



980/990MDb 3-5AXIS

Open PLC and software

With MPG

CONTENTS

Chapter 1	Preface	3
Chapter 2	System technical features	4
2.1	System structure	4
2.2	System technical parameter	4
2.3	System function	4
2.4	System operation condition	5
Chapter 3	Operation explanation	7
3.1	Panel layout and switch (Two types)	7
3.2	operation interface	9
3.3	Parameter	10
3.4	Programming	23
3.5	Manual	27
3.6	Automatic	30
3.7	Tool redeem	32
Chapter 4	Programming	34
4.1	Basic concepts	34
4.2	Program instruction	36
4.3	Preparation functions	44
Chapter 5	System installation and connection	85
5.1	System installation connection	85
5.2	System installation dimension	85
5.3	System rear view	86
5.4	interface connection graph	86
Chapter 6	System' s daily maintenance and repair	96
Chapter 7	Appendix	98

Chapter 1 Preface

This CNC control system is one middle class flush type CNC control system, aiming specially at lathe and grinding machine.

Based on modern computer technology, system move control core & PLC program running technology, and stable unique real time control engine subsystem PTAI, this system ensures the stabilization of operation. The use of high performance, low power consumption industrial grade ARM microprocessor as core of hardware, large scale FPGA integrate circuit, multiple layer (4,6) printed circuit, 32MB flash memory, 8 inch real colour LCD which provides friendly man-machine dialogue interface makes this system work to its best.

Note for “caution”:

1、“caution” reminds operator must be caution in the relative operation, otherwise the operation will fail or some action can not be effected.

2、“special caution” reminds operator must be special caution in the relative operation, otherwise it may break down the machine or give rise to accident.

Special hint:

This system has function to backup parameters. After debugging machine, it can backup all parameters of machine & system and PLC documents to computer. It is convenient not only for mass debugging, but also for machine recovery to normal after changing system.

Note :

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

Chapter 2 System technical features

2.1 System constructions

32 bits high performance, low power consumption industrial grade ARM microprocessor.

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 2-3 axes, liner 2-4 axes.

pulse equivalent: X、Y、Z、A axes:0.001mm.

max speed: X、Y、Z、A:60000mm/min.

cutting speed: 1-10000mm/min.

min input unit: 0.001mm.

program size range: \pm 99999.999.

99 tools management.

program code: ISO-840 international standard.

program coordinate system definition: ISO-841.

chassis protection complies with regulation of IP43.

2.3 System function

2.3.1 Auto-diagnosis function

All around diagnosis of CPU, storer, LCD, I/O interface, 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.
multiple tool offset instruction.
program cycle, jump, call and different program ending.
multiple positioning instruction: starting point, setting fixed point, etc.
linear, circular, spiral line interpolation instruction.
program management instructions: program cycle, call, transfer and different program ending method, etc.
6 workpieces coordinates system .

2.3.4 Chinese/English menu, full screen edition

Easy operation, convenient 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 Program 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.

2.4 System operation condition

2.4.1 Power supplying

AC 220V(+10%/-15%), Frequency 50Hz \pm 2%. power: \leq 200W.

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

2.4.2 Climate condition

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

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

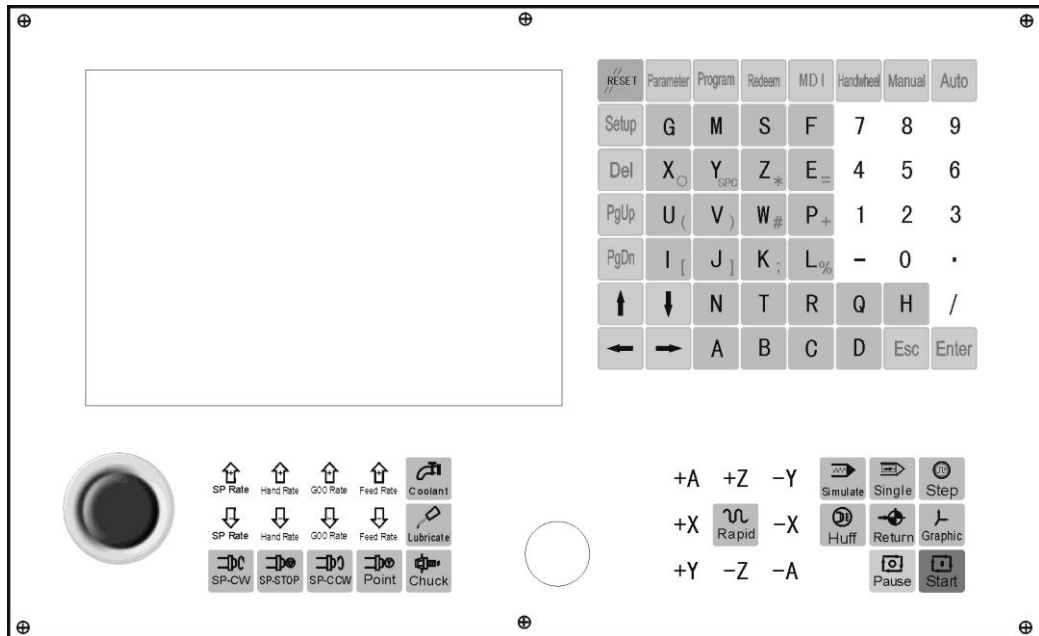
atmosphere pressure: 86-106kpa.

2.4.3 operation enviorment:


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

Chapter 3 Operation explanation

3.1 Panel layout and switch

























switch introduction:

Switch	Functions
	Emergency stop Driver and motor stop immediately, turns off the spindle, coolant, waits for the rise of button, and initializes values

buttons:

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	<p>“Esc” returning to upper level or stop a operation</p> <p>“Enter” selecting sub-menu and changing a newline</p> <p>“Del” delete program</p> <p>“program” entering program edition</p> <p>“parameter” entering parameter setting</p> <p>“manual” entering manual status</p> <p>“handwheel” for starting or stopping handwheel function</p> <p>“Setup” for confirming current tool ‘s position in machine tool coordinates system.</p> <p>“Redeem” for amending tool change errors</p> <p>“Auto” entering automatic status</p> <p>“MDI” entering MDI function</p> <p>“” selecting auto-coordinates/diagram machining</p> <p>“” for single segment or constant work</p> <p>“” for coordinates mode or diagram mode speedy simulating</p> <p>“” for manual increment or constant work</p>
-----------------	--

Control key	<p>“   ” spindle cw, ccw rotation</p> <p>“” coolant on/off</p> <p>“” for the shift between hand-driven continuous high speed and low speed.</p> <p>“” all axes return to datum point</p> <p>“” for spindle loosens tool</p> <p>“” for lubrication on/off</p> <p>“” for huff on/off</p> <p>“ ” handwheel gear selection</p> <p>“ ” adjusting spindle speed</p> <p>“ ” adjusting feed speed</p> <p>“ ” adjusting G00 speed</p>	
Feed key	+X -X +Y -Y +Z -Z +A -A	For X、Y、Z、A axes direction feed

3.2 operation interface

Whole system adopts multi-leveled menu full screen operation, user-friendly interface, providing comprehensive information. It enters into main interface when electrified:

Man Con		N0000		2013-03-28 08:56									
N106G0 G54G90X3.Y-33.25S1500M3 N108G43H1Z10.M8 N110Z2.			Program 02171.NC Instruction code G54 T01H0D0										
<table> <tr> <td><input type="radio"/> X</td> <td>0.000</td> </tr> <tr> <td><input type="radio"/> Y</td> <td>0.000</td> </tr> <tr> <td><input type="radio"/> Z</td> <td>0.000</td> </tr> <tr> <td><input type="radio"/> A</td> <td>0.000</td> </tr> </table>			<input type="radio"/> X	0.000	<input type="radio"/> Y	0.000	<input type="radio"/> Z	0.000	<input type="radio"/> A	0.000	Machine Status M05 M09 M10 M78 M33 M70 G00 X100% F500 X150% S5000 X100%		
<input type="radio"/> X	0.000												
<input type="radio"/> Y	0.000												
<input type="radio"/> Z	0.000												
<input type="radio"/> A	0.000												
No Alarm			Machine Coord X 0.000 Y 0.000 Z 0.000 PartTime 0:0 PartNo 1 SPrpm 0										

3.3 Parameters

Man Con		N00000		2013-03-28 08:57	
1,Cycle d of G7	5.000	Program	02171.NC		
2,Cycle d of G8	5.000	Instruction code	G54		
17,Running prog	0		T01H0D0		
18,Set M20 the	-1	Machine Status			
19,Set part cou	1	M05	M09	M10	
21,G01/G02/G03	0	M78	M33	M70	
22,G00 line del	0	G00	X100%		
200,system prot	1	F500	X150%		
202,ststem inne	1	S5000	X100%		
		Machine Coor			
		X	0.000		
		Y	0.000		
		Z	0.000		
N-User	T-Speed	R-Axis	Q-Tool	PartTime	0:0
A-Other	B-Coor	C-PASSWD	D-CANCEL	PartNo	1
No Alarm				SPrpm	0

In main menu, pressing “Parameter” function key, it enters para setting status, including “User”, “Speed”, “Axis”, “Tool”, “Other”, “Coor>>”, “Passwd”, seven function. Choose pressing “N、T、R、Q、A、B、C” choose

Except for special note, all data are using mm.

3.3.1 User parameter

- 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
- 19,Set part count
- 21,G01/G02/G03 line delay(ms) [>100]
- 22,G00 line delay(ms) [>100]
- 200,system protect times
[>=2minutes]
- 202,ststem inner parameter

3.3.2 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, Feed axis's manual speed(mm/min)
- 10, Spindle's manual speed(rpm)
- 11, Beginning feed speed(mm/min)
- 12, Jump speed at continuous track(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
- 19, Y acceleration
- 20, Z acceleration
- 21, A acceleration
- 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
[0 mean line, 8 mean curve]

- 31, curve ini acceleration
[>=10]
- 32, curve acceleration
[>=10]
- 33, curve max acceleration
[>=500]
- 34, X go home rampit speed(mm/min)
- 35, X go home reverse speed(mm/min)
- 36, Y go home rampit speed(mm/min)
- 37, Y go home rampit speed(mm/min)
- 38, Z go home rampit speed(mm/min)
- 39, Z go home reverse speed(mm/min)
- 40, A go home rampit speed(mm/min)
- 41, A go home rampit speed(mm/min)
- 42, Spindle first max speed(rpm)
- 43, Spindle second max speed(rpm)
- 44, Spindle third max speed(rpm)
- 45, Spindle forth max speed(rpm)
- 46, Second Spindle max speed(rpm)
- 47, G02/G03reverse compensation mode(0 mean A; 8 mean B)
- 48, mode B reverse compensation speed(mm/min)
- 48-1, mode B reverse compensation Beginning feed speed(mm/min) [>10]
- 48-2, mode B reverse compensation acceleration(mm/min)/s [>10]
- 49, speed Mode(1 Yes, 0 No)
- 50, Handwheel stop speed(mm/min) [>100]
- 58, Forcely limit drop speed critical(mm/min)

3.3.3 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, Z-axis`s negative scope(mm)
- 6, Z-axis`s positive scope(mm)
- 7, Spindle stop time(10ms)
- 8, Spindle stop long signal
[0 mean No, 1 mean Yes]
- 9, Check SP encode

[1 mean Yes, 0 mean No]
 10, SP encode pulse
 [4 times encode thread]
 11, Soft limit invalid
 [D2X;D3C(Y);D4Z;D5A;1mean invalidation;0 mean validation]
 12, X-axis's reverse compensation(um)
 [radius]
 13, Z-axis's reverse compensation(um)
 14, X-axis's direction signal
 [1 mean normal, 0 mean reverse]
 15, Z-axis's direction signal
 [1 mean normal, 0 mean reverse]
 16, Close feed electron gear
 [1 mean Yes, 0 mean No]
 17, X-axis's electron gear numerator(1-999999)
 18, X-axis's electron gear denominator(1-999999)
 19, Z-axis's electron gear numerator(1-999999)
 20, Z-axis's electron gear denominator(1-999999)
 21, XZ positive limit
 [0 open, 1 close]
 22, XZ negative limit
 [0 open, 1 close]
 23, float zero bit paramter
 [D3X;D4C(Y);D5Z;D6A;0 mean machine Zero;1 mean float Zero]
 24, X coor float zero set
 25, Z coor float zero set
 26, Feed axis home
 [1 mean No use, 0 mean clew, 8 compulsion, 9 must compulsion]
 27, Feed axis home mode
 [0 reverse check, 1 reverse No check, 2 No reverse check, 3 No reverse No
 check]
 28, Home reverse direction
 [D2X;D3C(Y);D4Z;D5A;D8=1fristZ;0Positive;1Neqative]
 29, Home switch set
 [D0X;D1C(Y);D2Z;D3A;1Close;0 Open]
 30, X check zero max lenth(100um)
 [radius]
 31, Z check zero max lenth(100um)
 32, X Home offset(10um)
 33, Z 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, Spindle manual point M04 [8 mean M04]
80, XZ axis coordinate plan
[D2Zwordpiece, D3Xwordpiece, D4Ztool, D5Xtool, D6Zcircumrotate, D7Xcircumrotate]
100, system inner parameter
101, lathe third axis name
[0 mean Y, 1 mean C]
102, lathe C axis
[0 mean circumrotate axis, 1 mean line axis]
103, lathe C is circumrotate axis
[0 null; 1 absolute coordinate plan; 2 tool coordinate plan; 3 all]
104, C(Y) motor direction(0 reverse, 1 normal)
105, C(Y)-axis's electron gear numerator(1-999999)
106, C(Y)axis's electron gear denominator(1-999999)
107, C(Y)-axis`s reverse compensation(um)
108, C(Y) G00 speed (mm/min)
109, C(Y) G1G2G3 Max speed(mm/min)
110, C(Y) acceleration
111, Handwheel C(Y) limit speed(mm/min)
112, C axis home encode zero speed(° /min)
113, C(Y)go home rampit speed(mm/min)
114, C(Y)go home reverse speed(mm/min)
115, Y check zero max lenth(100um)
116, Y Home offset(10um)
117, C(Y)-axis`s negative scope(mm)
118, C(Y)-axis`s positive scope(mm)
119, C(Y) coor float zero set
200, system inner parameter
201, lathe A axis
[0 mean circumrotate axis, 1 mean line axis]
202, lathe A is circumrotate axis
[0 null; 1 absolute coordinate plan; 2 tool coordinate plan; 3 all]
203, A motor direction(0 reverse, 1 normal)
204, A-axis's electron gear numerator(1-999999)
205, A-axis's electron gear denominator(1-999999)
206, A-axis`s reverse compensation(um)
207, A G00 speed (mm/min)
208, A G1G2G3 Max speed(mm/min)
209, A acceleration
210, Handwheel A limit speed(mm/min)
211, A go home rampit speed(mm/min)
212, A go home reverse speed(mm/min)

