

ECN*000TL SERIES USER MANUAL
CNC LATHE CONTROLLER



www.hybridservo.com

©2016 shenzhen ECON technology co.,ltd

This user manual describes all items concerning the operation of the system in detail as much as possible. However, it is impractical to give particular descriptions of all unnecessary and/or unavailable operations of the system due to the manual content limit, product specific operations and other causes. Therefore, the operations not specified herein shall be considered impossible or unallowable.

This user manual is the property of Shenzhen ECON technology Co.,Ltd
All rights are reserved. It is against the law for any organization or individual to publish or reprint this manual without the express written permission of ECON and the latter reserves the right to ascertain their legal liability.

FOREWORD

Dear user,

We are really grateful for your patronage and purchase of this **ECN*000T Turning/Lathe CNC system** made by Shenzhen ECON technology Co.,Ltd. The user manual describes the programming, operation, installation and connection of this **ECN*000T Turning/Lathe CNC system**. Please read it carefully before operation in order to get the safe and effective working.

Warning

This system can only be operated by authorized and qualified personnel as improper operations may cause accidents. Please carefully read this user manual before use!

Note: The power supply installed on (in) the cabinet is exclusive to ECON'S CNC systems. The power supply form is forbidden to be used for other purposes. Otherwise, there may be extreme danger!

This user manual shall be kept by final user.

Notes

III

Notes

■ Delivery and storage

Packing box over 6 layers in pile is unallowed.

Never climb the packing box, neither stand on it, nor place heavy objects on it.

Do not move or drag the product by the cables connected with it.

Forbid collision or scratch to the panel and displayer.

Packing box should be protected from damping, insolation and raining.

■ Open packing box to check

Ensure things in packing box are the required ones.

Ensure the product is not damaged in delivery.

Ensure the parts in packing box are in accordance to the order.

Contact us in time if the product type is inconsistent with the order, there is short of accessories, or product damage in delivery.

■ Connection

Only qualified persons can connect the system or check the connection.

The system must be earthed, its resistance must be less than 4 Ω and the ground wire cannot be replaced by zero wire.

Connection must be correct and firm to avoid the product to be damaged or other unexpected result.

Connect with surge diode in the specified direction to avoid the damage to the system.

Switch off power supply before pulling out plug or opening electric cabinet.

■ Troubleshooting

Switch off power supply before troubleshooting or changing components.

Troubleshoot and then startup the system when there is short circuit or overload.

Do not switch on or off it frequently and an interval is 1 minute at least after the system is powered on again.

All specification and designs are subject to change without further notice.

Warning !

Please read this user manual and a manual from machine builder completely before installation, programming and operation; do operate the system and machine according to user manuals, otherwise it may damage the system, machine, workpiece and even injure the operator.

Cautions !

Functions, technical indexes described in this user manual are only for the system. Actual functions and technical performance of machine tool with this CNC system are determined by machine builder's design, so refer to its user manual.

The system is employed with integrated machine control panel and the key on machine control panel are defined by PLC program. Functions of key in this user manual are for standard PLC program. Please notice it!

Refer to user manual from machine manufacturer about functions and meanings of key on machine control panel.

Announcement !

This manual describes various items as much as possible. However, operations allowable or unallowable can not be explained one by one due to so many possibilities that may involve with, so the contents that are not specially stated in this manual shall be considered to be unavailable.

Summary

I Programming

ECN*000T CNC Technical Specification, Product Type, Command and Program Format

II Operation

ECN*000T CNC Operation Use

III Installation and Connection

ECN*000T CNC Installation, Connection and Setting

Appendix

PLC Function Allocation, Alarm Message Table

Safety Responsibility

Manufacturer's safety responsibility

- The manufacturer should be responsible for the cleared or the controlled safety in the design and the structure of the CNC system and the accessories.
- The manufacturer should be responsible for the CNC system and the accessories.
- The manufacturer should be responsible for the message and the suggestion for the user.

User's safety responsibility

- The user should study and train the system safety operation, master the safety operation content.
- The user should be responsible for the danger caused by increasing, changing or modifying the CNC system, the accessories by itself.
- The user should be responsible for the danger because of the mistaken operation, regulation, maintenance, installation and storage.

CONTENTS

I Programming

CHAPTER	1
PROGRAMMING.....	1
1.1	ECN*000T
Introduction	1
1.1.1 Product introduction.....	1
1.1.2 Technical specification	2
1.1.3 Environment and conditions.....	4
1.1.4 Power supply	5
1.1.5 Guard.....	5
1.2 CNC System of Machine Tools and CNC	
Machine Tools.....	5
1.3.Programming.Fundamentals	7
1.3.1 Coordinates definition	7
1.3.2 Machine coordinate system, Machine Zero and machine reference point	7
1.3.3 Workpiece coordinate system and Program Zero.....	8
1.3.4 Interpolation function	9
1.3.5 Absolute programming and incremental programming	10
1.3.6 Diameter programming and radius programming	10
1.4 Structure of an NC program.....	11
1.4.1 General structure of a program.....	11
1.4.2 Main program and subprogram.....	15
1.5 Program run.....	15
1.5.1 Sequence of program run	15
1.5.2 Execution sequence of word.....	16
1.6 Basic Axis Incremental system.....	17
1.6.1 Incremental system speed of basic axis	17
1.6.2 Incremental system unit of basic axis	17
1.6.3 Incremental system data range of basic axis	18
1.6.4 Incremental system data range and unit of basic axis	18
1.6.5 Program address value unit and range of incremental system of basic axis.....	20
1.7 Additional Axis Incremental	
system	21
1.7.1 Additional axis being the current incremental system.....	21
1.7.2 Additional axis being IS-A incremental system.....	21
CHAPTER 2 MSTF	
COMMAND	22
2.1 M (Miscellaneous function).....	22
End of program M02.....	22
End of program run M30.....	22
Subprogram call M98.....	23

Return from subprogram M99.....	23
Macro program call M9000~M9999.....	24
M commands defined by standard PLC ladder diagram.....	24
Program stop M00	25
Program optional stop M01	25
Spindle CW, CCW and stop control M03, M04, M05.....	26
Cooling control M08, M09.....	26
Tailstock control M10, M11	26
Chuck control M12, M13.....	26
Spindle position/speed control switch M14, M15.....	26
Spindle clamped/released M20, M21	27
The 2nd spindle position/speed switch M24, M25.....	27
Lubricating control M32, M33.....	27
Spindle automatic gear change M41, M42, M43, M44.....	27
Spindle 8-point orientation M50~M58	27
The 2nd spindle rotation CCW, rotation CW , stop M63, M64, M65.....	27
2.2 Spindle function.....	28
2.2.1 Spindle speed switching value control	28
2.2.2 Spindle speed analog voltage control	28
2.2.3 Constant surface speed control G96, constant rotational speed control G97.....	29
2.2.4 Spindle override.....	31
2.2.5 Multiple spindle control function.....	32
2.2.6 Cs contour control function.....	33
2.3 Tool Function	33
2.3.1 Tool control	33
2.3.2 Tool life management	37
CHAPTER 3 G COMMANDS.....	46
3.1 Commands	46
3.1.1 Modal, non-modal and initial mode.....	47
3.1.2 Omitting words.....	48
3.1.3 Related definitions	49
3.2 Rapid Traverse Movement G00	49
3.3 Linear Interpolation G01.....	50
3.4 Circular Interpolation G02, G03.....	51
3.5 Three-point Circular Interpolation G05.....	54
3.6 Ellipse Interpolation G6.2, G6.3	56
3.7 Parabola Interpolation G7.2, G7.3.....	59
3.8 Plane Selection G17~G19.....	61
3.9 Polar Coordinate Interpolation G12.1,G13.1.....	62
3.10 Cylindrical Interpolation G7.1.....	65
3.11 Chamfering Function.....	68
3.11.1 Linear chamfering.....	68
3.11.2 Circular chamfering.....	70
3.11.3 Special cases.....	72

3.12 Dwell G04.....	74
3.13 Machine Zero Function	74
3.13.1 Machine 1st reference point G28	74
3.13.2 Machine 2nd, 3rd, 4th reference point G30.....	75
3.14 Skip Interpolation G31.....	77
3.15 Automatic Tool Offset G36, G37.....	79
3.16 Workpiece Coordinate System G50	81
3.17 Local Coordinate System.....	82
3.18 Workpiece Coordinate System G54~G59	84
3.19 Fixed Cycle Command	87
3.19.1 Axial cutting cycle G90.....	87
3.19.2 Radial cutting cycle G94.....	90
Contents	
III	
3.19.3 Caution of fixed cycle commands.....	92
3.20 Multiple Cycle Commands.....	93
3.20.1 Axial roughing cycle G71	93
3.20.2 Radial roughing cycle G72.....	99
3.20.3 Closed cutting cycle G73.....	103
3.20.4 Finishing cycle G70	107
3.20.5 Axial grooving multiple cycle G74	108
3.20.6 Radial grooving multiple cycle G75.....	111
3.21 Thread Cutting Commands.....	114
3.21.1 Thread cutting with constant lead G32.....	115
3.21.2 Rigid thread cutting G32.1.....	117
3.21.3 Thread cutting with variable lead G34.....	119
3.21.4 Z thread cutting G33.....	121
3.21.5 Rigid tapping G84, G88.....	122
3.21.6 Thread cutting cycle G92.....	125
3.21.7 Multiple thread cutting cycle G76.....	128
3.22 Constant Surface Speed Control G96, Constant Rotational Speed Control G97.....	132
3.23 Feedrate per Minute G98, Feedrate per Rev G99	133
3.24. Additional Axis Function.....	135
3.24.1 Additional axis start.....	135
3.24.2 Motion of additional axis.....	135
3.24.3 Additional axis coordinates display.....	135
3.25 Macro Commands.....	135
3.25.1 MACRO variables	136
3.25.2 Operation and jump command G65	140
3.25.3 Program example with macro command.....	143
3.26 Statement Macro Command.....	146
3.26.1 Arithmetic and logic operation.....	146
3.26.2 Transfer and cycle.....	147

3.27 Metric/Inch Switch.....	150
CHAPTER 4 TOOL NOSE RADIUS COMPENSATION (G41, G42)	152
4.1 Application	152
4.1.1 Overview.....	152
4.1.2 Imaginary tool nose direction	152
4.1.3 Compensation value setting.....	156
4.1.4 Command format	157
4.1.5 Compensation direction	157
4.1.6 Notes	159
4.1.7 Application	160
4.2 Tool Nose Radius Compensation Offset Path.....	161
4.2.1 Inner and outer side.....	161
4.2.2 Tool traversing when starting tool.....	162
4.2.3 Tool traversing in Offset mode.....	163
4.2.4 Tool traversing in Offset canceling mode	167
4.2.5 Tool interference check.....	168
4.2.6 Commands for canceling compensation vector temporarily.....	170

II Operation

CHAPTER 1 OPERATION MODE AND DISPLAY

INTERFACE.....	173
1.1 Panel Division.....	173
1.1.1 State indication	174
1.1.2 Edit keypad.....	174
1.1.3 Menu display	175
1.1.4 Machine panel	176
1.2 Summary of Operation.....	179
1.3 Display Interface	179
1.3.1 POS interface	179
1.3.2 PRG interface.....	184
1.3.3 TOOL OFFSET Interface	189
1.3.4 ALARM interface	193
1.3.5 Setting interface.....	194
1.3.6 PARAMETER Interfaces.....	196
1.3.7 Diagnosis Interface.....	198
1.3.8 Graph Interface.....	199
1.3.9 Ladder Interface	200
CHAPTER 2 POWER ON/OFF AND PROTECTION.....	205
2.1 System Power-on	205
2.2 System Power-off	205
2.3 Overtravel Protection	206
2.3.1 Hardware overtravel protection.....	206
2.3.2 Software Overtravel Protection.....	206
2.4 Emergency Operation.....	207

2.4.1 Reset	207
2.4.2 Emergency stop.....	207
2.4.3 Feed hold.....	207
2.4.4 Power-off	207
CHAPTER 3 MANUAL OPERATION	209
3.1 Coordinate Axis Move.....	209
3.1.1 Manual feed.....	209
3.1.2 Manual rapid traverse.....	209
3.1.3 Speed tune	210
3.2 Other Manual Operations	211
3.2.1 Spindle CCW, CW, stop control	211
3.2.2 Spindle jog.....	211
3.2.3 Cooling control.....	212
3.2.4 Lubricating control	212
3.2.5 Chuck control.....	212
3.2.6 Tailstock control	213
3.2.7 Hydraulic control.....	213
3.2.8 Manual tool change	213
3.2.9 Spindle override.....	214
CHAPTER 4 MPG/STEP OPERATION	215
4.1 Step Feed	215
Contents	
4.1.1 Increment selection.....	215
4.1.2 Moving direction selection	216
4.2 MPG(handwheel) Feed.....	216
4.2.1 Increment selection.....	217
4.2.2 Moving axis and direction selection	217
4.2.3 Other operations	218
4.2.4 Explanation items	219
CHAPTER 5 MDI OPERATION	220
5.1 Block Input	220
5.2 Block Execution	221
5.3 Parameter Setting	221
5.4 Data Alteration	222
5.5 Other Operations	222
CHAPTER 6 PROGRAM EDIT AND MANAGEMENT	223
6.1 Program Creation.....	223
6.1.1 Creating a block number.....	223
6.1.2 Inputting a program.....	223
6.1.3 Searching a character.....	226
6.1.4 Inserting a character	229
6.1.5 Deleting a character.....	230
6.1.6 Altering a character.....	230
6.1.7 Deleting a single block.....	231

6.1.8 Copying and pasting a block.....	232
6.1.9 Canceling and recovering a program.....	233
6.1.10 Program save	233
6.1.11 Macro program edit.....	234
6.1.12 Creating and modifying a program annotation.....	234
6.2 Deleting Programs	235
6.2.1 Deleting a program.....	235
6.2.2 Deleting all programs.....	237
6.3 Selecting a Program	238
6.3.1 Searching.....	238
6.3.2 Scanning.....	239
6.3.3 Direct confirmation.....	239
6.4 Executing a Program	240
6.5 Renaming a Program.....	241
6.6 Copying a Program.....	242
6.7 Program Management	244
6.7.1 Program list	244
6.8 Other Operations Available in Edit Mode	244
CHAPTER 7 TOOL OFFSET AND SETTING	246
7.1 Tool Positioning Setting.....	246
7.2 Trial Toolsetting.....	247
7.3 Toolsetting by Machine Zero Return	248
7.4 Coordinates Record	251
7.5 Setting and Altering the Offset Value	252
7.5.1 Offset setting.....	252
7.5.2 Offset alteration	252
7.5.3 Offset alteration in communication mode.....	253
7.5.4 Clearing the offset values	253
7.5.5 Setting and altering the tool wear	253
7.5.6 Locking and unlocking the offset value	254
7.5.7 No.0 tool offset moving workpiece coordinate system.....	254
CHAPTER 8 AUTO OPERATION.....	256
8.1 Automatic Run	256
8.1.1 Selection of a program running.....	256
8.1.2 Start of automatic run	256
8.1.3 Stop of automatic run.....	256
8.1.4 Automatic run from an arbitrary block.....	257
8.1.5 Adjustment of the feedrate, rapid rate	258
8.1.6 Spindle speed adjustment.....	258
8.2 Running State	259
8.2.1 Single block execution.....	259
8.2.2 Dry run.....	259

VI

8.2.3 Machine lock.....	259
8.2.4 MST lock.....	260
8.2.5 Block skip	260
8.3 MPG Trial-cut.....	260
8.3.1 Switching MPG trial-cut mode	260
8.3.2 Command speed in MPG trial-cut mode.....	261
8.3.3 Notes in MPG trial-cut mode.....	262
8.3.4 Temporarily invalid in MPG trial-cut mode	262
8.4 Other Operations	263
CHAPTER 9 ZERO RETURN OPERATION	264
9.1 Program Zero Return.....	264
9.1.1 Program Zero	264
9.1.2 Program zero return steps	264
9.2 Machine Zero Return	265
9.2.1 Machine Zero.....	265
9.2.2 Machine Zero return steps.....	265
9.3 Other Operations in Zero Return	266
CHAPTER 10 DATA SETTING	267
10.1 Data Setting	267
10.1.1 Switch setting	267
10.1.2 Level setting	267
10.1.3 Parameter operation.....	270
10.2 Clock Setting.....	271
10.3 Graphic Display	272
10.4 Parameter Setting.....	275
10.4.1 State parameter	275
10.4.2 Data parameter, pitch compensation	278
10.4.3 Often used parameters	280
CHAPTER 11 U OPERATION FUNCTION	283
11.1 File Management Function Introduction.....	283
Contents	
VII	
11.2 Often Used File Operation Function Introduction	283
11.2.1 File extension and return.....	283
11.2.2 File selection and cancellation of file selection.....	284
11.2.3 File copy.....	286
11.3 Data Copy and Resume.....	286
11.3.1 Entering backup/resume page	286
11.3.2 Backup/resume operation	287
11.3.3 Operation path selection	287
11.3.4 Format of data backup/resume file.....	288
11.3.5 Level explanation of data backup/resume operation.....	289
11.4 Notes.....	289
CHAPTER 12 COMMUNICATION.....	290

12.1 ECN*000T Communication Software ECONComm.....	290
12.1.1 ECONComm's system (PC) requirements.....	290
12.1.2 Software interface.....	290
12.1.3 Receiving files (CNC→PC).....	291
12.1.4 Sending files (CNC→PC).....	293
12.1.5 Part program management	295
12.2 Preparatory before Communication	296
12.2.1 Communication cable connection	296
12.2.2 Communication setting baud rate	296
CHAPTER 13 MACHINING EXAMPLES	298
13.1 Programming	299
13.2 Program Input	301
13.2.1 View a saved program	301
13.2.2 Creating a new program	301
13.3 Checkout a Program.....	302
13.3.1 Graphic setting	302
13.3.2 Program check.....	302
13.4 Toolsetting and Running	303

III Connection

CHAPTER 1 NSTALLATION LAYOUT.....	307
1.1 ECN*000T System Connection	307
1.1.1 ECN*000T rear cover interface layout	308
1.1.2 Interface explanation	308
1.2 ECN*000T Installation.....	308
1.2.1 ECN*000T external dimensions.....	308
1.2.2 Preconditions of the cabinet installation.....	308
1.2.3 Measures against interference.....	308
CHAPTER 2 DEFINITION & CONNECTION OF INTERFACE SIGNALS.....	310
2.1 Connection to Drive Unit	310
2.1.1 Drive interface definition	310
2.1.2 Code pulse and direction signals.....	310
2.1.3 Drive unit alarm signal nALM.....	310
VIII	
2.1.4 Axis enable signal nEN.....	311
2.1.5 Pulse disable signal nSET.....	311
2.1.6 Zero signal nPC.....	311
2.1.7 Connection to a drive unit	313
2.2 Being Connected with Spindle Encoder.....	314
2.2.1 Spindle encoder interface definition.....	314
2.2.2 Signal explanation	314
2.2.3 Being connected with spindle encoder interface.....	314

2.3 Being Connected with MPG (Manual Pulse Generator).....	315
2.3.1 MPG interface definition	315
2.3.2 Signal explanation	315
2.4 Spindle Interface.....	316
2.4.1 Spindle interface definition.....	316
2.4.2 Connection to inverter	317
2.5 ECN*000T / ECN*000T-V being Connected with PC	317
2.5.1 Communication interface definition	317
2.5.2 Communication interface connection.....	317
2.6 Power Interface Connection	318
2.7 I/O Interface Definition	318
2.7.1 Input signal	320
2.7.2 Output signal	321
2.8 I/O Function and Connection	323
2.8.1 Stroke limit and emergency stop.....	323
2.8.2 Tool change control	325
2.8.3 Machine zero return.....	332
2.8.4 Spindle control	338
2.8.5 Spindle switching volume control.....	341
2.8.6 Spindle automatic gearing control.....	342
2.8.7 Spindle eight-point orientation function.....	343
2.8.8 Spindle Cs axis control function.....	347
2.8.9 Multiple spindle function	350
2.8.10 Rigid tapping function	354
2.8.11 External cycle start and feed hold.....	355
2.8.12 Cooling control.....	356
2.8.13 Lubricating control	356
2.8.14 Chuck control.....	358
2.8.15 Tailstock control	360
2.8.16 Low pressure detection.....	361
2.8.17 Hydraulic control (only applied to 980TDc-V)	362
2.8.18 Safety door detection.....	363
2.8.19 CNC macro variables.....	363
2.8.20 External override	364
2.8.21 External MPG.....	364
2.8.22 Tri-color Lamp.....	365
2.8.23 Extended M Function.....	365
2.8.24 HOLD Control.....	367
2.8.25 Gear/tool number display	367
2.9 Commonly Use Symbol of Electricity Drawing.....	367
CHAPTER 3 PARAMETERS.....	369
Contents IX	
3.1 Parameter Description (by sequence).....	369
3.1.1 Bit parameter	369

3.1.2 Data parameter.....	378
3.1.3 PLC parameters (standard PLC definition)	393
3.2 Parameter Description (by function sequence)	397
3.2.1 X, Z, Y, 4 th ,5 th axis control logic.....	397
3.2.2 Acceleration & deceleration control	398
3.2.3 Precision compensation.....	400
3.2.4 Machine protection	402
3.2.5 Machine zero return.....	403
3.2.6 Threading function	406
3.2.7 Spindle control	406
3.2.8 Tool compensation.....	409
3.2.9 Tool life management function	409
3.2.10 Tool wear parameter	409
3.2.11 Edit and display.....	410
3.2.12 Communication setting	410
3.2.13 MPG Parameters	410
3.2.14 PLC axis control function	411
3.2.15 Skip function	411
3.2.16 Automatic tool setting function	411
3.2.17 Input and output function in metric and inch system	412
3.2.18 Parameters related to arc turning	412
3.2.19 Parameters related to the additional	413
CHAPTER 4 MACHINE DEBUGGING METHODS AND MODES	415
4.1 Emergency Stop and Limit	415
4.2 Drive Unit Configuration.....	415
4.3 Gear Ratio Adjustment.....	415
4.4 ACC&DEC Characteristic Adjustment.....	416
4.5 Mechanical (Machine) Zero Adjustment.....	417
4.6 Spindle Adjustment	419
4.6.1 Spindle encoder	419
4.6.2 Spindle brake.....	419
4.6.3 Switch volume control of spindle speed.....	420
4.6.4 Analog voltage control of spindle speed	420
4.7 Backlash Offset.....	420
4.8 Tool Post Debugging.....	421
4.9 Step/MPG Adjustment.....	422
CHAPTER 5 DIAGNOSIS MESSAGE	423
5.1 CNC Diagnosis	423
5.1.1 I/O status and data diagnosis message.....	423
5.1.2 CNC motion state and data diagnosis message	423
5.1.3 Diagnosis key.....	424
5.1.4 Others.....	425
5.2 PLC State.....	425
5.2.1 X address (machine→PLC , defined by standard PLC ladders.....	425

5.2.2 Y address (PLC→machine, defined by standard PLC ladders)	427
5.2.3 Machine panel	429
X	
5.2.4 F address(CNC→PLC).....	431
5.2.5 G address (PLC→CNC)	435
5.2.6 Address A (message display requery signal, defined by standard PLC ladders) ..	440
5.2.7 K address (K parameter, standard PLC definition)	441
5.3 PLC Data	443
5.3.1 Timer address T(defined by standard PLC ladders)	443
5.3.2 Counter address C(Defined by standard PLC Ladders)	444
5.3.3 Timer presetting address DT(Defined by standard PLC ladders).....	444
5.3.4 Counter presetting address DC	445
CHAPTER 6 MEMORIZING PITCH ERROR COMPENSATION	446
6.1 Function Description	446
6.2 Specification	446
6.3 Parameter Setting.....	446
6.3.1 Pitch compensation	446
6.3.2 Pitch error compensation number.....	446
6.3.3 Pitch error compensation interval	447
6.3.4 Offset value	447
6.4 Notes of Offset Setting.....	447
6.5 Setting Examples of Offset Parameters.....	447

IV Appendix

Appendix ECN*000T,dimension.....	451
Appendix II ECN*000T-B outline dimension	452
Appendix III Outline Dimension of Accessional Panel AP01.....	452
Appendix IV Outline Dimension of Accessional Panel AP02.....	453
Appendix V Outline Dimension of Accessional Panel AP03.....	453
Appendix VI Outline Dimension of I/O deconcentrator MCT01A.....	454
Appendix VII Outline Dimension of I/O deconcentrator MCT02	454
Appendix VIII Delivery standard parameter	455
Appendix IX Alarm list.....	462
Appendix X Operation list	470

I Programming

Chapter 1 Programming

CHAPTER 1 PROGRAMMING

1.1 ECN*000T Introduction



1.1.1 Product introduction

ECN*000T is a new upgraded software, its software, hardware based on ECON, its structure horizontal and vertical, matched with 8.4 inch color LCD, with 5 feed axes (including C axis),

2 analog spindles, least input increment 0.1 μ m. It uses the graphic interface, friendly human-machine operation window. It can realize the PLC on-line display, real-time monitoring and MPG trial-cut function.

Being upgraded product of ECON MST71X0 series, ECN*000T CNC turning is the better choice.

ECN*000TT

X, Z, Y, 4th, 5th ; axis name and axis type of Y, 4th, 5th can be defined

2ms interpolation period, control precision 1 μ m, 0.1 μ m

Max. speed 60m/min (up to 24m/min in 0.1 μ m)

Adapting to the servo spindle to realize the spindle continuously positioning, rigid tapping, and the rigid thread machining

Built-in multi PLC programs, and the PLC program currently running can be selected

G71 supporting flute contour cycle cutting

Statement macro command programming, macro program call with parameter

Metric/inch programming, automatic toolsetting, automatic chamfer, tool life management function

Chinese, English, display can be selected by parameters.

USB interface, U disc file operation, system configuration and software

2-channel 0V~10V analog voltage output, two-spindle control
 1-channel MPG input, MPG function
 41 input signals and 36 output signals
 Appearance installation dimension and command system are compatible with those of
 ECON, ECON980TDa

1.1.2 Technical specification

Controllable axes
 Controllable axes: 2 (X, Z)
 Link axes: 2
 PLC controllable axes: 3 (X, Z)
Feed axis function
 Least input increment: 0.001mm (0.0001inch) and 0.0001mm (0.00001inch)
 Least command increment: 0.001mm (0.0001inch) and 0.0001mm (0.00001inch)
 Position command range: $\pm 99999999 \times$ least command unit
 Rapid traverse speed: max. speed 60m/min in 0.001mm command unit, max. speed
 24m/min in 0.0001mm command unit
 Rapid override: F0, 25%, 50%, 100%
 Feedrate override: 0~150% 16 grades to tune
 Interpolation mode: linear interpolation, arc interpolation(three-point arc interpolation),
 thread interpolation, ellipse interpolation, parabola interpolation and rigid tapping
 Automatic chamfer function

Thread function
 General thread(following spindle)/rigid thread
 Single/multi metric, inch straight thread, taper thread, end face thread, constant pitch
 thread and variable pitch thread
 Thread run-out length, angle, speed characteristics can be set
 Thread pitch: 0.01mm~500mm or 0.06 tooth/inch~2540 tooth/inch

Acceleration/deceleration function
 Cutting feed: linear
 Rapid traverse: linear, S
 Thread cutting: linear, exponential
 Initial speed, termination speed, time of acceleration/deceleration can be set by
 parameters

Spindle function
 2-channel 0V~10V analog voltage output, two-spindle control
 1-channel spindle encoder feedback, spindle encoder line can be set (100p/r~5000p/r)
 Transmission ratio between encoder and spindle: (1~255) : (1~255)
 Spindle speed: it is set by S or PLC, and speed range: 0r/min~9999r/min
 Spindle override: 50%~120% 8 grades tune
 Spindle constant surface speed control
 Rigid tapping

Tool function
 Chapter 1 Programming
 Tool length compensation

Tool nose radius compensation (C)
Tool wear compensation
Tool life management
Toolsetting mode: fixed-point toolsetting, trial-cut toolsetting, reference point return toolsetting, automatic toolsetting
Tool offset execution mode: modifying coordinate mode, tool traverse mode

Precision compensation

Backlash compensation
Memory pitch error compensation

PLC function

Two-level PLC program, up to 5000 steps, the 1st program refresh period 8ms
PLC program communication download
PLC warning and PLC alarm
Many PLC programs (up to 16PCS), the PLC program currently running can be selected

Basic I/O: 41 input signals /36 output signals

Man-machine interface

8.4" wide screen LCD, resolution: 640×480
Chinese, English, Spanish, Russian display
Planar tool path display
Real-time clock

Operation management

Operation mode: edit, auto, MDI, machine zero return, MPG/single, manual, program zero return

Multi-level operation privilege management
Alarm record

Program edit

Program capacity: 40MB, 384 programs (including subprograms and macro programs)
Edit function: program/block word search, modification, deletion
Program format: ISO command, statement macro command programming, relative coordinate, absolute coordinate and compound coordinate programming
Program call: macro program call with parameter, 4-level program built-in

Communication function

RS232: two-way transmitting part programs and parameters, PLC program, system software serial upgrade

USB: U file operation, U file directly machining, PLC program, system software U upgrade

Safety function

Emergency stop
Hardware travel limit
Software travel check
Data backup and recovery

G command table

code	Function	code	function
G00	Rapid traverse(positioning)	G50	Float coordinate setting
G01	Linear Interpolation	G54- G59	Set workpiece coordinate
G02	Clockwise circular interpolation	G65	Macro calling(no modal)
G03	Anti-clockwise circular interpolation	G71	Axial roughing cycle
G04	Pause/ Orientation	G72	Radial roughing cycle
G17	Plane selection	G73	Closed cutting cycle
G18	Plane selection	G70	Finishing cutting cycle
G19	Plane selection	G74	Axial grooving cycle
G10	Data input available	G75	Radial roughing cycle
G11	Data input cancel	G76	Multiple threading cutting
G20	Inch unit	G80	Rigid tapping status cancel
G21	Metric input	G84	Axial rigid tapping
G28	Return to home position	G88	Radial rigid tapping
G30	Return to machine 2,3,4 ref.point	G90	Axial cutting cycle
G31	Skip function	G92	Threading cycle
G32	Constant pitch threading	G94	Radial cutting cycle
G33	Z axes tapping cycle	G96	Constant speed cutting
G34	Variable pitch threading	G97	Constant speed cutting cancel
G40	Cancel tool nose radius compensation	G98	Feed per minute
G41	Left tool nose radius compensation	G99	Feed per revolution
G42	Right tool nose radius compensation		

1.1.3 Environment and conditions

ECN*000T storage delivery, working environment as follows:

Item Working conditions Storage delivery conditions

Ambient temperature 0℃~45℃ -40℃~+70℃

Ambient humidity ≤90%(no freezing) ≤95%(40℃)

Atmosphere pressure 86 kPa~106 kPa 86 kPa~106 kPa

Altitude ≤1000m ≤1000m

Chapter 1 Programming

1.1.4 Power supply

ECN*000T can normally run in the following AC input power supply.

Voltage: within(0.85~1.1)×200V (rated AC input voltage);

Frequency: 49Hz~51Hz continuously changing

1.1.5 Guard

ECN*000T guard level is not less than IP20.

1.2 CNC System and CNC Machine

CNC machine tool is an electro-mechanical integrated product, composed of Numerical Control Systems of Machine Tools, machines, electric control components, hydraulic components,

pneumatic components, lubricant, cooling and other subsystems (components), and CNC systems of machine tools are control cores of CNC machine tools. CNC systems of machine tools are made up of computerized numerical control(CNC), servo (stepper) motor drive devices, servo (or stepper) motor etc.

Operational principles of CNC machine tools: according to requirements of machining technology, edit user programs and input them to CNC, then CNC outputs motion control commands to the servo (stepper) motor drive devices, and last the servo (or stepper) motor completes the cutting feed of machine tool by mechanical driving device; logic control commands in user programs to control spindle start/stop, tool selections, cooling ON/OFF, lubricant ON/OFF are output to electric control systems of machine tools from CNC, and then the electric control systems control output components including keys, switches, indicators, relays, contactors and so on. Presently, the electric control systems are employed with Programmable Logic Controller (PLC) with characteristics of compact, convenience and high reliance. Thereof, the motion control systems and logic control

systems are the main of CNC machine tools.

ECN*000T Turning Machine CNC system has simultaneously motion control and logic control function to control two axes of CNC machine tool to move, and has nested PLC function. Edit PLC programs (ladder diagram) according to requirements of input and output control of machine tool and then download them to ECN*000T Turning Machine CNC system, which realizes the required electric control requirements of machine tool, is convenient to electric design of machine tool and reduces cost of CNC machine tool.

Software used to control ECN*000T Turning Machine CNC system is divided into system software (NC for short) and PLC software (PLC for short). NC system is used to control the display, communication, edit, decoding, interpolation and acceleration/deceleration, and PLC system for controlling explanations, executions, inputs and outputs of ladder diagrams. Standard PLC programs are loaded (except for the special order) when ECN*000T Turning Machine CNC System is delivered, concerned PLC control functions in following functions and operations are described according to control logics of standard PLC programs, marking with “Standard PLC functions” in . Refer to Operation

Manual of machine manufacturer about functions and operations of PLC control because the machine manufacturer may modify or edit PLC programs again.



Fig. 1-1

Programming is a course of workpiece contours, machining technologies, technology parameters and tool parameters being edit into part programs according to special CNC programming G codes. CNC machining is a course of CNC controlling a machine tool to complete machining of workpiece according requirements of part programs.

Technical flow of CNC machining is as following Fig. 1-2.

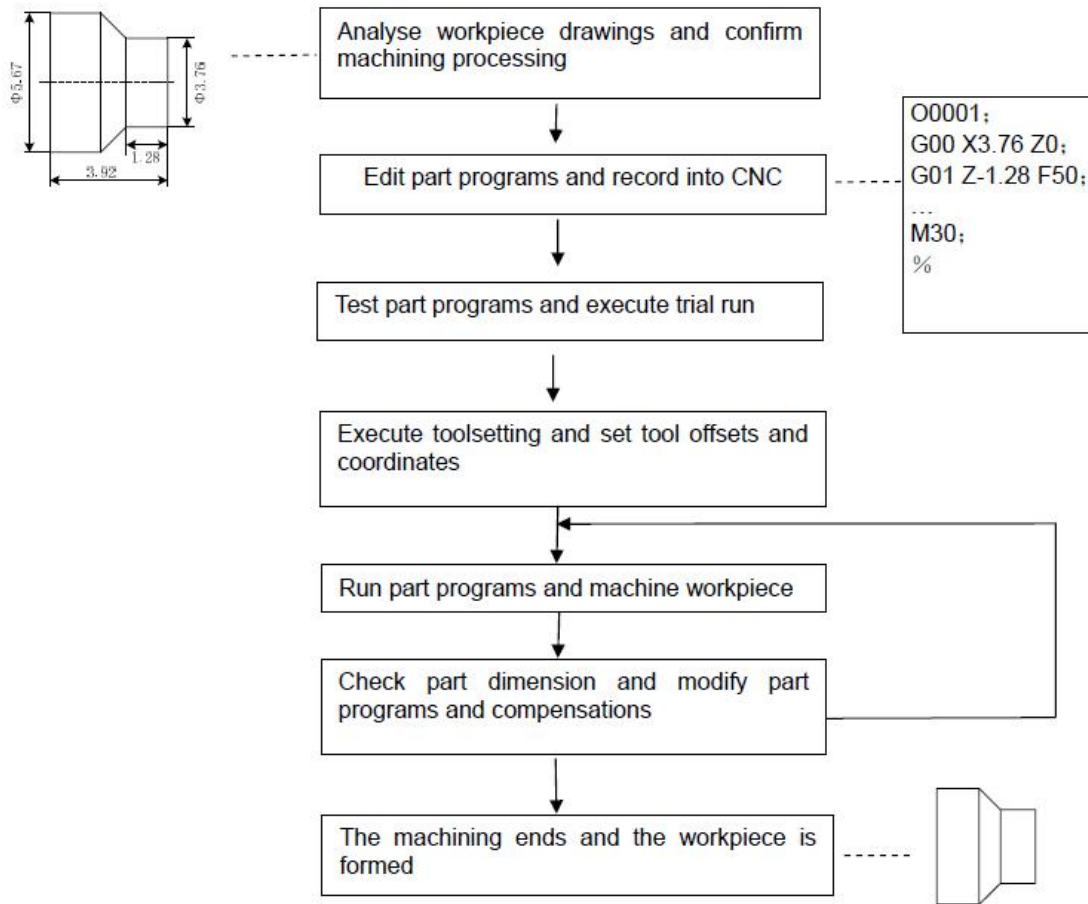


Fig. 1-2

1.3 Programming Fundamentals

1.3.1 Coordinates definition

Sketch map of CNC turning machine is as follows:

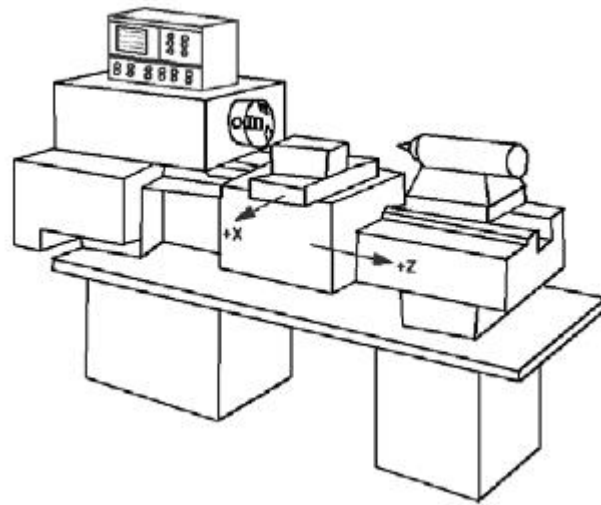


Fig. 1-3

ECN*000T uses a rectangular coordinate system composed of X, Z axis. X axis is perpendicular with axes of spindle and Z axis is parallel with axes of spindle; negative directions of them approach to the workpiece and positive ones are away from it.

There is a front tool post and a rear tool post of NC turning machine according to their relative position between the tool post and the spindle, Fig. 1-4 is a coordinate system of the front tool post and Fig. 1-5 is a rear tool post one. It shows exactly the opposite of X axes, but the same of Z axes from figures. In the manual, it will introduce programming application with the front tool post coordinate system in the following figures and examples.

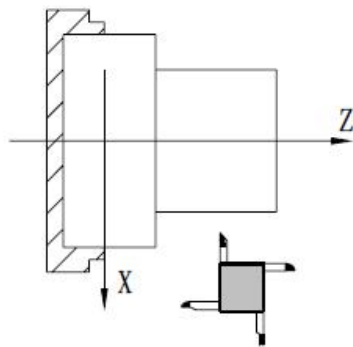


Fig.1-4 Front tool post coordinate system

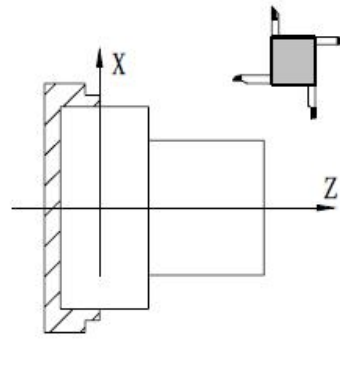


Fig.1-5 Rear tool post coordinate system

1.3.2 Machine coordinate system, Machine Zero and machine reference point

Machine tool coordinate system is a benchmark one used for CNC counting coordinates and a fixed one on the machine tool. **Machine tool zero** is a fixed point which position is specified by zero switch or zero return switch on the machine tool. Usually, the zero return switch is installed on max. stroke in X, Z positive direction. Machine reference point is located at the position at which the machine zero value adding the data parameter No.114/No.115 value. When No.114/No.115 value is 0, the machine reference point coincides with the machine zero. The coordinates of machine reference point is the No.120/No.121 value. Machine zero return/G28 zero return is to execute the machine reference point return. After the machine zero return/machine reference point return is completed,

ECN*000T machine coordinate system which takes No.120 value as the reference point, which is referred to I Programming, Section 3.13.

Note: Do not execute the machine reference point return without the reference point switch installed on the machine, otherwise, the motion exceeds the travel limit and the machine to be damaged.

1.3.3 Workpiece coordinate system and Program Zero

The workpiece coordinate system is a rectangular coordinate system based on the part drawing, also called floating coordinate system. After the workpiece is installed on the machine, the absolute coordinates of tool's current position is set by G50 according to the workpiece's measure, and so the workpiece coordinate system is established in CNC. Generally, Z axis of the workpiece coordinate system coincides with the spindle axis. The established workpiece is valid till it is replaced by a new one. The system can set 6 workpiece coordinate systems G54~G59 in advance. Refer to I Programming, Section 3.18 about the details of workpiece coordinate system.

A sub workpiece coordinate system is created in a workpiece coordinate system, which is called as a local coordinate system. Refer to I Programming, Section 3.17 about the details of the local coordinate system.

The current position of workpiece coordinate system set by G50 is the program zero.

Note: Do not execute the machine reference point return without using G50 to set the workpiece coordinate system after power on, otherwise, the alarm occurs.

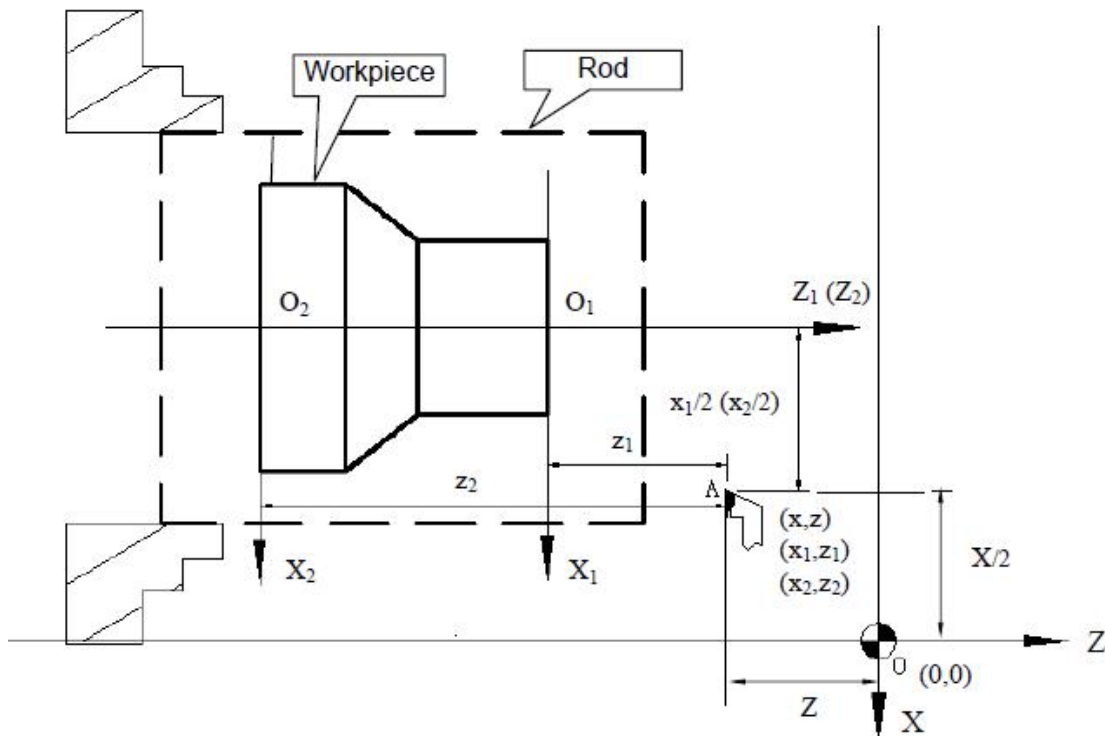


Fig. 1-6

In the above figure, XOZ is the coordinate system of machine tool, $X_1O_1Z_1$ is the workpiece coordinate system of X axis located at the heading of workpiece, $X_2O_2Z_2$ is the one of X axis located at the ending of workpiece, O point is the machine reference point, A point is the tool nose and Fig. 1-6

coordinates of A point in the above-mentioned coordinate systems is as follows:

A point in the machine tool coordinate system: (x,z);

A point in $X_1O_1Z_1$ coordinate system: (x_1,z_1) ;

A point in $X_2O_2Z_2$ coordinate system: (x_2,z_2) .

1.3.4 Interpolation function

Interpolation is defined as a planar or three dimensional contour formed by path of 2 or multiple axes moving at the same time, also called **Contour control**. The controlled moving axis is called link axis when the interpolation is executed. The moving distance, direction and speed of it are controlled synchronously in the course of running to form the required Composite motion path. Positioning control is defined that motion end point of one axis or multiple axes instead of the motion path in the course of running is controlled.

ECN*000T X and Z axis are link axes and 2 axes link CNC system. The system possesses linear, circular and thread interpolation function.

Linear interpolation: Composite motion path of X, Z axis is a straight line from starting point to end point.

Circular interpolation: Composite motion path of X, Z axis is arc radius defined by R or the circle center (I, K) from starting point to end point.

Thread interpolation: Moving distance of X or Z axis or X and Z axis is defined by rotation angle of spindle to form spiral cutting path on the workpiece surface to realize the thread cutting. For thread interpolation, the feed axis rotates along with the spindle, the long axis moves one pitch when the spindle rotates one rev, and the short axis and the long axis directly interpolate.

Example:

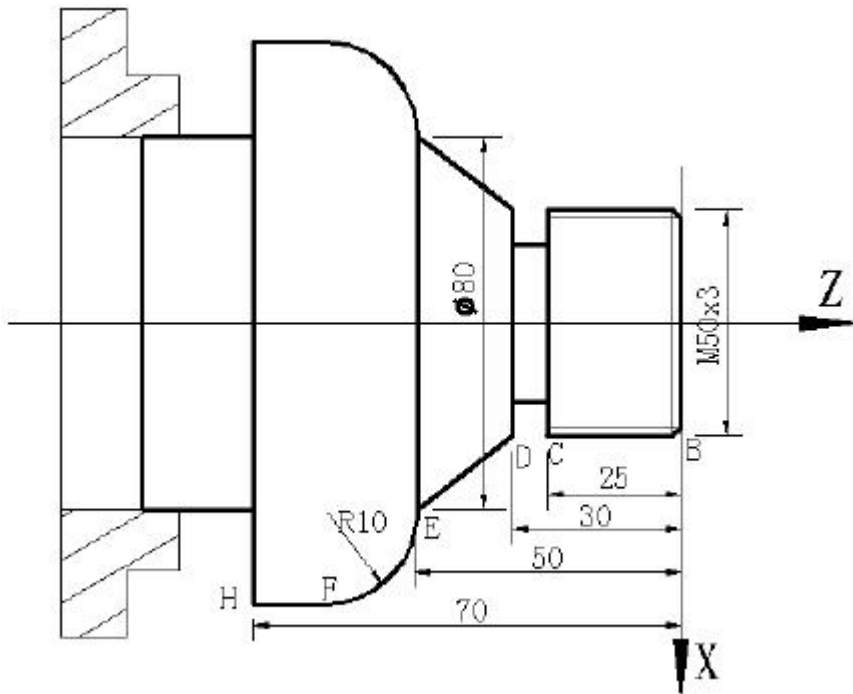


Fig. 1-7

...

G32 W-27 F3; (B→C; thread interpolation)

G1 X50 Z-30 F100;
 G1 X80 Z-50; (D→E; linear interpolation)
 G3 X100 W-10 R10; (E→F; circular interpolation)

...
 M30;

1.3.5 Absolute programming and incremental programming

Specify coordinate values of path's end point or target position in programming and there are 3 kinds of programming method according to coordinate values in programming: absolute programming,

incremental programming and compound programming.

Programming with X/Z axis absolute coordinate value to program (present with X, Z) is defined to be the absolute programming;

Programming with X/Z axis incremental movement (present with U, W) is defined to be the incremental programming;

In the system, X, Z axis separately uses the absolute programming and incremental program, which is called the compound programming.

Example: A→B linear interpolation

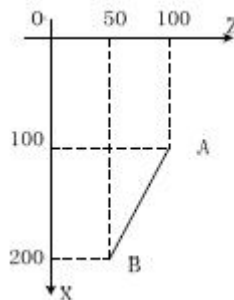


Fig.1-8

Absolute programming: G01 X200 Z50;

Incremental programming: G01 U100 W-50;

Compound programming: G01 X200 W-50; or G01 U100 Z50

Note: When there are command address X/ U or Z/ W at the same time, X/Z command value is valid.

Example: G50 X10 Z20;

G01 X20 W30 U20 Z30; 【End point of the block (X20, Z30)】

1.3.6 Diameter programming and radius programming

Programming methods of X coordinate values are divided into: diameter programming and radius programming.

Diameter programming: when NO.001 Bit2 is 0, X input command value is in diameter and X coordinate is in diameter at the moment;

Radius programming: when NO.001 Bit2 is 1, X input command value is in radius and X coordinate is in radius at the moment.

Addresses relevant to diameter or radius programming

	Address	Explanation	Diameter programming	Radius programming
Addresses relevant to diameter or radius	X	X coordinate	In diameter	In radius
		G50 setting X coordinate		
	U	X increment	In diameter	In radius
		X finishing allowance in G71, G72, G73	In diameter	In radius

	Address	Explanation	Diameter programming	Radius programming
programming		Moving distance of tool retraction when cutting to the end point in G74	In diameter	In radius

Except for addresses and data in Table 1-1, others (arc radius, taper in G90) are unrelated to diameter or radius programming, and their input values in X direction are defined by the radius.

Note: The diameter programming is used except for the special description in the following explanation.

1.4 Structure of an NC Program

User needs to compile part programs (called program) according to command formats of CNC system. CNC system executes programs to control the machine tool movement, the spindle starting/stopping, the cooling and the lubricant ON/OFF to complete the machine of workpiece. Program example:

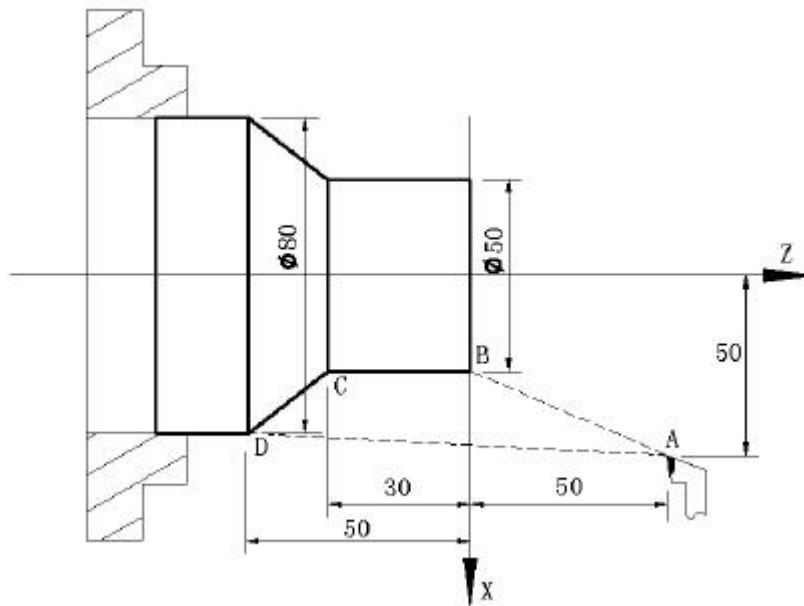


Fig. 1-9

The tool leaves the path of A→B→C→D→A after the above-mentioned programs are executed.

1.4.1 General structure of a program

A **program** consists of blocks. A block begins with a block number (it can be omitted) and several words ending with “;”. General structure of a program is shown in Fig.1-10:

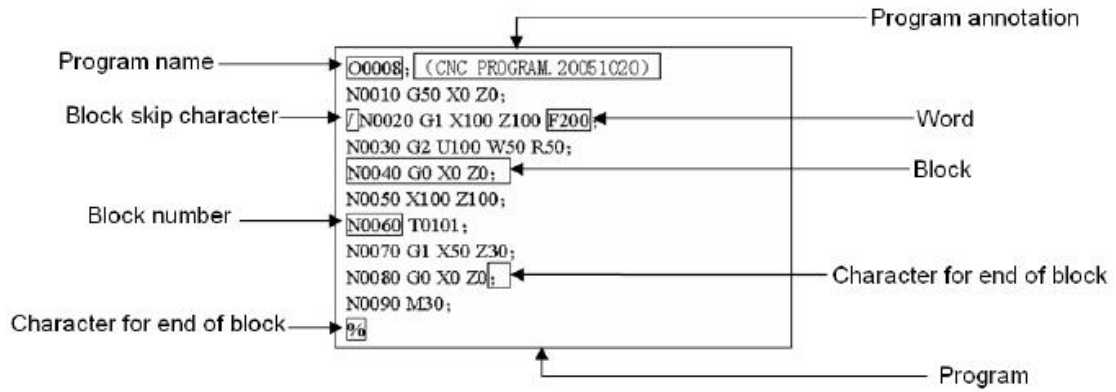
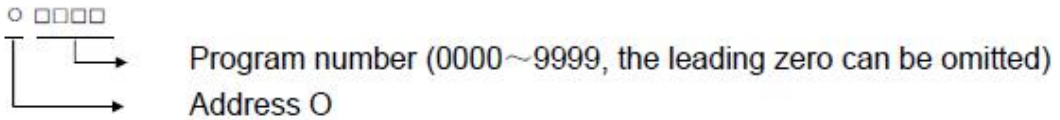


Fig. 1-10 Structure of a program

Program name

There are most 384 programs stored in ECN*000T. To identify it, each program has only one program name (there is no the same program name) beginning with command address O and the following 4 digits.



Word

A word is the basic command unit to command CNC system to complete the control function, composed of an English letter (called command address) and the following number (operation command with/without sign). The command address describes the meaning of its following operation

command and there may be different meaning in the same command address when the different words are combined together. All words of ECN*000T are in Table 1-2.

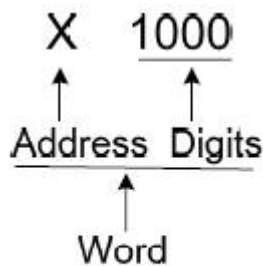


Table 1-2 Word table

Address Command value range Function meaning Unit

Address	Command value range	Function meaning	Unit
O	0~9999	Program name	
N	0~9999	Block number	
G	00~99	Preparatory function	
X	-99999999~99999999	X coordinate	Relevant to IS-B, IS-C
	0~99999.999(s)	Pause time	
Z	-99999999~99999999	Z coordinate	Relevant to IS-B, IS-C
Y	-99999999~99999999	Y coordinate	Relevant to IS-B, IS-C
U	-99999999~99999999	X increment	Relevant to IS-B, IS-C
	0~99999.999(s)	Pause time	
	-.99999~.99999	X finishing allowance in G71,G72, G73	Relevant to IS-B, IS-C

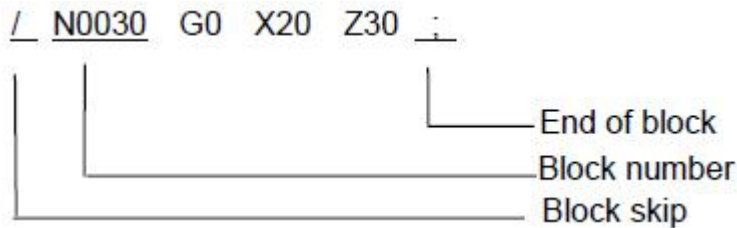
Address	Command value range	Function meaning	Unit
	1~99999	Cutting depth in G71	Relevant to IS-B, IS-C
	-.99999999~99999999	X tool retraction clearance in G73	Relevant to IS-B, IS-C
W	-99999999~99999999	Z increment	Relevant to IS-B, IS-C
	1~99999	Cutting depth in G72	Relevant to IS-B, IS-C
	-.99999~.99999	Z finishing allowance in G71,G72, G73	Relevant to IS-B, IS-C
	-99999999~99999999	Z tool retraction in G73	Relevant to IS-B, IS-C
V	-99999999~99999999	Y increment	Relevant to IS-B, IS-C
R	-99999999~99999999	Arc radius	Relevant to IS-B, IS-C
	1~99999	Tool retraction in G71, G72	Relevant to IS-B, IS-C
	1~9999 (times)	Roughing cycle times in G73	
	1~99999	Tool retraction clearance in G74, G75	Relevant to IS-B, IS-C
	1~99999	Tool retraction clearance from end point in G74, G75	Relevant to IS-B, IS-C
	1~99999999	Finishing allowance in G76	Relevant to IS-B, IS-C
	-99999999~99999999	Taper in G90, G92, G94, G96	Relevant to IS-B, IS-C
I	-99999999~99999999	X vector between arc center and starting point	Relevant to IS-B, IS-C
	0.06~25400 (tooth/inch)	Metric thread teeth	

K	-99999999~99999999	Z vector between arc center and starting point	Relevant to IS-B, IS-C
F	0~8000 (mm/min)	Feedrate per minute	
	0.0001~500(mm/r)	Feedrate per rev	
	0.001~500 (mm)	Metric thread lead	
S	0~9999 (r/min)	Spindle speed specified	
	00~04	Multi-gear spindle output	
T	01~32	Tool function	
M	00~99	Miscellaneous function output, program execution flow	
	9000~9999	Subprogram call	
P	0~9999999 (0.001s)	Pause time	
	0~9999	Calling times of subprogram number	
	0~999	Subprogram call times	
	0~9999999	X circle movement in G74, G75	Relevant to IS-B, IS-C
		Thread cutting parameter in G76	
	0~99999	Initial block number of finishing in the compound cycle command	
	1~9999999	Parabola mouth size in G7.2, G7.3	Relevant to IS-B, IS-C

Address	Command value range	Function meaning	Unit
Q	0~9999	End block number of finishing in the compound cycle	
	0~9999999	Z circle movement in G74, G75	Relevant to IS-B, IS-C
	1~9999999	First cut-in depth in G76	Relevant to IS-B, IS-C
	1~9999999	Min. cut-in depth in G76	Relevant to IS-B, IS-C
	0~360000	Offset angle between one-turn signal and starting point of thread cutting at the initial angle in G32	
	0~9999	Angle between long axis of the ellipse and Z in G6.2, G6.3	
	0~9999	Angle between long axis of the ellipse and Z in G7.2, G7.3	
A	0~99999999	Length of long radius of ellipse in G6.2, G6.3	Relevant to IS-B, IS-C
B	0~99999999	Length of short radius of ellipse in G6.2, G6.3	Relevant to IS-B, IS-C
H	01~99	Operand in G65	

Block

A block which is basic unit of CNC program consists of a sequence of words, ending with “;” or “*” . There is the character “;” or “*” between blocks. “;” is used to separate blocks in the manual as follows:



One block may be with a number of words or only with “;” ending character(EOB) instead of words. There must be one or more blank space between many words.

There is only one for other addresses except for N, G, S, T, H, L in one block, otherwise the system alarms. The last word in the same address is valid when there are more N, G, S, T, H, L in the same block. The last G code is valid when there are more G codes which are in the same group in one block.


Block number

A block number consists of an address N and its following 4-digit: N0000~N9999, and the leading zero can be omitted. The block number must be at the beginning of block, otherwise the block is invalid.

The block number can be omitted, but there must be the block number when the program calls/skips the target block. The increment of block number is at will and it better to increase or decrease the sequence of block number in order to conveniently search and analyze programs. When “Automatic number” in the switch window is set to “ON”, block numbers will be automatically created incrementally and their increment is defined by No.42.

Character for block skip



Insert “/” in the front of block and startup  when some block cannot be executed (cannot be deleted), and the system skips the block and executes the next one. The block with “/” in the front of it is executed if the block skip switch is not started.

Character for end of a program

“%” is an ending character of program. “%” is a mark of communication ended when the program is transmitted. The system will automatically insert “%” at the end of program.

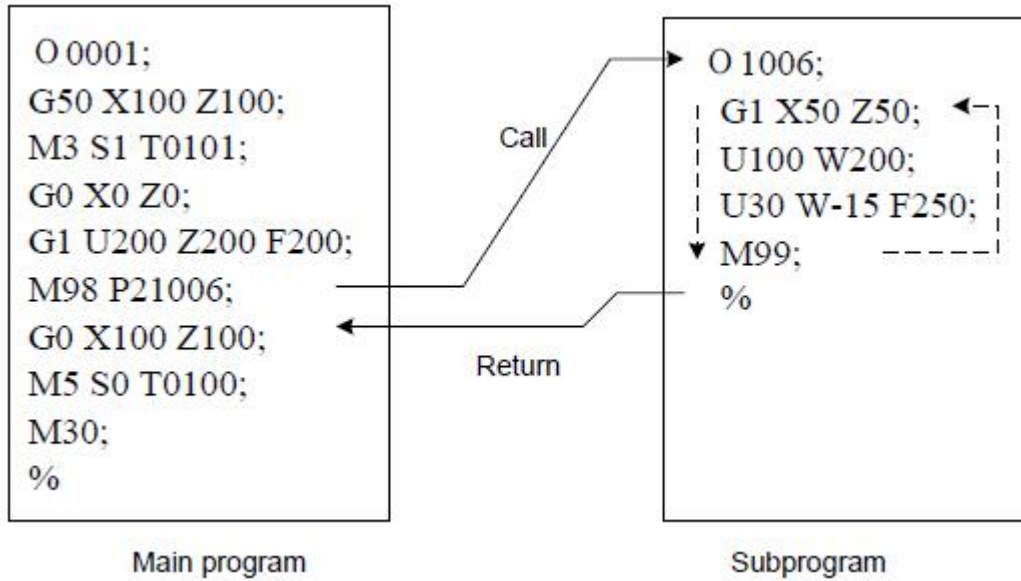
Program annotation

A program annotation has less than 20 characters (10 Chinese characters) for each program lies in a bracket following its program name and is expressed only in English and digitals in CNC system; it can be edited in Chinese in PC and displayed in Chinese in CNC system after being downloaded.

1.4.2 Main program and subprogram

To simply the programming, when the same or similar machining path and control procedure is used many times, its program commands are edited to a sole program to call. A program which calls the program is the main program and the called program (end with M99) is subprogram. They both take up the program capacity and storage space of system. The subprogram has own name, and can be called at will by the main program and also can run separately. The system returns to the main program to continue when the subprogram ends as follows.


Main program




1.5 Program Run

1.5.1 Sequence of program run

Running the current open program must be in Auto mode. ECN*000T cannot open two or more programs at the same, and runs only program any time. When the first block is open, the cursor is located in the heading of the first block and can be moved in Edit mode. In the run stop state in

Auto mode, the program starts to run by the cycle start signal ( is pressed or external cycle start signal) from a block pointed by current cursor, usually blocks are executed one by one according to their programming sequence, the program stops running till executing M02 or M30. The cursor moves along with program running and is located at the heading of the current block. Sequence and state of program running are changed in the followings:


The program stops running after pressing  or emergent stop key;


The program stops running when the system or PLC alarms;


The program runs and single block stops (the program run stops after the current block runs completely) in Edit, MDI mode, and then a block pointed by the current cursor starts running

after the system switches into Auto mode,  is pressed or external cycle start signal is switched on;


The program stops running in Manual(Jog), Handwheel (MPG), Single Block, Program Reference Point Return, Machine Reference Point Return mode and it continuously runs

from current position after the system is switched into Auto mode and  is pressed or the external cycle start signal is switched on;

The program pauses after pressing  or the external cycle start signal is switched off,

and it continuously runs from current position after pressing  or the external cycle start signal is switched on;

When Single Block is ON, the program pauses after every block is executed completely,

and then it continuously runs from the next block after  is pressed or the external cycle start signal is switched on;

Block with “/” in the front of it is not executed when the block skipping switch is ON;

The system skips to the target block to run after executing G65;

Please see I Programming, Section Three G Commands about execution sequence of G70~73;

Call corresponding subprograms or macro program to run when executing M98 or M9000~M9999; the system returns to main program to call the next block when executing

M99(if M99 specifies a target block number, the system returns to it to run) after the subprograms or macro programs run completely;

The system return to the first block to run and the current program is executed repetitively

when M99 is executed in a main program.

1.5.2 Execution sequence of word

There are many words (G, X, Z, F, R, M, S, T and so on) and most of M, S, T is transmitted to PLC by NC explaining and others are directly executed by NC. M98, M99, M9000~M9999, S word

used to specify the spindle speed r/min, m/min is directly executed by NC.

NC firstly executes G and then M commands when G codes and M00, M01, M02 and M30 are in the same block.

NC firstly executes G and then M commands(without transmitting M signal to PLC) when G codes and M98, M99, M9000~M9999 are in the same block.

When G codes and M, S, T executed by PLC are in the same block, PLC defines M, S, T and G to be executed simultaneously, or execute M, S, T after G codes. Please see User Manual of machine manufacturer for execution sequence of commands.

Execution sequence of G, M, S, T in the same block defined by ECN*000T standard PLC program is as follows: M3, M4, M8, M10, M12, M32, M41, M42, M43, M44, S□□, T□□□□ and G codes are executed simultaneously;

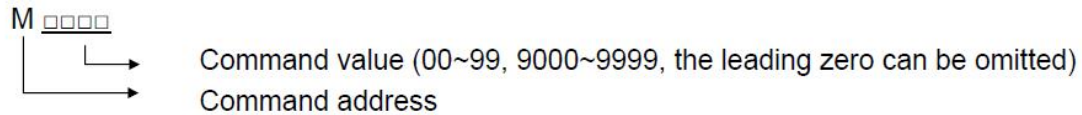
M5, M9, M11, M13, M33 after G codes are executed;

M00, M01, M02, M30 after other commands of current block are executed.

CHAPTER 2 MSTF COMMAND

2.1 M (Miscellaneous Function)

M command consists of command address M and its following 1~2 or 4 bit digits, used for controlling the flow of executed program or outputting M commands to PLC.



M98, M99, M9000~M9999 is executed by NC separately and NC does not output M commands to PLC.

M02, M30 are for ending of programs defined by NC, and NC outputs M commands to PLC which can control spindle OFF, cooling OFF and so on.

M98, M99, M9000~M9999 are for calling programs, M02, M30 are for ending of program which are not changed by PLC. Other M commands output to PLC and their function are defined by PLC.

Please refer to User Manual from machine manufacturer.

There is only one M command in one block, otherwise the system alarms.

Table 2-1 M commands to control program execution

Commands Functions

Commands	Functions
M02	End of program
M30	End of program
M98	Call subprograms
M99	Return from a subprogram; it is executed repeatedly when the program ends in M99(the current program is not called by other programs)
M9000~M9999	Call macro programs(their program numbers are more than 9000)

End of program M02

Command format: M02 or M2

Command function: In Auto mode, after other commands of current block are executed, the automatic run stops, and the cursor stops a block in M02 and does not return to the start of program. The cursor must return to the start of program when the program is executed again.

Besides the above-mentioned function executed by NC, M02 function is also defined by PLC ladder diagram as follows: current output of CNC is reserved after M02 is executed.

End of program run M30

Command format: M30

Command function: In Auto mode, after other commands of current block are executed in M30, the automatic run stops, the amount of workpiece is added 1, the tool nose radius compensation is cancelled and the cursor returns to the start of program (whether the cursor return to the start of program or not is defined by parameters).

If No.005 Bit 4 is set to 0, the cursor does not return to the beginning of program, and the cursor

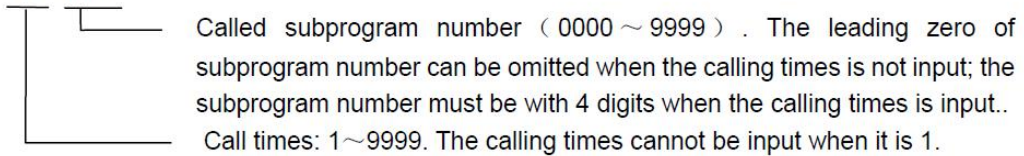
returns immediately after the program is executed completely when No.005 Bit 4 is set to 1. Except for the above-mentioned function executed by NC, M30 function is also defined by PLC

ladder diagram as follows: the system closes M03, M04 or M08 signal output and outputs M05 signal after M30 is executed.

Subprogram call M98

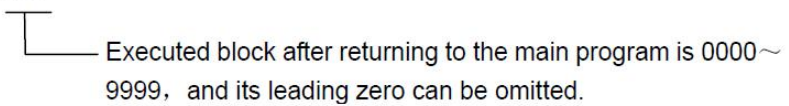
Command format:

M98 P○○○○□□□□



Return from subprogram M99

Command format: M99 P○○○○



Command function: After other commands of current block in the subprogram are executed, the system returns to the main program and continues to execute next block specified by P, and calls a block following M98 of current subprogram when P is not input. The current program is executed repeatedly when M99 is defined to end of program (namely, the current program is executed without calling other programs). M99 is invalid in MDI mode.

Example: Execution path of calling subprogram (with P in M99) as Fig. 2-1.

Execution path of calling subprogram (without P in M99) as Fig. 2-2.

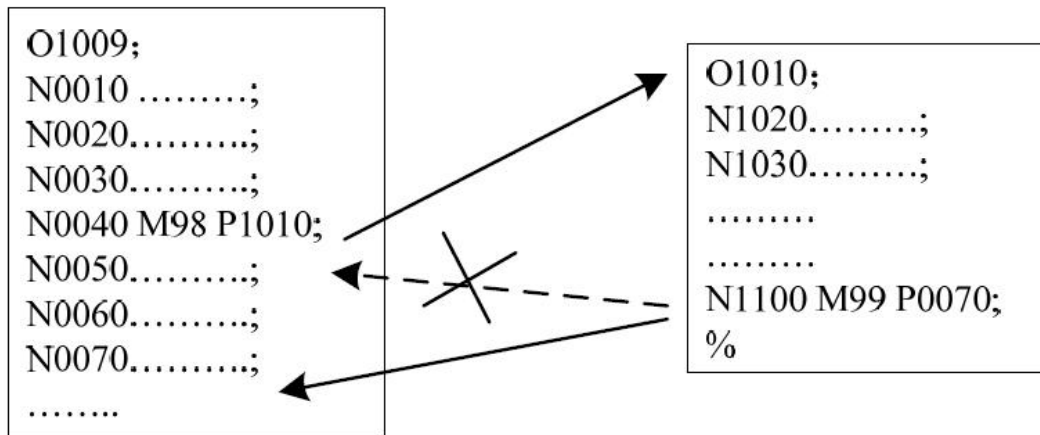


Fig. 2-1

Called subprogram number (0000 ~ 9999) . The leading zero of subprogram number can be omitted when the calling times is not input; the subprogram number must be with 4 digits when the calling times is input..
Call times: 1~9999. The calling times cannot be input when it is 1.

Executed block after returning to the main program is 0000~

9999, and its leading zero can be omitted.

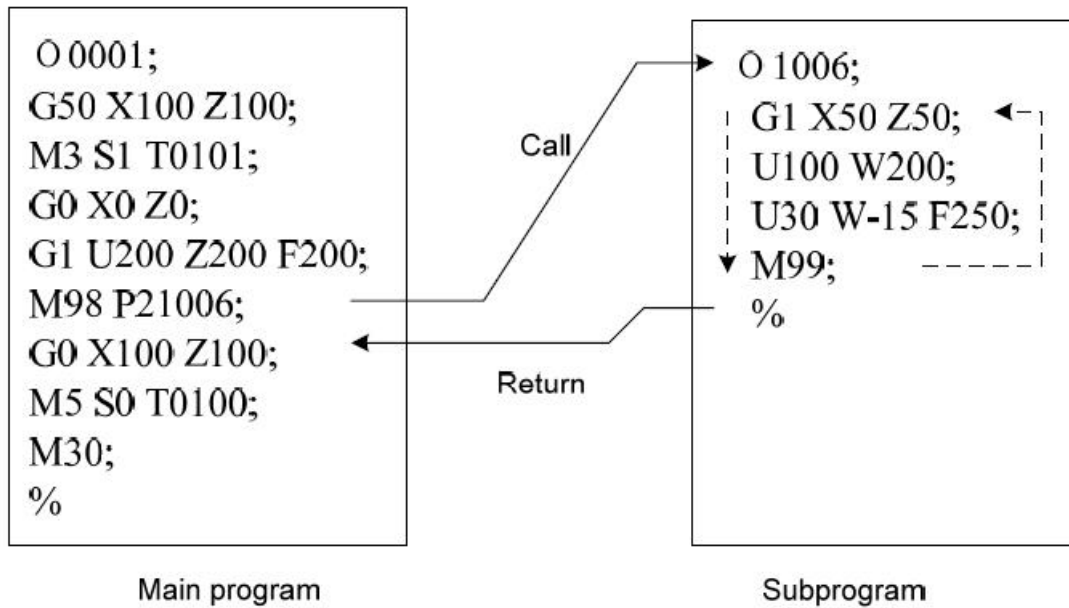


Fig. 2-2

Subprogram calls can be nested up to four levels as shown in Fig. 2-3.

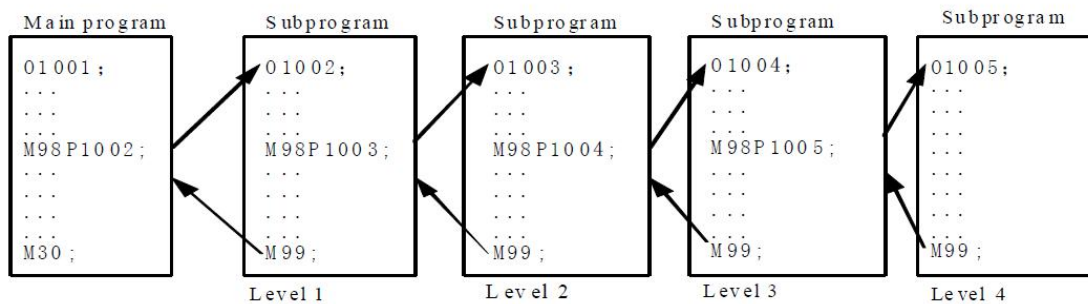


Fig. 2-3 Subprogram nesting

Macro program call M9000~M9999

Command format: $M\boxed{\quad}\boxed{\quad}\boxed{\quad}\boxed{\quad}$
└──────────┘ **9000~9999**

Command function: call macro programs corresponding to command values (O9000~O9999).

Macro programs: O9000~O9999 programs are for machine manufacturer, used for editing subprogram with special functions, called macro programs. The system must have 2-level operation

level (machine manufacturer) when editing O9000 ~ O9999, and macro programs calling commands

are executed to call with 3~5 operation level. M9000~M9999 are invalid in MDI mode.

M commands defined by standard PLC ladder diagram

Other M commands are defined by PLC except for the above-mentioned ones(M02, M30, M98, M99, M9000 ~ M9999). The following M commands are defined by standard PLC, and ECN*000T Turning Machine CNC system is used for controlling machine tool. Refer to

commands of machine manufacturer about functions, significations, control time sequence and logic of M commands.

M commands defined by standard PLC ladder diagram.

Code	Function	Remark
M00	Program pause	
M00	Program option stop	
M03	Spindle clockwise	Functions interlocked and states reserved
M04	Spindle counterclockwise	
*M05	Spindle stop	
M08	Cooling ON	Functions interlocked and states reserved
*M09	Cooling OFF	
M10	Tailstock forward	Functions interlocked and states reserved
M11	Tailstock backward	
M12	Chuck clamping	Functions interlocked and states reserved
M13	Chuck releasing	
M14	Spindle position control	Functions interlocked and states reserved
*M15	Spindle speed control	
M20	Spindle clamping	Functions interlocked and states reserved
*M21	Spindle releasing	
M24	The	Functions interlocked and states reserved
*M25	The	
M32	Lubricating	Functions interlocked and states reserved
*M33	Lubricating	
M63	The 2nd spindle rotation CCW	Functions interlocked and states reserved
M64	The 2nd spindle rotation CW	
*M65	The 2nd spindle stop	
*M41, M42, M43, M44	Spindle automatic gear shifting	Functions interlocked and states reserved

Table 2-2 M commands

Note: Commands with “*” defined by standard PLC is valid when power on.

Program stop M00

Command format: M00 or M0

Command function: After M00 is executed, the program stops and the system displays “Pause”, and then the program continuously runs after the cycle start key is pressed.

Program optional stop M01

Command format: M01 or M1

Command function: in AUTO, MDI mode, it is valid. Press and its indicator lights and the system enters the optional stop state, at the moment, the program stops run and the system displays “PAUSE” after M01 is executed, after the cycle start key is pressed, the program continuously runs. When the program optional

stop switch is not open, the program does not pause even if M01 runs.

Spindle CW, CCW and stop control M03, M04, M05

Command format: M03 or M3

M04 or M4;

M05 or M5.

Command function: M03: Spindle CW rotation;

M04: Spindle CCW rotation;

M05: Spindle stop.

Note: Refer to time sequence of output defined by standard PLC ladder in III INSTALLATION & CONNECTION.

Cooling control M08, M09

Command format: M08 or M8;

M09 or M9;

Command function: M08: Cooling ON;

M09: Cooling OFF.

Note: Refer to time sequence and logic of M08, M09 defined by standard PLC ladder in III INSTALLATION & CONNECTION.

Tailstock control M10, M11

Command format: M10;

M11;

Command function: M10: tailstock going forward;

M11: tailstock going backward.

Note: Refer to time sequence and logic of M10, M11 defined by standard PLC ladder in III INSTALLATION & CONNECTION

Chuck control M12, M13

Command format: M12;M13;

Command function: M12: chuck clamping;

M13: chuck releasing.

Note: Refer to time sequence and logic of M12, M13 defined by standard PLC ladder in III INSTALLATION & CONNECTION.

Spindle position/speed control switch M14, M15

Command format: M14; M15;

Command function: M14: spindle is in the position control mode from speed control mode;

M15: spindle is in speed control mode from the position control mode.

Note: Refer to time sequence and logic of M14, M15 defined by standard PLC ladder in III INSTALLATION Chapter 2 MSTF Command& CONNECTION.

Spindle clamped/released M20, M21

Command format: M20;

M21;

Command function: M20: spindle clamped

M21: spindle released

Note: Refer to time sequence and logic of M20, M21 defined by standard PLC ladder in III INSTALLATION& CONNECTION.

The 2nd spindle position/speed switch M24, M25

Command format: M24; M25;

Command function: M24: The 2nd spindle is switched from the speed control mode to the position control mode;

M25: The 2nd spindle is switched from the position control mode to the speed control mode.

Note: Refer to time sequence and logic of M24, M25 defined by standard PLC ladder in III INSTALLATION& CONNECTION.

Lubricating control M32, M33

Command format: M32; M33;

Command function: M32: lubricating ON;

M33: lubricating OFF.

Note: Refer to time sequence and logic of M32, M33 defined by standard PLC ladder in III INSTALLATION& CONNECTION.

Spindle automatic gear change M41, M42, M43, M44

Command format: M4n; (n=1, 2, 3, 4)

Command function: When the system executes M4n, the spindle changes to gear n.

Note: Refer to time sequence and logic of M41, M42, M43, M44 defined by standard PLC ladder in III INSTALLATION&CONNECTION.

Spindle 8-point orientation M50~M58

Command format: M5n; (n=0~8)

Command function: M50: cancel orientation state;

M5n(n=0~8): the spindle oriented to No. n point.

Note: Refer to time sequence and logic of M50~M58 defined by standard PLC ladder in III INSTALLATION & CONNECTION.

The 2nd spindle rotation CCW, rotation CW , stop M63, M64, M65

Command format: M63; M64; M65;

Command function:

M63: spindle rotation CCW;

M64: spindle rotation CW;

M65: spindle stop.

Note 1: The sequence of M63, M64, M65 defined by the standard PLC is the same that of M03, M04, M05.

Note 2: The function is enabled when the 2nd spindle function is valid.

2.2 Spindle Function

S command is used for controlling spindle speed and this ECN*000T has two modes to control it:
Spindle speed switching value control: S□□(2 digits command value)is executed by PLC, and PLC outputs switching value signal to machine tool to change spindle speed with grades.

Spindle speed analog voltage control: S□□□□(4 digits command value)specifies actual speed of spindle and NC outputs 0~10V analog voltage signal to spindle servo or converter to realize stepless spindle speed.

2.2.1 Spindle speed switching value control

Spindle speed is controlled by switching value when No.001 BIT4 is set to 0. There is only one S command in a block, otherwise the system alarms.

Their executing sequence is defined by PLC when S command and word for moving function are in the same block. Please refer to *User Manual* from machine manufacturer.

When spindle speed is controlled by switching value, ECN*000T Turning CNC system is used for machine tool and the time sequence and logic of executing S command is according to *User Manual* from machine manufacturer. Refer to S command defined by standard PLC of ECN*000T as follows:

Command format: S□□

00~04(the leading zero can be omitted): No.1~No.4 gear of spindle speed is controlled by switching value.

In spindle speed switching value control mode, after S signal transmits to PLC, the system dwells time defined by No.081, then return FIN signal, and the dwell time is called runtime of S command.S01, S02, S03, S04 output are reserved when resetting CNC.

S1~S4 output are invalid when CNC is switched on. The corresponding S signal output is valid and reserved, and others are cancelled at the same time when executing one of S01, S02, S03, S04. When executing S00, S1~S4 output are cancelled and only one of S1~S4 is valid at the same time.

2.2.2 Spindle speed analog voltage control

Spindle speed is controlled by analog voltage when No.001 BIT4 is set to 1.

Command format: S □□□□

0000~9999 (the leading zero can be omitted.):Spindle speed

Dwell time Start to execute S command Start to execute the following word or block

analog voltage control

Command function: The spindle speed is defined, and the system outputs 0~10V analog voltage to control spindle servo or converter to realize the stepless timing. S

command value is not reserved, and it is 0 after the system is switched on.

When the spindle speed analog voltage control is valid, there are 2 methods to input the spindle speed: the spindle fixed speed is defined by S command(r/min), and is invariant without changing S command value, which is called constant speed control(G97 modal); other is the tangent speed of tool relative to the outer circle of workpiece defined by S command, which is called constant surface speed control (G96 modal), and the spindle speed is changed along with the absolute coordinates value of X absolute coordinates in programming path when cutting feed is executed in the constant surface speed.

Please refer to Section 2.2.3.

The system can execute 4 gears spindle speed. Count the analog voltage value corresponding to the specified speed according to setting value(corresponding to No.037~No.040) of max. spindle speed (analog voltage is 10V)of current gear, and then output to spindle servo or converter to ensure that the spindle actual speed and the requirement are the same.

After the system is switched on, the analog output voltage is 0V. The analog output voltage is reserved (except that the system is in cutting feed in the surface speed control mode and the absolute value of X absolute coordinates is changed) after S command is executed. The analog output voltage is 0V after S0 is executed. The analog output voltage is reserved when the system resets and emergently stops.

Parameters relative to the analog voltage control of spindle speed:

System parameter No.021: offset value of output voltage with max. spindle speed (the analog output voltage is 10V);

System parameter No.036: offset value of output voltage with spindle speed 0 (the analog output voltage is 10V);

System parameter No.037~No.040: max. spindle speed (the analog output voltage is 10V) with spindle 1~4 gears(corresponding to M41~M44).

2.2.3 Constant surface speed control G96, constant rotational speed control G97

Command format: G96 S__; (S0000~S9999, the leading zero can be omitted.)

Command function: The constant surface speed control is valid, the cutting surface speed is defined (m/min) and the constant rotational speed control is cancelled.

G96 is modal G code. If the current modal is G96, G96 cannot be input.

Command format: G97 S__; (S0000~S9999, the leading zero can be omitted.)

Command function: The constant surface speed control is cancelled, the constant rotational speed control is valid and the spindle speed is defined (r/min). G96 is modal G code. If the current modal is G97, G97 cannot be input.

Command format: G50 S__; (S0000~S9999, the leading zero can be omitted.)

Command function: define max. spindle speed limit (r/min) in the constant surface speed control and take the current position as the program reference point.

G96, G97 are the modal word in the same group but one of them is valid. G97 is the initial word and the system defaults G97 is valid when the system is switched on.

When the machine tool is turning it, the workpiece rotates based on the axes of spindle as the center line, the cutting point of tool cutting workpiece is a circle motion around the axes, and the instantaneous speed in the circle tangent direction is called **cutting surface** (for short **surface speed**). There are different surface speed for the different workpiece and tool with different material.

When the spindle speed controlled by the analog voltage is valid, the constant surface control is valid. The spindle speed is changed along with the absolute value of X absolute coordinates of programming path in the constant speed control. If the absolute value of X absolute coordinates adds, the spindle speed reduces, and vice versa, which make the cutting surface speed as S command value. The constant speed control to cut the workpiece makes sure all smooth finish on the surface of workpiece with diameter changing.

Surface speed=spindle speed \times |X| \times $\pi \div 1000$ (m/min)

Spindle speed: r/min

$|X|$: absolute value of X absolute coordinate value, mm

$\pi \approx 3.14$

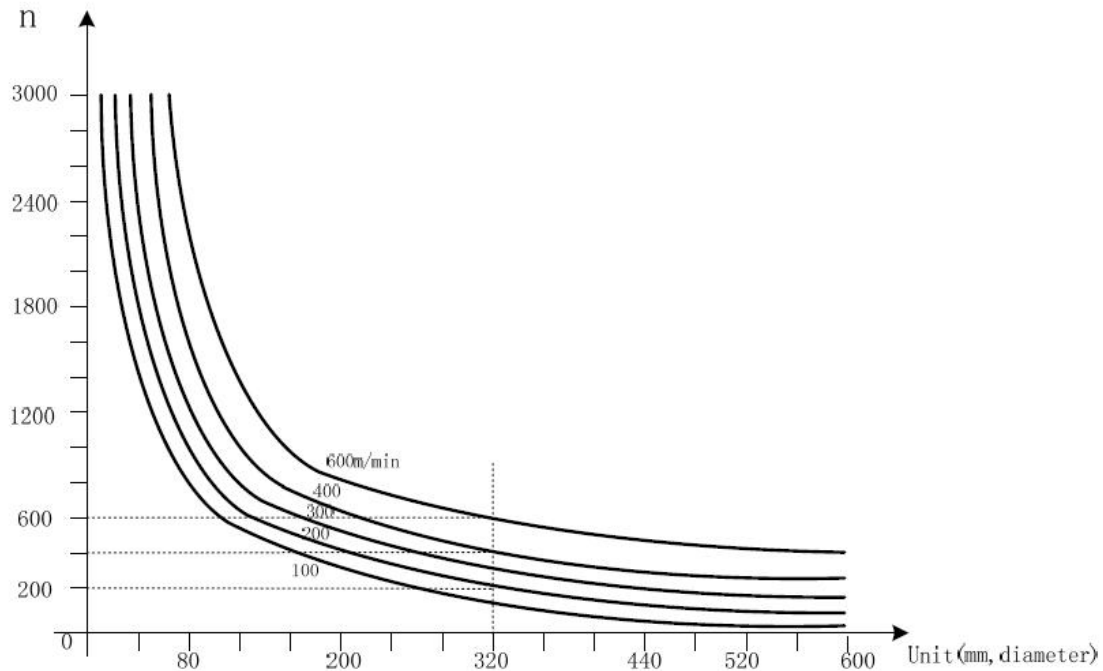


Fig. 2-4

In G96, the spindle speed is changed along with the absolute value of X absolute coordinates value of programming path in cutting feed (interpolation), but it is not changed in G00 because there is no actual cutting and is counted based on the surface speed of end point in the program block.

In G96 (constant surface speed control), Z coordinates axis of workpiece system must consist with the axes of spindle (rotary axis of workpiece), otherwise, there is different between the actual surface speed and the defined one.

G96 control is valid, G50 S_ can limit max. spindle speed (r/min). The spindle actual speed is the limit value of max. speed when the spindle speed counted by the surface speed and X coordinates value is more than the max. spindle speed set by G50 S_. After the system powers on, max. spindle speed limit value is not defined and its function is invalid. Max. spindle speed limit value defined by G50 S_ is reserved before it is defined again and its function is valid in G96. Max. spindle speed defined by G50 S_ is invalid in G97 but its limit value is reserved.

Note: When NO.043 (lowest spindle speed in constant surface speed control) is set to 0 and G50 S0 is executed, the spindle speed is limited to 0 r/min (the spindle does not rotate).

When the constant surface speed is controlled by the system parameter No.043, the spindle speed is lower limit, which is higher than one counted by the surface speed and X axis coordinates value.

