NK868T Lathe Numerical Control System

Manufacturer Manual

(1st Edition)

Weihong Electronic Technology Co., Ltd.

The copyright of this manual belongs to Weihong Electronic Technology Co., Ltd. (hereinafter referred to as Weihong Company). This manual and any image, table, data or other information contained in this manual may not be reproduced, transferred, or translated without any prior written permission of Weihong Company.

Due to the hysteresis of the presswork, some updated contents of the product may not be input timely. Please pardon us if the above-said absence brings you any inconvenience. You can login to the official website of Weihong Company www.weihong.com.cn/en to download the latest PDF edition for free.

Preface

About This Manual

This manual is to describe the programming, operation and installation of NK868T lathe numerical control system.

For safe, normal and effective working of product, please read this manual carefully before using the product.

This manual can be divided into 4 parts, as follows:

1) Part1: programming, this part gives an introduction to general information of program foundation, G function, MST code, tool offset C function, macro program and Graph function.
2) Part 2: operating, this part gives an introduction to the operation of NK868T CNC.
3) Part 3: connecting, this part gives an introduction to the installation, connection and usual function of NK868T CNC, parameters and monitor.
4) Part 4: appendix, this part gives an introduction to the installation dimension of NK868T CNC and its attachment, the standard parameter, alarm info and additional information of NK868T CNC.

Contact Us

You can contact us by the following info for technical support and pre-sales/after-sales service:

Company Name: Weihong Electronic Technology Co., Ltd.

Headquarters Address: No. 1590, Huhang Rd., Fengxian, Shanghai, PRC 201401

Tel: +86-21-33587550

Fax: +86-21-33587519

Website: www.weihong.com.cn/en

Contents

Programming 1

Chapter 1 Program foundation 2

1.1 PRODUCT INTRODUCTION 2

1.2 FOUNDATION 5

1.2.1 Definition 5

1.2.2 Machine tool coordinate system 5

1.2.3 Workpiece coordinate system 6

1.2.4 Insert function 6

1.2.5 Absolute/relative coordinate 8

1.2.6 Diameter / radius programming 9

Chapter 2 G function 10

2.1 G00—LOCATING 10

2.2 G01—LINE INTERPOLATION 11

2.3 G01—CHAMFERING FUNCTION 12

2.4 G02, G03—ARC INTERPOLATION 13

2.5 G04—TIME DELAY 15

2.6 G28—RETURN TO MACHINE ZERO 16

2.7 G26—RETURN TO PROGRAM ZERO 17

2.8 G31—JUMP PROCESSING 17

2.9 G32—SINGLE-TOOL THREAD PROCESSING CYCLE 18

2.10 G33—RIGIDITY THREADING CYCLE 19

2.11 G34—SCREW THREAD CUTTING CHANGEABLE PITCH 20

2.12 G92—THREAD CUTTING SINGLE CYCLE 20

2.13 G86—METRIC THREAD CUTTING COMPOUND CYCLE 23

2.14 G87—ENGLISH SYSTEM PROCESSING CYCLE 24

2.15 G40,G41,G42—CUTTING TOOL RADIUS COMPENSATION 25

2.16 G50—SETTING WORK PIECE COORDINATE SYSTEM 25

2.17 G98—FEED PER MINITE 25

2.18 G99—FEED PER ROTATION 26

2.19 G96,G97—CONSTANT LINEAR SPEED CUTTING 26

2.20 G94—SURFACE CUTTING CYCLE 28

2.21 G70—FINISH MACHING CYCLE 30

2.22 G71—EXCIRCLE/INNER CIRCLE ROUGH TURNING CYCLE 31

2.23 G72—END FACE ROUGH TURNING CYCLE 34

2.24 G73—CLOSE CUTTING CYCLE 38

2.25 G75—END SURFACE DEEP HOLE PROCESSING CYCLE 41

2.26 G76—EXCIRCLE/INNER CIRCLE GROOVING/CUT OFF CYCLE 43

2.27 G77—MULTIPLE THREAD CUTTING CYCLE 44

2.28 THE OTHER DECRIPTION OF THE FIXED CYCLE 48

2.29 G54,G55——COORDINATE CHANGE 48

2.30 G90/G91—PROGRAMMING WITH ABSOLUTE/INCREMENT MODE 50

2.31 AUTO ACCELERATE AND DECELERATE 51

2.31.1 Speed control in the corner of the program 51

Chapter 3 MST code 53

3.1 M CODE 53

3.1.1 M00—Program stop 54

3.1.2 M01—Condition pause 54

3.1.3 M03—Spindle positive rotation 55

3.1.4 M04—Spindle reverse 55

3.1.5 M05—Spindle stop 56

3.1.6 M08 M09—Cooling fluid 57

3.1.7 M10 M11—Workpiece clamp, loosen control 57

3.1.8 M14 M15—Servo spindle speed/position mode transition 57

3.1.9 M20, M21, M22—Specified relay control 57

3.1.10 M26, M27, M28—Rotation axis(Y axis) speed control 58
3.1.11 M30—End of program 58
3.1.12 M31—Workpiece count 59
3.1.13 M32 M33—Lubricate open, close 59
3.1.14 M35—Automatically repeat loading funcction 59
3.1.15 M78 M79—Tail enter, retreat control 60
3.1.16 M91 M92—Jump function 60
3.1.17 M98 M99—Sub program call and return 60
3.1.18 M63/M64/M65—The second spindle anticlockwise/clockwise/stop control 62
3.1.19 Assist function code call sub program 62

3.2 S FUNCTION 63
3.2.1 Spindle speed command 63

3.3 T FUNCTION 64
3.3.1 Cutting tool changing process 64
3.3.2 Cutting tool changing parameter 64
3.3.3 Cutting tool compensation function 65

Chapter 4 Tool offset C function 66

4.1 BASIC CONCEPTION 66
4.1.1 Image tool nose concept 66
4.1.2 Image tool nose direction 68
4.1.3 Set compensation amount 70
4.1.4 Relative position of tool and workpiece 70
4.1.5 Inside\outside 72
4.1.6 G41, G42 and G40 code format 72

4.2 TOOL OFFSET CONDITION 73
4.2.1 Tool nose radius compensation track resolve 73

4.3 TOOL OFFSET CANCLE 78
4.3.1 Change compensation direction during tool offset process 79
4.3.2 Tool offset cancle 80
4.3.3 Tool offset has non-moving command 82
4.3.4 Tool offset intervene check 84

4.4 TOOL OFFSET C NOTE 86

4.5 TOOL OFFSET C EXAMPLE 87

Chapter 5 Macro program 88

5.1 DEFINITION 88

5.2 USER MACRO CODE 88

5.3 USER MACRO PROGRAM ONTOLOGY 89

5.4 MACRO VARIABLE 89

5.5 CALCULATE COMMAND AND TRANSFER COMMAND G65 91

5.6 CALCULATE COMMAND 95

5.7 TRANSFER COMMAND 97

5.8 ATTENTION PROBLEMS 98

5.9 MACRO PROGRAM EXAMPLES 99

Chapter 6 Graph function 100

6.1 OPERATION DESCRIPTION 100

Operating 101

Chapter 1 Operation and display 102

1.1 PANEL 102

1.1.1 Keyboard introduce 103

1.1.2 State display 110

1.2 OPERATING MODE OVERVIEW 110

1.3 DISPLAY 111

1.3.1 Position page 113

1.3.2 Program page 117

1.3.3 Tool offset, macro variables page 120

1.3.4 Alarm page 123

1.3.5 Set page 125

1.3.6 Graph page 126

1.3.7 Parameter page 127

1.3.8 Diagnose page 129
1.3.9 U disk page 130

Chapter 2 Safe operation 131

2.1 STARTING UP 131

2.2 POWER OFF 131

2.3 OVERRUN PROTECT 131
2.3.1 Hardware 131
2.3.2 Software 131

2.4 EMERGENCY OPERATION 132
2.4.1 Emergency 132
2.4.2 Reset 132
2.4.3 Feed to keep 132
2.4.4 Cut off the power supply 132

Chapter 3 Manual 133

3.1 AXIS MOBILE 133
3.1.1 Manual feed 133
3.1.2 Manual fast moving 133
3.1.3 Speed modify 134

3.2 OTHER MANUAL OPERATION 134
3.2.1 The spindle rotation control 134
3.2.2 Spindle pulse control 134
3.2.3 Coolant 135
3.2.4 Lubrication 135
3.2.5 Chuck 135
3.2.6 Tailstock 135
3.2.7 Change tool manually 135
3.2.8 Spindle rate adjustment 135

Chapter 4 Handwheel and single step 136

4.1 SINGLE STEP FEED 136
4.1.1 Incremental choice 136

4.1.2 Direction selection 136

4.2 HANDWHEEL FEED 136

4.2.1 Incremental choice 136

4.2.2 Axis and direction choice 136

Chapter 5 MDI 137

5.1 THE TRADITIONAL MDI 137

5.2 QUICK WAY OF MDI 138

Chapter 6 Program operation 140

6.1 THE PREPARATION BEFORE PROGRAM STORAGE, EDIT OPERATION 140

6.1.1 Preparations before edit: 140

6.1.2 Transfer data using RS232 serial communication port: 140

6.2 THE COMPOSITION OF THE PROGRAM 140

6.2.1 The program generally constitute 140

6.3 THE MAIN PROGRAM /SUBROUTINE 141

6.4 ESTABLISH A NEW PROGRAM 142

6.5 THE PROGRAM NAME TO RETRIEVE 143

6.6 DELETE AND COPY OF THE PROGRAM 144

6.7 PROGRAM CHARACTER SEARCH AND LINE JUMP 147

6.8 APPLICATION NOTE 147

6.9 THE NUMBER OF STORED PROCEDURES AND STORAGE CAPACITY 147

6.9.1 Program memory information display 148

6.10 ABOUT THE IMPLIED PROGRAM AND USE MXX TO CALL SUBROUTINE 148

6.10.1 Function definition 148

Chapter 7 Tool offset and tool setting 150

7.1 SHIFT TOOL PRINCIPLE 150

7.2 TOOL SETTING 150

7.2.1 X/Z tool setting separate 151

7.2.2 X/Z tool setting at the same time 152

7.3 THE SETTING AND MODIFICATION OF THE TOOL OFFSET VALUE 152
7.3.1 Tool offset 152
7.3.2 Toolpost offset 153
7.4 TOOL COMPENSATION RESET 153

Chapter 8 Automatic operation 154

8.1 AUTOMATIC RUNNING 154
8.1.1 Program selection 154
8.1.2 Start 154
8.1.3 Stop 155
8.1.4 Starting from the arbitrary segment 155
8.1.5 Feed keeping 155
8.1.6 Feed, fast speed adjustment 156
8.1.7 The spindle speed adjustment 156
8.1.8 Three switch functions 156
8.2 RUNNING STATE 157
8.2.1 Single segment 157
8.2.2 Machine tool lock 157
8.2.3 Jump Period 158

Chapter 9 Back to zero operation 159

9.1 THE PROGRAM BACK TO ZERO 159
9.1.1 The program zero point 159
9.1.2 Back to zero step 159
9.2 THE MACHINE TOOL BACK TO ZERO 160
9.2.1 The machine tool zero point 160
9.2.2 The back to zero step 160

Chapter 10 The data of setting, backups and recovery 163

10.1 SETTINGS 163
10.1.1 Switch 163
10.1.2 Time 163
10.1.3 Password 163

10.2 THE DATA RECOVERY AND BACKUPS 163

10.3 RECOVERY, BACKUP OF SYSTEM FILES 164

Chapter 11 U disk operation 165

11.1 HOW TO EXPORT TO USB 166

11.1.1 Program files 166

11.1.2 Parameter file 166

11.2 HOW TO IMPORT TO THE SYSTEM 167

11.2.1 Program files 167

11.2.2 Parameter files 167

Connecting 168

Chapter 1 Installation layout 169

1.1 SYSTEM CONNECTION 169

1.1.1 System composition 169

1.1.2 System installation and connection 169

1.1.3 Back cover interface layout 170

1.1.4 Interface specification 170

1.2 SYSTEM INSTALLATION 171

1.2.1 Outline dimension 171

Chapter 2 Interface signal definition and connection 172

2.1 CONNECTION WITH DRIVER UNIT 172

2.1.1 Interface signal connection 172

2.1.2 Pulse and direct signal(nCP,nCW) 172

2.1.3 Alarm signal(nALM) 173

2.1.4 Enable signal(nEN) 173

2.1.5 Pulse forbidden signal(nSET) 174

2.1.6 Zero signal 174

2.1.7 Connection with driver unit 174

2.2 SPINDLE ENCODER 175

2.2.1 Interface definition 175

2.2.2 Description 176

2.3 HANDWHEEL CONNECTION 177

2.3.1 Handwheel interface definition 177

2.4 SPINDLE CONNECTION 178

2.4.1 Definition 178

2.4.2 Transducer connection 178

2.4.3 Connect with servo spindle 179

2.5 HAND-SET SUB CONNECTION: 182

2.6 IMPORT/OUTPORT 184

2.6.1 Pin definition 184

2.6.2 Input port 186

2.6.3 Output port 188

Chapter 3 Usual function description 191

3.1 SETTING ELECTRONIC GEAR RATIO PARAMETER 191

3.2 LINEAR DECELERATION TIME CONSTANT 192

3.3 SETTING SPINDLE PARAMETER 193

3.4 CHUCK CONTROL 193

3.5 HANDWHEEL 195

3.6 TAILSTOCK 195

3.7 LUBRICATION 196

3.8 SOFT LIMIT 197

3.9 BACKLASH COMPENSATION 197

3.10 TIMING AND COUNTING 198

3.11 M26 ROTATION FUNCTION 198

3.12 PARAMETER STORAGE FUNCTION 198

3.13 KEYBOARD DIGNOSE FUNCTION 198

3.14 PRINT SCREEN FUNCTION 198

3.15 HANDWHEEL TRY CUT FUNCTION 198

3.16 BRIGHTNESS ADJUSTING FUNCTION 198

Chapter 4 Parameter 199

4.1 DESCRIPTION 199

4.1.1 Bit parameters 199

4.1.2 Data parameters 199

Chapter 5 Monitor 200

5.1 I/O FIXED ADDRESS 200

5.1.1 Import state 200

5.1.2 Output port state 205

5.1.3 Date state 209

5.2 KEYBOARD DIGNOSE 209

Chapter 6 Pitch compensation parameters 210

6.1 PITCH COMPENSATION FUNCTION 210

6.2 THE STEPS OF THE PITCH COMPENSATION PARAMETERS SETTED 210

6.3 THE ATTENTION ABOUT PITCH ERROR COMPENSATION 210

6.4 PITCH ERROR COMPENSATION FOR EXAMPLE 211

Appendix 212

Appendix: 1 Standard parameter list 213

Appendix: 2 Alarm list 233

Appendix: 3 Common alarm solve method 238

Appendix: 4 System upgrade instructions 240

Appendix: 5 Connected with servo 241

Appendix: 6 Connected with Lio Shing Turret 242

Appendix: 7 Second tool carrier use method..... 245

Programming

Chapter 1 Program foundation

1.1 Product introduction

This system is CNC lathe machine tool dedicated control system. It is new product which updated software and hardware based on NK868T. This system adopts 32-bit flushbonading CPU and super large scale programmable device CPLD, and it applies real time multitask control technique and hardware insert technique. It realises μm Grade precision moving control. Its LCD ressolution is 800× 600. It is Chinese or English operation interface. Operation is so easy that improves processed efficiency, presition and surface quality. As a updated product of NK868T, this system is a best choice of economical CNC lathe machine tool system. Segment interim processing technic of system's program is in the leading position of the same trade that provides vast users' technique update a larger image space and valid terrace.

Main technique parameters:

Function	Description	Parameter
Control axis	Control axis	3-axis （x , z , c）
	Universal driving shaft	3-axis
Input command	Min set unit	X : 0.001mm z : 0.001mm
	Min moving unit	0.001mm
	Max set value	±99999.999mm
Feed	Max moving speed	60000mm/min
	Lead of screw thread	0.001mm～500.000mm
	Acc/dec auto	Line
	Feed speed rate	0～150%
	Fase speed rate	Fo～100%,fo depend on paramter
Manual	Manual feed	X,z ; set by keyboard
	Return to machine tool zero point	Manner: b:pulse interrupt, c:positioning switch, a: floating zero
	Return to program zero point	Process starting point
	Step increment feed	0.001mm,0.01mm,0.1mm,1mm
	Handwheel feed	Rate: x1,x10,x100 axis: x, z, c
Insert	Location,insert function	Line, circle, thread feed, tapping cycle, drilling, excircle, surface complex cycle

Function	Description	Parameter
Storage and edit	Program storage capacity	32m byte
	Storage program number	480
	Program edit	Insert,modify,delete,copy
	Parameter storage	Backup,factory data reset,import and export parameter
Display	Lcd display	8”
	Positon,program, ,alarm,dignose,parameter,setting,u disk,figure	Rich and ocular in content
U disk	Import and export program	Have
	Mport and export parameter	Have
	U disk update	Have
Serial communicate	RS232	19200bps, in-out progam
M, s, t function	Input port	54 switch, optoelectronic isolation
	Output port	48 switcch
	Spindle	Transducer analog control or s1～s4 gears control, output rate can adjust from 0 to 150.
	Tool	Tool num:t01-t08 ,compensation num:01-24. electric tool carrier,platoon tool carrier or special tool carrier,modify tool compensation by program.
	Aid	Have
	Aid m function	Have
Mdi manner	Fast mdi	Input command directly at position page.
	Traditional mdi	Input command at mdi page
Compensation function	Compensation	Tool compensation,reverse blacklash compensation, thread pitch mistake compensation.
Cutting cycle	G94	End surface cutting cycle
	G92	Thread cycle
	G86, G87	Thread compound cycle
	G70,G71,G72,G73	Compound cycle
	G74	Surface drilling
	G75	Grooving or cut off cycle

Function	Description	Parameter
	G76	Multiple thread cut cycle
	G33	Rigidity threading cycle
Other thread functions	G32	Solo drive thread cycle
	G34	Varying pitch thread cycle
Chamfer function	G01 L/R	Line or circle chamfer
Signal jump function	G31	Signal jump in service
Segment smooth transition	G61, G64	Dynamic adjust
Infinite\ finite cycle	M92	Partial program cycle
Condition jump	M91	External signal
Extend output port	M20,M21,M22	Output port or pulse output mode control
Condition pause	M01	Run when external signal input
Export repeat	M35	Upper and lower material
Rotation axis （y）	M26,M27,M28	Rotation speed and direct control
Others	Chuck	Inside or outside, keyboard or foot switch input.
	Lubrication	Continue or interval
	Count by time	Current time and total time
	Count by pieces	Current amount and total amount
	Three position switch	Have
	Indicator of run,pause,alarm state	Have
	External start,pause	Have
Dignose display	Import state	Have
	Outport state	Have
	Axis pulse	Have
	Encoder line number	Have
	Analogue voltage	Have
	Import connection	Have
	Export connection	Have
	Keyboard dignose	Have
Safety function function	Hardware limit	Have
	Software limit	Have
	Emg	Have

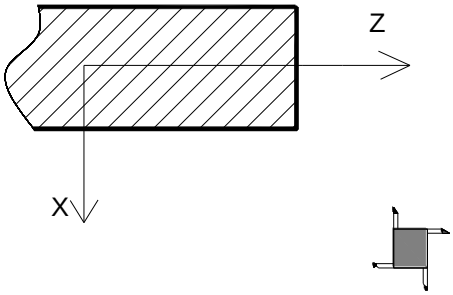
Function	Description	Parameter
	Customer alarm	Have
Debug	Single operation machine tool lock	Have
	Hand wheel trial cut	Have
Driver interface	Servo or stepping	Control mode : direct plus pulse
Spindle	Servo spindle/normal spindle	Have
	Double spindle	Have

1.2 Foundation

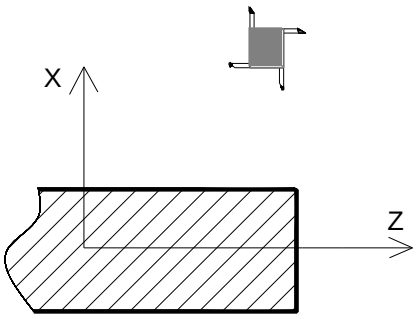
1.2.1 Definition

This lathe CNC system controls two axes, X axis and Z axis component rectangular coordinate system which is used for location and insert movement.

X axis is fore and aft, Z axis is left-right direct, close to workpiece is negative direct, while is positive direct. As shown front and back tool apron in pictures, X direct is different, but Z direct is the same. The following content is described as front tool apron.



Front tool apron



Back tool apron

1.2.2 Machine tool coordinate system

Machine tool coordinate system is the basic system which is used for cnc calculating.

Machine tool zero is a fixed point, usually it is set in the largest travel place. In the place, there is a block calls machine tool zero switch. If there is no block, the return to machine tool zero function (G28) can not be used. Or you can set P006 Bit0～Bit2 zero to close axis return zero function.

1.2.3 Workpiece coordinate system

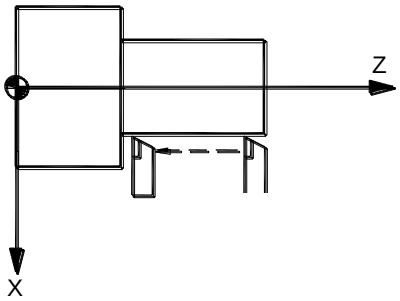
Workpiece coordinate system is a rectangular coordinate set by DWG drawing , also can be called float coordinate system. When workpiece is fixed to machine tool, command G50 to set current absolute coordinate. Usually Z axis of workpiece coordinate system is dead in line with spindle.X axis is in the top or bottom of workpiece. This workpiece coordinate system is valid until the next one appeared.

1.2.4 Insert function

Linear,circular motion and thread cutting function is called insert.

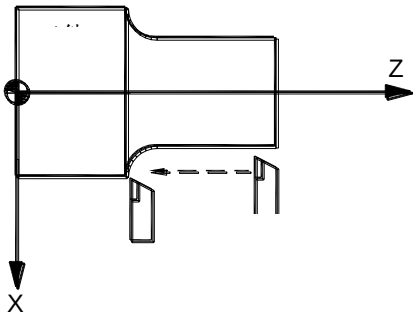
Command G01, G02, G92 is called prepared function which indicate the insert category.

1. Linear motion



Command: G01 Z__

2. Circular motion

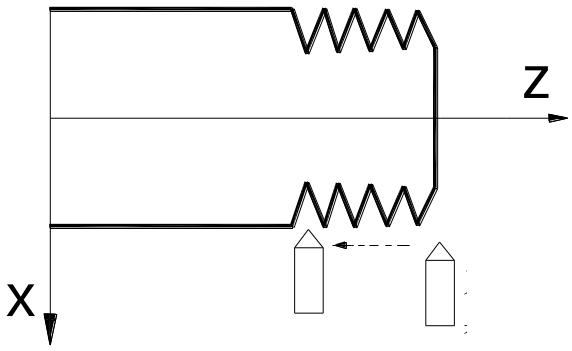


Command: G02 X__ Z__ R__; or G03 X__ Z__ R__;

3. Thread cutting

Depend on lead of screw thread, tool motion and spindle rotation is synchronizing.

(1) Straight thread

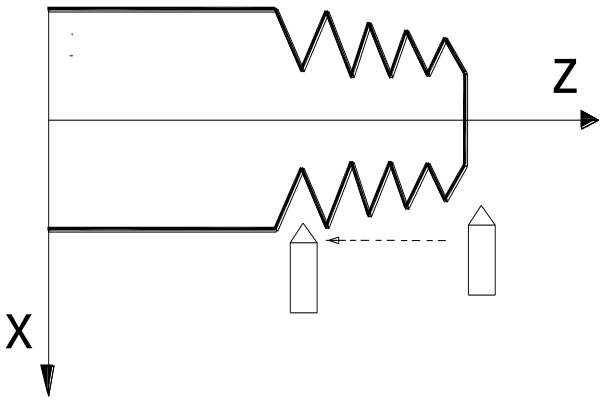


Command:

Thread circular processing: G92 X(U) Z（W） F/I

Solo drive thread cycle: G32 Z（W） F/I

(2) Taper thread



Command

Thread circular processing : G92 X(U) Z（W）R F/I

Solo drive thread cycle : G32 X(U) Z（W）R F/I

1.2.5 Absolute/relative coordinate

After workpiece coordinate system is set, all position coordinate value is relative to it. Absolute coordinate(X,Z) ,relative coordinate(U,W) and mix coordinate(X,U) all can be used as program value.

1. Absolute coordinate value

Absolute coordinate program is programmed with absolute coordinate value.

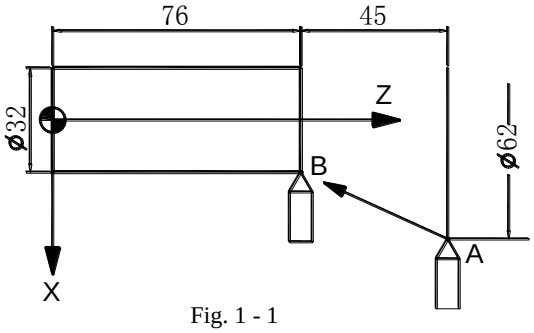


Fig. 1 - 1

As shown in pictures, tool moves from A point to B point, command is:

X32.0 Z76.0;

2. Relative coordinate program

Relative coordinate program is programmed with relative coordinate value.

As shown in pictures, tool moves from A point to B point, command is:

U-30.0 W-45.0;

3. Mix coordinate program

Mix coordinate program is programmed with absolute coordinate value and relative coordinate value at the same segment.

As shown in pictures, cutting tool moves from A point to B point, command is:

U-30.0 Z76.0; or X32.0 W-45.0;

1.2.6 Diameter / radius programming

There are two methods for lathe cnc program like diameter and radius programming. As shown in fig. 1-2.

When diameter programming is selected, value after X or U stand for diameter value.

While it means radius value.

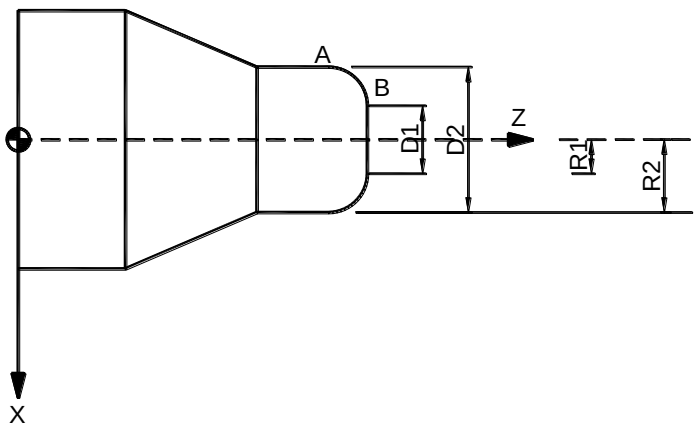


Fig. 1-2

Diameter programming attention:

Project	Attention
Z command	None of program method's business
X command	Diameter
Coordinate system set(G50)	Diameter
Tool compensation	Diameter
G92,G94 cutting depth	Radius
Circular insert command	Radius
X direction feed speed	Radius

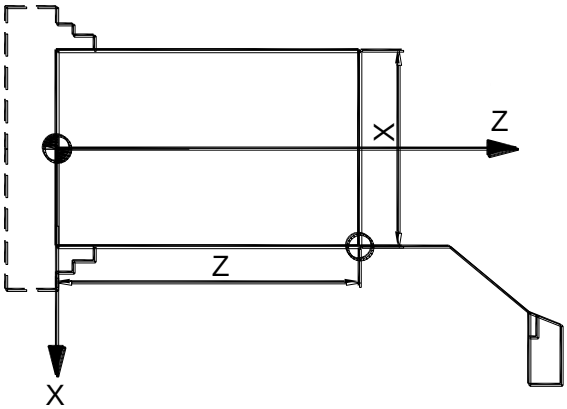
Chapter 2 G function

G code is composed of G with one or two digits following it. It is used to define the track's geometry and CNC working status.

2.1 G00—Locating

Format: G00 X (U) __Z (W) __;

Locating with G00, tool moves to definition location as fast moving speed, each axis has its own set speed.



Example: fig. 2-1, fast locating programming as follows:

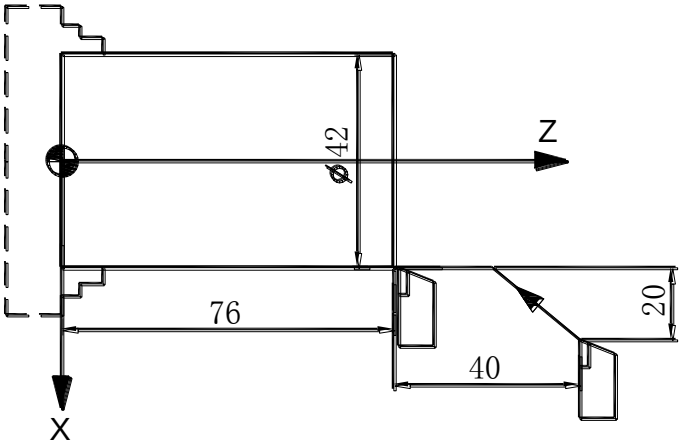


Fig.2-1

G0 X42.0 Z76.0; or G0 U-20.0 W-40.0;

Note: when G00 the fast moving speed is defined by manufacturer(decided by P035,P036,P037). Controlled by rapid rate (F0, 25%, 50%, 100%). It has none business with F value.

2.2 G01—Line interpolation

Format : G01 X(U)__Z(W)__F__ ;

Line interpolation G01, value of X Z or U W can be written with absolute value or increased value; its feed speed operates according to F value. F value is modality, when function is programmed, it is always valid until it's replaced by another modality function when the same quality.

Example:Fig. 2-2 tool track line interpolation (diameter program):

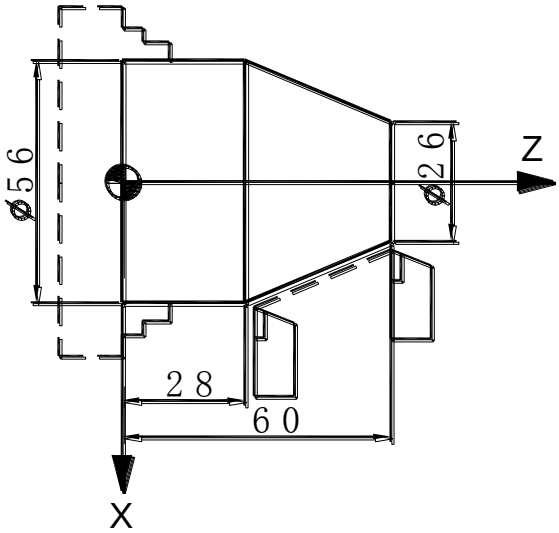


Fig. 2-2

G01 X56.0 Z28.0 F100 ; or

G01 U30.0 W-32.0 F100 ;

G01 line interpolation, feed speed calculate as follows:

G01 Uα Wβ Ff

X axis feed speed : Fx= $\frac{a}{L}$ * f

Z axis feed speed : Fz= $\frac{\beta}{L}$ * f

L= $\sqrt{\alpha^2+\beta^2}$

2.3 G01—Chamfering function

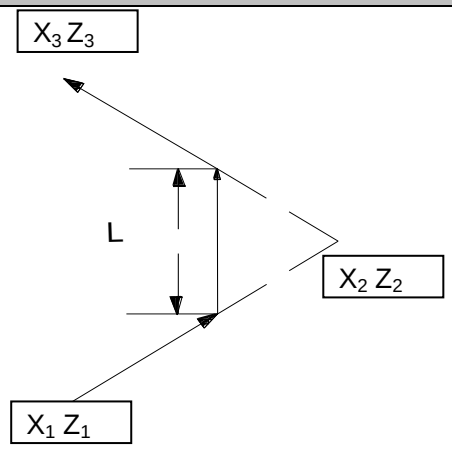
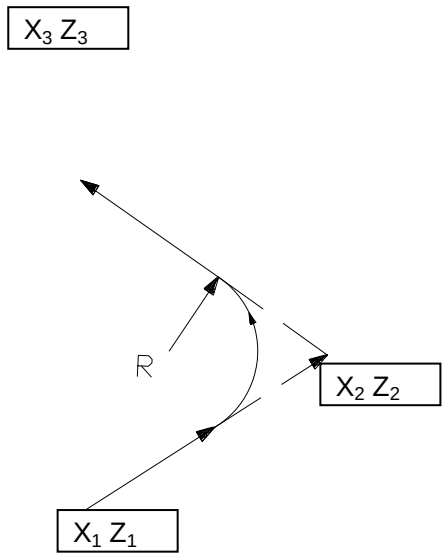
Format : G01 X(U)__Z(W)__L__F__ ;

G01 X(U)__Z(W)__R__F__ ;

L value is chamfer length.

R value is chamfer arc radius, its symbol:（-）G02 or （+）G03.

G01 line interpolation, between the two line programs which two have a cross, it can realize chamfer by programming. Format:

Chamfer manual	Program manual	Chamfer track
Line chamfer	**G01 X(U) Z(W) L F** G0 X_1 Z_1 ; G01 X_2 Z_2 L F100 ; G01 X_3 Z_3 ;	
Arc chamfer	**G01 X(U) Z(W) R F** G0 X_1 Z_1 ; G01 X_2 Z_2 -R F100 ; G01 X_3 Z_3 ;	

Note: program of instruct chamfer must be G01 command.

2.4 G02, G03—Arc interpolation

Format : G02 X_Z_ R_F;

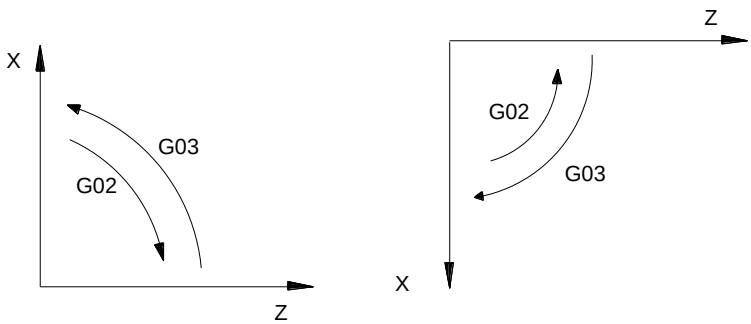
G02 X_Z_ I_K_F;

G03 X_Z_ R_F;

G03 X_Z_ I_K_F;

Word	Content	Significance
G02	Rotation direction	Clockwise arc CW
G03	Rotation direction	Inverse circle CCW
X, Z	Absolute value	Destination coordinate
U, W	Relative value	Distance frome start to end
I.K	Center of the circle coordinate	Distance from Center of the circle to start
R	radius	Distance from the point on the circle to the center of the circle
F	Feed speed	Speed along circle

Clockwise or inverse is according to the standard coordinate system provision,as follows:



Right hand rectangular

G02 X.. Z.. I.. K.. F..

or

G02 X.. Z.. R.. F..

(absolute)

(diameter)

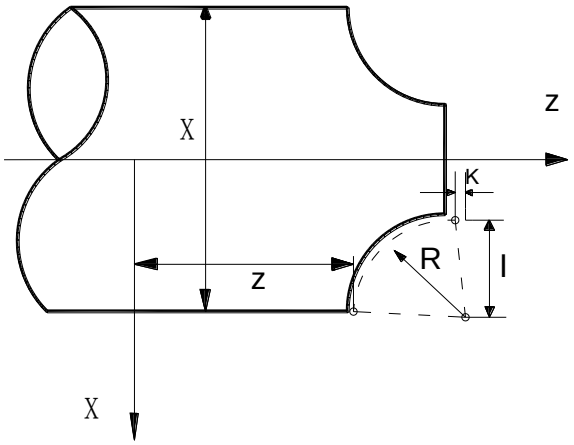
G03 X.. Z.. I.. K.. F..

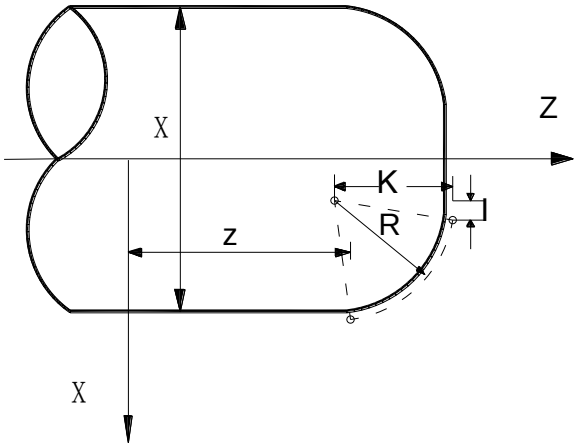
or

G03 X.. Z.. R.. F..

(absolute)

(diameter)





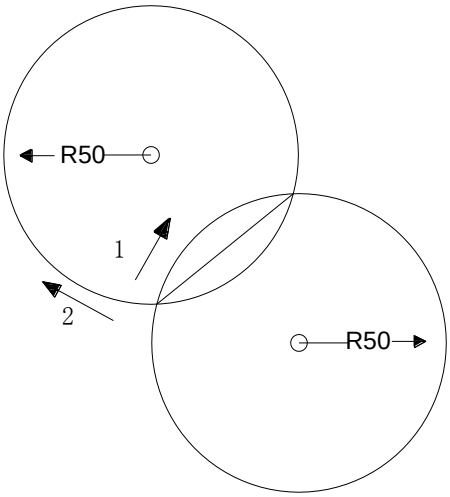
X/Z or U/W is destination to the circle which showed as absolute value or increase value.

I , K is the number with symbol . R value also can be used to replase. As follows:

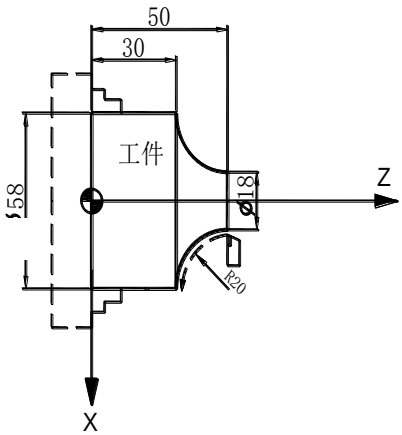
G02 X_Z_R_F_ ;

G03 X_Z_R_F_ ;

We can draw two circles as follows like greater than 180° and less than 180°. Greater than 180° can't be specified.



As follows, cutting tool is doing arc interpolation :



program by absolute and increased manners respectively:

program with I, K:

G02 X58.0 Z30.0 I20.0 K0 F30 ; or

G02 U40.0 W-20.0 I20.0 K0 F30 ;

program with R :

G02 X58.0 Z30.0 R20 F30 ; or

G02 U40.0 W-20.0 R20. F30 ;

The feed speed of aic interplotation specifies F which is the speed along circle.

Note 1: when program with I , K, system will validate the starting point, the ending point and the center of the circle; if the ending point isn't in circle, the radius from the ending point to the center of the circle differs to the radius from the starting point to the center of the circle more than value of Para 217, the system will prompt alert 117, "the ending point incorrect". It can directly program over the quadrant circle and full circle.

Note 2: the full circle can't be programmed with R.

Note 3: R is the arc's radius which is the number with symbol, "+" signifies that the arc angle is less than or equal to 180°; "-" signifies that the arc angle is greater than 180°.

Note 4:when program with R, if diameter is less than the distance from the starting point to the ending point, the system will prompt alert : "the ending point incorrect".

Note 5: when over the quadrant, X axis or Z axis maybe reverse, if clearance offset in the parameter region differs much to the practical reverse clearance of machine tool, which will generate obvious cut mark on the work piece.

Note 6:when value of x or z isn't programmed, default is last coor value. Default I,K zero.

2.5 G04—Time delay

Format :

G04 X__ ; //time delay command

G04 U__ ; // time delay command

G04 P__ ; // time delay command

G04 ; // time delay command

X,U,P is the delay unit :

Address	X	U	P
Unit	second	second	0.001 second

Time delay the next program carried on, time is decided by block.

Time range :0.001～99999.999s

like : G04 X10 ; //delay 10s

G04 P10 ; //delay 0.010s

If there is no other instructions in the G04 block, it means stop correctly.

As follows :

N0010 G64 ;

N0020 G01 U－10 F100 ;

N0030 G04

N0040 W－20 ;

Between N0020 and N0040, inserted N0030 G04, it means after finishing N0020, when speed is 0, N0040 can be carried on.

If there is no N0030 G04, the system will solve the speed between N0020 and N0040, that can produce an aic in the corner.

2.6 G28—Return to machine zero

Format : G28 X（U）__Z(W)__ ;

This command can make the axis return to reference point,

X（U）__Z(W)__ specify the middle point return to reference through of the way, use absolute or increased value.

1) Locating from the current position to the middle point （A—B）as the speed of rapid return zero(set by Para 101), picture 2-3.
2) Move from the middle point to the reference point （B—R）as the speed of rapid return zero.
3) After detected the reduce signal, check the fine positioning signal as the low speed (set by Para100).
4) After detected the fine positioning, return to machine tool zero coordinate finished, light of zero is on.

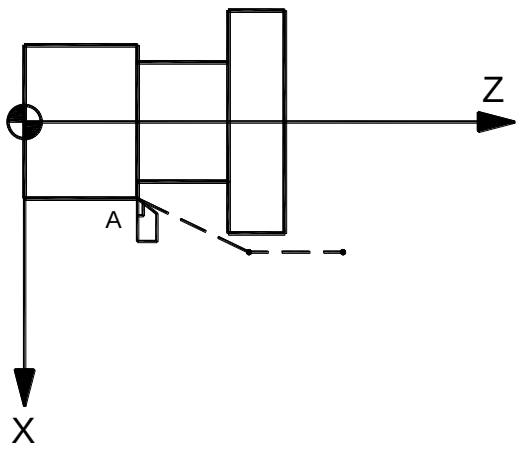


Fig. 2-3

Note 1:the move direct from middle point to zero point is decided by P.005 Bit0, Bit1, Bit2.

Note 2 :if the starting point is disagree with the reference, after return zero, it can return to program zero(defined by P111, P112, P113) by fast locating(G0).

2.7 G26—Return to program zero

Format : G26 X Z ;

There is no other instructions after X or Z, the program zero coordinate is decided by P111, P113.

When carry on the block G26, return to program zero point from the current position as the rapid manual speed.

2.8 G31—Jump processing

Format : G31 X(U)__Z(W)__L/K__ F__

X(U), Z(W) : the destination coordinates

F : feeding speed

L : detect the low level valid import

K : detect the high level valid import

Description : When carry on the block G31, feeding as the speed of F before detected the exterior level valid. If detecting the level valid before returns to the ending point, it will stop feeding and jump to the next block; if it has not detected the level valid before returns to the ending point, it will not carry on the next block until the current block is over. The import number can be checked in diagnostic interface. The details of the method see chapter five.

Example 1:Fig. 2-4, trajectory, A – B – D': no jump trajectory

Carry on G31 W160 L8 F100

G0 U60

The program continues on as the speed of F100, and check the eighth import at the same time. When system detects low level valid, program will stop and jump to carry on G0 U60. So the trajectoryis A－C－D.

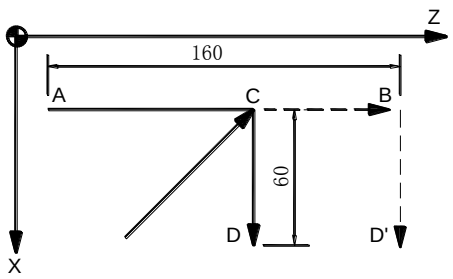


Fig. 2-4

2.9 G32—Single-tool thread processing cycle

G32 function can cut Straight thread and Taper thread of the same de-trailing length.

Straight thread format : G32 Z(W)__F/I__ ;

Taper thread format : G32 X（U）__Z(W)__F/I__ ;

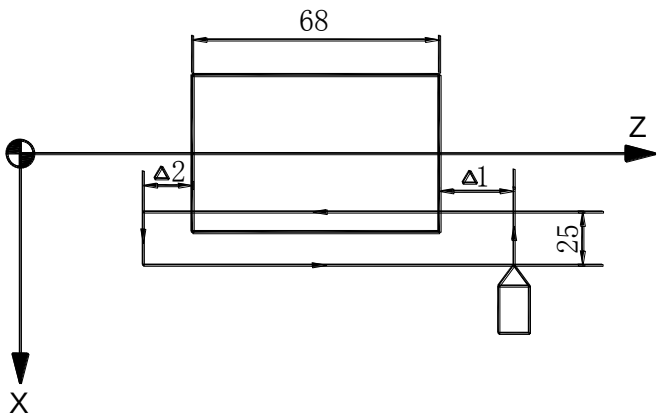
X（U）Z（W） : thread terminal coordinate ;

F : Metric system, cut de-trailing length（0.001—500.000mm）.

I : English system, teeth per inch (1—25400）

At the beginning and the end of cutting , for the reason of acc and dec of speed, some of cut de-trailing length is incorrect. For this reason , instruction thread length should be longer than necessary.

Example 1 : Straight thread



Z direction : △1=3mm, △2=1.5mm

De-trailing length : 3mm

Teeth deep : 2mm（Metric system, diameter） :

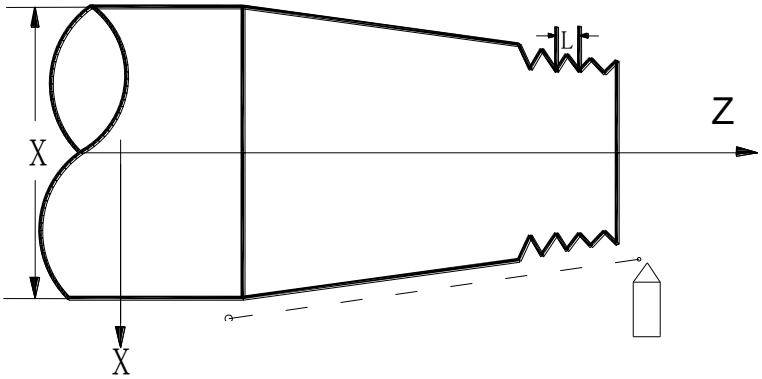
G00 U-25.0 //position

G32 W-72.5 F3.0 //length 72.5mm

G00 U25.0 //X direction

W72.5 //Z direction

Example 2 : Taper thread



Assume that de-trailing length is 4mm, starting point is（15, 40）, ending point is （30, 10）,

G00 Z40

X15.0 //position

G32 X30.0 Z10 F4 // Taper thread

G00 X40.0

Note 1 : Feeding speed rate and spindle rate have no business to thread cutting .

Note 2 : During thread cutting spindle cannot stop, feed to keep is invalid.

2.10 G33—Rigidity threading cycle

Format : G33 Z(W)__ F(I)__ J__ ;

Z, W : threading bottom absolute coordinate

F : Metric system rigidity threading teeth pitch（unit : mm）

I : English system rigidity threading teeth number per inch

J : When the threading feed arrives at Z(W) value , the system will give off spindle stop signal, and when spindle reduces to rotating speed J , the system will give off spindle reverse signal , which the reversing time is saved. If J isn't compiled, spindle reduces to zero then give off spindle reverse signal. When spindle has large inertia and reduces speed low, J is need in order to reduce following distance.

G33 action order :

Start spindle before cutting. When Z axis gets to the bottom of hole, it will output stop spindle signal. If J has value, when rotation speed lower than J, it will output reverse signal,while it will output signal when rotation speed equals to 0. When Z axis gets to the starting point, speed down to 0.

You can adjust Z direct follow-deviation by parameter 173.

2.11 G34—Screw thread cutting changeable pitch

Format : G34 X(U) __Z(W) __F/I__K__

Among them: X(U) : absolute (relative) coordinate of thread ending point in X direction

Z(W) : absolute (relative) coordinate of thread ending point in Z direction

F: metric-system thread, initial metric thread, screw threads initial lead（0.001-500.000mm）.

I : english-system thread, , thread the initial number of teeth per inch（1—25400teeth/inch）

K : thread increase distance or decrease distance per rotation, range : 0.001～500mm or 0.0001～9.9999 inch/tooth.

2.12 G92—Thread cutting single cycle

G92 can program thread cutting like straight thread,taper thread,multiple thread and random fixed feed angular. Because G92 can set screw tail length, thread cutting don't need relief groove.

Among them : Z(W) : absolute (relative) coordinate of thread ending point in Z direction, modal

X(U) : absolute (relative) coordinate of thread starting point in X direction

R : distance of thread head radius relative to thread ending radius value, for taper thread, modal

K: Z axis tail length, lead of thread screw,modal,unit mm.

J: X axis tail length,radium program, default is back to the cutting point, unit mm.

F: metric thread length, modal,unit mm

L: multiple thread head number, default single head, modal.

Q: thread start angular, default is 0, non-modal.

When thread cutting, the same track need thread cutting for many times. Adopt G92 can make thread cutting much more easier. Thread cutting starts from detected the spindle encoder zero pulse signal. Therefore even if cutting for many times, the starting point is the same and the track is the same too. From rough turning to finishing turning, the rotate speed should be stable,or else thread will exist deviation.

Among G92 command, the value is modal. As long as the first segment set thread parameter, the next can be leave out. Such as process straight thread of 1.2mm screw lead, 10mm length:

After each cycle, tool return to cutting point.

……

N0090 G0 X10 Z0 ;

N0100 G92 X9.5 Z-10 F1.2

N0110 X9.0

N0120 X8.9

N0130 G0 Z10 ;

Explan some kinds of thread cycle program below:

(a) straight thread cutting cycle

As shown in fig. 2-6, G92 cutting order of execution:

Quick position -> wait for head pulse -> cutting feed -> quick tail -> tool quickly back

1————————————————————2————3————4

Track 1: quick position from current location to the location that X (U) set, and wait for head pulse.

Track 2: cutting feed depend on spindle rotation position and speed.

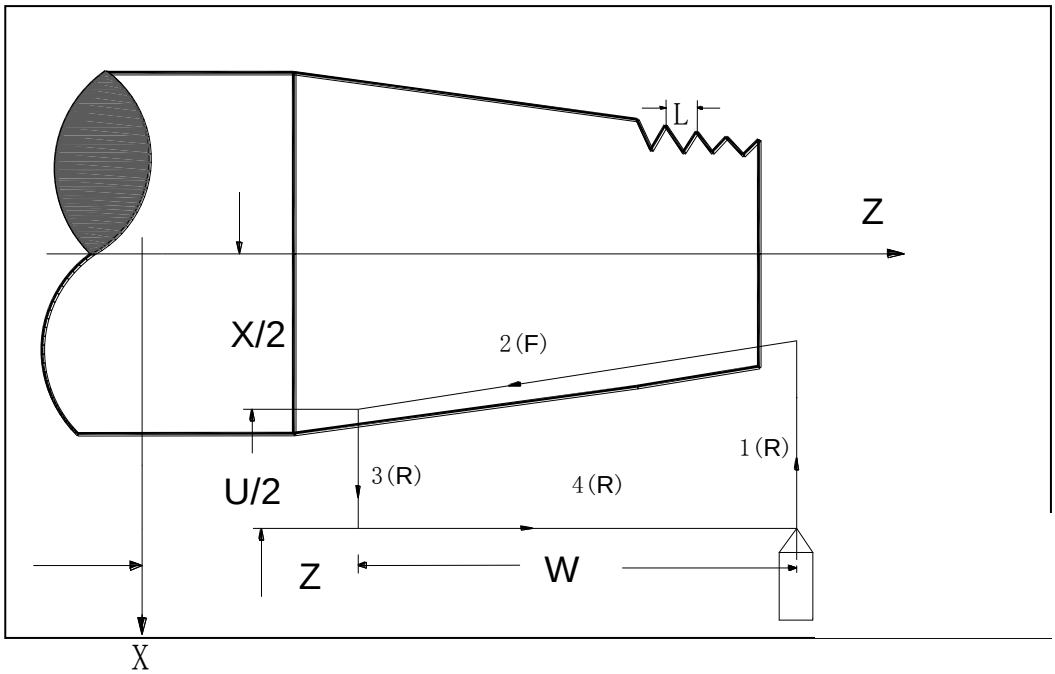
Track 3: begin to quick tail and X axis tool quick back .

Track 4: quick position to the cutting point, if next segment is G92, loop,while thread processing completed.

(b) taper thread cutting cycle

G92X（U）__Z(W)___R__F__ ;

G92X（U）__Z(W)___R__I__ ;



R is radius difference between thread head and tail.

Quick position -> wait for head pulse -> cutting feed -> quick tail -> tool quickly back

1————————————————————2————3————4

Track 1: quick position from current location to the location that X (U) set, and wait for head pulse.

Track 2: cutting feed along taper direct depend on spindle rotation position and speed.

Track 3: begin to quick tail and X axis tool quick back .

Track 4: quick position to the cutting point, if next segment is G92, loop,while thread processing completed.

(c) set thread cutting start angle

G92 can set thread cutting start angle(relative to spindle encoder head pulse position), set by parameter Q.

Q : thread cutting start angle, range 0-360. Default 0. non-modal.

When spindle rotation to Q, thread begin to cut feed.

(d) multiple thread

Parameter L is used for thread head number setting, default 1.

Cutting order of execution:

Quick position -> wait for head pulse -> cutting feed -> quick tail -> tool quickly back

1————————————————————2————3————4

-> quick position ->wait for scale division -> cutting feed -> quick tail -> tool quickly back

1————————————————————2————3————4

-> quick position ->wait for scale division

1————————————

Cycle until process is over.

For example: process 4-head thread of 8mm screw lead, 48mm length, workpiece diameter is 20.5mm, divided to 5 times:

G0 X22 Z2

G92 X20 W－50 F8 L4 // cut depth 0.5mm, divided to 4 times, every time feed angle is 90 degrees.

X19.6 // cut depth 0.4mm, divided to 4 times, every time feed angle is 90 degrees

X19 // cut depth 0.6mm, divided to 4 times, every time feed angle is 90 degrees

X18.6 // cut depth 0.4mm, divided to 4 times, every time feed angle is 90 degrees

X18.4 // cut depth 0.2mm, divided to 4 times, every time feed angle is 90 degrees

G0 X30

Note 1: feed speed rate and spindle rate are nothing to thread cutting.

Note 2: spindle cannot stop in the process of cutting, feed hold is invalid in thread cutting.

Note 3: feed hold is invalid in thread cutting.

Note 4: thread cutting in order of 1,2,3,4 to single carry on if single function is open.

2.13 G86—Metric thread cutting compound cycle

Format : G86 X(U) Z(W) K R I/D J L P Q

Among them : X(U) : absolute (relative) coordinate of thread ending point in X direction

Z(W) : absolute (relative) coordinate of thread ending point in Z direction

K : thread pitch, unit : mm

R : The diameter difference between thread outside diameter and root diameter, which is positive value(unsigned number)

I : the de-trailing length after thread is completed in X direction, the positon is decided by I.

D : screw in distance value, diameter, I and D cannot exist at the same time.

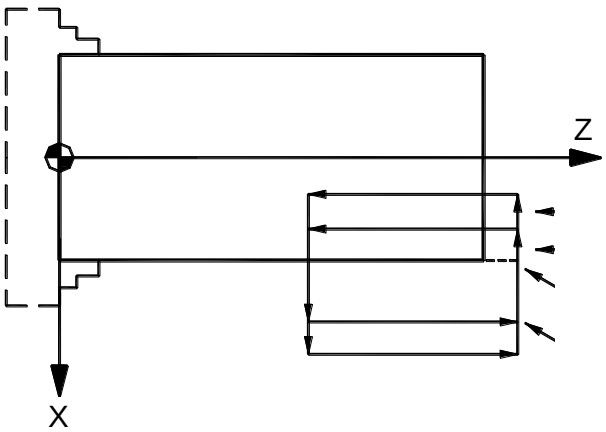
J : Z direction de-trailing length（plus positive value）. If program has not specified the value, it is cetermined by Para 165.

L : cycle number

P : thread head number. If program has not specified the value, it is default 1.

Q : thread feeding depth,diameter, unit : mm

G86 description :



1) general thread processing cycle for offset point on the outer diameter of thread, thread X retract amount is decided by the I value; I value positive means external thread cutting, I value is negative means internal thread cutting, I value size determines the thread X to return distance.
2) when the need for thread X to gradually cut , using the parameter D to set the distance of the cutter tooth top X direct, take attention when compiling D: screw feed point position must be located on the outside surface distance D value position, if the distance is less than the D value, it will bump or thread first cut out size; D also means the retract tail distanc. X to gradually cut way for thread Z to feed in central bar or inadequate for thread cutting.

3) system based on P172 parameter to decide whether to increase the last cut finish maching thread surface. P172 = 0, will not do ; P075 is not zero, will do, cutting amount is set by P075 ,unit: um.
4) (4) when need threaded ends in advance, can use retract tail function, format is the increase in a for G86 instruction value of J, J value represents the tail length of Z direct, namely when the tip from the threaded end distance behind for J (Z direction), X start to retract tail. J values are not made up, general X can not retract tail until Z close to the end.
5) P170 definites X retract rail speed, generally set the range of 3000 mm/min - 3000 mm/min.
6) G86 thread cutting cycle equidistant feed, the feed quantity can be set to the first cut feed depth can also be specified, surplus of equidistance feed. When parameters P172 = 0: if you don't specify the first cut feed quantity, the amount of each feed to R/L; If specified the first cut feed quantity, the first cut feed quantity for the Q, residual circulation feed quantity of (R, Q)/(L - 1); When parameter P172 indicates 0: if not specified the first cut feed quantity, every time the amount of feed for (R - P075)/L, finally add a cut laser scalpel; If specified the first cut feed quantity, the first cut feed quantity for the Q, residual circulation of feed quantity of (R - Q - P075)/(L - 1);
7) thread machining process, the start and end at a speed in the process, thread lead is not accurate, so the actual processing must avoid these two areas. Parameter P166 defines accelerate time constant of Z screw thread processing.
8) when the need for thread X to gradually cut, using parameter D set the crest of the relative distance of a tool, should be paid attention to when compiling D: the tip of the screw feed must be in the outside surface distance D value, if the distance is less than the D value, will bump or thread first cut too deep; D also said the distance X to the back end. X to gradually cut way for thread Z to feed on rods or inadequate central thread cutting.
9) when screw thread processing step/Z axis servo motor speed should not exceed a certain value, such as 8000 mm/min, the speed and motor power, and the size of machine tool is determined by the parameters P168.
10) thread before cutting, CNC system testing encoder signal of zero, while detecting the spindle rotational speed fluctuation degree, to meet the conditions of cutting tool to exercise, so in the beginning of the thread processing have a pause.

2.14 G87—English system processing cycle

G87 command is the same with G86, only the K is teeth/Inch（1 Inch＝25.4mm）. Other parameters are the same.

2.15 G40,G41,G42—Cutting tool radius compensation

Descriptions about tool nose radius offset are shown in Chapter 4.

2.16 G50—Setting work piece coordinate system

Format : G50 X* Z *****

This command set coordinate system to make the point's which on tool like tool nose coordinate is (***,***). This coordinate is called work piece coordinate system. Once coordinate system is set, the locations of absolute and relative are all depend on it.

Note : x means diameter in mode diameter and means radius in moder radius.

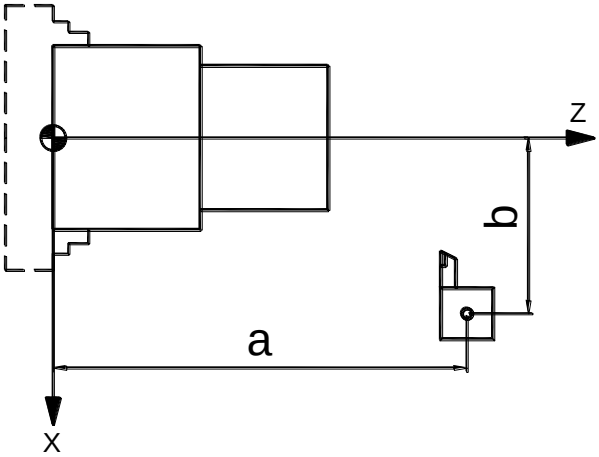


Fig. 2-7

Fig. 2-7, diameter mode program , set work piece coordinate system using G50 X2b Za

raidus mode program, set work piece coordinate system using G50 Xb Za

2.17 G98—Feed per minite

Format : G98

G98 is feed per minite mode, in mode G98, tool feed speed is decided by F.

G98 is modify, if specified G98, before G99 appeared, it is effective.

Default G98 mode is available when system power on.

2.18 G99—Feed per rotation

G99 is feed per rotation mode, in mode G99, spindle tool feed amount is decided by F.

G99 is modify, if specified G99, before G98 appeared , it is effective.

Form 2-2 feed per minite and feed per rotation

	Feed per minite	Feed per rotation
Specify address	F	F
Specify code	G98	G99
Specify range	1～60000mm/min (F1～F60000)	0.01～500.00mm/re (F0.01～F500)

Note 1 : if encode speed below 1 r/min, speed is not uniform. The slower speed is ,the more serious will be.

Note 2 : G98, G99 is modify, if specified , before the other one appeared, it is effective.

Note 3 : encode must be used in speed per rotation mode in spindle.

2.19 G96,G97—Constant linear speed cutting

So called the constant linear speed cutting is the speed after S is no longer changes. As the tool's position changes, to make cutter instantaneous position keeping on the relative constant linear velocity opposed to workpiece surface by calculating spindle speed according to the linear velocity, and outputting corresponding voltage to spindle controller.

Linear speed unit : m/min

Format :

G96 S__ ;

S is specified linear speed value.

Cancel constant linear speed cutting

Format :

G97 S__ ;

S is specified spindle rotation speed value

The constant linear speed cutting , the axis of rotation must be specified Z axis.

（1）Max spindle rotating speed

The instruction after G50 can decide the max spindle rotating speed (r/min).

G50 S__ ;

When spindle rotating speed reaches rated maximal rotating speed of system, the spindle rotating speed no longer changes.

（2）Constant linear speed cutting （G00）

As for G00 function, tool's positon is not change, so constant linear speed cutting only calculate linear speed of program ending point.

Note 1 : When power on, no limit state is the state has not specify spindle maximal rotation speed.

Note 2 : limiting is only suitable for G96 not for G97.

Note 3 : G50 S0 means be limited to 0 m/min.

