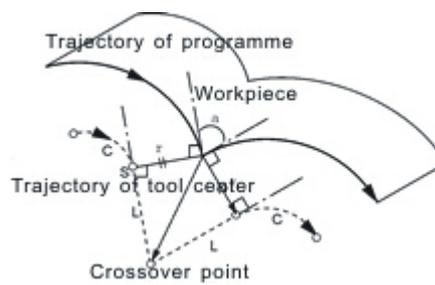
Straight line ->Arc



Arc-> Straight line

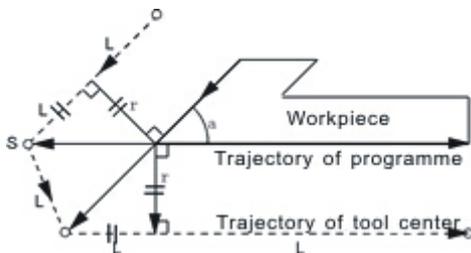


Arc->Arc

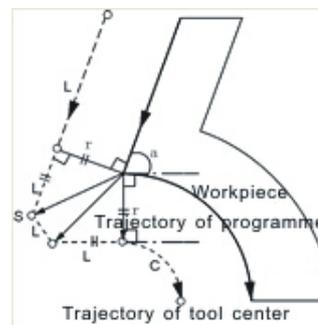


(3)The tool motion around the outside corner of acute angle ($\alpha < 90$)

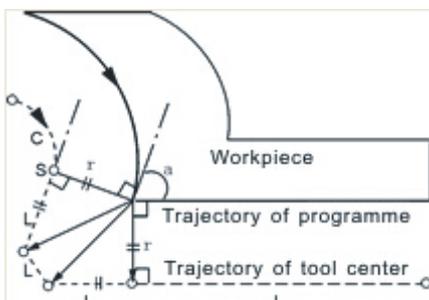
Straight line->Straight line



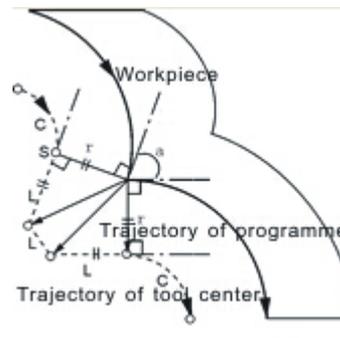
Straight line->Arc



Arc-> Straight line



Arc->Arc

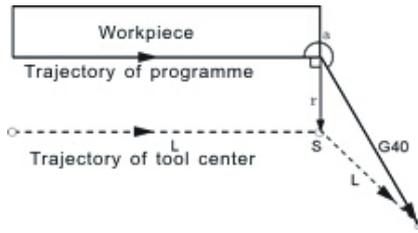


3.5.4 Tool motion in offset-cancel mode

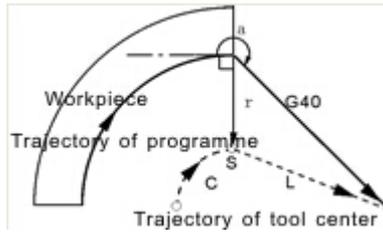
(1) Tool motion around the inside corner ($180 \leq \alpha$)

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

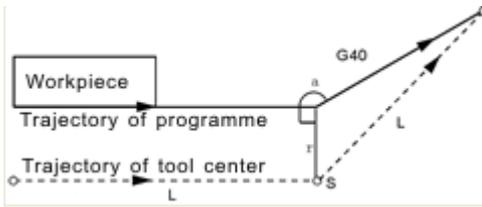


(2) The tool motion around the outside corner of obtuse angle ($90 \leq \alpha < 180$)

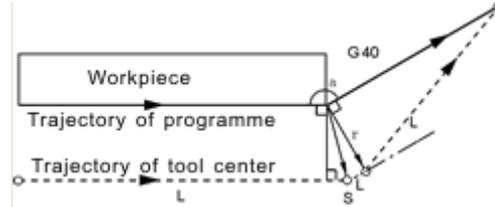
The tool center will move to the tool vector radius vertex of the starting point in next program line.

Straight line → Straight line

A type

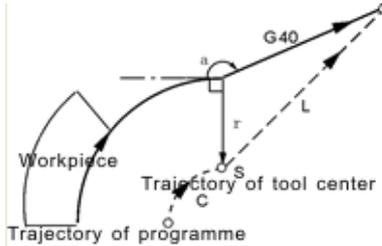


B type

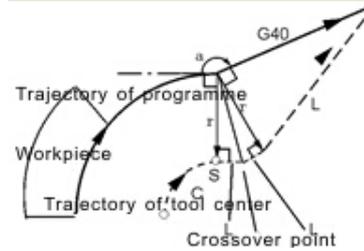


Arc → Straight line

A type



B type

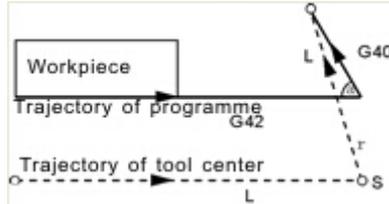


(3) The tool motion around the outside corner of acute angle ($\alpha < 90$)

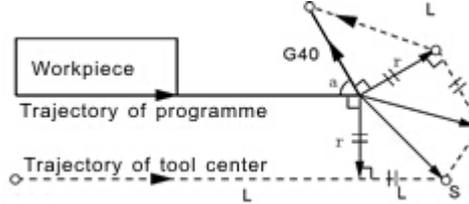
The tool center will move to the tool vector radius vertex of the starting point in next program line.

Straight line → Straight line

A type



B type

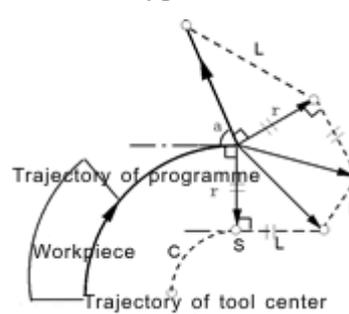


Arc → Straight line

A type



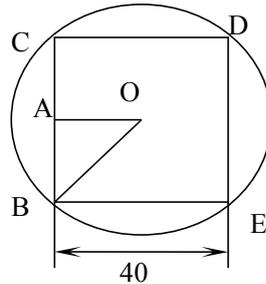
B type



3.6 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 milling to select the correct cutting allowance, for example:

Example 1: Cut square and cut circle

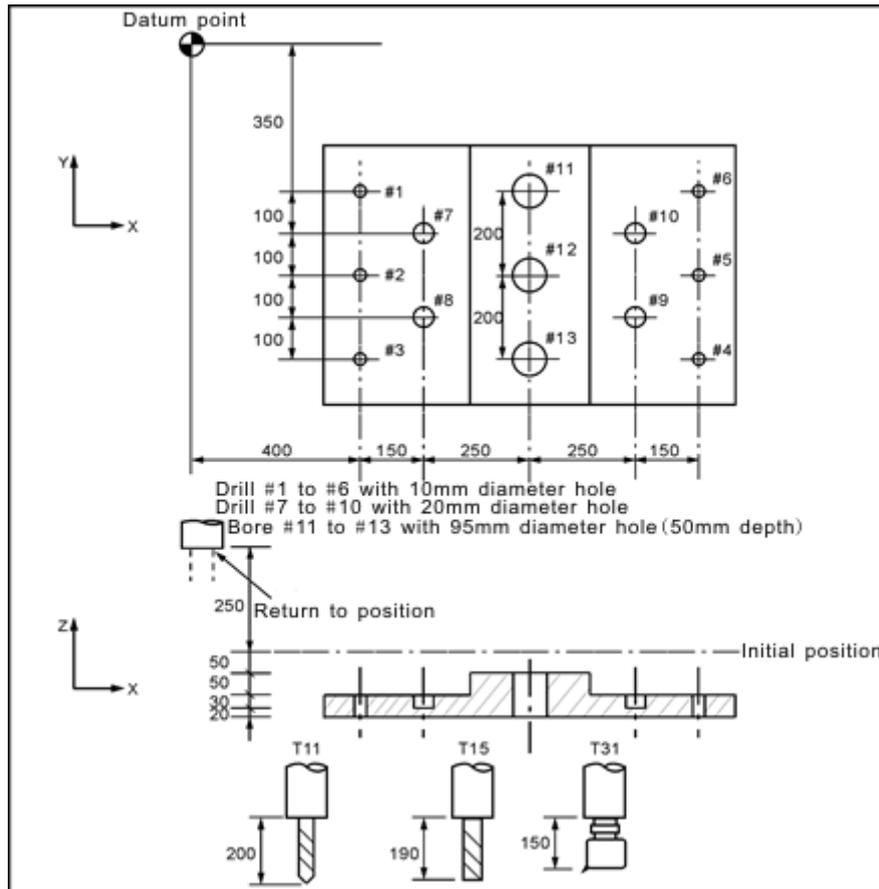


From the center to begin, the center coordinate is G54 X0 Y0 Z50

The tool radius in the D01 of the T01 parameter

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

Example 2:



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

N1 G54 X0 Y0 Z0	Set workpiece coordinate at the datum point
N2 G90 G00 Z250.0 T11	Tool exchange
N3 G43 Z0 H1	The offset length in initial position
N4 S30 M3	Turn on spindle
N5 G99 G81 X400.0 Y-350.0 Z-153.0 R-97.0 F120	To drill #1 hole
N6 Y-550.0	To drill #2 hole and return to initial position
N7 G98 Y-750.0	To drill #3 hole and return to initial position
N8 G99 X1200.0	To drill #4 hole and return to initial position
N9 Y-550.0	To drill #5 hole and return to initial position
N10 G98 Y-350.0	To drill #6 hole and return to initial position
N11 G00 X0 Y0 M5	Return to the datum point and stop spindle
N12 G49 Z250.0 T15	Cancel the tool length offset to change tool
N13 G43 Z0 H1	Tool length offset in initial position
N14 S20 M3	Turn on spindle
N15 G99 G82 X550.0 Y-450.0 Z-130.0 R-97.0 F500	To drill #7 hole and return to initial position
N16 G98 Y-650.0	To drill #8 hole and return to initial position

N17 G99 X1050.0	To drill #9 hole and return to initial position
N18 G98 Y-450.0	To drill #10 hole and return to initial position
N19 G00 X0 Y0 M5	Return to the datum point and stop spindle
N20 G49 Z250.0T31	Cancel the tool length offset to change tool
N21 G43 Z0 H1	Tool length offset in initial position
N22 S10 M3	Turn on spindle
N23 G85 G99 X800.0 Y-350.0 Z-153.0 R47.0 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

3.7 Usage for automatic tool setting gauge

1. Note for parameter:

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

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

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

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

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

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

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

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

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

#388: The minimal milling 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 automatic 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.

3.8 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.

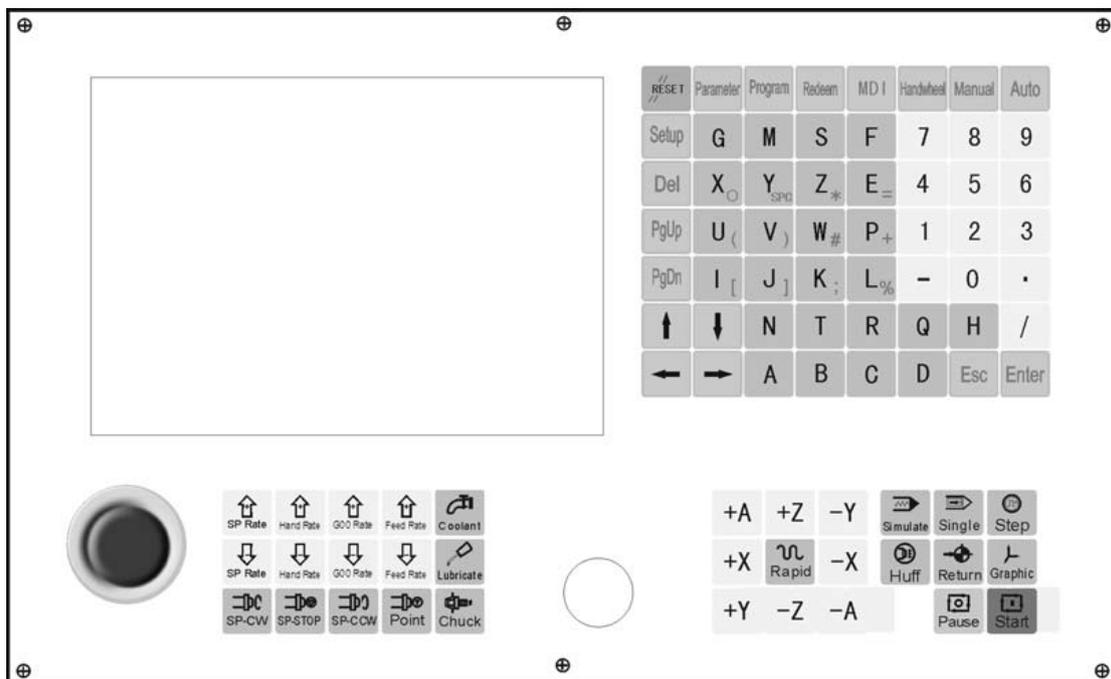
Chapter 4 Operation explanation

4.1 Summary

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

4.2 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.



4.3 Keyboard description

4.3.1 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.

4.3.2 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.