Example:

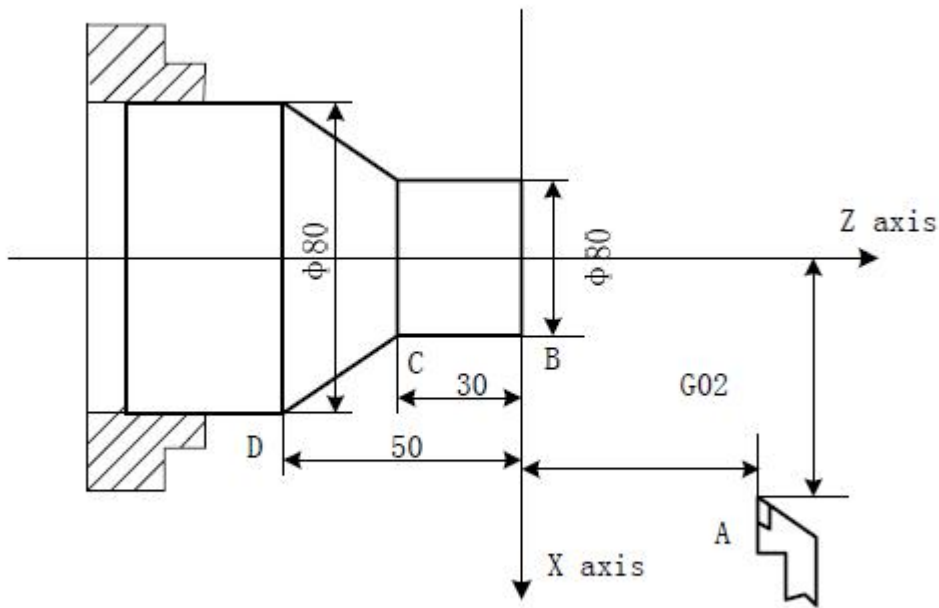


Fig. 2-5

O0001 ; (Program name)

N0010 M3 G96 S300; (Spindle rotates clockwise, the constant surface speed control is valid and the surface speed is 300 m/min)

N0020 G0 X100 Z100; (Rapid traverse to A point with spindle speed 955 r/min)

N0030 G0 X50 Z0; (Rapid traverse to B point with spindle speed 1910 r/min)

N0040 G1 W-30 F200; (Cut from B to C with spindle speed 1910 r/min)

N0050 X80 W-20 F150; (Cut from C to D with spindle speed 1910 r/min and surface speed 1194 r/min)

N0060 G0 X100 Z100; (Rapid retract to A point with spindle speed 955 r/min)

N0110 M30; (End of program, spindle stopping and cooling OFF)

N0120 %

Note 1: S value commanded in G96 is also reserved in G97. Its value is resumed when the system is in G96 again;

Example:

G96 S50; (Cutting surface speed 50m/min)

G97 S1000; (Spindle speed 1000 r/min)

G96 X3000; (Cutting surface speed 50m/min)

Note 2: The constant surface speed control is valid when the machine tool is locked (X, Z do not move when their motion command are executed);

Note 3: To gain the precise thread machining, it should not be adopted with the constant surface speed control but the constant rotational speed (G97) in the course of thread cutting;

Note 4: From G96 to G97, if none of S command (r/min) is commanded in the program block in G97, the last spindle speed in G96 is taken as S command in G97, namely, the spindle speed is not changed at this time;

Note 5: In G96, when the spindle speed counted by the cutting surface speed is more than max. speed of

current spindle gear (system parameter No.037~No.040), at this time, the spindle speed is limited to max. one of current spindle gear.

2.2.4 Spindle override

When the spindle speed analog voltage control is valid, the spindle actual speed can be tuned real time by the spindle override and is limited by max spindle speed of current gear after the spindle override is tuned, and it also limited by limited values of max. and min. spindle speed in constant surface speed control mode.

The system supplies 8 steps for spindle override (50% ~ 120% increment of 10%). The actual steps and tune of spindle override are defined by PLC ladder and introductions from machine manufacturer should be referred when using it. Refer to the following functions of ECN*000T standard PLC ladder.

The spindle actual speed specified by ECN*000T standard PLC ladder can be tuned real time by the spindle override tune key at 8 steps in 50% ~ 120% and it is not reserved when the spindle override is switched off. Refer to the operations of spindle override in II **OPERATION**.

2.2.5 Multiple spindle control function

ECN*000T can control up to two analog spindles. One S code is used to command one of them which are selected by PLC signal and which have the gear change function.

Because ECN*000T has only one spindle encoder interface, the 2nd spindle has no encoder feedback and the spindle speed is not displayed.

Being the speed command, S code is sent to the spindle selected by the spindle selection signal (SWS1, SWS2 <G27 # 0, G27 # 1>), and each spindle rotates with the specified speed. When the spindle has not received the spindle selection signal, it rotates with the previous speed, which makes the spindle rotates with different speed in different time. Each spindle has its own stop signal and enabling signal.

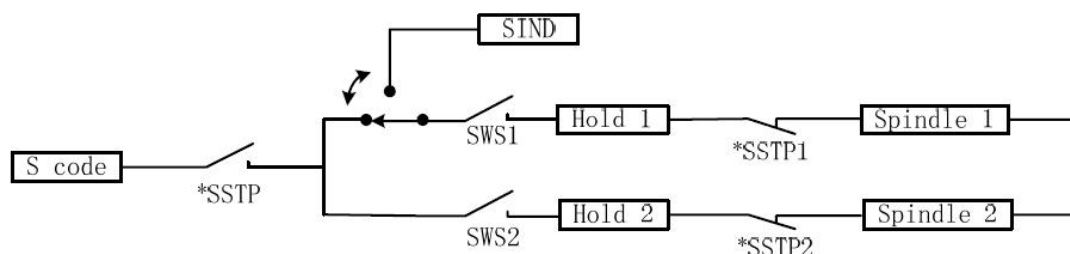
When №001 # 4 is set to 1, setting MSEN (№196 # 4) to 1 can start the multiple spindle control function.

The spindle control has several methods which are set by MSI (№196 # 7) as follows:

Multiple spindle control method A

When SWS1 signal selects the 1st spindle, SIND signal is used to determine that the spindle analog voltage is controlled by PLC or CNC, R011 to R121 signals are used to set the spindle analog voltage. These signals do not influence the 2nd spindle.

Multiple spindle control method A is shown below.

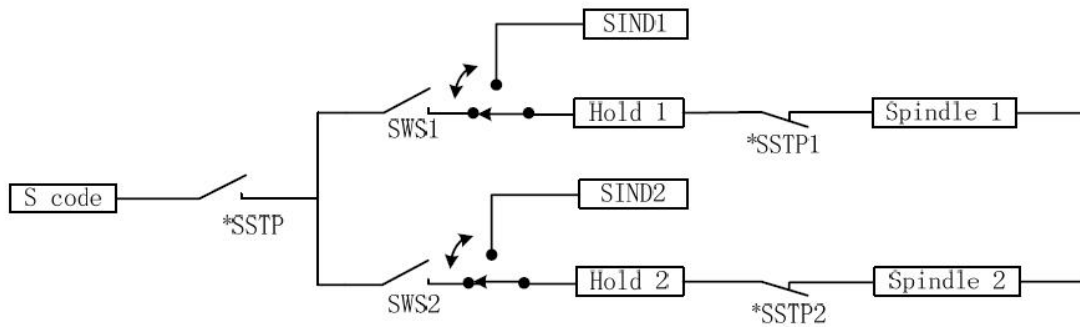


Multiple spindle control method B

Each spindle has separate SIND signal.

When the spindle selection signal, the 1st spindle or the 2nd spindle SIND signal is set to 1, SIND signals separately determine each spindle to be controlled by PLC or CNC.

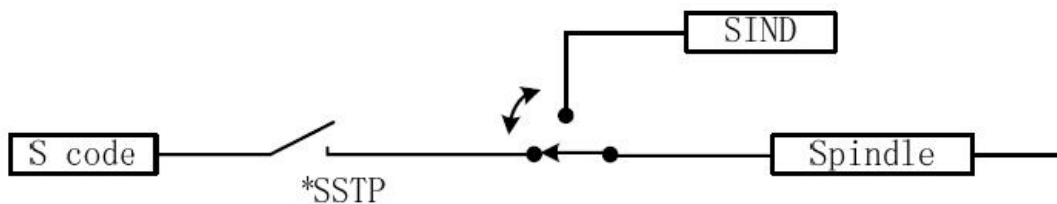
Multiple spindle control method B is shown below.



Target tool number (01-32, the leading zero cannot be omitted)

Multiple spindle control function being invalid

When the multiple spindle control is invalid, the control method is shown below.



2.2.6 Cs contour control function

Controlling a spindle speed is called as the spindle rotation control (the spindle rotates by the speed command), and controlling the spindle position is called as the spindle contour control (the spindle rotates by the movement command). The function of the spindle contour control is that of Cs contour control. Being the servo feed axis, the spindle rotates and orients by the position movement command, and executes the interpolation with other feed axes to machine the contour curve.

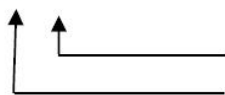
2.3 Tool Function

2.3.1 Tool control

T functions of ECN*000T: automatic tool change and executing tool offset. Control logic of automatic tool change is executed by PLC and tool offset is executed by NC.

Command format:

T□□○○



Tool offset number (00-32, the leading zero cannot be omitted)

Target tool number (01-32, the leading zero cannot be omitted)

Command function: The automatic tool post rotates to the target tool number and the tool offset of tool offset number commanded is executed. The tool offset number can be the same as the tool number, and also cannot be the same as it, namely, one tool can corresponds to many tool offset numbers. After executing tool offset and then T□□00, the system reversely offset the current tool offset and the system its operation mode from the executed tool length compensation into the non-compensation, which course is called the canceling tool offset, called canceling tool compensation. When the system is switched on, the tool offset number and the tool offset number displayed by T command is the state before the system is switched off.

Only one T command is in a block, otherwise the system alarms.

Toolsetting is executed to gain the position offset data before machining (called tool offset), and the system automatically executes the tool offset after executing T command when programs are running. Only edit programs for each tool according to part drawing instead of relative position of each tool in the machine coordinate system. If there is error caused by the wearing of tool, directly modify the tool offset according to the dimension offset.

Tool offset number (00-32, the leading zero cannot be omitted)

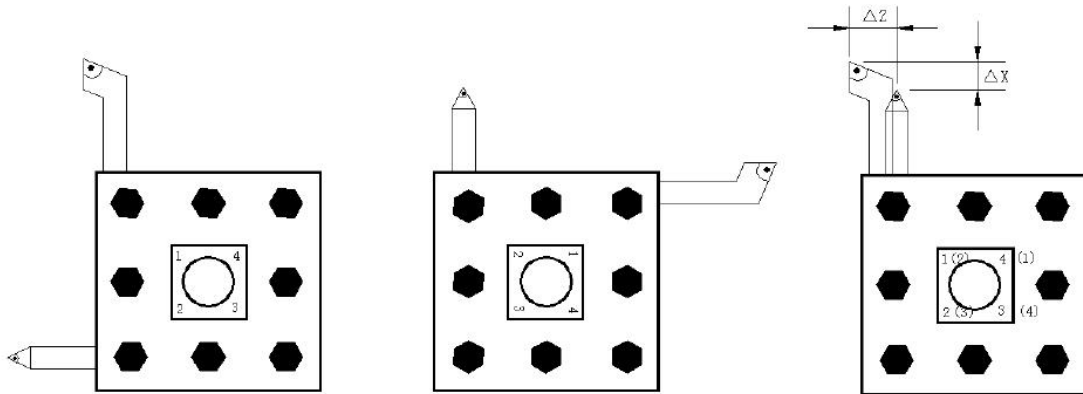


Fig.2-6 tool offset

The tool offset is used for the programming. The offset corresponding to the tool offset number in T command is added or subtracted on the end point of each block. Tool offset in X direction in diameter or radius is set by No.004 Bit4. For tool offset in diameter or radius in X direction, the external diameter is changed along with diameter or radius when the tool length compensation is changed.

Example: When the state parameter No.004 Bit4 is set to 0 and X tool length compensation value is 10mm, the external diameter of workpiece is 10mm; when No. 004 is set to 1 and X tool length compensation value is 10mm, the external diameter of workpiec is 20mm. Fig.2-5 is the course of creating, executing and canceling tool offset in traverse mode.

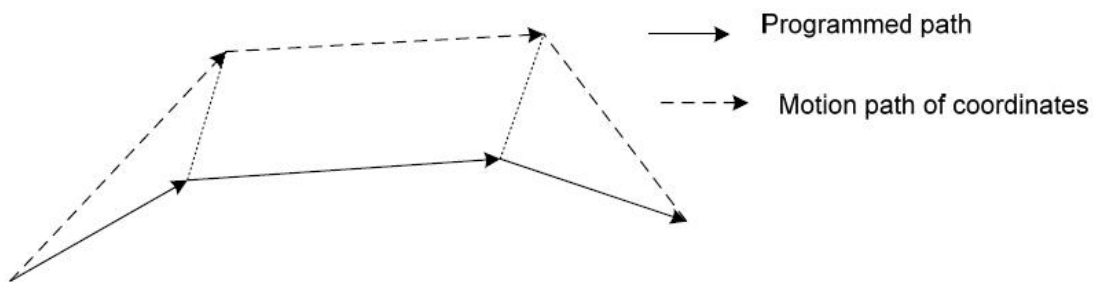


Fig. 2-7 Creation, execution and cancellation of tool length compensation

G01 X100 Z100 T0101; (Block 1, start to execute the tool offset)

G01 W150; (Block 2, tool offset Block 2, tool offset)

G01 U150 W100 T0100; (Block 3, canceling tool offset)

There are two methods defined by No.003 Bit4 to execute the tool length compensation:

Bit4=0: The tool length compensation is executed by the tool traversing;

Bit4=1: The tool length compensation is executed by modifying the coordinates;

Example:

Table 2-4

Tool offset number	X	Z
00	0.000	0.000
01	0.000	0.000
02	12.000	-23.000
03	24.560	13.452

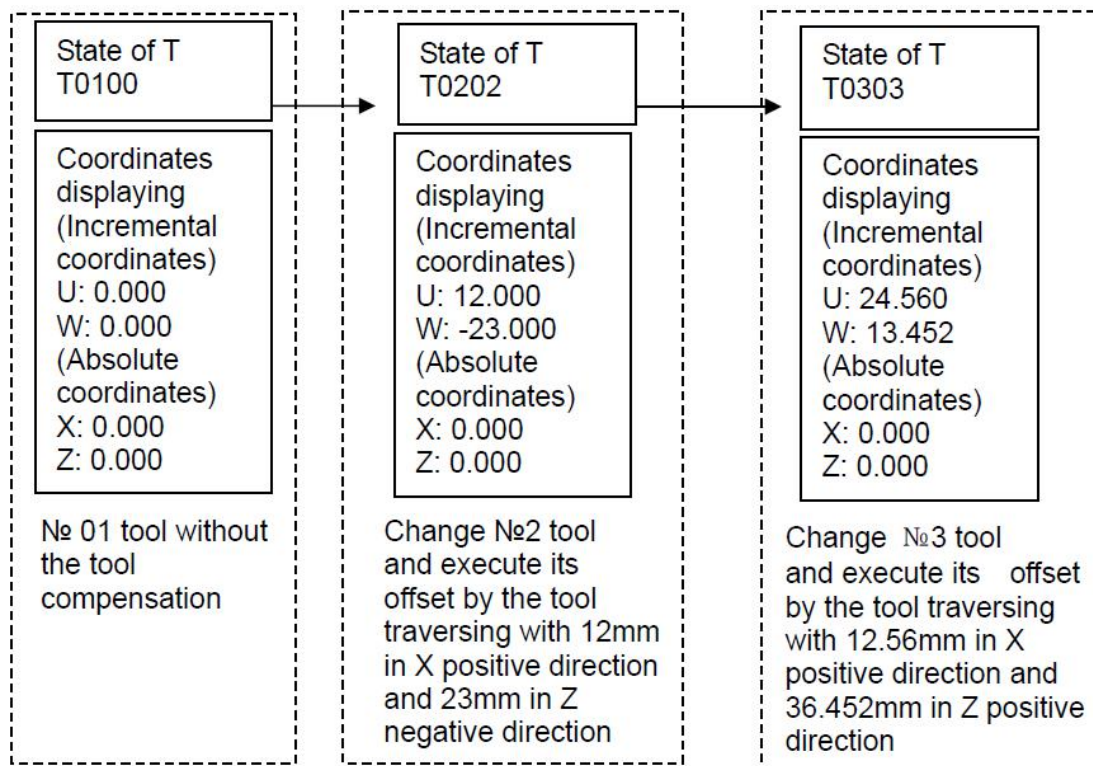


Fig. 2-7 Tool traversing mode

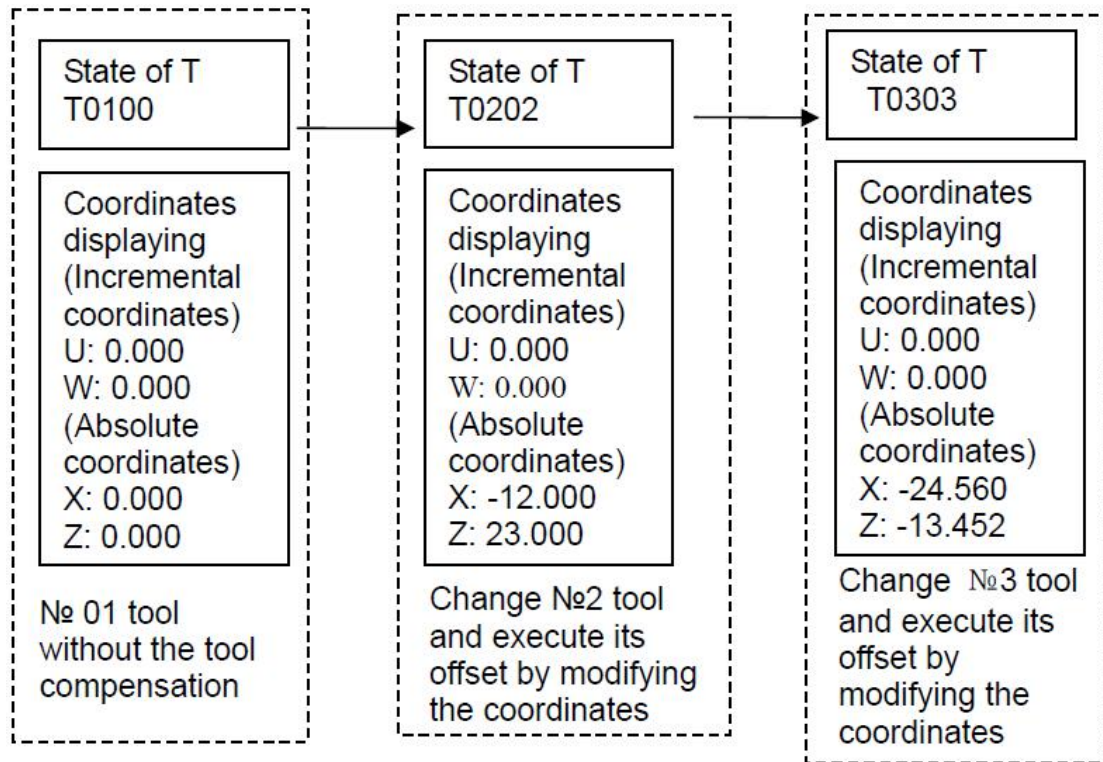


Fig. 2-8 Modifying the coordinates mode

In Edit and Auto mode, a sole T word in executing tool offset (it is not with the motion command in the same block) is relative to No.004 BIT3 setting (as Fig.2-6 and Fig.2-7). When No.003 Bit4=1

and a sole T command is executed, the tool offset number is displayed in poor, which is cleared out(tool offset number is still displayed in poor when tool offset is not executed for one axis, the previous bit of tool offset number is for X axis tool compensation and the next one is for Z axis tool compensation) after executing tool offset.

executing Z axis as follows:

Executing a sole T0101 and tool offsets of two axes are not executed

Executing W0 after T0101, X tool offset is not executed but that of Z is done

When T command and the motion command are in the same block and execute tool offset by modifying coordinates, the motion command and T command are executed at the same time, the system executes by adding the current tool offset to coordinates of motion command and whether the

traverse speed is employed the cutting feedrate or the rapid traverse speed defined by the motion command.

When T command and the motion command are in the same block and execute tool offset by traversing tool, the motion command or T command is executed separately. Firstly tool change is executed and then the motion command is executed. The tool offset is executed at current rapid traverse speed.

The tool offset is cancelled after one of the following operations is executed:

2. Execute G28 or manual machine reference point return (only the tool offset of coordinate axis which is executed machine reference point return is cancelled, and another one which is not

executed machine reference point return is not cancelled);

When No.084 is not 1 (2~32) and target tool number is not equal to current display tool number, the control sequence and logic of tool post is defined by PLC ladder diagram after commanding T command, please see User Manual of machine tool manufacturer. ECN*000T standard PLC ladder diagram defines as follows: clockwise rotation for selecting tool, counterclockwise rotation for tool post clamping, directly inputting tool selection signal for tool change. Please refer to III **INSTALLATION & CONNECTION.**

When the system is employed with line-up tool post, No.084 should be set to 1 and different tool number is executed by different tool offset as T0101, T0102, T0103.

2.3.2 Tool life management

1. Starting tool life management function

002								Tool life management
-----	--	--	--	--	--	--	--	----------------------

The state parameter No.002 Bit0 (TLIF) is the market whether the tool life management function is valid or not, the corresponding tool life management window is not displayed when it is invalid.

002 Tool life management

Bit 0 Tool life management function is not valid.

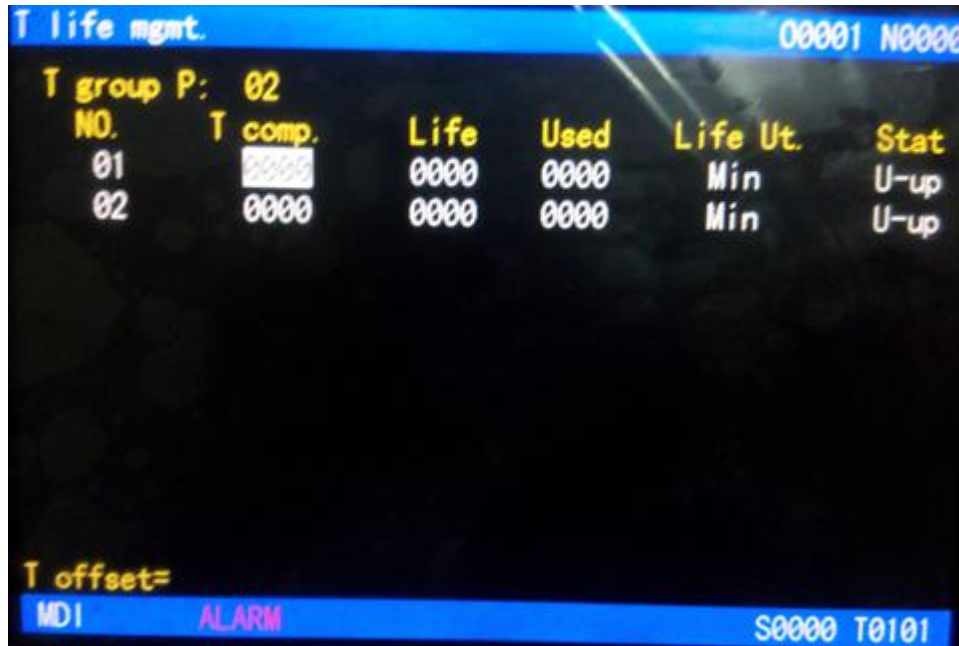
1 Tool life management function is valid.

2. Tool life management display window



Press **OFT** repetitively into the tool life management display window.

1) “Tool compensation->tool life” window



The current window displays the life management data of current used tool life management and defined tool groups. The window is shown below:

Current tool state: display the current used tool life management data;

Tool number: current used tool and tool compensation number;

Group: the group where the tool is;



Life: tool life data, the specified value can be time or number of times according to the different count N value; press to record the time or clear the times.

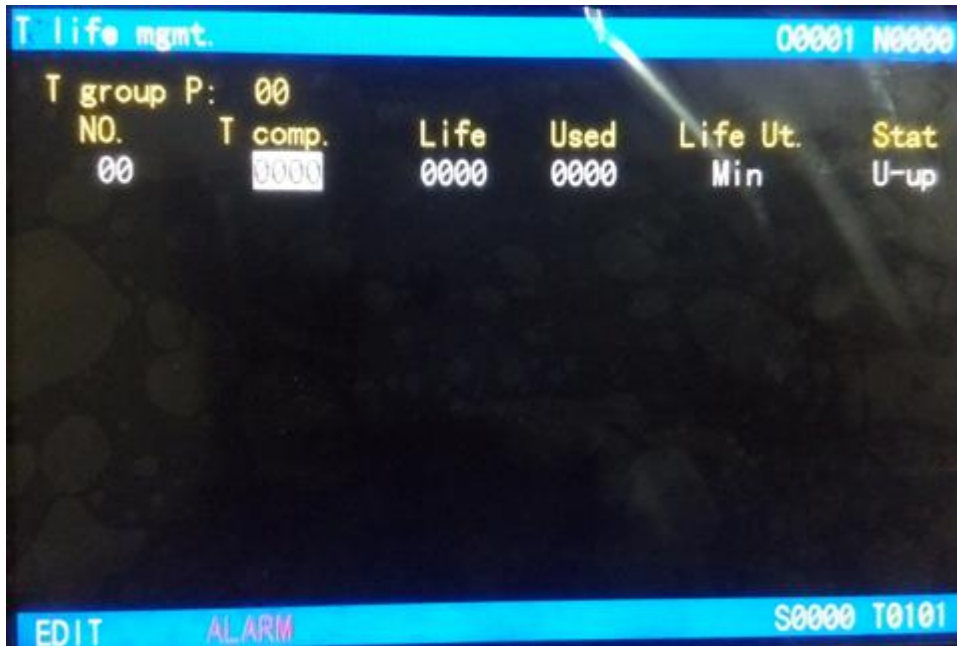
Used: used tool life data.

Life unit: count unit of tool life, N1 is the used time (unit: minute), N0 is the used times (unit:times) .

State: display the tool state (0—not be used, 1—being used , 2—used, 3—_p04_ÜÏ_<skip).

2) Creating and displaying tool group number

A. In tool group state display window, press  , group number,  , and the system display the tool group life data, and when the group does not exist, it is defined to the new group number(the parameter switch is ON in MDI mode).



Note: After the new group is defined, ECN*000T will automatically define the first tool, for example, the new defined group number is 01, the display is as follows:

B. Press to move the cursor to Defined Group Number.

C. Press or to select the group number in Defined Group Number, and to gradually display content of each group number.

3. Definition of tool life data

There are two ways to set the tool life data:

- 1) compile NC programs and run the program setting;
- 2) input directly in the tool life management window.

1) Compile NC programs and run program setting

Command format: G10 L3

Command function: set to be the tool life manage data input mode

Command format: G11

Command function: cancel the tool life management data input mode

Program	Meaning	Remark
O0020 (O0020)		T_: tool and tool offset number;

G10 L3;	Set to be the tool life management data input mod	
P01;	Tool group number, too group number setting range(1~32)	N_: tool life count mode, N0 is the used time(minute) to count tool life and N1 is the used number of times to count the tool life(unit: min)
T0101 L500 N0;	Tool number, life, mode(number of times) setting	
T0201 L600 N1;	Tool number, life and mode(min) setting	
P02;	Another tool group number	
T0303 L200 N0;		L_: tool life data, the specified value can be time or number of times according to the different count N value
T0304 L300 N0;		
G11;	Cancel the tool life management data input mode	
M30;		

L_: tool life data, the specified value

can be time or number of times

according to the different count

N value

Note 1: The tool group numbers specified by P may not be the continuous, but it is better to gradually increase the sequence number to easily search the tool group number.

Note 2: The tool life is 0 when the life data L_ is omitted, and the tool mode is 0 (minute) when the specified mode N_ is omitted, at the time, the system only counts and does not alarm for the output.

Note 3: Other commands between G10 L3 and G11 are ignored.

Note 4: Running the tool life preset program (such as O0020) completely clear all previous life data and preset the life data according to the program requirements.

Note 5: Prevent the life data from being modified manually when the part programs are running until the run

state stops (except for running the tool life preset programs).



Note 6: All tool life data is stored when power-down.

2) Input tool life management data in the tool life management window

Input directly the tool life management data in tool compensation- >tool life window in MDI mode, parameter switch ON and 3-level operation limit.

A. alter data:

In tool compensation -> tool life window, press  to move the tool to the life position, input

directly data, such as 500, press  to confirm the input. Press  to complete data input (tool offset, life, used, status) .

B. Inserting data:

Insert any serial number in the current page, press **N#** ——> [01~08] ——> **IN** ,insert a new line, and the initial value definition is as follows:

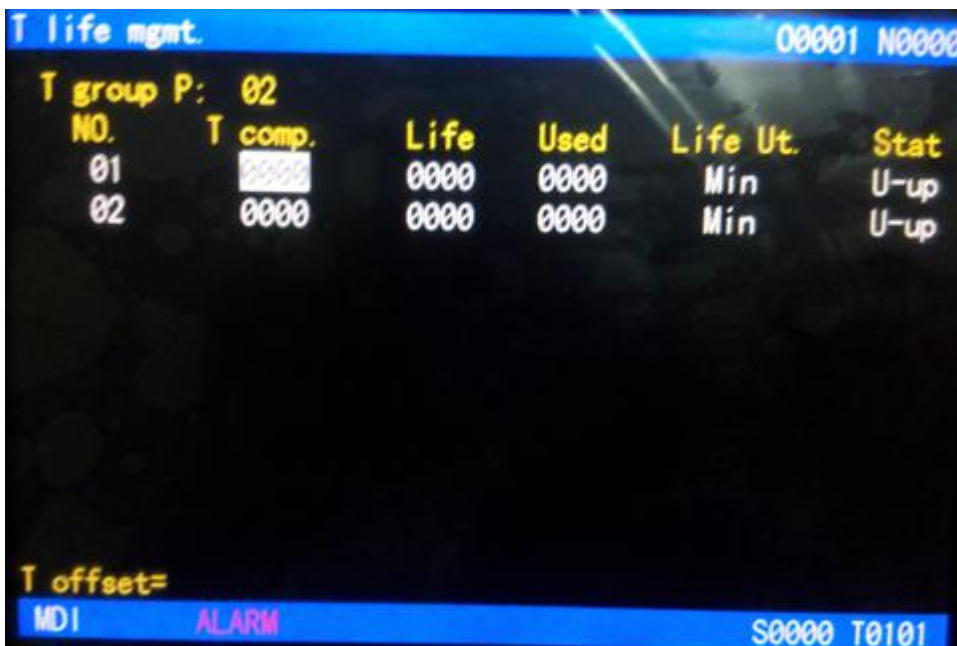
Serial number	Tool offset	Life	Used	Mode	State
N	0000	0	0	0	Over

Serial number Tool offset Life Used Mode State

a)Insert the front and the previous serial number moves backward

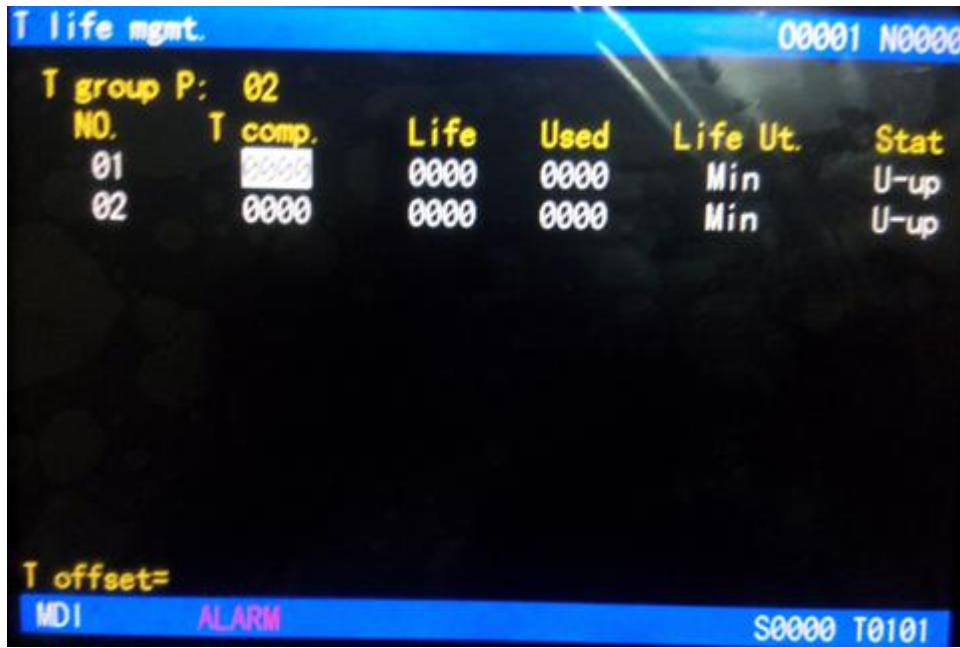


Before insert

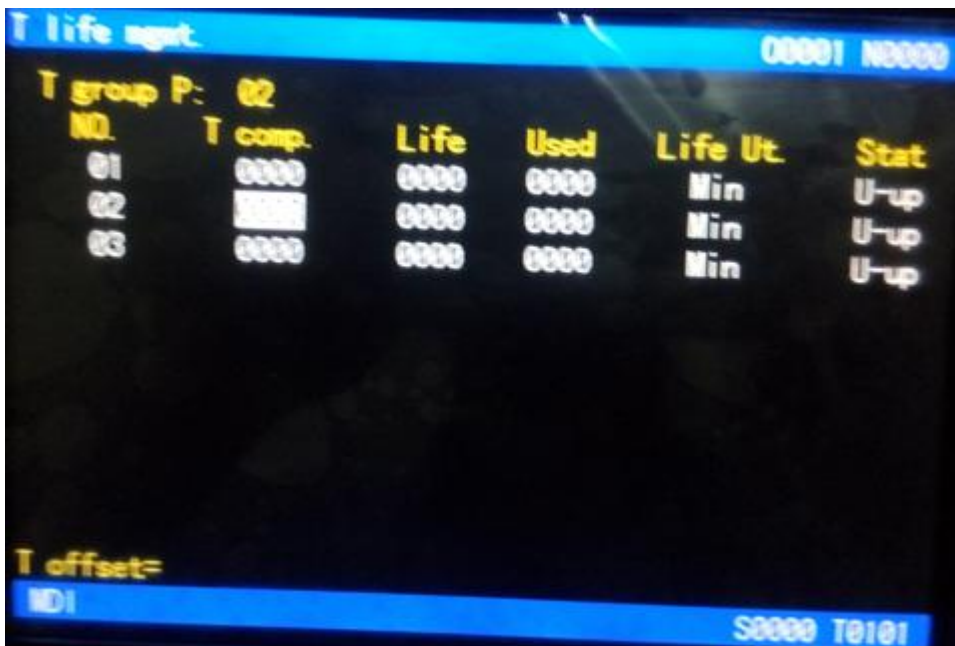


After insert

b) Insert the middle and the previous sequence number moves backward.



Before insert

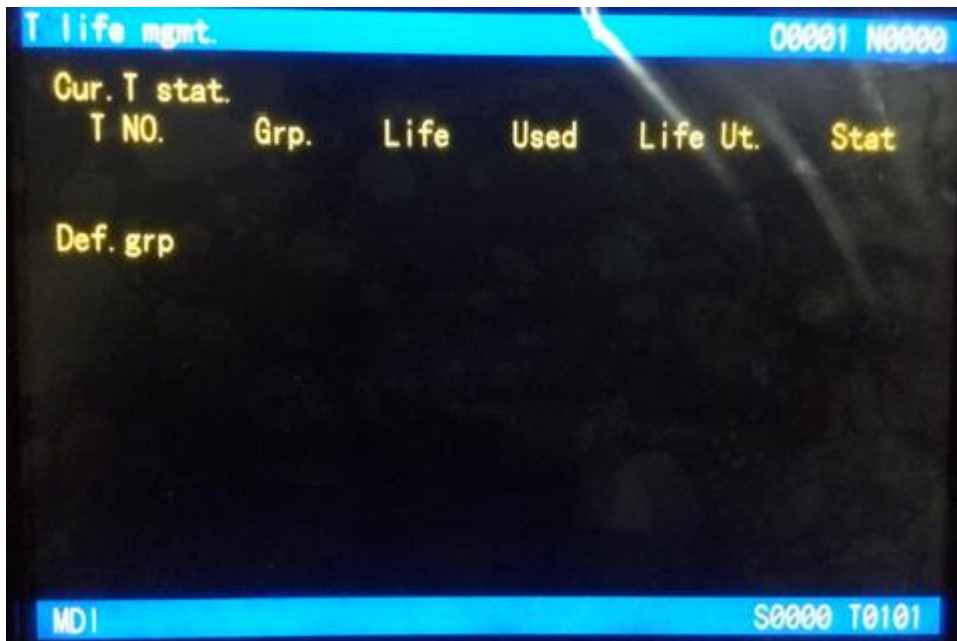


After insert

C. Delete data:

a) Delete data in all groups.

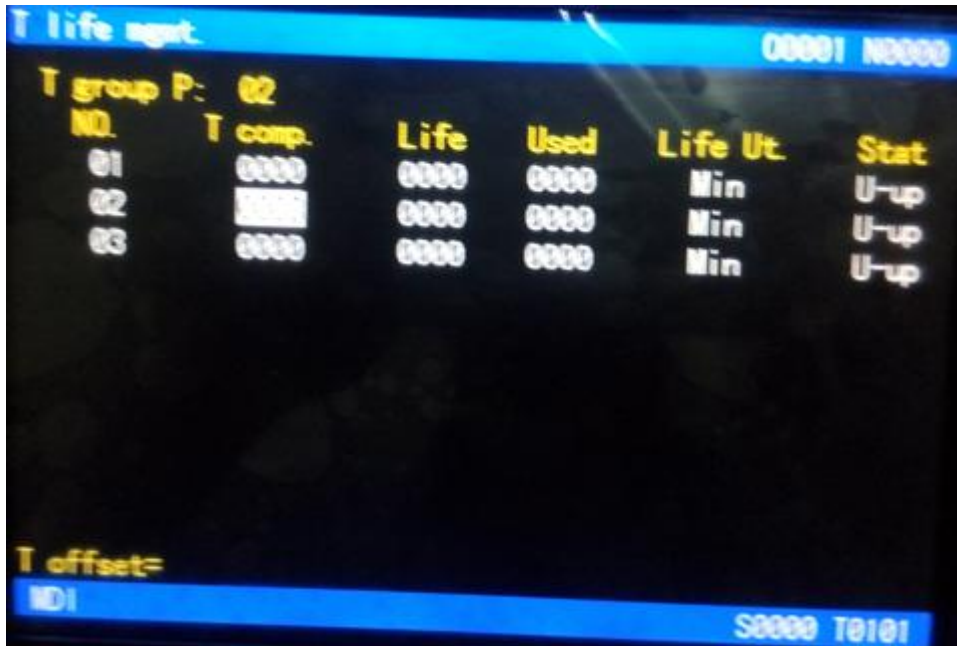
In tool compensation-> tool life window, press **CAN** + **G*** to delete all defined data.

Before operation**After operation**

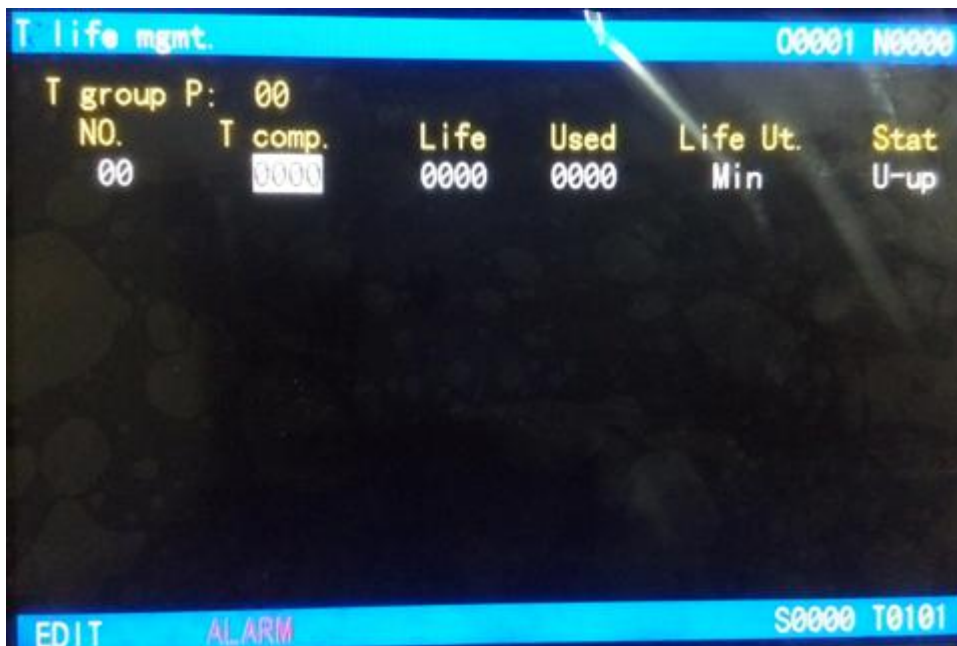
b) Delete data in any group

Press **P_a** —> input serial number [02] —> **DEL** ;

Before operation



After deletion



c) Delete any serial number in the current page(the parameter switch is ON)

Press **N**# ——> input serial number [04] ——> **DEL**

Before deletion

T life mgmt. 00001 N0000

T group P: 02

NO.	T comp.	Life	Used	Life Ut.	Stat
01	0000	0000	0000	Min	U-up
02	0000	0000	0000	Min	U-up
03	0000	0000	0000	Min	U-up

T offset=

MDI S0000 T0101

After deletion

T life mgmt. 00001 N0000

T group P: 02

NO.	T comp.	Life	Used	Life Ut.	Stat
01	0000	0000	0000	Min	U-up
02	0000	0000	0000	Min	U-up

T offset=

MDI ALARM S0000 T0101

4. Tool life function use

Command format:

Txx99: end of current used tool group, start the tool and execute the life management in XX group.

Txx88: cancel the tool offset in XX group

Two examples are as follows:

Application example:

O0000 (O0000)

...

T0199; End of previous tool group, and start the tool in 01 group

...

T0188; Cancel tool offset in 01 group(current used tool offset)

...

T0508; Use No.05 tool and 08 tool offset without life management

...

T0500; Cancel No.05 tool offset

...

T0299; End of No.05 tool and start the tools in 02 group

...

T0199;

End of tools in No.02 group, start tools in No.02 group, and start the next tool when there are many tools in No.01 group.

...

5. Tool life count:

When the count result is for the used life value being more than or equal to the setting value of the life data, the next tool group number selects the standby tool in the commanded selection tool group and the new selected tool will be counted, the count will be continuously executed and the system alarms to output to PLC when all tool life in the tool groups reaches and there is no standby tools. Executing the counting in MDI mode is determined by No.002 Bit3 (MDITL).

002					M T L			
-----	--	--	--	--	-------	--	--	--

002 MDITL

MDITL=0 Tool life management is invalid in MDI mode.

MDITL=1 Tool life management is valid in MDI mode.

The tool life count has two methods including time and number of times.

A. Time count

Use the time (minute) to count the tool life in cutting feed mode(such as G01, G02, G03, G32, G33, G34 and so on), and do not count it in G00, G04, single block stop, machine lock, auxiliary lock, dry run and so on.

B. Number of times count

There are two methods about the number of times count which is determined by No.002 Bit2

002					L I F C			
-----	--	--	--	--	---------	--	--	--

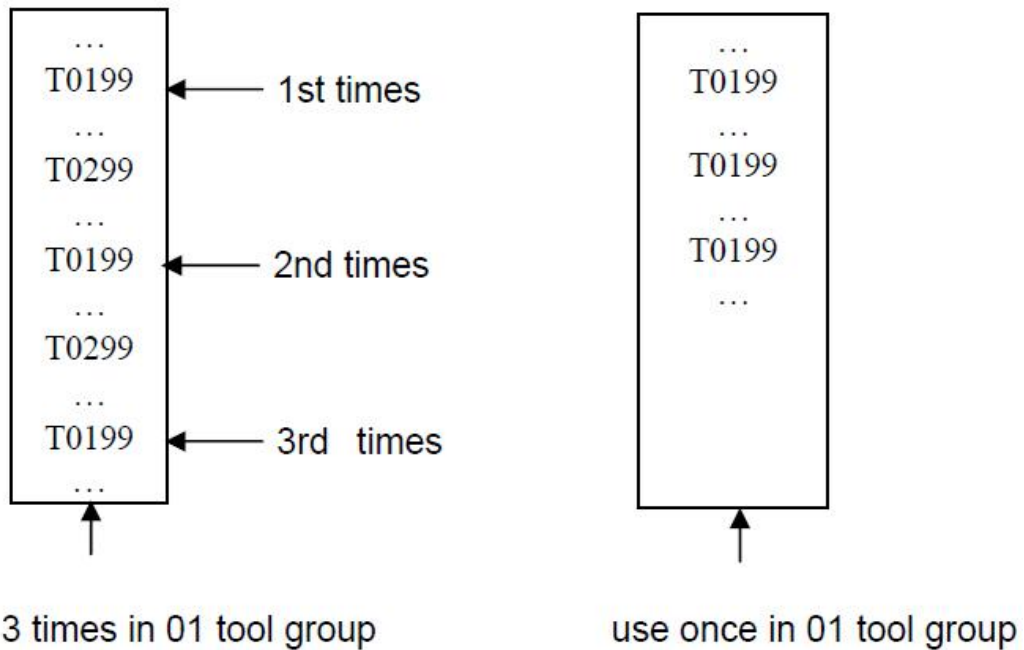
(LIFC).

002 LIFC

LIFC=0 Tool life management count mode 1

Execute the tool select (Txx99) to change the tool number, execute the count in the cutting feed mode (except for machine lock, auxiliary lock, dry run state). The count is not executed when the tool number is changed and the system is not in the cutting feed mode.

Application example:



LIFC=1 Tool life management count 2

The tool group adds when the machining program runs to M30(M99), when the system resets halfway, the number of time does not add, the count is not executed in machine lock, auxiliary lock and dry run mode.

6. G, F signals relevant to tool life function

a) Tool change signal TLCH (F064#0)

[Type] output signal

[Function] inform PLC that the last tool life ends of in this group

[Output condition] the signal is set to 1, the last tool life in one group ends, all other tool life ends.

b) New tool selection signal TLNW (F064#1)

[Type] output signal

[Function] inform a new tool in some tool group has been selected

[Output condition] the signal is set to 1 and a new tool in some tool group has been selected.

c) Tool change reset signal TLRST (G048#7)

[Type] input signal

[Function] clear all execution data

[Operation] when the signal is set to 1, the control unit clears the used tool life data of in all groups; the tool state is reset to the unused.

Note: when the automatic operation signal OP is "0", the tool change reset signal TLRST is valid.

d) Tool skip signal TLSKP (G048#5)

[Type] input signal

[Function] use the following two methods to change the tools which have not reached the life:

- 1) When LIFJ (NO.2#4) is set to 1, the selection signals of the tool group number (G47#0~#4) specify the tool group number, then, the tool skip signal TLSKP becomes "1", the next T command skips the tool which is being used of the

current group, and the tool which life has not reach in the group specified by G47#0~#4 is used.

2) When LIFJ (NO.2#4) is set to 0, the group numbers specified by the tool group number signal (G47#0~#4) are invalid, the tool skip signal TLSKP becomes "1", the machine skips to the next tool in the current used tool group.

[Operation] when the signal is set to 1, the operation described in [Function] is executed.

Note: When cycle start light signal(STL) and feed pause light signal(SPL) must be "0", inputting TLSKP signal

is valid.

e) Tool group number selection signal TL01~TL16 (G47#0~#4)

[Type] input signal

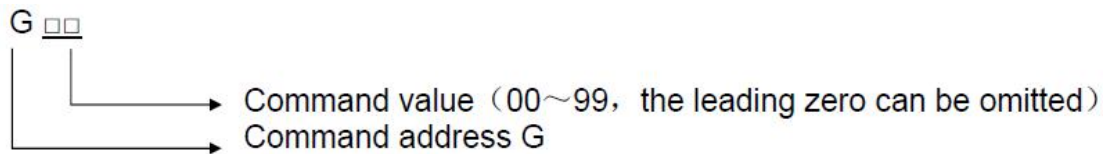
[Function] when TLSKP is input, using the tool group selection signal TL01~TL16 specifies the tool group number with the binary system. The specified tool group number = G47#4~#0 + 1.

[Operation] specify the selected tool group

CHAPTER 3 G COMMANDS

3.1 Commands

G command consists of command address G and its following 1~2 bits command value, used for defining the motion mode of tool relative to the workpiece, defining the coordinates and so on. Refer to G commands as Fig. 3-1.



G words are divided into 9 groups (00, 01, 02, 03, 06, 07, 12, 14, 16, 21). Except that commands in the group 01 and 00 are not in the same block, G words in the different groups can be input to the same block and the last one is valid when two or more G words in the same group are input. The words in the different groups without the same parameter (word) can be in the same block and their functions are valid without sequence at the same time. The system alarms when G words do not belong to Table 3-1 or they are optional functions without being supplied.

Table 3-1 G command list

Word	Group	Function	Remark
G00		Rapid traverse movement	Initial modal G command
G01	01	Linear interpolation	Modal G commands
G02		Circular interpolation(CW)	
G03		Circular interpolation(CCW)	
G05		Three-point circular interpolation	
G6.2		Ellipse interpolation (CW)	
G6.3		Ellipse interpolation (CCW)	
G7.2		Parabola interpolation (CW)	
G7.3		Parabola interpolation (CCW)	
G32		Thread cutting	
G32.1		Rigid thread cutting	
G33		Z tapping cycle	
G34		Variable pitch thread cutting	
G90		Axial cutting cycle	
G92		Thread cutting cycle	
G84		End rigid tapping	
G88		Side rigid tapping	
G94		Radial cutting cycle	
G04		00	
G7.1	Cylinder interpolation		
G10	Data input		
G11	Data input cancel		
G28	Machine reference point automatic return		
G30	Machine 2nd, 3rd , 4th reference point		
G31	Skip interpolation		
G36	Automatic tool compensation X		
G37	Automatic tool compensation Z		

G50		Setting workpiece coordinate system	
G52		Setting local coordinate system	
G65		Macro command	
G70		Finishing cycle	
G71		Axial roughing cycle	
G72		Radial roughing cycle	
G73		Closed c	
G74		Axial grooving cycle	
G75		Radial grooving cycle	
G76		Multiple thread cutting cycle	
G20	06	Inch select	Modal G command
G21		Metric select	Initial mode G command
G96	02	Constant surface speed ON	Modal G command
G97		Constant surface speed OFF	Initial mode G command
G98	03	Feed per minute	Initial mode G command
G99		Feed per rev	Modal G command
G40		Cancel cutter radius compensation	Initial mode G command
G41	07	Tool nose radius compensation left contour (option)	Modal G command
G42		Tool nose radius compensation right contour(option)	
G66	12		Modal G command
G67			
G54	14	Workpiece coordinate system 1	Modal G command
G55		Workpiece coordinate system 2	
G56		Workpiece coordinate system 3	
G57		Workpiece coordinate system 4	
G58		Workpiece coordinate system 5	
G59		Workpiece coordinate system 6	
G17	16	XY plane	Modal G command
G18		ZX plane	Initial mode G command
G19		YZ plane	Modal G command
G12.1	21	Polar coordinate interpolation	Non-modal G command
G13.1		Polar coordinate interpolation cancel	

3.1.1 Modal, non-modal and initial mode

G commands are divided into group 00, 01, 02, 03, 06, 07, 16, 21.

After G commands are executed, their defined functions and states are valid until they are changed by others in the same group, which commands are called modal G commands. After the modal G words are executed, and before their defined functions and states are changed, the G command cannot be input again when they are executed by the following block.

The defined function and state are valid one time after G command is executed, and the G word must be input again when it is executed every time, which command is called non-modal G command.

After the system is switched on, the valid modal G commands which are not executed their functions or states are called initial mode G command. Take it as the initial mode G command to be executed when it is not be input after the system is switched on.

3.1.2 Omitting words

To simplify the programming, their command values are reserved after executing words in Table 3-2. If the words are contained in the previous blocks, they cannot be input when the words are used with the same values and definitions in the following blocks.

Command address	Function	Initial value when power-on
U	Cutting depth in G71	No.51 parameter value
U	Move distance of X tool retraction in G73	No.53 parameter value
W	Cutting depth in G72	No.51 parameter value
W	Move distance of X tool retraction in G73	No.54 parameter value
R	Move distance of tool retraction in G71, G72 cycle	No.52 parameter value
R	Cycle times of stock removal in turning in G73	No.55 parameter value
R	Move distance of tool retraction after cutting in G74, G75	No.56 parameter value
R	Allowance of finishing in G76	No.60 parameter value
R	Taper in G90, G92, G94, G96	0
(G98) F	Feedrate per minute(G98)	No.30 parameter value
(G99) F	Feedrate per rev (G99)	0
F	Metric pitch(G32, G92, G76)	0
I	Inch pitch(G32, G92)	0
S	Spindle speed specified(G97)	0
S	Spindle surface speed specified(G96)	0
S	Spindle speed switching value output	0
P	Finishing times of thread cutting in G76; Tool retraction width of thread cutting in G76 Angle of tool nose of thread cutting in G76;	No.57 parameter value No.19 parameter value No.58 parameter value
Q	Min. cutting value in G76	No.59 parameter value

Table 3-2

Note 1: For the command addresses with functions (such as F, used for feedrate per minute, feedrate per revolution and metric pitch and so on), they can be omitted not to input when executing the same function to definite words after the words are executed. For example, after executing G98 F_ without executing the thread command, the pitch must be input with F word when machining metric thread.

Note 2: They can be omitted not to input when the address characters X(U) , Z(W) are the coordinates of end

point of block and the system defaults the current absolute coordinates in X or Z direction to the coordinate value of end point of block.

Note 3: The corresponding words must be input when the command addresses which are not in Table 3-2 are used.

Example 1:

O0001;

G0 X100 Z100; (rapid traverse to X100 Z100; the modal G0 is valid)

X20 Z30; (rapid traverse to X20 Z30; the modal G0 is not input)

G1 X50 Z50 F300; (linear interpolation to X50 Z50, feedrate 300mm/min; the modal G1 is valid)

X100; (linear interpolation to X100 Z50, feedrate 300mm/min; Z coordinate is not input and is the current coordinates Z50; F300 is kept, G1 is modal and is not input)

G0 X0 Z0; (rapid traverse to X0 Z0 and the modal G0 is valid)

M30;

Example 2:

O0002;

G0 X50 Z5; (rapid traverse to X50 Z5)

G04 X4; (dwell 4 seconds)

G04 X5; (dwell 5 seconds again, G04 is non-modal and is needed to input again)

M30;

Example 3 (the first run after power-on) :

O0003;

G98 F500 G01 X100 Z100; (Feedrate per minute 500mm/min in G98)

G92 X50 W-20 F2 ; (F value is a pitch and must be input in thread cutting)

G99 G01 U10 F0.01 (Feedrate per revolution in G99 must be input again)

G00 X80 Z50

M30;

3.1.3 Related definitions

In the user manual, the definitions of Word are as follows except for the especial explanations:

Starting point: position before the current block runs;

End point: position after the current block ends;

X: X absolute coordinates of end point;

U: different value of absolute coordinates between starting point and end point;

Z: Z absolute coordinates of end point;

W: different value of absolute coordinates between starting point and end point;

F: cutting feedrate.

3.2 Rapid Traverse Movement G00

Command format: G00 X(U) Z(W) ;

Command function: X, Z rapidly traverses at the respective traverse speed to the end points from their starting point. G00 is initial command as Fig.3-1.

X, Z traverses at the respective traverse speed, the short axis arrives the end point and the length axis continuously moves to the end point and the compound path may be not linear.

Command specification: G00 is initial mode;

X, U, Z, W range: $\pm 99999999 \times$ least input increment;

Can omit one or all command addresses X(U), Z(W). The coordinate values of starting point and end point are the same when omitting one command address; the end point and the starting point are in the same position when all are omitted. X, Z are valid, and U, W are invalid when X,U, Z and W are in the same one block. X, U, Z, W rang is referred to Table 1-2 of Section 1.4.1, unit: mm//inch.

Command path:

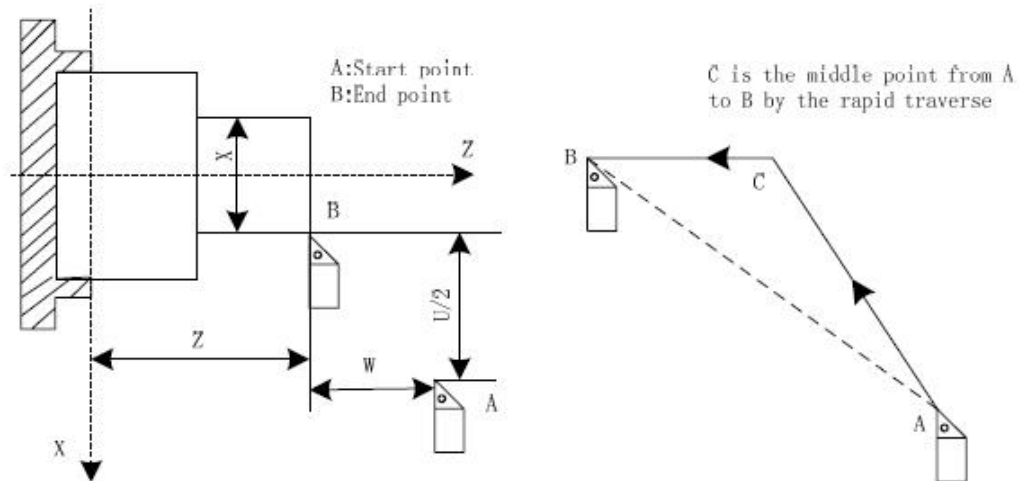


Fig. 3-1

The respective rapid traverse speed of X, Z is defined by the system parameter No.022, No.023, and their traverse speed can be changed by rapid override key on the machine control panel.

Example: The tool rapidly traverses to B from A as Fig. 3-2.

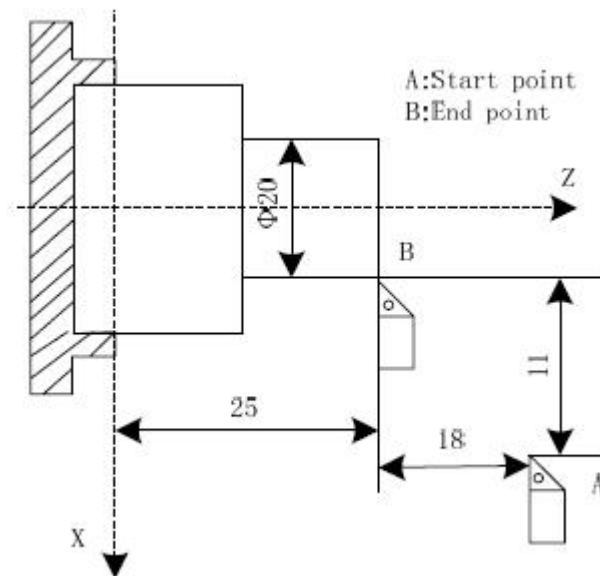


Fig. 3-2

G0 X20 Z25; (absolute programming)

G0 U-22 W-18; (incremental programming)

G0 X20 W-18; (compound programming)

G0 U-22 Z25; (compound programming)

3.3 Linear Interpolation G01

Command format: G01 X(U)_Z(W)_F_;

Command function: The movement path is a straight line from starting point to end point as Fig.3-3.

Command specification: G01 is modal.

Can omit one or all command addresses X (U), Z (W). The coordinate values of starting point and end point are the same when omitting one command address; the end point and the starting point are in the same position when all are omitted. X, U, Z, W range is referred to Table 1-2 of Section 1.4.1, unit: mm//inch.

F command value is the vector compound speed of X and Z instantaneous speed and the actual cutting feedrate is the product between the feedrate override and F command value.

After F command value is executed, it has been reserved unless the new one is executed. Do not repeat it when the following G commands adopt functions of F word. Its range is referred to Table 1-2.

Note: In G98, F max. value cannot exceed the value set by the data parameter No.027(upper speed of cutting feed), otherwise, the system alarms.

Command path:

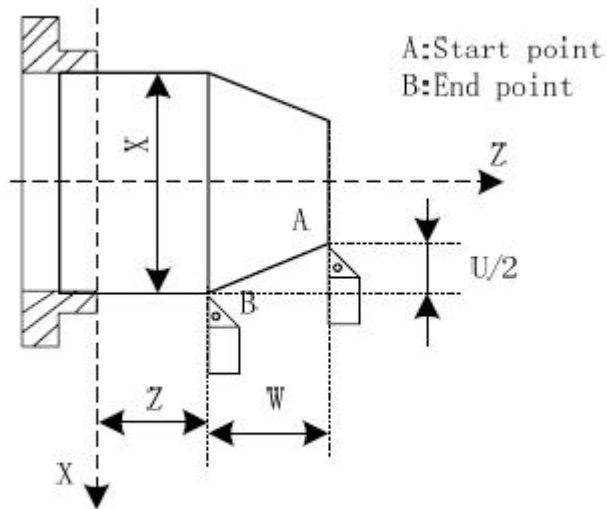


Fig. 3-3

Example: Cutting path from $\Phi 40$ to $\Phi 60$ as Fig.3-4:

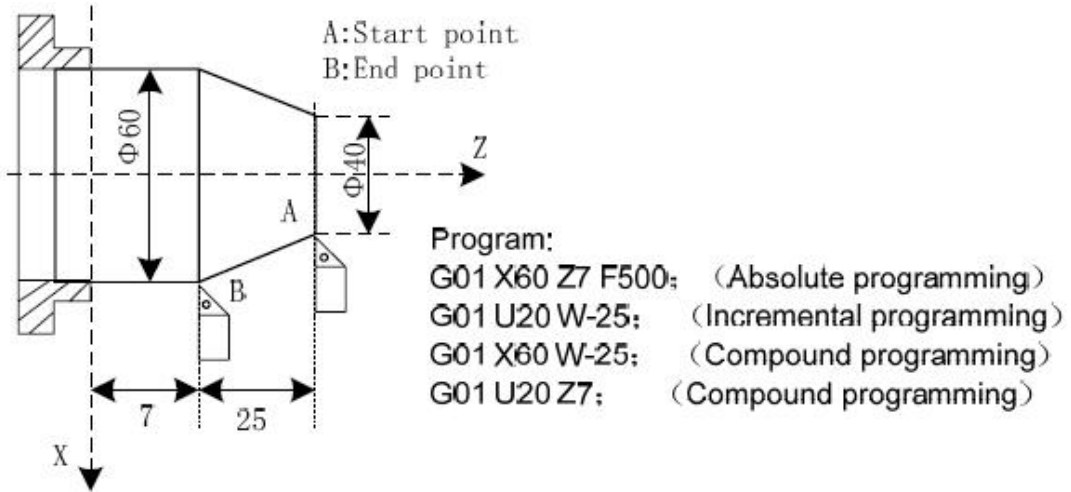


Fig. 3-4

3.4 Circular Interpolation G02, G03

Command format:

G02	}	X(U) _ Z(W) _	{	R _
G03				I _ K _

Command function:

G02 movement path is clockwise (rear tool post coordinate system)/counterclockwise (front tool post coordinate system) arc from starting point to end point as Fig. 3-5.

G03 movement path is counterclockwise (rear tool post coordinate system)/clockwise (front tool post coordinate system) arc from starting point to end point as Fig. 3-6.

Command path:

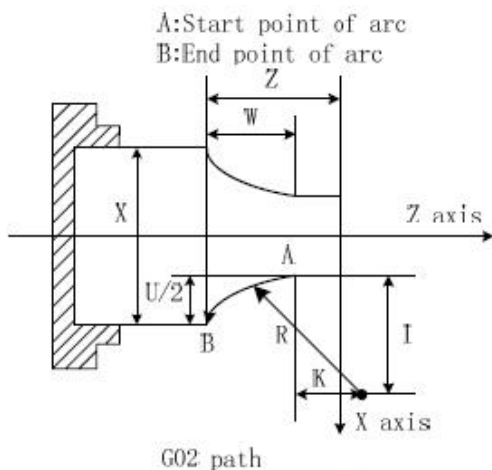


Fig. 3-5 G02 path

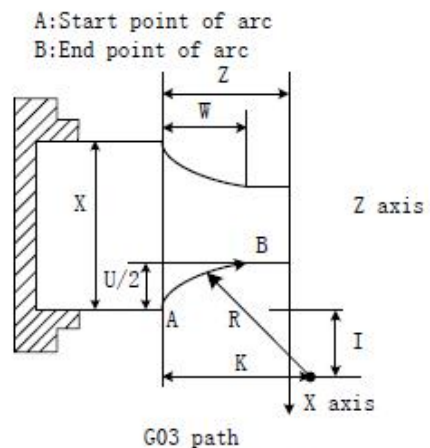


Fig.3-6 G03 path

Command specification:

G02, G03 are modal,

R is arc radius, range: ±99999999× least input increment;

I: X difference value between circle center and starting point of arc;
 K: Z difference value between circle center and starting point of arc;
 Center point of arc is specified by address I, K which separately corresponds to X, Z, I, K
 expresses the vector (it is the increment value) from starting point to center point of arc as the
 following figure;
 I=Coordinates of center point-that of starting point in X direction; K= Coordinates of center
 point-that of starting point in Z direction;
 I, K are with sign symbol. When directions of I, K are the same as those of X, Z, they are positive,
 otherwise, they are negative.
 X, U, Z, W, R, I, K range is referred to Table 1-2 of Section 1.4.1, unit: mm/inch.

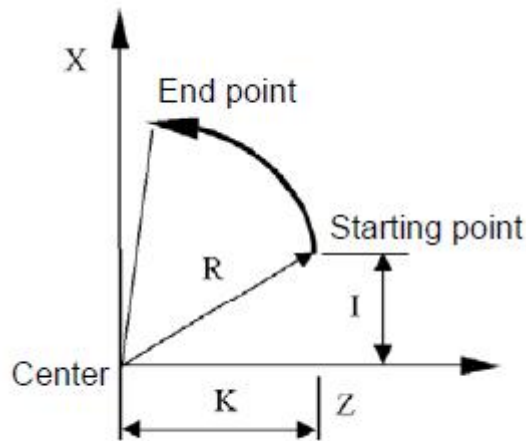


Fig. 3-6-1

Arc direction: G02/G03 direction (clockwise/counterclockwise) is opposite on the front tool post coordinate system and the rear one as Fig.3-7:

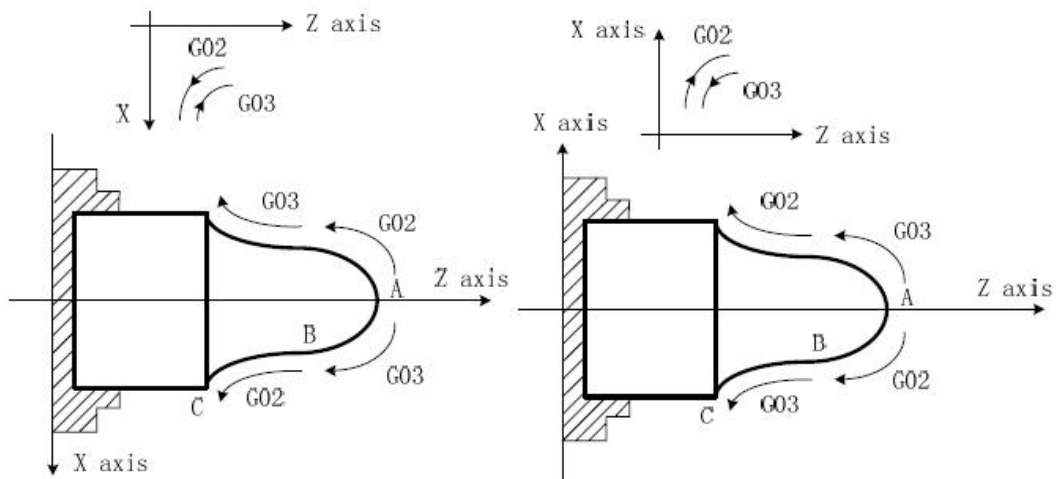


Fig. 3-7

Notes:

When $I = 0$ or $K = 0$, they can be omitted; one of I, K or R must be input, otherwise the system alarms.

R is valid and I, K are invalid when they are input at the same time.

R value must be equal to or more than half distance from starting point to end point, and

The system alarms if the end point is not on the arc defined by R command;

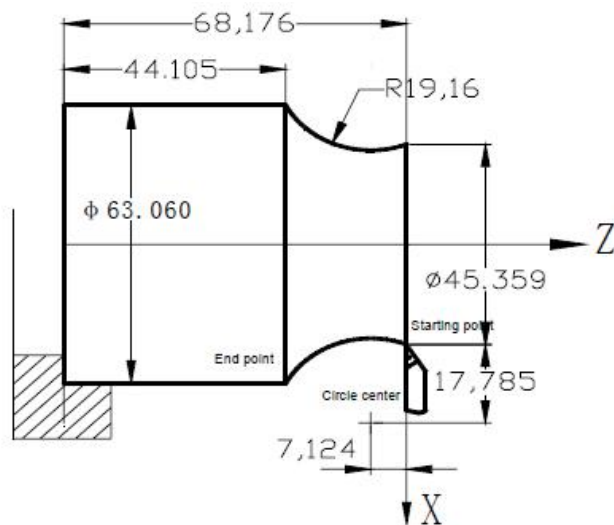
Omit all or one of X(U), Z(W); coordinates of starting point and end point of this axis are the same when omitting ones, the path is a full circle(360°) in G02/G03 when center point are specified by I,K; the path is 0(0°) when center point is specified by R.

R should be used for programming. The system executes in $R = \sqrt{I^2 + K^2}$ to ensure Starting point and end point of arc path are the specified ones in I, K programming.

When the distance from center point to end point is not equal to R($R = \sqrt{I^2 + K^2}$) in I,K programming, the system automatically adjusts position of center point to ensure starting point and end point of arc path are the specified ones; when the distance from center point to end point is more than 2R, and the system alarms.

Arc is less than 360° when R is commanded, the arc is more than 180° when R is negative, and it is less than or equal to 180° when R is positive.

Example: Arc cutting path from $\Phi 45.25$ to $\Phi 63.06$ shown in Fig. 3-8.



Program:

```
G02 X63.060 Z-24.071 R19.16 F300 ; or
G02 U17.701 W-24.071 R19.16 F300 ; or
G02 X63.060 Z-24.071 I17.785 K-7.124 ; or
G02 U17.701 W-24.071 I17.785 K-7.124 F300
```

Fig. 3-8

Compound programming in G02/G03:

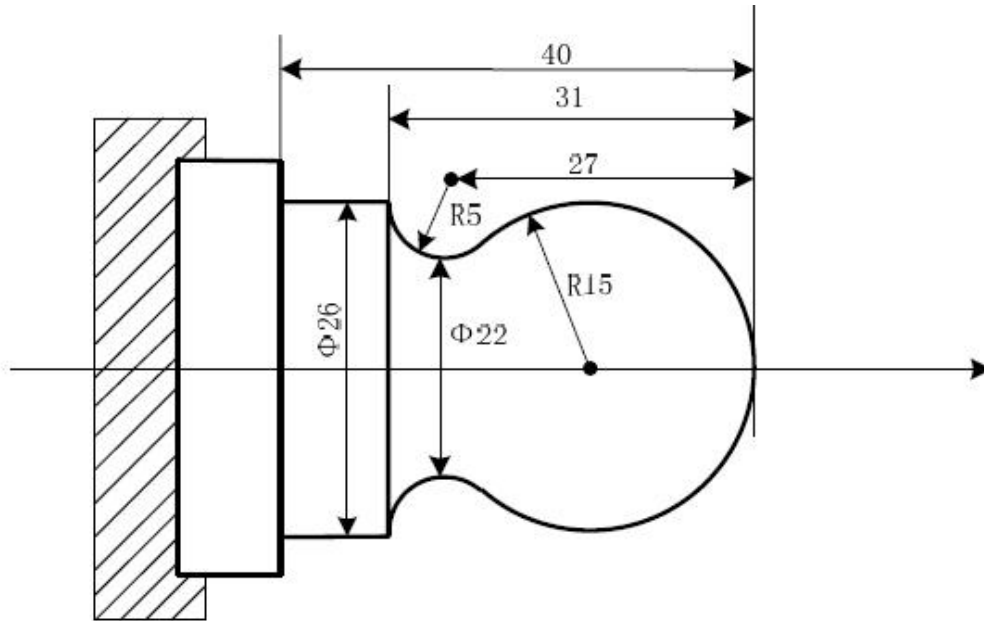


Fig. 3-9 Circular programming example

```

Program: O0001
N001 G0 X40 Z5; (Rapidly traverse)
N002 M03 S200; (Start spindle)
N003 G01 X0 Z0 F900; (Approach workpiece)
N005 G03 U24 W-24 R15; (Cut R15 arc)
N006 G02 X26 Z-31 R5; (Cut R5 arc)
N007 G01 Z-40; (Cut φ26)
N008 X40 Z5; (Return to starting point)
N009 M30; (End of program)

```

3.8 Plane Selection G17~G19

Command format:

G17.....XY plane

G18.....ZX plane

G19.....YZ plane

Command function: use G commands to select the plane of the arc interpolation or the one of the cutter compensation

Command explanation: G17, G18, G19 are modal, and the plane does not change in the block without the command.

Notes:

Firstly set the basic axis Y when the system selects G17, G19 plane;

Cannot switch the planes in C tool compensation;

G71~G76, G90, G92, G94 can be used in G18 plane;

The plane selection code can be in the same block with G codes in the other groups;

The movement command is not relevant to the plane selection;

Diameter or radius programming: currently, because there is only one bit parameter No 1.2 to select the diameter or the radius programming and is valid to only X axis, Z and Y axis use the only radius programming in G2, G3, and X axis is selected by the parameter;

The tool nose direction of C tool compensation is 0 in G17, G19.

3.11 Chamfering Function

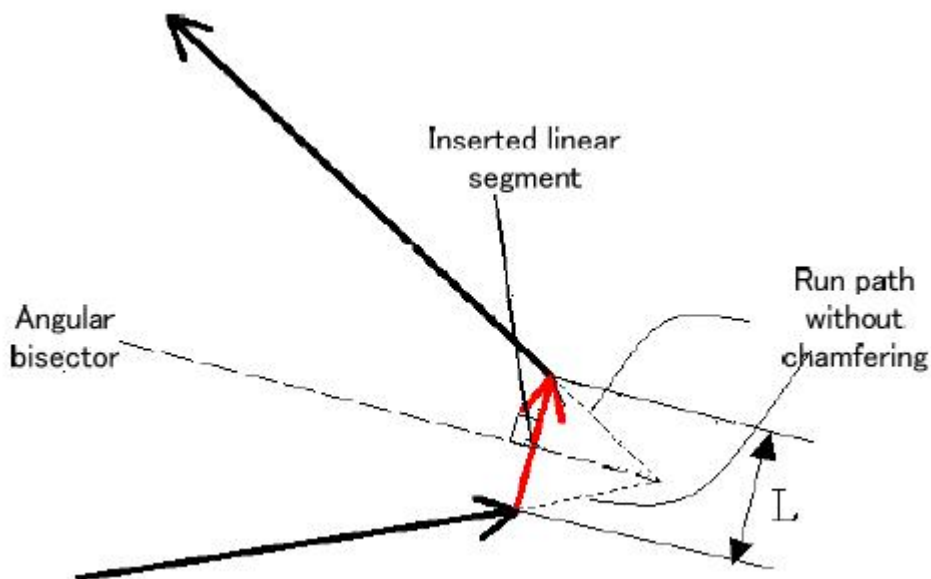
Chamfering function is to insert one straight line or circular between two contours to make the tool smoothly transmit from one contour to another one. ECN*000T uses the linear and circular chamfering functions.

3.11.1 Linear chamfering

Linear chamfering: insert one straight line in the linear contours, arc contours, linear contour and arc contour. The command address of linear chamfering is L, behind which data is the length of chamfering straight line. The linear chamfering must be used in G01, G02 or G03 command.

A. Linear to linear

Command format: G01 X(U)_ Z(W)_ L_ ;
G01 X(U)_ Z(W)_ ;



Command function: insert one straight line between two linear interpolation blocks.

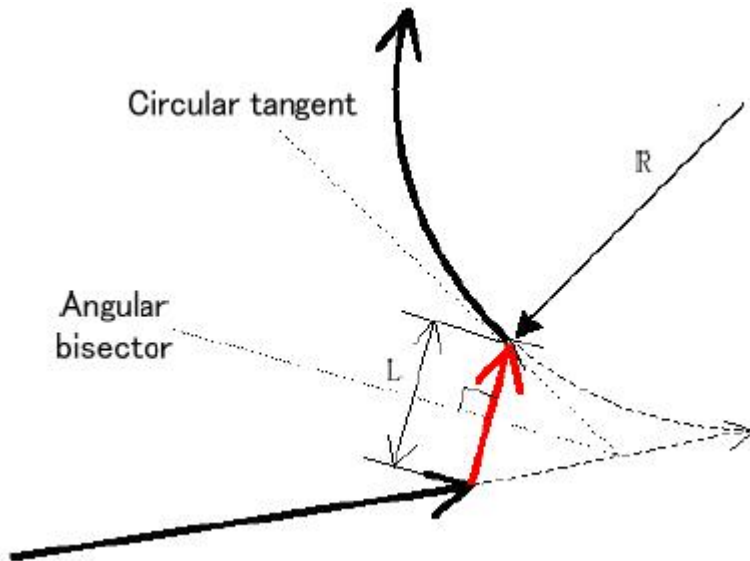
B. Linear to circular

Command format: G01 X(U)_ Z(W)_ L_ ;
G02/G03 X(U)_ Z(W)_ R_ ;

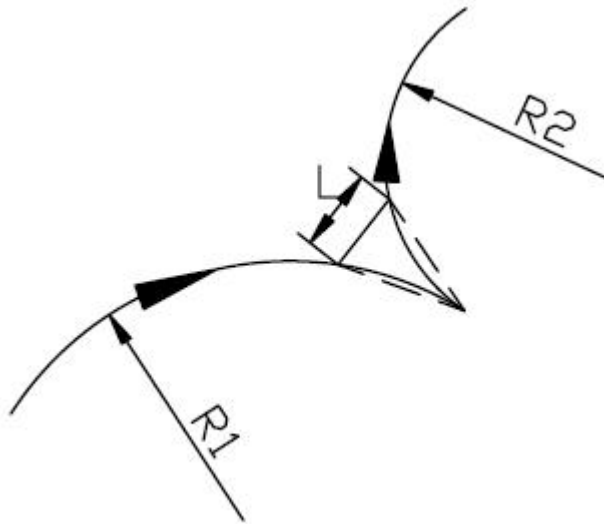
Or

G01 X(U)_ Z(W)_ L_ ;
G02/G03 X(U)_ Z(W)_ I_ K_ ;

Command function: insert one straight line between the linear and circular interpolation blocks.



C. Circular to circular



Command format:

G02/G03 X(U)_Z(W)_R_L_;

G02/G03 X(U)_Z(W)_R_;

Or

G02/G03 X(U)_Z(W)_I_K_L_;

G02/G03 X(U)_Z(W)_I_K_;

Command function: insert one straight line between two circular interpolation blocks.

D. Circular to linear

Command format: G02/G03 X(U)_Z(W)_R_L_;

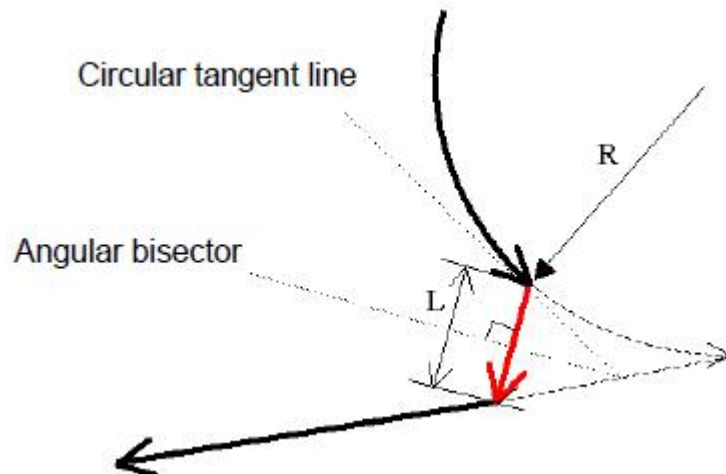
G01 X(U)_Z(W)_;

Or

G02/G03 X(U)_Z(W)_I_K_L_;

G01 X(U)_Z(W)_;

Command function: insert one straight line block between circular and linear interpolation block.



3.11.2 Circular chamfering

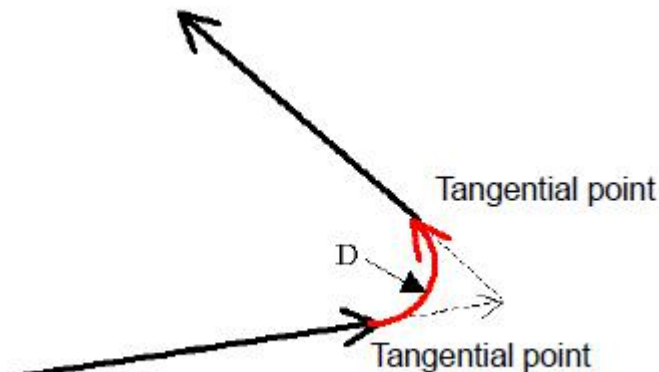
Circular chamfering: insert one circular between linear contours, circular contours, linear contour and circular contour, the circular and the contour line are transited by the tangent. The command of circular chamfering is D, and the data behind the command is the radius of chamfering circular. The circular chamfering must be used in G01, G02 or G03.

A. Linear to linear

Command format:

G01 X(U)_ Z(W)_ D_;

G01 X(U)_ Z(W)_;



Command function: insert one circular between two straight lines, the inserted circular block and two straight lines are tangent, the radius is the data behind the command address D.

B. Linear to circular

Command format:

G01 X(U)_ Z(W)_ D_;

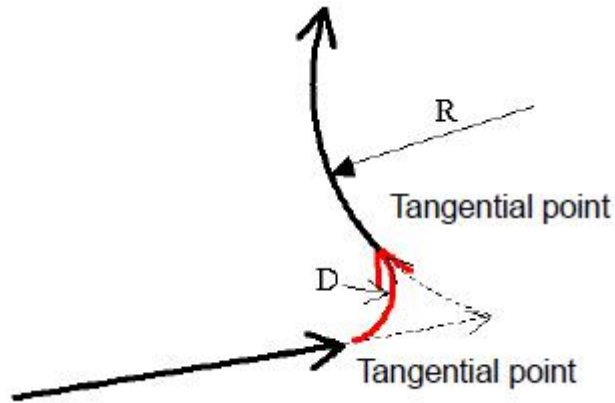
G02/G03 X(U)_ Z(W)_ R_;

or

G01 X(U)_ Z(W)_ D_;

G02/G03 X(U)_ Z(W)_ I_ K_;

Command function: insert one circular between linear and circular, the inserted circular is tangent to the linear and the circular, and the radius is the data behind the command address D.



C. Circular to circular

Command format: G02/G03 X(U)_Z(W)_R_D_;

G02/G03 X(U)_Z(W)_R_;

or

G02/G03 X(U)_Z(W)_R_D_;

G02/G03 X(U)_Z(W)_I_K_;

or

G02/G03 X(U)_Z(W)_I_K_D_;

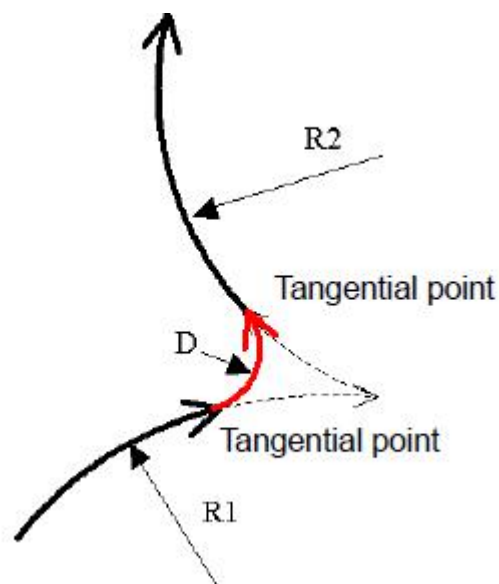
G02/G03 X(U)_Z(W)_I_K_;

or

G02/G03 X(U)_Z(W)_I_K_D_;

G02/G03 X(U)_Z(W)_R_;

Command function: insert one circular between two circular blocks, the inserted circular is tangent to the two circular blocks, and the radius is the data behind the command address D.



D. Circular to linear

Command format: G02/G03 X(U)_Z(W)_R_D_;

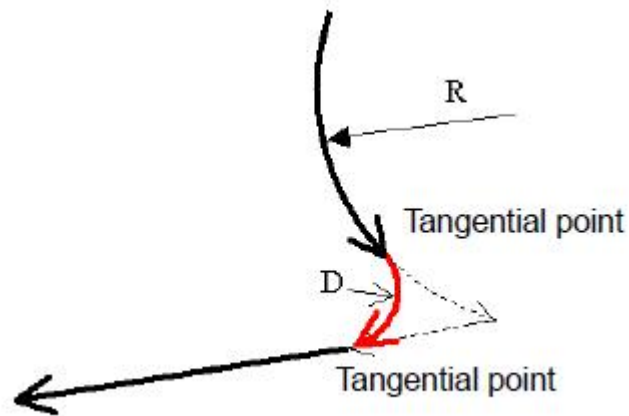
G01 X(U)_Z(W)_;

Or

G02/G03 X(U)_Z(W)_I_K_D_;

G01 X(U)_Z(W)_;

Command function: insert one circular block between the circular and the linear, the inserted circular block is tangent to the circular and the linear, and the radius is the data behind the command address D.



3.11.3 Special cases

The chamfering function is invalid or alarms as follows:

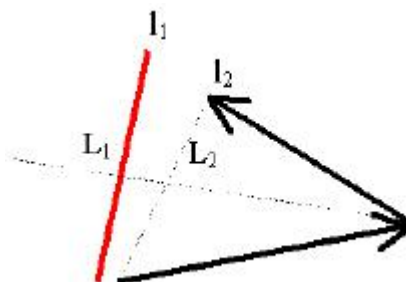
1) Linear chamfering

A. The chamfering function is invalid when two interpolation straight lines are in the same linear.



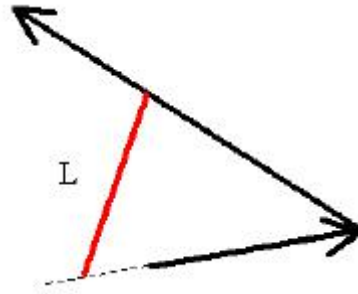
B. CNC alarms when the chamfering linear is too long.

L_1 is the chamfering linear, and the length is L_1 ; l_2 is the third edge of the triangle which is formed by two interpolation straight lines, the length is L_2 , CNC alarms when L_1 is bigger



C. Some linear block is too short

The chamfering linear length is L , CNC alarms when other end of the calculated chamfering linear is not in the interpolation linear (in the extension line of the interpolation linear).



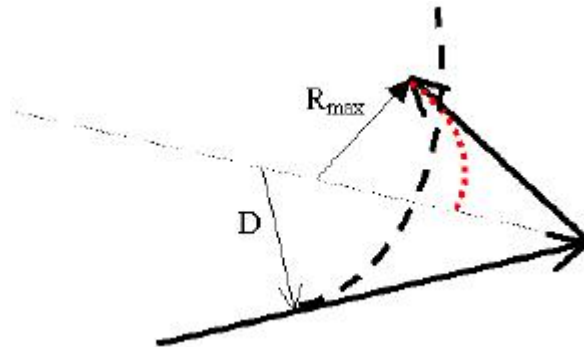
2) Circular chamfering

A. The circular chamfering function is invalid when two interpolation straight lines are in the same block.



B. CNC alarms when the chamfering circular radius is too big.

CNC alarms when the chamfering circular radius is D , max. circular radius of the tangential linear lines is R_{max} which is less than D as follows.



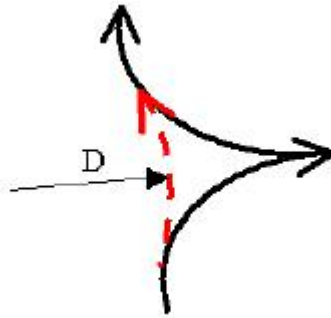
C. The circular chamfering function is invalid when the linear and the circular, or the circular and the linear are tangential.



D. The circular chamfering function is invalid when one circular and another one are tangential.



The circular chamfering function is valid when the circular tangency is as follows:



3.12 Dwell G04

Command format: G04 P__ ; or

G04 X__ ; or

G04 U__ ; or

G04;

Command function: each axis stops the motion, the modal of G commands and the reserved data, state are not changed, and execute the next block after dwelling the defined time.

