CONTENTS

Chapter 1 Preface

This CNC control system is one middle class flush type CNC control system, aiming sepcially 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 convertion speed control.

Manual pulse generator.

2.2 System technical parameter

controllable axes: X、Z、C/Y、A、B five axes.

linkage axes: Arc 2-3 axes, liner 2-5 axes.

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

max speed: X、Z、C/Y、A、B:60000mm/min.

cutting speed: 1-10000mm/min.

min input unit: 0.001mm.

program size range: ± 99999.999.

99 tools management.

controllable liner vertical type or revolving disc type tool changer.

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.
mutiple tool offset instruction.
program cycle, jump, call and different program ending.
multiple positioning instruction: starting point, setting fixed point,etc.
linera, circular, spiral line interpolation instruction.
program management instructions: program cycle, call, transfer and different program ending method, etc.
6 workpeices coordinates system .

2.3.4 Chineses/English menu, full screen edition

Easy operation, conveinent viewing.

2.3.5 Abundant debugging functions

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

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

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

2.4 System operation condition

2.4.1 Power supplying

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

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

2.4.2 Climate condition

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

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

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

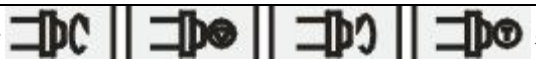
switch introduction:

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

buttons:

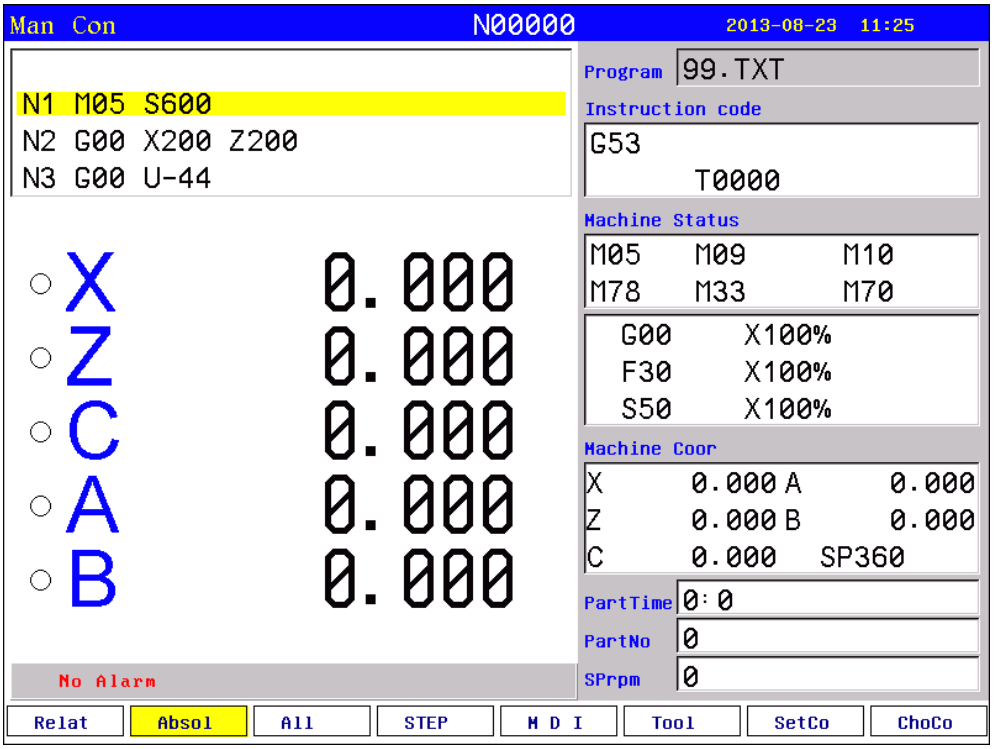
Keyboards	Functions
Letter key Number key	ABCSEFGHIJKLMNOPQRSTUVWXYZ123456789.-: for program instructions, parameters' edition; number keys are used for inputting data and selecting sub-menu.
Edit key	"↑、↓、→、←、Del、PgUp、PgDn" for programming, direction keys can be used for selecting menu.

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

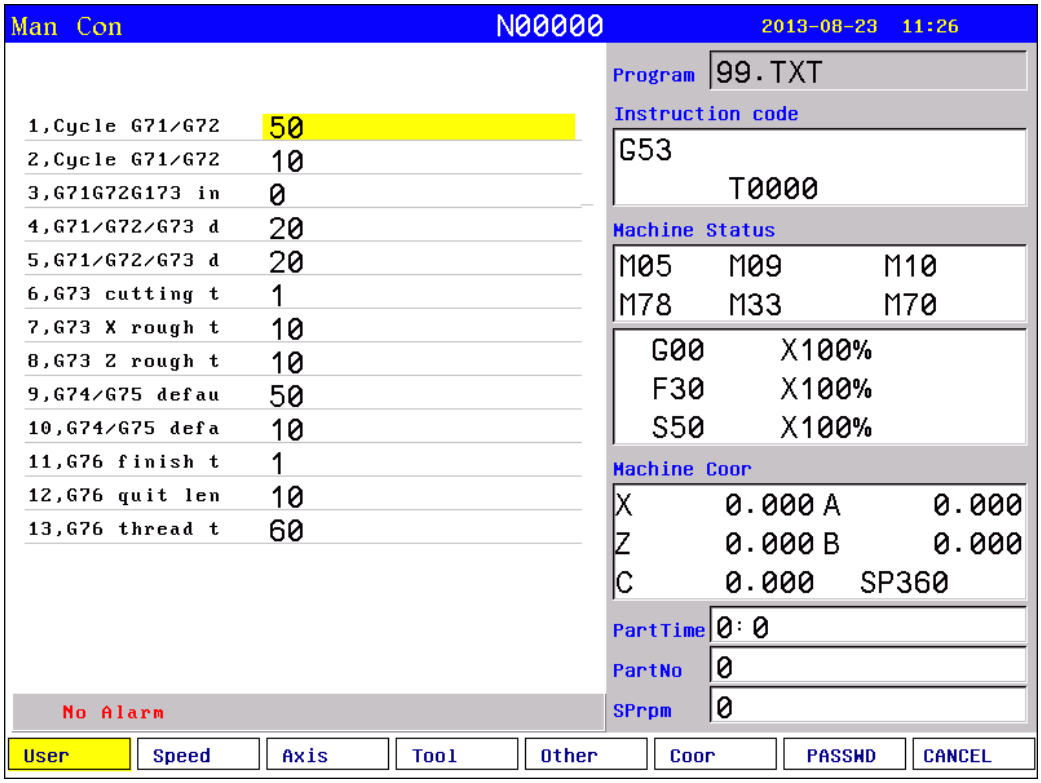
Control key	“  ” spindle cw, ccw rotation “  ” coolant on/off “  ” for the shift between electric tool carrier and gang tool carrier “  ” for the shift between hand-driven continuous high speed and low speed. “  ” all axes return to datum point “  ” for spindle chuck on/off “  ” for lubrication on/off “  ” for thumbstall on/off “  ” adjusting spindle speed “  ” adjusting feed speed “  ” adjusting G00 speed	
Feed key	 +C +A - C - A+ B-B	For X、Y/C、Z 、A 、 B 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:



3.3 Parameters



In main menu, pressing "Parameter" function key, it enters para setting status, including "User", "Speed", "Axis", "Tool", "Other", "Coor", "Passwd", seven function. Choose pressing "F1、F2、F3、F4、F5、F6、F7、F8" choose Except for special note, all data are using mm.

Except for special note, all data are using mm.

3.3.1 User parameter

1,Cycle G71/G72 default feed thickness(10um)

[X axis radius]

Cycle G71 X axis feed thickness; Cycle G72 Z axis feed thickness;

2,Cycle G71/G72 default backward distance(10um)

[X axis radius]

Cycle G71 X axis backward thickness; Cycle G72 Z axis backward thickness.

3,G71G72G173 instruction

[1 mean Yes,0 mean No]

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

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

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

6,G73 cutting times
7,G73 X rough thickness(10um)
[X axis radius]
8,G73 Z rough thickness(10um)
9,G74/G75 default feed thickness(10um)
[X axis radius]
10,G74/G75 default backward distance(10um)
[X axis radius]
11,G76 finish turn times
12,G76 quit length(1/10 lead)
13,G76 thread tooth angle(degree)
[0～180°]
14,G76 minimal cutting depth(10um)
[X axis radius]
15,G76 finish turn remaining(10um)
16,X program mode
[1 mean Radius,0 mean Diamete]
17,Running program need Sp run
[1 mean Yes,0 mean No]
18,Set M20 the time of auto-running
[Neqative mean immensity loop]
19,Set part count
20,G92 quit length(1/10 lead)
21,G01/G02/G03 line delay(ms)[>100]
22,G00 line delay(ms)[>100]
23,Handwheel acceleration[50-100]
24,G76 Q
[8 mean thick forward number]
200,system screen protect times
[>=2minutes]
201,G92/G76 delay time(ms)[>100]
202,system inner parameter

3.3.2 Speed parameter

1,X-axis's G00 speed(mm/min)
2,Z-axis's G00 speed(mm/min)
3,Manual maxminum feed speed(mm/min)
4,Auto Maximum feed speed(mm/min)

5,G01/G02/G03 default speed(mm/min)
6,Null run speed(mm/min)
7,Feed axis`s manual speed(mm/min)
8,Spindle`s manual speed(rpm)
9,Beginning feed speed(mm/min)
10,Jump speed at continuous track(mm/min)
11,Limit G1G2G3 axis speed
[1 mean Yes,0 mean No]
12,X G1G2G3 max speed(mm/min)
13,Z G1G2G3 max speed(mm/min)
14,X acceleration
[1～99999]
15,Z acceleration
[1～99999]
16,Auto run acceleration
[1-500]
17,Handwheel acceleration
[500--30000]
18,Run program Handwheel acceleration
[>500]
19,Run program Handwheel G00 speed(mm/min)
[>10]
20,Handwheel X limit speed(mm/min)
21,Handwheel Z limit speed(mm/min)
22,Make thread Z acceleration
[Servo motor is 0]
23,Make thread X acceleration
[Servo motor is 0]
24,Servo motor screw thread X axis Back speed
25,Step motor screw thread X axis Back rise speed
26,Step moter screw thread X axis Back Max speed
27,acceleration type
[0 mean line,8 mean curve]
28,curve ini acceleration
[>=10]
29,curve acceleration
[>=10]
30,curve max acceleration

[>=500]
31,X go home rampit speed(mm/min)
32,X go home reverse speed(mm/min)
33,Z go home rampit speed(mm/min)
34,Z go home reverse speed(mm/min)
35,G96 spindle min speed(rpm)
36,Spindle first max speed(rpm)
37,Spindle second max speed(rpm)
38,Spindle third max speed(rpm)
39,Spindle forth max speed(rpm)
40,Second Spindle max speed(rpm)
41,G02/G03reverse compensation mode(0 mean A; 8 mean B)
42,mode B reverse compensation speed(mm/min)
42-1,mode B reverse compensation Beginning feed speed(mm/min)
42-2,mode B reverse compensation acceleration(mm/min/s)
43,Handwheel stop speed(mm/min)[>100]
58,Forcedly 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,Y-axis`s negative scope(mm)
6,Y-axis`s positive scope(mm)
7,Z-axis`s negative scope(mm)
8,Z-axis`s positive scope(mm)
9,A-axis`s negative scope(mm)
10,A-axis`s positive scope(mm)
11,Spindle stop time(10ms)
12,Spindle stop long signal
[0 mean No,1 mean Yes]
13,Soft limit invalid
[D2X;D3Y;D4Z;D5A;0 inavaild,0 availd]
14,X-axis`s reverse compensation(um)
15,Y-axis`s reverse compensation(um)
16,Z-axis`s reverse compensation(um)
17,A-axis`s reverse compensation(um)
18,X-axis's direction signal

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

[1 mean open,0 mean close]
51,Spindle class speed(1/100rpm)
52,Spindle class direction
[0 mean M03,1 mean M04]
53,Spindle class stop time(10ms)
54,Spindle class time(10ms)
55,Spindle stop time(10ms)
56,Check SP encode
[1 mean Yes,0 mean No]
57,SP encode pulse
80,XZ axis coordinate plan[D2Zwordpiece,D3Xwordpiece,D4Ztool,D5Xtool,D6Zcircumrotate,D7Xcircumrotate]
81, Y axis
[0 mean circumrotate axis,1 mean line axis]
82,Y is circumrotate axis work coordinate
[0 No;1 plan]
83,Y is circumrotate axis machine coordinate
[0 No;1 plan]
100,Inner paramter
101,A-axis function
[0 mean circumrotate axis,1 mean line axis]
102,A-axis is circumrotate axis machine coordinate
[0 No;1 plan]
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,95 B]
411,Interpolation tap mode[2 follow encode;3 interpolation to SP]
412,SP tooth number(<P413)
413,Encode number(>P412)

3.3.4 Tool parameter

1,Active tool function
[1 mean Yes,0 mean No]
2,Active tool number
3,Lather type
4,Tool positive rotate max-time(s)
5,Delay time after tool positive rotate(ms)
6,Delay time after tool stop(ms)

7,Tighten time of tool reverse rotate(ms)
9,Have total signal TOK(1 mean have)
10,C Tool radius compensation's establish(0 mean A,1 mean B)
11,C Tool radius compensation's cancel(0 mean A,1 mean B)
20,Active tool mode
[1 mean normal,0 mean coding tool]

3.3.5 Other parameter

1,Set sub-panel type
[1 hand hold,0 panel]
2,lather outside chuck
[1 extroversion,0 diffidence]
3,use control switch
[1 Yes, 0 No]
4,Have auto lubricate(0 yes/1 no)
5,Auto lubricate time(0.01s)
6,Auto lubricate stop time(s)
7,Door switch checking M12(0 no,1 yes)
8,Door switch(0 open,1 close)
9,bit paramter
D0: Null;
D1:“1”Start cnc system clear part Number.;
D2:“1”Automatic space before letter when editting program;
D3: Null;
D4: Null;
D5:“1”Do not stopping SP and cooling when pressing “Restet”;
D6:“1”G00 XZ’ speed by oneself;
D7:“1”Tool’ redeem by oneself;
D8:“1”Save SP chuck(M10/M11) state when power off;
D9: Tool redeem input Mode1 or Mode2;
D10:“1” Program edit automatic compositor Line;
D11:“1”Frist SP +10V output from second output port;
D12:“1”Shield skip function (“/”is invalidation);
D13:“1”Sheld go home function;
D14:“1”Sheld “run” key;
D15:“1” Tool redeem display relative,“0”absolute;
10,Auto count part
[1 mean Yes,0 mean No]

11,Program edit number increase
12,Inner paramter
13,Does lock for Spindle & chuck(0 mean no)
14,Is availabe keys of lub&cool as runing(0 mean no)
15,Chuck clamp M10/loose M11 checking(1 mean need)
16,Finial forward M79/backward M78 checking(1 mean need)
17,servo ALM1 (0 open,1 close)
18,SP ALM2 (0 open,1 close)
19,Tool ALM3 (0 open,1 close)
20,Chuck control signal(0 single,1 doubleM10/M71)
21,finial control signal(0 single,1 doubleM79/M73)
22,Outside chuck control(0 no,1 yes)
23,Outside finial control(0 no,1 yes)
24,M10M11 short signal time(s)
25,M79M78 short signal time(s)
26,Emerge Stop(0 open,1 close)
27,Emerge Stop2(0 open,1 close)
28,Run status output M69 STOP output M65(0 invalid,1 valid)
29,Alarm status output M67(0 invalid,1 valid)
30,Set language(1表中文, 0 mean English)
31,Is enable I/O PLC program
32,Is enable High speed I/O PLC program
33,HY make run signal
[1 mean Yes,0 mean No]
34,HA make stop signal
[1 mean Yes,0 mean No]
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,C(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,C(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,C(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,C(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,C(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,C(Y) of work coordinates G59(mm)
28,Z of work coordinates G59(mm)
29,A of work coordinates G59(mm)

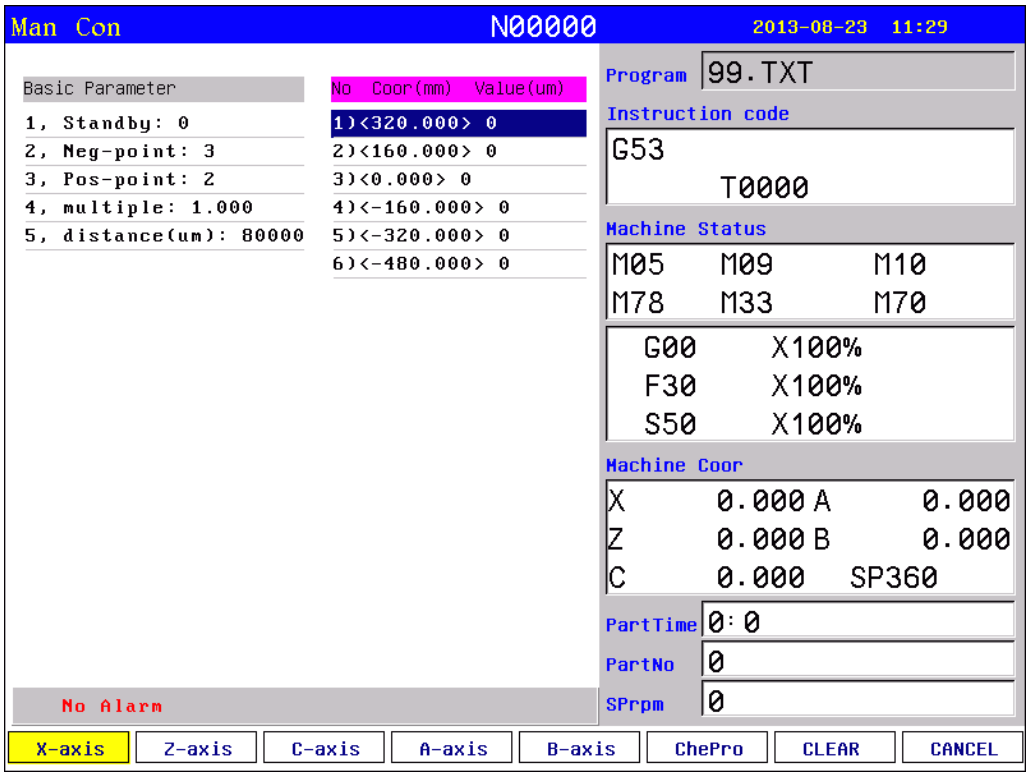
3.3.7 Password

password setting includs:
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:



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 cmpensation point can be set optionally, Compensation parameters include:

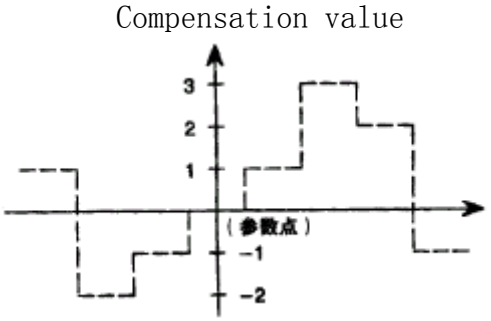
Compensation point NO. of reference point.

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) .



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

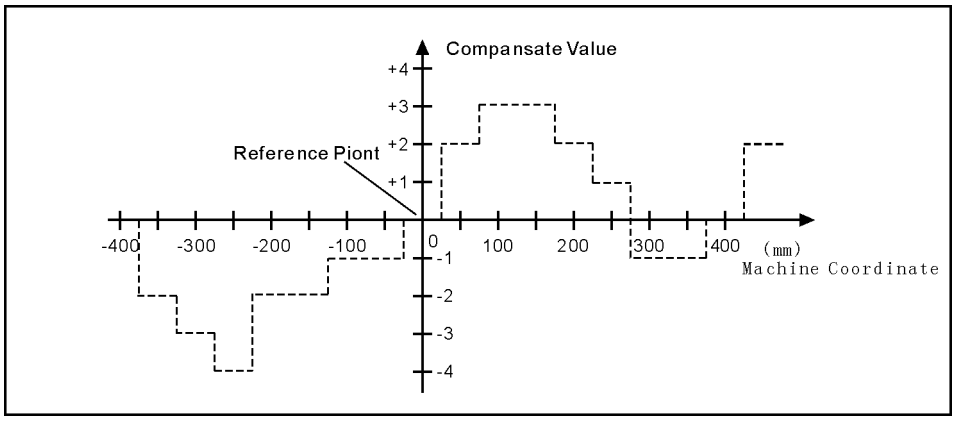
Com.point NO.of farest end in negative direction:30

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

Compensation percentage:1

Compensation point interval:50000

Compensation point and value contrast:

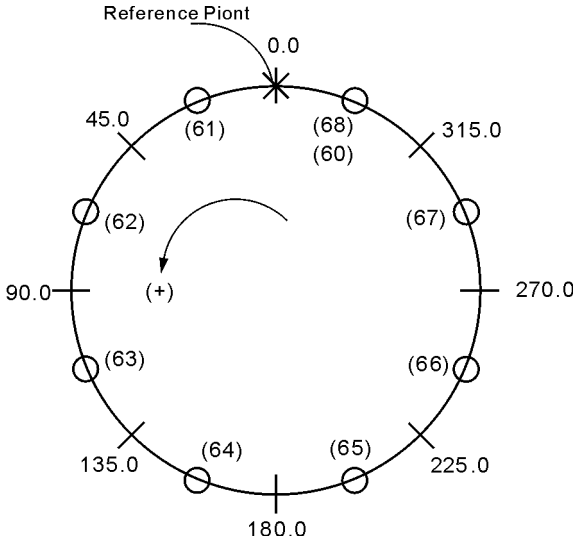


Example 2:rotor axis: when movement per revolution is 360°, interval of points 45°,reference point compensation NO. 60, Com.point NO.of farest end in negative direction is usually same as reference point com.point NO.