4.3.3 Others keys

Keyboards	Functions
Letter key Number key	ABCDEFGHIJKLMNOPQRSTUVWXYZ123456789 . -: 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 “Setup” for confirming current tool ‘s position in machine tool coordinates system. “Redeem” for amending tool change errors “Auto” entering automatic status “MDI” entering MDI function “  -Graphic” selecting auto-coordinates/diagram machining “  -Single” for single segment or constant work “  -Simulate” for coordinates mode or diagram mode speedy simulating “  -Step” for manual increment or manual continuous
Control Key	“     ” spindle cw, ccw rotation “  -Coolant” coolant on/off “  ” for the shift between electric tool carrier and gang tool carrier

Control Key		“Chuck” for spindle’s chuck clamp/loose tool
		“Huff ” for Huff On/Off.
		“Lubricate” for Lubricate On/Off
		” all axes return to datum point
		“-Rapid” for the shift between hand-driven continuous high speed and low speed.
		”handwheel gear selection
		” adjusting spindle speed
Feed key		” adjusting feed speed
		” adjusting G00 speed
	+X -X +Y -Y +Z -Z +A -A	Feeding with X/Y/Z/A axes

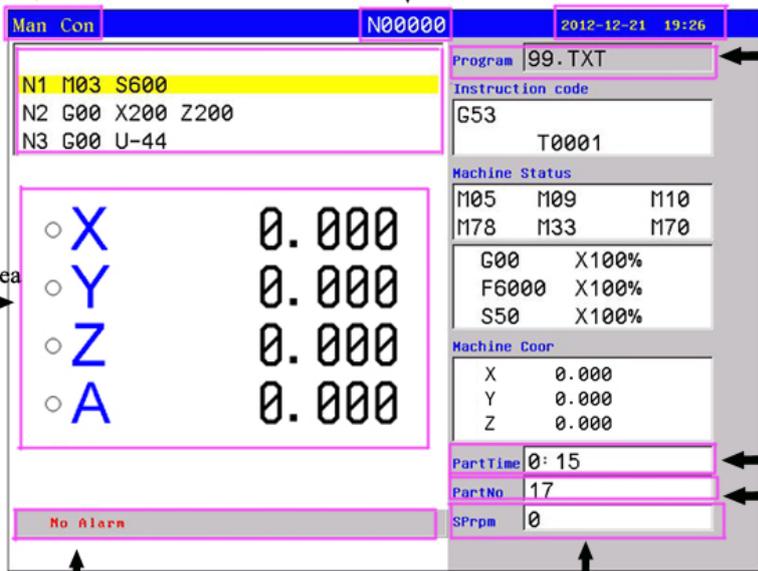
4.4 Manual operation

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

Powering the system is to enter the main interface

Control Mode:Manual Continuous,Manual Increment,Automatic Continuous,
Automatic Single-Step,Automatic Dry,Handwheel.

Line No. of current processing program Time



The screenshot displays the following information:

- Control Mode:** Man Con
- Line No. of current processing program:** N00000
- Time:** 2012-12-21 19:26
- Program:** 99. TXT
- Instruction code:** N1 M03 S600, N2 G00 X200 Z200, N3 G00 U-44
- Machine Status:** M05 M09 M10, M178 M133 M170, G00 X100%, F6000 X100%, S50 X100%
- Machine Coord:** X 0.000, Y 0.000, Z 0.000
- PartTime:** 0: 15
- PartNo:** 17
- SPrpm:** 0
- Alarm Hint:** No Alarm

Annotations in the image:

- “Name of Program” points to the Program field.
- “Processing Time” points to the PartTime field.
- “No. of workpiece” points to the PartNo field.
- “Spindle-Speed” points to the SPrpm field.
- “Displaying Area Of Axis” points to the X, Y, Z, A coordinate display area.

4.4.1 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.

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

(3)“S”: Set the speed of the first spindle. The range is from 0 to 99999, the max depends on P36 in Speed parameter.

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

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

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

(7)“  ”: Press once to increase or decrease the spindle override 10% when P2 in Axis parameter is 0. The range is form 5% to 150%,16 gears totally.

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

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

(10)“Auto”: Select automatic mode.

(11)“Manual”: Select manual mode.

(12)Spindle controlled: “ ||  ||  ||  ”Controlling spindle on clockwise, counterclockwise, stop, corresponding to instructions M03,M04,M05.

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

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

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

(16) “Lubricate”: Lubricate key On/Off, correspond to instructions M32, M33.

(17) “Huff”: Huff key On/Off , correspond to instructions M59,M58.

(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/Y/Z/A key, the X/Y/Z/A-axis goes back to the datum point automatically; Press “0” , Z axis firstly and then XYA axis return home; Press “Esc” to cancel the construction. The speed controlled by P30,P92,P34 in Speed parameter, the direction is up to P35 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 “+X/-X,+Y/-Y,+Z/-Z,+A/-A” correspond to X/Y/Z/A axis in positive or negative direction.

(22) “Speed-Cut”: Press  , the feeding speed change to the speed which is set by P1,P2,P3 in Speed parameter , loosen it that will be the previous speed.If the speed is higher than the speed set by parameters, it will be with the feeding speed.

(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.

Pay Attention: Milling coordinate clear: Press “E” in parameter and then press “Enter”.

4.4.2 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 change to the speed set by P1,P2,P3 in Speed parameter. If the specify feeding speed is higher than the speed set by parameter, Feeding speed will be P1,P2,P3 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 P4 in Speed parameter, when the setting speed is higher than the value of parameter, then the running speed will be set by P4.

When P38=8 in Other parameter, “” is change into a switch, press once to turn on (no more to always press), press again to turn off.

4.4.3 Manual increment

This operation is to set the value of increment as the basis, press feeding key once to run one 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) multi the rate.

4.4.4 Back to datum point (reference point/Home)

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

Switch for datum point:

Back to datum point operation is to feed every axis to Milling'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 machine 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 Milling'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 P38 in Axis parameter, it can be prompt or force;
2. The way and type of detecting signal can be set by P39 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 P40 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 P41 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 P37, P38, P39 in Axis parameter, the value must less than the distance of a cycle.
6. The shifting distance after backing to datum point can be set by P46, P47, P48 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 P33 in Axis parameter, setting P34, P35, P36 in Axis parameter to make sure the floating point of X Y Z axis, the datum point of machine.

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 milling coordinate.

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

If it is 15, the current milling coordinate of X axis is 15.000, the distance to Milling'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 milling process. The system power off unusually or in an accident, it must back to the datum point, otherwise could cause trouble.

4.4.5 Manual Pulse Generator(Handwheel)

Two types: hand held and panel, P1 in Other parameter to set the type.

Panel : Press “Handwheel” and “X” “Y” “Z” “A” to select an axis, “ ” to adjust the ratio.

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 should be not too fast when the system cooperate with stepper 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 P17 in Speed parameter(the bigger the faster). The maximum speed is controlled by P23(X axis) , P24(Y axis) ,P25(Z axis).

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

4.4.6 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.

4.5 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”.

4.5.1 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 milling and slant-bed Milling, No.3 parameter in tool parameter to control.

“”: The program is speedy simulate, the axis of coordinate don't move.

4.5.2 Processing at arbitrary program line or with arbitrary tool

A. Run at actual line

At the condition of Automation, press “—” to popup a dialog box, input a number of line, press “Enter” to confirm, system will start from the input line.

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 P6 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 if no input line no. , 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 P6 in Speed parameter (G01/G02/G03), then process the program normally.

4.5.3 Start program

Press “Auto” to cut to automatical mode to run program, two methods as follows.

- (1) Press “Run”
- (2) Switch on the Run of external signal.

4.5.4 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) Press “Pause” in the panel and stop the program.
- (4) Switch on the Halt of external signal.
- (5) Press “Reset” to stop all the actions of program.(Like spindle, tools and others)

4.5.5 Real-time control in automatical process

(1) “” : Press once to increase or decrease 10% feed speed when P1=0 in Axis parameter, the range is from 0% to 150%,16 gears totally; When P1=1 in Axis parameter, 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 P2=0 in axis parameter . The range is form 5% to 150%,16 gears totally. When P2=1 in Axis parameter, 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.

4.5.6 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.

4.5.7 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.

4.5.8 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, otherwise defeat.

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.

4.6 Operate safety, prompt alarm

4.6.1 Emergency stop

Press “” when emergency accidents happening, the system will stop all the actions of milling machine and shows “Emergency Stop” on the interface. Wait for the button up. M67 output effective signal when P29 in Other parameter to be set effectively.

Press “” in the processing or running, system coordinate and machine's position may change, make sure the position of system coordinate again before

processing, it is best to backing to the datum point to make coordinate same as the machine’s position.

The button can be external which is controlled by P27 in Other parameter to set it to normal open or normal close.

4.6.2 Reset system

Press “Reset” to stop current operation in anytime when the system is running, especially stop all the actions of machine(working table spindle, tools etc.) in automatical or manual mode, but the coordinate won’t lose, so don’t need to Homing.

4.6.3 Alarm

The screen shows error information and twinkles when the milling has alarm, the program is stop running, the coordinate stop moving, check the reason for alarm and clear troubles to run again. M67 signal is effective when P29=1 in Other parameter.

(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 Milling’s spindle is effective.(ALM1)

(4) No.0 alarm: The alarm signal of Milling’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: Alarm signal of M12(door switch) is effective.

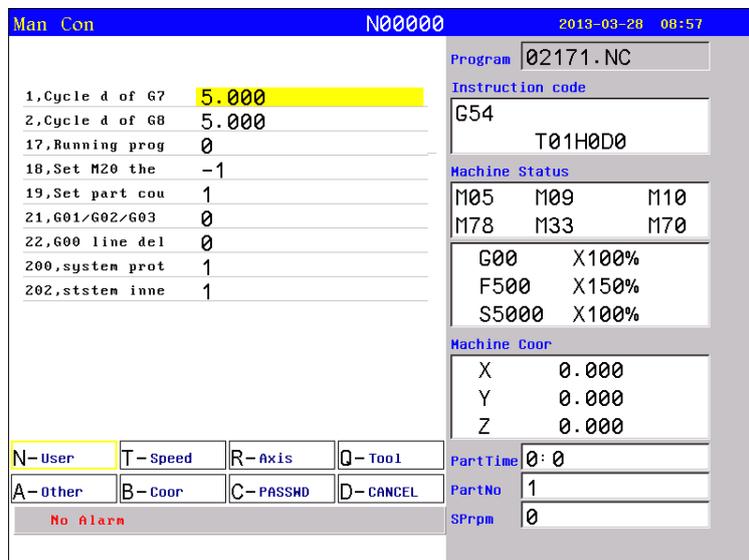
(7) +5V is under voltage: Supply voltage is low, +5V of the system is low.

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

4.7 Parameter operation

At any status, press “Parameter” to enter the status to set the parameter. Parameter includes “User”“Speed”“Axis”“Tool”“Other”“Coordinate”“Password”,7 kinds totally.

Press N、T、R、Q、A、B、C corresponding to corresponding interface in 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:

4.7.1 User parameter (processing)

- 1, Cycle d of G73(mm)
- 2, Cycle d of G83(mm)
- 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]
- 200, system screen protect times[>=2minutes]
- 202,system inner parameter.

Introduction:

- 1, Cycle d of G73(mm)
It is for set the value of d about G73.detailed introduction is in G73 program.
- 2, Cycle d of G83(mm)
It is for set the value of d about G83.detailed introduction is in G83 program.
- 17,Running program need Sp run [1 mean Yes,0 mean No]
It is for interlock between run program and run spindle,1 means Run program only when Spindle is running. 0 means Run program no need to check if spindle running.
- 18,Set M20 the times of auto-running [Negative number mean immensity loop]
It is see for times of use M20 code in program, minus mean cycle countless times.
- 19,Set part count
It is for display and set the times of running M20,also workpiece number.
- 21,G01/G02/G03 line delay(ms)[>100]
It is for set delay time between internal of G01/G02/G03 segment, which is for solve the over-cutting in the corner.
- 22,G00 line delay(ms)[>100]
It is for set delay time after running G00 ,it is effective that more than 100ms.
- 23,Handwheel acceleration[50-100]
It is for set the constant of handwheel smoothly acc/dec-eleration. the smaller it is,the faster the acc/dec-eleration is,but much vibration.
- 200,system screen protect times [>=2minutes]
It is the time that enter protection screen when system stay in main screen and without dialog,don't enter screen protection if less than 2 minutes,press any keys to return back.

4.7.2 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)

Introduction:

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)

The max value of G00 is 30000 . (unit:mm/min)

Attention:the value is related to machine's configuration,wrong set is very dangerous.

5,Manual maximum feed speed(unit: mm/min)

It is max feeding speed in Manual ,(suggestion speed=G00 speed*50%)

6,Auto Maximum feed speed(unit: mm/min)

It is the max feeding speed in Auto.the speed could faster than G00 speed.

7,G01/G02/G03 default speed(unit: mm/min)

It is the default speed when it isn't specify in first G01/G02/G03 codes.Max:5000.

8,Null run speed(unit: mm/min)

It is speed of simulate run. (press "simulate" is null run) Max:30000.

9,Feed axis`s manual speed(unit: mm/min)

It is the speed of feeding axis in Manual.Range: <Max feeding speed in Manual.

Attention:in Manual,press "F" key and set,refresh the parameter value automatically.

10,Spindle`s manual speed(rpm)

It is speed of spindle in manual.Unit:rpm.

Attention: in Manual,after press "S"key and set ,refresh the parameters automatically.

11,Beginning feed speed(mm/min)

It is beginning speed of feeding axis when acc/dec-eleration.when it is smaller than acceleration/ deceleration ,accelerate/decelerate start from the beginning feed speed.when it is bigger than acceleration/deceleration,the speed reached directly.

Attention:it is related to machine configuration,in general,stepper system is less than 100,servo system is less than 500.

12,Jump speed at continuous track(mm/min)

It is for increase the continuous when running multiaxial track-interpolation.

Example:when it is 300,the speed of X axis(multiaxial track-interpolation)up from F800 to F1600,800=(1600-800)>300,so the process is up from F800 to F1100,and then F1600.