Command specification: G04 is non-modal.

G04 dwell time is defined by the word P__, X__ or U__.

Range of -99999999~99999999 (unit: ms) .

X, U range is -99999999~99999999 x least input unit (unit: s)

Notes:

The exact stop is executed between the blocks when P, X, U are not input.

The system exactly stop a block when P, X, U are not input or P, X, U specify negative values.

P is valid when P, X, U are in the same block; X is valid when X, U are in the same block.

When the system executes the feed hold in G04, dwell can be executed after the Current delay time.

3.13 Machine Zero Function

3.13.1 Machine 1st reference point G28

Command format: G28 X(U) Z(W) ;

Command function: the tool rapid traverses to the middle point defined by X(U), Z(W) from starting point and then return to the machine zero.

Command specifications:

G28 is non-modal.

X, Z, Y: absolute coordinates of middle point;

U, W, V: Z absolute coordinates of middle point;

W: Difference value of absolute coordinates between middle point and starting point in Z direction. Omit all or one of X(U), Z(W) as follows:

Table 3-4

Command	Function
G28 X(U) __	X returns to machine zero and Z axis remains in the previous position
G28 Z(W) __	Z returns to machine zero and X axis remains in the previous position
G28	in the previous positions and continuously execute the next block
G28 X(U) __ Z(W) __	X, Z return to machine zero simultaneously

Running path(as Fig. 3-12) :

- (1) Rapid traverse to middle point of specified axis from current position(A point→ B point) ;
- (2) Rapid traverse to reference point from the middle point(B point→ R point) ;
- (3) If the machine is not locked, LED is ON when the machine reference point return is completed.

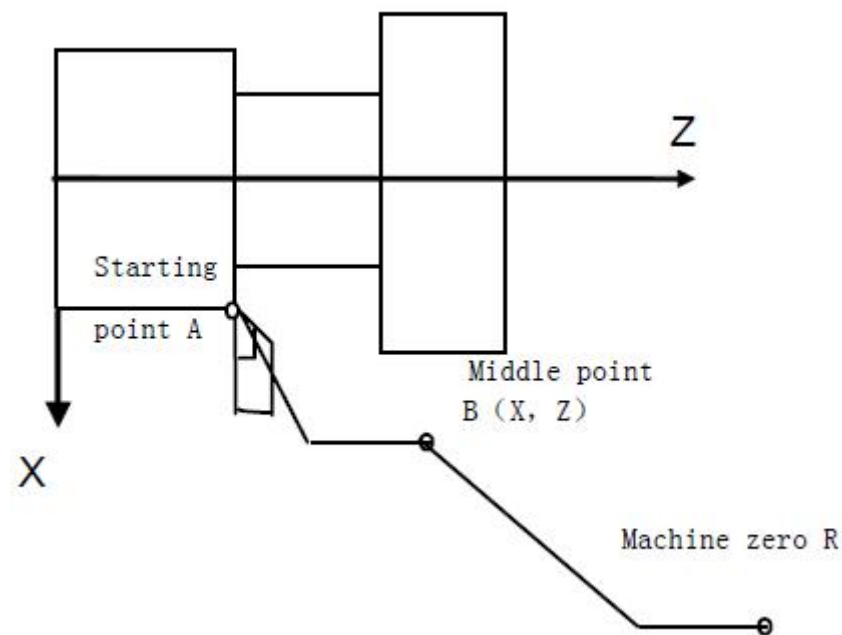


Fig.3-12

Note 1: Do not execute G28 and machine zero return without the zero switch on the machine.

Note 2: Machine zero returns in Jog mode and in G28 are the same and their deceleration signals and the signal every rotation must be detected;

Note 3: X and Z move at the respectively rapid traverse speed from A to B and from B to R, and so the path is not always a straight line;

Note 4: The system cancels the tool length compensation after executing G28 to perform the machine zero return;

3.13.2 Machine 2nd, 3rd, 4th reference point G30

Machine zero is fixed point in the machine tool, decided by the zero switch and zero return switch installed on the machine tool.

Machine reference point is located at the position after the machine zero offsets No.114 or No.115 value, when No.114, No.115 setting value is 0, the machine reference point coincides with the machine zero. The coordinates of machine reference point are No.120, No.121 setting value. Executing machine zero return is considered to executing the machine reference point return.

ECN*000T has machine 2nd, 3rd, 4th reference point functions. Use separately No.122~No.127

Command Function

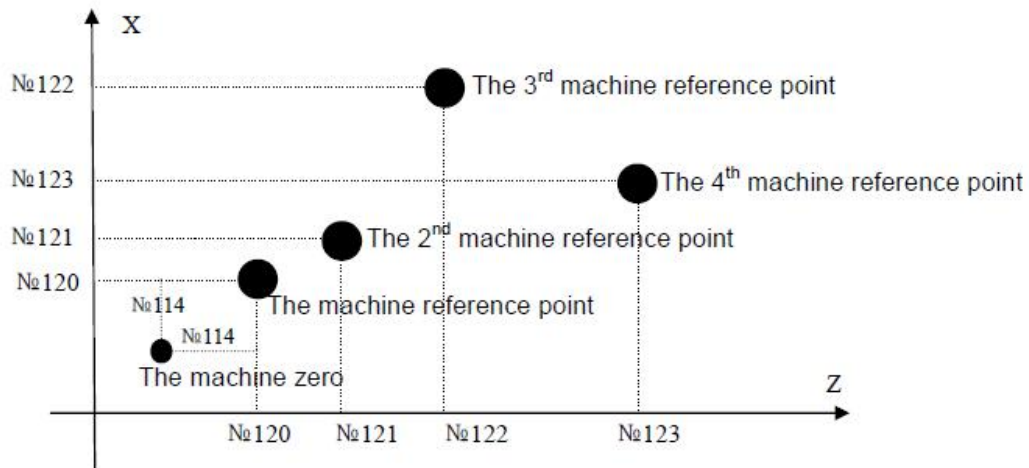
G28 X(U) X returns to machine zero and Z axis remains in the previous position

G28 Z(W) Z returns to machine zero and X axis remains in the previous position

G28 in the previous positions and continuously execute the next block

G28 X(U) Z(W) X, Z return to machine zero simultaneously to set X, Z machine coordinates of the machine 2nd, 3rd, 4th reference point.

The relationship between the machine zero, machine reference point, machine 2nd, 3rd, 4th reference point is as follows:



Command format:

G30 P2 X(U) __ Z(W) __;

G30 P3 X(U) __ Z(W) __;

G30 P4 X(U) Z(W) ;

Command function: the tool rapidly traverses with the rapid traverse speed to the middle point specified by X(U) , Z(W)

Command specifications: G30 is non-modal.

X: X absolute coordinate of the middle point;

U: difference value of X absolute coordinate value between the middle point and starting point;

Z: Z absolute coordinate of the middle point;

W: difference point of Z absolute coordinate between the middle point and starting point.

Omit one or all of X(U) , Z(W) as follows:

Command Function

G30 Pn X(U) X returns to the machine nth reference point, Z axis retains

G30 Pn Z(W) Z return to the nth machine reference point, X axis retains

G30 X and Z retain, go on executing the next program block

G30 Pn X(U) Z(W) X and Z return to the machine nth reference point simultaneously

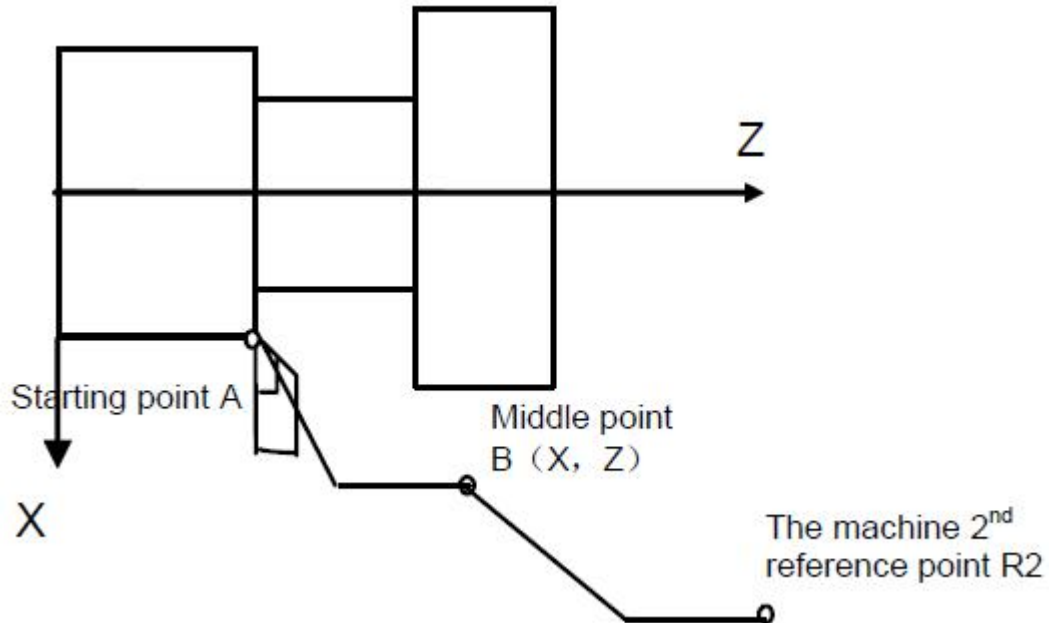
Command	Function
G30 P _n X(U) __	X returns to the machine nth reference point, Z axis retains
G30 P _n Z(W) __	Z return to the nth machine reference point, X axis retains
G30	X and Z retain, go on executing the next program block
G30 P _n X(U) __ Z(W) __	X and Z return to the machine nth reference point simultaneously

Note 1: n in the above table is 2, 3 or 4;

Note 2: Do not check the deceleration, zero signal when you execute the machine 2nd, 3rd, 4th reference point.

Command operations: (taking example of returning to machine 2nd reference point as follows):

- (1) Rapidly traverse to the middle position of command axis from the current position (A point →B point);
- (2) Traverse from the middle point with the speed set by No.113 to the 2nd reference point set by No.122 and No.123 (B point →R2 point);
- (3) When CNC is not in the machine lock state, the completion signal of reference point return ZP21 Bit0, Bit1 is high.



Note 1: Execute the machine 2nd, 3rd, 4th reference point return after you manually execute the machine reference point return or G28 (machine reference point return).

Note 2: A→B and B→R2, two axes separately traverse, and so their trails are linear or not.

Note 3: CNC cancels the tool length compensation after you execute G30 to return 2nd, 3rd, and 4th reference point.

Note 4: Must not execute G30 (machine 2nd, 3rd, 4th reference point return) when the zero switch is not installed on the machine.

Note 5: Do not set the workpiece coordinate system when you execute the 2nd, 3rd, and the machine 4th reference point return.

3.14 Skip Interpolation G31

Command format: G31 X(U)_Z(W)_F_;

Command function: in executing the command, when the outside skip signal (X3.5) is input, the system stops the command to execute the next block. The function is used to the dynamic measure (such as milling machine), toolsetting measure and so on of workpiece measure.

Command explanations: non-modal G command (00 group);

Its address format is same that of G01;

Cancel the tool nose radius compensation before using it;
 Feedrate should not be set to too big to get the precise stop position;
 a. following block execution after skip:

1. The next block of G31 is the incremental coordinate programming shown in Fig. 3-13:

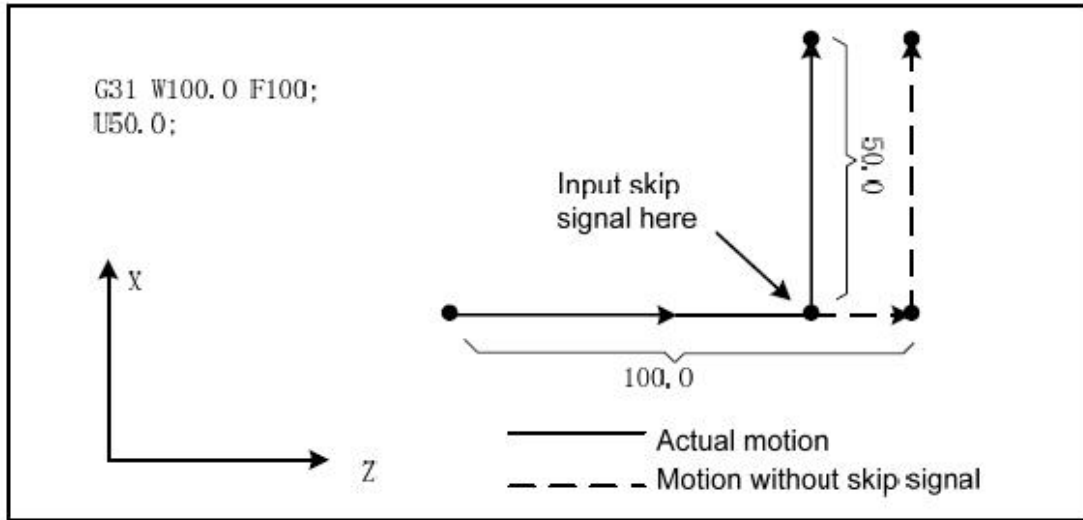


Fig. 3-13

2. The next block of G31 is the absolute coordinate programming of one axis as Fig. 3-14:

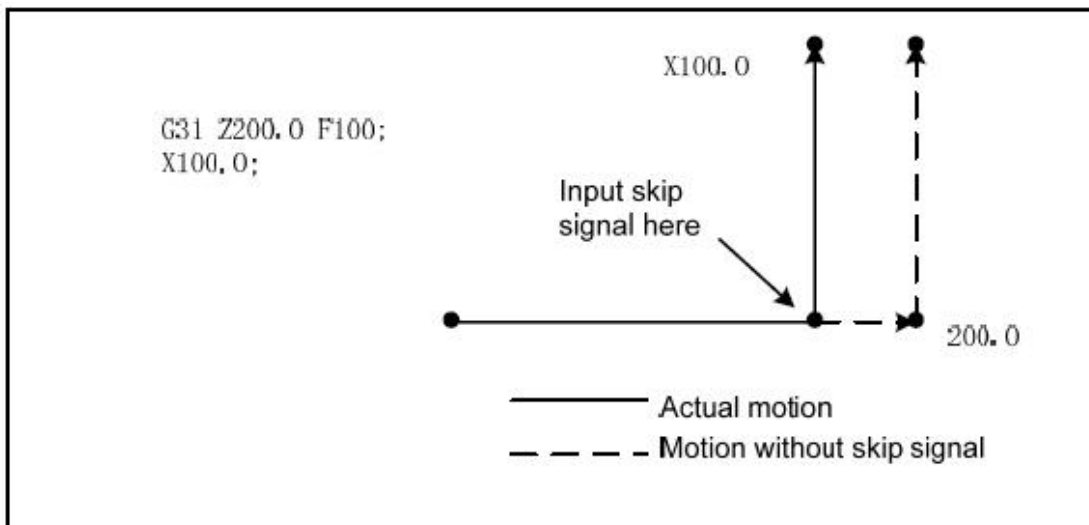


Fig. 3-14

3. The next block of G31 is the absolute coordinate programming of two axes shown in

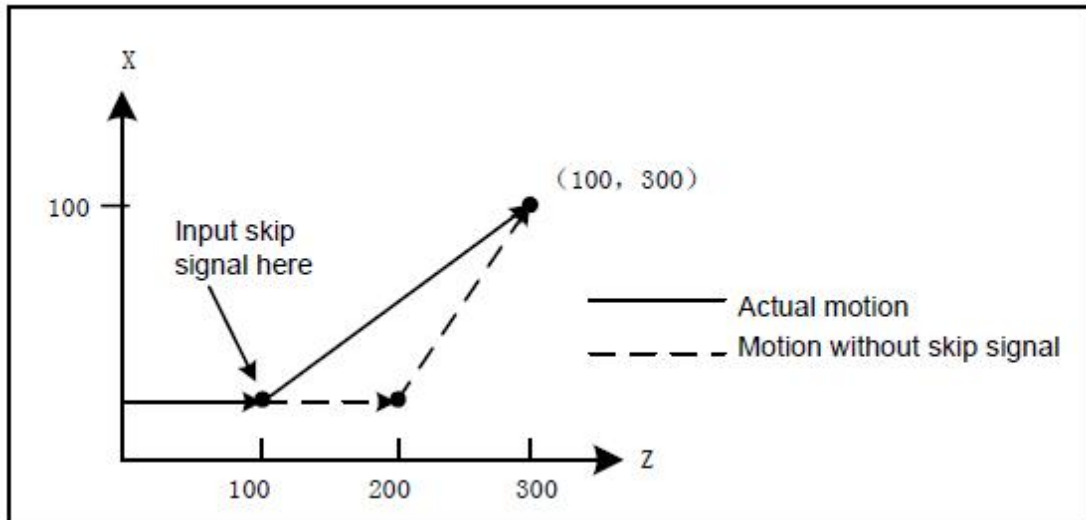


Fig. 3-15:

Program:

G31 Z200 F100

G01 X100 Z300

b. Signals relevant to G31

Skip signal:

SKIP:G6.6

Type: input signal

Function: G6.6 ends the skip cutting. I.e. in a block containing G31, the skip signal becoming the absolute coordinate position of "1" is to be stored in the macro variable (#5011~#5015 separately corresponds to X, Z, Y)

Operation: when the skip signal becomes "0", CNC executes as follows:

When the block is executing G31, CNC stores the current absolute coordinates of each axis. CNC stops G31 to execute the next block, the skip signal detects its state instead of its RISING EDGE. So when the skip signal is "1", it meets the skip conditions.

Note:

CNC immediately stops the feed axis (without acceleration/deceleration execution), and G31 feedrate should be as low as possible below 1000 mm/min to get the precise stop position.

3.16 Workpiece Coordinate System G50

Command format: G50 X(U) Z(W) ;

Command function: define the absolute coordinates of current position and create the workpiece coordinates system (called floating coordinates system) by setting the absolute coordinates of current position in the system. After G50 is executed, the system takes the current position as the program zero (program reference point), and the system returns to the point after executing the program zero return. After the workpiece coordinate system is created, input the coordinate values with the coordinate system in the absolute coordinates programming until the next workpiece coordinate system is created again (using G50).

Command specifications:

G50 is non-modal;

X: New absolute coordinates of current position in X direction;

U: Different value between the new absolute coordinates of current position in X direction

And the absolute coordinates before executing commands;

Z: New absolute coordinates of current position in Z direction;

W: Different value between the new absolute coordinates of current position in X direction and the absolute coordinates before executing commands;

In G50, when X(U) or Z(W) are not input, the system does not change current coordinates position as program zero; when X(U) and Z(W) are not input, the system takes the previous setting position as program zero.

Example:

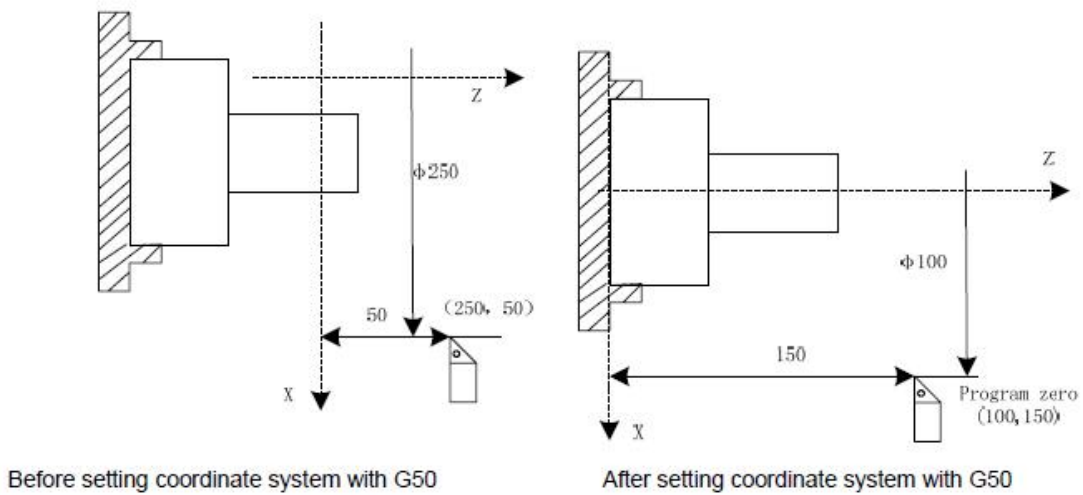


Fig.3-16

As Fig.3-16, create the above-mentioned workpiece coordinate system and set (X100 Z150) to the reference point of program after executing “G50 X100 Z150”.

Note: When No.003 Bit4 is 1(executing tool compensation by coordinates offset), T function is executed, motion command is not executed and the system creates workpiece coordinate system with G50, the displayed coordinate value are ones which are defined by G50 adding or subtracting tool compensation value which is not executed.

Current tool compensation state	Executing motion command	Coordinate value after executing G50 X20 Z20	No. 01 tool compensation value
T0100 or T0101	G0 X_ Z	X: 20 Z: 20	X: 12 Z: 23
	Motion command not executed	Coordinate value after executing G50 X20 Z20	
	※※※	X: 8 Z: -3 or X: 32 Z: 43	

3.18 Workpiece Coordinate System G54~G59

Command format:

G54 workpiece coordinate system 1

G55 workpiece coordinate system 2

G56 workpiece coordinate system 3

G57 workpiece coordinate system 4

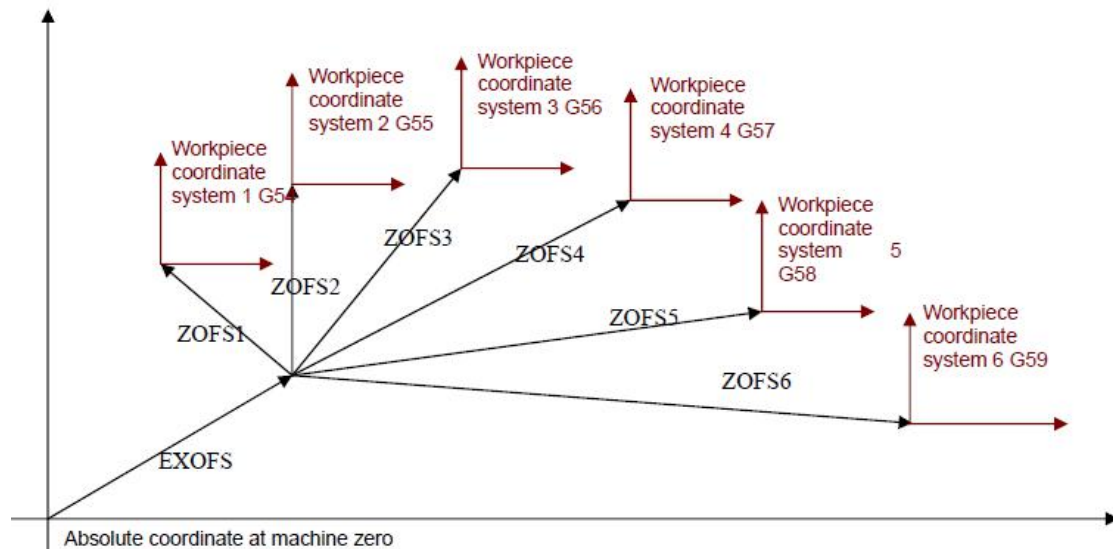
G58 workpiece coordinate system 5

G59 workpiece coordinate system 6

Command function: one of G54~G59 is specified, one of workpiece coordinate system 1~6 is selected. After a workpiece coordinate system is specified, the specified point is in the specified workpiece coordinate system till a new workpiece coordinate system is set.

Explanation: G54~G59 are modal.

In the system, each coordinate system should correspond to ZOFS1~ ZOFS6 of zero offset value of one workpiece coordinate system, relation between ZOFS1~ ZOFS6 and previous workpiece is shown below:



When №11.7 (APRS) is set to 1, an absolute coordinate system is set after machine zero return, and then EXOFS and ZOFSn setting values are offset. Current workpiece coordinate setting after zero return is shown: new workpiece coordinates=current absolute coordinates- (ZOFSn + EXOFS) .

G54~G59 workpiece coordinate system switch

Specifying G54~G59 can switch 6 workpiece coordinate systems to make the system work in different workpiece systems. Absolute coordinate variation of current position is the origin offset value between the new workpiece coordinate system and the old. Namely : new absolute coordinates =current absolute coordinates - (ZOFS new – ZOFS old)

Example:

Example 1:

Absolute coordinates after zero return is (0,0) ; EXOF=(0,0) ; ZOFS1=(-10,-10);

Absolute coordinate values after zero return in G54: $0 - (-10 + 0) = 10$;

Example 2:

Absolute coordinate values after zero return: (20,20) ; EXOF=(5,5) ; ZOFS2=(10, 10);

Absolute coordinate values after zero return in G55: $20 - (10 + 5) = 5$;

Example 3:

Current absolute coordinates (10,10); EXOF=(5,5) ; ZOFS1=(-10,-10); ZOFS2 = (-30,-30); absolute coordinate values from G54 to G55: $10 - (-30 - (-10)) = 30$

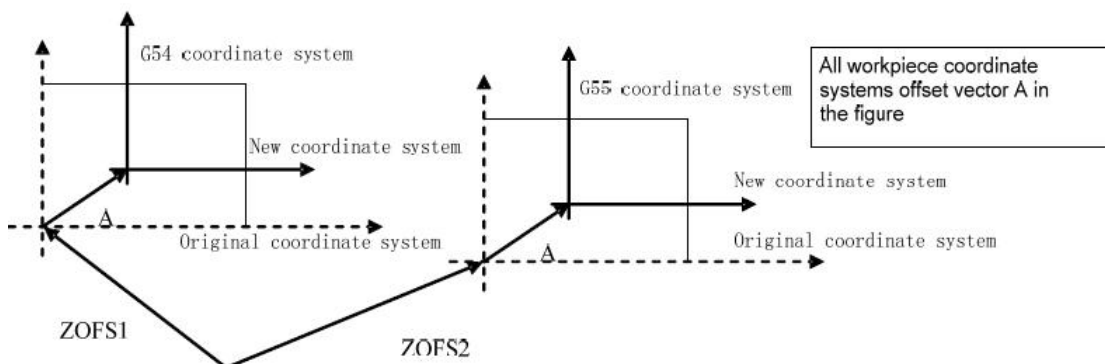
EXOFS: external workpiece coordinate zero (it is added to zero offset value of each workpiece coordinate system, its value is a part of tool compensation->setting in the workpiece coordinate system)

ZOFSn: zero offset value of each workpiece coordinate system ZOFS1~ZOFS6 (its value is a part of tool compensation->setting in the workpiece coordinate system)

86

Notes:

- When №11.7 (APRS) is set to 0, the absolute coordinate system after machine zero return is not set, and the EXOF and ZOFSn offset cannot be executed because the workpiece coordinate system is not set again.
- Modification setting of EXOFS and ZOFSn are valid only in non-run state, namely, setting values of each workpiece coordinate system cannot be modified when a program is running.
- The workpiece coordinate system offset is immediately valid after values of EXOFS and ZOFSn are modified.
- Whether the incremental coordinates include offset values of the coordinate system is set by №05.1 (PPD) .
- In tool nose radius compensation mode, compensation is cancelled temporarily in G54~G59, and is recovered in the next movement command.
- When G54~G59 and G50 are in the same block, G54~G59 are invalid, but G50 is valid.
- When G50 sets the coordinate system, origin of the current workpiece system is modified. So, there is an offset value of origins between a new coordinate system set by G50 and previously current coordinate system. The offset value should be added to offset values of origins of all workpiece coordinate systems, i.e. all workpiece coordinate systems move the same values, which are shown below: Offset value A are all workpiece coordinate systems, which are shown below:



3.19 Fixed Cycle Command

To simplify programming, the system defines G command of single machining cycle with one block to complete the rapid traverse to position, linear/thread cutting and rapid traverse to return to the starting point:

G90: axial cutting cycle;

G92: thread cutting cycle;

G94: radial cutting cycle;

G92 will be introduced in section Thread Function.

3.19.1 Axial cutting cycle G90

Command format: G90 X(U) __ Z(W) __ F __; (cylinder cutting)

G90 X(U) __ Z(W) __ R __ F __; (taper cutting)

Command function: From starting point, the cutting cycle of cylindrical surface or taper surface is completed by radial feeding(X) and axial (Z or X and Z) cutting.

Command specifications:

G90 is modal;

Starting point of cutting: starting position of linear interpolation(cutting feed)

End point of cutting: end position of linear interpolation(cutting feed)

X: X absolute coordinates of cutting end point

U: different value of X absolute coordinate between end point and starting point of cutting

Z: different value of Z absolute coordinate between end point and starting point of cutting

W: different value of Z absolute coordinate between end point and starting point of cutting

R: different value (radius value) of X absolute coordinates between end point and start point of cutting. When the signs of R is not the same that of U, $R \leq |U/2|$; when $R = 0$ or the input is default, the cylinder cutting is executed as Fig.3-17, otherwise, the cone cutting is executed as Fig. 3-18; unit: mm.

Ranges of X, U, Z, W, R are referred to Table 1-2 of Section 1.4.1, unit: mm/inch.

Cycle process:

- ① X rapidly travrses from starting point to cutting starting point;
- ② Cutting feed (linear interpolation) from the cutting starting point to cutting end point;
- ③ X executes the tool retraction at feedrate (opposite direction to the above-mentioned), and return to the position which the absolute coordinates and the starting point are the same;
- ④ Z rapidly traverses to return to the starting point and the cycle is completed.

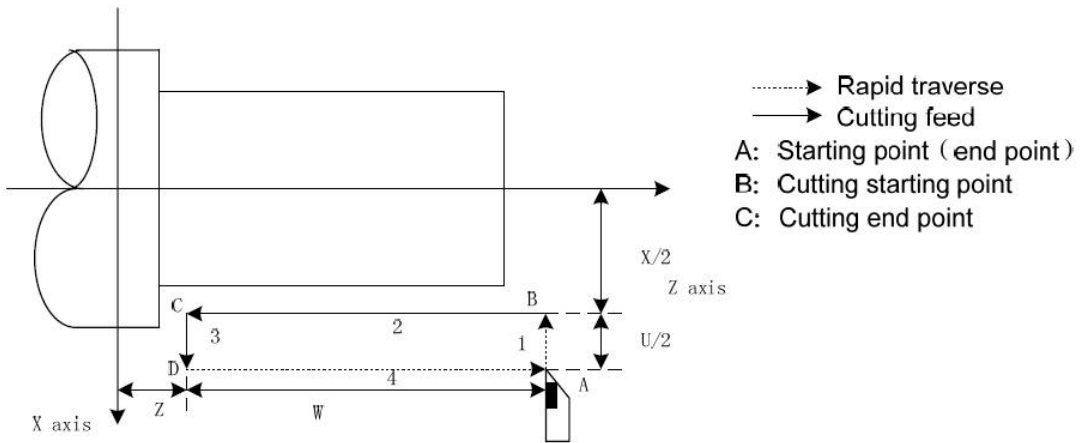


Fig. 3-17

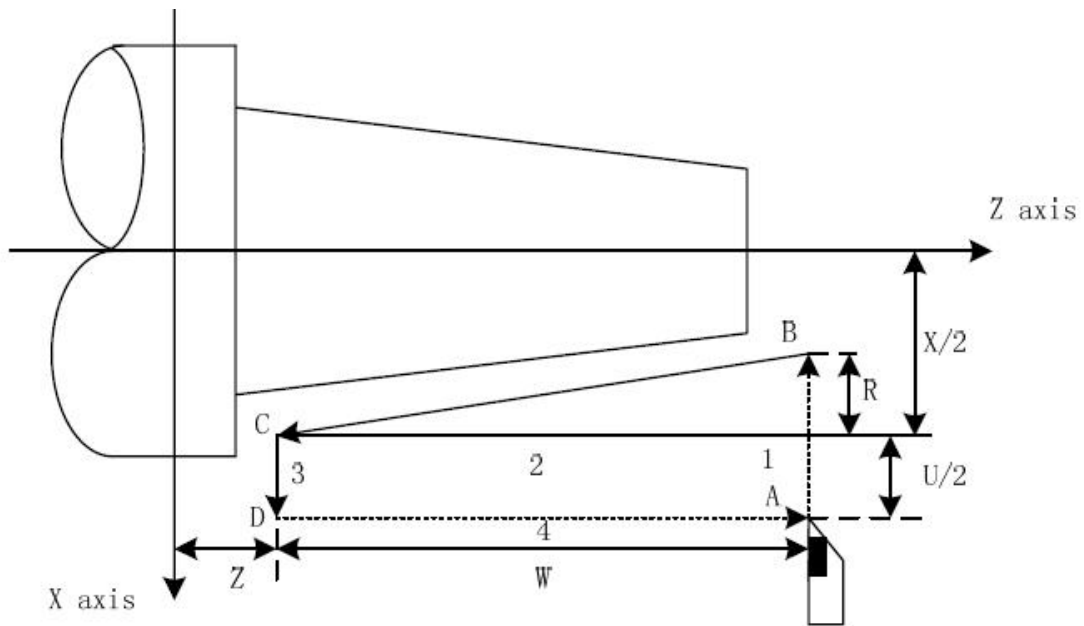
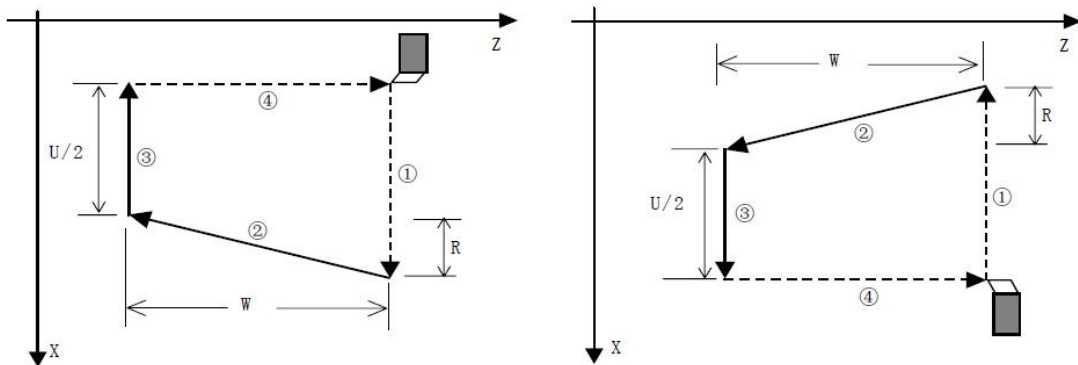


Fig. 3-18

Cutting path: Relative position between cutting end point and starting point with U, W, R, and tool path of U, W, R with different signs are shown in Fig. 3-19:

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

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



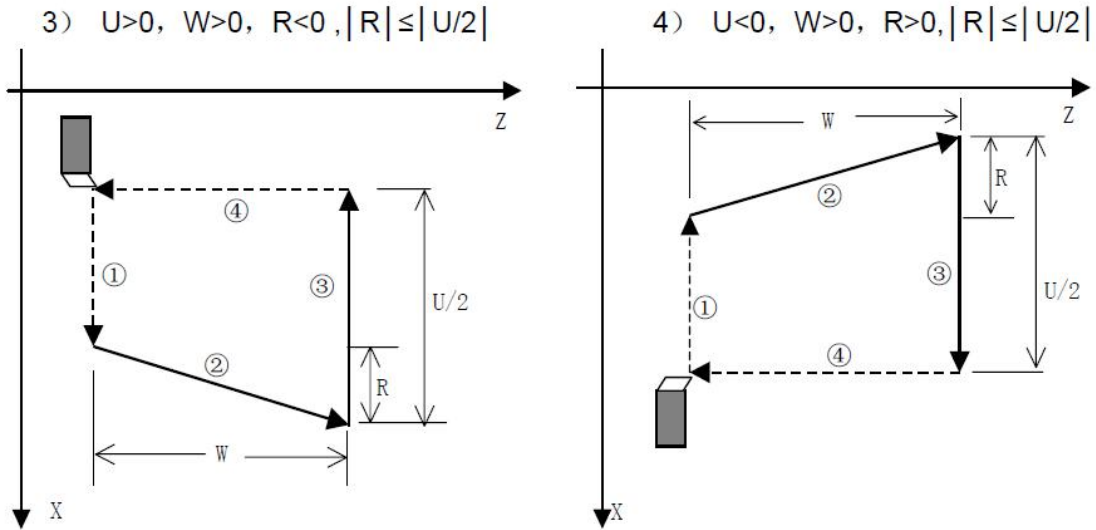


Fig. 3-19

Example: Fig. 3-20, rod $\Phi 125 \times 110$

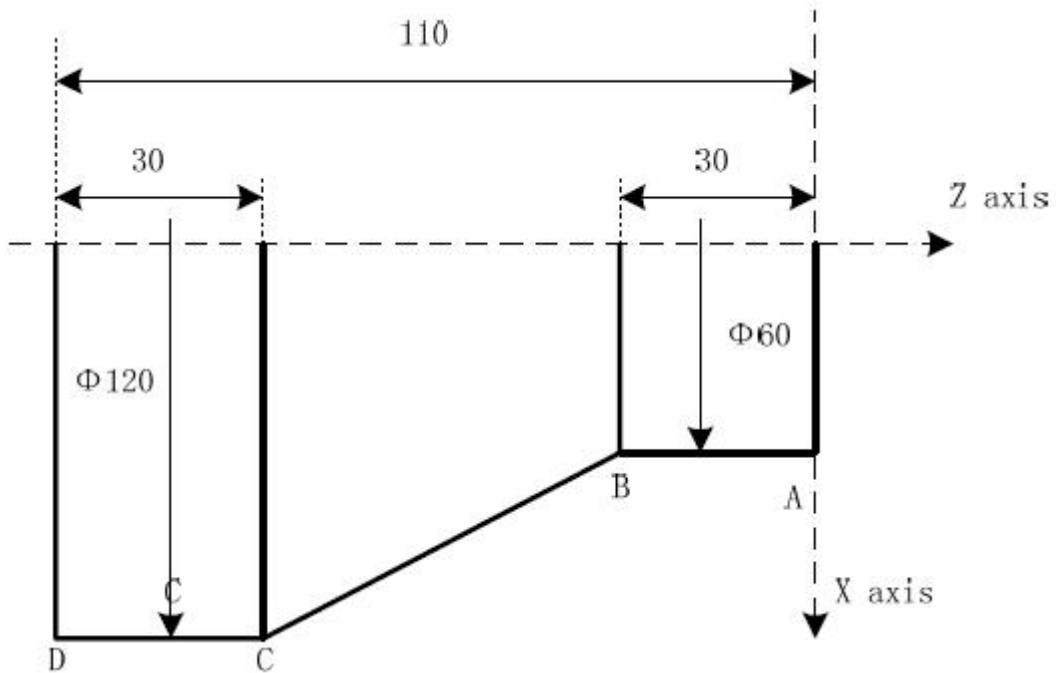


Fig. 3-20

```

Program : O0002;
M3 S300 G0 X130 Z3;
G90 X120 Z-110 F200;      (A→D, cut Φ120)
X110 Z-30;
X100;
X90;
X80;
X70;
X60;
G0 X120 Z-30;
G90 X120 Z-44 R-7.5 F150;
Z-56 R-15
Z-68 R-22.5
Z-80 R-30
M30;

```

} (A→B, 6 times cutting cycle Φ60, increment of 10mm)

} (B→C, 4 times taper cutting)

3.19.2 Radial cutting cycle G94

Command format: G94 X(U) __ Z(W) __ F __; (face cutting)

G94 X(U) __ Z(W) __ R __ F __; (taper face cutting)

Command function: From starting point, the cutting cycle of cylindrical surface or taper surface is completed by radial feeding(X) and axial (Z or X and Z) cutting.

Command specifications:

G94 is modal;

Starting point of cutting: starting position of linear interpolation (cutting feed). Unit: mm;

End point of cutting: end position of linear interpolation (cutting feed). Unit: mm;

X: X absolute coordinate of end point of cutting. Unit: mm;

U: Different value of absolute coordinate from end point to starting point of cutting in X direction .Unit: mm;

Z: Z absolute coordinates of end point of cutting, Unit: mm;

W: Different value of X absolute coordinate from end point to starting point of cutting, Unit: mm;

R: Different value(R value) of X absolute coordinates from end point to starting point of cutting. When the sign of R is not the same as that of U, R, $|R| \leq |W|$.

Radial linear cutting is shown in Fig. 3-21, radial taper cutting is as Fig. 3-22.

Ranges of X, U, Z, W, R are referred to Table 1-2 of Section 1.4.1, unit: mm/inch.

Cycle process:

- ① Z rapidly traverses from starting point to cutting starting point;
- ② Cutting feed (linear interpolation) from the cutting starting point to cutting end point;
- ③ Z executes the tool retraction at the cutting feedrate (opposite direction to the above-mentioned ①), and returns to the position which the absolute coordinates and the starting point are the same;
- ④ The tool rapidly traverses to return to the starting point and the cycle is completed.

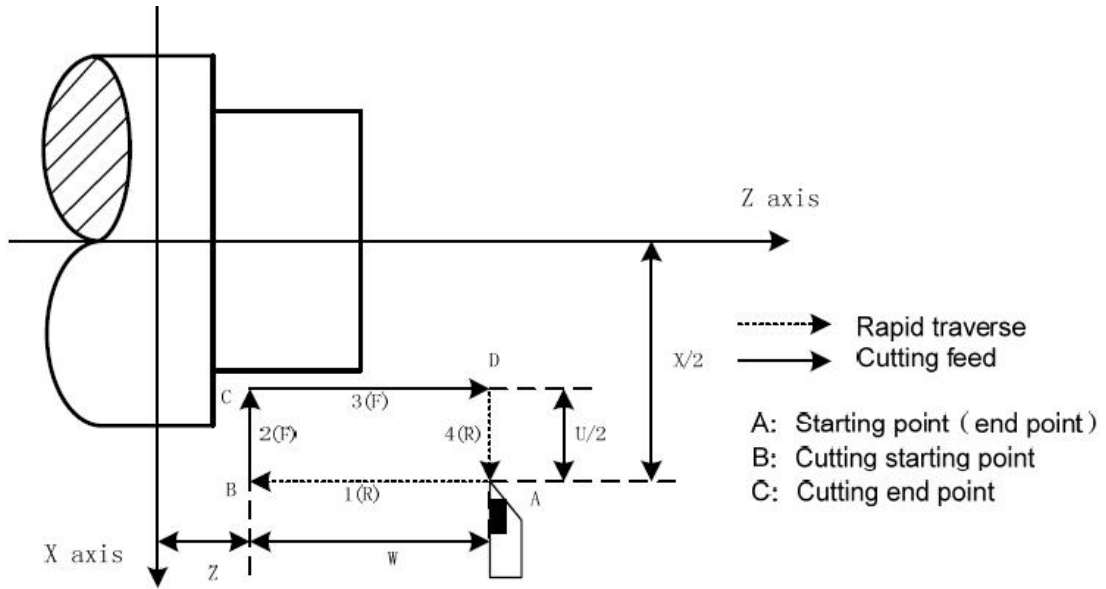


Fig. 3-21

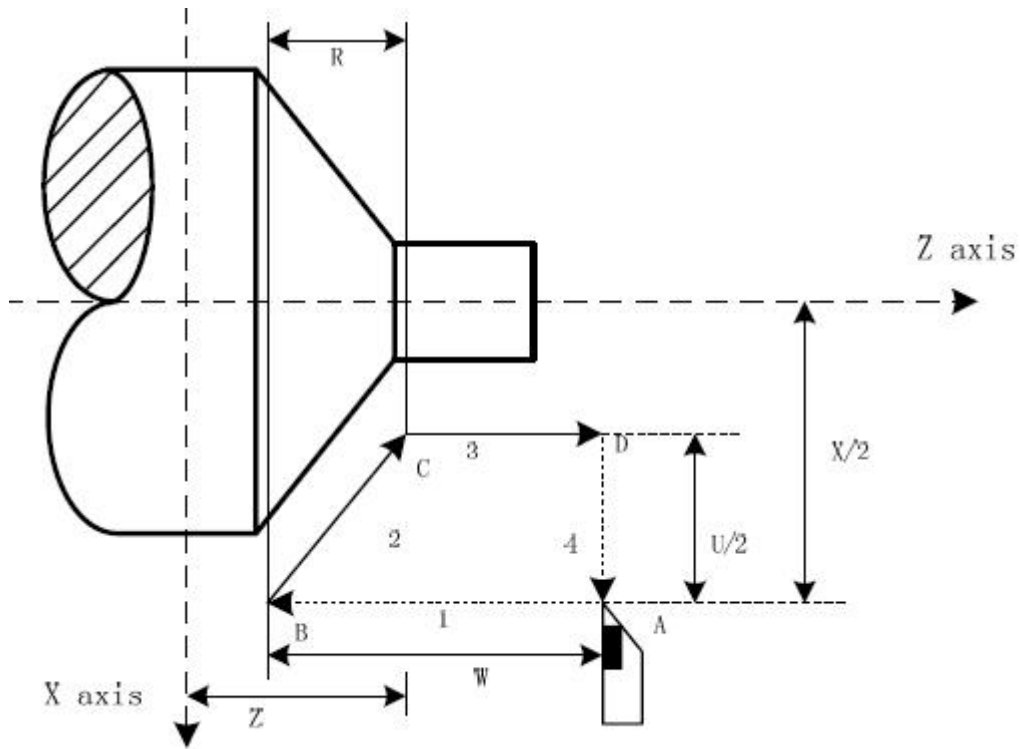


Fig. 3-22

Cutting path: Relative position between cutting end point and starting point with U, W is shown in Fig.3-23:

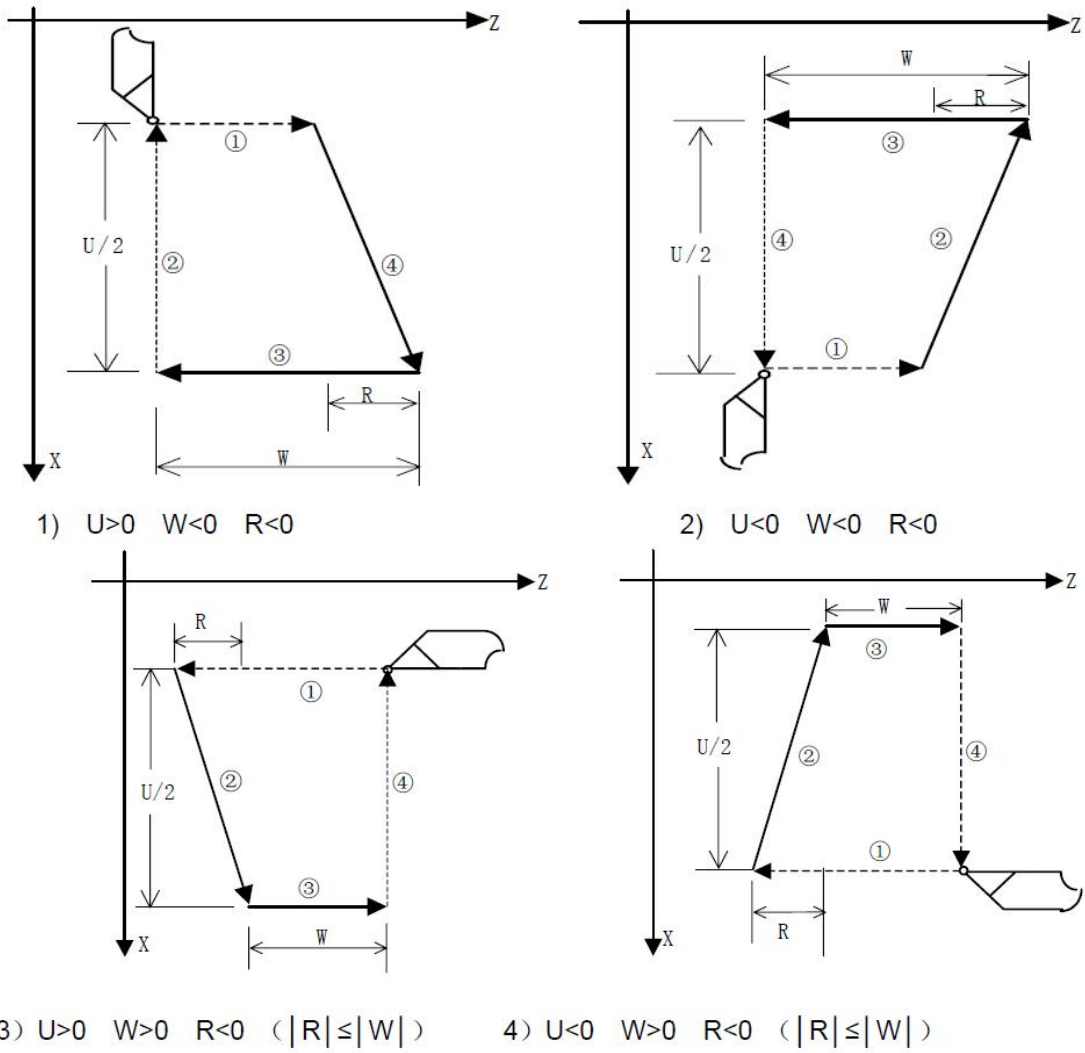


Fig. 3-23

Example: Fig. 3-24, rod $\Phi 125 \times 112$

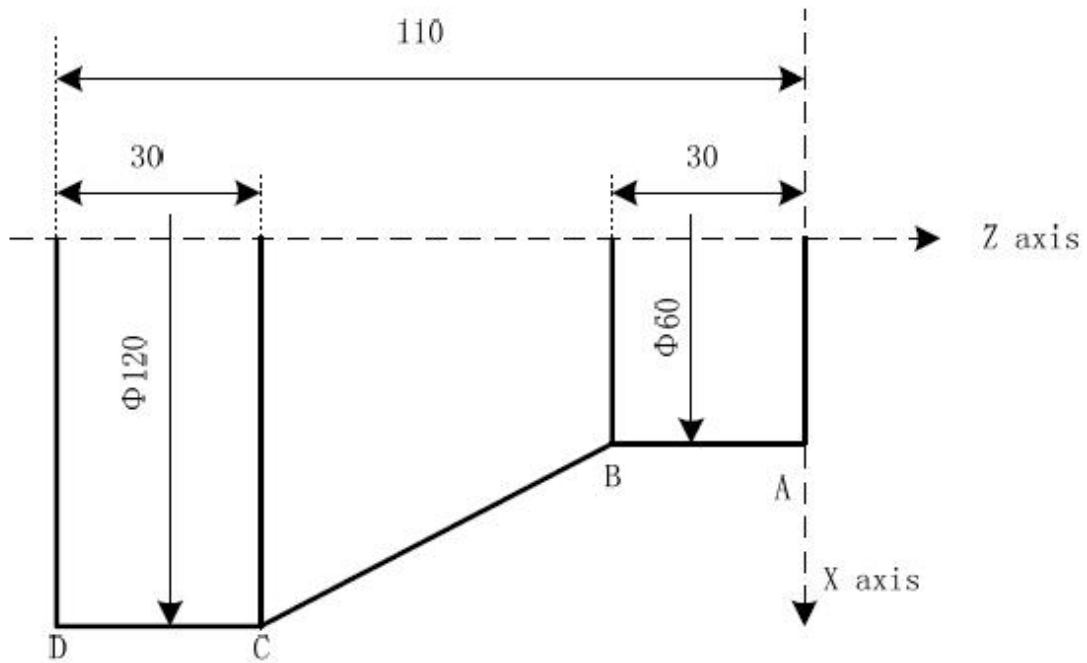


Fig. 3-24

Program: O0003;

```

G00 X130 Z5 M3 S1;
G94 X0 Z0 F200
X120 Z-110 F300;
G00 X120 Z0
G94 X108 Z-30 R-10
X96 R-20
X84 R-30
X72 R-40
X60 R-50;
M30;

```

End face cutting
(Outer cutting $\Phi 120$)

(C→B→A, cutting $\Phi 60$)

3.19.3 Caution of fixed cycle commands

1) After X(U) , Z(W) , R are executed in the canned cycle command, their command values are value if X(U) , Z(W) ,R are not redefined by executing a new canned cycle commands. The command values of X(U) ,Z(W) ,R are cleared if non-modal G command(00 Group) except for G04 or G00,G01, G02, G03, G32 is executed.

2) In MDI mode, the previous canned cycle can be executed by pressing the cycle start key after the canned cycle is completed.

3) One cycle cannot be executed repetitively in G90~G94 when the next block of G90~G94 is M, S, T command; the previous cycle is executed repetitively in G90~G94 when the next block is ended(EOB;).

Example: ...

N010 G90 X20.0 Z10.0 F400;
 N011 ; (execute G90 one time repetitively)
 ...

4) Pause or single block is executed in G90, G94, the single block stops after the tool moves end point of current path.

(Outer cutting $\Phi 120$)
 (C→B→A, cutting $\Phi 60$)

3.20 Multiple Cycle Commands

Multiple cycle commands of the system includes axial roughing cycle G71, radial roughing cycle G72, closed cutting cycle G73, finishing cycle G70, axial grooving multiple cycle G74, axial grooving multiple cycle G75 and multiple thread cutting cycle G76. When the system executes these commands, it automatically counts the cutting times and the cutting path according to the programmed path, travels of tool infeed and tool retraction, executes multiple machining cycle (tool infeed → cutting →retract tool→ tool infeed), automatically completes the roughing, finishing

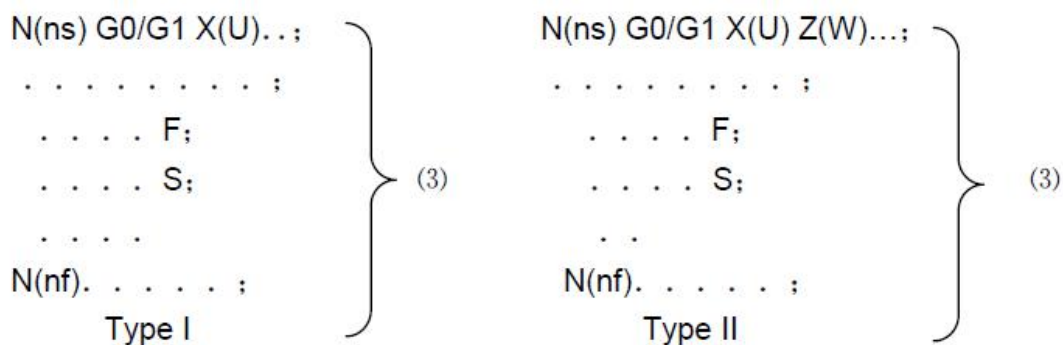
workpiece and the starting point and the end point of command are the same one.

3.20.1 Axial roughing cycle G71

G71 has two kinds of roughing cycle: type I and type II.

Command format: G71 U(Δd) R(e) F S T ; (1)

G71 P(ns) Q(nf) U(Δu) W(Δw) K0/1 J0/1; (2)



Command function: G71 is divided into three parts:

- (1) 1st blocks for defining the travels of tool infeed and retract tool, the cutting feedrate, the spindle speed and the tool function when roughing;
- (2) 2nd blocks for defining the block interval, finishing allowance;
- (3) 3rd blocks for some continuous finishing path, counting the roughing path without being executed actually when executing G71.

According to the finishing path, the finishing allowance, the path of tool infeed and tool retract, the system automatically counts the path of roughing, the tool cuts the workpiece in paralleling with Z,

and the roughing is completed by multiple executing the cutting cycle tool infeed→cutting→tool retraction. The starting point and the end point are the same one. The command is applied to the formed roughing of non-formed rod.

Relevant definitions:

Finishing path: The above-mentioned Part 3 of G71(ns~nf block)defines the finishing path, and

the starting point of finishing path (starting point of ns block) is the same as these of starting point and end point of G71, called A point; the first block of finishing path (ns block) is used for X rapid traversing or tool infeed, and the end point of finishing path is called B point; the end point of finishing path (end point of nf block) is called C point. The finishing path is $A \rightarrow B \rightarrow C$.

Roughing path: The finishing path is the one after offsetting the finishing allowance ($\Delta u, \Delta w$) and is the path contour formed by executing G71. A, B, C point of finishing path after offset corresponds separately to A', B', C' point of roughing path, and the final continuous cutting path of G71 is B' \rightarrow C' point.

Δd : It is each travel (unit: mm, radius value) of X tool infeed in roughing, its value:

0.001~99.999 (unit: mm/inch, radius value) without sign, and the direction of tool infeed is defined by move direction of ns block. The command value Δd is reserved after executing U(Δd) and the value of system parameter No.051 is rewritten to $\Delta d \times 1000$ (unit: 0.001 mm).

The value of system parameter No.051 is regarded as the travel of tool infeed when U(Δd) is not input.

e: It is travel (unit: mm/inch, radius value) of X tool retraction in roughing its value: 0.001~99.999 (unit: mm, radius value) without sign, and the direction of tool retraction is opposite to that of tool infeed, the command value e is reserved and the value of system parameter No.052 is rewritten to $e \times 1000$ (unit: 0.001 mm) after R(e) is executed. The value of system parameter No.052 is regarded as the travel of tool retraction when R(e) is not input.

ns: Block number of the first block of finishing path.

nf: Block number of the last block of finishing path.

Δu : X finishing allowance is $\pm 99999999 \times$ least input increment with sign symbol (diameter, unit: mm/inch, with sign). X coordinate offset of roughing path compared to finishing path, i.e. the different value of X absolute coordinates between A' and A. The system defaults $\Delta u = 0$ when U(Δu) is not input, i.e. there is no finishing allowance in X direction for roughing cycle.

Δw : Z finishing allowance is $\pm 99999999 \times$ least input increment with sign symbol (diameter, unit: mm/inch, with sign). the Z coordinate offset of roughing path compared to finishing path, i.e. the different value of Z absolute coordinate between A' and A. The system defaults $\Delta w = 0$ when W(Δw) is not input, i.e. there is no Z finishing allowance for roughing cycle.

K: When K is not input or is not 1, the system does not check the program monotonicity except that the Z value of starting point and end point of the arc or ellipse or parabola or the arc is more than 180 degree; K=1, the system checks the program monotonicity.

F: Feedrate; S: Spindle speed; T: Tool number, tool offset number.

M, S, T, F: They can be specified in the first G71 or the second ones or program ns~nf. M, S, T, F functions of M, S, T, F blocks are invalid in G71, and they are valid in G70 finishing blocks.

Type I:

1) Execution process: (Fig. 3-25)

- ① X rapidly traverses to A' from A point, X travel is Δu , and Z travel is Δw ;
- ② X moves from A' is Δd (tool infeed), ns block is for tool infeed at rapid traverse speed with G0, is for tool infeed at feedrate F with G71, and its direction of tool infeed is that of A \rightarrow B point;
- ③ Z executes the cutting feeds to the roughing path, and its direction is the same that of Z coordinate A \rightarrow B point;

- ④ X, Z execute the tool retraction e (45° straight line) at feedrate, the directions of tool retraction is opposite to that of too infeed;
- ⑤ Z rapidly retracts at rapid traverse speed to the position which is the same that of Z coordinate;
- ⑥ After executing X tool infeed ($\Delta d+e$) again, the end point of traversing tool is still on the middle point of straight line between A' and B'(the tool does not reach or exceed B'), and after executing the tool infeed ($\Delta d+e$) again, execute ③; after executing the tool infeed ($\Delta d+e$) again, the end point of tool traversing reaches B' point or exceeds the straight line between A'→B' point and X executes the tool infeed to B' point, and then the next step is executed;
- ⑦ Cutting feed from B' to C' point along the roughing path;
- ⑧ Rapid traverse to A from C' point and the program jumps to the next clock following nf block after G71 cycle is ended.

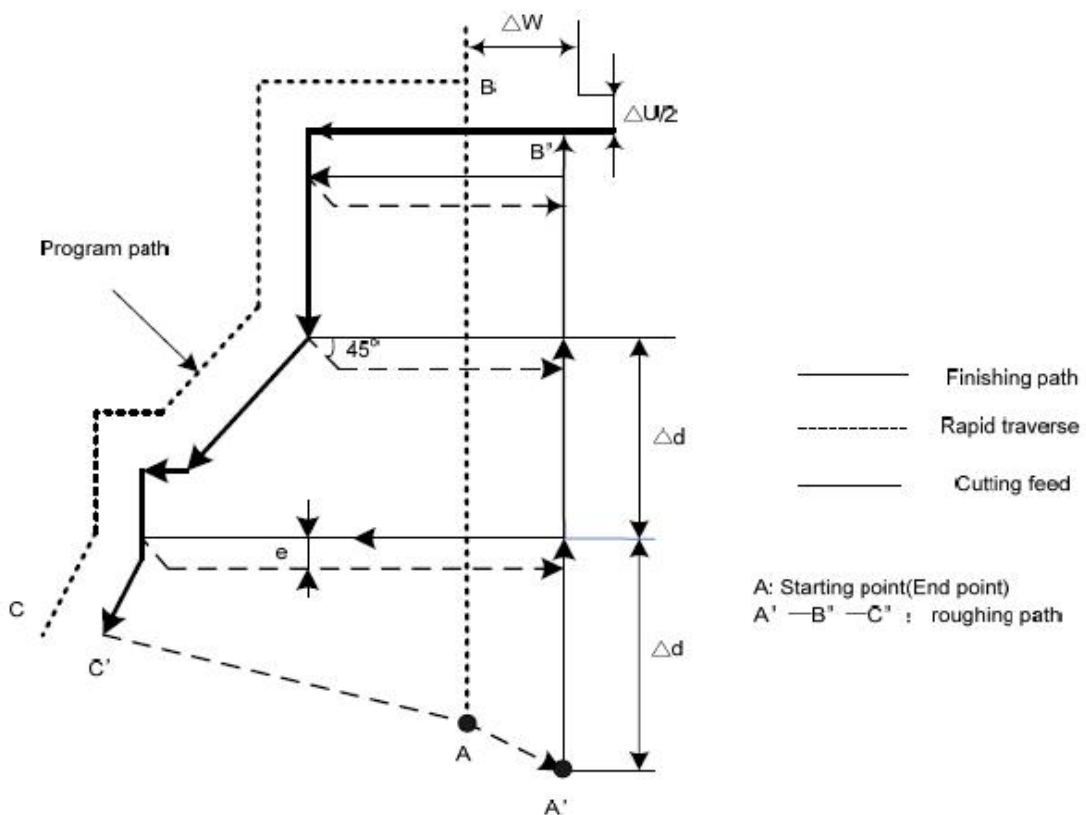


Fig. 3-25 G71 cycle path

2) Coordinate offset direction with finishing allowance:

Δu , Δw define the coordinate offset and cut-in direction in finishing, and their sign symbol are as follows Fig. 3-26: B→C for finishing path, B'→C' for roughing path and A is the tool start-up point.

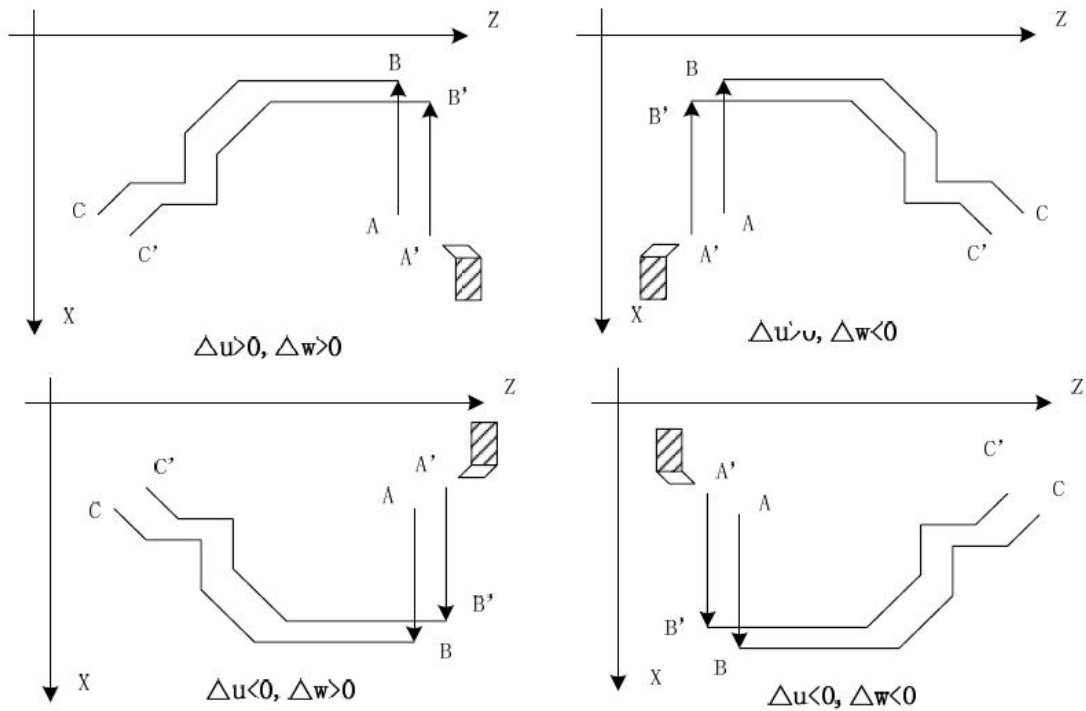


Fig.3-26

Type II:

The type II is different from the type I as follows:

1) Relative definition: more one parameter than the type I .

J: When J is not input or J is not 1, the system does not execute the run along the roughing contour;

J=1: the system executes the run along the roughing contour.

2) The system does not execute the monotonous increasing or the monotonous decreasing along X external contour, and the workpiece can be up to 10 grooves as follows:

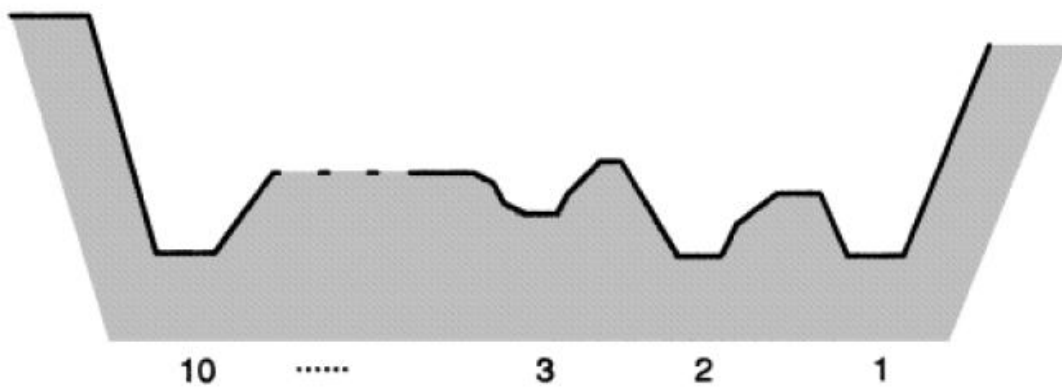


Fig. 3-26-1 (type II)

But, the Z external contour must be the monotonous increasing or the monotonous decreasing, and the following contour cannot be machined:

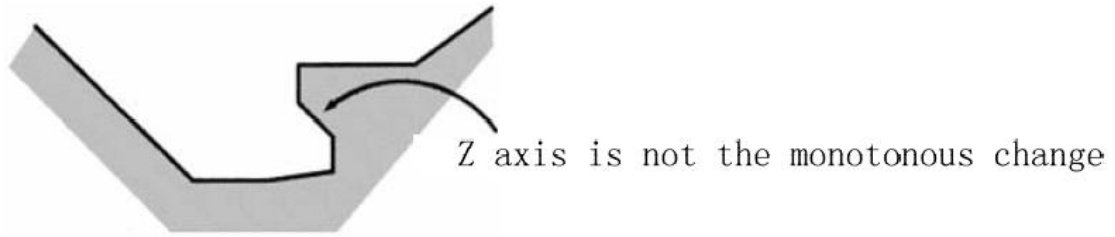


Fig. 3-26-2 (type II)

3) The first tool cutting need not the vertical: the machining can be executed when Z is the monotonous change shape as follows:

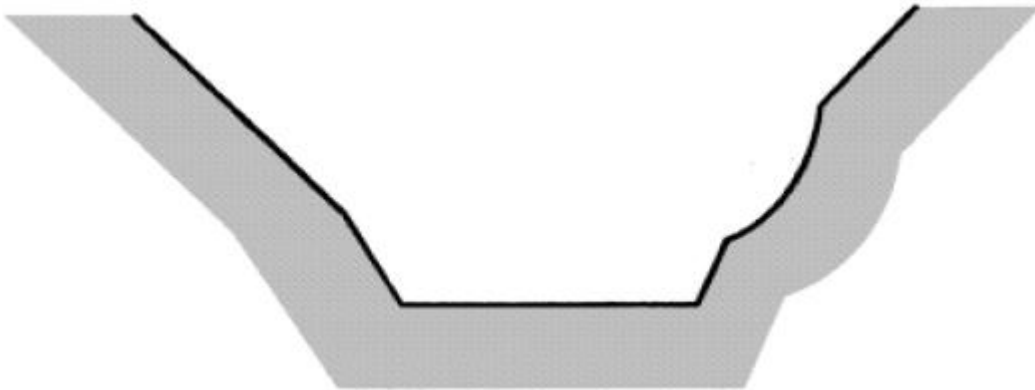


Fig. 3-26-3

4) After the turning, the system should execute the tool retraction, the retraction travel is specified by R (e) or No. 52 as follows:

θ is set by parameter

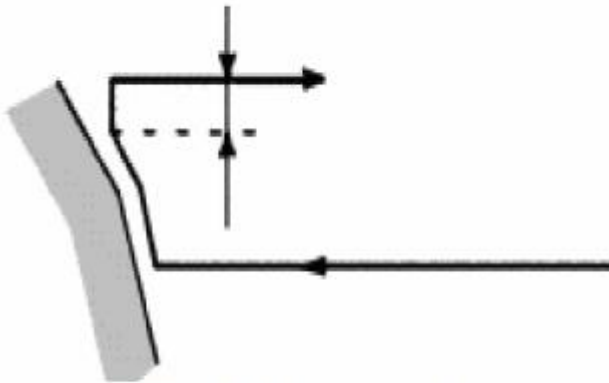


Fig. 3-26-4 (type II)

5) **Command execution process:** roughing path A->H

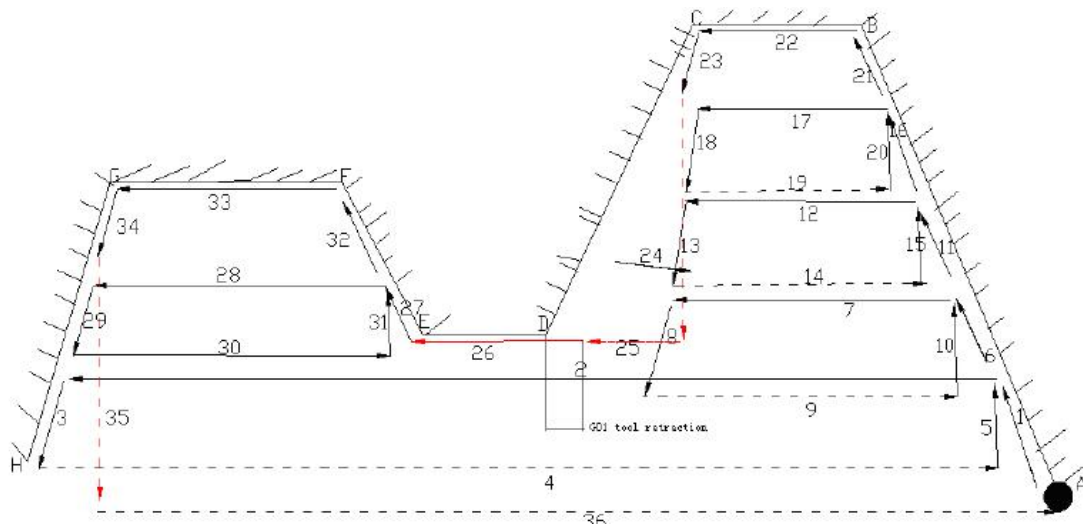


Fig. 3-26-5 (type II)

Notes:

- ns block is only G00, G01. When the workpiece is type II, the system must specify the two axes X(U) and Z(W), and W0 must be specified when Z does not move;
- For type II, only X finishing allowance can be specified; when Z finishing allowance is specified, the whole machining path offsets, and it can be specified to 0;
- For type II, after the current grooving is completed to execute the next, the tool approaches the workpiece(remark 25 and 26) in the remainder tool retraction distance at G1 speed; when the tool retraction is 0 or the remainder distance is less than the tool retraction, and the tool approaches the workpiece at G1 speed;
- Some workpiece without remarking the type I or the type II adapts the both;
- For the finishing path(ns~nf block), Z dimension must be monotonous change(always increasing or decreasing), X dimension in the type I must be monotonous change and does not need in the type II;
- ns ~ nf blocks in programming must be followed G71 blocks. If they are in front of G71 blocks,the system automatically searches and executes ns ~ nf blocks, and then executes the next program following nf block after they are executed, which causes the system executes ns ~ nf blocks repetitively;
- ns ~ nf blocks are used for counting the roughing path and the blocks are not executed when G71 is executed. F, S, T commands of ns ~ nf blocks are invalid when G71 is executed, at the moment, F, S, T commands of G71 blocks are valid. F, S, T of ns ~ nf blocks are valid when executing ns~nf to command G70 finishing cycle;
- In ns ~ nf blocks, there are only G commands: G00, G01, G02, G03, G04, G05, G6.2, G6.3,G7.2, G7.3, G96, G97, G98, G99, G40, G41, G42 and the system cannot call subprograms(M98/M99);
- G96, G97, G98, G99, G40, G41, G42 are invalid when G71 is executed, and are valid when G70 is executed;
- When G71 is executed, the system can stop the automatic run and manual traverse, but return to the position before manual traversing when G71 is executed again, otherwise, the following path will be wrong;
- When the system is executing the feed hold or single block, the program pauses after the system

has executed end point of current path;

- $\Delta d, \Delta u$ are specified by the same U and different with or without being specified P, Q

commands;

- G71 cannot be executed in MDI, otherwise, the system alarms;
- There are no the same block number in ns~nf when compound cycle commands are executed repetitively in one program;
- The tool retraction point should be high or low as possible to avoid crashing the workpiece.

Example: Fig. 3-73 (Type I)

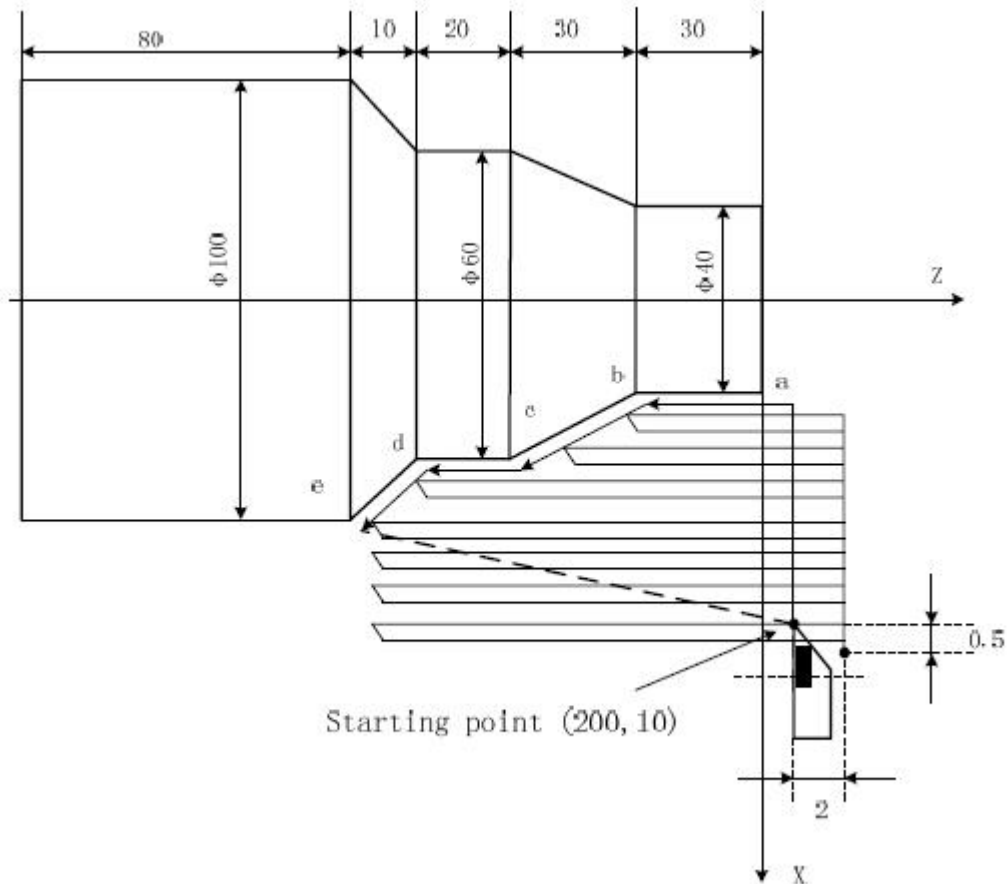


Fig. 3-27

Program: O0004;

G00 X200 Z10 M3 S800; (Spindle clockwise with 800 r/min)

G71 U2 R1 F200; (Cutting depth each time 4mm, tool retraction 2mm [in diameter])

G71 P80 Q120 U0.5 W0.2; (roughing a---e, machining allowance: X, 1mm;Z, 2mm)

N80 G00 X40 S1200;	(Positioning)	} a→b→c→d→e blocks for finishing path
G01 Z-30 F100 ;	(a→b)	
X60 W-30;	(b→c)	
W-20;	(c→d)	
N120 X100 W-10;	(d→e)	
G70 P80 Q120;	(a---e blocks for finishing path)	
M30;	(End of block)	

3.20.2 Radial roughing cycle G72

Command format: G72 W(Δd) R(e) F__ S__ T__ ; (1)

G72 P(ns) Q(nf) U(Δu) W(Δw); (2)

N__(ns) ;
 ;
 F ;
 S ;
 ;
 .
 N__(nf). ; (3)

Command function: G72 is divided into three parts:

- (1) 1st blocks for defining the travels of tool infeed and tool retraction, the cutting speed, the spindle speed and the tool function in roughing;
- (2) 2nd blocks for defining the block interval, finishing allowance;
- (3) 3rd blocks for some continuous finishing path, counting the roughing path without being executed actually when G72 is executed.

According to the finishing path, the finishing allowance, the path of tool infeed and retract tool, the system automatically counts the path of roughing, the tool cuts the workpiece in paralleling with Z, and the roughing is completed by multiple executing the cutting cycle tool infeed→cutting feed→tool retraction. The starting point and the end point of G72 are the same one. The command is applied to the formed roughing of non-formed rod.

Relevant definitions:

Finishing path: the above-mentioned Part (3) of G71(ns~nf block) defines the finishing path, and the starting point of finishing path (i.e. starting point of ns block) is the same these of starting point and end point of G72, called A point; the first block of finishing path(ns block) is used for Z rapid traversing or cutting feed, and the end point of finishing path is called to B point; the end point of finishing path(end point of nf block) is called to C point. The finishing path is A→B→C.

Roughing path: The finishing path is the one after offsetting the finishing allowance(Δu , Δw) and is the path contour formed by executing G72. A, B, C point of finishing path after

offset corresponds separately to A', B', C' point of roughing path, and the final continuous cutting path of G72 is B'→C' point.

Δd : it is Z cutting in roughing, its value: 0.001~99.999(unit: mm/inch) without sign symbol, and the direction of tool infeed is determined by ns block traverse direction. the specified value Δd is reserved and the data value is switched to the corresponding value to save to No.051 after W(Δd) is executed. The value of system parameter No.051 is regarded as the tool infeed clearance when R(e) is not input.

e: it is Z tool retraction clearance in roughing, its value: 0~99.999(unit: mm) without sign symbol, and the direction of tool retraction is opposite to that of tool infeed, the specified value e is reserved and the data value is switched to the corresponding value to save to No.052 after R(e) is executed. The value of system parameter No.052 is regarded as the tool retraction clearance when R(e) is not input.

ns: Block number of the first block of finishing path.

nf: Block number of the last block of finishing path.

Δu : it is X finishing allowance in roughing, its range: $\pm 99999999 \times$ least input increment(X coordinate offset of roughing contour corresponding to the finishing path, i.e. X absolute coordinate difference between A' and A.(diameter, unit: mm/inch, with sign symbol).

Δw : it is Z finishing allowance in roughing, its range: $\pm 99999999 \times$ least input increment(Z coordinate offset of roughing contour corresponding to the finishing path, i.e. Z absolute coordinate difference between A' and A.(diameter, unit: mm/inch, with sign symbol).

F: Cutting feedrate; S: Spindle speed; T: Tool number, tool offset number.

M, S, T, F: They can be specified in the first G72 or the second ones or program ns~nf. M, S, T, F functions of M, S, T, F blocks are invalid in G72, and they are valid in G70 finishing blocks.

Execution process: Fig. 3-28

- ① X rapidly traverses to A' from A point, X travel is Δu , and Z travel is Δw ;
- ② X moves from A' is Δd (tool infeed), ns block is for tool infeed at rapid traverse speed with G0, is for tool infeed at G72 feedrate F in G1, and its direction of tool infeed is that of A→B point;
- ③ X executes the cutting feeds to the roughing path, and its direction is the same that of X coordinate B→C point;
- ④ X, Z execute the tool retraction e (45° straight line) at feedrate, the directions of tool retraction is opposite to that of tool infeed ;
- ⑤ X rapidly retracts at rapid traverse speed to the position which is the same that of Z coordinate;
- ⑥ After Z tool infeed ($\Delta d+e$) again is executed, the end point of traversing tool is still on the middle point of straight line between A' and B' (the tool does not reach or exceed B'), and after Z executes the tool infeed ($\Delta d+e$) again, ③ is executed; after the tool infeed ($\Delta d+e$) is executed again, the end point of tool traversing reaches B' point or exceeds the straight line between A'→B' point and Z executes the tool infeed to B' point, and then the next step is executed;
- ⑦ Cutting feed from B' to C' point along the roughing path;
- ⑧ Rapidly traverse to A from C' point and the program jumps to the next block following nf block after G71 cycle is completed.

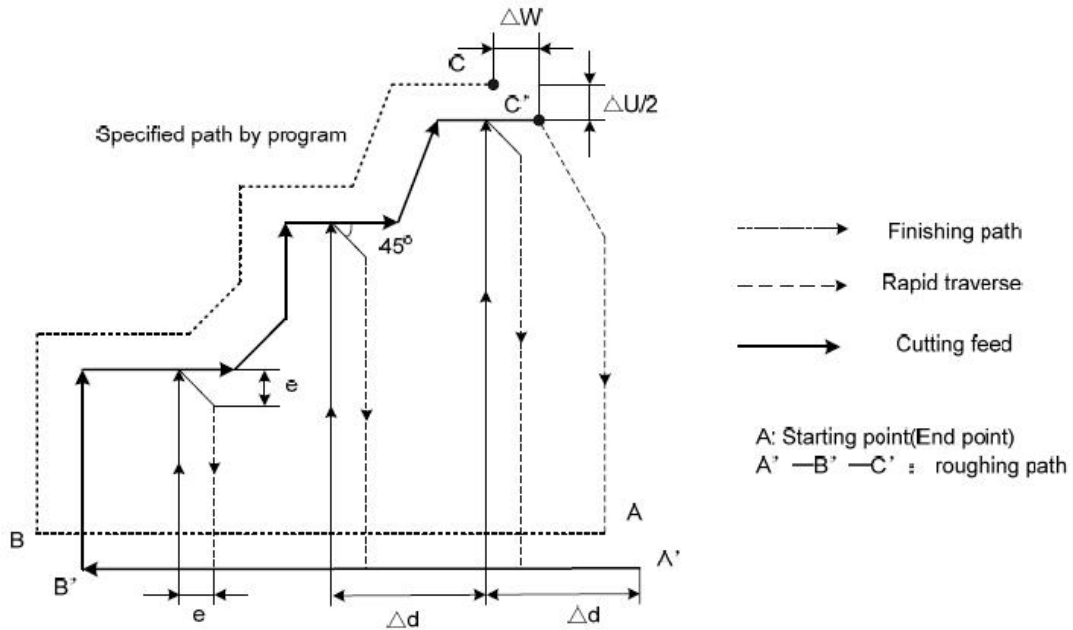


Fig. 3-28

Command specifications:

- ns ~ nf blocks in programming must be followed G72 blocks. If they are in the front of G72 blocks, the system automatically searches and executes ns ~ nf blocks, and then executes the next program following nf block after they are executed, which causes the system executes ns ~ nf blocks repetitively;
- ns ~ nf blocks are used for counting the roughing path and the blocks are not executed when G72 is executed. F, S, T commands of ns ~ nf blocks are invalid when G72 is executed, at the moment, F, S, T commands of G72 blocks are valid. F, S, T of ns ~ nf blocks are valid when executing ns ~ nf to command G70 finishing cycle;
- There are G00, G01 without the word X(U) in ns block, otherwise the system alarms;
- The dimensions in X, Z direction must be changed monotonously (always increasing or reducing) for the finishing path;
- In ns ~ nf blocks, there are only G commands: G01, G02, G03, G04, G05, G6.2, G6.3, G7.2, G7.3, G96, G97, G98, G99, G40, G41, G42 and the system cannot call subprograms(M98/M99);
- G96, G97, G98, G99, G40, G41, G42 are invalid when G72 is executed, and are valid when G70 is done;
- When G72 is executed, the system can stop the automatic run and manual traverse, but return to the position before manual traversing when G72 is executed again, otherwise, the following path will be wrong;
- When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path;
- Δd , Δu are specified by the same U and different with or without being specified P, Q

commands;

- There are no the same block number in ns~nf when compound cycle commands are executed repetitively in one program;
- G72 cannot be executed in MDI, otherwise, the system alarms;
- The tool retraction point should be high or low as possible to avoid crashing the workpiece.

Coordinate offset direction with finishing allowance:

Δu , Δw define the coordinate offset and its direction of cut-in in finishing, and their sign symbol are as follows Fig. 3-29: B→C for finishing path, B'→C' for roughing path and A is the tool start-up point.

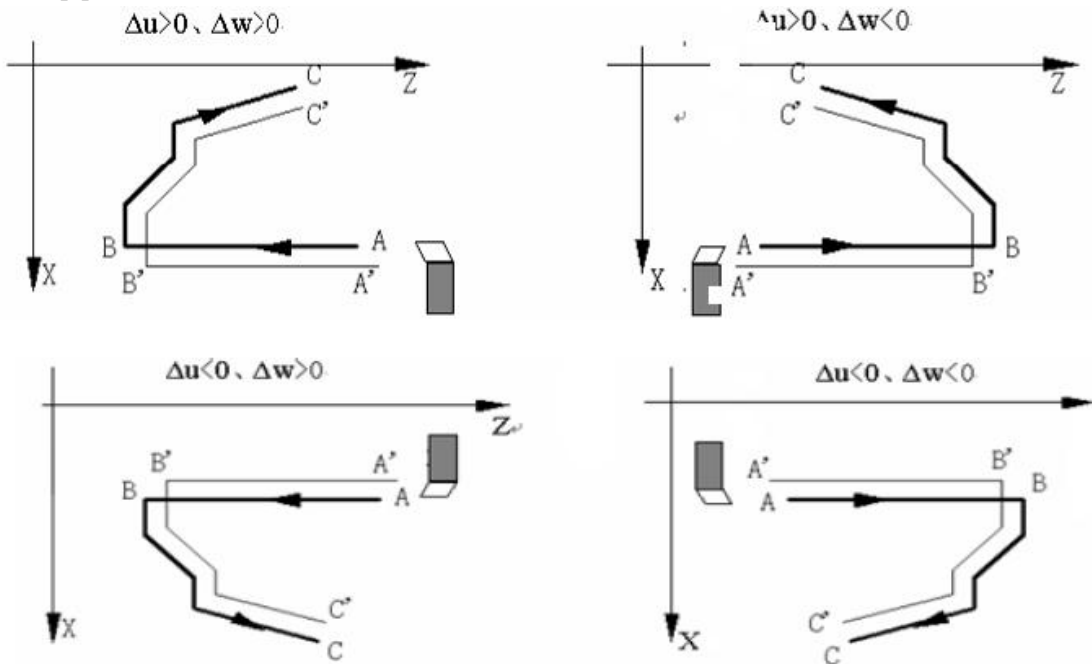


Fig.3-29

Example: Fig. 3-30

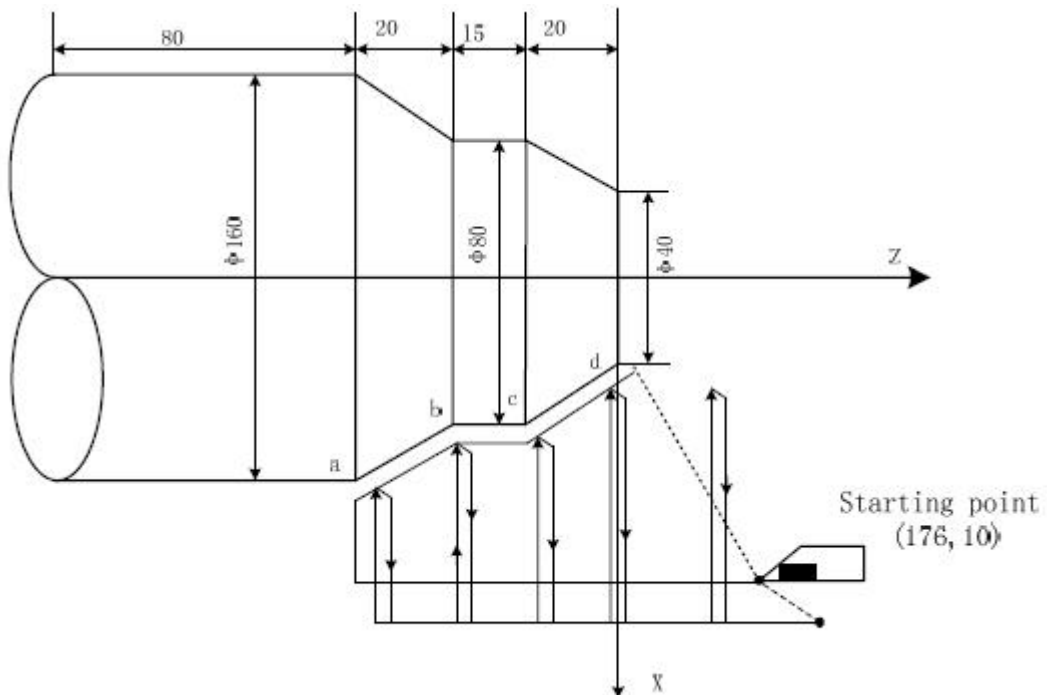


Fig.3-30

Program:

O0005;

G00 X176 Z10 M03 S500 (Change No.2 tool and execute its compensation, spindle CW rotation with 500 r/min)

G72 W2.0 R0.5 F300; (Tool infeed 2mm, tool retraction 0.5mm)

G72 P10 Q20 U0.2 W0.1; (Roughing a--d,X roughing allowance 0.2mm and Z 0.1mm)

N10 G00 Z-55 S800 ; (Rapid traverse)

G01 X160 F120; (Infeed to a point)

X80 W20; (Machining a—b) Blocks for finishing path

W15; (Machining b—c)

N20 X40 W20 ; (Machining c—d)

G70 P050 Q090 M30; (Finishing a—d)

3.20.3 Closed cutting cycle G73

Command format: G73 U(Δi) W (Δk) R (d) F__ S__ T__ ; (1)

G73 P(ns) Q(nf) U(Δu) W(Δw) ; (2)

N__(ns) ;

. ;

. . . . F;

. . . . S;

. ;

.