Note 4 : S set in G96 state is hold on in G97, when return to G96 again , its value recovered.

G96 S50 ; （50m/min）

G97 S1000 ; （1000r/min）

G96 X3000 ; （50 m/min）

Note 5 : when machine tool locked, machine tool don't move . constant linear speed control is based on x coordinate changes in program.

Note 6 : if S is not specified in G97, S in G96 can also replace S in G97.

N100 G97 S800 ; （800 r/min）

…

N200 G96 S100 ; （100 m/min）

…

…

N300 G97 ; （XXX r/min）

XXX r/min is rotating speed before N300, from G96 to G97, speed will not change.

2.20 G94—Surface cutting cycle

Format: G94 X（U）__Z(W)__R__F__ ;

Description: G94 command can realise surface and taper surface single cycle processing, after processing, tool return to starting point, as shown in picture 2-10 and 2-11. (F) line means feed cutting, (R) line means fast moving.

1. use following command can achieve surface cutting cycle:

G94 X（U）__Z(W)__F__ ;

Among them: X\Z : surface cutting end point coordinate, unit :mm, X is modal.

U\W: surface cutting end point coordinate difference related to starting point, unit: mm, U is modal.

F : surface cutting feed speed, modal.

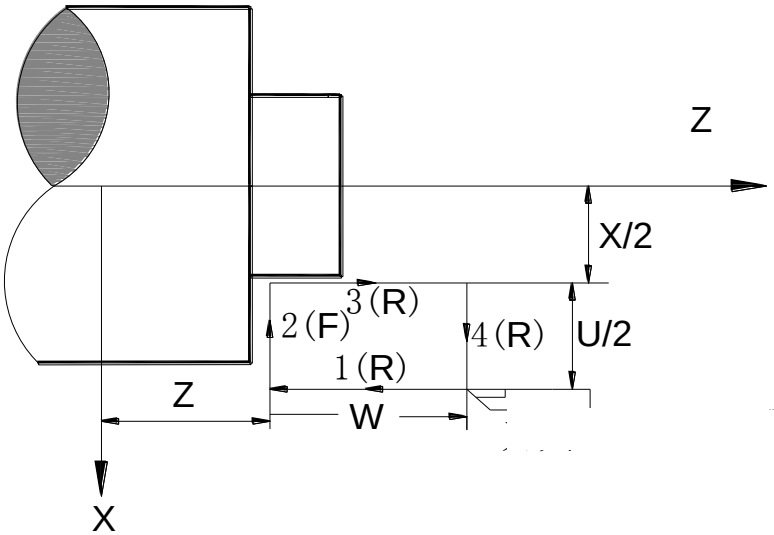


Fig. 2-10

X\Z value is end point position coordiante of track two. The Z axis plus-minus of this position related to starting point will decide moving direct of track one; the X axis plus-minus of this position related to starting point will decide moving direct of track two.

U \W value is difference of end point position coordiante related to starting point of track two. The plus-minus of U value will decide moving direct of track two; the plus-minus of W value will decide moving direct of track one. Above cycle ,U is negative, so is W.

If single function is open, pressing cycle start key, then program will carry on in the order 1→2→3→4→1 …….

G94 is modal, it can be leaved out when manifold cycles, so do X ,U ,F.

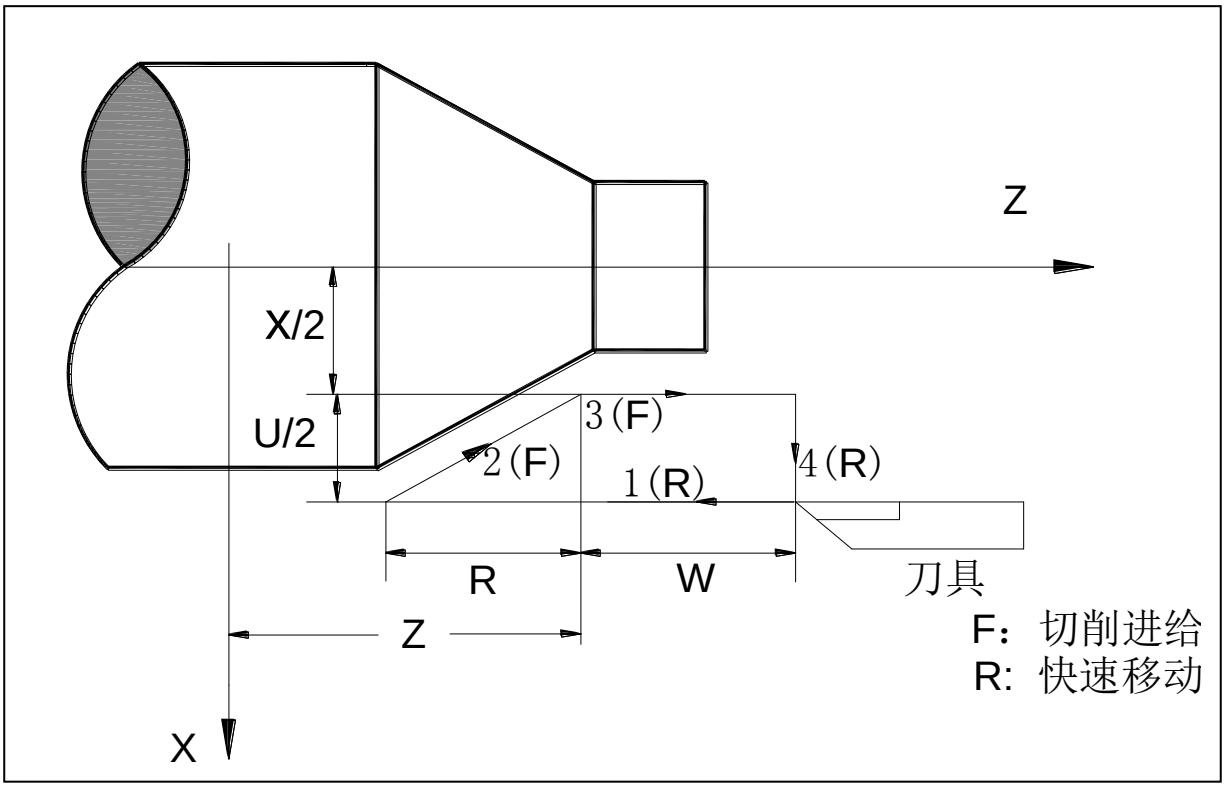
2. use following command can achieve taper surface cutting cycle:

G94 X（U）__Z(W)__R__F__ ;

Among them: X\Z : taper surface cutting end point coordinate, unit :mm, X is modal.

U\W: taper surface cutting end point coordinate difference related to starting point, unit: mm, U is modal.

R: difference of taper surface cutting starting related to starting point in Z direction.unit:mm, modal



F : taper surface cutting feed speed, modal.

Fig.2-11

G94 circular cone's cutting command is the same with cylndrical's, but the starting point Z axis position is decided by Z value and R value.

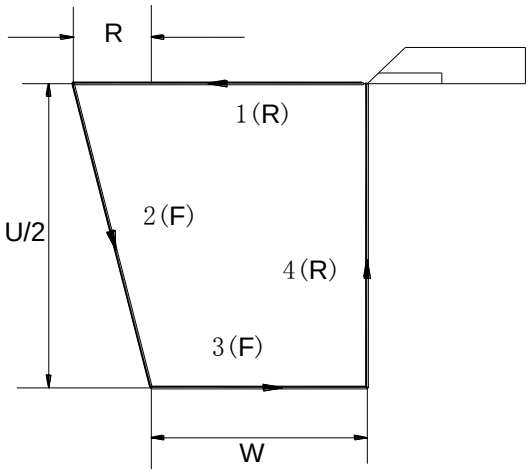
If single function is open, pressing cycle start key, then program will carry on in the order 1→2→3→4→1 …….

G94 is modal, it can be leaved out when manifold cycles, so do Z ,W ,R ,F.

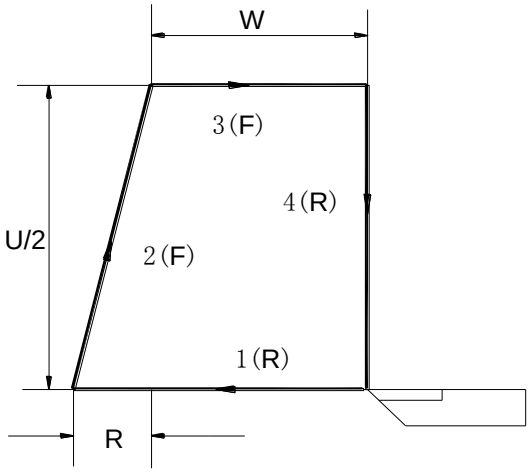
tool

As follows, depend on starting point ,G94 code has four tracks :

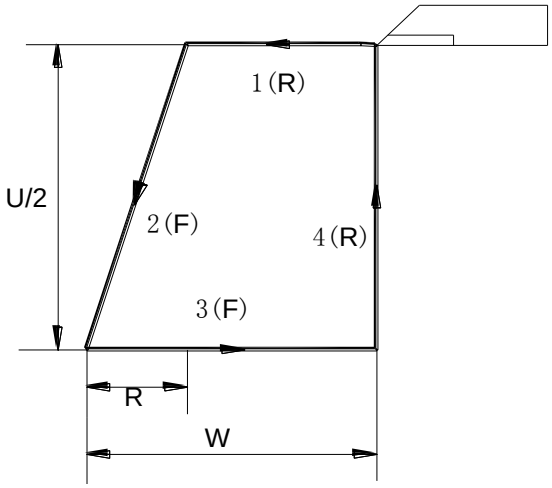
1) U>0, W<0, R<0



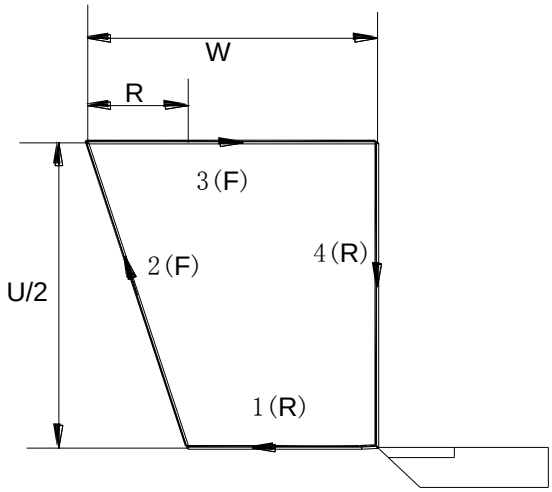
2) U<0, W<0, R<0



3) U>0, W<0, R>0 ($|R| \le |W|$)



4) U<0, W<0, R>0 ($|R| \le |W|$)



2.21 G70—Finish maching cycle

Format : G70　P（ns）　Q(nf)

Among them:

ns : the first segment of finish maching track;

nf : the last segment of finish maching track

G70 code track is decided by program trak from segment ns to nf. Relationship is as follows:

Code function: process finish maching from starting point along program segment from ns to nf. After rough machining of G71 G72 or G73, process finish maching of G70, process single complete allowance for finish of finish maching. After G70 cycle, tool return to starting point and carry on the next segment.

Code description:

- G70 must programmed after ns to nf.
- F\S\T code is valid in G70 finish maching segment of ns to nf.
- Program paused when current track is completed in single mode.
- G70 cannot be used in MDI mode.
- The same segment can not be existed in same compound cycle program of ns to nf.

2.22 G71—Excircle/inner circle rough turning cycle

Format :

```
G71 U(Δd) R(e)   ;
G71 P(ns) Q(nf) U(Δu) W(Δw) F    ;
N(ns) G0/G1 X(U)..   ;
........   ;
....  F   ;
....   ;
....   ;
N(nf) .......    ;
```

Among them:

Δd: rough turning feed amount of X axis, unsigned, radius, feed direct is decided by movement of ns program segment. If the value is not specified, system will regard parameter 230's value as cutting amount.

e: X ,Z retract amount, radius , unit mm, retract direct is opposite to feed direct. If the value is not specified, system will regard parameter 231's value as X ,Z retract amount.

ns: the first program segment num of finish maching track.

nf: the last program segment num of finish maching track.

Δu: allowance and direct for finish maching of X axis, signed, coordinate offset of rough turning outline related to finish maching track of X axis. If U(Δu) is not specified, Δu equals 0. It means that rough turning cycle X axis has not finish maching allowance amount.

Δw: allowance and direct for finish maching of Z axis, signed, coordinate offset of rough turning outline related to finish maching track of Z axis. If W(Δw) is not specified, Δw equals 0. It means that rough turning cycle Z axis has not finish maching allowance amount.

F: cutting feed speed.

M,S,F,T function is invalid in program segment of ns to nf, only valid in G70 finish maching cycle.

Code description:

G71 code can be devided to three parts:

(1): cutting and retract amount of rough turning

(2): program section and allowance program of defined finish maching track

(3): define finish maching track continues program segment, when carrying on G71, these segment only used for rough turning track but not be carried on.

Rough turning route automatic is depend on finish machine track, allowance,feed amount, retract amount and so on. Feed direct is along Z axis. Rough turning is in order of feed -> cutting ->feed ->retract. G71 tract's starting point and end point is the same. This code is fit for bar stock molding rough turning.

Track description:

Finish maching track: workpiece finish maching track decided by code part three. The starting point is the same with starting and end point of G71 code. It is called A point for short. The first segment should be fast moving or cutting feed of X axis, the end point of ns segment is called B point for short. The end point of finish maching track is called C point for short. Finish maching track is A point -> B point -> C point.

Rough turning outline: finish maching track is depend on offset of finish maching allowance, and it is track outline carried on G71. A,B,C of finish maching track correspond to A',B',C' of rough turning by excursion. The last continus cutting track is B' -> C'.

Code carried on process:

1) fast moving from A point to A' point, X axis moves length of Δu, Z axis moves length of Δw.
2) Moves length of Δd from A' point, segment of ns is tool fast moving speed in G0 mode, in mode G0 as cutting feed speed F of G71, feed direction is the same with A point to B point.
3) Coordinate range of Z axis B point to C point is the same with Z axis cutting feed to rough turning outline.
4) X axis and Z axis retract as cutting feed speed,retract direct is opposite to feed direct.
5) Z axis retract to the same position with absolute coordinate as the fast moving speed.
6) If after X axis feeding (Δd+e) once more, end point still in the middle of A' point and B' point, X axis will feed again, and carry on (3); if after X axis feeding (Δd+e) once more, end point reaches B' point or out of it, X axis cutting feed to B' point, and carry on (7).
7) Cutting feed from B' point to C' point along rough turning outline.
8) Fast moving from C' point to A point, G71 cycle is over, program will jump to the next segment of nf.

Finish maching allowance coordinate offset direct:

Δu , Δw means finish maching coordinate offset value and direct, there have four kinds depend on Δu and Δw's symbol, as shown in picture 2.12-2, B to C is finish maching track, B' to C' is rough turning outline, A is starting point.

Attention:

- ns segment can only have G00\G01 code.
- X\Z axis size must be change monotonous in finish maching track.
- Segment of ns to nf must be programmed following G71.
- Segment of ns to nf only can be used for calculating rough outline, not for program segment when G71 is carried on. F,S,T is invalid while is valid in G70 finish maching cycle.
- Segment of ns to nf can only can have G function: G00\G01\G02\G03.
- Feed hold and single function will be paused when current track is complete.
- △d, △u should be addicted by the same address.the only difference is if it has P or Q code.
- G71 can not be carried on in MDI mode.
- Segment of ns to nf can not have the same segment when compound cycle code is need in the same program.

program :

```
O0001 ;
M03 S800 ;
G00 X200 Z10 ;
G71 U2 R1 ;
G71 P80 Q120 U1 W2 F100 ;
N80 G00 X40 ;
G01 Z-30 F100   ;
X60 W-30 ;
W-20 ;
N120 X100 W-10 ;
T0202;
G70 P80 Q120 ;
M30 ; %
```

2.23 G72—End face rough turning cycle

Format: G72 W(Δd) R(e) ;

G72 P(ns) Q(nf) U(Δu) W(Δw) F ;

N (ns) . . . . . ;

. . . . . ;

. . . . F ;

. . . ;

. . . ;

N (nf). . . . . ;

finish maching

Among them:

Δd: rough turning feed amount of X axis, unsigned, radius, feed direct is decided by movement of ns program segment. If the value is not specified, system will regard parameter 230's value as cutting amount.

e: X ,Z retract amount, radius , unit mm, retract direct is opposite to feed direct. If the value is not specified, system will regard parameter 231's value as X ,Z retract amount.

ns: the first program segment num of finish maching track.

nf: the last program segment num of finish maching track.

Δu: allowance and direct for finish maching of X axis, signed, coordinate offset of rough turning outline related to finish maching track of X axis. If U(Δu) is not specified, Δu equals 0. It means that rough turning cycle X axis has not finish maching allowance amount.

Δw: allowance and direct for finish maching of Z axis, signed, coordinate offset of rough turning outline related to finish maching track of Z axis. If W(Δw) is not specified, Δw equals 0. It means that rough turning cycle Z axis has not finish maching allowance amount.

F: cutting feed speed.

M,S,F,T function is invalid in program segment of ns to nf, only valid in G70 finish maching cycle.

Code description:

G72 code can be devided to three parts:

(1): cutting and retract amount of rough turning

(2): program section and allowance program of defined finish maching track

(3): define finish maching track continues program segment, when carrying on G72, these segment only used for rough turning track but not be carried on.

Rough turning route automatic is depend on finish machine track, allowance,feed amount, retract amount and so on. Feed direct is along Z axis. Rough turning is in order of feed -> cutting ->feed ->retract. G72 tract's starting point and end point is the same. This code is fit for bar stock molding rough turning.

Track description:

Finish maching track: workpiece finish maching track decided by code part three. The starting point is the same with starting and end point of G72 code. It is called A point for short. The first segment should be fast moving or cutting feed of Z axis, the end point of ns segment is called B point for short. The end point of finish maching track is called C point for short. Finish maching track is A point -> B point -> C point.

Rough turning outline: finish maching track is depend on offset of finish maching allowance, and it is track outline carried on G72. A,B,C of finish maching track correspond to A',B',C' of rough turning by excursion. The last continus cutting track is B' -> C'.

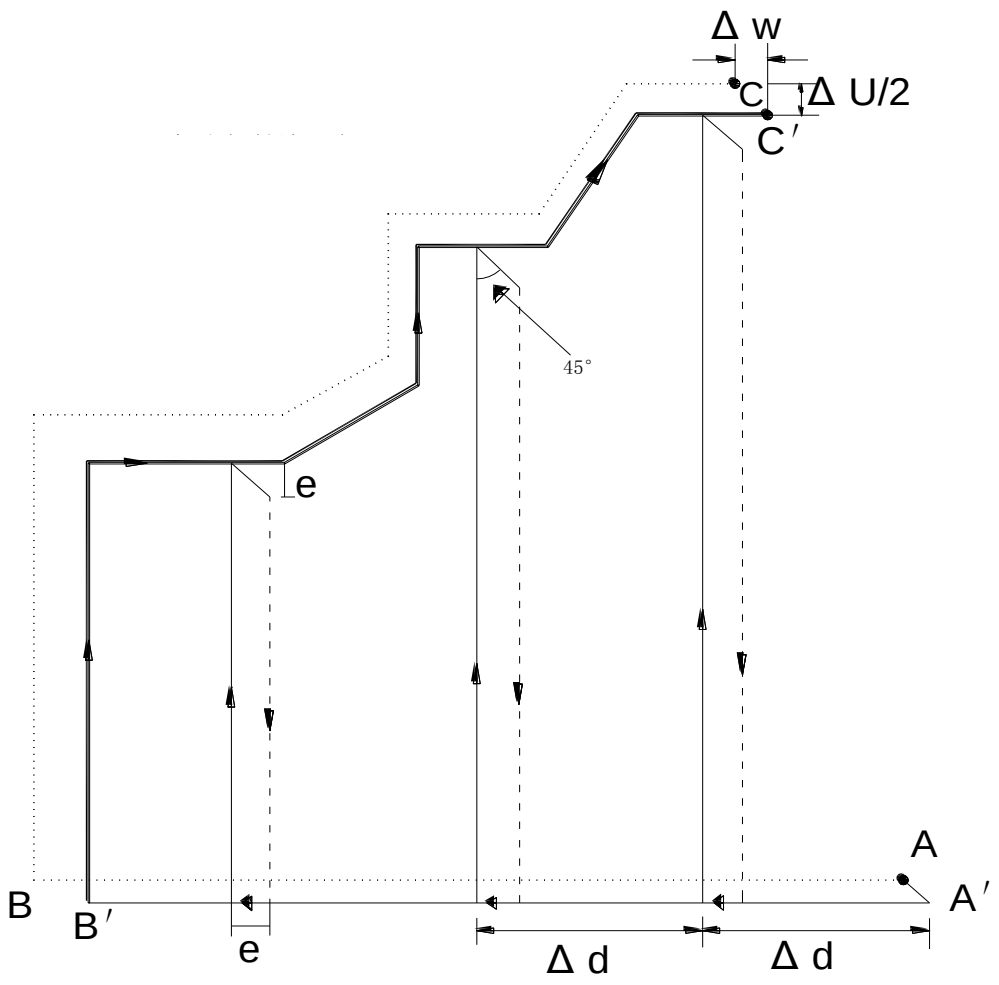


Fig. 2.13-1

Code carried on process:

1) fast moving from A point to A' point, X axis moves length of Δu, Z axis moves length of Δw.
2) Moves length of Δd from A' point, segment of ns is tool fast moving speed in G0 mode, in mode G0 as cutting feed speed F of G72, feed direction is the same with A point to B point.
3) Coordinate range of X axis B point to C point is the same with X axis cutting feed to rough turning outline.
4) X axis and Z axis retract as cutting feed speed,retract direct is opposite to feed direct.
5) X axis retract to the same position with absolute coordinate as the fast moving speed.
6) If after Z axis feeding (Δd+e) once more, end point still in the middle of A' point and B' point, Z axis will feed again, and carry on (3); if after Z axis feeding (Δd+e) once more, end point reaches B' point or out of it, Z axis cutting feed to B' point, and carry on (7).
7) Cutting feed from B' point to C' point along rough turning outline.
8) Fast moving from C' point to A point, G72 cycle is over, program will jump to the next segment of nf.

Finish maching allowance coordinate offset direct:

Δu , Δw means finish maching coordinate offset value and direct, there have four kinds depend on Δu and Δw's symbol, as shown in picture 2.12-2, B to C is finish maching track, B' to C' is rough turning outline, A is starting point.

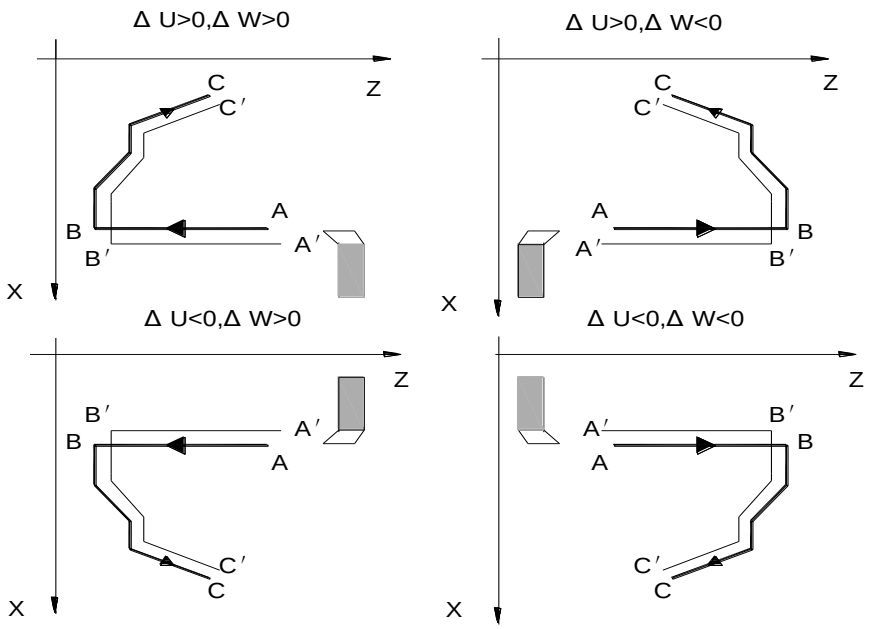


Fig. 2.13-2

Attention:

- ns segment can only have G00\G01 code.
- X\Z axis size must be change monotonous in finish maching track.
- Segment of ns to nf must be programmed following G72.
- Segment of ns to nf only can be used for calculating rough outline, not for program segment when G72 is carried on. F,S,T is invalid while is valid in G70 finish maching cycle.
- Segment of ns to nf can only can have G function: G00\G01\G02\G03.
- Feed hold and single function will be paused when current track is complete.
- △d, △w should be addicted by the same address.the only difference is if it has P or Q code.
- G72 can not be carried on in MDI mode.
- Segment of ns to nf can not have the same segment when compound cycle code is need in the same program.

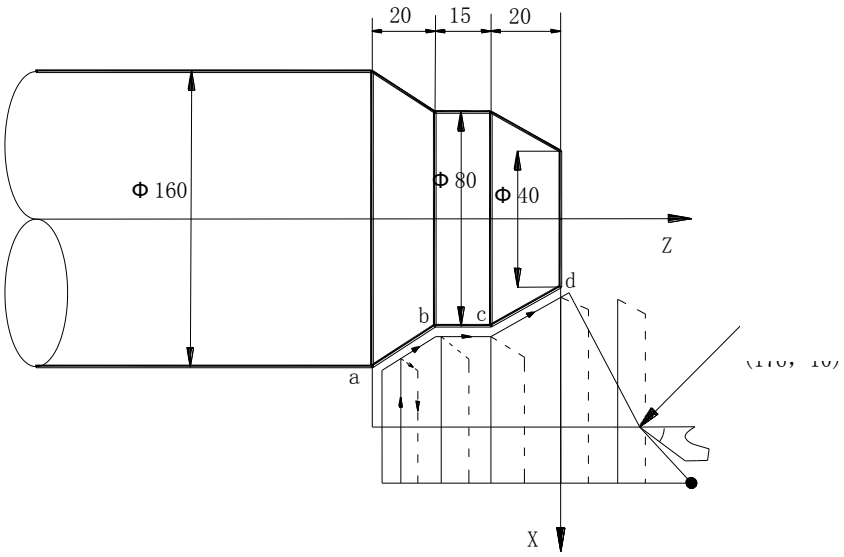


Fig. 2.13-3

Program : O0002 ;

```
G00 X176 Z10
M03 S500
G72 W2.0 R0.5 ;
G72 P10 Q20 U0.2 W0.1 F300 ;
N10 G00 Z-55 S800   ;
G01 X160 F120 ;
X80 W20 ;
W15 ;
N20 X40 W20   ;
G70 P010 Q020 ;
M30; %
```

2.24 G73—Close cutting cycle

Format: G73 U(Δi) W (Δk) R (d) F S T ;

G73 P(ns) Q(nf) U(Δu) W(Δw) ;

N (ns) . . . . . ;

. . . . . ;

. . . . F ;

. . . . ;

. . . . ;

N (nf). . . . . ;

Among them:

Δi: rough turning retract amount of X axis, signed, radius. Δi equals X axis coordinate offset amoount of A1 point related to Ad point, X axis total cutting amount of rough turning is |Δi|. The cutting direct is opposite to Δi.if Δi isgreater than 0, cutting along X axis negative direct. If U(Δi) is not specified, system will regard parameter 232's value as X axis rough turning cutting amount.

Δk: rough turning retract amount of Z axis, signed. Δk equals Z axis coordinate offset amoount of A1 point related to Ad point, Z axis total cutting amount of rough turning is |Δk|.The cutting direct is opposite to Δk.If Δk isgreater than 0, cutting along Z axis negative direct. If W(Δk) is not specified, system will regard parameter 233's value as Z axis rough turning cutting amount.

d: cutting times, range:1-9999(unit:times), R5 means completing close cutting cycle needs five times. If R (d) is not specified, system will regard parameter 234's value as cutting times. If it equals one, system will regard as two.

ns: the first program segment num of finish maching track.

nf: the last program segment num of finish maching track.

Δu: allowance and direct for finish maching of X axis, signed, the last coordinate offset of rough turning outline related to finish maching track of X axis. If U(Δu) is not specified, Δu equals 0. It means that rough turning cycle X axis has not finish maching allowance amount.

Δw: allowance and direct for finish maching of Z axis, signed, the last coordinate offset of rough turning outline related to finish maching track of Z axis. If W(Δw) is not specified, Δw equals 0. It means that rough turning cycle Z axis has not finish maching allowance amount.

F: cutting feed speed.

M,S,F,T function is invalid in program segment of ns to nf, only valid in G70 finish maching cycle.

Code description:

G73 code can be devided to three parts:

(1): program specified retract amount , cutting times, cutting speed, spindle rotation and tool function.

(2): program section and allowance program of defined finish maching track

(3): define finish maching track continues program segment, when carrying on G73, these segment only used for rough turning track but not be carried on.

Automatic calculating rough turning offset amount, single feed amount and track is depend on finish machine allowance, and each time cutting track is offset of finish maching reack.tthe last time cutting track is finish maching track according to allowance offset. G73 tract's starting point and end point is the same. This code is fit for molding rough turning.

Track description:

Finish maching track: workpiece finish maching track decided by code part three. The starting point is the same with starting and end point of G73 code. It is called A point for short. The first segment end point is called B point; The end point of finish maching track is called C point for short. Finish maching track is A point -> B point -> C point.

Rough turning outline: track amount is the same with cutting times. A,B,C of finish maching track correspond to An, Bn, Cn of rough turning by excursion. The first cutting coordinate offset amount is (Δi×2+Δu, Δw+Δk)(diameter) related to finish maching track. The last cutting coordinate offset amount is(Δu, Δw) related to finish maching track. Each time cutting offset amount related to the last cycle is

$$(-\frac{\Delta i \times 2}{1000 \times d - 1}, -\frac{\Delta k}{1000 \times d - 1})$$

(1) A→A1 : fast moving

(2) the first rough turning, A1→B1→C1

A1→B1 : ns segment is as fast moving speed in G0 mode and as G73 cutting feed speed in G1 mode.

B1→C1 : cutting feed.

(3) C1→A2 : fast moving.

(4)the second rough turning, A2→B2→C2

A2→B2 : ns segment is as fast moving speed in G0 mode and as G73 cutting feed speed in G1 mode.

B2→C2 : cutting feed.

(5) C2→A3 : fast moving.

……

The Nth rough turning, An→Bn→Cn :

An→Bn : ns segment is as fast moving speed in G0 mode and as G73 cutting feed speed in G1 mode.

Bn→Cn : cutting feed.

Cn→Ad : fast moving.

……

The last rough turning, Ad→Bd→Cd :

Ad→Bd : ns segment is as fast moving speed in G0 mode and as G73 cutting feed speed in G1 mode.

Bd→Cd : cutting feed.

Cd→A : return to starting point.

Attention:

Segment of ns to nf must be programmed following G73.

Segment of ns to nf only can be used for calculating rough outline, not for program segment when G73 is carried on. F,S,T is invalid while is valid in G70 finish maching cycle.

ns segment can only have G00\G01 code.

Segment of ns to nf can only can have G function: G00\G01\G02\G03.

Feed hold and single function will be paused when current track is complete.

Δi, △u should be addicted by the same address U, and Δk, Δw should be addicted by the same address W . The only difference is if it has P or Q code.

G73 can not be carried on in MDI mode.

Segment of ns to nf can not have the same segment when compound cycle code is need in the same program.

Finish maching allowance coordinate offset direct:

Δi , Δk means rough turning coordinate offset value and direct, while Δu , Δw mean the finish maching. There have four kinds depend on Δi , Δk , Δu , Δw, Δi and Δu have the same symbol, so do Δk and Δwas. As shown in picture 2.14-2.

B to C is workpiece outline, B' to C'is rough turning outline , B'' to C'' is finish maching track, A is starting point.

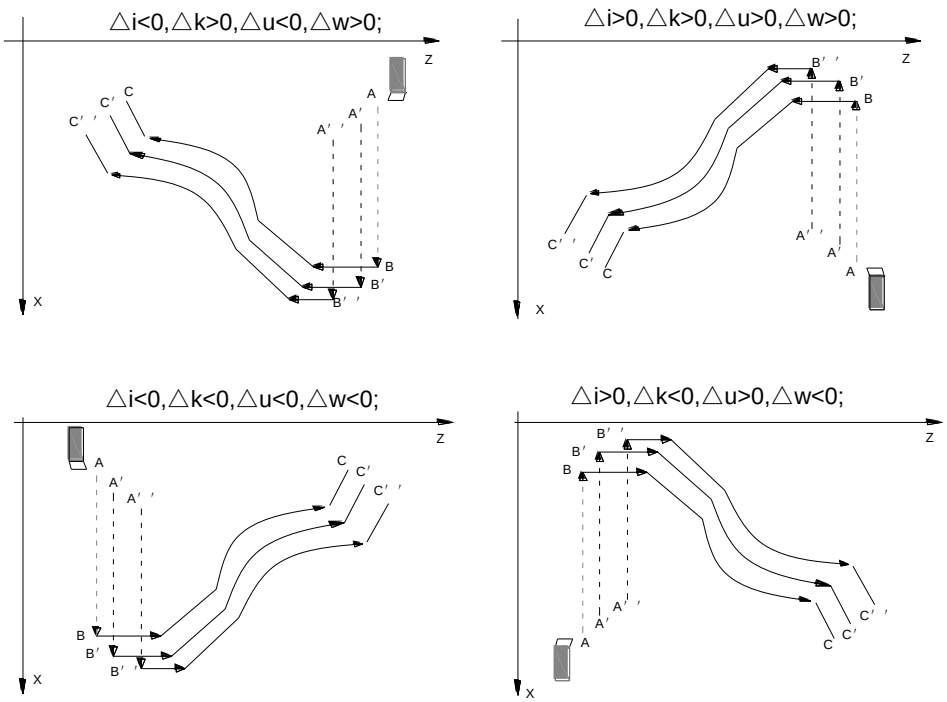


Fig. 2.14-2

Program :

```
O0073 ;
G00 X200 Z10
M03 S500;
G73 U1.0 W1.0 R3 ;
G73 P14 Q19 U0.5 W0.3 F200 ;
N14 G00 X80 Z0 ;
G01 W-20 ;
X120 W-10 ;
W-20 ;
G02 X160 W-20 R20 ;
N19 G01 X180 W-10 ;
G70 P14 Q19;
M30 ;
%
```

2.25 G75—End surface deep hole processing cycle

Format : G74 Z（W） I J/K R F ;

Among them :

Z（W） : coordinate of hole bottom

I : the movement (unsigned number) of single cycle in the Z direction.

J : The distance from fast positioning point to last hole bottom(unsigned number). K is available in mode A.

K : The distance from previous hole bottom (unsigned number) from fast feed to work feed when performing second feeding after retracting every time. K is available in mode B.

F : Feed speed

R : Hole bottom delay time

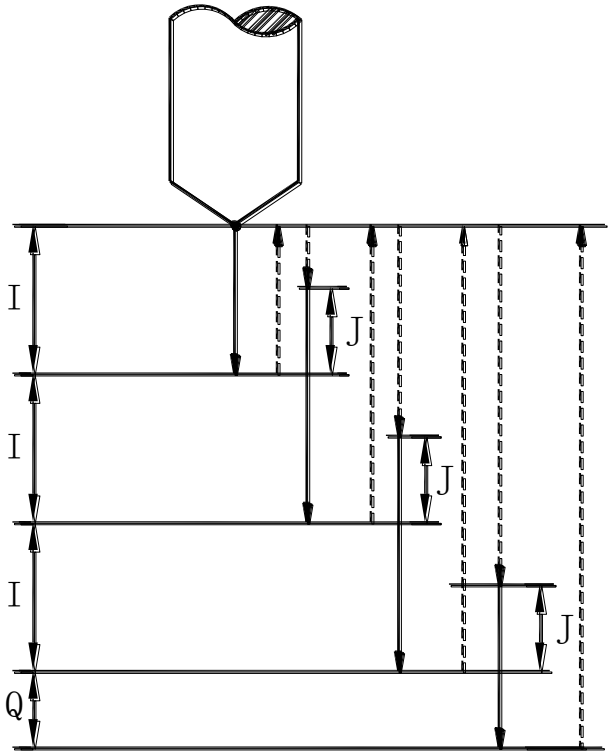
Note 1 : mode is decided by para J and para K, J means mode A ,while K means mode B. Attention J and K can not exist at the same tiome.

Note 2 : mode A characteristic : each cycle return processing start point

mode B characteristic : each cycle return to the point decided by para K .

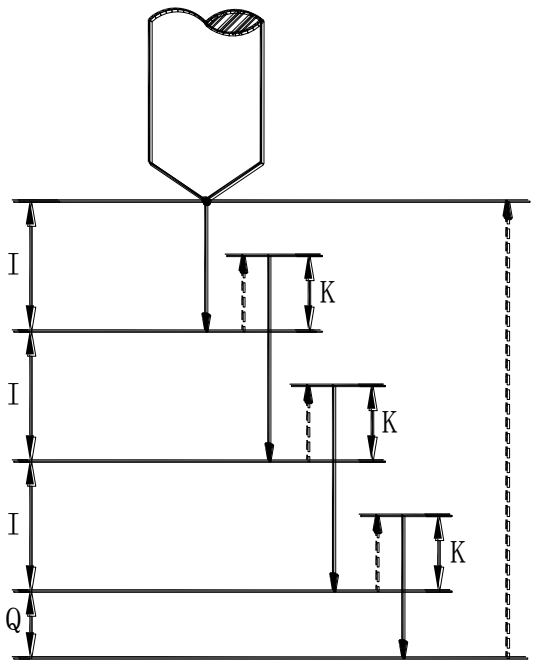
Note 3 : if adopt mode A., J cannot greater than I, otherwise system alert 121.

1. Mode A deep hole processing cycle order :



Process : (1) Z axis current position >>>(2) cut feed depth I as feed speed F(feed direction is decided by Z coordinate) >>>(3)rapid returning Z processing start point >>>(4) rapid positioning to the position the distance from which to hole bottom is J. >>>(5) cut feed depth I plus J as feed speed F >>>(6) rapid returning Z processing start point >>> …(repeat (4)～(6)cycle) >>>(7)cut feed to the hole bottom, if R is available, delay time R. >>>(8) rapid returning to hole top, G74 cutting completed.

2. Mode B deep hole processing cycle order :



Process : (1) Z axis current position >>>(2) cut feed depth I as feed speed F(feed direction is decided by Z coordinate) >>>(3)rapid returning distance K >>>(4) cut feed depth I plus J as feed speed F >>>(5) rapid returning distance K >>> …(repeat (4)～(5)cycle) >>>(6)cut feed to the hole bottom, if R is available, delay time R. >>>(7) rapid returning to hole top, G74 cutting completed.

2.26 G76—Excircle/inner circle grooving/cut off cycle

Format : G75 X（U） I K R F

Among them:

X（U）:X axis absolute (relative)coordinate of bottom of slot.

I :single feed cutting amount(unsigned) of X direct.

K: backspace distance(unsigned) after single feed cutting completed.

R: standing time of cutting to the bottom of slot.

F: cutting feed speed.

G75 cycle is make up of single feed and single retract. This make chip breaking and chip removal function more easier. And it is fit for deep groove and cutting off process.

G75 processing order:

(1) X axis current starting point >>>(2)cutting feed the length of I as the speed of F(feed direct is decided by X or U coordinate) >>> (3) fast retract the length of K >>>(4) cutting feed the length of I+K as the speed of F >>> (5) fast retract the length of J >>> …(doing (4)～(5) cycle over and over again) >>>(6) cutting feed to the bottom of slot, if R is specified, delay time of R. >>> (7) fast return to starting point. G75 cutting is completed.

2.27 G77—Multiple thread cutting cycle

Format: G76 P(m)(r)(a)Q(Δdmin)R(d);

G76 X(U)__Z(W)__R(i)P(k)Q(Δd)F(l)__ J__;

Function: complete tooth height processing thread by thread rough turning repeatedly and thread finish maching. If thread angle is not 0°, rough turning thread cutting point moves from top of thread to root of thread. It make the adjacent teeth thread angle value is the angle set. G76 code can process straight thread and taper thread which have thread retreat tail function. Achievable one-sided blades thread cutting, milling depth drop off, and it is make for protecting tool and increasing thread accurancy. G76 can not process end face thread. Track is as shown in picture 2-15(a).

Definition:

Starting point(end point): before and after operation position, called as A point.

Thread end point: defined by X(U) Z(W), called as D point. If it has retract tail, cutting end point of long axis direct is thread end point, cutting end point of short axis direct is retract tail position.

Thread starting point: Z axis absolute coordinate is the same with A point. The difference between X axis coordinate and D point is i(radius, thread taper) , called as C point. If thread angle is not 0°, C point can not be reached.

Thread cutting depth reference point, Z axis absolute coordinate is the same with A point. The difference between X axis coordinate and C point is k(radius, total thread cutting depth) , called as B point. Is B equals 0, the point will be the reference point each time.

Thread cutting depth: cutting cycle depth each time. Intersection point of extension line and straight BBC of cutting track. Thread cutting depth is difference between the point and X absolute coordinate. J is the first rough turning thread cutting depth, the second is $\sqrt{n}$ ×△d, n is current rough turning cycle times.

Thread cutting amount: the difference between this time the cutting depth value and the last value.

Retrack end point: after each time the actual cutting point of thread rough turning cycle and finish maching cycle, is called as Bn point(n is cutting cycle time),B1 is the first thread cutting point while Bf is the last one. Be is thread finish maching cutting point.

Among them:

X: X axis absolute coordinate of thread end point,unit mm.

U: the difference between X axis thread end point and starting point. Unit mm.

Z: Z axis absolute coordinate of thread end point,unit mm.

W: the difference between Z axis thread end point and starting point. Unit mm

P(m): thread finish maching times 01~99（unit:times）, m is valid after carried on. Cutting feed amount equals thread finish maching cutting amount each time.

P(r): thread retract tail length 1-99(unit: 0.1xL, L is screw pitch), r is not specified, it is addicted by P165.

P(a): angle of adjacent teeth, range 00-90, unit(°). Thread angle is decided by tool angle, so a is the same with tool angle. Usually equals 80°, 60°, 55°, 30°, 0° .

Q(△dmin): thread rough turning min cutting amount, range 0-9999999(radius),unit mm. if (△d×$\sqrt{N}$ - △d×$\sqrt{N-1}$)<△dmin, △dmin is this time rough turning amount. △dmin is set used for avoiled cutting amount undersize or cutting times over much caused by thread rough cutting amount decrease.

R(d): thread finish maching allowance, unit mm. non-modal.

R(i): thread taper, the head radius difference related to tail radius, unit mm.

P(k): thread tooth height, (X axis radius), unit mm.

Q(Δd): after the second cutting depth, radius, unit mm.

F: lead of screw thread,unit mm.

I: thread tooth num per inch.

J: the first tool cutting depth. Unit mm. radius. If J is addicted, the first cut cutting depth is only decided by J value, after the second cut is still decided by Q(Δd).

A :starting point

B:cutting depth reference point.

C: thread starting point.

D: end point.

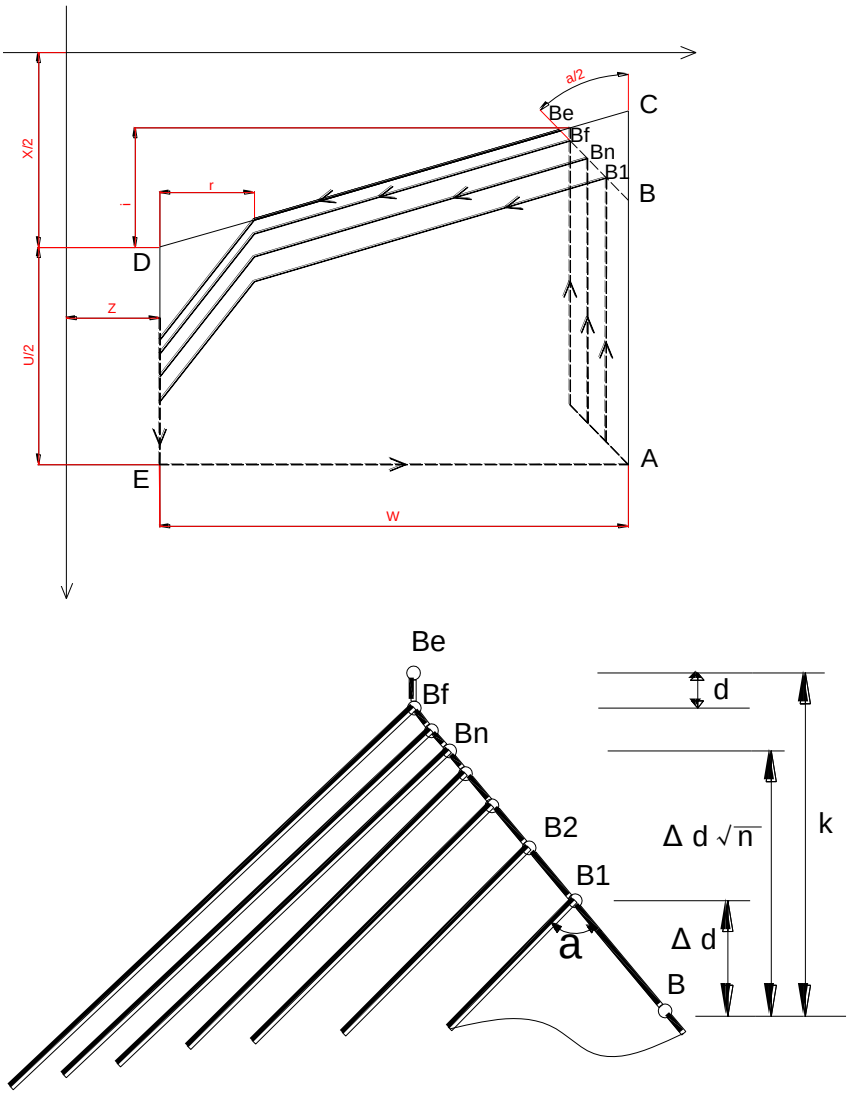


Fig. 2-15(a)

Code processing order:

1) fast moving from starting point to B1, thread depth is △d. if a equals 0, only moves X axis, while moves X and Z axis, the direct is the same with A to D.
2) thread cutting along direct of C to D to the cross point of D to E.
3) X axis fast moving to E point.
4) Z axis fast moving to A point. The single rough turning cycle is completed.
5) fast moving to Bn, cutting depth is the max of ($\sqrt{n}$ ×△d) and ($\sqrt{n-1}$ ×△d+△dmin).if it less than (k-d), goto (2). While feed cut to Bf, goto (6).
6) thread cutting along direct of C to D to the cross point of D to E.
7) X axis fast moving to E point.
8) Z axis fast moving to A point. The rough turning cycle is completed.
9) Fast moving to Be.Begin to process finish maching. At last return to A point.
10) If cycle times less than m, goto (9), cutting depth is k, cutting amount is 0, while cycle times equals m, G76 complex thread cutting cycle is completed.

Note:

- If feed hold is carried on during thread cutting, when thread cutting is completed, it shows “pause”,program paused.
- If single mode is open, when thread cutting is completed, program stoped.
- Reset, emg or drive alarm can make cutting slow down to stop.
- M,r,a use the same code P inout, attention each value should be two numbers.
- Coordinate and starting point decided by XU\ZW decide the direction of A→C→D→E, it corresponds four tracks.
- The sign of R(i) deciding C→D direction.

Fig. 2-16, thread standard : M68×6.

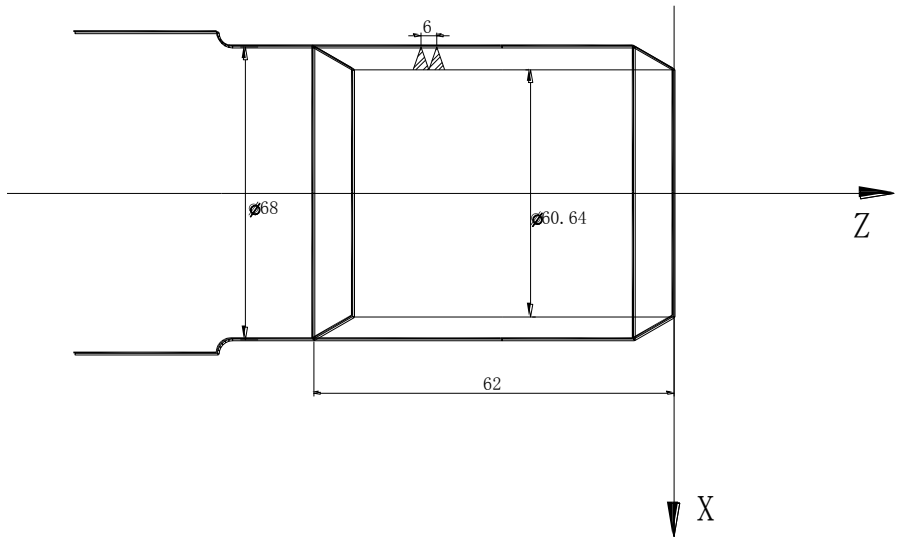


Fig. 2-16

Program: O0013 ;

G50 X100 Z50 M3 S300 ;

G00 X80 Z10 ;

G76 P020560 Q0.15 R0.1 ;

G76 X60.64 Z-62 P3.68 Q1.8 F6 ;

G00 X100 Z50 ;

M30 ;

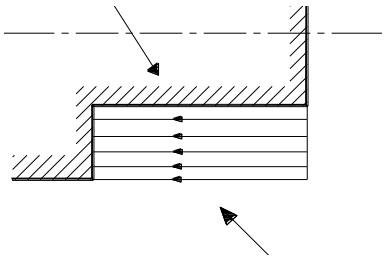
2.28 The other decription of the fixed cycle

In some special process , because of the larger amount of cutting ,you can use the fixed cycle to cut in the same path again and again , one program can finish the work whicn is finished by several programs . it is useful to reduce program , you only want to chang the value while repet the same cutting .

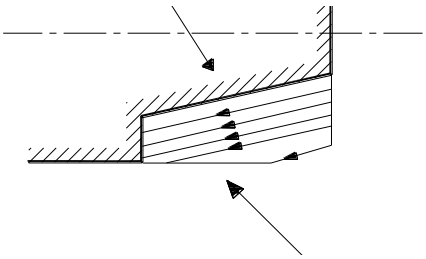
The method of use for the single fixed cycle:

You can choose the suitable fixed cycle ,base on the shape of the workblank and the shape of part.

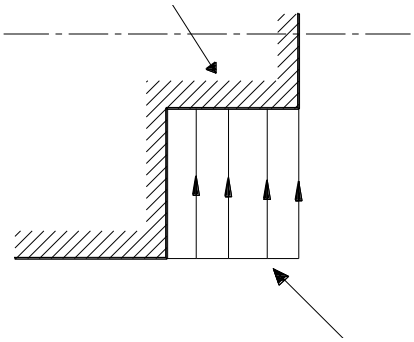
（1）cylindrical cutting cycle（G90）



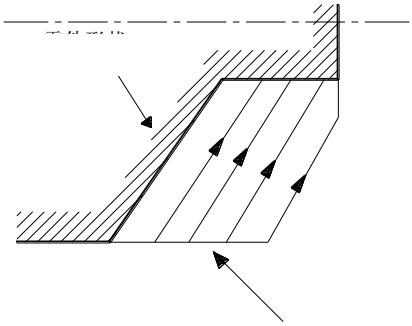
（2）cone cutting cycle（G90）



（3）end face cutting cycle（G94）



（4）end face cone cutting cycle（G94）



2.29 G54,G55——Coordinate change

Default in G54 state when system is power on, at this time, the coordinate can be restored from memory, or decided by G50. While G55 is different from the traditional coordinate change.

G54 : coordinate when power on

G55 : coordinate change command, the new programming zero is decided by instructions after G55.

Note：when program is over, remember carry on G54 to restore the initial value, incase make the coordinate cinfused.

Under the MDI mode, suppose not to use G54,G55, in order to avoid error.

G54 or G55 is displayed in G monifiy window.

G54 – Cancel zero bias, restore working coordinate system

Format : G54

Description : (1) After zero bias, G54 function will restore the processing part programming zero to the initial programming zero.

(2) G54 function will cancle all coordinate bias function before.

G55 –zero bias

Format : G55 X Z Absolute zero bias

G55 U W relative zero bias

G55 U0 W0 no change

Description : (1) G55 function can move programming zero to specified coordinates.

(2) X and Z can both move, or one of them can move. If there is no instructions after them, it means programming zero will not change.

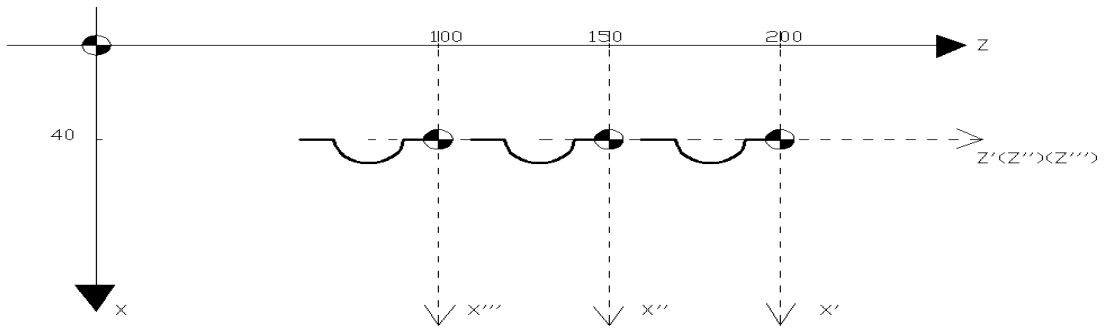
(3) G55 function cannot appeared with othe commands.

(4) The programs after G55 will not affected from the origin zero.

(5) dynamic coordinate is still the original coornate.

(6）G55 is not a move command.

Example 1 :



O3001 ;

N0010 G0 X40 Z200;

N0015 G55 W0;

N0020 M98P0013002; 01 X40 Z-10 F300;

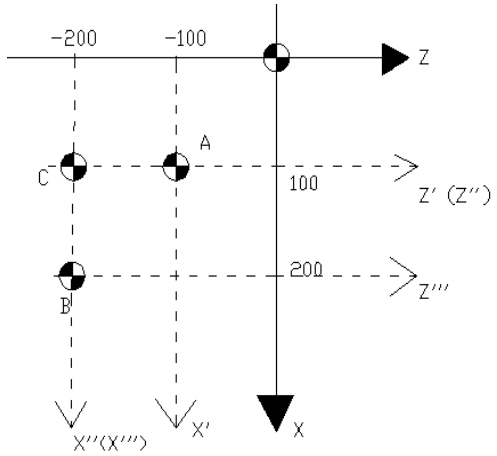
N0030 G55 Z150;

N0040 G0 X40 Z0;

N0050 M98P0013002;

N0060 G55 Z100

N0070 G0 X40 Z0;



```
N0080   M98P0013002;
N0120   G54;
N0130   M30 ;
N0140   %

O3002;
G01 X40 Z-10 F300;
G02 X40 Z-30 R10;
G01 X40 Z-40 ;
M99;
%
```

Example 2 :

```
O3003 ;
G01X100 Z-100 F500;
G55 X100 Z-100;
G01 X100 Z-100;
G55 U-100 W0;
G01 X0 Z100 ;
G55 X200 Z-200 ;
G01 X-200 Z200 ;
G54   ;
M30;
%
```

2.30 G90/G91—Programming with absolute/increment mode

Spindle is defined as C axis,if Incremental program is used for C axis ,you can use the G91 command.

Such as : G90 G01 C10 -- means the C axis move to the position of ten degree in the forward direction.

G91 G01 C10 -- means the C axis move ten degree in the forward direction

During the program is running , G90/G91 is a kind of work mode.

This command is also used for A axis while it is working in the line mode.

2.31 Auto accelerate and decelerate

At the beginning and end of the axis moving, the cnc system do the accelerate and decelerate automatically , so the axis can start an stop smoothly . at the same time ,if the move speed of the axis is changed ,the cnc system do the accelerate and decelerate automatically, so the move speed of the axis can change smoothly. In the process of program .you can ignore the accelerate and decelerate.

The style and the parameter of accelerate and decelerate:

Rapid feed: linear accelerate and linear decelerate (set the characteristic time of accelerate and decelerate of every axis by the cnc system parameter)(parameter p40,p42)

Cutting feed: linear accelerate and linear decelerate (set the general characteristic time of accelerate and decelerate of every axis by the cnc system parameter)(parameter p51)

Jog feed: linear accelerate and linear decelerate (set the general characteristic time of accelerate and decelerate of every axis by the cnc system parameter)(parameter p40,p42)

2.31.1 Speed control in the corner of the program

G61 , G64 modal code is used to control the transient mode in the program.

G61 style : while current command is finish and the feed is slow down to zer0 , the next command will perform ;

G64 syle : while current command approach to the end , the speed is under the setting smooth transition speed (set by parameter p54 or p55) , the cnc system will dispose automatically ,during the execution of command the speed is not need to slow down to zero , but it can formate circle transition area , under the G64 style you can use accurate stop (G04) command to remove the transition area.

For example : The forward command is only to move X axis , the next command is only to move Z axis , while the speed of X axis is decelerating and the speed of Z axis is accelerating , the path of tool as follow :

If you put accurate stop command in the program ,the path of the movement of the tool like the picture above . otherwise while the cutting feed speed is greater or the characteristic time of accelerate and decelerate is greater or the setted smooth transition speed is greater , the radian of the coner will comes greater .while under the arc command , the real radius of the circular arc for the tool path is smllaer than the radius which the program provide. If want to reduce the error in the coner , must to reduce the characteristic time of accelerate and decelerate under the mechanical system allowed .

Note: the cnc system will do the following command between two program segments:

Last / Next	Point position	Cutting feed	Unmoved
Point position	×	×	×
Cutting feed	×	○	×
Unmoved	×	×	×

×: will execute the next command till the feed speed of the period command slow down to zero.

○: will execute the next command till the feed speed of the period command slow down to the setting smooth transition speed.

Chapter 3 MST code

3.1 M code

Aid function (M function) is used for controlling tool machine electrical on and off action, input state inspection and control process order and so on. M function is make up of address character M and two-bit integer. Moving command and M code is at the same segment and carried on at the same time.

Such as : N1 G01 X50.0 Z-50.0 F100 M05; when carried on, G01 and M05 are carried on at the same time.

Used M function is shown in list:

Command	Function	Format
M00	Lppause, wait for “Lpstart”	M00
M01	Lppause, wait for external signal	M01 Lxx/Kxx J##
M03	Spindle clockwise rotation	M03
M04	Spindle anticlockwise rotation	M04
M05	Close spindle	M05
M08	Open coolant	M08
M09	Close coolant	M09
M10	Workpiece clamp	M10
M11	Workpiece loose	M11
M14	Change speed mode to location mode	M14
M15	Change location mode to speed mode	M15
M78	Tail enter	M78
M79	Tail retreat	M79
M20	Output low level signal to port	M20 Kxx
M21	Close port low level signal	M21 Kxx
M22	Output pulse signal to port	M22 Kxx J##
M26	Y axis rotation control(clockwise)	M26 Sxxxx
M27	Y axis rotation control (anticlockwise)	M27 Sxxxx
M28	Y axiss rotation stop	M28
M30	End of program	M30
M31	Workpiece num add one	M31
M32	Lubricate on	M32
M33	Lubricate off	M33
M35	Load material repeat automatically	M35 Lxx/Kxx Jxx Ixx Rxx Pxx
M63	Second spindle anticlockwise rotation	M63
M64	Second spindle clockwise rotation	M64

Command	Function	Format
M65	Second spindle stop	M65
M78	Tail forward	M78
M79	Tail recede	M79
M91	Program jump conditional	M91 Lxx/Kxx Nxxxx
M92	Program jump unconditional	M92 Nxxxx/M92 Nxxxx L***
M98	Subroutine call	M98 P***xxxx
M99	Subroutine return	

Note1: M and G are at the same segment, and carried on at the same time.

Note2: one segment only has one M function.

3.1.1 M00—Program stop

Format:M00

M00 can make program pause to do other work until pressing Lpstart key.

3.1.2 M01—Condition pause

Format: M01 Lxx/Kxx J##

Amont them : Lxx/Kxx is input port to be detected.

J## is waiting time(unit : second)

M01 make program to pause, waiting for external input port signal, prgoram will carring on if available signal is checked. If available signal is not checked during setted time Jxx,it will alarm.

Lxx means waiting for this port low level(connect to ground), Kxx means waiting for high level(disconnect to ground).

For example: M01 L07; waiting for No.7 input port signal.

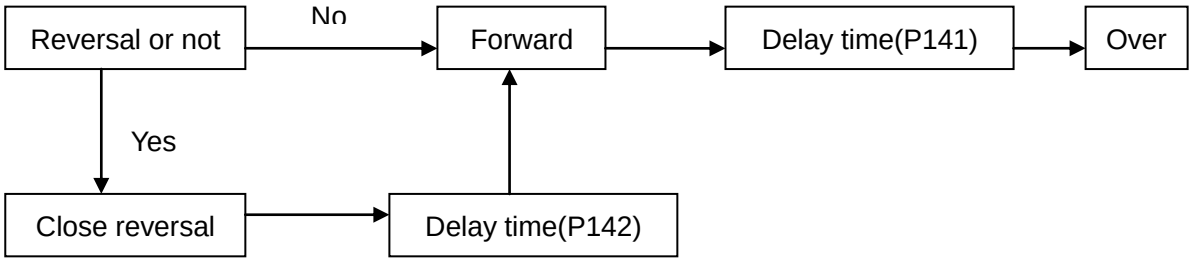
M01 K08 J5 ; waiting for No.8 input port signal, if the signal is not checked during five seconds, it will alarm.

Every input port has its own program port number, enter dignose page to check.

3.1.3 M03—Spindle positive rotation

Format: M03

Description:



(1) M03 makes spindle forward relay actuation, then output S function analogue, and control spindle forward rotation.