13,Limit G1/G2/G3 axis speed [1 mean Yes,0 mean No]

It is for whether limit the max speed of each axis when G1/G2/G3 interpolating.

14,X G1G2G3 max speed(mm/min)

It is for the Max running speed of X-axis when set G1/G2/G3 interpolation.

15,Y G1G2G3 max speed(mm/min)

It is for the Max running speed of Y-axis when set G1/G2/G3 interpolation.

16,Z G1G2G3 max speed(mm/min)

It is for the Max running speed of Z-axis when set G1/G2/G3 interpolation.

17,A G1G2G3 max speed(mm/min)

It is for the Max running speed of A-axis when set G1/G2/G3 interpolation.

18,X acceleration [1~99999]

It is time constant of X-axis acc/dec-eleration,the bigger it is ,the faster the ace/dec-eleration is.

Attention:This value depends on the machine structure,the heavier the load is ,the smaller the value is.With stepper system,the value should less than 15000.

19,Y acceleration [1~99999]

It is time constant of Y-axis acc/dec-eleration,the bigger it is ,the faster the ace/dec-eleration is.

Attention:This value depends on the machine structure,the heavier the load is ,the smaller the value is.With stepper system,the value should less than 15000.

20,Z acceleration [1~99999]

It is time constant of Z-axis acc/dec-eleration,the bigger it is ,the faster the ace/dec-eleration is.

Attention:This value depends on the machine structure,the heavier the load is ,the smaller the value is.With stepper system,the value should less than 15000.

21,A acceleration [1~99999]

It is time constant of A-axis acc/dec-eleration,the bigger it is ,the faster the ace/dec-eleration is.

Attention:This value depends on the machine structure,the heavier the load is ,the smaller the value is.With stepper system,the value should less than 15000.

22,Auto run acceleration [1-500]

It is for set constant of acc/dec-eleration in auto.the range is 1-500.It is mainly for distinguish Auto and Manual,only the difference is too much,set it is effective.

23,Handwheel acceleration [500--32000]

It is for set constant of acc/dec-eleration of Handwheel.the range is 500-32000.

24,Run program Handwheel acceleration [>500]

It is for set constant of acc/dec-eleration of Handwheel when running program.the range is from 500-32000.when the value is less than 500,it is invalid.

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

It is the G00 speed when triggered by Handwheel whitin testing.Invalid when <10.

26,Handwheel X limit speed(mm/min)

It is for limit the handwheel max speed of X-axis when use handwheel in manual.

Attention:it is valid when >100,otherwise invalid.

27,Handwheel Y limit speed(mm/min)

It is for limit the handwheel max speed of Y-axis when use handwheel in manual.

Attention:it is valid when >100,otherwise invalid.

28,Handwheel Z limit speed(mm/min)

It is for limit the handwheel max speed of Z-axis when use handwheel in manual.

Attention:it is valid when >100,otherwise invalid.

29,Handwheel A limit speed(mm/min)

It is for limit the handwheel max speed of A-axis when use handwheel in manual.

Attention:it is valid when >100,otherwise invalid.

30,acceleration type [0 mean line,8 mean curve]

It is for set type of ace/dec-eleration.set 0 means line type.set 8 means curve type.

Attention:In normal condition,set line type in step system;set curve type in servo system.

31,curve ini acceleration [≥ 10]

It is initial ace/dec-eleration constant when set curve type.It is valid when ≥ 10 .

32,curve acceleration [≥ 10]

It is second ace/dec-eleration constant when set curve type.It is valid when ≥ 10 .

33,curve max acceleration [≥ 500]

It is Max ace/dec-eleration constant when set curve type.

It is valid when ≥ 500 ,otherwise the ace/dec-eleration constant is with line type of each axis.

34,X go home rampit speed(Unit: mm/min)

It is speed of X-axis when go home in forward direction.the range is less than the G00 speed of X-axis.

35,X go home reverse speed(Unit: mm/min)

It is speed of X-axis when go home in reverse direction.the range is 20-500.

Attention:it is for ensure accuracy.the smaller it is ,the higher the accuracy is.when set well,don't change it forever.

36,Y go home rampit speed(Unit: mm/min)

It is speed of Y-axis when go home in forward direction.the range is less than the G00 speed of Y-axis.

37,Y go home reverse speed(Unit: mm/min)

It is speed of Y-axis when go home in reverse direction.the range is 20-500.

Attention:it is for ensure accuracy.the smaller it is ,the higher the accuracy is.when set well,don't change it forever.

38,Z go home rampit speed(Unit: mm/min)

It is speed of Z-axis when go home in forward direction.the range is less than the G00 speed of Z-axis.

39,Z go home reverse speed(Unit: mm/min)

It is speed of Z-axis when go home in reverse direction.the range is 20-500.

Note:it is for ensure accuracy.the smaller it is ,the higher the accuracy is.when set well,don't change it forever.

40,A go home rampit speed(Unit: mm/min)

It is speed of A-axis when go home in forward direction.the range is less than the G00 speed of A-axis.

41,A go home reverse speed(Unit: mm/min)

It is speed of A-axis when go home in reverse direction.the range is 20-500.

Attention:it is for ensure accuracy.the smaller it is ,the higher the accuracy is.when set well,don't change it forever.

42,Spindle first max speed(Unit: rpm)

It is the first max speed of spindle,it is also the speed when voltage is 10V.

43,Spindle second max speed(Unit: rpm)

It is the second max speed of spindle,it is also the speed when voltage is 10V.

44,Spindle third max speed(Unit: rpm)

It is the third max speed of spindle,it is also the speed when voltage is 10V.

45,Spindle forth max speed(Unit: rpm)

It is the forth max speed of spindle,it is also the speed when voltage is 10V.

46,Second Spindle max speed(Unit: rpm)

It is the max speed of the second spindle,it is also the speed when voltage is 10V.

47,reverse compensation mode(0 mean A; 8 mean B)

It is reverse compensation mode of arc gap.

0 means A mode.(A mode is the bigger it is ,the faster the speed is.the speed should not be bigger than 1000mm/min,the speed also is related with the value of reverse gap compensation.)

8 means B mode.(B mode is the speed depends on the related parameters.)

48,mode B reverse compensation speed(Unit: mm/min)

It is the speed of reverse compensation in B mode.

48-1,mode B reverse compensation Beginning feed speed(Unit: mm/min)

It is beginning speed of reverse compensation in B mode.it is valid when it >10.

48-2,mode B reverse compensation acceleration(Unit: (mm/min)/s)

It is the constant of reverse compensation acceleration It is valid when it >=10.

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

It is for whether system activate the function that prior to process speed when in short-line processing,1 means yes,open the function.

50,Handwheel stop speed(Unit: mm/min)[>100]

It is the speed when handwheel stop.the bigger it is ,the shorter the stop time is.

58,Forcedly limit drop speed critical(Unit: mm/min)

It is starting drop speed when it is force limit.when servo system, it is 1.

4.7.3 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
- 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 milling 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 milling coordinate [0 No;1 plan]

Introduction:

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

It is set for operation way of alter feeding axis's rate.1 mean select external band switch to alter.0 mean use Feed rate+/- in panel.

Explanation: When set to 1,the ports of external band switch are VDK0(OFF), VDK1(*100),VDK2(*10),VDK3(*1) of CN11 connector.

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

It is for operation way of alter spindle axis's rate.1 mean selection external band switch to alter.0 mean use SP rate+/- in panel.

Explanation: When set to 1,the ports of external band switch are VDS0(HCOM), VDS1(Z),VDS2(Y),VDS3(X) of CN11 connector .

3,X-axis's negative scope(Unit: mm)

It is the coordinate value of X-axis soft limit in max scope of negative direction.

4,X-axis's positive scope(Unit: mm)

It is the coordinate value of X-axis soft limit in max scope of positive direction.

5,Y-axis's negative scope(Unit: mm)

It is the coordinate value of Y-axis soft limit in max scope of negative direction.

6,Y-axis's positive scope(Unit: mm)

It is the coordinate value of Y-axis soft limit in max scope of positive direction.

7,Z-axis's negative scope(Unit: mm)

It is the coordinate value of Z-axis soft limit in max scope of negative direction.

8,Z-axis's positive scope(Unit: mm)

It is the coordinate value of Z-axis soft limit in max scope of positive direction.

9,A-axis's negative scope(Unit: mm)

It is the coordinate value of A-axis soft limit in max scope of negative direction.

10,A-axis's positive scope(Unit: mm)

It is the coordinate value of A-axis soft limit in max scope of positive direction.

11,Spindle stop time(Unit: 10ms)

It is the braking time of spindle,the shorter it is,the faster the brake is.

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

When it is 1,the signal of spindle's stop is long signal,when it is 0,it is short signal.

13,Soft limit invalid [D2X;D3Y;D4Z;D5A;0: valid,1: invalid]

It is ,bit parameter, set for if it is valid of soft limit function.Each axis is set alone. D2:X;D3:Y(C); D4:Z, D5:A.coresponding bit set to 1 mean valid,0 means invalid.

Example:the soft limit of X-axis is valid,the bit parameter is 00000100.

14,X-axis's reverse compensation(um)

It is the value of reverse compensation(radius),when X-axis is running in negative direction.when X-axis run in negative direction,system compensate with the value.

15,Y-axis's reverse compensation(um)

It is the value of reverse compensation(radius),when Y-axis is running in negative direction.when Y-axis run in negative direction,system compensate with the value.

16,Z-axis's reverse compensation(um)

It is the value of reverse compensation(radius),when Z-axis is running in negative direction.when Z-axis run in negative direction,system compensate with the value.

17,A-axis's reverse compensation(um)

It is the value of reverse compensation(radius),when A-axis is running in negative direction.when X-axis run in negative direction,system compensate with the value.

18,X-axis's direction signal [1 mean normal,0 mean reverse]

It is for change the direction of X-axis.when it is 0,the direction of code is opposite to the direction of moving.when it is1,the direction is same.

19,Y-axis's direction signal [1 mean normal,0 mean reverse]

It is for change the direction of Y-axis.when it is 0,the direction of code is opposite to the direction of moving.when it is1,the direction is same.

20,Z-axis's direction signal [1 mean normal,0 mean reverse]

It is for change the direction of Z-axis.when it is 0,the direction of code is

opposite to the direction of moving.when it is1,the direction is same.

21,A-axis's direction signal [1 mean normal,0 mean reverse]

It is for change the direction of A-axis.when it is 0,the direction of code is opposite to the direction of moving.when it is1,the direction is same.

22,Close feed electron gear [1 mean Yes,0 mean No]

It is for whether close the electron gear of feeding axis.1:close,0: no close.

23,X-axis's electron gear numerator(1-999999)

It is the numerator of X-axis's electron gear,Multiplication ratio of axis X's instruction(X_CMV)

24,X-axis's electron gear denominator(1-999999)

It is the denominator of X-axis's electron gear ,Frequency-division coefficient of axis X's instruction(X_CMD)

25,Y-axis's electron gear numerator(1-999999)

It is the numerator of Y-axis's electron gear,Multiplication ratio of axis Y's instruction(Y_CMV)

26,Y-axis's electron gear denominator(1-999999)

It is the denominator of Y-axis's electron gear ,Frequency-division coefficient of axis Y's instruction(Y_CMD)

27,Z-axis's electron gear numerator(1-999999)

It is the numerator of Z-axis's electron gear,Multiplication ratio of axis Z's instruction(Z_CMV)

28,Z-axis's electron gear denominator(1-999999)

It is the denominator of Z-axis's electron gear,Frequency-division coefficient of axis Z's instruction(Z_CMD)

29,A-axis's electron gear numerator(1-999999)

It is the numerator of A-axis's electron gear,Multiplication ratio of axis A's instruction(A_CMV)

30,A-axis's electron gear denominator(1-999999)

It is the denominator of A-axis's electron gear,Frequency-division coefficient of axis A's instruction(A_CMD)

P23-P30 parameters:

Effective Range: 1-999999

Unit:non

User:Upon operating administrators

Initialization:1

Effective time:Immediately

Explain:

When lead screws with different screw pitches are configured with motors of various step angles,or with servo motors of different pulse number per round,or connections are realized through different gears,the programmed values can remain consistent with the actual moved distance by setting the parameter of the electronic gear ration of the system.

$CMR/CMD = P / (L * 1000)$

CMR:Numerator of gear ratio

CMD:Denominator of gear ratio

P: pulse number per motor round

L: Moved distance per motor round(mm)

The value of CMD/CMR is the pulse equivalent,which tells the moved distance per pulse ,with its unit as 0.001mm.

Example1:

The motor rotates one circle very 5000 pulses,after which the machine tool moves 5mm,then:

$$\text{CMR}/\text{CMD} = 5000 / (5 * 1000) = 1 / 1$$

That is to say,we can set the values as :CMR=1,CMD=1.

Here ,the pulse equivalent is 0.001mm.

Example2:

The motor rotates one circle very 5000 pulses,after which the machine tool moves 10mm.

$$\text{CMR}/\text{CMD} = 5000 / (10 * 1000) = 1 / 2$$

That is to say,we can set the values as :CMR=1,CMD=2/

Here ,the pulse equivalent is 0.002mm.

31,XYZA positive limit [0 open,1 close]

It is for type of limit switch of XYZA-axis in positive direction.0 means the switch is normal open,1 means it is normal close.

32,XYZA negative limit [0 open,1 close]

It is for type of limit switch of XYZA-axis in negative direction.0 means the switch is normal open,1 means it is normal close.

33,float zero bit parameter [D3X;D4(C)Y;D5Z;D6A;0 machine Zero;1 float Zero]

It is bit parameter,set for whether the float zero function is valid.Each axis is set alone. D3:X; D4:C(Y); D5:Z; D6:A;1 means the axis is float zero point,0 mean with switch of machine zero point.

