Your Excellency：

We are honored by your purchase of GSK980TDi turning machine CNC system

The user manual introduces the programming, the operation, the installation and the connection of GSK980TTC turning machine CNC system in details.

To ensure safe and effective running, please read this manual carefully before installation and operation.

WARNING

Accident may occur by improper connection and operation！This system can only be operated by authorized and qualified personnel.

Special caution:

The power supply fixed on/in the cabinet is exclusively used for the CNC system

It can't be applied to other purposes, or else it may cause serious danger!

CAUTIONS

■ Delivery and storage

- Packing box over 6 layers in pile is unallowed.
- Never climb the packing box, stand on it or place heavy objects on it.
- Do not move or drag the products by the cables connected to it.
- Forbid collision or scratch to the panel and display screen.
- Avoid dampness, insolation and drenching.

■ Open-package inspection

- Confirm that the products are the required ones.
- Check whether the products are damaged in transit.
- Confirm that the parts in packing box are in accordance with the packing list.
- Contact us in time if any inconsistence, shortage or damage is found.

■ Connection

- Only qualified personnel can connect the system or check the connection.
- The system must be earthed, and the earth resistance must be less than 0.1Ω. The earth wire cannot be replaced by zero wire.
- The connection must be correct and firm to avoid any fault or unexpected consequence.
- Connect with surge diode in the specified direction to avoid damage to the system.
- Switch off power supply before plugging out or opening electric cabinet.

■ Troubleshooting

- Switch off power supply before troubleshooting or changing components.
- Check the fault when short circuit or overload occurs. Restart can only be done after troubleshooting.
- Frequent switching on/off of the power is forbidden, and the interval time should be at least 1 min.

Announcement

- This manual describes various possibilities as much as possible. However, operations allowable or unallowable cannot 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 as unallowable.

Warning !

- Before installing, connecting, programming and operating, please carefully read the product user manual and the manual from the machine tool manufacturer and strictly operate accordance with the regulations in the manual; otherwise, the product or the machine tool may be damaged, the workpiece may get rejected, even the personal injury may occur.

Caution !

- Functions, technical indexes (such as precision and speed) described in this user manual are only for this system. Actual function deployment and technical performance of the machine tool are designed by the machine tool manufacturer, so function configuration and technical indexes are subject to the user manual from the machine tool manufacturer.
- The system is with the standard machine operation panel, but the function of each button on the machine panel is defined by the PLC program (ladder diagram). Please pay attention to that the function of the buttons in the manual is described based on the standard PLC programs!
- Refer to the user manual from the machine tool manufacturer for function and meaning of each button on the machine panel.

All specifications and designs herein are subject to change without notice.

Section Ⅰ Programming

Introduce the technical specification, the product introduction, the command code and the program format.

Section Ⅱ Operation

Introduce the operation of CNC system of GSK980TDi turning machine.

Section Ⅲ Installation and Connection

Introduce the installation, connection and setting method of CNC system of GSK980TDi turning machine.

Appendix

Introduce the overall installation dimensions of CNC system of GSK980TDi turning machine and those of its accessories and the alarm information list, etc.

SAFETY RESPONSIBILITY

Manufacturer's Responsibility

——Be responsible for the danger which should be eliminated and/or controlled on design and configuration of the provided CNC systems and accessories.

——Be responsible for the safety of the provided CNC systems and accessories.

——Be responsible for the provided message and advice for the users.

User's Responsibility

——Be responsible for being familiar with and mastering the safety operation procedures through training with the safety operation of the CNC system.

——Be responsible for the dangers caused by adding, changing or altering the original CNC systems and the accessories.

——Be responsible for the dangers caused by failing to observe the provisions in the manual for operation, adjustment, maintenance, installation and storage.

This manual is kept by the end user.

CNCmakers Limited
Address: No.168, Xiadu Road, Haizhu District, Guangzhou, China 510300
Email: info@CNCmakers.com Website: www.CNCmakers.com Tel: +86-138-24444158 Fax: +86-20-84185336

CONTENTS

I Programming 1
CHAPTER 1 PROGRAMMING 3
1.1 GSK980TDi Introduction.....3
1.1.1 Product Introduction.....3
1.1.2 Technical Specification.....4
1.1.3 Environment & Conditions 7
1.1.4 Power Supply.....7
1.1.5 Guard.....7
1.2 CNC System of Machine Tools & CNC Machine Tools 7
1.3 Programming Fundamentals 9
1.3.1 Coordinates Definition.....9
1.3.2 Machine Coordinate System, Machine Zero & Machine Reference Point.....10
1.3.3 Workpiece Coordinate System & Program Zero 10
1.3.4 Interpolation Function 11
1.3.5 Absolute Programming & Incremental Programming 12
1.3.6 Diameter Programming & Radius Programming 13
1.4 Structure of an NC Program 13
1.4.1 General Structure of Program 14
1.4.2 Main Program & Subprogram 17
1.5 Program Run 18
1.5.1 Sequence of Program Run 18
1.5.2 Execution Sequence of Word.....19
1.6 Basic Axis Incremental System.....19
1.6.1 Incremental system speed of basic axis 20
1.6.2 Incremental System Unit of Basic Axis.....20
1.6.3 Incremental System Data Range of Basic Axis 20
1.6.4 Incremental System Data Range & Unit of Basic Axis 21
1.6.5 Program Address Value Unit & Range of Incremental System of Basic Axis 23
1.7 Additional Axis Incremental System.....23
1.7.1 Additional Axis of Current Incremental System 24
1.7.2 Additional Axis of IS-A Incremental System 24
CHAPTER 2 MSTF COMMAND 25
2.1 M (Miscellaneous Function).....25
2.1.1 End of Program M02 25
2.1.2 End of Program run M30 25
2.1.3 Subprogram Call M98 26
2.1.4 Return From Subprogram M99.....26
2.1.5 Macro Program Call M9000～M9999 27
2.1.6 M commands Defined by Standard PLC Ladder Diagram 27
2.1.7 Program Stop M00 28
2.1.8 Program Optional Stop M01.....29
2.1.9 Spindle CW, CCW & Stop Control M03, M04, M05.....29
2.1.10 Cooling Control M08, M09.....29
2.1.11 Tailstock Control M10, M11.....29
2.1.12 Chuck Control M12, M13.....29
2.1.13 Spindle Position/Speed Control Switch M14, M15 30
2.1.14 Spindle Clamped/Released M20, M21.....30
2.1.15 The 2nd Spindle Position/Speed Switch M24, M25 30
2.1.16 Lubricating Control M32, M33.....30
2.1.17 Spindle Automatic Gear Change M41, M42, M43, M44 30
2.1.18 Spindle 8-point Orientation M50～M58 31
2.1.19 The 2nd Spindle Rotation CCW/CW, Stop M63, M64, M65.....31
2.2 Spindle Function.....31
2.2.1 Spindle Speed Switching Value Control 31
2.2.2 Spindle Speed Analog Voltage Control.....32

2.2.3 Spindle Speed Series Control 32
2.2.4 Constant Surface Speed Control G96, Constant Rotational Speed Control G97 33
2.2.5 Spindle Override 35
2.2.6 Multiple Spindle Control Function 35
2.2.7 Cs Contour Control Funciton 37
2.2.8 Spindle Positioning Function 37
2.3 Tool Function 37
2.3.1 Tool Control 37
2.3.2 Tool Life Management 41
CHAPTER 3 G COMMANDS 51
3.1 Commands 51
3.1.1 Modal, One-Shot & Initial Mode 53
3.1.2 Omitting Words 53
3.1.3 Related Definitions 55
3.2 Rapid Traverse Movement G00 55
3.3 Linear Interpolation G01 56
3.4 Circular Interpolation G02, G03 57
3.5 Three-point Circular Interpolation G05, G05.1 60
3.6 Ellipse Interpolation G6.2, G6.3 62
3.7 Parabola Interpolation G7.2, G7.3 66
3.8 Plane Selection G17～G19 70
3.9 Polar Coordinate Interpolation G12.1, G13.1 71
3.10 Cylindrical Interpolation G7.1 74
3.11 Chamfering Function 79
3.11.1 Linear Chamfering 79
3.11.2 Circular Chamfering 81
3.11.3 Special Cases 83
3.12 Exact Stop Function Between Blocks 84
3.13 Function of Directly Inputting Graphic Dimension 86
3.14 Dwell G04 89
3.15 Machine Zero Function 90
3.15.1 Machine 1st Reference Point G28 90
3.15.2 Machine 2nd, 3rd, 4th Reference Point G30 91
3.16 Skip Interpolation G31 92
3.16.1 Following Block Execution After Skip 93
3.16.2 Signals Relevant to G31 94
3.16.3 Parameters Relevant to G31: Refer to Ⅲ INSTALLATION & CONNECTION, Section 3.2.15. 94
3.17 Automatic Tool Offset G36, G37 94
3.17.1 Signals Relevant to G36, G37 Automatic Tool Offset: 94
3.17.2 Parameters Relevant to G36, G37: Refer to INSTALLATION, Section 3.2.16. 95
3.17.3 G36, G37 Automatic Tool Offset Command Use 95
3.18 Workpiece Coordinate System G50 96
3.19 Local Coordinate System 97
3.20 Workpiece Coordinate System G54～G59 99
3.21 Fixed Cycle Command 101
3.21.1 Axial Cutting Cycle G90 101
3.21.2 Radial Cutting Cycle G94 104
3.21.3 Caution of Fixed Cycle Commands 106
3.22 Multiple Cycle Commands 106
3.22.1 Axial Roughing Cycle G71 107
3.22.2 Radial Roughing Cycle G72 114
3.22.3 Closed Cutting Cycle G73 118
3.22.4 Finishing Cycle G70 123
3.22.5 Axial Grooving Multiple Cycle G74 124
3.22.6 Radial Grooving Multiple Cycle G75 127
3.22.7 Drilling Cycle G83 or G87 130
3.23 Thread Cutting Commands 133