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

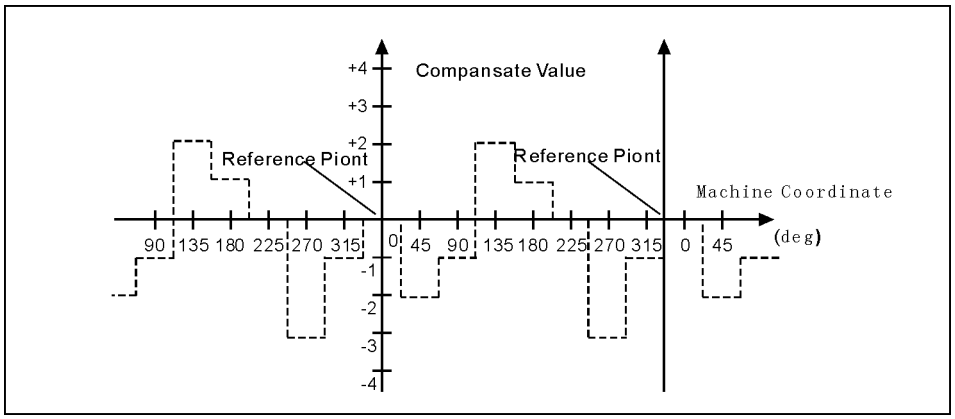
Com.point NO.of farest end in negative direction:60

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

Compensation percentage:1
Compensation point interval:45000
Output compensation value at corresponding point:

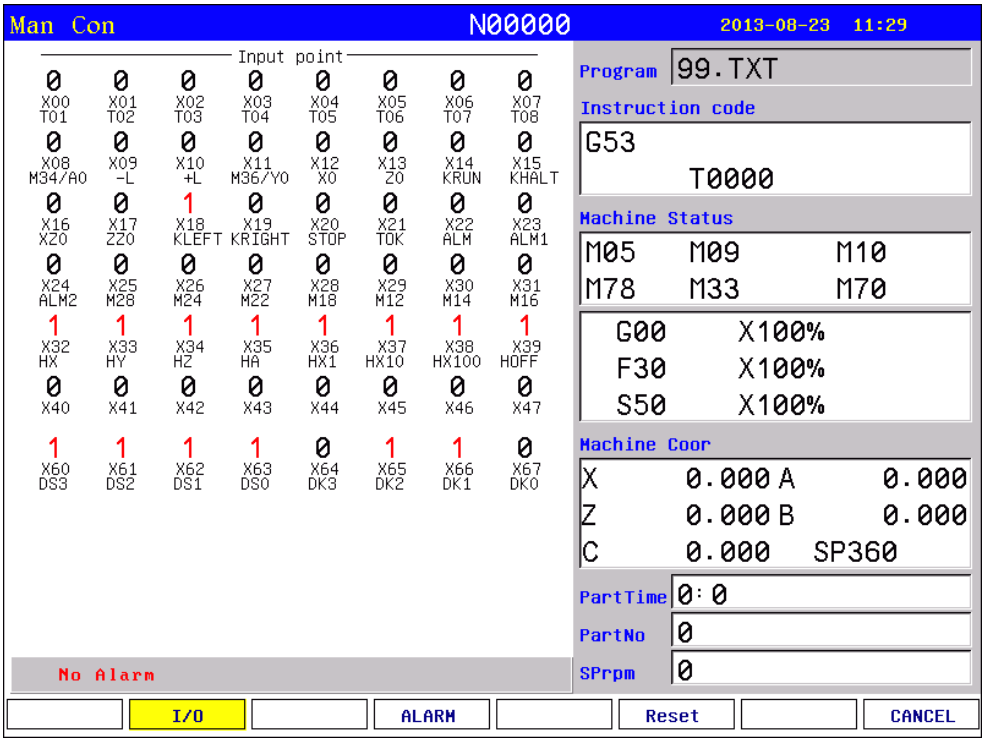
NO.	60	61	62	63	64	65	66	67	68
VALUE	+1	-2	+1	+3	-1	-1	-3	+2	+1

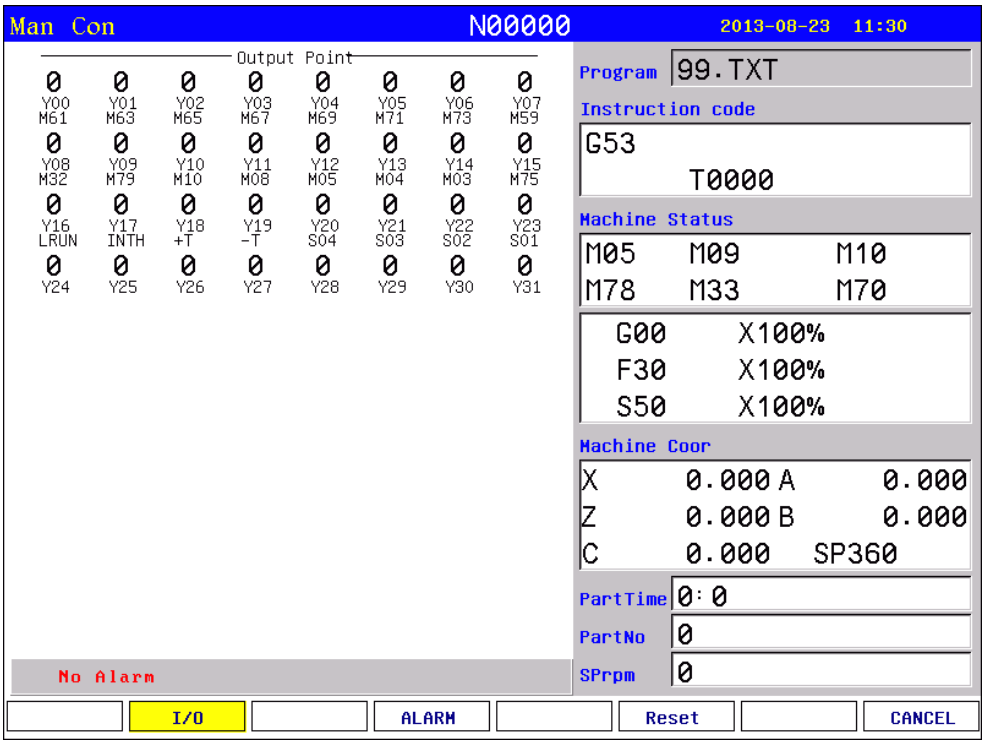
Compensation point and value contrast:



3.3.9 Input/output diagnosis

Presses “Diagnosis” key :

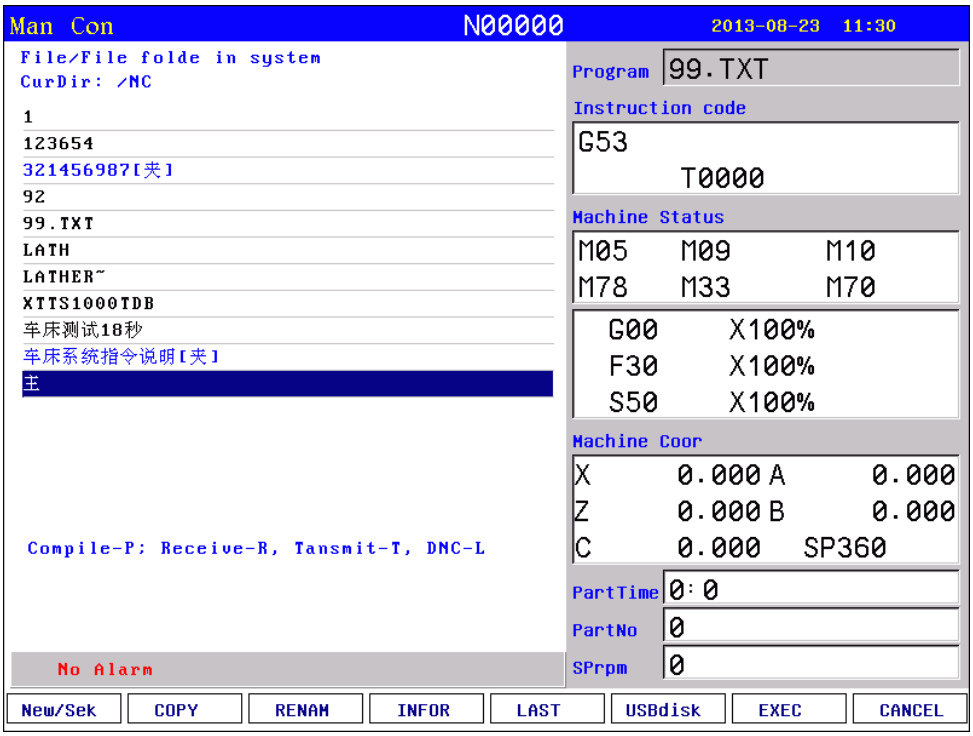

Man Con
N00000
2013-08-23 11:29
Input point
X00 T01
X01 T02
X02 T03
X03 T04
X04 T05
X05 T06
X06 T07
X07 T08
X08 M34/A0
X09 -L
X10 +L
X11 M36/Y0
X12 X0
X13 Z0
X14 KRUN
X15 KHALT
X16 XZ0
X17 ZZ0
X18 KLEFT
X19 KRIGHT
X20 STOP
X21 TOK
X22 ALM
X23 ALM1
X24 ALM2
X25 M28
X26 M24
X27 M22
X28 M18
X29 M12
X30 M14
X31 M16
X32 HX
X33 HY
X34 HZ
X35 HA
X36 HX1
X37 HX10
X38 HX100
X39 HOFF
X40
X41
X42
X43
X44
X45
X46
X47
X60 DS3
X61 DS2
X62 DS1
X63 DS0
X64 DK3
X65 DK2
X66 DK1
X67 DK0
No Alarm
Program
99.TXT
Instruction code
G53
T0000
Machine Status
M05 M09 M10
M78 M33 M70
G00 X100%
F30 X100%
S50 X100%
Machine Coor
X 0.000 A 0.000
Z 0.000 B 0.000
C 0.000 SP360
PartTime 0:0
PartNo 0
SPrpm 0
I/O
ALARM
Reset
CANCEL


Man Con
N00000
2013-08-23 11:30
Output Point
Y00 M61
Y01 M63
Y02 M65
Y03 M67
Y04 M69
Y05 M71
Y06 M73
Y07 M59
Y08 M32
Y09 M79
Y10 M10
Y11 M08
Y12 M05
Y13 M04
Y14 M03
Y15 M75
Y16 LRUN
Y17 INTH
Y18 +T
Y19 -T
Y20 S04
Y21 S03
Y22 S02
Y23 S01
Y24
Y25
Y26
Y27
Y28
Y29
Y30
Y31
No Alarm
Program
99.TXT
Instruction code
G53
T0000
Machine Status
M05 M09 M10
M78 M33 M70
G00 X100%
F30 X100%
S50 X100%
Machine Coor
X 0.000 A 0.000
Z 0.000 B 0.000
C 0.000 SP360
PartTime 0:0
PartNo 0
SPrpm 0
I/O
ALARM
Reset
CANCEL

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.



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 "F1、F2、F3、F4、F5、F6、F7、F8" 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 diaplayed. 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 “F6” 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.

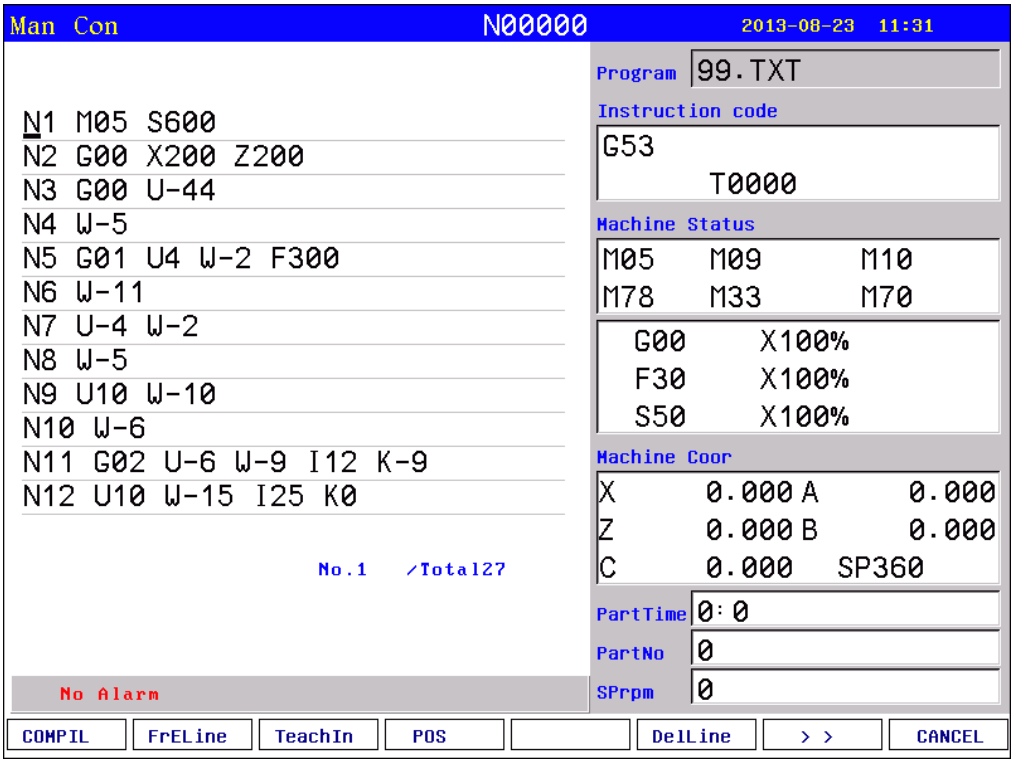
3.4.7 Serial port transmission program

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 transmiting; presse"R" 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 "T", now the system is waiting for transmiting;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



The edition mainly uses to edit,insert,modify,delet 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 "F1、F2、F3、F4、F5、F6、F7")"compile", "first line", "Teaching", "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" ("F7" 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, The edition mainly uses to edit, insert, modify, delet 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"F1、F2、F3、F4、F5、F6、F7")"compile", "first line", "Teaching", "pose", "del line"," >> "("del block" "copy block" "array" "serch" "alter" "aalter" "<<"),"cancel", etc. " 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, The edition mainly uses to edit, insert, modify, delet 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"F1、F2、F3、F4、F5、F6、F7")"compile", "first line", "Teaching", "pose", "del line"," >> "("del block" "copy block" "array" "serch" "alter" "aalter" "<<"),"cancel", etc. 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 lightat 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.

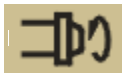
At the manual condition, presses and select X, Y, Z, A, B to go back to the reference point. When chooses X, Y, Z, A, B, only returns to this axis the reference point. Chooses A, returns to the reference point in turn.

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  SP chuck to turn on or turn off.

Presse  in choosing the position cutting tool.

Presses  thumbstall to turn on or turn 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.

7) presses "T", choosing the position cutting tool.

3.5.6 Tool setting

Tool setting is essential in operation. When two and more tools are required, tool setting is needed. To make accuracy, please follow the following steps:

Method 1:

1, tighten workpiece roughcast, choose proper spindle rotation and feed speed. Start spindle.
2, select the tool which is to be set, such as: T0101.
3, cut a short of excircle or hole in the continuous mode.
4, stop feed, press "Tool" to store the current cutting point coordinates.
5, X, Z axes withdraw, turn off spindle.
6, measure workpiece's diameter.
7, input the measured values into X value box, "enter" to confirm. If the tool is beyond the center of spindle, input negative value. Press "PgUp" to choose the previous tool, and "PgDn" to next tool.
8, likewise, cutting the head face of roughcast.
9, measure the distance between the head face and spindle collet.
10, press "tool", input the measured values into Z value box.
At this time, T0101 tool has finished its setting. Repeat steps 1-10 to set other tools.

Method 2:

1, tighten workpiece roughcast, choose proper spindle rotation and feed speed. Start spindle.
2, select the tool which is to be set, such as: T0101.
3, cut a short of excircle or hole in the continuous mode.
4, Z axis withdraws (X axis can not move), turn off spindle.
5, measure workpiece's diameter.
6, press "tool", input the measured values into X value box, "enter" to confirm. If the tool is beyond the center of spindle, input negative value. Press "PgUp" to choose the previous tool, and "PgDn" to next tool.
7, likewise, cutting the head face of roughcast.
8, measure the distance between the head face and spindle collet.
9, press "tool", input the measured values into Z value box.
At this time, T0101 tool has finished its setting. Repeat steps 1-9 to set other tools.

3.6 automatic

Cancels manual and turnes to automatical,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 shows the workpiece coordinates and the compositive 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 returns to the initial graph status. Furthermore, we can look at the entire tool track before the machining. under the runing 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 runing 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 coooridate mode. But all axises and other machine motion will be stopped.

3.6.6 Keep feed status

Under the program hold satus, 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 axia 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 "N" 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.

3.6.10 Begin from program some tool number

Pressing the "G" key, CNC will break out a dialog box, after input mark tool number(example:T0101 must input 01) and press the "run" key, CNC will execute program from the input tool number.

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 cooridnate or the relative coordinates value.Under the work cooridnate display mode configure work cooridnate; Under the synthesis cooridnate display mode configure relative cooridnate, in the course of program running also can configure relative cooridnate.

Note: The machine coordinates cannot be configured.

Choice coordinates:

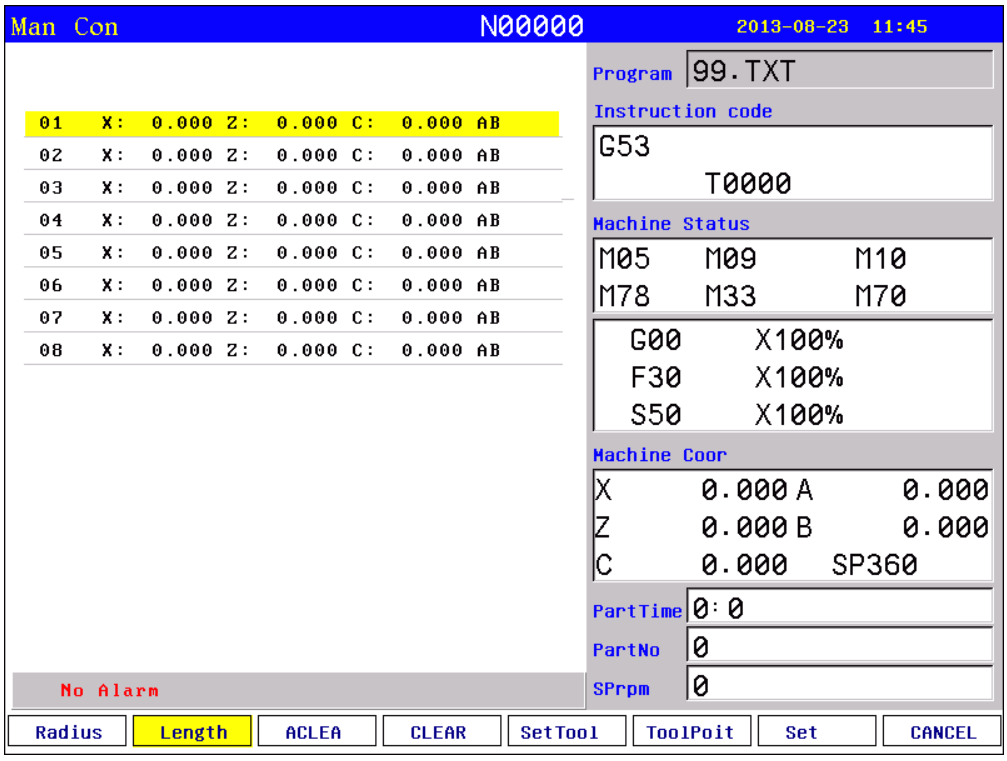
After pressing "MDI" input G53/G59 may choose G53, G54,G55,G56,G57,G58,G59 work coordinate. Corresponding work coordinate status is displaying in the top right corner interface.