Example:X set float zero point,the bit parameter is 00001000.

34,X coordinate float zero set

It is the coordinate value of X-axis float zero point.

35,Y coordinate float zero set

It is the coordinate value of Y-axis float zero point.

36,Z coordinate float zero set

It is the coordinate value of Z-axis float zero point.

37,A coordinate float zero set

It is the coordinate value of A-axis float zero point.

38,Feed axis home [1 mean No use, 0 mean clew, 8 compulsion , 9 must compulsion]

It is request that feeding axis go home.there is four kinds way of go home as follow:

1 means no request.when boot every time,no prompt and no limitation;

0 means prompt,when boot every time,there will be a prompted screen;

8 means enforcement,when boot every time,there will a prompted screen,and then,if the system don't go home,it will note "feed axis don't go home" ,and don't run the program;

9 means much enforcement,when boot every time,there will a prompted screen,and then,if the system don't go home,it will note "feed axis don't go home" ,and feed axis don't move.

39,Feed axis home mode

[0 reverse check,1 reverse No check ,2 No reverse check,3 No reverse No check]

It is mode that check the switch and Z pulse of motor's encoder when feeding axis go home:

When it is 0,go home after hit the switch,move in reverse direction until check the switch is disengaged,and then check the Z pulse of encoder.

When it is 1,go home after hit the switch,move in reverse direction until check the switch is disengaged.

When it is 2,go home after hit the switch,move forward until check the switch is disengaged,and then check the Z pulse of encoder.

When it is the rest,go home after hit the switch,move forward until check the switch is disengaged.

40,Home reverse direction [D1check tool chuck is position D2X;D3Y;D4Z;D5A]

It is for the homing direction of feeding axis.It is bit parameter,each axis is set

alone.D2:X;D3:Y;D4:Z;D5:A;1 mean negative direction,0 mean positive,1 means Z-axis first,0 means X-axis first.

Example:when set X-axis Homing in negative direction,Bit parameter is 100000100.

Whether check tool chuck clamp/loose in position.When P40_D1=1,means check,P40_D1=0,means no check.If set as check,After clamp M10 is in position ,system will check if M14 is valid automatically,it will enter next step only M14 is valid; After loose M11 is in position,system will check if M12 is valid automatically, it will enter next step only when M12 is valid.

41,Home NC switch bit set [D0X;D1C(Y);D2Z;D3A;D4B;D7:Manual/Auto cut automatically ;1: Normal Close ; 0: Normal Open]

It is for set the type of home switch,set alone,it is bit parameter. D0:X;D1:Y;D2:Z; D3:A; 1 means NC,0 means NO. D7:1 means that cut Manual/Auto automatically, enter Manual condition after program is finished automatically.Press 'Start' button In Manual ,system will cut to Auto and run program automatically.

Example: If X/Y/Z-axis all are NC switch ,the bit parameter is 000000111 .

42,X check zero max length(unit:100um)

It is the length that check zero pulse of encoder when X-axis go home and after disengaged switch.

Attention:the value must be less than the length of one rev,otherwise,go wrong home.

43,Y check zero max length(unit:100um)

It is the length that check zero pulse of encoder when Y-axis go home and after disengaged switch.

Attention:the value must be less than the length of one rev,otherwise,go wrong home.

44,Z check zero max length(unit:100um)

It is the length that check zero pulse of encoder when Z-axis go home and after disengaged switch.

Attention:the value must be less than the length of one rev,otherwise,go wrong home.

45,A check zero max length(unit:100um)

It is the length that check zero pulse of encoder when A-axis go home and after disengaged switch.

Attention:the value must be less than the length of one rev,otherwise,go wrong home.

46,X Home offset(unit:10um, Range: -9999~+9999)

It is offset that X-axis move in G00 speed after homing ,after check zero pulse.

47,Y Home offset(unit:10um, Range: -9999~+9999)

It is offset that Y-axis move in G00 speed after homing ,after check zero pulse.

48,Z Home offset(unit:10um,Range: -9999~+9999)

It is offset that Z-axis move in G00 speed after homing ,after check zero pulse.

49,A Home offset(unit:10um,Range: -9999~+9999)

It is offset that A-axis move in G00 speed after homing,after check zero pulse.

50,Have Spindle class control : [1 mean open,0 mean close]

It is if the spindle is booting when change gears.1 mean spindle is Yes,0 mean not.

51,Spindle class speed(1/100rpm)

It is the speed that the spindle boot when the spindle change gears.

52,Spindle class direction : [0 mean M03,1 mean M04]

It is the direction that the spindle boot when SP change gears,1 means reverse,0 means forward.

53,Spindle class stop time(unit:10ms)

It is the time that the spindle stop (M05) when SP change gears.

54,Spindle class time(unit:10ms)

It is the time that the spindle run in low class.unit:10ms

55,Spindle stop time(unit:10ms)

It is the delay time between cancel M03/M04 and boot M05.unit:10ms.

56,Check SP encode [1 mean Yes,0 mean No]

It is for whether system detect the signal of spindle 's encoder.when 1 ,means detect,0 mean no detect.It is for display and open related functions.but first of all,the gear ratio between spindle and encoder is 1:1.

57,SP encode pulse

It is for set the pulse of every rev of spindle's encoder.the value is Lines of SP *4.
80,XZ axis coordinate plan[D2Zworkpiece,D3Xworkpiece,D4Ztool,D5Xtool,D6 Z circle rotate ,D7Xcircumrotate]

It is bit parameter,D2:Z axis in workpiece coordinate system;D3:X axis in workpiece coordinate system;D4 is Z axis in machine coordinate system;D5 is X axis in machine coordinate system.D6 is Z axis whether is rotation axis;D7 is X axis whether is rotation axis.1 means valid/yes;0 means invalid/no.

81, Y axis [0 mean circle rotate axis,1 mean line axis]

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

It is for whether Y axis is in workpiece coordinate system when it is rotate axis.

83,Y is circle rotate axis machine coordinate [0 No;1 plan]

It is for whether Y axis is in machine coordinate system when it is rotate axis.

100,Inner parameter

300,Inner parameter

101,The fourth axis setting function [0 mean rotating axis,1 mean line axis]

102,The fourth axis is rotating axis milling coordinate [0 No;1 plan]

It is set for A-axis is base on workpiece coordinate system or machine coordinate sytem. 1 means workpiece coordinate system,0 means machine coordinate system.

4.7.4 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)

4.7.5 Other parameter

1,Set sub-panel type

3,use control switch

4,Have auto lubricate(0 yes/1 no)

5,Auto lubricate time(0.01s)

6,Auto lubricate stop time(0.01s)

7,Door switch checking(0 no,1 yes)

8,Door switch(0 open,1 close)

9,bit paramter

10,Auto count part

[1 mean Yes,0 mean No]

11,Program edit number increase

12,Inner paramter

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

14,Is availabe keys of lub&cool as runing

17,ALM1 (0 open,1 close)

18,ALM2 (0 open,1 close)

19,ALM3 (0 open,1 close)

20,Chuck control signal(0 single,1 double M10/M71)

22,Outside chuck control(0 no,1 yes M16)

24,M10M11 short signal time(s)

- 26,Emerge Stop(0 open,1 close)
- 27,Emerge Stop2(0 open,1 close)
- 28,Run status outputM(0 invalid,1 valid M69 run M65 stop)
- 29,Alarm status output M67(0 invalid,1 valid)
- 30,Set language(1 表中文, 0 mean English)
- 31,Is enable PLC program
- 32,Is enable High 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,Lock Manual rampit func key [8 Yes]
- 39,Special paramter
- 40,Special paramter
- 41,Bake current paramter
- 42,Resume original paramter
- 200, X axis feedback alarm error(Pulse)[>1]
- 201, Y axis feedback alarm error(Pulse)[>1]
- 202, Z axis feedback alarm error(Pulse)[>1]
- 203, A axis feedback alarm error(Pulse)[>1]
- 205, X axis stop feedback alarm error(pulse)[>1]
- 206, Y axis stop feedback alarm error(pulse)[>1]
- 207, Z axis stop feedback alarm error(pulse)[>1]
- 208, A axis stop feedback alarm error(pulse)[>1]
- 210, X-axis's electron gear numerator[auto count: L screw(um)M encoder pulse]
- 211, Y-axis's electron gear numerator[auto count: L screw(um)M encoder pulse]
- 212, Z-axis's electron gear numerator[auto count: L screw(um)M encoder pulse]
- 213, A-axis's electron gear numerator[auto count: L screw(um)M encoder pulse]
- 215, X-axis's electron gear denominator[auto count: L screw(um)M encoder pulse]
- 216, Y-axis's electron gear denominator[auto count: L screw(um)M encoder pulse]
- 217, Z-axis's electron gear denominator[auto count: L screw(um)M encoder pulse]
- 218, A-axis's electron gear denominator[auto count: L screw(um)M encoder pulse]
- 300,absolute encoder servo motor[X-D2,Z-D4,0 mean No, 1 mean Yes.]
- 301,absolute encoder adress of lower 16bits muti-turn data
- 302,absolute encoder address of higher 16bits one-revolution data
- 303, absolute encoder address of lower 16bits one-revolution data
- 304, X-axis one resolution pulse
- 305, Y-axis one resolution pulse
- 306, Z-axis one resolution pulse
- 307, A-axis one resolution pulse
- 309, X-axis one-revolution coordinate(nm)
- 310, Y-axis one-revolution coordinate(nm)
- 311, Z-axis one-revolution coordinate(nm)
- 312, A-axis one-revolution coordinate(nm)

- 314, X-axis muti-turn coordinate offset
- 315, Y-axis muti-turn coordinate offset
- 316, Z-axis muti-turn coordinate offset
- 317, A-axis muti-turn coordinate offset
- 601,Make current to Step Motor Parameter
- 602,Make current to Step Servo Paramete

Introduction:

- 1,Set sub-panel type : [1 hand hold,0 panel]

It is the type of handwheel,1 means hand held type,0 means panel type.

Attention:when the parameter is 1(P1=1),CN11(handwheel port) couldn't be used to alter axis,so P1,P2 only set as 0.A/X/Y/Z to select axis,X1/X10/X100/OFF to select grade.

- 3,use control switch : [1 Yes, 0 No]

It is for if the intervention switch of panel is valid.1 means valid,0 mean invalid.

- 4,Have auto lubricate(0 Yes/ 1 No)

It is for whether the auto lubricate is valid.1 means valid,0 mean invalid.

Attention:auto lubricate is decided by working time.

- 5,Auto lubricate time(Unit: 0.01s)

It is the holding time of auto lubricate,also time that M32 is valid.unit:0.01s.

- 6,Auto lubricate stop time(Unit: s)

It is the interval that lubricate every time,also the interval that twice M32 is valid.

- 7,Door switch checking M12(0 no,1 yes)

It is for whether the system check the signal of safe-door.0 means there isn't safe-door,1 mean there is safe-door and check it.

Attention:for the check of safe-door,it is realized by M12.

When set check and M12 is valid,system could work in manual,but it cannot in Auto.

- 8,Door switch(Model :0 open,1 close)

Type of safe-door.0 means NO(normal open)type,1 means NC(normal close) type.

- 9,bit parameter

It is bit parameter,each bit have different functions,detailes as following:

D0: Null; the default is 1,it cannot be altered.

D1: "1"clear part number when booting;0 mean keep part no. unchange.

D2: "1"Automatic insert space before letter when edit program; 0 mean not.

D3: Null;the default is 0,it cannot be altered.

D4: Null;the default is 0,it cannot be altered.

D5: "1"Do not stop SP and coolant when pressing "Restet";0 means yes.

D6: "1"G00 speed is set by self of each axis,non-linear , little coordinate will reach firstly; "0" means linkage ,reach at same time. Default is 1.

D7: Null; the default is 0,it cannot be altered.

D8: "1"Save the state of SP chuck(M10/M11) when power off;it will recover this state will rebooting. "0"means system will be at state of clamp tool when rebooting.

D9: Null; the default is 1,it cannot be altered.

D10: "1" Program edit automatic compositor Line;

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

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

D13: “1”Shield go home function;

D14: “1”Shield “run” key;

D15: “1” Tool redeem display relative,“0”display absolute value;

Attention:some bits of this bit parameter cannot be altered , otherwise maybe system will work abnormal.

10,Auto count part : [1 mean Yes,0 mean No]

It is for whether count workpiece automatically,1 means count,0 meas not.

11,Program edit number increase

It is incremetal of exchange line when edit program.

12,Inner parameter

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

It is for interlock between running spindle and chuck.1 means interlock,0 means no.

14,Is available keys of coolant as running(0 mean no)

It is for whether the coolant is valid in Auto.when P14=0,it is invalid.1 means valid.

17,servo ALM (0 open,1 close)

It is the type that system check ALM of servo drive(Pin12 in CN5),1:NC, 0: NO.

18,SP ALM1 (0 open,1 close)

It is the type that system check ALM1 of Spindle(Pin5 in CN3),1: NC,0 : NO.

19,Machine ALM2 (0 open,1 close)

It is the type that system check ALM2 of machine(Pin2 in CN10),1:NC,0: NO.

20,Chuck control signal(0 single,1 doubleM10/M71)

It is for set the chuck control signal is single or double signal.

When set as 0,only use M10 to control chuck.when M10 is valid,tighten chuck, otherwise invalid ,loose chuck.

When set as 1,use M10&M71 to control chuck.when M10 is valid and M71 is invalid,tighten tool,otherwise when M10 is invalid and M71 is valid,loose tool.(M10 output M10,M11 output M71)

Check if it is in place,P40_D1 in Axis parameter,1 means check,0 means no.