3.23.1 Thread Cutting with Constant Lead G32 133
3.23.2 Rigid Thread Cutting G32.1 136
3.23.3 Arc Threading Cutting Code G32.2 or G32.3 138
3.23.4 Thread Cutting with Variable Lead G34 139
3.23.5 Z Thread Cutting G33 141
3.23.6 Rigid Tapping G84, G88 142
3.23.7 Thread Cutting Cycle G92 145
3.23.8 Rigid Thread Cutting Cycle G92.1 148
3.23.9 Thread Reparation Function G92 P1 or G92.1 P1 152
3.23.10 Multiple Thread Cutting Cycle G76 154
3.23.11 Enhancement Thread Cutting Cycle G78 157
3.24 Constant Surface Speed Control G96, Constant Rotational Speed Control G97 163
3.25 Feedrate per Minute G98, Feedrate per Rev G99 163
3.26. Additional Axis Function 165
3.26.1 Additional Axis Start 165
3.26.2 Motion of Additional Axis 165
3.26.3 Additional Axis Coordinates Display 166
3.27 Macro Commands 166
3.27.1 Macro Variables 166
3.27.2 Operation & Jump Command G65 170
3.27.3 Program Example with Macro Command 174
3.28 Statement Macro Command 176
3.28.1 Arithmetic & Logic Operation 176
3.28.2 Transfer & Cycle 177
3.29 Metric/Inch Switch 179
3.30 Stored Stroke Limit 2 Detection G22 or G23 180
3.31 G Code Macro Program Call 181
3.32 Editable Parameter Read/Write G10 L52 182
CHAPTER 4 TOOL NOSE RADIUS COMPENSATION (G41, G42) 187
4.1 Application 187
4.1.1 Overview 187
4.1.2 Imaginary Tool Nose Direction 187
4.1.3 Compensation Value Setting 191
4.1.4 Command Format 192
4.1.5 Compensation Direction 192
4.1.6 Notes 194
4.1.7 Application 195
4.2 Tool Nose Radius Compensation Offset Path 196
4.2.1 Inner & Outer Side 196
4.2.2 Tool Traversing when Starting Tool 196
4.2.3 Tool Traversing in Offset Mode 198
4.2.4 Tool Traversing in Offset Canceling Mode 202
4.2.5 Tool Interference Check 204
4.2.6 Commands for Canceling Compensation Vector Temporarily 205
II Operation 209
CHAPTER 1 OPERATION MODE AND DISPLAY INTERFACE 211
1.1 Panel Division 211
1.1.1 State Indication 212
1.1.2 Edit Keypad 212
1.1.3 Menu Display 213
1.1.4 Machine Panel 214
1.2 Summary of Operation Mode 217
1.3 Display Interface 218
1.3.1 POS Interface 218
1.3.2 PRG Interface 220
1.3.3 TOOL OFFSET Interface 222
1.3.4 ALARM Interface 224
1.3.5 Setting Interface 225
1.3.6 Parameter Interfaces 227

1.3.7 Diagnosis Interface 229
1.3.8 Graph Interface 230
1.3.9 Ladder Interface 231
CHAPTER 2 POWER ON/OFF AND PROTECTION 235
2.1 System Power-on 235
2.2 System Power-off 235
2.3 Overtravel Protection 236
2.3.1 Hardware Overtravel Protection 236
2.4 Emergency Operation 237
2.4.1 Reset 237
2.4.2 ESP 238
2.4.3 Feed Hold 238
2.4.4 Power-Off 238
CHAPTER 3 MANUAL OPERATION 239
3.1 Coordinate Axis Move 239
3.1.1 Manual Feed 239
3.1.2 Manual Rapid Traverse 239
3.1.3 Speed Regulation 240
3.2 Other Manual Operations 241
3.2.1 Spindle CCW, CW, Stop Control 241
3.2.2 Spindle JOG 241
3.2.3 Cooling Control 241
3.2.4 Lubricating Control 242
3.2.5 Chuck Control 242
3.2.6 Tailstock Control 243
3.2.7 Hydraulic Control 243
3.2.8 Manual Tool Change 243
3.2.9 Spindle Override 243
CHAPTER 4 MPG/STEP OPERATION 245
4.1 Step Feed 245
4.1.1 Increment Selection 245
4.1.2 Moving Direction Selection 246
4.2 MPG Feed 246
4.2.1 Increment Selection 247
4.2.2 Moving Axis & Direction Selection 247
4.2.3 MPG Feedrate 248
4.2.4 Other Operations 248
4.2.5 Explanation Items 249
CHAPTER 5 MDI OPERATION 251
5.1 Block Input 251
5.2 Block Execution 252
5.3 Parameter Setting 252
5.4 Data Alteration 252
5.5 Other Operations 253
CHAPTER 6 PROGRAM EDIT & MANAGEMENT 255
6.1 Program Creation 255
6.1.1 Creating a Block Number 255
6.1.2 Inputting a Program 255
6.1.3 Searching a Character 257
6.1.4 Inserting a Character 260
6.1.5 Deleting a Character 261
6.1.6 Altering a Character 261
6.1.7 Deleting a Single Block 262
6.1.8 Copying & Pasting a Block 263
6.1.9 Canceling & Recovering a Program 263
6.1.10 Circle Command Aided Programming 264
6.1.11 Program Save 268
6.1.12 Macro Program Edit 268
6.1.13 Creating & Modifying a Program Annotation 269

6.2 Deleting Programs 270
6.2.1 Deleting a Program 270
6.2.2 Deleting All Programs 272
6.3 Selecting a Program 272
6.3.1 Searching 272
6.3.2 Scanning 273
6.3.3 Direct Confirmation 274
6.4 Executing a Program 274
6.5 Renaming a Program 274
6.6 Copying a Program 275
6.7 Program Management 276
6.8 Counter Function 277
6.9 Font Scaling 278
6.10 Program Preview Function 279
6.10.1 Interface Display of Path Preview 279
6.10.2 Operation of Path Preview 279
6.10.3 Command Display of Path Preview 281
6.10.4 Precautions 283
6.11 Teaching function 283
6.12 Other Operations Available in Edit Mode 284
CHAPTER 7 TOOL OFFSET AND SETTING 287
7.1 Tool Positioning Setting 287
7.2 Trial Tool-setting 288
7.3 Tool-Setting by Machine Zero Return 289
7.4 Coordinates Record 292
7.5 Setting & Altering an Offset Value 293
7.5.1 Offset Setting 293
7.5.2 Offset Alteration 293
7.5.3 Offset Alteration in Communication Mode 294
7.5.4 Offset Values Clearing 294
7.5.5 Setting & Altering a Tool Wear 295
7.5.6 Locking & Unlocking an Offset Value 295
7.5.7 No.0 Tool Offset Moving Workpiece Coordinate System 295
7.6 High Efficiency Tool-Change 296
7.6.1 Breif 296
7.6.2 Tool-Change Safety Area Setting 297
7.6.3 Setting Clear in Tool-Change Safety Area 299
7.6.4 High-Efficiency Too-Change Programming Format 299
7.6.5 High-Efficiency Tool-Change Procedure Explanation 300
7.6.6 Precaution of High-Efficiency Tool-Change 300
CHAPTER 8 COORDINATE SYSTEM 303
8.1 Coordinate System Setting 303
8.1.1 Directly Input 303
8.1.2 Measure Input 304
8.1.3 Machine Tool Coordinate Record 304
8.2 Coordinate System Modification 304
8.3 Coordinate System Clearing 305
CHAPTER 9 AUTO OPERATION 307
9.1 Automatic Run 307
9.1.1 Selection of a Program Running 307
9.1.2 Start of Automatic Run 307
9.1.3 Stop of Automatic Run 307
9.1.4 Automatic Run From an Arbitrary Block 308
9.1.5 Adjustment of Feedrate & Rapid Rate 309
9.1.6 Spindle Speed Adjustment 309
9.2 Running State 309
9.2.1 Single Block Execution 309
9.2.2 Dry Run 310
9.2.3 Machine Lock 310