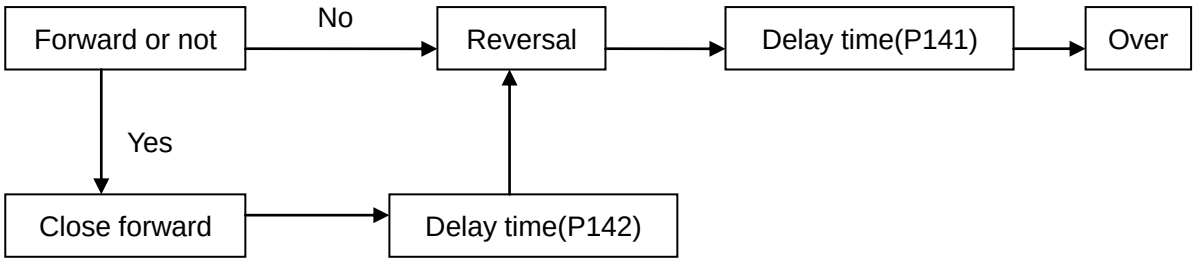
(2) if parameter 147 equals 0, M03 hold output level, or else output pulse, pulse width is decided by parameter 147.

(3) if parameter 150 is not 0, it will check rotate speed when spindle starting, when rotation reached value addicted by parameter 150, spindle forward will complete, or else it will alarm.

3.1.4 M04—Spindle reverse

Format: M04

Description:



(1) M04 makes spindle forward relay actuation, then output S function analogue, and control spindle reversal rotation.

(2) if parameter 148 equals 0, M04 hold output level, or else output pulse, pulse width is decided by parameter 148.

(3) if parameter 150 is not 0, it will check rotate speed when spindle starting, when rotation reached value addicted by parameter 150, spindle forward will complete, or else it will alarm.

3.1.5 M05—Spindle stop

Format: M05

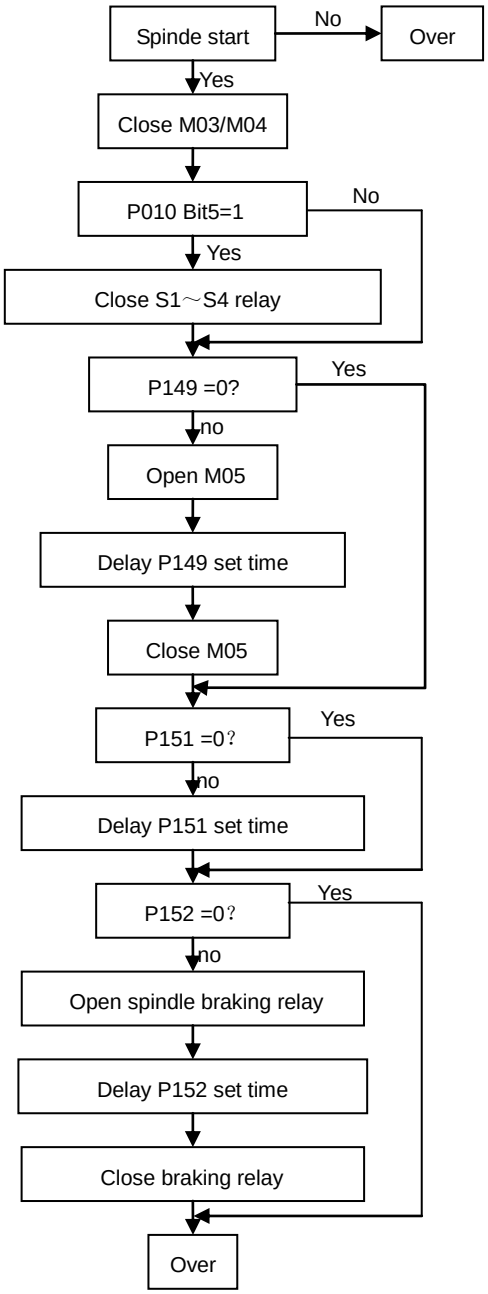
(1) M05 close spindle forward or reversal control relay, stop output analogue, spindle rotation stop.

(2) if parameter 149 equals 0, M05 is hold level output, or else is pulse output, pulse width is decided by parameter 149.

(3) if parameter 152 is not 0, system will output pulse signal to braking relay for spindle braking function.

(4) if parameter P010 Bit5 equals 1, then carry on M05 and close S1～S4 relay at the same time. If Bit5 =0, then will not.

(5) executing procedure as:



3.1.6 M08 M09—Cooling fluid

Format: M08

M09

M08 open cooling liquid.

M09 close cooling liquid.

M08 and M09 output as level mode.

3.1.7 M10 M11—Workpiece clamp, loosen control

Format : M10

M11

M10 command to clamping workpiece.

M11 command to loosen workpiece.

(1) M10 and M11 function output port are m10 and m11.

(2) M10 and M11 can be controlled by pulse and level decided by parameter P190.

(3) M10 , M11 and spindle have interlocking relationship, specific descripted in section 5.4 of charpter five.

3.1.8 M14 M15—Servo spindle speed/position mode transition

Format : M14

M15

M14 makes servo spindle speed mode switch to position mode.

M15 makes servo spindle position mode switch to speed mode.

3.1.9 M20, M21, M22—Specified relay control

Format : M20 Kxx

M21 Kxx

M22 Kxx J##

1. Kxx addicts output port, xx range from 1 to 32.
2. M20 makes xx port OC effective.
3. M21 makes xx port OC cut off.
4. M22 makes xx port OC output pulse. Pulse width is decided by J##. Don’t output pulse when J## is 0.

M20 and m21 output level signal, while m22 outputs pulse signal.

Every output port has its own program port number, enter dignose page to check.

When carry on m20 in auto mode, the port will be set, when press RESET or EMG, these ports will be reset.

3.1.10 M26, M27, M28—Rotation axis(Y axis) speed control

Format : M26 Sxxxx

M27 Sxxxx

M28

Among them:

1.M26, M27, M28 are rotation axis control special command, Sxxxx is addicted rotation, control axis is Y axis. This command is used to control movement of stepping or servo motor, and not effect following program carried on such as spindle movement.

2. M26 Sxxxx makes Y axis forward as speed of xxxx r/min.

3. M27 Sxxxx makes Y axis reversal as speed of xxxx r/min.

4. M28 makes Y axis to stop.

5. Y axis is stepping or servo drive units need to set as subdivide mode.

6.corresponding parameters:

0	1	1		ANGZ	ANGY	ANGX			SVRP	

ANGZ : =1 : Z Axis Angle Programing =0 : Length

ANGY : =1 : Y Axis Angle Programing =0 : Length

ANGX : =1 : X Axis Angle Programing =0 : Length

SVRP : =1 : open M26~M28 function =0 : no

Value : 0000 0100

89	Rotation axis screw lead（mm）	10	——
90	Rotation axis max speed（rpm）	500	——
91	Rotation axis set X : 0 Z : 1 Y:2	2	——

3.1.11 M30—End of program

Format : M30

M30 means end of program, process as follows:

(1) main program is over, pointer return to starting point and stop operation.

(2) close cooling and spindle(decided by P004 Bit6).

(3) number of cases add one, time stop.

(4) output m30 state(decided by P015 Bit5).

3.1.12 M31—Workpiece count

Format : M31

1. m31 makes current workpiece count value and accumulative total value add one at the same time.

2. if m31 is not addicted, when carried on m30 system will add workpiece count number automatically, while will not add.

3.1.13 M32 M33—Lubricate open, close

Format : M32

M33

M32 makes lubricate open.

M33 makes lubricate close.

(1) m32 or m33 function output port is m32.

(2) m32 or m33 is controlled by level or pulse which is decided by parameter P013 Bit2, P197 and P198.

(3) m32 or m33 specific application is descriptied in section 7 of charpter three.

3.1.14 M35—Automatically repeat loading funcction

Format : M35 Lxx/Kxx Jxx Ixx Rxx Pxx

Among them:

Lxx: external conditional signal input port, low level effective

Kxx: external conditional signal input port, high level effective

Jxx: max waiting time of checking input port signal,unit: second

Ixx: external output port of loading control

Rxx: delay time from open loading output port to send back. unit:second

Pxx: repeat loading times.

Description:

When carried on m35, system will wait for input port single of Lxx or Kxx, if detected effective signal, m35 is finished. If it has not detected effective signal during Jxx time, system will close output port Ixx, send back workpiece, delay time of Rxx and then open output port Ixx and loading again, after that checking input port of Lxx or Kxx, if there is no signal ,it repeat loading and sending back until repeat times reached set value of Pxx, if there is still no signal, system will produce Err 029, m35 is finished, while carried on the next segment.

M35 function is suit for automatical loading, when loading err, it can send back automatically, and loading again, this can improve loading success rate and process efficiency.

After m35 overtime alarm system enters whether stop or pause state is decided by P21 Bit1.

3.1.15 M78 M79—Tail enter, retreat control

Format : M78

M79

M78 means tail enter.

M79 means tail retreat.

(1) m78 and m79 output port are m78 and m79.

(2) m78 or m79 is controlled by pulse or lever decided by P194 and P195.

(3) m78 or m79 specific application is descriptied in section 5,6 of charpter five.

3.1.16 M91 M92—Jump function

Format : M91 Lxx Nxxxx

M91 Kxx Nxxxx

M92 Nxxxx

M92 Nxxxx L***

Among them:

1. m91 is conditional jump command.

Lxx: when input port xx is low level, program jumps to segment nxxxx, or else carry on the next segment.

Kxx: when input port xx is high level, program jumps to segment nxxxx, or else carry on the next segment.

2. m92 nxxxx is unlimited cycle jump, in case each cycle coordinate do not have offset, the command track should be close, or else it may range of worktable.

3. m92 nxxxx l*** is limited cycle jump. When carried on *** times jump, it will carry on the next segment.

3.1.17 M98 M99—Sub program call and return

Format : M98 P*＃＃＃＃**

M99

Among them:

P: character, can not be leave out.

＃＃＃＃: sub program name, must be four-figures.

***: sub program call times, call one time when leave out, max is 999.

The program which appeared repeatly and exist some fixed order is called sub program. Each need this fixed order can call sub program.

The last segment of sub program must be M99. then program return to carry on the next segment of called sub pogram in main program.

For example : main program O0001

```
N0010 M03 S1000
……
N0080 G0 X10
N0090 M98 P0005
N0100 G0X30
……
N0150 M30
```

Sub program O0005

```
N0010 G01 X10 F100
……
N0060 G0 Z30
N0070 M99        ; return
```

When carry on main program O0001, process is:

```
N0010 M03 S1000
……
N0080 G0 X10
N0010 G01 X10 F100
……
N0060 G0 Z30
N0100 G0X30
……
N0150 M30
```

3.1.18 M63/M64/M65—The second spindle anticlockwise/clockwise/stop control

Format : **M63 ;**

M64 ;

M65 ;

Among them:

M63: anticlockwise rotation

M64: clockwise rotation

M65: spindle stop

Note 1: m63\m64\m65 control time series is like m03\m04\m05.

Note 2: this function takes effect only if the second spindle function is available.

Function description:

multiple spindle control function is valid, default the first spindle control.

carry on m03,m04,m05, the first spindle signal output and set to control. And then the following S command will change rotation speed. In the similar way, carry on m63,m64,m65, the second spindle output and set to control. And then the following S command will change rotation speed.

panel keys of spindle forward, reversal, stop and rate only valid to the first spindle. If the second spindle is controlled at the current time, the panel key can still control the first spindle.

the second spindle rate fixed is 100%.

the second spindle parameter of delay time is the same with the first spindle's. Braking output port is decided by parameter.

3.1.19 Assist function code call sub program

If P004 Bit4（CM98）is set one, when outside of M,S,T code is carried on, system calles the sub program replace alarm. Combining input port and output port control code, user extend assist function code as needed.

1. M code calls sub program

M code, when outside of standard M code is carried on, the sub program is :

Mxx: O90xx

Note1: unstandard M,T code is carried on, the sub program must exist, or else system will alarm 051.

Note2: unstandard M,T code can be carried on in MDI mode.

Note3: sub program can exist movement command, and control output point.

Note4: 6.6.10 of passage two is about imply program and Mxx call sub program.

3.2 S function

3.2.1 Spindle speed command

Sent code signal to machine tool by address character S and following data which is used to control machine tool spindle rotation.

Format : S**** or S**

P001 Bit4 is used to test spindle whether analog control or other methods.

1. double speed~ four speed control mode(P001 Bit4 ＝0)

When S value less than 5, S function is electrically machine gear stage control command, S1~S4 control output port S1~S4 respectively, total four gear stages. S0 cancles all gear stages output signal.

S1~S4 code execute time are set by parameter P143～P146.

Set range: 0～1000

Set time = set value * 4 ms.

Set value equals 0 means long signal output; equals not 0 means pulse signal, width is set by parameter.

2．Spindle transducer mode（P001 Bit4 ＝1）

Sxxxx is transducer analogue control command, unit: r/min; system output 0-10V DC signal control transducer to realise spindle electrically machine continues viable.

S function controls transducer spindle, output 10V regards spindle max rotation is decided by P134, P135, P136, P137 and M41,M42,M43,M44:

If M41 is effective, 10V regards speed is decided by P134.

If M42 is effective, 10V regards speed is decided by P135.

If M43 is effective, 10V regards speed is decided by P136.

If M44 is effective, 10V regards speed is decided by P137.

System gear stage nums is default two, controlled by P133, the max value is four, system can switch gear stage by M41-M44 or S1-S4, at the time need is the gear stage of P140 multiplied by 4ms.

Data of P140 range from 0 to 10000.

3. spindle switch gear stage needs check switch in place signal

Parameter:

P021 Bit0 decides whether check in place signal

P162 sets switch gear stage in place delay time, if there has not signal during this time, system will alarm 72.

P163 set input port of M43 in place signal.

P164 set input port of M44 in place signal.

M41 and M42 input port are M41I and M42I.

3.3 T function

Choose cutting tool number and compensation number by four digits behind T. In one sentence can only use one T code.

Format:

T **xx

Among them: ** regards tool num, xx regards tool repairing num.

System controls tool num is 8, tool repairing num is 24.

3.3.1 Cutting tool changing process

When T code is carried on, the first, output tool holder forward signal(TL+), tool holder rotates, until received in place signal, close tool holder signal, delay time of T1, tool holder reversal(TL-) , and begin to check lock signal *TCP, when accived thos signal, delay time of P183 set, and then close tool holder reversal signal(TL-), tool changing over. Program turn to the next segment. If the current tool num is the same with command need, the tool changing command is over and program turn to the next segment.

After system output tool holder reversal signal, if system has not received *TCP signal during time set by P184 , system will alarm and close tool holder reversal signal.

If system has not received tool num during time set by P182, system will alarm and close tool holder forward and reversal signal.

If tool hold is SWD(P010 Bit0 = 1), there is no tool changing action, system only changes tool reparing num and adjust coordinate.

3.3.2 Cutting tool changing parameter

1. P010 Bit0

Tool holder type set:

TSS ＝0 : electric toolpost

＝1 : SWD toolpost

2. P009 Bit0, Bit1

Tool holder level set:

Tool holder in place signal (T8～T1) whether low level or high level valid is decided by P009 Bit1 TSGN.

TSGN ＝0: tool holder in place signal high leve valid.

=1: tool holder in place signal low leve valid.

Tool holder lock signal(*TCP) whether low level or high level valid is decided by P009 Bit0 TCPS.

TCPS ＝0: tool holder lock signal high leve valid.

=1: tool holder lock signal low leve valid.

Note1: If there is no tool holder lock signal, TCPS can be set high level replace of connected.

3. P180

Tool num choose:

Set value 0-8

Default: 4

4. P181

Delay time from tool holder forward stop to reversal lock.

Set value: 0-10000(0-40s), unit: 4 ms.

Default : 10

5.P183

Time from receives TCP signal to tool hold reversal.

Set value: 0-10000(0-40s), unit: 4ms.

Default: 240.

Note: if the value is too small, maybe lead tool holder can not lock. So turn up this parameter.

6. P182

Max time from the first tool changing to the last tool.

Set value: 0-10000(0-40s), unit: 4ms.

Default: 10000.

7. P184

Max time of receiving the tool hold lock signal.

Set value: 0-10000(0-40s), unit: 4ms.

Default: 400.

3.3.3 Cutting tool compensation function

In the actual processing, usually needs a few tools, but every tool cutting point is not the same, it is necessary to calculate coordinate offset of each tool cutting point to guarantee all tools moving track are the same. When call different tools, system automatically repairs offset between two tools to guarantee all tools moving track are the same. The offset value is called tool repairing value, the file record all tool repairing values is called tool repairing list.

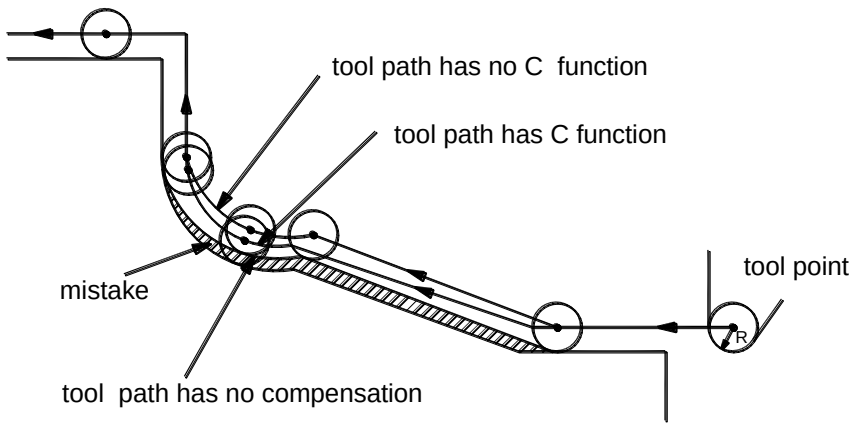
Tool repairing num regards tool repairing value of tool repairing list, which is used to calculate coordinate. Tool offset value input by key, one tool offset num regards X axis and Z axis tool offset value.

This system affords twenty four offset nums, tool offset num in this range is valid.

Chapter 4 Tool offset C function

Exactly tool point is not a point, it is a segment of circle. So exactly exist mistakes between process result and workpiece program, tool offset C function can realize removing this mistake. C function will take effected if relevant parameter is set, P002 Bit3 is one.

0	0	2					CCMP			



4.1 Basic conception

4.1.1 Image tool nose concept

Tool point A is a image point, exactly is not exist, so is called image tool point. Setting image tool point makes setting tool point radius center at starting position easier. Image tool point is not need to consider tool point radius.

Note: as for machine tool has machine zero, a standard point like tool holder center can be a starting point. Distance from this standard point to tool point radius center or image tool point center is called tool offset value.

Distance from this standard point to tool point radius center is like setting tool point radius center as starting point, while Distance from this standard point to image tool point is like setting image tool point center as starting point. In order to set tool offset value, usually set distance from standard point to image tool point as tool offset value because it is measured easier.

When tool holder center set as starting point, tool offset value is as fig. 4-1-1-2:

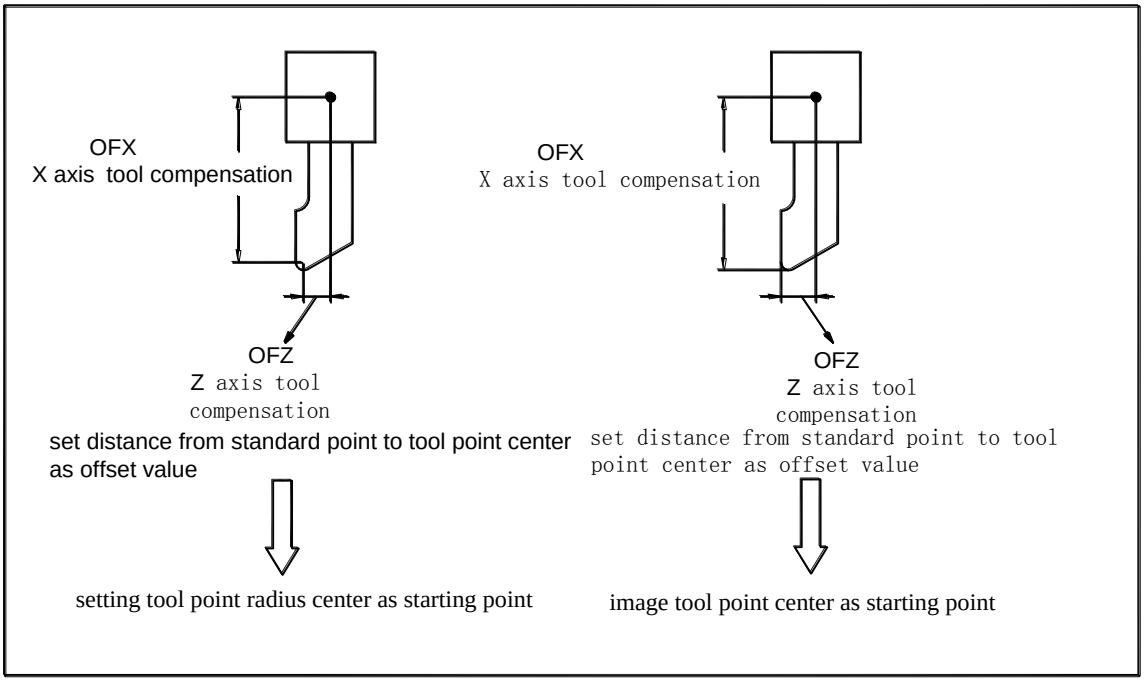


Fig. 4-1-1-2

Fig. 4-1-1-3 and 4-1-1-4 are tool tracks of tool point center program and image tool point program respectively. Left figure is path with non tool point radius compensation, while right figure is not.

Tool point track is the same with program track precision cutting.

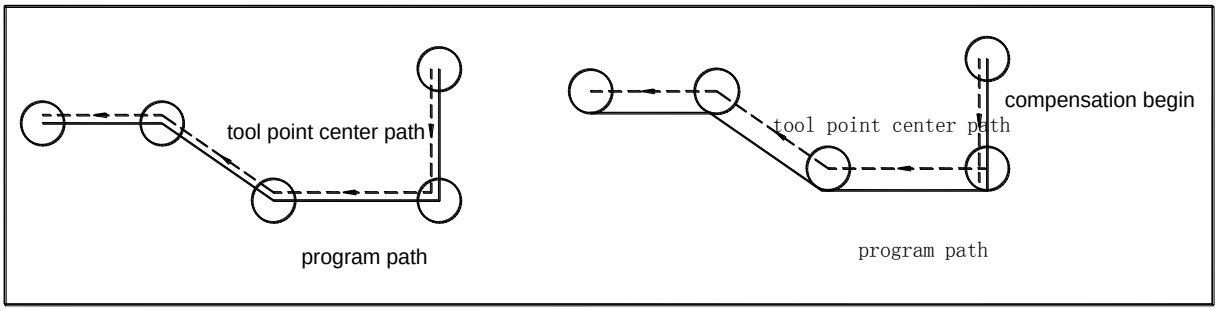


Fig. 4-1-1-3

Tool point track is the same with program track precision cutting.

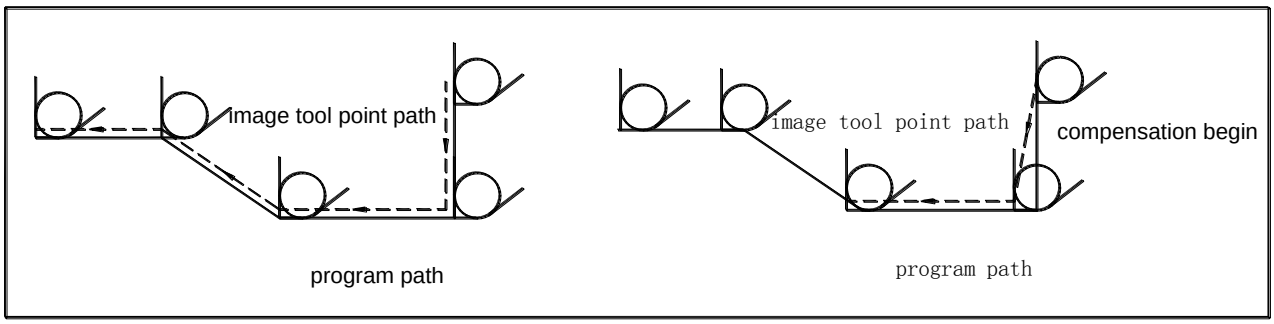


Fig. 4-1-1-4

4.1.2 Image tool nose direction

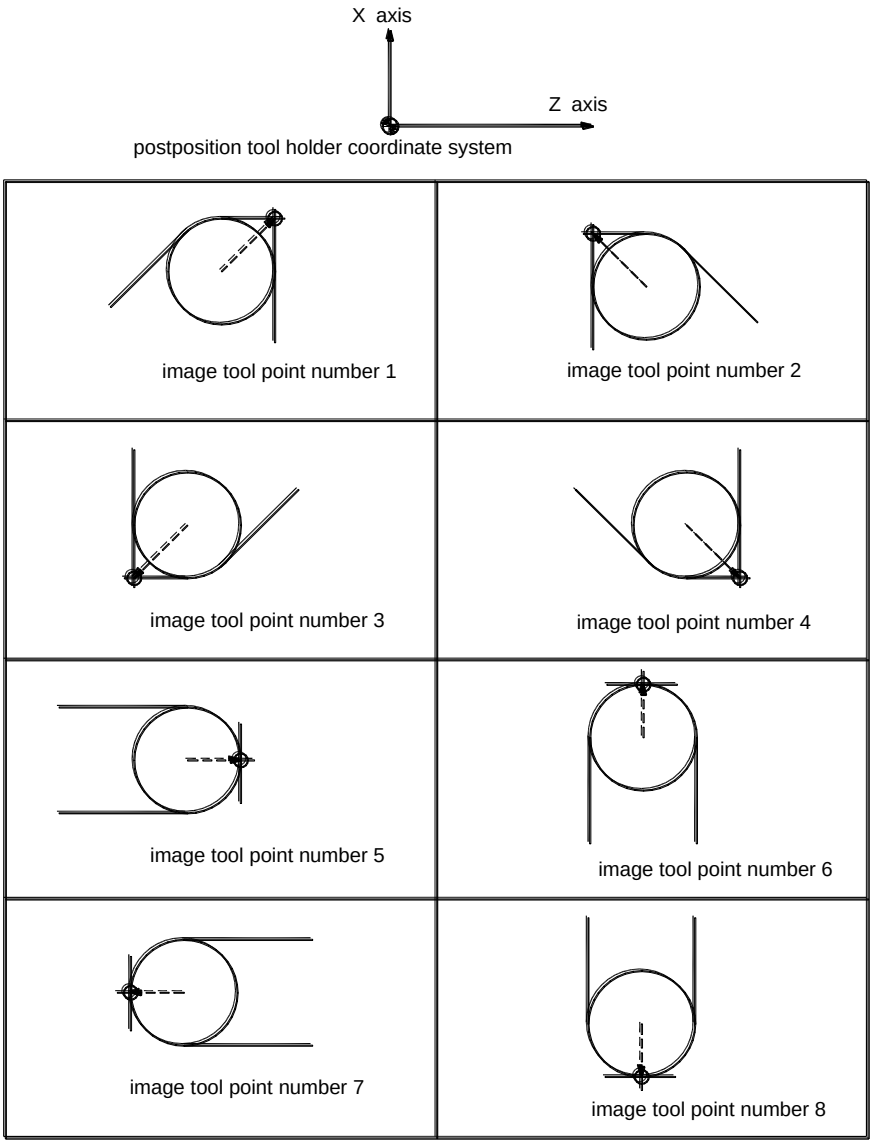


Fig. 4-1-2-1

Exactly processing, tool and workpiece have the different position system.The direction of look form tool point to image tool point is decided by cutting tool direction.

Image tool point number defines position relationship between image tool point and tool point circle center. Image tool point number totally has 10 kinds, that represents 9 kinds directions. Image tool point number direction can be choosed from 8 kinds . these pictures describe relationships between tool and starting point. End of arrows is image tool point.

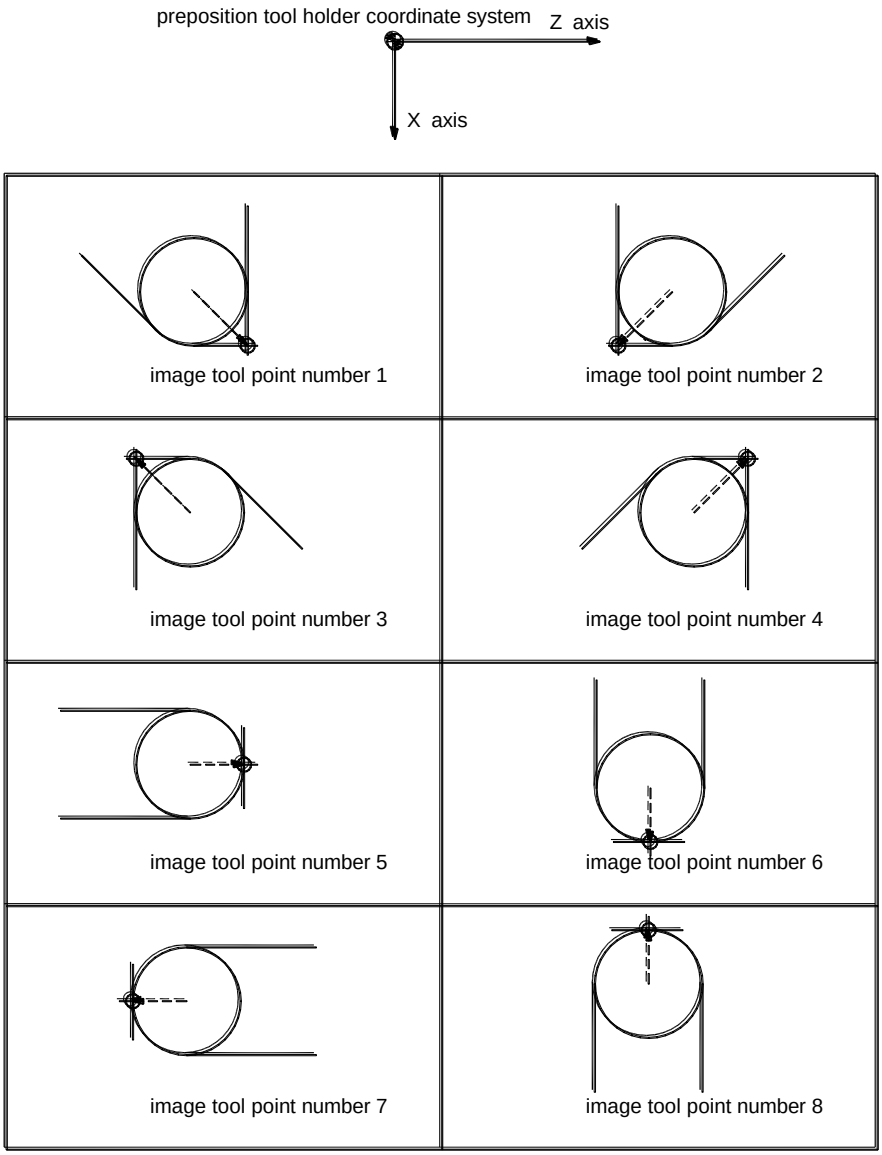
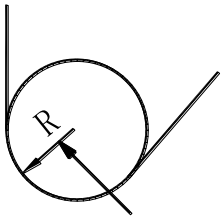


Fig. 4-1-2-2

When tool point center and starting point are the same, sets tool point number is 0 or 9, as shown in 4-1-2-3.

Corresponding each tool compensation, set each tool image tool point using address T.

4.1.3 Set compensation amount



tool point radius compensation amount

Fig. 4-1-3

Tool point radius compensation need to set: X,Z,R,T. Among this X, Z are tool offset from tool holder center to tool point of X and Z axis. R is image tool point radius compensation amount. T is image tool point num. Each group correspond one tool offset num. Set in tool offset display. As shown in list 4-1-3.

List 4-1-3 display

Num	X	Z	R	T
001	0.020	0.030	0.020	2
002	0.060	0.060	0.016	3
..	..	..	..	..
..	..	..	..	..
..	..	..	..	..
011	0.030	0.026	0.18	9
012	0.050	0.038	0.20	1

4.1.4 Relative position of tool and workpiece

When tool point radius compensation, system must addict relative position of tool and workpiece. In postposition coordinate sysem, tool center track in the right side of program track is called right tool offset, code is G42; while in the left is called left tool offset, code is G41. preposition is opposite to postposition. Relative position specify description is as list 4-1-4:

List 4-1-4

Code	Function	Remark
G40	cancle tool point radius compensation	Fig. 4-1-4-1 Fig. 4-1-4-2
G41	Means tool point radius left offset in postposition holder coordinate system, while tool point radius right offset in preposition holder coordinate system.	
G42	Means tool point radius right offset in postposition holder coordinate system, while tool point radius left offset in preposition holder coordinate system.	

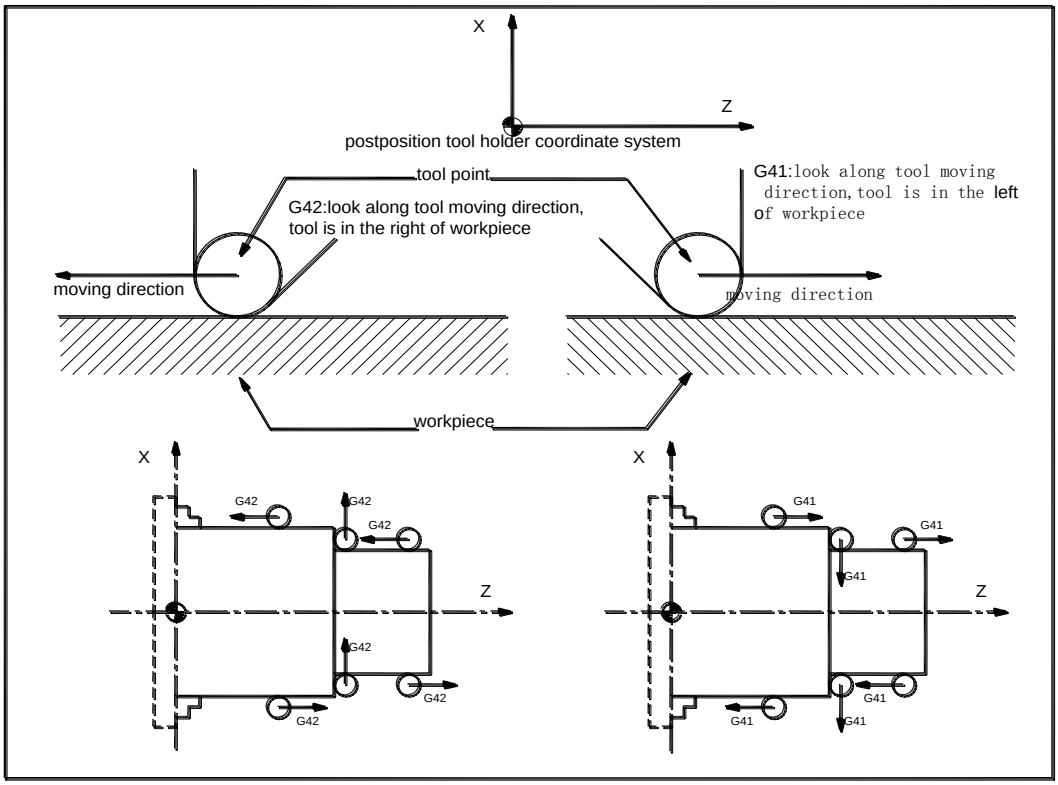


Fig. 4-1-4-1

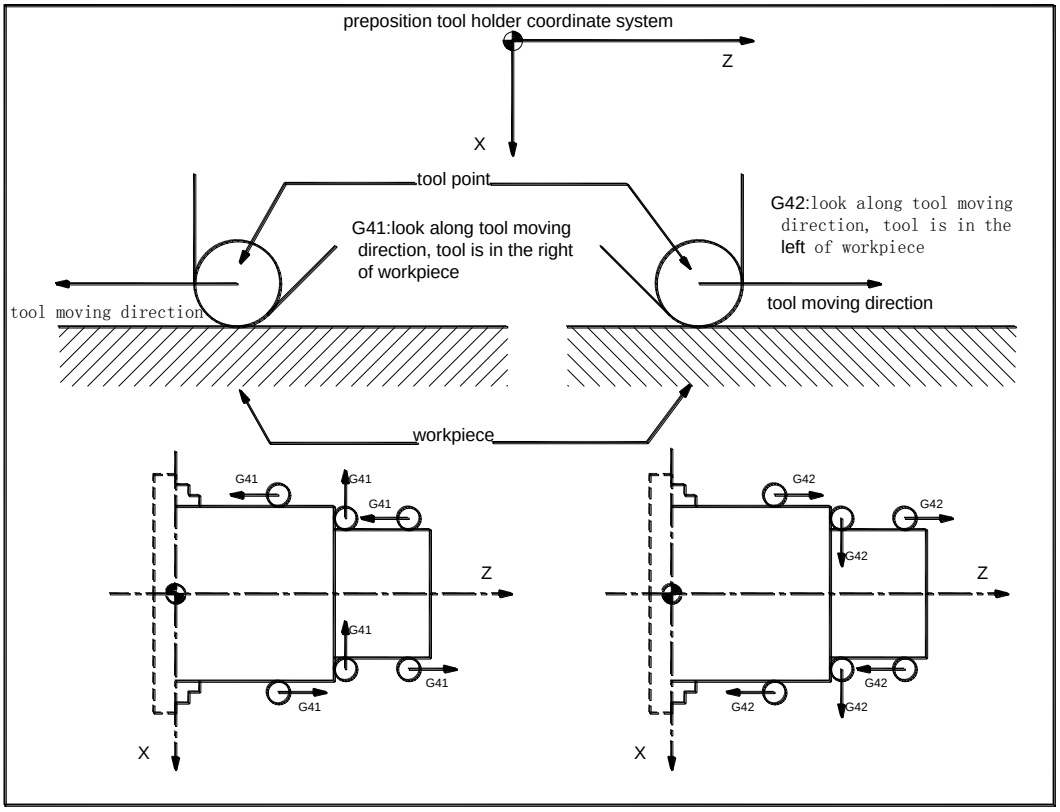


Fig. 4-1-4-2

4.1.5 Inside\outside

When tool point radius compensation, turning angles of front and behind program tracks are not the same, the angles greater than or equal to 180 is called “inside”, between 0 and 180 is called “outside”.

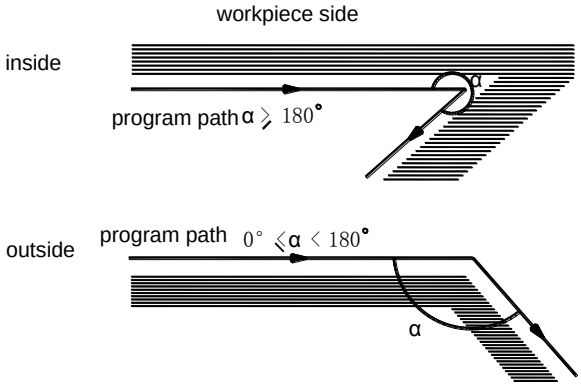


Fig. 4-1-5

4.1.6 G41, G42 and G40 code format

Format:

$$\begin{Bmatrix} G40 \\ G41 \\ G42 \end{Bmatrix} \begin{Bmatrix} G00 \\ G01 \end{Bmatrix} X- \quad Z-$$

Note1: G40,G41,G42 are modal.

Note2: after set tool offset, G41 or G42 can be followed by G02 or G03.

Note3: after set tool offset, only G00/G01/G02/G03 can be carried on in program. If it has other commands, system will alarm 142.

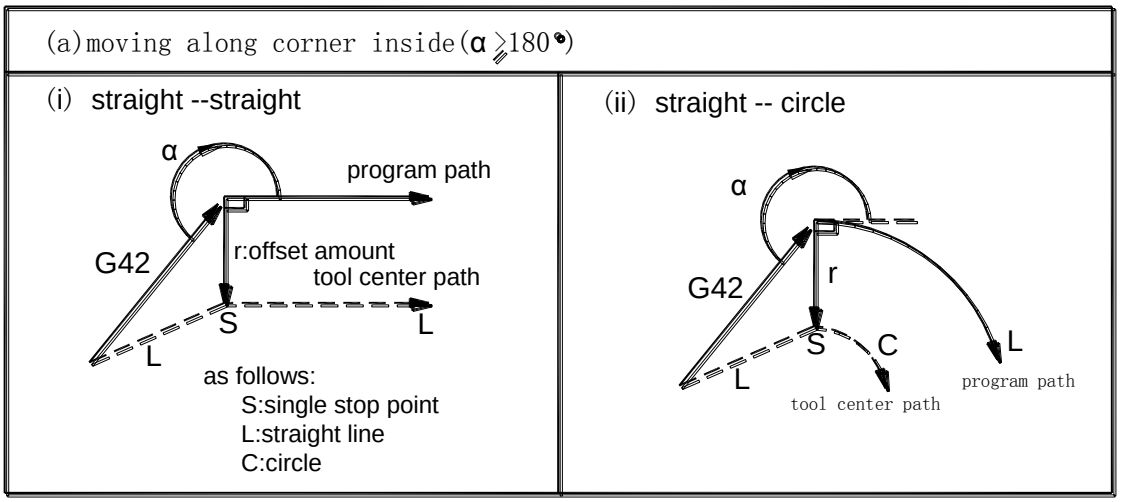
4.2 Tool offset condition

4.2.1 Tool nose radius compensation track resolve

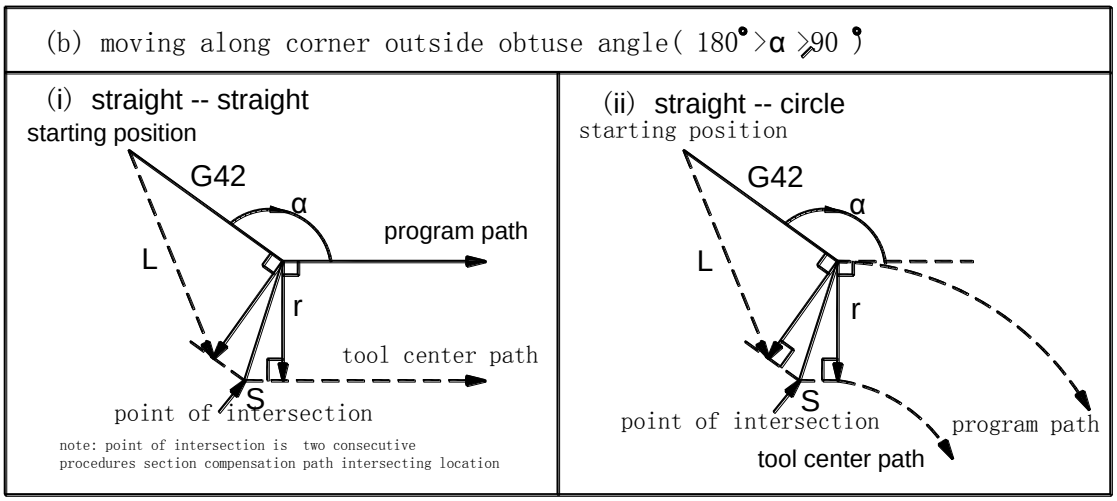
Usually it needs three steps to realize tool radius compensation: tool offset set, process,cancle.

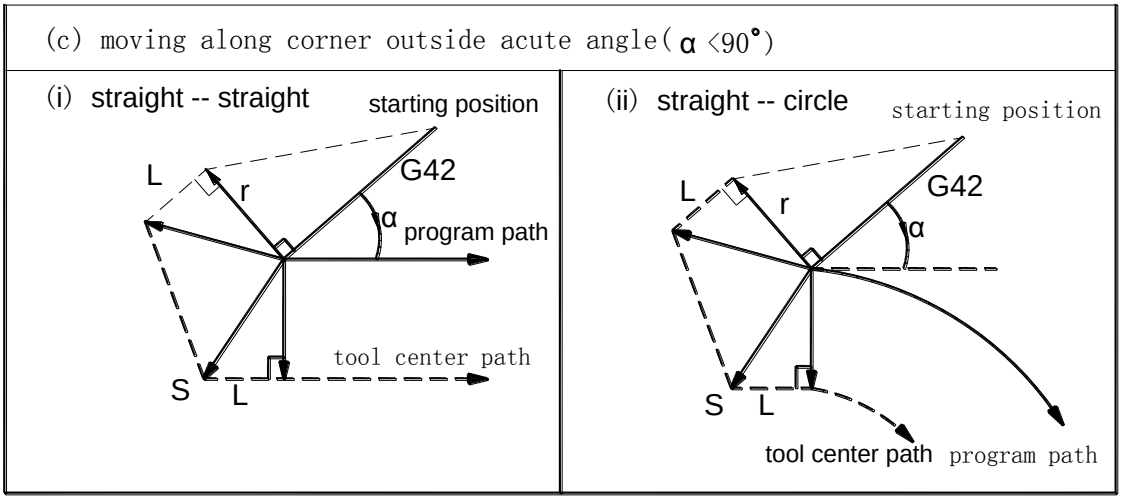
1. Tool offset set

Manner changes from offset cancle to offset is called tool offset set.



Specific tool offset set:





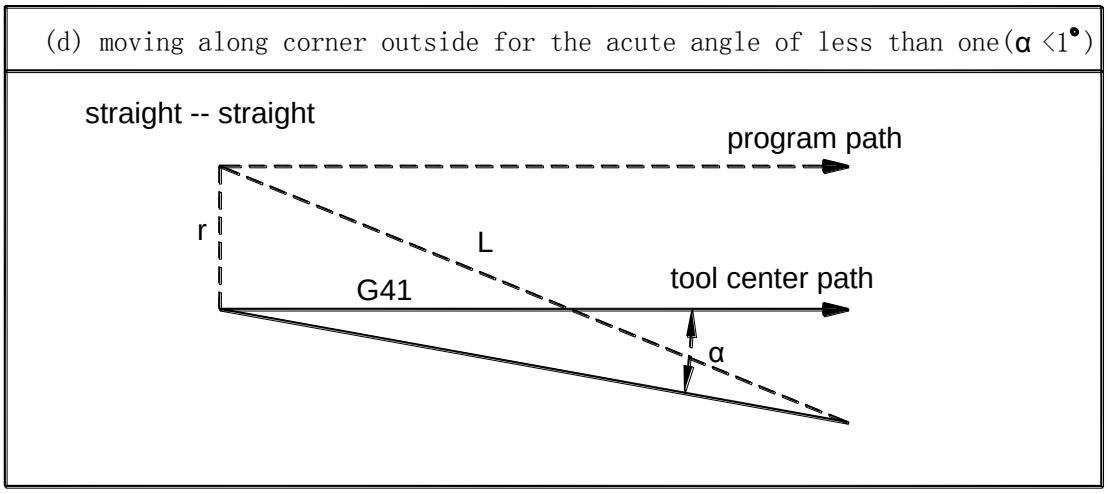


Fig. 4-2-1-1 tool offset set

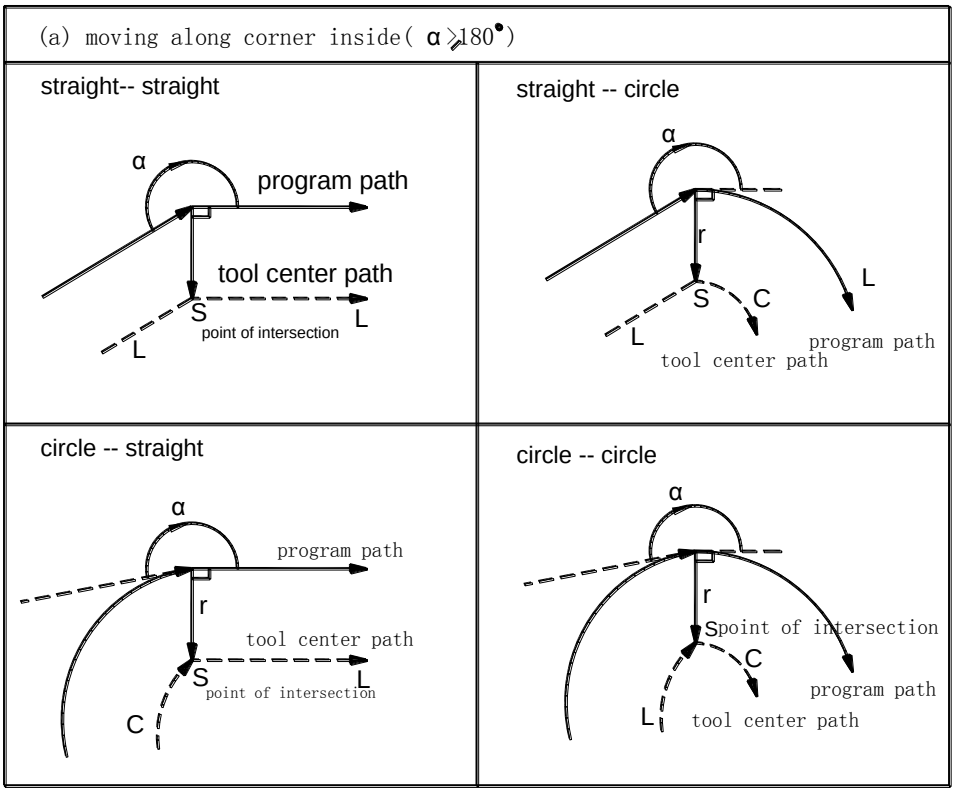
Note1: when set offset , if there has not addicted tool offset num or the num is 0, program will alarm 138.

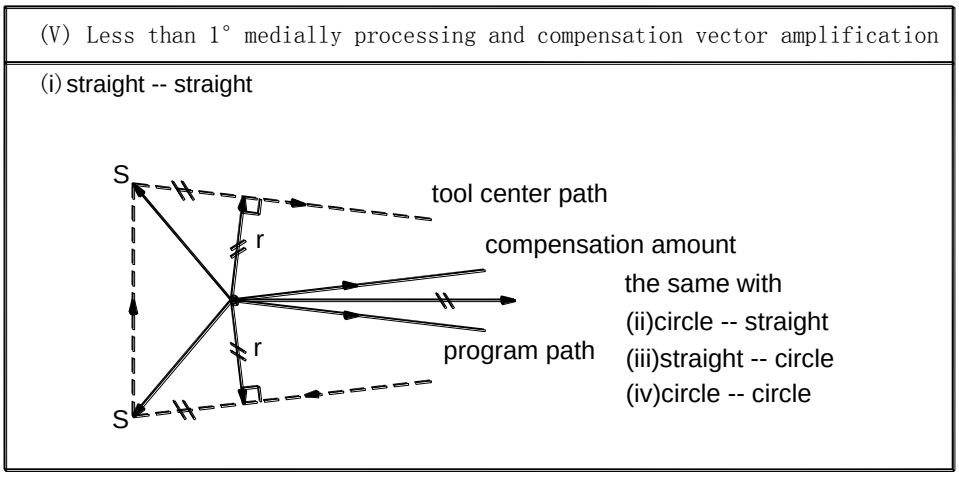
Note2: when set offset, it is need moving command G0 or G1 to carried on, if carry on circle, program will alarm 139.

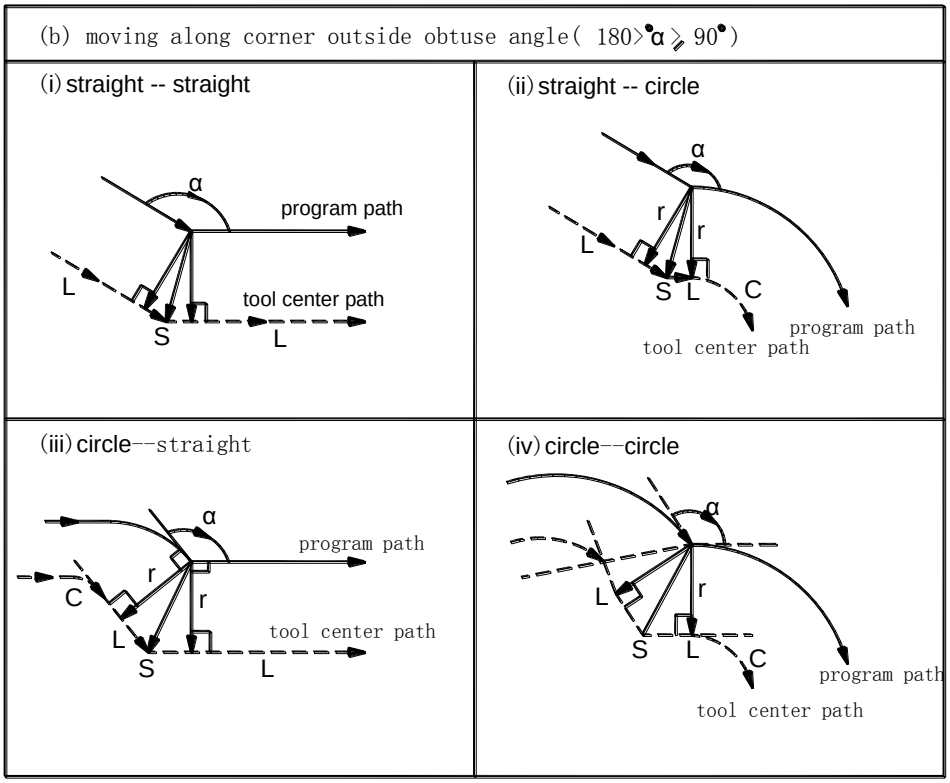
2. Tool offset process

Offset track from tool offset set to offset cancle is called tool offset process.

Specific tool offset process:







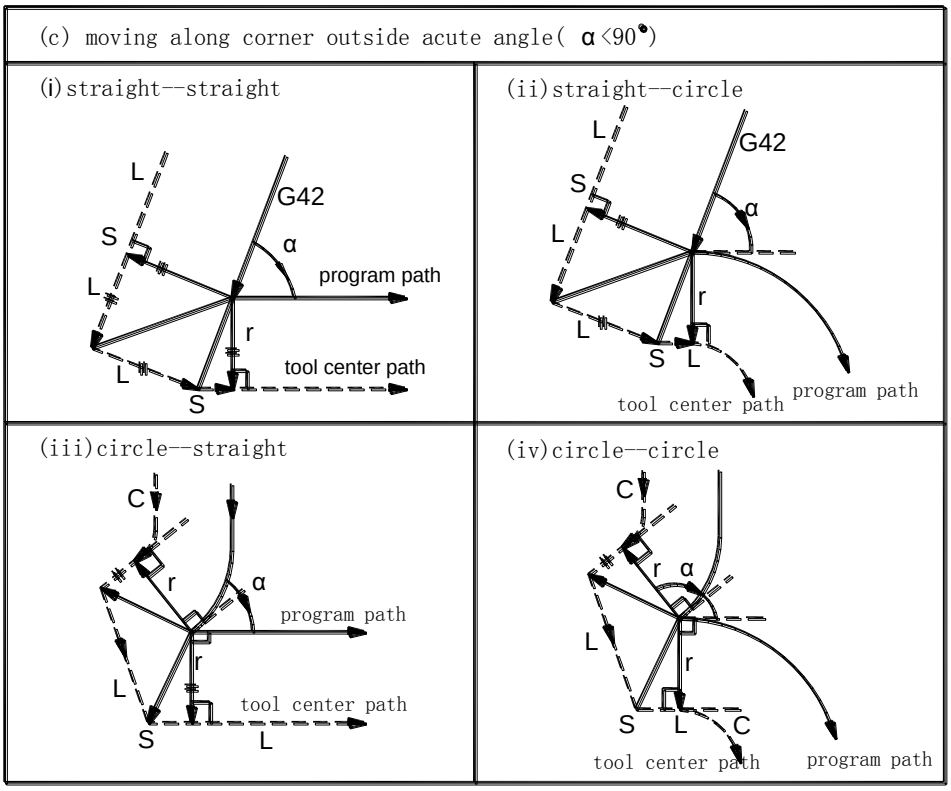


Fig. 4-2-1-2 tool offset process

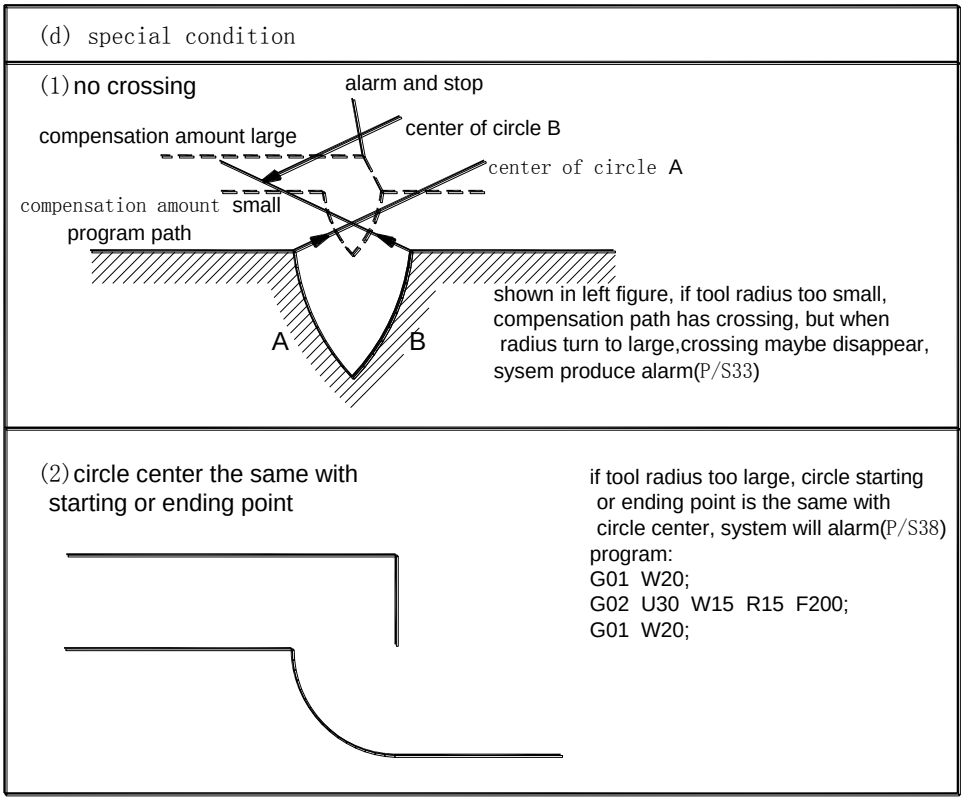


Fig. 4-2-1-3 tool offset process②

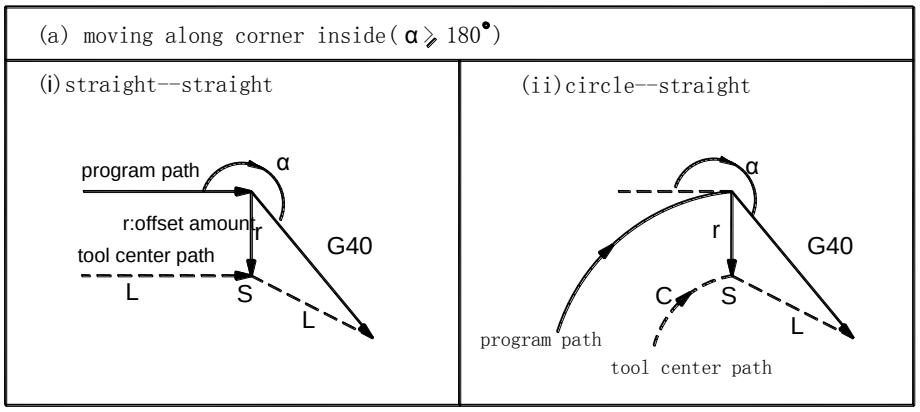
4.3 Tool offset cancle

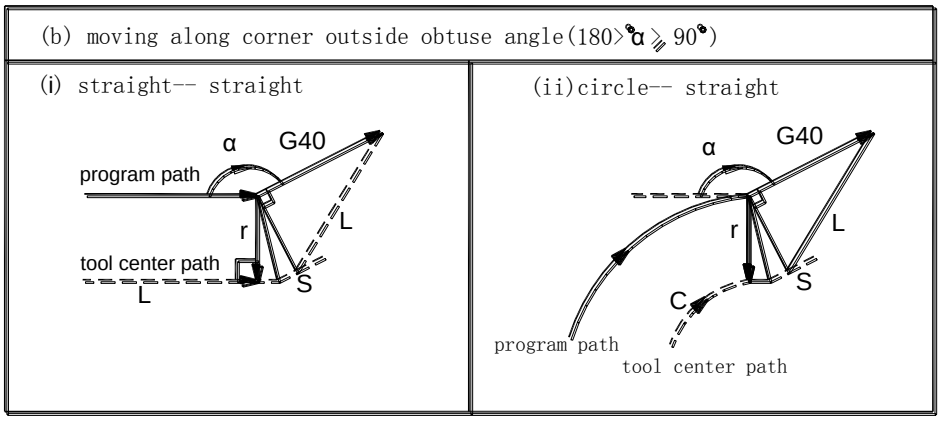
In compensation mode, if program satisfies some one condition, system will enter offset cancle mode, this program segment is called tool offset cancle.

(1) use code G40 cancle C tool offset, during this, can not use circle command (G2 and G3). If command circle, system will alarm 139 and tool stop.

(2) tool radius compensation num is addicted 0.