213, A check zero max lenth(100um)
214, A Home offset(10um)
215, A-axis`s negative scope(mm)
216, A-axis`s positive scope(mm)
217, A coor float zero set
404, SP motor direction(0 reverse, 1 normal)
405, SP-axis`s electron gear(0 Yes, 1 No)
406, SP-axis`s electron low gear numerator(1-999999)
407, SP-axis`s electron low gear denominator(1-999999)
408, SP-axis`s electron high gear numerator(1-999999)
409, SP-axis`s electron high gear denominator(1-999999)
410, Interpolation tap SP name[91 X, 92 Y/C, 93 Z, 94 A]
411, Interpolation tap mode[0 follow encode;4 interpolation to SP]
412, SP tooth number(<P413)
413, Encode number(>P412)

3.3.4 Tool parameter

1, C Tool radius compensation`s establish
2, C Tool radius compensation`s cancel

3.3.5 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
- 601, Make current to Step Motor Parameter
- 602, Make current to Step Servo Parameter

3.3.6 Work coordinater parameter

- 1, X of work coordinates G54(mm)
- 2, Y of work coordinates G54(mm)
- 3, Z of work coordinates G54(mm)
- 4, A of work coordinates G54(mm)
- 6, X of work coordinates G55(mm)
- 7, Y of work coordinates G55(mm)
- 8, Z of work coordinates G55(mm)
- 9, A of work coordinates G55(mm)
- 11, X of work coordinates G56(mm)
- 12, Y of work coordinates G56(mm)
- 13, Z of work coordinates G56(mm)
- 14, A of work coordinates G56(mm)
- 16, X of work coordinates G57(mm)
- 17, Y of work coordinates G57(mm)
- 18, Z of work coordinates G57(mm)
- 19, A of work coordinates G57(mm)

- 21, X of work coordinates G58 (mm)
- 22, Y of work coordinates G58 (mm)
- 23, Z of work coordinates G58 (mm)
- 24, A of work coordinates G58 (mm)
- 26, X of work coordinates G59 (mm)
- 27, Y of work coordinates G59 (mm)
- 28, Z of work coordinates G59 (mm)
- 29, A of work coordinates G59 (mm)

3.3.7 Password

password setting includes:

- 1, Is enable CNC Co. ' s password ?
- 2, Is enable Machine Co. ' s password ?
Original password ia "NEWNEW" .
- 3, Is enable User' s password ?
Original password ia "KERKER" .
- 4, Modify CNC Co. ' s password:
- 5, Modify Machine Co. ' s password:
- 6, Modify User' s password:
- 7, curry word time: (days)

3.3.8 Pitch error compensation

It is used for pitch error automatic compensation, due to the effect of screw pitch error on machine transmission accuracy. system adopts store pitch error compensation: when debugging, it measures out the screw error curve based on machine zero point a strating point, makes out revised curve on the basis of error curve, then inputs the revised curve into revised parameters table, and compensates according to this table.

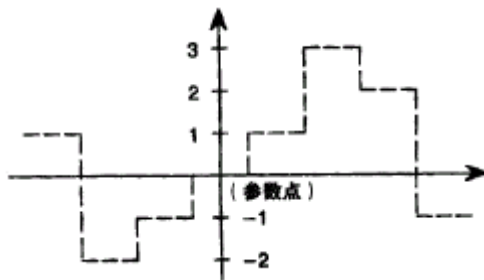
In parameter menu, pressing "Parameter" key enter into:

Man Con		N0000		2013-04-03 16:08	
Basic Parameter		No	Coord(mm)	Value(um)	Program
1, Standby: 0		1)	<160.000>	0	02171.NC
2, Neg-point: 3		2)	<80.000>	0	Instruction code
3, Pos-point: 2		3)	<0.000>	0	G54
4, multiple: 1.000		4)	<-80.000>	0	T01H0D0
5, distance(um): 80000		5)	<-160.000>	0	Machine Status
		6)	<-240.000>	0	M05 M09 M10
					M78 M33 M70
					G00 X100%
					F500 X150%
					S5000 X100%
					Machine Coord
					X 0.000
					Y 0.000
					Z 0.000
N-X-axis	T-Y-axis	R-Z-axis	Q-A-axis	PartTime	0:0
A-	B-ChePro	C-CLEAR	D-CANCEL	PartNo	1
No Alarm				SPrpm	0

By using cursor key, it enters into basic parameters setting area, selects parameter through up/down arrows, and presses Enter to pop up dialog box of inputting parameters.

The number of compensation point can be set optionally, Compensation parameters include:

- Compensation point NO.of reference point.
- Com.point NO.of farest end in negative direction.
- Com.point NO.of farest end in positive direction.
- compensation percentage.
- interval between compensation point (um) .
- Compensation value



compensation point

System automatically figures out each axis pitch error

compensation point position according to basic parameters. Each axis pitch error compensation point is distributed with equal interval; users can input each point compensation value.

The interval of compenstion point is set on the each axis,
For example:

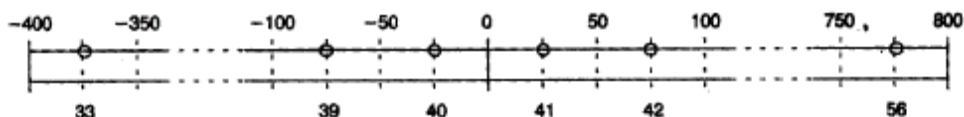
Example 1:Linear axis:when length of travel is -400mm ~ +800mm, interval of points 50mm, reference point compensation NO. 40, it can figure out that Com.point NO.of farest end in negative direction is:

Machine negative travel/point interval +1=40-400/50+1=33.

Com.point NO.of farest end in positive direction is:

Machine positive travel/point interval +1=40+800/50=56.

Machine coordinate and compensation point NO.correspondence is:



output compensation value in 0 position

parameters set as follows:

compensation point NO.of reference point:40

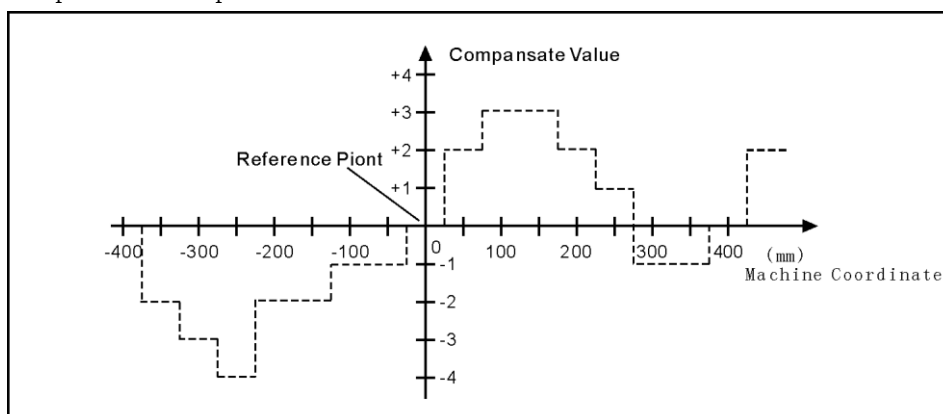
Com.point NO.of farest end in negative direction:30

Com.point NO.of farest end in positive direction:56

Compensation percentage:1

Compensation point interval:50000

Compensation point and value contrast:

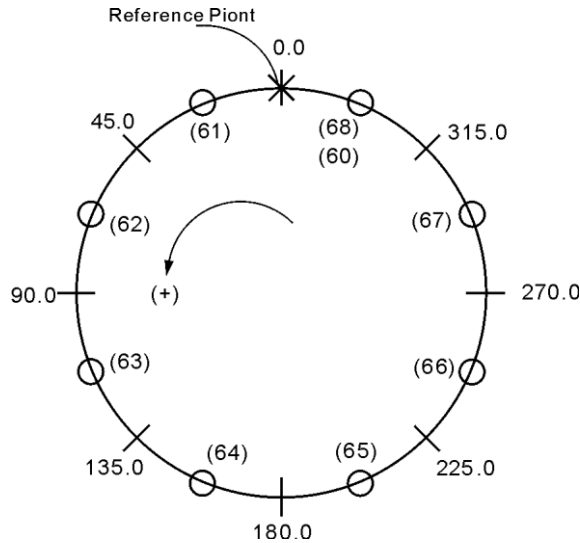


Example 2:rotor axis: when movement per revolution is 360°, interval of points 45°, reference point compensation NO. 60, Com.point NO.of farest end in negative direction is usually same as reference point com.point NO.

Com. point NO. of farest end in positive direction is:

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

Machine coordinate and compensation point NO. correspondence is:



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

Parameter sets as follows:

compensation point NO. of reference point:60

Com. point NO. of farest end in negative direction:60

Com. point NO. of farest end in positive direction:68

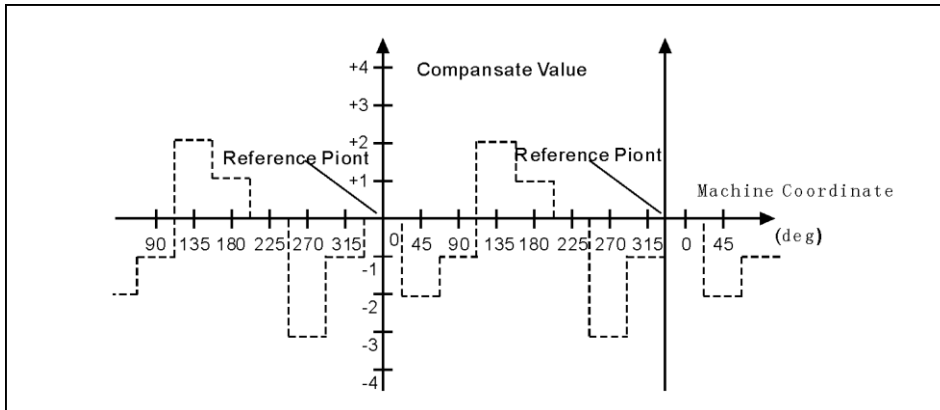
Compensation percentage:1

Compensation point interval:45000

Output compensation value at corresponding point:

NO.	60	61	62	63	64	65	66	67	68
VALUE	+1	-2	+1	+3	-1	-1	-3	+2	+1

Compensation point and value contrast:



3.3.9 Input/output diagnosis

Presses “Parameter” key :

Man Con		N0000		2013-03-28 08:57																																																																																																																											
<table border="1"> <tr> <td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td> </tr> <tr> <td>X00</td><td>X01</td><td>X02</td><td>X03</td><td>X04</td><td>X05</td><td>X06</td><td>X07</td> </tr> <tr> <td>T01</td><td>T02</td><td>T03</td><td>T04</td><td>T05</td><td>T06</td><td>T07</td><td>T08</td> </tr> <tr> <td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td> </tr> <tr> <td>X08</td><td>X09</td><td>X10</td><td>X11</td><td>X12</td><td>X13</td><td>X14</td><td>X15</td> </tr> <tr> <td>M34/A0</td><td>-L</td><td>+L</td><td>M36/Y0</td><td>X0</td><td>Z0</td><td>KRUN</td><td>KHALT</td> </tr> <tr> <td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td> </tr> <tr> <td>X16</td><td>X17</td><td>X18</td><td>X19</td><td>X20</td><td>X21</td><td>X22</td><td>X23</td> </tr> <tr> <td>XZ0</td><td>ZZ0</td><td>KLEFT</td><td>KRIGHT</td><td>STOP</td><td>TDK</td><td>ALM</td><td>ALM1</td> </tr> <tr> <td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td><td>0</td> </tr> <tr> <td>X24</td><td>X25</td><td>X26</td><td>X27</td><td>X28</td><td>X29</td><td>X30</td><td>X31</td> </tr> <tr> <td>ALM2</td><td>M28</td><td>M24</td><td>M22</td><td>M18</td><td>M12</td><td>M14</td><td>M16</td> </tr> <tr> <td>1</td><td>1</td><td>1</td><td>1</td><td>1</td><td>1</td><td>1</td><td>1</td> </tr> <tr> <td>X32</td><td>X33</td><td>X34</td><td>X35</td><td>X36</td><td>X37</td><td>X38</td><td>X39</td> </tr> <tr> <td>HX/DS8</td><td>HY/DS4</td><td>HZ/DS2</td><td>HA/DS0</td><td>HX1/K8</td><td>HX10/K</td><td>HX100/</td><td>HOFF/K</td> </tr> </table>						0	0	0	0	0	0	0	0	X00	X01	X02	X03	X04	X05	X06	X07	T01	T02	T03	T04	T05	T06	T07	T08	0	0	0	0	0	0	0	0	X08	X09	X10	X11	X12	X13	X14	X15	M34/A0	-L	+L	M36/Y0	X0	Z0	KRUN	KHALT	0	0	0	0	0	0	0	0	X16	X17	X18	X19	X20	X21	X22	X23	XZ0	ZZ0	KLEFT	KRIGHT	STOP	TDK	ALM	ALM1	0	0	0	0	0	0	0	0	X24	X25	X26	X27	X28	X29	X30	X31	ALM2	M28	M24	M22	M18	M12	M14	M16	1	1	1	1	1	1	1	1	X32	X33	X34	X35	X36	X37	X38	X39	HX/DS8	HY/DS4	HZ/DS2	HA/DS0	HX1/K8	HX10/K	HX100/	HOFF/K	Program 02171.NC Instruction code G54 T01H0D0 Machine Status M05 M09 M10 M78 M33 M70 G00 X100% F500 X150% S5000 X100% Machine Coord X 0.000 Y 0.000 Z 0.000 PartTime 0:0 PartNo 1 SPrpm 0	
0	0	0	0	0	0	0	0																																																																																																																								
X00	X01	X02	X03	X04	X05	X06	X07																																																																																																																								
T01	T02	T03	T04	T05	T06	T07	T08																																																																																																																								
0	0	0	0	0	0	0	0																																																																																																																								
X08	X09	X10	X11	X12	X13	X14	X15																																																																																																																								
M34/A0	-L	+L	M36/Y0	X0	Z0	KRUN	KHALT																																																																																																																								
0	0	0	0	0	0	0	0																																																																																																																								
X16	X17	X18	X19	X20	X21	X22	X23																																																																																																																								
XZ0	ZZ0	KLEFT	KRIGHT	STOP	TDK	ALM	ALM1																																																																																																																								
0	0	0	0	0	0	0	0																																																																																																																								
X24	X25	X26	X27	X28	X29	X30	X31																																																																																																																								
ALM2	M28	M24	M22	M18	M12	M14	M16																																																																																																																								
1	1	1	1	1	1	1	1																																																																																																																								
X32	X33	X34	X35	X36	X37	X38	X39																																																																																																																								
HX/DS8	HY/DS4	HZ/DS2	HA/DS0	HX1/K8	HX10/K	HX100/	HOFF/K																																																																																																																								
N-	T-I/O	R-	Q-ALARM																																																																																																																												
A-	B-Reset	C-	D-CANCEL																																																																																																																												
No Alarm																																																																																																																															

Man Con		N00000								2013-03-28 08:57		
Output Point										Program 02171.NC		
0	0	0	0	0	0	0	0	0	0	Instruction code		
Y00	Y01	Y02	Y03	Y04	Y05	Y06	Y07	Y08	Y09	G54		
M61	M63	M65	M67	M69	M71	M73	M75	M77	M79	T01H0D0		
0	0	0	0	0	0	0	0	0	0	Machine Status		
Y16	Y17	Y18	Y19	Y20	Y21	Y22	Y23	Y24	Y25	M05 M09 M10		
LRUN	INTH	+T	-T	S04	S03	S02	S01			M78 M33 M70		
										G00 X100%		
										F500 X150%		
										S5000 X100%		
										Machine Coord		
										X 0.000		
										Y 0.000		
										Z 0.000		
N-	T-I/O	R-	Q-ALARM								PartTime	0:0
A-	B-Reset	C-	D-CANCEL								PartNo	1
No Alarm										SPrpm	0	

3.4 Program

Program management adopts documents management mode, due to NAND FLASH, this system can store 32MB program. user poogram can be protected by password. Edition is made by full screen mode.