9.2.4 MST Lock 311
9.2.5 Block Skip 311
9.3 MPG Trial-Cut 311
9.3.1 Switching MPG Trial-cut Mode 311
9.3.2 Command Speed in MPG Trial-Cut Mode 312
9.3.3 Notes in MPG Trial-Cut Mode 312
9.3.4 Temporarily Invalid in MPG Trial-cut Mode 313
9.4 MPG Interrupted Function 313
9.4.1 MPG Interrupted Operation 313
9.4.2 Affection of Coordinate System from MPG Interruption 314
9.4.3 MPG Interruption Value Cancel 315
9.4.4 MPG Interruption Display Page 316
9.4.5 Precaution of MPG Interruption 317
9.5 Other Operations 317
CHAPTER 10 ZERO RETURN OPERATION 319
10.1 Program Zero Return 319
10.1.1 Program Zero 319
10.1.2 Program Zero Return Steps 319
10.2 Machine Zero Return 320
10.2.1 Machine Zero 320
10.2.2 Machine Zero Return Steps 320
10.3 Other Operations in Zero Return 321
CHAPTER 11 DATA SETTING 323
11.1 Data Setting 323
11.1.1 Switch Setting 323
11.1.2 Level Setting 324
11.1.3 Parameter Operation 326
11.2 Clock Setting 327
11.3 Graphic Display 327
11.4 Parameter Setting 330
11.4.1 State Parameter 331
11.4.2 Data Parameter, Pitch Compensation 333
11.4.3 Classification Parameter 334
11.4.4 Often Used Parameters 335
11.5 Macro Variable Note 336
11.5.1 Macro Variable Note Page 336
11.5.2 Note List 336
11.5.3 Macro Variable Note File 337
CHAPTER 12 U OPERATION FUNCTION 339
12.1 File Management Function Introduction 339
12.2 Often Used File Operation Function Introduction 339
12.2.1 File Extension & Return 339
12.2.2 File Selection & Cancellation of File Selection 340
12.2.3 File Copy 341
12.3 Data Copy & Resume 341
12.3.1 Entering Backup/Resume Page 341
12.3.2 Backup/Resume Operation 342
12.3.3 Operation Path Selection 342
12.3.4 Format of Data Backup/Resume File 343
12.3.5 Level Explanation of Data Backup/Resume Operation 344
12.4 System Update 344
12.5 Notes 345
CHAPTER 13 COMMUNICATION 347
13.1 GSK980TDi Communication Software GSKComm 347
13.1.1 GSKComm's System (PC) Requirements 347
13.1.2 Software Interface 347
13.1.3 Receiving Files (CNC→PC) 348
13.1.4 Sending Files (CNC→PC) 349
13.1.5 Part Program Management 351

13.2 Preparatory Before Communication 352
13.2.1 Communication Cable Connection.....352
13.2.2 Communication Setting Baud Rate 352
CHAPTER 14 MACHINING EXAMPLES 355
14.1 Programming.....356
14.2 Program Input.....357
14.2.1 View a Saved Program 357
14.2.2 Creating a New Program 358
14.3 Checkout a Program.....358
14.3.1 Graphic Setting 358
14.3.2 Program Check.....359
14.4 Tool-setting & Running 359
III Connection.....363
CHAPTER 1 INSTALLATION LAYOUT.....365
1.1 GK980TDi System Connection.....365
1.1.1 GK980TDi Rear Cover Interface Layout 365
1.1.2 Interface Explanation 365
1.2 GSK980TDi Installation 366
1.2.1 GSK980TDi External Dimensions.....366
1.2.2 Preconditions of Cabinet Installation.....366
1.2.3 Measures Against Interference 366
CHAPTER 2 DEFINITION & CONNECTION OF INTERFACE SIGNALS.....369
2.1 GSKLink Bus Interface 369
2.2 Being Connected with Spindle Encoder.....369
2.2.1 Spindle Encoder Interface Definition.....369
2.2.2 Signal Explanation 369
2.2.3 Spindle Encoder Interface Connection.....370
2.3 MPG Connection 371
2.3.1 MPG Interface Definition.....371
2.3.2 Signal Explanation 371
2.4 Analog Spindle Interface.....373
2.4.1 Analog Spindle Interface Definition 373
2.4.2 Inverter Connection.....373
2.5 GSK980TDi / GSK980TDi-V being Connected with PC.....374
2.5.1 Communication Interface Definition 374
2.5.2 Communication Interface Connection 374
2.6 Network Interface 375
2.7 Power Interface Connection 375
2.8 I/O Interface Definition.....376
2.8.1 Input Signal.....380
2.8.2 Output Signal 381
2.9 I/O Function & Connection 382
2.9.1 Stroke Limit & ESP.....382
2.9.2 Tool Change Control 384
2.9.3 Machine Zero Return 392
2.9.4 Spindle Control 400
2.9.5 Spindle Wwitching Volume Control 403
2.9.6 Spindle Automatic Gearing Control 403
2.9.7 Spindle 8-point Orientation Function.....405
2.9.8 Spindle Cs Axis Control Function.....406
2.9.8 Multiple Spindle Function.....408
2.9.9 Rigid Tapping Function 410
2.9.10 Spindle Clamped Function 410
2.9.11 Spindle Positioning Function 411
2.9.12 External Cycle Start & Feed Hold.....412
2.9.13 Cooling Control 413
2.9.14 Lubricating Control.....414
2.9.15 Chuck Control 416
2.9.16 Tailstock Control.....418

2.9.17 Low Pressure Detection....420
2.9.18 Hydraulic Control....420
2.9.19 Safety Door Detection....421
2.9.20 CNC Macro Variables....422
2.9.21 External Override....423
2.9.22 External MPG....423
2.9.23 Tri-color Lamp....424
2.9.24 Extended M Function....425
2.9.25 HOLD Control....425
2.9.26 Handle Interrupt....426
2.9.27 Chip-Removal Control....426
2.9.28 Working Indicator....427
2.9.29 Gear/tool Number Display....427
2.10 Commonly Use Sign of Electricity Drawing....427
CHAPTER 3 PARAMETERS....429
3.1 Parameter Description (by Sequence)....429
3.1.1 Bit Parameter....429
3.1.2 Data Parameter....439
3.1.3 PLC Parameters（Standard PLC Definition）....457
3.2 Parameter Description (by Function Sequence)....462
3.2.1 X, Z, Y, 4th,5th Axis Control Logic....462
3.2.2 Acceleration & Deceleration Control....462
3.2.3 Precision Compensation....464
3.2.4 Machine Protection....466
3.2.5 Machine Zero Return....468
3.2.6 Thread Function....471
3.2.7 Spindle Control....472
3.2.8 Tool Compensation....476
3.2.9 Tool Life Management Function....476
3.2.10 Tool Wear Parameter....477
3.2.11 Edit & Display....477
3.2.12 Communication Setting....478
3.2.13 MPG Parameters....478
3.2.14 PLC Axis Control Function....479
3.2.15 Skip Function....479
3.2.16 Automatic Tool Setting Function....479
3.2.17 Input & Output Function in Metric/Inch System....480
3.2.18 Parameters Related to Arc Turning....480
3.2.19 Parameters Related to Addition....481
3.2.20 GSKLink Bus Related to Absolute Encoder....482
CHAPTER 4 MACHINE DEBUGGING METHODS AND MODES....483
4.1 ESP & Limit....483
4.1.1 ESP & Limit Series-Connection....483
4.1.2 Separately Connect of ESP & Limit....483
4.2 Drive Unit Configuration....484
4.3 Gear Ratio Adjustment....485
4.4 ACC&DEC Characteristic Adjustment....486
4.5 Mechanical (Machine) Zero Adjustment....486
4.6 Spindle Adjustment....488
4.6.1 Spindle Encoder....488
4.6.2 Spindle Brake....488
4.6.3 Switch Volume Control of Spindle Speed....488
4.6.4 Analog Voltage Control of Spindle Speed....488
4.6.5 Spindle Speed Series Control....489
4.7 Backlash Offset....489
4.8 Tool Post Debugging....490
4.9 Step/MPG Adjustment....491
CHAPTER 5 DIAGNOSIS MESSAGE....493
5.1 CNC Diagnosis....493