Specific tool offset cancle :





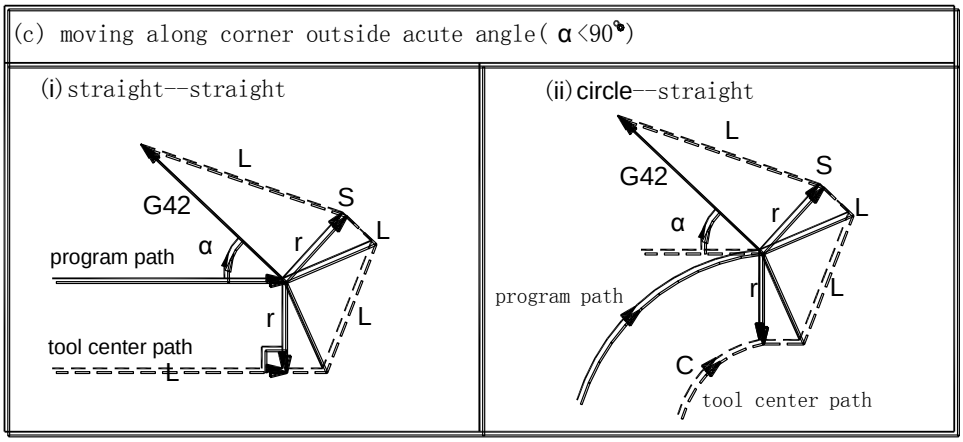


Fig. 4-2-1-4 tool offset cancle

4.3.1 Change compensation direction during tool offset process

Tool radius compensation G code (G41 and G42) decide compensation direction. Amount character is as follows:

Character / G code	+	-
G41	Left side compensation	Right side compensation
G42	Right side compensation	Left side compensation

In special occasion, compensation direction can be changed in compensation mode. But it can not be changed in the initial program. If compensation direction changes, it has no concept of inside and outside. Image following compensation is positive:

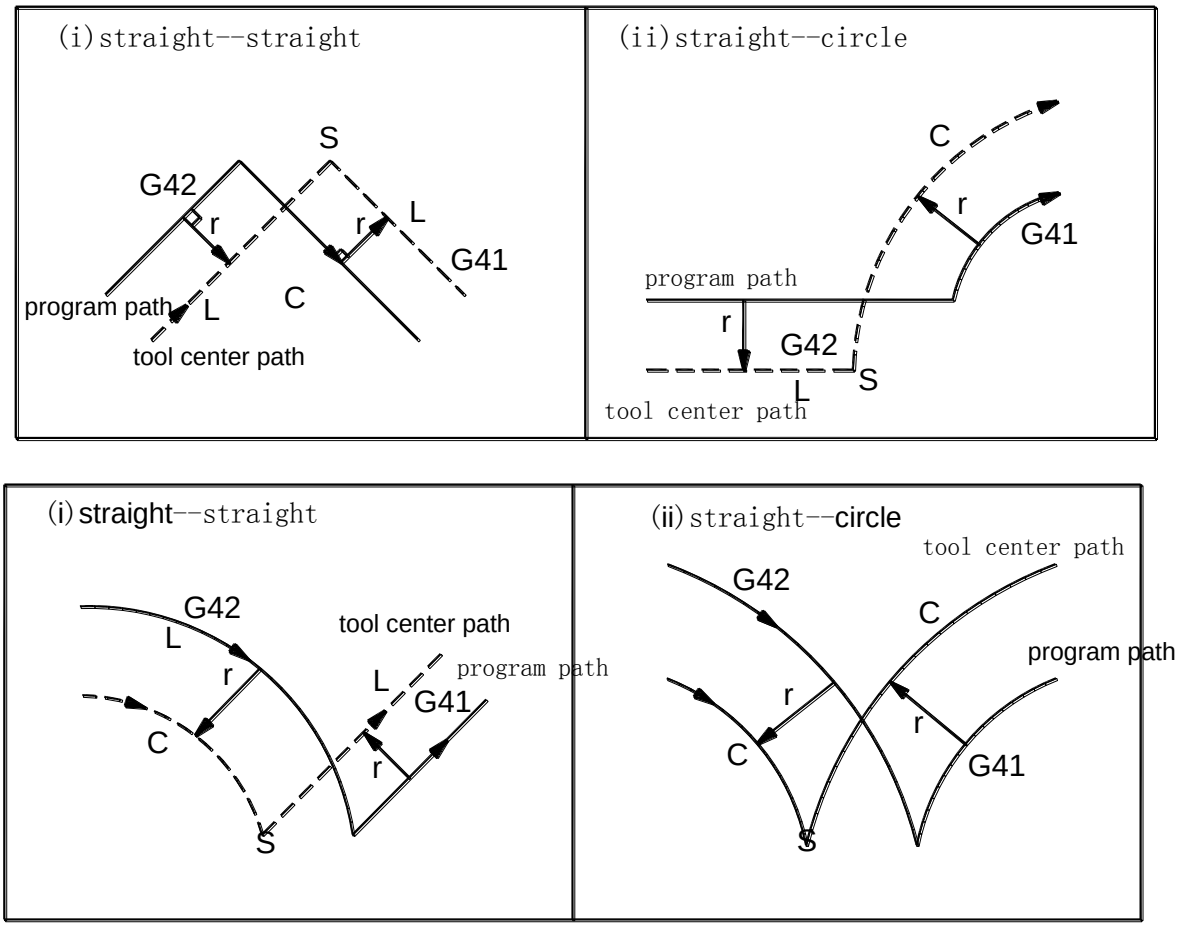
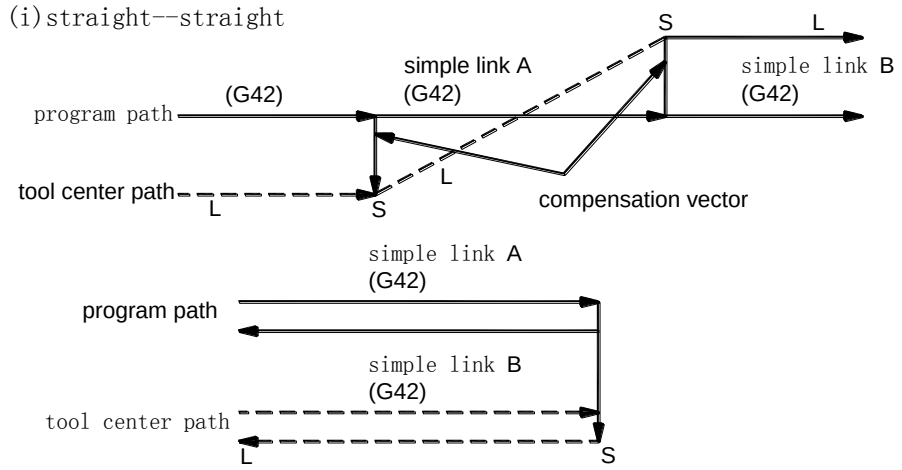


Fig. 4-2-2-1 change compensation direction during tool offset process

If there is no crossing, when use G41 and G42 changing program segment A and B compensation direction, starting point of segment B is regarded as vector perpendicular to segment B if crossing of path is not needed.



(ii)straight--circle

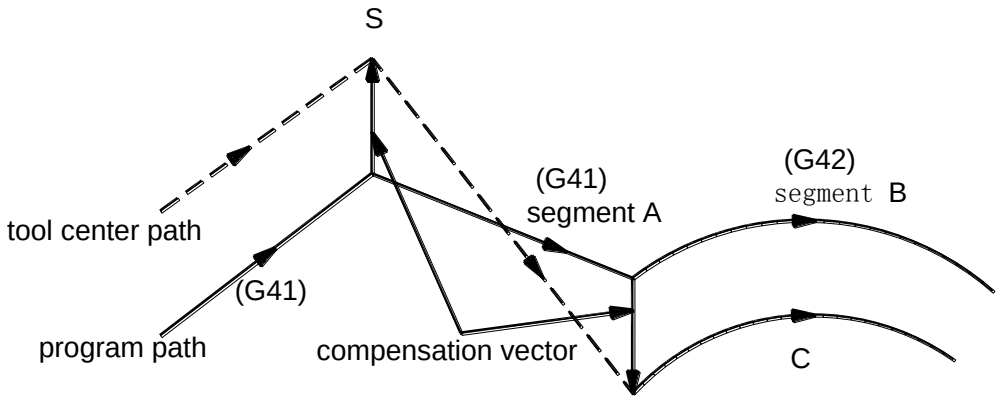


Fig. 4-2-2-2 straight—circle \ no crossing

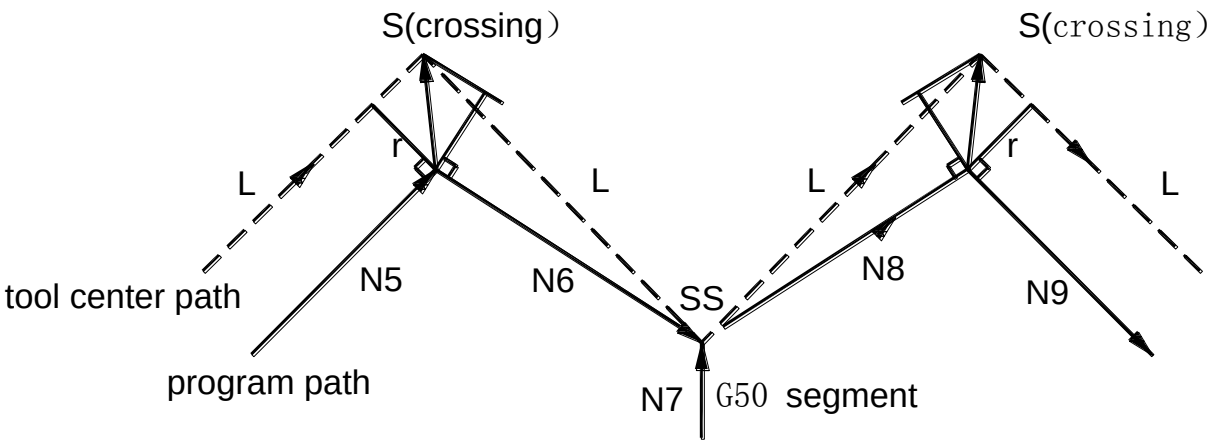
4.3.2 Tool offset cancle

In compensation mode, if addicted following command, tool offset will be cancled at the moment, then it will be restored automatically. At this time, tool moves from crossing point to cancle command point directly. When compensation mode is restored, tool moves back to crossing again.

1. coordinate system set（G50）

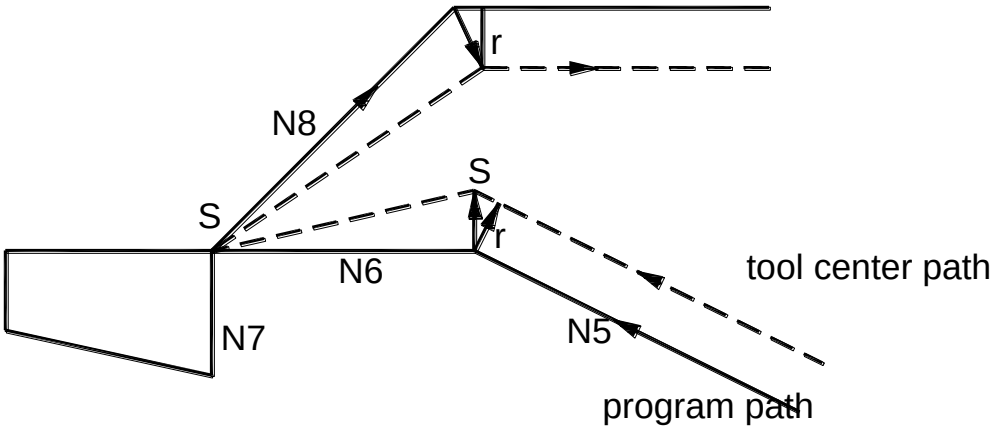
```
N1 T0101 ;
N2 G42 G00 X0 Z0 ;
N3 G01 U-30 W30 ;
N4 U30 W30 ;
N5 G50 X0 Z60 ;
N6 G01 U-30 W30 ;
N7 G01 U30 W30 ;
N8 G00 X0 Z0 ;
```

N9 M30 ;



Note: SS means tool stop the point twice in single mode.

2. G90, G92, G94 fixed cycle, G71～G76 fixed, thread code G32/G33/G34



N1 T0101

N2 G0 X100 Z100

N3 G0 X0 Z0

N4 G42 G90 X-20 W-50 F500

N5 G0 X50 Z50

N6 G0 X100 Z100

N7 M30

Note: G90/G94 tool offset only realize when G41/G42 at the same segment with G90/G94. Or lese system will regard as G90/G94 normal tool offset automatically.

4.3.3 Tool offset has non-moving command

1. at the beginning

If there is no tool moving command at the beginning, it will not produce compensation vector.

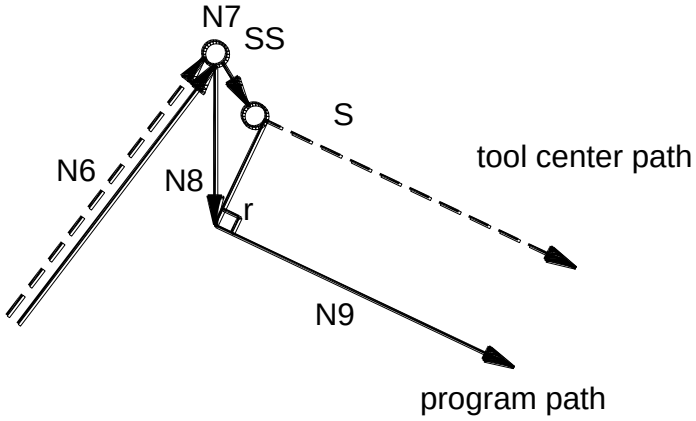


Fig. 4-2-4-1

N1 T0101 ;

N2 G0 X0 Z0 ;

N3 G01 U-30 W20 F500 ;

N4 G42 U0 ;

N5 U30 ;

N6 U20 W20 ;

N7 G40 G0 X100 Z100 ;

N8 M30 ;

2. In compensation mode

If there is only one none tool moving command at the beginning,vector and tool center path are the same with non-moving command. This non tool moving segment is carried on at the single stop point.

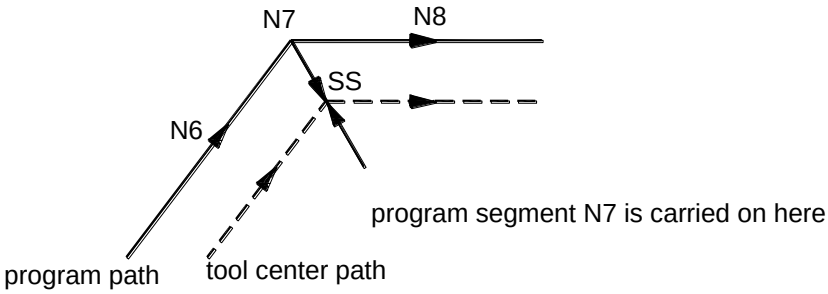


Fig. 4-2-4-2

N3 T0101 ;

N4 G0 X1000 Z100 ;

N5 G41 G01 X0 Z0 ;

N6 U-30 W20 ;

N7 G04 X5 ;

N8 W30 ;

N9 G40 G0 X100 Z100 ;

N10 M30 ;

3．At tool cancle time

When there is no tool moving command together with compensation cancle, it will produce a vector whose length is compensation amount and direction is perpendicular to formal segment moving direction. This vector will cancled in the next moving command.

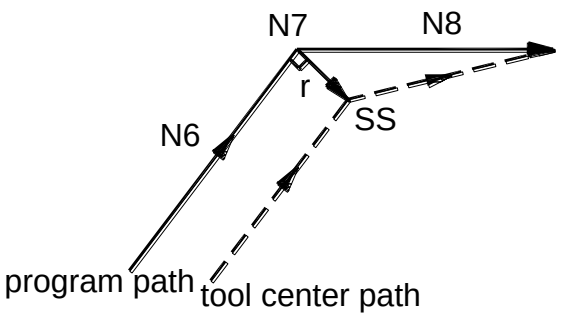


Fig. 4-2-4-3

N3 T0101 ;

N4 G0 X100 Z100 ;

N5 G41 G01 X0 Z0 F500 ;

N6 U-30 W20 ;

N7 G04 ;

N8 G0 X100 Z100 ;

N9 M30 ;

4.3.4 Tool offset intervene check

Tool cutting override is called intervene, tool offset intervene check can pre-check tool intervene cutting condition, even if not it still check intervene.

（a） basic condition

1) tool path direction is different with program path.

2) When process circle, except above conditions, angle between starting point and ending point of tool center path has a big difference with angle of program path.

Example 1:

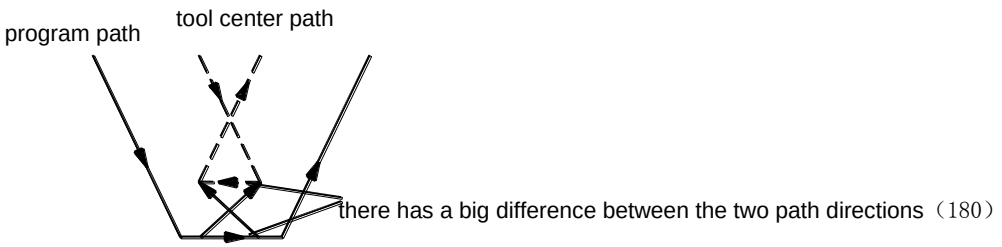


Fig. 4-2-5-1

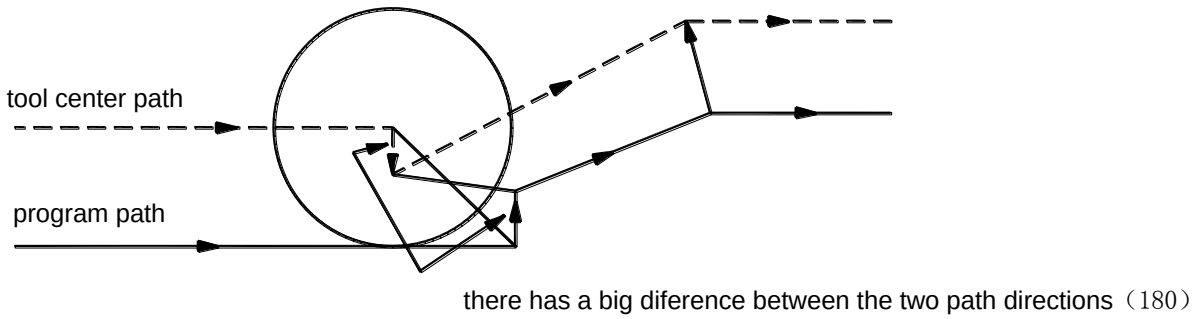


Fig. 4-2-5-2

（b）intervene example

（1）one shallow depth, depth is less than compensation amount

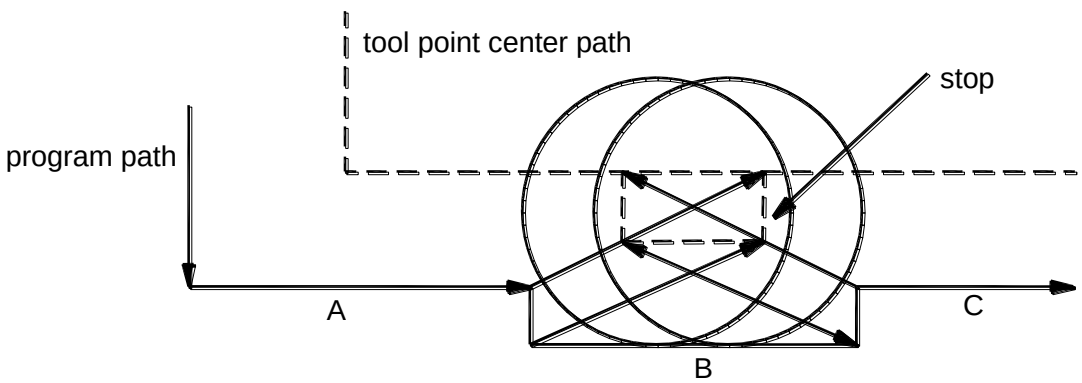


Fig. 4-2-5-3

Program :

N1 T0101 （R≤10）

N2 G0 X0 Z30

N3 G42 G01 X50 Z0 F500

N4 U50

N5 W20

N6 U10

N7 W20

N8 U-10

N9 W20

N10 G40 G0 X0 Z30

N11 M30

As shown in above program, tool point radius of num 01 is less than 10, when it larger than 10, system will produce alarm, program B direction and tool radius compensation path are different.

（2）chase depth is less than compensation amount

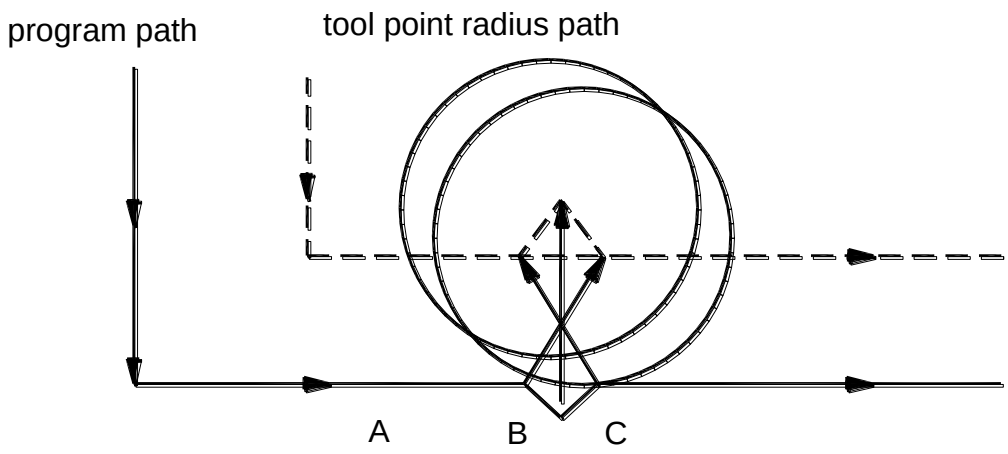


Fig. 4-2-5-4

Program :

N1 T0101

N2 G0 X0 Z30

N3 G42 G01 X50 Z0 F500

N4 U50

N5 W20

N6 U10 W10

N7 U-10 W10

N8 W20

N9 G40 G0 X0 Z30

N10 M30

As shown in above program, tool point radius of num 01 is less than or equal to 25, when it is larger than 25, system will produce alarm, because program C direction and tool radius compensation path are different.

4.4 Tool offset C note

1. only G00-G03 can appear during compensation process.

2. tool point radius offset will not be carried on in MDI mode.

3. tool point offset set and candled can only by G00 or G01 code, while not circle code(G02 or G03).

4. before call sub program(M8), system should in offset cancle mode. After enter sub program, offset can be open, but before return to main program(M9) , system should in offset cancle mode too.

5. if offset value(R) is negative, that means G41 and G42 exchanges. If set tool center moves along workpiece outside, it will moves along workpiece inside. V.V. That is because ifoffset value character changes, the offset direction will change, but image tool point direction will not change.

6. change offset value usually in cancle mode, if not, the new offset value will valid after tool changing.

7. during carry on tool offset program, alarm for various reasons, G code will hold, the original is G41,it will be G41, the original is G42,it will be G42. At this time, tool offset should be cancled, so in MDI mode, caryy on G0 can cancle tool offset state.

8. G33/G34 will not carry on tool point radius compensation.

4.5 Tool offset C example

Process workpiece and size are as shown in 4-5-1, tool point radius R=1, the first tool.

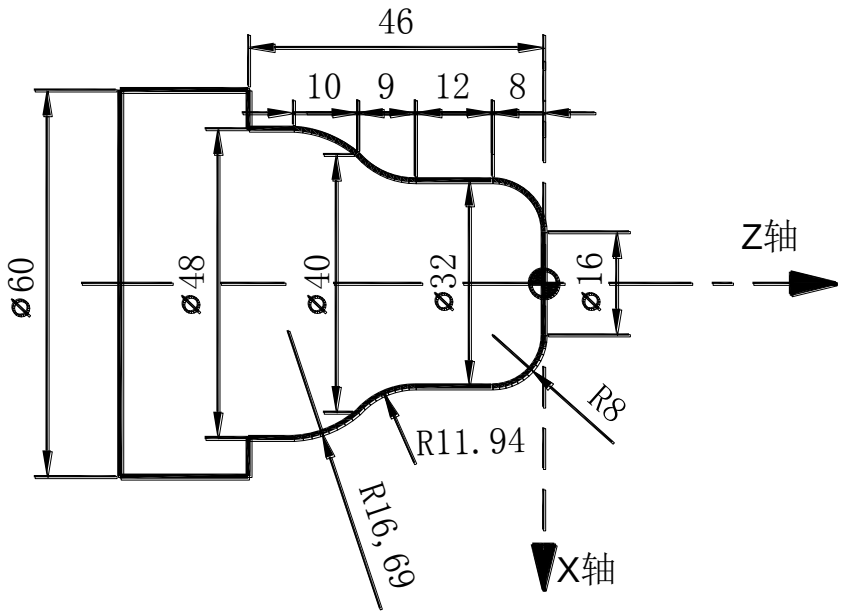


Fig. 4-5-1

Program :

O0001 ;

N010 G50 X100.0 Z100.0（set coordinate system）

N020 M3 S1200 ; （spindle forward,rotation:1200r/min）

N030 M08 ; （open cold）

N040 T0101 ; （change to num one tool）

N050 G00 X16.0 Z5.0 ; （fast locating）

N060 G42 G1 Z0 F80 ; （begin to carry on tool point offset）

N070 G3 X32 Z-8 R8 ;

N080 G1 Z-20 ;

N090 G02 X40 Z-29 R11.94 ;

N100 G3 X48 Z-39 R16.69 ;

N110 G1 Z-46 ;

N120 X60 ;

N130 G40 G0 X80 Z80 ; （cancle tool point offset） ;

N140 M09 ; （close cold）

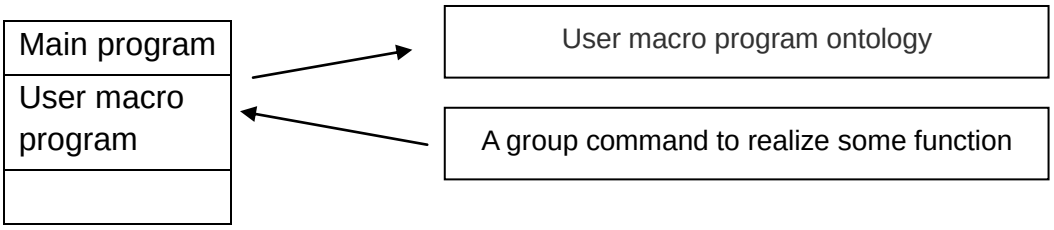
N150 G00 X100.0 Z100.0 T0200 ; （fast return to safe position,cancle tool offset）

N160 M30 ; （over）

Chapter 5 Macro program

5.1 Definition

A kind of function like sub program saved to memory in advance, and regarded it by a command. If the command is written, it can realize this function. Command is called user macro program.



5.2 User macro code

User macro code is command to call user macro program ontology by command M98 and M99.

Format: M98 P***＃＃＃＃ < variable > ;

M99 ;

＃＃＃＃: sub program name/ macro program name, four figures.

***: call times of sub program/macro program, default one time. the max number is 999.

Variable : Data moved to macro program, its value assigned to according part variable.

Use command, it can call macro program ontology addicted by P.

M98 can have four level nests.

Variable addicted:

Use Ii\Ji\Ki (i is 1 or 2), depend on using letter and appeared times(I \J \K) deciding variable according number automatically. The max number is 6.

Address	Number
I1	#01
J1	#02
K1	#03
I2	#04
J2	#05
K2	#06

Note1: subscript of I,J,K is used to show variable order, can not be written in program.

Note2: system will recognize variable by appeared order and times.

Example:

M98 P9010 I14 J15 K6 I7 J9 K11;

It will called program O9010, and pass variable I,J,K to part variable, relationship is like:

#01=14, #02=15, #03=6, #04=7, #05=9, #06=11 ;

5.3 User macro program ontology

On user macro program ontology, it can use normal CNC command, it can also use variable, operation,and transfer command.

User macro program ontology starting as followed O program number and end as M99.

08000;	Program number
G65 H01... ...;	Calculate command
G00 X#101... ..	Using variable CNC command
…	
…	
G65 H82...;	Transfer command
M99	End

5.4 Macro variable

Using method:

Variable can addicted address of user macro program ontology. Variable can set by main program assign or through keyboard. It can also assigned when macro program ontology is carried on.

These variables are differential by number.

1. addicted

   By “#” + number

   Format: # i(i=100, 102, 103, ……) ;

   Example: #105, #109, #125

2. called

   Replace value followed by address.

   If it has “<address> # i ”or “<address－# i >”, variable value will regard as address.

   (example) F#203… when #203 is 15, it is the same meaning with F15.

   Z-#210… when #210 is 250, it is the same meaning with Z-250.

   G#230… when #203 is 3, it is the same meaning with G3.

Replace variable number:

（example） #200 = 205, #205 = 500

X#9200 is the same with X500

X-#9200 is the same with X-500

Note1: address O and N can not quote variable.such as O#200,N#220.

Note2: if it is more than max command value, it can not be used.

Note3: it can be displayed on LCD screen. It can also be input by keyboard.

Number	Type	Function
#1--#31	Part	It is used to transfer parameter, storage data such as calculate result. Variable addict part variable when macro program called. Value default 0.
#200~#439	Public	It is public. It can be used in every macro program. When power off, value of #200~#439 is formatted to 0, while value of #500~#579 is to saved.
#500~#579		
#1000~	System	System variable

Note: #0 is empty, can not be used.

Public variable input value range from -99999999 to 99999999, int and decimal part should total have 8 numbers, or else it will alarm. When calculate results range of -99999999 to 99999999 and stored in, system will display “***”, if this value is used, the command will be ignored. But middle result during calculating can be more than effective input bits.

System variable is fixed. Details as dignose list.

1) interface input signal #1000-#1072

2) interface output signal #1100 - #1164

3) macro program alarm

program can produce alarm addicted by user and it can only be written.

Number	Function
#1200	When carried on #1200 system stop and alarm. Alarm number is value of #1200 plus 500, range of 500 to 550, if value of #1200 less than 0, alarm number is 500; while if it more than 200, alarm number is 550.

Such as:

G65 H01 P#1200 Q06;

When carries on this segment, system will stop and alarm E506. Because of buffer exist, system will alarm in advance.

4) tool offset variable:

Offset num	X axis offset amount	Z axis offset amount	Y axis offset amount	A axis offset amount
1	#2001	#2101	#2201	#2301
•••	•••	•••	•••	•••
24	#2024	#2124	#2224	#2324

5) processed piece number :

Number	Function
#3901	Have been processed pieces number

6）coordinate position message:

Number	Position signal	Coordinate system	Tool offset	Read operation during movment
#5021~#5024	Current positon	Machine tool coordinate system	include	Impossible
#5041~#5044	Current positon	Workpiece coordinate system		

Note: the above table list messages correspond X ,Y, Z, A in order. For example: #5021 means X axis message,#5022 means Y axis message,#5023 means Z axis message,#5024 means A axis message.

Finished piece number, spindle rotation speed amount and coordinate message are read only.

5.5 Calculate command and transfer command G65

Normal format:

G65 H<u>m</u> P<u># i</u> Q<u># j</u> R<u># k</u> ;

Among them: m: kind of function

i: store calculate result

j: variable 1, can be constant

k: variable 2, can be constant

Code significance:

i = #j O # k

Symbol ,decided by Hm.

For example:

P#100 Q#101 R#102 ….. #100 = #101 O #102 ;

P#100 Q#101 R15 …. #100 = #101 O 15 ;

P#100 Q-100 R#102 ….. #100 = -100 O #102 ;

Description: when variable is constant it can not have “#”.

Macro calculate list

Format	Function	Definition
G65 H01 P#I Q#J ;	Assignment	# i = # j ; give variable #j’s value to variable #i
G65 H02 P#i Q#j R#k ;	Decimalism add operation	# i = # j + # k
G65 H03 P#i Q#j R#k ;	Decimalism minus operation	# i = # j - # k
G65 H04 P#i Q#j R#k ;	Decimalism multiplication operation	# i = # j×# k
G65 H05 P#i Q#j R#k ;	Decimalism division operation	# i = # j÷# k
G65 H11 P#i Q#j R#k ;	Binary system add operation	# i = # j OR # k
G65 H12 P#i Q#j R#k ;	Binary system multiplication operation	# i = # j AND # k
G65 H13 P#i Q#j R#k ;	Binary system XOR	# i = # j XOR # k
Format	Function	Definition
G65 H21 P#i Q#j ;	Decimalism sqrt	# i = $\sqrt{\# j}$
G65 H22 P#i Q#j ;	Decimalism absolute value	# i = |# j |
G65 H23 P#i Q#j R#k ;	Decimalism remainder	# i = (#j÷# k) remainder
G65 H24 P#i Q#j ;	Decimalism change to Binary system	# i = BIN(# j)
G65 H25 P#i Q#j ;	Binary system change to decimalism	# i = BCD(# j)
G65 H26 P#i Q#j R#k ;	Decimal arithmetic	# i = # i×# j÷# k
G65 H27 P#i Q#j R#k ;	Composite square root	# i = $\sqrt{\# i^2 + \# j^2}$
G65 H31 P#i Q#j R#k ;	Sin	# i = # j×sin(# k)
G65 H32 P#i Q#j R#k ;	Cos	# i = # j×cos(# k)

Format	Function	Definition
G65 H01 P#I Q#J ;	Assignment	# i = # j ; give variable #j's value to variable #i
G65 H02 P#i Q#j R#k ;	Decimalism add operation	# i = # j + # k
G65 H03 P#i Q#j R#k ;	Decimalism minus operation	# i = # j - # k
G65 H04 P#i Q#j R#k ;	Decimalism multiplication operation	# i = # j×# k
G65 H05 P#i Q#j R#k ;	Decimalism division operation	# i = # j÷# k
G65 H11 P#i Q#j R#k ;	Binary system add operation	# i = # j OR # k
G65 H12 P#i Q#j R#k ;	Binary system multiplication operation	# i = # j AND # k
G65 H13 P#i Q#j R#k ;	Binary system XOR	# i = # j XOR # k
Format	Function	Definition
G65 H21 P#i Q#j ;	Decimalism sqrt	# i = $\sqrt{\# j}$
G65 H22 P#i Q#j ;	Decimalism absolute value	# i = |# j |
G65 H23 P#i Q#j R#k ;	Decimalism remainder	# i = (#j÷# k) remainder
G65 H24 P#i Q#j ;	Decimalism change to Binary system	# i = BIN(# j)
G65 H25 P#i Q#j ;	Binary system change to decimalism	# i = BCD(# j)
G65 H33 P#i Q#j R#k ;	Tan	# i = # j×tan(# k)
G65 H34 P#i Q#j R#k ;	cot	# i = ATAN(# j / # k)
G65 H80 Pn ;	Unconditional transfer	Jump to segment n
G65 H81 Pn Q#j R#k ;	Conditional transfer 1	If # j equals # k, then jump to segment n,or else program run in order
G65 H82 Pn Q#j R#k ;	Conditional transfer 2	If # j not equals to # k, then jump to segment n,or else program run in order
G65 H83 Pn Q#j R#k ;	Conditional transfer 3	If # j more than # k, then jump to segment n,or else program run in order

Format	Function	Definition
G65 H01 P#I Q#J ;	Assignment	# i = # j ; give variable #j's value to variable #i
G65 H02 P#i Q#j R#k ;	Decimalism add operation	# i = # j + # k
G65 H03 P#i Q#j R#k ;	Decimalism minus operation	# i = # j - # k
G65 H04 P#i Q#j R#k ;	Decimalism multiplication operation	# i = # j×# k
G65 H05 P#i Q#j R#k ;	Decimalism division operation	# i = # j÷# k
G65 H11 P#i Q#j R#k ;	Binary system add operation	# i = # j OR # k
G65 H12 P#i Q#j R#k ;	Binary system multiplication operation	# i = # j AND # k
G65 H13 P#i Q#j R#k ;	Binary system XOR	# i = # j XOR # k
Format	Function	Definition
G65 H21 P#i Q#j ;	Decimalism sqrt	# i = $\sqrt{\# j}$
G65 H22 P#i Q#j ;	Decimalism absolute value	# i = |# j |
G65 H23 P#i Q#j R#k ;	Decimalism remainder	# i = (#j÷# k) remainder
G65 H24 P#i Q#j ;	Decimalism change to Binary system	# i = BIN(# j)
G65 H25 P#i Q#j ;	Binary system change to decimalism	# i = BCD(# j)
G65 H84 Pn Q#j R#k ;	Conditional transfer 4	If # j less than # k, then jump to segment n,or else program run in order
G65 H85 Pn Q#j R#k ;	Conditional transfer 5	If # j not less than # k, then jump to segment n,or else program run in order
G65 H86 Pn Q#j R#k ;	Conditional transfer 6	If # j not more than # k, then jump to segment n,or else program run in order
G65 H99 Pn ;	Produce user alarm	Produce number (500+n) alarm

5.6 Calculate command

1) Set variable value : # I = # J

G65 H01 P#I Q#J

(example)G65 H01 P#101 Q1005 ; (#101 = 1005)

G65 H01 P#101 Q#110 ; (#101 = #110)

G65 H01 P#101 Q-#102 ; (#101 = -#102)

2) Decimalism add operation: # I = # J+# K

G65 H02 P#I Q#J R#K

(example)G65 H02 P#101 Q#102 R15 ; (#101 = #102+15)

3) Decimalism minus operation: # I = # J—# K

G65 H03 P#I Q#J R# K

(example)G65 H03 P#101 Q#102 R#103 ; (#101 = #102—#103)

4) Decimalism multiplication operation: # I = # J×# K

G65 H04 P#I Q#J R#K

(example)G65 H04 P#101 Q#102 R#103 ; (#101 = #102×#103)

5) Decimalism division operation: # I = # J÷# K

G65 H05 P#I Q#J R#K

(example)G65 H05 P#101 Q#102 R#103 ; (#101 = #102÷#103)

6) Binary system add operation: # I = # J.OR. # K

G65 H11 P#I Q#J R#K

(example)G65 H11 P#101 Q#102 R#103 ; (#101 = #102.OR. #103)

7) Binary system multiplication operation: # I = # J.AND. # K

G65 H12 P#I Q#J R#K

(example)G65 H12 P# 101 Q#102 R#103 ; (#101 = #102.AND.#103)

8) Binary system XOR: # I = # J.XOR. # K

G65 H13 P#I Q#J R#K

(example)G65 H13 P#101 Q#102 R#103 ; (#101 = #102.XOR. #103)

9) Decimalism sqrt : # I = $\sqrt{\#j}$

G65 H21 P#I Q#J

(example)G65 H21 P#101 Q#102; (#101 = $\sqrt{\#102}$)

10) Decimalism absolute value : # I = | # J |

G65 H22 P#I Q#J

(example)G65 H22 P#101 Q#102 ; (#101 = | #102 |)

11) Decimalism remainder: # I = # J－TRUNC(#J/#K)×# K, TRUNC : take the decimals

G65 H23 P#I Q#J R#K

(example)G65 H23 P#101 Q#102 R#103 ; (#101 = #102- TRUNC (#102/#103)×#103

12) Decimalism change to Binary system : # I = BIN (# J)

G65 H24 P#I Q#J

(example)G65 H24 P#101 Q#102 ; (#101 = BIN(#102))

13) Binary system change to decimalism : # I = BCD (# J)

G65 H25 P#I Q#J

(example)G65 H25 P#101 Q#102 ; (#101 = BCD(#102))

14) Decimal arithmetic : # I =(# I×# J)÷# K

G65 H26 P#I Q#J R# k

(example)G65 H26 P#101 Q#102 R#103 ; (#101 =(# 101×# 102)÷# 103)

15) Composite square root : # I = $\sqrt{\#i^2 + \#j^2}$

G65 H27 P#I Q#J R#K

(example)G65 H27 P#101 Q#102 R#103 ; (#101 = $\sqrt{\#102^2 + \#103^2}$)

16) sin: # I = # J•SIN(# K)(unit: °)

G65 H31 P#I Q#J R#K

(example)G65 H31 P#101 Q#102 R#103 ; (#101 = #102•SIN(#103))

17) cos: # I = # J•COS(# K)(unit: °)

G65 H32 P#I Q#J R# k

(example)G65 H32 P#101 Q#102 R#103 ; (#101 =#102•COS(#103))

18) tan: # I = # J•TAM(# K)(unit: °)

G65 H33 P#I Q#J R# K

(example)G65 H33 P#101 Q#102 R#103 ; (#101 = #102•TAM(#103))

19) cot: # I = ATAN(# J /# K)(unit: °)

G65 H34 P#I Q#J R# k

(example)G65 H34 P#101 Q#102 R#103 ; (#101 =ATAN(#102/#103))

5.7 Transfer command

1) Unconditional transfer

G65 H80 Pn ; n : running number

(example) G65 H80 P120; (jump to N120)

2) Conditional transfer 1 #J.EQ.# K (=)

G65 H81 Pn Q#J R# K ; n : running number

(example) G65 H81 P1000 Q#201 R#202 ;

If #201 equals to #202, jump to N1000 segment, otherwise,running in order.

3) Conditional transfer 2 #J.NE.# K (≠)

G65 H82 Pn Q#J R# K ; n : running number

(example) G65 H82 P1000 Q#101 R#102 ;

If #101 equals to #102, jump to N1000 segment, otherwise, running in order.

4) conditional transfer 3 #J.GT.# K (＞)

G65 H83 Pn Q#J R# K ; n : running number

(example) G65 H83 P1000 Q#101 R#102 ;

If #101 more than #102, jump to N1000 segment, otherwise,running in order.

5) conditional transfer 4 #J.LT.# K (＜)

G65 H84 Pn Q#J R# K ; n : running number

(example) G65 H84 P1000 Q#101 R#102 ;

If #101 less than #102, jump to N1000 segment, otherwise,running in order.

6) conditional transfer 5 #J.GE.# K (≥)

G65 H85 Pn Q#J R# K ; n : running number

(example) G65 H85 P1000 Q#101 R#102 ;

If #101 not less than #102, jump to N1000 segment, otherwise,running in order.

7) conditional transfer 6 #J.LE.# K (≤)

G65 H86 Pn Q#J R# K ; n : running number

(example) G65 H86 P1000 Q#101 R#102 ;

If #101 not more than #102, jump to N1000 segment, otherwise,running in order.

8) P/S alarm

(example) **G65 H99 P15**; i: alarm num plus 500.

P/S alarm 515.

Note: running number can be addicted by variable. Such as: G65 H81 P#100 Q#101 R#102; program can jump to segment addicted by #100.

5.8 Attention problems

Note 1 : address O and N can not be addicted by variable, O#200 or N#220 is not allowed;

Note 2 : if the value is more than address addicted max code value, it will not be used. Such as: if #230 equals 120, M#230 is more than the largest code value;

Note 3: -0 and +0 can not be identified. That means if #4 equals -0, X#4 is regarded as X0;

Note 4: under the effective bit values are rounded when variable is used for address data;

Note 5: formula also can be used, such as “code address[<formula>]”or“code address -[<formula>]”is program, the formula value or its negative value is as address code value;

Note 6: decimal point can be left out, such as if defines #1 as 123, #1 exactly value is 123.000;

Note 7: ‘-’should be in front of ‘#’,such as G00X-#1;

Note 8: reset will empty macro value from #1 to #33 and #100 to #199;

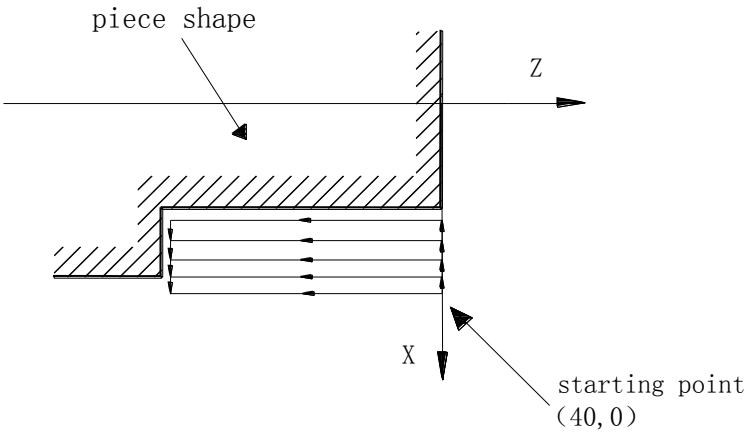
Note 9: if variable run over, variable code address will be ignored.

5.9 Macro program examples

```
O0009;
G0 X45 Z0;
M98 P0010010 I40 J0 K-20 I-20 J-5;（call program 0010 1 times, pass parameter to #01, #02, #03, #04,#05）
M30;
%

O0010;
G65 H01 P#200 Q0;
G01 X#01 Z#02 F500;
G65 H03 P#204 Q0 R#04;
G65 H03 P#205 Q0 R#05;
G65 H02 P#200 Q#200 R#05;
N0010 G01 U#200;            (the first feeding distance is -5,and -10,-15,-20 in order)
G01 W#04;                   (W cutting distance is -20）
G65 H03 P#205 Q0 R#200;
G01 U#205;                  （X retract）
G01 W#204;                  （Z retract）
G65 H02 P#200 Q#200 R#05;
G65 H86 P0010 Q#03 R#200;   （adjust return whether after reaching the setting point）
M99;
%
```

Program track:

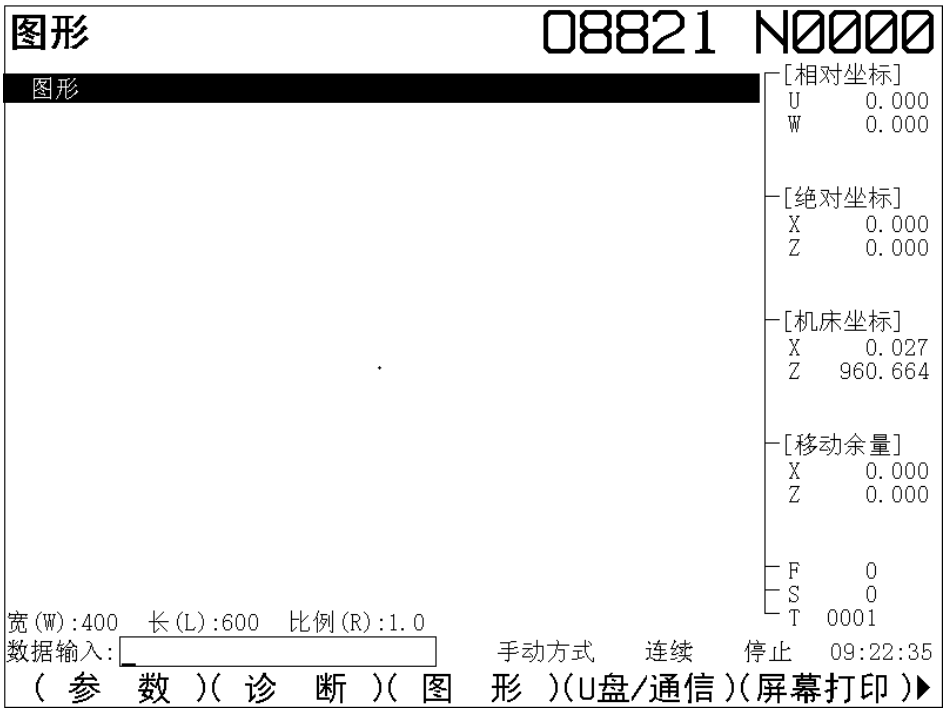


Chapter 6 Graph function

Graph analog function means displaying tool running track, which is used to check program track.

6.1 Operation description

Press GRA key, system enter graph interface, as shown below:



Before graph analog, user should set analog working table width and length, which is used to calculate setting suitable screen size.

Working table width set: press W, input size, and enter input , unit : mm

Working table length set: press L, input size, and enter input ,unit: mm

Scale set: press R, input scale value, and enter input

Cls: enter graph, press F5 clear screen.

Basic point set: default in screen center, before program running moving cursor key.

Display district and size can be adjust.

Above the screen display the current process segment.

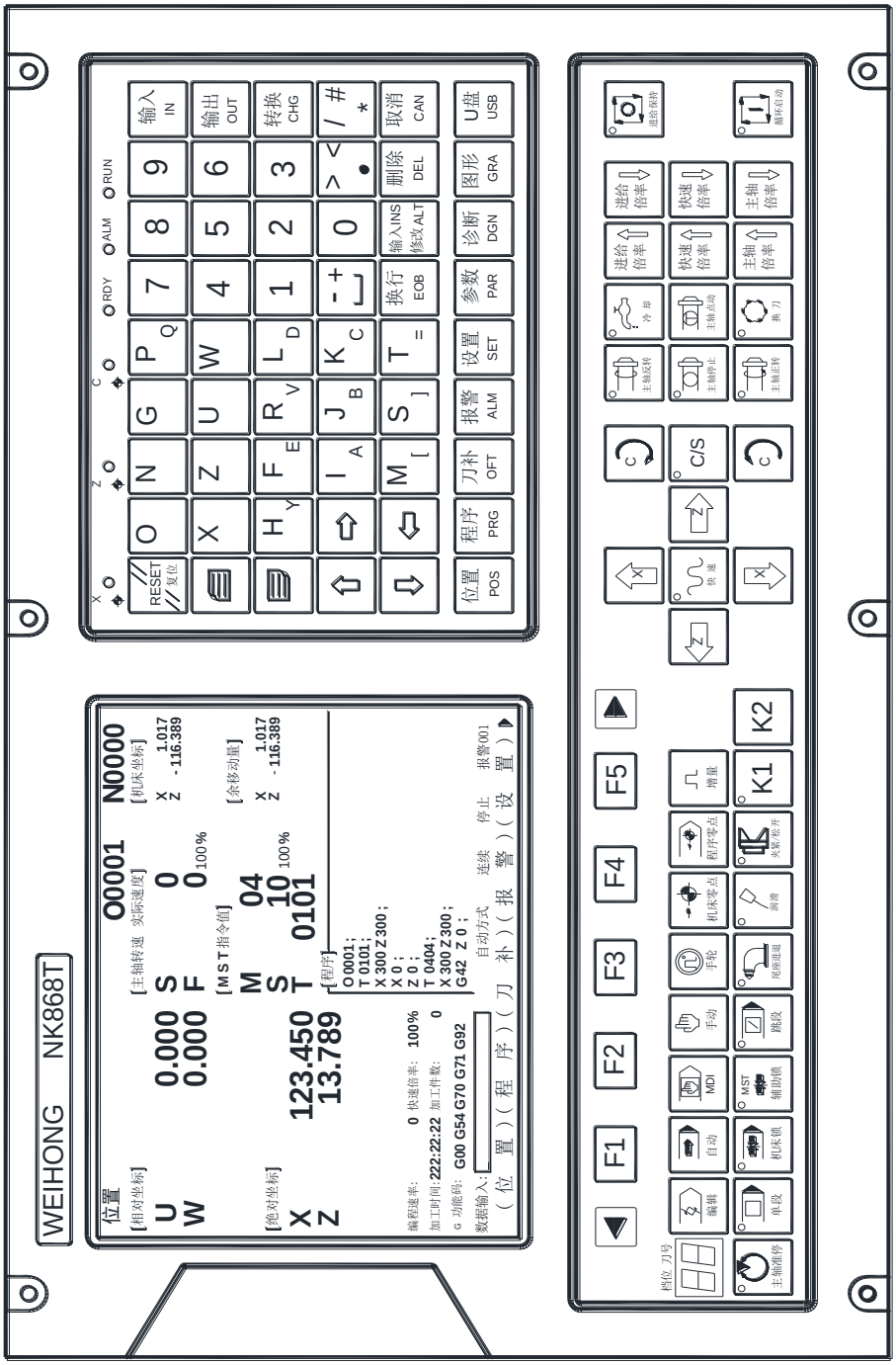
Note 1: in graph analog mode, keys of start,pause,manual,spindle and cold is effective.

- If it is necessary to adjust working table width, scale of width and length is 2:3
- Graph analog mode can only display cutting process, because of resolution ratio, it is not precise
- Parameter P010 Bit6 is suggest set 1, enter lock to lock pulse export.

Operating

Chapter 1 Operation and display

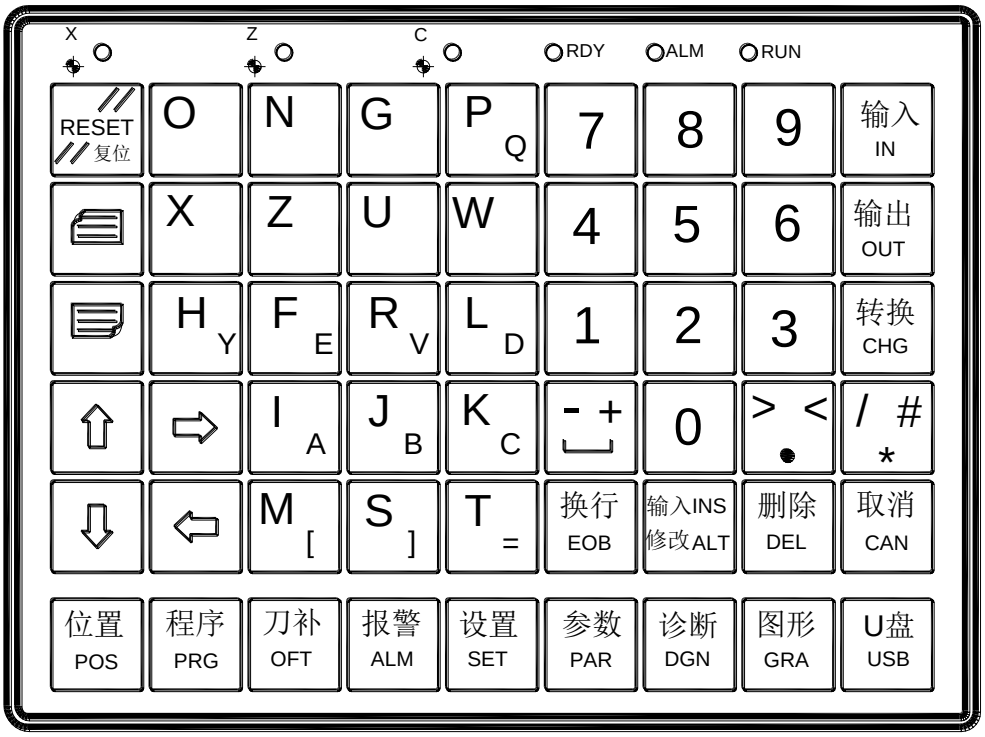
1.1 Panel

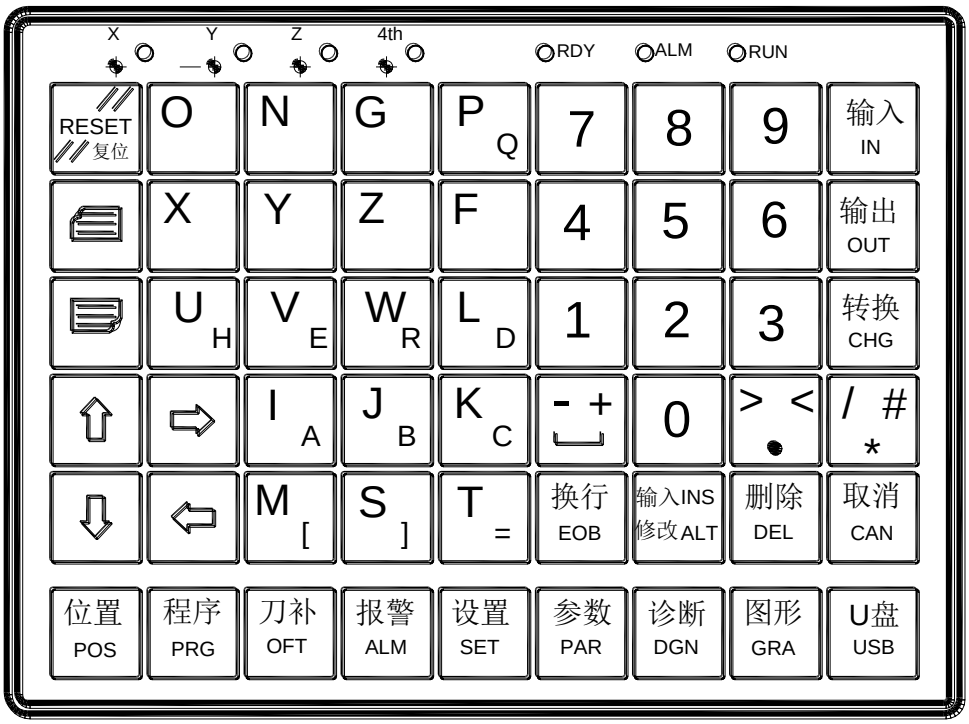


1.1.1 Keyboard introduce

These buttons can be divided into three kinds, such as for display, for input, and for operation.

Page showing key and character digital edit key figure:





1）buttons for display

Includes : POS, PRG, OFT, ALM, SET, PAR, DGN, GRH, USB.

Figure	Key name	Describe	Function
位置 POS	Position	POS	Display position, there're 4 pages: relative coordinate,absolute, comprehensive, program. You can press flip-page key or press POSto flip page.
程序 PRG	Program	PRG	Display and edit program, there're 3 pages: MDI mode, program, directory/ storage.
刀补 OFT	Tooloffset	OFT	Display setted tool offset
报警 ALM	Alarm	AM	Display alarm information and history alarm records
设置 SET	Set	SET	Display parameter switch set, program switch set,time set, password modify, memory format and so on.
参数 PAR	Parameter	PAR	Display system parameter and pitch offset parameter, 3 pages totally. You can press press PARto flip page.
诊断 DGN	Diagnose	DGN	Display input and output port state, encoder line number,pulse,analogue voltage, accumulate amount,pin number,program number.
图形 GRA	Graph	GRA	Enter figure page, and display moving trail.
U盘 USB	U disk	USB	Display U disk files, program and parameter, making importing and exporting documents easily.

2）character number edit key

Character number edit include all numbers, letters, and EOB, INS, ALT , DEL, CAN, CHG, IN, OUT, cursor, pageup,pagedown.

Figure	Key name	Describe	Function
RESET 复位	reset	RESET	CNC reset, stop feeding process, remove alarm, stop serial signal output.
输入 IN	input	IN	Confirm input data or MDI program segment.
输出 OUT	output	OUT	Input and output file from serial port or U disk.
取消 CAN	cancle	CAN	Delete character or MDI program segment.
输入INS 修改ALT	insert	INS	Insert means input word before selected word.
输入INS 修改ALT	alter	ALT	Alter means input word replace it.
删除 DEL	delete	DEL	Delete cursor current name in edit page; delete the last character in parameter page or MDI page; delete file in USB file page.
转换 CHG	change	CHG	In U disk interface, press CHGself-replication, while in edit mode, press for program copy.

Figure	Key name	Describe	Function
	Page up page down	PGUP PGDN	Edit mode or parameter interface scrolling display.
	cursor		Move cursor
O N G X Z U	address		Address input
F E R V L D I A J B K C			Double address key, repeated button, switch between the two.
- + ␣ / # *	symbol		Double address key, repeated button, switch between the two.
7 8 9 4 5 6 1 2 3 0	number		Number input
> < ●	point		The decimal input
换行 EOB	end	EOB	End mark character

Operation key:

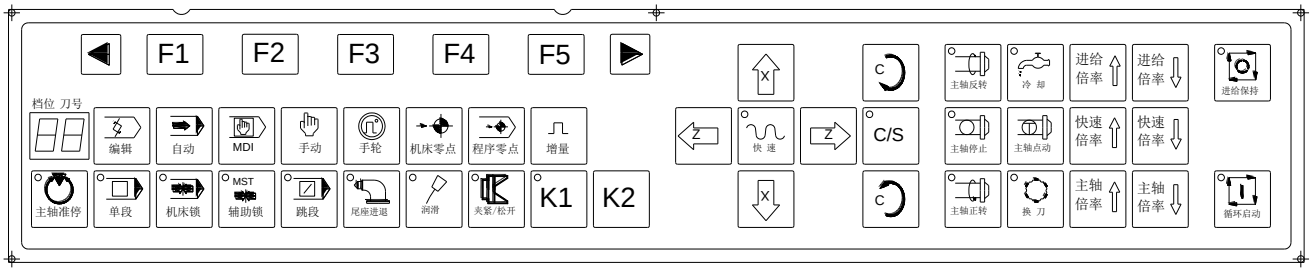


Figure	Key name	Describe	Function
自动	Auto	AUTO	Program automatic continuous operation mode.
手动	Manual	MANUAL	Manual control machine tool feed mode.
编辑	Edit	EDIT	Edit mode
MDI	MDI	MDI	MDI mode, in which to input parameters or MDI data
增量	Increase	INC	Increase switch, unit of 0.001,0.01, 0.1mm feed
手轮	Handwheel	handwheel	Handwheel feed function key.
机床零点	Machine tool zero	Machine zero	Back to machine tool zero point.
程序零点	Program zero	Program zero	Back to program zero point.

Figure	Key name	Describe	Function
主轴准停	Spindle stop	Must stop	Output spindle stop signal.
单段	Single	SINGLE	Single segment Operation mode switch.
MST 辅助锁	Assist lock	MST LOCK	Control MST output.
机床锁	Machine tool lock	LOCK	Control output system feeding axis pulse.
K1	K1	K1	Self-definition
K2	K2	K2	
循环启动	Cycle start	start	Start program
进给保持	Hold key	pause	Feeding pause
换刀	Tool change	Shift tool	Manual tool change
润滑	Lubricate	lubricate	Lubricating oil switch
冷却	Cool	Cool	cool on/off

Figure	Key name	Describe	Function
主轴反转	Spindle reversion	CCW	Spindle reversion
主轴停止	Spindle stop	stop	Spindle stop
主轴点动	Spindle crawl	Crawl	Spindle crawl on/off
夹紧/松开	Chuck control	Chuck	Chuck clamp/loosen
尾座进退	Tailstock control	Tail	Tailstock control
C	C CW	C CW	C CW
C/S	Switch position and speed	C/S	Switch position and speed
C	C CCW	C CCW	C CCW
进给倍率↑ 进给倍率↓	Feed override	FRATE↑ FRATE↓	Set automatic feed Override and manual move override
快速倍率↑ 快速倍率↓	Rapid rate	RRATE↑ RRATE↓	Set hand rapid rate
主轴倍率↑ 主轴倍率↓	Spindle override	SRATE↑ SRATE↓	Set spindle override

Figure	Key name	Describe	Function
快 速	Rapid feed	ACC	Rapid manual feeding
X X	X+/X- feed	X+ X−	Manual control X axis.
Z Z	Z+/Z- feed	Z+ Z−	Manual control Z axis.

1.1.2 State display

X ○ Z ○ C ○	Axis back to zero point indicating lamp
○ RDY	Ready lamp
○ ALM	Alarm lamp
○ RUN	Run lamp

1.2 Operating mode overview

There’re 8 operate maners like auto, edit, MDI, machine tool zero, single, handwheel, manual, program zero and so on.

Edit: set, delete, modify program.

Auto: operate program automatically.

MDI: input parameter and perform command.