N__(nf) ;

(3)

Command functions: G73 is divided into three parts:

- (1) Blocks for defining the travels of tool infeed and tool retraction, the cutting speed, the spindle speed and the tool function when roughing;
- (2) Blocks for defining the block interval, finishing allowance;
- (3) Blocks for some continuous finishing path, counting the roughing path without being executed actually when executing G73.

According to the finishing allowance, the travel of tool retraction and the cutting times, the system automatically counts the travel of roughing offset, the travel of each tool infeed and the path of roughing, the path of each cutting is the offset travel of finishing path, the cutting path approaches gradually the finishing one, and last cutting path is the finishing one according to the finishing allowance. The starting point and end point of G73 are the same one, and G73 is applied to roughing for the formed rod. G73 is non-modal and its path is shown in Fig.3-31.

Relevant definitions:

Finishing path: The above-mentioned Part 3 of G73 (ns~nf block) defines the finishing path, and the starting point of finishing path (start point of ns block) is the same these of starting point and end point of G73, called A point; the end point of the first

block of finishing path(ns block)is called B point; the end point of finishing path(end point of nf block) is called C point. The finishing path is A→B→C.

Roughing path: It is one group of offset path of finishing one, and the roughing path times are the same that of cutting. After the coordinates offset, A, B, C of finishing path separately corresponds to A_n, B_n, C_n of roughing path(n is the cutting times, the first cutting path is A_1, B_1, C_1 and the last one is A_d, B_d, C_d). The coordinates offset value of the first cutting compared to finishing path is $(\Delta i \times 2 + \Delta u, \Delta w + \Delta k)$ (diameter programming), the coordinates offset value of the last cutting compared to finishing path is $(\Delta u, \Delta w)$, the coordinates offset value of each cutting compared to the previous one is as follows:

Δi : It is X tool retraction clearance in roughing, and its range is $\pm 99999999 \times$ least input increment (radius, unit: mm/inch, with sign symbol), Δi is equal to X coordinate offset value (radius value) of A1 point compared to Ad point. The X total cutting travel(radius value) is equal to $|\Delta i|$ in roughing, and X cutting direction is opposite to the sign of Δi : $\Delta i > 0$, the system executes X negative cutting in roughing. It is reserved after Δi specified value is executed and the data is switched to the corresponding value to save to NO.053. The No.053 value is regarded as X tool retraction clearance in roughing when U(Δi) is not input.

Δk : It is Z tool retraction clearance in roughing, and its range is $\pm 99999999 \times$ least input increment (radius, unit: mm/inch, with sign symbol), Δk is equal to Z coordinate offset value (radius value) of A1 point compared to Ad point. Z total cutting travel(radius value) is equal to $|\Delta k|$ in roughing, and Z cutting direction is opposite to the sign of Δk : $\Delta i > 0$, the system executes Z negative cutting in roughing. It is reserved after Δk specified value is executed and the data is switched to the corresponding value to save to NO.054. The No.054 value is regarded as Z tool retraction clearance in roughing when W(Δk) is not input.

d: It is the cutting times 1~9999 (unit: times). R5 means the closed cutting cycle is completed by 5 times cutting. R (d) is reserved after it is executed and NO.055 value is rewritten to d (unit: times). No.055 value is regarded as the cutting times when R(d) is not input. When the cutting times is 1, the system completes the closed cutting cycle based on 2 times cutting.

ns: Block number of the first block of finishing path.

nf: Block number of the last block of finishing path.

Δu : It is X finishing allowance and its range is $\pm 99999999 \times$ least input increment (diameter, unit: mm/inch, with sign symbol) and is the X coordinate offset of roughing path compared to finishing path, i.e. the different value of X absolute coordinates of A1 compared to A. $\Delta u > 0$, it is the offset of the last X positive roughing path compared to finishing path. The system defaults $\Delta u = 0$ when U(Δu) is not input, i.e. there is no X finishing allowance for roughing cycle.

Δw : It is Z finishing allowance and its range is $\pm 99999999 \times$ least input increment (diameter, unit: mm/inch, with sign symbol) and is the X coordinate offset of roughing path compared to finishing path, i.e. the different value of Z absolute coordinates of A1 compared to A. $\Delta w > 0$, it is the offset of the last X positive roughing path compared to finishing path. The system defaults $\Delta w = 0$ when W(Δw) is not input, i.e. there is no Z

finishing allowance for roughing cycle.

F: Feedrate; **S:** Spindle speed; **T:** Tool number, tool offset number.

M, S, T, F: They can be specified in the first G73 or the second ones or program ns~nf. M, S,T, F functions of M, S, T, F blocks are invalid in G73, and they are valid in G70 finishing blocks.

Execution process: (Fig. 3-31)

① A→A1: Rapid traverse;

② First roughing A1→B1→C1 :

A1→B1: Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;

B1→C1: Cutting feed.

③ C1→A2: Rapid traverse.

④ Second roughing A2→B2→C2 :

A2→B2: Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;B2→C2: Cutting feed.

⑤ C2→A3: Rapid traverse:

.....

No. n times roughing, An→Bn→Cn :

An→Bn: ns Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;

Bn→Cn: Cutting feed.

Cn→An+1: Rapid traverse;

.....

Last roughing, Ad→Bd→Cd :

Ad→Bd: Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;

Bd→Cd: Cutting feed.

Cd→A: Rapid traverse to starting point;

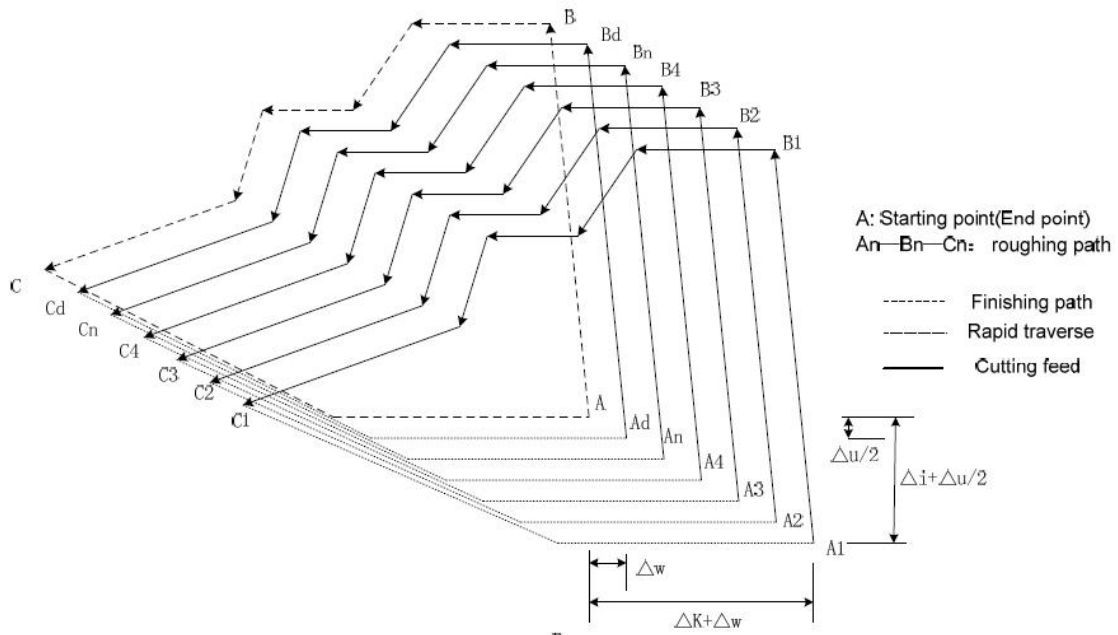


Fig. 3-31 G73 path

Command specifications:

- ns ~ nf blocks in programming must be followed G73 blocks. If they are in the front of G73 blocks, the system automatically searches and executes ns ~ nf blocks, and then executes the next program following nf block after they are executed, which causes the system executes ns ~ nf blocks repetitively.
- ns ~ nf blocks are used for counting the roughing path and the blocks are not executed when G73 is executed. F, S, T commands of ns ~ nf blocks are invalid when G71 is executed, at the moment, F, S, T commands of G73 blocks are valid. F, S, T of ns ~ nf blocks are valid when executing ns ~ nf to command G70 finishing cycle.
- There are only G00, G01 in ns block.
- In ns ~ nf blocks, there are only G commands: G00, G01, G02, G03, G04, G05, G6.2, G6.3, G7.2, G7.3, G96, G97, G98, G99, G40, G41, G42 and the system cannot call subprograms(M98/M99)
- G96, G97, G98, G99, G40, G41, G42 are invalid when G73 is executed, and are valid when G70 is executed.
- When G73 is executed, the system can stop the automatic run and manual traverse, but return to the position before manual traversing when G73 is executed again, otherwise, the following path will be wrong.
- When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path.

- Δi , Δu are specified by the same U and Δk , Δw are specified by the same U, and they are

different with or without being specified P,Q commands.

- G73 cannot be executed in MDI, otherwise, the system alarms.
- There are no the same block number in ns~nf when compound cycle commands are executed repetitively in one program.
- The tool retraction point should be high or low as possible to avoid crashing the workpiece.

Coordinate offset direction with finishing allowance:

Δi , Δk define the coordinates offset and its direction of roughing; Δu , Δw define the coordinate offset and the cut-in direction in finishing, and their sign symbols are as follows Fig. 3-32: A is tool start-up point, B→C for workpiece contour, B'→C' for roughing contour and B''→C'' for finishing path.

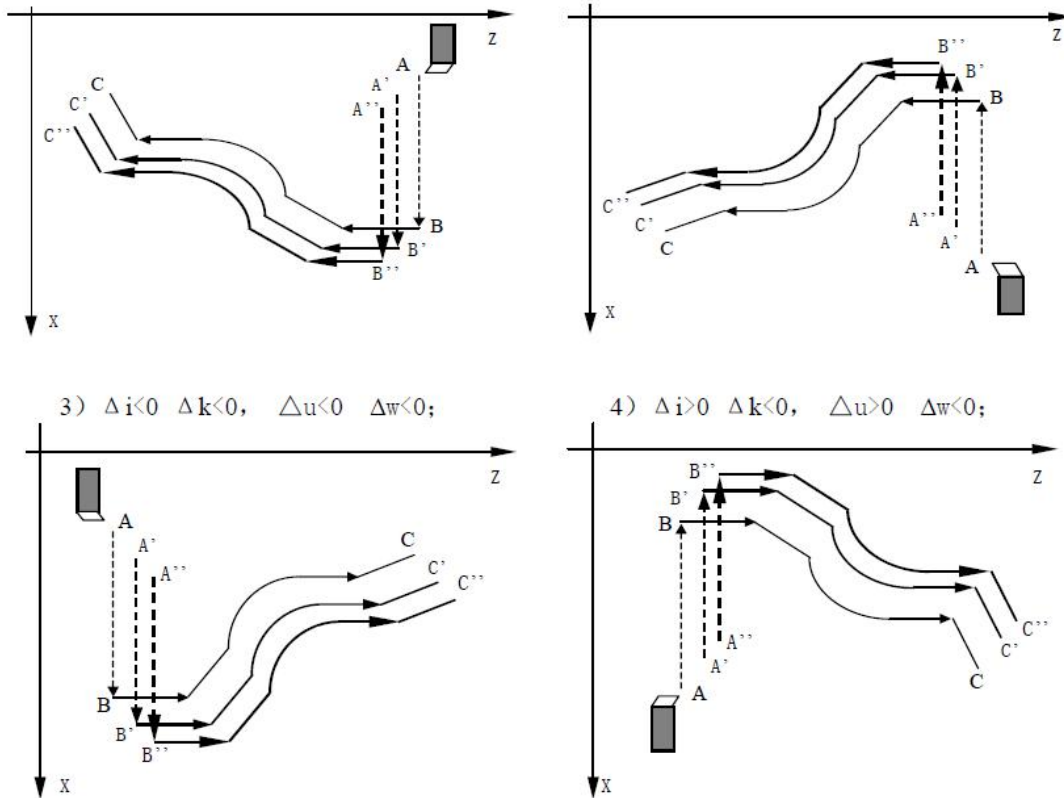


Fig-3-32

Example: Fig. 3-33

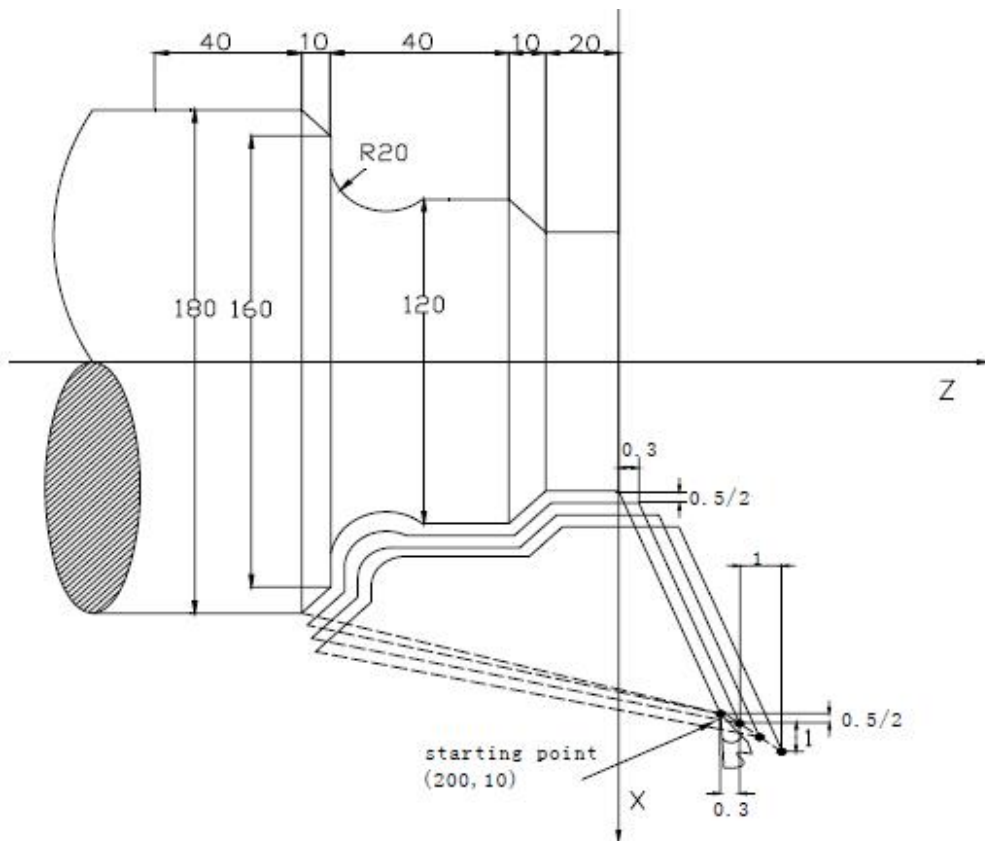


Fig.3-34

```

Program: O0006;
G99 G00 X200 Z10 M03 S500;      (Specify feedrate per rev and position starting point and
                                start spindle)
G73 U1.0 W1.0 R3 ;              (X tool retraction with 2mm, Z 1mm)
G73 P14 Q19 U0.5 W0.3 F0.3 ;    (X roughing with 0.5 allowance and Z 0.3mm)
N14 G00 X80 W-40 ;
G01 W-20 F0.15 S600 ;
X120 W-10 ;
W-20 ;
G02 X160 W-20 R20 ;
N19 G01 X180 W-10 ;
G70 P14 Q19 M30;                (Finishing)
    
```

} Blocks for finishing

3.20.4 Finishing cycle G70

Command format: G70 P(ns) Q(nf) ;

Command function: The tool executes the finishing of workpiece from starting point along with the finishing path defined by ns~nf blocks. After executing G71, G72 or G73 to roughing, execute G70 to finishing and single cutting of finishing allowance is completed. The tool returns to starting point and execute the next block following G70 block after G70 cycle is completed.

ns: Block number of the first block of finishing path.

nf: Block number of the last block of finishing path.

G70 path is defined by programmed one of ns~nf blocks. Relationships of relative position of ns, nf block in G70~G73 blocks are as follows:

```

G71/G72/G73 .....;
N__(ns) . . . . .
. . . . .
. F
. S
.
.
N__(nf).....
. . .
G70 P(ns) Q(nf);
    
```

} Blocks for finishing path

Command specifications:

- ns ~ nf blocks in programming must be followed G70 blocks. If they are in the front of G71 blocks, the system automatically searches and executes ns~nf blocks, and then executes the next program following nf block after they are executed, which causes the system executes ns ~ nf blocks repetitively.
- F, S, T in ns~nf blocks are valid when executing ns~nf to command G70 finishing cycle.

- G96, G97, G98, G99, G40, G41, G42 are valid in G70;
- When G70 is executed, the system can stop the automatic run and manual traverse, but return to the position before manual traversing when G70 is executed again, otherwise, the following path will be wrong.
- When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path.
- G70 cannot be executed in MDI, otherwise, the system alarms.
- There are no the same block number in ns~nf when compound cycle commands are executed repetitively in one program.
- The tool retraction point should be high or low as possible to avoid crashing the workpiece.

3.20.5 Axial grooving multiple cycle G74

Command format: G74 R(e) ;

G74 X(U) Z(W) P(Δ i) Q(Δ k) R(Δ d) F ;

Command function: Axial (X axis) tool infeed cycle compounds radial discontinuous cutting cycle: Tool infeeds from starting point in radial direction(Z), retracts, infeeds again, and again and again, and last tool retracts in axial direction, and retracts to the Z position in radial direction, which is called one radial cutting cycle; tool infeeds in axial direction and execute the next radial cutting cycle; cut to end point of cutting, and then return to starting point (starting point and end point are the same one in G74), which is called one radial grooving compound cycle. Directions of axial tool infeed and radial tool infeed are defined by relative position between end point X(U) Z(W) and starting point of cutting. G75 is used for machining radial loop groove or column surface by radial discontinuously cutting, breaking stock and stock removal.

Relevant definitions:

Starting point of axial cutting cycle: starting position of axial tool infeed for each axial cutting cycle, defining with $A_n(n=1,2,3,\dots)$, Z coordinate of A_n is the same that of starting point A, the different value of X coordinate between A_n and A_{n-1} is Δi . The starting point A_1 of the first axial cutting cycle is the same as the starting point A, and the X coordinate of starting point (A_f) of the last axial cutting cycle is the same that of cutting end point.

End point of axial tool infeed: starting position of axial tool infeed for each axial cutting cycle, defining with $B_n(n=1,2,3,\dots)$, Z coordinate of B_n is the same that of cutting end point, X coordinate of B_n is the same that of A_n , and the end point (B_f) of the last axial tool infeed is the same that of cutting end point.

End point of radius tool retraction: end position of radius tool infeed (travel of tool infeed is Δd) after each axial cutting cycle reaches the end point of axial tool infeed, defining with $C_n(n=1,2,3,\dots)$, Z coordinate of C_n is the same that of cutting end point, and the different value of X coordinate between C_n and A_n is Δd ;

End point of axial cutting cycle: end position of axial tool retraction from the end point of radius tool retraction, defining with $D_n(n=1, 2, 3,\dots)$, Z coordinate of D_n is the same that of starting point, X coordinate of D_n is the same that of C_n (the different value of X coordinate between it and A_n is Δd);

Cutting end point: it is defined by X(U) Z(W), and is defined with B_f of the last axial tool infeed. R(e) : it is the tool retraction clearance after each axial(Z) tool infeed, and its range is 0~99.999(unit: mm) without sign symbols. The specified value is reserved validly after R(e) is executed and the data is switched to the corresponding value to save to NO.056. The NO.056

value is regarded as the tool retraction clearance when R(e) is not input.

X: X absolute coordinate value of cutting end point B_f(unit: mm).

U: Different value of X absolute coordinate between cutting end point B_f and starting point.

Z: Z absolute coordinate value of cutting end point B_f(unit: mm).

W: Different value of Z absolute coordinates between cutting end point B_f and starting point.

P(Δ i) : radial(X) cutting for each axial cutting cycle , range: $0 < \Delta i \leq 9999999 \times$ least input increment (unit: least input increment, diameter value, without sign symbol).

Q(Δ k): radial(Z) cutting for each axial cutting cycle, range: $0 < \Delta k \leq 9999999 \times$ least input increment (unit: least input increment, diameter value, without sign symbol).

R(Δ d) : radial (X) tool retraction after cutting to end point of axial cutting, range: $0 \sim 99999999 \times$ least input increment (unit: mm/inch, diameter value, without sign symbol).The radial (X) tool retraction clearance is 0 when the system defaults the axial cutting end point. The system defaults the tool retraction is executed in positive direction when X(U) and P(Δ i) are omitted.

Execution process: (Fig. 3-34)

① Axial (Z) cutting feed Δk from the starting point of axial cutting cycle, feed in Z negative

direction when the coordinates of cutting end point is less than that of starting point in Z direction, otherwise, feed in Z positive direction;

② Axial (Z) rapid tool retraction e and its direction is opposite to the feed direction of ①;

③ X executes the cutting feed ($\Delta k + e$) again, the end point of cutting feed is still in it between starting point A_n of axial cutting cycle and end point of axial tool infeed, Z executes the cutting feed ($\Delta k + e$) again and execute ②; after Z executing the cutting feed ($\Delta k + e$) again, the end point of cutting feed is on B_n or is not on it between A_n and B_n cutting feed to B_n in Z direction and then execute ④;

④ Radial(X) rapid tool retraction Δd to C_n, when X coordinate of B_f(cutting end point) is less

than that of A (starting point), retract tool in X positive, otherwise, retract tool in X negative direction;

⑤ Axial(Z axial) rapid retract tool to D_n, No. n axial cutting cycle is completed. If the current axial cutting cycle is not the last one, execute ⑥ ; if it is the previous one before the last axial cutting cycle, execute ⑦;

⑥ Radial(X axial)rapid tool infeed, and it direction is opposite to ④ retract tool. If the end point of tool infeed is still on it between A and A_f(starting point of last axial cutting cycle)

after X executes the tool infeed ($\Delta d + \Delta i$) , i.e. D_n→A_{n+1} and then execute ① (start the next

axial cutting cycle); if X end point of tool infeed is not on it between D_n and A_f after tool

infeed ($\Delta d + \Delta i$), rapidly traverse to A_f and execute ① to start the first axial cutting cycle;

⑦ X rapidly traverse to return to A, and G74 is completed.

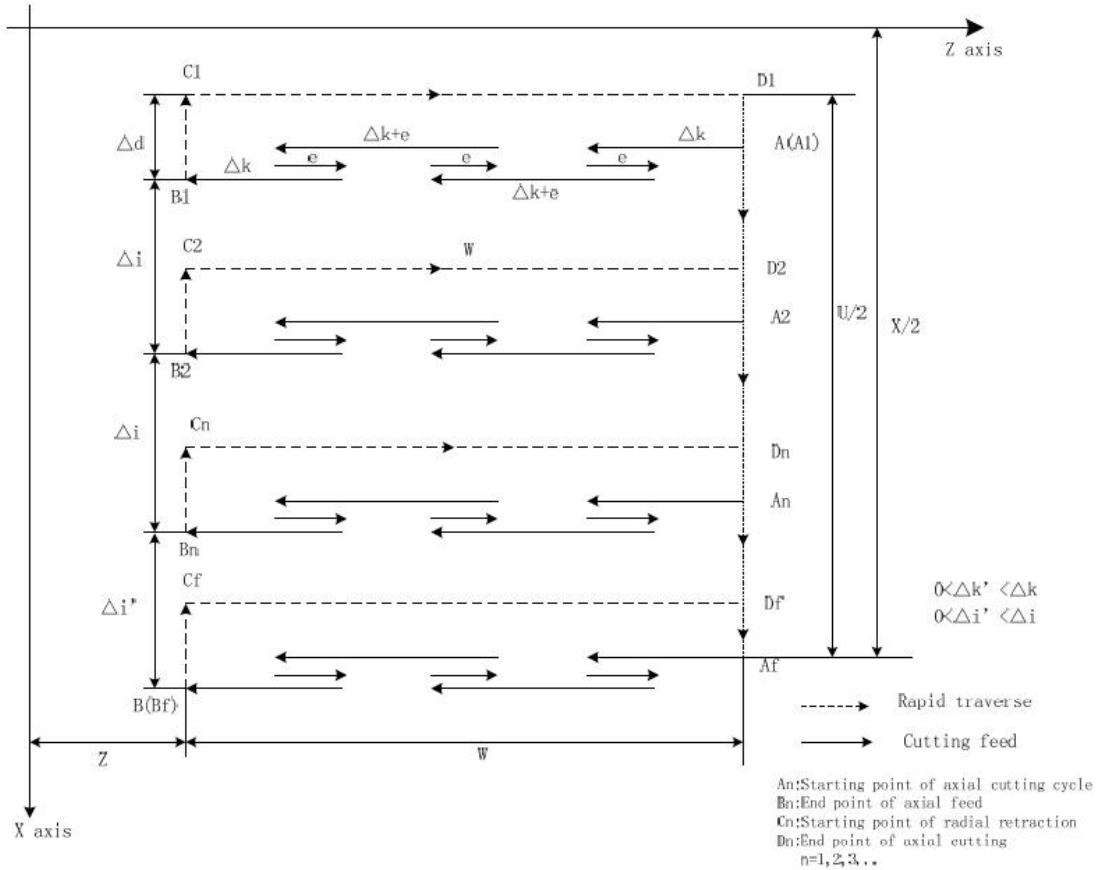


Fig. 3-34 G74 path

Command specifications:

- The cycle movement is executed by Z(W) and P(Δk) blocks of G74, and the movement is not executed if only “G74 R(e);” block is executed;
- Δd and e are specified by the same address and whether there are Z(W) and P(Δk) word or not in blocks to distinguish them;
- The tool can stop in Auto mode and traverse in Manual mode when G74 is executed, but the tool must return to the position before executing in Manual mode when G74 is executed again, otherwise the following path will be wrong.
- When the single block is running, programs dwell after each axial cutting cycle is completed.
- R(Δd) must be omitted in blind hole cutting, and so there is no distance of tool retraction when the tool cuts to axial end point of cutting.

Example: Fig. 3-35

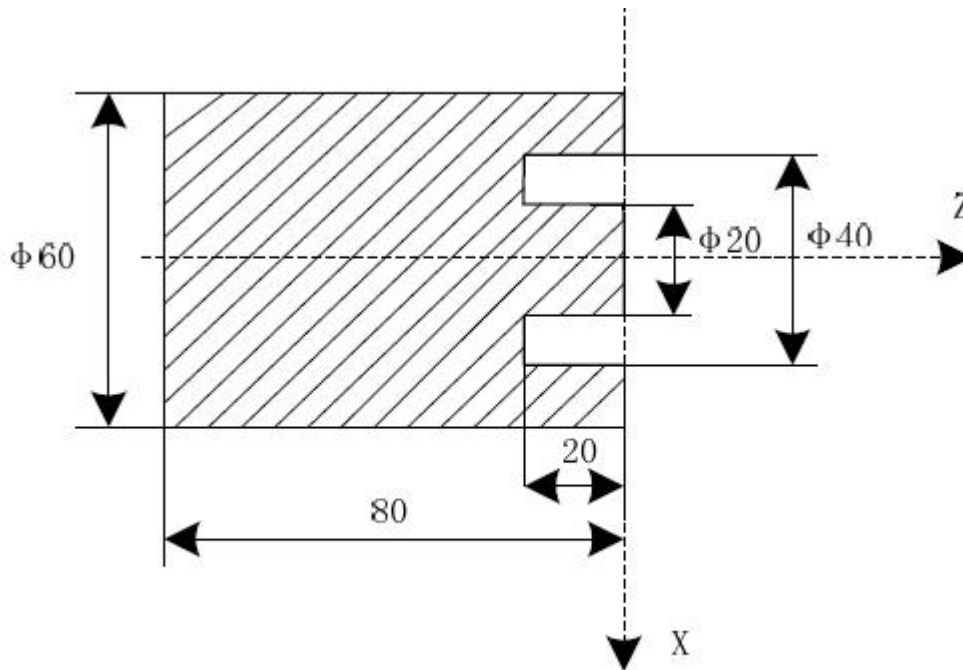


Fig.3-35

Program (suppose that the grooving tool width is 4mm, system least increment is 0.001mm):

O0007;

G0 X40 Z5 M3 S500; (Start spindle and position to starting point of machining)

G74 R0.5 ; (Machining cycle)

G74 X20 Z60 P3000 Q5000 F50; (Z tool infeed 5mm and tool retraction 0.5mm each time; rapid return to starting point (Z5) after cutting feed to end point(Z-20), X tool infeed 3mm and cycle the above-mentioned steps)

M30; (End of program)

3.20.6 Radial grooving multiple cycle G75

Command format: G75 R(e);

G75 X(U) Z(W) P(Δi) Q(Δk) R(Δd) F ;

Command function: Axial (Z) tool infeed cycle compounds radial discontinuous cutting cycle:

Tool infeeds from starting point in radial direction, retracts, infeeds again, and again and again, and last tool retracts in axial direction, and retracts to position in radial direction, which is called one radial cutting cycle; tool infeeds in axial direction and execute the next radial cutting cycle; cut to end point of cutting, and then return to starting point (starting point and end point are the same one in G75), which is called one radial grooving compound cycle. Directions of axial tool infeed and radial tool infeed are defined by relative position between end point X(U) Z(W) and starting point of cutting. G75 is used for machining radial loop groove or column surface by radial discontinuously cutting, breaking stock and stock removal.

Relevant definitions:

Starting point of radial cutting cycle: Starting position of axial tool infeed for each radial cutting cycle, defined by $A_n(n=1, 2, 3, \dots)$, X coordinate of A_n is the same that of starting point A, the different value of X coordinate between A_n and A_{n-1} is Δk . The starting point A_1 of the first radial cutting cycle is the same as the starting point A, and Z starting point (A_f) of the last axial cutting cycle is the same that of cutting end point.

End point of radial tool infeed: Starting position of radial tool infeed for each radial cutting cycle, defined by B_n ($n=1, 2, 3, \dots$), X coordinates of B_n is the same that of cutting end point, Z coordinates of B_n is the same that of A_n , and the end point (B_f) of the last radial tool infeed is the same that of cutting end point.

End point of axial tool retraction: End position of axial tool infeed (travel of tool infeed is Δd) after each axial cutting cycle reaches the end point of axial tool infeed, defining with C_n ($n=1, 2, 3, \dots$), X coordinate of C_n is the same that of cutting end point, and the different value of Z coordinate between C_n and A_n is Δd ;

End point of radial cutting cycle: End position of radial tool retraction from the end point of axial tool retraction, defined by D_n ($n=1, 2, 3, \dots$), X coordinate of D_n is the same that of starting point, Z coordinates of D_n is the same that of C_n (the different value of Z coordinate between it and A_n is Δd);

Cutting end point: It is defined by $X(U) Z(W)$, and is defined with B_f of the last radial tool infeed. $R(e)$: It is the tool retraction clearance after each radial(X) tool infeed, its range is 0~99.999 (unit:mm, radius value) without sign symbols. The specified value is reserved validly after $R(e)$ is executed and the data is switched and saved to No.056. NO.056 value is regarded as the tool retraction clearance when $R(e)$ is not input.

X: X absolute coordinate value of cutting end point B_f (unit: mm).

U: Different value of X absolute coordinate between cutting end point B_f and starting point.

Z: Z absolute coordinate value of cutting end point B_f (unit:mm).

W: Different value of Z absolute coordinate between cutting end point B_f and starting point A (unit:mm).

$P(\Delta i)$: Radial(X) discontinuous tool infeed of each axial cutting cycle, its range: $0 < \Delta i \leq 9999999(IS_B) / 0 < \Delta k \leq 99999999 (IS_C)$ (unit: least input increment, without sign symbol).

$Q(\Delta k)$: Axial(Z) discontinuous tool infeed of each radial cutting cycle, its range: $0 < \Delta k \leq 9999999(IS_B) / 0 < \Delta k \leq 99999999 (IS_C)$ (unit: least input increment, without sign symbol)..

$R(\Delta d)$: Axial (Z) tool retraction clearance after cutting to end point of radial cutting, its range: $0 \sim 99999999 \times \text{least input increment}$ (unit: mm/inch, without sign symbol).

The system defaults the tool retraction clearance is 0 after the radial cutting end point is completed when $R(\Delta d)$ is omitted.

The system defaults it executes the positive tool retraction when $Z(W)$ and $Q(\Delta k)$ are omitted.

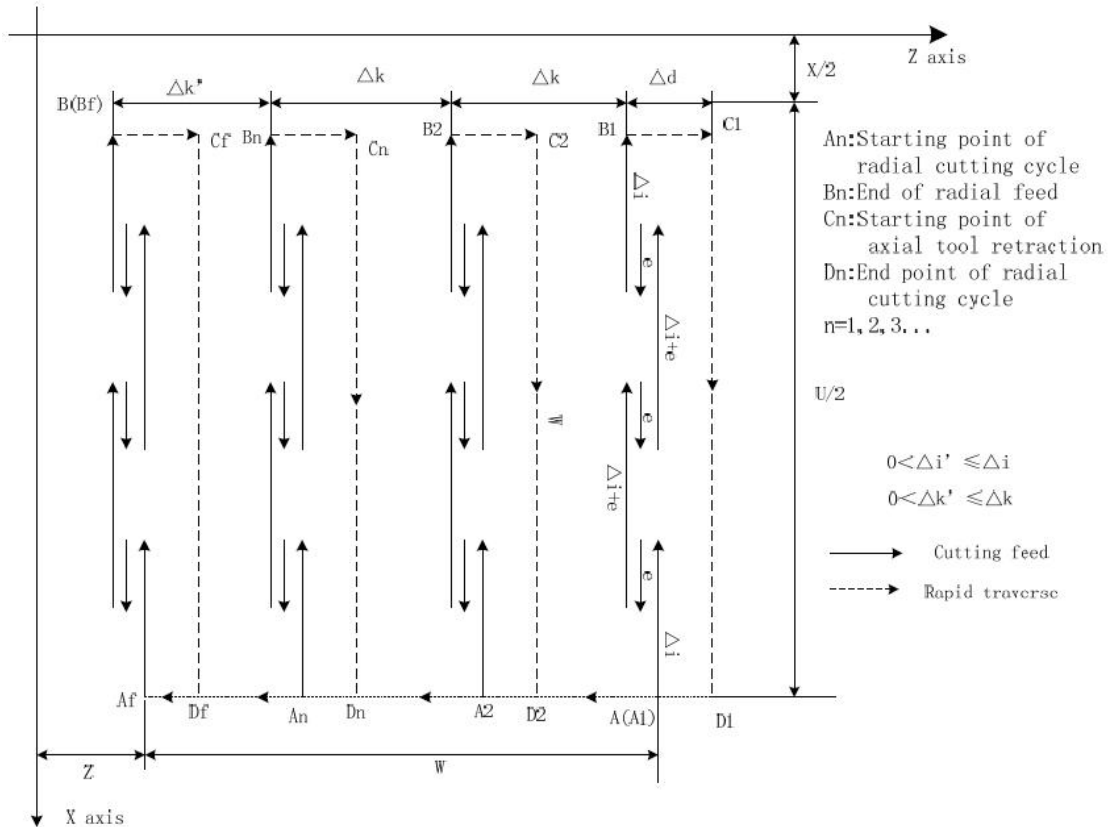


Fig. 3-36 G75 path

Execution process: (Fig. 3-36)

- ① Radial (X) cutting feed Δi from the starting point of radial cutting cycle, feed in X negative direction when the coordinates of cutting end point is less than that of starting point in X direction, otherwise, feed in X positive direction;
- ② Radial(X) rapid tool retraction e and its direction is opposite to the feed direction of ①;
- ③ X executes the cutting feed $(\Delta k+e)$ again, the end point of cutting feed is still in it between starting point A_n of radial cutting cycle and end point of radial tool infeed, X executes the cutting feed $(\Delta i+e)$ again and executes ②; after X cutting feed $(\Delta i+e)$ is executed again, the end point of X cutting feed is on B_n or is not on it between A_n and B_n cutting feed to B_n and then execute ④;
- ④ Axial(Z) rapid tool retraction Δd to C_n , when Z coordinate of B_r (cutting end point) is less than that of A (starting point), retract tool in Z positive, otherwise, retract tool in Z negative direction;
- ⑤ Radial (Z) rapid retract tool to D_n , No. n radial cutting cycle is completed. The current radial cutting cycle is not the last one, execute ⑥; if it is the previous one before the last radial cutting cycle, execute ⑦;
- ⑥ Axial(X) rapid tool infeed, and it direction is opposite to ④ retract tool. If the end point of

tool infeed is still on it between A and A_f (starting point of last radial cutting cycle) after Z

tool infeed ($\Delta d + \Delta k$), i.e. $D_n \rightarrow A_{n+1}$ and then execute ① (start the next radial cutting cycle); if

the end point of tool infeed is not on it between D_n and A_f after Z tool infeed ($\Delta d + \Delta k$), rapidly

traverse to A_f and execute ① to start the first radial cutting cycle;

⑦ Z rapidly traverses to A, and G75 is completed.

⑧ **Explanation:**

- The cycle movement is executed by X(W) and P(Δi) blocks of G75, and the movement is not executed if only "G75 R(e);" block is executed;
- Δd and e are specified by the same address R and whether there are X(U) and P(Δi) words or not in blocks to distinguish them;
- The tool can stop in Auto mode and traverse in Manual mode when G75 is executed, but the tool must return to the position before executing in Manual mode when G75 is executed again, otherwise the following path will be wrong;
- When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path;
- R(Δd) must be omitted in grooving, and so there is no tool retraction clearance when the tool cuts to radial cutting end point.

Example: Fig.3-37

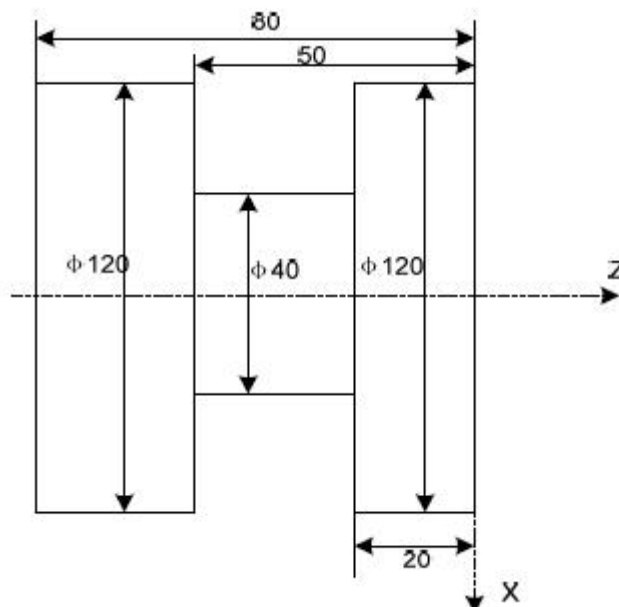


Fig. 3-37 G75 cutting

Program (suppose the grooving tool width is 4mm, the system least increment is 0.001mm):

O0008;

G00 X150 Z50 M3 S500; (Start spindle with 500 r/min)

G0 X125 Z-20; (Position to starting point of machining)

G75 R0.5 F150; (Machining cycle)

G75 X40 Z-50 P6000 Q3000; (X tool infeed 6mm every time, tool retraction 0.5mm, rapid returning to starting point (X125) after infeeding to end point (X40), Z tool infeed 3mm and cycle the above-mentioned steps to continuously run programs)

G0 X150 Z50; (Return to starting point of machining)

M30; (End of program)

3.21 Thread Cutting Commands

ECN*000T CNC system can machine many kinds of thread cutting, including metric/inch single, multi threads, thread with variable lead and tapping cycle. Length and angle of thread run-out can be changed, multiple cycle thread is machined by single sided to protect tool and improve smooth finish of its surface. Thread cutting includes: continuous thread cutting G32, thread cutting with variable lead G34, Z thread cutting G33, Thread cutting cycle G92, Multiple thread cutting cycle G76.

The machine used for thread cutting must be installed with spindle encoder whose pulses are set by No.070m. Drive ratio between spindle and encoder is set by No.110 and No.111. X or Z traverses to start machine after the system receives spindle signal per rev in thread cutting, and so one thread is machined by multiple roughing, finishing without changing spindle speed.

The system can machine many kinds of thread cutting, such as thread cutting without tool retraction groove. There is a big error in the thread pitch because there are the acceleration and the deceleration at the starting and ending of X and Z thread cutting, and so there is length of thread lead-in and distance of tool retraction at the actual starting and ending of thread cutting. X, Z traverse speeds are defined by spindle speed instead of cutting feedrate override in thread cutting when the pitch is defined. The spindle override control is valid in thread cutting. When the spindle speed is changed, there is error in pitch caused by X and Z acceleration/deceleration, and so the spindle speed cannot be changed and the spindle cannot be stopped in thread cutting, which will cause tool and workpiece to be damaged.

3.21.1 Thread cutting with constant lead G32

Command format: G32 X(U)_ Z(W)_ F(I)_ J_ K_ Q_

Command function: The path of tool traversing is a straight line from starting point to end point as Fig.3-33; the longer moving distance from starting point to end point (X in radius value) is called as the long axis and another is called as the short axis. In course of motion, the long axis traverses one lead when the spindle rotates one revolution, and the short axis and the long axis execute the linear interpolation. Form one spiral grooving with variable lead on the surface of workpiece to realize thread cutting with constant lead. Metric pitch and inch pitch are defined respectively by F, I. Metric or inch straight, taper, end face thread and continuous multi-section thread can be machined in G32.

Command specifications:

G32 is modal;

Pitch is defined to moving distance when the spindle rotates one rev (X in radius);

Execute the straight thread cutting when X coordinates of starting point and end point are the same one (not input X or U);

Execute the end face thread cutting when X coordinates of starting point and end point are the

same one(not input Z or W);

Execute the cutting taper thread when X and Z coordinates of starting point and end point are different;

Related definitions:

F: Metric pitch is moving distance of long axis when the spindle rotates one rev: 0.001 mm ~ 500mm. After F is executed, it is valid until F with specified pitch is executed again.

I: Teeth per inch. It is ones per inch(25.4 mm) in long axis, and also is circles of spindle rotation when the long axis traverses one inch(25.4 mm) :0.06tooth/inch ~ 25400tooth/inch. After I is executed, it is valid until I with specified pitch is executed again. The metric, inch input both express the teeth per inch thread.

J: Movement in the short axis in thread run-out, its range: $\pm 99999999 \times$ least input increment with negative sign; if the short axis is X, its value is specified with the radius; J value is the modal parameter.

K: Length in the long axis in thread run-out, its range: $\pm 99999999 \times$ least input increment. If the long axis is X, its value is in radius without direction; K is modal parameter.

Q: Initial angle(offset angle)between spindle rotation one rev and starting point of thread cutting: 0 ~ 360000 (unit: 0.001 degree). Q is non-modal parameter, must be defined every time,otherwise it is 0.

Q rules:

1. Its initial angle is 0° if Q is not specified;
2. For continuous thread cutting, Q specified by its following thread cutting block except for the first block is invalid, namely Q is omitted even if it is specified;
3. Multi threads formed by initial angle is not more than 65535;
4. Q unit : 0.001°. Q180000 is input in program if it offsets 180° with spindle one-turn; if Q180 or Q180.0, it is 0.18°.

Difference between long axis and short axis is shown in Fig. 3-38.

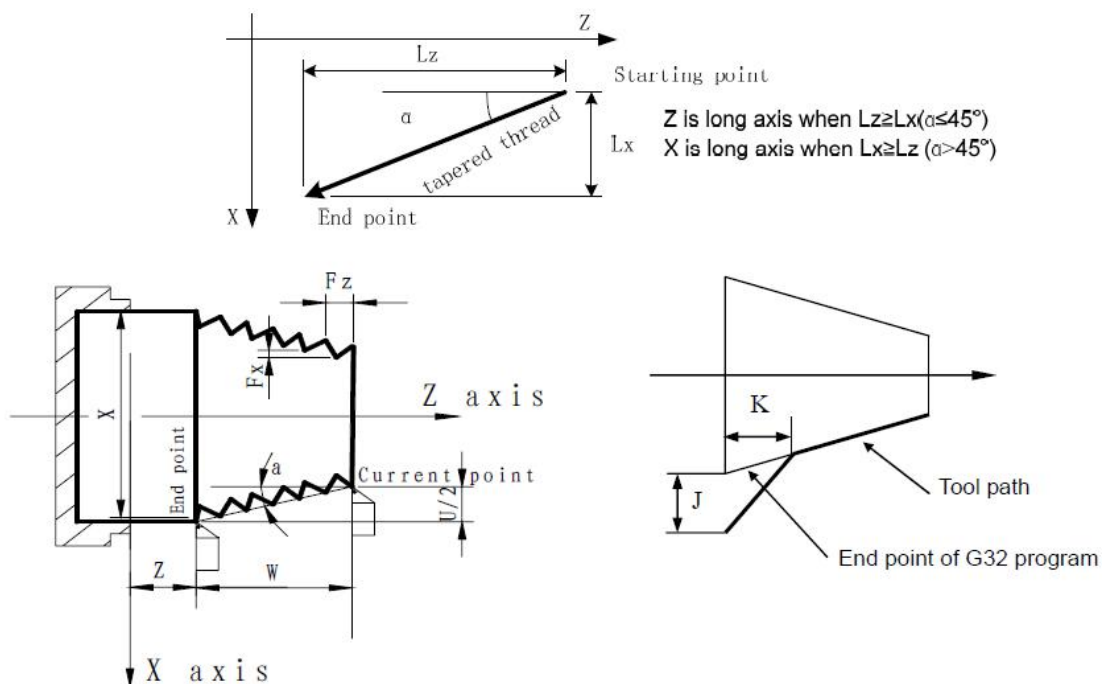


Fig. 3-38 G32 path

Notes:

- J, K are modal. The thread run-out is previous J, K value when they are omitted in the next block in continuous thread cutting. Their mode are cancelled when no thread cutting are executed;
- There is no thread run-out when J, or J, K are omitted; $K=J$ is the thread run-out value when K is omitted;
- There is no thread run-out when $J=0$ or $J=0, K=0$;
- The thread run-out value $J=K$ when $J \neq 0, K=0$;
- There is no thread run-out when $J=0$ or $K \neq 0$;
- If the current block is for thread and the next block is the same, the system does not test the spindle encoder signal per rev at starting the next block to execute the direct thread cutting, which function is called as continuous thread machining;
- After the feed hold is executed, the system displays “Pause” and the thread cutting continuously executes not to stop until the current block is executed completely; if the continuous thread cutting is executed, the program run pauses after thread cutting blocks are executed completely;
- In Single block, the program stops run after the current block is executed. The program stops running after all blocks for thread cutting are executed;
- The thread cutting decelerates to stop when the system resets, emergently stop or its drive unit alarms.

Example: Pitch: 2mm. $\delta 1 = 3\text{mm}, \delta 2 = 2\text{mm}$, total cutting depth 2mm divided into two times cut-in.

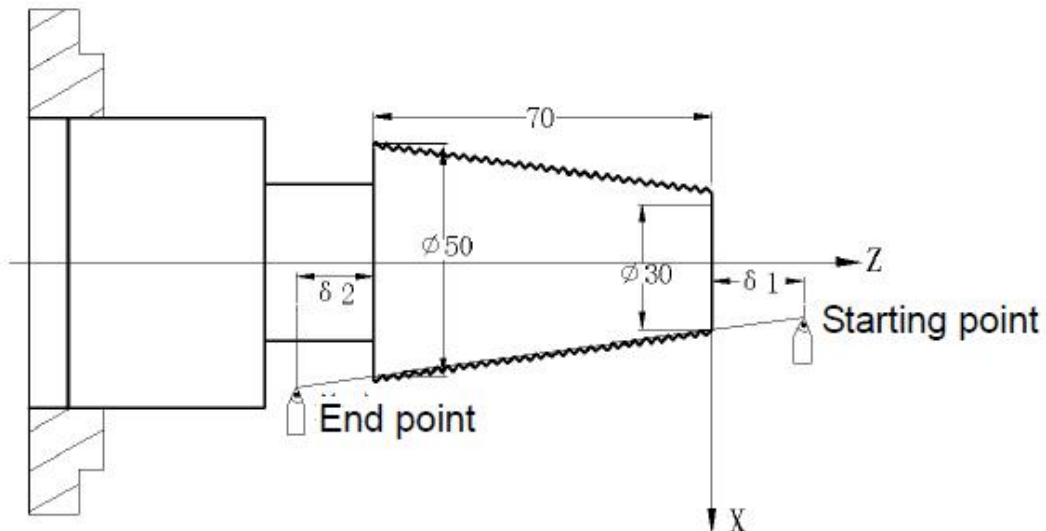


Fig-3-39

Program:

O0009;

G00 X28 Z3; (First cut-in 1mm)

G32 X51 W-75 F2.0; (First taper cutting)

G00 X55; (Tool retraction)

W75; (Z returns to the starting point)

X27; (Second tool infeed 0.5mm)

G32 X50 W-75 F2.0; (Second taper thread cutting)

G00 X55; (Tool retraction)

W75 ; (Z returns to the starting point)

M30;

3.21.3 Thread cutting with variable lead G34

Command format: G34 X(U)__ Z(W)__ F(I)__ J__ K__ R__ ;

Command function: The motion path of tool is a straight line from starting point of X, Z to end point specified by the block, the longer moving distance from starting point to end point(X in radius value) is called as the long axis and another is called as the short axis. In course of motion, the long axis traverses one lead when the spindle rotates one rev, the pitch increases or decreases a specified value per rev and one spiral grooving with variable lead on the surface of workpiece to realize thread cutting with variable lead. Tool retraction can be set in thread cutting.

F, I are specified separately to metric, inch pitch. Executing G34 can machine metric or inch straight, taper, end face thread with variable pitch.

Command specifications:

G34 is modal;

Meanings of X(U) , Z(W) , J, K are the same that of G32;

F: Specify lead, and its range is referred to Table 1-2;

I: Specify thread teeth per inch, and its range is referred to Table 1-2;

R: Increment or decrement of pitch per rev, $R=F1- F2$, with direction; $F1>F2$, pitch decreases when R is negative; $F1<F2$, pitch increases when R is positive (as Fig. 3-41);

R: $\pm 0.001\sim\pm 500.000$ mm/pitch (metric thread);

$\pm 0.060\sim\pm 25400$ tooth/inch (inch thread).

The system alarms when R exceeds the above-mentioned range or the pitch exceeds permissive value or is negative owing to R increases or decreases.

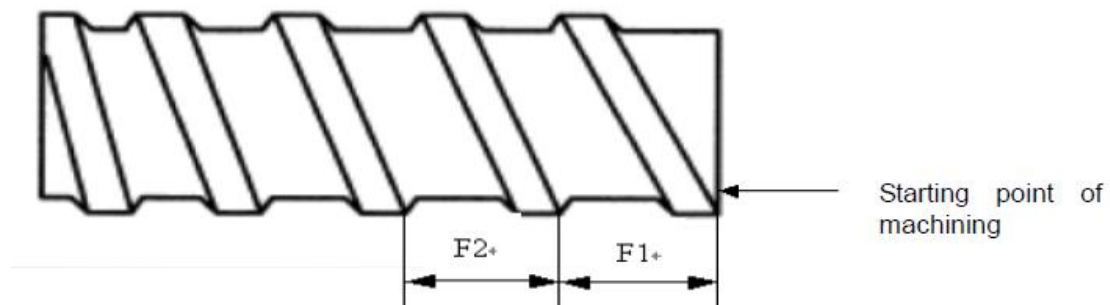


Fig. 3-41 Variable pitch thread

Note: It is the same as that of G32.

Example: First pitch of starting point: 4mm, increment 0.2mm per rev of spindle.

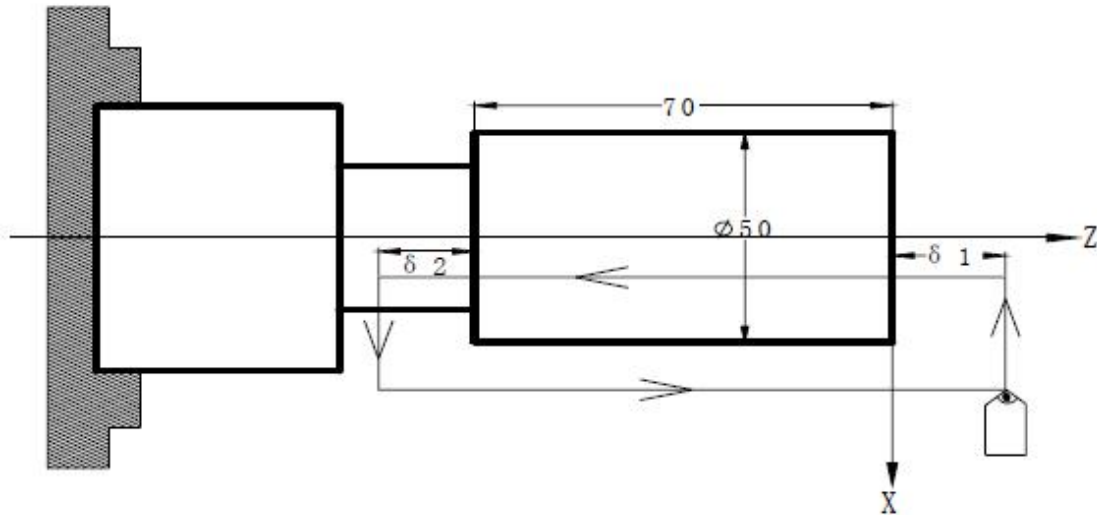


Fig. 3-42 Variable pitch thread machining

Use macro variables to simplify programming when G34 is used many times. $\delta_1 = 4\text{mm}$, $\delta_2 = 4\text{mm}$, total cutting depth 4mm, total cutting cycle 15 times; first tool infeed 0.8mm, gradual decreasing cutting every time 0.2mm, min. infeed 0.2mm.

```

Program: O0010;
G00 X60 Z4 M03 S500;
G65 H01 P#102 Q800;
G65 H01 P#103 Q0;
N10 G65 H02 P#104 Q#103 R1;
G65 H01 P#103 Q#104;
G65 H81 P30 Q#104 R15;
G00 U-10;
G65 H01 P#100 Q#102;
G00 U-#100;
G34 W-78 F3.8 J5 K2 R0.2;
G00 U10;
Z4;
G65 H03 P#101 Q#100 R200;
G65 H01 P#102 Q#101;
G65 H86 P20 Q#102 R200;
G65 H80 P10;
N20 G65 H01 P#102 R200;
G65 H80 P10;
N30 M30;
First tool infeed: assignment #102=0.8mm
Cycle count: assignment #103=0
Cycle count starting: #104=#103+1
#103=#104
Total cutting cycle times: #104=15,jump to block N30
Tool infeed to Φ50

```

Cutting infeed: #100=#102
 Tool infeed
 Variable pitch cutting
 Tool retraction
 Z returns to starting point
 Decreasing of cutting feed again: #101=#100-0.2
 Assignment again #102=#101
 Infeed: Jump to block N20 when #102 \leq 0.2mm
 Unconditionally jump to block N10
 Min. infeed: #102=0.2
 Unconditionally jump to block N10
 Starting point of machining

3.21.4 Z thread cutting G33

Command format: G33 Z(W)___ F(I)___ L___ ;

Command function: Tool path is from starting point to end point and then from end point to starting point. The tool traverses one pitch when the spindle rotates one rev, the pitch is consistent with pitch of tool and there is spiral grooving in internal hole of workpiece and the internal machining can be completed one time.

Command specification: G33 is modal command;

Z(W): When Z or W is not input and starting point and end point of Z axis are the same one, the thread cutting must not be executed;

F: Thread pitch, and its range is referred to Table 1-2;

I: Teeth per inch thread 0.06 ~ 25400 teeth/inch; its range is referred to Table 1-2. It is single thread when L is omitted.

Cycle process:

- ① Z tool infeed (start spindle before G33 is executed);
- ② M05 signal outputs after Z reaches the specified end point in programming;
- ③ Test spindle after completely stopping;
- ④ Spindle rotation (CCW) signal outputs(reverse to the original rotation direction);
- ⑤ Z executes the tool retracts to starting point;
- ⑥ M05 signal outputs and the spindle stops;
- ⑦ Repeat the steps ①~⑤ if multi threads are machined.

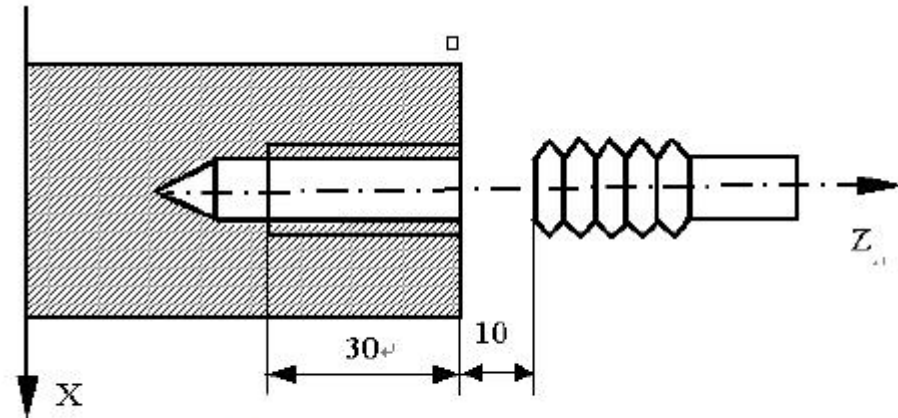


Fig. 3-89

Example: Fig. 3-89, thread M10×1.5

Program:

O0011;

G00 Z90 X0 M03; Start spindle

G33 Z50 F1.5; Tap cycle

M03 Start spindle again

G00 X60 Z100; Machine continuously

M30

Note 1: Before tapping, define rotation direction of spindle according to tool rotating. The spindle stops rotation after the tapping is completed and the spindle is started again when machining thread continuously.

Note 2: G33 is for rigid tapping. The spindle decelerates to stop after its stop signal is valid, at the moment, Z executes continuously infeeds along with the spindle rotating, and so the actual cutting bottom hole is deeper than requirement and the length is defined by the spindle speed and its brake in tapping.

Note 3: Z rapid traverse speed in tapping is defined by spindle speed and pitch is not relevant to cutting feedrate override.

Note 4: In Single block to feed hold, the tapping cycle continuously executes not to stop until the tool returns to starting point when the system displays "Pause".

Note 5: The thread cutting decelerates to stop when the system resets, emergently stop or its driver alarms.

3.21.5 Rigid tapping G84, G88

Command format: End face rigid tapping G84 X(U)__ C(H)__ Z(W)__ P__ F(I)__ K__ M__;

Side rigid tapping G88 Z(W)__ C(H)__ X(U)__ P__ F(I)__ K__ M__;

Command explanations: modal G codes

G84: End face tapping cycle G code

G88: Side tapping cycle G code

(X, C): Tapping hole position; -----G84

Z: Hole bottom position of tapping; -----G84

(Z, C): Tapping hole position; -----G88

X: Hole bottom position of tapping hole; -----G88

P: Pause time (ms) when tapping to the hole bottom

F(I): Thread lead, F (I) > 0 right-hand tapping, F (I) < 0 left-hand tapping

K: Repetitive count of tapping. When the incremental programming is used to the tapping hole

position, the tapping is performed in the different hole position

M: Used to clamp the graduation spindle.

Rigid tapping method:

M29 S_‾ before G84/G88 is specified below

M29 S_‾;

G84(G88) X_‾ C_‾ (Z_‾ C_‾) Z_‾ (X_‾) P_‾ F_‾ K_‾ M_‾;

M29 S_‾ in the same block is commanded below

G84(G88) X_‾ C_‾ (Z_‾ C_‾) Z_‾ (X_‾) R_‾ P_‾ F_‾ K_‾ M29 S_‾;

Notes :

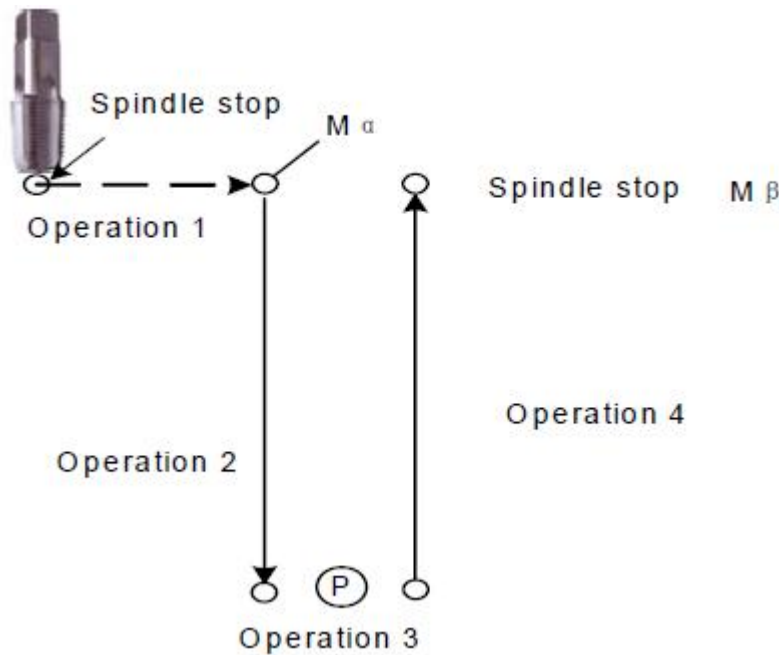
1) For the above 2nd method, the machine zero return is executed before the spindle executes the tapping, otherwise, the method cannot be used. In M29, CNC cannot select a spindle to execute the tapping. The rigid tapping needs to be clamped when the spindle positioning is completed, the method also cannot be used because M codes cannot be in the same block;

2) The axis movement command cannot be use between M29 and G84/G88;

3) M29 cannot be specified repetitively in rigid tapping;

4) In multiple spindle rigiding tapping, a spindle used to the rigid tapping must be selected before M29, and it cannot be switched before the rigid tapping state is cancelled;

Operations is shown below:



Execution

explanations:

Operation 1: positioning to the hole position (starting point of rigid tapping) ;

Before starting operation 2, the system outputs M α when it has M code used to clamp the spindle;

Operation 2: start the rigid tapping;

Operation 3: pause time P in rigid tapping in hole bottom;

Operation 4: the rigid tapping returns to the hole bottom (starting point of rigid tapping;

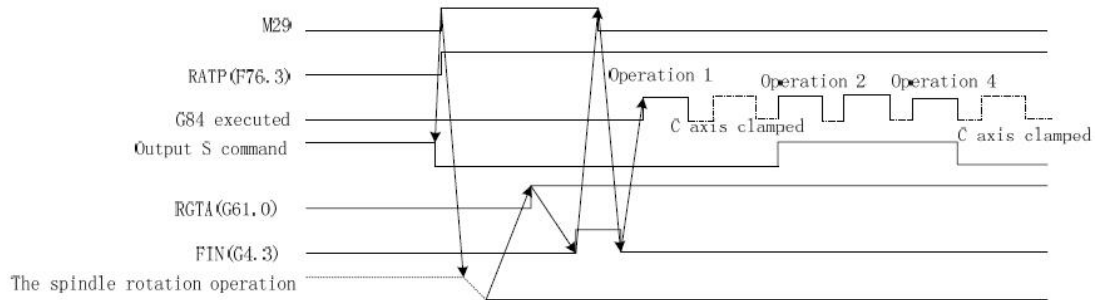
When the block for rigid tapping specifies M code used to clamp the spindle, M β is output.

Note: α value is set in №170, $\beta = \alpha + 1$, so, these corresponding M codes in PLC are executed.

Rigid tapping sequence

Taking example of G84 right-hand tapping, explain the creation, execution and cancellation of rigid tapping.

Creation and execution of rigid tapping



The spindle rotation operation is defined that the rotary axis is switched to the position control mode(i.e. send the position mode switch signal to the servo spindle), and the system checks the position mode arrival signal of the servo spindle.

Rigid tapping cancellation

The cancellation methods of rigid tapping are shown below:

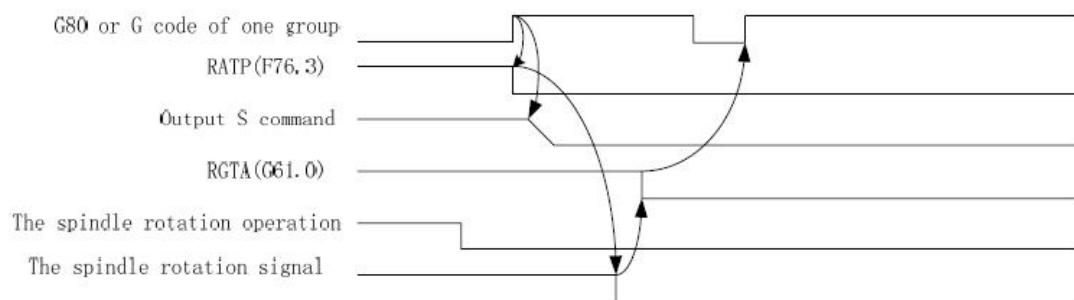
- 1) G80 cancels the rigid tapping mode;
- 2) G codes commands other cycles;
- 3) Other G codes in Group 01;
- 4) CNC resets.

The falling edge of F76.3 signal cancels the rigid tapping mode signal.

(a) RTCRG(№186 #2) is set to '1' when:

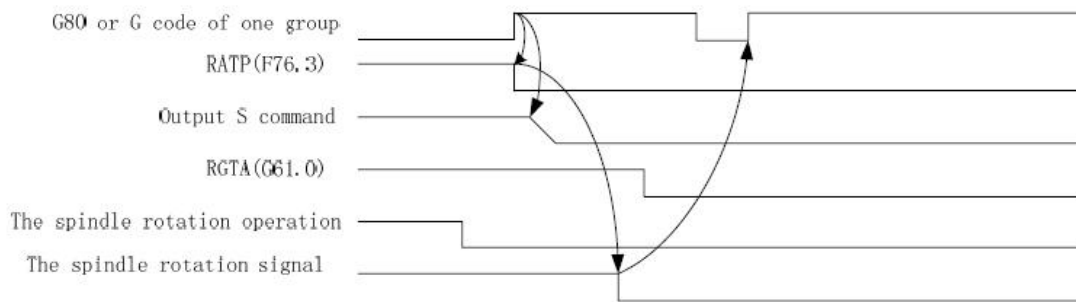
The system executes directly the next block instead of waiting the rigid tapping mode signal RGTAP<G61 #0> to become '0'.

Rigid tapping cancellation is shown below:



(b) RTCRG(№186 #2) is set to '0' when:

Wait the rigid tapping mode signal RGTAP<G61 #0> to become '0' and then execute the next block. Rigid tapping cancellation is shown below:

**Notes:**

- In rigid tapping, when the tapping direction is changed (i.e. G84 and G88 are switched), the system specifies again the hole bottom position of the tapping, otherwise, the unexpected result rises;
- The rigid tapping G command is in Group 01, it is cancelled temporarily in the rigid tapping state, and then is recovered it after the tapping is done;
- The dry run function is disabled in rigid tapping;
- In the rigid tapping, the machine lock function is valid; the tapping axis and the spindle do not move when the machine lock function is enabled;
- When the reset is executed in the rigid tapping, the rigid tapping state is released and the spindle returns to the previous state before the rigid tapping;
- In rigid tapping, when the tapping block is executed and the tapping returns, the feed hold/single block run function is disabled temporarily till the tapping return completes;
- In rigid tapping mode, the backlash compensation is performed to compensate the dry run of the spindle rotation (CW, CCW). It is set in parameters (№33 ~ №34 or №180 ~ №182). The backlash compensation along the tapping axis is performed in general method;
- In multiple spindle tapping, the 1st spindle is used to the graduation and the 2nd to the tapping. After the 1st spindle graduation is completed, the spindle must be clamped on the machine and the machine clamping/releasing M code can be specified in the rigid tapping. M code used to clamp the spindle is added in G84/G88, i.e. the system can output two kinds of M code. M code used to clamp the spindle is set in the data parameter (№170). M code used to the releasing is the setting value +1 of №170;
- Setting RTORI (state parameter №186 # 7) to 1 can execute the reference point return of the spindle before the system starts the rigid tapping.

3.21.6 Thread cutting cycle G92

Command format: G92 X(U) _ Z(W) _ F_ J_ K_ L ; (Metric straight thread cutting cycle)

G92 X(U) _ Z(W) _ I_ J_ K_ L ; (Inch straight thread cutting cycle)

G92 X(U) _ Z(W) _ R_ F_ J_ K_ L ; (Metric taper thread cutting cycle)

G92 X(U) _ Z(W) _ R_ I_ J_ K_ L ; (Metric taper thread cutting cycle)

Command function: Tool infeeds in radial (X) direction and cuts in axial (Z or X, Z) direction from starting point of cutting to realize straight thread, taper thread cutting cycle with constant thread pitch. Thread run-out in G92: at the fixed distance from end point of thread cutting, Z executes thread interpolation and X retracts with exponential or linear acceleration, and X retracts at rapidly traverse speed after Z reaches to end point of cutting as Fig. 3-44.

Command specifications:

G92 is modal;

Starting point of cutting: starting position of thread interpolation;

End point of cutting: end position of thread interpolation;

X: X absolute coordinate of end point of cutting, unit: mm;

U: different value of X absolute coordinate from end point to starting point of cutting, unit: mm;

Z: Z absolute coordinate of end point of cutting, unit: mm;

W: Different value of X absolute coordinate from end point to starting point of cutting, unit: mm;

R: Different value(radius value) of X absolute coordinate from end point to starting point of cutting. When the sign of R is not the same that of U, $R \leq |U/2|$, unit: mm;

F: Thread lead, its range: $0 < F \leq 500$ mm. After F value is executed, it is reserved and can be omitted;

I: Thread teeth per inch, its range: $0.06\text{tooth/inch} \sim 25400\text{tooth/inch}$, it is reserved and it can be omitted not to input after I specified value is executed;

J: Movement in the short axis in thread run-out, its range $0 \sim 99999999 \times$ least input increment, unit: mm/inch, without direction (automatically define its direction according to starting position of program), and it is modal parameter. If the short axis is X, its value is specified by radius;

K: Movement in the long axis in thread run-out, its range: $0 \sim 99999999 \times$ least input increment, unit: mm/inch, without direction (automatically define its direction according to starting position of program), and it is modal parameter. If the long axis is X, its value is specified by radius;

L: Multi threads: $1 \sim 99$ and it is modal parameter. (The system defaults it is single thread when L is omitted).

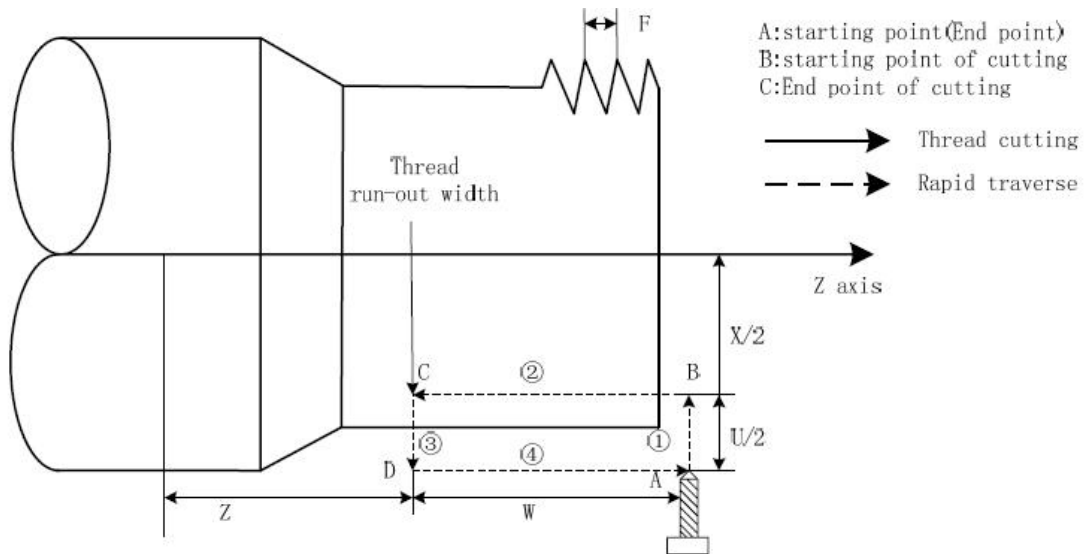


Fig. 3-44

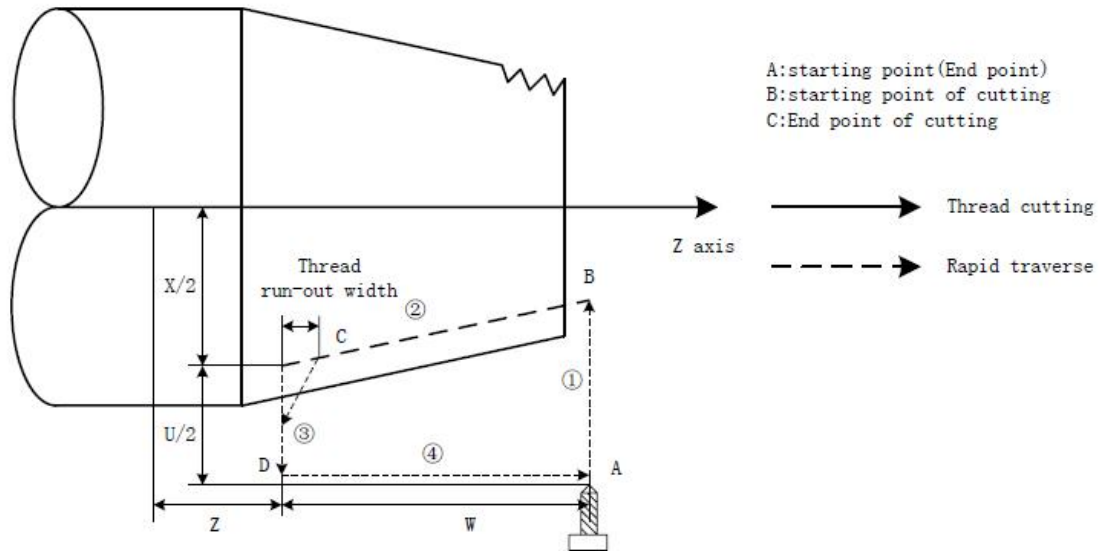


Fig. 3-45

The system can machine one thread with many tool infeed in G92, but cannot do continuous two thread and end face thread. Definition of thread pitch in G92 is the same that of G32, and a pitch is defined that it is a moving distance of long axis(X in radius) when the spindle rotates one rev.

Pitch of taper thread is defined that it is a moving distance of long axis(X in radius). When absolute value of Z coordinate difference between B point and C point is more than that of X (in radius), Z is long axis; and vice versa.

Cycle process: straight thread as Fig.3-44 and taper thread as Fig.3-45.

- ① X traverses from starting point to cutting starting point;
- ② Thread interpolates (linear interpolation) from the cutting starting point to cutting end point;
- ③ X retracts the tool at the cutting feedrate (opposite direction to the above-mentioned), and return to the position which X absolute coordinate and the starting point are the same;
- ④ Z rapidly traverses to return to the starting point and the cycle is completed.

Notes:

- Length of thread run-out is specified by №019 when J, K are omitted;
- Length of thread run-out is K in the long direction and is specified by №019 when J is omitted;
- Length of thread run-out is J=K when K is omitted;
- There is no thread run-out when J=0 or J=0, K=0;
- Length of thread run-out is J=K when J≠0, K=0;
- There is no thread run-out when J=0, K≠0;
- After executing the feed hold in thread cutting, the system does not stop cutting until the thread cutting is completed with **Pause** on screen;
- After executing single block in thread cutting, the program run stops after the system returns to starting point(one thread cutting cycle is completed);
- They are executed as the positive values when J, K negative values are input;
- Thread cutting decelerates to stop when the system resets, emergently stops or its driver alarms.

Command path: relative position between thread cutting end point and starting point with U, W,

R and tool path and thread run-out direction with different U, W, R signs below:

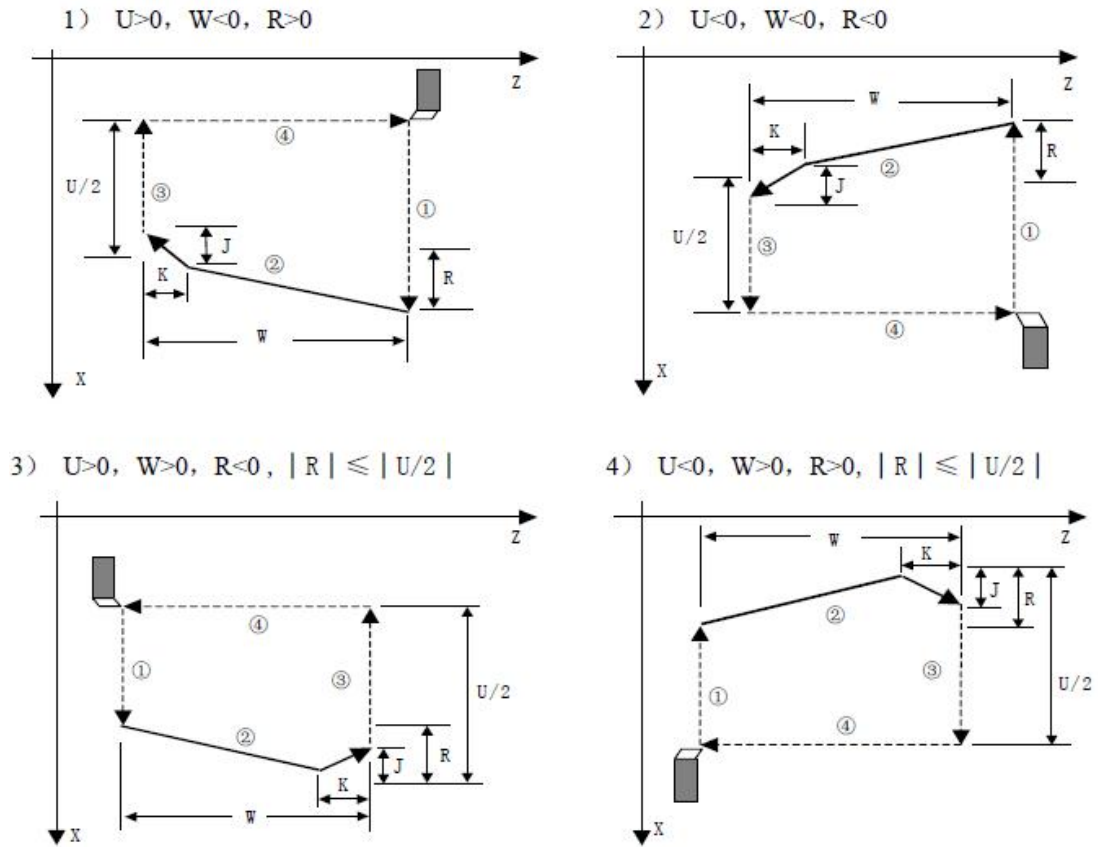


Fig.3-46

Example: Fig.3-47

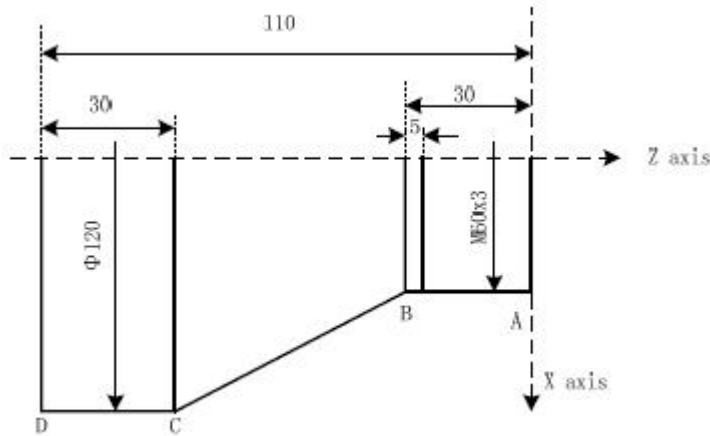


Fig.3-47

Program:

O0012;

M3 S300 G0 X150 Z50 T0101; (Thread tool)

G0 X65 Z5; (Rapid traverse)

G92 X58.7 Z-28 F3 J3 K1; (Machine thread with 4 times cutting, the first tool infeed 1.3mm)

X57.7; (The second tool infeed 1mm)

X57; (The third tool infeed 0.7mm)

X56.9; (The fourth tool infeed 0.1mm)

M30;

3.21.7 Multiple thread cutting cycle G76

Command format: G76 P(m) (r) (a) Q(Δ dmin) R(d) ;

G76 X(U) Z(W) R(i) P(k) Q(Δ d) F(I) ;

Command function: Machining thread with specified depth of thread (total cutting depth) is completed by multiple roughing and finishing, if the defined angle of thread is not 0° , thread run-in path of roughing is from its top to bottom, and angle of neighboring thread teeth is the defined angle of thread. G76 can be used for machining the straight and taper thread with thread run-out path, which is contributed to thread cutting with single tool edge to reduce the wear of tool and to improve the precision of machining thread. But G76 cannot be used for machining the face thread. machining path is shown in Fig. 3-48(a):

Relevant definitions:

Starting point(end point): Position before block runs and behind blocks run, defined by A point;

End point of thread(D point): End point of thread cutting defined by X(U) Z(W) .The tool will not reach the point in cutting if there is the thread run-out path;

Starting point of thread: Its absolute coordinates is the same that of A point and the different value of X absolute coordinates between C and D is i(thread taper with radius value). The tool cannot reach C point in cutting when the defined angle of thread is not 0° ;

Reference point of thread cutting depth (B point) : Its absolute coordinates is the same that of A point and the different value of X absolute coordinate between B and C is k(thread taper with radius value).The cutting depth of thread at B point is 0 which is the reference point used for counting each thread cutting depth by the system;

Thread cutting depth: It is the cutting depth for each thread cutting cycle. It is the different value(radius value, without signs) of X absolute coordinate between B and intersection of reversal extension line for each thread cutting path and straight line BC. The cutting depth for

each roughing is $n \times \Delta d$, n is the current roughing cycle times, Δd is the thread cutting depth of

first roughing;

Thread cutting amount: Different value between the current thread current depth and the

previous one: $(n - n - 1) \times \Delta d$;

End point of tool retraction: It is the end position of radial (X) tool retraction after the thread cutting in each thread roughing, finishing cycle is completed, defining with E point;

Thread cut-in point: B_n (n is the cutting cycle times) is the actual thread cutting starting point in each thread roughing cycle and finishing cycle, B_1 is the first thread roughing cutting-in point, B_r is the last thread roughing cut-in point, B_e is the thread finishing cutting-in point. B_n is X, Z replacement formula corresponding to B.

$$tg \frac{a}{2} = \frac{Z \text{ replacement}}{X \text{ replacement}}$$

a: thread angle;

X: X absolute coordinate (unit: mm) of thread end point;

U: Different value (unit: mm) of X absolute coordinate between thread end point and starting point;

Z: Z absolute coordinate (unit: mm) of thread end point;

W: Different value (unit: mm) of Z absolute coordinate between thread end point and starting point;

P(m): Times of thread finishing: 00 ~ 99 (unit: times). It is valid after m specified value is executed, and the system parameter №057 value is rewritten to m . The value of system parameter №057 is regarded as finishing times when m is not input. In thread finishing, every feed cutting amount is equal to the cutting amount d in thread finishing dividing the finishing times m ;

P(r): Width of thread run-out 00~99(unit: $0.1 \times L$, L is the thread pitch). It is valid after r specified value is executed and the system parameter №019 value is rewritten to r . The value of system parameter №019 is the width of thread run-out when r is not input. The thread run-out function can be applied to thread machining without tool retraction groove and the width of thread run-out defined by system parameter №019 is valid for G92, G76;

P(a): Angles at taper of neighboring two tooth, range: 00~99, unit: deg($^{\circ}$). It is valid after a specified value is executed and the system parameter №058 value is rewritten to a . The system parameter №058 value is regarded as angle of thread tooth. The actual angle of thread in defined

by tool ones and so a should be the same as the tool angle; $\Delta Q(\Delta d_{min})$: Minimum cutting travel of

thread roughing, range: 0 ~ 999999(IS-C)/ 0 ~ 99999(IS-B),(unit: least input increment,

radius value). When $(n - n - 1) \times \Delta d < \Delta d_{min}$, Δd_{min} is regarded as the cutting travel of current

roughing, i.e. depth of current thread cutting is $(n - 1 \times \Delta d + \Delta d_{min})$. Setting Δd_{min} is to avoid

the too small of roughing amount and too many roughing times caused by the cutting amount

deceleration in thread roughing. After $Q(\Delta d_{min})$ is executed, the specified value Δd_{min} is valid

and the system data parameter NO. 059 value is rewritten to Δd_{min} (unit: 0.001). when $Q(\Delta d_{min})$

is not input, the system data parameter NO.059 value is taken as the least cutting amount;

R(d): It is the cutting amount in thread finishing, range: 00~99.999 (unit: mm/inch, radius value without sign symbols), the radius value is equal to X absolute coordinates between cut-in point B_e of thread finishing and B_f of thread roughing. After R(d) is executed, the specified value d is reserved and the system parameter №060 value is rewritten to $d \times 1000$ (unit: 0.001 mm) . The value of system parameter №060 is regarded as the cutting travel of thread finishing when R(d) is not input

R(i): It is thread taper and is the different value of X absolute coordinate between thread starting point and end point, rang: -99999.999~99999.999(IS_B) /-9999.9999~9999.9999(IS_C) (unit: mm/inch, radius value). The system defaults $R(i)=0$ (straight thread) when R(i) is not input;

P(k): Depth of thread tooth, the total cutting depth of thread, range: 1~99999999 (unit: least input increment, radius value, without sign symbols). The system alarms when P(k) is not input;

Q(Δd): Depth of the 1st thread cutting, range: 1~99999999 unit: least input increment, radius

value, without sign symbols). The system alarms when Δd is not input;

F: metric thread pitch, its range is referred to Table 1-2;

I: thread teeth per inch for inch thread, its range is referred to Table 1-2.