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”:



Presses F1, set tool’s radius.
Presses F2, set tool’s redeem.
Presses F3, clear all value.
Presses F4, clear current tool value.
Presses F5, tool posit.
Presses F6, G41/G42 compensation parameter.
Presses F7, set tool’s number.
Presses F8, 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

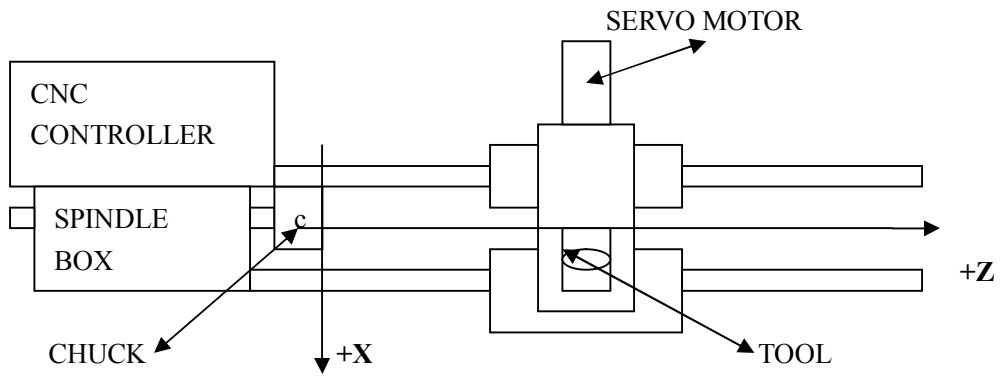
program segment: is a complete command line consisted of instruction segment and data segment.

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

Machine coordinate system: see fig Lathe top view.

Machine coordinate stipulates machine coordinate and direction, z axis parallels spindle, direction away from spindle is positive. X axis is vertical to spindle, the opposite of knifepoint is positive.

Lathe top view



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

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

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

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

Starting point: the place where the tool begins to move when program

starts.

Name of program: the name of machining program.

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

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

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

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

4.2 Program instruction

4.2.1 Functional meaning of address symbol, data list.

Functions	Address symbol	Meaning	Data range
Program segment No	N	No of program segment	
Preparation function	G	Content and mode of designalated instruction operation	00-99
Auxiliary function	M	Auxiliary operation instruction	00-99
Tool function	T	Tool changing instruction	Tool T0101-9999
Spindle function	S	The first spindle speed	0-65000
Spindle function	SS	The second spindle speed	0-65000
Cutting speed	F	Speed per minute, per	1-30000mm/min,

		rotation	0.001-9.999mm/n
Coordinates character	XU ZW	The coordinates value of X Z axes	±99999.9999mm
Lead of screw thread	F(I)	F for metric, I for imperial	0.1-1000mm, 1-99teeth/inch
Core coordinates	IK	X Z axes coordinate increment value	±99999.9999mm
Arc radius	R	Arc radius value, tool radius value	0.001-99999.999mm
Delay time	XUP	Delay time of designalated delay	0.001-65s
Program entrance	P	Entrance of calling program segment	0000-99999
Repeat times	L	Times of cycle or subprogram calling L can be used as numbers of multiply screw thread	1-65000 1-99
Program skip	/	There is “/”befor N, this line does not run.	

4.2.2 Program instruction table

Group	Nature	Code	Functions	Origin	Mode	remarks
1	Main control functions	G00	Rapid point positioning	√	√	
		G01	Linear interpolation		√	
		G02	Circular—clockwise		√	
		G03	Circular—counterclockwise		√	
		G32	spiral interpolation			
		G332	G02 spiral			
		G333	G03 spiral			
		G31	Check jump No alm			
		G311	Check jump alm			

	Loop comand	G70	Finish manching loop		√	
		G71	Cylindrical face thick loop		√	
		G72	end face thick loop		√	
		G73	close loop		√	
		G74	end face deep hole loop		√	
		G75	slot loop		√	
		G76	complex screw thread loop		√	
		G90	Circular loop		√	
		G92	screw thread loop		√	
		G93	tap loop		√	
		G94	end face loop		√	
		G22	loop end		√	
		G800	cancel loop		√	
	Go start	G26	Z、X go start			
		G261	X go start			
		G262	Y go start			
		G263	Z go start			
		G264	A go start			
		G265	B go start			
	Go G25	G61	XZ Go G25			
		G611	X Go G25			
		G612	Y Go G25			
		G613	Z Go G25			
		G614	A Go G25			
		G615	B Go G25			
	Save	G25	Save current coor		√	

	Go home	G28 G281 G282 G283 G284 G285 M800	XZ go home X go home Y/C go home Z go home A go home B go home C go encode Zero		√	
	set	G50	Setup coor system		√	
		G52	Setup part coor ststem		√	
		G184	Setup current Tool coor			
		G185	Setup all Tool coor			
2	line	G96	Constant line speed cutting		√	
		G97	cancel	√	√	
	feed	G98	Set feed per minute	√	√	
		G99	Set feed per revolution		√	
	mode	G15	cancel	√	√	
		G16	bally coor program		√	
		G21	metric program	√	√	
		G20	imperil		√	
3	delay	G04	Programmed dwell			
4	cut	G60 G64	Nicety stop Continuum part	√	√ √	
5	compensation	G40 G41 G42	Cancel cutter compensation Cutter in the left of workpiece Cutter in the right of workpiece	√	√ √ √	

6	Work coor	G53 G54 G55 G56 G57 G58 G59	Machine coor Work coor1 Work coor 2 Work coor 3 Work coor 4 Work coor 5 Work coor 6	√	√ √ √ √ √ √ √	
7	SP	M03 M04 M05	Spindle on clockwise Spindle on counterclockwise Spindle off	√	√ √ √	
8	Cooling	M08 M09	Coolant on Coolant off	√	√ √	
9	Chuck	M10 M11	Chuck tightens Chuck looses	√	√ √	
10	Tailstock	M79 M78	Tighten Loose	√	√ √	M79
11	Lubrication	M32 M33	Lubrication on Lubrication off	√	√ √	M32
12	Huff	M59 M58	Huff on Huff off	√	√ √	M59 output
12	user-definde output	M61 M60	user-definde1		√ √	M61 output
		M63 M62	user-definde2		√ √	M63 output
		M65 M64	user-definde3		√ √	M65 output
		M67 M66	user-definde4		√ √	M67 output
		M69 M68	user-definde5		√ √	M69 output

		M71 M70	user-definde6		√ √	M71 output
		M73 M72	user-definde7		√ √	M73 output
		M75 M74	user-definde8		√ √	M75 output
13	user-definde input	M12 M13	Check M12 valid Check M12 invalidate		√ √	M12 INPUT
		M14 M15	Check M14 valid Check M14 invalidate		√ √	
		M16 M17	Check M16 valid Check M16 invalidate		√ √	
		M18 M19	Check M18 valid Check M18 invalidate		√ √	
		M28 M29	Check M28 valid Check M28 invalidate		√ √	
		M22 M23	Check M22 valid Check M22 invalidate		√ √	
		M24 M25	Check M24 valid Check M24 invalidate		√ √	
14	SP Gear	M41 M42 M43 M44	first gear second gear third gear fourth gear			
15	jump	M97 M98 M99	jump call return			
16	Program	M00 M01 M02 M30 M20	pause M22 availpause Program end M05、M09 end Loop go start			
17	SP speed	S SS	First SP Second SP		√ √	S=0-65000 SS=0-65000

18	Tool	Tab	A:xx B:xx		√	A、b=00-12

4.3 Preparation functions

4.3.1 Programming stipulation

1, multiply instruction exist in a segment simultaneously: one program line allows multiply instructions in order to reduce the lines, but the same group of instruction can not share one segment.
2, within a program segment, instruction and parameters can be arranged optionally.
Such as: G01 U10 W-30 can be written: U10 G01 W-30
3, no repeat of instruction within a program segment.
4, no irrelative parameters and operation in a segment.
5, “0” before a instruction is allowed to delete, such as: G01 G03 can be written as G1 G3.
6, the command of optional point, line start or that after tool changing instruction must be programmed by absolute coordinates.

4.3.2 Instructions

（1）Rapid motion(G00)

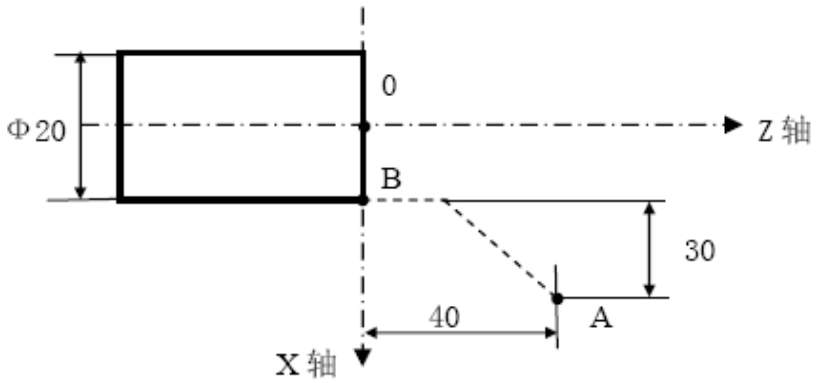
Tool move to instructive postion according to G00 speed in paramter。
As absolute method, use section end point coordinate to program;
As increase method, use motion distance to program。

Format: G00 X/U- Y/V- Z/W- A- B- (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。



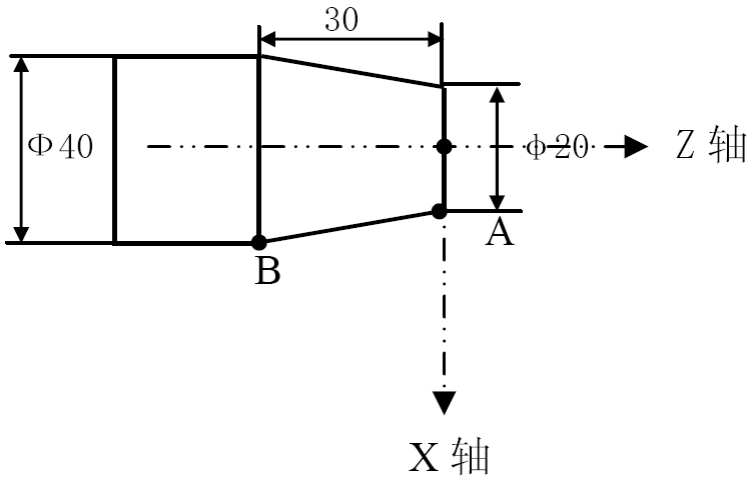
Example：from A to B.
Absolute program: G00 X20 Z0;
Relative program: G00 U－60 W－40;

（2）Line interpolation(G01)

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

Format：G01 X/U- Y/V- Z/W- A- B- 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.



Example: from A to B.

Absolute program: G01 X40 Z-30 F100

Relative program: G01 U20 W-30 F100

（3）Arc interpolation(G02/G03)

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

Format:

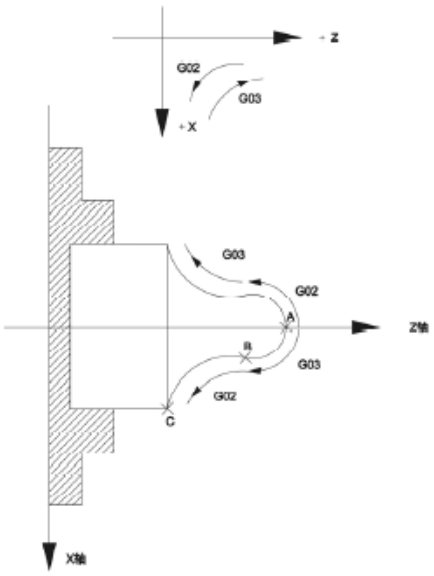
G02 X（U）__ Z（W）__ I__ K__ F__;

G03 X（U）__ Z（W）__ I__ K__ F__;

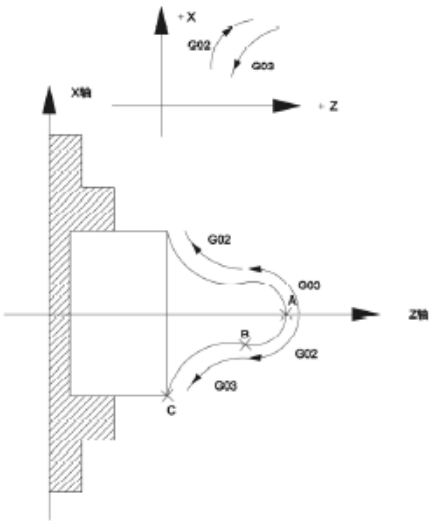
G02 X（U）__ Z（W）__ R__ F__;

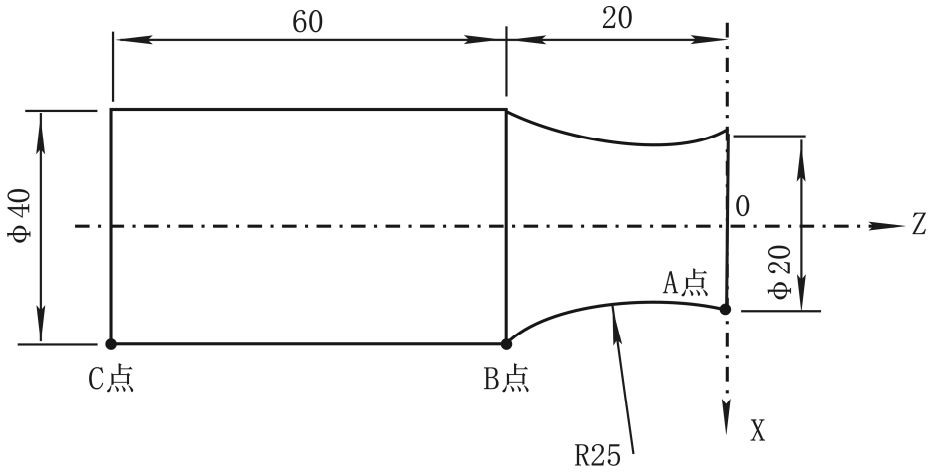
G03 X（U）__ Z（W）__ R__ F__;

前刀架系统



后刀架系统





Example：from A to B.
Absolute program: G02 X40 Z-20 I25 K0 ;
Relative program: G02 U20 W-20 I25 K0 ;
R program: G02 X40 Z-20 R25 ;
G02 U20 W-20 R25 。

（4）Screw thread (G32)

Format：G32 Z（W）- X（U）- F（I）- SP-

G32 is the spiral interpolation machining instruction. Z/W is the length of Z axis, it cuts strait thread;X/U is the length of X axis,it cuts head face thread; F is metric lead, range is 0.1-1300mm; I is imperial lead, range is 1-99 teeth/inch; L is multiple thread head numbers, range is 1-999, default value is 1.

The use of thread machining instruction must be under the condition that machine has quipped with photoelectric encoder; otherwise the system is in the status of standby. When spindle rotates clockwise, it machinings plus thread in the Z axis negative direction, and left-hand thread in positive direction. The cutting feed speed F=KxN is appropriate to the machining thread, overhigh speed will destroy the teeth, this system requires N≤2000n/min, F≤3000mm/min.

For example:

Strait thread: N0000 G32 W-30 K1.5 ; strait thread of length 30, lead 1.5

Metric thread: N0000 G32 W-30 I10.2 ; thread of each inch 101/2

Head face thread: N0000 G32 U-50 K2 ; thread of length 50, lead 2

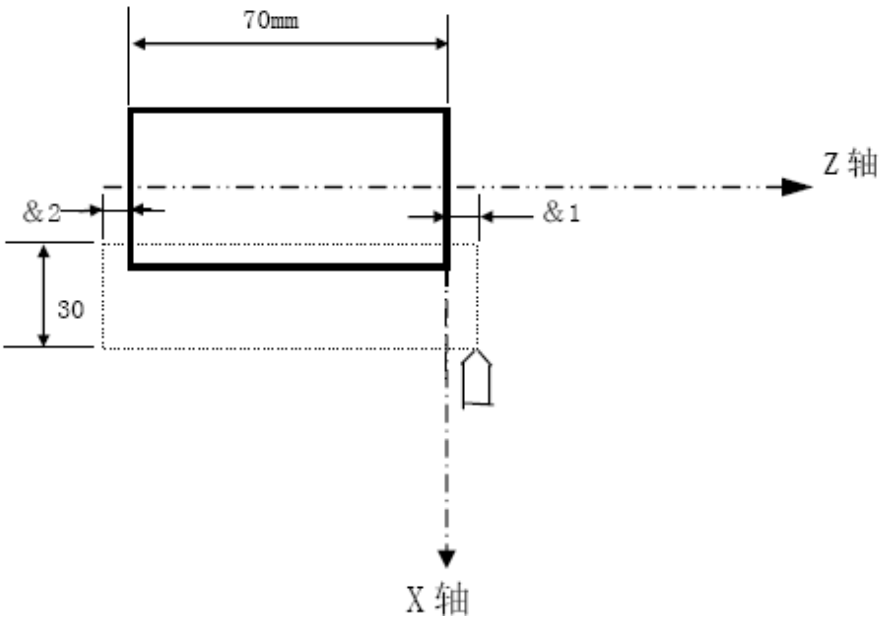
If rough or fine machining is needed, it can add fine machining rotating speed into thread fine machining instruction.

For example1:

```
N0000 T0101 S1200      ; T1 is tool for rough machining, rotating
                              speed is 1200
N0010 G00 U-1            ; advance of tool
N0020 G32 W-30 K1.5      ; rough machining
```

For example2:

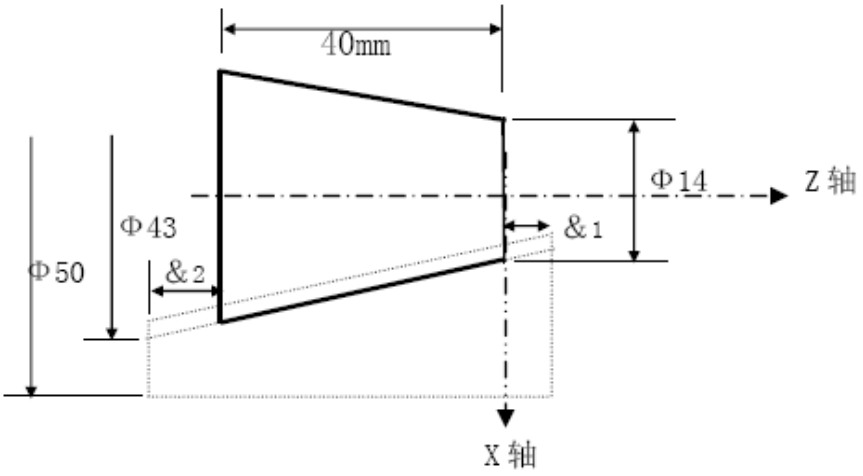
```
G00 U-62.0;
G32 W-74.5 F4.0;
G00 U62;
W74.5;
U-64;
G32 W-74.5;
G00 U64.0;
W74.5;
  ……
```



For example3:

```
G00 X12 Z3.0;
G32 X41.0 Z-41.5 F3.5;
G00 X50;
Z3;
```

```
X10;
G32 X39 Z-41.5;
G00 X50;
Z3;
    ……
```



（5） Circularity screw thread(G332、G333)

Format：G332/G333 Z（W）- X（U）- R - F（I）- SP-

Use method refer G02、G03、G32instruction.