If set as check,M10 is in place,system will check automatically if M14 is valid ,when M14 is valid,do next step. Loose chuck ,M11, is in place,and system also check if M12 is valid automatically,when M12 is valid,do next step.

22,Outside chuck control(0 no,1 yes)

It is for if use external switch(button) to control tighten/loose tool,it is reciprocating signal,one time is valid,tighten tool,next time is invalid,loose tool.when set 0 means without external switch. When with external switch,the signal is M16.

23,Outside finial control(0 no,1 yes)

24,M10M11 short signal time(Unit: s)

It is for stay time when output short signal,M10,M71. 0 means they are long signal.

26,Emerge Stop(0 open,1 close)

It is the type of emerge stop1-switch in operation panel.for safe,advice to set 1-NC.

27,Emerge Stop2(0 open,1 close)

It is the type of emerge stop2-switch in handwheel or external switch.(Pin5 in CN11).for safe,advice to set 1-NC.

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

It is set for if output the running signal,0 means no output running signal;1 means output running signal,M69(CN10-PIN21). Output Stop signal,M65(CN10-PIN20).

Attention: These signals can be used as show condition of machine.

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

It is set for if output the alarm signal,0 means no output alarm signal output; 1 means output alarm signal,M67(CN10-PIN8)

Attention: The signals can be used as machine-protection or show condition of machine.

30,Set language(1 表中文, 0 mean English)

31,Is enable I/O PLC program

It is for set running condition of system inner PLC,1 means ON,0 means OFF.

Attention:it is usually used as adjusting parameter,it must be ON when actual use, otherwise system will abnormal.

32,Is enable High speed I/O PLC program

It is for if start speed of system's inner PLC,1 means ON-High Speed,0 means OFF-High Speed.

Attention:it is usually used as adjusting parameter,it must be ON-High Speed when actual use, otherwise system will abnormal.

35,soft-limit without home as manual :[1 Yes,0 No]

It is set for if soft-limit is valid when not homing,1 means valid,0 mean invalid.

Attention:the set of this parameter is related operation habit of user.

36,Set system time :[year-month-day-hour-minute]

It is for set time and date of system,after set,system will take this setting time as basic, according to inner timer count time and shows in displayer.

37,Velocity of RS232

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

Attention:The bigger the value is,the unstabler it is.The bits of both ends must keep same.

38,Lock Manual rampit func key :[8 Yes]

It is for type of Rapid key's using. 8 means Switch cycle condition of rapid key.

39,Special parameter

40,Special parameter

41,Bake current parameter

It is defined to ex-factory value set.it is used for bake current parameter after test system well.

Attention: select this paramter,press "Enter" twice,finish bake.

42,Resume original parameter

200, X axis feedback alarm error(Pulse)[>1]

If follow error bigger than this parameter,system will alarm.(the parameter is effective when it >1). #200/#201/#202/#203 stand for X/Y/Z/A-axis. Press "G"key in Diagonal to clear alarm and command position & feedback position after alarm.

205, X axis stop feedback alarm error(pulse)[>1]

If follow error bigger than this parameter when axis stop,system will alarm.(the parameter is valid when it >1). #205/#206/#207/#208 stand for X/Y/Z/A-axis. Press "G"key in Diagonal to clear alarm and command & feedback's position after alarm.

210, X-axis's electron gear numerator[auto count: L screw(um)M encoder pulse]

The parameter is numerator of X-axis electron gear, and also input screw lead and encoder lines, for example, when screw lead is 6mm, encoder's resolution is 2500PPR, input : L6000M2500. #210/#211/#212/#213 stand for X/Y/Z/A-axis.

Special Attention:

1) when input L***M***, it needs to set numerator and denominator of electron gear of each axis.

2) when series port read data of absolute encoder, it must input in this way.

215, X-axis's electron gear denominator[auto count: L screw(um)M encoder pulse]

The parameter is denominator of X-axis electron gear, and also input screw lead and encoder lines, for example, when screw lead is 6mm, encoder's resolution is 2500PPR, input : L6000M2500. #210/#211/#212/#213 stand for X/Y/Z/A-axis.

Special attention:

1) when input L***M***, it needs to set numerator and denominator of electron gear of each axis.

2) when series port read data of absolute encoder, it must input in this way.

300, absolute encoder servo motor[X-D2,Z-D4, 0 mean No, 1 mean Yes.]

The parameter is set for if the axis is with motor with absolute type encoder. #301, absolute encoder address of lower 16bits multi-turn data(Example: set to 92); #302, absolute encoder address of higher 16bits one-revolution data(Example: set to 91); #303, absolute encoder address of lower 16bits one-revolution data.

Special Attention: System is connected with driver with RS485 by CN13 port, communication protocol is as follow:

1) Driver's Satation Address of RS485(P56): 1 -X Axis; 2 -Y Axis; 3 -Z Axis; 4 -A Axis .

2) Baud Rate of RS485: 19200; Data is 8 bits; Stop bit is 1 bit; Format is RTU; Parity bit is odd. {1start+8data+1stop+odd}

304, X-axis one resolution pulse

The parameter is pulses of absolute encoder of X-axis. The pulses of 17bits absolute encoder is 131072. #304/#305/#306/#307 stand for X/Y/Z/A-axis.

309, X-axis one-revolution coordinate(Unit: nm)

The parameter is distance of absolute encoder of X-axis. Minus means count's direction is negative. Example: lead screw of X-axis is 4mm, so P309=4000000 or -4000000. #309/#310/#311/#312 stands for X/Y/Z/A-axis.

314, X-axis multi-turn coordinate offset

The parameter is multi-turns offset of X-axis, in order to solve when the values of multi-turns or coordinate is too big, appear overflow etc. problems. Input 'E' clear the value to 0. #314/#315/#316/#317 stand for X/Y/Z/A-axis.

319, Servo driver current/speed/power(284/283/435)16bits address

601. One key to define step.

Press "Enter" key to fit all parameters for step system.

602. One key to define servo.

Press "Enter" key to fit all parameters for step system.

Attention: the default value is set as servo system.

4.7.6 Coordinate system

The parameter has the function of multiple coordinate system, includes 6 workpiece coordinate system and a machine 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 milling 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 milling 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]
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]
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]
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]
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]
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]

Introduction:

4.6.6.1 How to set up the workpiece coordinate system?

We set up the workpiece coordinate in the condition of Manual, steps are following:

- a). Press “MDI” key, select corresponding workpiece coordinate system (G54-G59), **Example, select G55 coordinate, input G55, Press ‘Enter’, ‘Start’, selected G54 Coordinate.**
- b). Move machine to suitable position that easy to measure in manual, and measured the related coordinate value between this point (zero point in the workpiece) to Home of G53 coordinate system (also machine coordinate system).
- c). Press “Setup”, press “X” key and ‘Enter’, ‘insert the measured value’, and ‘Enter’.
- d). Press “Setup”, press “Y” key and ‘Enter’, ‘insert the measured value’, and ‘Enter’.
- e). Press “Setup”, press “Z” key and ‘Enter’, ‘insert the measured value’, and ‘Enter’.

Done well now. Enter different workpiece coordinate system, it will show the corresponding value, which also is offset value between workpiece coordinate system and machine coordinate system (G53).

4.6.6.2 How to adjust the offset value after set well?

If set up workpiece coordinate system well,when it needs to adjust the offset value,it could be set by enter the workpiece coordinate parameter,steps is as follow:

In the coordinate parameter screen,selected the parameter,press “Enter”,and pop up dialog,input the offset value(also Increments,example:offset 10mm in negative direction,also input -10),press “Enter”.It is okay.

Explanation:1.when the parameter is altered well,the coordinate main screen will refresh the corresponding coordinate value soon.

2.brackets in these parameters,it means the sum ,which is offset or adjust every time.It is suitable to look for the offset every time.

4.7.7 Password

In this system in order to prevent from the parameter modification in accident, make sure the milling working, the system adopts the parameter setting of classify the authority. Divided into "CNC factory" and "Milling factory", "User factory" three level authority. The "CNC factory" set for the function of the system,belong to internal parameter;"Milling factory" set equipment configuration of milling 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,"Milling 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 milling manufacturer
3. Whether to turn on the password protection of user manufacturer
4. Modify the password of CNC manufacturer
5. Modify the password of milling manufacturer
6. Modify the password of user manufacturer
7. The using time of system
8. Software version

4.8 Set parameter of tool redeem

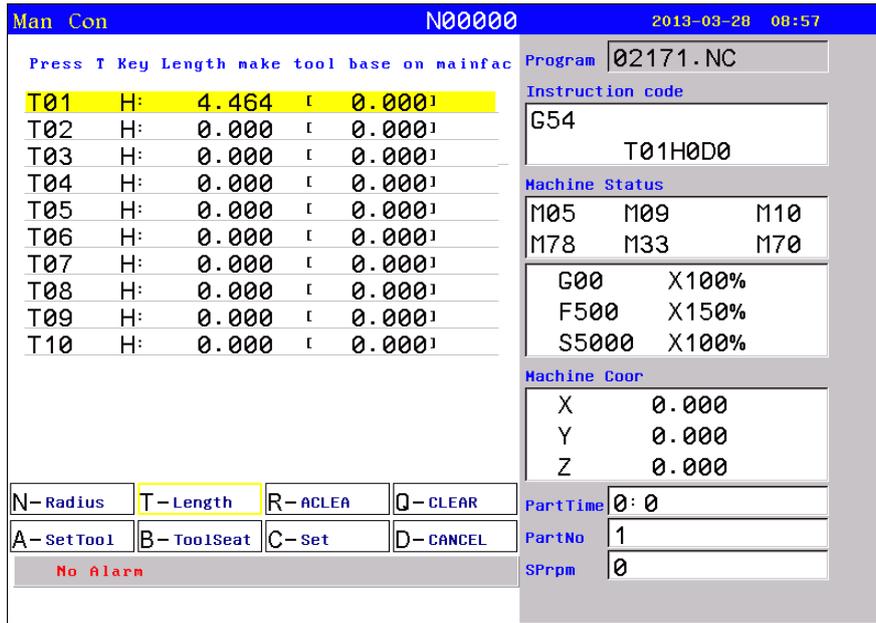
Press “Redeem” to enter interface of redeem in any interface, including “Radius compensation” “Length” “Clear all value” “Clear current value” “Set tool” “Posit tool” “Set”, total 7 functions, correspond to press“N”、“T”、“R”、“Q”、“A”、“B”、“C” to enter corresponding interface, press “Esc” to back the primary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

4.8.1 Radius Compensation

Press “N” to enter radius compensation interface in Redeem. 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.



4.8.2 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 milling 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.

4.8.3 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.

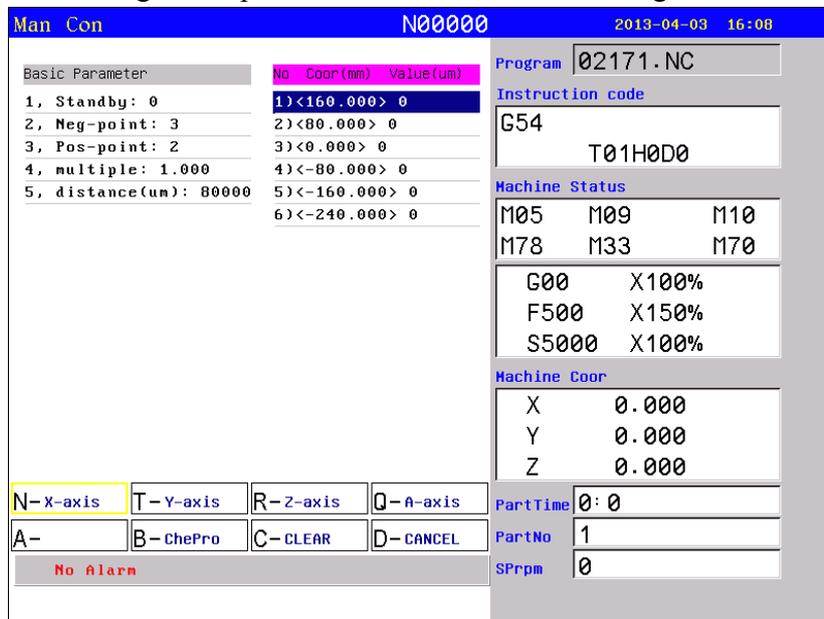
4.8.4 Set quantity

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.