J: movement amount (run-out amount) (unit: mm/inch, without direction) in the short axis when thread run-out; when the short axis is X, its value is designated by radius and is simple parameter;

K: movement amount (run-out amount) (unit: mm/inch, without direction) in the long axis when thread run-out; when the long axis is X, its value is designated by radius and is simple parameter.

Note: J, K are not compiled in G76, its thread run-out method is the same that of the previous, i.e. the thread

run-out is executed according to P® or No.19; when J or, K, or J, K are compiled, the thread run-out method is the same those of G32, G92.

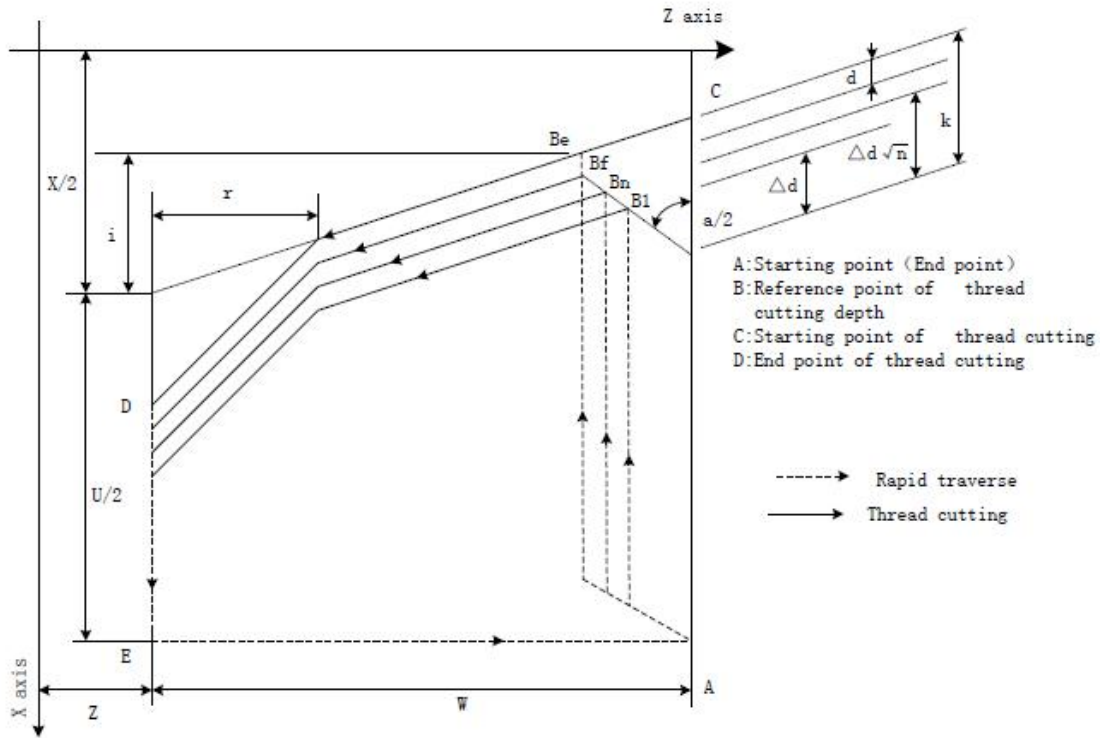


Fig. 3-48(a)

Cut-in method as follows: Fig. 3-48(b):

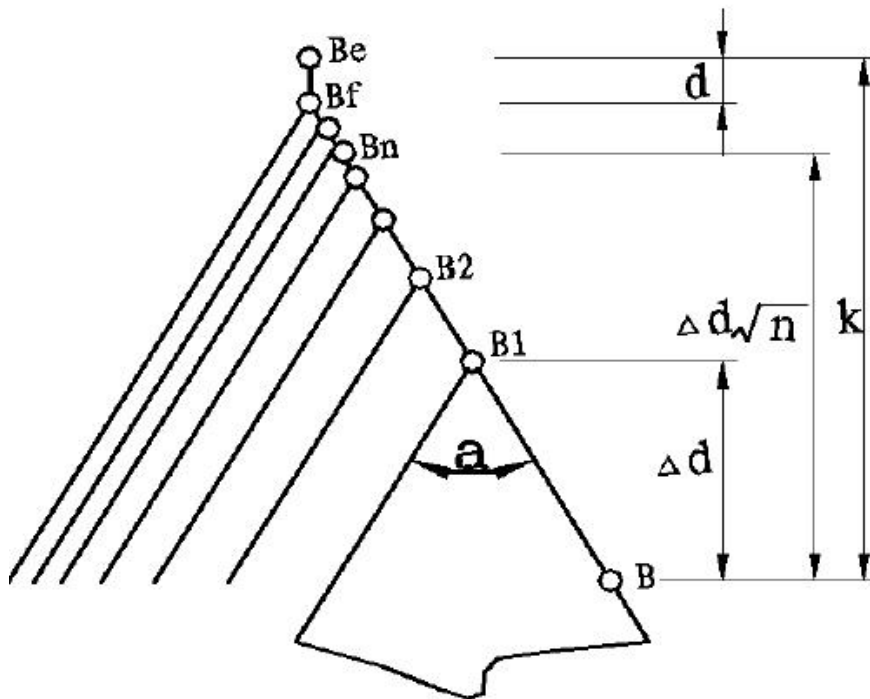


Fig. 3-48(b)

Pitch is defined to moving distance (X radius value) of long axis when the spindle rotates one rev. Z is long when absolute value of coordinate difference between C point and D point in Z direction is more than that of X direction (radius value, be equal to absolute value of i); and vice versa

Execution process:

- ① The tool rapidly traverses to B₁, and the thread cutting depth is Δd . The tool only traverses in

X direction when $a=0$; the tool traverses in X and Z direction and its direction is the same that of A→D when $a \neq 0$;

- ② The tool cuts threads paralleling with C→D to the intersection of D→E ($r \neq 0$: thread run-out);
 ③ The tool rapidly traverses to E point in X direction;
 ④ The tool rapidly traverses to A point in Z direction and the single roughing cycle is completed;
 ⑤ The tool rapidly traverses again to tool infeed to B_n (is the roughing times), the cutting depth

is the bigger value of $(n \times \Delta d)$, $(n - 1 \times \Delta d + \Delta d_{min})$, and execute ② if the cutting depth is

less than $(k-d)$; if the cutting depth is more than or equal to $(k-d)$, the tool infeeds $(k-d)$ to B_r, and then execute ⑥ to complete the last thread roughing;

- ⑥ The tool cuts threads paralleling with C→D to the intersection of D→E ($r \neq 0$: thread run-out);
 ⑦ X axis rapidly traverses to E point;
 ⑧ Z axis traverses to A point and the thread roughing cycle is completed to execute the finishing;
 ⑨ After the tool rapidly traverses to B(the cutting depth is k and the cutting travel is d), execute the thread finishing, at last the tool returns to A point and so the thread finishing cycle is completed;
 ⑩ If the finishing cycle time is less than m, execute ⑨ to perform the finishing cycle, the thread cutting depth is k and the cutting travel is 0; if the finishing cycle times are equal to m, G76 compound thread machining cycle is completed.

Notes:

- In thread cutting, execute the feed hold, the system displays **Pause** after the thread cutting is executed completely, and then the program run pauses;
 - Execute single block in thread cutting, the program run stops after returning to starting point(one thread cutting cycle is completed);
 - The thread cutting decelerates to stop when the system resets and emergently stop or the driver alarms;
- Omit all or some of G76 P(m) (r) (a) Q(Δd_{min}) R(d) . The omitted address runs according to

setting value of parameters;

- m, r, a used for one command address P are input one time. Program runs according to setting

value of №57, 19, 58 when m, r, a are all omitted; Setting value is a when address P is input with 1 or 2 digits; setting values are r, a when address P is input with 3 or 4 digits;

- The direction of A→C→D→E is defined by signs of U,W , and the direction of C→D is defined by the sign of R(i) . There are four kinds of sign composition of U, W corresponding to four kinds of machining path as Fig. 3-46.

Example: Fig. 3-49, thread M68×6.

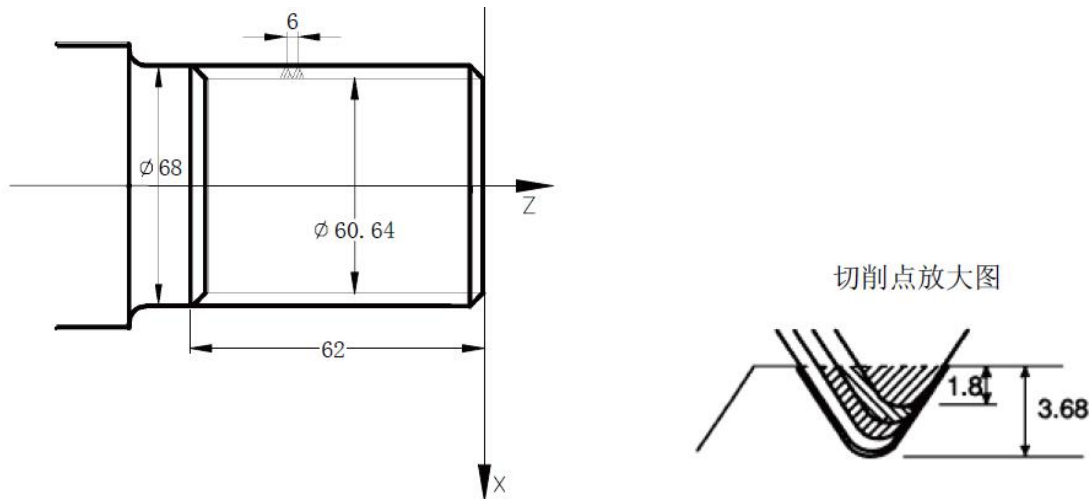


Fig.3-49

Program:

O0013;

G50 X100 Z50 M3 S300; (Set workpiece coordinate system, start spindle and specify spindle speed)

G00 X80 Z10; (Rapid traverse to starting point of machining)

G76 P020560 Q150 R0.1; (Finishing 2 times, chamfering width 0.5mm, tool angle 60°, min. cutting depth 0.15, finishing allowance 0.1)

G76 X60.64 Z-62 P3680 Q1800 F6; (Tooth height 3.68, the first cutting depth 1.8)

G00 X100 Z50 ; (Return to starting point of program)

M30; (End of program)

3.22 Constant Surface Speed Control G96, Constant Rotational

Speed Control G97

The detailed is referred to Section 2.2.3.

3.23 Feedrate per Minute G98, Feedrate per Rev G99

Command format: G98 F_; (its range is referred to Section 1.6.5, the leading zero can be omitted, feed rate per minute is specified, mm/min)

Command function: cutting feed rate is specified as mm/min, G98 is the modal G command. G98 cannot be input if the current command is G98 modal.

Command format: G99 F_; (its range is referred to Section 1.6.5, the leading zero can be omitted)

Command function: Cutting feed rate is specified as mm/min, G99 is the modal G

command.G99 input may be omitted if current state is G99. The actual cutting feedrate is gotten by multiplying the F command value (mm/r) to the current spindle speed(r/min). If the spindle speed varies, the actual feedrate changes too. If the spindle cutting feed amount per rev is specified by G99 FXXXX , the even cutting texture on the surface of workpiece will be gotten. In G99 state, a spindle encoder should be fixed on the machine tool to machine the workpiece. G98, G99 are the modal G commands in the same group and only one is valid. G98 is the initial state G command and the system defaults G98 is valid when the system turns on.

Reduction formula of feed between per rev and per min:

$$F_m = F_r \times S$$

F_m : feed per min (mm/min) ;

F_r : feed per rev (mm/r) ;

S : spindle speed (r/min) .

After the system turns on, the feedrate is ones set by No030 and F value is reserved after F is executed. The feed rate is 0 after F0 is executed. F value is reserved when the system resets and emergently stops. The feedrate override is reserved when the system is turned off.

Note: In G99 modal, there is the uneven cutting feed rate when the spindle speed is lower than 1 r/min; there is the follow error in the actual cutting feed rate when there is the swing in the spindle speed. To gain the high machining quality, it is recommended that the selected spindle speed should be not lower than min. speed of spindle servo or converter.

Relative Parameter:

No.027: cutting speed upper limit value **P**

No.029: cutting feed and manual feed index Acc&Dec time

No.030: cutting feed start and end speed

3.25 Macro Commands

ECN*000T provides the macro command which is similar to the high language, and can realize the variable assignment, and subtract operation, logic decision and conditional jump by user macro command, contributed to compiling part program for special workpiece, reduce the fussy counting and simplify the user program.

3.25.1 MACRO variables

Presentation of macro variables

Present with “#”+ macro variables number.;

Format: # i(i=100,102,103,.....) ;

Example: #105, #109, #125.

Variable Type

The variable is divided into four types according to the variable number:

Number NO. Variable type Function

Number NO.	Variable type	Function
#0	Null variable	The variable is null and is not valued.
#1~#33	Local variable	The local variable is used to store data in the macro program, such as result. When the system is turned off, the local variable is initialized to be null. When the macro program is called, the argument values to the local.
#100~#199 #500~#999	Share variable	The share variable has the same meaning in the different macro program. When the system is turned off, the variable #100~#199 is initialized to be null, #500~#999 is saved and is not lost.
#1000~	System variable	System variable

Macro variables reference

宏变量				00001 N0000	
序号	数据	序号	数据	序号	数据
100	125.200	110		120	
101		111	1.000	121	
102		112	3.125	122	
103		113		123	
104		114		124	
105		115		125	
106		116		126	
107		117		127	
108		118		128	
109		119		129	
公用变量					
序号 101 =					
				录入方式	S0000 T0000

1. Macro variables can replace command values

Format: < Address > +“# i” or < Address > +“-# I”. It shows the system takes variable value or negative value of variable value as address value.

Example: F#103...when #103=15, its function is the same that of F15;

Z-#110...when #110=250, its function is the same that of Z-250;

2. Macro variables can replace macro variables values.

Format: “#”+“9”+macro variables number

Example: if #100 = 205, #105 = 500,

The command function of X#9100 is the same as X500;

The command function of X-#9100 is the same as X-500

Note 1: The address O, G and N cannot refer macro variables. For example, O#100, G#101, N#120 are illegal;

Note 2: If macro variables values exceed the maximum rang of command values, they cannot be used. For example: #130 = 120, M#130 exceeds the maximum command value.

Variable display

- (1) In macro window, the variable being displayed to the null means it is null, i.e. it is not defined. The variable being displayed to “*****” means it exceeds the displayable range.
- (2)The share variable (#100~#199, #500~#999) values are displayed in the macro variable

window, and is also displayed the window, the data is input directly to value the share variable.

(3)The local variable (#1~#33) and the system variable values are not displayed. Some local variable or system variable value is displayed by assigned with the share variable.

System variable

- 1) Input signal: #1000 --- #1047 (Read PMC input signal)
- 2) Output signal #1100 --- #1147 (output to PMC signal)
- 3) X length compensation #1500 --- #1531 (radius ,can read/write)
- 4) Z length compensation #1600 --- #1631 (can read/write)
- 5) Y length compensation #1700 --- #1731 (can read/write)
- 6) Radius compensation #1800 --- #1831 (can read/write)
- 7) X wear compensation #1900 --- #1931 (radius ,can read/write)
- 8) Z wear compensation #2000 --- #2031 (can read/write)
- 9) Y wear compensation #2100 --- #2131 (can read/write)
- 10) Radius wear compensation #2200 --- #2231 (can read/write)
- 11) Alarm #3000
- 12) User data list table #3500 --- #3755 (read only)
- 13) Modal info.#4000 --- #4030 (read only)
- 14) position info #5001 --- #5030 (read only)

(1) Interface signal: CNC only executes G and F signals. Whether there are I/O to correspond to it is defined by PLC.

Variable No. Function

Variable No.	Function
#1000~#1015	Correspond G54.0~G54.7, G55.0~G55.7 signal states
#1032	Correspond G54, G55 signal states
#1100~#1115	Correspond F54.0~G54.7, F55.0~F55.7 signal states
#1132	Correspond F54, F55 signal states
#1133	Correspond F56, F57, F58, F59 signal states

(2) Tool compensation system variable:

Compensation No.	X compensation value		Z compensation value		Tool tip radius compensation value		Imagery tool tip position T	Y compensation value	
	Wear	Geometric shape	Wear	Geometric shape	Wear	Geometric shape		Wear	Geometric shape
1	#2001	#2701	#2101	#2801	#2201	#2901	#2301	#2401	#2451
...
32	#2032	#2732	#2132	#2832	#2232	#2932	#2332	#2432	#2482

(3) Machined workpiece number:

#3901 Machined workpiece number(completion)

(4) System modal information variable

Variable No.	Function	
#4001	G00, G01, G02, G03, G05, G32, G33, G34, G80, G84, G88, G90, G92, G94, G124(G06.2), G126(G06.3), G132(G32.1), G144(G07.2), G146(G07.3)	No. 1 group
#4002	G96, G97	No. 2 group
#4005	G98, G99	No. 3 group
#4006	G20, G21	No. 6 group
#4007	G40, G41, G42	No. 7 group
#4012	G66, G67	No. 12 group
#4013	G54, G55, G56, G57, G58, G59	No. 14 group
#4016	G17, G18, G19	No. 16 group
#4109	F command	
#4113	M command	
#4114	Serial No.	
#4115	Program No.	
#4119	S command	
#4120	T command	

(5) system variable of coordinate position information:

Variable No.	Position signal	Coordinate system	Tool compensation value	Read in running
#5001~#5005	End point of block	Workpiece coordinate system	Not including	Possible
#5021~#5025	Current position	Machine coordinate system	Including	impossible
#5041~#5045	Current position	Workpiece coordinate system		

Note: The position listed in the above table separately corresponds orderly to X, Y, Z, 4th, 5th axis

For example:

#5001 meanings to be X position information, #5002 meanings to be Y position information, #5003 meanings to be Z position information and #5004 meanings to 4th position information and #5005 meanings to 5th position information.

(6) zero offset value and workpiece coordinate systems

G54: #5206 ~ #5210

G55: #5211 ~ #5215

G56: #5216 ~ #5220

G57: #5221 ~ #5225

G58: #5226 ~ #5230

G59: #5231 ~ #5235

(7) address and local variable relationship

Independent Var.	Local Var.	Independent Var.	Local Var.	Independent Var.	Local Var.
A	#1	E	#8	U	#21
B	#2	F	#9	V	#22
C	#3	M	#13	W	#23
I	#4	Q	#17	X	#24
J	#5	R	#18	Y	#25
K	#6	S	#19	Z	#26
D	#7	T	#20		

3.2.5.2 Operation and jump command G65**Command format:**

G65 Hm P# i Q# j R# k;

m: operation or jump command, range 01~99.

I: macro variables name for storing values.

j: macro variables name 1 for operation, can be constant.

k: macro variables name 2 for operation, can be constant.

Command significance: # i = #j O # k

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;

Note: Macro variable name has no “#” when it is presented directly with constant.

Macro command list

Command format	Functions	Definitions
G65 H01 P#i Q#j	Assignment	# i = # j assign value of j to i
G65 H02 P#i Q#j R#k;	Decimal add operation	# i = # j + # k
G65 H03 P#i Q#j R#k;	Decimal subtract operation	# i = # j - # k

Command format	Functions	Definitions
G65 H04 P#i Q#j R#k;	Decimal multiplication operation	# i = # j×# k
G65 H05 P#i Q#j R#k;	Decimal division operation	# i = # j÷# k
G65 H11 P#i Q#j R#k;	Binary addition	# i = # j OR # k
G65 H12 P#i Q#j R#k;	Binary multiplication(operation)	# i = # j AND # k
G65 H13 P#i Q#j R#k;	Binary exclusive or	# i = # j XOR # k
G65 H21 P#i Q#j;	Decimal square root	# i = $\sqrt{\# j}$
G65 H22 P#i Q#j;	Decimal absolute value	# i = # j
G65 H23 P#i Q#j R#k;	Decimal remainder	Remainder of # i = (#j÷# k)
G65 H24 P#i Q#j;	Decimal into binary	# i = BIN(# j)
G65 H25 P#i Q#j;	Binary into decimal	# i = DEC(# j)
G65 H26 P#i Q#j R#k;	Decimal multiplication/division operation	# i = # i×# j÷# k
G65 H27 P#i Q#j R#k;	Compound square root	# i = $\sqrt{\# j^2 + \# k^2}$
G65 H31 P#i Q#j R#k;	Sine	# i = # j×sin(# k)
G65 H32 P#i Q#j R#k;	Cosine	# i = # j×cos(# k)
G65 H33 P#i Q#j R#k;	Tangent	# i = # j×tan(# k)
G65 H34 P#i Q#j R#k;	Arc tangent	# i = ATAN(# j / # k)
G65 H80 Pn;	Unconditional jump	Jump to block n

G65 H81 Pn Q#j R#k;	Conditional jump 1	Jump to block n if # j = # k, otherwise the system executes in order
G65 H82 Pn Q#j R#k;	Conditional jump 2	Jump to block n if # j ≠ # k, otherwise the system executes in order
G65 H83 Pn Q#j R#k;	Conditional jump 3	Jump to block n if # j > # k, otherwise the system executes in order
G65 H84 Pn Q#j R#k;	Conditional jump 4	Jump to block n if # j < # k, otherwise the system executes in order
G65 H85 Pn Q#j R#k;	Conditional jump 5	Jump to block n if # j ≥ # k, otherwise the system executes in order
G65 H86 Pn Q#j R#k;	Conditional jump 6	Jump to block n if # j ≤ # k, otherwise the system executes in order
G65 H99 Pn;	P/S alarm	(500+n) alarms

1 Operation commands

1) Assignment of macro variables: # 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) Decimal 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) Decimal subtract 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) Decimal 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) Decimal 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 logic add(or) : # 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 logic multiply(and) : # I = # J.AND. # K

G65 H12 P#I Q#J R#K

(Example) G65 H12 P# 201 Q#102 R#103; (#101 = #102.AND.#103)

8) Binary executive or: # 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) Decimal square root: # I = #J

G65 H21 P#I Q#J

(Example) G65 H21 P#101 Q#102 ; (#101= #102)

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

G65 H22 P#I Q#J

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

11) Decimal remainder: # I = # J - TRUNC(#J/#K) × # K, TRUNC: omit decimal fraction

G65 H23 P#I Q#J R#K

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

12) Decimal converting into binary: # I = BIN (# J)

G65 H24 P#I Q#J

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

13) Binary converting into decimal: # I = BCD (# J)

G65 H25 P#I Q#J

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

14) Decimal multiplication/division operation: # 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) Compound square root: # I = # J₂ + #K₂

G65 H27 P#I Q#J R#K

(Example) G65 H27 P#101 Q#102 R#103; (#101 = #102₂ + #103₂)

16) Sine: # 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) Cosine: # I = # J • COS(# K) (Unit: ‰)

G65 H32 P#I Q#J R# k

(Example) G65 H32 P#1Q#102 R#103; (#101 = #102•COS(#103))

18) Tangent: # 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) Cosine: # 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))

2 Jump commands

1) Unconditional jump

G65 H80 Pn; n: Block number

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

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

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

(Example) G65 H81 P1000 Q#101 R#102;

The program jumps N1000 when # 101=#102 and executes in order when #101 ≠#102.

3) Conditional jump 2 #J.NE.# K (≠)

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

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

The program jumps N1000 when # 101 ≠ #102 and executes in order when #101 = #102.

4) Conditional jump 3 #J.GT.# K (>)

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

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

The program jumps N1000 when # 101 > #202 and executes in order when #101 ≤#102.

5) Conditional jump 4 #J.LT.# K (< =)

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

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

The program jumps N1000 when # 101 < #102 and executes in order when #101 ≥#102.

6) Conditional jump 5 #J.GE.# K (≥)

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

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

The program jumps N1000 when # 101 ≤ #1 and executes in order when #101 < #102.

7) Conditional jump 6 #J.LE.# K (≤)

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

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

8) P/S alarm

G65 H99 Pi; i: alarm number +500

(Example) G65 H99 P15;

P/S alarm 515.

Note: Block number can be specified by variables. Such as: G65 H81 P#100 Q#101 R#102;

The program jumps to block that its block number is specified by #100.

3.25.3 Program example with macro command

Differences between user macro program call (G65, G66) and subprogram call (M98) are as follows:

1. G65, G66 can specify the argument data and send them to macro program and M98 has no such

function.

2. G65, G66 can change the level of local variable and M98 has no such function.
3. G65, G66 only follows N and only P or H follows them.

3.25.3.1 Non-modal call G65

Command format: G65 P_ L_ <argument> _;

Macro program specified by P is called, the argument (data) is send to the user macro program body.

Command explanation:

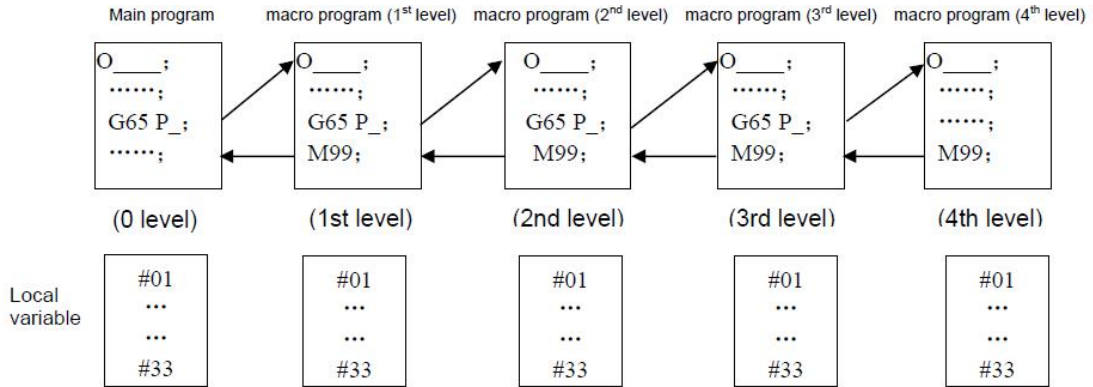
P —— called macro program number

L —— called times (it is 1 when it is omitted, it can be the repetitive times from 1 to 9999)

<argument> —— data sent to macro program is valued with the corresponding local variable.

Nest call: G65 call has four-level nest.

Main program macro program (1st level) macro program (2nd level) macro program (3rd level) macro program (4th level)



Specifying argument: the argument can be specified by two forms.

Method 1: use the letter besides G, L, O, N, P, and each is only specified one time, and the last which is specified many times is valid.

Argument address and corresponding variable No. table in method 1

Address	Variable No.	Address	Variable No.	Address	Variable No.
A	#1	I	#4	T	#20
B	#2	J	#5	U	#21
C	#3	K	#6	V	#22
D	#7	M	#13	W	#23
E	#8	Q	#17	X	#24
F	#9	R	#18	Y	#25
H	#11	S	#19	Z	#26

Note: The addresses which are not needed to specify can be omitted, the corresponding local variable of the omitted address is valued by <null>.

Method II: use A, B, C and li, Ji, Ki (I is 1~10), the used letter and executed times(I, J, K) automatically decides the corresponding variable number of argument. The argument in the method specifies A, B, C one time for each and I, J, K up to 10 times (10 times replacing the more).

Argument address and corresponding variable No. table in method II

Address	Variable No.	Address	Variable No.	Address	Variable No.
A	#1	K ₃	#12	J ₇	#23
B	#2	I ₄	#13	K ₇	#24
C	#3	J ₄	#14	I ₈	#25
I ₁	#4	K ₄	#15	J ₈	#26
J ₁	#5	I ₅	#16	K ₈	#27
K ₁	#6	J ₅	#17	I ₉	#28
I ₂	#7	K ₅	#18	J ₉	#29
J ₂	#8	I ₆	#19	K ₉	#30
K ₂	#9	J ₆	#20	I ₁₀	#31
I ₃	#10	K ₆	#21	J ₁₀	#32
J ₃	#11	I ₇	#22	K ₁₀	#33

Note 1: The subscripts of I, J, K are used to confirming the specified sequence of argument, and are not written in the actual programming.

Note 2: The system can identify the variable number according to the present sequence and times of I, J, K in the method.

If the block has: G65 P9010 A1 B2 C3 I14 J15 I6 J7 K9 K11 K12 J30;

Call program O9010, transfer the argument A1 to the local variable, and their corresponding relationship as follows:

#1=1, #2=2, #3=3, #4=14, #5=15, #7=6, #8=7, #6=9, #9=11, #12=12, #11=30;

The specified method I, II compound of argument: CNC can automatically identifies the specified method I and II of the argument. When the two methods are specified together, the later specifies is valid.

Argument specifying mode I, II compound:

CNC internal can automatically identify the argument specifying mode I and II. When the two modes are specified, the later specified is valid.

3.27 Metric/Inch Switch

Command format: G20; (inch input)

G21; (metric input)

Explanation: input/output unit of CNC system is divided into two, i.e., metric unit: mm and inch unit:

inch. Input unit of modifying the system has also:

Modifying BIT0 (metric/inch): 0: metric input 1: inch input No. 001# completely corresponds to G20/G21, namely, the parameter also changes when G20/G21 is executed; G20/G21 mode also changes when the parameter is modified, and the alarm prompt occurs after modification. Parameters relevant to metric/inch are referred to Installation & Connection, Section 3.2.17.

Notes

① **No.001 # 0(INI) input increment unit change**

① After the input increment unit is changed (inch/metric input), the following unit system is changed: (i.e.: mm<>inch; mm/min<>inch/min):

- F specifies the feedrate (mm/min<>inch/min), thread lead (mm <>inch)
- position command (mm<>inch)
- tool compensation value (mm<>inch)
- MPG graduation unit (mm<>inch)
- movement distance in incremental feed (mm<>inch)
- some data parameters, including NO.49~NO.54, NO.56, NO.59, NO.60, NO.114~ NO.116, NO.120~ NO.127, NO.140, No.141, No.154; the unit is 0.001mm(IS-B) in the metric input system, is 0.0001inch(IS-B) in the inch input system. For example, the same parameter NO.49 setting value is 100m, it means to be 0.1mm in the metric input system (G21), and it means 0.01inch in the inch input system (G20).

② The machine coordinates will automatically switch after the input increment unit change is switched:

(2) **No.004#0(SCW) output command unit change**

SCW=0: the system minimal command increment uses the metric output (0.001mm)

SCW=1: the system minimal command increment uses the inch output (0.0001inch)

Some data parameter meanings will be changed when the output control bit parameter SCW is changed:

① Speed parameter:

Metric machine: mm/min

Inch machine: 0.1 inch/min

Example: when the speed is set to 3800, the metric machine is 3800 mm/min and the inch machine is 380 inch/min.

Speed parameters: No.22, No.23, No.27, No.28~No.31, No.32, No.33, No.41, No.107, No.113;

② Position(length) parameter

metric machine: 0.001 mm

inch machine: 0.0001 inch

When the setting is 100, the metric machine is 0.1mm and the inch machine is 0.01 inch. Position parameters: No.34, No.35, No.37~No.40, No.45~No.48, No.102~No.104 and all pitch error compensation parameter;

Note 1: When the minimal input increment unit and the minimal command unit are different, the maximal error

is the half of minimal command unit. The error cannot be accumulated.

Note 2: The current system increment is IS-B in the above explanation.

CHAPTER 4

TOOL NOSE RADIUS COMPENSATION (G41, G42)

4.1 Application

4.1.1 Overview

Part program is compiled generally for one point of tool according to a workpiece contour. The

point is generally regarded as the tool nose A point in an imaginary state (there is no imaginary tool nose point in fact and the tool nose radius can be omitted when using the imaginary tool nose point to program) or as the center point of tool nose arc (as Fig. 4-1). Its nose of turning tool is not the imaginary point but one arc owing to the processing and other requirement in the practical machining.

There is an error between the actual cutting point and the desired cutting point, which will cause the over- or under-cutting affecting the part precision. So a tool nose radius compensation is needed in machining to improve the part precision.

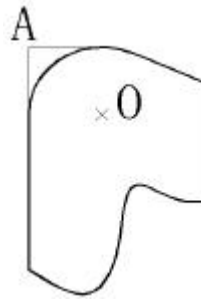


Fig. 4-1 Tool

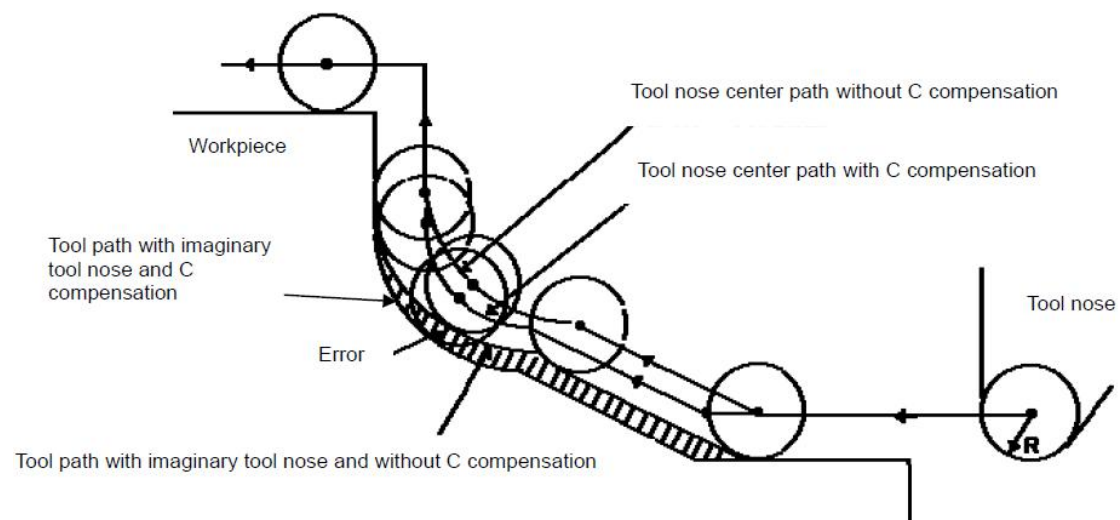


Fig. 4-2 Tool nose center path

4.1.2 Imaginary tool nose direction

Suppose that it is generally difficult to set the tool nose radius center on the initial position as Fig.4-3; suppose that it is easily set the tool nose on it as Fig. 4-4; The tool nose radius can be omitted in programming. Fig. 4-5 and Fig.4-6 correspond separately to the tool paths of tool nose center programming and imaginary tool nose programming when tool nose radius is executed or not.

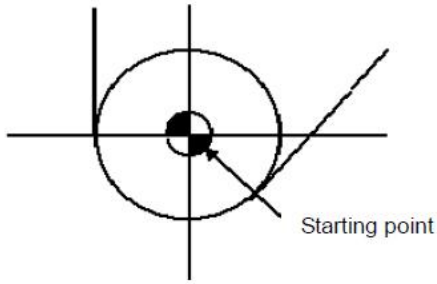


Fig. 4-3 Programming with tool nose

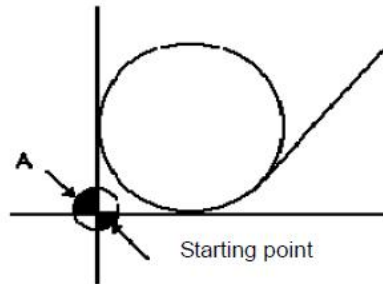


Fig. 4-4 Programming with imaginary tool nose

Tool path is shown in Fig.4-5 when the tool nose center programming is executed.

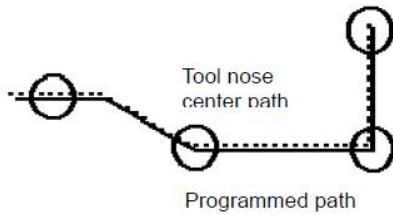


Fig. 4-5-1 the tool nose center path is the same that of programmed path without radius compensation

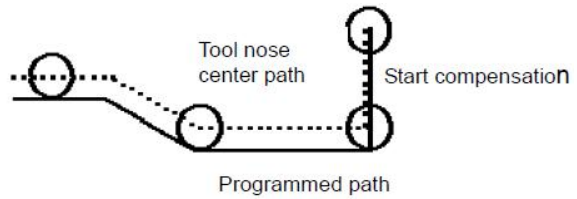


Fig. 4-5-2 precise cutting with tool radius compensation

The tool path is shown in Fig. 4-6 when imaginary tool nose programming is executed

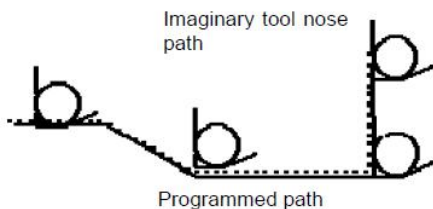


Fig. 4-6-1 the imaginary tool nose path is the same that of programmed path without tool radius compensation

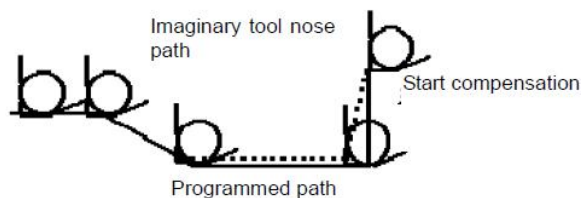


Fig. 4-5-2 precise cutting with tool radius compensation

The tool is supposed to one point in programming but the actual cutting blade is not one ideal point owing to machining technology. Because the cutting blade is not one point but one circular, machining error is caused which can be deleted by tool nose circular radius compensation. In actual machining, suppose that there are different position relationship between tool nose point and tool nose circular center point, and so it must create correct its direction of imaginary tool nose.

From tool nose center to imaginary tool nose, set imaginary tool nose numbers according to tool direction in cutting. Suppose there are 10 kinds of tool nose setting and 9 directions for position relationship. The tool nose directions are different in different coordinate system (rear tool post coordinate system and front tool post coordinate system) even if they are the same tool nose direction numbers as the following figures. In figures, it represents relationships between tool nose

and starting point, and end point of arrowhead is the imaginary tool nose; T1~T8 in rear tool post coordinate system is as Fig. 4-7; T1~T8 in front tool post coordinate system is as Fig. 4-8.
 The tool nose center and starting point for T0 and T9 are shown in Fig. 4-9.

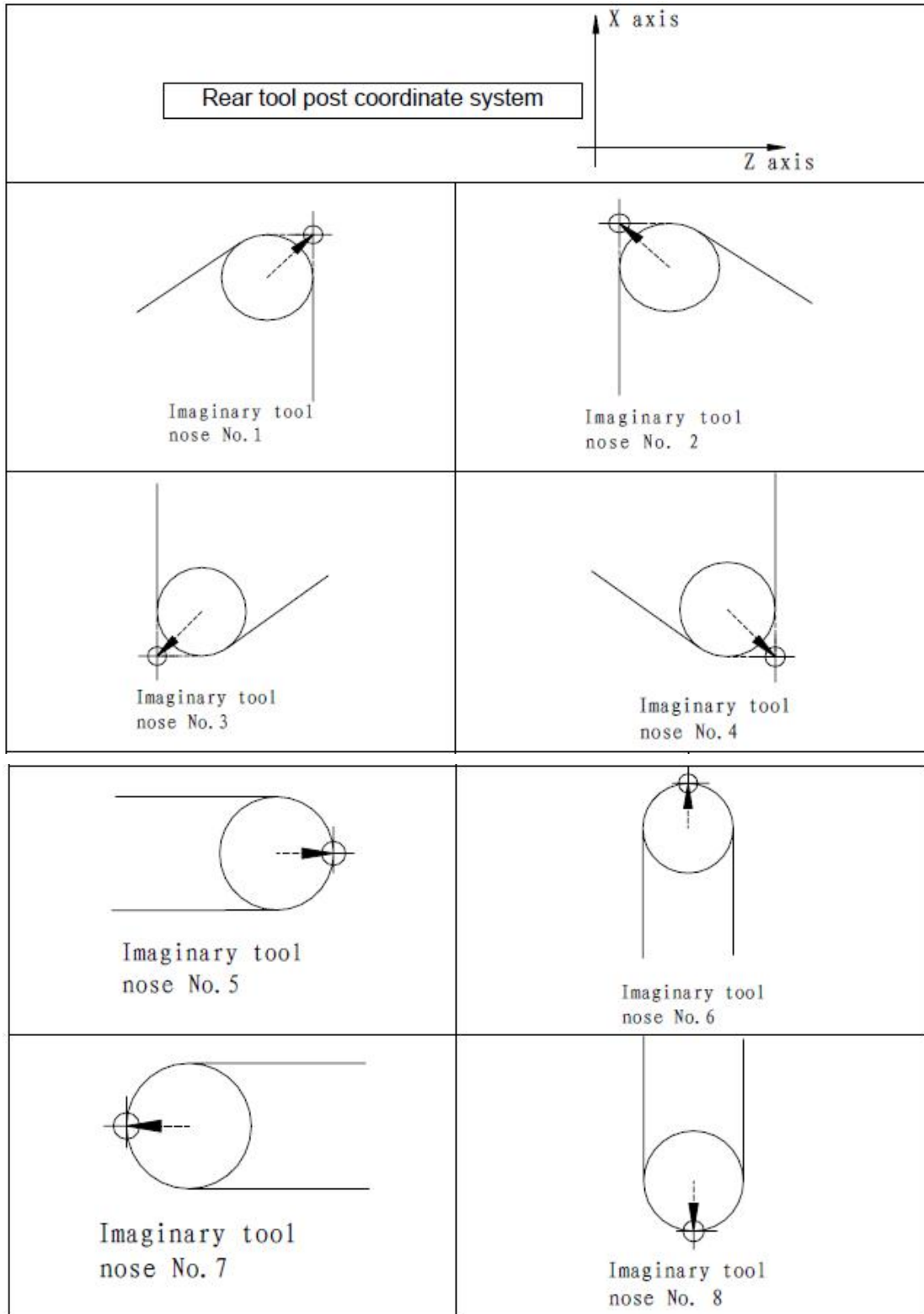


Fig. 4-7 Imaginary tool nose number in rear tool post coordinate system

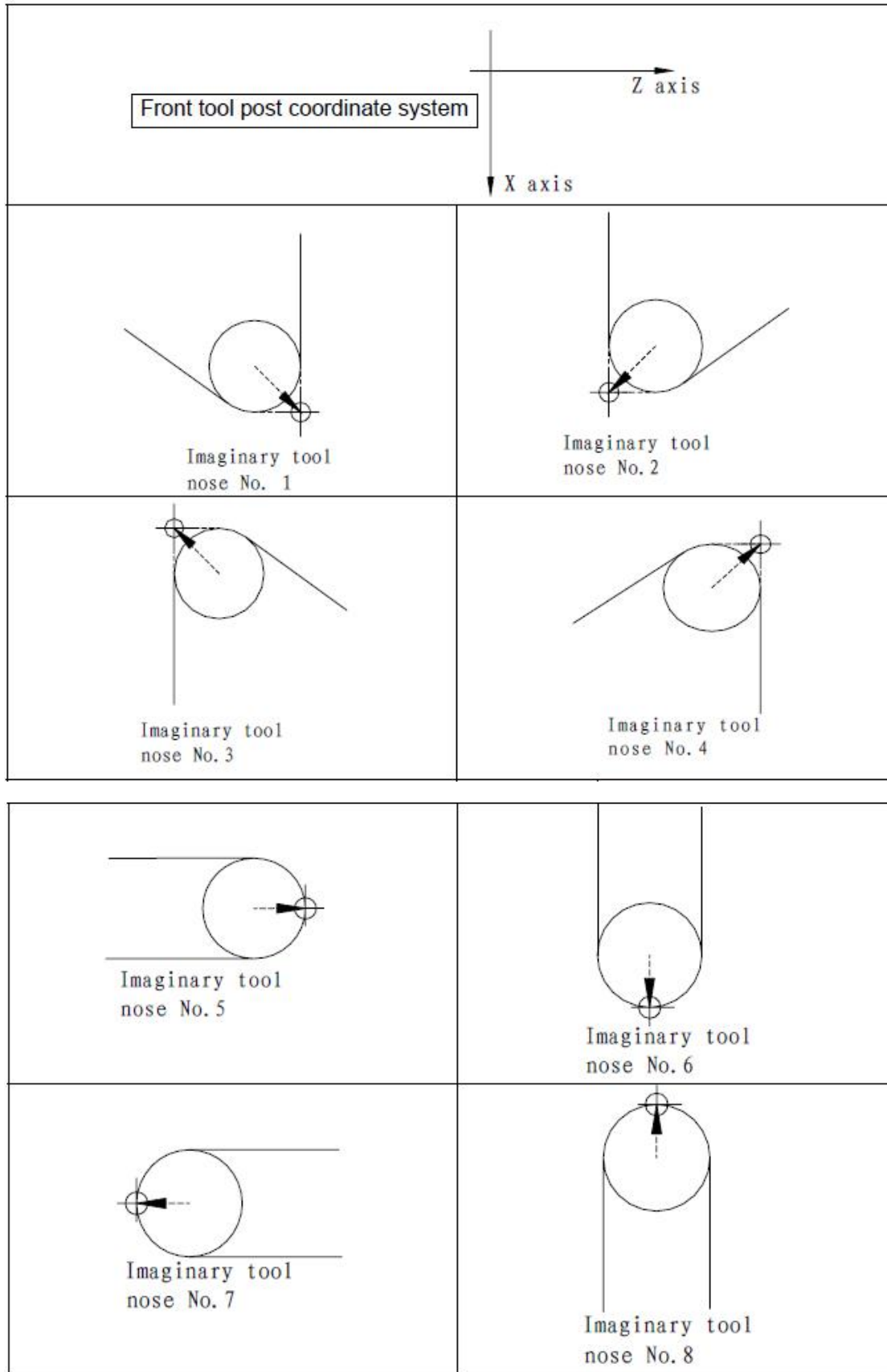


Fig. 4-8 Imaginary tool nose number in front tool post coordinate system

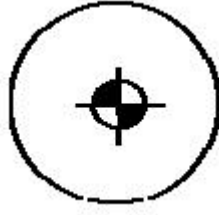


Fig. 4-9 Tool nose center on starting point

4.1.3 Compensation value setting

Preset imaginary tool nose number and tool nose radius value for each tool before executing tool nose radius compensation. Set the tool nose radius compensation value in **OFFSET** window (as Fig. 4-1), R is tool nose radius compensation value and T is imaginary tool nose number.

number	X	Z	R	T
000	0.000	0.000	0.000	0
001	0.020	0.030	0.020	2
002	1.020	20.123	0.180	3
...
032	0.050	0.038	0.300	6

Table 4-1 CNC tool nose radius compensation value display window

Note: X tool offset value can be specified in diameter or radius, set by No.004 Bit4 ORC, offset value is in radius when ORC=1 and is in diameter when ORC=0.

In toolsetting, the tool nose is also imaginary tool nose point of T_n (n=0~9) when taking T_n(n=0~9) as imaginary tool nose. For the same tool, offset value from standard point to tool nose radius center (imaginary tool nose is T₃) is different with that of ones from standard point to imaginary

tool nose (imaginary tool nose is T₃) when T₀ and T₃ tool nose points are selected to toolsetting in rear tool post coordinate system, taking tool post center as standard point. It is easier to measure distances from the standard point to the tool nose radius center than from the standard point to the imaginary tool nose, and so set the tool offset value by measuring distance from the standard point to the imaginary tool nose

(tool nose direction of T₃).

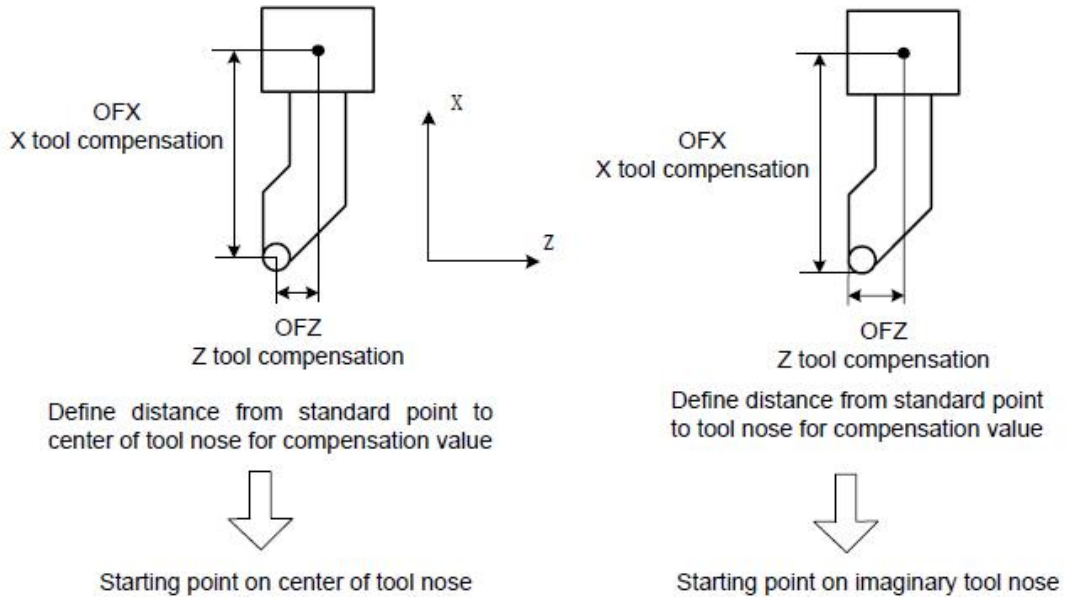


Fig. 4-10 Tool offset value of tool post center as benchmark

4.1.4 Command format

$$\left. \begin{matrix} G40 \\ G41 \\ G42 \end{matrix} \right\} \left\{ \begin{matrix} G00 \\ G01 \end{matrix} \right\} X_ Z_ T_ ;$$

Commands	Function specifications	Remark
G40	Cancel the tool nose radius compensation	See Fig.4-11 and 4-12
G41	Tool nose radius left compensation is specified by G41 in rear tool post coordinate system and tool nose radius right compensation is specified by G41 in front tool post coordinate system	
G42	Tool nose radius right compensation is specified by G42 in rear tool post coordinate system and tool nose radius left compensation is specified by G42 in front tool post coordinate system	

4.1.5 Compensation direction

Specify its direction according to relative position between tool nose and workpiece when executing tool nose radius compensation is shown in Fig. 4-11 and Fig.4-12.

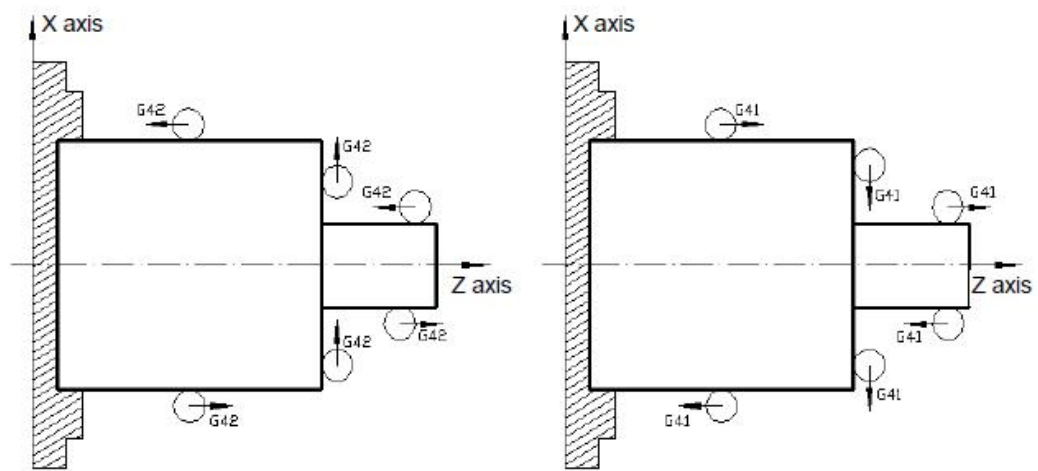
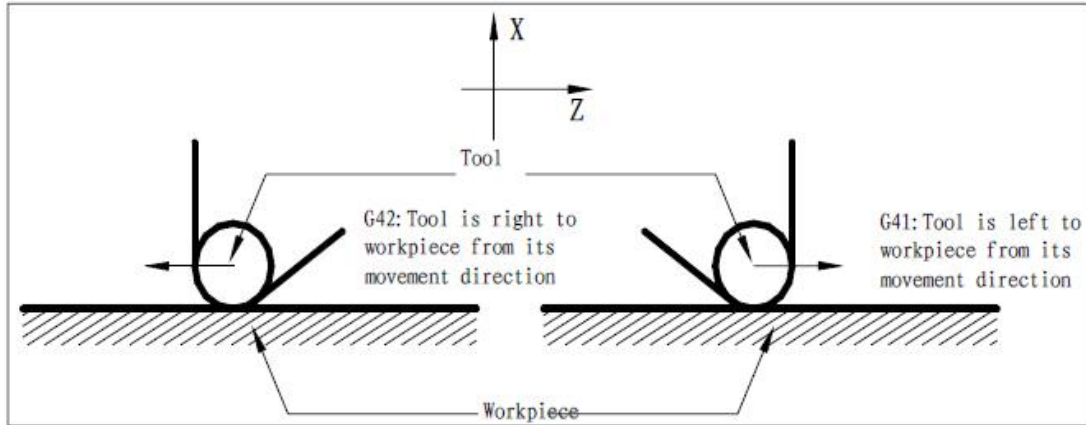
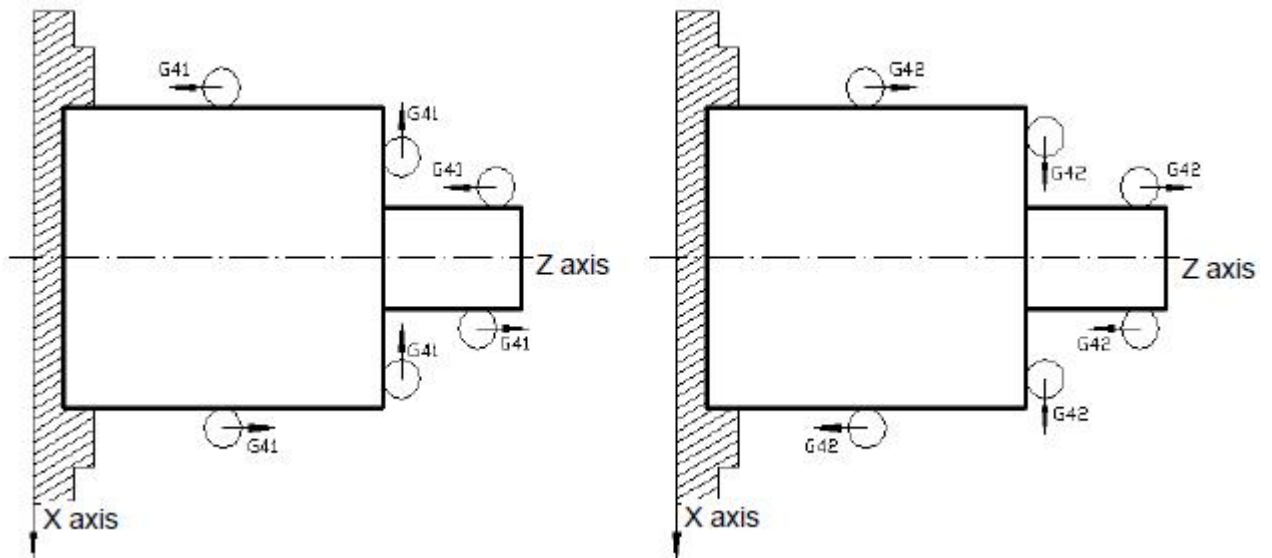
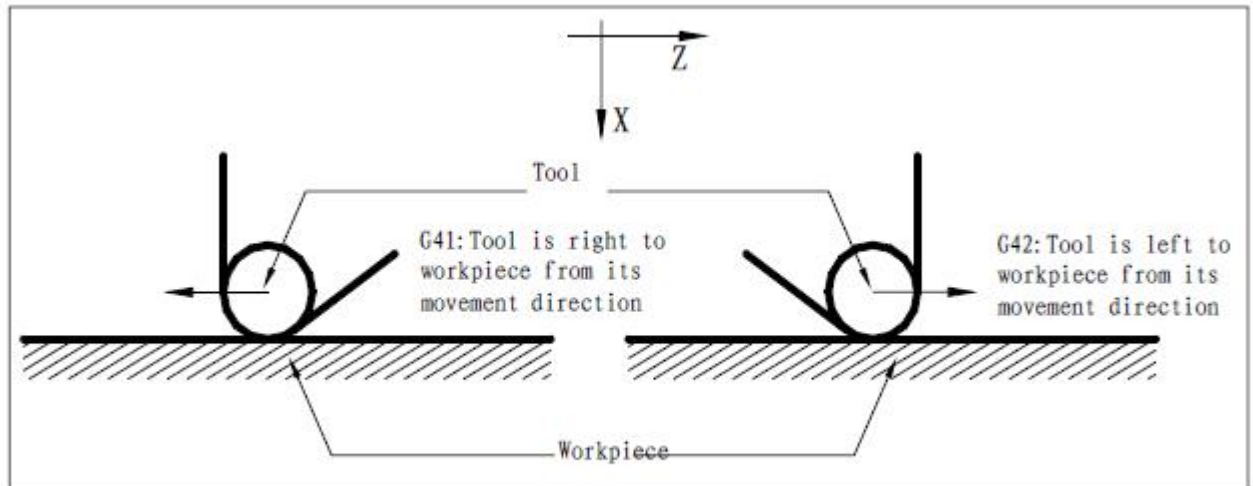


Fig. 4-11 Compensation direction of rear coordinate system



4.1.6 Notes

The system is in tool nose radius compensation mode at initial state, and starts to create tool nose radius compensation offset mode when executing G41 or G42. When the system starts to execute compensation, it pre-read two blocks, and the next block is saved to storage for tool nose radius compensation when executing one of them. The system reads two blocks in **“Single”** mode and stops after executing end point of the first block.

In tool nose radius compensation mode, the tool nose center moves to end point of previous block and is vertical to its path when the system executes two block or more than blocks without motion Command.

The system cannot create and cancel tool nose radius compensation.

Tool nose radius R is without negative value, otherwise there is a mistake running path.

Tool nose radius compensation is created and cancelled in G00 or G01 instead of G02 or G03, otherwise, the system alarms.

The system cancels the tool nose radius compensation mode when pressing RESET key.

G40 must be specified to cancel offset mode before the program is ended, otherwise the tool

path offsets one tool nose radius.

The system executes the tool nose radius compensation in main program and subprogram but must cancel it before calling subprogram and then create it again in the subprogram.

The system does not execute the tool nose radius compensation in G71, G72, G73, G74, G75, G76 and cancel it temporarily.

The system executes the tool nose radius compensation in G90, G94, it offsets one tool nose radius for G41 or G42.

4.1.7 Application

Machine a workpiece in the front tool post coordinate system as Fig. 4-13. Tool number: T0101, tool nose radius R=2, imaginary tool nose number T=3.

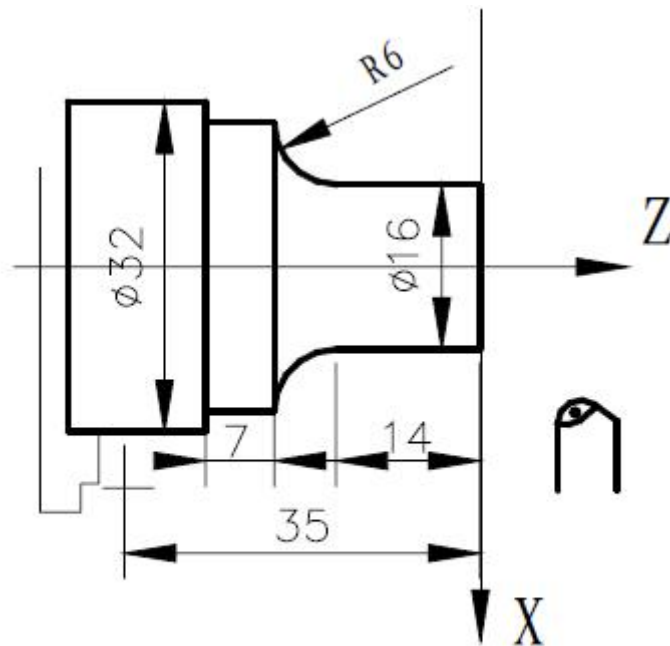


Fig. 4-13

For toolsetting in Offset Cancel mode, after toolsetting, Z axis offsets one tool nose radius and its direction is relative to that of imaginary tool nose and toolsetting point, otherwise the system excessively cuts tool nose radius when it starts to cut.

Set the tool nose radius R and imaginary tool nose direction in “**TOOL OFFSET&WEAR**” window as following:

Table 4-3

No.	X	Z	R	T
001			2.000	3
002
...
007
008

Program:

G00 X100 Z50 M3 T0101 S600; (Position, start spindle, tool change and execute

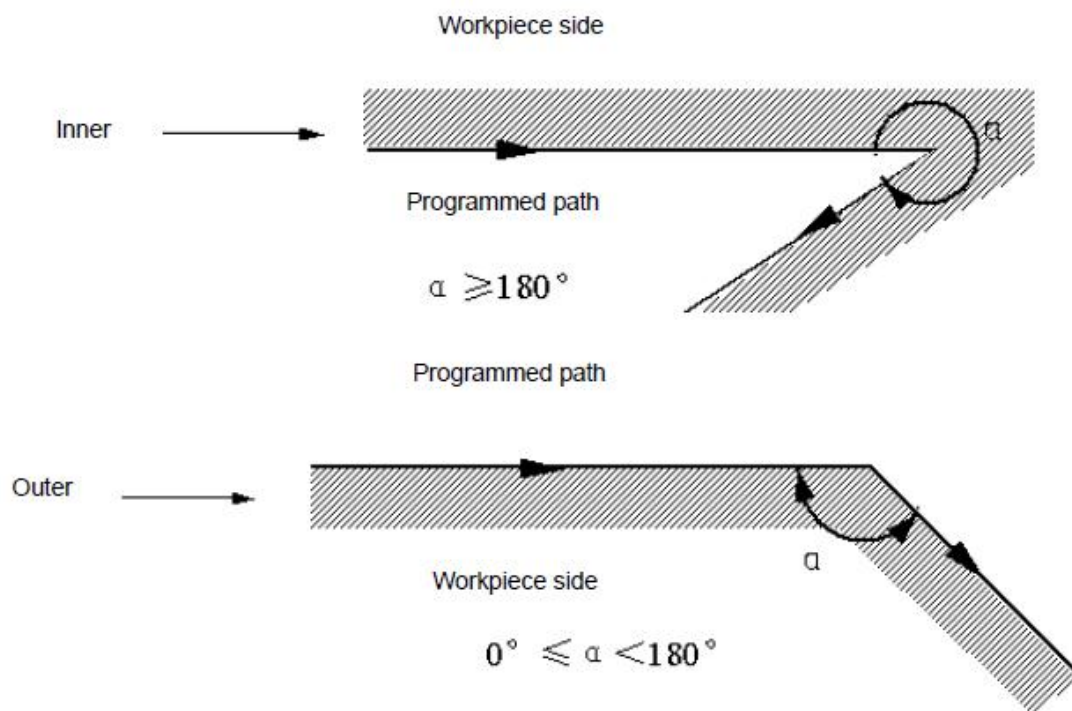
tool compensation)
 G42 G00 X0 Z3; (Set tool nose radius compensation)
 G01 Z0 F300; (Start cutting)
 X16;
 Z-14 F200;
 G02 X28 W-6 R6;
 G01 W-7;
 X32;
 Z-35;
 G40 G00 X90 Z40; (Cancel tool nose radius compensation)
 G00 X100 Z50 T0100;
 M30;

4.2 Tool Nose Radius Compensation Offset Path

4.2.1 Inner and outer side

Inside is defined that an angle at intersection of two motion blocks is more than or equal to 180° ;

Outside is $0^\circ \sim 180^\circ$.



3 steps to execute tool nose radius compensation: tool compensation creation, tool compensation execution and tool compensation canceling.

Tool traverse is called tool compensation creation (starting tool) from offset canceling to G41 or G42 execution.

Note: Meanings of S, L, C in the following figures are as follows: S—Stop point of single block; L—linear; C—circular.

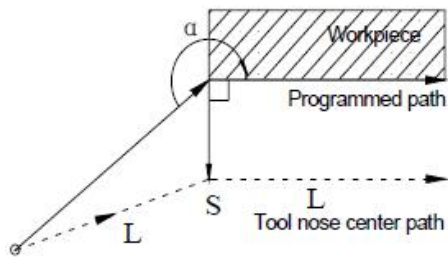
Note: Tool movement path is marked with the tool nose direction θ in the following figures.

Note: Tool path is described in rear tool post coordinate system and tool compensation direction G41 in the following figures.

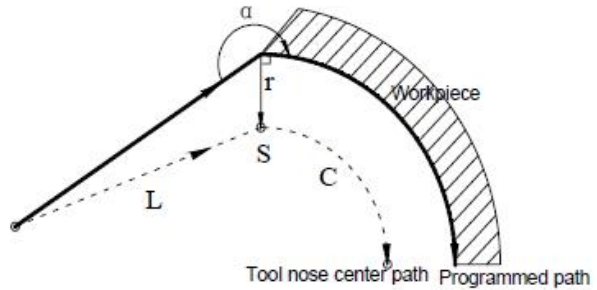
4.2.2 Tool traversing when starting tool

(a) Tool traversing inside along corner ($\alpha \geq 180^\circ$)

1) Linear \rightarrow linear

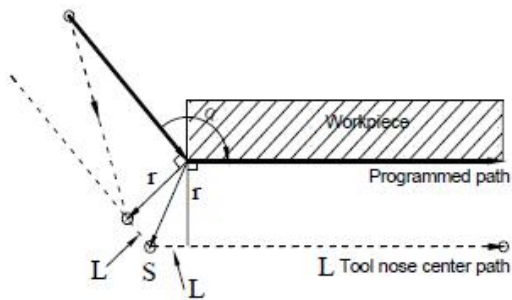


2) linear \rightarrow circular

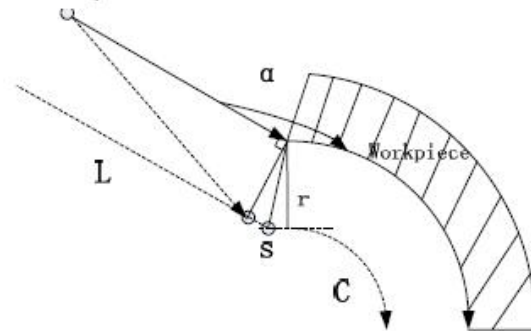


(b) Tool traversing inside along corner ($180^\circ > \alpha \geq 90^\circ$)

1) Linear \rightarrow linear

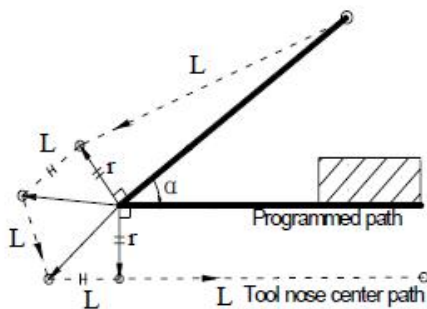


2) linear \rightarrow circular

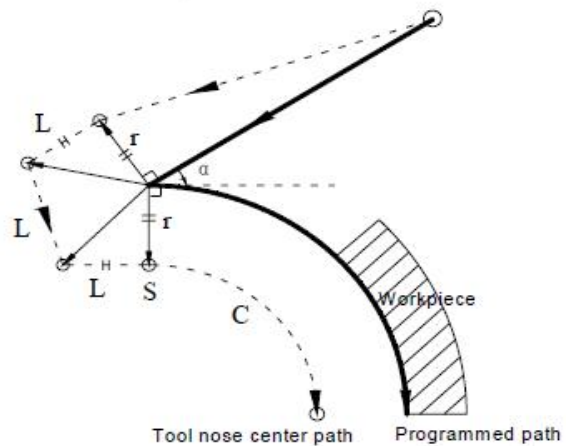


(c) Tool traversing inside along corner ($\alpha < 90^\circ$)

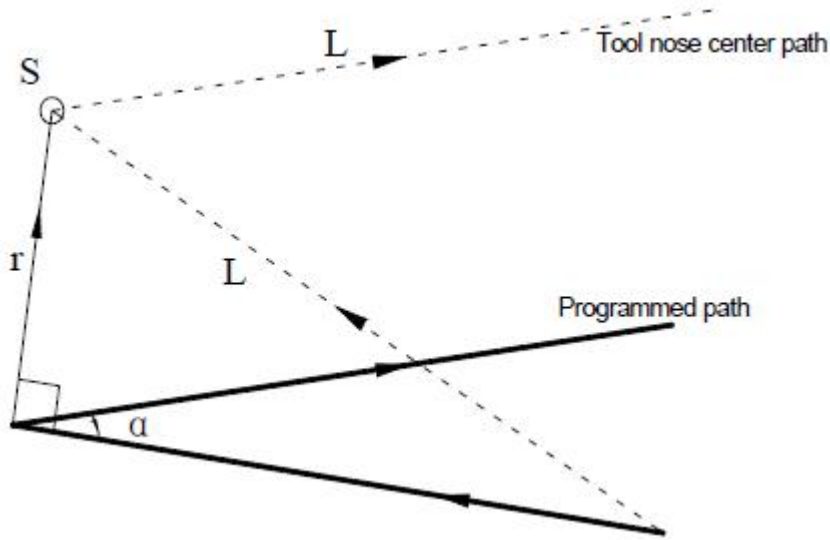
1) Linear \rightarrow linear



2) Linear \rightarrow circular



(d) Tool traversing outside along corner , linear \rightarrow linear ($\alpha \leq$ angle set by No. 237)



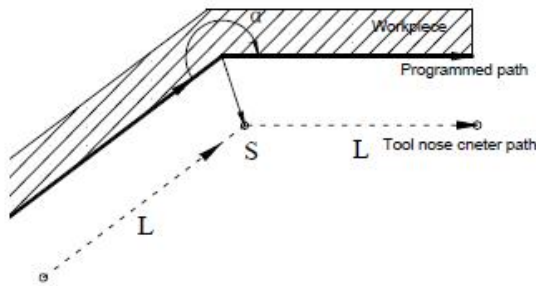
4.2.3 Tool traversing in Offset mode

Offset mode is called to ones after creating tool nose radius compensation and before canceling it.

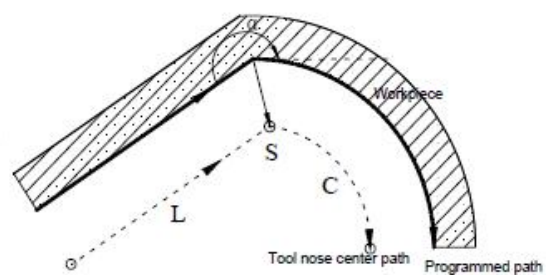
Offset path without changing compensation direction in compensation mode

(a) Tool traversing inside along corner ($\alpha \geq 180^\circ$)

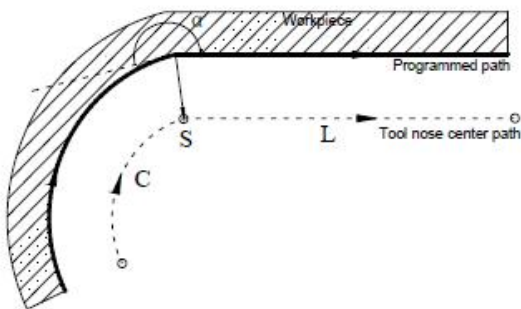
1) Linear \rightarrow linear



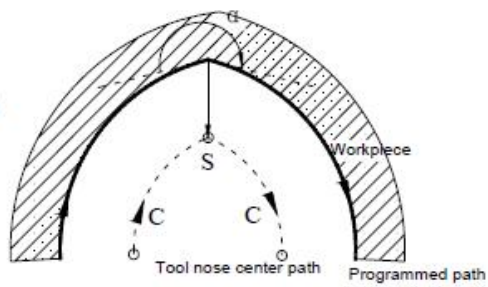
2) Linear \rightarrow circular



3) Circular \rightarrow linear



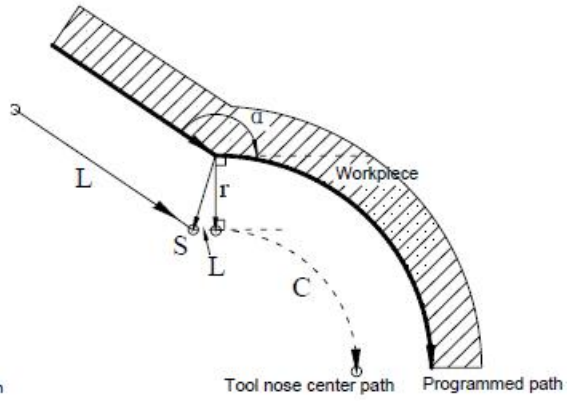
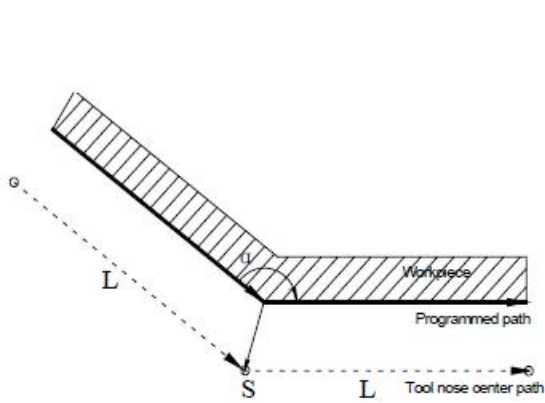
4) Circular \rightarrow circular



(b) Tool traversing outside along corner ($180^\circ > \alpha \geq 90^\circ$)

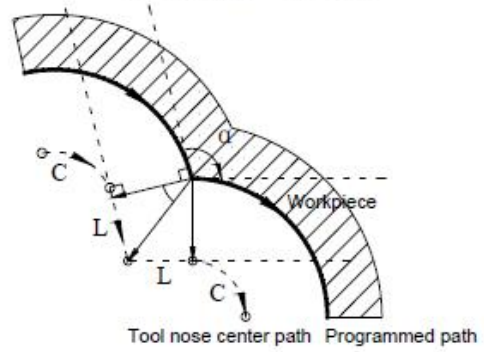
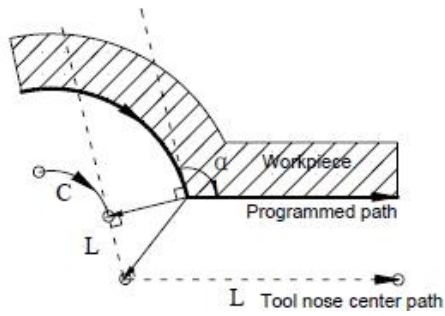
1) Linear \rightarrow linear 2)

2) Linear \rightarrow circular



3) Circular→Linear

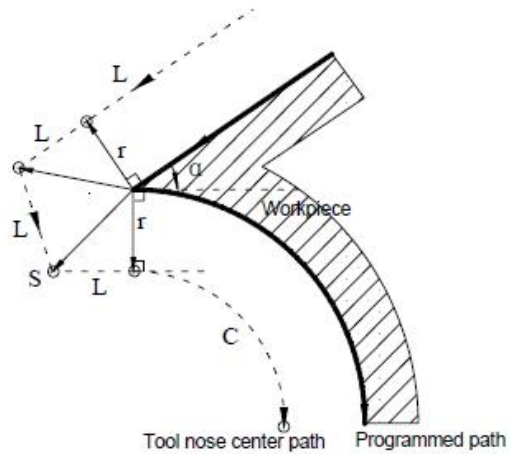
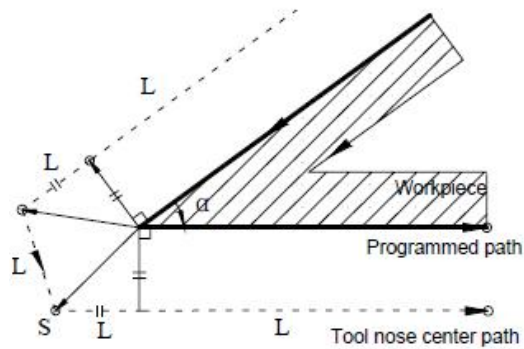
4) Circular→Circular



(c) Tool traversing outside along corner($\alpha < 90^\circ$)

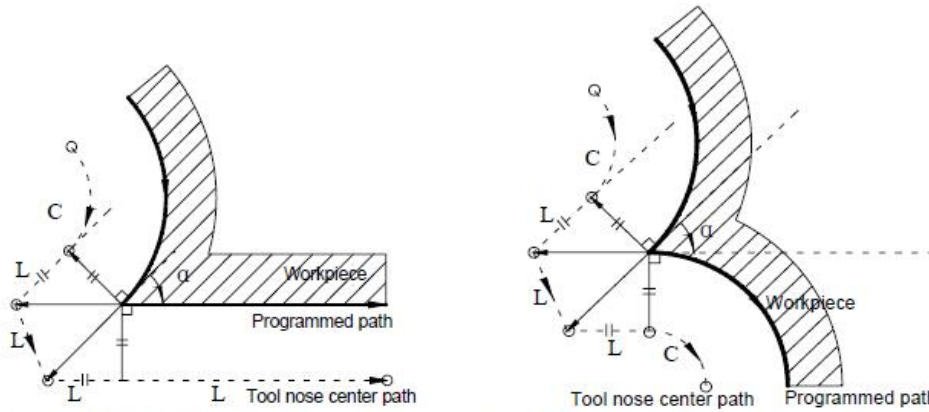
1) Linear→linear

2) linear→circular

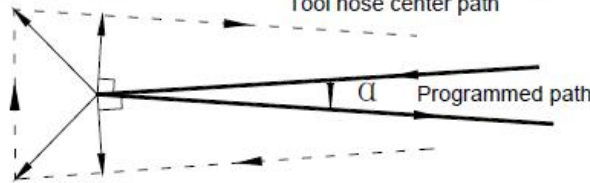


3) Circular→linear

4) Circular→circular



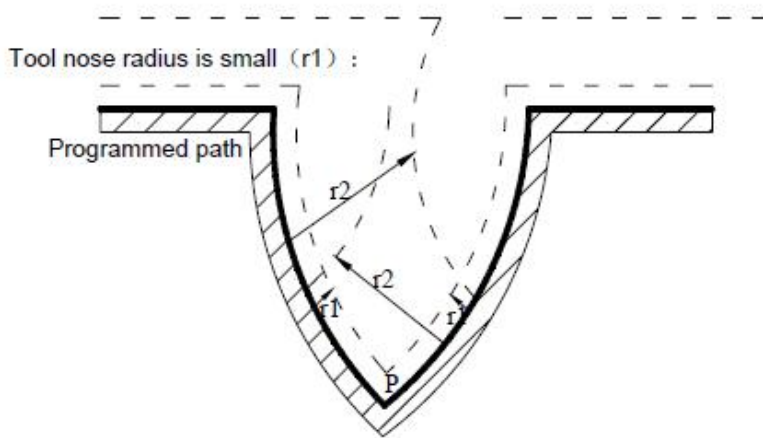
5) Inner side machining and scaling up compensation valid for acute angled being less than α (α : set by No. 237) .
Tool nose center path



(d) Special cutting

1) Without intersection

Tool nose radius is big (r_2):



There is an intersection P between arc and arc compensation path when the tool radius is small; none when the radius is big, and the system alarms.

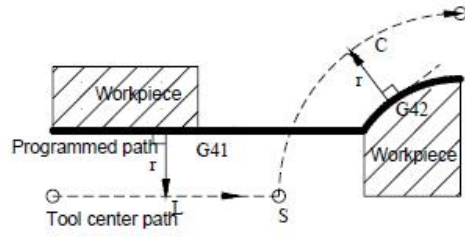
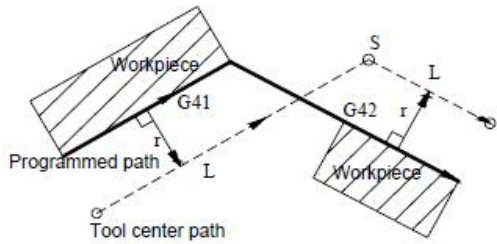
Fig. 4-14 Paths without intersection after offset

Offset path of changing compensation direction in compensation mode

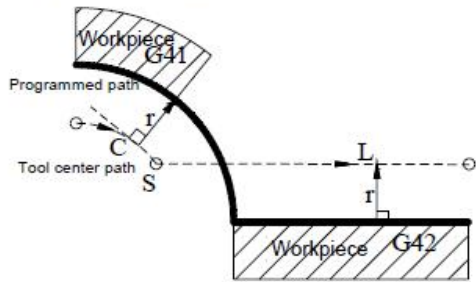
The compensation direction can be changed in compensation mode in special cutting. There is no inside and outside cutting when the system changes the compensation direction. The following are the path of tool running when the compensation changes are changed:

1) Linear→linear

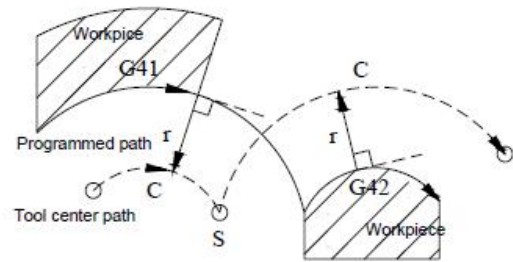
2) Linear→circular



3) Circular→linear



4) Circular→circular



5) No intersection when compensation is executed normally

When the system executes G41 and G42 to change the offset direction between block A and B, a vector perpendicular to block B is created from its starting point.

i) Linear---Linear

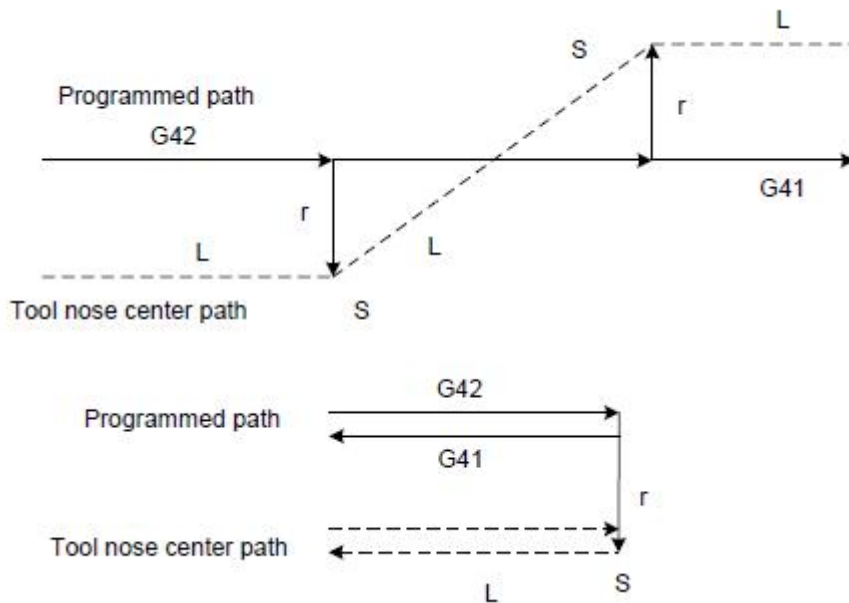


Fig. 4-15a Linear—linear, no intersection (changing compensation direction)

ii) Linear ---circular

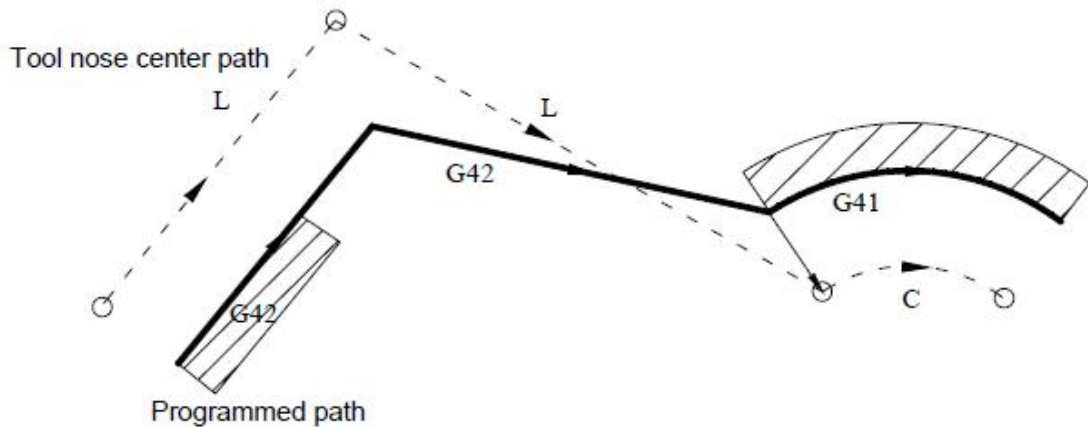


Fig. 4-15b Linear—circular without intersection (changing compensation direction)

iii) Circular—circular

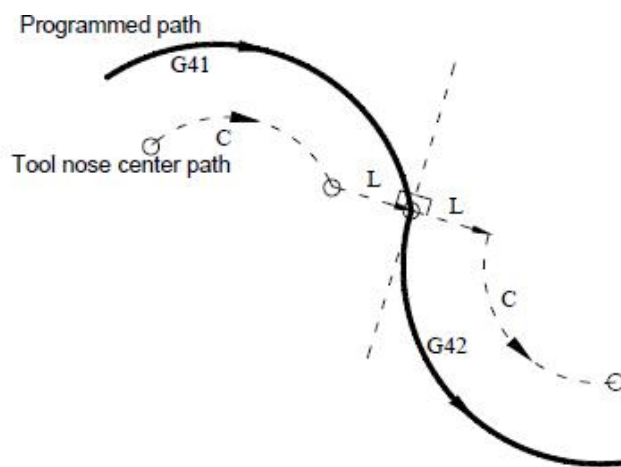


Fig. 4-15c Circular—circular without intersection (changing compensation direction)

4.2.4 Tool traversing in Offset canceling mode

In compensation mode, when the system executes a block with one of the followings, it enters compensation canceling mode, which is defined to compensation canceling of block.

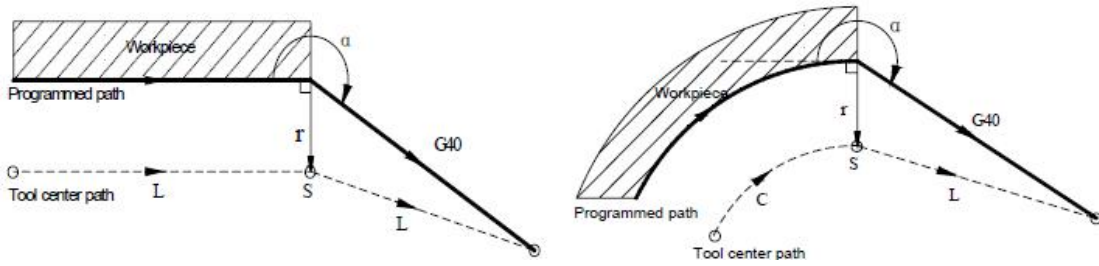
1. Execute G40 in a program;
2. Execute M30.

The system cannot execute G02 and G03 when canceling C tool compensation (tool nose radius compensation), otherwise the system alarms and stops run.

In compensation canceling mode, the system executes the block and ones in the register for tool nose radius compensation. At the moment, the run stops after one block is executed when single block is ON. The system executes the next one but does not read its following one when pressing **CYCLE START** key again.