5.1.1 I/O Status & Data Diagnosis Message 493
5.1.2 CNC Motion State & Data Diagnosis Message 493
5.1.3 Diagnosis Keys 495
5.2 PLC State 495
5.2.1 X Address (Machine→PLC , Defined by Standard PLC Ladders) 496
5.2.2 Y Address (PLC→Machine, Defined by Standard PLC Ladders) 497
5.2.3 Machine Panel 499
5.2.4 F Address(CNC→PLC) 500
5.2.5 G Address (PLC→CNC) 507
5.2.6 Address A (Message Display Requiery Signal, Defined by Standard PLC Ladders) 514
CHAPTER 6 MEMORIZING PITCH ERROR COMPENSATION 517
6.1 Function Description 517
6.2 Specification 517
6.3 Parameter Setting 517
6.3.1 Pitch Compensation 517
6.3.2 Pitch Error Compensation Number 517
6.3.3 Pitch error compensation interval 518
6.3.4 Offset Value 518
6.4 Notes of Offset Setting 518
6.5 Setting Examples of Offset Parameters 518
6.5.1 Rotation Axis Pitch Error Compensation 521
IV Appendix 523
APPENDIX I GSK980TDi,GSK980TDi-V APPEARANCE DIMENSION 525
1.1 GSK980TDi Appearance Dimension 525
1.2 GSK980TDi-V Appearance Dimension 526
APPENDIX Ⅱ OUTLINE DIMENSION of ACCESSIONAL PANEL 527
2.1 Outline Dimension of Accessional Panel AP01A 527
2.2 Outline Dimension of Accessional Panel AP03A 527
APPENDIX Ⅲ OUTLINE DIMENSION of I/O Deconcentrator MCT01B 529
APPENDIX Ⅳ OUTLINE DIMENSION of I/O Deconcentrator MCT06R 531
APPENDIX Ⅴ DELIVERY STANDARD PARAMETER 533
APPENDIX VI ALARM LIST 541

I Programming

CHAPTER 1 PROGRAMMING

1.1 GSK980TDi Introduction

1.1.1 Product Introduction

Being upgraded product of GSK980TDi, GSK980TDi CNC turning is the better choice.

The updated development GSK980TDi Turning Machine CNC System made by CNCmakers is the new product based upon the GSK980TDi updated software/hardware, which owns the vertical and horizontal structures. GSK980TDi uses the GSKLINK bus technology, matches with the 17bit absolute encoder, carries out the automatic recovery of the system power-on, no-block mechanical zero return, servo parameter on-line modification or servo state on-line diagnosis, etc.

GSK980TDi

GSK980TDi-V

- 5 feed axes，2 analogy spindle
- control precision 0.1μm，Max. speed 120m/min
- GSK980TDi standard configuration with GS2000T-LA1 series strong overloading servo unit, it is used the GSKLINK bus technology, matched with 17Bit absolute encoder
- The spindle orientation can be carried out by adapting the servo spindle, Rigid tapping, Cylindrical Interpolation or Polar Coordinate Interpolation.
- Used the GSKLINK bus technology，carried out the functions such as the power-on automatically recover of the coordinate system, Non-block mechanical zero return, Servo parameter on-line modification and servo state on-line diagnosis, etc.
- Built-in multi PLC programs，PLC ladder on-line display and real time monitor, as well the series I/O unit.
- It owns the MPG trial cutting, Teachine, MPG interruption functions.
- It owns the functions, such as the on-line machine tool debugging guide, miscellaneous programming, diagram dimension directly input and polygon turning, etc.
- MDI multi-block performance
- 64M program store space, 10000 components program, and support the U disk program

machining.

- 6-level operation Authorization Management, Multi-level Timed stop
- 8.4 inch true-color LCD supports multi-language display, such as Chinese, English and Russian, etc.

1.1.2 Technical Specification

- Controllable axes
  - Controllable axes: 5（X, Z, Y , 4th,5th）
  - Link axes：3
  - PLC controllable axes：5（X, Z, Y, 4th,5th）
- **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: ±99999999× least command unit
  - Rapid traverse speed：120m/min
  - 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,Cylindrical Interpolation、Polar Coordinate 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**
  - The Max. spindle controllable number is 3, the top series spindle number is 3, the upmost analog spindle number is 2; support the multi-spindle control.
  - 2-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

- 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：48 input signals /38 output signals

Man-machine interface

- 8.4″ wide screen LCD，resolution: 800×600
- Chinese, English display
- Planar tool path display
- Real-time clock

Programming miscellaneous

- Aided programming
- Grammar check
- Two-dimensional tool path
- Path preview

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: 64MB，10000 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
- RJ45 access: File dual-direction transportation, such as the Component program,

parameters, PLC program, etc.

- 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

Command	Function	Command	Function	Command	Function
G00	Rapid traverse (positioning)	G28	Reference point return	G66	Macro program modal call
G01	Linear interpolation	G30	2nd, 3rd, 4th reference point return	G67	Macro program modal call cancel
G02	CW arc interpolation	G31	Skip function	G71	Axial roughing cycle (flute cycle)
G03	CCW arc interpolation	G32	Constant pitch thread cutting	G72	Radial roughing cycle
G04	Dwell, exact stop	G32.1	Rigid thread cutting	G73	Closed cutting cycle
G05	Three-point arc interpolation	G33	Z tapping cycle	G70	Finishing cycle
G6.2	Ellipse interpolation (CW)	G34	Thread cutting with variable lead	G74	Axial grooving cycle
G6.3	Ellipse interpolation (CCW)	G36	Automatic tool compensation X	G75	Radial grooving cycle
G7.2	Parabola interpolation (CW)	G37	Automatic tool compensation Z	G76	Multiple thread cutting cycle
G7.3	Parabola interpolation (CCW)	G40	Tool nose radius compensation cancel	G80	Rigid tapping state cancel
G12.1	Polar coordinate interpolation	G41	Tool nose radius compensation left	G83	Axial drilling cycle
G7.1	Cylinder interpolation	G42	Tool nose radius compensation right	G84	Axial rigid tapping
G15	Polar coordinate command cancel	G50	Workpiece coordinate system setting	G87	Radial drilling cycle
G16	Polar coordinate command	G52	Local workpiece coordinate system	G88	Radial rigid tapping
G17	Plane selection command	G54	Workpiece coordinate system 1	G90	Axial cutting cycle
G18	Plane selection command	G55	Workpiece coordinate system 2	G92	Thread cutting cycle
G19	Plane selection command	G56	Workpiece coordinate system 3	G92.1	Rigid thread cutting cycle
G10	Data input ON	G57	Workpiece coordinate system 4	G94	Radial cutting cycle
G11	Data input OFF	G58	Workpiece coordinate system 5	G96	Constant surface speed control
G20	Input in inch	G59	Workpiece coordinate system 6	G97	Constant surface speed control cancel

G21	Input in metric	G61	Exact stop begins between blocks	G98	Feed per minute
G22	Stored travel check ON	G64	Cancle the exact stop between blocks	G99	Feed per revolution
G23	Stored travel check OFF	G65	Macro command non-modal call		

1.1.3 Environment & Conditions

GSK980TDi 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

1.1.4 Power Supply

GSK980TDi 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

GSK980TDi guard level is not less than IP20.

1.2 CNC System of Machine Tools & CNC Machine Tools

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 buttons, 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.

GSK980TDi 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 GSK980TDi 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 GSK980TDi 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 GSK980TDi 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 GSK980TDi Turning CNC System User Manual. 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.