4.9 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 Milling. The system adopts storage mode of screw compensation: Making the Milling’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

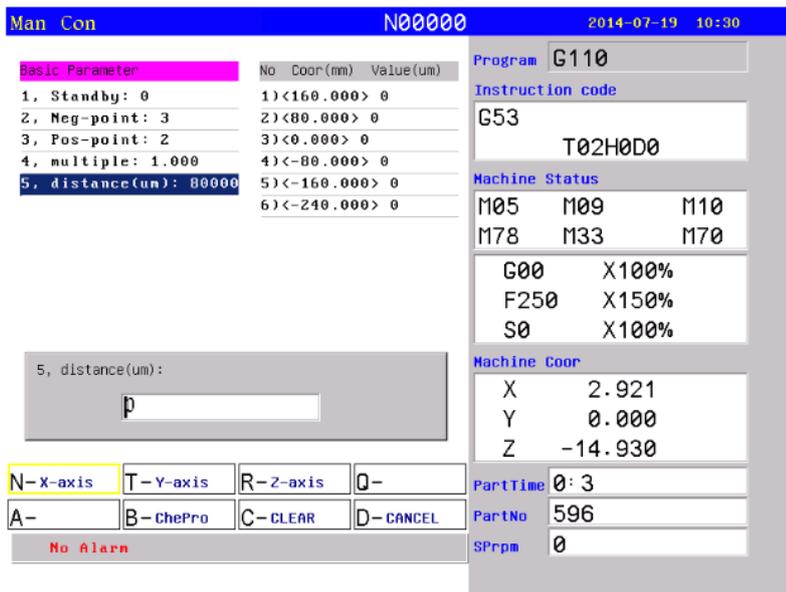
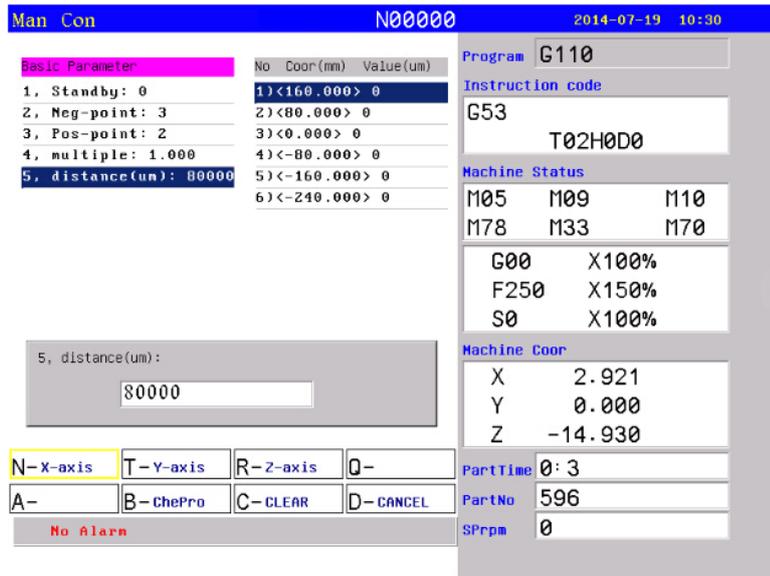
Screw compensation by the axis as the unit to set storage, set X Y Z A axis separately, by pressing “N” “T” “R” “Q” to switch; Every axis of screw compensation interface has tow areas(basic parameter and set the compensation), by pressing “→ ←”to move the cursor to realize.

Basic parameter:

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

Set compensation value:

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

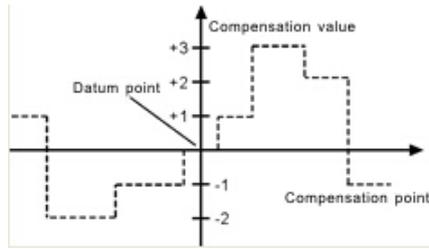


Test program generation automatically

Automatical generate a program of laser interferometer to check the screw compensation. Enter the screw pitch interface and set basic parameter, press checking program to popup a dialog box and press “Enter” to generate corresponding checking program of screw compensation.

The number of compensation points can be set freely, the maximum number of each axis is 300. The basic parameter of every axis’ error compensation of screw pitch includes as follows:

1. Reserve.
2. Backward checking points.
3. Forward checking points.
4. Compensation override.
5. The spacing of compensation points (um).



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

The compensation points are uniform, set the spacing into each axis.

For example:

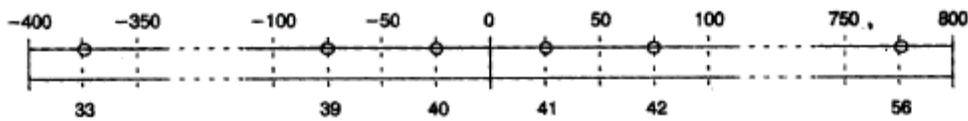
Example 1:Linear axis: when length of travel is -400mm~+800mm, spacing of points is 50mm,reference point compensation isNo.40, it can figure out that Compensation point of farthest end in negative direction is:

$$\text{Machine negative travel/point interval} +1 = 40 - 400/50 + 1 = 33.$$

Compensation point of farthest end in positive direction is:

$$\text{Machine positive travel/point interval} +1 = 40 + 800/50 = 56.$$

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

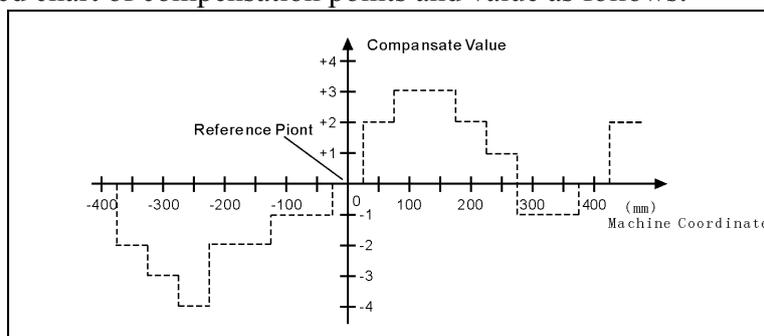
Compensation point interval: 50000

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	50	51	52	53	54	55	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:



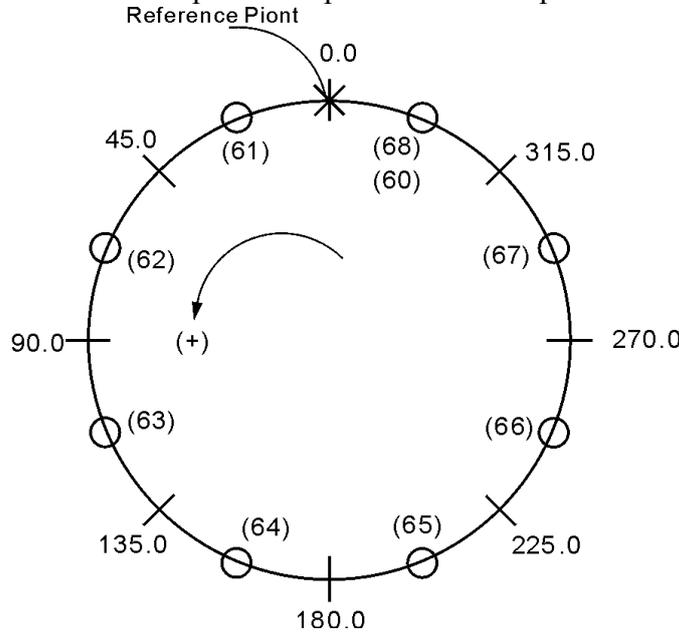
Example 2:rotor axis: when movement per revolution is 360°, interval of points

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:

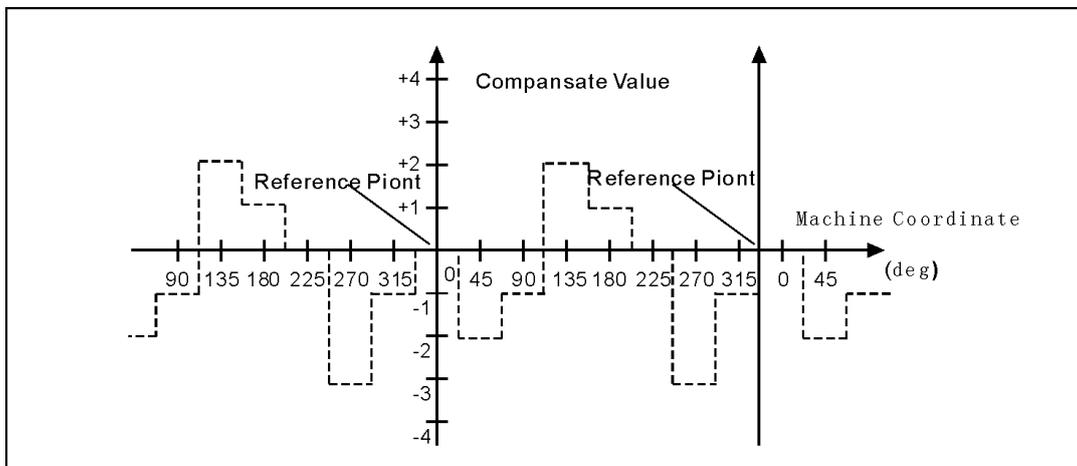


Note: input value in small circle. If the total amount from 61 to 68 doesn't equal 0, accumulated pitch error per revolution will deviate, so same value shall be put in 60 and 68.

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:



4.10 Diagnosis

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.

4.11 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”.

4.11.1 Editing

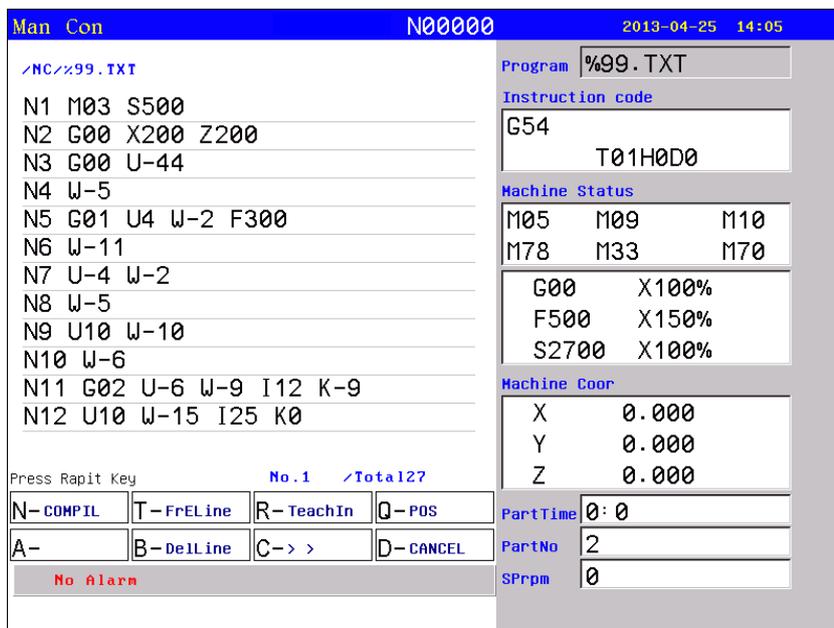
Select “New file/search” to popup a dialog box to import the name of program, if the name is existent, the quondam program is called up; If the name is inexistent, the system will build a new file.

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 milling 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 first 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).

D, “”+“Q”: located to the specified line.

E, “”+“A”: no operation.

F, “”+“B”: delete the current line.

G, “”+“C”: the first 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.

4.11.2 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.

4.11.3 Delete

Press “↑ ↓” in program main interface to select program which need to delete and press “Del” to delete the program.

Attention: The operation of delete need to be careful, it can't be recovery after deleting.

4.11.4 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.

4.11.5 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.

4.11.6 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.

4.11.7 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.

4.11.8 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.

4.11.9 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.

4.11.10 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, wait 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.

Chapter 5 System installation and connection

5.1 System installation and connection

At first, users should check whether the hardware is complete, unwounded and compatible, such as: CNC system, spindle 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.

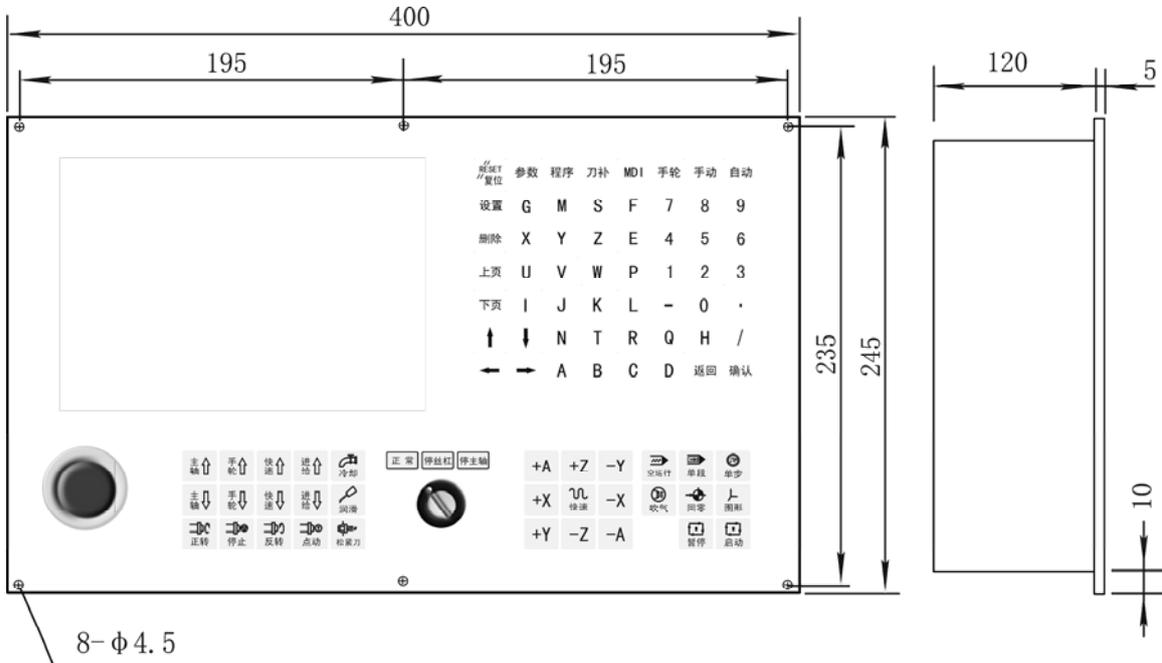
In installation of CNC system, panel should avoid hurting by hard and sharp materials. If the painting of other part of machine is needed, please take off CNC system to keep it clean.