Machine tool zero: in this mode, axis can back to machine tool zero.

Single: operate substep.

Handwheel: feed as selected step.

Manual: fast feeding, adjust rate, spindle control, cool, lubrite, step-move, shift tool.

Program zero: in this mode, axis can back to program zero.

1.3 Display

It has 9 pages as like position, program and so on, and each page has many sub pages. Every page and operate mode are independent. Hierarchical structure is as follows:

Interface	Page
Main menu	Position → Program → Offset → Alarm → Set → Para → Monitor → Figure → U disk → Print screen
Position	Multiple page → Relative page → Machine tool → Absolute page
Program	Program (F1) → Address value (F2) → Catalog (F3)
Tooloffset	Offset (F1) → Measure (F2) → Micro value (F3)
Parameter	Parameter set (F1) → Thread set (F2)
Monitor	In (F1) → Output (F2) → Others (F3) → Keyboard dignose (F4)
Alarm	Current alarm (F1) → Alarm recode (F2)

Interface	Page
Figure	Graph (F1)
Set	Para switch → Password set → Format → Time set → Factory Value
U disk	File (F1)

1.3.1 Position page

Press POS to enter position interface. It has four sub pages as like multiple, relative, machine tool, absolute. We can shift each page through PGUP ,PGDNorPOS.

1） Multiple page

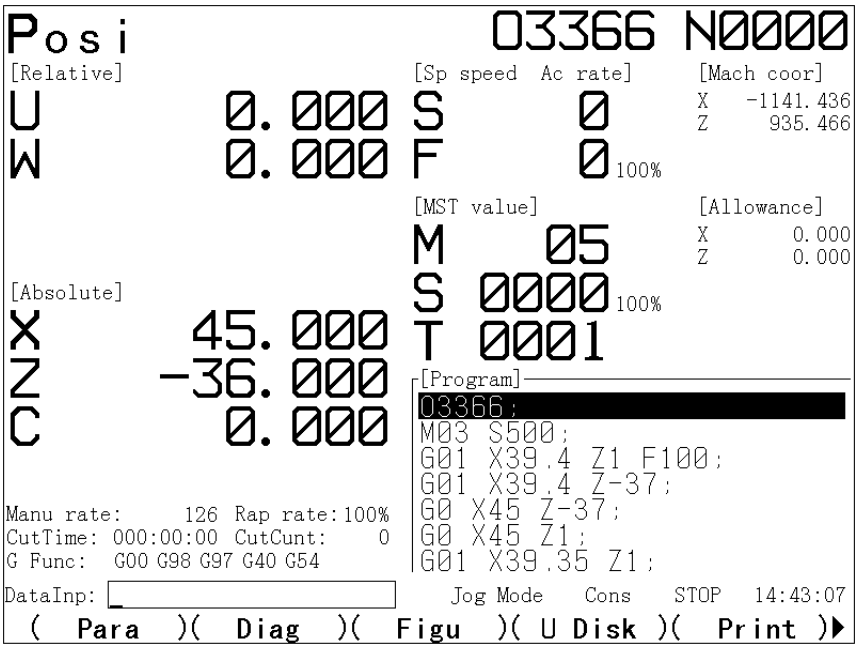
Display relative, absolute, machine tool coordinate, program and remain move amount and so on at the same time.

Machine tool show value is current coordinate in the machine tool coordinate system. It is set by returning to machine tool zero point.

Remain move amount is difference between target position and current position.

During program running, segment is refreshed dynamically, the cursor is in the current segment.

Display page:



Display feeding time and amount:

Press K, switch to accumulate time.

Press L, time flicker, and then pressCAN, time cleared.

Amount: when program carries on M30, the value plus 1.In infinite loop, when carries on M31,the value plus 1 too.

Press J, switch to accumulate amount.

Press R, amount flicker, and then press CAN amount cleared.

2） Relative page

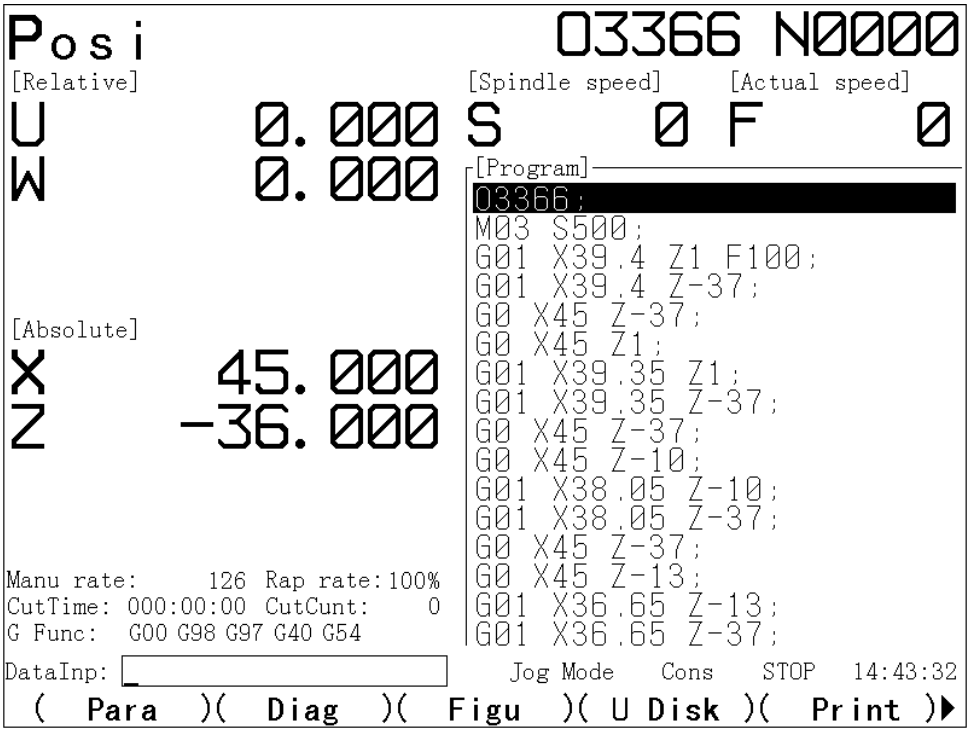
Display relative, absolute, program and so on at the same time.

Coordinate clear mechod:

Press U, U flicker, and then press CAN, U coordinate cleared;

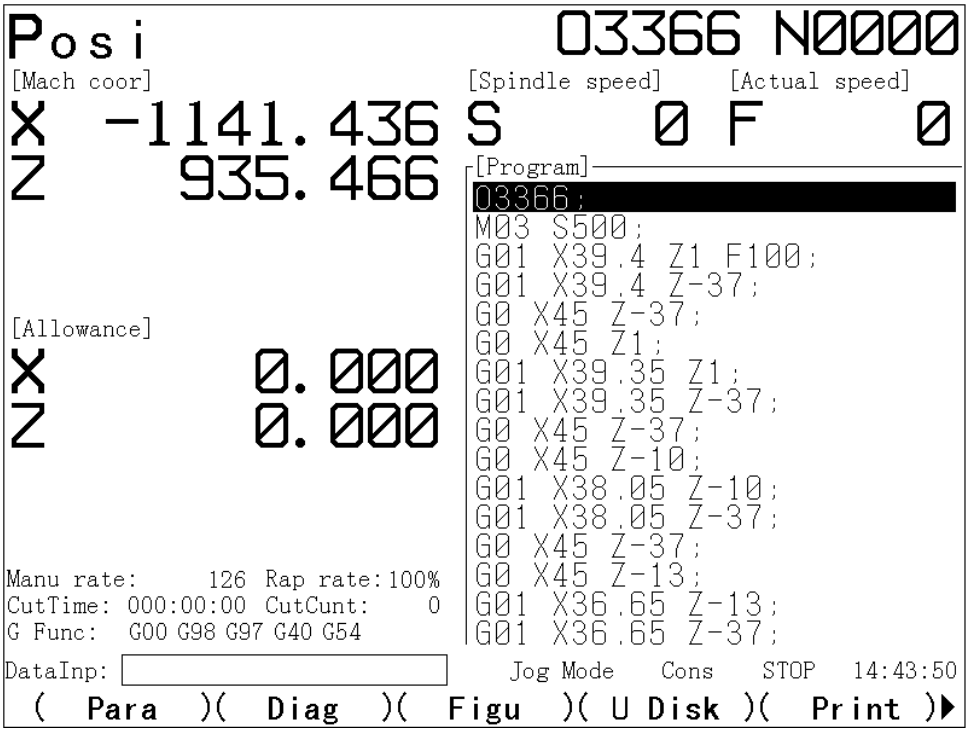
Press W, W flicker, and then press CAN, W coordinate cleared;

Note: the method is suit for Y, 4th.



3） Machine tool page

Display machine tool coordinate, remain move amount, program and so on at the same time.



4） Absolute page

Workpiece coordinate system is desided by G50.

Display mode: edit, auto, MDI: program rate

Machine tool zero, program zero, manual: manual rate

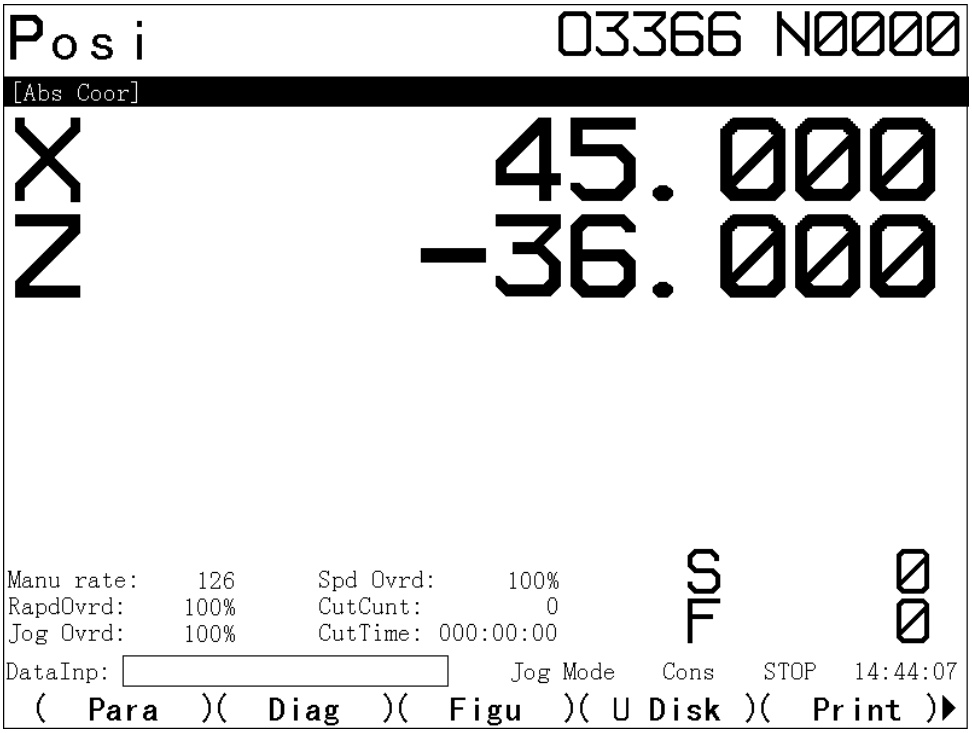
Handwheel: handwheel rate

Single step: single rate

Rapid rate: display current rapid rate.

Feed rate: display selected feed rate switch.

Spindle rate: selected by spindle rate switch.



Note1: display spindle actual speed should have position encoder on the spindle.

Note2: actual speed=program F speed×rate .

Note3: rate is invalid in thread cutting.

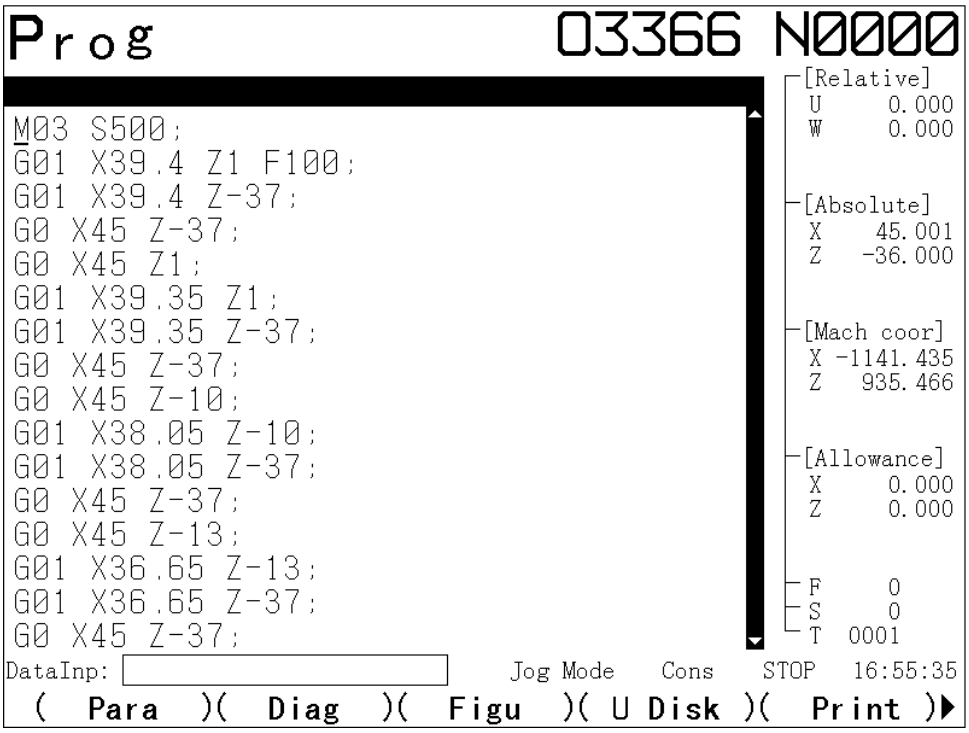
Note4: display feed rate per rotate only when command appoints.

1.3.2 Program page

Press PRG to enter program interface, it has 3 sub pages as like program, address value and list. We can shift each page though PRG.

1）program page

Display current program, in this mode, we can view program content though PGUPorPGDN



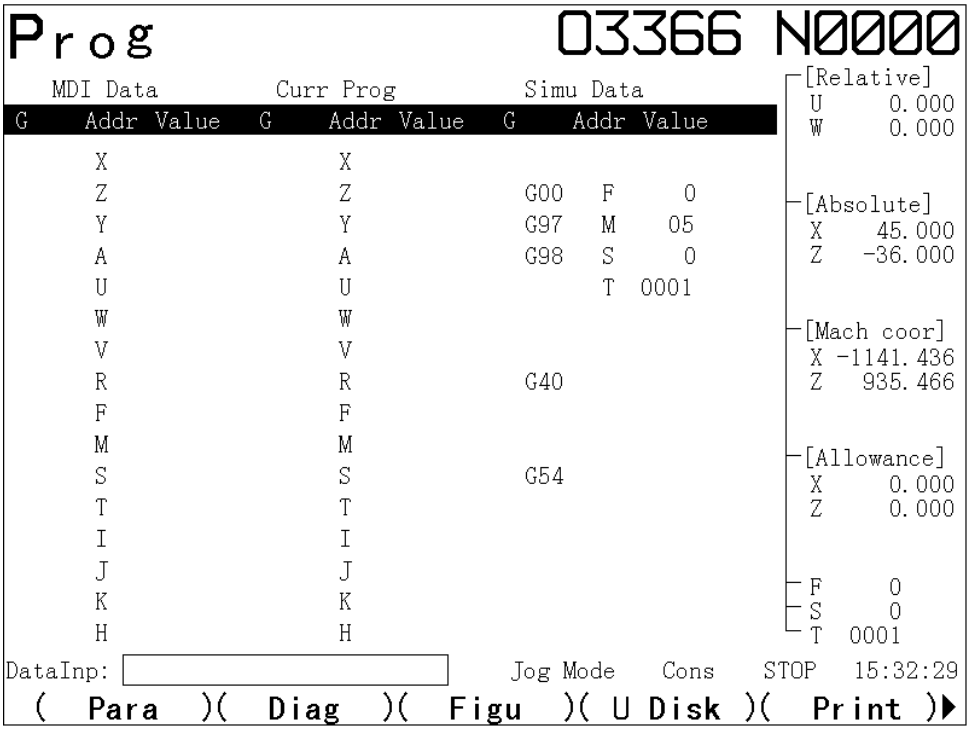
2）address value page

In MDI mode, input single segment.It is used for machine tool debug or workpiece trail cut.

Note: 1. cannot cancle modify G code

2. cancle step:

For example: cancle Z2005, method: press Z , RESET in order .

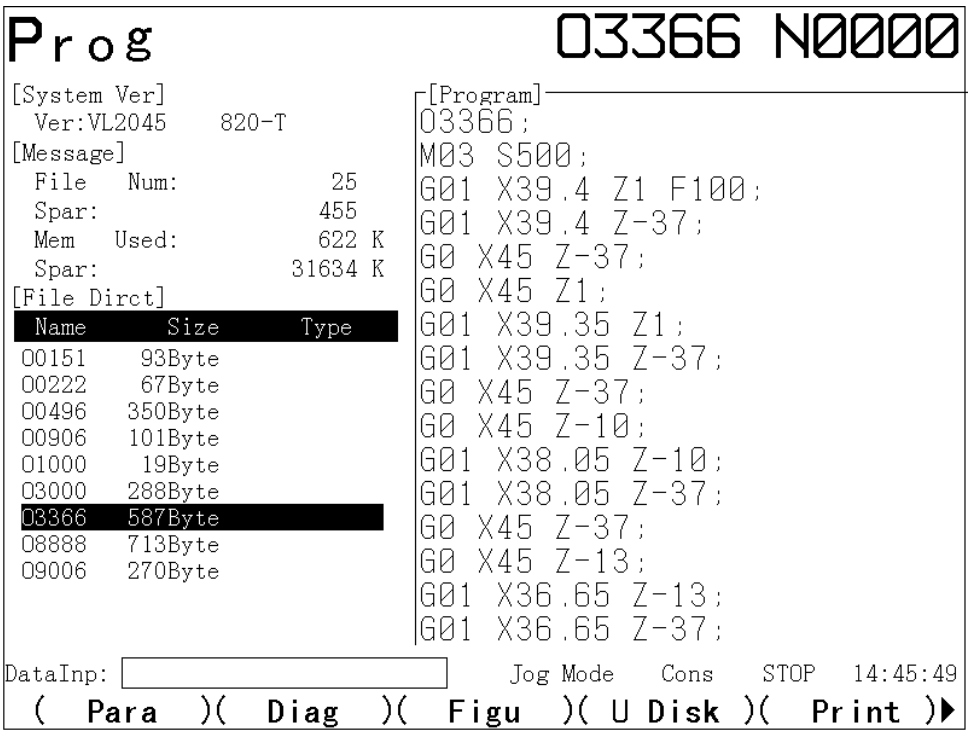


3) The directory display page

The number of the stored procedures and the storage capacity:

System standard configuration can store 480 programs.

Program memory capacity is 32M bytes,in the internal system reserve 16 Kbyte space for parameter file,tool compensation,pitch compensation files,the rest of the space for the user' program storage .



The content of the show include:

System version information: display the current version of the system using

Existing file number: display the number of documents stored in CNC (including subroutine)

Remaining file number: display the rest of file number can be stored in CNC

Existing storage: display the stored parts program in CNC take up the storage capacity

The remaining storage: display the CNC the rest available storage capacity.

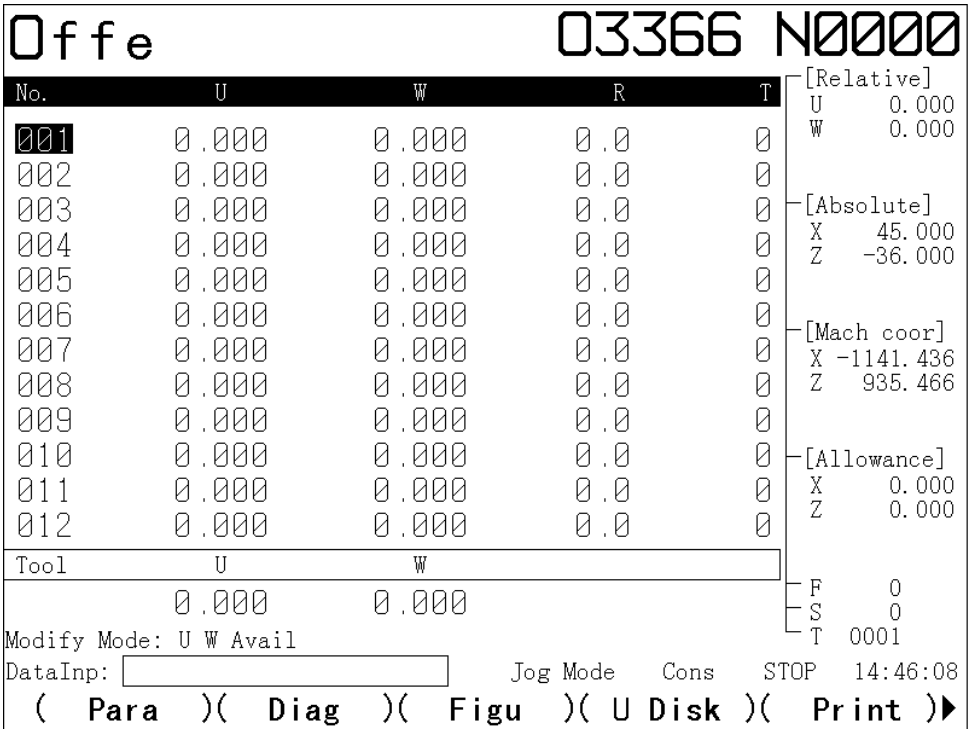
File directory: according to the size of the part program name show in turn the application number of the parts program.

1.3.3 Tool offset, macro variables page

Press the tool compensation key into the tool compensation interface,the interface has three pages, abrasion, measurement, macro variables,you can press the pageup or pagedown or tool compensation key to switch in each page.

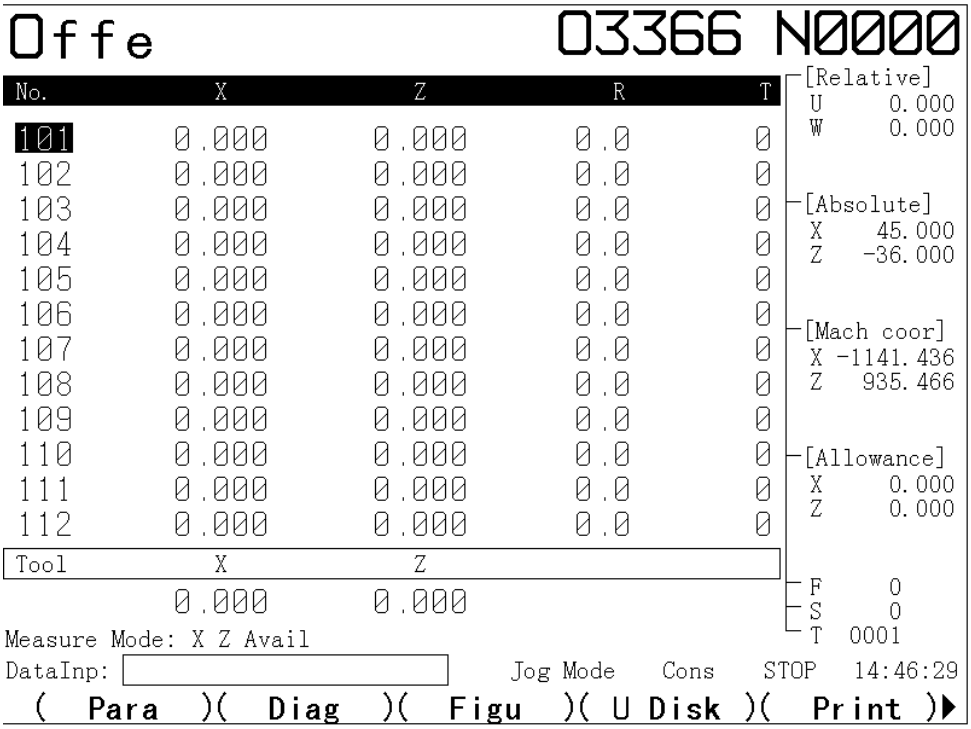
1) abrasion page display

In the actual processing find a tool process the workpieces size small or large,can use tool solder a callback function to complement the compensate.



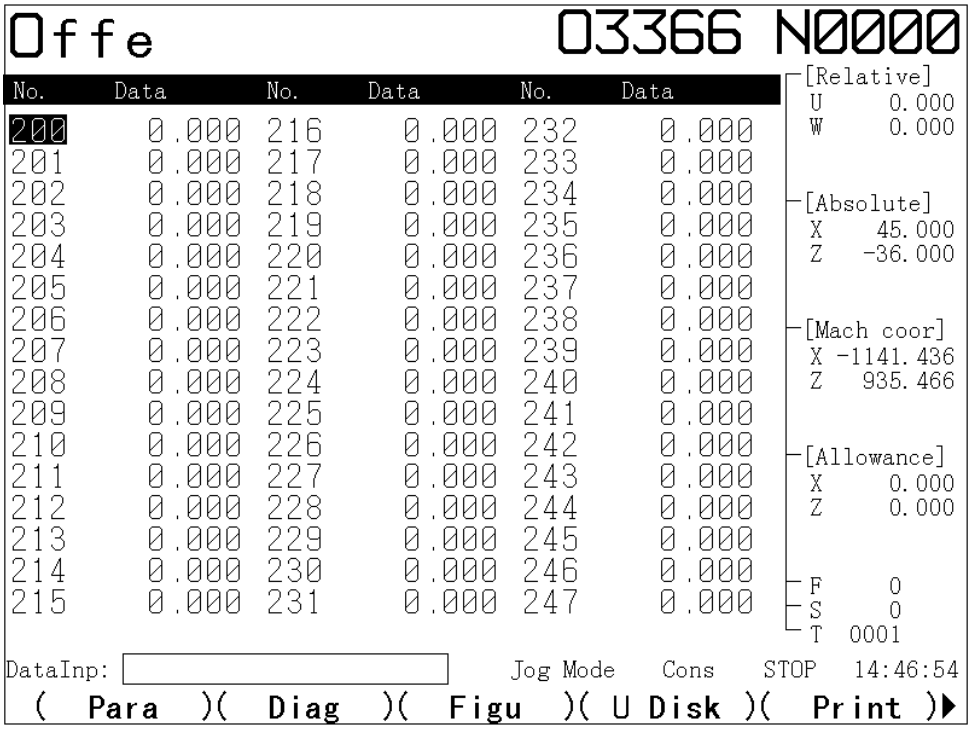
2）measurement page display

This system sets up 001 ~ 024, a total of 24 groups tool compensation values, each group contains the X axis and Z axis tool compensation data and the tip radius data , tool point phase data.



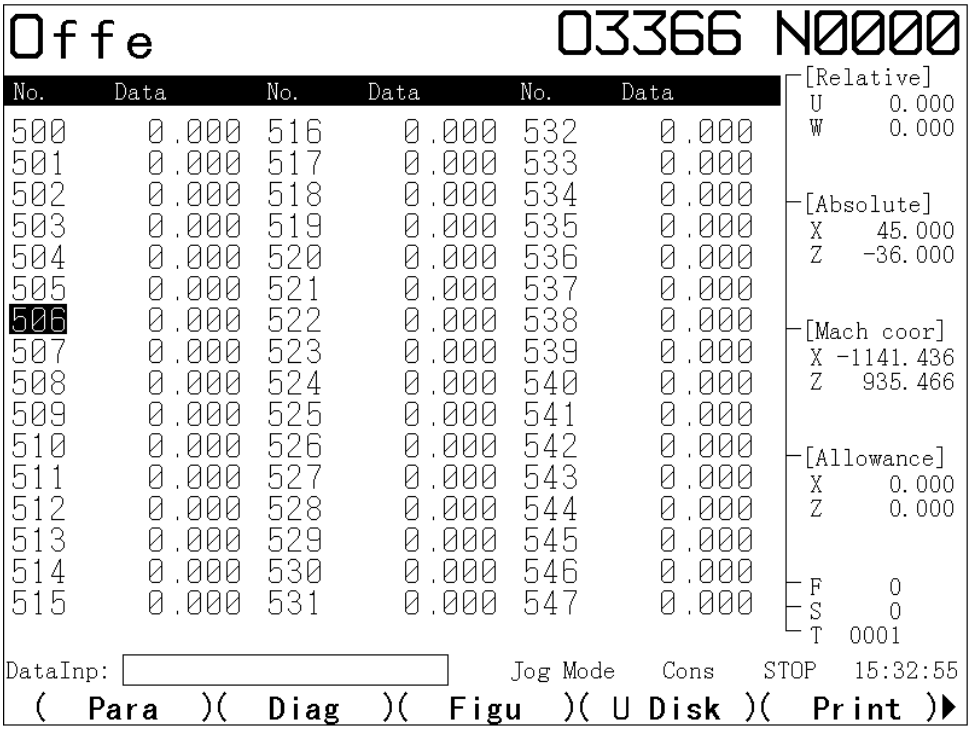
3）macro variables page 1

Display CNC macro variables, there are 240 numbers on one page, they can be addicted by codes or keyboard.



4）macro variables page 2

Variables on this page can be saved if power off.



1.3.4 Alarm page

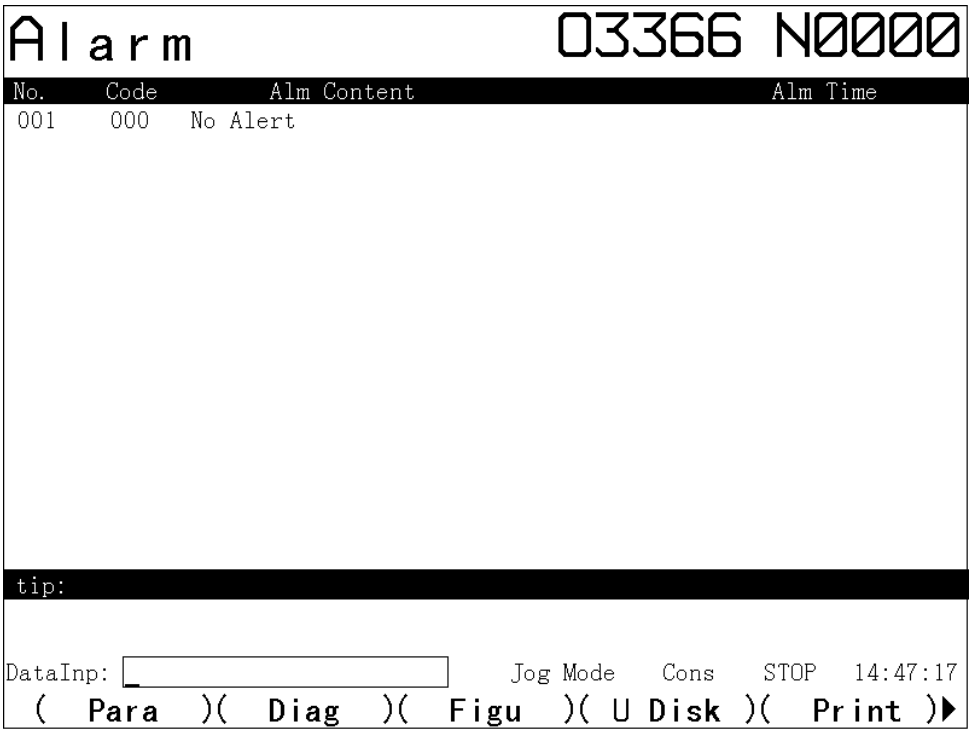
When alarm, the alarm number appears at the bottom right of screen. Press ALM key, current alarm number and content can be displayed. You may check it by pressing PGUP , PGDN or ALM key.

The meanings of alarm numbers refered to appendix 3.

In alarm screen, display the current alarm number detail content, and gives out the solution prompt, and also display history alarm recode.

Press RESET key or CAN key to cancle current alarm, but system will keep alarm until external alarm is cancled.

1）alarm page display:

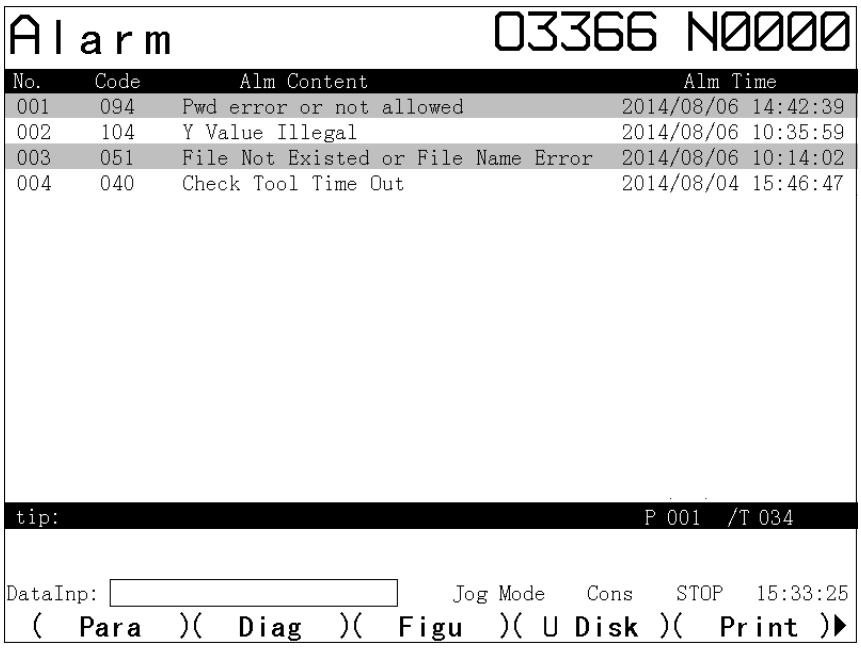


2）alarm blog:

Check alarm blog by pressing PGUPorPGDN key.

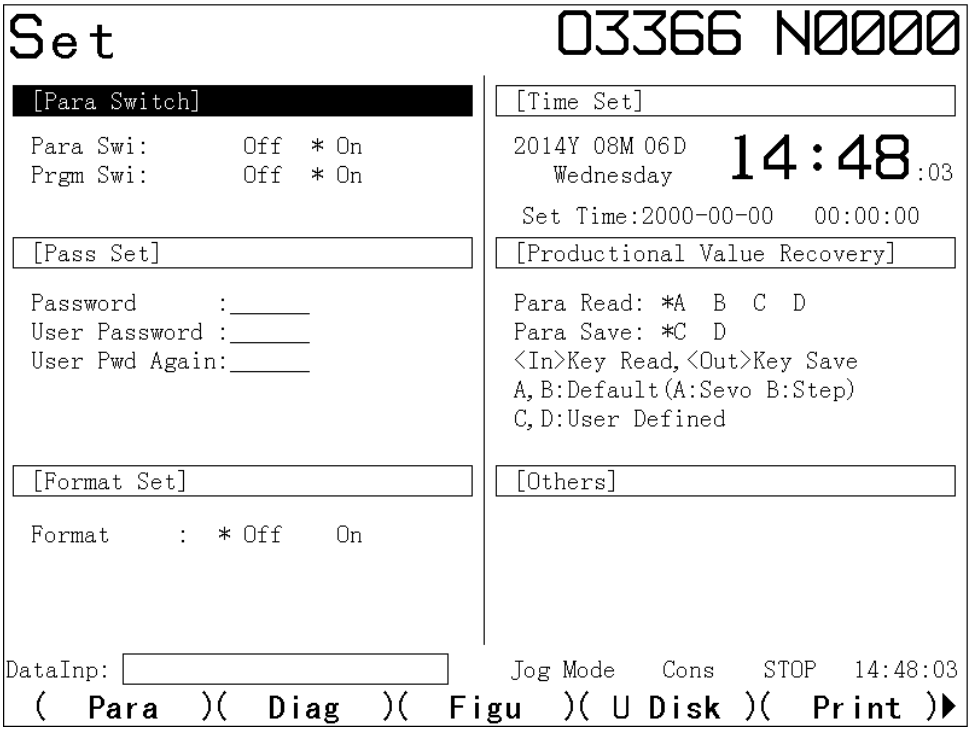
Order: the newest alarm blog message in the front of page one. The max storage number is 1000.

Delete alarm recode: press delete, input passward: 877350.



1.3.5 Set page

Set interface is divided into six parts, include parameter switch, passward set, format, time set, factory set, others. Check it by pressing PGUP /PGDN key. If the box is selected, it invert.



Parameter switch: default is on, only switch is on can input parameter, modify switch state with moving cursor.

Program switch: default is on, only switch is on can edit or copy program, modify switch state with moving cursor.

Passward set: modify passward. Only after input old passward correctly, you can input new passward.

Format: press right cursor key, switch on, press IN key, then input correct passward, press IN key, then system begins to format.

Note: after system formated, all user programs and parameter file, tool fime, thread file will be cleared. It needs user to restore.

Time set: move cursor to addicted line, press number key to set current time, format is : ××××—××—××××—××—××,that means ××××year××month××day××hour××min××second.

After that, press IN key, time take effect. If time format error, system will alarm 093.

Factory value:

System has four districts for reading parameter, such as:

A : configure servo driver parameter

B : configure stepper driver parameter

C : user self-definition

D : user self-definition

System also has two districts for storing parameter, such as:

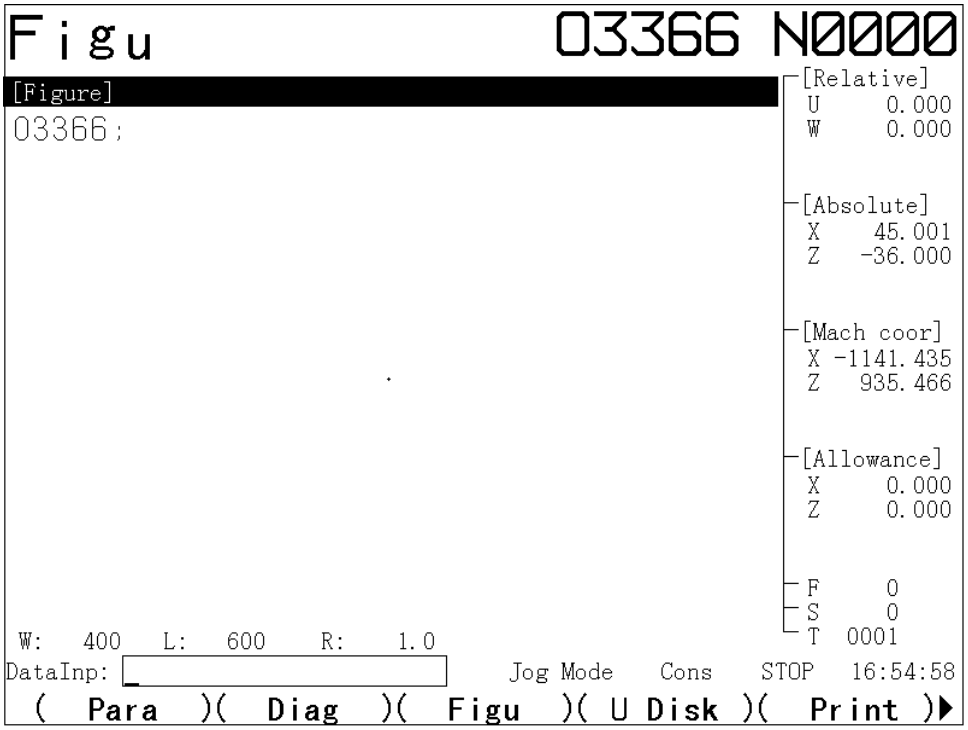
C : user self-definition

D : user self-definition

IN means read disk, OUT means store disk. Specific operation see second quarter chapter ten.

1.3.6 Graph page

Press GRH key enter graph interface, as follows:



During producing, program current command is displayed at the top left corner, and analog path is shown too.

1.3.7 Parameter page

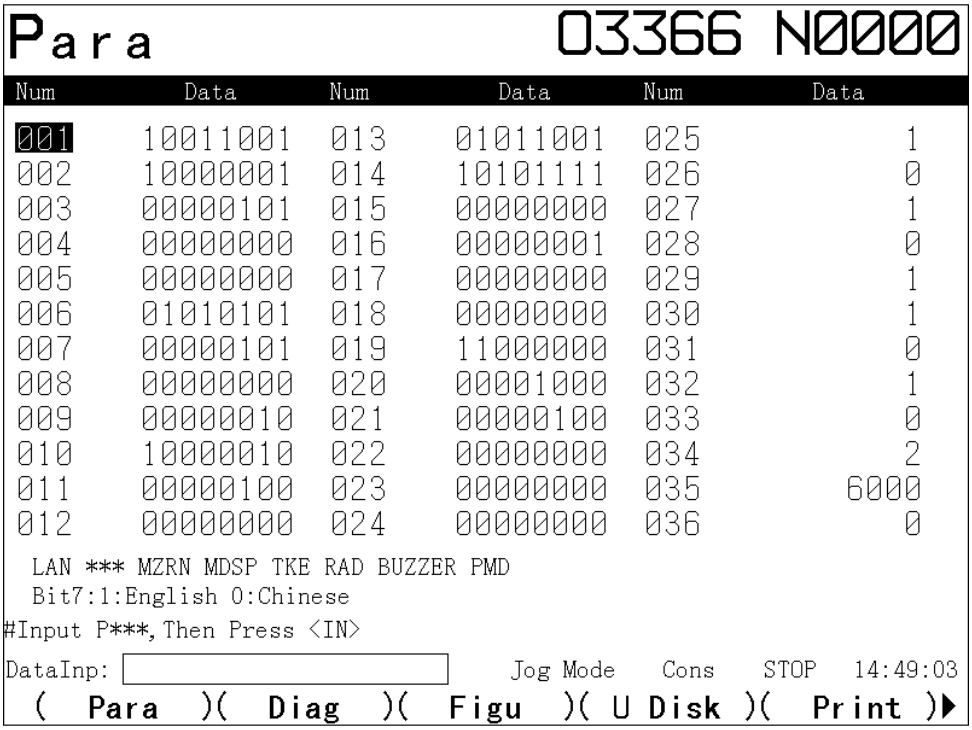
Press PAR to enter parameter page, it has 3 sub pages as like parameter set, X axis thread offset and Z axis thread offset. We can shift each page through PAR.

This system has 288 parameters, the section will introduce parameter display and set.

1）parameter display

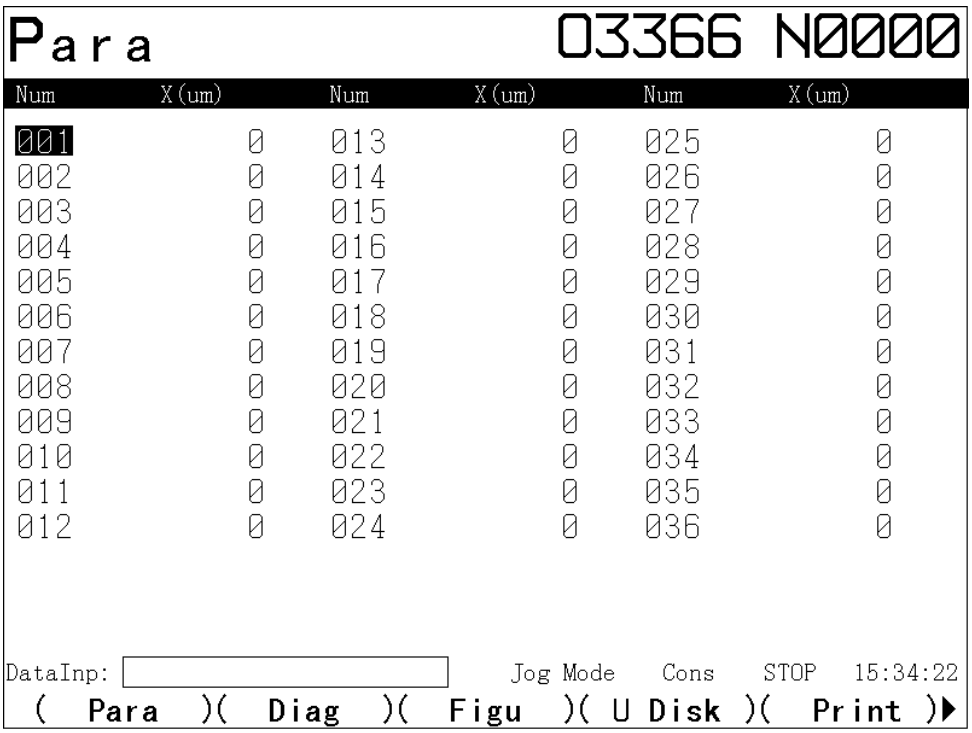
System has two kinds: bit parameter and data parameter.

See parameter list:

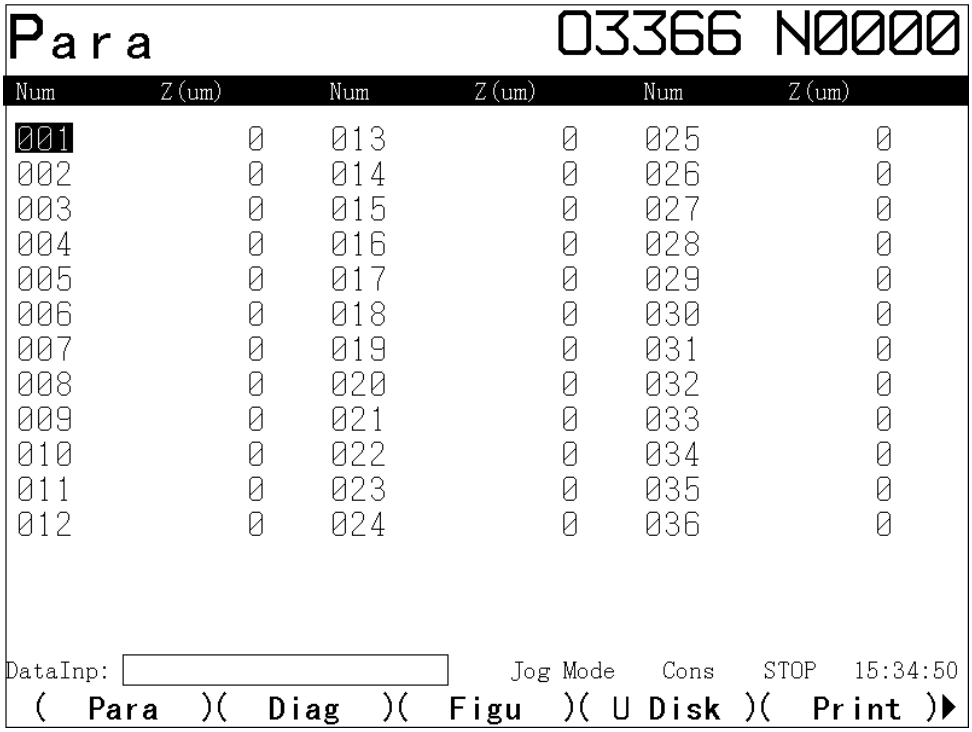


2）X axis thread offset page

Thread offset is used for mistake caused by self uniform precision. Each axis can have 256 mistake offset point.



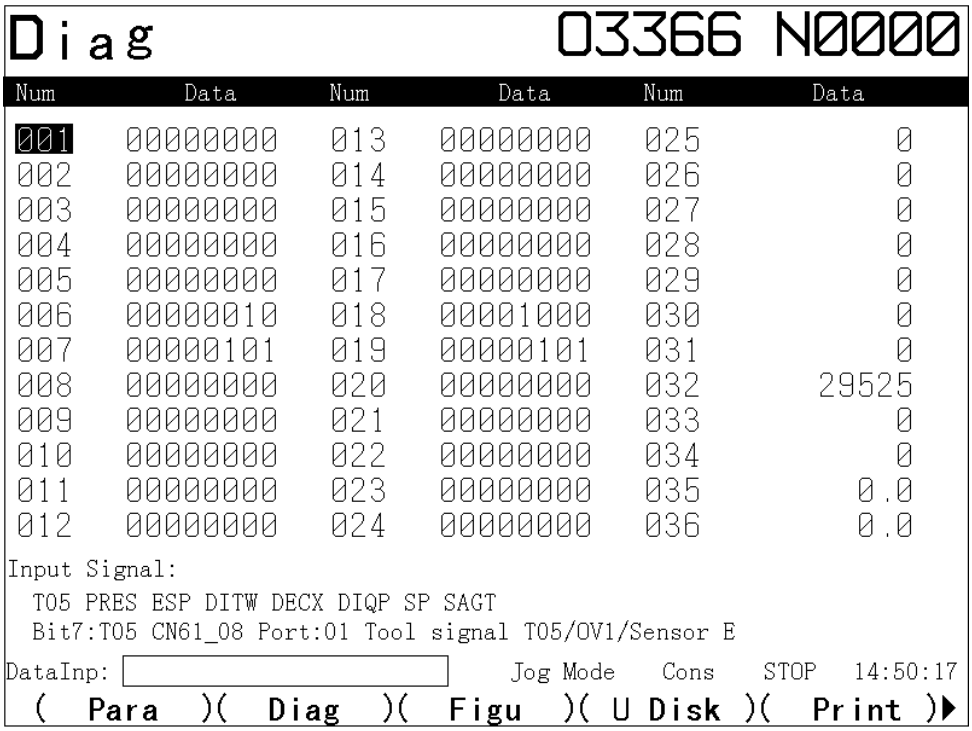
3）Z axis thread offset page



1.3.8 Diagnose page

Press DGN to enter diagnose page, it has 3 kinds as like input, output and others. Input and output ports' state can check on this page. Each port's significance is shown at the bottom of screen.

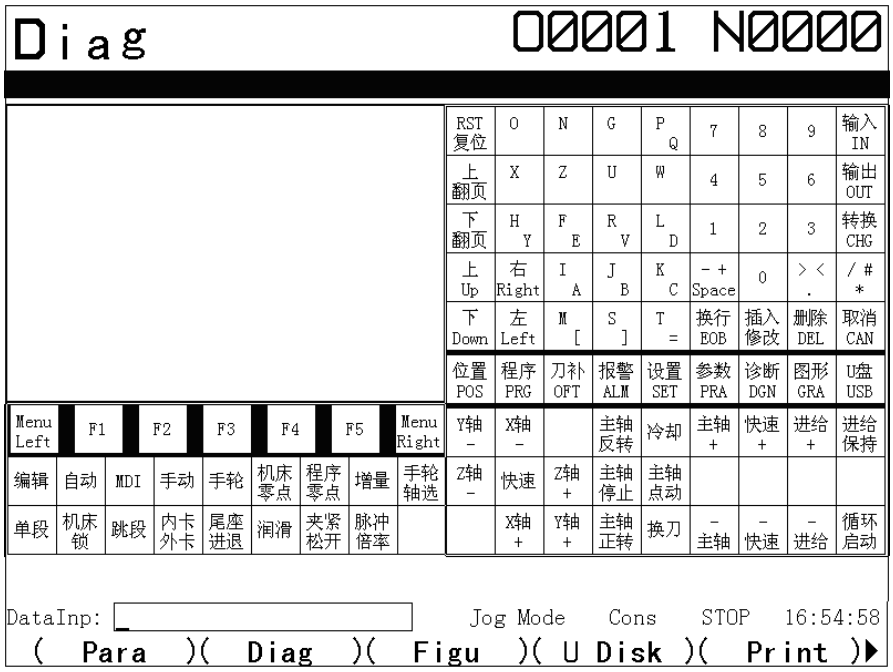
1）input, output, others



2）key diagnose page

This page can only be displayed or closed by F5.

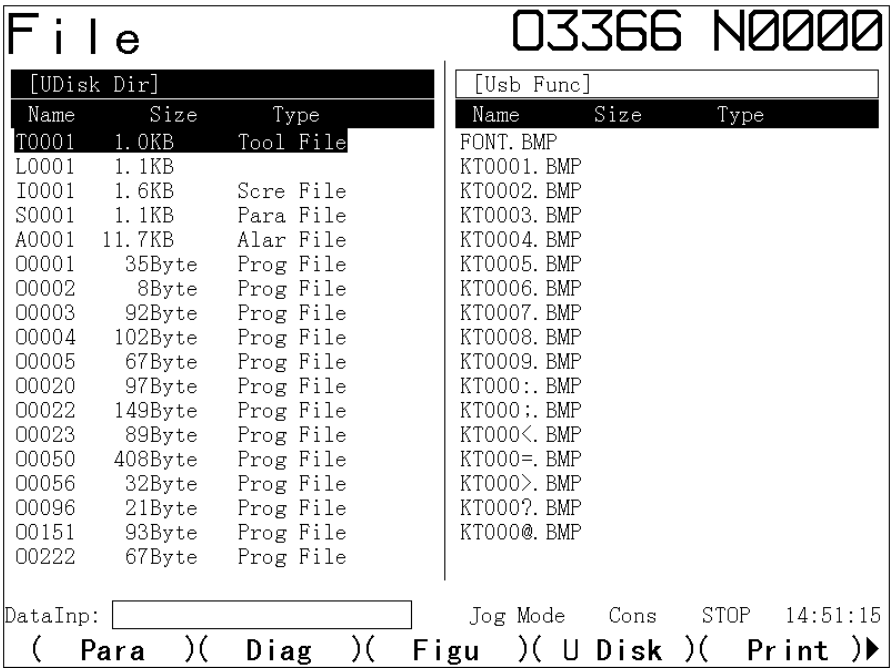
This page is correspond with system keyboard, when press down, the corresponding key will show in red. If there is no appearance, it must have problems to be solved.



1.3.9 U disk page

File manage page includes 2 kinds as like system list and U disk list.

system list includes 4 kinds as like thread offset file, parameter file, tool offset file and program file.



Chapter 2 Safe operation

2.1 Starting up

Before power on, confirm:

1. Machine tool state OK.
2. Power voltage is OK.
3. Connection correct , firm

2.2 Power off

Before power off, confirm: :

1. CNC feeding axis is stop.
2. the auxiliary function closed
3. cut off CNC power first, then cut off machine tool power.

2.3 Overrun protect

In case run out of range of X axis,Y axis, Z axis to destroy machine tool, machine tool must take overrange protect measures.

Overrange control is divided into hardware limit and software limit.

2.3.1 Hardware

Hardware limit need user fix limit switch on positive and negative limit position. Switch should be connected to system limit port. When the system detects positive and negative limit signal then decelerate to stop and alarm.

2.3.2 Software

Software overtravel limit need users according to the axis of Positive or negative limit coordinate position (coordinates) to set the corresponding parameter（P120,P125,P122,P127）. If the tool into the forbidden area (machine tool coordinate travel limit) prescribed by the parameters, the CNC display over travel alarm, the tool slow down to stop.

Specific over travel range, please refer to the instruction book which issued by machine tool manufacturer.

2.4 Emergency operation

2.4.1 Emergency

Pressing the emergency button, the machine tool will immediately stop moving, all the output ports such as the spindle, coolant, toolpost ports are all closed down. Rotate the emergency button to remove emergency state, but all output ports should be started again,at the same time the coordinate may be inconsistent with the physical location, it need to set tool again or return to machine tool zero point.

Note 1: before lifting the emergency and reboot the system, the machine tool abnormal factors should be eliminated.

2.4.2 Reset

The system output anomaly, the axis anomaly movement, pressing reset button to make it in a reset state:

1. all axes stop moving;
2. M, S function output invalid;
3. run automatically stop, modal function, maintain keep up.

2.4.3 Feed to keep

You can press feed key during the operation to suspend operation. Need to pay attention to when the thread cutting, cyclic code to run, this function does not make action immediately stop.

2.4.4 Cut off the power supply

The power supply can be immediately cut off when the machine tool running in dangerous or emergency,in case of the accident. But must pay attention,after cutting off the power supply the CNC display coordinates and the actual position may have larger deviation, must adjust the tool again, and so on.

Chapter 3 Manual

3.1 Axis mobile

In manual operation mode, can make the axis manually feed, manually fast moving.

3.1.1 Manual feed

Press the feed axis and the direction choosing key, X+ or X- can make X feed positive or negative direction,loosen the key the axis movement stop. Press Z+ or Z- key can make the Z axis to the positive or negative feed, loosen the key the axis movement stop

3.1.2 Manual fast moving

Fast moving has two modes: modal, non-modal;

If P010 Bit7 is 1, ACC key is modal;

Else if P010 Bit7 is 0, ACC key is non-modal.

In modal mode, when press ACC key, fast moving function switch from ‘open-close-open’. If function is open, the light is on, while it is off. If function is open, manual feed as a fast moving speed(set by P035,P037). In modal mode, fast feeding only need to press direct key.

In non-modal mode, fast moving function needs two key of axis direction key and ACC key. When loose ACC key system moves as speed of manual speed, press ACC key until light on, press X+ or X- key make X axis moves as fast speed, when loose it axis stop. Press Z+ key or Z- key makes Z axis moves as fast speed, when loose it axis stop.

Note1: Values of speed, time, acc and dec manner of fast moving are the same with G00.

3.1.3 Speed modify

Press RATE↑/ RATE↓ key, manual feed speed set as the following list:

Feed speed rate	Manual feed speed (mm/min)
0	0
10	2
20	3
30	5
40	7
50	12
60	20
70	32
80	50
90	72
100	126
110	220
120	320
130	460
140	790
150	1260

Current manual rate is list at the bottom-left of screen.

3.2 Other manual operation

3.2.1 The spindle rotation control

spindle negative : In manual operation mode, press this button, the spindle negative rotation;

spindle Positive : In manual operation mode, press this button, the spindle Positive rotation;

spindle stop : In manual operation mode, press this button, the spindle stop;

3.2.2 Spindle pulse control

Press Spindle point motion , the spindle is in dynamic state.

The open and close of spindle dynamic function need the main axis in the stop state. Spindle dynamic state, press the spindle Positive, move Positive point; press the spindle negative,move negative point.

3.2.3 Coolant

In any operation mode, press coolant key, cool fluid to switch between on and off. When the cool function is open, the key indicator is on.

3.2.4 Lubrication

Press lubrication key,lubrication function switch between 'on ->off ->on.' .When the lubricating oil supply, the key indicator is on. In interval lubrication mode, press lubrication key to trigger lubrication function to open,the system automatically change oil supply on and off. In continuous lubrication mode, press lubrication key to trigger lubrication function to open, system maintain oil supply on. In any interval lubrication mode or continuous lubrication mode,when the oil supply open at this time press lubrication key,the lubrication function will be closed.

3.2.5 Chuck

With any way,press clamp/loosen key, chuck will switch between the clamping and loosening.

3.2.6 Tailstock

In any way,press the tailstock key,the machine tool tailstock will switch between forward/backward.

3.2.7 Change tool manually

In manual operation mode,press tool change key,change the tool in order manually. (if the current tool is the first tool,click this button, change to the second tool; while if the current is the last,press this key,change tool to the first one).

3.2.8 Spindle rate adjustment

Spindle rate ↑: Press one time,the spindle ratio at 10% increase from the current ratio to incremental block,the spindle analog quantity will increase.

Spindle rate ↓: Press one time,the spindle ratio at 10% decrease from the current ratio to decreasing block,the spindle analog quantity will decrease.

Chapter 4 Handwheel and single step

In hand pulse and single step mode, the machine tool according to the incremental value selected to move.

4.1 Single step feed

Press the incremental key into a single step

4.1.1 Incremental choice

Press the soft keyboard according to one of the 0.001,0.01,0.1,1mm for the unit to feed.

4.1.2 Direction selection

Press X+ or X- button one time, can make the X axis to the positive or negative according to the single step increment to feed once; Press Z+ or Z- one time,can make the Z axis to the positive or negative according to the single step increment to feed once; Press Y+ or Y- button one time, can make the Y axis to the positive or negative according to the single step increment to feed once;

4.2 Handwheel feed

Press the hand wheel button to enter the handwheel feeding mode

4.2.1 Incremental choice

Press the soft keyboard according to one of 0.001, 0.01, 0.1mm for the unit to feed.

When parameter P253 is setted to 72, K2 key can operate the wheel ratio.

4.2.2 Axis and direction choice

Press the hand wheel axial key on the soft keyboard, until the axis needed to move display on the interface. The handwheel feed direction is determined by the hand wheel rotation direction. In general, the handwheel clockwise for positive feed, counterclockwise for negative feed. If sometimes the handwheel clockwise to negative feed, counterclockwise for positive feed, can exchange the handwheel end A, B signal or modify parameters P008 Bit5.

In the handwheel mode, you can press the X or Z direction key to choice the hand wheel axial function directly.

Chapter 5 MDI

The so-called MDI execution, refers to input a single segment and run of the program.when the machine tool debugging or workpiece trying to cut operation, MDI mode is more efficient operation.

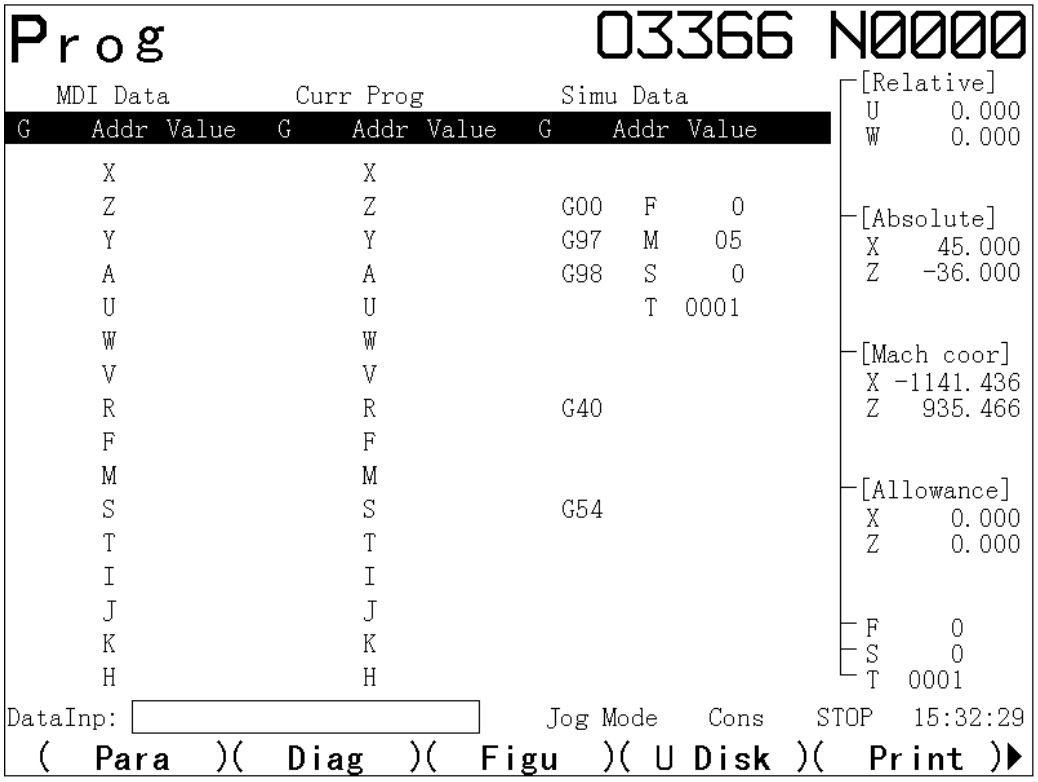
The system has two kinds of MDI input operation mode: traditional way of MDI input and quick input mode. Relative to traditional MDI input and execution methods, fast way of MDI more convenient. The following introduce two operation modes respectively.