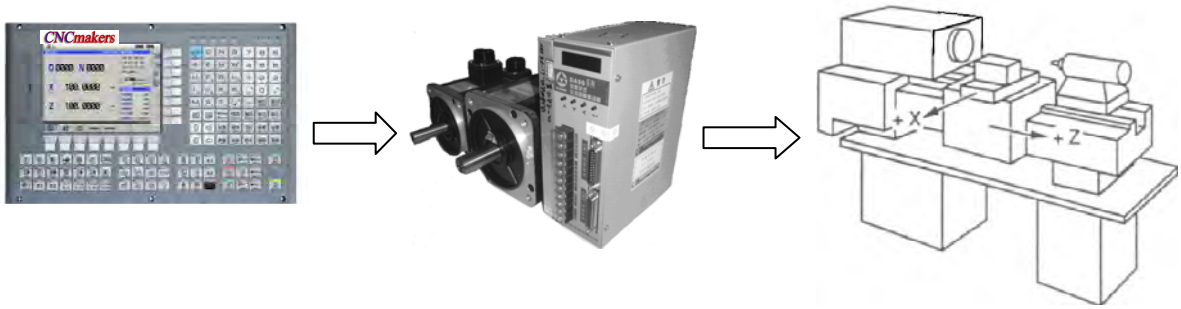


Fig. 1-1

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.

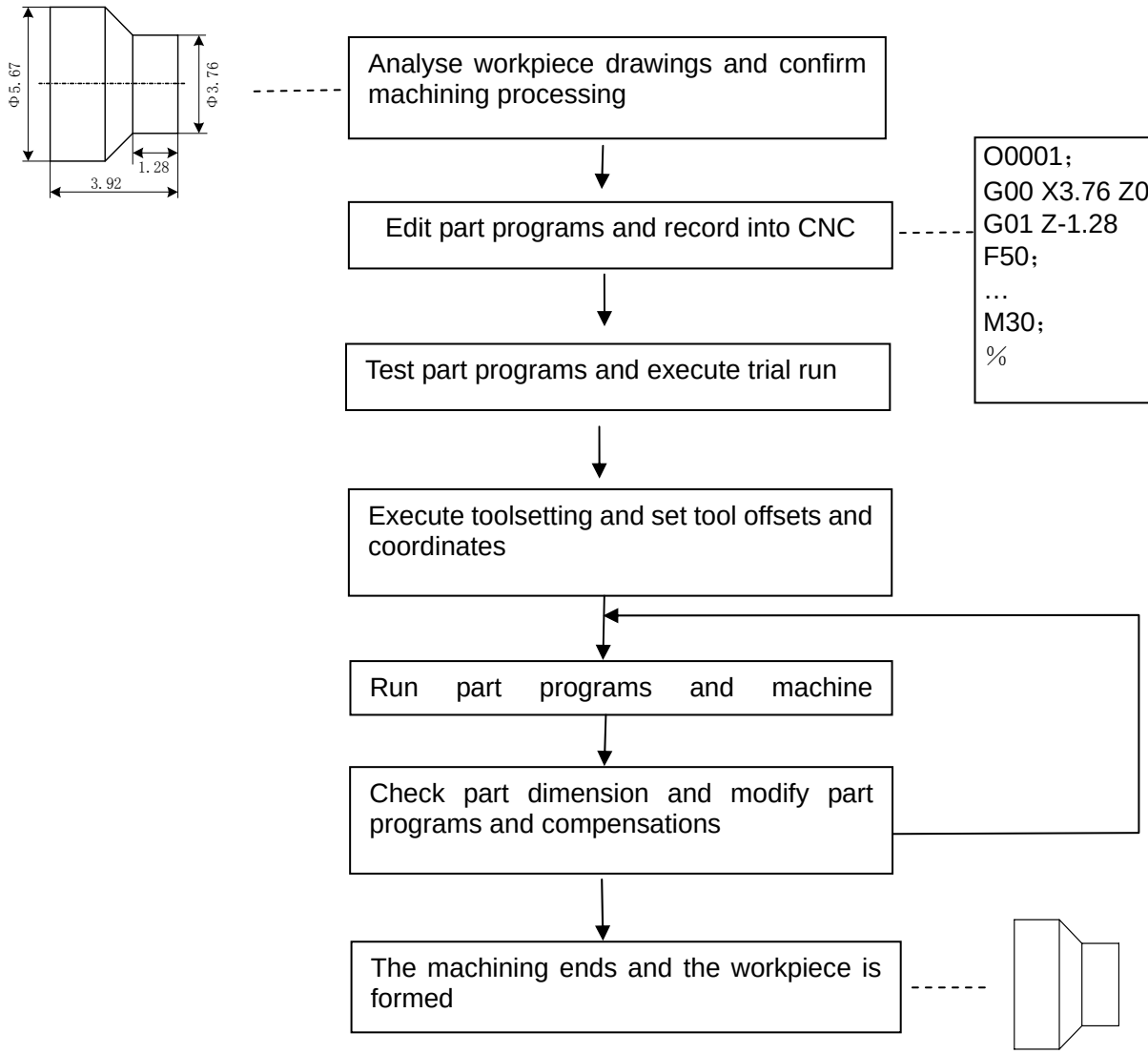


Fig. 1-2

1.3 Programming Fundamentals

1.3.1 Coordinates Definition

Sketch map of CNC turning machine is as follows:

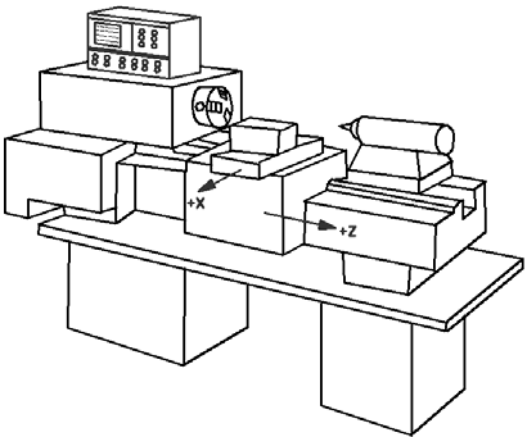


Fig. 1-3

GSK980TDi 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.

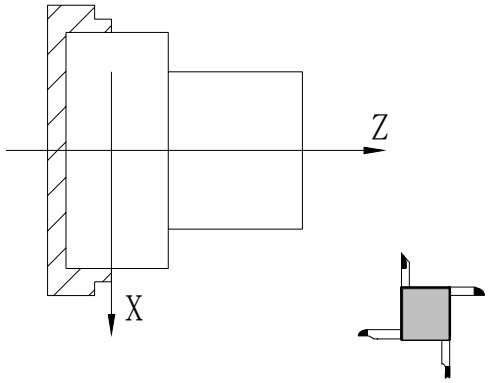


Fig.1-4 Front tool post coordinate system

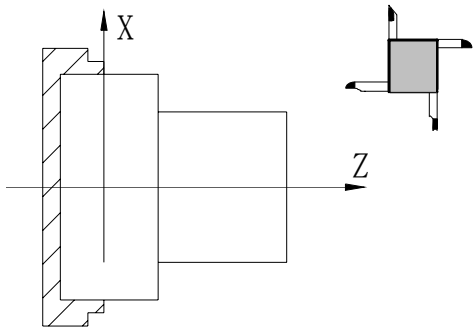


Fig.1-5 Rear tool post coordinate system

1.3.2 Machine Coordinate System, Machine Zero & 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, GSK980TDi 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 tool; otherwise, the motion exceeds the travel limit and the machine to be damaged.