In main interface, press “program” to pop up interface of choosing program.

Man Con				N00000		2013-03-28 08:56	
File/File folde in system				Program		02171.NC	
CurDir: /NC				Instruction code		G54	
02171.NC				Machine Status		T01H0D0	
1000.TXT				M05		M09	
1000X				M78		M33	
1001				M110		M70	
1002				G00		X100%	
120				F500		X150%	
1201				S5000		X100%	
121				Machine Coor		X 0.000	
1234				Y		0.000	
147				Z		0.000	
22				PartTime		0:0	
84				PartNo		1	
841				SPrpm		0	
98							
Compile-P: Receive-J, Tansmit-K, DNC-L							
N-New/Sek		T-COPY		R-RENAH		Q-INFOR	
A-LAST		B-USBdisk		C-EXEC		D-CANCEL	
No Alarm							

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

3.4.1 new file/search

when this button is pressed, it pops up the requirement to input the name of new/searched documents, it can be number, letter (no difference if it is capital letter or small letter) or other mixture of symbol (not include / \ : * ? “ < > | and), no limitation on length. Input document name, then press “enter” to confirm. if it exists in system, it will be found and reversely displayed, if not, it will be newly build and reversely diplayed. To build a new file.

3.4.2 copy

it is reduplicating current program to another program. Choose this item to pop up dialogue box, input new document name, if it exists, input is invalid, if not, this name will be the name of newly copied

document.

3.4.3 rename

for convenience of management, the original documents can be renamed. Choose this item to pop up dialogue box, input new document name, if it exists, input is invalid, if not, this name will be the name of original document.

3.4.4 delete

“Del” for deleting all content and name of current program.

3.4.5 infomation

This system provides users information column for each program, which is convenient for users to amend and set.

Length of document (uneditable)

Last time of document amending (uneditable) .

3.4.6 USBdisc

Press “B” open or close U disk.

note: before pulling out U, it must return to directory of doc name. otherwise newly copied data in U may lose.

3.4.7 Serial port transmission program

Besides U, can use RS232 port. In interface of choosing program, press R to receive program, press T to send program:

Then can communicate the program according to the interface. The following chart shows:

Transmit the program file from PC to CNC: Run CNC CO.’ s special series communication software on PC . Clicks the “transmits the CNC program file” button and select , clicks the “turns on” button, now PC is waiting for transmitting; presse”J” under the “program” interface, keys in the program filename. The PC begins to transmit.

Transmit the program file from CNC to PC: presses the key “↑” “↓” to selet the program filename under the interface of “program” , then presse “K”, now the system is waiting for transmitting; Run CNC CO.’ s special series communication software in PC. Click the “receives the CNC program file” button, key in the program

filename in the dialog box, clicks on the "save" button, now the system begins to transmit the program file.

3.4.8 editing

Man Con		N0000		2013-03-28 08:56	
/NC/02171.NC				Program 02171.NC	
N106G0 G54G90X3.Y-33.25S1500M3				Instruction code	
N108G43H1Z10.M8				G54	
N110Z2.				T01H0D0	
N112G1Z-.278F1000.				Machine Status	
N114Y33.25F1500.				M05 M09 M10	
N116Z-.656F1000.				M78 M33 M70	
N118Y-33.25F1500.				G00 X100%	
N120Z-1.033F1000.				F500 X150%	
N122Y33.25F1500.				S5000 X100%	
N124Z-1.411F1000.				Machine Coord	
N126Y-33.25F1500.				X 0.000	
N128Z-1.789F1000.				Y 0.000	
Press Rapit Key No.1 /Total2305				Z 0.000	
N-COMPIL	T-FirLine	R-EndLine	Q-POS	PartTime	0:0
A-	B-DeLine	C->>	D-CHINA	PartNo	1
No Alarm				SPrpm	0

The edition mainly uses to edit, insert, modify, delete and so on. After selects the program name and enter the entire screen edition system. The menu at the base of the screen includes (press "RAPIT" + "N, T, R, Q, B, C, D") "compile", "first line", "end line", "pose", "del line", ">>" ("del block" "copy block" "array" "serch" "alter" "aalter" "<<"), "cancel", etc.

Users can operate at the area of line number at the left side of the screen.

The program name to edit and the line number to point were clue at the top of the screen.

1) **pose the cursor:** change the cursor's position

"↑ ↓" the cursor moves up or down

"-> <-" The cursor shifts to left or right

"PgUp.PgDn" the cursor goes to last page or next page.

"Enter", to the next line.

press "pose" and key in line number can locate directly to the line which you key in.

Press "first line" locate directly to "the first line".

press "endline" locate directly to the end line.

When the located program line surpasses the page, it will automatically change to the next page and the located program line will be contained in the display .

2) **insert**: key in the insertion in front of the cursor, if they are letters, it will automatically produces blank space.

3) **delete**: presses "Del" can delete the character at the back of the cursor.

4) **shift KEY**: presse twice key in the shift character.

5) **delete line** :press "RAPIT+delete line" to delete the line.

6) **operate the block** : Contains copy block and delete block.

7) **compile**: compile the source program (ISO code) to the computer code procedure.

show error when compile, or show "OK" .

When enters the automatic main function, this system automatically carries on concealed compiling process .If there' s a mistake, the system clues on the error message.

"compile" includes " compile NC" and " compile MAC".

8) **search**: Uses to search the appointed character string.

9) **replace**: "alter" Uses to replace the appointed character string.

10) **all replace**: "aalter" Use to replace all appointed character string from the cursor to the ending of the program

press "Emergency brake" can stop carrying on " search", " alter", "all alter".

11) **exit**: press "Esc" or F8 returns to the main interface and save the program automatically.

3.4.9 Select the machining procedure

Select the machining procedure before the automatically machining.

The operating procedure is: Press "↑" "↓" to select the program and press "execute" ("C" key).

3.5 Manual


3.5.1 Continual mode

Continuous operation is based on the time of pressing down the keys, press down to, By using the keys "+X, -X, +Y, -Y, +Z, -Z, +4, -4" in the panel to make feed in the selected axis, feed speed equals handle speed times speed percentage.

When feed moves over the two hard limit points of the operating axes, it will stop, at this time it can only move reversely.

3.5.2 Increment

The increment way operation means set a increment with the keys "+X, -X, +Y, -Y, +Z, -Z, +4, -4". feed speed equals handle speed times speed percentage.


Presses the key  to change the increment. When feed moves over the two hard limit points of the operating axes, it will stop, at this time it can only move reversely. Presses "I" change increment value.

3.5.3 Handwheel pulse generator

Users can select the axis X, Y, Z and the fourth axis, and can select percentage X1, X10, X100. When you use it the green light at the right side of the interface lights up.

3.5.4 Back to the reference points

Going back to the reference points means to move each axis to machine datum point switch. When axis inspects the datum point signal, it will set the parameter as datum point data in accordance with the preferential reference points.

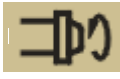
At the manual condition, presses  and select X, Y, Z, 4, A to go back to the reference point. When chooses X, Y, Z, 4, only returns to this axis the reference point. Chooses A, returns to the reference point in turn.


Presses "stops" returns to the reference point.


3.5.5 Other operation at the manual conditions


1) manual main axle condition:

Press  the principal axis veer, display M03.


Press  the principal axis reverse, display M04.

Press  the principal axis stop, display M05. At this condition, users can press the key to turn off or turn on.

Press  M03 turn on point for while.

Press  the coolant to turn on or turn off.



Press  for spindle loses tool on/off

Press  for lubrication on/off

Press  for huff on/off



2) Adjust the feed speed:

The feed speed percentage can be controlled by the wave band

switch or the key  , the percentage increases or decreases 10%. The scope is 0 -150%, 16 grades in all.

3) Control the principal axis speed:

The main axle speed percentage can be controlled by the wave band

switch or the key  , the percentage increases or decreases 10%. The scope is 0 -150%, 16 grades in all.

4) presses "stops": Stops the manual operation.

5) presses "F", there' s a dialog box used to alter the manual feed speed. That is convenient for cutting by single axis.

6) presses "S", alter the principle axis' s revolving speed.


3.5.6 Work Coordinate system setting

- 1, Presses "MDI" input G54/G59;
- 2, Presses "Setup" ,input X/Y/Z/A work coordinate.

3.6 automatic

Cancel manual and turn to automatic, The system compile the procedure automatically, it can show the error.

3.6.1 coordinates

The coordinates running show the tool' s position. It can show the workpiece coordinates and the composite coordinates. Shifted by  key.

3.6.2 graphics mode

The graphics running status means the tool path is displaying by the graphic method. Operator may rotate or translate graphics through the cursor key, and may enlarge or shorten the graphics through PageUp, PageDn key. By the Q key can return to the initial graph status. Furthermore, we can look at the entire tool track before the machining. Under the running or stop state operator can switch the coordinates/Graphics status, the coordinates/graphics switch key is



3.6.3 continuously mode

The continuous running state means the program unceasingly executes section after section.


3.6.4 Step mode

The Step mode means only runs the current program section, then waits for pressing running button.

Under the running or hold or stop state operator can switch the

step/Continual status, step/continual switch key is .

3.6.5 simulations

Under the status of simulation when presses key , After pressing the "run" button, program run by path graphic mode or coordinate mode. But all axes and other machine motion will be stopped.

3.6.6 Keep feed status

Under the program hold status, pressing "Manual" soft key can enter keep feed status, at this time, we can execute manual operate by manual continuously, manual increase, handwheel. Afterward, cancel "Manual" status and pressing the "run" button, CNC will move to the holding point by the speed of default G01/G02/G03. First move Z axis if forward, otherwise backward, other axis moving sequence is X→Y→A.

3.6.7 M D I method

When presses down the "MDI" soft key, CNC would spring the MDI dialog box, After input NC code, pressing "run" key, The CNC will carry out this section of program immediately.

3.6.8 Begin from program some actual line

Pressing the "-" key, CNC will break out a dialog box, after input actual line number and press the "run" key, CNC will execute program from the input line.

Specially pay attention: The CNC will first move to begin line point according speed of default G01/G02/G03, after all, begin to execute program.

3.6.9 Begin from program some mark line

Pressing the "M" key, CNC will break out a dialog box, after input mark line number and press the "run" key, CNC will execute program from the input line.

Specially pay attention: The CNC will first move to begin line point according speed of default G01/G02/G03, after all, begin to

execute program.

3.6.11 Set coordinates/Choice coordinates

Set coordinates:

The Set coordinates is used for configure any work coordinate or the relative coordinates value. Under the work coordinate display mode configure work coordinate; Under the synthesis coordinate display mode configure relative coordinate, in the course of program running also can configure relative coordinate.

Note: The machine coordinates cannot be configured.

Choice coordinates:

After pressing “ MDI ” input G53/G59 may choose G53, G54, G55, G56, G57, G58, G59 work coordinate. Corresponding work coordinate status is displaying in the top right corner interface.

3.6.12 large capacity molds program

Because this CNC have 32MB flash for saving user NC program, therefore the NC program can not longer 32MB. At the same time , if the program is larger than 3000 lines, cannot use G22 and other cycle instruction.

3.7 Tool redeem

Presses “Redeem” :

Man Con		N0000		2013-03-28 08:57									
Press T Key Length make tool base on mainfac				Program	02171.NC								
T01	H:	4.464	[0.000]	Instruction code									
T02	H:	0.000	[0.000]	G54									
T03	H:	0.000	[0.000]	T01H0D0									
T04	H:	0.000	[0.000]	Machine Status									
T05	H:	0.000	[0.000]	M05	M09 M10								
T06	H:	0.000	[0.000]	M78	M33 M70								
T07	H:	0.000	[0.000]	G00 X100%									
T08	H:	0.000	[0.000]	F500 X150%									
T09	H:	0.000	[0.000]	S5000 X100%									
T10	H:	0.000	[0.000]	Machine Coord									
<table border="1"> <tr> <td>N-Radius</td> <td>T-Length</td> <td>R-ACLEA</td> <td>Q-CLEAR</td> </tr> <tr> <td>A-SetTool</td> <td>B-ToolSeat</td> <td>C-Set</td> <td>D-CANCEL</td> </tr> </table>				N-Radius	T-Length	R-ACLEA	Q-CLEAR	A-SetTool	B-ToolSeat	C-Set	D-CANCEL	PartTime	0:0
				N-Radius	T-Length	R-ACLEA	Q-CLEAR						
A-SetTool	B-ToolSeat	C-Set	D-CANCEL										
No Alarm				PartNo	1								
				SPrpm	0								

- Presses N, set tool' s radius.
- Presses T, set tool' s length.
- Presses R, clear all value.
- Presses Q, clear current tool value.
- Presses A, tool posit.
- Presses B, setup tool serial table.
- Presses C, set tool' s number.
- Presses D, cancel.

Chapter4 Programming

Programming refers to process of using cnc language to describe machining track and actions based on the machining blueprint and technique requirement.

4.1 Basic Concepts

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

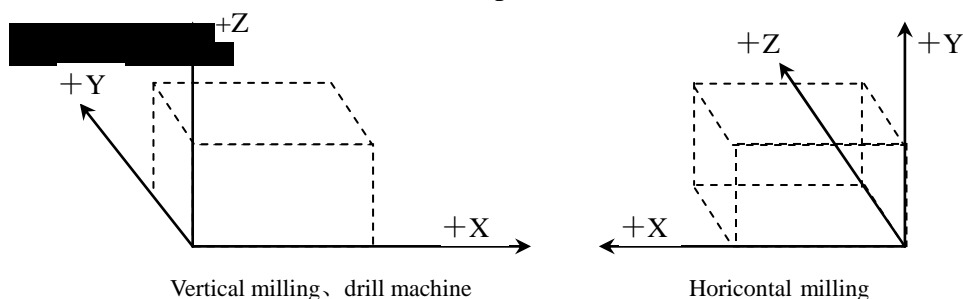
Program: is a congregation of program segment by machining logic structure in order to complete the machining of workpiece.

Machine Coordinate System: The establishment of coordinate is based on machine's zero point. The milling machine coordinate axis and its direction should follow to "IS0841" standard. The method as follow: Through right hand rule we can make the program coordinate, The Z axis is parallel as spindle, The X axis is horizontal, The Y axis is determined by right hand rule. The A, B, C are rotated axis or assistant axis which parallel as X, Y, Z axis. Furthermore, The coordinate axis direction is the increasing workpiece dimension direction.

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

•

Machine Coordinate & direction
sketch map



Work 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. Uses one of three methods to set the work piece coordinates:

USeG54 toG59: Use operating parameter set coordinate system may set 6 work piece coordinate system.