（6）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.
```

（7）Return Refernce(G28/G281/G285)

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

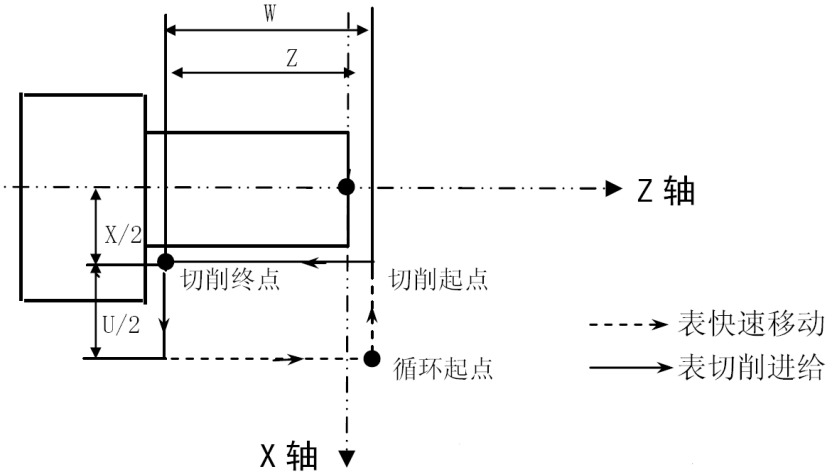
```
format:   G28 X/UY/VZ/W;ZXY return to reference
          G281       ;only X return to reference
          G282       ;only Y return to reference
          G283       ;only Z return to reference
          G284       ;only A return to reference
          G285       ;only B return to reference
```

（8）Setup workpiece coordinate system (G50)

Format： G50 X（x） Z（z）；

（9）Column or taper loop(G90)

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

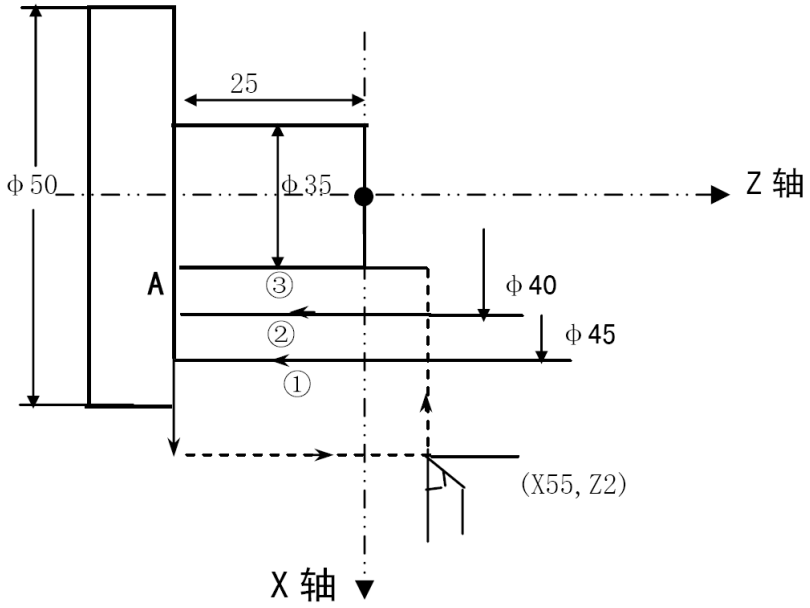


Column loop cotting

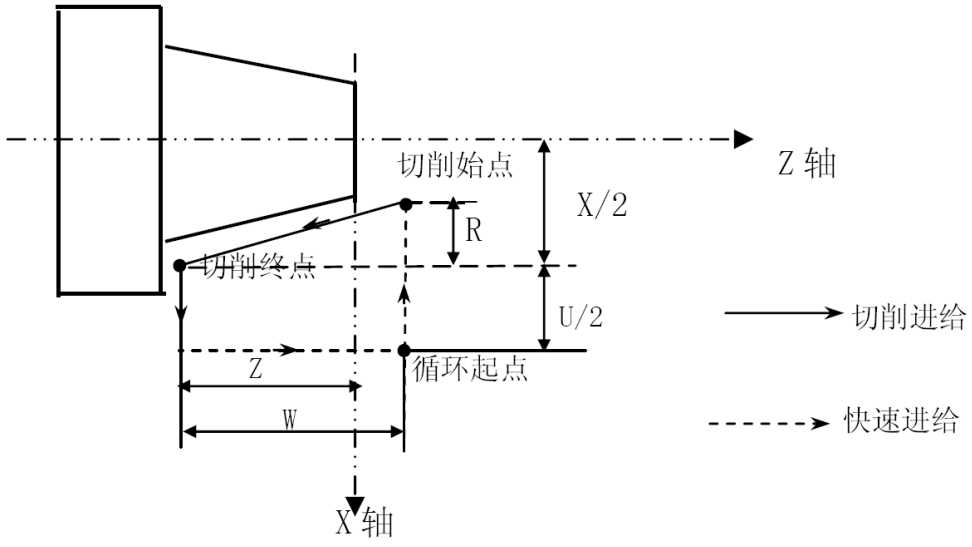
For example:

```
N10 T0101;
N20 G00 X55 Z4 M03;
N30 G01 Z2 F100 M08;
N40 G90 X45 Z-25;
N50 X40;
N60 X35;
N70 G00 X100 Z100;
N80 T0100 M09;
```

N90 M05;
N100 M30;



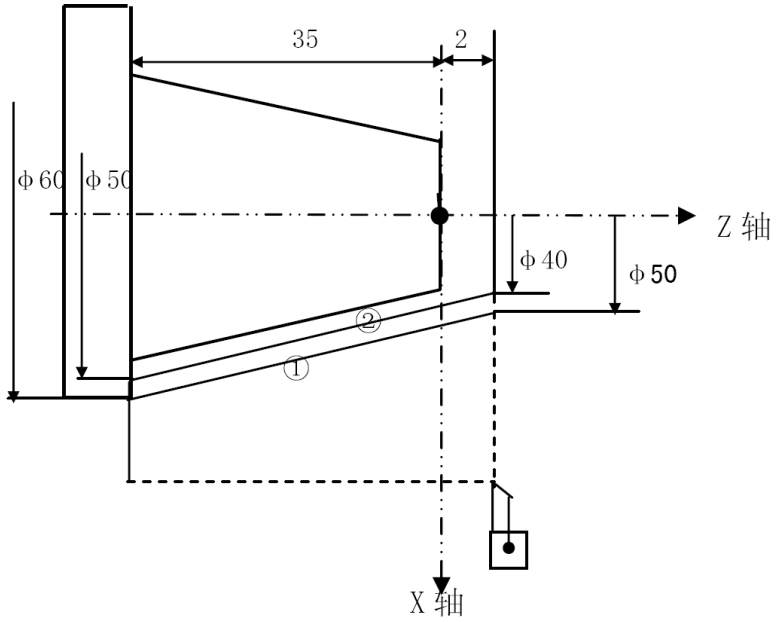
Taper loop format:G90 X(U)__ Z (W) __ R__ F__;



Taper loop cutting

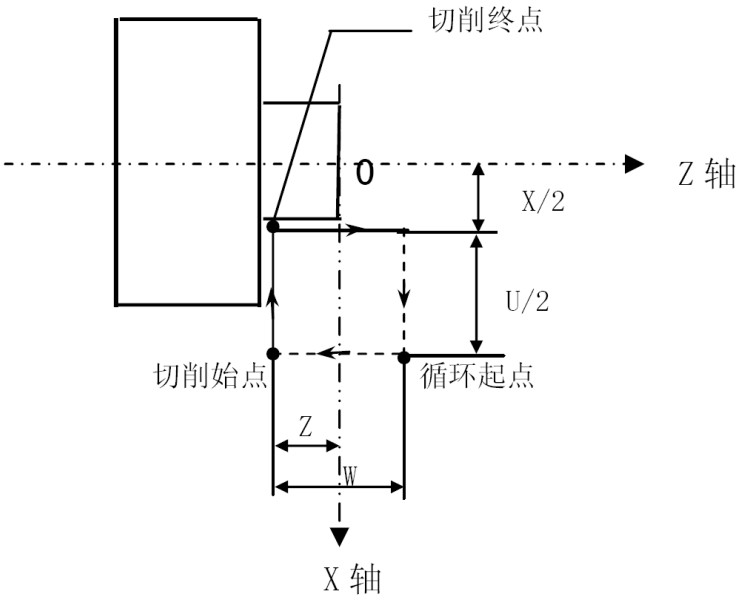
For example:
N10 M03 S1000;
N20 T0101;
N30 G00 X65 Z5;

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



（10）End face loop(G94)

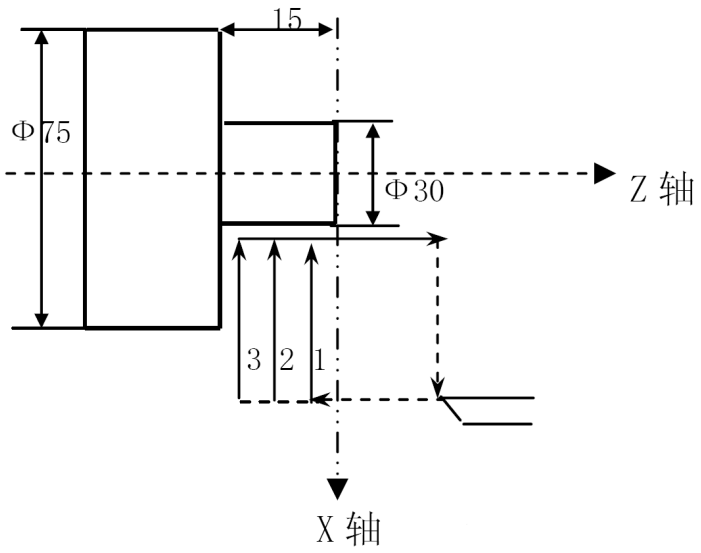
Format: G94 X(U)__ Z（W）__ F__;



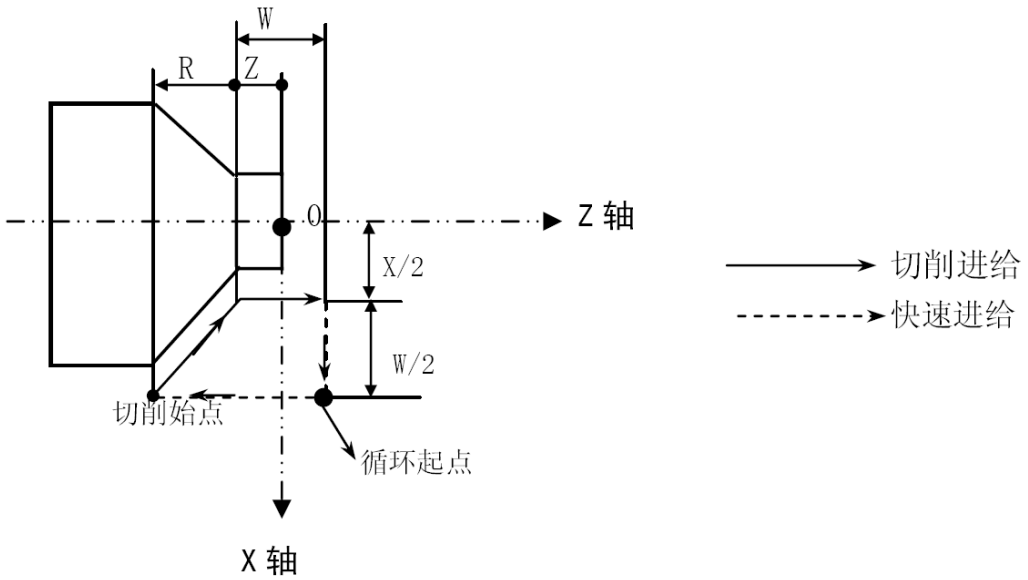
End face loop cutting

For example:

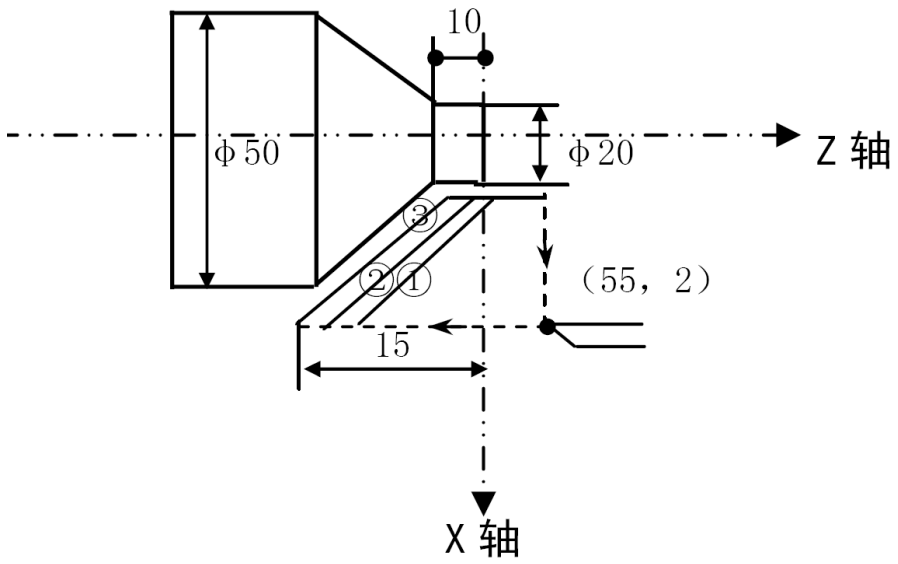
```
N10 M03 S1000;
N20 T0101;
N30 G00 X85 Z10 M08;
N40 G01 Z5 F200;
N50 G94 X30 Z-5 F100;
N60 Z－10;
N70 Z－15;
N80 G00 X100 Z60 M09;
N90 T0100 M05;
N100 M30;
```



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



Taper end face loop cutting



For example:

……

N40 G01 X55 Z2 F200;

N50 G94 X20 Z0 R-5 F100;

N60 Z-5;

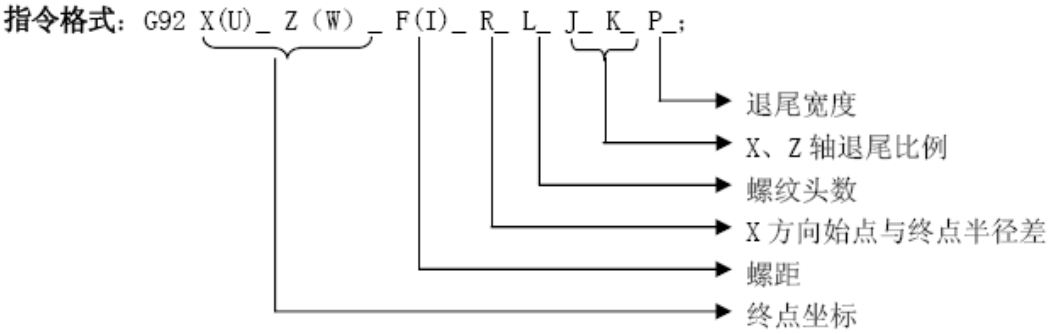
N70 Z-10;

N80 G00 X Z;

……

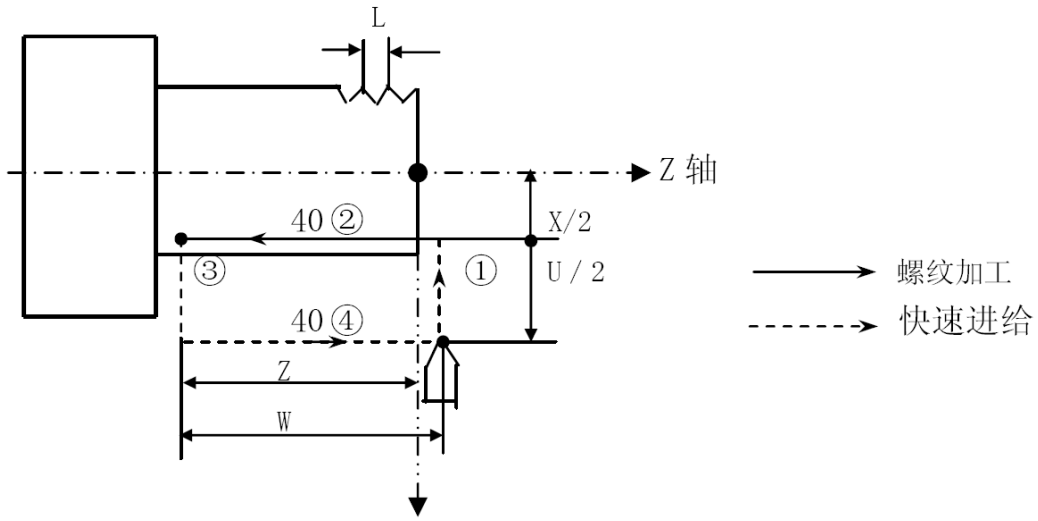
（11）Screw thread loop (G92)

Format:

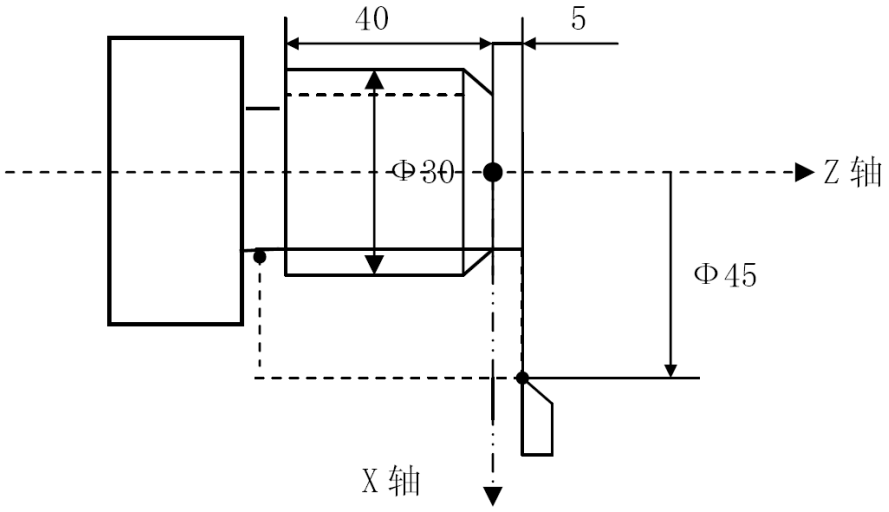


1）Straight screw thread loop format：

G92 X(U)__ Z（W）__ F/I__;



Straight screw thread loop



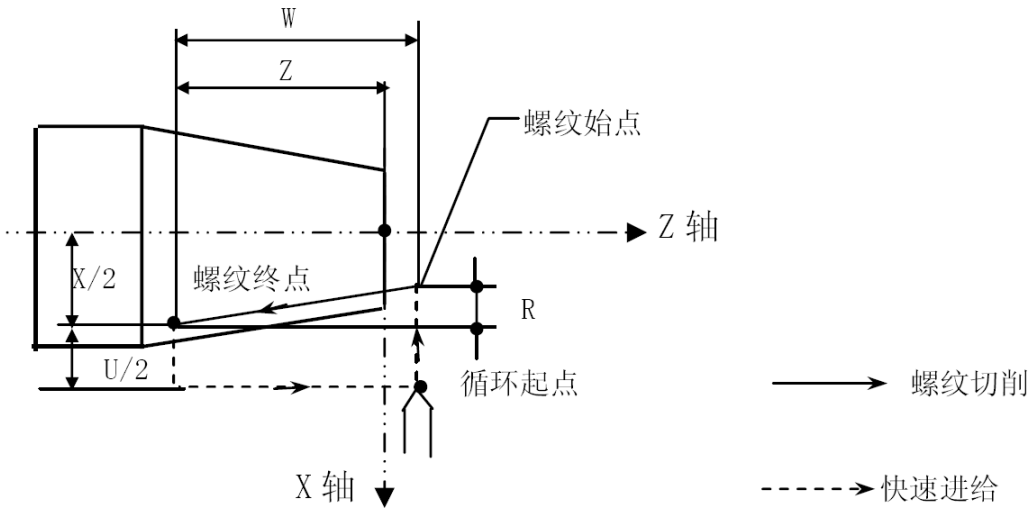
Format:

```
N10 M03 S××;
N20 T0101;
N30 G00 X45 Z5;
N40 G92 X29.2 Z-45 F1.5;
N50 X28.6;
N60 X28.2;
N70 X28.04;
N80 G00 X100 Z50;
```

N90 T0100 M05;
N100 M30;

2）Taper screw thread loop format：

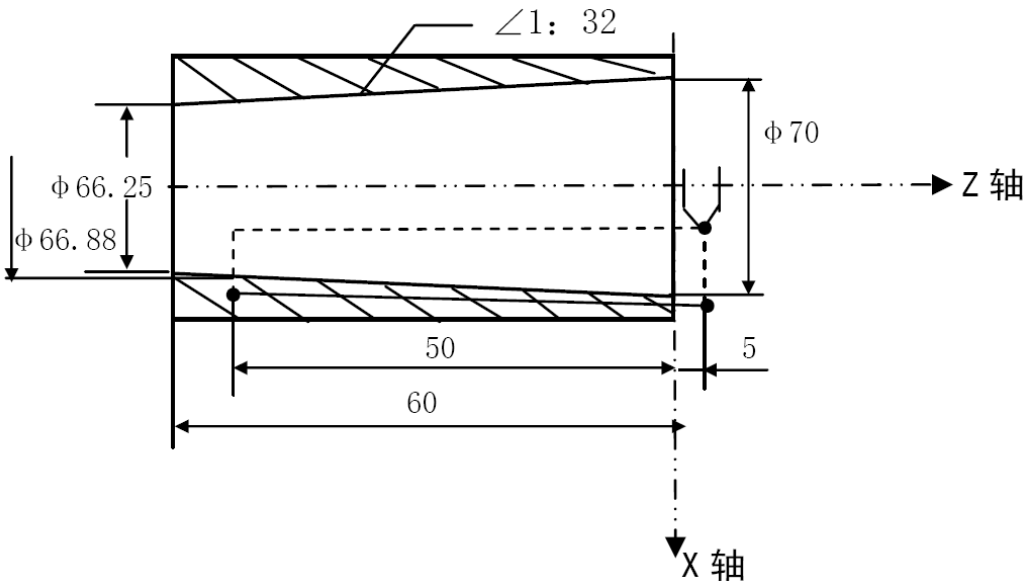
G92 X(U)__ Z（W）__ R__ F/I__;



Taper screw thread loop

For example:

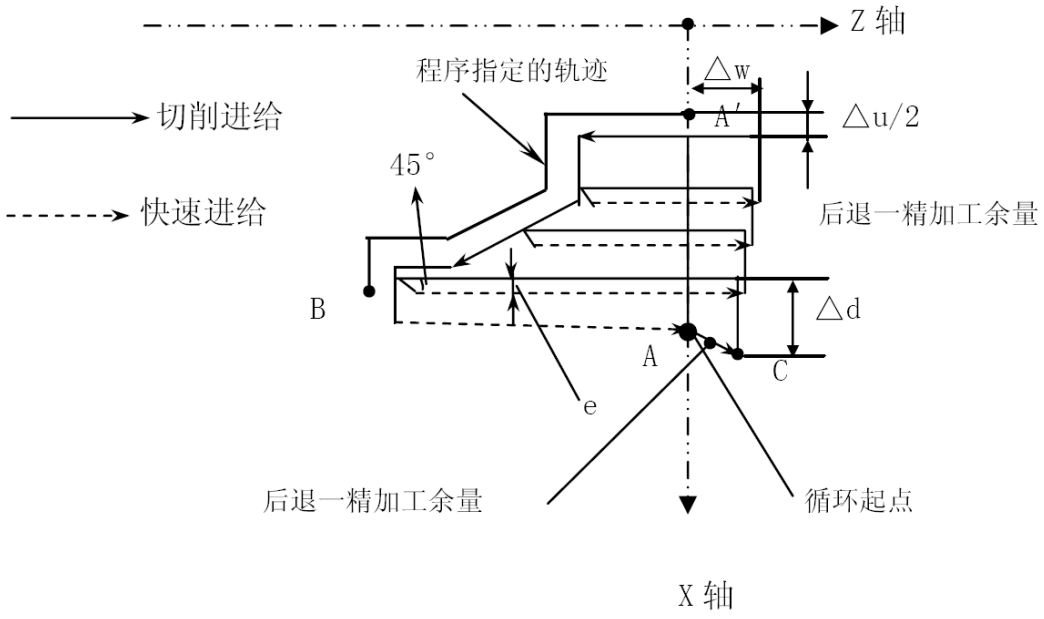
```
N10 M03 S××;
N20 T0101;
N30 G00 X55 Z10;
N40 G01 X60 Z5 F100;
N50 G90 X66.25 Z-60 R1.875;
N60 G92 X66.88 Z-50 R1.4 I11;
N70 X66.9 I11;
N80 X67 I11;
N90 X67.4 I11;
N100 X67.6 I11;
N110 X67.8 I11;
N120 G00 X100 Z50;
N130 T0100 M05;
N140 M30;
```



（12）Tap loop (G93)

Format: G93 Z（W） F/I ;
For example:
M03 S500
G04 X1
G93 Z-100 F5
G00 X50

（13）Column thick loop (G71)



Column thick loop

Format:

G71 U (△d) R(e);

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

△d: feed thickness, no signal; User parameter P1.

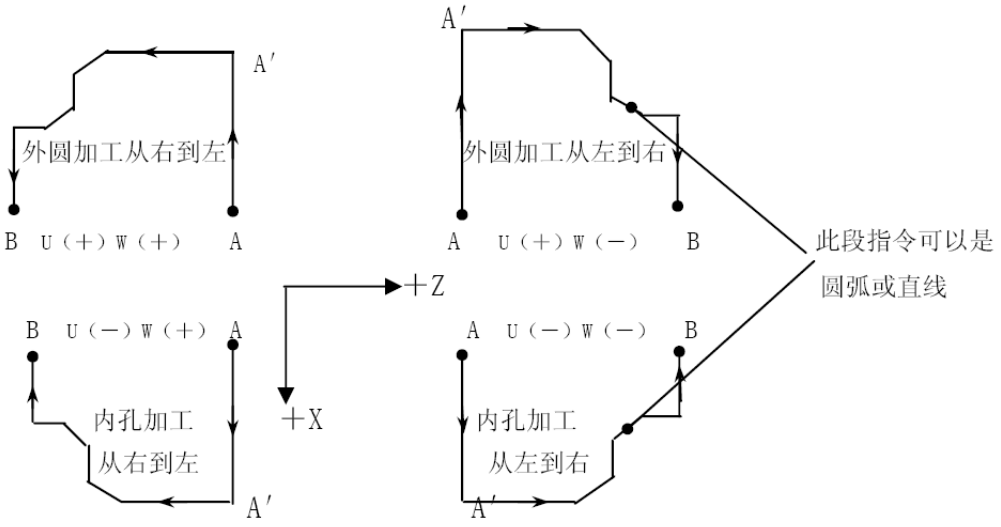
e: backward distance; User parameter P2.

ns: first N line.

nf: end N line.

△u: X remain; User parameter P4.

△w: Z remain; User parameter P5.



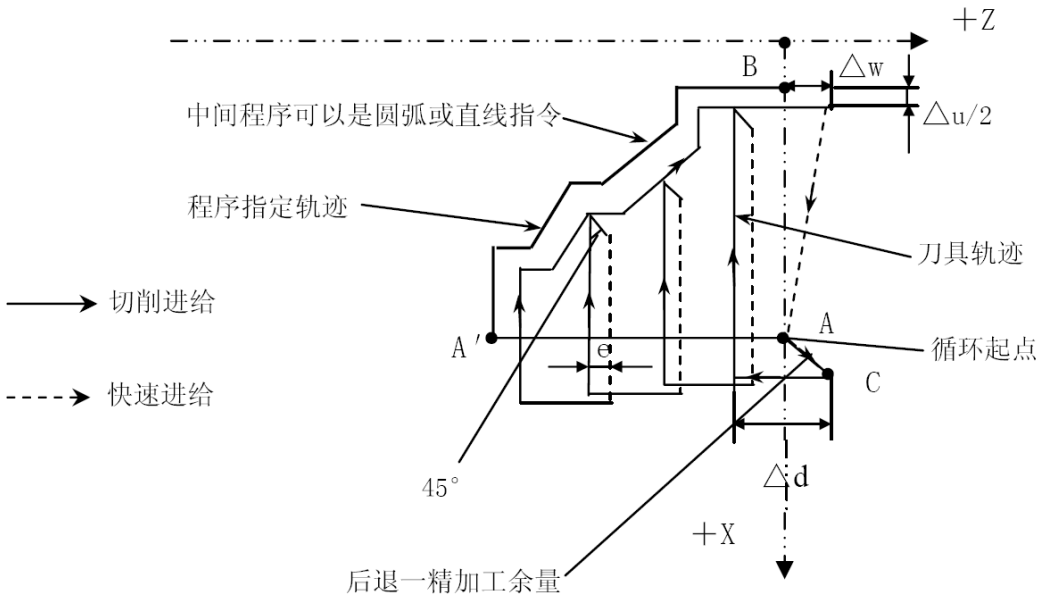
（14）End face thick loop(G72)

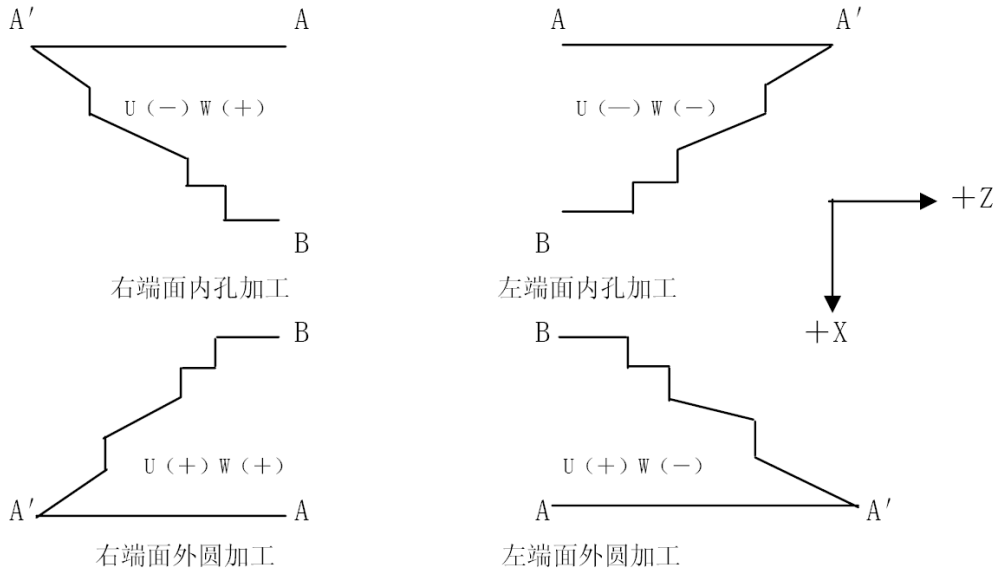
Format:

G72 W（△d） R（e）；

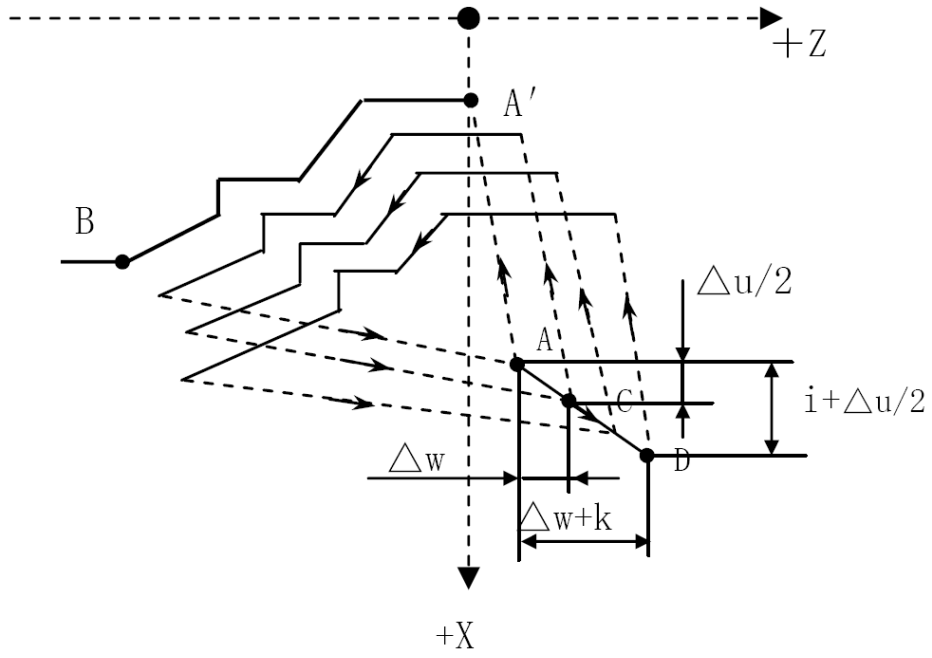
G72 P（ns） Q（ns） U（△u） W（△w） F（f） S（s） T（t）；

△d、e、ns、nf、△u、△w、f、s、t is same as G71.





（15）Close cutting loop(G73)



Format:

G73 U（i）__ W（k）__ R（d）__;

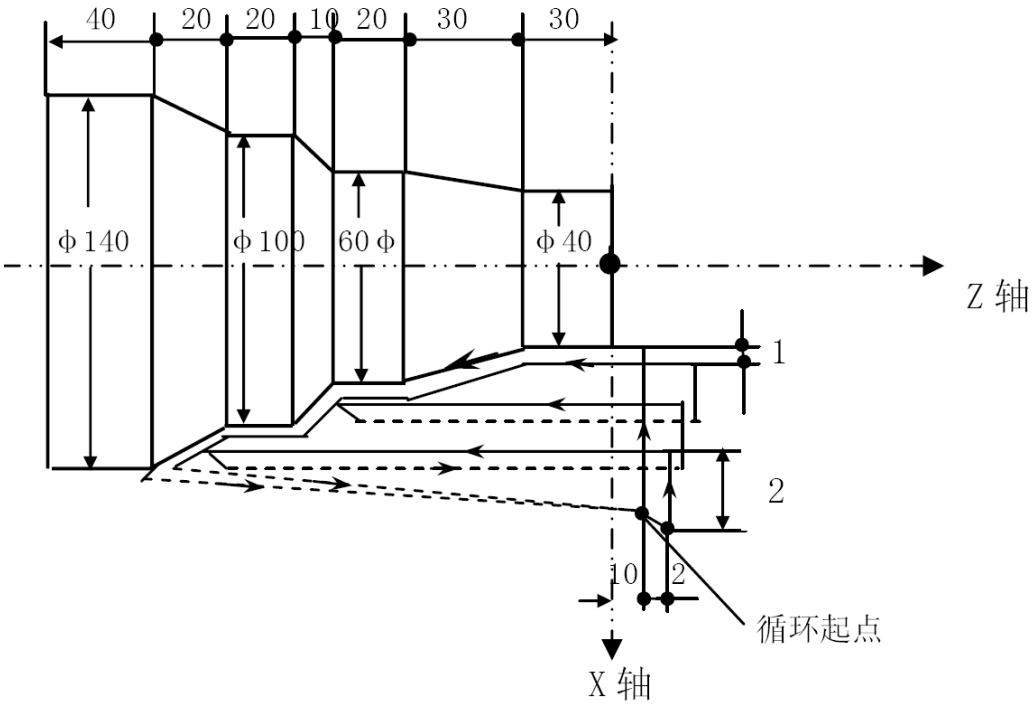
G73 P（ns）__ Q（nf）__ U（△u）__ W（△w）__ F（f）__ S（s）__ T（t）__;

```
N (ns) ……; --\
       ……;   |
         .     >  A→A′ →B ,ns to nf
         .     |
         .     |
         .     |
N (nf) ……; --/
```

i: X rough thickness; User parameter P7.
k: Z rough thickness; User parameter P8.
d: cutting times; User parameter P6.
Others is same as G71.

（16）Finish machining loop (G70)

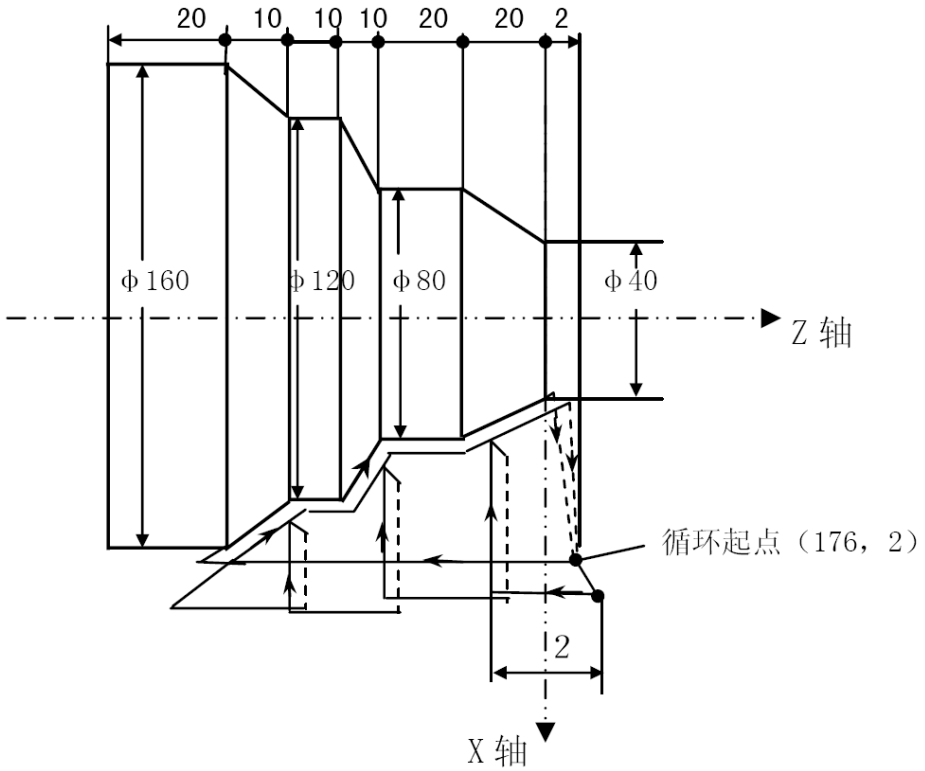
Format:
G70 P (ns) Q (nf)
For example1: G71 G70




```
N10 M03 S1500;
N20 T0101;
```

```
N30 G00 X160 Z10;
N40 G71 U2 R1;
N50 G71 P60 Q120 U2 W1 F100 S2000
N60 G00 X40;
N70 G01 Z-30 F80;
N80 X60 W-30;
N90 W-20;
N100 X100 W-10;
N110 W-20;
N120 X140 W-20;
N130 G70 P60 Q120;
N140 G00 X200 Z50;
N150 T0100 M05;
N160 M30;
```

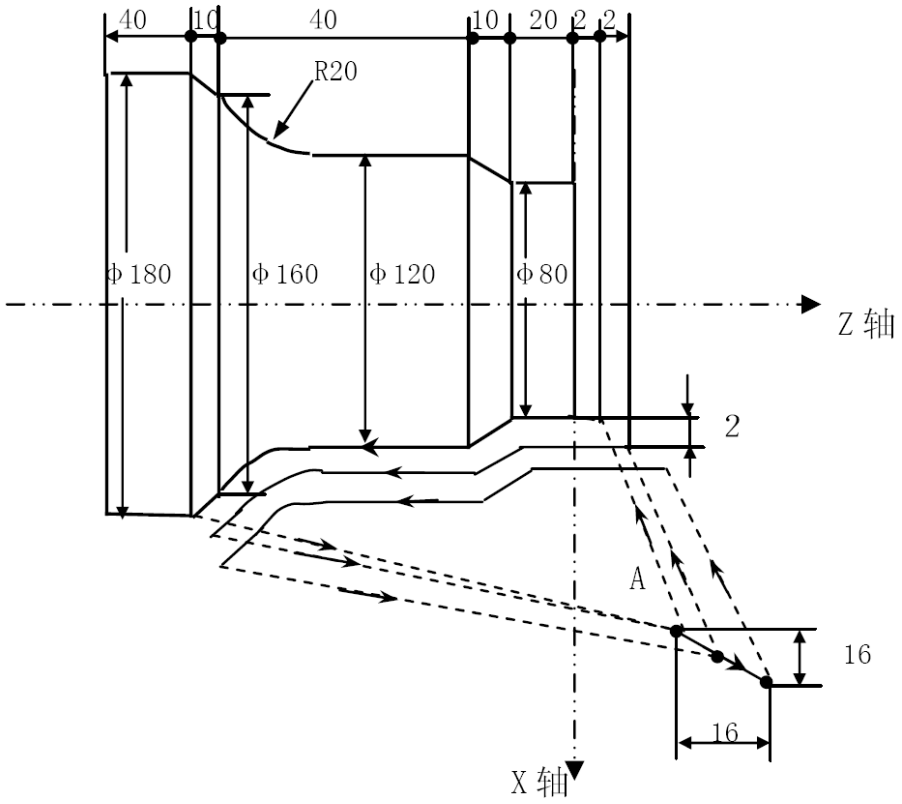
For example2：G72 G70




```
N10 M03 S2000;
N20 T0202;
N30 G00 X176 Z2;
N40 G72 W2 R1;
```

```
N50 G72 P60 Q120 U2 W1 F100 ;
N60 G00 Z-72;
N70 G01 X160 Z-70 F80;
N80 X120 W10;
N90 W10;
N100 X80 W10;
N110 W20;
N120 X36 W22.08;
N130 G70 P60 Q120;
N140 G00 X200 Z50;
N150 T0200 M05;
N160 M30;
```

For example3: G73 G70