1.3.3 Workpiece Coordinate System & 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.

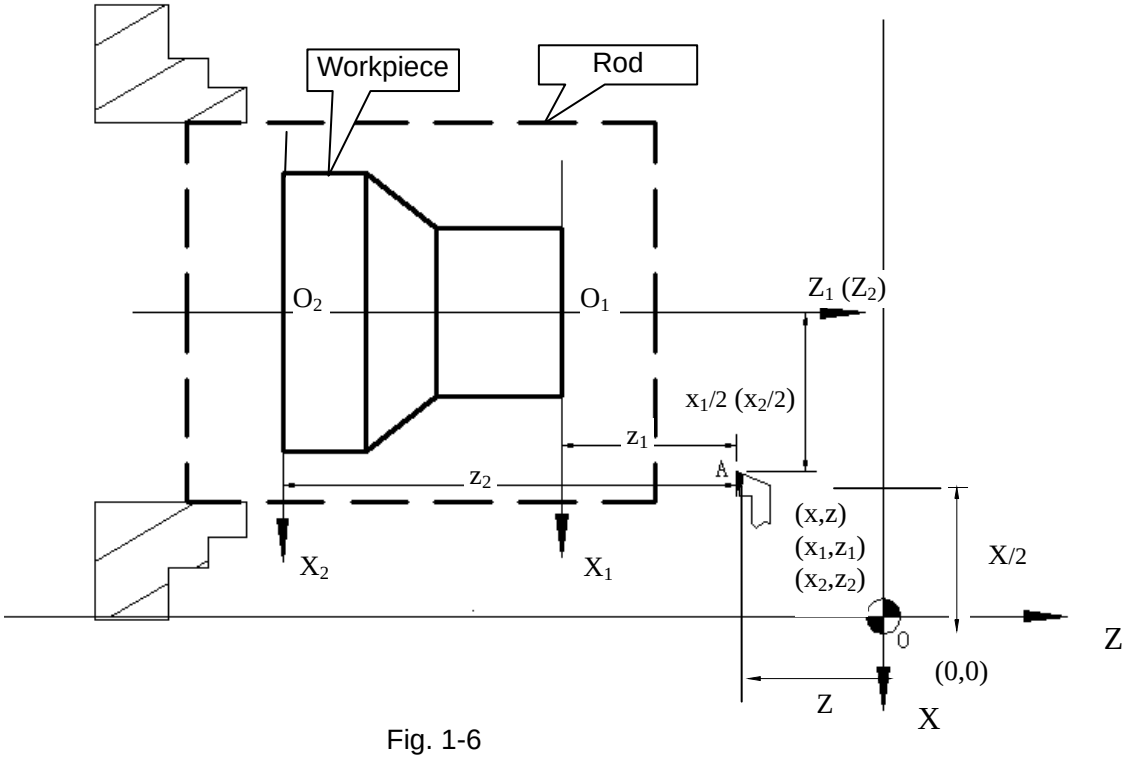


Fig. 1-6

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 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.

GSK980TDi 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:

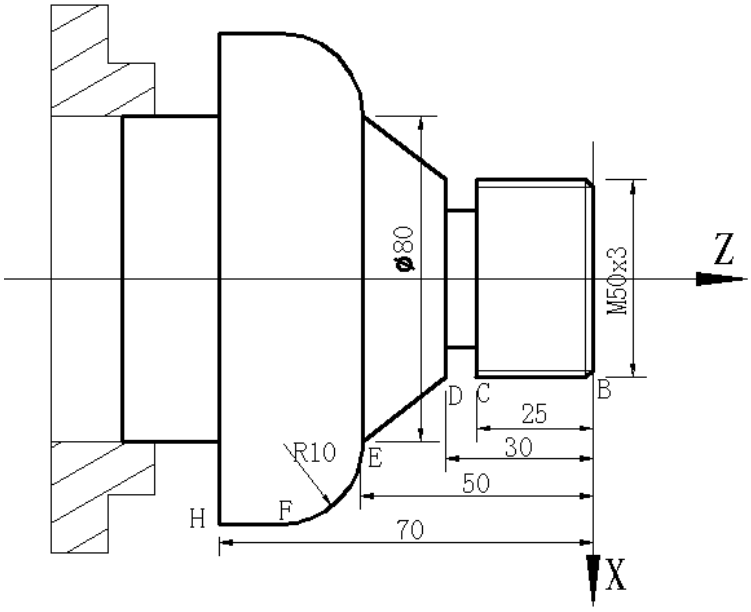


Fig. 1-7

```
…
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 & 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

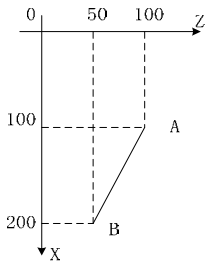


Fig.1-8

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 & 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
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
R	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:

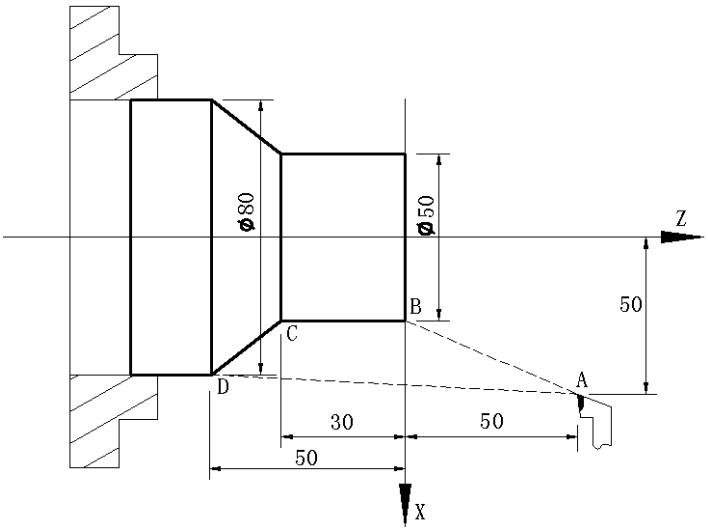


Fig. 1-9

O0001	;	(Program name)
N0005	G0 X100 Z50;	(Rapidly positioning to A point)
N0010	M12;	(Clamping workpiece)
N0015	T0101;	(Changing No.1 tool and executing its offset)
N0020	M3 S600;	(Starting the spindle with 600 r/min)
N0025	M8	(Cooling ON)
N0030	G1 X50 Z0 F600;	(Approaching B point with 600mm/min)
N0040	W-30 F200;	(Cutting from B point to C point)
N0050	X80 W-20 F150;	(Cutting from C point to D point)
N0060	G0 X100 Z50;	(Rapidly retracting to A point)
N0070	T0100;	(Canceling the tool offset)
N0080	M5 S0;	(Stopping the spindle)
N0090	M9;	(Cooling OFF)
N0100	M13;	(Releasing workpiece)
N0110	M30;	(End of program, spindle stopping and Cooling OFF)

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

1.4.1 General Structure of Program

A program consists of blocks. A block begins with a block number (it can be omitted) and several words ending with ";".

Program name

There are most 10000 programs stored in GSK980TDi. 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.

○ □□□□

Program number (0000～9999, the leading zero can be omitted)

Address O

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 GSK980TDi are in Table 1-2.

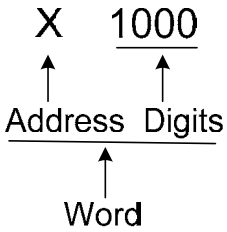


Table 1-2 Word table

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
	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

Address	Command value range	Function meaning	Unit
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
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:

/ N0030 G0 X20 Z30 ;

End of block

Block number

Block skip

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 N_o.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

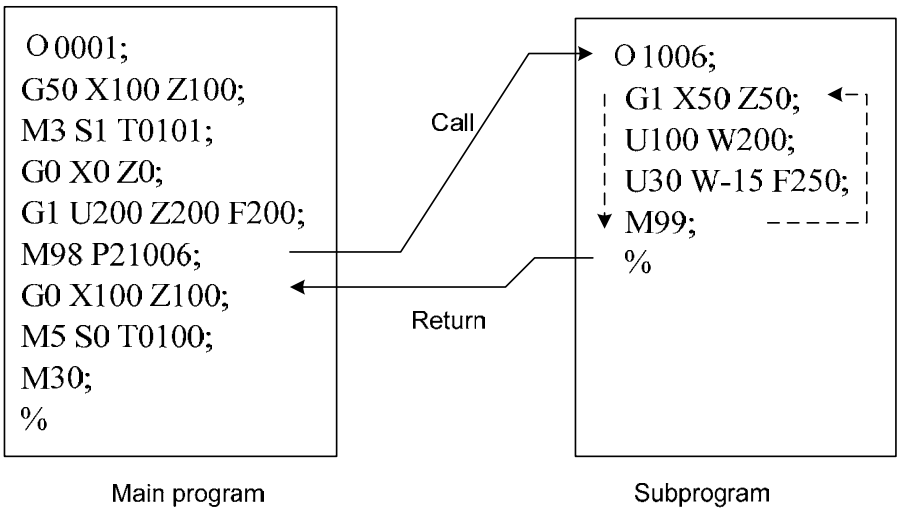
“%” is an ending character of program. “%” is a mark of communication ended when the program is transmitted. The system will automatically insert “%” at the end of program.

Program Note

To make it convenient for the user to search, read the program, the program note can be edited followed each program, and its note locates within the bracket following with the block. Only the English Letters and Numbers can be edited the notes on the CNC; but the Chinese can be performed on the PC. The CNC can be displayed the Chinese note after the program loads to the CNC.

1.4.2 Main Program & 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.



1.5 Program Run

1.5.1 Sequence of Program Run

Running the current open program must be in Auto mode. GSK980TDi 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 button;
- 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 GSK980TDi 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.

1.6 Basic Axis Incremental System

The incremental system includes least input increment (input) and least command increment (output). Least input increment is the least movement unit of programming movement distance, and least command increment is the least unit of tool traversing on the machine. The two increment systems use mm, inch or deg.

The basic axes include X, Z, and their incremental system has IS-B, IS-C selected by No.00.1 ISC.

001							ISC	

Increment 0－IS-B（μ level）

1－IS-C（0.1μ level）

1.6.1 Incremental system speed of basic axis

	μ level（IS-B）		0.1μ level（IS-C）	
	Metric machine (mm/min)	Inch machine (inch/min)	Metric machine (mm/min)	Inch machine (inch/min)
speed	120,000	12,000	120,000	12,000

1.6.2 Incremental System Unit of Basic Axis

In different incremental system and different metric/inch, the least input/output increment is different as follows:

μ level（IS-B）		Least input increment（input）	Least command increment（output）
Metric machine	metric input(G21)	0.001 (mm)	0.001 (mm)
		0.001 (deg)	0.001 (deg)
	inch input(G20)	0.0001 (inch)	0.001 (mm)
		0.001 (deg)	0.001 (deg)
Inch machine	metric input (G21)	0.001 (mm)	0.0001 (inch)
		0.001 (deg)	0.001 (deg)
	inch input (G20)	0.0001 (inch)	0.0001 (inch)
		0.001 (deg)	0.001 (deg)

0.1μ level（IS-C）		Least input increment（input）	Least command increment（output）
Metric machine	metric input (G21)	0.0001 (mm)	0.0001 (mm)
		0.0001 (deg)	0.0001 (deg)
	inch input (G20)	0.00001 (inch)	0.0001 (mm)
		0.0001 (deg)	0.0001 (deg)
Inch machine	metric input (G21)	0.0001 (mm)	0.00001 (inch)
		0.0001 (deg)	0.0001 (deg)
	inch input (G20)	0.00001 (inch)	0.00001 (inch)
		0.0001 (deg)	0.0001 (deg)

Least input increment (input) using the metric or the inch is specified by G20 or G21 or is completed by modifying #001 increment bit.

Least command increment (output) using the metric or the inch is determined by the machine, and is set by#004 output increment.

1.6.3 Incremental System Data Range of Basic Axis

In different incremental system and pulse output frequency, the corresponding different data range is as follows:

Incremental system		Command data input range	Data format
μ level（IS-B）	metric input (G21)	-99999.999 ~ 99999.999 (mm)	5.3
		-99999.999 ~ 99999.999 (deg)	5.3
	inch input (G20)	-9999.9999 ~ 9999.9999 (inch)	4.4

		-9999.999 ~ 9999.999 (deg)	4.3
0.1μ level（IS-C）	metric input (G21)	-9999.9999 ~ 9999.9999 (mm) -9999.9999 ~ 9999.9999 (deg)	4.4 4.4
	inch input (G20)	-999.99999 ~ 999.99999 (inch) -999.9999 ~ 999.9999 (deg)	3.5 3.4

Note : 5.3 in the above table means 5-bit integer and 3-bit decimal. And other data likes these.

1.6.4 Incremental System Data Range & Unit of Basic Axis

- **Speed parameter**

Linear axis speed parameter unit is determined by the machine type. i.e.: metric machine speed unit: mm/min, inch machine is: 0.1inch/min.

Linear axis speed parameter range is determined by the machine type and incremental system type.

Data parameter No. 027(cutting upper speed):

Machine type	Incremental system	Linear axis speed unit	Parameter range	Rotary axis speed unit
Metric machine	μ level（IS-B）	mm/min	10~ 15000	deg/min
	0.1μ level（IS-C）		10~ 15000	
Inch machine	μ level（IS-B）	0.1inch/min	5~58000	
	0.1μ level（IS-C）		5~5800	

Rotary axis does not execute the metric/inch conversion, the rotary axis speed unit is deg/min; the parameter range is the same that of metric machine.

The different incremental system switch may cause the set max. run speed in the data parameter exceeding max. range permitted by the system, so, the operator should modify the speed parameter to avoid the unexpected when the first power-on after switch.

- **Incremental parameter**

The unit and range of linear axis incremental parameter are determined by the machine type and incremental system type as follows:

Machine type	Incremental system	Linear axis incremental unit	Linear axis parameter range
Metric machine	μ level（IS-B）	0.001mm	-99, 999.999~ 99,999.999
	0.1μ level（IS-C）	0.0001 mm	-9, 999.9999~ 9,999.9999
Inch machine	μ level（IS-B）	0.0001inch	-9, 999.9999~ 9,999.9999
	0.1μ level（IS-C）	0.00001 inch	-999.99999~ 999.99999

The rotary axis does not execute the metric/inch switch, the unit of rotary axis incremental parameter is determined by the incremental system type. The range of rotary axis incremental parameter is the same that of the metric machine.

Machine type	Incremental system	Rotary axis speed unit	Rotary axis parameter range
Metric, Inch machine	μ level（IS-B）	0.001°	0~ 99999.999
	0.1 μ level（IS-C）	0.0001°	0~ 9999.9999

- **Programming value input range**

Linear axis coordinate data unit is determined by the metric/inch input system. i.e. metric input is mm; inch input is inch;

Linear axis coordinate data range is determined by metric/inch input system and incremental system. The range is the same that of the command data input range as follows:

Incremental system		Linear axis coordinate data range
μ level（IS-B）	Metric input (G21)	-99999.999 ~ 99999.999(mm)
	Inch input (G20)	-9999.9999 ~ 9999.9999(inch)
0.1μ level（IS-C）	Metric input (G21)	-9999.9999 ~ 9999.9999(mm)
	Inch input (G20)	-999.99999 ~ 999.99999(inch)