5.1 The traditional MDI

So-called traditional way of MDI is refers to the MDI mode of operation: press the enter key to enter input mode, press the program key, then press address values key, enter MDI interface (procedures section value interface), and then in the MDI input interface, in turn to enter a procedures section, and press the cycle start button to perform input procedures section.

For example : Perform G01 X100 Z150 F100.

1) the program [address values] interface;
2) Type G01, and press the enter key. G01 input is displayed, before pressing the enter key, found that the input error, can press cancel button, then enter again G and the correct values; Can also press the cancel key, delete the value before the cursor. If find the input error, again enter the correct numerical values. Input values in [MDI data] column;



3) Similar operation input X100 ;

4) Similar operation input Z150;

5) Similar operation input F100;

6) Press the cycle start key, start the execution. After the execution of the MDI numerical display in a column [now the procedures section value].

Note 1: can't cancel the modal G code, need to input the correct data.

Note 2:before pressing the cycle start button, cancel the part operation is as follows:

In order to cancel Z150, its method is as follows: press Z and RESET button in turn.

5.2 Quick way of MDI

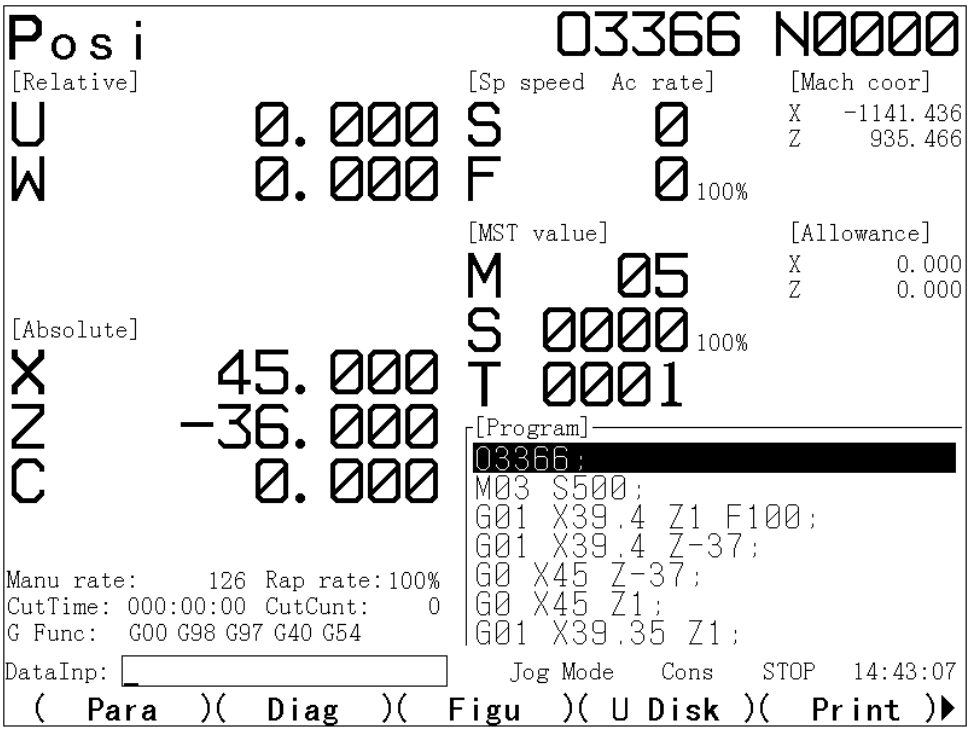
In integrated position display interface and in the nonautomatic mode,input the need to be executed code directly, and press the enter key to execute.

Such as to perform G0 X100 Z150 code:

At any position interface,in the data input box,after press G0X100Z150 key in turn, G0X100Z150 are displayed in central screen, then press the enter key, the system automatically execute this code segment.

For example if the tool is to change no.01 (electric toolpost) for no.04, as long as input T0404, press the enter key to Realize the changing tool function.

Such as control spindle forward at a speed of 1000 r/min, then input M03S1000, press the enter key to perform.



Quick MDI mode response to G, M, S, T input at the beginning of the procedures section, does not respond to other letters or Numbers at the beginning of the procedures section.

Quick MDI mode input how to modify the characters, such as to modify the input characters, can press delete button,the character before the cursor to be deleted. If you want to cancel the current whole segments of MDI input, press cancel key.

Quick MDI mode does not need to enter the program interface and switch to MDI entry mode, simplify the operation.

Note: quick way of MDI cannot perform in automatic mode.

Chapter 6 Program operation

6.1 The preparation before program storage, edit operation

6.1.1 Preparations before edit:

(1) the program switch is "open" (enter setup interface to operate).

(2) press the program key to display.

(3) press the edit key to set as edit mode.

6.1.2 Transfer data using RS232 serial communication port:

(1) Cut power off of the NC and the PC, interconnection the two with the serial communication lines.

(2) The program switch is "open".

(3) Press the program key, enter the program interface.

(4) Press the edit key, set to edit mode.

Note: 1. in order to keep the user program from being mistaken delete or edited, on the "Setting" interface is equipped with protection switch, only when the switch is "open", can only edit program.

2. the serial port communication line for 9 core straight wire, one end of needle,the other end for the hole.

6.2 The composition of the program

6.2.1 The program generally constitute

Program is composed of multiple procedures section, and procedures section is made up of words,separate each application period for over code ("; "). In the program, can't appear only a semicolon, there is no instruction format. Only when there is a semicolon line system will report error 114.

Process generally constitute by the program name (a single segment), application, end of the program instructions (general single period), program end code (single period).

6.3 The main program /subroutine

(1) The main program

The program is devided the main program and the subroutine. The CNC is usually movement in order according to the instructions of the main program, if the main program to execute commands to call subroutine, the CNC according to the subroutine movement, in a subroutine that executes instructions to return to the main program, CNC return to the main program to continue to execute. The end of the main program use M30 to end the operation of the process. If the end process has no M30, the system will alarm NO.140. after the M30 execution processing file pointer automatically return to the first.

The main program written format :

Oxxxx ; the main program name

…… ; //the main program segment

…… ;

…… ;

…… ;

…… ; // the main program segment

M30 ; the main program over

%

In CNC memory, the main program and subroutine combined storage 480 program, select one of the main program, can control of CNC machine tools according to the instructions.

(2) Subroutine

If the processing technology of some fixed action sequence and repeated, can separate them to a subroutine, and then call in the main program, so the programming become easier. Subroutines can be called in automatic mode, and the called subroutine can call another subroutine. Be called from the main program subroutine is called one baryon program, which calls for ten baryon program. Can use a subroutine calling instruction to call the same subroutine multiple repeately, can repeat calls up to 999 times.

The subroutine program written format :

Oxxxx ; the subroutine program name

…… ; // the subroutine program segment

…… ;

…… ;

…… ;

…… ; // the subroutine program segment

M99 ; the subroutine program over

%

After O write the subroutines name at the beginning of the subroutine,in the subroutine last paragraph for M99,said to the subroutine is over to return to the main program,should be a a single paragraph.

(3) Subroutine call

The subroutine called by the main program or other subroutine to be performed,subroutine call instruction format for:

M98P***####

Among them # # # # : the name of the subroutine is called

* * * : number of subroutine is called, if * * * is omitted, the default call 1 times.

Such as instruction M98 P51003, expressed as a subroutine of program called 1003 called 5 times continuously.

Note 1: M98 instructions can't with mobile instructions exist in the same logic program.

Note 2:The subroutine called by the subroutine is the same with it called in the main program.

Note 3: when can't detect the subroutine number addressed P specified , produce 120 report to the police.

Note 4: using MDI input M98 PXXXX, cannot call subroutine.

6.4 Establish a new program

There are three ways to create new program,respectively is: keyboard input,serial communication input,U disk input method.The following explain the operation method respectivelly.

1．Keyboard input

(1) Press The program key ;

(2) Press edit key to set to edit mode ;

(3) Key input address O ;

(4) Key input program, such as 0020 ;

(5) press EOB key ;

Through this operation, if the system have been input program number, system shows that the program contents; If there is no input program,the system establish this program.

In either case, then the contents of the program input pressing key, when the button exit the edit interface, the system automatically store the current program.

2. Serial communication input method

(1) Press The program key ;

(2) Press edit key to set to edit mode ;

(3) Key input address O ;

(4) Key input program, such as 0050 ;

(5) Press the enter key, the system waiting for PC transfer procedures, the bottom of the screen shows "receiving" character prompt; And then operated the PC sends procedures; The system is complete to receive, the bottom of the screen shows "receiving completely" prompt characters. When you want to cancel the transfer during transmission,you can press the RESET button to cancel the receiving.

(6) Receiving is completed, the system automatically save the received content.

3. U disk input method

Use U disk,the application of the U dish input the system, the specific operation see in operating chapter in 11.

6.5 The program name to retrieve

1. The program name input method

(1) Press program key, display program interface ;

(2) Press the edit key to select edit mode ;

(3) Press the address O ;

(4) Type to retrieve the program number, such as 1234 ;

(5) press EOB key ;

(6) If type the program number already exists, it displays the current program content, and on the upper right of the screen display program number, at the same time, system will take the current program as for wii be processed program.

2. According to the up and down cursor keys to retrieve program number method

(1) Press program key, display program interface ;

(2) Press the edit key to select edit mode ;

(3) Press the address O ;

(4) Press the cursor↓ or the cursor↑ keys, has shown the program content which deposited in, and take the current program as the will be processed program.

3. Indexes method

（1）Press the program key, then press the directory key, at the right hand page shows the current selected most of the file program ;

（2）Through the left and right cursor keys to check each file, and press the enter key to select, then this program is considered to will be processed program, and in the upper right corner shows the name of the file.

6.6 Delete and copy of the program

1）Delete a single program

1. Press program key, display program interface ;
2. Press the edit key to select edit mode ;
3. Press the address O ;
4. Type the program number ;
5. Press the delete key, then the corresponding programs number deleted from the memory.

2）Delete all programs

1. Press program key, display program interface ;
2. Press the edit key to select edit mode ;
3. Press the address O ;
4. Type -999, and press the delete key.

3）Program copy

1. Press program key, display program interface ;
2. Press the edit key to select edit mode ;
3. Press the address O, enter the name of the copied program ;
4. Press the switch key, if the copy is successful, the bottom of the screen shows "copy complete", or else display the "copy failure".

Note 1: if the input program number already exists, the system prompt "file already exist," cancel the copy.

Note 2: the cause of replication failure: storage space is full or storage file number is full .

4）Program output

1. Serial interface mode output program (output the program in the storage to the PC through the serial port):

(1) Use a serial port communication lines to connect system and PC (Don't charged operation) ;

(2) Press program key, display program interface ;

(3) Press the edit key to select edit mode ;

(4) Make the PC for input waiting state ;

(5) Press the address O, enter the name of the copied program ;

(6) After pressing the output key,the system will output the program to the PC.

Note 1: in the process of transmission, press the RESET button to cancel the output.

Note 2: PC serial port input details see appendix 4

2. U disk mode output program (the program in the storage output to the U disk) :

Can use U disk,output the system application to the U disk, the detailed operation description see chapter 11.

5）The characters insert, modify, delete, find

1. Press program key, display program interface ;

2. Press the edit key to select edit mode ;

3. Select the edited program ;

4. Set the cursor to the edited character ;

5. Character Modify, insert, delete, search,edit operation.

（1） Set the cursor on the edited characters

1. Press program key, display program interface ;

2. Press the edit key to select edit mode ;

3. Select the edited program ;

4. Set the cursor to the edited words, press the cursor keys (up and down, left and right cursor keys, flip key) move the cursor setting method :

↑ : Move the cursor up a line, if the current cursor on the number of columns is greater than the line of the total number of the up columns,after pressing the keys, the cursor moves to the last paragraph section（“;”） ;

↓ : The cursor moves down one line, if the cursor on the number of columns is greater than the line of the total number of the up columns,after pressing keys, the cursor moves to the end of the next line （“;”） ;

← : The cursor moves left a list,if the cursor at the beginning, then moves to the last paragraph of the tail ;

→ : The cursor moves to the right a column, if the cursor at the end of the line, then moves to the next paragraph of the beginning ;

Page up : page up, The cursor moves to the up page the first column first line ;

Page down : page down, The cursor moves to the next page first column first line.

（2） Insert the character

1．Set the cursor to the characters to be inserted ;

2．Press the insert/modify key,a column on the lower left corner of the page show insert and modify switching, when display inserting, enter the characters to be inserted, will display directly to be inserted in the program, the cursor indicates the character the same.

For example :

As shown in figure, if you want to insert Z50 before F600 :

Move the cursor to F directly, insert mode,enter Z50,the cursor is still on the F, press the reset button, automatically increase the space.

After modifying : Z50 F600 ;

When there are obvious mistakes in the statement, the row become red, and prompt error.

（3） Modification of the characters

1．Set the cursor to changed characters ;

2．Press the insert/modify key,on the lower left corner of the page show insert and modify switching, when display modifying, enter the characters to be modified, the original characters will be replaced, directly display in the program. The cursor automatically move to the next character.

For example :

Before modifying : N100 X100.0 Z120.0 T15 ;

Modify the T15 to M03, move the cursor to the T,in the modify state, then input M03.

After modifying : N100 X100.0 Z120.0 M03 ;

（4） Delete the characters

1．Set the cursor to the deleted character ;

2．Press the delete key, the current cursor under the characters to be deleted. The cursor automatically move to the next character.

For example :

Before deleting : N100 X100.0 Z120.0 M03 ;

Delete Z120.0, when press Del key four times :

After deleting : N100 X100.0 M03 ;

3. Press the cancel key, to delete the character which before the current cursor.

6.7 Program character search and line jump

1. Search character in program

(1) Open the program which characters will be searched

(2) Press the conversion key

(3) Input the found characters, for example M30

(4) Press ↓ key, the cursor will jump to M30

(5) Press the conversion key again to exit find/jump state

2. Jump to the specified row

(1) Open the program which characters will be searched

(2) Press the conversion key

(3) Input the jump line number, such as 115

(4) Press key input, the cursor will jump to the designated line 115

(5) Press the conversion key again to exit find/jump state

6.8 Application note

In the edit interface, input O/123, will increase annotation 123 for the program; press O/, the program annotation will be cleared.increase program annotation for easy management.

6.9 The number of stored procedures and storage capacity

System standard configuration can be stored program 480.

Program memory capacity is 32M bytes,internal system reserved parameter file,tool, pitch compensation, alarm files, use the space 16 kbyte, the rest of the space for the user.

6.9.1 Program memory information display

Use the following operation, can display the program memory usage state.

1. Press the program key, then press the directory key.

2. Memory information content contains :

(1) Exist file number : the number of program have been deposited(including subroutine and system parameter file).

Remaining: may be deposited of the program.

(2) Has been in storage: deposited the program to take up the storage capacity (unit KB; 1 KB = 1024 bytes).

Remaining: you can use the program storage capacity

(3) Program table contents: display in turn the procedures number, in order of the size to arrange automatically.

Under the directory status display page :

Press the left and right cursor keys: move the cursor between the program name, and in the right column displays the details of the program.

6.10 About the implied program and use MXX to call subroutine

6.10.1 Function definition

Function 1: implied file display function

0	0	4					CM98				

Parameter P004 Bit4 "CM98 0/1: no/open function of auxiliary function code",

When the parameter is setted to 1, open the auxiliary code function.at this time, Program O9000 - O9099 as reserves program, in the program files directory, U disk directory is not visible, program O9012 not found through pressing the EOB.

Function 2: call subroutine function

When parameter P004 Bit4 is setted to 1, the program can be through the M98 P0019012 to be obtained, also can obtain directly by M12 function. (note if executive M01 is waiting for instructions, rather than the M98 P0019001)

Function 3:when calls subroutine, not into the subroutine tracking page

When the parameter P004 Bit4 is setted to 1,use the M98 P0019012 or M12 commands into the subroutine, single step cannot enter subroutine debugging interface,when program running do not display subroutine, if something wrong happens to the subroutine, prompt alarm, the cursor displays always in the main program.

Function 4: reset and emergency custom call subroutine O9000 and O9001

When the parameter P004 Bit4 is setted to 1, if you execute reset and stop,the program will execute reset and stop at this time,and will call subroutine O9000 and O9001 to be executed. Facilitating the clients to customize various operations.

Note that this function is only in automatic state effective. when executing,the way of manual/automatic interface display will prompte to reset and stop execution.

Note: parameters P004 Bit4 Settings need to increase the password control, password level for the manufacturer.

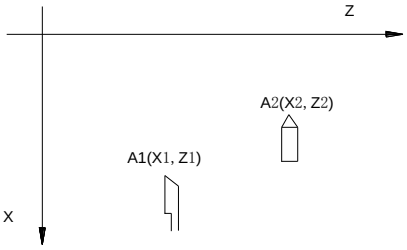
6.10.2 The operation of the encrypted file

CM98 parameters must be setted to 0, banning program encryption, the encrypted file like normal file operation, when operation is done, then set CM98 parameter to 1. At this time file into a encryption state.

Chapter 7 Tool offset and tool setting

7.1 Shift tool principle

For some complex work pieces, many tools are needed. Producing program is compiled by some one tool point, when shift tool, current tool point has offset to the formal tool point, that means even if carriage fixed, tool point position will change after shift tool, so tool offset is used for this change.



For example, the current tool is T1, its tool point position is A1; shift to tool(T2), its tool point position is A2, tool offset effect is exchanges tool point position from (X1,Z1) to (X2,Z2), the difference value can be measured in advance, the value is called tool offset value.

For example, along X Z direction, fixed point position of tool point just contacts the work piece as standard, because of each tool's length is different, the fixed point is different. CNC memories these fixed points. Coordinate values are differences between two tools. Differences between each tool's offset not a single tool offset make sense. In order to reduce memory amount, normal tool offset list value is memorized coordinate according to a basic point, and then calculates differences between the current tool to the last tool before shift tool.

7.2 Tool setting

This system adopts try cutting method to set tool offset list, establish work piece coordinate at the same. In order to operate, this system afford two ways of tool setting, it is decided by parameter P002 Bit4.

If P002 Bit4 equals 1, tool setting method is set offset value of X and Z axis at the same time. after one direction cutting, should press key and key to memory current coordinate, then retract tool, and the other direction cutting, press key and key to memory, after that, retract tool to fit position, enter tool offset list, input measure value.

If P002 Bit4 equals 0, tool setting method is set offset value of X and Z axis separate. In this method, after one direction cutting, there is no need to press X key or Z key to memory current coordinate, but before input measure value, cutting axis can not move. After one axis tool setting, the other axis can be set.

7.2.1 X/Z tool setting separate

Operating steps of tool setting separate and at the same time:

1. X direction tool setting

(1) enter manual mode, choose tool number and offset number(press shift tool key or input tool offset in MDI mode).

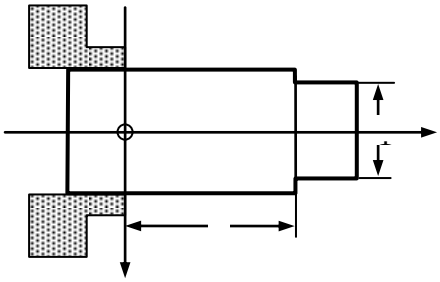
(2) start spindle, move tool rest, producing a surface on the work blank.

(3) retract tool along Z axis, don't move X axis, stop spindle, measure length L from cutting surface to basic surface, unit: mm.

(4) Press key enter tool offset page, and press key, "X Z is valid "at the bottom of screen.

(5) Press X key, input length L in step 3, and press IN key or INS key to confirm.

Note: if tool cuts in the other side, the diameter is negative.



2. Z direction tool setting

(1) enter manual mode, choose tool number and offset number(press shift tool key or input tool offset in MDI mode).

(2) start spindle, move tool rest, producing a surface on the work blank.

(3) retract tool along X axis, don't move Z axis, stop spindle, measure length L from cutting surface to basic surface,unit:mm.

(4) Press key enter tool offset page, and press key, "X Z is valid "at the bottom of scree.

(5) Press Z key, input length L in step 3, and press IN key or INS key to confirm.

Note: each tool must have the same basic surface, or else tool offset value is error.

7.2.2 X/Z tool setting at the same time

1. enter manual mode, choose set tool and tool offset.

2. start spindle, move tool rest, producing a surface by choosing tool, donnot move Z axis, pressing Z key to memory Z axis coordinate, retract X axis to fit position, producing an excricle, donnot move X axis, pressing X key to memory X axis coordinate.

3. retract tool, stop spindle, measuring cutting circle dynamic Φ and length from workpiece surface to basic surface.

4. pressing OFT key to enter tool offset page, neasure mode, “X Z available”is at the bottom of screen.

5. pressing X key, input the diameter Φ in step 3, confirm by pressing IN key and INS key, pressing Z key, input the diameter Φ in step 3, confirm by pressing IN key and INS key.

Note 1: tool parameter cursor stop the current tool offset line, measure value can be valued to the current line by pressing X key and Z key.

Note 2: if P002 Bit4 is 1, X axis and Z axis memory coordinate at the same time, if try cut has not pressed key to memory, system will alarm”Err:41”.

Note 3: if tool has not checked, system will retain the last tool offset number and alarm 042.

7.3 The setting and modification of the tool offset value

7.3.1 Tool offset

1. Press tool repairing key, and then press abrasion key, into the abrasion interface, the bottom of the screen shows "U W" effective.

2. If the cursor at the tool number is not need to modify number, can press the up and down cursor keys to select needed to modify number;

3. Fix the X axis direction, press U key,input the fixing value, and press the enter key or insert button to confirm. ;

Fix Z axis direction, press W key,input the fixing value , and press the enter key or insert button to confirm. ;

4. Within the system calculate and adjust existing tool offset value, the result of the adjustment as a new compensation shown.

(for example) has set the compensation amount 6.678

Keyboard input increment 2.5

A new set of offsets 9.178(=6.678+2.5)

To determine the tool offset value method :

1. When processing the workpiece outer diameter is larger need to input negative value,when the value is small input Positive.

2. When processing the workpiece inner diameter is larger need to input negative value,when the value is small input Positive. ;

3. When processing the workpiece Z axis is larger need to input negative value,when the value is small input Positive.

For example: if the X direction (outer diameter) is morer than 0.008 mm,after pressing the U key input deviation -0.008,press the enter key or insert button to ensure. If the Z direction length is larger than 0.015 mm, after pressing the W input deviation -0.015, press the enter key or insert button to ensure.

Note 1:when the system is automatically running at this time repair the tool offset value,the new compensation value cannot take effect immediately,must wait the specify compensation T code is executed,the new compensation value take effect.

Note 2: when in automatic operation,if they change tool number is the processing called number,prompted alarm 038,do not modify the current tool complement,only not the current called processing tool value can only be modified.

7.3.2 Toolpost offset

When the motor due to blocke, stuffy to make the coordinate overall migration will cause the workpiece coordinate system was damaged. At this point,if found the size in the X, Z direction changes, can use toolpost offset function input X, Z direction of the change value,input workpiece size as many changes, when the workpiece size increase input positive value,when the size reduce input negative value.

7.4 Tool compensation reset

Can be through reseting all the tool compensation function to reset all the tool compensation one-time.

Abrasion clear to zero operation steps:

1. Press the tool repairing button, then press abrasion key,then into the abrasion interface ;

2. Press the delete key, the system wait for password input, input the correct password, and press the enter key to ensure. ;

3. Correct password, the system will set all value to 0.

Tool offset value to clear to zero same with the above steps. reset the tool offset at the same time will remove abrasion and toolpost offset.

Chapter 8 Automatic operation

8.1 Automatic running

8.1.1 Program selection

Retrieval method :

1) Select edit or automatic operation mode ;
2) Press the program key, enter the program content display interface ;
3) Press to address key O, type the program number ;
4) Press the EOB key,the retrieved procedure displayed on the screen, if a program doesn't exist,the CNC alarm.

Scanning method :

1) Select edit or automatic operation mode ;
2) Press the program key, enter the program content display interface ;
3) Press to address key O ;
4) Press the page up or page down key,display the next or the last program ;
5) repeat steps 3 and 4),show the deposited program one by one.

Indexes method :

1) Choose the automatic operation mode ;
2) Press the directory key, enter the program directory display page ;
3) Press the left and right cursor keys to move the cursor around to will be choosed program name ;
4) Press the enter key.

8.1.2 Start

1）Press the automatic key to choose the automatic operation mode ;

2）Press the cycle start key to start the program, the program automatically running.

8.1.3 Stop

There are multiple ways of stop or suspend the automatic operation :

1. The pause command M00 M01

After executing the contains M00 program, stop working automatically, wait for the external start signal, after pressing the cycle start button, once again began to automatic operation.M01 instructions to make the program to suspend execution, waiting for the external input signal, if the effective signal is detected the program continues to run, otherwise waiting for the port signal.

2. The program over instructions M30

M30 show the end of the main program, automatic operation stop.

8.1.4 Starting from the arbitrary segment

Press the edit key to enter edit operation mode, press the page up or page down key to select program content pages :

1) Move the cursor to the segment that is ready to start running ;
2) such as the current cursor at the modal of the procedures section (G, M, T, F code)default, and does not agree to run the program mode, must perform the function of the corresponding modal then can continue to the next step ;
3) Press the automatic key into the automatic operation mode, press the cycle start key to start the program.

8.1.5 Feed keeping

In automatic operation,press the feed keeping key on the panel can make the operation automatically stop temporarily. After pressing the feed maintain button, the machine tool is in the following form :

1) When the machine tool moving, feed slow to stop.
2) After executing the M, S, T, stop.
3) After pressing the cycle start key, program continue to be performed.
4) After pressing the reset key the program execution over.

8.1.6 Feed, fast speed adjustment

When run automatically,can through adjusting the feed and fast moving ratio to change the speed, without needing to change the program and the speed value setted by the parameters.

The adjustment of the feed rate :

Press the feed rate ↑ and feed rate ↓ key,which can realize feed ratio 0 ~ 150% 16 level real-time adjustment.

Note 1: feed ratio adjustment the value F which specified in the program ;

Note 2: the actual feed speed = F specified value x feed ratio.

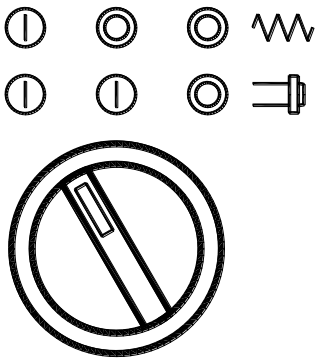
Quick ratio adjustment :

Press the quick ratio ↑ and quick ratio ↓ key,can realize rapid rate Fo, 50%, 75%, 100% four gears.The speed of Fo are setted in the parameter P053.

8.1.7 The spindle speed adjustment

Press the spindle rate ↑ and spindle rate ↓ key,can realize the spindle ratio 8 magnitude real-time adjustment of 50% ~ 120%.

8.1.8 Three switch functions



The three switch divided into the left, middle and right three position state, when the position on the left side dialed to the right position, again by the right position to the left when the control order :

Left → middle → right → middle → left

normal operation——Feed suspended——spindle suspend——spindle start——Feed start, normal operation

The parameter P013 Bit0 control whether open the three switch functions :

＝0 : Open the three switch functions

＝1 : Close the three switch functions, the system does not detect the three switch

When the three switch function open, the system in automatic mode, press the cycle start key to operate the processing program, system detect the three switch initial position, If not in the left position (normal operation),the CNC prompt alarm 036, needs to place it in the left position to start running.

8.2 Running state

8.2.1 Single segment

When execute a program for the first time, to prevent programming errors to occur accident, can choose the single period of operation.

Press the single segment key, the system switch between the single run and the continuous operation mode.

When the parameter P016 Bit4 is 0, when the system is in a state of single segment, then again press the single segment key switch for continuous mode, "continuous" display in the right below screen, the single procedures section lights off, the system run to the program continuously.

When the parameters P016 Bit4 is 1, the system is in state of single segment, need to press the single segment key, switch for continuous mode, "continuous" display in right below of the screen, the single procedures section lights off, then again press the cycle start,the system run to the current program.

8.2.2 Machine tool lock

Machine tool lock button is used to control the machine tool spindle feed pulse switch; When press the machine tool lock the machine tool lock switch is "open" (machine tool lock indicator is on), the machine tool spindle does not move, but the location coordinates display the same with the machine tool movement, and the M, S, T functions can be performed.

Related parameters :

0	1	0		MLCK					

MLCK : =1 : Open the machine tool lock function =0 : Don't open

Every press this button, the machine tool lock switch cut between "on-> off -> on. ', when for the' open ',the machine tool lock indicator is on, when off the light is out. When the machine tool lock is' open ', the system automatically memory the current coordinates and tool number, after that, whether the system coordinate values change or not, when the machine tool lock is "off", the system automatically restore previous memory coordinates and tool number,to ensure that the workpiece coordinate system and the machine tool position unchanged.

Machine tool lock function is used for routine checked,before checking the new program,open the lock function of the machine tool,and then automatically run the will be checked processing program,wait for being checked then close the machine tool lock function,the system automatically restore the before checking state.

Note: due to the function T can be executed when the machine tool is in lock state,so after the routine checking the tool number and the tool compensation number may be different with before. Closing machine tool lock function only recover tool compensation number,tool number not recover,so after the routine checking, before performing other cutting operation need to determine wheter the cutting tool number is correct or not.

8.2.3 Jump Period

In the program have don't want to perform a certain period of the program and don't want to delete it, can choose the jump function. When at the beginning of the program paragraphs with a "/" and the key jump section of the program segment switch opens, when automatically run this program period can be skipped for not running.

Chapter 9 Back to zero operation

9.1 The program back to zero

9.1.1 The program zero point

The so-called program zero, refers to the process of starting coordinates point. When the system back to the machine tool zero point after being powered on,when the processing starting location coordinates and the machine tool zero point position is not the same one, in order to simplify the operation for quick returning to the processing starting point position, this system provides the function of return to the program starting position, also be called return to program zero point function.

9.1.2 Back to zero step

If the processing tool point position at point A, the coordinates in the workpiece coordinate system (100,200), before each caller carrying on the processing, no matter what is currently coordinates, all need to be located to point A, and then start the program for processing.

You can use the following way to locate to point A rapidly :

1. Press the position key, and switch to the absolute position display or relative position display. ;

2. Press the program zero point key,the "program back to zero" is showed at the bottom right corner of the LCD screen, and in the central screen display G26 X Z

3. Press the enter key, the system performs the program to zero point, each axis back to program zero speed are setted by the parameter P035, P037.

4. After completing back to the program zero,the back to zero light on the panel is on.

The machining coordinates of the starting point is that point coordinates in the workpiece coordinate system,which defined by the parameter P111 and P113, therefore, before running back to the zero point program should first determine the workpiece coordinate system has been setted up correctly,otherwise the program will appear can not correctly back to the starting point of processing physical location or will have a accident.Back to the program zero point can also be done after backing to the machine tool zero point,but also need to determine whether the right workpiece coordinate system has been established when after backing to the machine tool zero point.

9.2 The machine tool back to zero

9.2.1 The machine tool zero point

The machine tool coordinate is the CNC calculate standard coordinate system,which is inherent in the machine tool coordinate system,the machine tool coordinate system origin is called the machine tool zero point,the machine tool zero poit is determined by the zero switch or back to zero switch installed in the machine tool,usually the zero switch or back to zero switch is installed in the positive direction of X axis and Z axis the biggest travel.

9.2.2 The back to zero step

1. Press the machine tool zero mode key,the "machine tool back to zero" displays at the bottom right corner of the LCD screen.
2. Press the X axis and Z axis key, until the back to zero indicator is on,complete the back to zero.

Before executing to back to the machine tool zero point operation, the user need to know complete the principle and parameter Settings of the back to the machine tool zero point,lest cause an accident or is unable to correctly back to zero.In section 3.5.2 introduce the back to zero correlate parameter function, this section introduce the backing to zero action process.

There are three ways A,B and C to back to mechanical zero point. The each axis back to zero setted by the parameter P005 Bit4 and parameter P006 Bit4, Bit5, Bit6. the way C is a single slow down speed switch at the same time used as the deceleration signal and back to the zero reference point. Method B is deceleration switch signal is only used to slow down and rough positioning, rely on the detection of servo encoder zero pulse (Zero pulse) signals as the precise positioning signal. The Way A for backing to float zero point way, requires the user to set the float zero point position.in the way B and C back to zero process, need to press the direction key always until back to zero over. For the convenience of the user operation, increase the set parameters 11,3, when the bit is 1, the machine tool back to 0 only need to press the direction key one times; When the bit is 0,back to the mechanical zero point need to hold the direction key always.

Way B back to zero movement process :

Press the X axis and Z axis direction key,the movement of the machine tool based on the system back to the zero setting direction (setted by the parameter P005 Bit0, Bit1, Bit2), has arrived at the machine tool zero can loosen the direction key. Before reaching the deceleration point,the machine tool fast moving (by the parameter P101 to set the rapid rate back to zero), After touching the deceleration switch,the machine tool slow down to stop, and at the speed of FL (setted by the parameter P100) left reference point move backward, when after leaving the deceleration switch continue to move until detecte the Zero pulse signal of the axis, (here if a period of time can not detecte zero signal, the system will prompt alarm, can't detect signal) run a short distance forward,then slow down to stop,and then back again with a small speed which configured within the system to run, when detect the axis zero pulse signal then slow to stop, complete to return to the mechanical zero, back to zero indicator is on.

Way C back to zero movement process :

Press the X axis and Z axis direction key,the machine tool move to the direction setted by the system back to zero (by the parameter P005 Bit0, Bit1, Bit2 to be setted),has arrived at the machine tool zero can loosen the direction key. Before reaching the deceleration point,the machine tool fast moving(by the parameter P101 to set back to zero rapid rate), after touching the deceleration switch,the machine tool speed slow down to stop, and at the speed of FL (setted by the parameter P100) reverse move to leave the reference point, After leaving the deceleration switch for walking a short distance then slow down to stop, and then back again with the very small speed setted by the internal system close to decelerate point, when detecte signal then stop,complete returnning to the mechanical zero,back to zero indicator is on.

Way A back to zero movement process :

Before backing to zero in the way A need to determine to establishe the floating zero beforehand. The process of the float zero point establishment: move each axis to the need to be setted location,press the position button to enter the comprehensive coordinate display,after pressing the key X and Z,the machine tool coordinate corresponding coordinate character twinkling,at this time press the cancel key,the corresponding machine tool coordinates are setted to 0,which is the float zero, way A back to zero is that return to the machine tool coordinate zero position. After setting the float zero point properly,the system automatically memory this zero point,as long as do not reset the float zero point,then way A back to zero all back to that point.

Under the condition of existing float zero point,press the X axis and Z axis one time, the system automatically move to float zero position, after reaching the zero point,the back to zero indicator light is lit, the axis back to zero accomplish.

Note:before using the way A to returne to zero,it is necessary to determine whether has set a float zero point correctly,otherwise may happen that the axis movement location is not correct,or even cause an accident.

The related parameters significance and matters which needed attention of the returnig to the machine tool zero point.

1．Whether open the function of backing to zero is decided by the parameter P005 Bit4 :

＝0 : Way B and C back to zero is effective ;

＝1 : Way A back to zero is effective ;

2．By the parameter P006 Bit0, Bit1, Bit2 to set the axis which needed to return to zero

＝0 : The axis back to zero function close ;

＝1 : The axis back to zero function open ;

Not install zero signal or do not need to return to zero axis, need set the corresponding bit to be 1,to shut down the axis back to zero function,for fear that when return to zero can't find the zero signal then happen accidents.

3．Set the mechanical zero point direction by the parameter P005 Bit0, Bit1, Bit2

Before backing to zero need to determine whether the parameters define the direction and the actual zero point direction are consistent.

4．At the end of backing to the machine tool zero point,the back to zero indicator is lit,lights go out in the following condition

A. Move out from the mechanical zero point(manually operation mode)

B. Press the emergency switch

5．After the completion of backing to zero the absolute coordinates value by the parameter P004 Bit5 to decide whether to set the coordinates.When the bit is 1,the parameter P106, P107 and P108 to set the absolute coordinates;When the bit is zero,the absolute coordinates are setted to 0.

Note: due to return to zero process the control conditions are more,therefore,suggest the user completely understand the principle of backing to zero and the parameters setting,lest cause accident or not right back to zero.

Chapter 10 The data of setting, backups and recovery

10.1 Settings

10.1.1 Switch

Press the set key,in the parameter switch options,display the parameters switch and the program switch state, Move the upward or the downward cursor to the need to setting project.Through the left or the right to modify the switch state. Only if the parameters switch is open can modify the parameters, the program switch is open can edit, copy, delete the program and so on.

10.1.2 Time

Press the set key, in the time settings, display the moment time which is date, month, year, week, second, minute, hour respectively, which can set the time through the time settings.

10.1.3 Password

Press the set key, in the password settings, in this page: input your old password firstly, and then enter the new password twice, which can complete the password changes.

The system initial password is 187350.

10.2 The data recovery and backups

Restore the parameters to the factory

move the flashing cursor to the reading parameter values row,move the left and right cursor key,after * letters on behalf of the content of the operation,move the * before A or B,after pressing the enter key,the system parameters restore to the factory.

A zone for servo configuration parameters, B zone for stepping configuration parameters, two kinds of parameters in the lifting speed time constant and top mobile or cutting speed has bigger difference,shall according to the actual machine tool motor drive configuration to recovery, otherwise may cause machine tool can not meet the best working state or even appear step out or vibration phenomenon.

Data backups

Move the flashing cursor to the parameter value saved line, move the left and right cursor, after * letters on behalf of the operated drive (C or D), press the output key, the current system parameters are saved to the setting disc area.

The backups content recovery for the current parameters

Move the flashing cursor to read the parameter values row,move the left and right cursor key,after the * letters on behalf of the operation of the disc,move the * to the front of C or D,press the enter key,the current parameter content is restored to the contents of the C or D area.

Note: before restoring,shall determine the parameters have a backup file in C or D plate.

10.3 Recovery, backup of system files

backup of files

In the system catalog tables, select the needed to backup file and press the transform key, and modify the file name, then press the enter key, completes the backups.

Recovery of the files

File backups is complete,after the modification of the parameters of the system parameters need to restore the parameters to the initial parameters,firstly select the backup file, press the switch key,then press the delete key and type a new filename, file name is S0001,then press the enter key,so the original file is replaced with the parameters of the backup file. Screw file I0001 and tool compensation T0001, alarm files A0001 the same with S0001 operation.

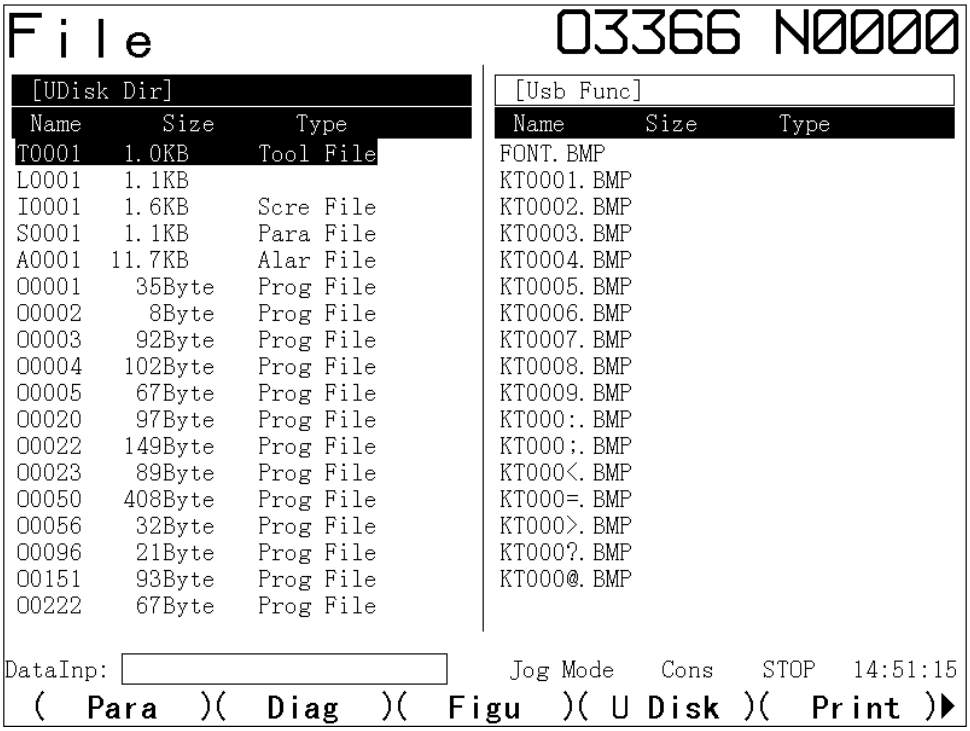
Delete files

System's own S, T, I, A, 0001 file is can not be deleted. In the system catalog tables, firstly choose which needed to be deleted file, press the delete key,the frame will display the deleted filename, pressing the enter key once again, the file is deleted.

Chapter 11 U disk operation

The system has read and write USB functions, convenient for the user to import and export programs and parameter files, and through the USB to upgrade the system software. This system supports the USB1.1 / USB2.0 protocol of usb memory,the U disk file format for FAT format. System default input/output filename suffix for .NC, such as O1234.NC, system can't identify the format of file name have no suffix, need to add. NC. Program files through the U disk being copied to your computer, can use notebook to open them, the same file edited by the notebook can also through the U disk being copied to the system.

The suffix of the system program files and parameter file is defaulted to .NC, upgrade software for .BIN format, boot interface for .BMP format.



11.1 How to export to USB

11.1.1 Program files

1 , In any pages to insert U disk to go to the USB interface.

2 , Through the pageup and pagedown key to move the cursor to the system catalog tables,press the up and down cursor keys to move the cursor to select the exported file,if the file number is large can flip the left and right key to operate,to select the file O0021.

3 , Press the output key,at the bottom of the screen shows the file name O0021, modify the file name is that type the file name after pressing the delete key,then press the enter key,in the U disk directory will appear the file.

4 , Press the output key,input O - 999, enter, will copy all the procedures to the U disk.

11.1.2 Parameter file

1 , In any page to insert U disk to go to the USB interface.

2 , Through the pageup and pagedown key to move the cursor to the system catalog tables,press the up and down key to move the cursor to select the exported documents, such as S0001.

3 , Press the output key,at the bottom of the screen shows the file name S0001, modify the file name is that type the file name after pressing the delete key,then press the enter key,in the U disk directory will appear the file.

Note 1 : Screw file I0001 and tool compensation T0001, alarm files A0001 the same with S0001 operation.

Note 2: the exported file name can not be modified.

Note 3: if the exported file have the same name with the existing file in U dish, there will be no alarm, but the new copy file will replace the original file in the USB.

Note 4: after firstly pressing the output key, if do not need to export, can through the cancel key to undo.

Note 5: the suffix of the system file name is.nc

11.2 How to import to the system

11.2.1 Program files

1. In any pages to insert U disk to go to the USB interface.
2. Through the pageup and pagedown key to move the cursor to the U disc catalog tables,press the up and down key to move the cursor to select the exported documents, such as O0001.
3. Press the output key,at the bottom of the screen shows the file name O0001,type the file name after pressing the delete key to modify the file name,then press the enter key again, the file will appear in the system directory.
4. Press the output key,input O - 999, enter,will copy all the procedures to the system.

11.2.2 Parameter files

1. In any pages to insert U disk to go to the USB interface.
2. Through the pageup and pagedown key to move the cursor to the U disc catalog tables,press the up and down key to move the cursor to select the exported documents, such as S0001.
3. Press the output key,at the bottom of the screen shows the file name S0001,type the file name after pressing the delete key to modify the file name,then press the enter key again, the file will appear in the system directory.

Imported into the system, covering the original S0001 file, become the current parameter file.If the user needs to retain the original parameter files,can backup the parameters in advance.

Note 1: Screw file I0001 and tool compensation T0001, alarm files A0001 the same with S0001 operation.

Note 2: file name can be modified.

Note 3: if the exported file have the same name with the existing file in U dish, there will be no alarm, but the new copy file will replace the original file in the system.

Note 4: after firstly pressing the output key, if do not need to export, can through the cancel key to undo.

Note 5: the suffix of the system file name is.NC

The system file operations see in operation chapter 10, section 3 of system files' backup, recovery.

Connecting

Chapter 1 Installation layout

1.1 System connection

1.1.1 System composition

Machine tool numerical control system is mainly composed of the following units, the following figure, including :

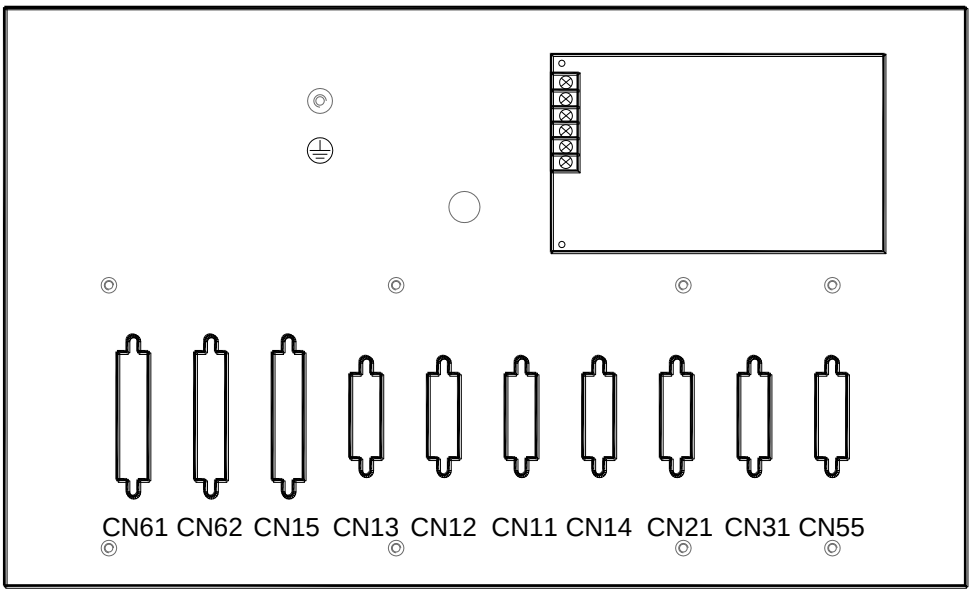
1. The NC
2. Digital AC servo drive unit (or stepping drive unit)
3. Servo motor（or step motor）
4. AC transformer

1.1.2 System installation and connection

CNC system and driver unit as part of the electrical machine tool,the work environment (including electric environment, temperature, humidity, vibration, etc.) good or bad has an important influence on it. When installate CNC system should pay attention to the following matters :

1. enough clearance should be kept between the back of numerical control device and the electric tank wall (not less than 10 cm), in order to connect the cable and heat dissipation.
2. CNC device must be reliable installed on the electric cabinet, fix bolts shall be installed to be complete, to avoid vibration.
3. CNC device should be installed where can avoid iron filings and cooling fluid can contact position.
4. The connection cable plug of CNC device should be tight with screw, to avoid the plug not firm or machine tool vibration impact contaction.
5. Around the CNC device should reduce the equipment which can bring high voltage and strong magnetic interference.

1.1.3 Back cover interface layout



1.1.4 Interface specification

CN61: input, 44 core type D needle socket, connected to machine tool input

CN62 : output, core type D hole socket, connected to machine tool output

CN15: spindle, 25 core type D hole socket, connected to spindle driver unit

CN13: Z axis, 15 core type D hole socket, connected to Z axis driver unit

CN12: Y axis, 15 core type D hole socket, connected to Y axis driver unit

CN11 : X axis, 15 core type D hole socket, connected to X axis driver unit

CN14 : A axis, 15 core type D hole socket, connected to A axis drive unit

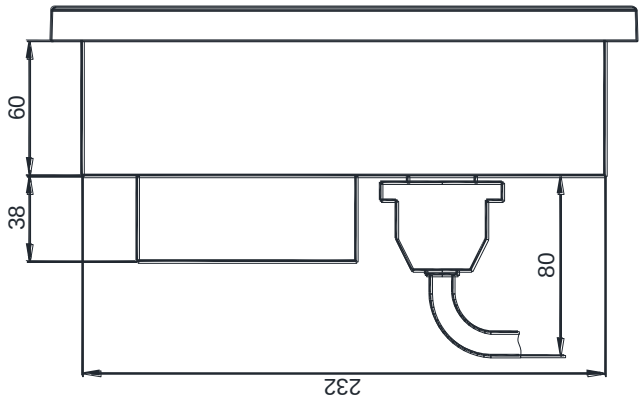
CN21: encoder, 15 core type D needle socket, connected to the spindle encoder

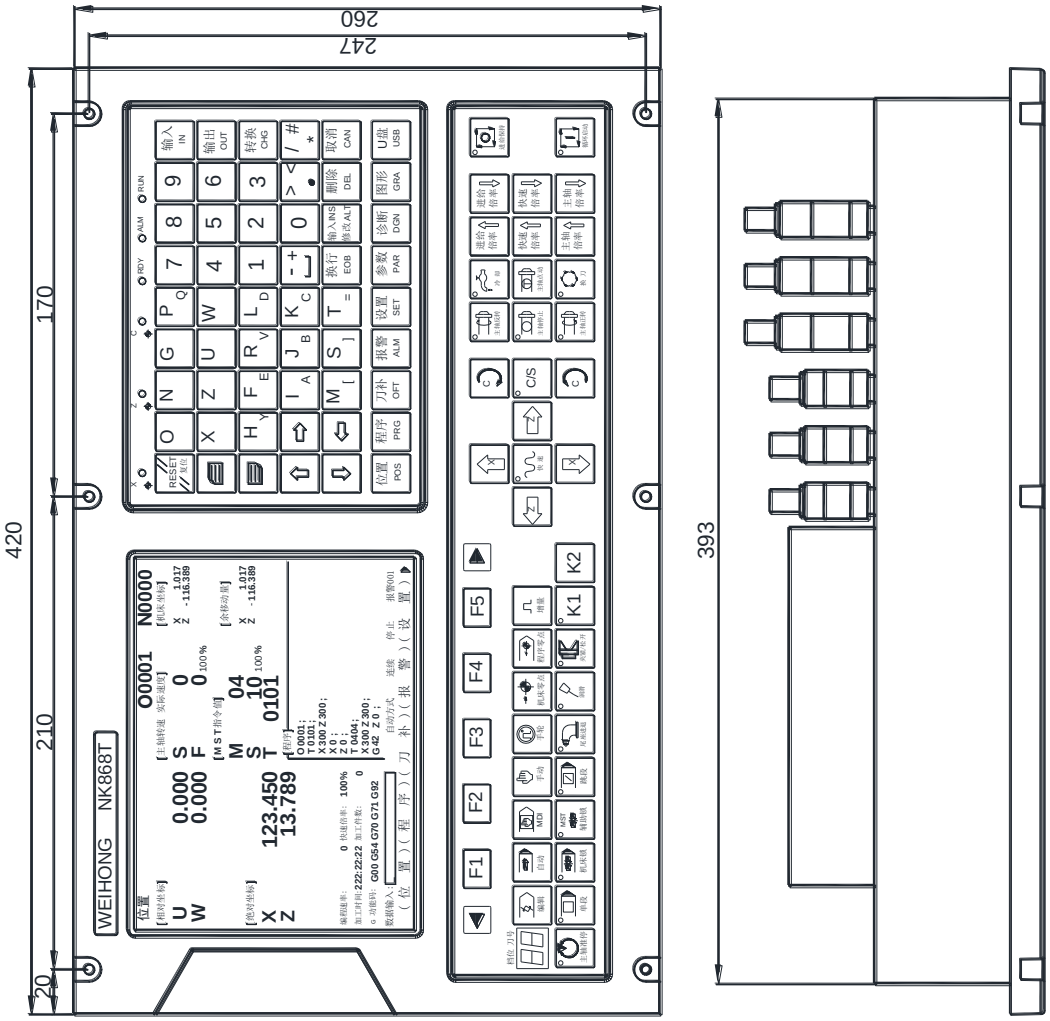
CN31: handwheel pulse, 26 core type D needle socket, connected to handwheel pulse

CN55: vice panel, 26 core type D hole socket, connected to vice panel

1.2 System installation

1.2.1 Outline dimension

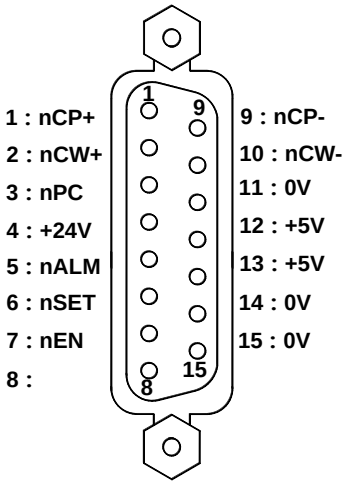




Chapter 2 Interface signal definition and connection

2.1 Connection with driver unit

2.1.1 Interface signal connection



CN11 , CN12 , CN13 , CN14（15 core D type hole socket）interface

2.1.2 Pulse and direct signal(nCP,nCW)

NCP +, nCP - for the pulse signal,ncw+,ncw- for direction signal, the two groups output signals are differential output(AM26LS31) , use AM26LS32 to receive outside, internal circuit below 2-2:

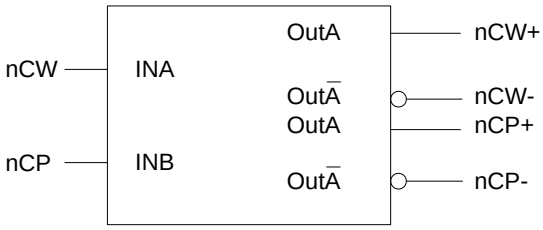
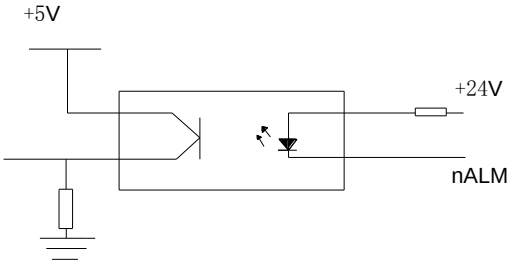


Fig. 2-2

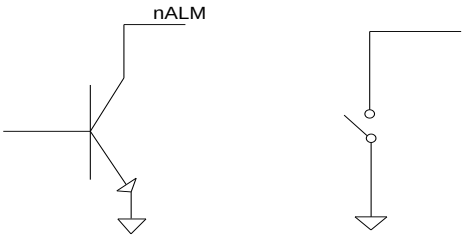
2.1.3 Alarm signal(nALM)

By the CNC parameters P008 Bit0, Bit1, Bit2, Bit3 set drive alarm level is low level or high level. Internal circuit as shown in fig. 2-3:



Pin	Clarify	X axis	Y axis	Z axis	A axis	Description
4	+24V					DC +24V
8	suspend					suspend
11, 14, 15	0V					Power 0V
12, 13	+5V					DC +5V
1	nCP+	XCP+	YCP+	ZCP+	ACP+	Pulse signal
9	nCP-	XCP-	YCP-	ZCP-	ACP-	
2	nCW+	XCW+	YCW+	ZCW+	ACW+	Direct signal
10	nCW-	XCW-	YCW-	ZCW-	ACW-	
3	nPC	XPC	YPC	ZPC	APC	Zero signal
5	nALM	XALM	YALM	ZALM	AALM	Driver unit alarm signal
7	nEN	XEN	YEN	ZEN	AEN	Enable signal
6	nSET	XSET	YSET	ZSET	ASET	Pulse forbid signal

The type of input circuit requirements drive unit adopts the chart 2-4 way to provide signal.

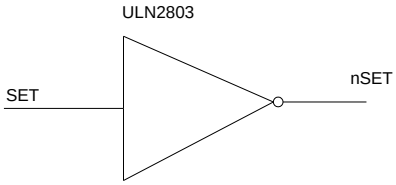


2.1.4 Enable signal(nEN)

CNC work normally the nEN signal output (nEN signal connecte with 0V) effectively, when the drive unit alarm, CNC close nEN signal output (nEN disconnecte with 0V). Internal interface circuit as below:

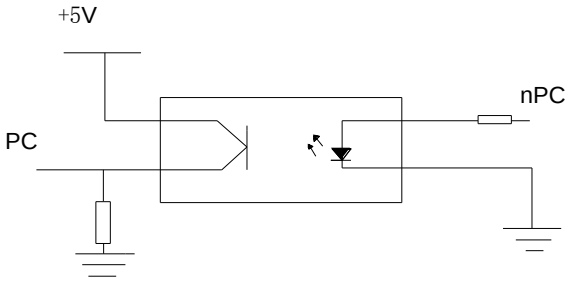
2.1.5 Pulse forbidden signal(nSET)

nSET signal is used to control the servo input, improve the anti-interference ability between CNC and drive unit, the signal is the high impedance state when CNC output pulse signal , without the pulse signal output for the low level. Internal interface circuit as below:



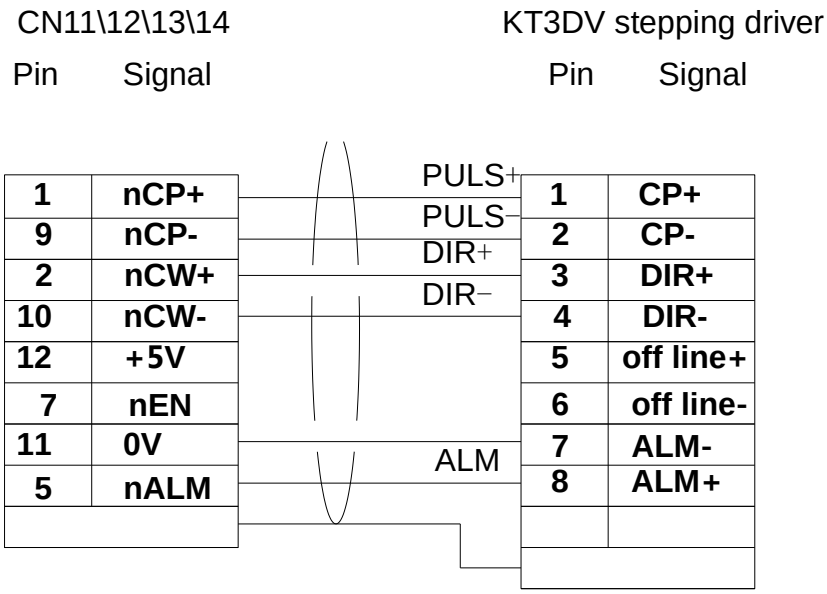
2.1.6 Zero signal

when machine tool back to zero use a motor encoder signals or close to switch signal as a zero signal. Internal connection circuit as below:

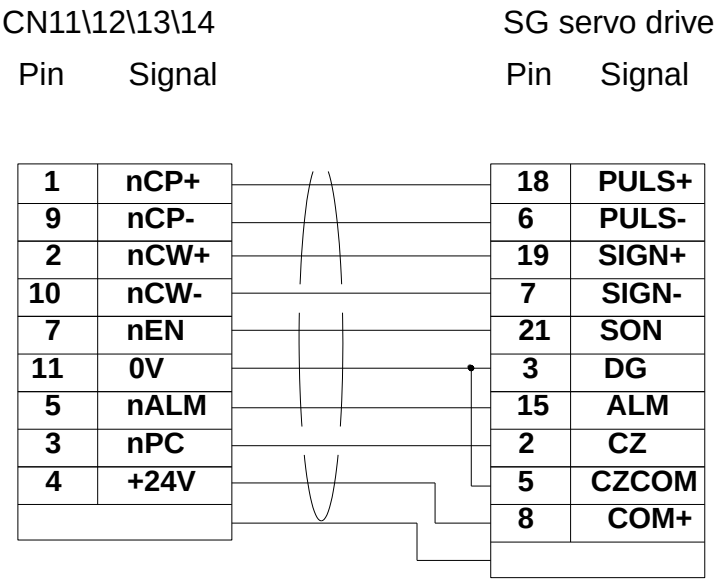


2.1.7 Connection with driver unit

(1) Stepping driver

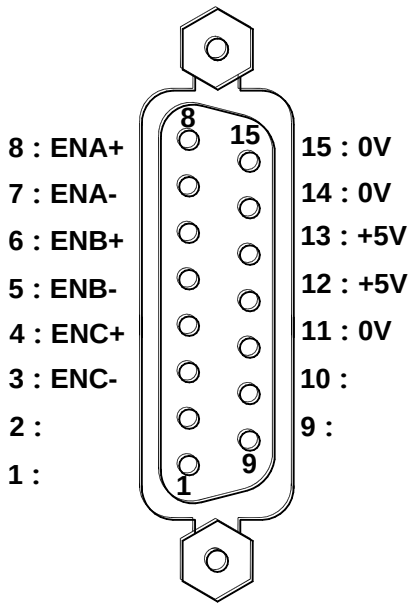


(2) Servo driver



2.2 Spindle encoder

2.2.1 Interface definition



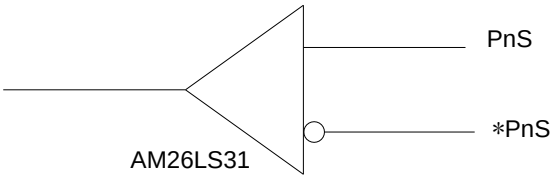
CN21（15core D pin）interface

Pin	Signal	Description
1, 2, 9, 10	suspend	suspend
11, 14, 15	0V	Power GND
12, 13	+5V	Power
7	ENA-	Encoder A pulse
8	ENA+	
5	ENB-	Encoder B pulse
6	ENB+	
3	ENC-	Encoder C pulse
4	ENC+	

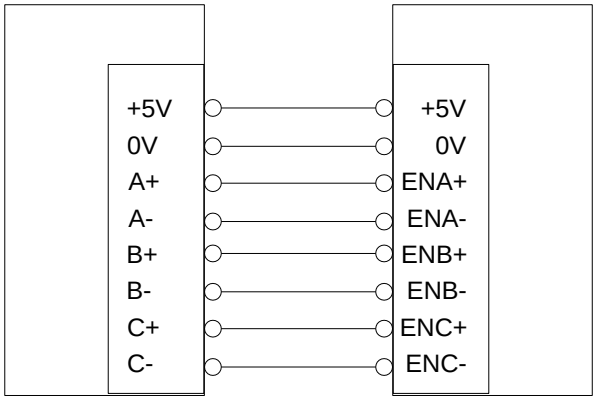
2.2.2 Description

ENA-/ENA+, ENB-/ENB+, ENC-/ENC+ respectively for the encoder of A, B, C phase differential input signal, use 26ls32 to receive; the wave of ENA-/ENA+, ENB-/ENB+ differ 90°quadrature wave, the highest signal frequency < 1 MHZ; Using encoder line number Setted by the parameter 130(range 100-5000).