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

Partial coordinate system: In work piece coordinate system for easy to programming it may establish the sub- coordinate system, this sub-coordinate system is called the partial coordinate system

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

Relative Programming(increment programming): It is distance and direction of operation end point ,compared with starting point. It is settment 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(spindleclockwise), M04(spindlecounter clockwise), M05(spindle stop). They are allModle 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 unstruction 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 ststus.

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

4.2 General description of program

%04, N04, G02, T02, H02, D02, M02, S04, F04, X-043, Y-043, Z-043, A-043, I-043, J-043, K-043, L04, P4, R043.

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

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

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

4.2 Program instruction

4.2.1 Functional meaning of address symbol, data list

Functions	Address symbol	meaning	Data range
Document No.	%	Name of machining workpiece	0-9、A-Z
Program segment No.	N	No. of program segment	0000-9999
Preparation function	G	Content and mode of designated instruction operation	00-99
Auxiliary function	M	Auxiliary operation instruction	00-99
Tool chosen	T	No. of Tool.	01-99
Tool compensation	H D	The length compensates No. of the radius compensates of the parameter	1- 4

Spindle function	S SP	The spindle speed; spindle localization	00-99999
Cutting speed	F	Speed per minute	1-3000mm/min
Coordinates character	X Y Z A(B/C/U /V/W)	The coordinates value of X Z and 4th axes.	□9999.999mm
Core coordinates	I J K	X Z axes and Z axes core coordinate increment value	□9999.999mm
Step length	R	Circular arc radius	0.001-999.999mm
Delay time	P	Delay time of designated delay	0.001-99.999s
Program entrance	P	Entrance of calling program name	0000-9999
Repeat times	L	Times of cycle or subprogram calling	1-9999

4.2.2 G、M Function instruction data list

Table 1 G Instruction-code and functiont

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

G02	01	Inserts along the circle makes up/the spiral line to insert makes up CW: The spiral motion spiral line inserts makes up the 2 circular arcs insert makes up the axis synchronization migration other axes. The instruction method only is simply adds on is not the circular arc inserts makes up the axis the shifting shaft
G03		The counter circle inserts makes up/the spiral line to insert makes up CCW
G04	00	pause
G15		Polar coordinate instruction cancellation

G16	17	<p>Polar coordinate instruction: The polar coordinate (radius and angle), the angle to is chooses the plane the first axis to anti-clockwise changes, but the negative direction is clockwise changes .</p> <p>Form :</p> <p>G** G## G16; G00 IP;</p> <p>G** Expresses the plane chosen</p> <p>G## 表示 G90 (Work piece coordinate system original point) 或 G91 (Current position) Assigns the polar coordinate and zero point</p>	
G17	02	Choose the X Yplane	X: X axis or its parallel axis
G18		Choose the Z X plane	Y: Y axis or its parallel axis
G19		Choose the Y Z plane	Z: Z axis or its parallel axis
G20	06	Inch input	
G21		Millimetre input	
G28/G281/G282/G283/G284		Go to first reference point	

G30/G301/G302/G303/G304	00	Go to 2,3,4 reference point
G26		ZXY axis go to program original point
G261		X axis go to program original point
G262		Y axis go to program original point
G263		Z axis go to program original point
G264		A axis go to program original point
G40	07	Cancel tool radius compensate
G41		tool radius compensate, left
G42		tool radius compensate, right
G43	08	Tool lenthen positive compensate
G44		Tool lenthen negative compensate
G45	00	Tool adding offset
G46		Tool subtact offset
G47		Tool adding two multiple offset
G48		Tool subtact two multiple offset
G49	08	Cancel tool lengthen compensate
G37	11	Cancel scale zoom
G36		Enable scale zoom: format: G36 X_Y_Z_R_
G12	22	Cancel programmer mirror
G11		Enable programmer mirror
G52	00	Set local coordinate

G53	14	Chocie coordinate	
G54		Chocie work coordinate 1	Note: These six work coordinate save in CNC, user may choice any one.
G55		Chocie work coordinate 2	
G56		Chocie work coordinate 3	
G57		Chocie work coordinate 4	
G58		Chocie work coordinate 5	
G59		Chocie work coordinate 6	
G60		15	
G64	Continue path work.		
G68	16	rotate coordinate. format: G17 } G18 } G68 a-b- R-; R:Angle G19 }	
G69		Cancel rotate coordinate	
G73	09	Drill deep hole cycle : format: G73 X-Y-Z-R-Q-F- L - Z: distance from R to hole bottom R: distance from original to R Q : feed depth every time F: feed speed L: repeat time	

G74		Left Tap cycle:。 format: G74X-Y-Z-R-P-F- L -
G80		Cancel cycle mode
G81		Drill cycle: format: G81 X-Y-Z-R-F- L -
G82		Drill cycle。 format: G82 X-Y-Z-R-P-F- L -
G83		Drill cycle。 format: G83 X-Y-Z-R-Q-F- L -P-
G84		Right Tap cycle: format: G84 X-Y-Z-R-P-F- L -
G85		Drill cycle:。 format: G85 X-Y-Z-R-F- L -
G86		Drill cycle:。 format: G86 X-Y-Z-R-F- L -
G89		Drill cycle:: G89 X-Y-Z-R-P-F-L-
G90	03	Absolute program
G91		Increase program
G98	10	Go back to original point
G99		Go back to R point
G22	19	Program cycle order
G800		Cancel Program cycle order
G65	12	Mode use macro program
G66		Non-Mode use macro program
G67		Cancel Mode use macro program
G180—G189		User self defined macro program

Table 2 M code and function

M02	Program over, stop auto run (default M02)。
-----	--

M30	Program over, turn off sprindle and cool.
M00	Program hold, press “run” to continue run.
M20	Program over, According paramter auto run , using for test CNC.
M98	Using sub-program
M99	sub-program over
M97	Program jump
M03	Spindle CW
M04	Spindle CCW
M05	Stop Spindle
M08	Turn on cool
M09	Turn off cool
M10	Tighten tool
M11	Loosen tool
M58	Turn off huff
M59	Turn on huff
M32	Turn on lubricate
M33	Turn off lubricate
M79	User self-defined1 output turn on
M78	User self-defined1 output turn off
M61	User self-defined2 output turn on
M60	User self-defined2 output turn off
M63	User self-defined3 output turn on
M62	User self-defined3 output turn off
M65	User self-defined4 output turn on
M64	User self-defined4 output turn off
M67	User self-defined5 output turn on
M66	User self-defined5 output turn off
M69	User self-defined6 output turn on
M68	User self-defined6 output turn off
M71	User self-defined7 output turn on
M70	User self-defined7 output turn off
M75	User self-defined8 output turn on
M74	User self-defined8 output turn off
M41	SP Speed first gear

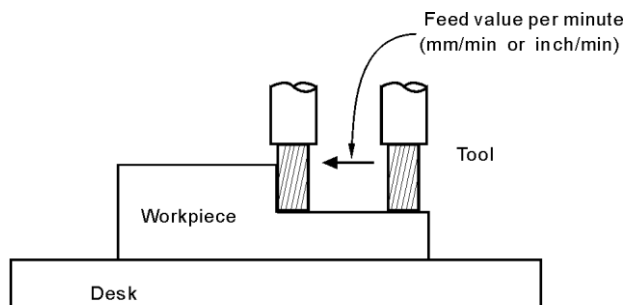
M42	SP Speed second gear
M43	SP Speed third gear
M44	SP Speed fourth gear
M12	Check M12 input valid
M13	Check M12 input invalidate
M14	Check M14 input valid
M15	Check M14 input invalidate
M16	Check M16 input valid
M17	Check M16 input invalidate
M18	Check M18 input valid
M19	Check M18 input invalidate
M28	Check M28 input valid
M29	Check M28 input invalidate
M22	Check M22 input valid
M23	Check M22 input invalidate
M24	Check M24 input valid
M25	Check M24 input invalidate

4.2.3 F function

In this CNC system, feed speed use F word. It is mode. True feed speed is the multiply of order feed speed and multiple.

Feed speed of line interpolation G01, arc interpolation G02, G03 are determined by "F" word.

feed value per minute sketch:



4.2.4 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.

4.2.5 S/SS function:

S/SS function can control spindle speed, this function is valid to all spindle which have frequency conversion speed control drive. In program we can use S/SS word to change speed. CNC provides analog voltage of 0~10V, and S/SS function is mode order. Spindle speed can use five number.

4.3 Preparation functions

4.3.1 Set coordinate(G53/G54/G55/G56/G57/G58/G59)

Note: we advice that general using this instuction in program. These instuction are used for choicing work or machine coordinate.

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

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.

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

Example 1:

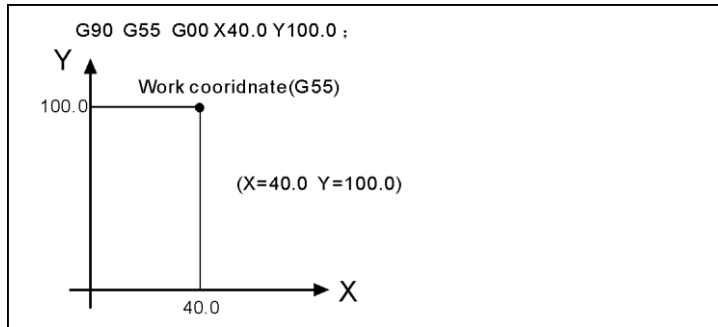
```
G01 X34
G54 X78
```

First section means moving to point of X34 in G53 machine coordinate through G01 instuction, Second section means moving to point of X78 in G54 work coordinate through G01 instuction.

Example 2:

```
G01 G56 Y64
G57
G00 Z178
```

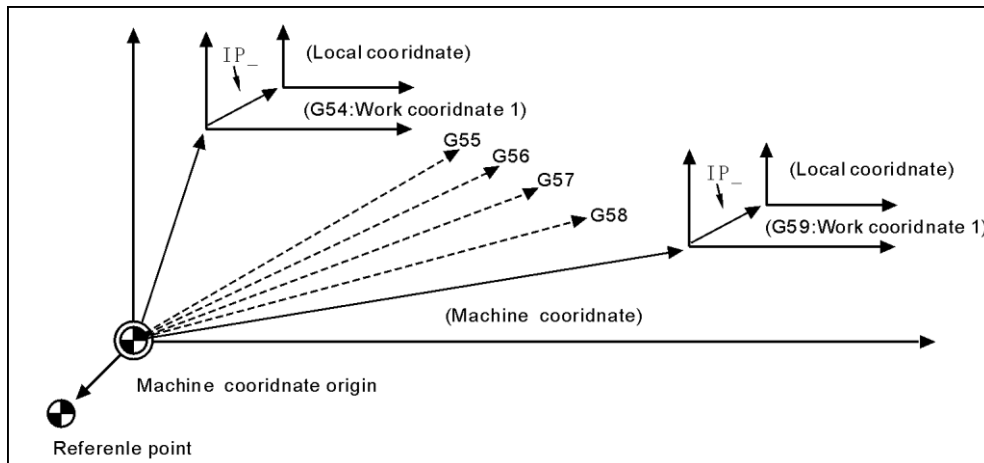
First section means moving to point of Y64 in G56 work coordinate through G01 instuction, Second section means entering G57 work coordinate, Third section means moving to point of Z178 in G57 work coordinate through G00 instuction. Demon:



4.3.2 Local coordinate(G52)

Format: G52 X- Y- Z- ; set (Mode)
G52 X0 (Y0 Z0); cancel.

Note: we advice that general doesnt using this instuction.
example:



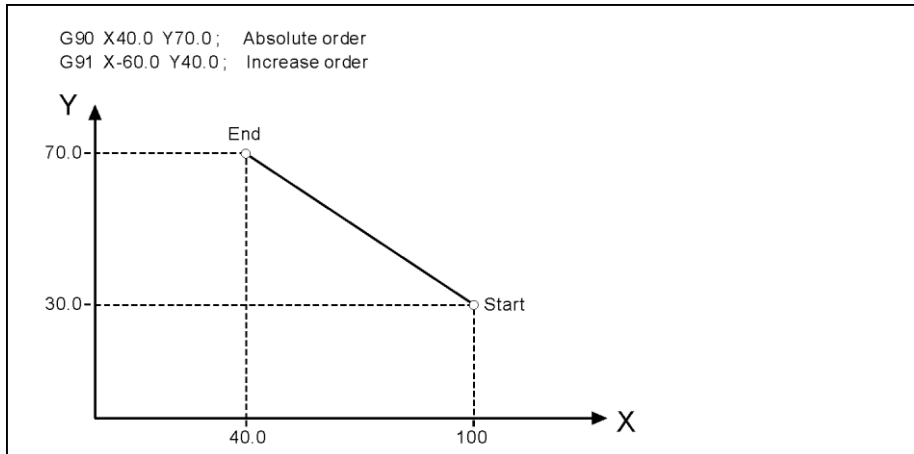
4.3.3 Program method(G90/G91)

There have two methods to move tool in program: absolute instuction and increase instuction. In absolute instuction, the number is coordinate value; but in increase instuction, the number is motion

distance. G90 and G91 are used for point out absolute or increase program.

Format: G90 (Mode)	; absolute program.
G91 (Mode, original)	; increase program.

Example:



In the example, First section means moving to point which is coordinate value X40.0Y70.0 by absolute program.

Second section means increase program, expressing that moving X distance is 60.0mm and Y distance is 40.0mm.

4.3.4 Select Plane(G17/G18/G19)

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

Using to point out arc interpolation plane.

Note: this instruction doesnot produce motion.

4.3.5 Rapid motion(G00)

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

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

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

G00 move to aim point according to line way.

Moving speed is determined by paramter.

4.3.6 Line interpolation(G01)

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

Format: G01 X- Y- Z- A- F- (Mode)

Note: X, Y, Z, A means motion axis. The data point out motion distance and direction by absolute or increase method. Motion speed is determined by F word. The F instuction is mode.

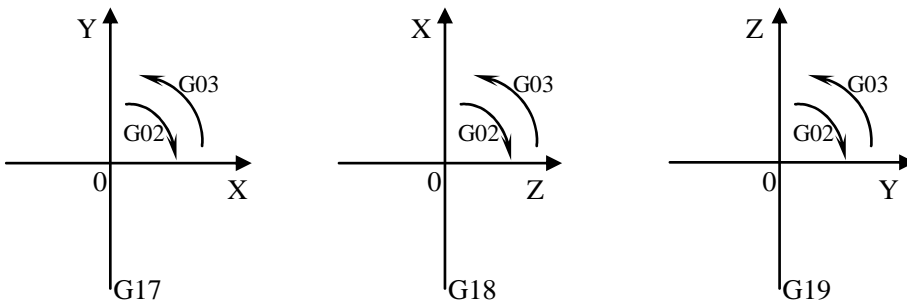
4.3.7 Arc interpolation(G02/G03)

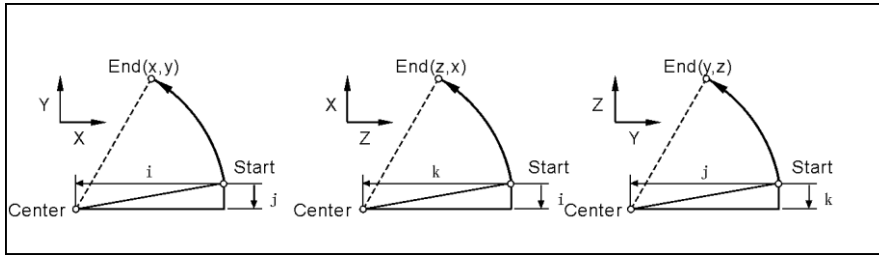
In the program plane, these instucciones execute G02 clockwise and G03 counter-clockwise arc interpolation.