```
N10 M03 S3000;
N20 T0303;
N30 G00 X220 Z40;
N40 G73 U14 W14 R0.010;
N50 G73 P60 Q110 U4 W2 F100;
```

```
N60 G00 X80 Z2;
N70 G01 Z-20 F80;
N80 X120 W-10;
N90 W-20;
N100 G02 X160 W-20 R20;
N110 G01 X180 W-10;
N120 G70 P60 Q110;
N130 G00 X250 Z50;
N140 T0300 M05;
N150 M30;
```

（17）End face deep hole loop(G74)

Format:

G74 R（e）；

G74 X（u）P（△i）Z（w） Q（△k） F（f）；

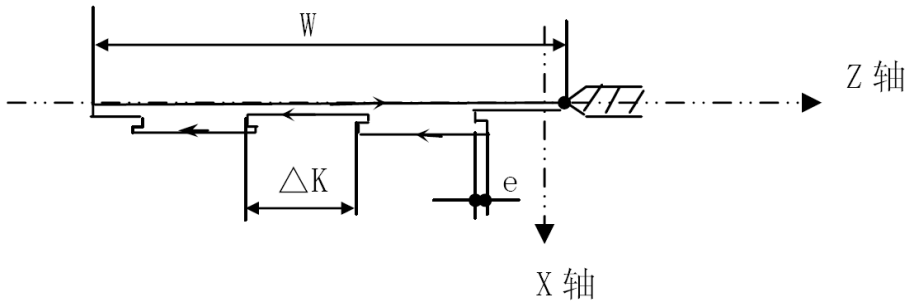
e: backward distance;User parameter P10.

Z（w） Z depth;

X（u） X end-point coordinate;

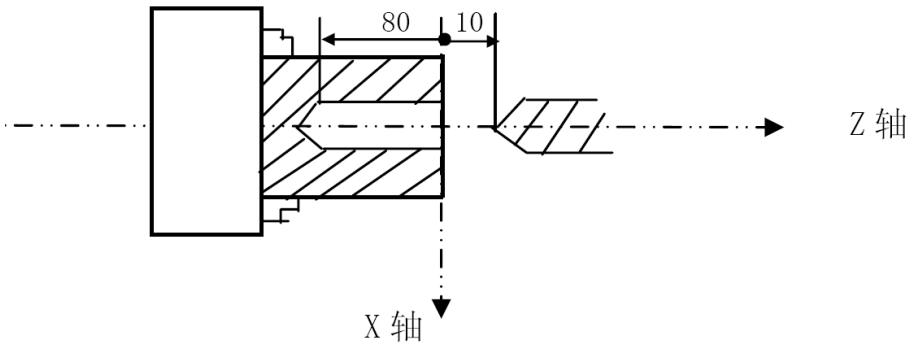
Δk： Z feed thickness;User parameter P9.

Δi： X feed thickness.



For example:

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



（18）Slot cutting loop(G75)

Format:

G75 R(e)__;

G75 X(U)__ P(Δi)__ Z（w）__ Q（△k）__F(f)__;

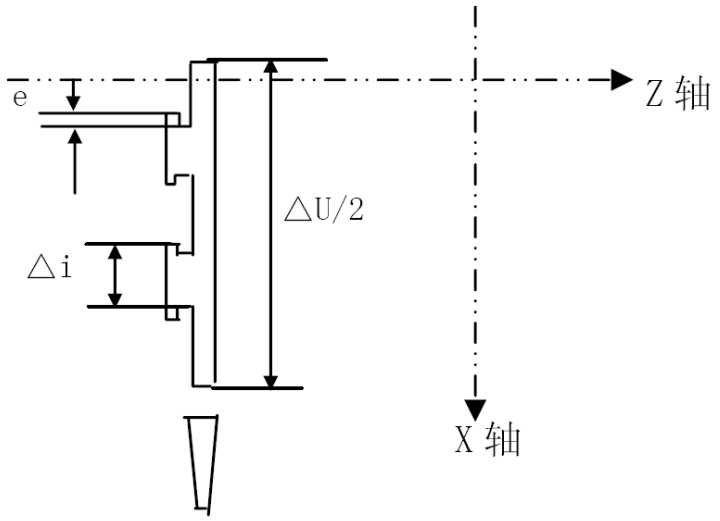
e: backward distance;User parameter P10.

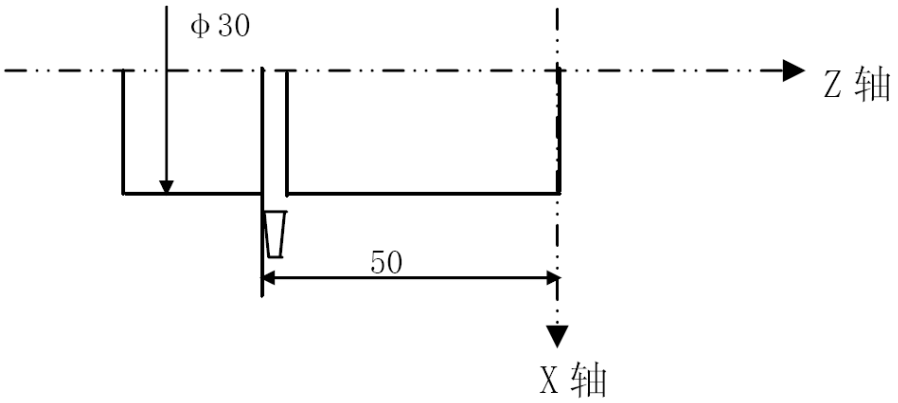
X（u） X depth;

Z（w） Z end-point coordinate;

Δi： X feed thickness;User parameter P9.

Δk： Z feed thickness.



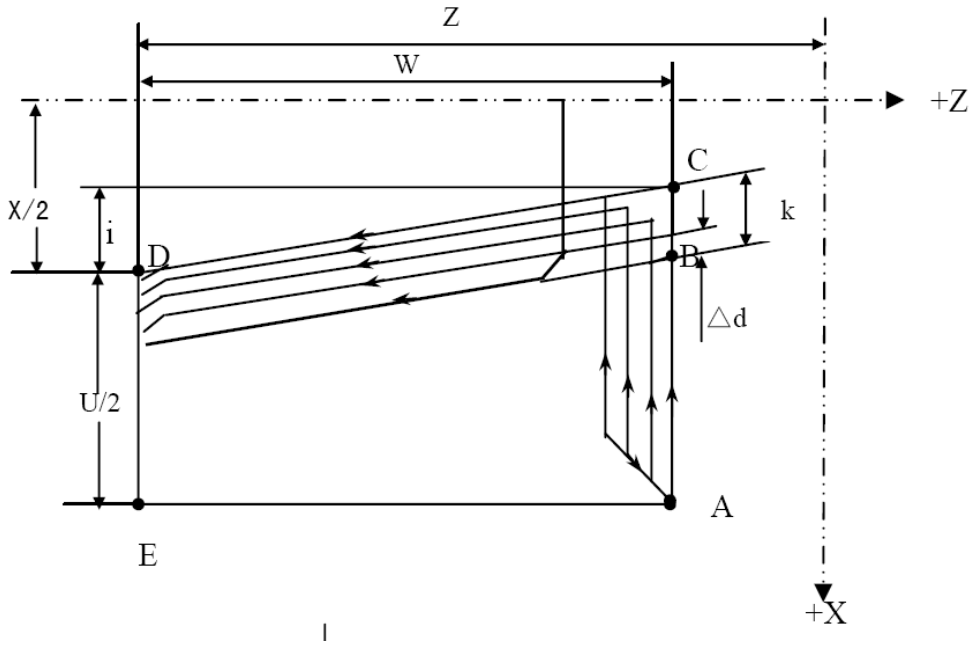


For example:

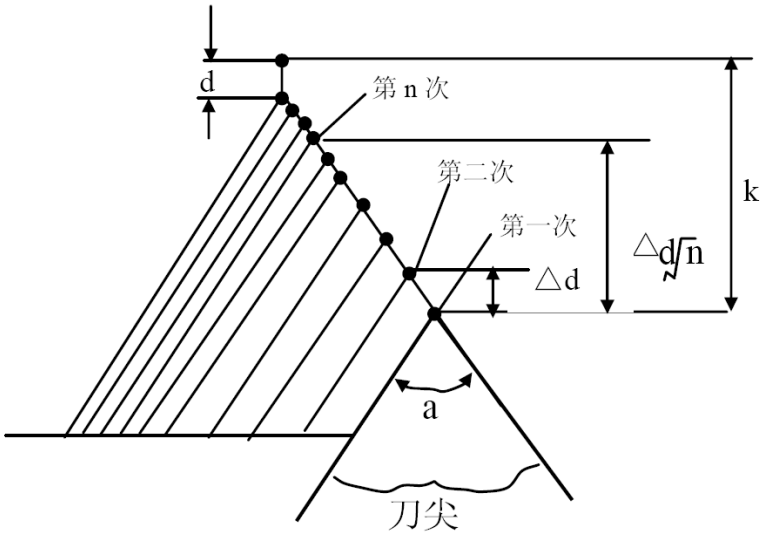
```
N10 M03 S1000;
N20 T0101;
N30 G00 X35 Z-50;
N40 G75 R1;
N50 G75 X-1 P5000 F60;
N60 G00 X100 Z50 M09;
N70 M05;
N80 T0100;
N90 M30;
```

（19）Complex screw thread loop(G76)

As fig:



Cutting method:



Format:

G76 P(b)(c)(m)(r)(a) Q(Δdmin) R(d) ;

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

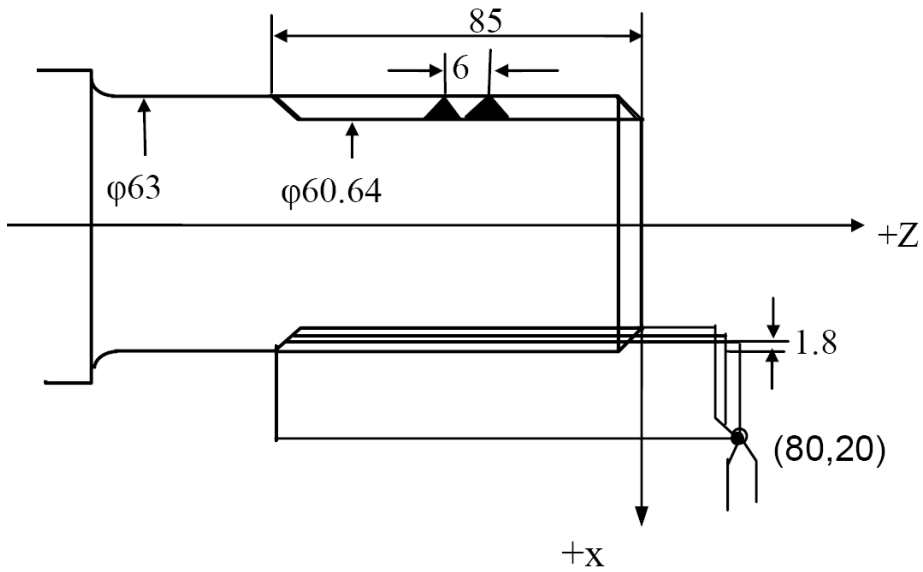
b: 0————degression feed;

1————equidistance feed;

c: 0————right enter;

1————left enter;

m: finish turn times,User parameter P11.
r: quit length,User parameter P12.
a: thread tooth angle(degree),Usr parameter P13.
Δdmin: minimal cutting depth,User parameter P14.
d: finish turn remaining,User parameter P15.
I: X taper screw thread feed measure.
f: metric lead.
L: multiple thread head numbers.
SP: start angle: 0-360°




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

（20）Fix loop(G22、G800)

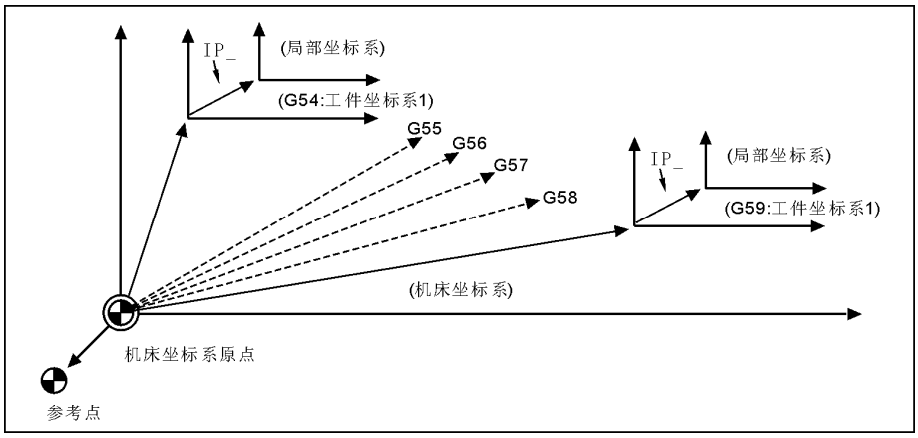
Format: G22 L-

.
.
.

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

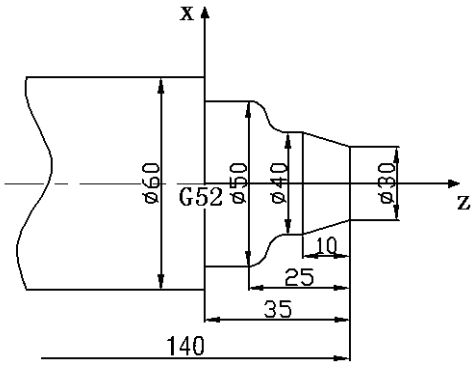
（21）Part coordinate setup(G52)，Setup tool coor（G184、G185）

```
Format:
G52 X_Z_; absolute coordinate  setup
G52 U_W_ ;relative coordinate  setup
G52 X0Z0 ; cancel
G184 X_Z_P_L_;setup current&P&L tool absolute coordinate
G184 U_W_P_L_;setup current&P&L tool relative coordinate
G185 X_Z_; setup all tool absolute coordinte
G185 U_W_ ; setup all tool relative coordinte
```



Part coordinate setup

For example:




```
N1 G00 X60 Z20
N2 G52 X0 Z-236
N3 T0101
N4 M03 S800 M08
N5 G01 Z35 F100
N6 X-1
N7 X70
N8 G71 U2 R1
N8 G71 P10 Q15 U0.5 W0.5 F150
N10 G01 X30
N11 X40 Z25
N12 Z20
N13 G02 X50 Z15 R5
N14 G03 X60 Z10 R5
N15 G01 Z0
N16 G00 X70
N17 G52 X0 Z0
N18 M05
N19 M30
```

（22）Back start point(G26、G261/G265)

Format:

G26、G261、G262、G263、G264、G265; XZ、X、Y、Z、A、B

Format:

```
N0000  G00 X120 Z300        ;
N0001  G01 X150 Z50 F160    ;
N0002  G26                  ;
N0003  M2                   ;
```

(23） Back to G25 point(G25、G61、G611/G615)

Format:
G25 ;Save current coordinate
G61、G611、G612、G613、G614、G615; XZ, X,Y,Z,A,B
For example:

```
N0000  G0 X20  Z80          ;
N0001  G01 U5 W－16 F200    ;
N0002  W－100               ;
N0003  G00 U10              ;
N0004  Z80                  ;
N0005  G25
N0006  G01 U10 W-30         ;
N0007  G0 X100 Z200         ;
N0008  G61                  ;
N0009  M2                   ;
```

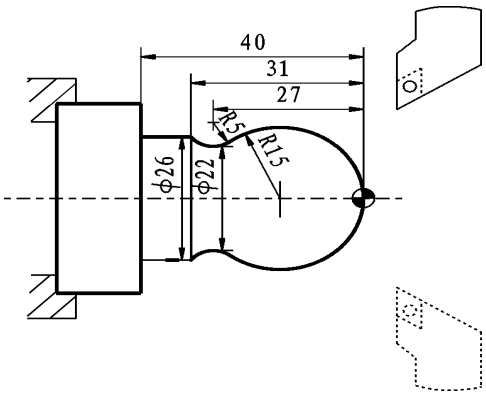
（24）continue feed cutting(G60/G64)

Format：G60 ; cancel
G64 ; continue feed

(25）Constant speed cutting(G96/G97)

Format：G96 S_ Constant speed cutting
G97 cancel

For example:




```
N1 T0102 X40 Z5
N2 M03 S400
N3 G96 S80
```

```
N4 G00 X0
N5 G01 Z0 F60
N6 G03 U24 W-24 R15
N7 G02 X26 Z-31 R5
N8 G01 Z-40
N9 X40 Z5
N10 G97 S300
N11 M30
```

（26）Feed mode(G98、G99)

Format:

G98 feed per minute instruction

G99 feed per revolution instruction

（27）Check skip(G31、G311)

Format：G31 X_ Z_ F_ P_ ;No alarm

G311 X_ Z_ 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.

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

Format：G53(G54/G55/G56/G57/G58/G59)

G53 machine coordinate

G54 workpiece coordinate1

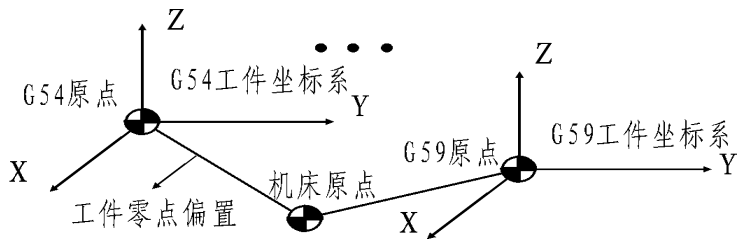
G55 workpiece coordinate 2

G56 workpiece coordinate 3

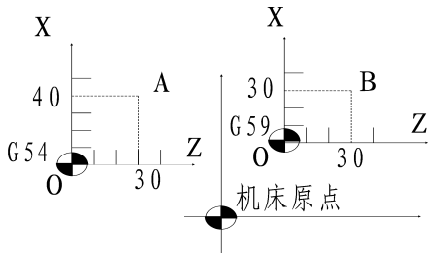
G57 workpiece coordinate 4

G58 workpiece coordinate 5

G59 workpiece coordinate 6



For example:



当前值——A——B

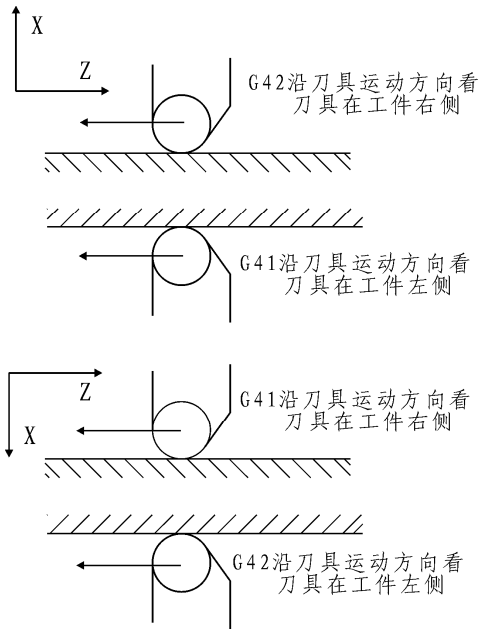
```
%ABC
N01 G54 G00 G90 X40 Z30
N02 G59
N03 G00 X30 Z30
N04 M30
```

(29）Tool radius compensate G40、G41、G42

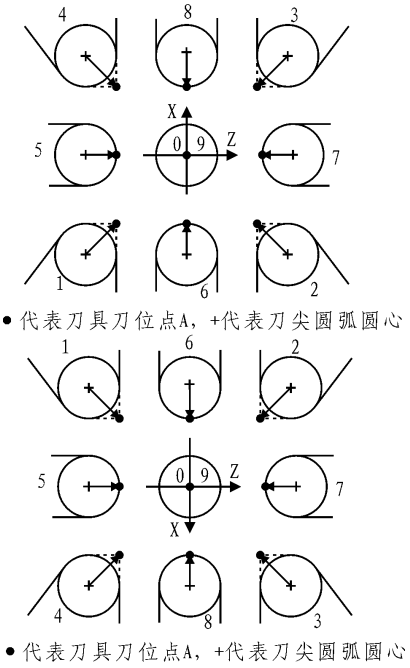
Format :G40 ; tool radius compensation cancellation
G41 ; tool radius compensation of cutter in the left of workpiece offset.
G42 ; tool radius compensation of cutter in the right of workpiece offset.

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

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

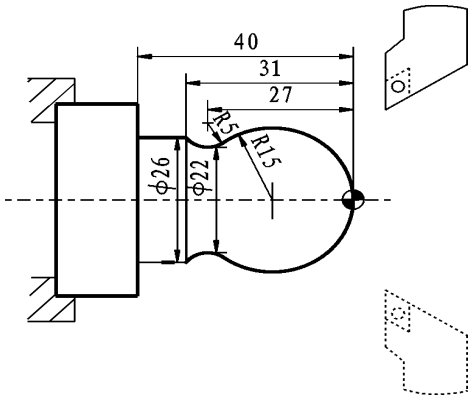


Left and right compensate



Tool parameter

For example：




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

N11 M30

（30） Pole coordinate program(G15/G16)

```
Format:
            G15          ; cancel
            G16 IP-(XZ)  ; pole coorninate
```

For example:

```
N1 G16 X0 Z0    ;
N2 G01 X30.0 Z100.0 F200.0
N3 X150.0       ;
N4 X270.0       ;
N5 G15          ;
N6 M02
```

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