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

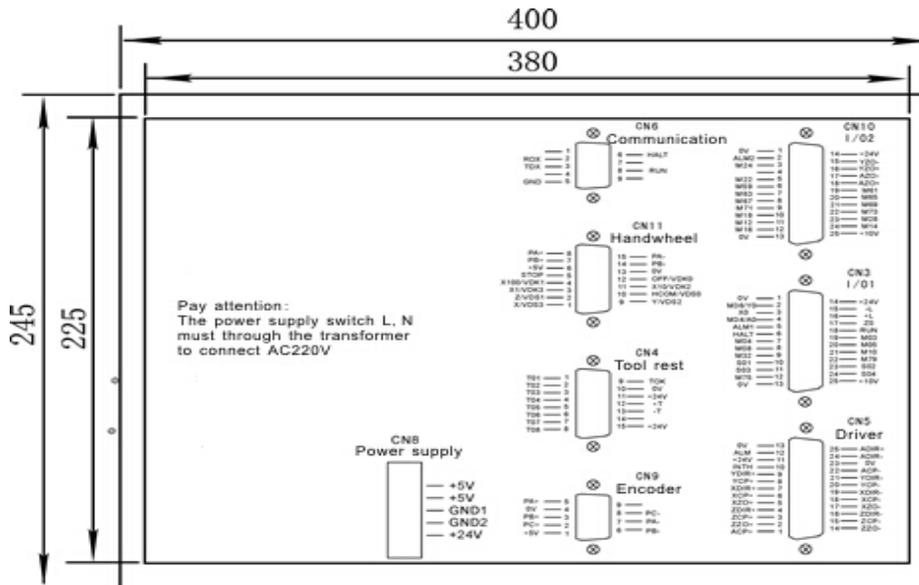
Pay attention:

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.

5.2 System Installation Dimension (400x245x120)



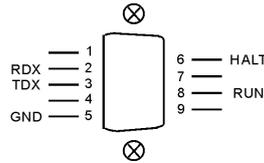
5.3 System Rear View



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

5.4 Interface connection graph

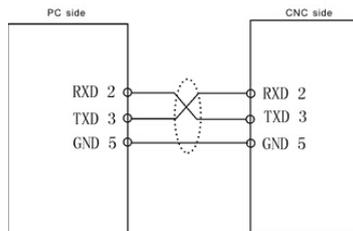
5.4.1 Communication socket CN6 connect to the hole socket DB9



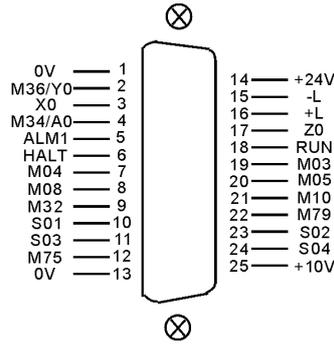
CN6 communication signal with the hole socket DB9				
Signal	Pin No.	I/O	Function	Effective voltage
0V	5	OUT	The ground of signal	0V
RXD	2	IN	The received data signal	
TXD	3	OUT	The transmission of data signal	
RUN	8	IN	Spindle of running	0V
HALT	6	IN	Spindle of stop	0V

- 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:

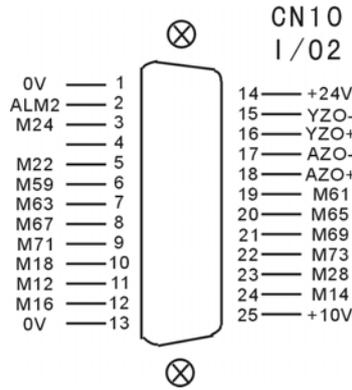


Pay attention: When PC programming, the files should be text files.
5.4.2 The milling machine CN3 connect to the hole socket DB25



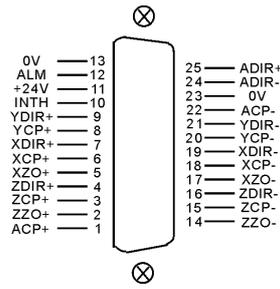
CN3 DB25(hole) I/O1 machine signal				
signal	pin	I/O	function	Availability
0V	1	OUT	0V	0V
+24V	14	OUT	+24V	+24V
M36/Y0	2	IN	M36/Y0	0V
X0	3	IN	X axis Zero	0V
Z0	17	IN	Z axis Zero	0V
-L	15	IN	Positive limit	0V
+L	16	IN	Negative limit	0V
M34/A0	4	IN	M34/A0	0V
ALM1	5	IN	Transducer alarm1	0V
HALT	6	IN	Pause	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	Analgo Voltage of First Spindle	0~10V
0V	13	OUT	Ground of converting	0V

5.4.3 Milling machine CN10 connect to hole socket DB25



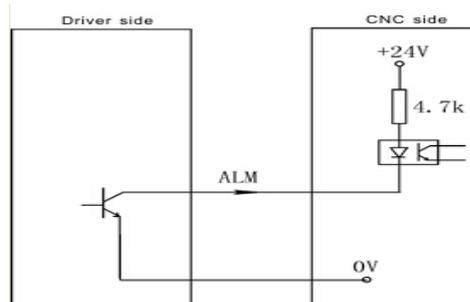
CN10 DB25 I/O2 milling signal				
Spindle	Pin	I/O	Function	Effective voltage
0V	1	OUT	Ground of the power supply	0V
+24V	14	OUT	24V power supply	+24V
ALM2	2	IN	Milling alarm2	0V
M24	3	IN	User-define Input7	0V
M22	5	IN	Stop M01 input	0V
M59	6	OUT	Huff output	0V
M61	19	OUT	User-defined output1	0V
M63	7	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	Analog Voltage of Second Spindle	0~10V
0V	13	OUT	Ground of frequency conversion	0V

5.4.4 Driver signal CN5 socket connect needle socket DB25



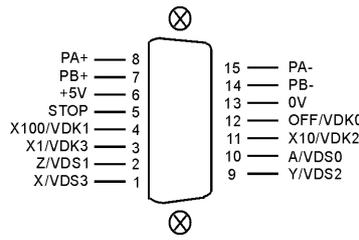
CN5 DB25(pin) servo drive signal				
signal	pin	I/O	Function	Availability
XCP+	6	OUT	X pulse signal +	5V
XCP-	18	OUT	X pulse signal -	
XDIR+	7	OUT	X direction signal +	5V
XDIR-	19	OUT	X direction signal -	
YCP+	8	OUT	Y pulse signal +	5V
YCP-	20	OUT	Y pulse signal -	
YDIR+	9	OUT	Y direction signal +	5V
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.Signal line must adopt shielded twisted pair cable,the length can't over 20m.
 2. Whether the alarm signal ALM is always open or close is set by P17 in Other parameter.



Servo alarm signal:

5.4.5 External switch, electrical handwheel (socket CN11,DB15)



CN11 DB15 I/O signal				
Signal	Pin	I/O	Function	Effective
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/1st rate of feeding Axis	0V
X100/VDK1	4	IN	*100 gears signal/2nd rate of feeding Axis	0V
X10/VDK2	11	IN	*10 gears signal/3rd rate of feeding axis	0V
X1/VDK3	3	IN	*1 gear signal/4th rate of feeding axis	0V
A/VDS0/HALT	10	IN	A-axis Selection/1st rate of SP/Stop signal	0V
Z/VDS1	2	IN	Z-axis Selection/2nd rate of SP	0V
Y/VDS2/RUN	9	IN	Y-axis Selection/3rd rate of SP/Running signal	0V
X/VDS3	1	IN	X-axis Selection/4th rate of SP	0V

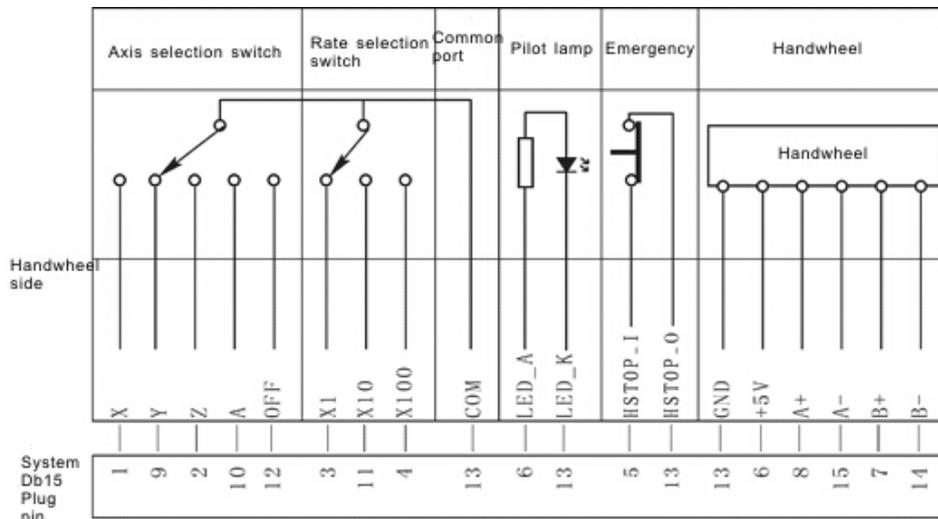
Attention: Inner power supply are all +5V of all signal, don't access voltage higher than 5V.

5.4.5.1 Usage for electrical handwheel(Manual pulse generator)

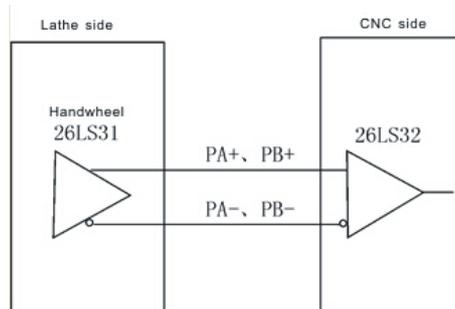
You can connect standard external handwheel when P1=1 in Other parameter and can not use band switch to adjust spindle, feed and external stop running button, so P1,P2 in Axis parameter only set to be "0"

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

Handwheel contact diagrammatic as:



- 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:



5.4.5.2 Usage for terminal band switch behind sub panel

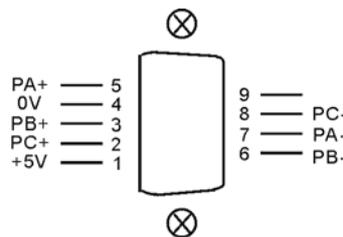
Could use the band switch when P1=P2=1 in "Axis parameter" ,P1 in "Other parameter" should be set to 0, A、Z、Y、X、OFF、X100、X10 and X1 are corresponding to select the feeding axis and the gear of the spindle.

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.

5.4.5.3 Using for external emergency button

STOP signal is the input signal of external emergency button, P27 in Other parameter controls the emergency button is always open or close.

5.4.6 Spindle encoder socket CN9 connect to needle socket DB9

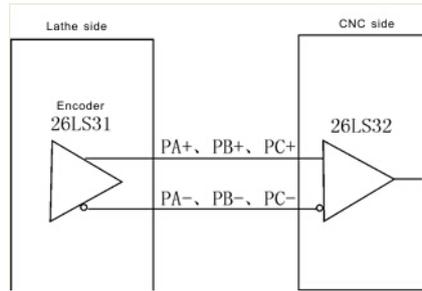


CN9 DB9(pin) spindle encoder				
signal	pin	I/O	function	availability
0V	4	OUT	0V	0V
+5V	1	OUT	+5V	+5V
PA+	5	IN	+A signal	5V
PA-	7	IN	-A signal	
PB+	3	IN	+B signal	5V
PB-	6	IN	-B signal	
PC+	2	IN	+Z signal	5V
PC-	8	IN	-Z signal	

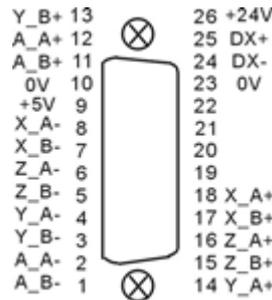
Attention:1. The output signal of encoder adopt the output way is line output, the power supply is +5V.

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

The input signal of encoder PA PB PC:



5.4.7 CN13 Position Feedback of coordinate axis (DB26 Optional Function)



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 A axis	5V
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	

P200-P220 in Other parameter set the function. Press "G" in Diagnosis to clear the instruction position and feedback position and clear the deviation alarm after alarm.

Attention: 1, The encoder or the grating output signal with long-line output mode (also RS422), the power supply is +5V.

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

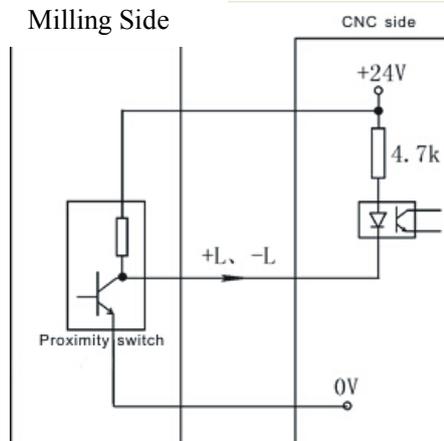
5.4.8 Trimming method for the system matches the absolute bus type of motor

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 P56 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 milling machine:
 - 1) manual move coordinate and remember the milling 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 P307,P310,P311,P312 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 Milling: manually move to the position which is nearby the zero point , set the P314,P315,P316,P317,P318 in Other parameter(Corresponds to X/Y/Z/A/B axis), press "Enter" and press "E" key to clear the milling coordinate;
9. Manual move every axis to the limit position of milling to set the soft limit value in axis parameter;
10. Set the P41 in Other parameter and bake up the current system parameter

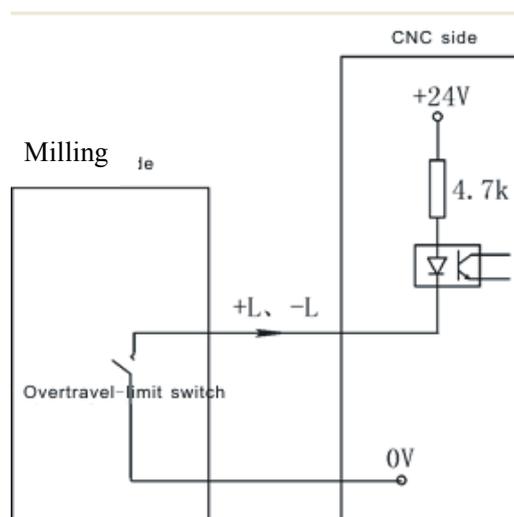
5.5 The Installation of Milling Machine

5.5.1 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:

P27 is for setting the type of hard limit switch for +L positive, 0 means always open, 1 means always close.