(a) Tool traversing inside along corner($\alpha \geq 180^\circ$)

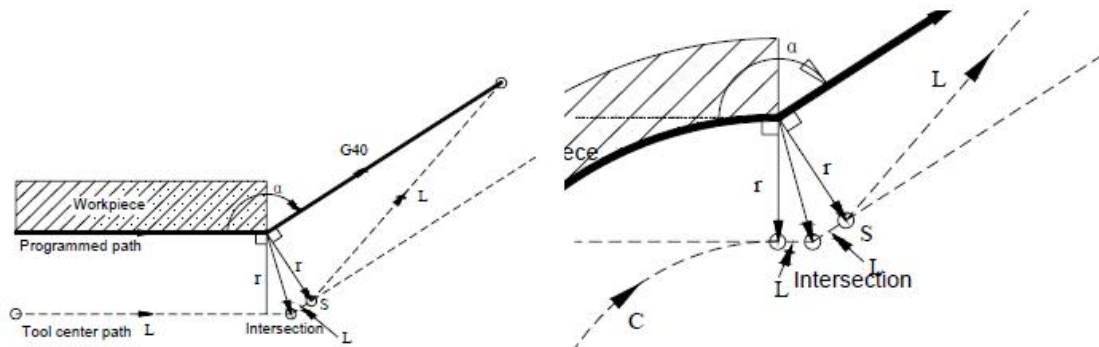
- 1) Linear→linear
- 2) Circular→linear



(b) Tool traversing outside along corner($180^\circ > \alpha \geq 90^\circ$)

1) Linear→linear

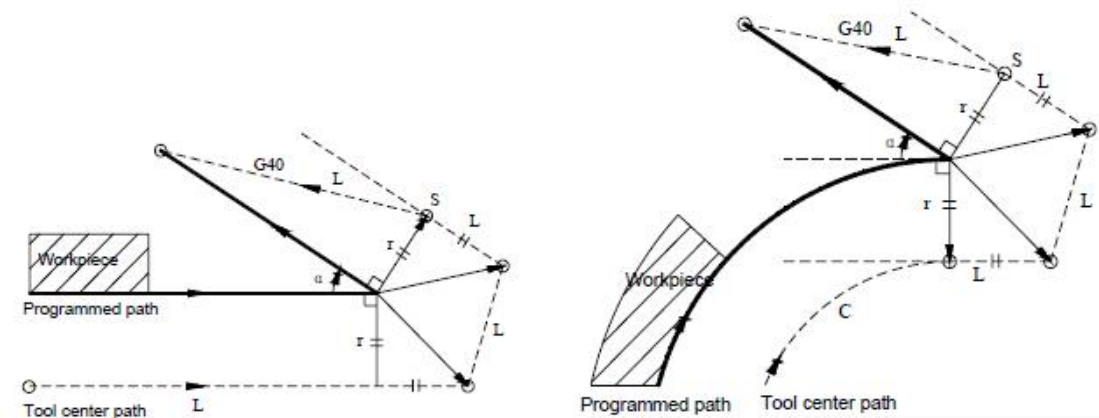
2) Circular→linear



(c) Tool traversing outside along corner($\alpha < 90^\circ$)

1) Linear→linear

2) Circular→linear



4.2.5 Tool interference check

“Interference” is defined that the tool cuts workpiece excessively and it can find out excessive cutting in advance, the interference check is executed even if the excessive cutting is not created, but the system cannot find out all tool interferences.

(1) Fundamental conditions

- 1) The tool path direction is different that of program path (angle is $90^\circ \sim 270^\circ$).
- 2) There is a big difference ($\alpha > 180^\circ$) for two angles between starting point and end point of tool nose center path, and between starting point and end point of program path.

Example: linear machining

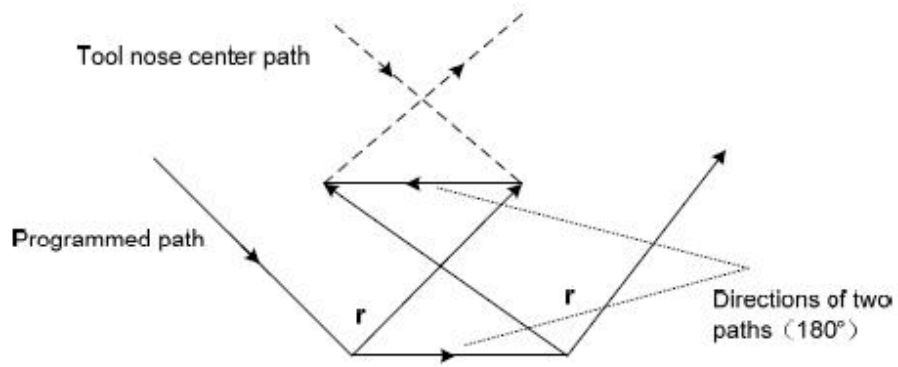
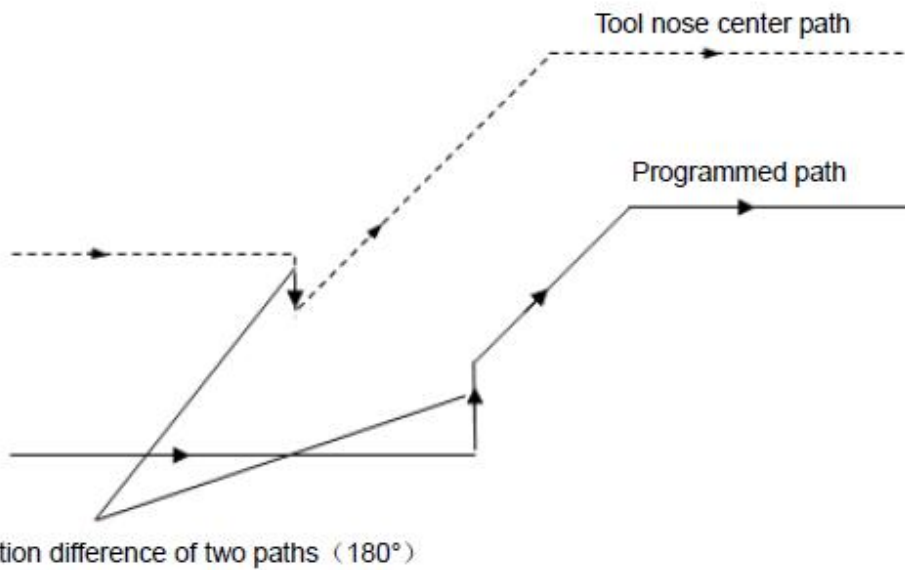


Fig. 4-16a Machining interference (1)



(2) Executing it without actual interference

1) Concave groove less than compensation value

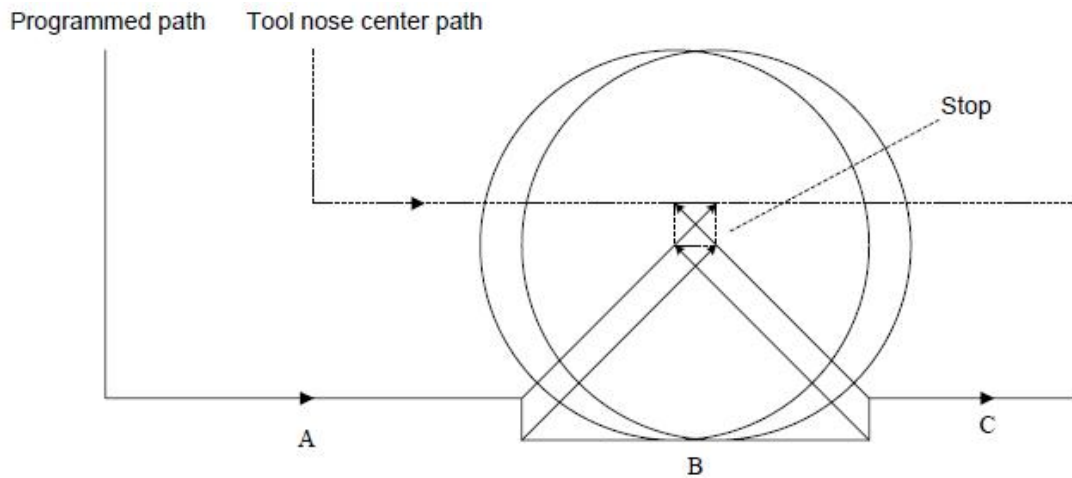


Fig. 4-17 Executing interference (1)

Directions of block B and tool nose radius compensation path are opposite without interference, the tools stops and the system alarms.

2) Concave channel less than compensation value

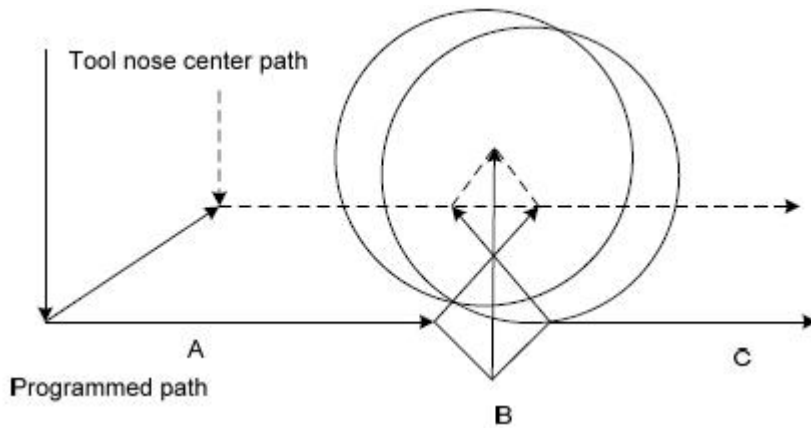


Fig. 4-18 Executing interference (2)

Directions of block B and tool nose radius compensation path are opposite without interference, the tool stops and the system alarms.

4.2.6 Commands for canceling compensation vector temporarily

In compensation mode, the compensation vector is cancelled temporarily in G50, G71~G76 and is automatically resumed after executing the commands. At the moment, the compensation is cancelled temporarily and the tool directly moves from intersection to a point for canceling compensation vector. The tool directly moves again to the intersection after the compensation mode is resumed.

Setting coordinate system in G50

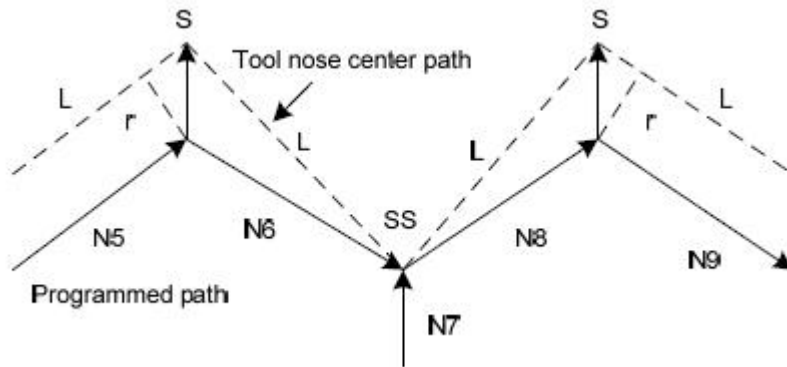


Fig. 4-19 Temporary compensation vector in G50

Note: SS indicates a point at which the tool stops twice in Single mode.

Reference point automatic return G28

In compensation mode, the compensation is cancelled in a middle point and is automatically resumed after executing the reference point return in G28.

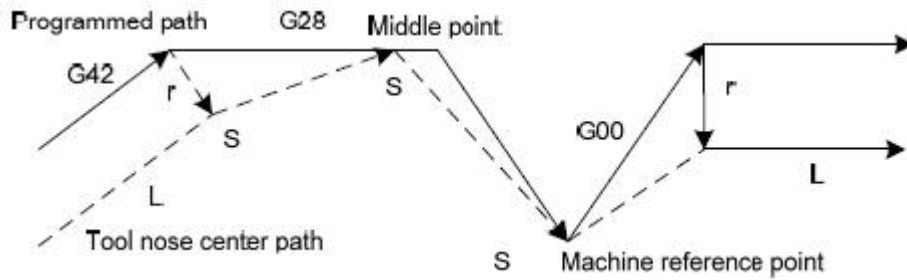


Fig. 4-20 Cancel compensation vector temporarily in G28

G71~G75 compound cycle; G76, G92 thread cutting

When executing G71~G76, G96 thread cutting, the system does not execute the tool nose radius compensation and cancel it temporarily, and there is G00, G01 in the following blocks, and the system automatically recovers the compensation mode.

G32, G33, G34 thread cutting

They cannot run in the tool nose radius compensation mode, otherwise, No.131 alarm occurs ".....CANNOT USED TO C COMPENSATION".

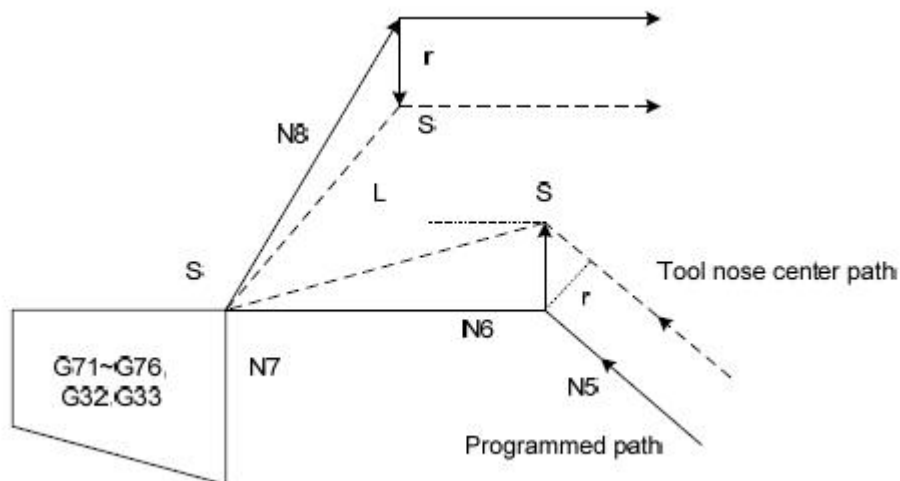


Fig. 4-21 Cancel compensation vector temporarily in G71~G76

G32, G33, G34 constant thread cutting

They cannot be executed in tool nose radius compensation mode, otherwise, No. 131 alarm occurs ".....the command cannot be used in C compensation mode."

G90, G94 (taking an example of G42)

Compensation method of tool nose radius compensation in G90 or G94:

- Cancel the previous tool nose radius compensation;
- Create the previous C compensation before cutting, and the path ① in the following figure creates the previous radius compensation mode;
- The paths 2, 3 in the following figure are the radius compensation cutting;
- The path 4 in the following figure can cancel the radius compensation, and the tool returns to the cycle starting point; there is G00, G01 in the following block, and the CNC automatically recovers the compensation mode.

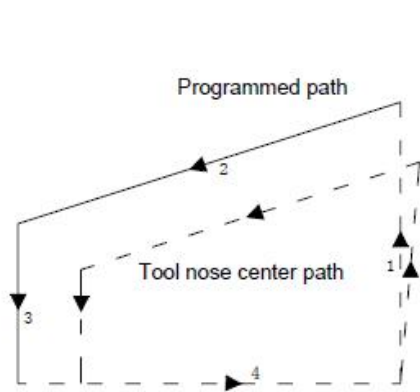


Fig. 4-22 Offset direction of G90 tool nose radius compensation

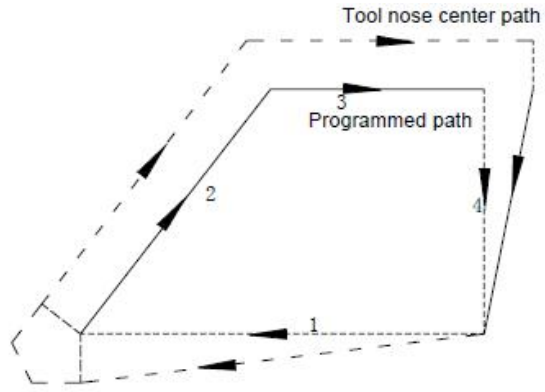
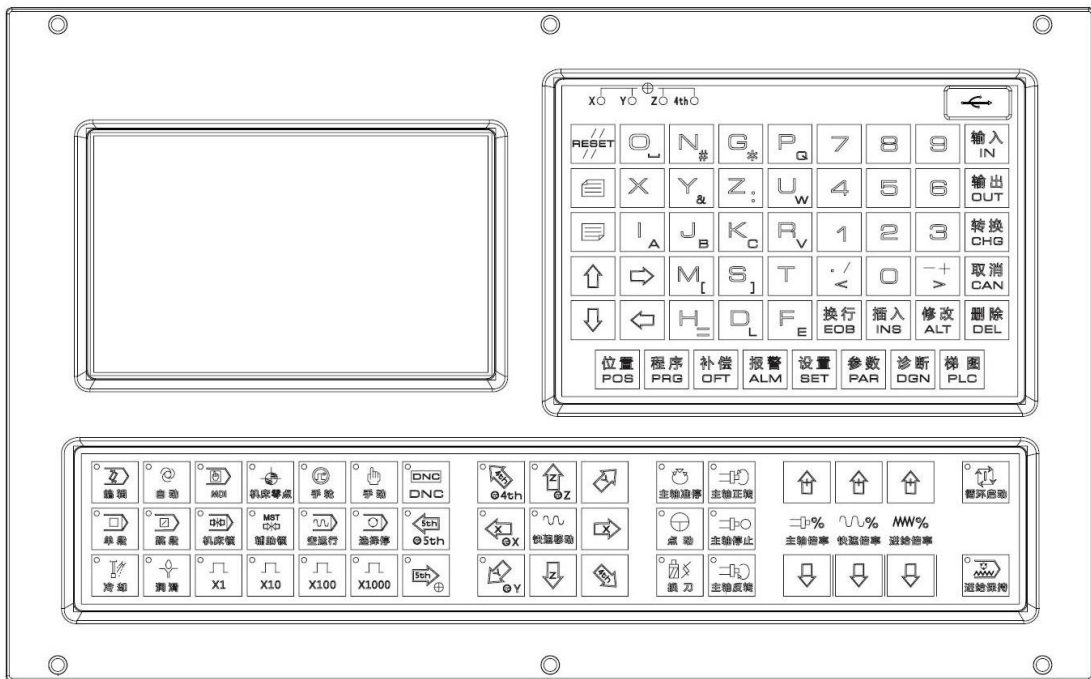


Fig. 4-23 Offset direction of G94 tool nose radius compensation

II Operation

CHAPTER 1 OPERATION MODE AND DISPLAY



INTERFACE



1.1 Panel Division

ECN*000T, ECN*000T-V CNC system uses an integrated panel, which is divided as follows:




1.1.1 State indication

	Indicator for axis zero return completion
	Three-color light

Indicator for axis zero return completion

Three-color light

1.1.2 Display Submenu













Menu	Remark
	<p>To enter Position interface including RELATIVE POS, ABSOLUTE POS, POS & PRG, INTEGRATED POS page.</p> <p>“ABSOLUTE POS” page can display the current absolute coordinates, clear out machining count, machining time;</p> <p>“POS & PRG” page can display current coordinate information and current run program;</p> <p>“RELATIVE POS” page can display current relative coordinates, clear relative coordinates;</p> <p>“INTEGRATED POS” can display current coordinate information and clear machine coordinates</p>
	<p>To enter Program interface including program content, program list, program state page</p> <p>Edit/search current open programs in program content page;</p> <p>Edit MDI programs, display coordinates, override and modal state in MDI page;</p> <p>Display, preview, open, copy machining programs in CNC in local catalog page;</p> <p>Display, preview, open and copy machining programs in U disk in U disk catalog page</p>
	<p>Enter tool offset interface including offset setting, macro variables, workpiece coordinate system and tool life page</p> <p>Set/search tool offset value and wear value in tool offset setting page;</p> <p>Set/search CNC macro variables in macro variable page;</p> <p>Set/search G54 ~ G59 coordinate system and zero offset value in workpiece coordinate system page;</p> <p>Set/search use information of current tool life</p>







ALM	<p>Enter alarm interface including alarm information, alarm log page</p> <p style="padding-left: 40px;">Search CNC alarm, CNC warning, PLC alarm, PLC warning in alarm information page;</p> <p style="padding-left: 40px;">Display alarm/warning history record in alarm log page</p>
SET	<p>Enter Setting interface including CNC setting, system clock, file management page</p> <p style="padding-left: 40px;">Set switch, level and parameters in CNC setting page;</p> <p style="padding-left: 40px;">Set system clock, data in system clock page;</p> <p style="padding-left: 40px;">Search/management files in CNC/U disk, copy/resume files in file management page</p>
PAR	<p>Enter Parameter interface including state parameters, data parameter, often used parameters and pitch compensation page.</p> <p style="padding-left: 40px;">Search/set state parameters in the state parameter page;</p> <p style="padding-left: 40px;">Search/set data parameters in the data parameter page;</p> <p style="padding-left: 40px;">Search/set user's customized parameters in the often used parameter page;</p> <p style="padding-left: 40px;">Search/set pitch error compensation data of each axis in the pitch compensation page.</p>
DGN	<p>Enter Diagnosis interface including the system diagnosis, system information page</p> <p style="padding-left: 40px;">Search the current diagnosis information of the CNC in the diagnosis page.</p> <p style="padding-left: 40px;">Search the product information, ladder information and ladder state.</p>
PLC	<p>PLC page includes three subpages: PLC state, PLC monitor, PLC data.</p> <p style="padding-left: 40px;">PLC state page can search states of X, Y, F, G, R, A, C,T;</p> <p style="padding-left: 40px;">PLC monitor page can monitors the execution state of current ladder on-line;</p> <p style="padding-left: 40px;">PLC data page can search/set K, D, DT, DC value.</p>

1.1.4 Machine panel

The key functions on MST-2000 machine panel are defined by PLC program (ladder), the detailed function meanings are referred to machine manufacturer manual.

The functions of this MST-2000 machine panel key defined by standard PLC program are as follows:

Key	Name	Function explanation	Operation mode
	Feed hold key	Dwell commanded by program, MDI code	Auto, MDI mode
	Cycle Start key	Cycle start commanded by program, MDI code	Auto, MDI mode
	Feedrate Override keys	Adjusting feedrate	Auto, MDI, Edit , Machine zero return, MPG, Step, Manual, Program zero return mode
	Rapid override keys	Adjusting rapid traverse	Auto, MDI, Machine zero return, Manual, Program zero return mode
	Spindle override keys	spindle speed adjustment (spindle analog control active)	Auto, Edit, MDI, Machine zero return, Manual, Step, MPG, Program zero return mode
	Single Block switch	For switching of block/blocks execution, Single block indicator lights up if Single mode is active	Auto, MDI mode
	Block Skip switch	For skipping of block headed with"/"sign, if its switch is set for ON, the Block Skip indicator lights up	Auto, MDI mode
	Machine Lock key	If the machine is locked, its indicator lights up, and X, Z axis output is inactive.	Auto, MDI, Edit, Machine zero return, Manual, Step, MPG, Program zero return mode
	M.S.T. Lock key	If the miscellaneous function is locked, its indicator lights up and M, S, T function output is inactive.	Auto, MDI mode
	Dry Run key	If dry run is active, the Dry run indicator lights up. Dry run for program/MDI codes	Auto, MDI mode
	Edit mode key	To enter Edit mode	Auto, MDI, Machine zero return, Manual, Step, MPG, Program zero return mode
	Auto mode key	To enter Auto mode	MDI, Edit, Machine zero return, Manual, Step, MPG, Program zero return mode

	MDI mode key	To enter MDI mode	Auto, Edit, Machine zero return, Manual, Step, MPG, Program zero return mode
	Machine zero return mode key	To enter Machine zero return mode	Auto, MDI, Edit, Manual, Step, MPG, Program zero return mode
	Step/MPG mode key	To enter Step or MPG mode (one mode by parameter)	Auto, MDI, Edit, Machine zero return, Manual, Program zero return mode
	Manual mode key	To enter Manual mode	Auto, MDI, Edit, Machine zero return, Step, MPG, Program zero return mode
	Program zero return mode key	To enter Program zero return mode	Auto, MDI, Edit, Machine zero return, Step, MPG, Manual mode
	MPG trial-cut selection key	To enter MPG trial-cut mode	Auto, MDI, Edit, Machine zero return, Step, MPG, Manual, MPG trial-cut mode

1.2 Summary of Operation Mode

There are 8 modes in ECN*000T, which are Edit, Auto, MDI, Machine zero, Step/MPG, Manual, Program Zero, and MPG trial-cut modes.

Edit mode

In this mode, the operation of part program setup, deletion and alteration can be performed.

Auto mode

In this mode, the program is executed automatically.

MDI mode

In this mode, the operation of parameter input, command blocks input and execution can be performed.

Machine zero mode

In this mode, the operation of X, Z machine zero return can be performed separately.

MPG / Step mode

In the Step/MPG feed mode, the moving is performed by an increment selected by CNC system.

Manual mode

In this mode, the operation of Manual feed, Manual Rapid, feedrate override adjustment, Rapid override adjustment and spindle ON/OFF, cooling ON/OFF, Lubricating ON/OFF, spindle jog, manual tool change can be performed.

Program zero return mode

In this mode, the operation of X, Z program zero return can be performed separately.

MDP trail-cut mode



In this mode, rotating the MPG can control program's execution speed, which can check whether the machining program is correct.

1.3 Display Interface

The chapter introduces page switch, relationship between operation input and soft key, and concrete operation methods.

ECN*000T has 9 function key including POS, PRG, SET, etc. on its edit keyboard. Each function key corresponds to one interface which has many pages and operation soft key, which are shown below:

1.3.1 POS interface

Press to enter POS interface which includes pages  and , and these pages can be viewed by corresponding soft function key or repetitively press .

1) ABSOLUTE POS display interface

The X, Z coordinates displayed are the absolute position of the tool in current workpiece coordinate system, these coordinates are memorized as power is down and the workpiece coordinate system is specified by G50.



Basic information in POSITION interface (taking an example of the above figure):



- G code : information of each G code;
- M : last executed M code;
- S : it is input spindle speed when analog spindle is executed, and input spindle's gear when the spindle is executed;
- L : call times of subprogram;
- F : the first half (0.0000mm/min) is the actual speed, and the second half (200mm/min) is the command speed;
- JOG F.(JOG speed): theory speed value of current override in JOG mode;
- FED OVRI(Feedrate override):override selected by feedrate override switch;
- RAD OVRI (Rapid override): display currently rapid override;
- SPI OVRI (Spindle override): display the spindle override when NO.001 Bit1 is set to 1;
- PART CNT(Part counting) : part counting adds 1 when M30 (M99 in the main program) is



executed;

CUT TIME: the system counts the time when the automatic run is started, time unit is hour, minute and second in order. The part counting and cut time are memorized at power-down, the clearing ways for them are shown below:

Press “**Absolute**” software key to display the following interface: Then, press “**PART CLS**” software key to clear the quantity of the machined work pieces, and press “**TIME CLS**” software key to clear the cutting time;

Method 2:

Quantity of processing reset: hold the  key, and  key.

Cutting time reset: hold the  key and  key.

2) RELATIVE POS display interface

Displayed U, W coordinates are the current position relative to the relative reference point, and they are held on when power on. They can be cleared at any time. When No.005 Bit1=1, and the absolute coordinates are set by G50, the relative coordinates are set simultaneously. Clearing U, W:

Method 1:

Press “**RELATIVE**” and the following page appears:

And then, press **U CLR** software key to clear U value, and press **W CLR** to clear W value;

Method 2:

Under the relative coordinate display page hold the key **U** until the page U flashing, press

 key, X coordinate values;

Under the relative coordinate display page hold the key **W** until the page W flashing, press

 key, X coordinate values;

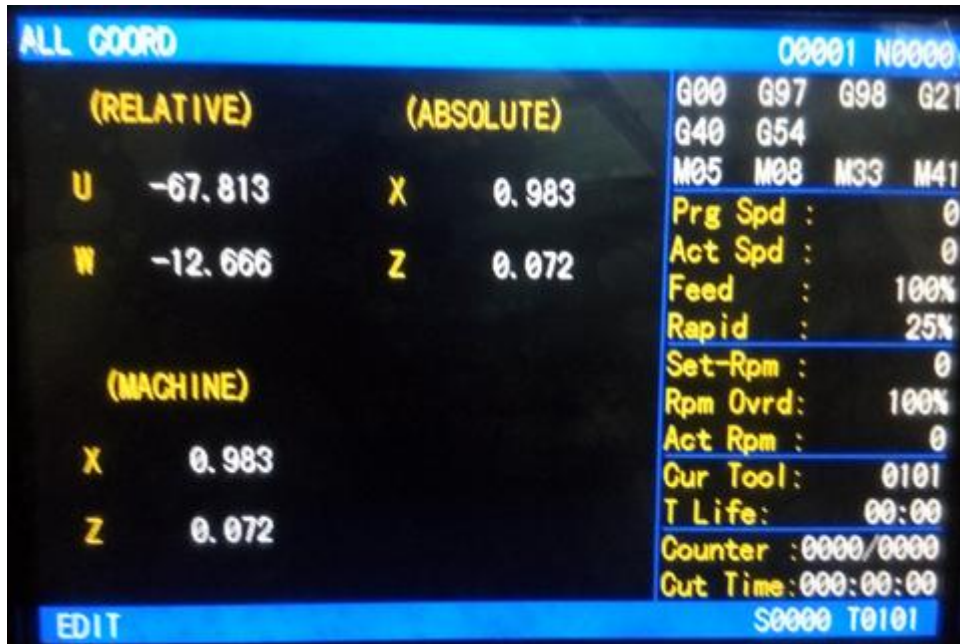
Note: Clearing method is the same when many axes are enabled.

3) Comprehensive coordinate display page

In the comprehensive position page, also showed relative coordinate, absolute coordinates, machine tool coordinate, the residual amount of mobile (you moves only in automatically and record the mode shows).

The indications of the machine tool coordinate for the coordinates of the current position in machine coordinate system, machine tool coordinate system was established through back to the machine zero point. Motion values are the difference value of program block or MDI code’s difference value of target location and current location.

Display the page as follows:





Clearing method of absolute coordinates are shown below:



4) POS&PRG display interface

In POS&PRG interface, it displays ABSOLUTE, RELATIVE coordinate of the current position (ABSOLUTE, DIST TO GO of current position and machine coordinates will be displayed when No.180 BIT0 is set to 1) as well as 7 blocks of current program together. During the program execution, the displayed blocks are refreshed dynamically and the cursor is located in the block being executed.

1.3.2 PRG interface

Press  key to enter PRG interface, which includes the subpages the interface program content, program state, directory, file directory four pages. Press  key to switch interfaces.

1) PRG page

In the page, the program content including current block can be displayed. Press  and  the program content can be viewed forward or backward by pressing or key.




2) Program status interface

In MDI mode, the page can be displayed, and its value when the corresponding soft function key is pressed as follows:



3) Local directory page

In application status page, press  to enter the directory page. Under the interface, lists all the processing procedures, for the party Users find want to select the program and system at the right shows the current page program before 12 line segments.

Display the contents of the program directory page:

(a) Existing application number:

Displayed in the CNC has been deposited in the number of programs (including subroutine)

(b) The remaining application number:

Display can also deposit in the CNC program

(c) Has been in storage:

According to CNC has been deposited in the part program takes up the amount of storage

(KB)

(d) The remaining amount:

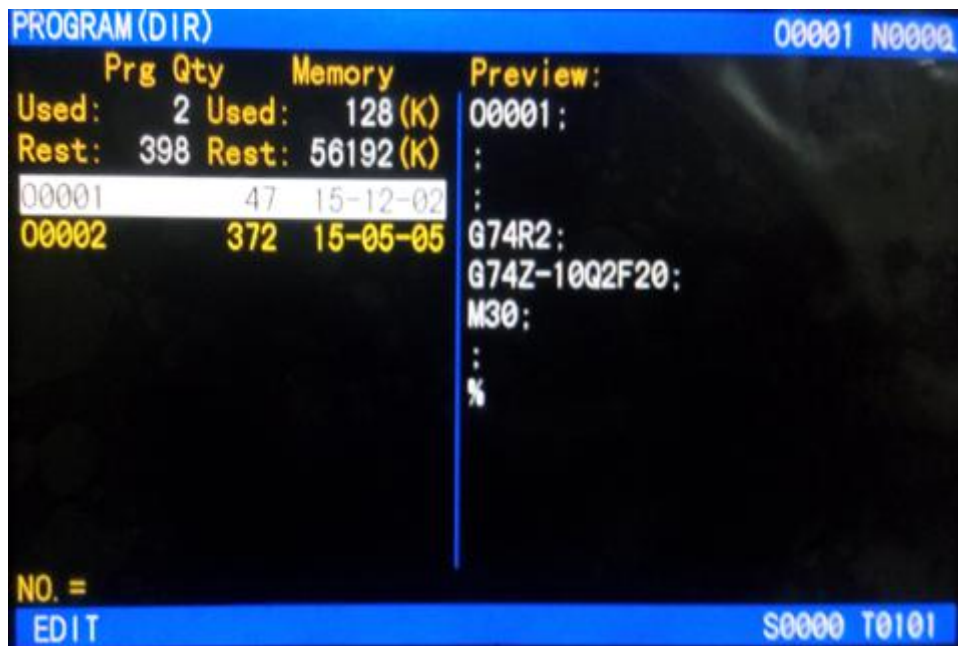
According to the residual capacity of CNC program storage parts of (KB)

(e) Program directory:

According to the size of the part program name showed in turn into parts the application number of the program

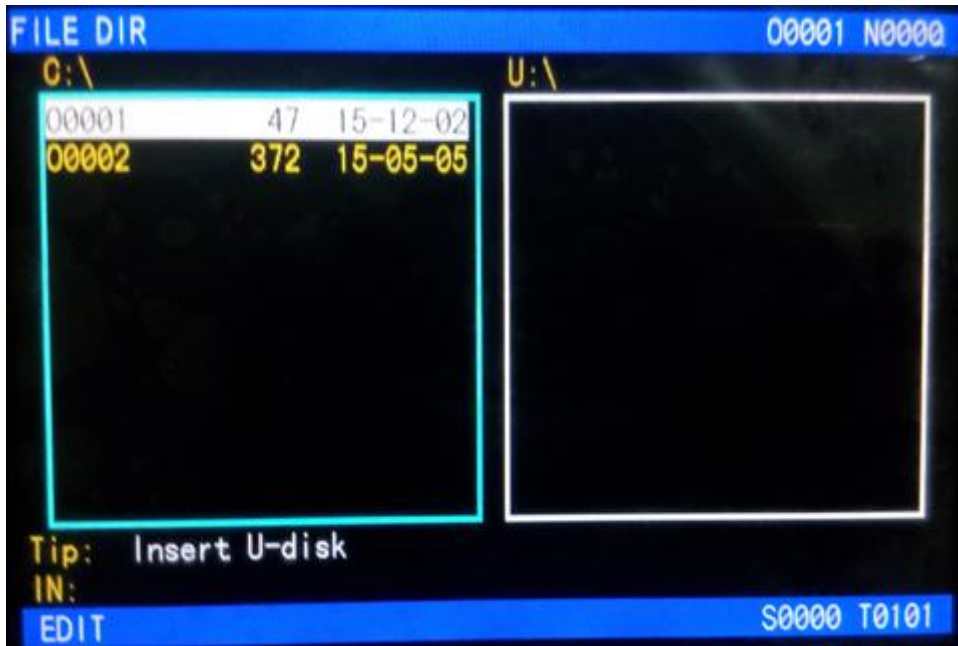
(f) Program size:

The size of the display of CNC program storage space




4) U disk directory page

Press to enter U disk directory page as follows:





1.3.3 TOOL OFFSET Interface

Press  to enter the TOOL OFFSET interface, and press it several times until it enters the interface when No. 002 Bit0 is set to 1.. Press repetitively or corresponding key to switch pages.

1) TOOL OFFSET&WEAR interface



There are 5 pages and 33 offset & wear No. (No.000~No.032) available for user in the interface,

which can be shown as follows by pressing  or  key.



Note: The axis names of Y, the 4th, the 5th axis is defined by No.225.

2) MACRO interface

There are 25 pages in this interface, which can be shown by pressing  or  key. In Macro window there are 600 (No.100~No.199 and No.500~No.999) macro variables which can be specified by macro command or set directly by keypad. The public variable page is shown below:

Macro						00001 N0000
No.	Data	No.	Data	No.	Data	
100		110		120		
101		111		121		
102		112		122		
103		113		123		
104		114		124		
105		115		125		
106		116		126		
107		117		127		
108		118		128		
109		119		129		
公用变量						
NO. 100 = 9						
EDIT						S0000 T0101




Note: The local variable and the system local are specified by macro codes instead of directly set by the keyboard in the page.

- 3) Workpiece coordinate system setting
- 4) Tool life management

T life mgmt.							00001 N0000
Cur. T stat.							
T NO.	Grp.	Life	Used	Life Ut.	Stat		
Def. grp							
MDI							S0000 T0101




1.3.4 ALARM interface

1) Alarm:

Press  key to enter alarm interface including  or  repetitively, or the corresponding soft function. The alarm interface is shown below:

1) Alarm information page





Note: Alarm clearing: It can clear alarms by pressing  .(N.O100 alarm can be cleaned by pressing  and )

2) Alarm log page



1.3.5 Setting interface

Press  to enter the SETTING interface which can be switched by pressing  repetitively or corresponding soft key. The interface is shown below:

1) Setting interface



Para SW: when this setting is on , the parameters can be modified, if off ,can not modify parameters

Prog SW: ON :can edit program; OFF: can not edit program

Auto NO:ON: program will create program number automatically, OFF: need create program number by manual.

2) Operation:

data in this page for CNC data (status parameters, data parameters, pitch compensation parameters, tool offset, etc.) backup and restore.

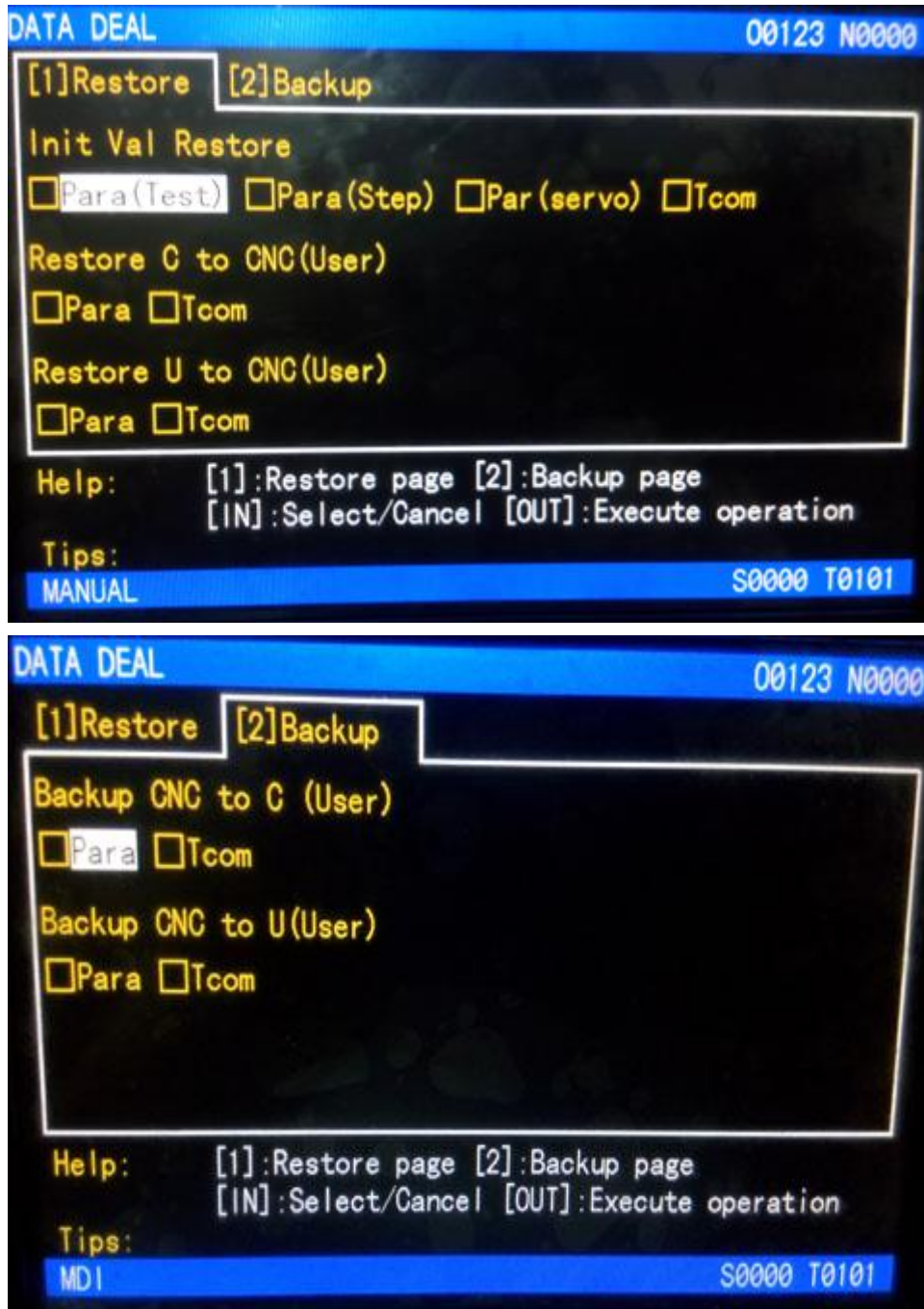
The initialization operation: can make parameters, tool offsets and pitch compensation parameters to default values.

C disk data reduction to the CNC: the backup data files in the system disk restore to the system.

U disk data reduction to CNC: the backup data in the U disk restore to the system.

CNC data backup to disk C: system current parameters, tool offsets, pitch compensation parameters and ladder diagram backup to system.

CNC data backup to usb: system current parameters, tool offsets, pitch compensation parameters and ladder diagram backup to U disk.



3) Permissions Settings:

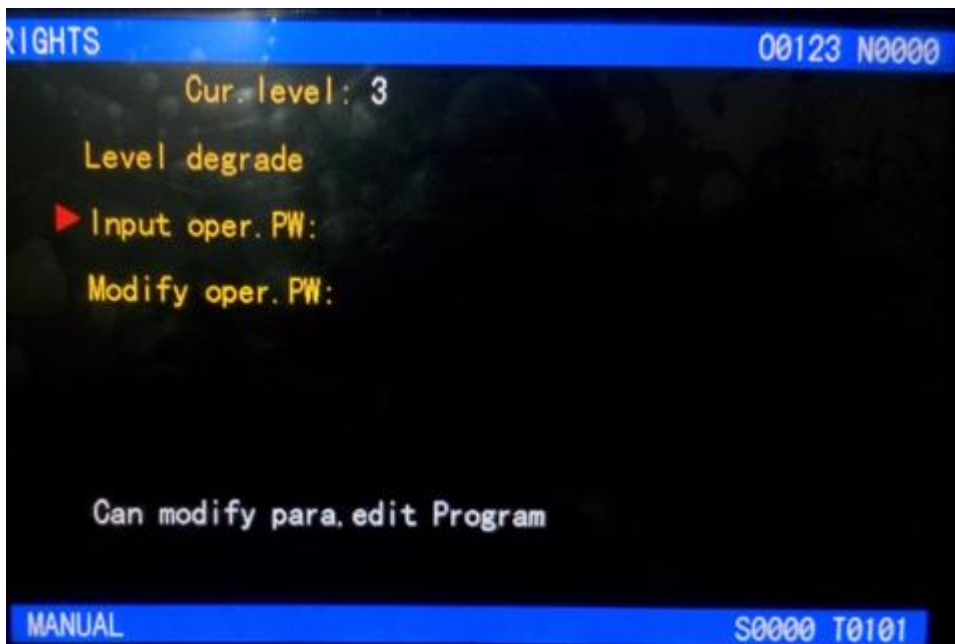
ECN*000T series password level is divided into four grade, from high to low were machine manufacturer level (level 2), the equipment management level (level 3), the engineer level (level 4), the operator level (level 5).

Machine manufacturers: allowed to modify the status parameters, data parameters, and pitch compensation parameters, tool offsets data, edit program (including macro program), edit the PLC ladder diagram, download upload ladder diagram;







Equipment management level: the initial password is 12345, allowed to modify the status parameters, data parameters, tool offsets parameters, and editing program;

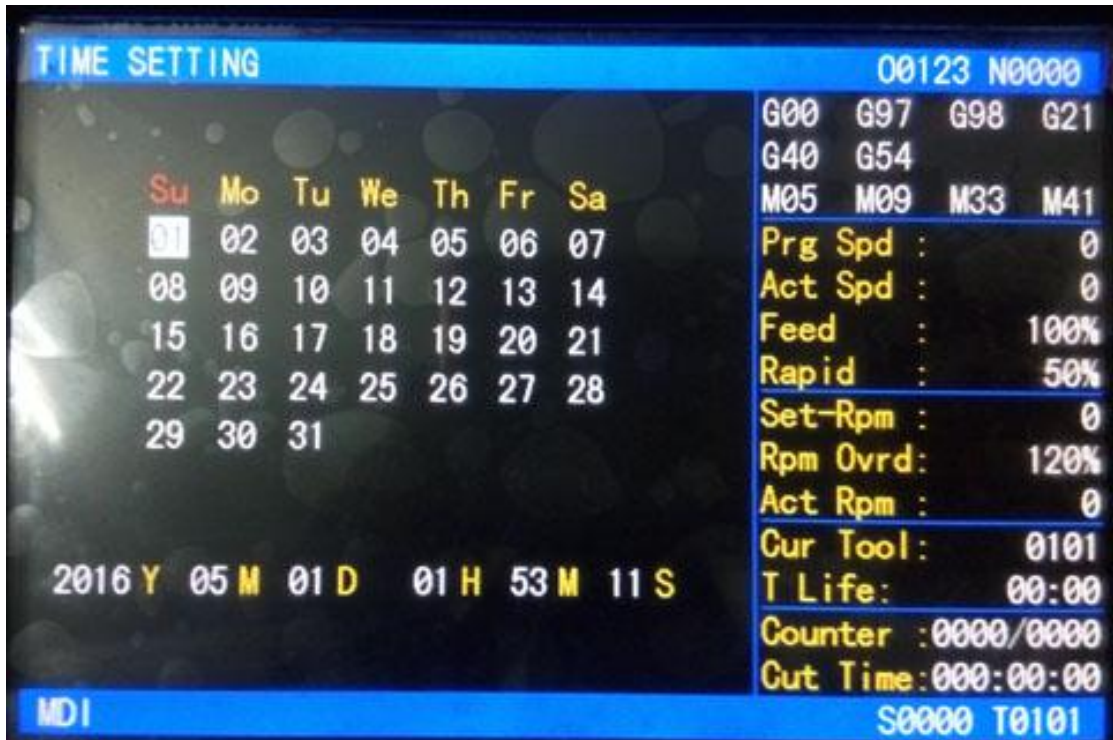
Engineer level: the initial password is 1234 .can modify the tool offsets (tool setting), macro variables, program edit,but can not modify the status parameters, data parameters and pitch compensation parameters.

Operator level: no password .can carry on the operate controller tool setting, choose workpieces program, but can not edit and modify the status parameters, data parameters, and pitch parameters.





Time Setting: display the current time and date.




Can press  key into the modified model, and the key   switch needed to modify year/month/day/minutes/seconds, press  , its value is modified, if you want to exit the modify mode, press the key  again.

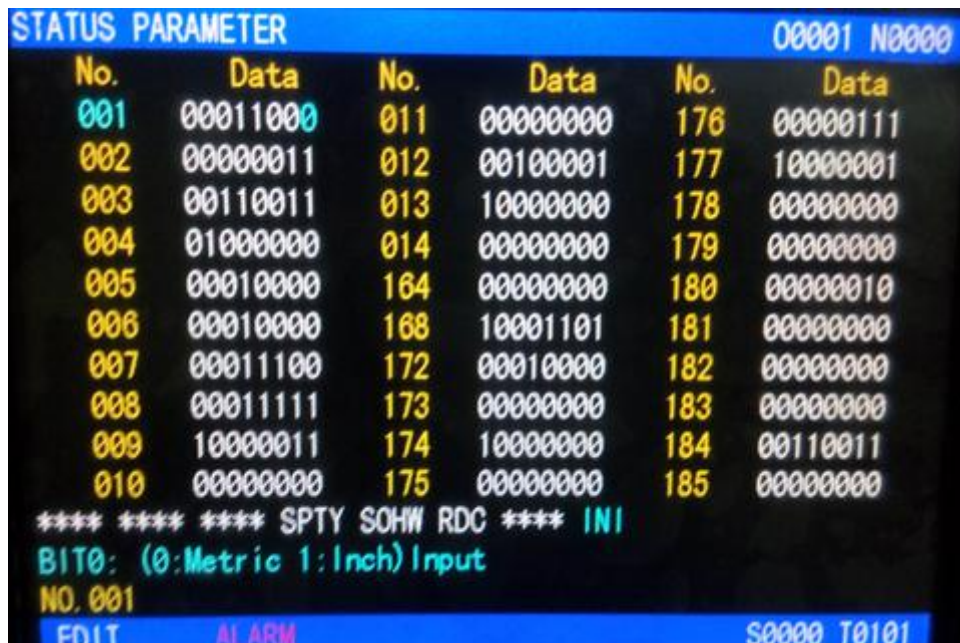


1.3.6 Parameter Interfaces



Press  to enter **parameter** interface including pages, which can be switched by pressing  repetitively or the corresponding soft key. The interface is shown below:

1) Status parameter page

Press  to enter **status** parameter interface, there are 48 parameters list on two pages, by switching  or  to display all parameters.





2) Data parameter page

Press  **PAR** to enter **date** parameters interface.(if the current interface is status parameter , to display data parameter by pressing )

DATA PARAMETER						00001 N0000
No.	Data	No.	Data	No.	Data	
015	1	025	100	035	0.0000	
016	1	026	100	036	0	
017	1	027	8000	037	6000	
018	1	028	50	038	6000	
019	5	029	60	039	6000	
020	0	030	1	040	6000	
021	0.0000	031	1260	041	40	
022	5000	032	400	042	10	
023	5000	033	40	043	100	
024	100	034	0.0000	044	115200	

X axis CMR
NO. 015
EDIT S0000 T0101





3) Pitch compensation parameter page

Press  **PAR** or  multi times to enter pitch compensation interface.

PITCH COMP						00001 N0000
No.	X	Z	No.	X	Z	
000	0	0	012	0	0	
001	0	0	013	0	0	
002	0	0	014	0	0	
003	0	0	015	0	0	
004	0	0	016	0	0	
005	0	0	017	0	0	
006	0	0	018	0	0	
007	0	0	019	0	0	
008	0	0	020	0	0	
009	0	0	021	0	0	
010	0	0	022	0	0	
011	0	0	023	0	0	



NO. 000
EDIT S0000 T0101



1.3.7 Diagnosis Interface

Press  **诊断**  to enter diagnosis interface including pages which can be switched by pressing  or  repetitively or the corresponding soft key.


1) System diagnosis page


The I/O status of CNC controller and machine, Transmission of signals between CNC and PLC, The PLC internal data and CNC internal status can be displayed through diagnosis. Press

 **诊断**  Key into CNC diagnosis page displays, CNC diagnostic page displays have a keyboard,

Status diagnosis and auxiliary function parameters, etc. By pressing key  or  to check.

In CNC diagnosis display page, in the bottom of the page there are two lines for detailed content,



the second line shows the current bit signal's meaning, can display the diagnosis of  or

 to change; The first line shows the current cursor in the diagnosis of all the English abbreviations.





CNC DIAG				00001 N0000	
No.	Data	No.	Data	No.	Data
000	00000000	010	00000000	020	00000000
001	00000000	011	00000000	021	00000000
002	00000000	012	00000000	022	00000000
003	00000000	013	00000000	023	00000000
004	00011111	014	00000000	024	00000000
005	00000000	015	00000000	025	00000000
006	00000000	016	00000000	026	00000000
007	00000000	017	00000000	027	00000000
008	00000000	018	00000000	028	00000000
009	00011111	019	00000000	029	00000000
ESP *** ** DEC5 DEC4 DECZ DECY DECX					
BIT0: X axis machine homing Dec signal					
NO. 000					
EDIT				S0000 T0101	

2) PLC information interface

There are several pages by pressing  **诊断**  to display PLC address, from X0000 ~ X0063, Y0000 ~ Y0047, F0000 ~ F063, G0000 ~ G063 and so on..the pages can be switched

by pressing  or .


Status page in PLC, details the bottom of the page has two lines, the second line shows the

current bit meaning in Chinese, can display the  or  to change the address bits.

The first line shows the current cursor location number all the abbreviations.




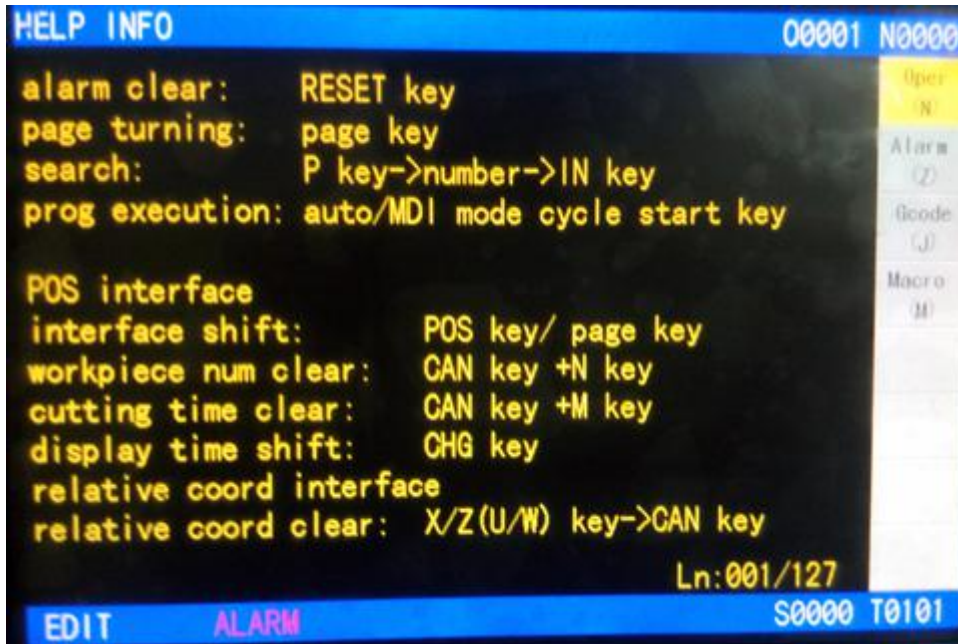
3) Soft key panel interface

Press  multi times to enter soft key panel interface .the corresponding function will be activated by operating the soft key in this interface .




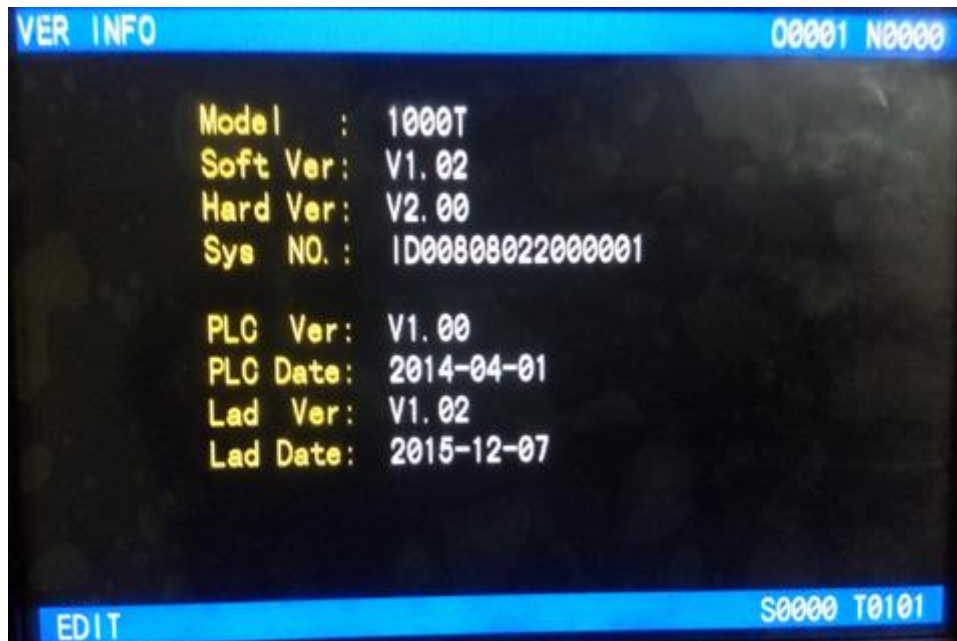
4) Help information interface

Press  to enter the help information interface. Can view the help page CNC current operating table clock, alarm table, G, macro instruction, such as information, displays the page as shown.



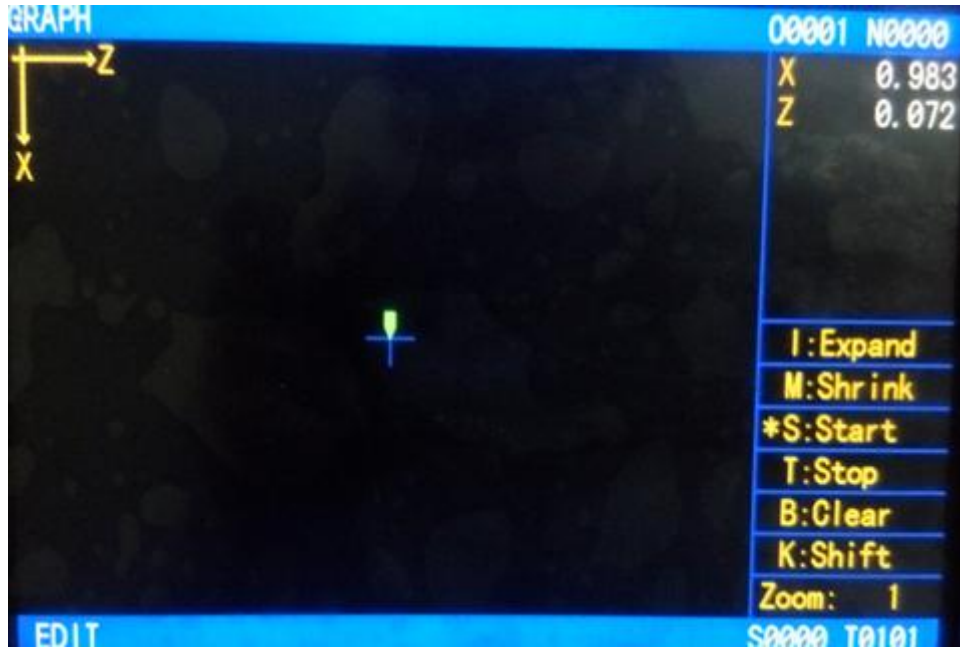
5) System information interface

Press  multi times to enter system version information which includes software/hardware version, system series number, PLC version, and son on.







1.3.8 Graph Interface

Press  multi times to enter GRAPH interface. The interface is shown below:



1.3.9 Ladder Interface

Press   to enter LADDER interface including pages , which can be switched by pressing

 or  repetitively or corresponding soft key. The interface is shown below:

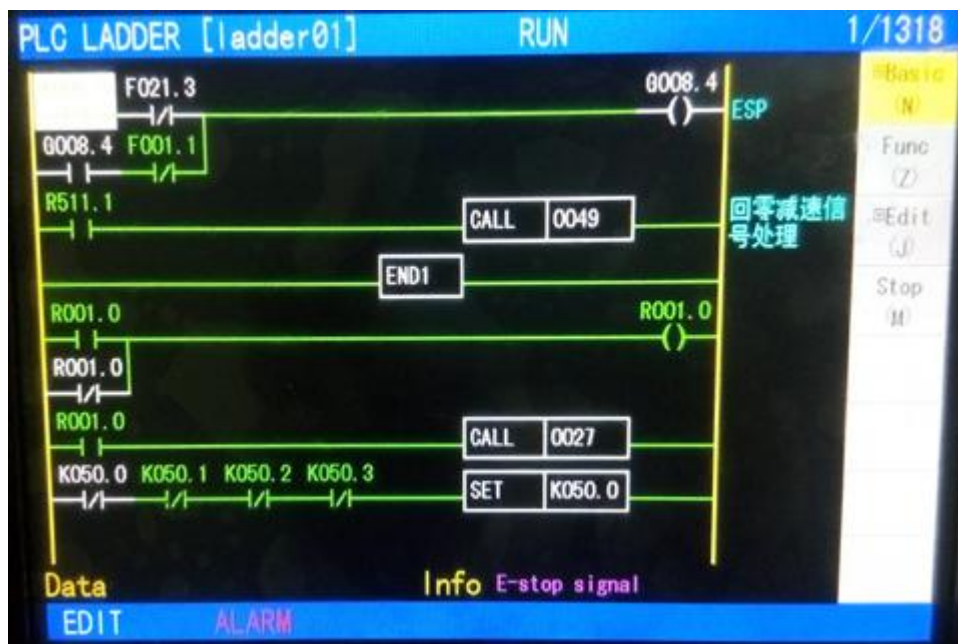


1) PLC DIAG (I/O status)page

This page can display PLC I/o state information.

PLC DIAG		RUN				
No.	Data	No.	Data	No.	Data	X sig
X000	00000000	X010	00000000	X020	00000000	N
X001	00000000	X011	00000000	X021	00000000	Y sig
X002	00000000	X012	00000000	X022	00000000	(Z)
X003	00000000	X013	00000000	X023	00000000	A sig
X004	00001000	X014	00000000	X024	00000000	(J)
X005	00000000	X015	00000000	X025	00000000	F sig
X006	00000000	X016	00000000	X026	00000000	(M)
X007	00000000	X017	00000000	X027	00000000	G sig
X008	00000000	X018	00000000	X028	00000000	(H)
X009	00000000	X019	00000000	X029	00000000	R sig
T05 **** ESP DITW DECX DIQP SP SAGT						(U)
BIT0: protection door detect signal						Trace
X000						(O)
MANUAL						

2) Ladder monitor page



The monitor page can view states of current contact, coil ON/OFF state, values of timer and counter. They are displayed with a green background when they are ON, otherwise, they are the same those of the page background. means the contact X0.5 is ON, means Y25.2 is OFF.

1. page program view

In MONITOR page, the system can monitor simultaneously three windows, the ladder blocks corresponded to each window can be viewed by separately pressing , at the moment, the ladder corresponded to the selected block is displayed on the screen.

2. Window block selection

- (1) Select the window which needs to select blocks, press respectively , to select the windows.
- (2) Press to select the window program. At the moment, the display page is shown below:
- (3) Move the cursor to select the ladder block for display.

(4) Press , and then return the previous menu, press to cancel the selection operation to return to the previous menu.

3. Find parameters, commands and network

- (1) Select the required block window for finding commands, parameters and network, make the corresponding block display in the window, and find the command, parameters and network in the window.
- (2) Respectively press to find the corresponding parameter, functional command and network in the current displayed block, and move the cursor to the corresponding position.
- (3) Press , to locate the cursor to the home or end of the current block.

3) PLC data page

PLC data includes K, D, DT, DC, which can be set in the page.

4) PLC information interface

The ladder file list page displays all ladders and their relevant information included in the current system, and the new ladder can run in the page. The program name marked with is the current running ladder.



Note: The file list page is displayed when the above 2-level privilege is executed

CHAPTER 2 POWER ON/OFF AND PROTECTION

2.1 System Power-on

Before ECN*000T power on, the following items should be confirmed:

1. The machine is in a normal state.
2. The power voltage conforms to the requirement of the machine.
3. The connection is correct and secure.

The following window is displayed after ECN*000T is turned on:

The current position (ABSOLUTE POS) page is displayed after ECN*000T automatic detection and initiation are finished.

2.2 System Power-off

Before power is off, ensure that:

1. The feeding axes of the CNC is at stop;
2. Miscellaneous functions (spindle, cooling etc.) are OFF;
3. Cut off CNC power prior to machine power cutting off.

Note: Please refer to the machine manufacturer's manual about turn-off the machine's power supply.

2.3 Overtravel Protection

Overtravel protection should be used to prevent machine from being damaged due to the overtravel of X, Y, or Z.

2.3.1 Hardware overtravel protection

The stroke switches are fixed at the positive and negative maximum travel of the machine X, Y, Z axis respectively, they are connected by the following figure. And the BIT2 of bit parameter No.215 must be set to 0. If the overtravel occurs, the stroke switch acts to make ECN*000T stop, and the emergency alarm is activated.

When the hardware overtravel occurs, there will be an "emergency stop" alarm in ECN*000T.

The steps to eliminate this alarm are: press the OVERTRAVEL key to switch to the ALARM page to view the alarm message, and reset the alarm and move the table reversely to detach the stroke switch (for positive overtravel, move negatively; vice versa) .

2.3.2 Software Overtravel Protection

When the Bit3 of bit parameter No.215 is set to 0, the software limit is active.

1. X, Z axis

The software strokes are set by data parameter NO.045、NO.046、NO.047、NO.048, they refer to the machine coordinates. As follows figure shows, X, Z are the machine coordinate system axes; NO.045、NO.046 are for X axis positive and negative strokes, NO.047、NO.048 are for Z axis positive and negative strokes, within the broken line is the software stroke scope.

If the machine position (coordinate) exceeds the area within broken line, overtravel alarm will be issued. The steps to eliminate this alarm are: press the overtravel alarm release key to clear the alarm, then moves reversely (for positive overtravel, move out negatively; vice versa).


2. Additional axis

The stroke range setting is the same that of the basic axis. When the additional axis is enabled, NO.192---NO.197 displays stroke range parameter of the corresponding additional axis.

2.4 Emergency Operation

During the machine working , Unexpected incidents may occur because of the user programming, operation and product fault etc. So the ECN*000T should be stopped immediately for these incidents. This section mainly describes the resolutions that the system is capable of under the emergency situation. Please see the relative explanation on these resolutions under the emergency by machine manufacturer.

2.4.1 Reset

Press  key to reset ECN*000T system when there are abnormal output and axis operations:

- 1 :All axes motion stops;
- 2 :M, S function output is inactive (the parameter sets whether the system automatically cuts off signals of spindle rotation, lubricating, cooling by pressing key, defined by PLC ladder);
- 3: Automatic run ends, modal function and state remain.

2.4.2 Emergency stop

During machine running, if the emergency key is pressed under the dangerous or emergent situation (external SP signal active), the CNC system enters into emergency status and the machine movement is stopped immediately. All the outputs such as the spindle running, cooling are cut off. If the emergency key is released, the emergency alarm is cancelled and the CNC resets. Its circuit wiring is shown in **Section 2.3.1** of the Chapter.

Note 1: Ensure the fault is eliminated before the emergency alarm is cancelled.


Note 2: Pressing down the Emergency key prior to power on or off may alleviate the electric shock to the machine system.

Note 3: Re-perform the machine zero return to get the correct position coordinate after the emergency alarm

is cancelled (machine zero return is forbidden if there is no machine zero on the machine.).

Note 4: Only Bit3 of the bit parameter No.172 is set to 0, is the external emergency stop active.


2.4.3 Feed hold

Press  key during the machine running to make the running pause. However, in threading cutting, tapping cycle state, the function cannot stop the running immediately.

2.4.4 Power-off

Under the dangerous or emergency situations during the machine running, the machine power should be cut off immediately to avoid the accidents. But it should be noted that there may be a large error between the CNC coordinates displayed and the actual position. So the toolsetting operation should be performed again.







CHAPTER 3 MANUAL OPERATION

Press  key, it enters Manual mode. In the mode, the system can perform the manual feed, spindle control, override adjustment, tool change etc..







3.1 Coordinate Axis Move

In Manual mode, 2 coordinate axes can perform manual feeding and rapid traverse.


3.1.1 Manual feed

Press  or  for moving x axis forward or backward, and the moving stops if the key is released; press  or  a for moving x axis forward or backward, and the moving stops if the key is released; press  key to make the indicator  activated and enter the manual rapid traverse mode.

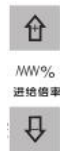
3.1.2 Manual rapid traverse


Press  key to make the indicator  activated, then Press  or  for moving x axis forward or backward, and the moving stops if the key is released; press  or  a for moving x axis forward or backward, and the moving stops if the key is released; In Manual mode, press key to make the indicator OFF, and the rapid traverse is disabled and JOG mode is executed.

Note 1: If no reference point return is performed after power on, as the rapid traverse switch is turned on (rapid indicator ON), the manual feedrate or rapid rate for the traverse is defined by the Bit0 of the bit parameter No.012 of this ECN*000T system.

Note 2: In Edit/MPG mode,  key is inactive.

3.1.3 Speed tune



In Manual mode, press  to alter the manual feed override which is divided into 16 steps. The relation of the override and the feedrate is as follows table when data parameter No.031 is set to 1260:

Feedrate override (%)	Feedrate (mm/min)
0	
10	126
20	252
30	378
40	504

Feederate override (%)	Feederate (mm/min)
50	630
60	756
70	882
80	1008
90	1134
100	1260
110	1386
120	1512
130	1638
140	1760
150	1890

Note : There is about 2% error for the data in the above table.

In the manual rapid traverse, it can press key to alter the rapid override, and there are 4 steps of F0, 25%, 50%,100% for the override.(F0 is set by data parameter No.032) The rapid override is active under the following conditions:

- (1) G00 rapid traverse (2) Rapid traverse in canned cycle
 (3) Rapid traverse in G28 (4) Manual rapid traverse

3.2 Other Manual Operations

3.2.1 Spindle CCW, CW, stop control



SPD.CW: In Manual mode, the spindle rotates counterclockwise if pressing this key;



SPD.SP: In Manual mode, the spindle stops if pressing this key;




SPD.CCW: In Manual mode, the spindle rotates clockwise if pressing this key.


3.2.2 Spindle jog





SPD.JOG: At the moment, the spindle is in JOG state.

Functional description:

Press  to enter JOG mode, and the spindle JOG function ON/OFF is executed only when the spindle is in the state of stop.

In spindle JOG mode, by pressing  key, the spindle rotates counterclockwise for jogging;

by pressing  key, the spindle rotates clockwise for jogging. The jog time and speed are set by data parameter No.108 and No.109 respectively.

When the spindle JOG rotates,  is pressed to stop the spindle JOG rotation, the spindle brake signal is not output when the JOG rotation stops.

K10.4 is set to 1, the spindle JOG is valid in any mode. In Auto or MDI mode, the spindle is in the JOG rotation state, the program closes the spindle JOG rotation and the JOG function.

Parameter setting:

PLC parameter K104 1/0: the spindle JOG is valid in any mode/Manual, MPG, Zero return mode.

Data parameter No.108: spindle JOG time

Data parameter No.109: rotary speed in spindle JOG.

3.2.3 Cooling control




In Manual mode, press this key, the cooling is switched on/off.


Parameter setting: PLC parameter K10.1 1/0: the spindle lubricating and cooling output remains/closes in reset.

3.2.4 Lubricating control

Function description:

1. Non-automatic lubricating (DT017 =0)

When PLC DT013=0, it is the lubricating turn output, Pressing  can execute the output. And the lubricating is cancelled by pressing it again. M32 is to execute lubricating output, and M33 is to cancel lubricating output.

When PLC DT013 >1, it is timing lubricating output. Pressing  can execute the output. And it is cancelled after a setting time by data parameter DT013; M32 is to execute lubricating output and the output is cancelled after a time set by DT013. If the setting time is not yet up, M33 is executed to cancel the lubricating output.

2. Automatic lubricating (DT017>0)

For automatic lubricating, the system executes the lubricating in the time set by DT017 and after time over, the output is off. After lubricating interval time set by DT016, the output is done again, it does repetitively like this. During the automatic lubricating, M32, M33 codes as well as the key are all active.

Parameter setting:

PLC parameter: K10.1 1/0: the spindle lubricating/cooling output remains/closes in reset;

PLC parameter:K16.2 1/0: whether the lubricating outputs in power-on when the automatic lubricating is valid;

PLC data DT005: M execution duration(ms);

PLC data DT013: lubricating start time (0-60000ms)(0:lubricating time is not limited);

PLC data: DT016 automatic lubricating interval time (ms);

PLC data: DT017: automatic lubricating output time (ms).

3.2.5 Chuck control



press it to switch the chuck releasing/clamping in any mode.

Function description:

When the chuck control is invalid, the system alarms in executing the chuck control M command;

When the system checks the chuck clamping (K12.1=1), the spindle cannot be started when the chuck is not clamped, and the chuck cannot be released after the spindle is started;

When using the input signal controls the chuck operation, the spindle should stop and delay the time set by DT21, the control is valid.

Parameter setting:

PLC parameter: K12.0 1/0: chuck control valid/invalid;

PLC parameter: K12.1 1/0: do not check/check the chuck clamping before the spindle start;

PLC parameter: K12.2 1/0: chuck outer/inner control mode;


PLC parameter: K12.3 1/0: check/not chuck in-position signal;

PLC data: DT018: chuck pulse output width(ms);

PLC data: DT021: spindle stops, chuck operation enabling delays(ms).

3.2.6 Spindle override


In Manual mode, if the spindle speed is controlled by analog voltage output, the spindle speed

may be overridden. Pressing  can change the spindle speed in real-time to adjust 8 steps of 50%~120% of the spindle override.


CHAPTER 4 MPG/STEP OPERATION


In MPG/Step mode, the machine moves by a specified increment.


4.1 Step Feed

Set the system parameter No.001 Bit3 to 0, and press  key to enter the STEP working mode.

4.1.1 Increment selection



Press  to select the move increment, the increment will be shown in the



page. When the BIT7(SINC) of K016 is 1,  step width value is inactive; when the BIT7 is 0,



all  are active. **Note!**

The key functions of this ECN*000T machine panel are defined by PLC program (ladders), please refer to the manuals by the machine builder for their significance. Please note that the following description for the key functions in this chapter is based on the ECN*000T standard PLC program!


4.1.2 Moving direction selection

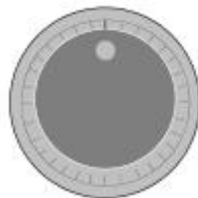
Pressing  or  key once can move X negatively or positively by a step increment;

pressing  or  key once can move Z negatively or positively by a step increment;

pressing  or  key once can move Y negatively or positively by a step increment.

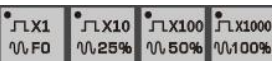
4.2 MPG(handwheel) Feed






Set the BIT3 of the system parameter No.001 to 1, and press  key to enter the MPG mode, The MPG figure is shown as follows:



The handwheel (MPG) figure

4.2.1 Increment selection

Press  key to select the move increment, the increment will be shown in

the window. When the BIT7(SINC) of PLC K016 is 1,  step is inactive; when BIT7 is 0, all     are active.


4.2.2 Moving axis and direction selection


In MPG mode, press     key, the corresponding axis will be selected.


The MPG feed direction is defined by its rotation direction. Generally, the MPG CW is for positive feed, and CCW for negative feed. In case of that MPG CW is for negative feed, CCW for positive feed, it may exchange the A, B signals of the MPG terminals. Bit0 ~ Bit4 of NO. 013 selects the feed direction of MPG rotation.

4.2.3 Other operations


1) Spindle CCW, CW, stop control

 SPD.CW: In Manual mode, the spindle rotates counterclockwise if pressing this key;


 SPD.SP: In Manual mode, the spindle stops if pressing this key;


 SPD.CCW: In Manual mode, the spindle rotates clockwise if pressing this key.


2) Spindle Jog


 SPD.JOG: At the moment, the spindle is in JOG state.

Functional description:

Press  to enter JOG mode, and the spindle JOG function ON/OFF is executed only when the spindle is in the state of stop.

In spindle JOG mode, by pressing  key, the spindle rotates counterclockwise for jogging;

by pressing  key, the spindle rotates clockwise for jogging. The jog time and speed are set by data parameter No.108 and No.109 respectively.

When the spindle JOG rotates,  is pressed to stop the spindle JOG rotation, the spindle brake signal is not output when the JOG rotation stops.

3) Cooling control

Refer to OPERATION, Section 3.2.3

4) Lubricating control

Refer to OPERATION, Section 3.2.4

5) Chuck control

Refer to OPERATION, Section 3.2.5

6) Spindle override tune

Refer to OPERATION, Section 3.2.5

4.2.4 Explanation items

The correspondence of the handwheel scale to the machine moving amount is as follows table:

	Moving amount of each MPG scale			
MPG increment	0.001	0.01	0.1	1
Specified coordinate value	0.001mm	0.01mm	0.1mm	1mm

Note 1: The MPG increment is related to the system's current metric/inch input state and the system's least input increment.


Note 2: The MPG speed cannot be more than 5r/s, otherwise, the scale value is inconsistent with the movement amount.




CHAPTER 5 MDI OPERATION

In MDI mode, the operations of parameter setting, code words input and execution can be performed.

5.1 Block Input

Select MDI mode to enter PROGRAM->MDI page, to input an block G50 X50 Z100, the steps are shown below:

1) Press  key to enter MDI mode;

2) Press , and then press  or  to enter MDI program page below:



The page is shown as follows after above operations are completed:



Note!

The key functions of this ECN*000T machine panel are defined by PLC program (ladders), please refer to the manuals by the machine builder for their significance. Please note that the following description for the key functions in this chapter is based on the ECN*000T standard PLC program!

5.2 Block Execution



key is pressed to execute the input block. During the execution, , and Emergency Stop key may be pressed to terminate these code words execution.

Note: The subprogram call codes (M98 P_ ; etc.), compound cutting cycle codes(G70, G71, G72, G73, G12.1, G7.1 and so on) is inactive in MDI mode.


5.3 Parameter Setting

In MDI mode, the parameter value can be altered after entering the parameter interface. The detailed is referred to **OPERATION, Chapter 10**.

5.4 Data Alteration

In PRG->MDI page, if there is an error during words inputting before executing the input block,




is pressed to cancel the display to alter the program, or  is pressed to clear all the input, then correct re-inputting is executed.

5.5 Other Operations

1. Adjusting spindle override


In MDI mode, the spindle speed can be adjusted when the analog voltage output controls the



spindle speed. Press  can adjust the spindle override to change the spindle speed, which can realize the 8 steps adjustment in real-time from 50%~120%.


2. Adjusting rapid override



Pressing  can execute the rapid traverse, which can realize the 4 steps adjustment in real-time.

3. Adjusting feedrate



In MDI mode, pressing  can adjust the feedrate override to change the feedrate, which can realize 16 steps adjustment from 0~150%.

CHAPTER 6 PROGRAM EDIT AND MANAGEMENT

In Edit mode, a program can be created, selected, altered, copied and deleted, and the bidirectional communication of CNC to CNC, or CNC to PC can also be done.

To prevent the program to be altered or deleted accidentally, a program switch is set up for this ECN*000T system. And it must be turned on before program editing. Please see details in Section 10.1.1 of the part.


The system provides multi-level user permission to facilitate the management. More than 4 levels (4 level, 3 level) can open the program switch to edit programs. See **OPERATION**, Section 10.1.2.

6.1 Program Creation


6.1.1 Creating a block number

In the program, a block number can be added or not, the program is executed by the sequence. When the “AUTO NO” switch in SWITCH SETTING window is OFF, CNC doesn’t generate the block number automatically, but the blocks may be numbered manually.

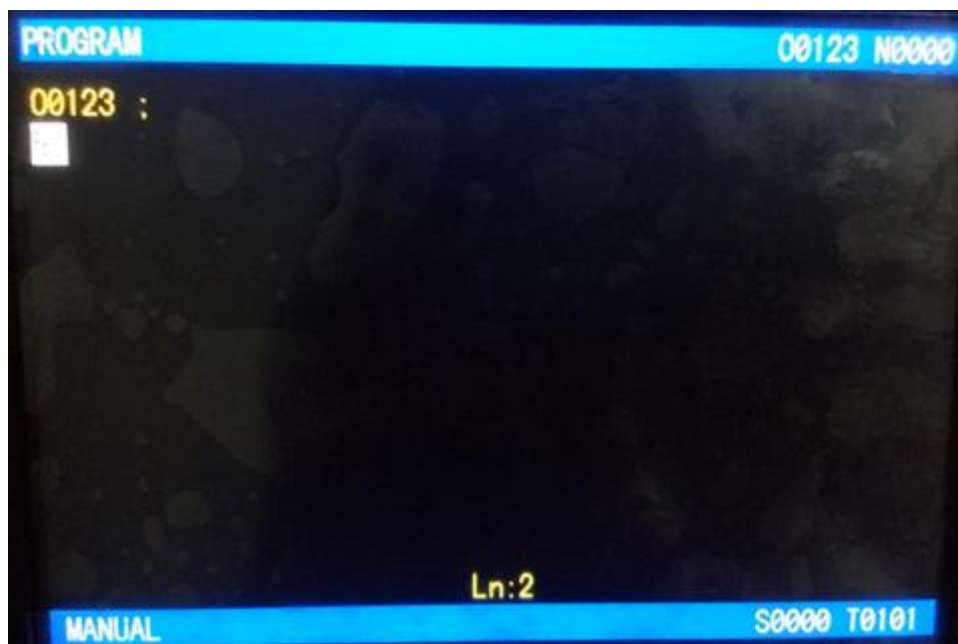
When the “AUTO NO” switch in SWITCH SETTING window is ON, CNC generates the block






number automatically, it automatically generates the next block number by pressing  in editing. The block number increment is set by the CNC data parameter No.042. (See details in Section 10.1.1 of this part.)

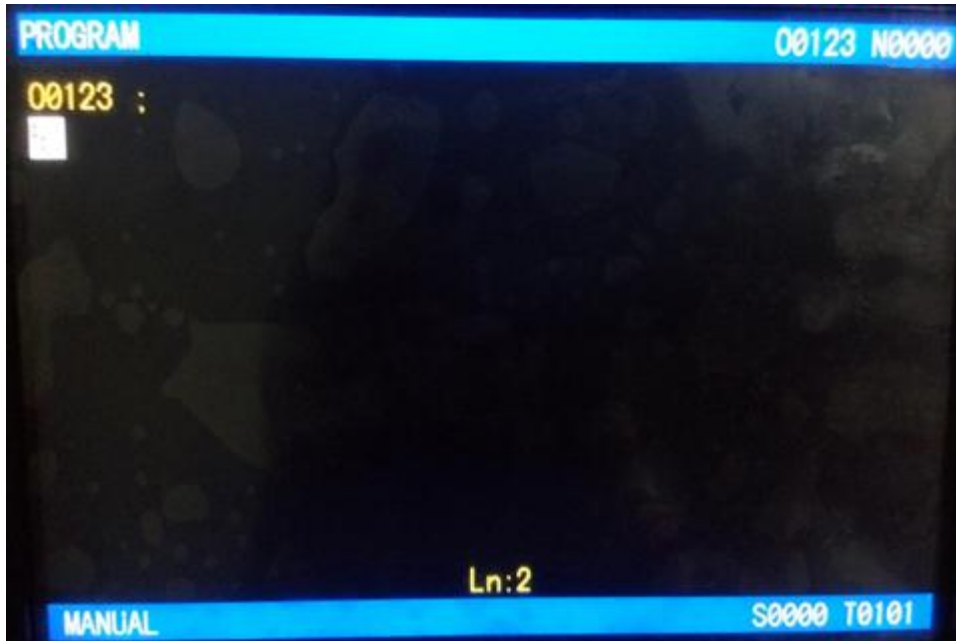
6.1.2 Inputting a program

Press  to enter the Edit mode. Press to enter the Program interface, create a machining program before inputting a machining program. The creating method is shown below:

Method 1:



Press address key , number key , , ,  and key by sequence (e.g. taking an example of creating program O0123, the leading 0 can be omitted when its name is input). Its display is shown below:



Note: When a program is needed to create, the input program name exists, the system opens the file, otherwise, it automatically create a new one.

Input the edited part program one by one, the character will be displayed on the screen immediately as it is input(as for compound key, press this key repeatedly for alternate input), after a block is finished, press or to terminate it.






Note: The unexpected power-off when the program is input, the program being edited cannot be saved.

At edit mode, the command should be completed instruction, like as G0 ,X100 ,only G or M word ,or number can not be recognized.

If there is wrong input, press  to delete instructions.





6.1.3 Searching a character

1. Scanning: To scan the character one by one by cursor

- 1) Press key  to enter the Edit mode, then press  key to enter the PRG CONTENT window;
- 2) Press  key, the cursor shifts a row upward; if the number of the column where the cursor locates is over the total columns of the previous row, the cursor moves to the previous block end (at “;” sign) after key  is pressed;
- 3) Press  key, the cursor shifts a row downward; if the number of the column where the


cursor locates is over the total columns of the next row, the cursor moves to the next block end (at

“,” sign) after the key  is pressed;

- 4) Press  key, the cursor shifts a column to the right; if the cursor locates at the row end, it moves to the head of the next block;
- 5) Press  key, the cursor shifts a column to the left; if the cursor locates at the row head, it moves to the end of the next block;
- 6) Press  key to window upward, the cursor moves to the 1st row and 1st column of the previous window, if it windows to the head of the program, the cursor moves to the 2nd row and 1st column;
- 7) Press  key to window downward, the cursor moves to the 1st row and 1st column of the next window, if it windows to the end of the program, the cursor moves to the last row and 1st column of the program;



Note : During the searching, it doesn't search the characters in the called subprogram, and the character in subprogram is searched in subprogram.

4. The method to return to the program head

- 1) In Program interface of the Edit mode, press  key, the cursor returns to the program head;
- 2) Search the program head character by the methods in OPERATION, Section 6.1.3.

6.1.4 Inserting a character

Steps:

- 1) Press  to enter PRG CONTENT page in Edit mode;
- 2) Press  key to enter the INS mode (the cursor is an underline)
- 3) Input the character to be inserted

Note 1: In the Insert mode, if the cursor is not located at the row head, a space will be automatically generated when inserting the code address; if the cursor is located at the row head, the space will not be generated, and it should be inserted manually.


Note 2: In the Insert mode, if the previous bit before the cursor is a decimal point and the cursor is not located at the row end, input an address word, the “0” will be added automatically following the decimal point.

Note 3: In the Insert state, if the previous bit before the cursor is a decimal point and the cursor is not located at the row end, the “0” will be added automatically following the decimal point by pressing key.

6.1.5 Word of insert



Operating procedures are as follows:

- 1) Select edit operation mode, the program content display page;

2) Input to insert the words, press the  key, the system will insert the input to the left of the cursor.


6.1.6 The deletion of the word

Operating procedures are as follows:

- 1) Select edit operation mode, the program content display page;
- 2) Put the cursor to the location of the need to delete,  key, the system will delete the contents of a cursor, if continuous key , will remove the cursor to the right of the program content continuously.



6.1.7 Word modify

Operating procedures are as follows:

- 1) Select edit operation mode, the program content display page;
- 2) Move the cursor to the place where need to modify and input to modify the word, and then key , system to replace the contents of the cursor to the content of the input.


6.1.8 The remove of the procedures section

Operating procedures are as follows:

- 1) Select edit operation mode, the program content display page;
- 2) Move the cursor to the first (column 1) delete the line of the procedures section, key , key  again to delete cursor.

6.1.9 To delete multiple procedures section

Operating procedures are as follows:

- 1) Select edit operation mode, the program content display page;
- 2) Move the cursor moves to remove the beginning of a line of the procedures section;
- 3) Input to remove part of the last the serial number of the procedures section;
- 4) Key , the cursor can be delete procedures section between address with the markers.

6.1.10block delete

Operating procedures are as follows:



- 1) Select edit operation mode, the program content display page;
- 2) Delete the cursor moves to block the first code word;
- 3) Input to remove block finally a code word;

- 4) Key  DEL, the cursor can be address block between deleted with the markers.

Note 1: if the deleted successfully, the input code word will disappear, or delete failed. If the specified character downs how much the same, is the distance from the current cursor on a recent by default.


6.1.11 The copy of the procedures section

Operating procedures are as follows:

- 1) Select edit operation mode, the program content display page;
- 2) Move the cursor to the copy of the beginning of a line of the procedures section;
- 3) Press the key , key  again, copy the cursor procedures section.


6.1.12 More than the copy of the procedures section

Operating procedures are as follows:

- 1) Select edit operation mode, the program content display page;
- 2) The cursor to the first want to copy the program code words;
- 3) The input serial number to copy program period of the last paragraph;
- 4) Procedures section between key , the cursor to the input character copy complete.

6.1.13 Block replication


Operating procedures are as follows:

- 1) Select edit operation mode, the program content display page;
- 2) The cursor moved to copy the first code block;
- 3) Input to replicate the end of the block a code word;
- 4) Key , the cursor to the input character between block replication is complete.

6.1.14 Block of the paste

Operating procedures are as follows:







- 1) Select edit operation mode, the program content display page;
- 2) Move the cursor to the program location of paste;

3) Key , to copy the last program content is inserted into the cursor before, to complete the paste operation.

6.2 The deletion of the program








6.2.1 To delete a single program

Steps are as follows:

- 1) Select edit operation mode, enter the program display the page;
- 2) Type the address in turn key , , , ,  the number key (O0001 program, for example)
- 3) Key , O 0001 program is deleted.

6.2.2 Delete all program






Steps are as follows:


- 1) Select edit operation mode, enter the program display the page;
- 2) Type the address key  in turn, the symbol key , the number key , , ,  ;
- 3) Key , all program are deleted.

6.3 The selection of the program





When CNC has multiple programs, you can through the following three methods selection process.

6.3.1 Retrieval method










- 1) Select edit or automatic operation mode;
- 2) The key , and enter the program content display screen;
- 3) Press the address , type the number program;
- 4) According to the press  or  or in an automated way , on the display screen displays the retrieved process, if the program does not exist, CNC in alarm.

Note: step 4, the edit operation mode, if the program does not exist, keys , CNC will create a new program.



6.3.2 Scanning method

- 1) Select edit or automatic operation mode;
- 2) Key , display screen and enter the program;
- 3) Press the address  ;
- 4) Press  or  to display next or on a program;
- 5) Repeat steps 3, 4, one at a show in the program.