```
Format:
          G20 ; inch;
          G21 ; millimeter;
```

（32）Call sub-program (M97/M98/M99)

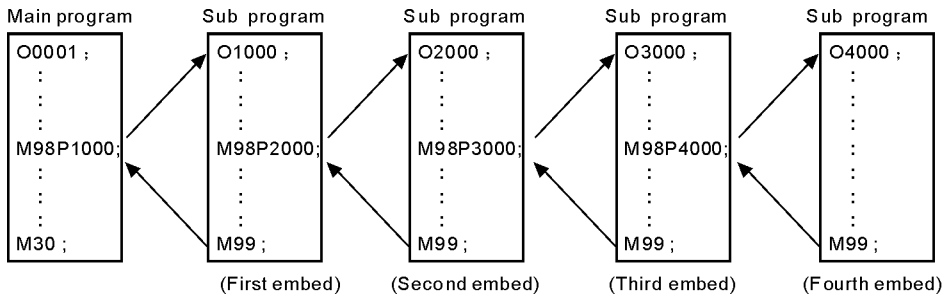
M97 P　　Non-condition to jump to P word

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

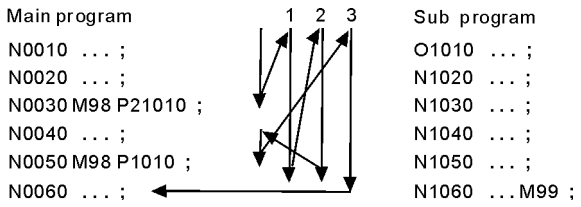
for example:　Psub\\%ab12 means the name of sub-program is CNC\\sub\\%ab12,　L word point out call times。

M99　　Back of sub-program

The Sub-program can embedded call as follow :



For example



（33）T tool

Format fuction
Tab a: Tool numbera，b: compensate number
Format:

```
N0000   T0101
N0001   G0  X30 Z500
N0002   T0303
N0003   G00  X50
N0004   T0505
N0005   M02
```

（34）S、SS SP speed

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

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

（35）F feed speed

F use for G01、G02、G03 etc. feed speed.
G98: F0.01-20000mm/min，G99: 0.001-500mm/r.

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

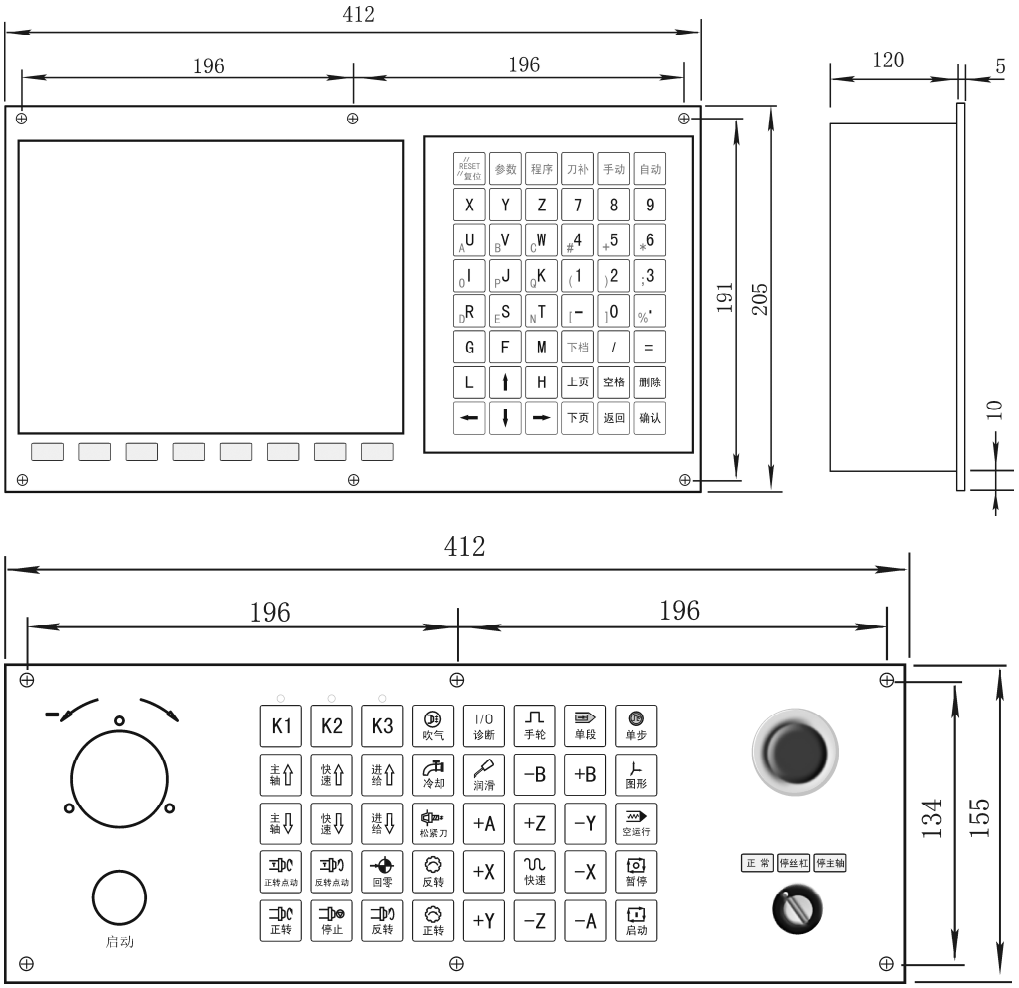
Fix all plugs with screw. Forbid to insert and extract all cables when power is on.

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.

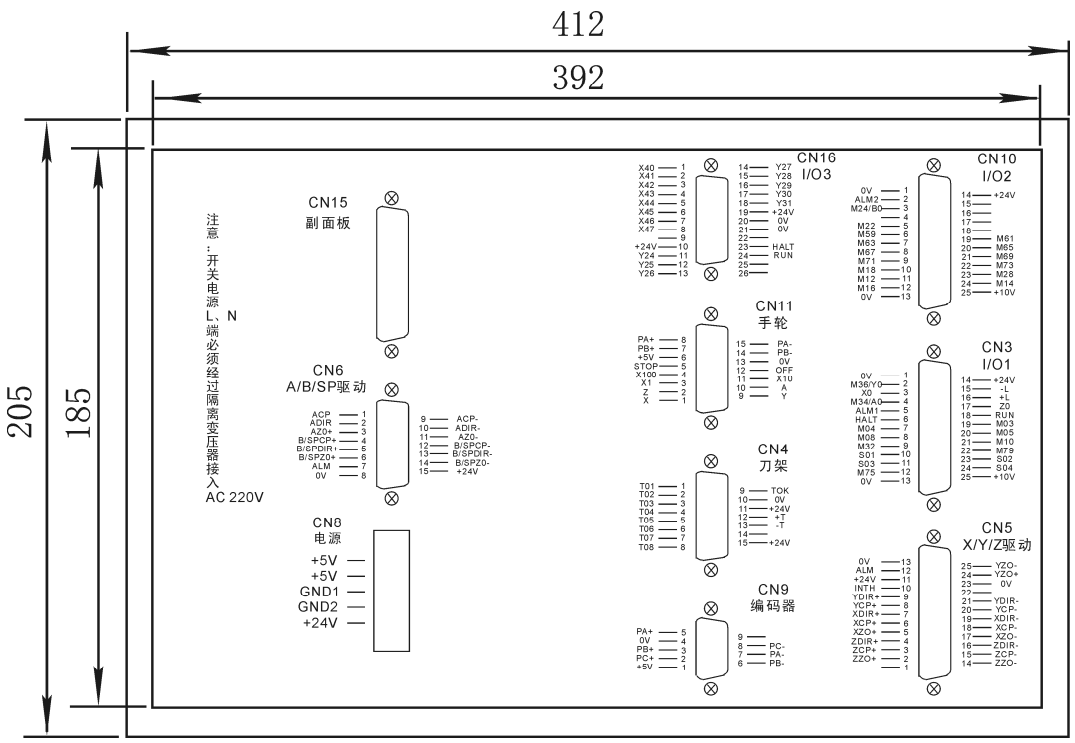
To ensure there is no strong magnet and current interference, keep away from inflammable, explosive and other danger materials.

5.2 system installation dimension

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

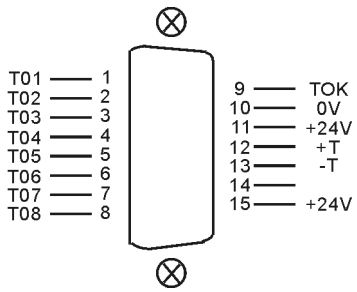


5.3 system rear view



5.4 interface connection graph

5.4.1 CN4 and electric tool carrier connection



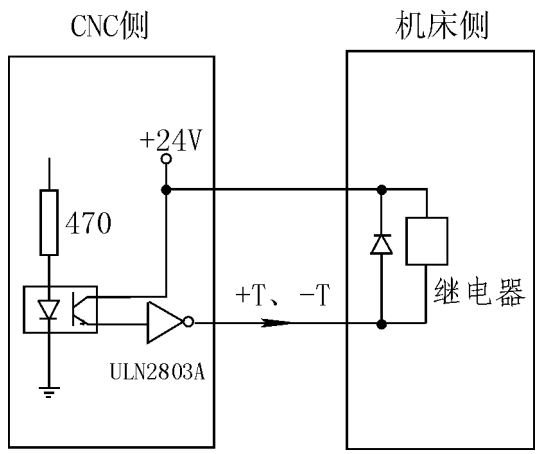
CN4 DB15(hole) electric tool carrier				
signal	pin	I/O	function	availability
0V	10	OUT	0V	0V
+24V	11、15	OUT	+24V	+24V
+T	12	OUT	Positive rotate	0V

-T	13	OUT	Reverse rotate	0V
T1	1	IN	T1 signal	0V
T2	2	IN	T2 signal	0V
T3	3	IN	T3 signal	0V
T4	4	IN	T4 signal	0V
T5	5	IN	T5 signal	0V
T6	6	IN	T6 signal	0V
T7	7	IN	T7 signal	0V
T8	8	IN	T8 signal	0V
TOK	9	IN	Lock signal	0V

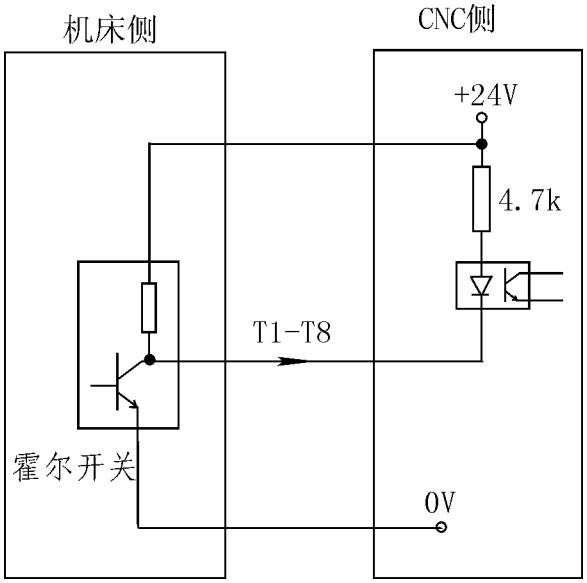
System can control 1-99 Tools. Active tools is 8 Tools,default is 4. Totality Tools setup in “Redeem” press “C”.Interfix Tool parameter as:

1,Active tool function
[1 mean Yes,0 mean No]
2,Active tool number
4,Tool positive rotate max-time(s)
5,Delay time after tool positive rotate(ms)
6,Delay time after tool stop(ms)
7,Tighten time of tool reverse rotate(ms)
9,Have total signal TOK(1 mean have)
10,C Tool radius compensation's establish(0 mean A,1 mean B)
11,C Tool radius compensation's cancel(0 mean A,1 mean B)
20,Active tool mode
[1 mean normal,0 mean coding tool]

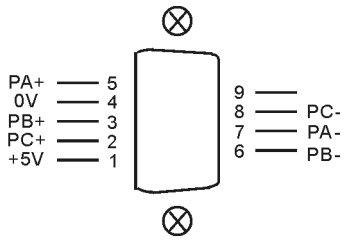
System output signal +T、-T:



Tool input T1～T8、TOK signal：

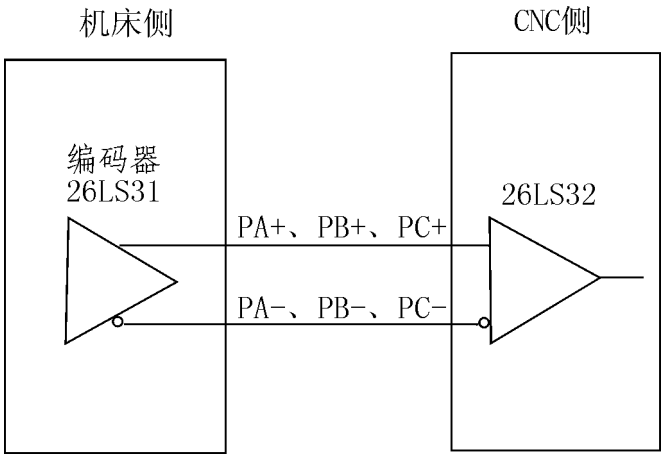


5.4.2 CN9 and spindle encoder connection

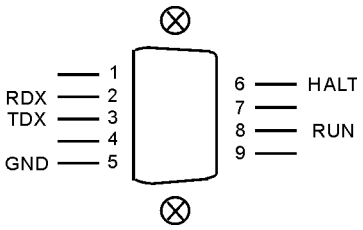


CN9 DB9(pin) spindle encoder				
signal	pin	I/O	function	availability
0V	4	OUT	0V	0V
+5V	1	OUT	+5V	+5V
PA+	5	IN	+A signal	5V
PA-	7	IN	-A signal	
PB+	3	IN	+B signal	5V
PB-	6	IN	-B signal	
PC+	2	IN	+Z signal	5V
PC-	8	IN	-Z signal	

Encode input signal PA、PB、PC:

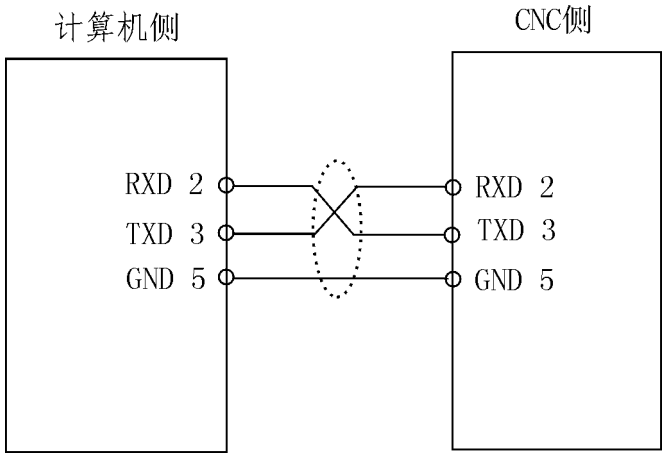


5.4.3 CN6 and computer system connection

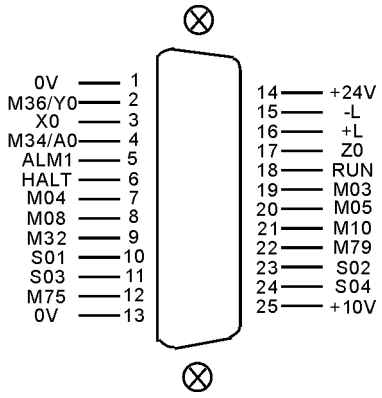


CN6 DB9(hole) RS232 communication				
signal	pin	I/O	function	availabilit
0V	5	OUT	0V	0V
RXD	2	IN	RXD	
TXD	3	OUT	TXD	
RUN	8	IN	run	0V
HALT	6	IN	pause	0V

CN6 connect fig:



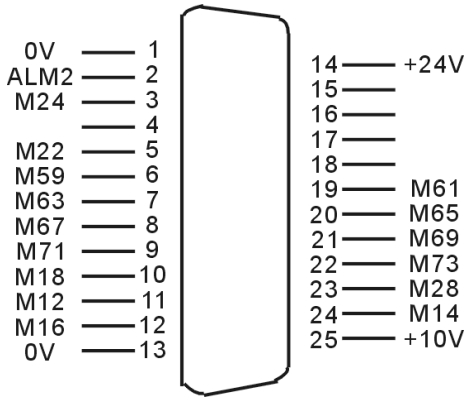
5.4.4 CN3 and machine electric device I/O1 connection



CN3 DB25(hole) I/O1 machine signal				
signal	pin	I/O	function	availabilit
0V	1	OUT	0V	0V
+24V	14	OUT	+24V	+24V
M36/Y0	2	IN	M36/Y0	0V
X0	3	IN	X axis Zero	0V
Z0	17	IN	Z axis Zero	0V
-L	15	IN	Positive limit	0V
+L	16	IN	Negative limit	0V
M34/A0	4	IN	M34/A0	0V
ALM1	5	IN	Transducer alarm1	0V
HALT	6	IN	Pause	0V
RUN	18	IN	Run	0V
M03	19	OUT	spindle clockwise	0V

M04	7	OUT	SP counter clockwise	0V
M05	20	OUT	SP stop	0V
M08	8	OUT	coolant	0V
M10	21	OUT	spindle chuck	0V
M32	9	OUT	lubricating	0V
M79	22	OUT	spindle tailstock	0V
S01	10	OUT	spindle first gear	0V
S02	23	OUT	spindle second gear	0V
S03	11	OUT	spindle third gear	0V
S04	24	OUT	spindle fourth gear	0V
M75	12	OUT	C axis mode	0V
+10V	25	OUT	the first spindle converting	0～10V
0V	13	OUT	0V	0V

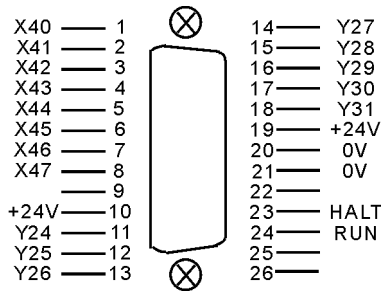
5.4.5 CN10 and machine electric device I/O2 connection



CN10 DB25(hole) I/O2 machine signal				
signal	pin	I/O	function	availability
0V	1	OUT	0V	0V
+24V	14	OUT	+24V	+24V
ALM2	2	IN	Machine alarm2	0V
M24/B0	3	IN	M24/B0	0V
M22	5	IN	M01 input	0V
M59	6	OUT	Huff	0V
M61	19	OUT	M61	0V
M63	7	OUT	M63	0V
M65	20	OUT	M65	0V

M67	8	OUT	M67	0V
M69	21	OUT	M69	0V
M71	9	OUT	M71	0V
M73	22	OUT	M73	0V
M18	10	IN	M18	0V
M28	23	IN	M28	0V
M12	11	IN	M12	0V
M14	24	IN	M14	0V
M16	12	IN	M16	0V
+10V	25	OUT	the second spindle converting	0～10V
0V	13	OUT	0V	0V

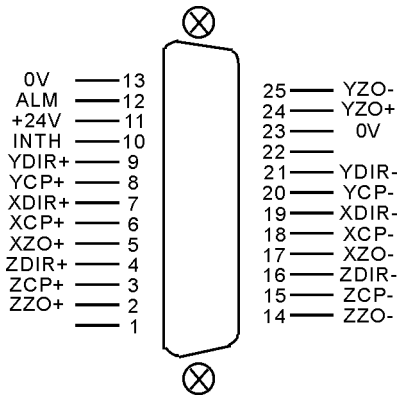
CN16 machine electric device connection



CN16 I/O3 DB26 (hole)				
signal	pin	I/O	function	availability
0V	20、21	OUT	0V	0V
+24V	10、19	OUT	+24V	+24V
X40	1	IN	inout0	0V
X41	2	IN	inout 1	0V
X42	3	IN	inout 2	0V
X43	4	IN	inout 3	0V
X44	5	IN	Inout4	0V
X45	6	IN	inout5	0V
X46	7	IN	inout 6	0V
X47	8	IN	inout 7	0V
Y24	11	OUT	output 0	0V
Y25	12	OUT	output 1	0V
Y26	13	OUT	output 2	0V

Y27	14	OUT	output 3	0V
Y28	15	OUT	output 4	0V
Y29	16	OUT	output 5	0V
Y30	17	OUT	output 6	0V
Y31	18	OUT	output 7	0V
RUN	24	IN	Run	0V
HALT	23	IN	Halt	0V

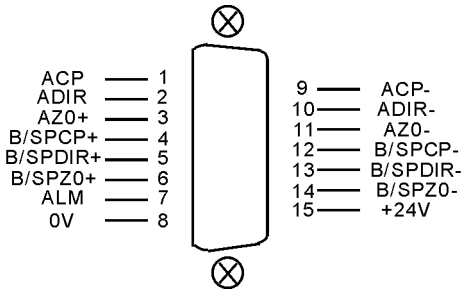
5.4.6 CN5 and servo drive & motor connection



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

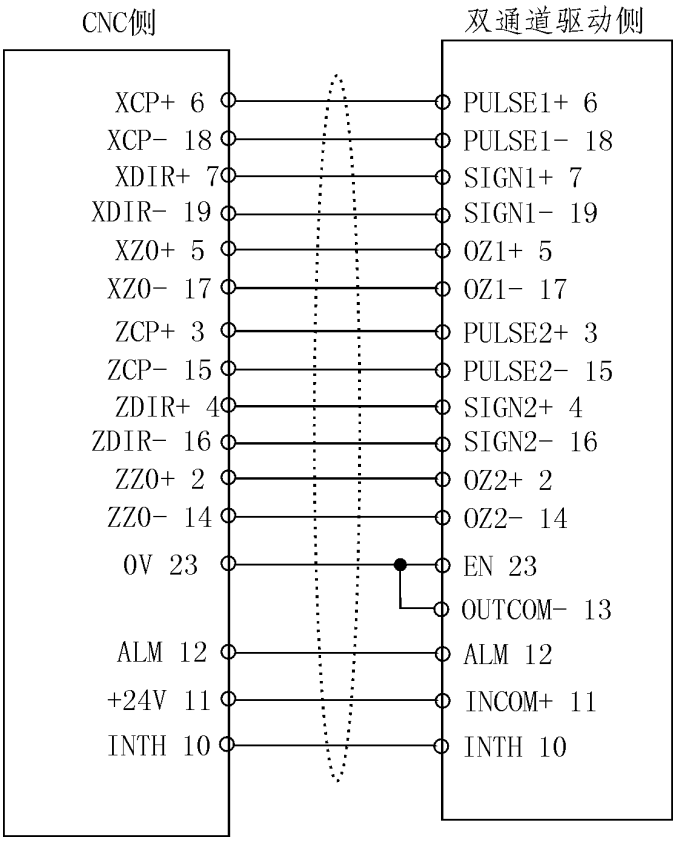
ZZO-	14	IN	Z motor Zero -	
YZO+	24	IN	Y motor Zero +	5V
YZO-	25	IN	Y motor Zero -	
0V	13、23	OUT	0V	0V
ALM	12	IN	Servo alarm	0V
+24V	11	OUT	+24V	24V
INTH	10	OUT	Clear alarm	0V

A/B (SP) servo drive CN6

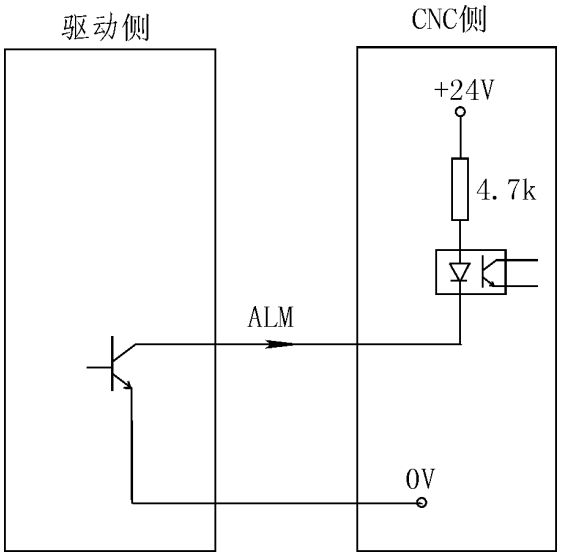


CN6 servo drive DB15(hole)				
signal	pin	I/O	Function	Availability
ACP+	1	OUT	A pulse signal+	5V
ACP-	9	OUT	A pulse signal-	
ADIR+	2	OUT	A direction signal +	5V
ADIR-	10	OUT	A direction signal -	
BCP+	4	OUT	B pulse signal+	5V
BCP-	12	OUT	B pulse signal-	
BDIR+	5	OUT	B direction signal +	5V
BDIR-	13	OUT	B direction signal -	
AZO+	3	IN	A motor Zero +	5V
AZO-	11	IN	A motor Zero -	
BZO+	6	IN	B motor Zero +	5V
BZO-	14	IN	B motor Zero -	
0V	8	OUT	0V	0V
ALM	7	IN	ALM	0V
+24V	15	OUT	+24V	24V

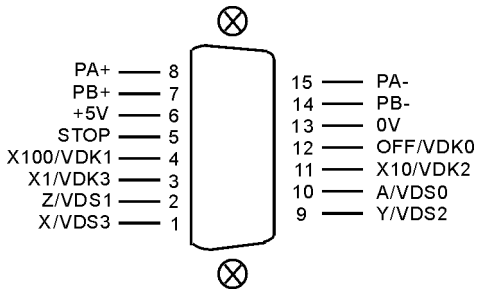
CN5 X、Z connect to our Co.' S servo drive:



Servo alarm signal:



5.4.7 CN11 and hand wheel, band switch connection

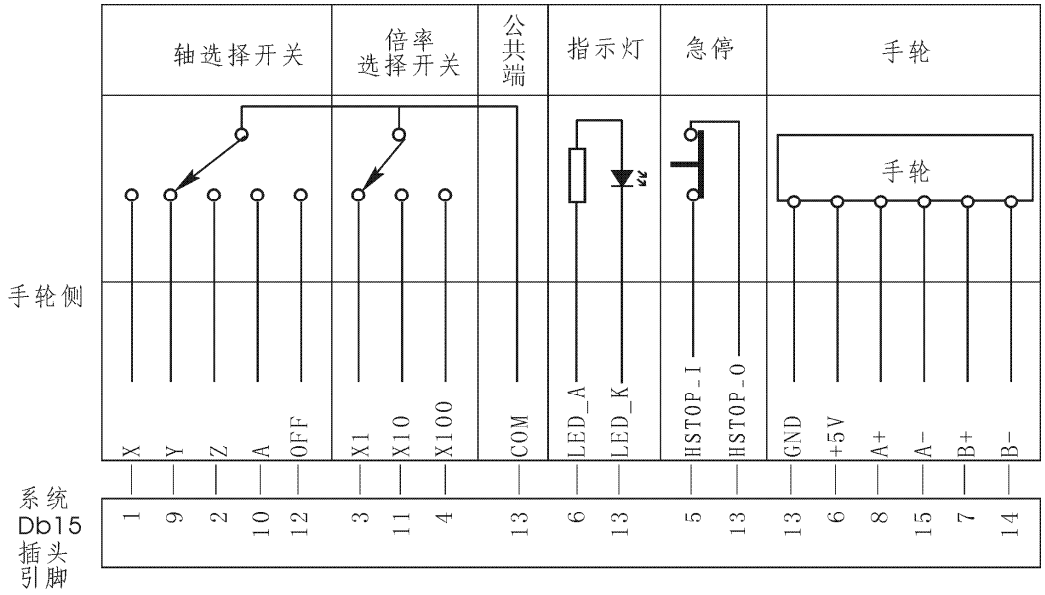


CN11 DB15(pin) hand wheel, band switch connection				
signal	pin	I/O	function	Availability
0V	13	OUT	0V	0V
+5V	6	OUT	+5V	+5V
PA+	8	IN	A signal +	5V
PA-	15	IN	A signal -	
PB+	7	IN	B signal +	5V
PB-	14	IN	B signal -	
STOP	5	IN	emergency stop	0V
OFF/VDK0	12	IN	Off/ feed amending 0	0V
X100/VDK1	4	IN	*100/ feed amending 1	0V
X10/VDK2	11	IN	*10/ feed amending 2	0V
X1/VDK3	3	IN	*1/ feed amending 3	0V
A/VDS0/HALT	10	IN	A/SP amending 0/halt stop	0V
Z/VDS1	2	IN	Z/SP amending 1	0V
Y/VDS2/RUN	9	IN	Y/SP amending 2/run	0V
X/VDS3	1	IN	X/SP amending 3	0V

5.4.7.1 hand wheel

When “Other parameter” P1=1, It will be pend handwheel,and do not use band switch. “Axis parameter”P1=0、P2=0. “Other parameter” P33=0、P34=0. Input sigal A、X、Y、Z、X1、X10、X100 is choice switch.

Handwheel contact diagrammatic as:



5.4.7.2 Band switch

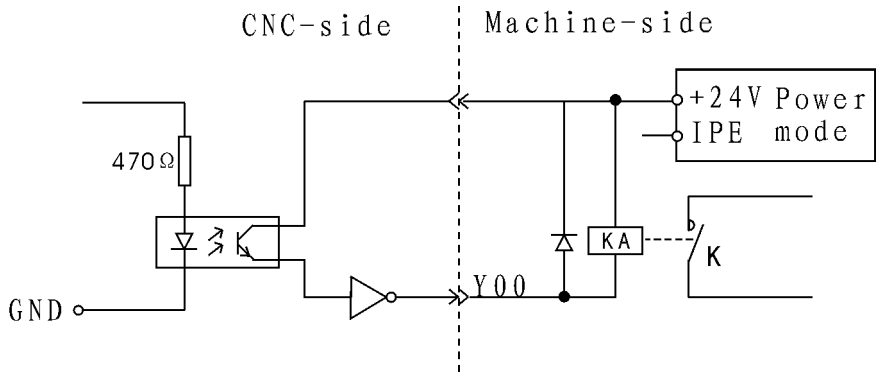
When “Axis parameter” P1=1、P2=1, It will be band switch.and do not use pend handwheel. “Other parameter” P1=0、P33=0、P34=0. Input sigal VDS0(A)、VDS1(Z)、VDS2(Y)、VDS3(X) are spindle speed adjust switch. VDK0(OFF)、VDK1(X100)、VDK2(X10)、VDK3(X1) are G01/G02/G03 speed adjust switch.

5.4.7.3 Emergency Stop

STOP sigal is extemal emergency stop input sigal. “Other parameter” P27 setup “CLOSE” or “OPEN”.

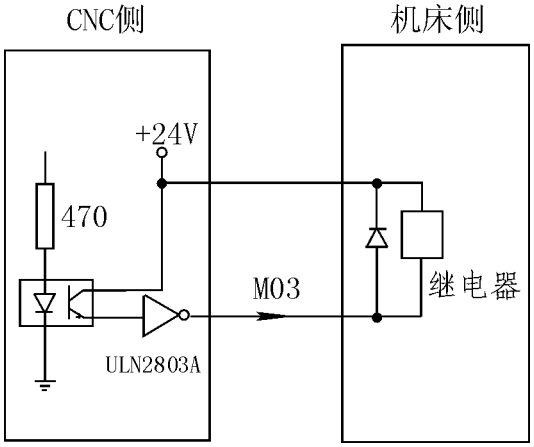
5.4.8 General, motion cotrol I/O output port principle which is availability by "0V"

Output port of Y00-Y23 are availability by "0V", the connection method as follow (take Y00 cotrol relay as example):



Specially pay attention: Because the output ports are the transistor output, thus the load electric current cannot be bigger than 150mA.

OUTPUT sigal example M03（M04、M05、M08、M10、M32、M79、M75、M59、M61、M63、M65、M67、M69、M71、M73、S01-S04）;



Notice:

1、IC ULN2803A controloutput sigal:

1)、U28: M59、M61、M63、M65、M67、M69、M71、M73

2)、U29: M03、M04、M05、M08、M10、M79、M32、M75

3)、U30: +T、-T、S01、S02、S03、S04、LRUN、INTH

2、User-defined M71/M70、M73/M72 will maybe stuck control sigal, “Other parameter” P20、P21 setup.

3、User-defined M65、M67、M69 will maybe stop\alarm\run output controlsigal; “Other parameter” P28、P29 setup.

4、All output sigal is valid by 0V.

5.4.8.1 CNC spindle control (M03/M04/M05)

Axis parameter:

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]
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)

Speed paramemter:

8,Spindle`s manual speed(rpm)
36,Spindle first max speed(rpm)
37,Spindle second max speed(rpm)
38,Spindle third max speed(rpm)
39,Spindle forth max speed(rpm)
40,Second Spindle max speed(rpm)

Other parameter:

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

5.4.8.2 CNC lubrication control (M32/M33)

Other parameter:

4,Have auto lubricate(0 yes/1 no)
5,Auto lubricate time(0.01s)
6,Auto lubricate stop time(s)

5.4.8.3 CNC stuck/ finial control (M10/M11、M79/M78)

Other parameter:

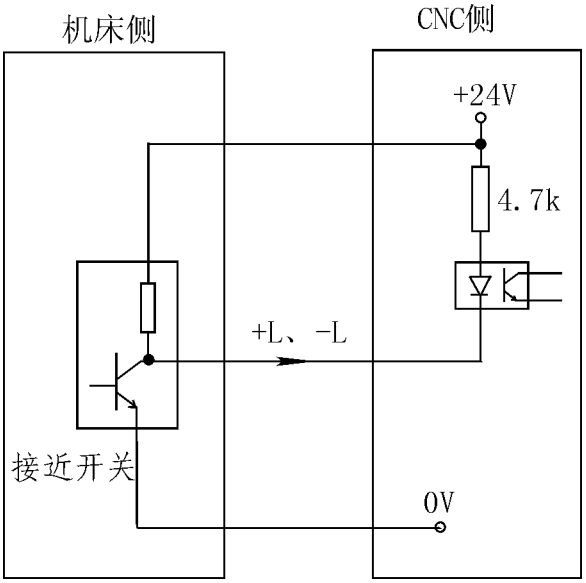
2,lather outside chuck
[1 extroversion,0 diffidence]
13,Does lock for Spindle & chuck(0 mean no)

15,Chuck clamp M10/loose M11 checking(1 mean need)
16,Finial forward M79/backward M78 checking(1 mean need)
20,stuck control signal(0 single,1 doubleM10/M71)
21,finial control signal(0 single,1 doubleM79/M73)
22,Outside chuck control(0 no,1 yes)
23,Outside finial control(0 no,1 yes)
24,M10M11 short signal time(s)
25,M79M78 short signal time(s)

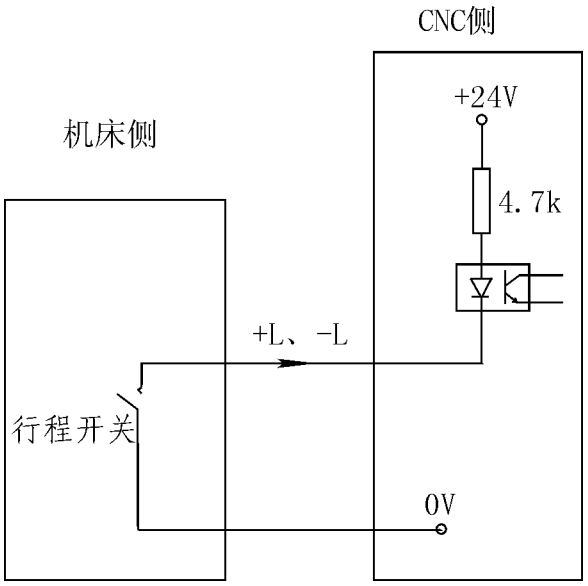
5.4.9 Reference points connections inpout port principle

5.4.9.1 Tool Limit

Take +L、-L axis as example
Mode1：NPN approach switch



Mode2：general switch

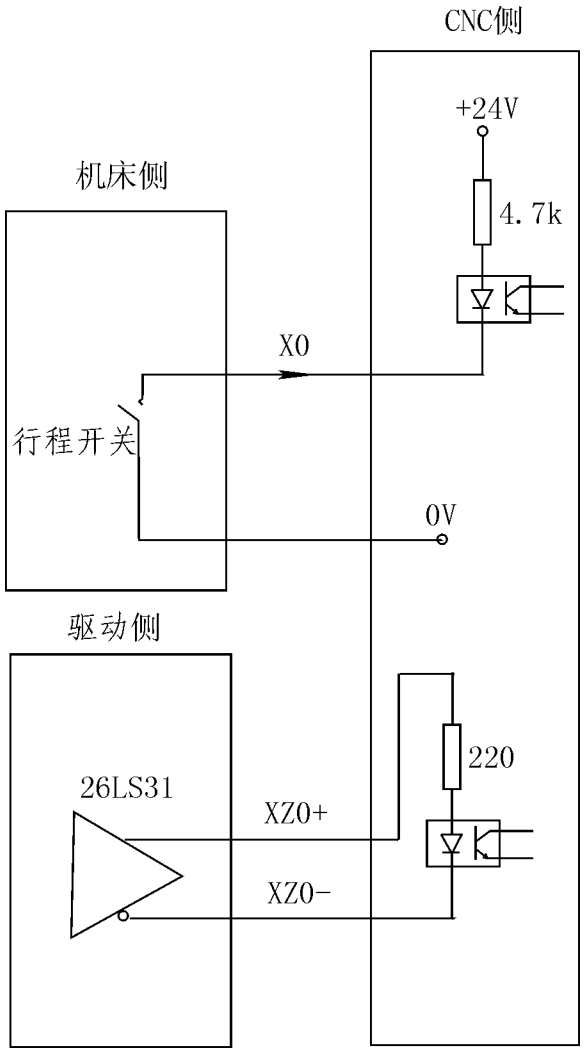


Axis parameter:

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

5.4.9.2 Tool Reference zero

Take X0、Z0 axis as example



Axis parameter:

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)

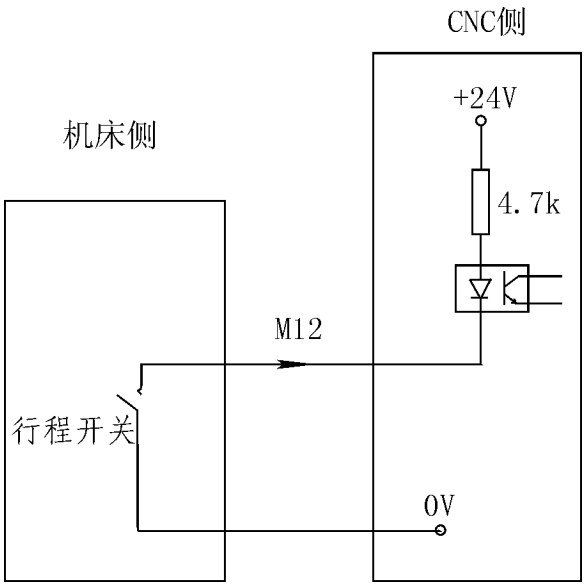
5.4.9.3 ALM、ALM1、ALM2、door alarm/M12、Emergency-stop

Other parameter:

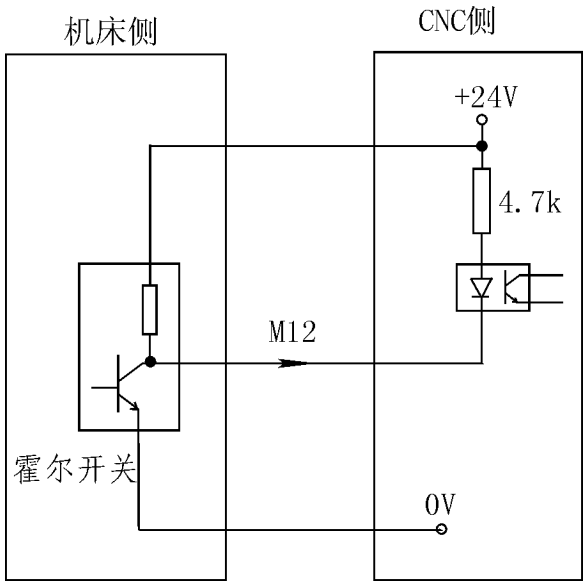
7,Door switch checking M12(0 no,1 yes)
8,Door switch(0 open,1 close)
17,servo ALM1 (0 open,1 close)
18,SP ALM2 (0 open,1 close)
26,Emerge Stop(0 open,1 close)
27,Emerge Stop2(0 open,1 close)
28,Run status output M69 STOP output M65(0 invalid,1 valid)
29,Alarm status output M67(0 invalid,1 valid)

5.4.9.4 User-defined M12 (M14、M16、M18、M28、M22、M24、RUN、HALT、Emergency-stop) sigal contact mode

Mode1: general switch



Mode2: NPN approach switch



Notice:

1、M12、M14、M16、M18、M28 are multifunctional sigale, only use one.

2、All input sigal is valid by 0V.

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.

6.2.3 System's control disorganize

1) Not correct operation.
2) The switch power's anti-jamming ability descend.
3) System's work circumstance become bad.

6.2.4 User's program lose

The DC battery on system main board can insure user's program and parameter don't lose。When system isn't used for half year or system has been used for over two years, the battery maybe invalidate, therefore, should exchange battery.

6.2.5 Machining precision is bad

1) CNC machine's reverse interval would change after using for a period of time, it needs to revise on time.
2) Best to revise base point before machining in order to insure the start point's precision.
3) Machining speed and cutting depth is improper.
4) Machine connector's prick melt falls off.
5) Tool isn't tightened.
6) Piece clamp isn't good.
7) Tool's giving up isn't equality because piece's dimension isn't uniformity.
8) Machine problem.

Chapter 7 Appendix

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

decimal	binary D7 ……D0		decimal	binary D7 ……D0
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.