Format: G02(G03) X- Y- I- J- F- ;XY plane (Mode)
 G02(G03) Z- X- K- I- F- ;ZX plane (Mode)
 G02(G03) Y- Z- J- K- F- ;YZ plane (Mode)

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

The arc track as follow:





The arc interpolation speed is determined by F word.
 Attention: I, J, K and R are the non-modality instruction.
 Demonstration:

1) absolute programming

```
N0000 G92 X200 Y40 Z0;
N0010 G90 G03 X160 Y40 I-20 J0;
N0020 G02 X120 Y40 R20;
N0030 G02 X120 Y40 R20;
N0040 G26 M02;
```

2) increase 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.

4.3.8 spiral interpolation (G02/G03)

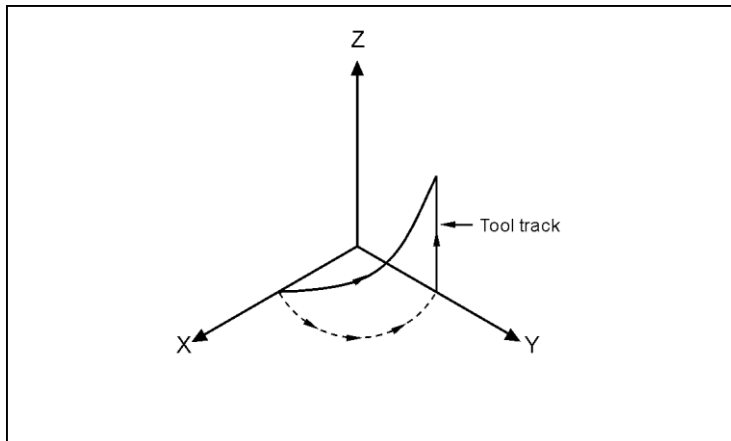
Spiral interpolation means arc interpolation adding another axis line interpolation, F instruction defines arc interpolation speed. therefore, the feed speed of line interpolation axis is as follow:

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

```
Format : G02(G03) X- Y- I- J- Z- F- ;XY plane(mode)
          G02(G03) Z- X- K- I- Y- F- ;ZX plane(mode)
          G02(G03) Y- Z- J- K- 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.



4.3.9 delay Instruction(G04)

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

Format: G04 P_ X_ U_

P word unit ms, means delay time.

X word unit S, means delay time.

U word unit S, means delay time.

For example:

G04 X1; delay 1s.

G04 P1000; delay 1s.

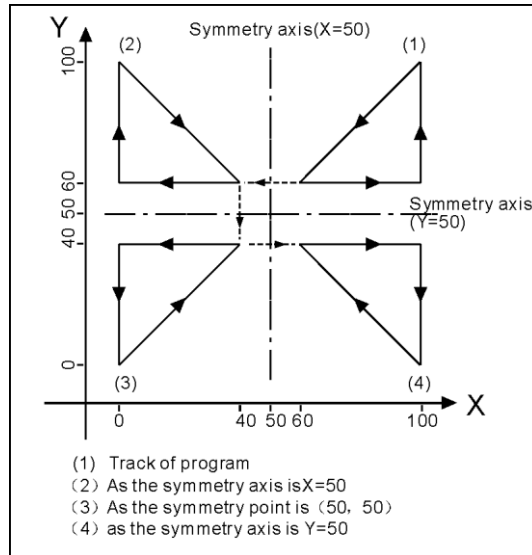
G04 U1; delay 1s.

4.3.10 Mirror instruction(G11/G12)

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

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

For example:



The mirror procedure gives an example

Sub program

%9000

G00 G90 X60.0 Y60.0;

G01 X100.0 F100;

G01 Y100.0;

G01 X60.0 Y60.0;

M99;

Min program

N10 G00 G90;

N20 M98 P%9000;

N30 G11 X50.0

N40 M98 P%9000;

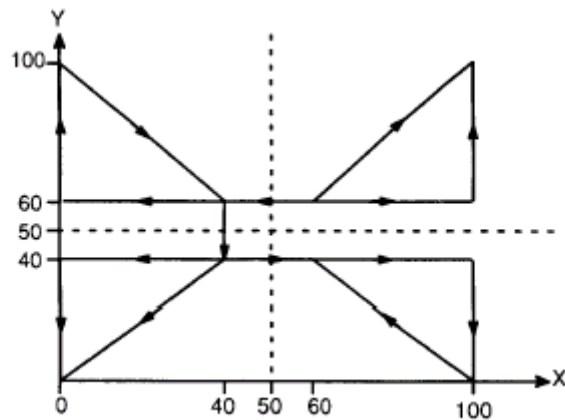
N50 G11 X50.0 Y50.0

N60 M98 P%9000;

N70 G11 Y50.0

N80 M98 P%9000;

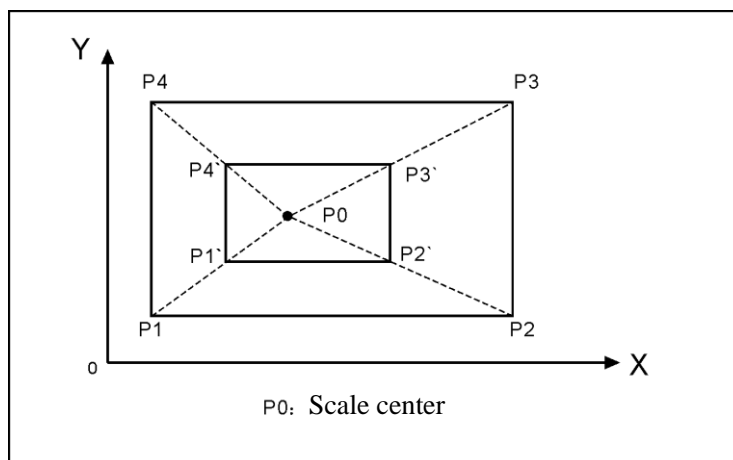
N90 G12;



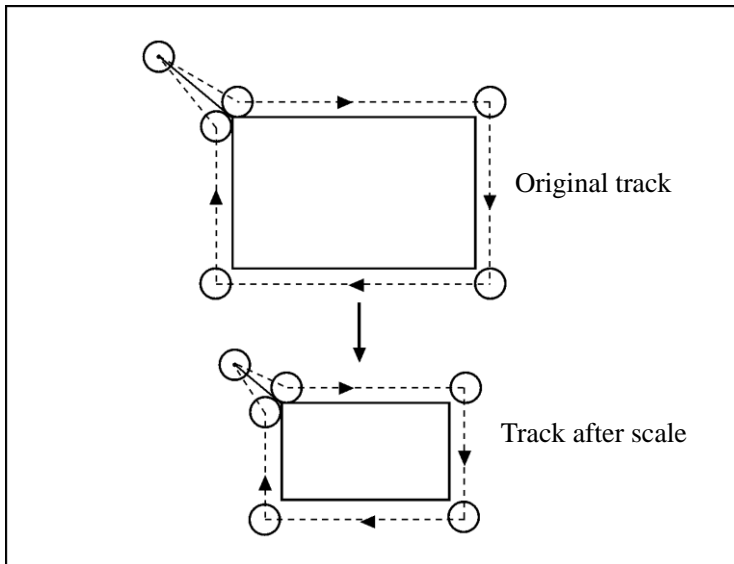
4.3.11 proportions scale instruction(G36/G37)

format: G36 X_Y_Z_R_ (mode)	;enable
G37 (mode, original)	;disable

Note: the scale coefficient is after R word.



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



4.3.12 Coordinate rotate(G68/G69)

```

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

```

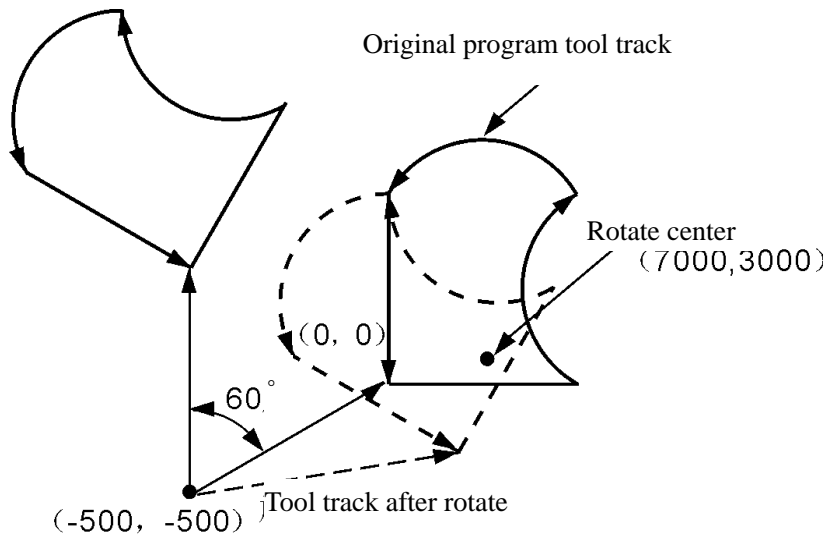
Note: The (G17)X-Y- or (G18)Z-X- or (G19) Y-Z- after G68 are used for pointing out rotate center.

R word is used for pointing out rotate angle.

Rotate angle

Rotate center

example 1:



```

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
G90 X-500Y-500
M02;

```

4.3.13 Return Refernce(G28/G281/G282/G283/G284)

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

format:	G28 X/Y/Z/ ;ZXY return to reference
	G281 ;only X return to reference
	G282 ;only Y return to reference
	G283 ;only Z return to reference
	G284 ;only A return to reference

4.3.14 Return Zero Refernce(G30/G301/G302/G303/G304)

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

format:

```
G30      ;XYZA return to Zero.
G301    ;only X return to Zero.
G302    ;only Y return to Zero.
G303    ;only Z return to Zero.
G304    ;only A return to Zero.
```

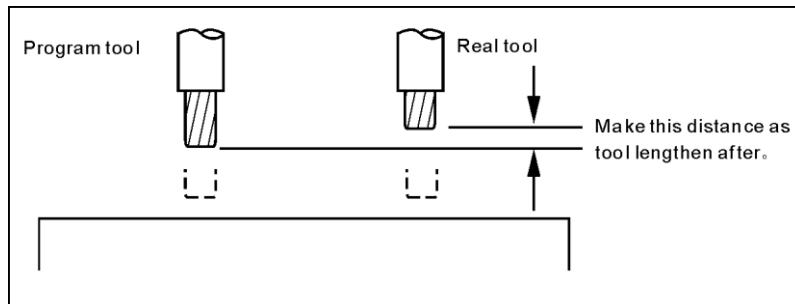
Note:

return to first reference G28 `s` sequence is Z->X->Y.

4.3.15 tool length compensate instruction(G43/G44/G49)

format:

```
G43 H-  ;Add tool length compensate.
G44 H-  ;subtract tool length compensate.
G49 or H0 ;cancel tool length compensate.
```



Example: N0000 G43 H2 X10 (H2 value is 5)

N0010 G44 H3 X20 (H3 value is 10)

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

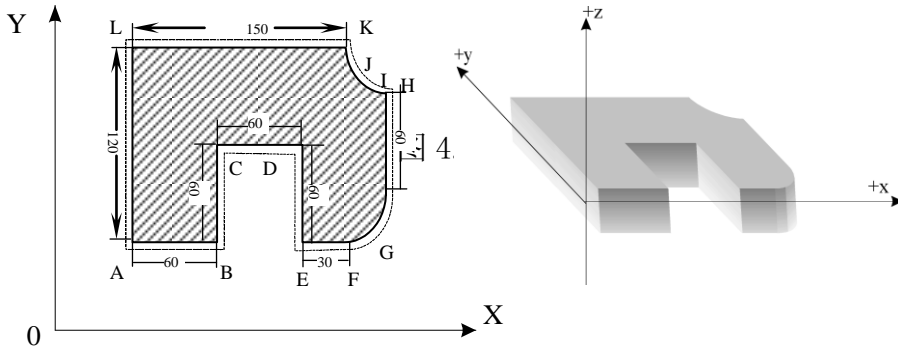
4.3.16 Offset tool radius instruction(G45/G46/G47/G48)

```
format:  G45 T-  ;Add one radius.
         G46 T-  ;subtract one radius.
         G47 T-  ;Add two radius.
         G48 T-  ;subtract two radius.
```

Note: These instructions cannot use with tool radius compensate instruction (G41, G42).

G45/G46/G47/G48 is the non-modality instruction.

For example



Program as follows:

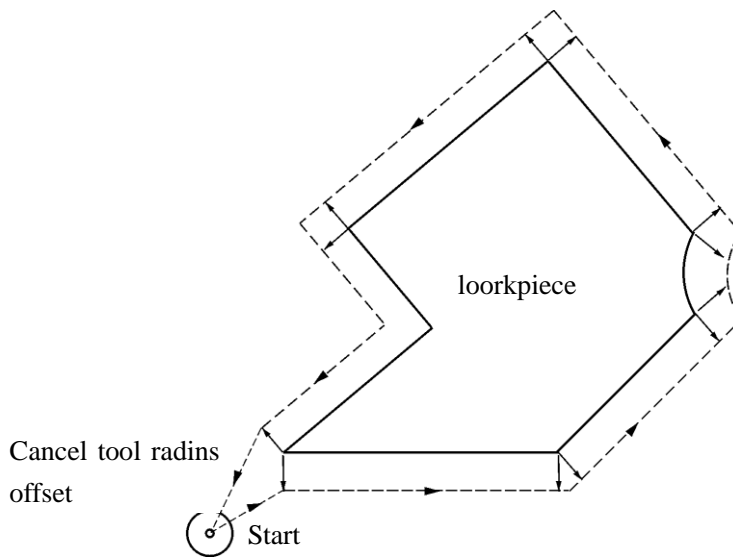
```