6.3.3 The cursor recognition method



- 1) Select automatic operating mode (must be in the running state);
- 2) Display the page key , enter the program directory;
-) Press  or  the cursor move up/down, or press  , up/down turn pages, or press   the cursor to the first/last program in the directory, move the cursor to be selected on the program name (the cursor at the same time, the program content changes);
- 4) Key  or a key 

6.4 Program name

- 1) Select edit operation mode, enter the program display the page content;
- 2) Press the address  , type the new program name;
- 3) key .


6.5 Copy of program

Save the current program:




- 1) Select edit operation mode, enter the program display the page content;
- 2) Address key  , type number;
- 3) Key .

6.6 Program management

6.6.1 Program directory

Key  to enter the show program directory page. In this page, in the form of table of contents shows CNC existing program name, one page only shows up to 10 program name, when the number of more than 10 existing applications, can press up/down page key displays another page program directory.

1) Open the program

Open the specified procedures:  + number + enter (or ) or serial number + enter (or ) .

In edit mode, if the input sequence number does not exist will create program.

2) The deletion of the program:

1. Edit mode According to  remove the cursor specified procedure.
2. Edit mode  + serial number+  Or the serial number+ .

6.6.2 The number of stored procedures

This system the number of stored procedures for a maximum of 400, the show CNC can the current program number has been stored and the rest of the parts.

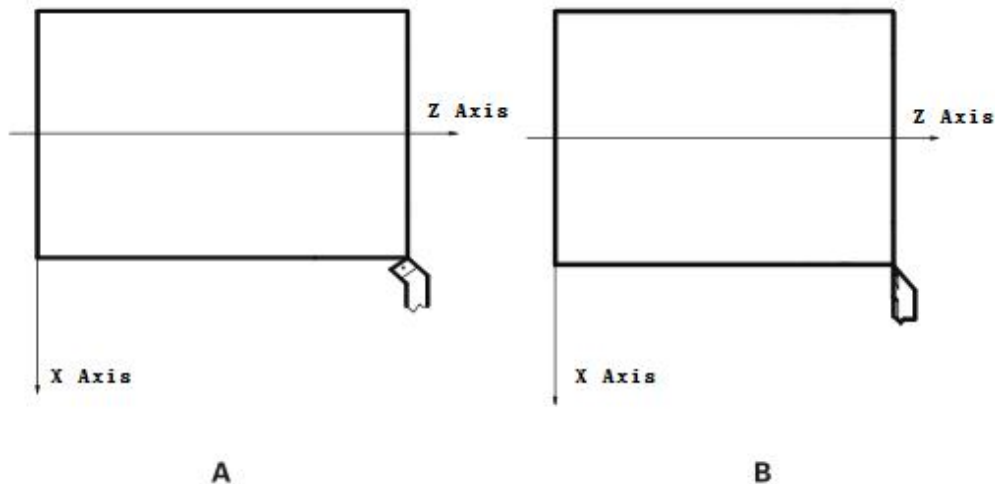
6.6.3 Memory capacity

The application of this system storage capacity (total capacity of 56 m), this shows the current has been occupied storage capacity and the remaining storage capacity.

CHAPTER 7 TOOL OFFSET AND SETTING

The actual location of tool can be overlooked in programming for simplifying programming. Three methods including positioning toolsetting, trial toolsetting and machine zero toolsetting are available in this ECN*000T system. The tool offset data are obtained from this toolsetting operation.

7.1 Tool Positioning Setting



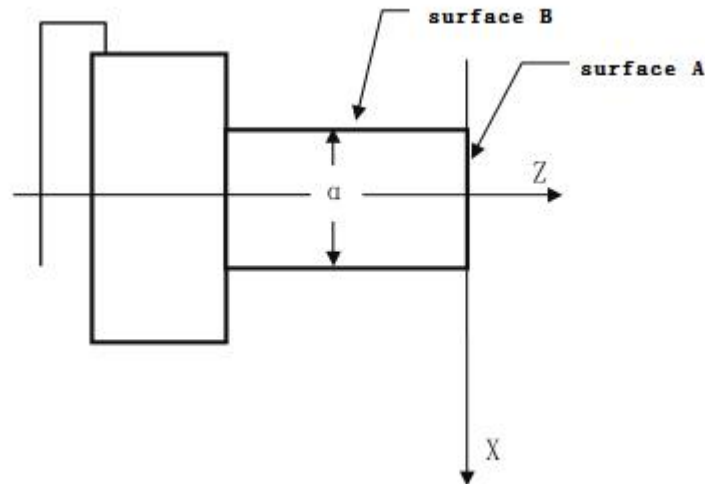
Steps:

1. Firstly determine if the offset values are zero in X, Z, if not, clear all the tool number offset values;
2. Set the offset No. for 00 (i.e. T0100,T0300), as for the offset value: (method: execute a move code or perform the machine zero return in T0100 state, then clear the offset value automatically as returning to the machine zero);
3. Select a tool by random (usually the 1st tool, this tool will be used as the reference tool);
4. Position the tool nose of the reference tool to a point (toolsetting point), as shown in A;
5. In PRG STATE page of the MDI mode, set up the workpiece coordinate system by the command G50 X__ Z__;
6. Clear the relative coordinate (U, W), see details in APPENDIX;
7. After the tool is moved to a safety height, select another tool and move it to the setting point, as shown in B;
8. Press **补偿** **OFT** key and move the cursor by **↑** , **↓** key to select the corresponding offset number of that tool;
9. Press address **U** key , then press **输入** **IN** key to input the tool offset value of X axis into the corresponding offset number;
10. Press address **W** key , then press **输入** **IN** key to input the tool offset value of Z axis into the corresponding offset number;
11. Repeat the steps from 7 to 10 to perform the toolsetting operation for other tools.

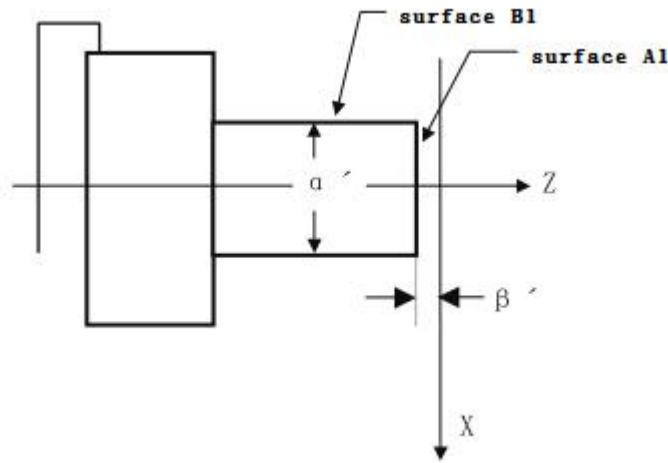
Note: For the fixed toolsetting, the original system tool offset should be cleared, input the new offset one time instead of many times by pressing , Refer to II OPERATION, Section 7.4.4 about the tool compensation clearing method.

7.2 Trial Toolsetting

Whether the method of trial toolsetting is inactive is defined by the system parameter No.012 Bit5. Steps (workpiece coordinate system by part end surface):



- 1) Select a tool by random and make it cut on Surface A;
- 2) Retract the tool along X axis without Z axis moving and stop the spindle;
- 3) Press **补偿** **OFT** key to enter the Offset interface, select the TOOL OFFSET page
- 4) Key in by sequence the address key **Z**, number key **0** and **输入** **IN** key;
- 5) Make the tool cut along Surface B;
- 6) Retract the tool along Z axis without the movement of X axis, and stop the spindle
- 7) Measure the diameter "α" (supposing $\alpha=15$) ;
- 8) Press **补偿** **OFT** key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing **↑**, **↓** key to select the corresponding offset number;
- 9) Key **X** in the address key by sequence, number key **1**, **5** and **输入** **IN** key;
- 10) Move the tool to a safety height to change for another tool;



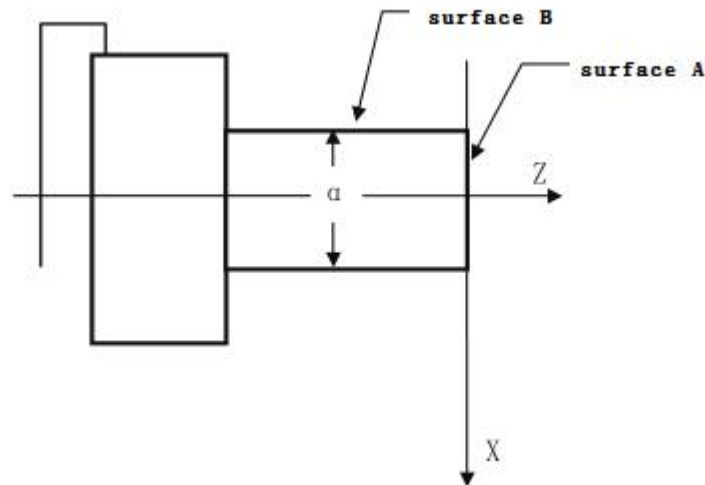
- 11) Make the tool to cut on Surface A1;
- 12) Retract the tool along X axis without Z axis moving and stop the spindle;
- 13) Measure the distance " β' " between the Surface A1 and the workpiece coordinate origin(supposing $\beta' = 1$);
- 14) Press **补偿** **OFT** key to enter the Offset interface, select the TOOL OFFSET page, then move the cursor by pressing **↑** , **↓** to select the corresponding offset number;
- 15) Key in by sequence the address key **Z** , symbol key **-+** , number key **1** , and **输入** **IN** key;
- 16) Make the tool to cut on Surface B1;
- 17) Retract the tool along Z axis without the movement of X axis, and stop the spindle;
- 18) Measure the distance " α' " (supposing $\alpha' = 10$);
- 19) Press **补偿** **OFT** key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing **↑** , **↓** to select the corresponding offset number;
- 20) Press orderly the address key **X** , number key **1** , **□** and **输入** **IN** key;
- 21) Repeat the execution from Step 10 to Step 20 to perform the toolsetting operation for other tools.
















Note: The offset value may be large by this toolsetting method, so the tool compensation should be done by the coordinate offset by the CNC system. (set the BIT4 of the CNC parameter No.003 to 1). Moreover, the tool lengths compensation should be performed by using the T code in the 1st block, or the 1st move block should contain the T code for the tool length compensation.

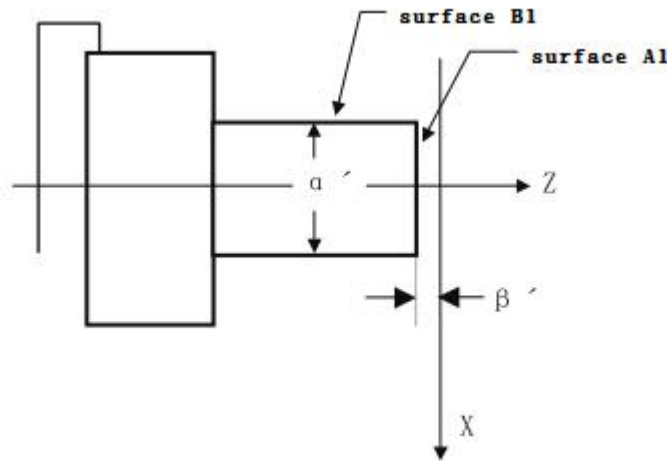
7.3 Toolsetting by Machine Zero Return

There is no reference tool in this toolsetting methods, when the tool is worn or to be adjusted, it only needs to be set again, and a machine zero return should be done before the toolsetting. The

machining could be continued by performing a machine zero return at power on after power-off, which is very convenient for the operation. Steps (workpiece coordinate system by part end surface):



- 1) Press  key to enter Machine Zero mode, move axes to machine zero;
- 2) Select a tool by random and set the offset number of the tool to 00 (e.g. T0100, T0300) ;
- 3) Make the tool to cut on Surface A;
- 4) Retract the tool along X axis without the movement of Z axis, and stop the spindle; directly press , and the CNC records the absolute values of the position, at the moment, directly move the tool;
- 5) Press  key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing ,  key to select the corresponding offset number;
- 6) Key in by sequence the address key , number key  and key  to set the offset value of Z axis;
- 7) Make the tool cut along Surface B;
- 8) Retract the tool along Z axis without the movement of X axis, and stop the spindle;
- 9) Measure the distance "α"(supposing $\alpha=15$);
- 10) Press  key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing ,  to select the corresponding offset number;
- 11) Key in by sequence the address key , number key ,  and  key the offset value of X axis;
- 12) Move the tool to a safety height for tool change;
- 13) Change for another tool, and set the tool offset number to 00 (i.e. T0100, T0300);



14) Make the tool to cut on Surface A1;

15) Retract the tool along X axis without Z axis moving and stop the spindle; measure the distance " β " between the Surface A1 and the workpiece coordinate system origin (supposing $\beta=1$);

16) Press **补偿** **OFT** key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing **↑** , **↓** to select the corresponding offset number;

17) Key in by sequence the address key **Z** , symbol key **-+** , number key **1** , and **输入** **IN** key;

18) Make the tool to cut on Surface B1;

19) Retract the tool along Z axis without the movement of X axis, and stop the spindle;

20) Measure the distance " α " (supposing $\alpha=10$);

21) Press **补偿** **OFT** key to enter the Offset interface, select the TOOL OFFSET window, then move the cursor by pressing **↑** , **↓** t key to select the corresponding offset number;

22) Key in by sequence the address key , number key , and key to set X offset value;

23) Move the tool to a safety position;

24) Repeat the execution from Step 12 to Step 23 to perform the toolsetting operation for other tools.

Note 1: Machine zero switch must be fixed for the toolsetting operation by machine zero return.

Note 2: The workpiece coordinate system setting can't be done by G50 code after toolsetting by machine zero return.




Note 3: The tool compensation should be done by coordinate offset by the CNC system (the system parameter No.003 Bit4 is set to 1), further more, the tool lengths compensation should be performed by using the T code in the 1st block, or the 1st move block should contain the T code for the toollength compensation.


coordinate system is (a,b), the setting value of system parameter No.047 should be close

7.4 Setting and Altering the Offset Value

Refer to Section 1.3.3 about methods of entering the offset setting page.




7.4.1 Offset setting

- 1) Press  key to enter the offset setting page, select  or  the desired window by pressing the , ;
- 2) Move the cursor to the location of the tool offset, wear number to be input.


scanning: Press  ,  to move the cursor in sequent

Searching: use the following press key to directly move the cursor to the input position:

Press  + offset number + 

- 3) After pressing the address key  or  , the numerical number may be keyed in (decimal point allowed);
- 4) Press a  and the CNC calculates the offset value automatically and displays the result in the window.


7.4.2 Offset alteration

- 1) Refer to OPERATION, Section 7.4.1, and move the cursor to the location of the offset number that is to be altered;
- 2) If the offset value of X axis is to be altered, key in U; as for that of Z axis, key in W; ;
- 3) Then input the incremental value;
- 4) Press  to add the current offset value to the value keyed in, the operation result will be displayed as a new offset value.

Example: The set X axis offset value is 5.678 The increment is U 1.5

Then the new offset value is 7.178 (=5.678+1.5)

7.4.3 Clearing the offset values

- 1) Move the cursor to the offset number to be cleared;
- 2) When the offset value of X axis is to be cleared,  is pressed, then is done, the offset will be cleared. Other axis clearing is done like that;

Note: The offset clearing in the tool offset page doesn't mean that the system in under the state with no offsetting, if the system is needed to be in this state, the offsetting is required to be executed,

Execute a positioning command in T0100 state or perform a machine zero return. After the offsetting is finished, the "□□" in "T□□□□" shown at the right bottom of the screen will not be backlighted.

7.4.4 Setting and altering the tool wear

To prevent the mistaken operation of the setting and alteration of the offset value (decimal point missed, mislocated etc.), which may cause the tool collision by oversize offset value, for the visual judgement for the tool wear by the operator, the TOOL WEAR window is set in this ECN*000T system. When the offset value is needed to be altered due to the inaccurate dimensions by the tool

wear, it may set or alter the wear value. The wear input range is defined by the data parameter No.140, and they are saved even at power down.

The setting and alteration methods for the tool wear are approximately identical to that of the tool offset, and the wear value is input by U(X axis), W(Z axis), V(Y axis).

7.4.5 No.0 tool offset moving workpiece coordinate system

When No.012 Bit 6 is set to 1, No. 0 tool offset moving workpiece coordinate system is valid.

After the value is input in No. 0 tool offset, the workpiece coordinate system executes the offset based on the input value.

Before inputting No.0 tool offset

After inputting No.0 tool offset

As the above figure, after X100, Z100 in No.0 tool offset is input, the workpiece coordinate system offsets X100, Z100.

Note: When No. 0 offset modification is valid real-time, No. 0 tool offset must be set before the system runs a program, otherwise, the run path will offset.



CHAPTER 8 AUTO OPERATION

8.1 Automatic Run


8.1.1 Selection of a program running

Refer to Section 6.3.

8.1.2 Start of automatic run

1. Press  to select the Auto mode;
2. Press  to start the program, and the program automatically runs.

Note: Since the program execution begins from the block where the cursor locates, before pressing the


, make a check whether the cursor is located at the block to be executed.

8.1.3 Stop of automatic run


Stop by code (M00)


1. M00

After the block containing M00 is executed, the auto run is stopped. So the modal function and



state are all reserved. Press the  or the external run key, the program execution continues.

2. M01

Press  and the optional stop indicator is ON and the function is valid. After the block with M01 is executed, the system stops the automatic run, the modal function and the state are saved.

Press  or the external run key, and the program continuously runs.

Stop by a relevant key

1. In Auto run, by pressing  or external pause key, the machine keeps the following state:
 - (1) The machine feed slows down to stop;
 - (2) The modal function and state are reserved;
 - (3) The program execution continues after pressing  the key.

2. Stop by Reset key

- (1) All axes movement is stopped.
- (2) M, S function output is inactive (the automatic cut-off of signals such as spindle CCW/CW,

lubricating, cooling by pressing  key can be set by the parameters)

- (3) Modal function and state is held on after the auto run.

3. Stop by Emergency stop key

If the external emergency key (external emergency signal active) is pressed under the dangerous or emergent situation during the machine running, the CNC system enters into emergency state, and the machine moving is stopped immediately, all the output (such as spindle rotation, cooling) are all cut off. If the Emergency key is released, the alarm is cancelled and CNC

systementers into reset mode.

4. Switching operation mode

When Auto mode is switched to the Machine zero, MPG/Step, Manual, Program zero mode, the current block “dwells” immediately; when the Auto mode is switched to the Edit, MDI mode in Automode, the “dwell” is not displayed till the current block is executed.

Note 1: Ensure that the fault has been resolved before cancelling the emergency alarm.

Note 2: The electric shock to the device may be decreased by pressing the Emergency key before power on and off.

Note 3: The Machine zero return operation should be performed again after the emergency alarm is cancelled to ensure the correctness of the position coordinates (but this operation is forbidden if there is no machine zero in the machine).

Note 4: Only the BIT3 (MESP) of the bit parameter No.172 is set to 0, could the external emergency stop be active.

8.1.4 Automatic run from an arbitrary block



Press key to enter the Edit mode, press key to enter the program content page;



1) Move the cursor to the block to be executed (for example, move the cursor to the 3rd row head if it executes from the 3rd row);

2) If the mode (G, M, T, F code) of the current block where the cursor locates is defaulted and inconsistent with the running mode of this block, the corresponding modal function should be executed to continue next step.



3) Press key to enter the Auto mode, then press key to start the execution.



8.1.5 Adjustment of the feedrate, rapid rate

In Auto mode, the running speed can be changed by adjusting the feedrate override, rapid override. It doesn't need to change the settings of the program and parameter.

Adjustment of the feedrate override



Press 16-level real time feedrate can be obtained.

Note 1: The actual feedrate value is specified by F in feedrate override adjustment;

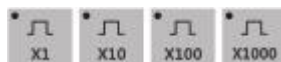
Note 2: Actual feedrate= value specified by F× feedrate override

Adjustment of rapid override

It can realize the F0, 25%, 50%, 100% 4-level real time rapid override adjustment by



pressing



or

Note 1: The rapid traverse speeds of X, Z axis are set by the system parameter No.022, No.023

X axis actual rapid traverse rate = value set by parameter No.022X×rapid override


Z axis actual rapid traverse rate = value set by parameter No.023Z×rapid override

Note 2: When the rapid override is F0, the min. rapid traverse rate is set by bit parameter No.032.

8.1.6 Spindle speed adjustment

While the spindle speed is controlled by the analog voltage output in Auto mode, it can be adjusted by spindle override.



Press  to adjust the spindle override for the spindle speed, it can realize 8-level real-time override adjustment between 50%~120%.

Note : The actual output analog voltage = analog voltage by parameter × spindle override.


Example: ParameterNO.037 is set 9999, if execute S9999, and select 100%, then the analog output is 10VDC.


8.2 Running State

8.2.1 Single block execution

When a program is executed firstly, the system selects the single block run mode to avoid the program error to cause the unexpected.



Press  to select the single block run function in Auto mode; in single block mode, after the

current block is executed, the CNC stops run; press  to execute the next block. Such repetition is executed till the program is done.

Note 1: The single block stops at the mid point of G28 code.


Note 2: For the single block state in the execution of canned cycle codes G90, G92, G94, G70~G76, refer to the 1st part PROGRAMMING.

Note 3: While the subprogram calling (M98 __), or subprogram calling return (M99) is being executed, the single block is inactive. But it is active except for N, O, P addresses in the block that contains M98 or M99 code.

8.2.2 Dry run

Before the program is to be executed automatically, in order to avoid the programming errors, it may select the Dry run mode to check the program.



In Auto mode, press  to enter the dry run state; in Dry run state, the machine feed and miscellaneous functions are both active (as machine lock, MST lock are both OFF), that means the dry run switch has nothing to do with the machine feeding, MST functions, so the feedrate by program is inactive and the CNC system runs at the speed described in the following table:

	Program command	
	Rapid traverse	Cutting feed
Rapid traverse switch ON	Rapid traverse	Max. manual feedrate
Rapid traverse switch OFF	Manual feedrate or rapid traverse(see note)	Manual feedrate


Note 1: The rate by manual feedrate or rapid rate is set by the BIT6 of the CNC system parameter No.004.

Note 2: The shift of rapid switch in Dry run mode doesn't affect the rate of the current block being executed, but that of the next block.

Note 3: The switch operation of Dry run is inactive if the ladder of this ECN*000T is defined to be in auto running state (Auto, MDI mode).

8.2.3 Machine lock




In Auto mode, press  to enter the machine lock; the machine lock and MST lock are usually used together to check the program. While the machine is in the lock run state:

1. The machine carriage doesn't move, the "MACHINE" in the INTEGRATED POS window of Position interface doesn't vary too. The RELATIVE POS and ABSOLUTE POS, DIST TO GO are refreshed continuously, which is the same as that the machine lock switch is OFF.
2. M, S, T commands can be executed normally.

8.2.4 MST lock



In Auto mode, press  to enter the miscellaneous function lock state; The machine carriage moves without the M, S, T code being executed. The machine lock and MST lock are usually used together to check the program.

Note: When the MST lock is active, it takes no effect to the execution of M00, M29, M30, M98, M99.

8.2.5 Block skip

When a block in program is not needed to be executed and not to be deleted, this block skip function can be used. When the block is headed with "/" sign and Block skip indicator lights up (panelkey active or external skip input active), this block is skipped without execution in Auto



mode. In Automode, press  to enter the state that the program skip is valid.


Note: While the block skip switch is off, the blocks headed with "/" signs are executed normally in Auto mode.

8.3 Other Operations




1. In Auto mode, press  key to switch on/off the cooling;



2. Press any of the  key to switch the operation modes;



3. Press  the key to reset this CNC system.

4. Automatic lubricating operation (Refer to **OPERATION, Chapter 3**).


CHAPTER 9 ZERO RETURN OPERATION

9.1 Program Zero Return

9.1.1 Program Zero

While the part is fixed on the machine, absolute coordinate of current tool position may be set by G50 code according to the relative position between the tool and the part, so a workpiece coordinate system is setup. The tool current position is called **program zero**, and this is the program zero return position.

9.1.2 Program zero return steps

1. Press  key, it enters the Program zero return mode, the bottom line of the window displays “P. ZERO”:
2. Press the direction key of X, Z, Y, 4th, C axis, it returns to the program zero of X, Z, Y, 4th or C axis;
3. The machine axis moves toward the program zero return, and the axis stops with the program zero return completion indicator ON after the axis returns to the program zero.



Note 1: The tool offset is not changed for the program zero return operation, if there is offset, the return position is the point set by G50.






Note 2: Whether the key is held on at program zero return is defined by the bit parameter No.011 BIT2 (zero return is locked automatically).

9.2 Machine Zero Return

9.2.1 Machine Zero

The **machine coordinate system** is a reference coordinate system for CNC coordinate operation. It is an inherent coordinate system of the machine. The origin of the machine coordinate system is called machine zero (or machine reference point). It is defined by the zero or zero return switch fixed on the machine. Usually this switch is fixed at the positive stroke point of each axis.

9.2.2 Machine Zero return steps

1. Press  key, it enters the Machine zero mode, the bottom line of the window displays “REF”:
2. Press , , ,  key to return to the machine zero of X, Z, Y, 4th or C axis;
3. The machine axis returns to the machine zero via the deceleration signal, zero signal detection. At the machine zero, the axis stops, and the corresponding machine zero return completion indicator lights up.



Note 1: If there is no machine zero on the machine, the machine zero operation is forbidden;

Note 2: The machine zero finish indicator is gone out on condition that: 1)The axis is moved out from

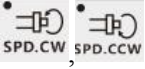
machine zero; 2) CNC is powered off.

Note 3: After the machine zero operation, the tool length compensation is cancelled by CNC;


Note 4: Parameters related to machine zero return are referred to INSTALLATION and CONNECTION.


Note 5: After the machine zero return is executed, the original workpiece coordinate system is set again with G50.


9.3 Other Operations in Zero Return


1) The spindle rotates  (CW, CCW), stops.


2) The cooling  is ON or OFF.

3) The lubricating contro  LUBE I.

4) The spindle override tuning .


5) Rapid override tuning .

6) Feedrate override tuning .

7) Hydraulic chuck control .

CHAPTER 10 DATA SETTING

10.1 Data Setting

Press  and then to enter the CNC setting page which includes the switch setting, level setting and parameter operation function as follows:



10.1.1 Switch setting

When the cursor is in the switch set column, the system can set parameters, programs, automatic sequence number ON/OFF state.

1) Switch explanation:





Parameter switch: parameters can be modified when the parameter switch is ON; they are forbidden to alter when it is OFF.

Program switch: programs can be edited when the program switch is ON; they are forbidden to edit when it is OFF.

Automatic block number: the block number is automatically generated when the automatic sequence number switch is ON and a program is edited; when the switch is OFF, the block number is input manually instead of being automatically generated.

2) Switch setting method:

a) Move the cursor to the setting item;

b) Press  (or ) to open the switch, press  (or ) to close it;

Note: When the parameter switch is shifted from “OFF” to “ON”, an alarm will be issued by CNC system.

The alarm can be cancelled by pressing  and . If the PARM SWT is shifted again, no alarm is issued. For security it should set the PARM SWT for “OFF” after the parameter alteration is finished.

10.1.2 Level setting

To prevent programs, CNC parameters from being modified at will, ECN*000T provides the level

setting function, and its password grade is divided into five, from high to low grade: 2nd grade (machine manufacturer), 3rd grade (device management), 4th grade (engineer), 5th grade (machining operation), 6th grade (operation limit). The current operation grade is displayed in “Current Level: _” in the level setting page. 2nd grade: it is for the machine manufacturer, which permits the machine manufacturer to alter the state parameters, data parameters, pitch compensation data, tool compensation data, to edit programs, to transmit PLC and so on.

3rd grade: it permits to alter state parameters, data parameters, tool compensation data and to edit programs.

4th grade: it permits to alter tool compensation data (toolsetting operation), macro variables, to edit programs, but not to alter state parameters, data parameters and pitch compensation data.

5th grade: it permits to alter tool compensation data, not to select and edit programs, and not to alter state programs, data programs and pitch compensation data.

6th grade: it has no level to operate only the machine operation panel, not to alter tool compensation data, not select and edit programs, not to alter state parameters, data parameter and pitch compensation data.

1) Operation grade entry

- a) Move the cursor to the line “INPUT PASSWORD” in the CNC setting page;
- b) Input the operation password (the system displays one “*” as soon as a number is input);
- c) Press and the operator can enter the corresponding grade operation;
- d) After the operator enters the corresponding operation grade page, the prompt column prompts the system’s current grade, and the corresponding operations as follows:

Note 1: The defined password length corresponds to the operation grade, and the user cannot increase or reduce the length at will, which is shown below:

Operation

grade Length Initial password

3rd grade 5 digits 12345

4th grade 4 digits 1234

5th grade 3 digits 123

6th grade None None

Note 2: When the operation level is less than or equal to the 3rd grade (3rd, 4th, 5th or 6th grade) and the system is turned on again, the operation level does not change. When the level is the 2nd grade and the system is turned on again, the operation level defaults to the 3rd grade.

2) Password modification

Steps:

- a) The operator enters the grade to alter the password according to the method “Operation grade entry” after the system enters the setting page;
- b) Move the cursor to the line “ALATER PASSWORD”;

Chapter 10 Data Setting, Backup and Restore

- c) Input the new password, and then press ;
- d) The CNC prompts “ Input the new password again”;
- e) Input the new password again and then press . When the two input passwords are identical, the CNC prompts “Password has been altered, please save the new password.” The password alteration is done successfully.

Note: The system prompts “ The new password is not identical with the confirmation” when the two

input passwords are not identical. At the moment, input the new password again.


3) Operation grade degradation

The degradation is to degrade from the high to the low, and its steps are shown below:

a) Input the password according the method of “Operation grade entry” after the system enters the setting page;

b) Move the cursor to the line “CURRENT LEVEL” and the system pops up

c) Press  , and the CNC prompts “Password degrades one grade”

d) Press  again, and the degradation is done successfully

Note 1: The degradation operation cannot be executed when the current level is the 6th grade.

Note 2: The current password can be altered after degradation. When the system is in the high level, the degradation can be executed to set to the low-level password.

10.1.3 Parameter operation

The parameter data (state parameter, data parameter) can be backup (stored) and recovered (read). Programs can be stored in the CNC when the backup or recovery is executed. The display is shown below:

STATUS PARAMETER				00001 N0000	
No.	Data	No.	Data	No.	Data
001	00011000	011	00000000	176	00000111
002	00000011	012	00100001	177	10000001
003	00110011	013	10000000	178	00000000
004	01000000	014	00000000	179	00000000
005	00010000	164	00000000	180	00000010
006	00010000	168	10001101	181	00000000
007	00011100	172	00010000	182	00000000
008	00011111	173	00000000	183	00000000
009	10000011	174	10000000	184	00110011
010	00000000	175	00000000	185	00000000

**** **** **** SPTY SOHW RDC **** INI
 BIT0: (0:Metric 1:Inch) Input
 NO. 001
 EDIT ALARM S0000 T0101

1) Option explanation:

Backup current parameters (user): the user to backup the CNC data (store)

Resume backup parameters(user): the user resumes the backup data (read)

Resume parameter 1(servo 1u level): the user reads the original parameter data matched with the servo drive with the precision 1um

Resume parameter 2(stepper 1u level): the user reads the original parameter data matched with the stepper drive



Resume parameter 3(servo 0.1u level): the user reads the original parameter data matched with the servo drive with the precision 0.1um

Note: When the level is the 2nd grade, the backuping current parameters (user) and resuming backup parameters (user) become backuping current parameters (machine manufacturer) and resuming backup

parameters (machine manufacturer), which is used to backup and resume the manufacturer's data.



10.1.4 Alteration of the Password

Steps for password alteration:

1. After entering the PASSWORD SETTING page, enter the password by the methods in Section 10.1.2;
2. Move the cursor to the "ALTER PASSWORD:" line;
3. Key in the new password, and press  key
4. The CNC system prompts "PLEASE INPUT USER PASSWORD AGAIN", the page is as follows:
5. After reinput the password, press  key, if the two passwords input are identical, CNC prompts "PASSWORD UPDATED". So the password alteration is successful.
6. If the two passwords input are not identical, CNC prompts "PASSWORD CHECKOUT ERROR".


10.1.5 Level degradation


The demotion of the operation level is used to enter a lower level from a higher level, the steps are as follows:

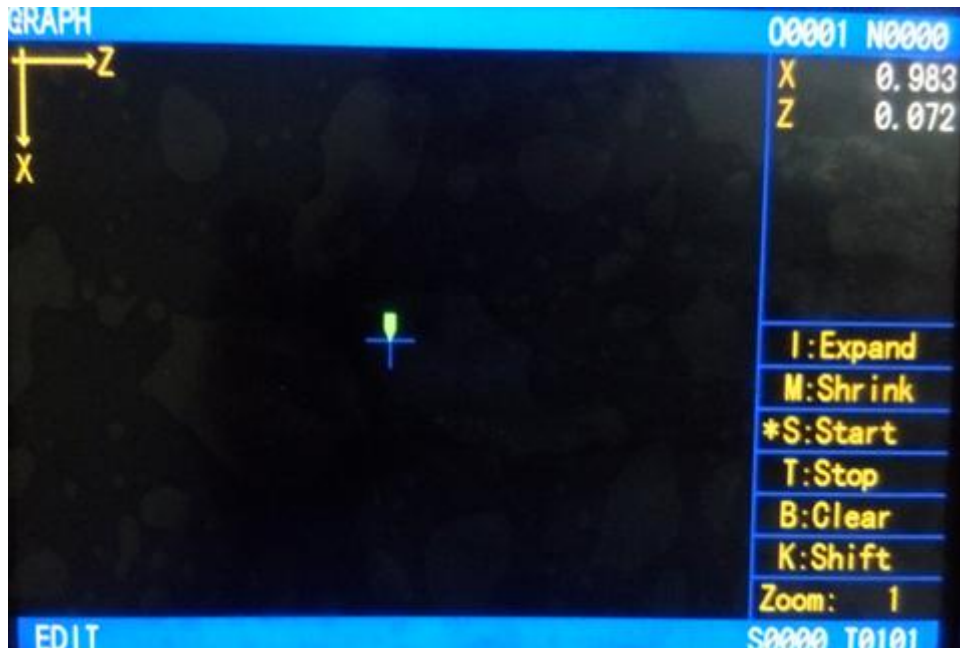
1. After entering the PASSWORD SETTING page, key in the password by the method in Section 10.1.2
2. Move the cursor to the "SET LOWER LEVEL" line, if the current CNC operation is the 3rd level, the page is as follows:
3. Press  key, the CNC prompts "CURRENT LEVEL TO 4, OK?" the page is as follows:
4. Press  key again, if the demotion is successful, the page is as follows:

Note: If the current level is the 5th level, the demotion operation is unallowed.

10.2 Graphic Display

Press  key to enter the path page

Press  to open the next page menu as follows:



1) The START, STOP and CLEAR of the graphic path

In the path display page, press the **S** and the system starts the drawing; press **T**, it stops drawing; press **J**, it clears the current graphic path.

When the path exceeds the display range during automatic graph, the system regulate the graphic center following the tool nose position in real time. Press **()** (or **()**) and the display area is locked, the system does not regulate the graphic center on when the path exceeds the display area. Press **()** (or **()**) and the system releases the lock.

2) Scaling up and down

In the path display page, press **I**, **M**, the system scales up/down the path in real time.

Press **I** each time, the graphic path is scaled up 2 fold, press **M** each time, the graphic path is scaled down 2 fold.

10.3 Parameter Setting

Parameter setting can adjust related characteristics of the drive unit and machine.



Press **参数** **PAR** to enter Parameter interface which includes State parameter, data parameter, pitch compensation


10.3.1 State parameter

1) Byte alteration:

- Turn on the parameter switch;
- Enter the MDI mode;
- Move the cursor to the parameter number to be set:


Method 1: Press **[]** or **[]** to enter the page containing the parameter to be set, press

 or  key to move the cursor to the parameter number to be set;

Method 2: Press  and input the parameter number in the pop-up dialogue box, then press



4) Key in a new parameter value;

5) Press  key, the parameter value is entered and displayed.

6) For security, the PARM SWT needs to be set to OFF after all parameter settings are finished.


10.3.2 Data parameter, pitch compensation

1) Turn on the parameter switch;

2) Enter the MDI mode;

3) Move the cursor to the No. of the parameter to be set;

4) Key in the new parameter value;

5) Press key , the value is entered and displayed;

6) After all parameters setting is finished, the PARM SWT needs to be set to OFF for security.

Explanation: The pitch-pitch parameter can only be altered under the 2 level password authority.








10.4 Restore and backup of data

The user data (such as state parameter, data parameter and screw data) of ECN*000T can be backup (stored) and recovery (read), it doesn't affect the parts program which stored in the CNC.

Data operation page display as below:



1. Turn on the parameter switch;

2. Press  press to input operation mode, press  (press key  or  when it is necessary) enter data operation page;
3. Press  enter data backup page, press  enter data recovery page;
4. Move cursor to the operation options and press  enter data recovery page;
5. Press execute all of the selected operation.

Note 1: Please don't cut off the power when backup or restore data, and it'd better not to do other operation until it finished.

Note 2: Both the 3 level and above password level users can backup and restore




11.45 Notes

The user should pay more attention to the followings during executing the data backup/resume operation:

- 1) Need more than the 2nd grade password(including the 2nd grade) to resume more than (including be equal to) 9000 part programs;
- 2) Backup/resume operation must be in MDI mode;
- 3) When the user executes the data backup operation, the system automatically replaces all files in “U:\ECN*000T_backup\user”, the user should save them as when they cannot be replaced;
- 4) Stop part programs running before data backup/resume;
- 5) They must not stop unless the data backup/resume operation is completed;
- 6) The system is turned on again after the data resume is done.




CHAPTER 11 OPERATION OF U-DISK

11.1 File Management Function Introduction

Press  key enter program page under un-edit state, press  or  key enter [documentation content] interface, insert U disk as following picture:





It displayed CNC disk contents information at left of page and display USB disk contents information at right of page. It only displays “.CNC”, “.NC” and “.txt” under U disk root directory.

CNC disk will switch to USB disk by press  cursor, press  or  can move cursor.

11.2 File copy

Select the file. When the cursor is in the U disk catalog window,

 is pressed and the files in the U disk is copied to the CNC; when the cursor is in the CNC

catalog window,  is pressed and the file in the CNC is copied to the U disk.

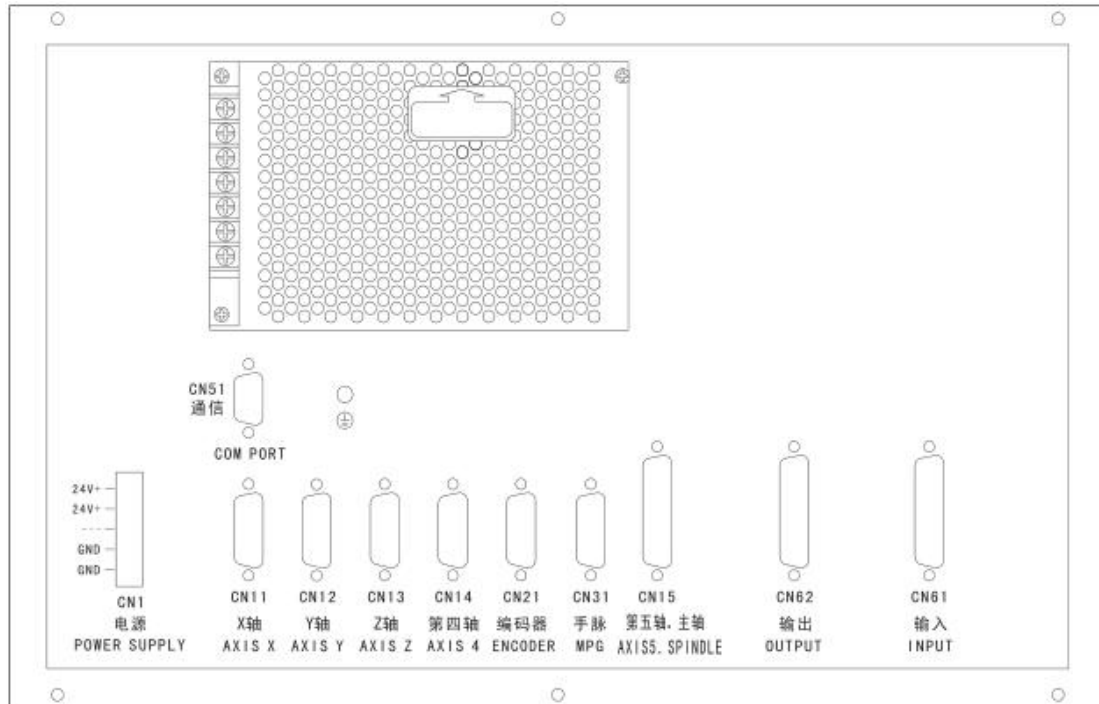
11.3 Data Copy and Resume

In the file management page, backup, resume the CNC data (including the system parameters, part parameter, tool compensation, pitch compensation, ladder parameters, ladder, alarm log and soon) via the U disk.

III Connection

Chapter 1 Installation Layout

1.1.1 ECON200T series Back Cover Interface Layout



1.1.2 Interface explanation

Power box: ECON-PB2, for +5V, +24V, +12V, -12V, GND power supply

Filter(optional): Input terminals for 220V AC power, PE terminal for grounding, output terminals to L, N terminals of ECON-PB2 power box

CN1: power supply interface

CN11: X axis, pin15 D female, connect with X drive unit

CN12: Y axis, pin15 D female, connect with Y drive unit

CN13: Z axis, pin15 D female, connect with Z drive unit

CN14: 4th axis, pin15 D female, connect with 4th drive unit

CN15: spindle, pin 25 D female, connect with spindle drive unit

CN21: encoder, pin15 D male, connect with spindle encoder

CN31: MPG, pin26 D male, connect with MPG

CN51: communication, pin9 D female, connect PC RS232 interface

CN61: input, pin44 D male, connect with machine input

CN62: output, pin44 D female, connect with machine output

1.2 ECN*000T Installation

1.2.1 ECN*000T external dimensions

Please refer the appendix 1 and 2.

1.2.2 Preconditions of the cabinet installation

The dust, cooling liquid and organic resolution should be effectively prevented from entering the cabinet;

The designed distance between the CNC back cover and the cabinet should be not less than 20cm, the inside and outside temperature difference of the cabinet should be not more than 10°C when the cabinet inside temperature rises;

Fans can be fixed in the cabinet to ventilate it;

The panel should be installed in a place where the cooling can't splash;

The external electrical interference should be taken into consideration in cabinet design to prevent it from interfering the CNC system.

1.2.3 Measures against interference

In order to insure the CNC stable working, the anti-interference technology such as space electromagnetic radiation shielding, impact current absorbing, power mixed wave filtering are employed in CNC design. And the following measures are necessary during CNC connection:

1. Make CNC far from the interference devices (inverter, AC contactor, static generator, high-voltage generator and powered sectional devices etc.);
2. To supply the CNC via an isolation transformer, the machine with the CNC system should be grounded, the CNC and drive unit should be connected with independent grounding wires at the grounding point;
3. To inhibit interference: connect parallel RC circuit at both ends of AC winding (Fig. 1-3), RC circuit should approach to inductive loading as close as possible; reversely connect parallel freewheeling diode at both ends of DC winding (Fig. 1-4); connect parallel surge absorber at the ends of AC motor winding (Fig. 1-5);

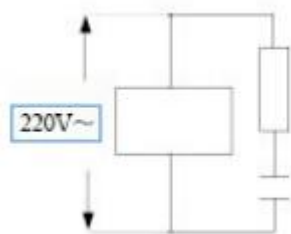


Fig. 1-3

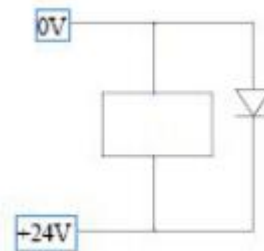


Fig. 1-4

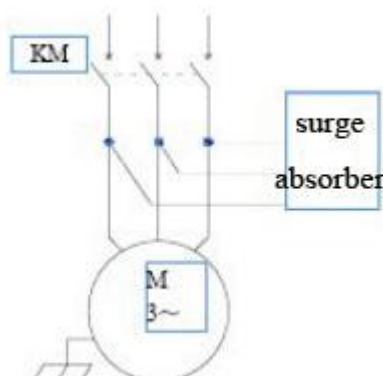


Fig. 1-5

4. To employ with twisted shield cable or shield cable for the leadout cable of CNC, the cable shield tier is grounded by single end at CNC side, signal cable should be as short as possible;
5. In order to decrease the mutual interference between CNC cables or CNC cables with

strong-power cables, the wiring should comply with the following principles:

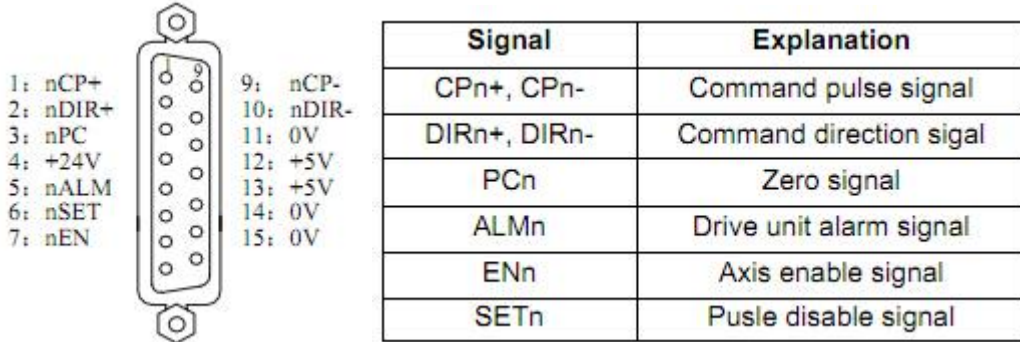
Group	Cable type	Wiring requirement
A	AC power line	Tie up A group cables with a clearance at least 10cm from that of B, C groups, or shield A group cables from electromagnetism
	AC coil	
	AC contactor	
B	DC coil (24VDC)	Tie up B and A group cables separately or shield B group cables; and the further B group cables are from that of C group, the better it is
	DC relay (24VDC)	
	Cables between CNC and strong-power cabinet	
	Cables between CNC and machine	
C	Cables between CNC and servo drive unit	Tie up C and A group cables separately, or shield C group cables; and the cable distance between C group and B group is at least 10cm with twisted pair cable applied.
	Position feedback cable	
	Position encoder cable	
	MPG cable	
	Other cables for shield	

CHAPTER 2 DEFINITION & CONNECTION OF

INTERFACE SIGNALS

2.1 Connection to Drive Unit

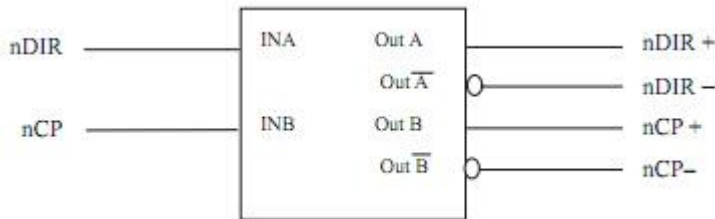
2.1.1 Drive Interface Definition



Picture 2-1 CN11, CN12, CN13, CN14 (15 core D hole socket)

2.1.2 Command Pulse and Direction Signals

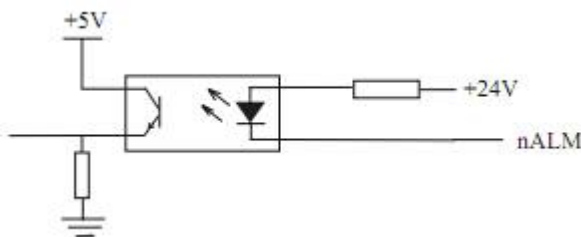
nCP+, nCP- are command pulse signals, nDIR+, nDIR- are command direction signals. These two group signals are both difference output (AM26LS31), the interior circuit for them is shown in Picture 2-2.



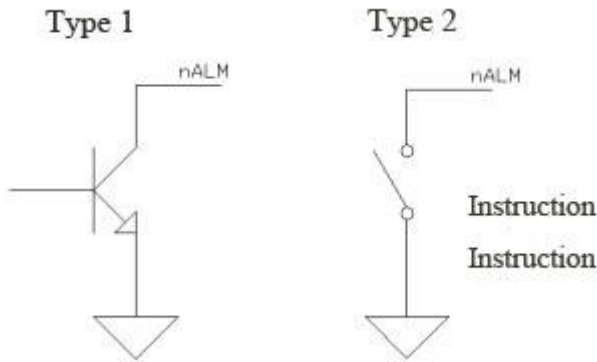
Picture 2-2 Interior circuit of command pulse and direction signals

2.1.3 Drive Unit Alarm Signal

The low or high level of the drive unit alarm level is set by the CNC bit parameter No.009 BIT0~BIT4, whose interior circuit is shown in Picture 2-3:



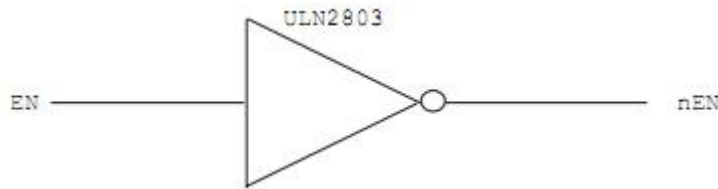
This input circuit requires that the drive unit transmits signal by the following types in Picture 2-4:



Picture 2-4 Signal types of drive unit

2.1.4 Axis Enable Signal ENn

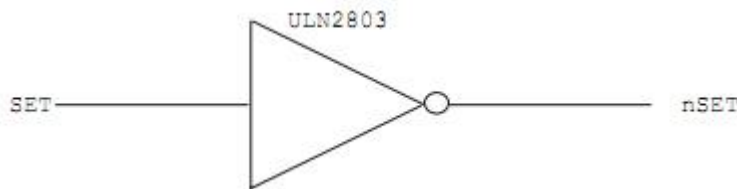
nEN signal output is valid as CNC works normally (nEN signal to 0V); when the drive unit alarm or emergency alarm occurs, CNC cuts off nEN signal output (nEN signal to 0V off). The interior interface circuit is shown in Picture 2-5:



Picture 2-5 interior interface circuit for axis enable signal

2.1.5 Pulse Disable Signal SET

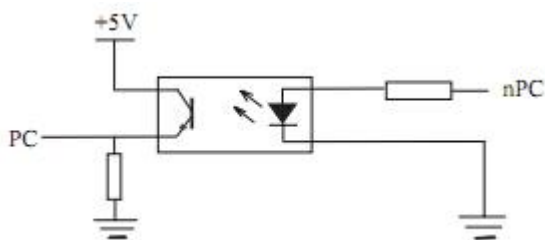
nSET signal is used to control servo input disable which can enhance the anti-disturbance capability between CNC and drive unit. This signal is at low level if there is pulse output from CNC, high resistance if not. The interior interface circuit of it is shown in Picture 2-6:



Picture 2-6 Interior interface circuit for pulse disable signal

2.1.6 Zero Signal nPC

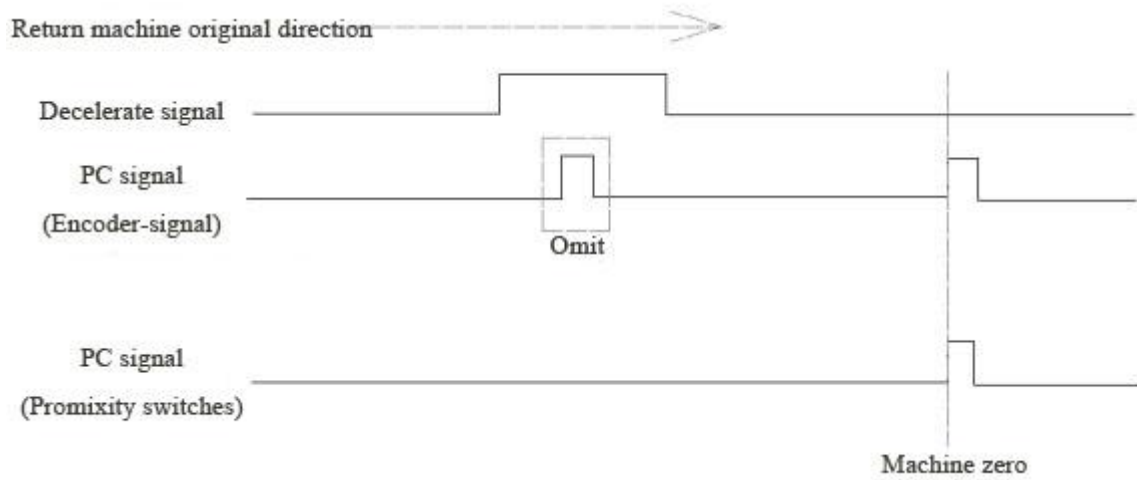
The one-rotation or approach switch signal is taken as zero signals for machine zero return. Its interior connection circuit is shown in Picture2-7.



Picture 2-7 Zero signal circuit

Note: nPC signal uses +24V level.

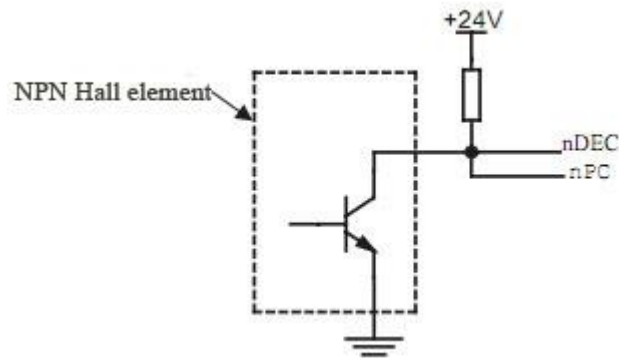
a) The wave shape that supplied by users should be like aspicture 2-8:



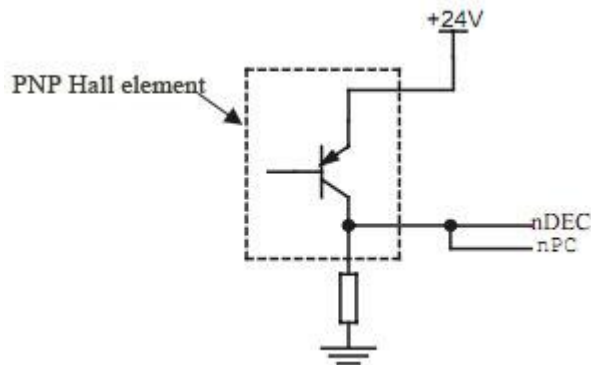
Picture 2-8 PC signal wave shape figure

Note: when machine tool backs to zero, CNC judges reference point though detect PC signal's jump after decelerate switch triped out, both rise edge and falling edge are valid.

b) The connection for NPN Hall elements taken as both deceleration signal and zero signals is shown as below:



c) The connection for PNP Hall elements taken as both deceleration signal and zero signals is shown as below:

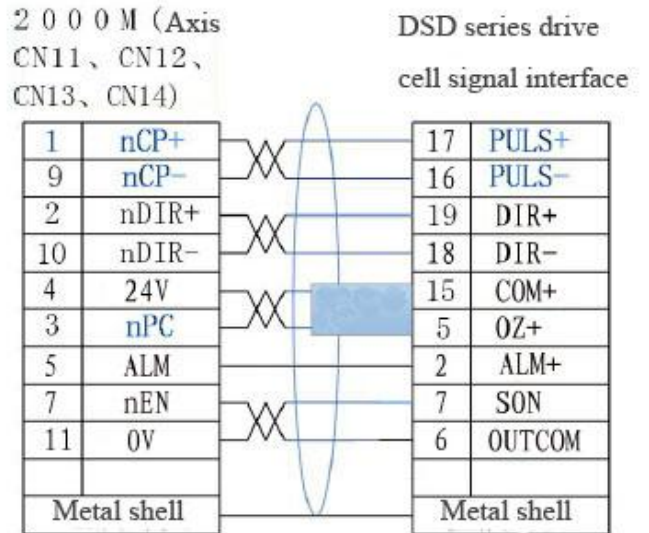
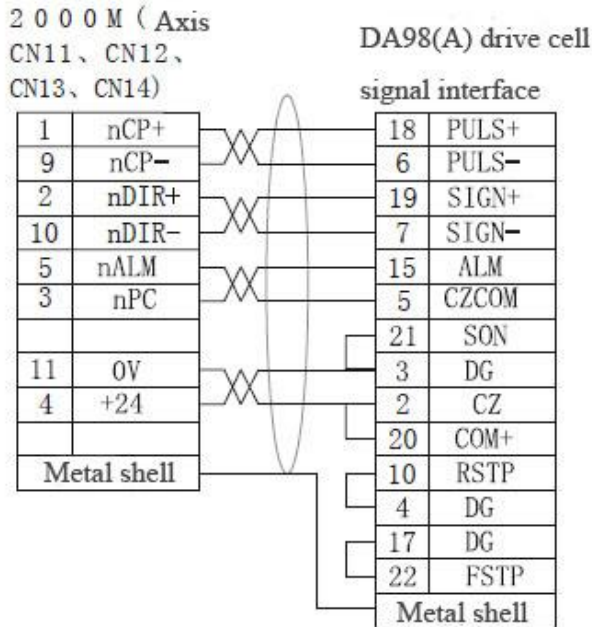


2.1.7 Connection to Drive Unit

The connection of ECN*000T to drive unit is shown in picture 2-10:

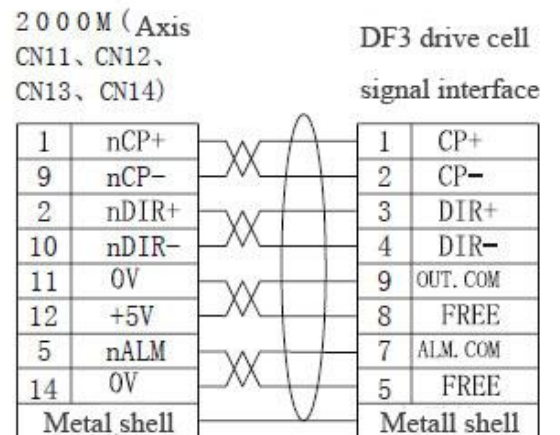
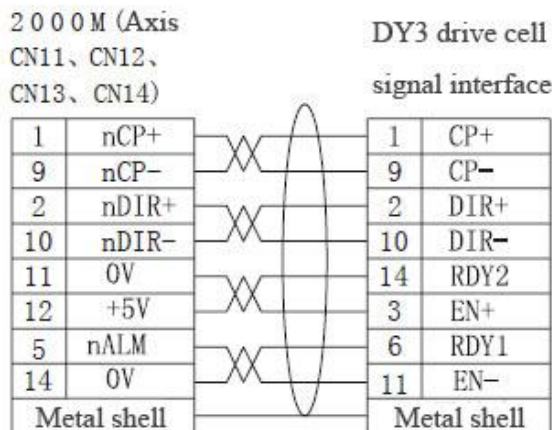
The connection of ECN*000T and DA98(A) drive cell

The connection of ECN*000T and DSD drive cell



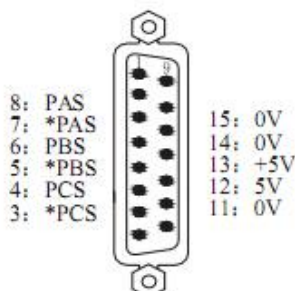
The connection of ECN*000T and DY3 (A) drive cell

The connection of ECN*000T and DF3 drive cell



2.2 Connection of spindle axis encoder

2.2.1 Spindle axis encoder interface definition

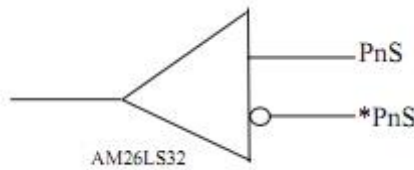


Name	Explanation
*PAS/PAS	Encoder A phase pulse
*PBS/PBS	Encoder B phase pulse

*PCS/PCS	Encoder C phase pulse
----------	-----------------------

2.2.2 Signal explanation

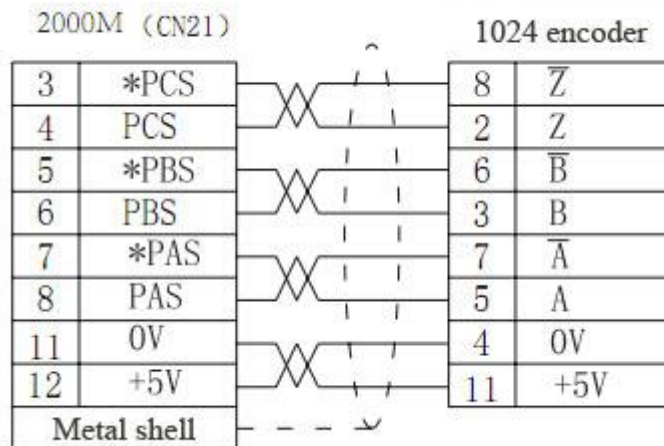
*PCS/PCS, *PBS/PBS, *PAS/PAS are differential input signal of C phase, B phase and A phase, adopts 26LS32 receives; *PAS/PAS, *PBS/PBS are orthogonality wave which differs 90, max sign frequency<1MHz; used encoder wire numbers are setting by parameter (range 100~5000) Internal connection circuit as picture 2-13 (n=A, B, C in picture)



Picture 2-13 Encoder signal circuit

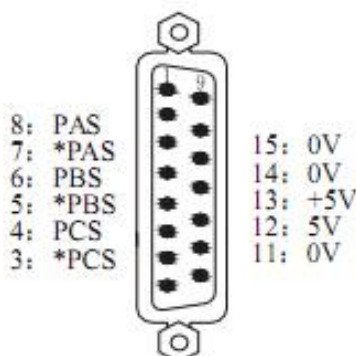
2.2.3 Connection of spindle encoder port

The connection of ECN*000T and spindle encoder as picture 2-14, it adopts twisted-pair. (Take an example of ZLF-12-102.4BM-C05D)



2.3 Connection to Handwheel

2.3.1 Handwheel Interface Definition



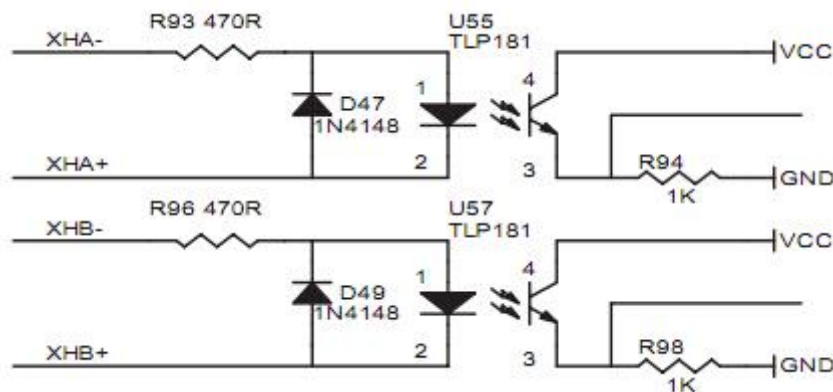
Signal	Explanation
HA+, HA-	Handwheel A phase signal
HB+, HB-	Handwheel B phase

	signal
X5.0	MPG X axis selection
X5.1	MPG Y axis selection
X5.2	MPG Z axis selection
X5.3	The 4 th axis selection
X5.4	The 5 th axis selection
X5.5	Increment *1
X5.6	Increment *10
X5.7	Increment *100
+24V	DC power supply
VCC, GND	

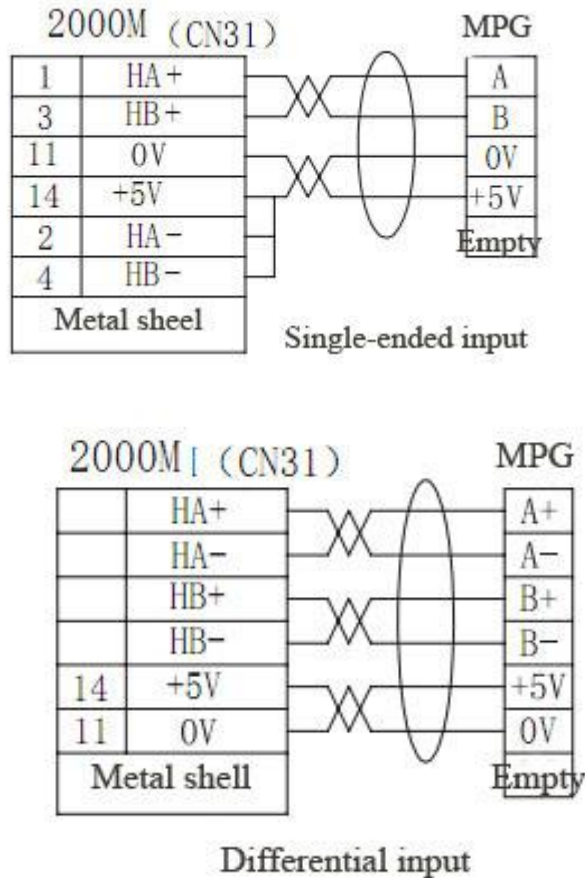
Picture 2-15 CN31 encoder port (26 cores DB type socket)

2.3.2 Signal explanation

HA, HB are A phase and B phase input signals respectively. Internal connection circuit as picture 2-16 shows:



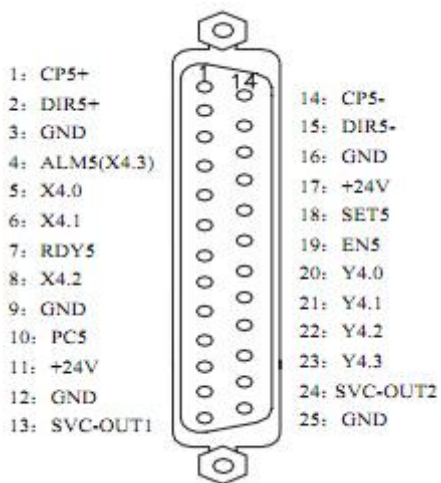
The connection of ECN*000T and MPG as picture 2-17 shows:



Picture 2-17 the connection of ECN*000T and MPG

2.4 Connection to Spindle Encoder

2.4.1 Spindle Encoder Interface Definition



CP5+, CP5-	Spindle pulse signal
DIR5+, DIR5-	Spindle direction signal
ALM5	Spindle alarm signal
RDY5	Spindle already signal
PC5	Spindle zero signal
SVC-OUT1	Analog voltage output 1
SVC-OUT2	Analog voltage output 2
SET5	Spindle setting signal
EN5	Spindle enable signal
X4.0~X4.3	PLC address, only the low

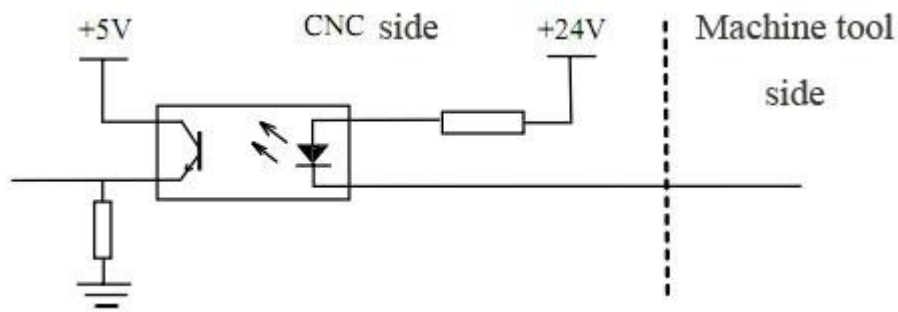
	level is valid
Y4.0~4.3	PLC address

Picture 2-18 CN15 spindle port (25 core DB hole)

Note 1: PC5 and 0V break over is valid, it is different from other feeding axis (breakover of CN11~CN14 axis port's PC and +24 is valid)

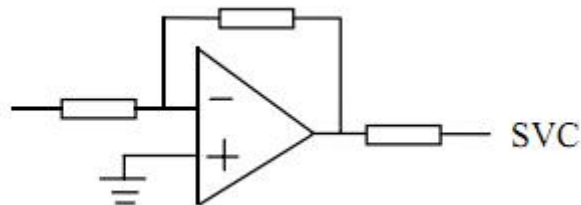
Note 2: X4.0 ~X4.3 is breakover with 0V and is different from other input signal (the breakover of other input signal is valid)

Note 3: The signal internal circuit of PC5, X4.0 ~ X4.3 as below:



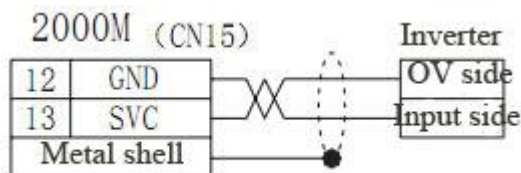
2.4.2 Connection of general inverter

Analog spindle port SVC port can output 0 ~ 10V voltage, internal circuit of signal as picture 2-20:



Picture 2-20 SVC signal circuit

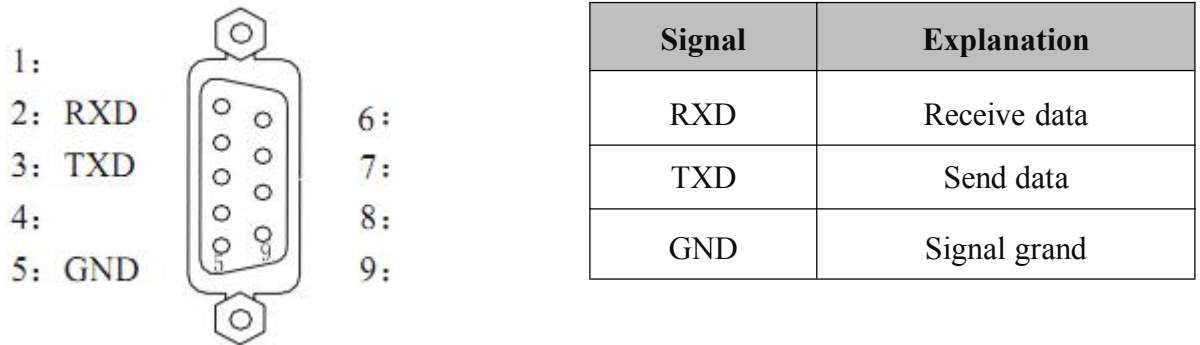
The connection of ECN*000T and inverter as below picture 2-21:



Picture 2-21 the connection of ECN*000T and inverter

2.5 Connection of ECN*000T to PC serial port

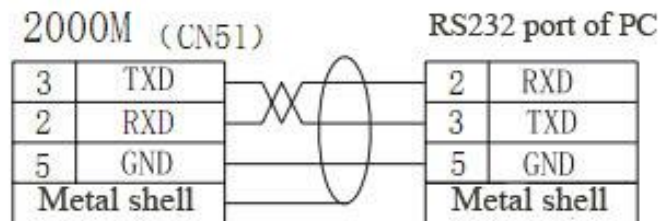
2.5.1 Communication Interface Definition



Picture 2-22 CN51 communication port (9 holes)

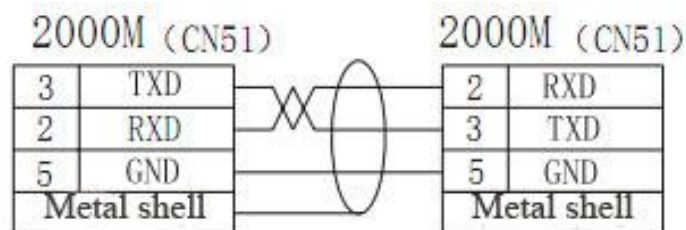
2.5.2 Communication Interface Connection

The communication between ECN*000T and PC can be done via RS232 interface (ECN*000T communication software needed);the connection of them is shown in picture 2-25:



Picture 2-25 connection of ECN*000T to PC

The communication of a ECN*000T to another ECN*000T can be made via their CN51 interfaces, and the connection of them is shown in picture 2-26:

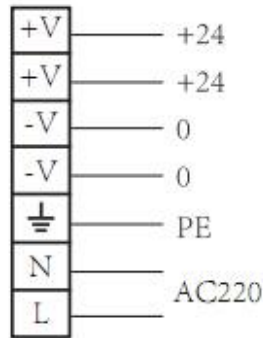


Picture 2-26 connection of ECN*000T

2.6 Connection of Power Interface

Power box has already connected with CN1 port before delivery; users only need to connect an

external 220V AC power.

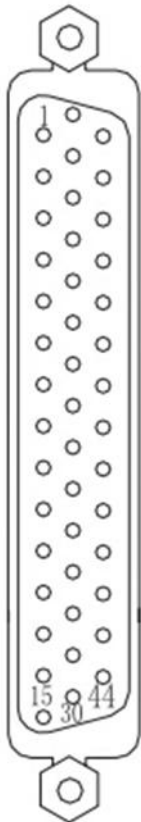


Picture 2-24 system power port CN1

2.6 I/O Interface Definitions

ECN*000T milling machine tool CNC doesn't mark the fixed address I/O function meaning defined by PLC program (ladder figure), when ECN*000T CNC assembles machine tool, I/O function decided by machine tool supplier, please refer user manual of machine tool supplier.

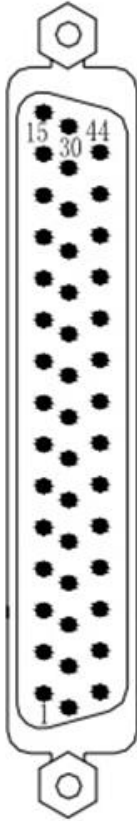
This section. doesn't mark fixed address I/O function is described by ECN*000T standard PLC program.



Subscript	Address	Function	Explanation
21 ~ 24	0V	Power interface	Power 0V side
18 ~ 20 25 ~ 28	empty	empty	empty
1	X0.0	SAGT	Protective door detection signal
2	X0.1	SP	External pause
3	X0.2	THAN	External tool clamp/unclamp
4	X0.3	DECX	X axis decelerate signal
5	X0.4		Reserve
6	X0.5	KYP	Emergency stop button
7	X0.6	LIMU	Overtravel release signal
8	X0.7	PRKY	Air pressure detect input signal

9	X1.0	TOPE	Spindle tool unclamp on position signal
10	X1.1	TCL0	Spindle tool clamp on position signal
11	X1.2	TZER	Tool changer homing key
12	X1.3	DECZ	Z axis decelerate signal
13	X1.4	ST	External cycle start signal
14	X1.5	M41I	Spindle the 1 st gear on position signal
15	X1.6	M42I	Spindle the 2 nd gear on position signal
16			
29	X2.0	TFRX	Tool changer forward on position signal
30	X2.1	TBAX	Tool changer backward on position signal
31	X2.2	TCUX	Tool changer tool counter signal
32	X2.3	DECY	Y axis decelerate signal
33	X2.4	DEC4	The 4 th axis decelerate signal
34	X2.5	TZEX	Tool changer zero position signal
35	X2.6	TRSW	Current tool detect signal
36	X2.7	TMSW	Spindle tool detect switch signal
37	X3.0	LMIX	X axis overtravel input
38	X3.1	LMIY	Y axis overtravel input
39	X3.2	LMIZ	Z axis overtravel input
40	X3.3	TCW	Tool changer reversal button signal
41	X3.4	TCCW	Tool changer forward button signal
42	X3.5	SKIP	C31 skip signal
43	X3.6	TFRX	Tool changer forward button signal
44	X3.7	TBAX	Tool changer backward button signal

Subscript	Address	Function	Explanation
17 ~ 19 26 ~ 28	0V	Power interface	Power 0V side
20 ~ 25	+24V	Power interface	Power 24V side



1	Y0.0	COOL	Coolant output
2	Y 0.1	M32	Lubrication output
3	Y 0.2	TCLA	tool unclamp/clamp
4	Y 0.3	M03	Spindle anticlockwise rotation (forward)
5	Y 0.4	M04	Spindle clockwise rotation (reversal)
6	Y 0.5	M05	Spindle stop
7	Y 0.6	SCLP	Spindle clamp
8	Y 0.7	SPZD	Spindle brake
9	Y 1.0	S1/M41	Spindle mechanical gear output 1
10	Y 1.1	S2/M42	Spindle mechanical gear output 2
11	Y 1.2	S3/M43	Spindle mechanical gear output 3
12	Y 1.3	S4/M44	Spindle mechanical gear output 4
13	Y 1.4		Reserve
14	Y 1.5		Reserve
15	Y 1.6		Reserve
16	Y 1.7		Reserve
29	Y 2.0	TLP	Spindle tool unclamp indicator
30	Y 2.1		Reserve
31	Y 2.2	CLPY	Three - color light – yellow light
32	Y 2.3	CLPG	Three – color light – green light
33	Y 2.4	CLPR	Three – color light – red light
34	Y 2.5		Reserve
35	Y 2.6		Reserve
36	Y 2.7		Reserve
37	Y 3.0	STAO	Spindle orientation
38	Y 3.1	TCCY	Tool changer CW
39	Y 3.2	TCWY	Tool changer CCW
40	Y 3.3	TFRY	Tool changer forward
41	Y 3.4	TBAY	Tool changer backward

42	Y 3.5	TBAL	Tool changer backward indicator light
43	Y 3.6		Reserve
44	Y 3.7		Reserve

Note 1: some input, output ports can definit multiply functions, it is “/” in the above form;

Note 2: When output is active, the output port is 0V. If output function is inactive, the output signal is cut off by high impedance;

Note 3: If input signal and+24V breakover, the input is active. When the input signal disconnect to +24V,input is inactive.

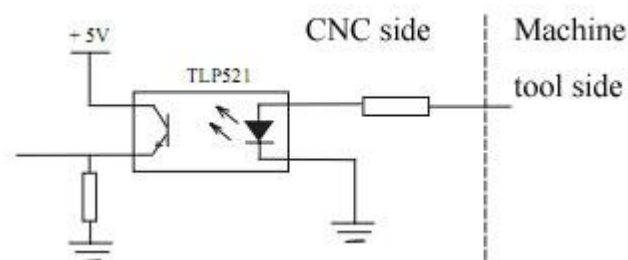
Note 4: The +24V, COM are same as ECN*000T power supply’s terminals that have the same name;

2.6.1 Input signal

Input signal means the signal from machine to CNC, when this signal connect with +24V, the input is valid; when it disconnect with +24V, the input is invalid. The contact point of input signal at machine side should meet the following conditions:

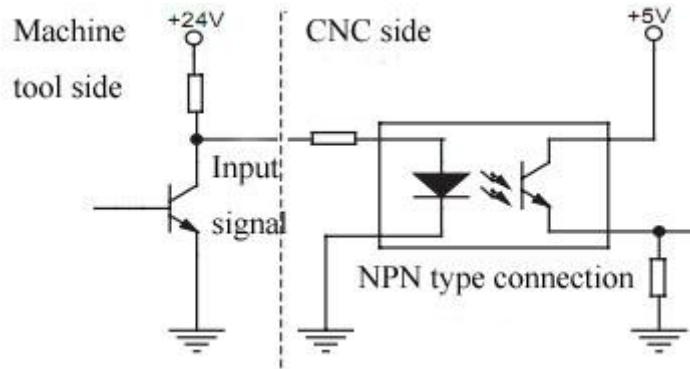
The capacity of the contact point: DC30V, 16mA above Leakage current between contact points in open circuit: 1mA below Voltage drop between contact points in closed circuit: 2V below (current 8.5mA, including cable voltage drop)

There are two external input types for input signals: one type is input by contactor whose signals are from key, stroke switch and contacts of relay at machine side, as is shown in picture 2-27:

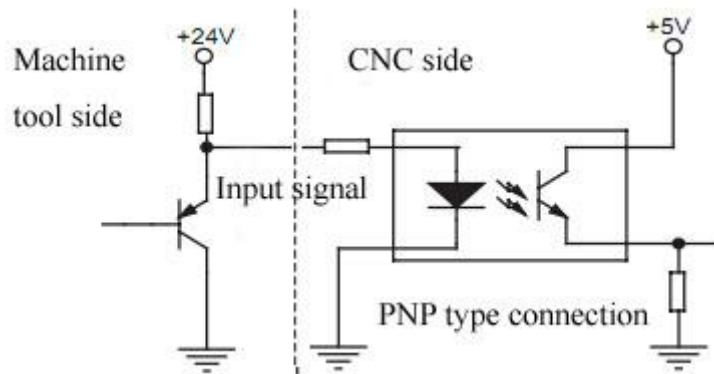


Picture 2-27

The other type is input by switch with no contactor (transistor), as is shown in Fig. 2-28A, 2-28B.



Picture 2-28A NPN type connection

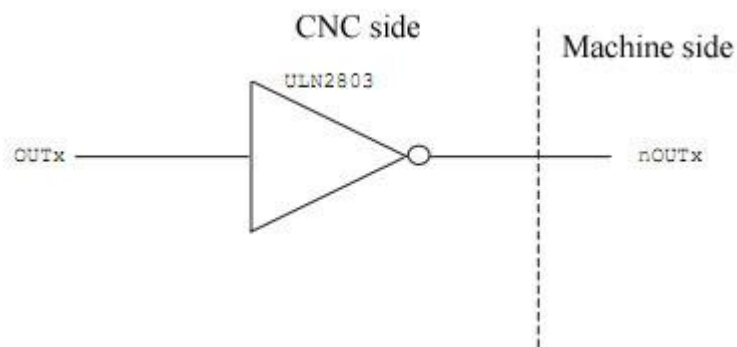


Picture 2-28B PNP type connection

2.6.2 Output signal

The output signal is used for the machine relay and indicator, if it is on with 0V, the output function is valid; if it is off with 0V, the output function is invalid. There are total 36 digital volume outputs in

I/O interface that they all have the same structure as is shown in Fig.2-29:



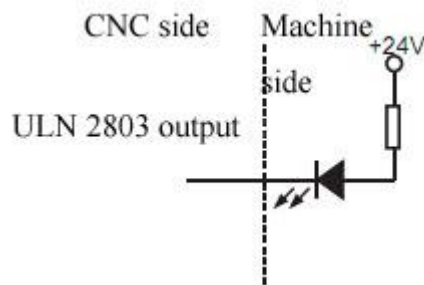
Picture 2-29 Circuit for digital volume output module

The logic signal OUTx output from the main board is sent to the input terminal of

inverter (ULN2803) via a connector. And there are 2 output types for nOUTx: output with 0V, or high impedance. Its typical application is shown in follows:

- **To drive OLED**

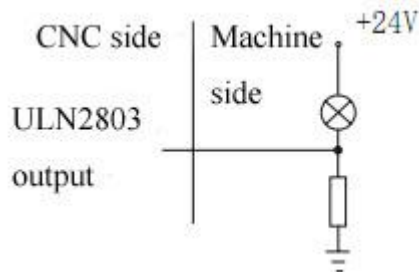
A serial resistance is needed to limit the current (usually 10mA) that goes through the OLED by using ULN2803 output to drive LED, which is shown in picture 2-30.



Picture 2-30

- **To drive filament indicator**

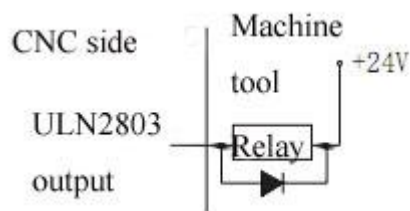
An external preheat resistance is needed to decrease the current impact at power on by using ULN2803 output to drive filament indicator, and this resistance value should be within a range that the indicator can't light up. It is shown in picture 2-31:



Picture 2-31

- **To drive inductive load (relay etc.)**

To use ULN2803 output to drive an inductive load, it requires connecting a freewheeling diode near the coil to protect output circuit and deduce interference. It is shown in picture 2-32:



Picture 2-32

The meanings of output signal in I/O port defined by PLC program, output signals which defined

by standard PLC program includes S1 ~ S4 (M41 ~ M44), M3 ~ M5, M8 and M32 etc signals.

2.7 I/O function and connection

2.7.1 Travel limit and scram

Relative signal

KYP: scram signal, stop alarming when disconnect with +24V;

LMIX: X axis travel limit detect input

LMIY: Y axis travel limit detect input

LMIZ: Z axis travel limit detect input

Data diagnose

0	0	0		ESP						
Interface pin				CN61.6						

Signal diagnose

Signal	KYP	LMIX	LMIY	LMIZ
Address diagnose	X0.5	X3.0	X3.1	X3.2
interface pin	CN61.6	CN61.38	CN61.38	CN61.39

Control parameter

State parameter

0	2	1							MESP		
----------	----------	----------	--	--	--	--	--	--	-------------	--	--

KYP=0: detect KYP scram signal

=1: don't detect KYP scram signal.

PLC state parameter

K	1	0		LIMIT	LMIS					
----------	----------	----------	--	--------------	-------------	--	--	--	--	--

LMIT=1: each axis travel limit detection function is valid.

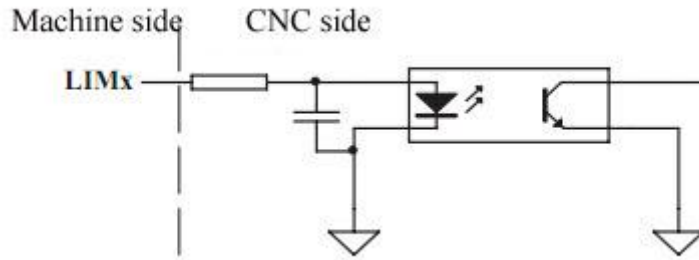
=0: each axis travel limit detection function is invalid.

LMI=1: overrun alarm when travel limit detect signal and +24V switch off.

=0: overrun alarm when limit detect signal and +24 break over.

Signal connection

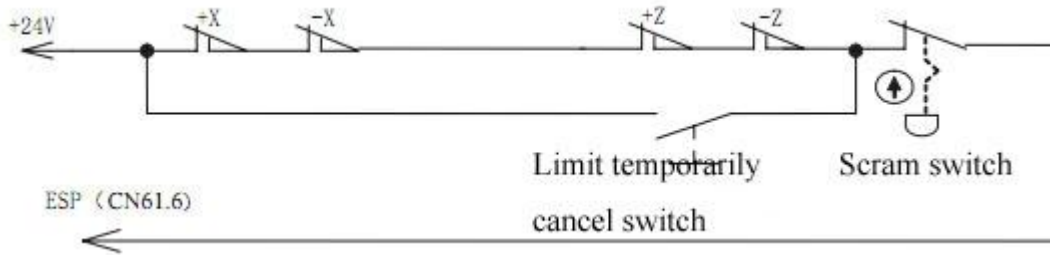
Overrun limit and scram signal (KYP) circuit as below picture 2-33:



Picture 2-33

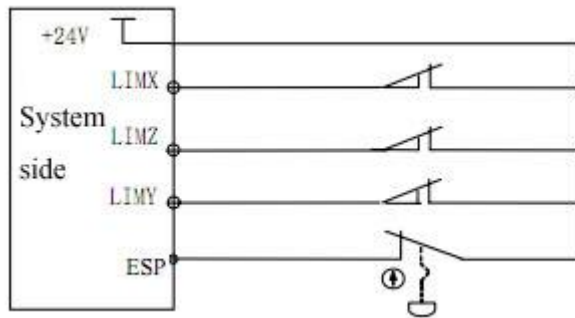
Machine external connection

① travel limit and scram connect in series, connection method as below picture2-34A:



Picture 2-34A scram, route switch connect in series

② travel limit and scram connect independent, connection method as below picture 2-34B:



Picture 2-34B scram, route switch connection in dependent

Control logic

① Travel limit and scram connect in series

CNC will appear “scram” alarm when appears overrun or press scram key, press overrun remove key if it is overrun, press reset key cancel alarm and reversal move can relies the overrun. CNC stop pulse output when it appears scram alarm. Except above CNC processing functions, PLC program can also definit other functions when scram alarm. Standard PLC program definits functions: close M03 or M04, M08 signal output when scram alarm, and

output M05 signal.

② Travel limit and scram connect independent

1. There is only one overrun touch point on each axis, it judges positive or negative overrun alarm through axis move direction.
2. It can be moved reversal when occur overrun alarm, move out of limit position and press reset to clear alarm.

Note: it must be sure that machine tool plunker between positive and negative route before start overrun limit function, or the reminder alarm will be not unconformity with reality.