Internal connection circuit:



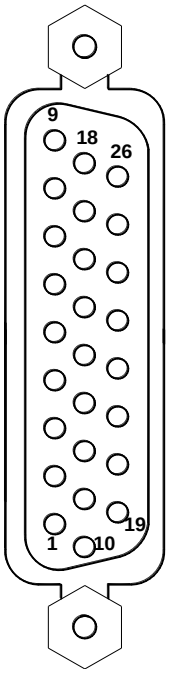
connection



2.3 Handwheel connection

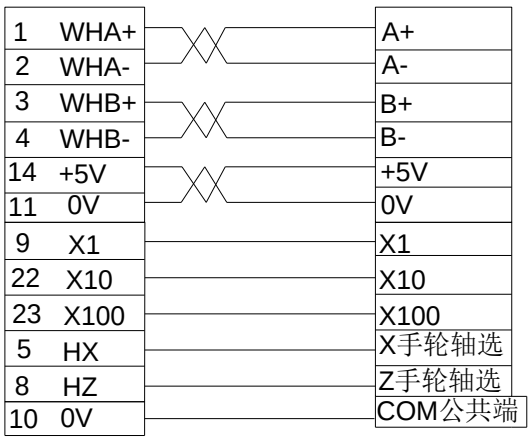
2.3.1 Handwheel interface definition

Connection:



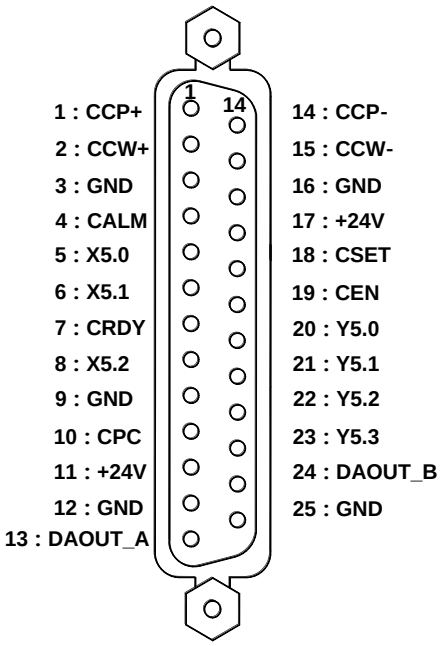
CN31（26 core D pin）interface

Pin	Signal	Description
1	WHA+	Handwheel A phase signal
2	WHA-	
3	WHB+	Handwheel B phase signal
4	WHB-	
5	XHAN	handwheel choose X axis
6	YHAN	Handwheel choose Y axis
8	ZHAN	Handwheel choose Z axis
9	X1	Increment×1
22	X10	Increment×10
23	X100	Increment×100
14~16	+5V	DC +5V
17~18	+24V	DC +24V
10, 11, 12, 13	0V	DC GND
19, 20, 21, 24	suspend	suspend
25, 26	Y7.6,Y7.7	Self definition



2.4 Spindle connection

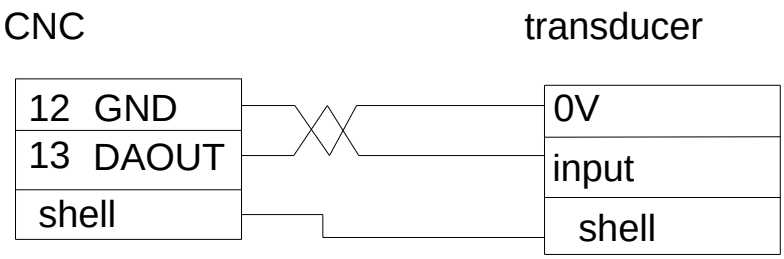
2.4.1 Definition



CN15（25 core D hole）interface

Pin	Signal	Description
1	CCP+	C axis pulse signal
14	CCP-	
2	CCW+	C axis direct signal
15	CCW-	
4	CALM	C axis anormal alarm signal
7	CRDY	C axis ready
10	CPC	C axis zero signal
13	DAOUT_A	Analog voltage output 1
24	DAOUT_B	Analog voltage output 2
18	CSET	Servo spindle motion mode
19	CEN	C axis enable signal
5	X5.0	Self definition input port
6	X5.1	Self definition input port
8	X5.2	Self definition input port
20	Y5.0	Accurate stop output signal
21	Y5.1	servo spindle forward control
22	Y5.2	servo spindle reverse control
23	Y5.3	Servo spindle external EMG stop control
3,9,12 16,25	GND	GND
11, 17	+24V	DDC +24V

2.4.2 Transducer connection



2.4.3 Connect with servo spindle

1: Parameters:

0	0	1				MDSP				

MDSP : =1 : Simulate Control =0 : Other

0	1	9	RHD5	SPOR	RSCS	CZSE	CALM	CDIR		

RHD5 =1: Allow Servo Spindle =0: No

SPOR =1 : Spindle Position Available =0 : No

RSCS =1 : No =0 : ESP Return Speed State

CZSE =1 : Zero Pulse Is CPC Signal =0 : No

CALM =1 : C Axis Alert Is High Level =0 : low Level

CDIR =1 : C Axis Position Pos =0 : negative

0	2	0	PRSS				SMSE	SALM	CRHL	CRDY

PRSS =1: Enable Servo Ready Stop When C/V Swicht

=0: Disable

SMSE =1: Servo Maner Engage Plus Position =0: No

SALM =1: ESP Close Servo Spindle =0: No

CRHL =1: Servo Spindle Ready High Level =0: Low

CRDY =1: Check Servo Spindle Ready =0: No

159	Spindle Stop Port Output Time(x4ms)	200	1~10000

When press the spindle must stop button, corresponding port pulse output time,the port output signal cannot be zero.

160	Tapping Define Output Port	0	1~72
161	Stop To Position Define Input port	0	1~72

when the system must stop the system will output the stop signal, if the parameter P161 set the ports must stop in place, then must stop signal always output,until must stop signal effectively. If the must stop signal is not detected for a period of time,system prompted alarm 73. if parameter P161 is not setted, it must stop signal output the pulse signal, the pulse width by the parameter P159 said.

174	Serve Spindle Pre_Stop All Time (x4ms)	100	1～10000

2: Command

M14: Spindle from the speed control mode switch to a position control mode

M15: Spindle from the position control mode switch to a speed control mode

3: Work process

1). The factory default servo is not enabled,parameters P019 BIT7 is 0. To make servo spindle enabled, must manually modify parameters.

2). When servo spindle is allowed, the CNC servo output can be setted to high, the servo spindle external alarm input signal is setted to high.

3). System default to speed control mode, analog control the speed output. System C axis display 0.000 always.

4). At this point, if execute M03 M04/M05 instruction or press the forward/reverse/stop on the panel. System to control the driver signal is the forward/reverse signal on the spindle interface. The original control the general inverter M03 M04, M05 output signal is invalid.

At this point can regulate speed by S instructions.

5). To execute commands M14,the CNC send C/S signal (through the SET pins speed/position control signal) to driver,after the driver receiving the signal, switch to the location mode. In front of the switch,the CNC will firstly execute spindle stop running and analog output is 0. Complete switch must wait for the spindle speed is 0, switch operation completelyPay attention to the M14 instruction execution,after switching to the position the state will perform must stop action, absolute coor is cleared to zero.

M15 instruction execution is the same as the M14, switch position mode to speed mode.

Press the switch key C/S on panel, the function of the key is same with the M14 instruction, when switch to the position mode ,the light is on. When switch to speed mode, the lights is off. (note after the reset, stop and alarm if the system restore to speed mode,the lamp will be off).

6). When switch to the position control mode,should perform back to 0 operations. The operations are done by instruction G28 C. After the instruction execution, C axis coordinate display 0.000. back to 0 test PC signal or encoder head pulse signal by the parameter P019 BIT4 decide.

If not back to zero, but can use servo must stop output function, make it back to 0, pay attention to after executing servo back to 0, display 0.000. Servo must stop function is only effective under the manual button. Under the condition of automatic execution M14 instructions also perform must stop instructions.

7). Pay attention to when the parameter P020 Bit7 bit is setted to 1,after switching to position mode system will automatically perform must stop.

8). Once by M15 instructions or on the front panel C/S switch button to switch to the speed instruction status, C axis remains the same.

4: Attention:

1). The C axis is rotary type only, the display data changes from 0 to 359.999.

2). Into position instruction, in the case of manual, can through the C key to rotate the C axis. MDI can make C axis move.

In automatic case, through programming can make C axis with other axis interpolation, such as G01X10Z10C10 instructions.

3). Under the position command,C axis can't rotate, only through switching for analog way to make C axis rotate.

4). After the stop,reset and servo alarm,the CNC will switch to the speed control mode.

5).When press the spindle must stop button on the panel,the system output a must stop signal to servo spindle,when servo spindle receive the signal,it will operate automatically to the setted position.

6).Under the mode of servo axis allows, the handwheel can control C axis rotation positive/negative. The direction of the pulse can be achieved by parameter P019 Bit2 regulation.

7).Can use G90/G91 incremental to programme, increment of C programming, at the same time also can do this to A.

8).The head of the pulse signal of the servo back to 0 is given by encoder, also can be given by CPC signal which decided by the parameter P019 Bit4.

9).Once detecte C axis alarm signal, when servo allow C axis switch back to the state of analog control,all output of the servo spindle is invalid,including forward/reverse/analog signals. Alarm number is 22.

10).If C axial is ready abnormally,the CNC prompt 55 alarm.

11).Parameters P020 Bit3 is servo spindle control way, corresponding to the two modes of servo driver, when to 0, the operating method of servo for forward and reverse, when the servo is speed controled mode, for enable + direction control mode.

under speed mode :

The system servo working mode	Operate way	Output port CEN	Output port Y5.2	Output port Y5.1
enable+direction Para P020Bit3 = 1	stop	0	0	0
	positive	1	0	1
	negative	1	1	0
positive/negative Para P020Bit3 = 0	stop	0	0	No
	positive	1	0	No
	negative	0	1	No

Note servo spindle is not for speed mode, such as location mode, must stop mode and other mode, the output of the CEN must be 1.

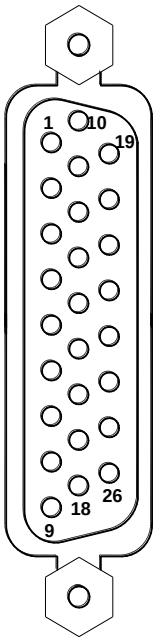
12).The output of analog controlled by S instruction,under the manual of forward/reverse operation does not affect the analog output. Reset close analog output.

13).Servo enable,when in position instruction,execute M03/M04/ M05 instructions,system alarm,in the speed mode,if G01 C similar instruction is executed,system alarm.

Alarm 159: servo spindle in position mode cannot perform M03 M04/M05 instructions

Alarm 160: servo spindle in speed mode cannot perform G01 C instructions

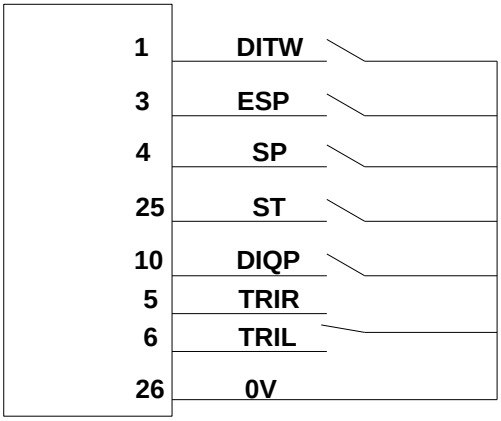
2.5 Hand-set sub Connection:



CN55（26 core D hole）interface

Pin	Signal	Description
1	DITW	Tailstock control signal
3	ESP	External EMG stop signal
4	SP	External pause signal
5	TRIR	Three position switch
6	TRIL	Three position switch
8	WHA+	Handwheel pulse signal
9	WHB+	
10	DIQP	Chuck input signal
21	T05/OV1	Feed rate switch input signal
22	T06/OV2	Feed rate switch input signal
23	T07/OV3	Feed rate switch input signal
24	T08/0V4	Feed rate switch input signal
25	ST	External start signal
2	+24V	+24V
7, 16	+5V	+5V
19, 20 26	0V	GND
11, 12, 13, 14, 15	Suspend	Suspend
17	485B	485 communication signal B
18	485A	485 communication signal A

Inner connection as follows:

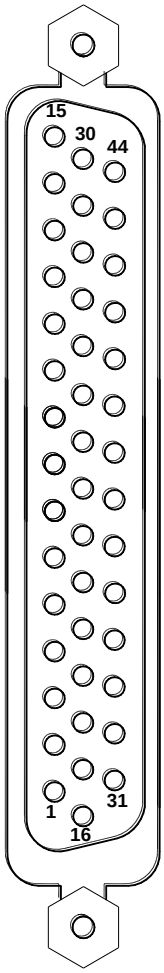


Description: input function is available, the input signal is conducted with 0V. while the input function is unavailable, the input signal is high inpedance off. So lower level available.

2.6 Import/outport

2.6.1 Pin definition

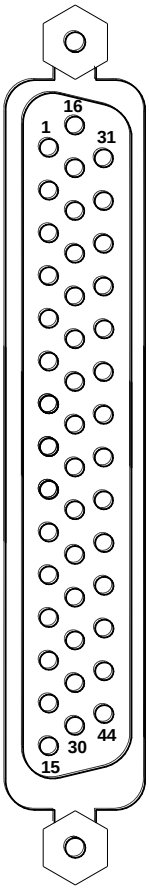
Import



CN61（44 core D pin）interface

Pin	Function	Description
21~24	Power	0v
17~20 25~28	Suspend	Suspend
1	SAGT	Protection door detection signal
2	SP	External feed hold signal
3	DIQP	Chuck input signal
4	DECX	X axis reduce signal
5	DITW	Tailstock control signal
6	ESP	External EMG stop signal
7	PRES	Check press signal
8	T05/OV1	Tool number signal 5/OV1
9	T06/OV2	Tool number signal 6/OV2/gating signal
10	T07/OV3	Tool number signal 7/OV3/preassigned degree close switch
11	T08/OV4	Tool number signal 8/OV4/tool table overheating check
12	DECZ(DEC3)	Z axis slow down signal
13	ST	External start signal
14	M41I	Shift one in place
15	M42I	Shift two in place
16	T01	Tool number 1
29	T02	Tool number 2
30	T03	Tool number 3
31	T04	Tool number 4
32	DECY(DEC2)	Y axis slow down signal
33	DEC4	4th axis slow down
34	DEC5	5th axis slow down
35	TCP	Tool rest locking signal
36	AEY/BDT	External jump
37	LIM+	Positive limit
38		A axis handwheel axis choose(CN31-7)
39	LIM-	Negative limit
40	WQPJ/ VPO2	Chuck loose in place
41	NQPJ/ SALM2	Chuck clamp in place
42	SKIP	G31 jump signal
43	AEX	X axis tool measure in place signal
44	AEZ	Z axis tool measure in place signal

Output port



CN62（44 core D hole）interface

Pin	Function	Description
17~19 26~28	Power	0V
20~25	Power	+24V
1	M08	Cool output
2	M32	Lubricate output
3	Y0.2	Self definition
4	M03	Spindle anticlockwise rotation
5	M04	Spindle clockwise rotation
6	M05	Spindle stop
7	Y0.6	Self definition
8	SPZD	spindle braking
9	S1/M41	Spindle mechanical gear 1
10	S2/M42	Spindle mechanical gear 2
11	S3/M43	Spindle mechanical gear 3
12	S4/M44	Spindle mechanical gear 4
13	DOQPJ (M10)	Chuck clamp output
14	DOQPS (M11)	Chuck loose output
15	TL+	Tool rest forward
16	TL-	Tool rest reverse
29	TZD	Tool table breaking
30	INDXS	Tool table preassigned degree of coil
31	YLAMP	Tri-color lamp yellow lamp
32	GLAMP	Tri-color lamp green lamp
33	RLAMP	Tri-color lamp red lamp
34	DOTWJ (M78)	Tailstock forward
35	DOTWS (M79)	Tailstock retreat
36~44	Y2.7, Y3.0~Y3.7	Self definition

2.6.2 Input port

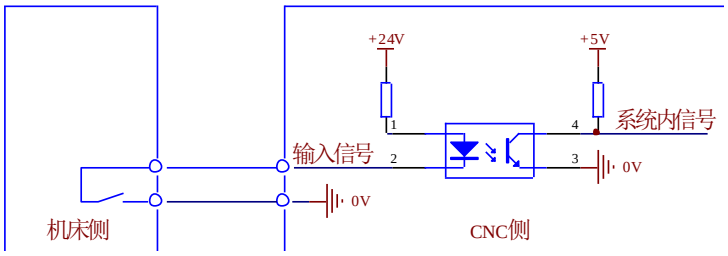
1, Input port signal

System have 54 road input ports, all the input mouth and circuit inside the the system be used the photoelectric isolation processing, each input port electrical specifications as follows:

（1）Photoelectric isolating circuit, maximum isolation voltage 2500 VRMS

（2）Input voltage range 0 v ~ 24 v DC

Input port electrical schematic diagram below:



The system input interface of the standard PLC definition function includes DECX , DECZ , DECY , DEC4 , DEC5 , ESP , ST , SP , SAGT , PRES , AEY/BDT , DITW , DIQP , OV1～OV4 , T01～T08 , TCP , LIM- , LIM+ , M41I , M42I , LMIX , LMIY , LMIZ , WQPJ , NQPJ , SKIP , AEX , AEZ and so on.

Input signal refers to the signal from machine tool to the CNC, when the input signal connected with 0 v,input valid ; When the input signal is connected with + 24 v, the input is invalid (all I/o port of this system are low level effective).The contact of input signal at the side of the machine tool should meet the following conditions :

Contact capacity: DC30V,more than 16mA

When open between contact leakage current: less than 1 mA

Pathways between the contact voltage drop: under 2 v,(current 8.5 mA, including the voltage drop of the cable).

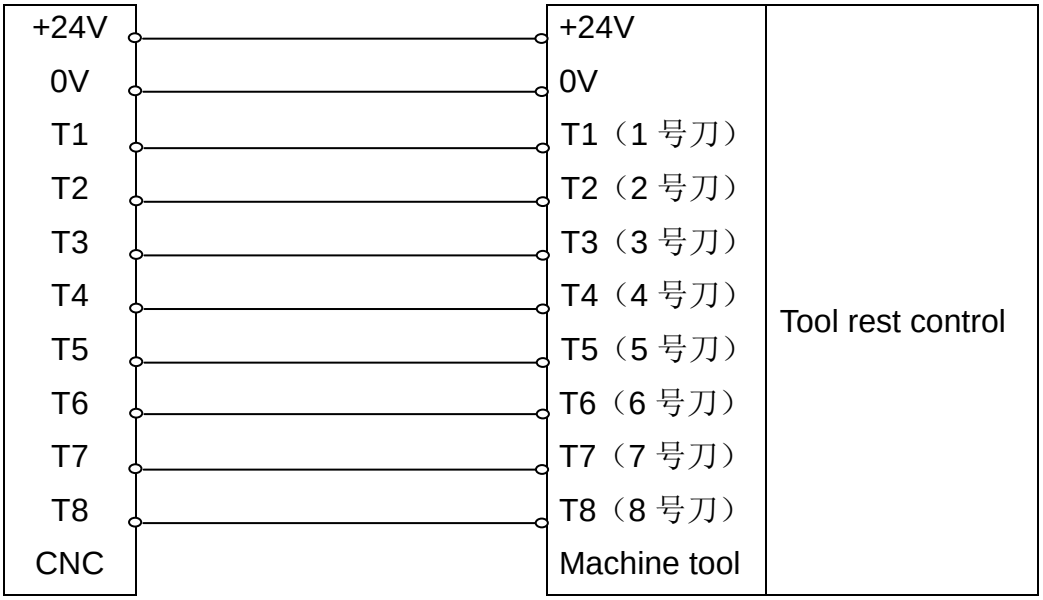
There are two ways to get the input signal of the external input: use a contact switch input, in this way the signal from the side of the machine tool buttons , limit switch and relay contacts,and so on, another use contactless switch (transistor) input.

2, Description

(1) T01～T08 Tool signal

Tool signal is high level or low level effectively,decided by the parameter P009 Bit1 Settings, 0 for high level effective, 1 for low level effective.when there is one signal of T01 - T08 effective,it is said in the knife position .

The wiring diagram below :



(2) TCP Toolpost Lock Status Signal

In the process of changing tool,after toolpost is in place the system output toolpost reversal signal (TL-), and started to detecte the lock input signal TCP, when received the signal,the time delayed by the parameters P183 setted,close toolpost reverse signals,tool change instruction to end,next program executed.When the system output toolpost reversal signal,if CNC didn't receive TCP signal in period of time setted by the parameter P0184,the system will generate alarm and close the toolpost reverse signals.If the toolpost controller without locking signal output,set the parameter P009 Bit0 to 0, don't lock signal detection tool don't need to detecte the toolpost lock signal.

(3) DECX, DECZ, DECY, DEC4, DEC5 deceleration switch signal

In back to the machine tool zero point mode, press the corresponding manual feed key, machine tool quickly (setted by the parameter P101) move to the setted direction back to zero, When deceleration switch signal is detected, the machine tool to slow down to stop, and low speed (setted by the parameter P100) leave deceleration switch in the opposite direction, When slow signal disappears, machine tool slow down to stop, after again at low speed near the speed switch, System to detect the Zero signal of the servo encoder pulse (Zero pulse) and deceleration switch signal,when receive signal then stop motion, the axis back to Zero completely,back to Zero indicator is light.System is to detect the Zero pulse or deceleration switch signals as the final back to Zero completly,decided by the parameter P006 Bit4, Bit5, Bit6.

(4) ST Cycle start, SP Feed keep signal

External loop start signals ST and external pause SP function keep the consistent with cycle start and feed keep function on panel.By the parameter P014 Bit0 to decide whether shield ST signal, parameter P014 Bit1 to decide whether shield SP signals.

ST and SP are low-level pulse signal effective.

(5) ESP Emergency stop signal

External emergency input signal, the low level effectively.By the parameter P015 Bit7 to decide whether shield ESP signal.When there is ESP signal, the machine tool feed emergency stop, spindle stop, cool off, stop changing tool.

(6) LIM＋, LIM－The positive and negative hardware limit signals

LIM＋, LIM－signal for low level effective, owned by all axis.Connecte each axis positive limit signal to access LIM +, the axis of negative limit signal to access LIM－.Limit switches shall be normally open mode.When produce go beyond alarm, system will stop the direction of feeding, you can manually reverse feed.

(7) SAGT Protective door detection signal

Protective door detecte signal SAGT effective level signal decided by the parameter P016 Bit6, when SAGT signals effective, the system to produce alarm prompt, and stop the current process.This function is only effective in automatic processing mode.

The wiring diagram :

(8) DIQP Chuck input signals and chuck loose/clamp in placeWQPJ/NQPJ

there are two signals of Chuck clamping whether in place :

NQPJ : chuck clamp in place signal,high level effective.

WQPJ : chuck loose in place signal,high level effective.Also used for outer chuck,chuck clamp input signal.

(9) Pressure detection signal

When detecte abnormal pressure alarm, low level effective.

2.6.3 Output port

1 , Summary of the output signal :

System have total of 48 road output, driving circuit for darlington tube OC(open collector output)output,the electrical specification for each road output :

(1) Output ON the maximum load current 500 mA

(2) Output ON the maximum saturation pressure drop, 350 mA is 1.6 V, the typical value of 1.1 V.

(3) Output OFF pressure below + 24 v.

(4) Output OFF when the leakage current under 100uA

Output electrical schematic diagram below :

Output signals include the following :

S1～S4, M03, M04, M05, M08, M10, M11, M32, M78, M79, TL-, TL+, SPZD, STM, WARN, M30, Y32, Y29, Y26, Y14, Y13, Y21, Y22,M10,M11,TZD,YLAMP,GLAMP,RLAMPand so on.（The system are all input and output low level effective）.

2 , Function description

「188」Connecting

(1) M03, M04 M05 spindle M function signal port

M03 for spindle Positive signal ;

M04 for spindle negative signal ;

M05 for spindle stope signal ;

(2) M08 Cool control port

M08 cool open ;

M09 cool close ;

(3) TL + for toolpost positive rotate signal port ;
TL－for toolpost negative rotate signal port ;

(4) S1～S4 The spindle variable speed signal

S1 ~ S4 instructions and the M41 ~ M44 Share S1 ~ S4 output, is used to control the spindle I ~ IV option.S1 ~ S4 signal port can only have one effective. S0 instructions close S1 ~ S4 signal.

(5) SPZD The spindle brake signal

When performe the M05,if parameters P151 and P152 are not 0, then the system output signal M05,delay the time decided by para P151 setting, output SPZD signal, the signal pulse width are setted by the parameter P152.

(6) RLAMP, GLAMP, YLAMP Three color light control signals

RLAMP is lamp output, when the parameters P015 Bit6 is setted to 1, if the system alarm, the signal output effective.after canceling alarm the signal is invalid.

YLAMP for standby light output, when the parameter P015 Bit5 is setted to 1, if not running processing program or run over or system just to be powered on, the signal port output effectivly.

GLAMP is the running light output,when the parameters P015 Bit4 is setted to 1, if the system is running process, the signal port output effective.When the program is running to suspend or end or alarm the signal port is invalid.

Usually RLAMP, YLAMP, GLAMP signal port respectively connect with three color light red yellow green control port.

(7) M10, M11 for chuck clamp/loose control port

When the way for the inner chuck, M10 is chuck clamping signal, M11 for chuck loosing signal ;

When the way for the outer chuck, M10 for chuck loosing signal, M11 for chuck clamping signal ;

(8) M78, M79 for tailstock top control port

M78 for the tailstock forward port,M79 for the tailstock backward port.

(9) M32 Lubrication control port

M32 instruction for lubrication open, M33 instructions for lubrication close ;

When the parameter P013 Bit2 is setted to 0, for continuous oil supply mode, M32 signal port continuously output signal ; When the parameter P013 Bit2 is setted to 1, for intermittent oil supply mode, fuel supply time setted by parameter P197, suspension fuel supply time setted by P198.

(10) Y 0. 2, Y 0. 6, Y 2. 7, Y 3. 0, Y 3. 1, Y 3. 2, Y 3. 3, Y 3. 4, Y 3. 5, Y 3. 6, Y 3. 7, Y 5. 0, Y 5. 1, Y 5. 2, Y 5. 3, Y 7. 6, Y 7. 7 for extend output port

Standby expand output port, the user can through the M20, M21 instruction to programme to control output port state.The serial number of output port within the system respectively are 8, 12, 21, 22 , 23, 24, 25, 26, 27, 28, 30, 34, 35, 36, 37, 46, 47 （Through diagnostic interface can view all output internal number）

use M20, M21 command to control output port Programming, the instruction format is as follows :

Y5.3 output port effective : M20 K37

Y5.3 output port invalid : M21 K37

use M22 command to control output pulse output, its programming instruction format is as follows :

Such as control Y5.3 port output 2 seconds wide pulse : M22 K37 J2

Chapter 3 Usual function description

3.1 Setting electronic gear ratio parameter

The function of the electronic gear is to make the change of the system coordinate values and the change of the location of the workbench the same, rather than for matching different pitch screw,different angle of the motor and different line matching of the servo motor to adjust motor and screw mechanical transmission ratio.

Parameters P025, P026, P027, P030, P031, P032 used to set the electronic gear ratio of X, Y, Z axis, each axis of electronic gear have two parameters ratio value and fraction value, the matching is used to feed the pulse signal of the regulating system of the output, in order to meet the coordinate values of the change and the change of the location of the workbench.

Ratio parameters expressed with MLT , fraction parameters expressed with DVT, the ball screw drive workbench its electronic gear ratio to satisfy the following relations:

$$\frac{MLT}{DVT} = \frac{Pmt \times Gf}{Pcn \times Gd}$$

Among them: Pmt: motor per pulse number, step motor: after subdivision steps

Servo motor: encoder line × several times frequency

Pcn : Screw pitch, Need to convert toμm.for example 6mm lead screw, corresponding Pcn value should be 6000 ;

Gf : the screw driven wheel teeth number of motor and mechanical transmission parts, If the direct connection Gf ＝1.

Gd : the screw driving wheel number of electrical and mechanical transmission parts, If the direct connection Gd ＝1.

MLT and DVT must take its minimal positive integer values, the range of MLT and DVT is 0 ~ 65535.

The corresponding parameters of MLT : P025, P027, P026

The corresponding parameters of DVT : P030, P032, P031

Machine tool users need on the basis of the above algorithm to calculate the ratio of X axis and Z axis value (MLT) and fraction (DVT), respectively, set up corresponding parameters.

Example 1: the Z axis is 6 mm lead screw, and stepping motor straight league,the driver for the three-phase hybrid stepping, 6000 segments:

The Pmt＝6000 ; Pcn ＝6mm x 1000=6000 ; Gf＝Gd＝1 ;

MLT/DVT =6000/6000 ＝ 1 ;

You can take MLT ＝ 1 ; DVT ＝1 ;

Set parameters P027＝1 ; P032 ＝ 1

Example 2: the X axis is 6 mm lead screw, and stepping motor straight league,the driver for the three-phase hybrid stepping, 10000 segments:

The Pmt ＝ 10000 ; Pcn ＝6mm x 1000=6000 ; Gf＝Gd＝1 ;

MLT/DVT =10000/6000 ＝ 5/3 ;

You can take MLT ＝ 5 ; DVT ＝3 ;

Set parameters P025 ＝ 5 P030 ＝ 3

Example 3: the Z axis is 4 mm lead screw, direct connects with servo motor, servo motor encoder 2500 line, 4 times servo drive internal encoder signal processed, which is 10000 segment:

The Pmt ＝ 10000 ; Pcn ＝4mm x 1000=4000 ; Gf＝Gd＝1 ;

MLT/DVT =10000/4000 ＝ 5/2 ;

You can take MLT ＝ 5 ; DVT ＝2 ;

Set parameters P027 ＝ 5 P032 ＝ 2

3.2 Linear deceleration time constant

The physical meaning of linear deceleration time constant is: movement speed from zero speed, at a constant acceleration, speed up to 10 meters/minute.

The smaller the deceleration time constant, the faster the speed;

The greater the deceleration time constant,the slower the speed;

System provides a fast moving accelerate and decelerate speed parameter Settings, cutting feed accelerate and decelerate speed parameter Settings, thread cutting accelerate and decelerate speed parameter Settings, the handwheel feed accelerate and decelerate speed parameter Settings, backlash compensation accelerate and decelerate speed parameter Settings, and so on. The user should be appropriate to the character according to the machine tool Settings, in order to achieve good working condition.

Fast moving accelerate and decelerate speed Settings : P040, P042, P41

cutting feed accelerate and decelerate speed Settings : P051

Thread cutting accelerate and decelerate speed Settings : P166, P169

The handwheel feed accelerate and decelerate speed Settings : P209

Backlash compensation accelerate and decelerate speed Settings : P066

3.3 Setting spindle parameter

Related parameters of spindle set see parameters of 130 to 158 in the appendix 1.

3.4 Chuck control

1．Related parameters:

0	1	5						KPDW		
0	1	6	NWKP						SLSP	SLQP

KPDW ＝1 : check clamp chuck in place signal

＝0 : not

NWKP ＝1 : Chuck Holds Outer

＝0 : Inner

SLSP ＝1 : Check Chuck Loosen State

＝0 : No

SLQP ＝1 : Chuck Function Available

＝0 : No

191	Delay Time For Chuck Close Time (×4ms)	100	0～10000

The KPDW ＝1; when chuck checked in place, the parameters are used to set delay detection time

190	M10 M11 output time（×4ms）	0	0～10000

Spindle Clamp/Loose time setting; ＝0 : level signal >0 : pulse signal

The parameters used to set the M10 and M11 output model,if need long signal level output ,the parameters should be setted to 0; If need a short signal output the parameters used to set the pulse width.

2．Input/output signal

NQPJ : Inner Chuck Clamp/Outer Chuck Loose Ready, specified by the P192 parameter, the factory default values for 28.

WQPJ : Inner Chuck Loose/Outer Chuck Clamp Ready, specified by the P193 parameter, the factory default values for 29.

DIQP:Chuck Clamp/Loose input port, general external pedal switch

M10 : Inner Chuck Clamp output, Outer Chuck Loose output

M11 : Inner Chuck loose output, Outer Chuck Clamp output

3．Sequence

(1) when SLQP ＝0

No chuck function,all chuck input and output is invalid

(2) when SLQP ＝1, SLSP＝0, KPDW＝0

control Chuck clamping and loosening output, don't check chuck clamping or loosening in place signal,when start the spindle don't check chuck state.

(3) when SLQP ＝1, SLSP＝1, KPDW＝0

Check the chuck state when spindle is started, chuck state is decided by whether the clamping or loosening of instructions executed, no check clamping or loosening in place of signal.the system according to whether triggers a pedal switch clamp input or panel chuck button clamp input or M10 instructions in the program is executed can decide whether to start the spindle.

(4) when SLQP ＝1, SLSP＝1, KPDW＝1

when spindle is started detect the chuck state, chuck states is decided by whether detecte the signal of chuck clamping in place. If not detecte clamping in place signal,there is alarm .

(5) when NWKP＝0

Inner Chuck model, this moment :

M10 : chuck clamp output

M11 : chuck loose output

NQPJ : the clamping in place input signal

WQPJ : the loosing in place input signal

(6) when NWKP＝1

Outer Chuck model, this moment :

M10 : chuck loose output

M11 : chuck clamp output

NQPJ : the loosing in place input signal

WQPJ : the clamping in place input signal

(7) the Settings of the parameter when the chuck clamping and loosing state signal only one way.

In inner chuck model, set the parameters P192 as clamping state input signal, set parameter P193 as 0.

In outer chuck model,set the parameter P193 as clamping state input signal,set the parameter P192 as 0.

Under this case the system only tested clamping state signal, a signal indicates clamping, no signal indicates loosing.

3.5 Handwheel

1．Related parameters

(1) P008 Bit4

＝1 : Handheld unit is available, ＝0 : not the handheld unit

When the handheld unit effectively,you can through the selector switch on the handheld unit to choose axis and ratio selection ;

When the handheld unit is invalid,you can through hand wheel shaft and pulse rate keys which on the system front panel to set.

(2) P008 Bit5

Set the handwheel rotating direction

(3) P204 – P207

Define the handwheel speed upper limit

(3) P209

Define the handwheel feeding time constant

2．Connection

The system default connection mode is seen in the third quarter of the second chapter of the third article.

3.6 Tailstock

Through the panel buttons, execute M78/M79 instructions or external input ports to control tailstock forward or backward.

1．The tailstock control output :

M78: forward control output port ;

M79: backward control output port ;

2．The tailstock control related parameters:

Para P194: M78 output time ;

Para P195: M79 output time ;

3．The tailstock control sequence :

When Para 194,195 are greater than 0 respectively, the tailstock controls two short signal (pulse) mode: Under this mode, execute M78 instruction or buttons control to execute the tailstock forward, M78 port output a pulse signal which time width dicided by P194 ; when execute M79 instruction or buttons to control tailstock backward,M79 port output a pulse signal which time width decided by P195;

When Para 194,195 are respectively equal to 0 ,the tailstock control two long signal (pulse) mode: : Under this mode,execute M78 instruction or buttons control to execute the tailstock forward, M78 port keep effective,M79 port keep invalid ; when execute M79 instruction or buttons to control tailstock backward,M79 port keep effective , M78 port keep invalid ;

4．In system, external input port CN61 5 pin for the outside tailstock input pin, when the input port have input signal, at this time equivalent the tailstock forward/backward button on the control panel is pressed. This function enabled by para16 bit 2 controled.

3.7 Lubrication

Through the panel buttons or execute M32/M33 instruction can control lubrication control mouth open and close.

M32 instructions for lubrication open, M33 instruction for lubrication close; Or press lubrication key between the lubrication function on or off to switch.

Lubrication function output: M32

1．Parameters:

Parameter P013 Bit2 = 0, for the continuous oil supply mode ;

The parameter P013 Bit2 = 1, for intermittent oil supply mode ;

Parameter P197: intermittent mode oil supply on duration, unit: second

Parameter P198: intermittent mode oil supply off duration, unit: second

2．Action order :

When for continuous oil supply mode,M32 instructions that are executed or or key lubrication function open, M32 output port maintain effective output ; Perform M33 instruction or key lubrication function close, keep M32 output invalid, stop the oil supply ;

M32 instructions that are executed or key lubrication function open when for intermittent oil supply mode, keep M32 output the setted time that decided by the parameter P197, and then shut down, delay the time setted by parameters P198, again keep output the time setted by parameters P197, and repeat ; Perform M33 instruction or key lubrication function close, keep M32 output invalid, stop the oil supply. In the intermittent oil supply model, system automatically open interval lubrication after the system is powered on.

When lubricating the lubrication indicator is powered on, shut lubrication function or lubrication pauses the button light is powered off.

3.8 Soft limit

1. Parameters:

Para P120,P121,P122,P125,P126,P127 : set X, Z, Y axis positive and negative direction limit coordinate ;

Para P014 Bit2 : Set whether check the positive and negative direction limit coordinate values ;

Para P014 Bit3 : Set the soft limit coordinate for machine tools coordinate or workpiece absolute coordinates ;

2. Description:

When the soft limit is limited by the machine tool coordinate, values of parameter P120, P121, P122, P125 and P126 are machine tool coordinate. So, when set the soft limit coordinate values need to first back to the machine tool zero point, and then set the corresponding limit coordinates.

While the soft limit is limited by absolute coordinate, values of parameter P120, P121, P122, P125 and P126 are absolute coordinate.

3.9 Backlash compensation

1. Parameters:

P013 Bit4 : The X axis backlash compensation function enabled

P013 Bit5 : The Y axis backlash compensation function enabled

P013 Bit6 : The Z axis backlash compensation function enabled

P067 : The X axis backlash, unit : um

P068 : The Y axis backlash, unit : um

P069 : The Z axis backlash, unit : um

P065 : The backlash compensation speed upper limit, unit :mm/min ;

P066 : The backlash compensation time constant

2. Description:

Before the backlash compensation, need to set P067, P068, P069 based on the actual measuring the clearance value of each axis ,at the same time, according to the compensation axis to set parameter P013 corresponding bit.Parameter P066 set the acceleration of compensation, the smaller the parameter P066,the faster the compensate speed, axial feeding speed to beat big;the greater the parameter P066,the slower the compensate speed, axial feeding speed to beat small.Parameter P065 for setting speed upper limit compensation, the larger the value, the faster the maximum compensation.you can adjust the para P065 and P066 based on the actual characteristics of the machine tool.

3.10 Timing and counting

See the second sheet first chapter section 1.3.1

3.11 M26 rotation function

See the first sheet third chapter section 3.1.15.

3.12 Parameter storage function

See the second sheet tenth chapter section 2.

3.13 Keyboard dignose function

See the second sheet first chapter section 1.3.8.

3.14 Print screen function

Insert U disk, set the need to print interface to the current interface, press the soft keyboard print screen, wait a moment, when the lower right corner of time operate normally, the screen has been printed and output to the usb drive, as BMP format picture.

3.15 Handwheel try cut function

Choose the handwheel buttons, press F3 again into the handwheel try cutting mode.

when program is running, the tool path is controlled by the process, movement speed controlled by the handwheel input, the speed is directly proportional to the speed of the input screw handwheel.

Select the handwheel try cut key, pulse rate can be setted, the selection of shaft of external wheel should be set in the OFF position. The actual speed is the product of the handwheel and programming of the input speed rate.

3.16 Brightness adjusting function

System working screen brightness adjusting parameters for P213 P214, when the system work normally the P213 parameters to be choosed for 100, when the system is working in the evening to select P214 parameters for 50,the evening time interval referred for the night time 19:00-7:00.

Chapter 4 Parameter

4.1 Description

4.1.1 Bit parameters

Bit parameters: P001 ~ P024 are bit parameters, the left for the highest Bit7, followed by Bit6, Bit5, Bit4 and Bit3, Bit2, Bit1,Bit0 ,there are all 8 bits, each one has its special significance.

Make the cursor up and down or the pagedown and pagesup to choose parameters press the left and right cursor key, flashing cursor move a bit in turn, in turn, at the same time the bottom of the screen shows the details of the bit.

Parameter details display row has two lines, one line is the abbreviation of the all bits parameters,another line is the detail display in Chinese of each bit.

4.1.2 Data parameters

Data parameters: P025 ~ P288 are data parameters, each parameter represents a specific meaning. Make the cursor up and down or the pageup and pagedown to choose parameters, at the bottom of the screen displays the current function. Details parameter display line only for one line.

Need to quickly find a parameter,at the parameter Setting page, enter the P * * *,after P is a parameter number. Such as: need to look at 185 parameter, input: P185, the cursor on the page selecte number 185.

See the appendix 1 for detail parameters.

Chapter 5 Monitor

5.1 I/O fixed address

5.1.1 Import state

001 ~ 009 state parameter display 54 way input port state, each row shows eight way input port.press the up and down keys, flashing cursor move a line, at the same time in the bottom of the screen shows the current input port definition.

Diagnosis X001	7	6	5	4	3	2	1	0
Terminal	CN61_08	CN61_07	CN61_06	CN61_05	CN61_04	CN61_03	CN61_02	CN61_01
Port number	1	2	3	4	5	6	7	8
Serial number	XIN10	XIN9	XIN8	XIN7	XIN6	XIN5	XIN4	XIN3
Macro Definition	#1001	#1002	#1003	#1004	#1005	#1006	#1007	#1008
Signal name	T05/OV1	PRES	ESP	DITW	DECX	DIQP	SP	SAGT

Instructions: T05/OV1 : tool input signal/ Ratio switch 1 input signal

PRES : Press Test Input Signal

ESP : ESP

DITW : TailStock Control Input

DECX : X Axis Dec Signal

DIQP : Chuck Control Input

SP : External SP

SAGT : Safety Door Signal

Diagnosis X002	7	6	5	4	3	2	1	0
Terminal	CN61_16	CN61_15	CN61_14	CN61_13	CN61_12	CN61_11	CN61_10	CN61_09
Port number	9	10	11	12	13	14	15	16
Serial number	XIN33	XIN17	XIN16	XIN15	XIN14	XIN13	XIN12	XIN11
Macro Definition	#1009	#1010	#1011	#1012	#1013	#1014	#1015	#1016
Signal name	T01	M42I	M41I	ST	DECZ	T08/OV4	T07/OV3	T06/OV2

Instructions: T01 : Tool signal

M42I : Auto Spindle Rearshift No.2 Tool Ready

M41I : Auto Spindle Rearshift No.1 Tool Ready

ST : External ST

DECZ : Z Axis Dec Signal

T08/OV4 : tool input signal / Ratio switch 4 input signal

T07/OV3 : tool input signal / Ratio switch 3 input signal

T06/OV2 : tool input signal / Ratio switch 2 input signal

Diagnosis X003	7	6	5	4	3	2	1	0
Terminal	CN61_36	CN61_35	CN61_34	CN61_33	CN61_32	CN61_31	CN61_30	CN61_29
Port number	17	18	19	20	21	22	23	24
Serial number	XIN27	XIN28	XIN29	XIN30	XIN31	XIN32	XIN18	XIN20
Macro Definition	#1017	#1018	#1019	#1020	#1021	#1022	#1023	#1024
Signal name	AEY/BDT	TCP	DEC5	DEC4	DECY	T04	T03	T02

Instructions: AEY/BDT : External Jump

TCP : Toolpost Lock Signal

DEC5 : C Axis Dec Signal

DEC4 : A Axis Dec Signal

DECY : Y Axis Dec Signal

T04 : Tool signal

T03 : Tool signal

T02 : Tool signal

Diagnosis X004	7	6	5	4	3	2	1	0
Terminal	CN61_44	CN61_43	CN61_42	CN61_41	CN61_40	CN61_39	Reserve	CN61_37
Port number	25	26	27	28	29	30		32
Serial number	XIN49	XIN48	XIN22	XIN23	XIN24	XIN25		XIN26
Macro Definition	#1025	#1026	#1027	#1028	#1029	#1030		#1032
Signal name	EZ	AEX	SKIP	NQPJ	WQPJ	LIM-		LIM+

Instructions: AEZ : Z Axis Tool Posi Ready (G37)

AEX : X Axis Tool Posi Ready (G36)

SKIP : G31 Jump Period"

NQPJ : Inner Chuck Clamp/Outer Chuck Loose Ready

WQPJ : Inner Chuck Loose/Outer Chuck Clamp Ready

LIM- : Negtive Limit

LIM+ : Positive Limit

Diagnosis X005	7	6	5	4	3	2	1	0
Reserve all								

Diagnosis X006	7	6	5	4	3	2	1	0
Terminal						CN15_08	CN15_06	CN15_05
Port number						46	47	48
Serial number						XIN51	XIN50	XIN53
Macro Definition						#1046	#1047	#1048
Signal name	***	***	***	***	***	X5.2	X5.1	X5.0

Instructions: X5.2/X5.1/X5.0 : SelfDefi Input

Diagnosis X007	7	6	5	4	3	2	1	0
Terminal		CN31_07	CN31_23	CN31_22	CN31_09	CN31_08	CN31_06	CN31_05
Port number		50	51	52	53	54	55	56
Serial number		XIN52	XIN45	XIN47	XIN44	XIN43	XIN42	XIN40
Macro Definition		#1050	#1051	#1052	#1053	#1054	#1055	#1056
Signal name	***	AHAN	X100	X10	X1	ZHAN	YHAN	XHAN

Instructions: AHAN : A HandWheel Axis

X100/X10/X1 : The outer wheel incremental input signal

ZHAN : Z HandWheel Axis

YHAN : Y HandWheel Axis

XHAN : X HandWheel Axis

Diagnosis X008	7	6	5	4	3	2	1	0
Terminal			CN15_07	CN15_10	CN14_03	CN13_03	CN12_03	CN11_03
Port number			59	60	61	62	63	64
Serial number			XIN35	XIN41	XIN46	XIN0	XIN1	XIN2
Macro Definition			#1059	#1060	#1061	#1062	#1063	#1064
Signal name	***	***	CRDY	CPC	APC	ZPC	YPC	XPC

Instructions: CRDY : C Axis Ready

CPC : C Axis Zero Point

APC : A Axis Zero Point

ZPC : Z Axis Zero Point

YPC : Y Axis Zero Point

XPC : X Axis Zero Point

Diagnosis X009	7	6	5	4	3	2	1	0
Terminal	CN55_06	CN55_05		CN15_04	CN14_05	CN13_05	CN12_05	CN11_05
Port number	65	66		68	69	70	71	72
Serial number	XIN21	XIN19		XIN34	XIN39	XIN36	XIN37	XIN38
Macro Definition	#1065	#1066		#1068	#1069	#1070	#1071	#1072
Signal name	TRIL	TRIR	***	CALM	AALM	ZALM	YALM	XALM

Instructions: TRIL : TriSw Left

TRIR : TriSw Right

CALM : C axis driver alarm

AALM : A axis driver alarm

ZALM : Z axis driver alarm

YALM : Y axis driver alarm

XALM : X axis driver alarm

***Reserved.

note : the following signal reserved

T08: tool overheat detection

TO7: Yantai toolpost preassigned input

T06 : Yantai toolpost gating signal

5.1.2 Output port state

013 ~ 020 state parameter display 48 way input port state, each row shows eight way input port.Press left or right key, a flashing cursor movement, at the same time in the bottom of the screen shows the current output definition.At the cursor blink press "0" or "1", output "0" to the corresponding output signal (disconnect with ground signal) or "1" signal (connect with ground signal). press the up and down keys, flashing cursor move a line, the bottom of the screen shows the current line output definition.

Diagnosis Y0000	7	6	5	4	3	2	1	0
Terminal	CN62_08	CN62_07	CN62_06	CN62_05	CN62_04	CN62_03	CN62_02	CN62_01
Port number	1	2	3	4	5	6	7	8
Serial number	MO7	MO8	MO9	MO10	MO11	MO12	MO13	MO14
Macro Definition	#1101	#1102	#1103	#1104	#1105	#1106	#1107	#1108
Signal name	SPZD	Y0.6	M05	M04	M03	Y0.2	M32	M08

Instructions: SPZD : Spindle Brake

M05 : Spindle Stop

M04 : Spindle ClockWise

M03 : Spindle Counter ClockWise

M32 : Grease Output

M08 : Cool Output

Y0.6/Y0.2 : SelfDefi

Diagnosis Y0001	7	6	5	4	3	2	1	0
Terminal	CN62_16	CN62_15	CN62_13	CN62_14	CN62_12	CN62_11	CN62_10	CN62_09
Port number	9	10	11	12	13	14	15	16
Serial number	MO15	MO0	MO2	MO1	MO3	MO4	MO5	MO6
Macro Definition	#1109	#1110	#1111	#1112	#1113	#1114	#1115	#1116
Signal name	TL-	TL+	DOQPJ/ M10	DOQPS/ M11	S4/M44	S3/M43	S2/M42	S1/M41

Instructions: TL+ : ToolPost ClockWise(Positive)

TL- : ToolPost ClockWise(Negtive)

DOQPJ/M10 : Chuck Clamp

DOQPS/M11 : Chuck Loose

S4/M44 : Spindle Gear 4

S3/M43 : Spindle Gear 3

S2/M42 : Spindle Gear 2

S1/M41 : Spindle Gear 1

Diagnosis Y0002	7	6	5	4	3	2	1	0
Terminal	CN62_36	CN62_35	CN62_34	CN62_33	CN62_32	CN62_31	CN62_30	CN62_29
Port number	17	18	19	20	21	22	23	24
Serial number	MO21	MO20	MO19	MO18	MO17	MO16	MO31	MO29
Macro Definition	#1117	#1118	#1119	#1120	#1121	#1122	#1123	#1124
Signal name	Y2.7	DOTWS / M79	DOTWJ/ M78	RLAMP	GLAMP	YLAMP	INDXS	TZD

Instructions: DOTWS/M79 : TailStock Retreat

DOTWJ/ M78 : TailStock Forward

RLAMP : Trichromatic lamp-Red

GLAMP : Trichromatic lamp-Green

YLAMP : Trichromatic lamp-Yellow

INDXS : Tool table for indexing coil

TZD : tool brake output signal

Y2.7 : SelfDefi

Diagnosis Y0003	7	6	5	4	3	2	1	0
Terminal	CN62_44	CN62_43	CN62_42	CN62_41	CN62_40	CN62_39	CN62_38	CN62_37
Port number	25	26	27	28	29	30	31	32
Serial number	MO30	MO28	MO27	MO26	MO25	MO24	MO23	MO22
Macro Definition	#1125	#1126	#1127	#1128	#1129	#1130	#1131	#1132
Signal name	Y3.7	Y3.6	Y3.5	Y3.4	Y3.3	Y3.2	Y3.1	Y3.0

Instructions: Y3.7~Y3.0 : SelfDefi

Diagnosis Y0004	7	6	5	4	3	2	1	0
Reserve all								

Diagnosis Y0005	7	6	5	4	3	2	1	0
Terminal					CN15_23	CN15_22	CN15_21	CN15_20
Port number					45	46	47	48
Serial number					MO37	MO36	MO35	MO34
Macro Definition					#1145	#1146	#1147	#1148
Signal name	***	***	***	***	Y5.3	Y5.2	Y5.1	Y5.0

Instructions: Y5.3/Y5.2/Y5.1/Y5.0 : SelfDefi

Diagnosis Y0006	7	6	5	4	3	2	1	0
Terminal				CN15_19	CN14_07	CN13_07	CN12_07	CN11_07
Port number				MO52	MO53	MO54	MO55	MO56
Serial number				MO33	MO44	MO38	MO40	MO42
Macro Definition				#1152	#1153	#1154	#1155	#1156
Signal name	***	***	***	CEN	AEn	ZEn	YEn	XEn

instructions : CEN : C Enable

AEn : A Enable

ZEn : Z Enable

YEn : Y Enable

XEn : X Enable

Diagnosis Y0007	7	6	5	4	3	2	1	0
Terminal	CN31_26	CN31_25		CN15_18	CN14_06	CN13_06	CN12_06	CN11_06
Port number	57	58		60	61	62	63	64
Serial number	MO47	MO46		MO32	MO45	MO39	MO41	MO43
Macro Definition	#1157	#1158		#1160	#1161	#1162	#1163	#1164
Signal name	Y7.7	Y7.6	***	CSET	ASET	ZSET	YSET	XSET

instructions : CSET : Main Axis POS/SPEED Change

ASET : A Pulse Forbid

ZSET : Z Pulse Forbid

YSET : Y Pulse Forbid

XSET : X Pulse Forbid

Y7.7/Y7.6 : SelfDefi

5.1.3 Date state

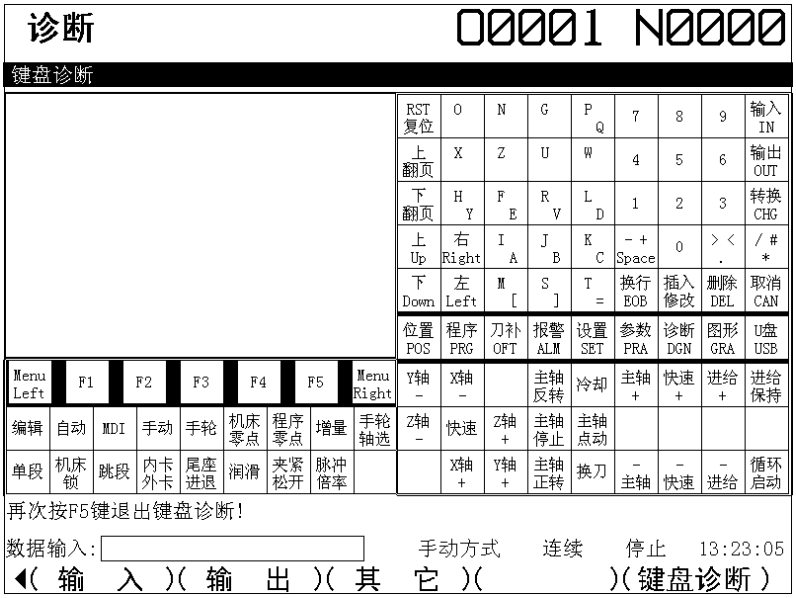
0 2 5	X Axis Output Pulse(Unit:a)
0 2 6	Y Axis Output Pulse(Unit:a)
0 2 7	Z Axis Output Pulse(Unit:a)
0 2 8	A Axis Output Pulse(Unit:a)
0 2 9	C Axis Output Pulse(Unit:a)
0 3 0	Spindle Speed(Unit:Revolutions Per Minute)
0 3 1	Spindle Encoder Num(Unit:Nums Per Revolution)
0 3 2	Spindle Encoder Feedback Pulse Num(Unit:a)
0 3 3	Spindle Head Pulse Num(Unit:a)
0 3 4	HandWheel Output Pulse Num(Unit:a)
0 3 5	Spindle Analog 1 Voltage Output(Unit:V)
0 3 6	Spindle Analog 2 Voltage Output(Unit:V)

Press the cursor up and down, move the cursor to the 025 ~ 029 lines, long press X,Z,Y,A or C key can cancel the corresponding pulse zero.

31 spindle encoder line number,the value can display properly only need the spindle rotation more than 3.

5.2 Keyboard dignose

Press the diagnostic key, and then press the keyboard diagnosis (F5), enter the keyboard diagnosis function page. This interface corresponding to the CNC on position and function of on keyboard, by pressing the keyboard, interface display a red light corresponding to the key pressed, such as: press clamp/loose button, if the button is normal, the corresponding keyboard will show a red light, if it has no appearance, there is a problem, you should resolve the problem according to the situation.



Chapter 6 Pitch compensation parameters

6.1 Pitch compensation function

Leadscrew is used to compensate the the uneven error caused by screw pitch own precision , system each axis can be entered up to 256 compensation points.

6.2 The steps of the pitch compensation parameters setted

Leadscrew X setup procedures:

1. Enter the Leadscrew interface;

2. Use the flip key and the cursor keys ← , ↑ , → , ↓Move the cursor to where need to set the parameters of Leadscrew positions;

3. Press the X key, press the number keys enter the Leadscrew value, if the input for the first time the password input box will pop-up,you need to input the correct password then enter the Leadscrew value.

4. If you need to put each axis leadscrew value zero, then press the delete key, pop-up password box, enter the correct password, the system will put all screw value zero.

Z axis is the same as above steps.

6.3 The attention about pitch error compensation

1. Input values for the spot measurement,the value that is to offset the error compensation , each point of the compensation scope is -6.000 ~ 6.000 mm,the point beyond the scope of compensation system consider that pitch error is zero.

2. Whether the X and Z axis leadscrew to be compensated decided by the parameter P002 Bit2, = 1 open leadscrew error compensation.

3. System must go back to the machine tool zero point then to compensate the leadscrew. Back to the machine tool zero point, machine tool coordinate is zero.

4. According to each axis length and precision of screw,workpiece size to determine the length and point of the axis error compensation, compensation points of each axis and the gap between the two

points determined by the parameter P072, P074, P077, P079 ,that in between the two compensation point system think screw error is linear changed.

5. The erro of each axis is measured with a laser interferometer at regular intervals starting at the machine tool zero point (set by parameter P077, P079).

6. the error to be measured will input to corresponding to the points of the leadscrew error parameters table.

7. Reboot the system, the pitch error compensation effect.

6.4 Pitch error compensation for example

For example : X axis screw effective length is 300 mm, total compensation for 100 points, the parameter P072 = 100, parameter P0077 = 300/100 = 3.With laser interferometer to measure pitch error: (first back to zero,make XP = 0).

Along -X to go to the -3 mm (system display), the actual measurement is -2.975,1 # pitch error is -0.005

Along -X to go to the -6 mm (system display), the actual measurement is -6.003,2 # pitch error is +0.003

Along -X to go to the -9 mm (system display), the actualmeasurement is -9.008,3 # pitch error is +0.008

Along -X to go to the -12 mm (system display), the actual measurement is -11.992,4 # pitch error is -0.008

Along -X to go to the -15 mm (system display), the actual measurement is -14.998,5 # pitch error is -0.002

Along -X to go to the -18 mm (system display), the actual measurement is -14.993,6 # pitch error is -0.007

…

…

Along -X to go to the -294 mm (system display), the actual measurement is -294.000,98 # pitch error is 0

Along -X to go to the -297 mm (system display), the actual measurement is -296.997,99 # pitch error is -0.003

Along -X to go to the -300 mm (system display), the actual measurement is -300.006,100 # pitch error is +0.006

write the 1 # ~ 100 # pitch error value into X pitch error table 1 ~ 100, CNC system will automatically compensate the pitch during processing

Appendix

Appendix: 1 Standard parameter list

1. Bit parameters

0	0	1	LAN		MZRN	MDSP	TKE	RAD	BUZZER	PMD

LAN : =1 : English =0 : Chinese

MZRN : =1 : Return to mach zero first =0 : No

MDSP : =1 : Simulate Control =0 : Other

TKE: =1 : Check Tool Num =0 : No

RAD : =1 : X Axis Radius =0 : Diameter

BUZZER : =1 : buzzer On =0 : buzzer Off

PMD: =1 : Open Low Vibration Mode =0 : close

Factory set : 0001 0001

0	0	2	AUN	RTCL		REMF	CCMP	SCRP	TLNEG	CLRC

AUN : =1 : Auto Serial Number In Program =0 : No

RTCL : =1 : Auto Clear Time =0 : No

REMF : =1 : Compensation Remember By X,Z =0 : No

CCMP : =1 : Tool C Compensation valid =0 : No

SCRP : =1 : Thread Pitch Compensation valid =0 : No

TLNEG : =1 : Invert TL+&TL- =0 : No

CLRC : =1 : Auto Clear Amount Power On =0 : No

Factory set : 0000 0001

0	0	3	DECA	DECZ	DECY	DECX	ENA	ENZ	ENY	ENX

DECA : =1 : A Return To Origin Dec Signal Is 1 =0 : 0

DECZ : =1 : Z Return To Origin Dec Signal Is 1 =0 : 0

DECY : =1 : Y Return To Origin Dec Signal Is 1 =0 : 0

DECX : =1 : X Return To Origin Dec Signal Is 1 =0 : 0

ENA : =1 : Output A Enable Signal Power On =0 : No

ENZ : =1 : Output Z Enable Signal Power On =0 : No

ENY : =1 : Output Y Enable Signal Power On =0 : No

ENX : =1 : Output X Enable Signal Power On =0 : No

Factory set : 0000 0101

0	0	4		M30	PPD	CM98	HDA	HDZ	HDY	HDX

M30 : =1 : M30 Close Spindle,Cool =0 : Not

PPD : =0 : Set Relative coordinates =1 : Not

CM98 : =0 : Into the system beyond the standard M, T code,system alarm

=1 : Into the system beyond the standard M, T code does not produce alarm, and automatically to invoke a subroutine

If CM98 ＝1, When performing standard outside Mxx code, the system automatically call subroutine O90xx ;

When performing the T20 ~ T99 code, the system automatically call subroutine O92 # #.