N0000 G01 Z-20 F400 G91      ;
N0010 G46 T01 X55 Y55       ;
N0020 G47 G01 X60 F200      ;
N0030 Y60                    ;
N0040 G48 X60                ;
N0050 Y-60                   ;
N0060 G45 X30                 ;
N0070 G45 G03 X30 Y30 R30    ;
N0080 G45 G01 Y60            ;
N0090 G46 X0                  ;
N0100 G46 G02 X-30 Y30 R30    ;
N0110 G45 G01 Y0              ;
N0120 G47 X-150               ;
N0130 G47 Y-120               ;
N0140 G46 X-55 Y-55          ;
N0150 G26                      ;
N0151 M02

```

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

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



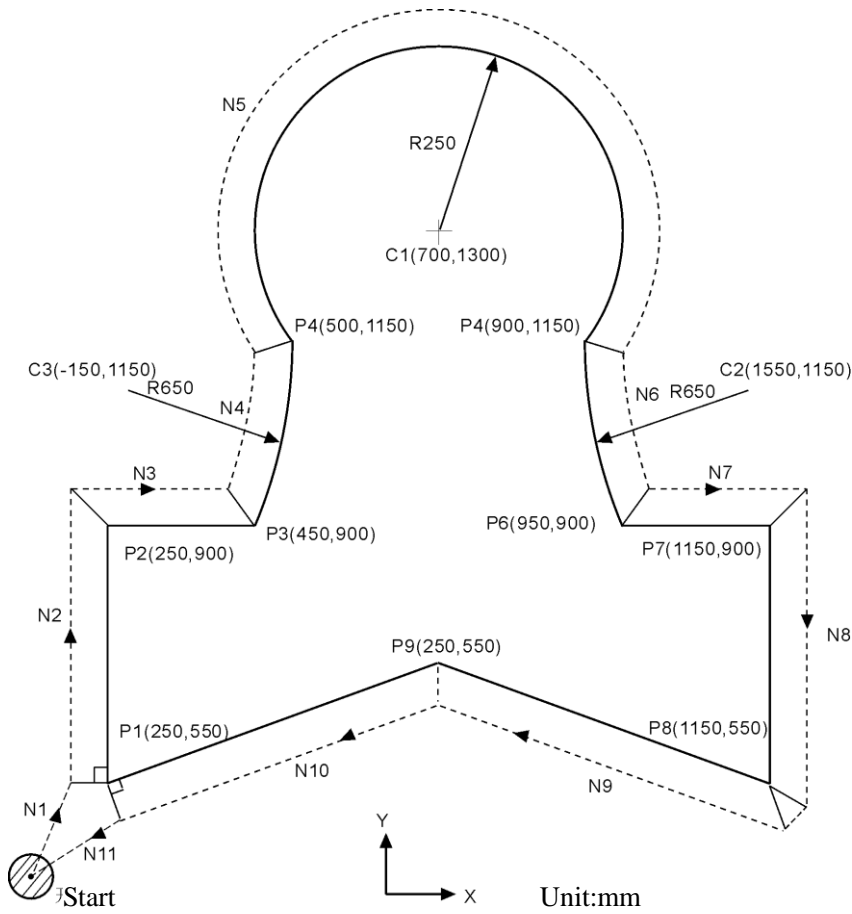
```

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.

```

Note:

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



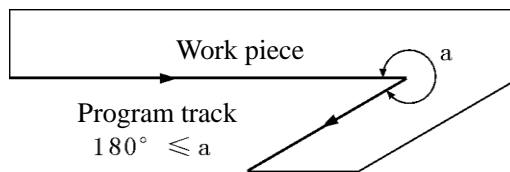
```
G54 X0 Y0 Z0;.....
N1 G90 G17 G00 G41 T15 D2 X250.0 Y550.0; establish compensation
N2 G01 Y900.0 F150; ..... from P1 to P2
N3 X450.0; .....from P2 to P3
N4 G03 X500.0 Y1150.0 R650.0; .....from P3 to P4
N5 G02 X900.0 R-250.0; .....from P4 to P5
N6 G03 X950.0 Y900.0 R650.0; .....from P5 to P6
N7 G01 X1150.0; .....from P6 to P7
N8 Y550.0; .....from P7 to P8
N9 X700.0 Y650.0; .....from P8 to P9
N10 X250.0 Y550.0; .....from P9 to P1
N11 G00 G40 X0 Y0; ..... cancel compensation
```

Tool radius compensation C:

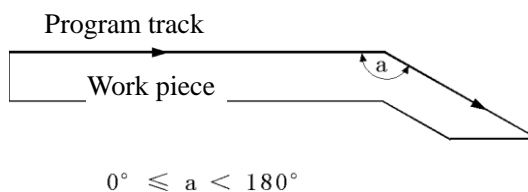
Tool radius compensation C is according to the last and next section to compute tool track.

(1) Inside and Outside

Inside:



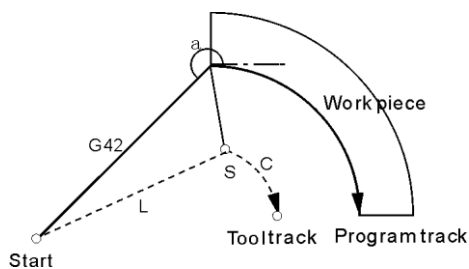
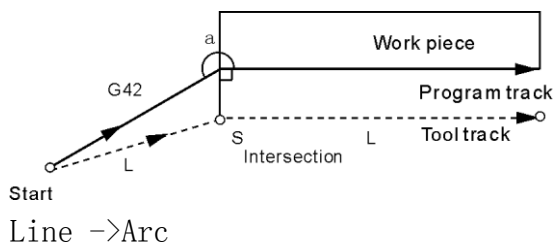
Outside:



(2) Establish tool radius compensation

(2.1) ($\alpha \geq 180$)

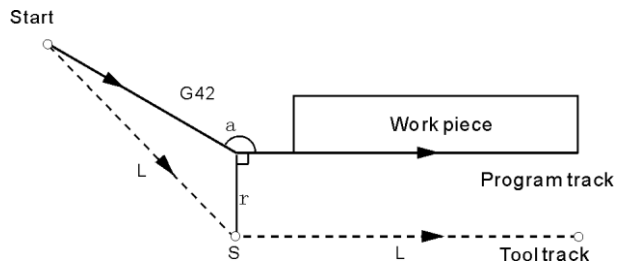
Line → Line



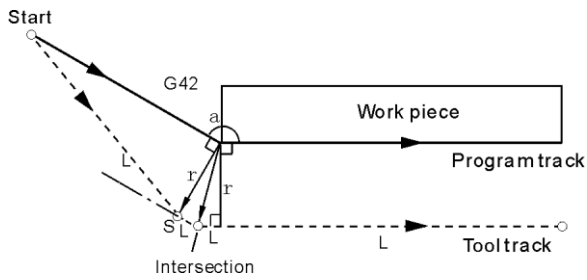
(2.2) ($90 \leq \alpha \leq 180$)

Line → Line

A type

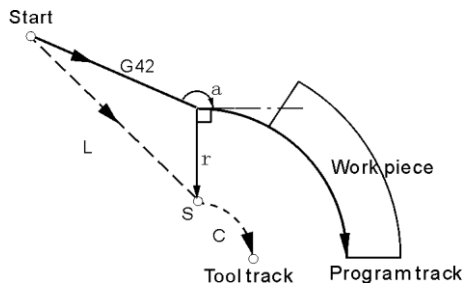


B type

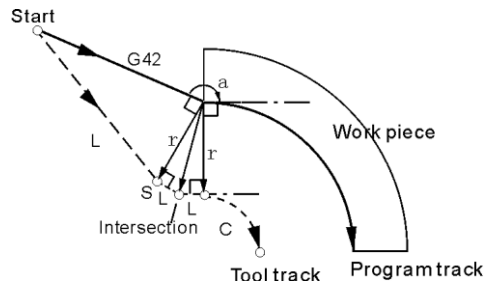


Line ->Arc

A type

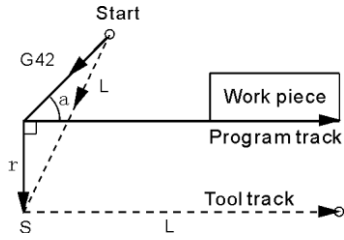


B type

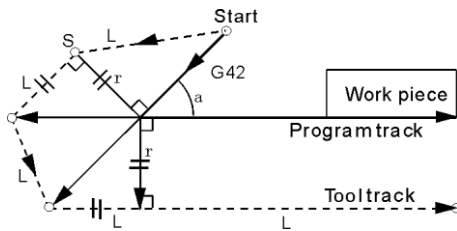


(2.3) ($\alpha \leq 90$)

Line→ Line
A type

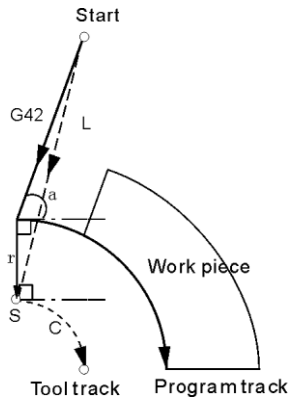


B type

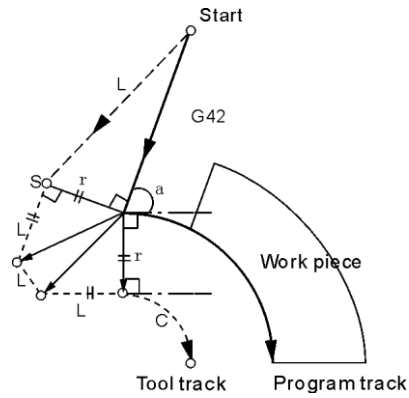


Line→Arc

A type



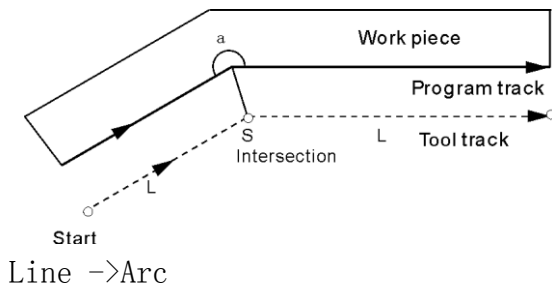
B type



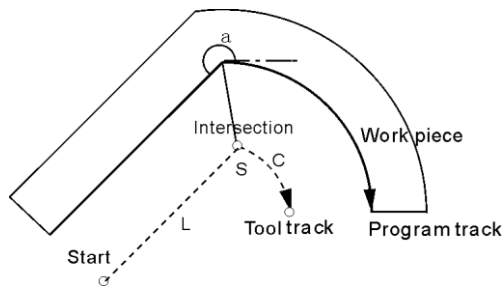
(3) Tool track compute in the course of tool compensation

(3.1) ($180 \leq \alpha$)

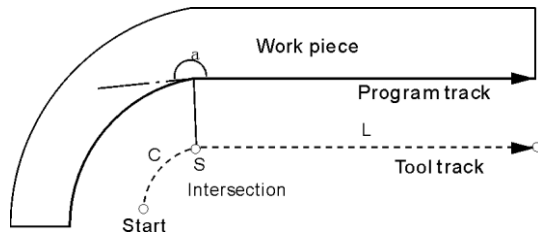
Line→Line



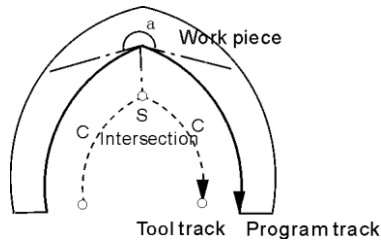
Line →Arc



Arc→ Line

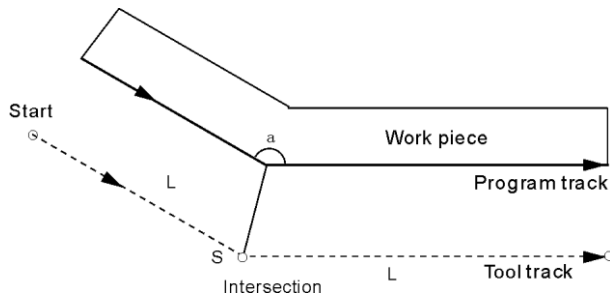


Arc→Arc

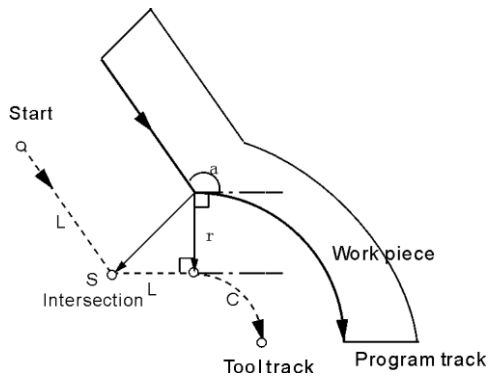


(3. 2) ($90 \leq \alpha < 180$)

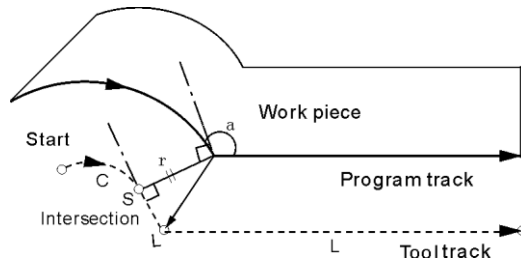
Line → Line



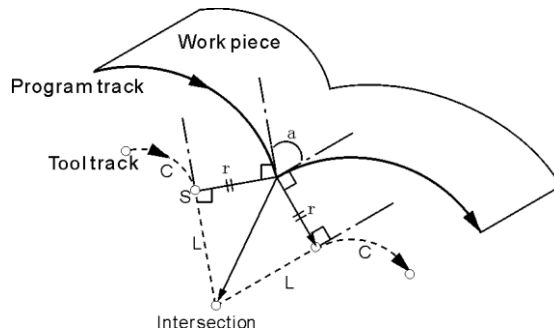
Line →Arc



Arc→ Line

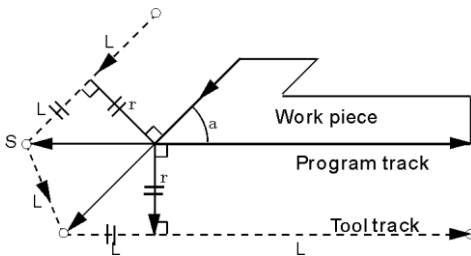


Arc → Arc

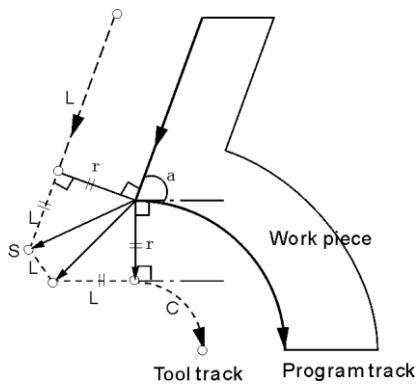


(3.3) ($\alpha < 90$)

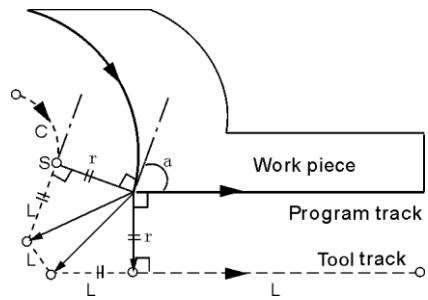
Line → Line



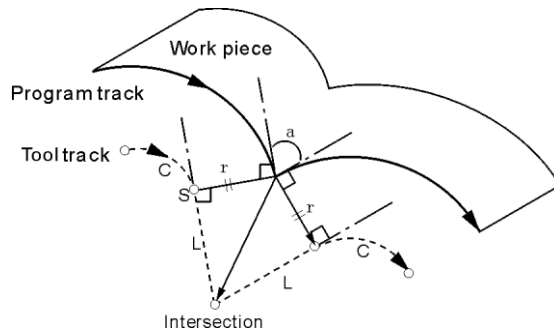
Line → Arc



Arc → Line



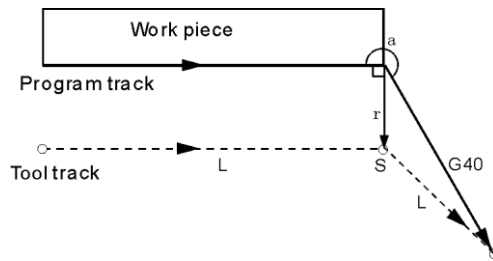
Arc→Arc



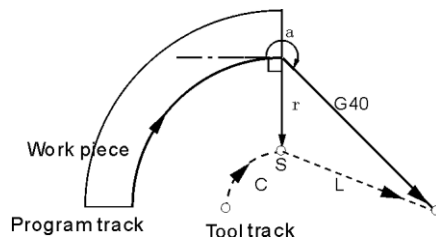
(4) Cancel tool radius compensation

(4.1) ($180 \leq \alpha$)

Line→Line



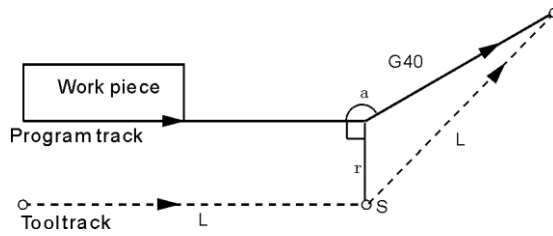
Arc→Line



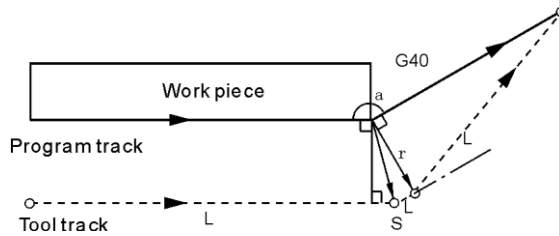
(4.2) ($90 \leq \alpha < 180$)

Line—>Line

A Type

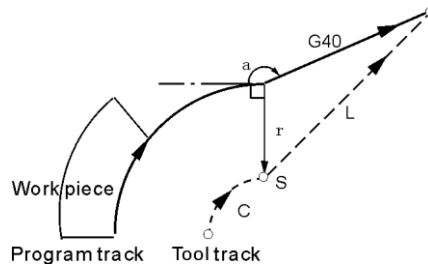


B type

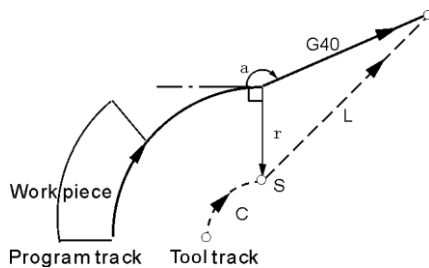


Arc—>Line

A type



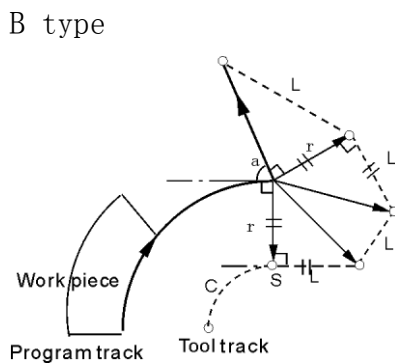
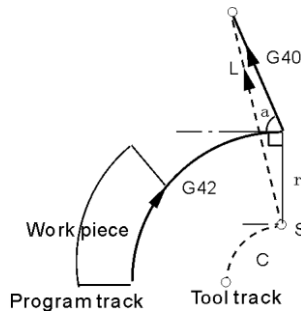
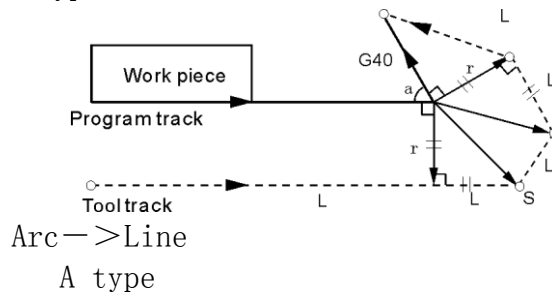
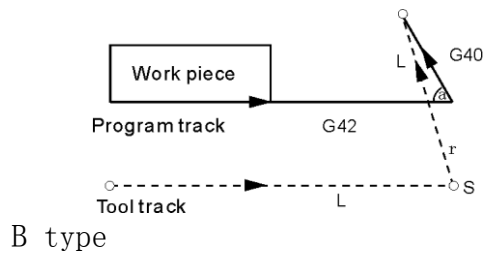
B type



(4.3) ($\alpha < 90$)

Line—>Line

A type



4.3.18 program circulation instruction (G22--G800)

This instruction is used for realizing program which have repeatable motion and track.

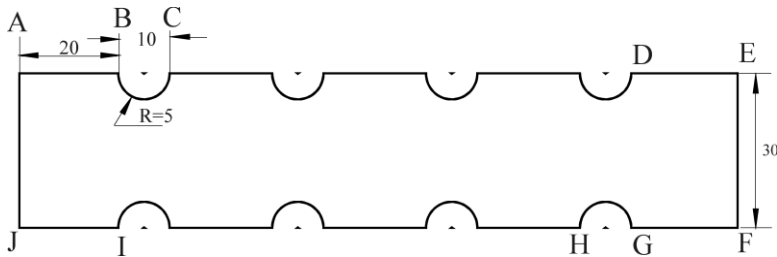
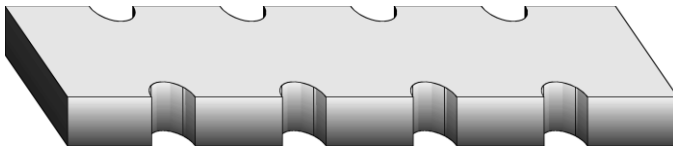
```