2.7.2 Machine zero

Relative signal

DECX: X axis deceleration signal;

DECY: Y axis deceleration signal;

DECZ: Z axis deceleration signal;

DEC4: The 4th axis deceleration signal;

DEC5: The 5th axis deceleration signal;

Data diagnose

0	0	0					DEC5	DEC4	DECZ	DECY	DECX
Interface pins							CN61.34	CN61.33	CN61.12	CN61.32	CN61.4

Control parameter

K	2	2		***	***	***	DEC5T	DEC4T	DECZ	DBCY	DECX
---	---	---	--	-----	-----	-----	-------	-------	------	------	------

DEC5T=0: 5th axis decelerates signal low level;

=1: 5th axis decelerates signal high level;

DEC4T=0: 4th axis decelerates signal low level;

=1: 4th axis decelerates signal high level;

DECZ=0: Z axis decelerates signal low level;

=1: Z axis decelerates signal high level;

DECY=0: Y axis decelerates signal low level;

=1: Y axis decelerates signal high level;

DECX=0: X axis decelerates signal low level;

=1: X axis decelerates signal high level;

0	0	6						ZPLS			ZMOD
---	---	---	--	--	--	--	--	------	--	--	------

ZMOD=1: zero mode choose front link stopper;

=0: zero modes choose after link stopper;

ZPLS=1: zero mode choice, there is a switch signal;

=0: zero mode choice, there is no switch signal;

0	1	2									ISOT
---	---	---	--	--	--	--	--	--	--	--	------

ISOT=1: at the time that after power on and after return machine tool zero point, manual rapidly move is valid;

=0: at the time that after power on and after return machine tool zero point, manual rapidly move is not valid;

0	2	6					MZR5	MZR4	MZRY	MZRZ	MZRZ	MZRZ
---	---	---	--	--	--	--	------	------	------	------	------	------

MZR_x=1: choose the zero returning direction is negative;

=0: choose the zero returning direction is positive;

Data parameter

0	8	0									ZRNFL
---	---	---	--	--	--	--	--	--	--	--	-------

ZRNFL: low speed rate of machine zero turning

0	7	0									ZRNFHX
---	---	---	--	--	--	--	--	--	--	--	--------

ZRNFHX: high speed machine zero turning of X axis

0	7	1									ZRNFHY
---	---	---	--	--	--	--	--	--	--	--	--------

ZRNFHY: high speed machine zero turning of Y axis.

0	7	2									ZRNFHZ
---	---	---	--	--	--	--	--	--	--	--	--------

ZRNFHZ: high speed machine zero turning of Z axis

0	7	3									ZRNFH4
---	---	---	--	--	--	--	--	--	--	--	--------

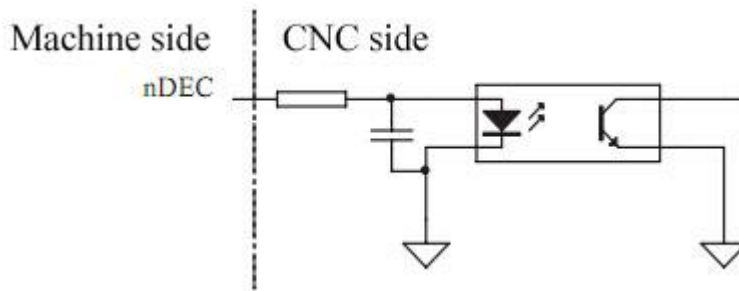
ZRNFH4: high speed machine zero turning of the 4th axis

0	7	4									ZRNFH5
---	---	---	--	--	--	--	--	--	--	--	--------

ZRNFH5: high speed machine zero turning of the 5th axis

Signal connection

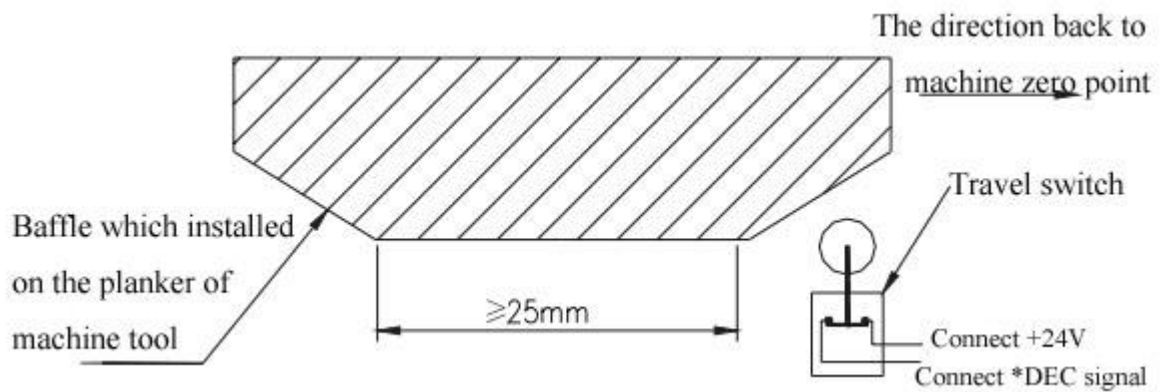
Deceleration signal internal connection circuit as below picture 2-38:



Picture 2-38

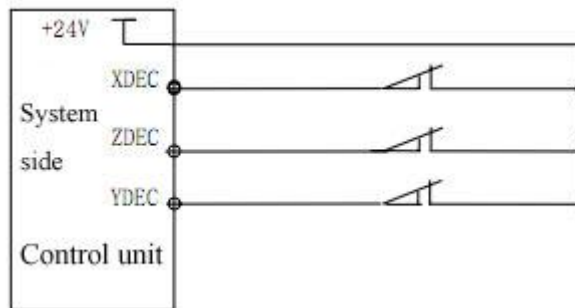
Machine zero return by regarding servo motor one-rotation signal as zero signals

① Its sketch map is shown as follows:



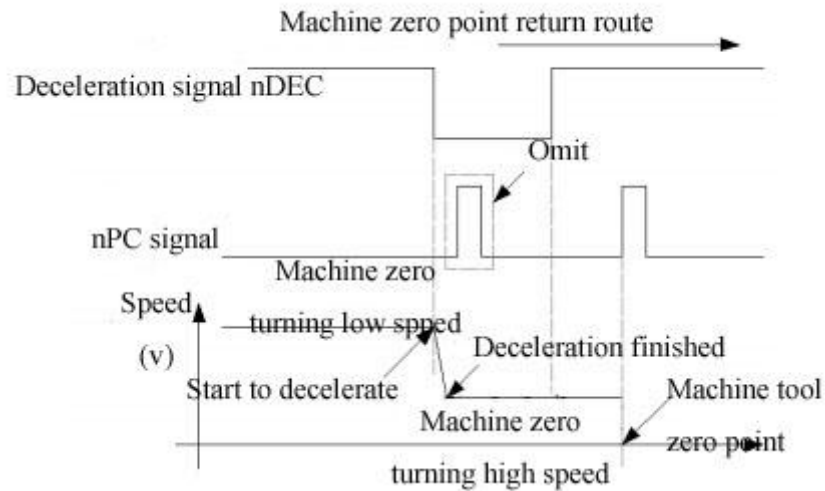
Picture 2-39

② The circuit of deceleration signal:



Picture 2-40

③ Action time sequence of machine zero return



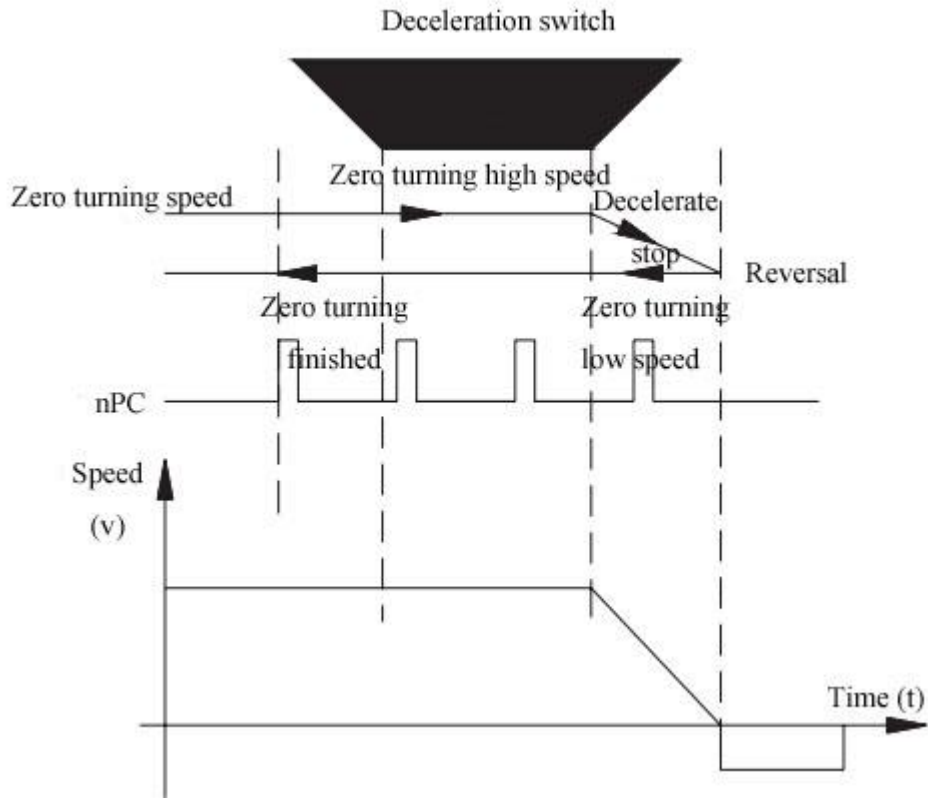
Picture 2-41-a

④ Machine zero return process

A: Select machine zero return mode, press the manual positive or negative feed key(machine zero return direction is set by bit parameter No.026), the corresponding axis moves to the machine zero by a rapid speed (parameter No. 70 ~ 74). As the axis press down the deceleration switch to cut off deceleration signal, the feed slows down immediately, and it continues to run in a fixed low speed (parameter No. 80).

B: When the deceleration switch is released, the deceleration signal contact point is closed again. And CNC begins to detect the encoder one-rotation signal, if the signal level changes, the motion will be stop. And the corresponding zero indicator on the operator panel lights up for machine zero return completion.

When the BIT1 (ZMOD) of state parameter No.006 sets to be 1, BIT4 of K022 to be 0, choose link stopper current mode to return machine zero point, low speed signal low level is valid. By this time, zero sequence of machine zero turning as following picture:



Picture 2-41-b

A: Select machine zero return mode, press the manual positive or negative feed key(machine zero return direction is set by bit parameter No.026), the corresponding axis moves to the machine zero by a rapid speed (parameter No. 70 ~ 74). As the axis press down the deceleration switch to cut off deceleration signal, the feed slows down immediately, and it continues to run in a fixed low speed .

B: When the deceleration switch is released, the deceleration signal contact point is closed again. And CNC begins to detect the encoder one-rotation signal, if the signal level changes, the motion will be stop. And the corresponding zero indicator on the operator panel lights up for machine zero return completion.

2.7.3 Spindle control

Relative signal (standard PLC program definition):

Signal type	Symbol	Signal port	Address	Function explanation	Remark
Input signal	SAR	CN15.6	X4.1	Spindle axis reaches signal	The signal 0V input is valid
	SALM	CN15.4	X4.3	Spindle axis abnormal alarm input	

Output signal	M03	CN62.4	Y0.3	Spindle axis anticlockwise rotation (forward)	
	M04	CN62.5	Y0.4	Spindle axis clockwise rotation (reversal)	
	M05	CN62.6	Y0.5	Spindle stop	
	SCLP	CN62.7	Y0.6	Spindle clamp	
	SPZD	CN62.8	Y0.7	Spindle brake	
	SVF	CN62.37	Y3.0	Spindle servo break up	
Instruction format	M03			Spindle axis anticlockwise rotation (forward)	
	M04			Spindle axis clockwise rotation (reversal)	
	M05			Spindle stop	
	M20			Spindle clamp	It is valid when analog spindle
	M21			Spindle brake	

Control parameter

State parameter

K	1	0								RSJG	
---	---	---	--	--	--	--	--	--	--	------	--

RSJG=1: press , cnc wouldn't close M03, M04, M08 and M32 output signal;=0: press , cnc close M03, M04, M08 and M32 output signal.

0	2	0			SAR						
---	---	---	--	--	-----	--	--	--	--	--	--

Bit6 1: Detect spindle SAR signal before cutting;

0: Doesn't detect spindle SAR signal before cutting.

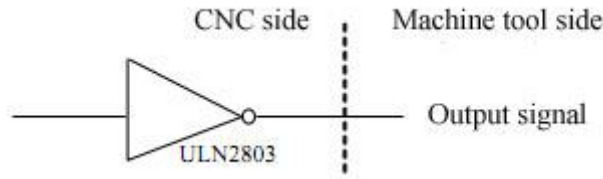
Data parameter

D99

Spindle zero speed output range (r/min)

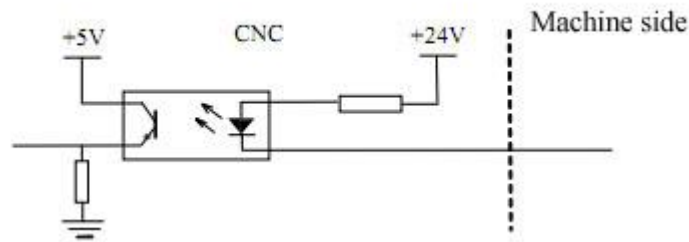
Signal connection

M03, M04, M05, SCLP, SPZD, SVF signal output circuit as below picture 2-45A shows:



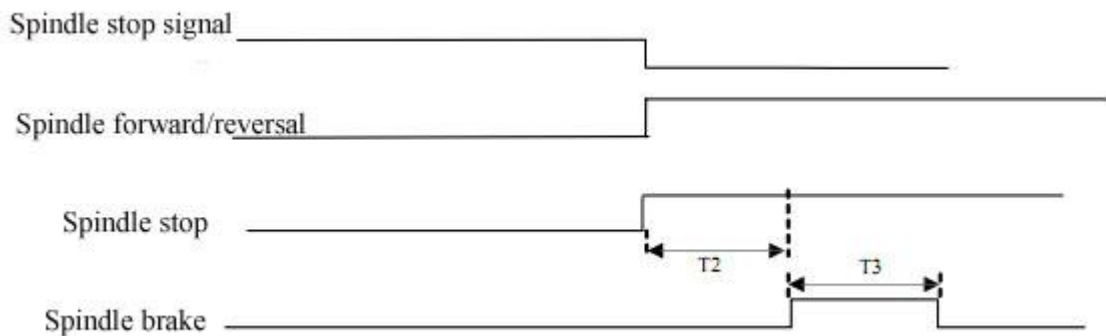
Picture 2-45A

SAR, SALM signal input circuit as below picture 2-45B shows:



Picture 2-45B

Action sequence (standard PLC program definition)



Picture 2-46

Note: T2 is the delay time from set out spindle stop signal to spindle brake signal, T3 is spindle brake hold time.

Function explanation (standard PLC program definition)

- ① M05 output is valid when CNC power on. When M05 output is valid, execute M03 or M04, M03 or M04 output is valid and continuous, and close M05 at the same time; M03 or M04 output is valid, execute M05, close M03 or M04 output, M05 output is valid and go on;
- ② M03 (M04) output is valid, execute M04 (M03) will alarm.

Note 1: when CNC scram, close M03 or M04, M08 signal output, and output M05 signal at the same time;

Note 2: when CNC reset, Bit 1 of K parameter NO.010 set if cancel the output of M03, M04:

When Bit 1 = 0, cnc reset and close the output of M03, M04;

When Bit 1 =1, cnc reset M03, M04's output state keep the same.

2.7.4 Spindle rotate speed switching value control

Relative signal (standard PLC program definition)

S01 ~ S04: Spindle rotate speed switch value control signal, S01 ~ S02 signal port which defined by standard PLC program is multiplex interface, S01 ~ S04 and M41 ~M44 share the port.

Signal diagnose

Signal	S4	S3	S2	S1
Diagnose address	Y1.3	Y1.2	Y1.1	Y1.0
Interface pins	CN62.12	CN62.11	CN62.10	CN62.09

Control parameter

State parameter

0	0	1					Analog spindle				
---	---	---	--	--	--	--	----------------	--	--	--	--

Bit 4 = 1: Spindle rotates speed analog voltage control;

= 0: Spindle rotates speed switch value control.

Control logic (standard PLC program definition)

S1 ~ S4 output is invalid when CNC power on. Execute one code of S01, S02, S03 and S04, corresponding S signal output is valid and hold on, and cancel the output of other S signal. Cancel output of S1 ~ S4 when execute S00 code, there is only one output of S1 ~ S4 is valid at the same time.

2.7.5 Spindle auto shift gears control

Relative signal (standard PLC program definition)

M41 ~ M44: spindle axis auto shift output signal, it can support 4 gears spindle shift automatically when choose spindle analog value control (0~10V analog voltage output)

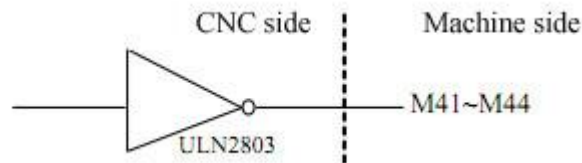
M41I, M42I: spindle axis auto shift the 1st and 2nd gears to position signal, it can support 2 gears shift position detection function,

Signal diagnose

Signal	M42I	M41I	M44	M43	M42	M41
Diagnose	X1.6	X1.5	X1.3	Y1.2	Y1.1	Y1.0
Interface pin	CN61.15	CN61.14	CN62.12	CN62.11	CN62.10	CN62.09

Signal connection

M41 ~ M44 circuit as below picture 2-47 shows:



Picture 2-47

Control parameter

State parameter

0	0	1					Analog spindle				
---	---	---	--	--	--	--	----------------	--	--	--	--

Bit 4 = 1: Spindle rotates speed analog control, it must be 1 when use spindle auto shift function;

= 0: Spindle rotates speed switch value control.

K	1	5					SHT	AGIM	AGIN	AGER
---	---	---	--	--	--	--	-----	------	------	------

AGER=1: spindle auto shift function is valid;

=0: spindle auto shift function is invalid;

AGIN=1: detect shift position signal M41I and M42I when spindle auto shift to 1st and 2nd

=0: wouldn't detect shift position signal M41I and M42I when spindle auto shift to 1st and 2nd

AGIM=1: shift position signal to M41I, M42I and +24V break up is valid;

=0: shift position signal to M41I, M42I and +24V connected is valid;

SHT=1: spindle shift power down memory;

=0: spindle shift power down non-memory.

Data parameter

2	1	0		GRMAX1
2	1	1		GRMAX2
2	1	2		GRMAX3

2	1	3	4	GRMAX4
---	---	---	---	---------------

GRMAX1, GRMAX2, GRMAX3 and GRMAX4: they are the spindle speed which corresponding to the 1st, 2nd, 3rd and 4th when Spindle analog voltage is 10V. when spindle auto shift is valid, they are the spindle highest speed corresponding to M41, M42, M43 and M44.

2	1	4	3	SFTREV
---	---	---	---	---------------

The output voltage (0 ~ 10000, unit is mV) when spindle shift

Function description (standard PLC program definition)

Spindle axis should be chosen under analog voltage control mode (Bit4 position of state parameter No.001 is 1), and Bit0 position of K parameter No.15 is 1, it is valid that spindle axis switches automatically; when spindle axis switches invalid, CNC will alarm when execute M41~M44. There is only one available of M41, M42, M43 and M44 at the same time.

Spindle auto switch function used for control auto switch spindle mechanical gear. When CNC execute S □□□□ code, calculate the output voltage according to current parameter (M41~44 corresponding data parameter No.210~213) which controlled by M4n, control the real rotation speed of spindle axis and S code are the same.

When power on, CNC controlled by parameter K of NO.15 Bit3 control whether to restore power in front of the main shaft gear. When the state parameters of NO. 001 of Bit4 is 0, with electricity, when the power is not memory spindle gear, default 1 gear main shaft gear, M41 ~ M44 NO output; when parameters of NO. 001 of Bit4 is 1, after power on electricity, main shaft gear memories. If you specify the gear is in line with current gear and not to shift. If you specify the gears do not agree with current gear, gear shifting, the standard of the definition of PLC shift process is as follows:

- ① Execute each code of M41, M42, M43, M44, output analog voltage to spindle servo or inverter according to the value (unit: mv) set by No.214;
- ② After delay data DT000 after (the shift time 1), shut down the original gear output signal output new shift signals at the same time;
- ③ When shift to 1 or 2, and Bit1 (AGIN) of № 15 K parameter is 1, the switch ④, otherwise the switch ⑤;
- ④ Check 1 or 2 files in the input signal M41I, M42I, if the shift in position switch ⑤; If the shift

does not reach the designated position, the CNC has been waiting for shift in position signal;

- ⑤ Delay data DT001 (2) shift time, according to the current gear according to the data parameter NO. 210 to NO. 213 (1 ~ 4) setting output spindle analog voltage, shift to an end.

Note: CNC reset, stopping, defined as the output state of M41 ~ M44 standard PLC stays the same

2.7.6 External cycle start and feeding hold

Relative signal (standard PLC program definition)

ST: External auto cycle start signal, it is of same function with the auto cycle start key of machine panel;

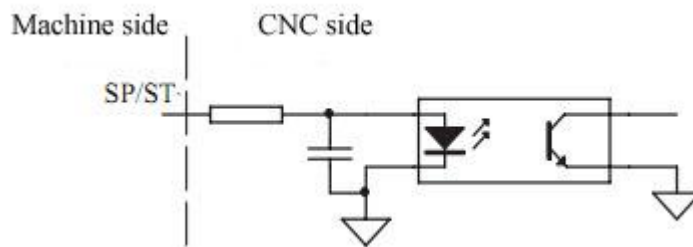
SP: External feeding hold signal, it is of same function with feeding hold key of machine panel.

Data diagnose

Signal	SP	ST
Diagnose address	X0.1	X1.4
Interface pin	CN61.2	CN61.13

Signal connection

SP/ST signal internal circuit as below picture 2-48



Picture 2-48

Control parameter

State parameter

0	2	1								MSP	MST
---	---	---	--	--	--	--	--	--	--	-----	-----

MST=1: External cycle start (ST) signal is invalid;

=0: External cycle start (ST) signal is valid.

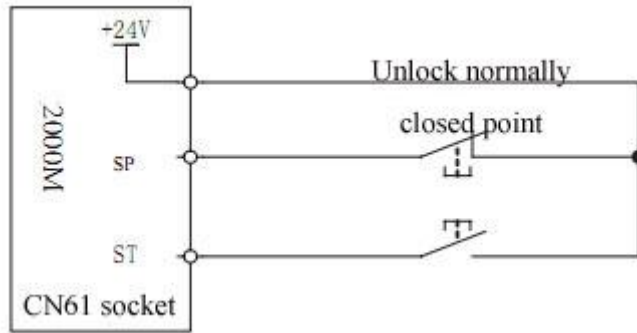
MSP=1: External pause signal is invalid;

=0: External pause signal is valid. It must connect external pause switch, or CNC display

“pause”

External connect circuit

SP, ST signal external connection as below picture 2-49



Picture 2-49

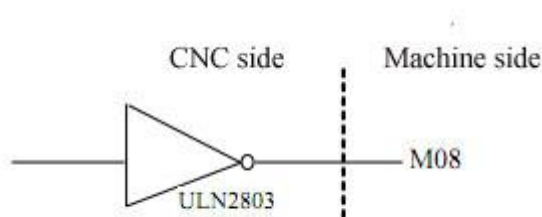
2.7.7 Cooling pump control

Relative instruction signal (standard PLC program definition)

Signal type	Symbol	Signal interface	Address	Function explanation	Remark
Output signal	M08	CN62.1	Y0.0	Cooling pump control output	
Instruction format	M08			Cooling liquid open	
	M09			Cooling liquid closed	

Signal connection

Internal circuit as below picture 2-50 shows:



Picture 2-50

Function description (standard PLC program definition)

M09 is valid and M08 is invalid after CNC power on. Execute M08, M08 is valid and cooling pump opened; execute M09, cancel M08 output, cooling pump closed.


Note 1: Cancel M08 output when CNC scram;

Note 2: When CNC reset, Bit 1 of CNC K parameter NO.10 sets if cancel M08 output;

Bit1=0: cancel M08 output when CNC reset;

Bit1=1: output state hold when CNC reset;

Note 3: M09 has no corresponding output signal, execute M09 cancel M08 output;

Note 4:  key of panel can control cooling pump switch, please refer the 2nd volume “operation explanation”

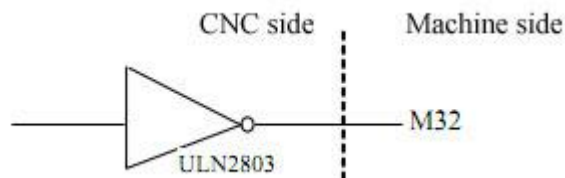
2.7.8 Lubrication control

Relative instruction signal (standard PLC program definition)

Signal type	Symbol	Signal interface	Address	Function explanation	Remark
Output signal	M32	CN62.2	Y0.1	Lubrication control output	
Instruction format	M32			Lubrication opened	
	M33			Lubrication closed	

Signal connection


Internal circuit as below picture 2-51 shows:



Control parameter

PLC state parameter

K 1 0 RSJG

RSJG=1: when press , CNC doesn't close M03, M04, M08 and M32 output signal.

=0: when press , CNC close M03, M04, M08 and M32 output signal

K	1	6							M32A		
---	---	---	--	--	--	--	--	--	------	--	--

M32A=1: when lubrication is valid, start up output lubrication

=0: when lubrication is valid, start up doesn't output lubrication

PLC data

D	T	0	5	3		
---	---	---	---	---	--	--

Auto lubrication interval time (0 ~ 65535ms)

D	T	0	1	3		
---	---	---	---	---	--	--


Auto lubrication output time (0 ~ 65535ms)



Function description

There are two lubrication functions of ECN*000T series standard PLC program definition, manual lubrication and auto lubrication set by parameter;


DT13=0: manual lubrication

>0: auto lubrication, lubrication time DT13 and interval time DT53 can be set

1. Manual lubrication function is lubrication reversal output, press  machine tool operation panel press, lubrication output cancel. When execute M32, lubrication output, lubrication output cancel through the time set by M33.

When DT13>1, lubrication timing output, press machine operation panel , lubrication output cancel through the time which set by DT13; lubrication output when execute M32, lubrication output cancel through DT13 setting. If the time that set by DT13 didn't come, now execute M33 or press  one more time, then lubrication output cancel.

2. Auto lubrication;

When K16.2 set to be 1, the time that set by DT13 started after system power on, and then stop output, repeat output lubrication through the time that set by DT53, recycle one by one. When auto lubrication, M32, M33 and  of machine operation panel are valid, lubrication time is the time that set by DT13.

Note 1: CNC scram, close lubrication output;

Note 2: CNC reset, Bit 1 of K parameter NO.010 set if cancels lubrication output;

When Bit1=0, CNC reset close lubrication output;

When Bit1=1, CNC reset lubrication output state hold on.

2.7.9 Protective door**Relative signal (standard PLC program definition)**

SAGT: protective door detect input signal

Signal diagnose

Signal	SAGT
Diagnose address	X0.0
Interface pins	CN61.1

Control parameter

State parameter

K	1	4						SPB4	PB4		
---	---	---	--	--	--	--	--	------	-----	--	--

PB4=0: protective door detection is invalid;

=1: protective door detection is valid;

SPB4 =0: when it is SAGT low level (break with +24V), the protective door is closed;

=1: when it is SAGT high level (connect with +24V), the protective door is closed;

Function description (standard PLC program definition)

- ① When PB4 = 1, SPB4 = 0, CNC can be confirmed that protective door closed when SAGT signal is broke with =24V;
- ② When PB4 = 1, SPB4 =1, CNC can be confirmed that protective door closed when SAGT signal connect with =24V;
- ③ Protective door detection function is valid in auto mode, all the modes will give alarm that “protective door is already opened” when protective door is open and wouldn’t affect other functions;
- ④ In auto mode, when auto cycle restart, it will alarm when CNC detects the protective opened;
- ⑤ During auto operation, if CNC detects protective door opened, then axis feeding paused, close spindle axis and cooling output.

2.7.10 CNC Marco variable

Relative signal

Marco output signal: standard PLC defined5 #1100 ~ #1105 macro output port;

Marco input signal: standard PLC defined 5 #1000 ~ #1015 macro input port;

Signal diagnose

Marco variable	#1105	#1104	#1103	#1102	#1101	#1100
----------------	-------	-------	-------	-------	-------	-------

No.						
Diagnose	Y3.7	Y3.6	Y3.5	Y3.4	Y3.3	Y3.2

Marco variable No.	#1007	#1006	#1005	#1004	#1003	#1002	#1001	#1000
Diagnose	X0.7	X0.6	X0.5	X0.4	X0.3	X0.2	X0.1	X0.0

Marco variable No.	#1015	#1014	#1013	#1012	#1011	#1010	#1009	#1008
Diagnose	X1.7	X1.6	X1.5	X1.4	X1.3	X1.2	X1.1	X1.0

Function description (standard PLC program definition)

Assignment a Marco variable to #1100 ~ #1105, can change the state of UO0 ~ UO5 output signal state. It outputs 0V when assignment is “1”; it closes its output signal when assignment is “0”.

The input states of interface X0.0 ~ X0.7, X1.0 ~ X1.7 can be learnt after detect the value of Marco variable #1000 ~ #1015.

2.7.11 Tricolor light

Relative signal and function definition (standard PLC program definition)

Y2.2 (CN62.31): Yellow light, it means normal state (non-running, non-alarm state)

Y2.3 (CN62.32): Green light, it means running state

Y2.4 (CN62.33): Red light, it means alarm state

2.7.12 External MPG

CN31 (MPG)	PLC address	Address symbol	Address definition function	Remark
5	X5.0	EHDX	X handwheel	Suit for PSG-100-05E/L, ZSSY2080 type MPG
6	X5.1	EHDY	Y handwheel	
8	X5.2	EHDZ	Z handwheel	
9	X5.3	EMPO	Increment *1	
22	X5.4	EMP1	Increment *10	
23	X5.5	EMP2	Increment *100	

11, 12, 13	GND			
14, 15	+5V			
17, 18	+24V			

Relative parameter

State parameter

0	0	1						SOHW			
---	---	---	--	--	--	--	--	------	--	--	--

Bit3 = 0: signal step work mode;

= 1: handwheel work mode.

PLC state parameter

K	1	6		SINC							
---	---	---	--	------	--	--	--	--	--	--	--

SINC = 0: handwheel, signal step mode *1000 gear increment is valid.

= 1: handwheel, signal step mode *1000 gear increment is invalid.

Function description

- ① MPG/sign step mode * 1000 increment gear chosen is invalid when SINC set to be 1, if it already chosen *1000 gear before amend the parameter, then it will change to *100mm gear automatically;
- ② When choose external MPG, axis choice of external MPG wouldn't clock automatically, namely when axis choice of MPG is invalid, it will be non-axis choice state;
- ③ When external MPG axis choice and gear choice input is valid, panel handwheel axis choice and gear choice key is invalid; when external MPG axis choice and gear choice input is invalid, panel MPG axis choice and gear choice key is valid and clock automatically.

2.7.12 External MPG

CN31 (MPG)	PLC address	Address symbol	Address definition function	Remark
5	X5.0	EHDX	X handwheel	Suit for PSG-100-05E/L, ZSSY2080 type MPG
6	X5.1	EHDY	Y handwheel	
8	X5.2	EHDZ	Z handwheel	
9	X5.3	EMPO	Increment *1	
22	X5.4	EMP1	Increment *10	

23	X5.5	EMP2	Increment *100	
11, 12, 13	GND			
14, 15	+5V			
17, 18	+24V			

Relative parameter

State parameter

0	0	1						SOHW			
---	---	---	--	--	--	--	--	------	--	--	--

Bit3 = 0: signal step work mode;

= 1: handwheel work mode.

PLC state parameter

K	1	6		SINC							
---	---	---	--	------	--	--	--	--	--	--	--

SINC = 0: handwheel, signal step mode *1000 gear increment is valid.

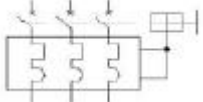
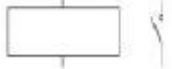
= 1: handwheel, signal step mode *1000 gear increment is invalid.

Function description

- ④ MPG/sign step mode * 1000 increment gear chosen is invalid when SINC set to be 1, if it already chosen *1000 gear before amend the parameter, then it will change to *100mm gear automatically;
- ⑤ When choose external MPG, axis choice of external MPG wouldn't clock automatically, namely when axis choice of MPG is invalid, it will be non-axis choice state;
- ⑥ When external MPG axis choice and gear choice input is valid, panel handwheel axis choice and gear choice key is invalid; when external MPG axis choice and gear choice input is invalid, panel MPG axis choice and gear choice key is valid and clock automatically.

2.8 Electric diagram common symbols comparison

In the design of ECN*000T of DC24V power supply and working current larger electromagnetic valve used DC24V power supply must be independent, Symbols of components can be stated below:

Name	Symbol	Diagram	Name	Symbol	Diagram
Air breaker	QF		Contact coil and assist	KM	

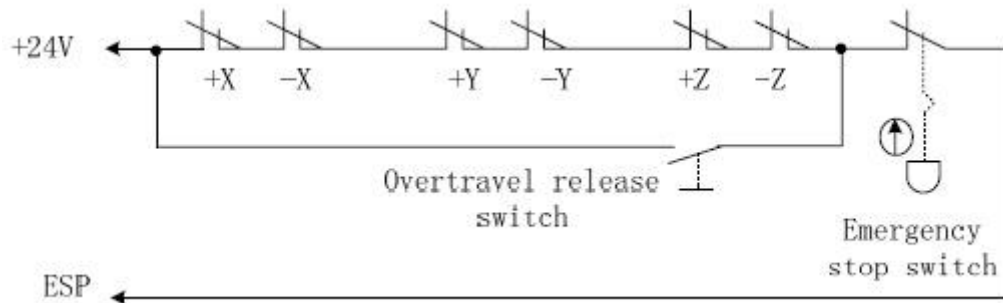
			terminal		
			Hot relay and terminal	FR	
Transformer	TC		Capacitance	C	
Bridge rectifier	VC		Resistance	R	
Motor	M		Hall element		
Diode	VD		Dynamic travel switch	SQ	
Solenoid valve clue	YV		Veneer jack		
Relay clue and contact	KA		Foot switch	SA	
			Fuse	FU	

Chapter 3 Machine Debugging methods and steps

The trial run methods and steps at initial power on for this ECN*000T series are described in this chapter. The corresponding operation can be performed after the debugging by the following steps.

3.1 Emergency stop and stroke limit

This ECN*000T series system has software limit function, it is suggested that stroke limit switches are fixed in the positive or negative axes for hardware limit. The connection is shown in follows: (The chart is designed for X, Y, Z axes)



Picture 3-1

So the BIT2 (MKYP) of bit parameter No.021 should be set to 0.

And the CNC diagnostic message DGN000.7 can monitor the state of emergency stop input signal.

In Manual or MPG mode, slowly move the axes to test the validity of stroke limit switch, correctness of alarm display, validity of over travel release key. When the over travel occurs or Emergency Stop key is pressed, “Emergency stop” alarm will be issued by CNC system. The alarm can be cancelled by pressing down the Over travel key and moving reversely.

3.2 Drive unit setting

Set BIT4~BIT0 of bit parameter No.009 according to alarm logic level of drive unit. The BIT4~BIT0 (5ALM, 4ALM, YALM, ZALM, XALM corresponding to 5th, 4th, Y, Z, X axis respectively) of bit parameter No.009 for our drive unit are all set for 1.

If the machine moving direction is not consistent with the moving command, modify the BIT4~BIT0 (DIR4, DIR5, DIRY, DIRZ, DIRX corresponding to 5th, 4th, Y, Z, X axis respectively) of

bit parameter No.008.

Manual moving direction can be changed through BIT4, BIT3, BIT2, BIT1 and BIT0 bit (5VAL, 4VAL, YVAL, ZVAL, XVAL corresponding to 5t, 4th, Y, Z, X axis moving key respectively) of No. 020.

3.3 Gear Ratio Adjustment

When different thread pitch lead pitch match with step motor in different step angle or servo motor in different pulse number, or connect through different ratio gear, it can make the program and real movement distance are the same through set electrical gear parameter.

Linear axis when configure step motors

$$\frac{CMR}{CMD} = \frac{360}{a \times L \times 1000}$$

CMR: command multiplier coefficient (parameter No. 000~004)

CMD: command fractional coefficient (parameter Np. 005~009)

a: step angle (degree)

L: step motor– turn to movement amount (millimeter) of corresponding machine tool

For example a=0.75 L=5

$$\frac{CMR}{CMD} = \frac{12}{125}$$

System minimum output unit is $CMD/CMR=125/12$ (unit: 0.001mm)

Note: no matter what step angle motor it configured, the minimum programming unit of system is 0.001mm, and the minimum output unit depends on a and L, the smaller the a and L are, the higher the resolution ratio is, but the speed will go down; on the contrary, the larger the a and L are, the higher the speed is, but the resolution ratio will go down.

Linear axis configure servo motors

$$\frac{CMR}{CMD} = \frac{P}{L \times 1000}$$

L: step motor – turn to movement amount (millimeter) of corresponding machine (millimeter)

P: motor – turn to corresponding pulse number of feedback

Rotation axis configure servo motors

$$\frac{CMR}{CMD} = \frac{P}{360000} \times \frac{ZM}{ZD}$$

P: motor – turn to corresponding pulse number of feedback

ZM: gears number of lead pitch end(Passive tooth)

ZD: gears number of motor end (driving tooth)

Tapping spindle axis

$$\frac{CMR}{CMD} = \frac{P}{1000} \times \frac{ZM}{ZD}$$

P: motor – turn to corresponding pulse number of feedback

ZM: gears number of lead pitch end(Passive tooth)

ZD: gears number of motor end (driving tooth)

CMR: command multiplier coefficient (parameter No. 230~232)

CMD: command fractional coefficient (parameter Np. 233~235)

3.4 Acceleration&deceleration Characteristic Adjustment

Adjust the relative CNC parameters according to the factors such as the drive unit, motor characteristics and machine load:

Data parameter №090~№094, №113~№117: X, Y, Z, 4th,and 5th axis rapid traverse rate;

Data parameter №120~№139: X,Y, Z, 4th, and 5th axis rapid traverse rate;

Data parameter №111 ~ №112: acceleration& deceleration time constant of each axis manual feeding;

Data parameter №154: proread method, acceleration speed (mm/s/s) before cutting feed

Data parameter №102 ~ №105: MPG speed and acceleration & deceleration time constant;

Data parameter №108: step feeding max strangulation speed;

Data parameter №110: each axis manual (JOG) feeding speeds in continuous.

The larger the acceleration&deceleration time constant is, the slower acceleration&deceleration is, the smaller the machine movement impact and the lower the machining efficiency is.

The principle for acceleration&deceleration characteristic adjustment is to properlyreduce the acceleration & deceleration time constant and increase the acceleration&decelerationstart/end speed to improve the machining efficiency on the condition that there is no alarm, motor out-of-step and obvious machine impact. If the acceleration&deceleration time constant is set too small, and the start/end speed is set too large, it is easily to cause drive unit alarm, motor out-of-step or machine vibration.

3.5 Machine zero adjust

Relative signal:

DECX: X axis deceleration signal;

DECY: Y axis deceleration signal;

DECZ: Z axis deceleration signal;

DEC4: the 4th axis axis deceleration signal;

DEC5: the 5th axis axis deceleration signal;

Diagnose data

0	0	0					DEC5	DEC4	DECZ	DECY	DECX
Interface pin							CN61.34	CN61.33	CN61.12	CN61.32	CN61.4

Control parameter

K	2	2		***	***	***	DEC5T	DEC4T	DECZ	DECY	DECX
----------	----------	----------	--	------------	------------	------------	--------------	--------------	-------------	-------------	-------------

DEC5T=0: the 5th axis deceleration signal low level;

=1: the 5th axis deceleration signal high level;

DEC4T=0: the 4th axis deceleration signal low level;

=1: the 4th axis deceleration signal high level;

DECY=0: Y axis deceleration signal low level;

=1: Y axis deceleration signal high level;

DECZ=0: Z axis deceleration signal low level;

=1: Z axis deceleration signal high level;

DECX=0: X axis deceleration signal low level;

=1: X axis deceleration signal high level;

0	0	6						ZPLS			ZMOD
----------	----------	----------	--	--	--	--	--	-------------	--	--	-------------

ZMOD=1: Back to zero pattern choice pieces before;

=0: Back to zero pattern choice pieces after;

ZPLS=1: Turn back to zero mode selection, a signal;

=0: Turn back to zero mode selection, no a signal.

0	1	2									ISOT
----------	----------	----------	--	--	--	--	--	--	--	--	-------------

ISOT=1: After electrify, back to the machine before midnight, manual fast moving is valid;

=0: After electrify, back to the machine before midnight, manual fast moving is invalid;

0	2	6					ZMI5	ZMI4	ZMIZ	ZMIY	ZMIX
---	---	---	--	--	--	--	------	------	------	------	------

ZMIX=1: Select the direction of shaft back to zero for negative direction back to zero;

=0: Select the direction of shaft back to zero for positive direction back to zero;

Data parameter

0	8	0		ZRNFL
---	---	---	--	-------

ZRNFL: Back to the mechanical zero point of the low speed rate;

0	7	0		ZRNFHX
---	---	---	--	--------

ZRNFHX: X axis and Z axis to mechanical zero point of the high speed;

0	7	1		ZRNFHY
---	---	---	--	--------

ZRNFHY: Y axis back to mechanical zero speed at a high speed;

0	7	2		ZRNFHZ
---	---	---	--	--------

ZRNFHZ: Z axis back to mechanical zero speed at a high speed;

0	7	3		ZRNFH4
---	---	---	--	--------

ZRNFH4: the 4th axis back to mechanical zero speed at a high speed;

0	7	4		ZRNFH5
---	---	---	--	--------

ZRNFH5: the 5th axis back to mechanical zero speed at a high speed;

According to the effective connection signal level, USES the way of back to zero, back to the direction of the zero adjustment related parameters:

№ 22 BIT4, BIT5, BIT6, BIT7 of K parameter: 4 axis X, Y, Z, return machine zero point, slow down the signal level effectively. State parameter BIT0 (ZMOD) of № 006 : back to the zero mode selection (0: gear block 1: after gear block before).

State parameter BIT3 of № 006: back to zero mode selection: (0: no. 1: yes) a turn signal.

Data parameters № 080: the low speed of each axis returns to machine zero deceleration process.

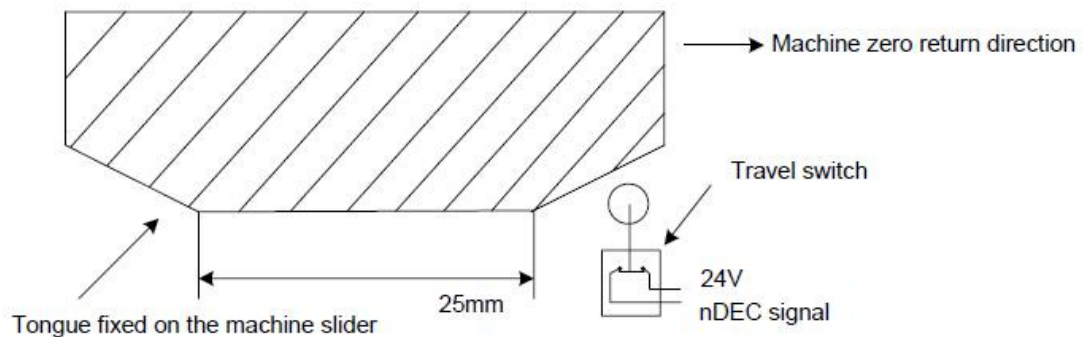
Data parameters № 070 ~ № 074: the axis return machine high speed velocity of zero.

№ 26 BIT0, BIT1, BIT2 (ZMIX ZMIY ZMIZ ZMI4 ZMI5) of State parameter: Each axis direction chooses back to zero, the positive direction back to zero, or negative direction back to zero

After the confirmation of super form a switch to the executable machine tools back to zero.

Usually install the machine zero point in the biggest trip, back to zero collision block effective stroke in more than 25 mm, to ensure the enough deceleration distance, can ensure the speed down, to ensure accurate back to zero. Perform the faster the speed of the machine back to zero, back to zero collision block to grow more, or you will make drag with CNC deceleration, such as machine inertia plate through back to zero speed after collision block failed to come down, there is not enough to slow down distance, affecting the precision of the back to zero. Usually there are two kinds of machine tools back to zero connection method:

1. Usually form a complete set of the ac servo motor connection: use a travel switch and servo motor a diagram of the turn signal



Picture 3-2

Using this connection, when back to the machine zero point after release of deceleration switch should be avoided at a encoder signal after the switch to release critical point position, turn a half circle of motor to turn the encoder signal, in order to improve the precision back to zero.

Parameter Settings (recommended) as follows:

BIT0 (ZMOD) of state parameter № 006=0

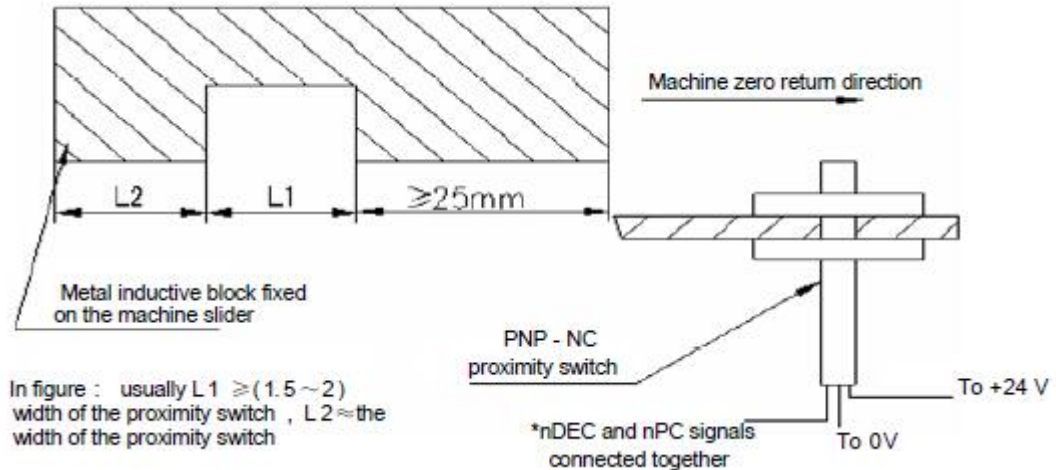
BIT3(ZPLS) of state parameter № 006=1

Data parameter № 080=200

State parameter № 026 的BIT0(ZMIX), BIT1(ZMIY), BIT2(ZMIZ), BIT3(ZMI4),

BIT4(ZMI5)=0

2. Usually form a complete set of stepping motor connection: use a proximity switch at the same time as the deceleration, zero signal diagram;



Form a complete set of stepping motor, parameter Settings (recommended) as follows:

BIT0 (ZMOD) of state parameter № 006=0

BIT3 (ZPLS) of state parameter № 006=0

State parameter № 026 的 BIT0(ZMIX), BIT1(ZMIY), BIT2(ZMIZ), BIT3(ZMI4),

BIT4(ZMI5)=0

Data parameter № 080=200

3.6 Spindle Adjustment

3.6.1 Spindle Encoder

Encoder with the linear number 100 p/r ~5000p/r is needed to be installed on the machine for threading. The linear number is set by data parameter No.109. The transmission ratio(spindle gear teeth/encoder gear teeth) between encoder and spindle is 1/255~255. The spindle gear teeth are set by CNC data parameter No.110 and the encoder gear teeth are set by data parameter No.111. Synchronous belt transmission should be applied for it (no sliding transmission)

3.6.2 Switch Volume Control of Spindle Speed

When multiple speed motor control is used, motor speed control command can be defined by ladder diagram as S01~S04. Relevant parameter is shown below.

Bit4=0 of parameter №001: select switching control of spindle speed.

3.6.3 Analog Voltage Control for Spindle Speed

This function can be obtained by the parameter setting of CNC. By interface outputting 0V~10V analog voltage to control inverter, the stepless shift can be obtained. And the related parameters are needed to be adjusted are:

Bit4=1 of parameter №001: for selection of spindle speed analog voltage control;

Data parameter №206: offset compensation value(mv) as spindle speed command voltage is 10V;

Data parameter №0210~№212: offset compensation value as spindle speed command voltage is 10V;

The inverter to adjust the basic parameters:

Positive &negative mode choice: determined by the terminal VF;

Frequency setting mode choice: determined by the terminal FR

If the speed by programming is not consistent with that detected by the encoder, it can be adjusted to be consistent with the actual one by adjusting the data parameter №210~№212.

Speed adjustment method: select the spindle first gear, input M03/M04 S9999 code in MDI mode to run the spindle, view the spindle speed shown on the right bottom of the screen, then reinput the displayed speed value into the parameter. The other spinle gear adjustment is identical with this.

When entering S9999 code, the voltage should be 10V, S0 is 0V. If there is an voltage error, adjust bit parameter №206 to correct the voltage offset value (corrected by manufacturer, usually not needed). When the current gear is the max.speed, if the analog voltage output by CNC is higher than 10V, set a smaller value for data parameter №100; when the S00 code is entered, if there is still slow rotation in the spindle, it means the analog voltage output by CNC is higher than 0V, so set a smaller value for data parameter №099.

If the machine is not fixed with an encoder, the spindle speed can be detected by a speed sensor, input S9999 in MDI mode to set the speed value displayed by sensor to the data parameter №210~212.

3.7 Backlash Offset

The backlash offset is input by diameter value with the unit 0.001mm, which is irrelevant to the programming by diameter or by radius. It can be measured by a dial indicator, a micrometer or a laser detector. Because the backlash offset can improve the machining precision only by accurate compensation, it is not recommended to measure it in MPG or Step mode, but the following method is suggested:

Program editing (as Z axis)

O0001;

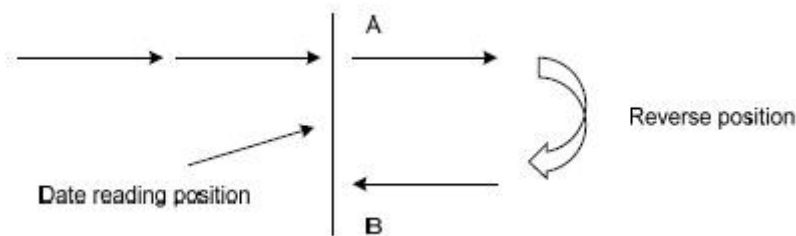
```

N10 G01 Z10 F800 G91;
N20 Z15;
N30 Z1;
N40 Z-1;
N50 M30.

```

Set the backlash error offset to 0 before measuring;

Run the program by single blocks, search the measuring benchmark A after 2 positioning operations, record the current data, move 1mm in the same direction, then move 1mm reversely to point B, read the current data.



Picture 3-4 Schematic map of backlash measuring methods

Backlash error offset value = | data of point A – data of point B |; Input the calculated data to the CNC data parameter №031 ~ №035.

Data A: dial-indicator data at point A

Data B: dial-indicator data at point B

Note 1: The backlash offset mode can be set by Bit 7 of CNC parameter No.011, and offset frequency can be set by of bit parameter №036~040;

Note 2: Check the machine backlash at regular intervals according to specific conditions to ensure machine precision

3.8 Step/MPG Adjustment

Operation panel key can be chosen to be the way for a single step or hand wheel operation, by the state parameter setting option № 001 Bit3.

Bit3=1: The handwheel operation effectively, a single step mode is invalid;

=0: Single-step operation effectively, the handwheel operation mode is invalid.

3.9 Other Adjustment

0	1	7		***	MST	MSP	MOT	MESP	***	***	***
---	---	---	--	-----	-----	-----	-----	------	-----	-----	-----

MST =1:External Cycle Start (ST) signal invalid.

=0:External Cycle Start(ST) signal valid;

MSP =1:External Dwell (SP) signal invalid.

=0: External Stop (SP) signal valid.

CHAPTER 4 DIAGNOSIS MESSAGE

4.1 CNC Diagnosis

This diagnosis section is used to check the CNC interface signals and internal running state and it can not be modified.

4.1.1 I/O Fixed address diagnose message

0	0	0		ESP	***	***	DEC5	DEC4	DECZ	DECY	DECX
Subscript				CN61.6			CN61.34	CN61.33	CN61.12	CN61.32	CN61.4
PCL fixed address				X0.5			X2.5	X2.4	X1.3	X2.3	X0.3

DECX, DECY, DECZ, DEC4, DEC5: X, Y, Z, 4th, 5th axis machine back to zero deceleration signal

KYP: scram signal

0	0	1		***	***	***	***	***	***	***	SKIP
Subscript											CN61.42
PCL fixed address											X3.5

4.1.2 Axes Moving State and Data Diagnosis Signal of CNC

0	0	4		***	***	***	EN5	EN4	ENZ	ENY	ENX
---	---	---	--	-----	-----	-----	-----	-----	-----	-----	-----

EN5 ~ ENX: axis enable signal

0	0	5		***	***	***	SET5	SET4	SETZ	SETY	SETX
---	---	---	--	-----	-----	-----	------	------	------	------	------

SET5 ~ SETX: axis pulse forbid signal

0	0	6		***	***	***	DRO5	DRO4	DROZ	DROY	DROX
---	---	---	--	-----	-----	-----	------	------	------	------	------

DRO5 ~ DROX: X, Y, Z, 4th and 5th axis movement direction output

0	0	9		***	***	***	5ALM	4ALM	ZALM	YALM	XALM
---	---	---	--	-----	-----	-----	------	------	------	------	------

5ALM ~ XALM: X, Y, Z, 4th and 5th axis alarm signal

4.1.3 Key diagnosis

DGN.016~DGN.022 are the diagnosis messages of MDI keypad key. When pressing a key in the operation panel, the corresponding bit displays “1”and“0”after releasing this key. If it displays reversely, it means there is a fault in the keypad circuit.

4.1.4 Others

1	4	5		PLC executes time (ms)
1	4	6		System runs total time (h)

4.2 PLC state

This part of diagnosis is used to detect the signal state of machine→PLC (X),

PLC→machine(Y),CNC→PLC(F),PLC→CNC (G) and alarm address A, state of internal relay (R, K).

4.2.1 F signal

F000		OP	SA	STL	SPL				
------	--	----	----	-----	-----	--	--	--	--

OP: Automatic signal

SA: Servo ready signal

STL: Light signal cycle started

SPL: Feed suspended light signal

F001		MA		TAP	ENB	DEN		RST	AL
------	--	----	--	-----	-----	-----	--	-----	----

MA: CNC ready signal

TAP: Tapping signal

ENB: Spindle enabling signal

DEN: Distribution over the signal

RST: Reset signal

AL: Alarm signal

F002		MDRN	CUT	MSTOP	SRNMV	THRD		RPDO	
------	--	------	-----	-------	-------	------	--	------	--

MDRN: Empty running detection signal

CUT: A cutting feed signal

MSTOP: Choose to stop testing signal

SRNMV: Program start signal

THRD: Thread cutting signal

RPDO: Rapid feed signal

F003			MEDT	MMEM	MRMT	MMDI	MJ	MH	MINC
------	--	--	------	------	------	------	----	----	------

MEDT: Editors' choice signal storage

MMEM: Run automatically select signal

MRMT: DNC operation select signal

MMDI: Manual data input selection signal

MJ: JOG Feed selection signal

MH: The handwheel feed selection signal detection

MINC: Incremental feed selection signal detection

F004			MPST	MREF	MAFL	MSBK	MABSM	MMLK	MBDT
------	--	--	------	------	------	------	-------	------	------

MPST: Back to the program start signal

MREF: A manual return reference point detection signal

MAFL: Assist lock detection signal

MSBK: Single segment detection signal

MABSM: Manual absolute value detection signal

MMLK: All the locked signal axis machine tool

MBDT: Skip the optional program segment detection signal

F007						TF	SF		MF
------	--	--	--	--	--	----	----	--	----

TF: Tool function gating signal

SF: The spindle speed gating signal

MF: Assist the gating signal

F008								SCHK	
------	--	--	--	--	--	--	--	------	--

SCHK: Syntax checking for signal

F009		DM00	DM01	DM02	DM30				RCT
------	--	------	------	------	------	--	--	--	-----

DM00: M00 decoded signal

DM01: M01 decoded signal2

DM01: M02decoded signal

DM30: M30 decoded signal

RCT: During the implementation of the tool change

F010		MB07	MB06	MB05	MB04	MB03	MB02	MB01	MB00
------	--	------	------	------	------	------	------	------	------

MB07: Auxiliary function code MB07

MB06: Auxiliary function code MB06

MB05: Auxiliary function code MB05

MB04: Auxiliary function code MB04

MB03: Auxiliary function code MB03

MB02: Auxiliary function code MB02

MB01: Auxiliary function code MB01

MB00: Auxiliary function code MB00

F014								DRUN	PDBG
------	--	--	--	--	--	--	--	------	------

PDBG: PLC enter debug mode

DRUN: Switching signal is prohibited

F015					EN5T	EN4T	ENZ		
------	--	--	--	--	------	------	-----	--	--

EN5T: the 5th axis selection

EN4T: the 4th axis selection

ENZ: Z axis selection

F016						ZP4	ZP3	ZP2	ZP1
------	--	--	--	--	--	-----	-----	-----	-----

ZP1: X axis returns zero point finish signals

ZP2: Y axis returns zero point finish signals

ZP3: Z axis returns zero point finish signals

ZP4: 4TH axis returns zero point finish signals

F018		AR07	AR06	AR05	AR04	AR03	AR02	AR01	AR00
------	--	------	------	------	------	------	------	------	------

AR07: The spindle actual speed AR07

AR06: The spindle actual speed AR06

AR05: The spindle actual speed AR05

AR04: The spindle actual speed AR04

AR03: The spindle actual speed AR03

AR02: The spindle actual speed AR02

AR01: The spindle actual speed AR01

AR00: The spindle actual speed AR00

F019		AR15	AR14	AR13	AR12	AR11	AR10	AR09	AR08
------	--	------	------	------	------	------	------	------	------

AR15: The spindle actual speed AR15

AR14: The spindle actual speed AR14

AR13: The spindle actual speed AR13

AR12: The spindle actual speed AR12

AR11: The spindle actual speed AR11

AR10: The spindle actual speed AR10

AR09: The spindle actual speed AR09

AR08: The spindle actual speed AR08

F020								BCLP	BUCLP
------	--	--	--	--	--	--	--	------	-------

BCLP: 4TH axis dividing the workbench clamping signals

BUCLP:4TH axis dividing the workbench loosening signals

F021			MST	MSP		MESP			
------	--	--	-----	-----	--	------	--	--	--

MST:Shielding external loop start signal

MSP: Shielding external signal

MKYP: Shielding external stop signal

F022		SB07	SB06	SB05	SB04	SB03	SB02	SB01	SB00
------	--	------	------	------	------	------	------	------	------

SB07:The spindle speed code signalSB07

SB06:The spindle speed code signalSB06

SB05:The spindle speed code signalSB05

SB04:The spindle speed code signalSB04

SB03:The spindle speed code signalSB03

SB02:The spindle speed code signalSB02

SB01:The spindle speed code signalSB01

SB00:The spindle speed code signalSB00

F026		TB07	TB06	TB05	TB04	TB03	TB02	TB01	TB00
------	--	------	------	------	------	------	------	------	------

TB07: Tool function code signal TB07

TB06: Tool function code signal TB06

TB05: Tool function code signal TB05

TB04: Tool function code signal TB04

TB03: Tool function code signal TB03

TB02: Tool function code signal TB02

TB01: Tool function code signal TB01

TB00: Tool function code signal TB00

F030		R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
------	--	------	------	------	------	------	------	------	------

R08O: S12 bit code signal R08O

R07O: S12 bit code signal R07O

R06O: S12 bit code signal R06O

R05O: S12 bit code signal R05O

R04O: S12 bit code signal R04O

R03O: S12 bit code signal R03O

R02O: S12 bit code signal R02O

R01O: S12 bit code signal R01O

F031						R12O	R11O	R10O	R09O
------	--	--	--	--	--	------	------	------	------

R12O: S12 bit code signal R12O

R11O: S12 bit code signal R11O

R10O: S12 bit code signal R10O

R09O: S12 bit code signal R09O

F032		X1000	X100	X10	X1			RGSPM	RGSP
------	--	-------	------	-----	----	--	--	-------	------

X1000: step width X1000soft key

X100: step width X100 soft key

X10: step width X10 soft key

X1: step width X1 soft key

RGSPM: Rigid tapping spindle in reverse

RGSP: Rigid tapping spindle is turned in

F033		MTAP	DTAP						RTAP
------	--	------	------	--	--	--	--	--	------

MTAP: G63 Tapping mode signal

DTAP: Rigid tapping the execution of the signal

RTAP: Rigid tapping mode signal

F034		SSTOP	SCW	Z-	Z+	Y-	Y+	X-	X+
------	--	-------	-----	----	----	----	----	----	----

SSTOP: The spindle stop soft key

SCW: The spindle is soft key

Z-: Z- soft key

Z+: Z+ soft key

Y-: Y- soft key

Y+: Y+ soft key

X-: X- soft key

X+: X+ soft key

F035		SCCW	MSTOP	AFLO	BDTO	SBKO	MLKO	DRNO	QFAST
------	--	------	-------	------	------	------	------	------	-------

SCCW: Spindle turn counterclockwise soft key

MSTOP: Select stop soft key

AFLO: Lock the soft key auxiliary function

BDTO: Jump period of soft key program

SBKO: Single paragraph soft key

MLKO: Machine tool lock soft key

DRNO: Empty running soft key

QFAST: Fast moving soft key

F036		S-	S+	FAST-	FAST+			FEED-	FEED+
------	--	----	----	-------	-------	--	--	-------	-------

S-: Spindle ratio decrease soft key

S+: Spindle ratio increase soft key

FAST-: Quick ratio decrease soft key

FAST+: Quick ratio increase soft key

FEED-: Feed rate minus soft key

FEED+: Feed rate plus soft key

F037					ZP5	ZP4	ZP3	ZP2	ZP1
------	--	--	--	--	-----	-----	-----	-----	-----

ZP5: End signal return reference point ZP5

ZP4: End signal return reference point ZP4

ZP3: End signal return reference point ZP3

ZP2: End signal return reference point ZP2

ZP1: End signal return reference point ZP1

F038									
------	--	--	--	--	--	--	--	--	--

MV5: Axis mobile signal MV5

MV4: Axis mobile signal MV4

MV3: Axis mobile signal MV3

MV2: Axis mobile signal MV

MV1: Axis mobile signal MV1

F039					MVD5	MVD4	MVD3	MVD2	MVD1
------	--	--	--	--	------	------	------	------	------

MVD5: Axis direction signalMVD5

MVD4: Axis direction signalMVD4

MVD3: Axis direction signalMVD3

MVD2: Axis direction signalMVD2

MVD1: Axis direction signalMVD1

F040					ZRF5	ZRF4	ZRF3	ZRF2	ZRF1
------	--	--	--	--	------	------	------	------	------

ZRF5: Reference point to establish the signal ZRF5

ZRF4: Reference point to establish the signal ZRF4

ZRF3: Reference point to establish the signal ZRF3

ZRF2: Reference point to establish the signal ZRF2

ZRF1: Reference point to establish the signal ZRF1

F041					ZP15	ZP14	ZP13	ZP12	ZP11
------	--	--	--	--	------	------	------	------	------

ZP15: 5TH axis end of shaft to return to the first reference point signal

ZP14: 4TH axis end of shaft to return to the first reference point signal

ZP13: Z axis end of shaft to return to the first reference point signal

ZP12: Y axis end of shaft to return to the first reference point signal

ZP11: X axis end of shaft to return to the first reference point signal

F042					PRO5	PRO4	PRO3	PRO2	PRO1
------	--	--	--	--	------	------	------	------	------

PRO5: Return the application zero signal PRO5

PRO4: Return the application zero signal PRO4

PRO3: Return the application zero signal PRO3

PRO2: Return the application zero signal PRO2

PRO1: Return the application zero signal PRO1

F043									MSPHD
------	--	--	--	--	--	--	--	--	-------

MSPHD: Spindle motion detection signal

F044					SIMSPL			FSCSL	
------	--	--	--	--	--------	--	--	-------	--

SIMSPL: Simulate the spindle effective

FSCSL: Cs Profile control end of the switch signal

F047		Total knife digits							
------	--	--------------------	--	--	--	--	--	--	--

F048			MST	MSP		MESP			
------	--	--	-----	-----	--	------	--	--	--

MST: Shielding external loop start signal

MSP: Shielding external signal

MKYP: Shielding external stop signal

F051					VAL5	VAL4	VALY	VALZ	VALX
------	--	--	--	--	------	------	------	------	------

VAL5: 5 direction of choice

VAL4: 4 direction of choice

VALY: Y direction of choice

VALZ: Z direction of choice

VALX: X direction of choice

F054		UO07	UO06	UO05	UO04	UO03	UO02	UO01	UO00
------	--	------	------	------	------	------	------	------	------

UO07: Macro output signal UO07

UO06: Macro output signal UO06

UO05: Macro output signal UO05

UO04: Macro output signal UO04

UO03: Macro output signal UO03

UO02: Macro output signal UO02

UO01: Macro output signal UO01

UO00: Macro output signal UO00

F055		UO15	UO14	UO13	UO12	UO11	UO10	UO09	UO08
------	--	------	------	------	------	------	------	------	------

UO15: Macro output signal UO15

UO14: Macro output signal UO14

UO13: Macro output signal UO13

UO12: Macro output signal UO12

UO11: Macro output signal UO11

UO10: Macro output signal UO10

UO09: Macro output signal UO09

UO08: Macro output signal UO08

F057					ZP25	ZP24	ZP23	ZP22	ZP21
------	--	--	--	--	------	------	------	------	------

ZP25: 5TH axis return to the end of the second reference point signal

ZP24: 4TH axis return to the end of the second reference point signal

ZP23: Z axis return to the end of the second reference point signal

ZP22: Y axis return to the end of the second reference point signal

ZP21: X axis return to the end of the second reference point signal

F058					ZP35	ZP34	ZP33	ZP32	ZP31
------	--	--	--	--	------	------	------	------	------

ZP35: 5TH axis return to the end of the third reference point signal

ZP34: 4TH axis return to the end of the third reference point signal

ZP33: Z axis return to the end of the third reference point signal

ZP32: Y axis return to the end of the third reference point signal

ZP31: X axis return to the end of the third reference point signal

F059					ZP45	ZP44	ZP43	ZP42	ZP41
------	--	--	--	--	------	------	------	------	------

ZP45: 5TH axis return to the end of the 4th reference point signal

ZP44: 4TH axis return to the end of the 4th reference point signal

ZP43: Z axis return to the end of the 4th reference point signal

ZP42: Y axis return to the end of the 4th reference point signal

ZP41: X axis return to the end of the 4th reference point signal

F061									ESEND
------	--	--	--	--	--	--	--	--	-------

KYEND: The required number of parts to the signal

4.2.2 G Sign

G004						FIN			
------	--	--	--	--	--	-----	--	--	--

FIN: Assist the end of the signal

G005		LEDT	AFL		LAXIS				
------	--	------	-----	--	-------	--	--	--	--

LEDT: Edit lock signal

AFL: Assist lock signal

LAXIS: The entire shaft interlocking signal

G006			SKIPP		OVC		ABSM	MSTOP	SRN
------	--	--	-------	--	-----	--	------	-------	-----

SKIPP: skip signal

OVC: Feed rate cancellation signal

ABSM: A manual signal absolute value

MSTOP: Select stop signal

SRN: Application restart signal

G007							ST		
------	--	--	--	--	--	--	----	--	--

ST: Cycle start signal

G008				SP	ESP				
------	--	--	--	----	-----	--	--	--	--

SP: Feed signal

KYP: scram signal

G009							M12	M32	COOL
------	--	--	--	--	--	--	-----	-----	------

M12: 0/1: spindle axis tool loose/clamp signal

M32: Lubrication signal

COOL: Cooling signal

G010		JV07	JV06	JV05	JV04	JV03	JV02	JV01	JV00
------	--	------	------	------	------	------	------	------	------

JV07: Manual mobile rate signal JV07

JV06: Manual mobile rate signal JV06

JV05: Manual mobile rate signal JV05

JV04: Manual mobile rate signal JV04

JV03: Manual mobile rate signal JV03

JV02: Manual mobile rate signal JV02

JV01: Manual mobile rate signal JV01

JV00: Manual mobile rate signal JV00

G011		JV15	JV14	JV13	JV12	JV11	JV10	JV09	JV08
------	--	------	------	------	------	------	------	------	------

JV08: Manual mobile rate signal JV08

JV09: Manual mobile rate signal JV09

JV10:Manual mobile rate signalJV10

JV11:Manual mobile rate signalJV11

JV12:Manual mobile rate signalJV12

JV13:Manual mobile rate signalJV13

JV14:Manual mobile rate signalJV14

JV15:Manual mobile rate signalJV15

G012		FV07	FV06	FV05	FV04	FV03	FV02	FV01	FV00
------	--	------	------	------	------	------	------	------	------

FV07: Feed speed ratio signal FV07

FV06: Feed speed ratio signal FV06

FV05: Feed speed ratio signal FV05

FV04: Feed speed ratio signal FV04

FV03: Feed speed ratio signal FV03

FV02: Feed speed ratio signal FV02

FV01: Feed speed ratio signal FV01

FV00: Feed speed ratio signal FV00

G014		RV08	RV07	RV06	RV05	RV04	RV03	RV02	RV01
------	--	------	------	------	------	------	------	------	------

RV08: Fast feed rate signal RV08

RV07: Fast feed rate signal RV07

RV06: Fast feed rate signalRV06

RV05: Fast feed rate signalRV05

RV04: Fast feed rate signalRV04

RV03: Fast feed rate signalRV03

RV02: Fast feed rate signalRV02

RV01: Fast feed rate signalRV01

G016					SAR				
------	--	--	--	--	-----	--	--	--	--

SAR: Spindle speed signal

G017						DECA	DECY	DECAZ	DECAX
------	--	--	--	--	--	------	------	-------	-------

DECA: 4TH axis back to zero speed signal detection

DECY: Z axis back to zero speed signal detection

DECZ: Y axis back to zero speed signal detection

DECX: X axis back to zero speed signal detection

G018						H4TH	HY	HZ	HX
------	--	--	--	--	--	------	----	----	----

H4TH: 4TH axis handwheel feed selection signal

HY: Z axis handwheel feed selection signal

HZ: Y axis handwheel feed selection signal

HX: X axis handwheel feed selection signal

G019		RT		MP2	MP1				
------	--	----	--	-----	-----	--	--	--	--

RT: Manual feed selection signal quickly

MP2: The handwheel ratio signal MP2

MP1: The handwheel ratio signal MP1

G021		SOV7	SOV6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0
------	--	------	------	------	------	------	------	------	------

SOV7: The spindle speed ratio signal SOV7

SOV6: The spindle speed ratio signal SOV6

SOV5: The spindle speed ratio signal SOV5

SOV4: The spindle speed ratio signal SOV4

SOV3: The spindle speed ratio signal SOV3

SOV2: The spindle speed ratio signal SOV2

SOV1: The spindle speed ratio signal SOV1

SOV0: The spindle speed ratio signal SOV0

G022		R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
------	--	------	------	------	------	------	------	------	------

R08I: Spindle motor speed code signal R08I

R07I: Spindle motor speed code signal R07I

R06I: Spindle motor speed code signal R06I

R05I: Spindle motor speed code signal R05I

R04I: Spindle motor speed code signal R04I

R03I: Spindle motor speed code signal R03I

R02I: Spindle motor speed code signal R02I

R01I: Spindle motor speed code signal R01I

G023		SIND	SGN			R12I	R11I	R10I	R09I
------	--	------	-----	--	--	------	------	------	------

SIND: Spindle motor speed code selection signal

SGN: Spindle motor speed code selection signal

R12I: Spindle motor speed code selection signalR12I

R11I: Spindle motor speed code selection signalR11I

R10I: Spindle motor speed code selection signalR10I

R09I: Spindle motor speed code selection signalR09I

G024		MRDYA							
------	--	-------	--	--	--	--	--	--	--

MRDYA: Machine tool ready signal

G025				SRVB	SFRB				
------	--	--	--	------	------	--	--	--	--

SRVB: spindle axis reverse signal

SFRB: spindle axis forward signal

G026		CON							
------	--	-----	--	--	--	--	--	--	--

CON: CS Profile control of switch signal

G027						+J4	+J3	+J2	+J1
------	--	--	--	--	--	-----	-----	-----	-----

+J4: Feeding axis and direction selection signal +J4

+J3: Feeding axis and direction selection signal +J3

+J2: Feeding axis and direction selection signal +J2

+J1: Feeding axis and direction selection signal +J1

G028						-J4	-J3	-J2	-J1
------	--	--	--	--	--	-----	-----	-----	-----

-J4: Feeding axis and direction selection signal-J4

-J3: Feeding axis and direction selection signal-J3

-J2: Feeding axis and direction selection signal-J2

-J1: Feeding axis and direction selection signal-J1

G030						+L4	+L3	+L2	+L1
------	--	--	--	--	--	-----	-----	-----	-----

+L4: Axis overtravel signal+L4

+L3: Axis overtravel signal+L3

+L2: Axis overtravel signal+L2

+L1: Axis overtravel signal+L1

G031						-L4	-L3	-L2	-L1
------	--	--	--	--	--	-----	-----	-----	-----

-L4: Axis overtravel signal-L4

-L3: Axis overtravel signal-L3

-L2: Axis overtravel signal-L2

-L1: Axis overtravel signal-L1

G036		BEUCL	BECLP						SPD
------	--	-------	-------	--	--	--	--	--	-----

BEUCL: Indexing table loosen the complete signal

BECLP: Indexing table clamp the complete signal

SPD: Spindle dynamic function of the signal

G037		NT07	NT06	NT05	NT04	NT03	NT02	NT01	NT00
------	--	------	------	------	------	------	------	------	------

NT07: The current tool number NT07

NT06: The current tool number NT06

NT05: The current tool number NT05

NT04: The current tool number NT04

NT03: The current tool number NT03

NT02: The current tool number NT02

NT01: The current tool number NT01

NT00: The current tool number NT00

G043		ZRN		DNC1			MD4	MD3	MD1
------	--	-----	--	------	--	--	-----	-----	-----

ZRN: The current work mode selection 4

DNC1: DNC run the selection signal

MD4: The current work mode selection 3

MD2: The current work mode selection 2

MD1: The current work mode selection 1

G044		HDT						MLK	BDT
------	--	-----	--	--	--	--	--	-----	-----

HDT: Manual tool change signal sequence

MLK: Machine tool lock signal (LC → CNC)

BDT: Program selected jump signal(PLC → CNC)

G046		DRN				KEY1		SBK	
------	--	-----	--	--	--	------	--	-----	--

DRN: Empty running signal

KEY1: Memory protection signal

SBK: Single block signals(PLC → CNC)

G048								CR2	CR1
------	--	--	--	--	--	--	--	-----	-----

GR2: The gear selection signal

GR1: The gear selection signal

G053		CD2	SMZ						
------	--	-----	-----	--	--	--	--	--	--

CDZ: Chamfering signal

SMZ: Error check signal

G054		UI07	UI06	UI05	UI04	UI03	UI02	UI01	UI00
------	--	------	------	------	------	------	------	------	------

UI07: Macro input signalUI07

UI06: Macro input signalUI06

UI05: Macro input signalUI05

UI04: Macro input signalUI04

UI03: Macro input signalUI03

UI02: Macro input signalUI02

UI01: Macro input signalUI01

UI00: Macro input signalUI00

4.2.3 An address (message displays request signal, standard PLC definition)

Address	Alarm number	Display
A0002.0	1216	The protective door is opened, not allowed to run automatically
A0002.1	1217	Low pressure alarm
A0002.3	1219	The spindle rotates, must not loosen the cutting tool
A0002.4	1220	The spindle rotates, tool clamping invalid alarm signal to reach the designated position
A0002.5	1221	Tool clamping in place when the signal is invalid, can not start the spindle
A0002.6	1222	The spindle tool to loosen, and may not start the spindle
A0004.0	1232	Illegal M code
A0004.1	1233	The current is not simulate the spindle, unable to perform the function of point move
A0004.2	1234	M03, M04 specified error code
A0004.4	1236	Spindle shift time is too long
A0004.5	1237	The spindle speed/position control switch time is too long
A0005.1	1241	Spindle servo abnormal alarm or frequency converter
A0007.1	1257	Door has been opened

CHAPTER 5 MEMORIZING PITCH-PITCHES ERROR COMPENSATION FUNCTION

5.1 Function Explanation

There are more or less precision errors in the pitch-pitch of machine axes lead pitch, it will definitely affect the parts machining precision. This ECN*000T series has the memorizing pitch-pitch error.

5.2 Specifications

1. The offset is concerned with the offset origin, offset clearances, offset point, mechanical moving direction etc;
2. After performing the machine zero return, take this reference point as the offset origin, and set the offset value into the parameters according to axes compensation intervals;
3. Points to be compensated: 256 points for each axis
4. Axis to be compensated: X, Y, Z, 4th, 5th axis
5. Offset range: $0 \sim \pm 99 \times$ for each offset point
6. Offset clearance: $1 \sim 9999999 \mu\text{m}$;
7. Offset of point N (N=0, 1, 2, 3,...255) is determined by the N, N-1 mechanical error;
8. The setting is the same as the CNC parameters input, see the explanation in the relative operation.

5.3 Parameter Setting

5.3.1 Pitch-Pitch Compensation

State parameter

0	0	3				PITCH					
---	---	---	--	--	--	-------	--	--	--	--	--

Bit5=1: itch error compensation function is valid;

Bit5=0: itch error compensation function is invalid.

5.3.2 Pitch error compensation the origin

Machine tools original point which corresponding to pitch error compensation in the table compensation howling pitch error compensation the origin (reference point); the origin point of

pitch error compensation set by parameter № 180 ~ № 184. According to the actual demand, each axis can set any place in 0 ~ 255.

Data parameter

1	8	0		Position number of X axis pitch error compensation original point
1	8	1		Position number of Y axis pitch error compensation original point
1	8	2		Position number of Z axis pitch error compensation original point
1	8	3		Position number of the 4th axis pitch error compensation original point
1	8	4		Position number of the 5th axis pitch error compensation original point

5.3.3 Offset interval

Pitch error compensation: № 190 ~ № 194;

Input Units: metric system machine: mm, British system machine: inch;

Setting range: 1 ~ 9999.9999;

State parameter

1	9	0		Spacing distance of X axis pitch error compensation original point
1	9	1		Spacing distance of Y axis pitch error compensation original point
1	9	2		Spacing distance of Z axis pitch error compensation original point
1	9	3		Spacing distance of the 4th axis pitch error compensation original point
1	9	4		Spacing distance of the 5th axis pitch error compensation original point

5.3.4 Compensation dosage

The axis pitch pitch error compensation, according to the following table parameter set, fixed amount of compensation to the radius value input, has nothing to do with programming in diameter or radius of programming, input value is mm (metric machine) or inch (british machine).

5.4 Precaution of offset value setting

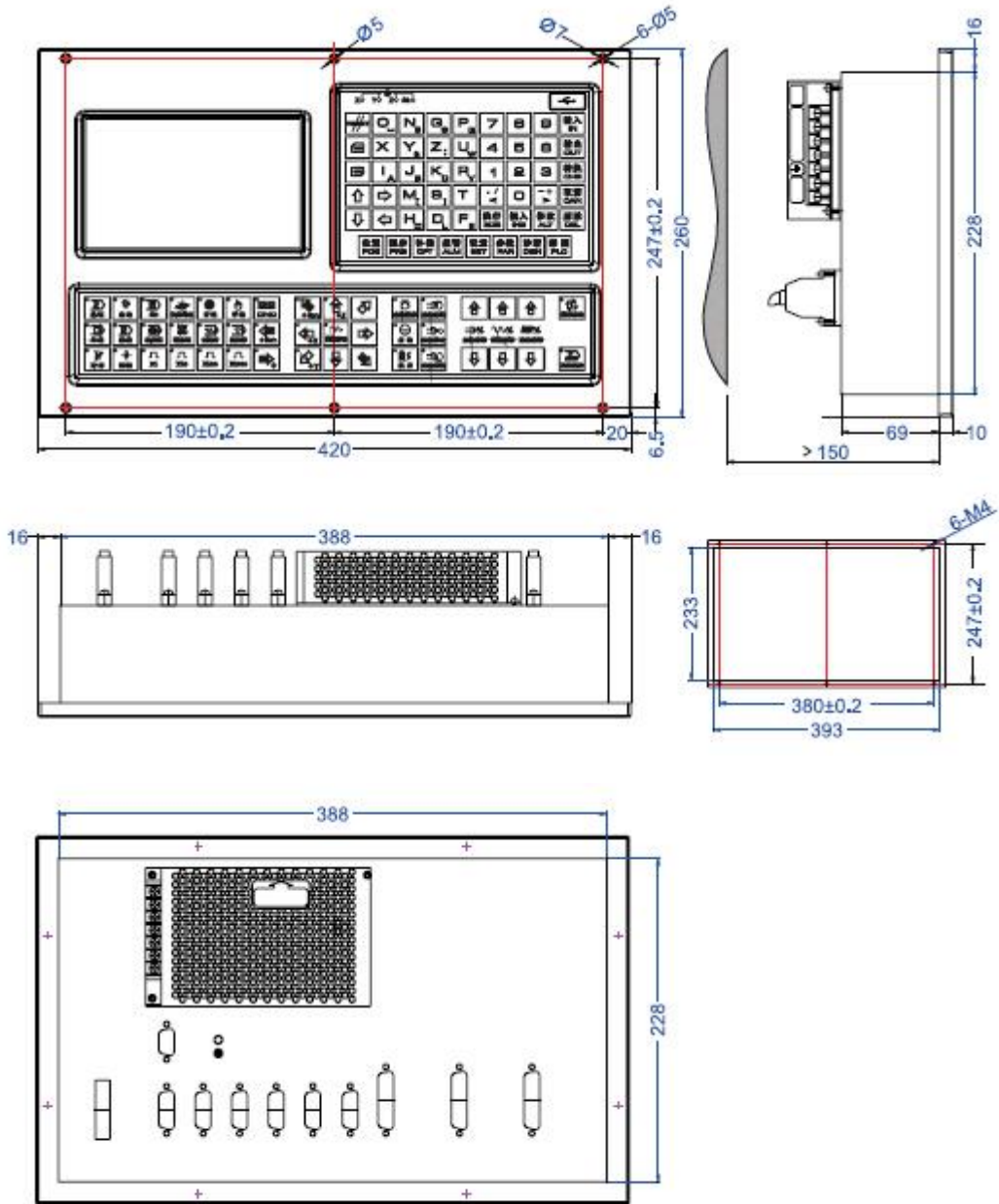
- ① Operation privileges must be level 2 password to set and amend pitch offset parameter;
- ② After setting the parameters of the pitch pitch error compensation, return the machine zero point can be carried out only after proper compensation.

5.5 Example of the compensation parameters setting

- ① Data parameter № 180 (pitch-pitch error origin point)=33, data parameter № 185 (pitch-pitch offset interval)=10.000mm, the pitch error offset point number is 33 in the following example.

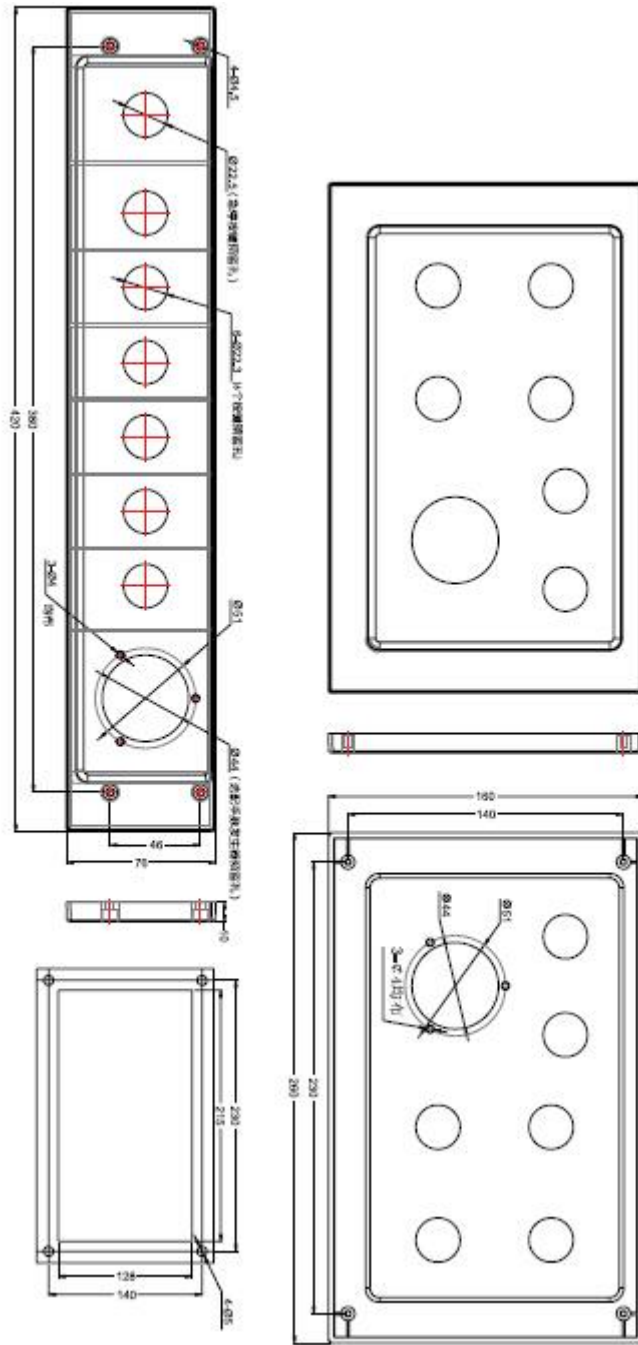
Appendix

Appendix 1 Outline Dimension of ECN*000T series





















ECN*000T series size


Appendix 2 Dimensions of Additional Panel




































Appendix 3 Common operations list

Classification	Function	Operation	Operation mode	Display page	Keyword level	Program switch	Parameter switch	Remark
Reset	Reset X axis relative coordinate			Relative coordinate				1.3.1 of Volume 2







	Reset Y axis relative coordinate	 , 		Relative coordinate				
	Reset Z axis relative coordinate	 , 		Relative coordinate				
	Reset processing quantity	 + 		Relative coordinate				
	Reset cutting time	 + 		or absolute coordinate				
Data setting	State parameter	Parameter value, 	Input mode	State parameter	Level 2, level 3		Open	
	Data parameter	Parameter value, 	Input mode	Data parameter	Level 2, level 3		Open	
	X axis pitch parameter input	 , Compensation value, 	Input mode	Pitch parameter	Level 2		Open	10.1.3 of Volume 2
	Y axis pitch parameter input	 , Compensation value, 	Input mode	Pitch parameter	Level 2		Open	
	Z axis pitch parameter input	 , Compensation value, 	Input mode	Pitch parameter	Level 2		Open	
	Marco variable value	Marco variable value, 			Marco variable	Level 2, level 3. Level4		

				value				2
	Tool offset	Compensation value, 		Tool offset	Level 2, level 3. Level4			7.4.2 of Volume 2

Classification	Function	Operation	Operation mode	Display page	Keyword level	Program switch	Parameter switch	Remark
Search	Search down from cursor	Symbol, 	Edit mode	Program context	Level 2, level 3. Level4	Open		6.1.3 of Volume 2
	Search up from cursor	Symbol, 	Edit mode	Program context				
	Search down from current place	 , 	Edit mode or auto mode	Program context, program catalogue or program state				6.4.1 of Volume 2
	Search up from current place	 , 						6.4.2 of Volume 2
	Search specified program	 , program name, 						
	Reach of state parameter, data parameter or pitch parameter	 , parameter number, 		Program relative page				10.1.3 of Volume 2
	PLC state, PLC data search	 , address number, 		PLC state and data				1.3.7 of Volume 2
Delete	Delete the symbol in cursor	 	Edit mode	Program context	Level 2, level 3. Level4	Open		6.1.6 of Volume 2
	Signal program	 , 						Program block includes block

	delete	SN, 						No. 6.1.7 of Volume 2
	Multiply program block delete							6.1.8 of Volume 2
	Block delete							6.1.9 of Volume 2
	Signal program delete	 , program name, 						6.3.1 of Volume 2
	Entire program delete	 ,  , 9999, 						6.3.2 of Volume 2
Rename	Rename of program	 , program name, 	Edit mode	Program content	Level 2, level 3, level 4			
Copy	Copy of program	 , program name, 						
Switch setting	Open parameter switch			Switch setting	Level 2, level 3			10.1.1 of Volume 2
	Open program switch				Level 2, level 3, level 4			
	Open auto SN switch							
	Close parameter switch				Level 2, level 3			
	Close program switch				Level 2, level 3, level 4			
	Close auto SN switch							

Description 1: In the "actions" column ", "said the operation between the two keys have successively order," + "means the operation between the two keys are at the same time.

Example:  ,  Said the first key  , again key  , and then press  +  said press the two key at the same time.

Explain 2: Operation mode, display page, password level, program, parameter switch and column blank said has nothing to do with the corresponding function.