Such as executing M59, the system automatically call subroutine O9059 ;

Such as executing T59, the system automatically call subroutine O9259;

Note 1: when performing non-standard M, T code, must be incorporated into the corresponding subroutine. Otherwise it will alarm 051.

Note 2: non-standard M, T code can run in the MDI mode (see details in MDI operation description)

Note 3: in the corresponding subroutine that can be programed into axis movement instruction, also can to control the output point (closing and opening), can also according to the input signal to jump or to cycle, or a particular input signals as the end of the M/T.

HDA : =1 : A Axis Bottom Left P =0 : Upper Right P

HDZ : =1 : Z Axis Left P =0 : Right P

HDY : =1 : Y Axis Upper Left P =0 : Bottom Right P

HDX : =1 : X Axis Up P =0 : Down P

Factory set : 0000 0000

0	0	5	EHLS	CPZ	IHCD	CMZ	ZMA	ZMZ	ZMY	ZMX

EHLS : =1 : External stop alarm signal high level effective =0 : low level effective

CPZ : =0 : Can Go To Program Zero =1 : No

IHCD : =1 : Tool compen. Cancel After Return Zero =0 : No

CMZ : =1 : Mode A Back To Machine tool Zero =0 : B,C

ZMA : =1 : A Axis Neg To Machine tool Zero =0 : P

ZMZ : =1 : Z Axis Neg To Machine tool Zero =0 : P

ZMY : =1 : Y Axis Neg To Machine tool Zero =0 : P

ZMX : =1 : X Axis Neg To Machine tool Zero =0 : P

Factory set : 0000 0000

0	0	6	ZCA	ZCZ	ZCY	ZCX	RTMA	RTMZ	RTMY	RTMX

ZCA : =1 : A Axis Return Zero Mode Is C =0 : B（Need slowdown switch and zero signal）

ZCZ : =1 : Z Axis Return Zero Mode Is C =0 : B（Need slowdown switch and zero signal）

ZCY : =1 : Y Axis Return Zero Mode Is C =0 : B（Need slowdown switch and zero signal）

ZCX : =1 : X Axis Return Zero Mode Is C =0 : B（Need slowdown switch and zero signal）

RTMA : =1 : A Axis Can Return Machine tool Zero =0 : No

RTMZ : =1 : Z Axis Can Return Machine tool Zero =0 : No

RTMY : =1 : Y Axis Can Return Machine tool Zero =0 : No

RTMX : =1 : X Axis Can Return Machine tool Zero =0 : No

Factory set : 0101 0101

0	0	7	BKA	BKZ	BKY	BKX	DIRA	DIRZ	DIRY	DIRX

BKA : =1 : En A Ax Open Motor Brake =0 : No

BKZ : =1 : En Z Ax Open Motor Brake =0 : No

BKY : =1 : En Y Ax Open Motor Brake =0 : No

BKX : =1 : En X Ax Open Motor Brake =0 : No

DIRA : =1 : Direction Of A Motor P =0 : No

DIRZ : =1 : Direction Of Z Motor P =0 : No

DIRY : =1 : Direction Of Y Motor P =0 : No

DIRX : =1 : Direction Of X Motor P =0 : No

Factory set : 0000 0000

0	0	8	LMT－	LMT+	HAND	HDSTL	ALMA	ALMZ	ALMY	ALMX

LMT－ : =1 : Not Check N Hardlimit =0 : No

LMT＋ : =1 : Not Check P Hardlimit =0 : No

HAND : =1 : Wheel Clockwise As Negtive =0 : P

HDSTL : =1 : Handheld Unit =0 : General Handwheel

ALMA: =1 : A Axis Driver Alarm Signal High =0 : Low

ALMZ: =1 : Z Axis Driver Alarm Signal High =0 : Low

ALMY : =1 : Y Axis Driver Alarm Signal High =0 : Low

ALMX : =1 : X Axis Driver Alarm Signal High =0 : Low

Factory set : 0000 0101

0	0	9	ACC4	ACC3	ACC2	ACC1	SEGE	SEGT	TSGN	TCPS

ACC4 , ACC3 , ACC2 , ACC1 : To control the acc/dec together

Such as : 0000 : Liner Acc/Dec (begin)

0001 : S Curve Acc/Dec (begin)

0100 : Liner Acc/Dec (end)

1000 : S Curve Acc/Dec (end)

SEGE : =1 : Default G61 Mode =0 : G64

SEGT : =1 : Speed Transe Low_Lim Static =0 : Dynamic

TSGN : =1 : Toolpost In-Position Signal L =0 : H

TCPS : =1 : Toolpost Lock Signal L =0 : H

Factory set : 0000 0010

0	1	0	RPDK	MLCK	MSS	TSS2	TSS1	MSSR	G0&T	TSS0

RPDK : =1 : Acc-Key Simulation State =0 : No

MLCK : =1 : Open Machine tool Lock Function =0 : No

MSS : =1 : M05 Close S1~S4 =0 : No

TSS2: =1: 2nd tool is SWD Tllopast =0: Electric Toolpost

TSS1: =1: SWD Toolpost =0: By TSS0 decide the toolpost type

MSSR: =1: RESET Close S1~S4 =0 : No

G0&T : =1 : G0/T Can't Execute In Same Time =0 : No

TSS0 : =1 : SWD Toolpost =0 : Electric Toolpost

Factory set : 1000 0010

0	1	1	ANGA	ANGZ	ANGY	ANGX	ZERO	THDM	SVRP	

ANGA : =1 : A Axis Angle Programing =0 : Length

ANGZ : =1 : Z Axis Angle Programing =0 : Length

ANGY : =1 : Y Axis Angle Programing =0 : Length

ANGX : =1 : X Axis Angle Programing =0 : Length

ZERO : =1 : Return Mach Zero Need Keep Pressing =0 : Not

THDM : =1 : Open Precision Thread Mode =0 : Not

SVRP : =1 : Open M26~M28 Function =0 : Not

Factory set : 0000 0100

0	1	2	WAR2	WAR1	WTP2	WTP1	WA2	WA1	ANG	RSJG

WAR2 : =1 : External Alert 2 Available =0 : No

WAR1 : =1 : External Alert 1 Available =0 : No

WTP2 : =1 : Ext Alert 2 Appear Pro Stopped =0 : No

WTP1 : =1 : Ext Alert 1 Appear Pro Stopped =0 : No

WA2 : =1 : Ext Alert 2 High Level Ava =0 : No

WA1 : =1 : Ext Alert 1 High Level Ava =0 : No

ANG : =1 : Simulate Amount -10V~10V =0 : 0~10V

RSJG : =0 : Reset Close Spindle,Cool,Grease =1 : No

Factory set : 0000 0000

0	1	3	CLRA	CLRZ	CLRY	CLRX	TRIM	JLB	RNMD	TRSW

CLRA : ＝1 : Open A Backlash Compensation ＝0 : No

CLRZ : ＝1 : Open Z Backlash Compensation ＝0 : No

CLRY : ＝1 : Open Y Backlash Compensation ＝0 : No

CLRX : ＝1 : Open X Backlash Compensation ＝0 : No

TRIM : = 1 : Three-Switch Pause To Run Need Start Key = 0 : No

JLB : = 1 : Macro Commom Is Cleared = 0 : No

RNMD : = 1 : Default Auto Mode =0 : Handle

TRSW : = 1 : Turn Off Three-Switch Function =0 : No

Factory set : 0101 1001

0	1	4	KEY1	SKEY	KEY2	LPKY	SLT	MOT	MSP	MST

KEY1 : =1 : Program Switch Is On =0 : Off

SKEY : =1 : Shield Screen Program Switch =0 : Not

KEY2 : =1 : Parameter Switch Is On =0 : Off

LPKY : =1 : Shield Cycle Start Key Function =0 : Not

SLT: =1: Software Limit Mac Coor =0: Abs

MOT : =1 : Not Check Software Limit =0 : Not

MSP : =1 : Shielding External SP Signal =0 : No

MST : =1 : Shielding External ST Signal =0 : No

Factory set : 1010 1111

0	1	5	EEMG	OWAR	OM30	OSTM	EES	KPDW	CRWS	RNSC

EEMG : =1 : Shielding External Emergency =0 : No

OWAR : =1 : Output Alarm Is Available =0 : No

OM30 : =1 : Output M30 Signal Is Available =0 : No

OSTM : =1 : Output MST Signal Is Available =0 : No

KPDW : =1 : Check Chuck In-Position Signal =0 : No

EES : =1 : The outside feed rate selection switch on =0 : off

CRWS : =1 : In Auto mode, Chuck Can Run if Spindle Run =0 : No

RNSC: =0 : Default Continue Mode =1: Single

Factory set : 0000 0000

0	1	6	NWKP	SFDV	SFDR	STCS	CHPR	SLTW	SLSP	SLQP

NWKP : =1 : Chuck Holds Outer =0 : Inner

SFDV : =1 : Safety Door High Signal =0 : Low

SFDR : =1 : Check Safety Door signal When Cycle Start =0 : No

STCS : =1 : Single to Continue Need Start Key =0 : No

CHPR : =1 : Chuck Can Action in Pause =0 : No

SLTW : =1 : Tailstock Function Available =0 : No

SLSP : =1 : Check Chuck Loosen State =0 : No

SLQP : =1 : Chuck Function Available =0 : No

Factory set : 0000 0000

0	1	7	FZRO	APRS			G31P			

FZRO : =1 : Back To Float Zero Coor Clear =0 : No

APRS : =1 : Set Coordinate Return Zero =0 : No

G31P: =1: G31 Stop Right Now =0 : No

Factory set : 0000 0000

0	1	8								

Factory set : 0000 0000

0	1	9	RHD5	SPOR	RSCS	CZSE	CALM	CDIR		

RHD5: =1: Allow Servo Spindle =0: No

SPOR: =1 : Spindle Position Available =0 : No

RSCS: =0 : ESP Return Speed State =1 : No

CZSE: =1 : Zero Pulse Is CPC Signal =0 : No

CALM: =1 : C Axis Alert Is High Level =0 : No

CDIR: =1 : C Axis Position Pos =0 : No

Factory set : 0000 0000

0	2	0	PRSS				SMSE	SALM	CRHL	CRDY

PRSS: =1: Enable Servo Ready Stop When C/V Swicht =0: Disable

SMSE: =1: Servo Maner Engage Plus Position =0: No

SALM: =1: ESP Close Servo Spindle =0: No

CRHL: =1: Servo Spindle Ready High Level =0: No

CRDY: =1: Check Servo Spindle Ready =0: No

Factory set : 0000 1000

0	2	1							MPOE	CSG

MPOE: =0: M01,M35 Time Out Enter Pause State =1: Stop State

CSG: =1: Check Gear In-Position Signal =0: No

Factory set : 0000 0000

0	2	2								

Factory set : 0000 0000

0	2	3								

Factory set : 0000 0000

0	2	4								

Factory set : 0000 0000

2. The data parameter

Feed axis, speed and other related parameters

025	Command Multiply For X Axis	1	1～65535
026	Command Multiply For Y Axis	1	1～65535
027	Command Multiply For Z Axis	1	1～65535
028	Command Multiply For A Axis	1	1～65535
029	Command Multiply For C Axis	1	1～65535
030	Command Denominator Of X Axis	1	1～65535
031	Command Denominator Of Y Axis	1	1～65535
032	Command Denominator Of Z Axis	1	1～65535
033	Command Denominator Of A Axis	1	1～65535
034	Command Denominator Of C Axis	1	1～65535

Set X , Y , Z , A axis electronic gear ratio

035	Speed At Rapid In XAxis（mm/min）（Auto Run）	7600	1～60000
036	Speed At Rapid In Y Axis（mm/min）（Auto Run）	7600	1～60000
037	Speed At Rapid In Z Axis（mm/min）（Auto Run）	7600	1～60000
038	Speed At Rapid In A Axis（mm/min）（Auto Run）	7600	1～60000
039	Speed At Rapid In CAxis（mm/min）（Auto Run）	7600	1～60000

Set X , Y , Z , A axis G00 speed

040	Time Of Liner Acc/Dec In X Axis（Rapid）	100	10～4000
041	Time Of Liner Acc/Dec In Y Axis（Rapid）	100	10～4000
042	Time Of Liner Acc/Dec In Z Axis（Rapid）	100	10～4000
043	Time Of Liner Acc/Dec In A Axis（Rapid）	100	10～4000
044	Time Of Liner Acc/Dec In C Axis（Rapid）	300	10～4000

Set X , Y , Z , A axis when fast moving the acceleration value of the linear Acc/Dec speed.

045	Time Of S Curve Acc/Dec In X Axis（Rapid）	100	10～4000
046	Time Of S Curve Acc/Dec In Y Axis（Rapid）	100	10～4000
047	Time Of S Curve Acc/Dec In Z Axis（Rapid）	100	10～4000
048	Time Of S Curve Acc/Dec In A Axis（Rapid）	100	10～4000
049	Time Of S Curve Acc/Dec In C Axis（Rapid）	300	10～4000

Set the X, Y, Z, A axis when fast moving the S type speed Acc/Dec acceleration coefficient.

050	Upper Limit For Cutting Speed（mm/ms）	8000	1～60000

Limit the highest feed of cutting feed speed, this parameter limit the G01 highest cutting speed and highest cutting feed speed of the composite cycle.

051	Time Of Linear Acc/Dec For Cutting	150	10～4000

The linear acc/dec time constant of cutting feed.

052	Time Of S Curve Acc/Dec For Cutting	150	10～4000

Set the X and Z axis of cutting feed speed of the S type acc/dec coefficients.

053	Lower Speed Limit For F0 (mm/min)	100	0～60000

Fast moving set the minimum speed, its speed gear is Fo, 25%, 50%, 75%, 100% ;

054	L Lower Cutting Speed Limit(mm/min)	200	0～60000

When the program is setted up for the period of transition between speed mode (G64), and the parameter P009 Bit2 to 1 (speed transition floor to static mode), is decided by the parameters of transition between two cutting section speed limit; The smaller this value, the smaller the transition arc, the actual trajectory more approximate programming outline; The axis is larger, the bigger the transition arc.

055	L Percentage Cutting Speed Limit	100	0～100

When the program is setted up for the period of transition between speed mode (G64), and the parameter P009 Bit2, 0 (transition rate lower limit for the dynamic mode), when the speed of the transition between the two cutting section of the lower point decided by this parameter and the current section of the programming cutting speed ; The smaller this value, the smaller the transition arc, the actual trajectory more approximate programming outline; The axis is larger, the bigger the transition arc.

056	Transition Dec For Cutting	4	1～30

057	Fast Speed for X Axis（mm/min）（Jog Mode）	3000	1～60000
058	Fast Speed for Y Axis（mm/min）（Jog Mode）	3000	1～60000
059	Fast Speed for Z Axis（mm/min）（Jog Mode）	3000	1～60000
060	Fast Speed for A Axis（mm/min）（Jog Mode）	3000	1～60000
061	Fast Speed for C Axis（mm/min）（Jog Mode）	3000	1～60000

Set the X, Y, Z, A axis manually quick speed (when press the fast button)

062	Max Rev For Feed Per Revolution（μm/rev）	1000	0～1000

063	L Limit In Constant Linear Speed（rpm）	100	1～60000
064	H Limit In Constant Linear Speed（rpm）	2000	1～60000

065	Backlash Compensation Speed（mm/min）	200	0～60000

066	Acc/Dec Time In Backlash Compensation	300	10～4000

067	X Axis Blacklash Compensation（um）	0	－9999～9999
068	Y Axis Blacklash Compensation（um）	0	－9999～9999
069	Z Axis Blacklash Compensation（um）	0	－9999～9999
070	A Axis Blacklash Compensation（um）	0	－9999～9999
071	Reserved	0	——

072	X Pitch Error Compensation Points	0	0～256
073	Y Pitch Error Compensation Points	0	0～256
074	Z Pitch Error Compensation Points	0	0～256
075	A Pitch Error Compensation Points	0	0～256
076	Reserved	0	--

077	X Pitch Error Compensation Interval（μm）	0	0～10000
078	Y Pitch Error Compensation Interval（μm）	0	0～10000
079	Z Pitch Error Compensation Interval（μm）	0	0～10000
080	A Pitch Error Compensation Interval（μm）	0	0～10000
081	Reserved	0	--

082	Delay Time Off Brake Relative Servo(×4ms)	5	4 ～ 10000
083	Delay Time On Brake Relative Servo (×4ms)	30	0～10000

when need motor brake control,the system output servo enabled signal, delay the time of the parameters, then output motor brake open signal.

084	X Axis Brake Control Outport	0	0～72
085	Y Axis Brake Control Outport	0	0～72
086	Z Axis Brake Control Outport	0	0～72
087	A Axis Brake Control Outport	0	0～72
088	Reserved	0	--

The rotation axis setted parameters

089	Lead Screw Of Rotation Axis（mm）	10	0～10000
090	Max Speed Of Rotation Axis（rpm）	500	0～10000
091	Set Rotation Axis X:0 Z:1 Y:2 A:3	1	0～3

092	The Amount Of Rotation Axis's Movement	360000	0 – 360000

Unit 0.001mm

…… ……

099	Reserved	0	--

Back to zero correlation parameters

100	Lower feedrate In Origin Return（mm/min）	120	1～60000

This parameter to zero common way B/C mode; In the process of back to the machine tool zero point, after receive the deceleration switch signals, looking for servo drive zero pulse (Z pulse) signal (B) or (C) location of switch signal. Suggest the value is not greater than 200.

101	X Axis Rapid Go Back To Reference Point (mm/min)	3000	1～60000
102	Y Axis Rapid Go Back To Reference Point (mm/min)	3000	1～60000
103	Z Axis Rapid Go Back To Reference Point (mm/min)	3000	1～60000
104	A Axis Rapid Go Back To Reference Point (mm/min)	3000	1～60000
105	C Axis Rapid Go Back To Reference Point (mm/min)	3000	1～60000

Back to zero mode, before checking the slowdown signal, the speed of the machine tool fast move to the direction of reference point.

106	X Is Set After Return Zero (um)	0	－99999999～99999999
107	Y Is Set After Return Zero (um)	0	－99999999～99999999
108	Z Is Set After Return Zero (um)	0	－99999999～99999999
109	A Is Set After Return Zero (um)	0	－99999999～99999999
110	Reserved	0	--

To complete the program/machine tool to zero, X, Y, Z, A axis coordinate (absolute) value, the default value is 0.

111	Pro Beginning X Coor (um)	0	－9999999～9999999
112	Pro Beginning Y Coor (um)	0	－9999999～9999999
113	Pro Beginning Z Coor (um)	0	－9999999～9999999
114	Pro Beginning A Coor (um)	0	－9999999～9999999
115	Reserved	0	--

Used to set back to the program zero point coordinates.

116	Reserved	0	--

…… ……

119	Reserved	0	--

Machine tool safety

120	X Positive Limit (um)	99999999	－99999999～99999999
121	Y Positive Limit (um)	99999999	－99999999～99999999
122	Z Positive Limit (um)	99999999	－99999999～99999999
123	A Positive Limit (um)	99999999	－99999999～99999999
124	Reserved	0	——
125	X Negtive Limit (um)	- 99999999	－99999999～99999999
126	Y Negtive Limit (um)	- 99999999	－99999999～99999999
127	Z Negtive Limit (um)	- 99999999	－99999999～99999999
128	A Negtive Limit (um)	- 99999999	－99999999～99999999
129	Reserved	0	——

Soft limit scope of coordinates of the axis, when the coordinate beyond the upper and lower limit, alarm system, and slow down to stop.

The spindle related parameters

130	Spindle Analog Adjustment	1024	1～65535

When threading, rigid tapping and spindle speed display functions such as all need this parameter;after the spindle rotation, enter diagnose interface for checking Spindl encode number.

131	Gear Ration:Spindle Gear Number	1	1～65535
132	Gear Ration:Encoder Gear Number	1	1～65535

For the ration between the encoder and the spindle is not 1,in order to accurately display the current spindle speed.

133	Spindle Stall Numbers	2	0～4

Set the spindle effective number of gear, the maximum is 4; This parameter controls the effective number of S1 ~ S4 output.

134	Spindle Speed For 10V,1st Gear (rpm)	2000	0～60000
135	Spindle Speed For 10V,2nd Gear (rpm)	2000	0～60000
136	Spindle Speed For 10V,3rd Gear (rpm)	1000	0～60000
137	Spindle Speed For 10V,4th Gear (rpm)	500	0～60000

Used to calibrate the machine tool high speed gear analog output.

138	Spindle Analog Adjustment (－10～10)	0	－10～10

used to accurate spindle analog control applications. Due to loss of electric transmission and discrete deviation device, when the set of spindle speed value corresponding to the analog output trace deviation can use this parameter to adjust. Input range - 10 ~10 (corresponding to the full range 10V, the voltage adjusting range of about -0.4 V ~ 0.4 V).

139	Reserved	0	——

140	Spindle Shift Delay Time（×4ms）	200	0～10000

Used to modify the time intervals when spindle shift, close the gear before to open the setted gear, the default time is 800 ms.

141	Spindle Turn On Last Time (×4ms)	10	0～10000

when Spindle is started, the system gives signal M03 or M04, delay the parameters setted time before execution of period,to ensure that the spindle speed reaches the set value.

142	Spindle Reverse Last Time (×4ms)	80	0～10000

when Spindle to the opposite, the system close the M03 or M04 signal, delay time after the parameter setting and then give M04 or M03 signal, in order to eliminate the lagging response of spindle inverter.

143	M41/S01 Output Time (×4ms)	0	0～10000
144	M42/S02 Output Time (×4ms)	0	0～10000
145	M43/S03 Output Time (×4ms)	0	0～10000
146	M44/S04 Output Time (×4ms)	0	0～10000

Set M41-44/S01-S04signal output mode : ＝0 : level mode, keep output ; >0 : pulse mode ;

147	Spindle CW Output Time (×4ms)	0	0～10000
148	Spindle CCW Output Time (×4ms)	0	0～10000
149	Spindle Stop Output Time (×4ms)	0	0～10000

Set the spindle rotation signal output mode : ＝0 : level mode, keep output ; >0 : pulse mode ;

150	Spindle Speed Error range	60	0～100

when Spindle starts, check the speed whether equal to the speed of the para 150 setted then continue running, otherwise will wait, until timeout alarm.

151	Spindle Command End To Brake Issue (×4ms)	150	0～10000

when System to execute the spindle stop commands ,close M03, M04, delay the time of the parameter setted,output the signal SPZD.

152	OutputTime Of Spindle Brake Signal (×4ms)	150	0～10000

Set the spindle brake pulse width.

The second Spindle control signal

153	Second Spindle Positive Output Port	0	0～72
154	Second Spindle Negtive Output Port	0	0～72
155	Second Spindle Stop Output Port	0	0～72
156	Second Spindle Break Output Port	0	0～72
157	Second Spindle Alert Input Port	0	0～72

Set the second spindle brake port signal

158	Second Spindle Speed For 10V,Second Spindle Gear (rpm)	2000	1~10000

159	Spindle Stop Port Output Time (×4ms)	200	1~10000

When press the spindle must stop button, corresponding pulse output time of the port.

160	Tapping Define Output Port	0	1~72
161	Stop To Position Define Input port	0	1~72

162	M4x Code Shift Delay Time(x4ms)	500	1~10000
163	M43 Code Shift Import	0	1~72
164	M44 Code Shift Import	0	1~72

Thread cutting parameters

165	Thread Cutting Back Scale Factor (0～60)	5	0～60

When not make back tail commands in threaded program, the system default set this parameter to the proportion of the value back end.

The default return tail length : P165 × 0.1 ×Screw thread

166	Time of linear Acc/DEC in thread cutting	150	10～4000

Because of speed Acc/Dec, screw up to the beginning and the end part of the thread pitch deviation : The faster the speed, the shorter the deviation part, the slower speed, the longer the deviation part ; Para 069 used in the set screw speed Acc/Dec, the smaller the value, the faster the thread speed, the shorter the distance deviation ; the greater the value, the slower thread speed, the longer the deviation distance.

167	Thread Cut Low-Vibrate Mode Factor (0～60)	5	1～60

Thread cutting, because there are a more or less spindle speed fluctuations, contributed to the Z axis feed not smooth. This parameter is used to set the Z axis response the sensitive coefficient of spindle rotation speed fluctuation, the smaller the value, the more sensitive response and Z axis large noise, high precision thread; the greater the value the response is gentle, Z axis low noise, low thread precision ;

168	Thread Cutting Max Speed Limit (mm/min)	6000	1～60000

Setting thread cutting length axis feed speed limit, when the spindle speed by screw thread to calculate the feed speed is greater than the parameters setting, alarm system. This value should be set according to the maximum feed speed machine tool.

169	Thread Back Acc And Dec Time Factor	150	10～4000

Set at the end of the thread back acceleration curve, the smaller the value, the faster the thread ends. This value should be setted according to the machine tool axis Acc/Dec speed feature.

170	Thread Back Max Speed Limit (mm/min)	6000	1～60000

Set screw back to the end, backing down tail axis's biggest feed speed; The greater the value, the faster the back end.

171	Thread Cutting Speed Reach Set (0%-100%)	60	0～100

Thread processing need stable spindle rotation, when the spindle speed is not up to the percentage setting speed, system alarm 124, do not respond to thread processing.

172	The Last Cutting Amount Of G86,G87 (um)	0	0～10000

Set the last cutting amount of G86, G87 thread loop , for the last thread cutting amount.

173	G33 Pulse Adjust(um)	12	0～10000

G33 tapping to reverse exits start at the bottom of the bore, The default value of the adjustment of the axix distance is 12.

174	Serve Spindle Pre_Stop All Time (x4ms)	300	1～65535

…… ……

179	Reserved	0	——

The para about tool

180	Select Tool Number	4	1～8

input range from 1 to 8.

181	Delay Time Tool Begin To Reverse (×4ms)	10	0～10000

After system to find effective tool, close (TL +) signal, and time after time delay parameter setting, give off lock (TL-) signal.

182	U Limit To Rotate Toolpost To Last (×4ms)	2000	0～10000

Tool change, if not found in the parameters set by the time set by the tool, system alarm, 040, and stop the tool change.

183	Tool Reverse Time Receive TCP SIGL	240	0～10000

Setting toolpost biggest reversal locking time.

184	Alert Time For Checking The *TCP (×4ms)	400	0～10000

When toolpost lock state output function, after sending the locking signal system, if not detected in the parameter setting time locking signal alarm occurs in 064, and stop the tool change. To do not have lock the toolpost of the state output function, can be set parameters P009 Bit0 1 to implement TCP normal signal detection.

185	Delay Time For Checking Loose Tool (×4ms)	40	0～10000
186	Times For Checking The *TCP	0	0～10
187	Times For Checking Tool Inposition	0	0～10

188	Reserved	0	——
189	Reserved	0	——

Chuck, tailstock, cooling, lubrication parameters

190	Level Output/Pulse Output（×4ms）	0	0～10000

set chuck lock/loose time ; ＝0 : level signal >0 : pulse signal

191	Delay Time For Chuck Close Time (×4ms)	100	0～10000

192	Clamp Chuck In Place Import（Inner）	28	0～72

Set chuck clamp in place (card) input port or loosen in place (wild) input port.

193	Loosen Chuck In Place Import（Inner）	29	0～72

Set chuck loosen in place (card) input port or clamped in place (wild) input port

194	M78 Output Time (×4ms)	0	0～10000

set M78 output mode : ＝0 : level mode, keep output ; >0 : pulse mode ;

195	M79 Output Time (×4ms)	0	0～10000

set M79 output mode : ＝0 : level mode, keep output ; >0 : pulse mode ;

196	Reserved	0	——

197	Lubricate Work Time（s）	5	0～10000

Set time each lubrication open interval lubrication mode; Unit: second ;

198	Lubricated Close Time（s）	3600	0～999999

Set the interval lubrication mode lubrication pause time every time; Unit: second ;

…… ……

203	Reserved	0	——

The handwheel related parameters

204	Max Handwheel Feedrate For X（mm/min）	6000	0～60000
205	Max Handwheel Feedrate For Y（mm/min）	6000	0～60000
206	Max Handwheel Feedrate For Z（mm/min）	6000	0～60000
207	Max Handwheel Feedrate For A（mm/min）	6000	0～60000
208	Max Handwheel Feedrate For C（mm/min）	1000	0～60000

The handwheel mode, the maximum of feed speed of each axis.

209	Handwheel Feed Time Cons（ms）	400	10～4000

Handwheel mode, the deceleration time constant of the axial feed; This value should be set to the appropriate bigger, in order to improve the smoothness of machine tool spindle feed.

The other parameters

210	Time For Executing M Codes (×4ms)	1	0～10000
211	Time For Executing S Codes (×4ms)	1	0～10000

212	Inc Of Sequence Number Is Auto Inserted	10	1～1000

When programming the segment num increase automatically,the para 002 bit7 setted 1 is valid.

213	TFT Brightness	80	40～100
214	TFT Brightness In Night	0	0～100

The TFT night time is from the night seven clock to the next day morning seven clock.

215	Boot Screen Start Time (×4ms)	120	1～10000

Set the time of home screen display before entering operation display,after the time the CNC enter operation display automatically.

216	Signal Remove Jitter Frequency	3	1～12

In the PLC interrupt cycle, the same level signal will be readed during the para continuously setted times, the CNC make sure it effective. when the external electrical interference is serious, the appropriate para will filter out external interference signal.

217	Circular Contour Error Range （um）	10	0～10000

Arc contour maximum error setting ; System for inscribed strings way of circular interpolation , Always keep the strings in the process of interpolation and arc on the top of the maximum error is not beyond the parameter value .When, in accordance with the process of contour error of arc feed speed in stoppage time beyond this parameter value , The system automatically adjust arc feed speed , To ensure effective contour error .

218	M30 Output Time (×4ms)	0	0～10000

Set M30 signal output mode : ＝0 : level mode, keep outport ; >0 : pulse mode ;

219	MST Output Time (×4ms)	0	0～10000

Set MST signal output mode : ＝0 : level mode, keep outport ; >0 : pulse mode ;

220	WARN Output Time (×4ms)	0	0～10000

Set WARN signal output mode : ＝0 : level mode, keep outport ; >0 : pulse mode ;

221	Start Key Alarm Addicted Time（×4ms）	200	180～1000

Set the longest time after pressing the start key, when the time out of the range the key still pressed, the CNC alarm 037 ; The function will avoid the wrong start because of the Circulate Button Function or external start switch stuck.

222	The Number of AXIS	2	2-4
223	Serial Communication Band	19200	0--50000

224	Wait for Input Signal Delay Alert	1000	0--10000

225	Speed Test Delay	400	0--10000

Modify the checking time to adapt the transducer on/off.

G instructions related parameters

230	Cycle Command Depth Of Cut（um）	1500	0～10000
231	Retract Amount Of Cycle Command（um）	1000	0～10000
232	Distance Of X Rough Cutting In G73（um）	2000	0～10000
233	Distance Of Z Rough Cutting In G73（um）	2000	0～10000
234	G73 Cycle Times	10	0～50

The para on port setted

250	Work Lamp OutPort	0	0~72

The para is setted outport, When the CNC System is turned on, the port is setted H level.when the para is setted zero, there is not output.

251	Key K1 Outport	0	0~72
252	Key K1 Output Width Of Pulse（×4ms）	0	0～10000
253	Key K2 Outport	0	0~72
254	Key K2 Output Width Of Pulse（×4ms）	0	0～10000

257	External Alert 1 Import	0	0～72
258	External Alert 2 Import	0	0～72

External Alert Import, need to set para 012.

259	Feed Rate Import1	1	0～72
260	Feed Rate Import2	16	0～72
261	Feed Rate Import3	15	0～72
262	Feed Rate Import4	14	0～72
265	max tool num of the 2nd toolpost	0	0～8
266	the 2nd toolpost positive outport	0	0～72
267	the 2nd toolpost negtive outport	0	0～72
268	the signal of the 2nd toolpost NO.1 tool arrived inport	0	0～72
269	the signal of the 2nd toolpost NO.2 tool arrived inport	0	0～72
270	the signal of the 2nd toolpost NO.3 tool arrived inport	0	0～72
271	the signal of the 2nd toolpost NO.4 tool arrived inport	0	0～72
272	the signal of the 2nd toolpost NO.5 tool arrived inport	0	0～72
273	the signal of the 2nd toolpost NO.6 tool arrived inport	0	0～72
274	the signal of the 2nd toolpost NO.7 tool arrived inport	0	0～72
275	the signal of the 2nd toolpost NO.8 tool arrived inport	0	0～72
276	the 2nd toolpost locked signal port	0	0～72
277	Wheel Led Output	0	0～72
288	Reserved	0	——

Appendix: 2 Alarm list

Alarm	Description	Solve way
001	Emergency	check ESP
002	X Axis Alert	check driver or X alarm input L/H level setting
003	Y Axis Alert	check driver or Y alarm input L/H level setting
004	Z Axis Alert	check driver or Z alarm input L/H level setting
005	A Axis Alert	check driver or A alarm input L/H level setting
006	Axis Lim+ Alert	check whether Draggingplate out of range or not or the L/H level set of the import
010	Axis Lim- Alert	check whether Draggingplate out of range or not or the L/H level set of the import
014	X Axis Soft Lim+ Alert	check the coor whether out of the range of the para setting or not
015	Y Axis Soft Lim+ Alert	check the coor whether out of the range of the para setting or not
016	Z Axis Soft Lim+ Alert	check the coor whether out of the range of the para setting or not
017	A Axis Soft Lim+ Alert	check the coor whether out of the range of the para setting or not
018	X Axis Soft Lim- Alert	check the coor whether out of the range of the para setting or not
019	Y Axis Soft Lim- Alert	check the coor whether out of the range of the para setting or not
020	Z Axis Soft Lim- Alert	check the coor whether out of the range of the para setting or not
021	A Axis Soft Lim- Alert	check the coor whether out of the range of the para setting or not
022	C Axis Alert	Check driver or the CNC para L/H level of C Axis Alert input setting
023	Go Zero Disabled	open para 6 bit0 to 2 back to zero point
024	Go Zero Point Before Run	set para001 bit5 1,return machine tool zero followed by enterring auto mode
025	Go Zero Point With A Way,the point dosn't set	set para5 bit4 0 to return zero
026	T Code Illegal	tool num or toolcomp num illegal,max of tool num is 8,toolcomp is 16
028	Go Program Zero Point Disabled	set para5 bit6 0
029	M35 Test Timeout	M35 avail signal has not been checked during set time

Alarm	Description	Solve way
030	M01 Test Timeout	M01 avail signal has not been checked during set time
031	User No.1 Alert	
032	User No.2 Alert	
033	Protective door Need Closed Before Run	Close Safety Door:check the para setting of the alarm level
034	Spindle Gear Control Order , S0 ~ S4	check para1 bit4,S is between 0 to 4 when other control method
035	Can't Set Tools Compensation When Running	stop program followed by modify tool comp
036	TriSw not in Start State	turn triSw left
037	Circulate Button locked Continually	Check Circulate Button Function or external start switch stuck
038	Can't Modify When Tool Used	
039	Key Locked	check wether button stuck
040	Check Tool Time Out	check toolpost Hall device and dish,check para182
041	Not Remember Coor	set tool comp after pressing x or z to memory coor
042	Not Find Valid Tool	check toolpost Hall device and dish
043	Tool Type Error	
044	Can't Loosen Chock	
045	Circulate Button Function Disable	para14 bit4 is to set open cycle start function
046	No Clap Signal	
047	No Loosen Signal	
048	Screw Less Than 0	
049	peed Limit Error When Thread Process	screw program data error or para168 set error
050	Don't Fined UDisk	
051	File Not Existed or File Name Error	
052	Search Top Or Bottom	check end tip
053	File Full	
054	Illegal Size	
055	No Axis C Ready Sign	check C Axis Ready Sign
056	Failed To Read File	
057	Failed To Write File	
058	File Existed or File Name Error	
059	File Delete Failed	
060	Failed To Read Para	set again or restore product value
061	Failed To Read Tool Para	tool comp reset or set again
062	Failed To Read Thread File	screw comp reset or set again
063	Para File Can't Del	

Alarm	Description	Solve way
064	Failed To Check Tool Locked Signal	check lock signal, wiring and set of para9 bit0 and bit1
065	Not Open G50 Setting Relative Coordinate Function	check set of para4 bit5
066	File Converted Failed	
067	U Disk operated After Oper Ended	
068	Tool Number Error	tool comp port is beyond of 1 to 16 when C comp beyond 100000 alarm 68
069	Spindle Speed Error	There is large difference between Spindle Speed and setting speed,check wether the control of the Spindle is right or not
070	Program is Disable,When System running	Program is Disable,When System running
071	the file num beyond the max num	
072	Spindle Gear Change Error	
073	The Time of Servo Spindle Is Up	
074	prog code is illegal	
093	Time Setted Error	time format set illegal
094	Pwd error or not allowed	
096	Can't Edit Now	program processing can't be edit
099	System Exception	
100	Para Switch ON	press reset or cancle key
101	G Code Illegal	G function code is not exist
102	Segment Too Long	max single period number is 78
103	X Value Illegal	X prog numb illegal
105	Z Value Illegal	Z prog numb illegal
107	F Value Illegal	F prog numb illegal
108	X Multi Defined	data can't define repeatedly
110	Z Multi Defined	data can't define repeatedly
112	Data Illegal	set data precision vail
113	M Code Illegal	M function code is not exist
114	Segment Code Illegal	code is not exist
115	Arc Plane Illegal	circle para illegal,check program para
116	Para Multi Defined	command data define repeatedly
117	Arc EndPoint Error	circle data illegal
118	T Code Illegal	command format is T0101
119	Sub File Called Illegal	subroutine nested series less than 10
120	Open Sub File Failed	make sure subroutine is exist
121	Para Error	define repeatedly or beyond the range or lack of para

Alarm	Description	Solve way
122	Not Find Jump Segment	can't find period to jump when doing M91 , 92, set Jump Segment
124	Spindle Speed Slow When Thread Oper	the Spindle start early
125	Jump Loop Illegal	
126	Chamfer Command Para Error	check Command Para
127	R Should Not Used For Whole Circle	circle program data illegal
128	Thread Segment Para Illegal	Thread circle data illegal
129	No Spindle Encode Signal	check encoder and wiring
130	Arc Radius Equal To 0	circle program data illegal
131	No Chock Clap Signal	not check clamp chuck in place signal
132	Spindle Not Clap	start spindle when chuck not clamped
133	U Value Error	data error or U repeatedly
135	W Value Error	data error or W repeatedly
136	Code T And G00 Can't The Same Line	need open para 10 bit 1
137	Cutting Cycle Interference	check program data and coor location
138	T And Cutting Code Can't In Same Line	only G00 can be with T code in the same period,neither can G01
140	End Of File Missing M30	
141	C Cutting Modify At The End	
142	C Cutting Modify Data Building Error	
143	C Cutting Modify Radius Or Tip Position Error	
144	Can't G00 When C Cutting	
145	Circle Center Coincidence With Beginning Or End	
146	Code P Or Q Not Exist In Cycle	can't find line of P and Q in cycle
147	Cut Depth Zero Or Negative	deepth designated error in cycle
148	G01-G03Only G01-G03 In Cycle	Only G01-G03 In Cycle
149	Coor Not Monotonous In Cycle	X and Z coor drab only in cycle
150	Program that Loop Command P Indicates Not Adjacent	not P command followed cycle
151	T-thread Programming Parameter Error	T screw para error in G92
152	G73 Repeatition times Error	repeat numbers in G73 must between 2 to 50
153	XZ Retract Error In G73 Cycle	Recede cutter quantity of XZ set error
154	Too Many Cycle in G71/G72	cycle too many times

Alarm	Description	Solve way
155	Wrong Tool Num	check if tool in position is the same with command
156	can not find AXIS zero sign	check the sending disc installation right or not
157	M30 can not same line with others	M30 must in the last and other command can't in the same line
158	Power 24VDC Issue	Power 24VDC Issue
159	When Servo Axis In POS Mode , M03,M04,M05 can not run	Check Servo State
160	When Servo Axis In SPEED Mode , G0,G01 C Order can not run	Check Servo State

Appendix: 3 Common alarm solve method

1．Alarm 002～004, driver alarm

The cause : the driver has alarm output or driver alarm h/l level detection logic instead ;

Remove method :

1）Check whether there is any alarm on drive to produce (drive alarm lights) ;

2）check the axis drive alarm level Settings,set parameters P008 Bit0, Bit1, Bit2 properly ;

2．Alarm 042, not detected effective tool

The cause : toolpost type error or not receive the sending disc of toolpost signal ;

Remove method :

1）Check toolpost type setting, parameter P010 Bit0 setting row knife or power toolpost ;

2）Check the electric toolpost hall sensing device or toolpost sending disc ;

3）Set parameters 009 Bit1 depending on the type of toolpost (tool H/L level signal) ;

3．Alarm 036, The trisw is not started

The cause : the TriSw is not started when the CNC system is on;

Remove method : 1）turn the TriSw on (Left) ;

4．Alarm 001, Emergency

The cause : the panel emergency button be pressed or an external stop signal input (ESP) ;

Remove method :

1）pull the emergency button ;

2）check the external emgecy input signal (external emgecy signal should be normally open mode) ;

5．Alarm 006, 010, Axis Lim+/Lim- alarm

The cause : the signal of LIM＋or LIM－ input ;

Remove method :

1）Check whether there is any signal on the axis LIM＋or LIM－ switch ;

2）The axis of LIM＋or LIM－ switch shall be NPN or normally open ;

6．Alarm 014～020, The axis of LIM＋ or LIM－ soft limit alarm

The cause : The axis of LIM＋ or LIM－coordinates beyond the range of the para setted ;

Remove method : 1）move to the opposite ;

7．Alarm 037, Circulate Button be pressed continuously

The cause : the time of the circulate button be pressed is too long or the button not release after contact ;

Remove method :

1）check whether the time of the para 221 setted is too short ;

2）check whether the button stucked

8．Alarm 060, the system para file is not checked

The cause : the system para file S0001 is missing ;

Remove method :

1）Under the parameter interface, press the switch key, to restore the factory values or restore the backup operation ;

2）U disk mode import file S0001 ;

9．Alarm 061, ToolOffset file not be checked

The cause : ToolOffset file T0001 is missing ;

Remove method :

1）Under the tooloffset interface, input tooloffset or repair the tooloffset, the system automatically establish a tooloffset file ;

Appendix: 4 System upgrade instructions

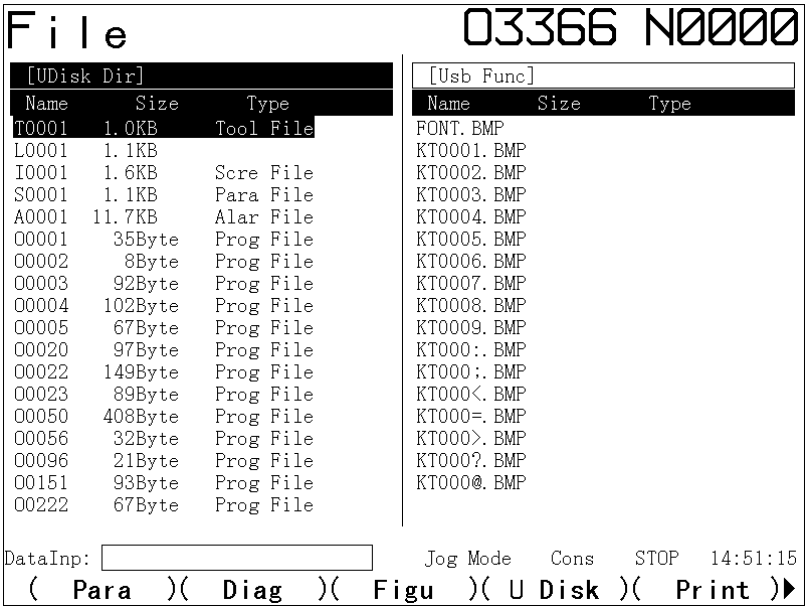
1. By U disk to upgrade the CNC system software

By U disk to upgrade the CNC system, operation steps are as follows :

（1）First store the upgrade file in U packing directory, file format for the suffix is "bin", insert the USB into the CNC system.

（2）Press the transformation key, don't loose, and then the numerical control system is powered on, until the CNC system popup the password input interface then release the button, input the password "KT7350", and press the enter key.

(3) after the password being entered correctly, the system enter the interface of the upgrade file management.



(4) in the U disk directory will show the bin file in U dish, at the bottom left page have a prompt, by moving up and down keys to select upgrade file, and then press the enter key to upgrade the software . System begin to read the upgrade file, and display the progress of the file.after receiving the Upgrade file begin to programme upgrade code,and display the burning process.

(5) if the detected file type is not correct, will display the current file type cannot recognize in the system upgrade information.

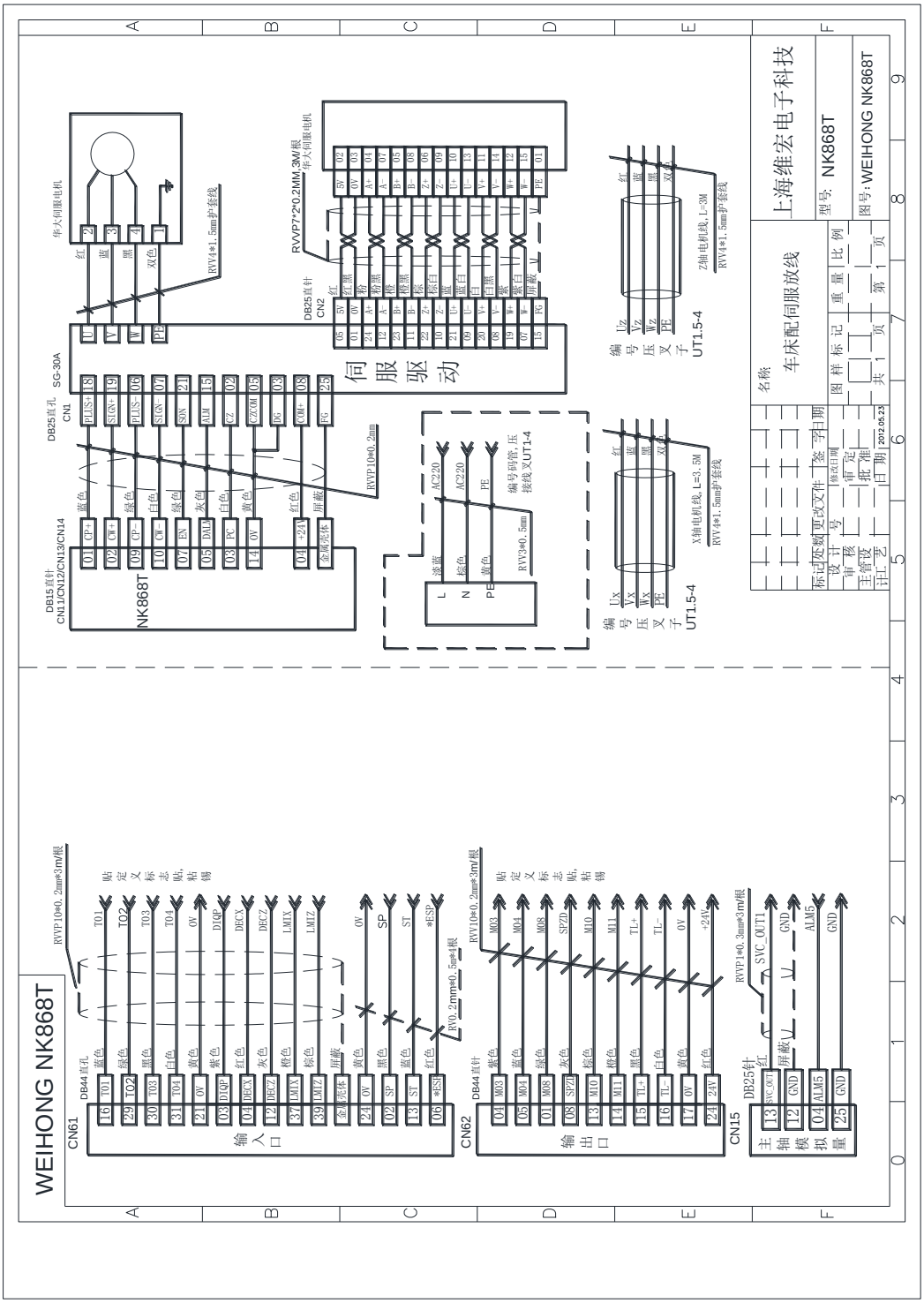
(6)after being upgraded the CNC system prompt complete message. If upgrade failed system prompt not successfully.

2. By U disk to upgrade the CNC system boot Screen

the steps of the U disk upgrade the interface and the system software are the same.

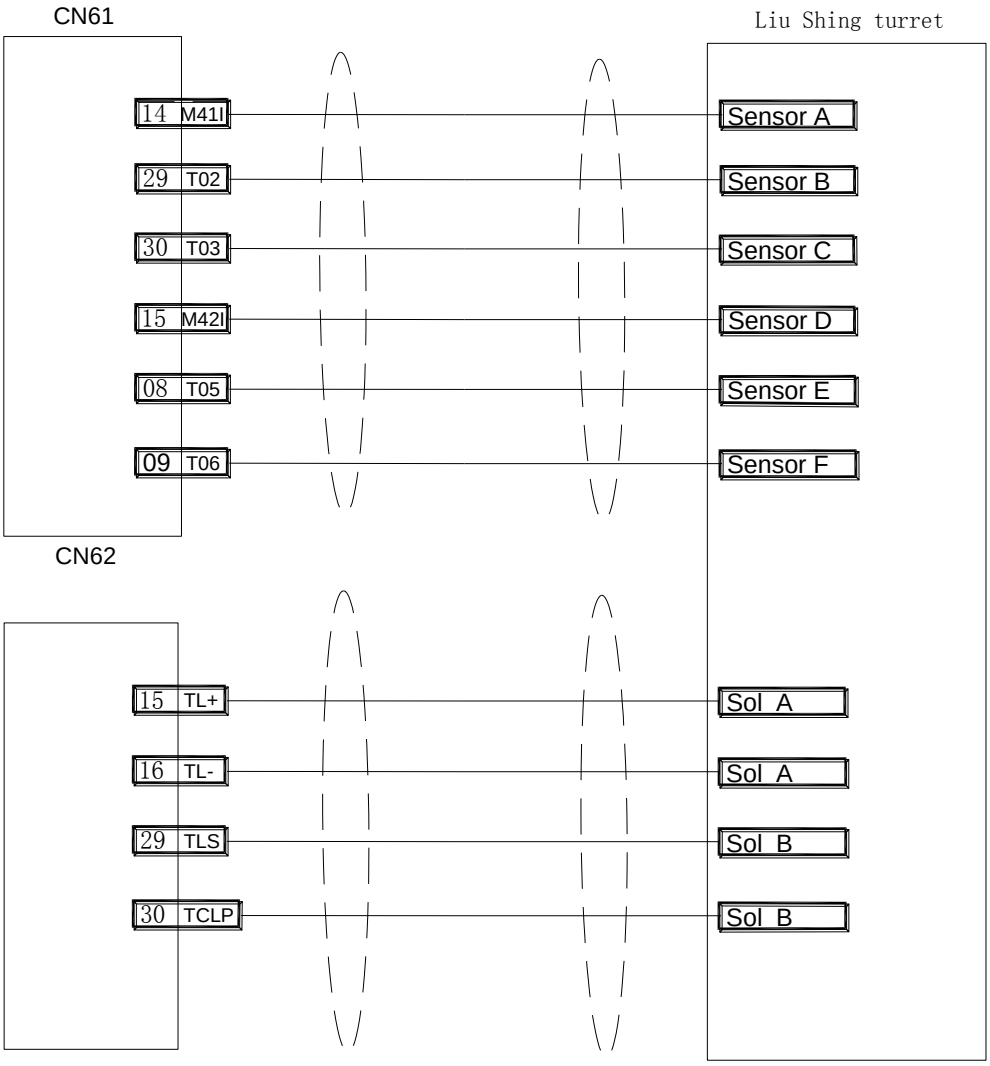
Boot interface image format for 800 x600 pixels, 24 bit color BMP format.

Appendix: 5 Connected with servo



Appendix: 6 Connected with Lio Shing Turret

1 : wiring diagram



2 : parameter

0	1	0				TSS0					TSS0

TSS0,TSS1 = 0,0 : motor tool（default value）

TSS0,TSS1 = 0,1 : horizontal tool

TSS0,TSS1 = 1,0 : Liu Shing turret

0	0	9									TCPS

TCPS : =1 : tool rest lock signal low level available =0 : high level available

180	Total tool number	4	1～12

Input range 1～12

182	Time limit shift from the first tool to last tool (×4ms)	2000	0～10000

185	Hydraulic tool rest delay time (x4ms)	20	0～10000
186	Hydraulic tool rest check in place times	0	0～10000
187	Check shift tool times after horizontal tool in place	0	0～10000

3 : related alarm

040	Check tool overtime	Can not find available tool.
064	Have not checked tool lock signal	Check lock signal,wiring and parameter set

4 : work flow（number one）

1) command input (T0505) or press shift tool key.
2) TLS (tool loose signal) effective.
3) check whether SENSORF signal loose.
4) tool rest rotation, direction is decided by tool number.
5) check number five tool signal and SENSORF signal are both effective, tool rest rotation in place.
6) loose tool rest rotation signal.
7) TCLP (tool lock signal) effective.
8) check whether SENSORF signal locking.
9) shift tool is completed.

Liu Shing turret signal list:

6 station:

	1	2	3	4	5	6
SENSERA(T01)	○		○	○	○	
SENSERA(T02)		○		○	○	○
SENSERA(T03)			○		○	
SENSERA(T04				○		○

8 station:

	1	2	3	4	5	6	7	8
SENSERA(T01)			○		○	○	○	
SENSERA(T02)	○				○		○	○
SENSERA(T03)				○	○	○		○
SENSERA(T04		○				○	○	○

10 station:

	1	2	3	4	5	6	7	8	9	10
SENSERA(T01)			○		○	○	○		○	
SENSERA(T02)	○				○		○	○	○	
SENSERA(T03)				○	○	○		○		○
SENSERA(T04		○				○	○	○		○

12 station:

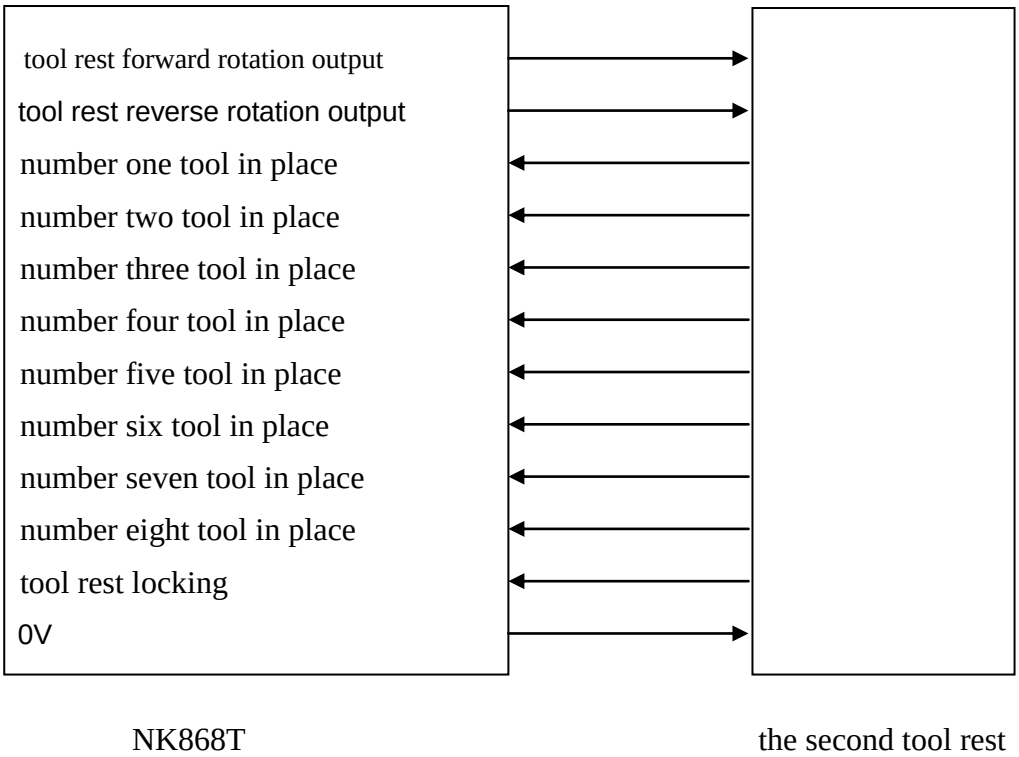
	1	2	3	4	5	6	7	8	9	10	11	12
SENSERA(T01)	○			○	○		○	○	○			
SENSERA(T02)			○			○	○		○	○	○	
SENSERA(T03)		○			○	○		○	○	○		
SENSERA(T04				○			○	○		○	○	○

Note: circle means high level, input system is invalid.

Description: in the wiring, M41,M42,T03,T04,T05,T06 should add resistor of 3K,1/4W to input port.

Appendix: 7 Second tool carrier use method

Wiring figure



1. Parameter

0	1	0					TSS2				

TSS2: =1: the second tool rest is horizontaltool =0: motor tool

265	Max tool number of the second tool rest	0	0～8

If this parameter is 0, the second tool rest is forbidden.

266	Forward output port of the second tool rest	0	0～72
267	Reverse output port of the second tool rest	0	0～72
268	Number one tool in place signal input port of the second tool rest	0	0～72
269	Number two tool in place signal input port of the second tool rest	0	0～72
270	Number three tool in place signal input port of the second tool rest	0	0～72
271	Number four tool in place signal input port of the second tool rest	0	0～72
272	Number five tool in place signal input port of the second tool rest	0	0～72
273	Number six tool in place signal input port of the second tool rest	0	0～72
274	Number seven tool in place signal input port of the second tool rest	0	0～72
275	Number eight tool in place signal input port of the second tool rest	0	0～72
276	Lock signal input port of the second tool rest	0	0～72

3. Using description

(1) wiring according to figure.

(2) set parameters according to actual need.

(3) whether high or low level is effective is managed by the first tool rest parameter.

(4) the first tool rest is followed by the second, for example, the first tool rest has four tools, so the number one tool of second tool rest is number 5 tool of system.

(5) when the second tool rest is used, the first tool rest is motor tool as the second.

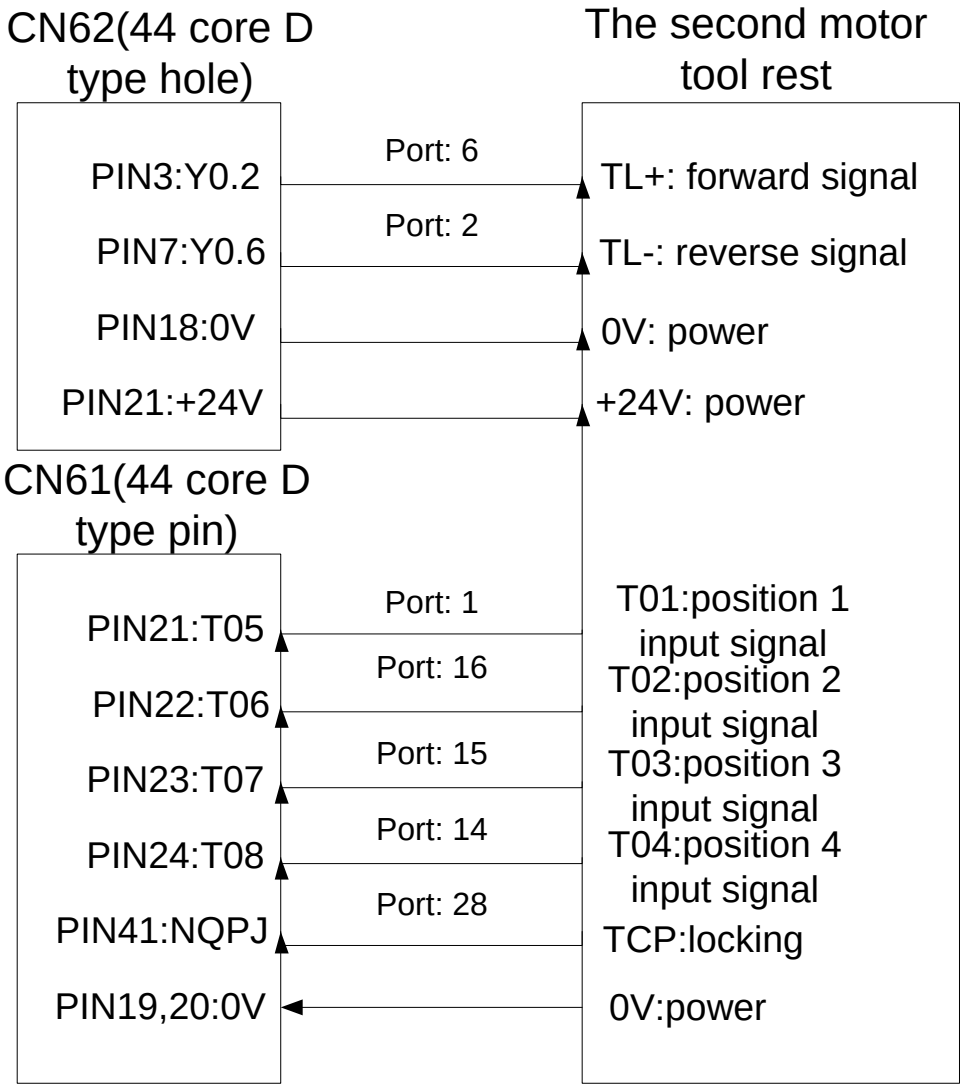
4. Action description

image the first and the second tool rest are both four-tool motor tool.

when tool shift between number 1 and 4, the second tool rest is not action.

when tool shift between number 5 and 8, the first tool rest is not action.

when tool number from 1 to 4, shift to 5 to 8, the first tool rest is not action.

when tool number from 5 to 8, shift to 1 to 4, the second tool rest is not action.

5. Typical electronic connection