Format:  G22 L2                ;begin
          :      }              ;body
          :
          :
          G800                  ;end

```

Note: G22 and G800 are used by the way of partnership, moreover, we can embed another circulation or sub-program in the between of G22 and G800, the time of circulation is determined by L word.

for example 3.5



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

4.3.19 accurate localizations/Continual way processing (G60/G64)

According to adds the craft the request, may pass G60/The G64 instruction assigns between the segment the connection way.

Instruction format: G60; Accurate localization (modality) G64; Continual way processing (modality, initial state)

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

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

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

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

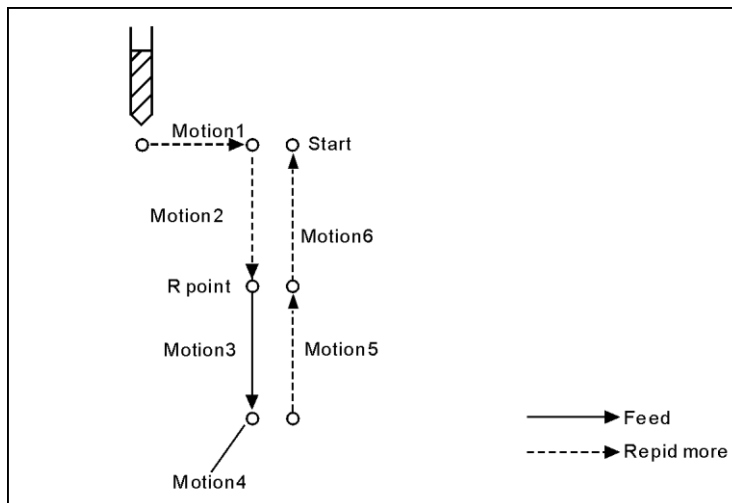
Circle instruction table

G code	Feed method	Motion in the bottom of hole	withdraw	application
G73	Intermission feed	No	Rapid move	High speed drill deep hole
G74	Continue feed	Stop-Spindle ClockWise	Cutting feed	Left tap cycle
G80	Continue feed	No	No	Cancel cycle
G81	Continue feed	No	Rapid move	Drill cycle
G82	Continue feed	Stop	Rapid move	Drill cycle
G83	Intermission feed	No	Rapid move	Drill deep hole cycle
G84	Continue feed	Stop-Spindle ClockWise	Cutting feed	Tap cycle

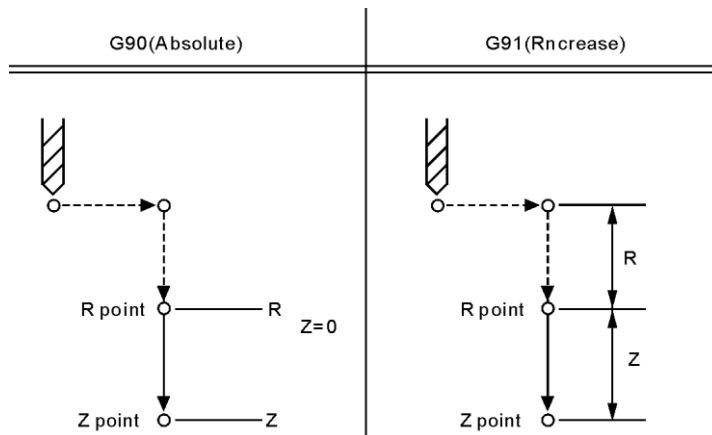
G85	Continue feed	No	Cutting feed	Drill hole cycle
G86	Continue feed	Spindle stop	Rapid move	Drill hole cycle
G87	Continue feed	Spindle ClockWise	Rapid move	Drill hole cycle
G89	Continue feed	Spindle stop	Cutting feed	Drill hole cycle

Cycle instruction is consist of six motions

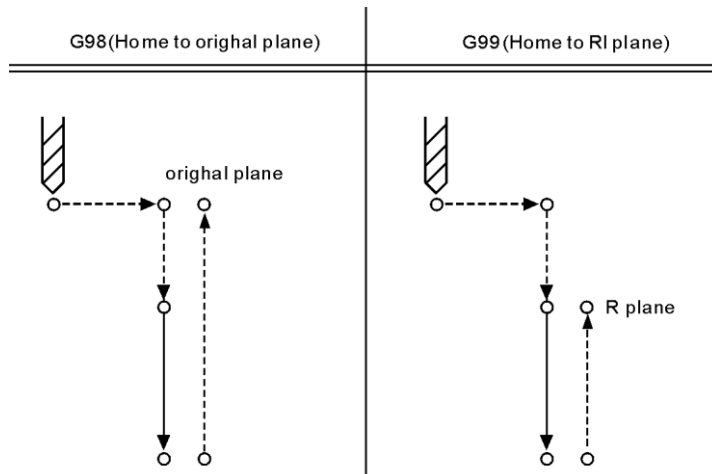
- Motion 1 location of X and Y axis
- Motion 2 rapid move to R point
- Motion 3 machining hole
- Motion 4 action in the bottom of hole
- Motion 5 withdraw to R point
- Motion 6 rapid move to original point



The difference of G90 and G91 as follow:



The difference of G98 and G99 as follow:



Use the L word to set cycle time, the maximum value is 9999, the default value is 1;

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

4.3.20.1 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 timw
 F-:cutting speed
 L-:repeat time

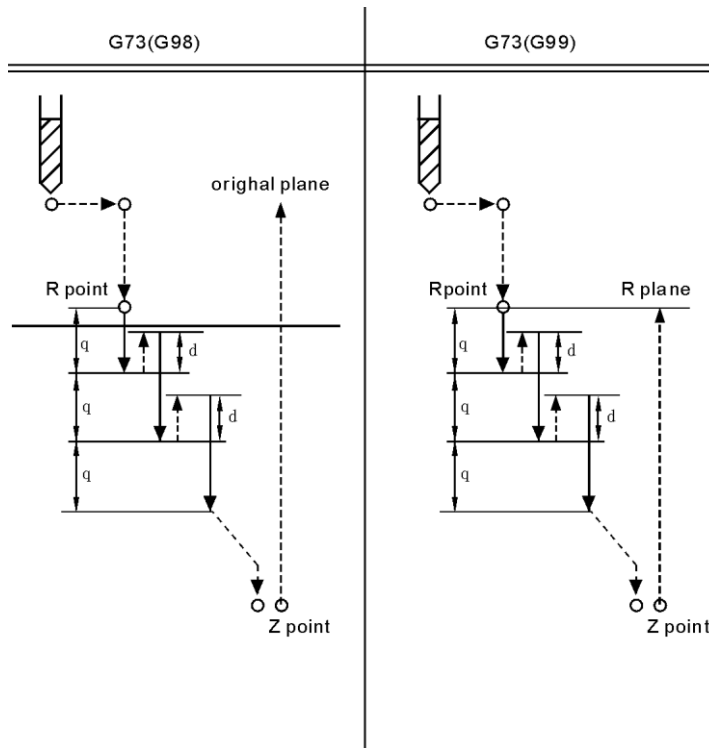


图 4.24

Note:

Please set the withdraw d in the other paramter

Please use M03 to rotate SP before G73 instruction.

For example:

M3 S2000

G90 G99 G73 X300. Y-250. Z-150. R-100. Q15. F120.

Y-550. ;

Y-750. ;

```

X1000. ;
Y-550. ;
G98 Y-750. ;
G80
M5;

```

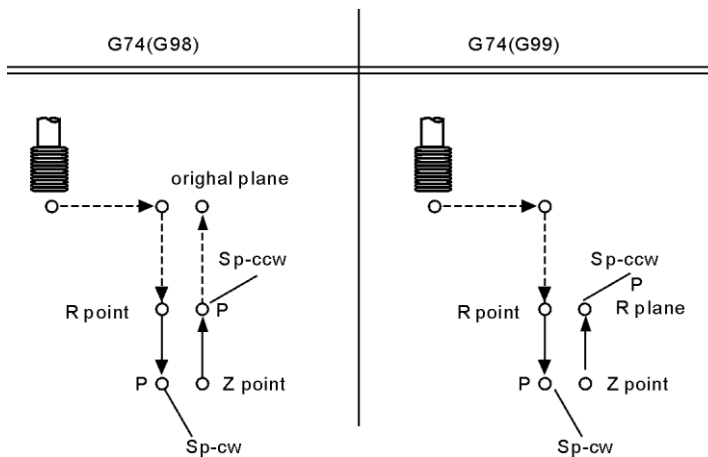
4.3.20.2 Left tap cycle(G74)

This instruction inquire the spindle drive have function of pulse cotrol.