P28 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.

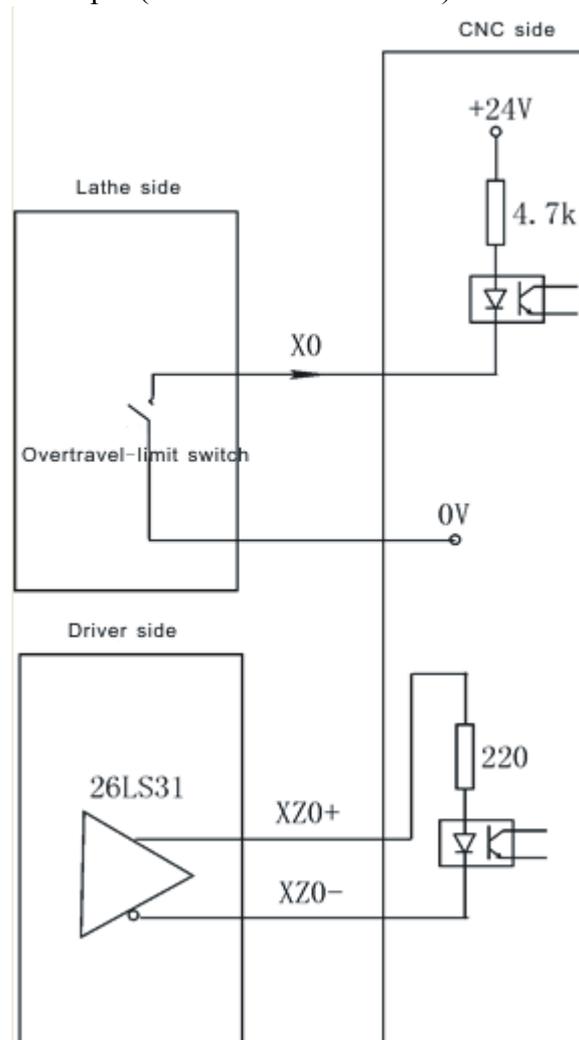
2. Could select our electrical appliance plate of Milling.

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 ladder and program of PLC.

5.5.2 Datum point of Machine (Reference point or Home)

Take X axis as example (the same as Y Z A axis)



Backing to the datum point

When setting floating datum point is invalid, backing to the datum point need to check signal of proximity switch and motor's Z pulse. P33 in Axis parameter need to be set as "00000000".

P38 in Axis parameter set the function of Homing:

There are four ways to set backing to the datum point when turn on the system:

When P38= 1: **Needn't**; system doesn't prompt and no limits when turn on the system every time.

When P38= 0: **Prompting way**; system popup a dialog box to prompt user to process operation of backing to the datum point, it has no limits.

When P38= 8: **Compulsory mode**; 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.

When P38= 9: **Super compulsory mode**; System moves feeding axis each time, it will popup a dialog box for backing to datum point, it will prompt "Feeding axis doesn't back to the datum point" and don'ting act if don't backing to the datum point.

P39 in Axis parameter set the mode of checking signal of Homing

When P39= 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 P39= 1: After hitting the datum point switch when backing to the datum point, run reverse to check the switch off.

When P39= 2: After hitting the datum point switch when backing to datum point, continue running to check the switch off and check Z pulse signal of motor's encoder.

When P39= 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 get promise.

P40 in Axis parameter set the direction of Homing

Bit parameter ,each axis is to set separately. D2 controls the homing direction of X axis, D3 controls the homing direction of Y axis,D4 controls the homing direction of Z axis, 1 means negative direction, 0 means positive direction;

P41 in Axis parameter set the type of homing switch:

Bit parameter, each axis is to set separately. D0 -- X axis,D1 -- Y axis,D2 -- Z axis, 1 means NC(normal close), 0 means NO(normal open).

P42~P45 in Axis parameter is set to check the processing length of motor's 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(P42) Y(P43) Z(P44) A(P45) 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.

P34/P36/P38/P40 in Speed parameter set the speed of hitting to homing switch:

The speed of reaching to datum point switch when X(P34) Y(P36) Z(P38) A(P40) axis backing to positive datum point. Unit: mm/min. Range: < G00 speed.

P35/P37/P39/P41 in Speed parameter set the speed of checking motor's Z pulse when homing:

When the X (P35) Y (P37) Z (P39) A (P41) axis backing to the datum point, the speed of checking the motor's Z pulse signal after disengaging the homing switch . Range: 20-500.

Pay attention: The parameter value influence with the accuracy of backing to the datum point, the smaller the value the higher the accuracy. This value has been set which not to change anymore, otherwise it will affect the reference point too.

P46~P49 in Axis parameter set the offset of finishing back to the datum point:

Used to set the offset distance after X(P46) Y(P47) Z(P48) A(P49) axis homing and checking the zero pulse signal of servo motor. Unit: 0.01mm. Range:-99999~+99999.

The parameter value is related to the install position of datum point and coordinate.

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

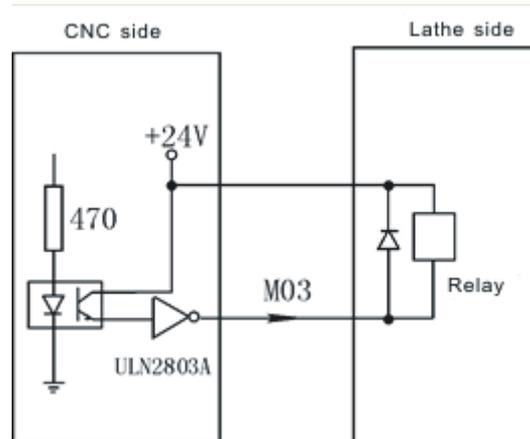
1. The speed reduce switch is also can use NPN switch.

2. Must consider the length of speed reduce when installing the speed reduce switch.Must be not over 25mm(<=25mm).

5.5.3 Controlling signal of switch value:

Take M03 as example (the same as M04 M05 M08 M10 M32 M79 M75 M59 M61 M63 M65 M67 M69 M71 M73 S01~S04)

M03 signal control:



As the picture shows, it will form a circuit with 24V when system outputting M03, The intermediate relay is working and a group of normal open contact form a circuit with spindle rotation AC contactor.

It is effective when all of output signal is low level(also 0V).

Pay attention:

1. When the relays and others load, must connected with the diode to absorb the reverse current so as not to damage the system, if use the electromagnetic contactor, then plus resistive and capacitive spark circuit.

2. Chip ULN2803A corresponds output ports:

- 1). U28 corresponds to M59 M61 M63 M65 M67 M69 M71 M73
- 2). U29 corresponds to M03 M04 M05 M08 M10 M79 M32 M75
- 3). U30 corresponds to +T -T S01 S02 S03 S04 LRUN INTH

3. When user-defined signal M71/M70 is used for input signal of spindle chuck, it can't be the other usages. P20 in Other parameter to set.

4. When user-defined signal M65 M67 M69 is used for onput signal of machine's status, it can't be the other usages. M65:output the status of program stop,M69:output the status of program run, M67: output when system alarm. P28,P29 in Other parameter to set.

5.5.3.1 Spindle Control of System (M03/M04/M05)

System output controlling signal of two analog spindles (First spindle, second spindle) , relative parameter as follows:

Axis parameter

P9: Set the braking time of spindle, also hold time of output, the less the faster.

P10: Set the braking signal is long signal 1 or short signal 0.

P43: Whether turn on the spindle or not when spindle shifting [1: on, 0: off]

P44: The turning speed of motor when spindle shifting (1/100rpm)

P45: The turning direction when spindle shifting (0: positive, 1: negative)

P46: The stopping time when spindle shifting (10ms)

P47: Turning time of low speed when spindle shifting (10ms)

P48: Stopping delay time of spindle (10ms)

Speed parameter

P9: To set the speed of spindle setting value at manual condition. Unit: rpm.

P42: To set the highest speed of spindle, that's the turning speed of corresponding 10V instruction voltage.

P43: 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.

P44: To set the highest speed of spindle (Third gear), that's the turning speed of corresponding 10V instruction voltage. Unit: r/min

P45: To set the highest speed of spindle (Fourth gear), that's the turning speed of corresponding 10V instruction voltage. Unit: r/min

P46: To set the highest speed of second spindle, that's the turning speed of corresponding 10V instruction voltage. Unit: r/min

5.5.3.2 System lubrication control (M32/M33)

P4 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:

P13: 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 milling and user's service condition, but consider for safe, suggest setting 1.

P20: 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 Milling.

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.

P22: 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.

P24: 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.

5.5.4 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: Normal open, 1: normal close.

P17: To set the type of system checking the servo alarm signal (twelfth pin of CN5 ALM), 0 means always open, 1 means always close.

P18: To set the type of system checking the spindle alarm signal of milling (fifth pin of CN3 ALM1), 0 means always open, 1 means always close.

P19: To set the type of system checking the alarm signal of milling (second pin of CN19 ALM2), 0 means always open, 1 means always close.

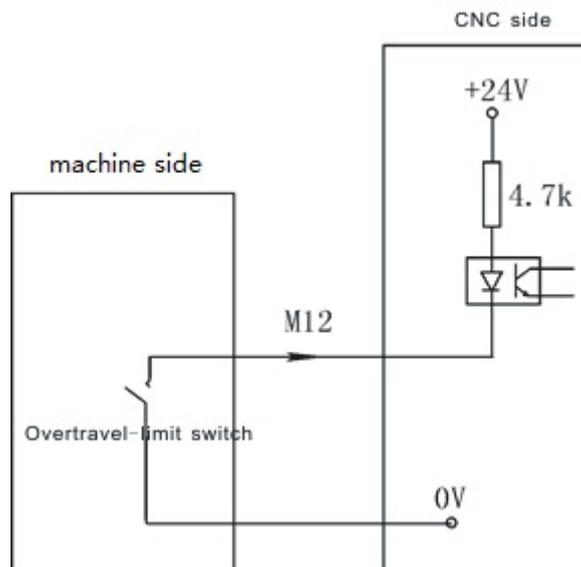
P27: To set the Emergency always open or close of system CN11, suggest setting always close for safe.

Emergency STOP: Press “Emergency” when appearing emergent accident, the milling 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 P29 in Other parameter is effective. This output signal can be used to protect the milling (Cut off power supply).

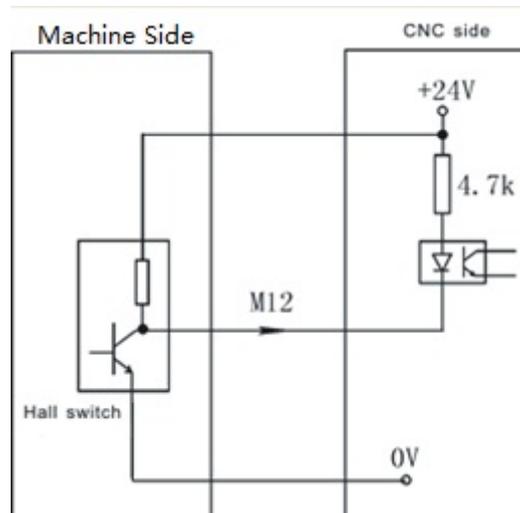
P29 in Other parameter is effective when appearing alarm, the output signal M67 is effective.

5.5.5 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:



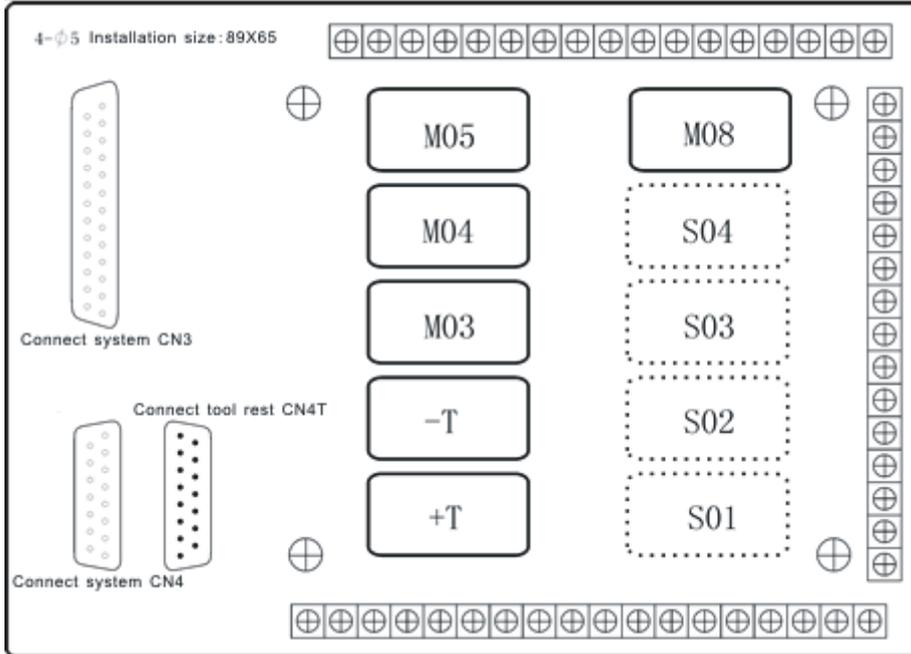
M12 switch also can use NPN as checking witch:



5.6 Electrical appliance plate of Milling

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

I/O1 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.