```

format: G74 X-Y-Z-R-P-K-S-L- ;
      X-Y-: hole position data
      Z-: the distance(G91) or coorinate(G90)
from R point to hole bottom
      R-: the distance(G91) or coorinate(G90)
from original point to R point
      P-: pause time
      K-: screw paramter
      S-: spindle rotate speed
      L-: repeat time

```



For example:

```

M4 S100
G90 G99 G74 X300. Y-250. Z-150. R-100. K5 S100
Y-550. K5;
Y-750. K5;
X1000. K5;

```

```

Y-550. K5;
G98 Y-750. K5;
G80
M5;

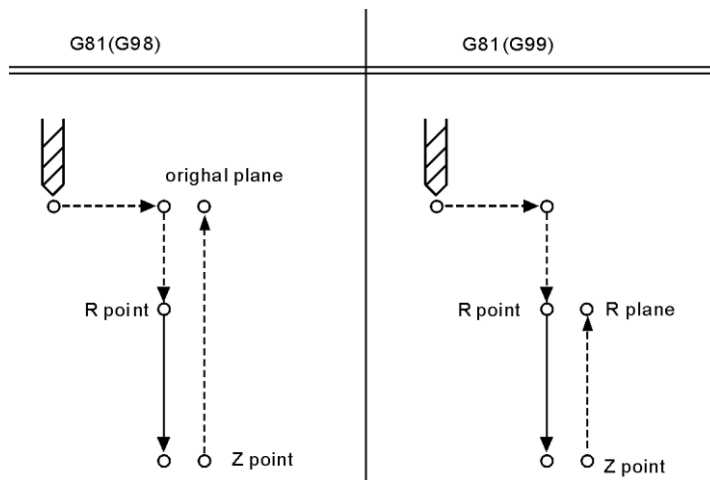
```

4.3.20.3 Drill cycle(G81)

```

format: G81 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-:repeat time

```



For example:

```

M3 S2000
G90 G99 G81 X300. Y-250. Z-150. R-100. Q15. F120.

```

```

Y-550. ;
Y-750. ;
X1000. ;
Y-550. ;
G98 Y-750. ;
G80
M5;

```

4.3.20.4 drill cycle(G82)

format: G82 X-Y-Z-R- P-F-L- ;

X-Y-: hole position data

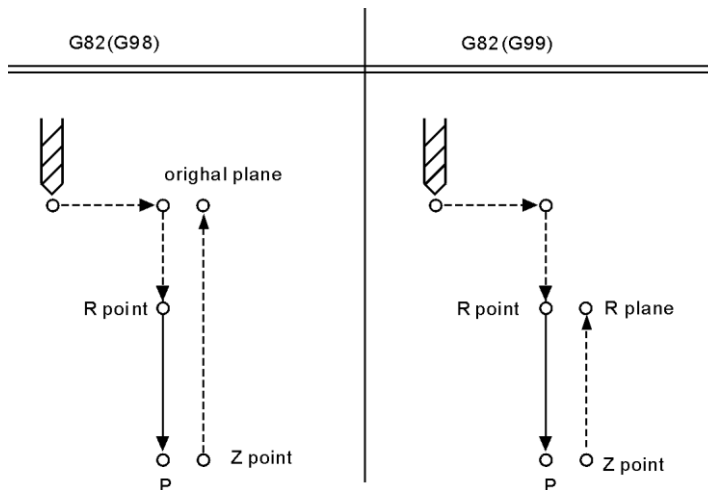
Z-: the distance(G91) or coordinate(G90)
from R point to hole bottom

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

P-:pause time

F-:cutting speed

L-:repeat time



For example:

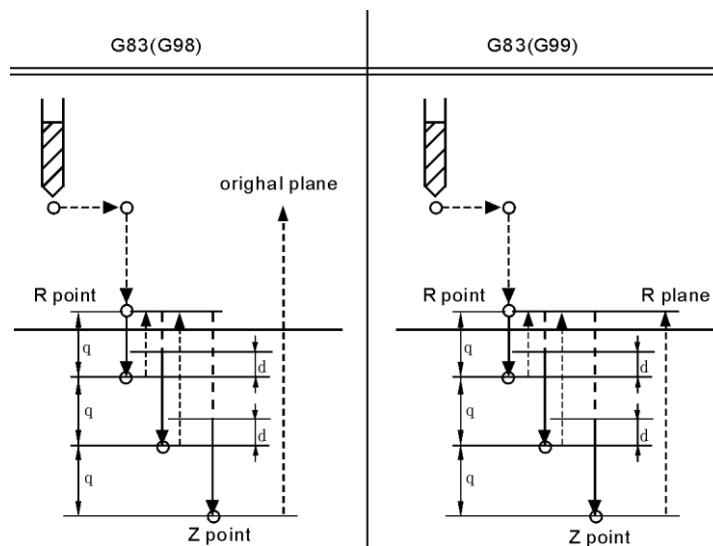
```

M3 S2000
G90 G99 G82 X300. Y-250. Z-150. R-100. P1000 F120.
Y-550. ;
Y-750. ;
X1000. ;
Y-550. ;
G98 Y-750. ;
G80
M5;

```

4.3.20.5 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-:repeat time



For example:

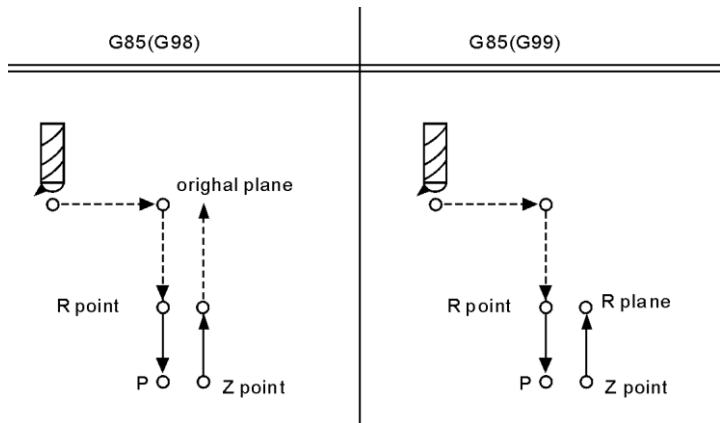
```
M3 S2000
G90 G99 G83 X300. Y-250. Z-150. R-100. Q15. F120.
Y-550. ;
Y-750. ;
X1000. ;
Y-550. ;
G98 Y-750. ;
G80
M5;
```

4.3.20.6 Right tap cycle(G84)

The condition is that spindle must have encode feedback or servo function.

In this cycle, spindle will counter clockwise rotate as reaching

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-:repeat time



for example:

```
M3 S100
G90 G99 G85 X300. Y-250. Z-150. R-120. F120.

Y-550. ;
Y-750. ;
X1000. ;
Y-550. ;
G98 Y-750. ;
G80
M5;
```

4.3.20.8 Drill cycle(G86)

For example:

```
M3 S100
G90 G99 G88 X300. Y-250. Z-150. R-120. F120.
Y-550. ;
Y-750. ;
X1000. ;
Y-550. ;
G98 Y-750. ;
G80
G28;
M5;
```

4.3.21 Pole coordnate instruction(G15/G16)

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

Format :

```
G15 Cancel Pole coordnate;
(G17/G18/G19) (G90/G91) G16 IP- ;establish
```

Note:

G17/G18/G19 point out the selection of plane.

Set current position as pole coordnate original point. As follow:

Note: G26 movtion is according to linkage type.

4.3.24 Check skip(G31、 G311)

Format: G31 X_Y_Z_A_ F_ P_ ;No alarm

G311 X_Y_Z_A_ F_ P_ ;alarm

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

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

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

4.3.25 Call sub-program (M97/M98/M99)

M97 P Non-condition to jump to P word

M98 P L Call sub-program. P word point out the name of sub-program.

for example: Psub\\%ab12 means the name of sub-program is

CNC\\sub\\%ab12, L word point out call times.

M99 Back of sub-program

The Sub-program can embedded call as follow :

For example

4.3.26 S、 SS SP speed

The first SP use “S” ,speed parameter P42 control the highest speed,output 0-10V frequency conversion voltage.

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, 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, weak 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 earthing.

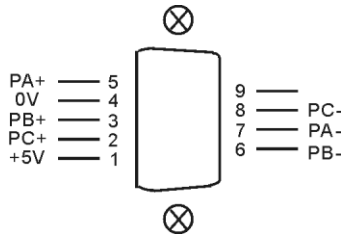
Fix all plugs with screw. Forbid to insert and extract all cables when power is on.

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.

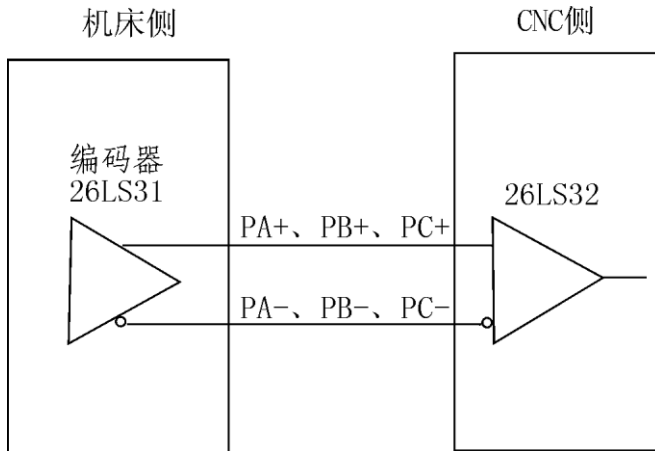
5.2 system installation dimension

This system has two types of installation, except that the installation dimension are different, the other functions are same.



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	

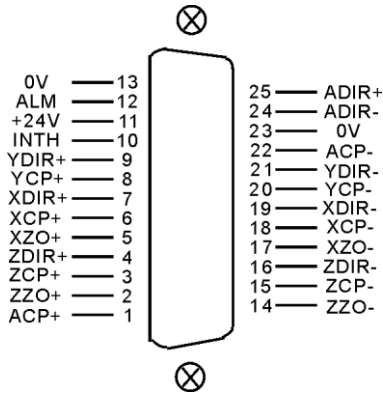
Encode input signal PA、PB、PC:



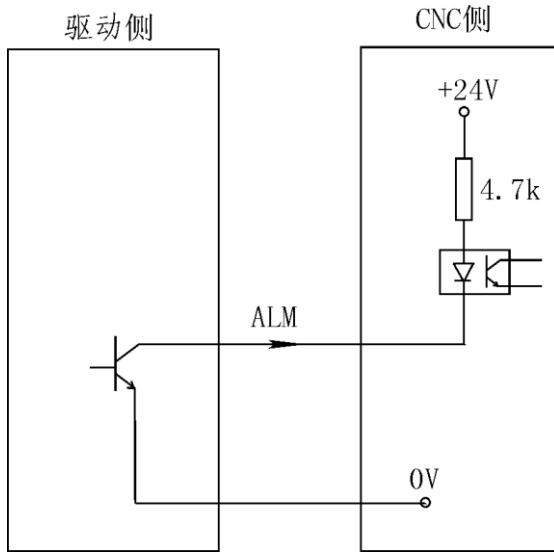
5.4.2 CN6 and computer system connection

			signal	
+10V	25	OUT	the second spindle converting	0~10V
0V	13	OUT	0V	0V

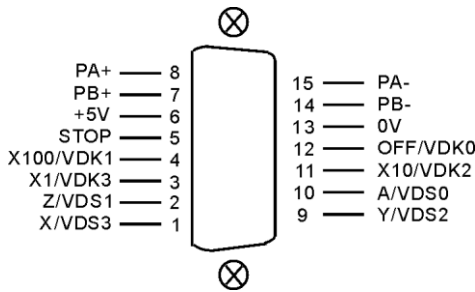
5.4.5 CN5 and servo drive & motor connection



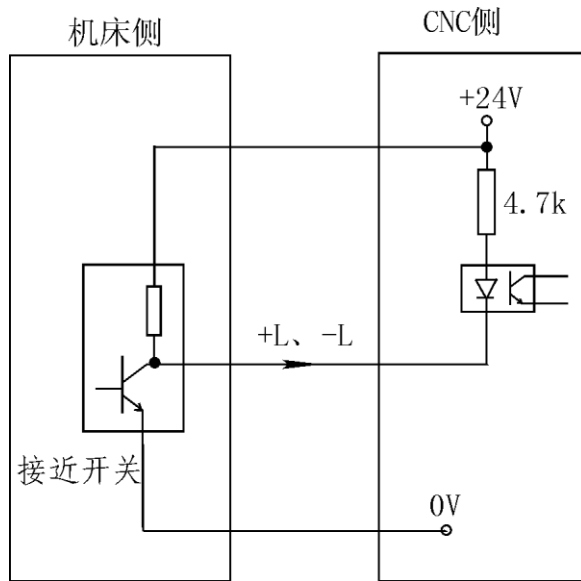
CN5 DB25(pin) servo drive signal				
signal	pin	I/O	Function	Availability
XCP+	6	OUT	X pulse signal +	5V
XCP-	18	OUT	X pulse signal -	
XDIR+	7	OUT	X direction signal +	5V
XDIR-	19	OUT	X direction signal -	
YCP+	8	OUT	Y pulse signal +	5V
YCP-	20	OUT	Y pulse signal -	
YDIR+	9	OUT	Y direction signal +	5V
YDIR-	21	OUT	Y direction signal -	
XZO+	5	IN	X motor Zero +	5V
XZO-	17	IN	X motor Zero -	
ZCP+	3	OUT	Z pulse signal +	5V
ZCP-	15	OUT	Z pulse signal -	
ZDIR+	4	OUT	Z direction signal +	5V
ZDIR-	16	OUT	Z direction signal -	
ZZO+	2	IN	Z motor Zero +	5V
ZZO-	14	IN	Z motor Zero -	
ACP+	1	OUT	A pulse signal +	5V
ACP-	22	OUT	A pulse signal -	
ADIR+	25	OUT	A direction signal +	5V



5.4.6 CN11 and hand wheel, band switch connection



CN11 DB15(pin) hand wheel, band switch connection				
signal	pin	I/O	function	Availability
0V	13	OUT	0V	0V
+5V	6	OUT	+5V	+5V
PA+	8	IN	A signal +	5V
PA-	15	IN	A signal -	
PB+	7	IN	B signal +	5V
PB-	14	IN	B signal -	



Mode2: general switch

Chapter 6 System's daily maintenance and repair

In order to plenty use CNC system's function and promote efficiency, the most important work is correctly using system, and notice system's daily maintenance work, promote Mean Time Between Failures MTBF. Now this system's maintenance method is introduced as follows:

6.1 System's maintain

6.1.1 System's using must be under the good circumstance.

6.1.2 Operator、programmer and repairer must be familiar with NC machining technology, and according the require of user book correctly use, do one's best to avoid improper operation.

6.1.3 Everyday operator should clean the system's box and panel in case for corrupt thing and sundries to damnify it.

6.1.3 When CNC system's using time is over three month, operator should open the system box and clean inside.

6.1.4 If not using system for long time, should boot the system one time every week.

6.2 Ordinary trouble

6.2.1 System can't boot

- 1) check if power is normal.
- 2) check if power switch is turn on.
- 3) check insurance.

6.2.2 No display as boot

- 1) Boot again or reset.
- 2) Check if switch power's +5V、+12V、-12V、-24V are normal.
- 3) Check if transformer is bad.
- 4) Check if LCD's bright adjust and connection are normal.
- 5) Check if computer main board is normal.

Chapter 7 Appendix

Appendix: binary、decimal switch table (0 — 15)

decimal	binary D7D0		decimal	binary D7D0
0	0 0 0 0 0 0 0 0		8	0 0 0 0 1 0 0 0
1	0 0 0 0 0 0 0 1		9	0 0 0 0 1 0 0 1
2	0 0 0 0 0 0 1 0		10	0 0 0 0 1 0 1 0
3	0 0 0 0 0 0 1 1		11	0 0 0 0 1 0 1 1
4	0 0 0 0 0 1 0 0		12	0 0 0 0 1 1 0 0
5	0 0 0 0 0 1 0 1		13	0 0 0 0 1 1 0 1
6	0 0 0 0 0 1 1 0		14	0 0 0 0 1 1 1 0
7	0 0 0 0 0 1 1 1		15	0 0 0 0 1 1 1 1

Note: Because of many kinds of reasons this Manual book may have some mistakes. Our company will provide the high quality service and the technical support for every customer.