The rotary axis does not execute the metric/inch switch, the rotary axis coordinate data unit is deg. The range of programmed value is shown below:

Input type	Incremental system	Rotary axis coordinate data range
Metric, inch input	μ level（IS-B）	-99999.999 ~ 99999.999（deg）
	0.1μ level（IS-C）	-9999.9999 ~ 9999.9999(deg)

- **Offset data**

Offset data unit is determined by metric/inch input system. i.e. metric input is mm; inch input is inch.

Offset data range is limited to 9999999 and is determined by the metric/inch input system and incremental system as follows:

Input type	Incremental system	Offset data unit	Offset data range
Metric input(G21)	μ level（IS-B）	mm	±9999.999
	0.1μ level（IS-C）		±999.9999
Inch input (G20)	μ level（IS-B）	inch	±999.9999
	0.1μ level（IS-C）		±99.99999

- **Pitch data compensation**

Linear axis pitch compensation unit and range are determined by the machine type and incremental system type as follows：

Machine type	Incremental system	Linear axis data unit	Linear axis pitch compensation data range
Metric machine	μ level（IS-B）	0.001mm	-2550～2550
	0.1μ level（IS-C）	0.0001mm	-2550～2550
Inch machine	μ level（IS-B）	0.0001inch	-2550～2550
	0.1μ level（IS-C）	0.00001inch	-2550～2550

The rotary axis does not execute the metric/inch switch; the rotary axis pitch compensation unit is determined by the least input unit as follows.

Machine type	Incremental system	Rotary axis pitch compensation unit	Rotary axis pitch compensation range
Metric, inch machine	μ level（IS-B）	0.001°	0～255
	0.1μ level（IS-C）	0.0001°	0～2550

1.6.5 Program Address Value Unit & Range of Incremental System of Basic Axis

- **Pitch definition and range:**

	Command	μ level （IS-B）	0.1μ level（IS-C）	Unit
Metric input （G21）	F	0.001~500.000	0.0001~500.00	mm/tooth[lead]
	I	0.06~25400	0.06~2540	tooth[lead]/inch
Inch input（G20）	F	0.0001~50.00	0.00001~50.0	inch/tooth[lead]
	I	0.06~2540	0.06~254	tooth[lead]/inch

- **Speed F definition**

G98 m/min: feed per minute: F unit: mm/min

G99 feed per rev: F definition and range are as follows:

Incremental system	Address	Metric input	Inch input
IS-B system	F（G98）	0～99999.999 (mm/min)	0～99999.999 (mm/min)
IS-C system		0～9999.9999 (mm/min)	0～9999.9999 (mm/min)
IS-B system	F（G99）	0～500 (mm/r)	0～500 (mm/r)
IS-C system		0～50 (mm/r)	0～50 (mm/r)

1.7 Additional Axis Incremental System

Least incremental system in μ level（IS-B） or 0.1μ level（IS-C）, the additional axis does not execute the link, and is not used alone. When the least incremental output of additional axis is 0.01 in the low precision requirement and the feedrate must be fast, and so the working efficiency largely increases. The least incremental system of additional axis does not sometime consist with the current least incremental system. The system adds the optional function of the least incremental system of the additional axis (Y, 4th , 5th axis) .

Additional axis incremental system is set by No..187 as follows:

187	nIS1	nIS0						

nIS1, nIS0：select the least incremental system of each additional axis (n means the axis name of each additional axis)

nIS1	nIS0	Incremental system of axis	Least input/output
0	0	Same with current incremental system of basic axis (XY)	
0	1	IS-A	0.01
1	0	IS-B	0.001
1	1	IS-C	0.0001

Note: The least I/O in the above table is expressed without considering the metric/inch and rotary axis.

1.7.1 Additional Axis of Current Incremental System

IS-B or IS-C: The relative speed and data range of additional axis is the same that the described in Section 1.6.

1.7.2 Additional Axis of IS-A Incremental System

IS-A: The max. speed of additional axis is separate 10 times and 100 times of IS-B and IS-C. The relative data and parameter range are same those of the incremental system of current basic axis (refer to Section 1.6).

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.

M □□□□

Command value (00~99, 9000~9999, the leading zero can be omitted)

Command address

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

M02, M03 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
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)

2.1.1 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.

2.1.2 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.

2.1.3 Subprogram Call M98

Command format:

M98 P○○○○□□□□

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.

Command function: In Auto mode, after other commands are executed in M98, CNC calls subprograms specified by P, and subprograms are executed 9999 times at most. M98 is invalid in MDI mode.

2.1.4 Return From Subprogram M99

Command format: M99 P○○○○

Executed block after returning to the main program is 0000～9999，and its leading zero can be omitted.

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.

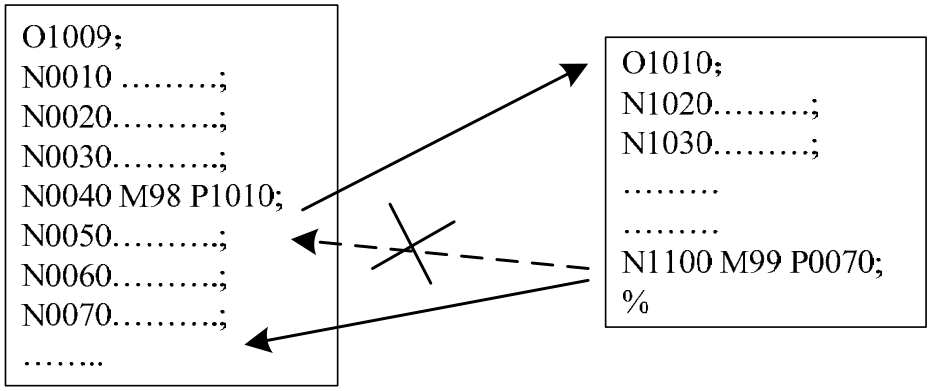


Fig. 2-1

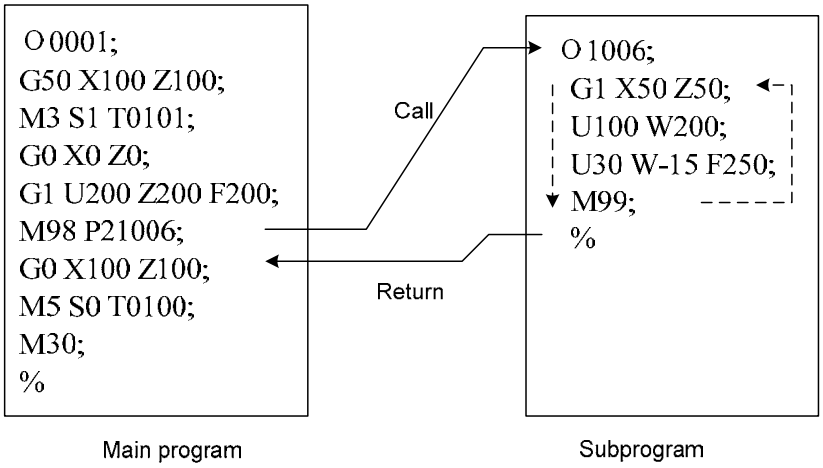


Fig. 2-2

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

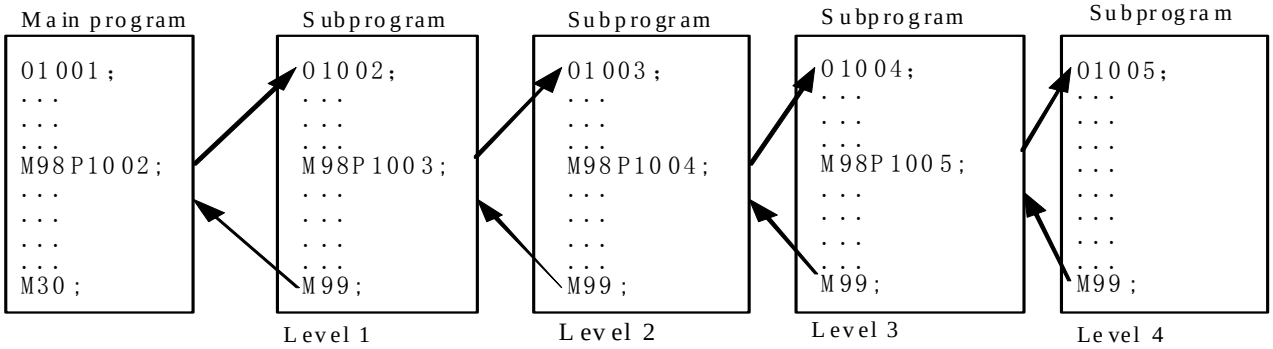


Fig. 2-3 Subprogram nesting

2.1.5 Macro Program Call M9000～M9999

Command format: M□□□□

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.

2.1.6 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 GSK980TDi 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.

Table 2-2 M commands

Command	Function	Remark
M00	Program pause	
M01	Program optional stop	
M03	Spindle clockwise (CW)	Functions interlocked and states reserved
M04	Spindle counterclockwise (CCW)	
*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 2nd spindle position control	Functions interlocked and states reserved
*M25	The 2nd spindle speed control	
M32	Lubricating ON	Functions interlocked and states reserved
*M33	Lubricating OFF	
*M50	Spindle orientation cancel	Functions interlocked and states reserved
M51	Spindle orientate to No. 1 point	
M52	Spindle orientate to No. 2 point	
M53	Spindle orientate to No. 3 point	
M54	Spindle orientate to No. 4 point	
M55	Spindle orientate to No. 5 point	
M56	Spindle orientate to No. 6 point	
M57	Spindle orientate to No. 7 point	
M58	Spindle orientate to No. 8 point	
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

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

2.1.7 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.

2.1.8 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.

2.1.9 Spindle CW, CCW & 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.

2.1.10 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.

2.1.11 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

2.1.12 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.

2.1.13 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 & CONNECTION.

2.1.14 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.

2.1.15 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.

2.1.16 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.

2.1.17 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.

2.1.18 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.

2.1.19 The 2nd Spindle Rotation CCW/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 GSK980TDi 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.

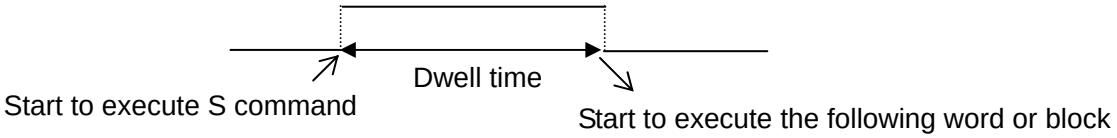
Their executing sequence is defined by PLC when S command and word for moving function are in the same block. Please refer to *User Manual* from machine manufacturer.

When spindle speed is controlled by switching value, GSK980TDi 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 GSK980TDi 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 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 2.2.4**.**

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.

2.2.3 Spindle Speed Series Control

When the Bit 4 of the state parameter No. 001 sets to 1, and the Bi7 of the No. 195 sets to 1, the spindle speed is the series control.

Command format:S □□□□

└── 0000～9999 (the leading zero can be omitted.): Spindle speed series control

Command function: The spindle speed is defined, CNC directly controls the spindle servo drive unit by delivering the series data based upon the bus to carry out the stepless speed

of the spindle. 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 2.2.4**.**

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.

2.2.4 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 verse, 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 × |X| × π ÷1000 (m/min)

Spindle speed: r/min

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

π ≈3.14

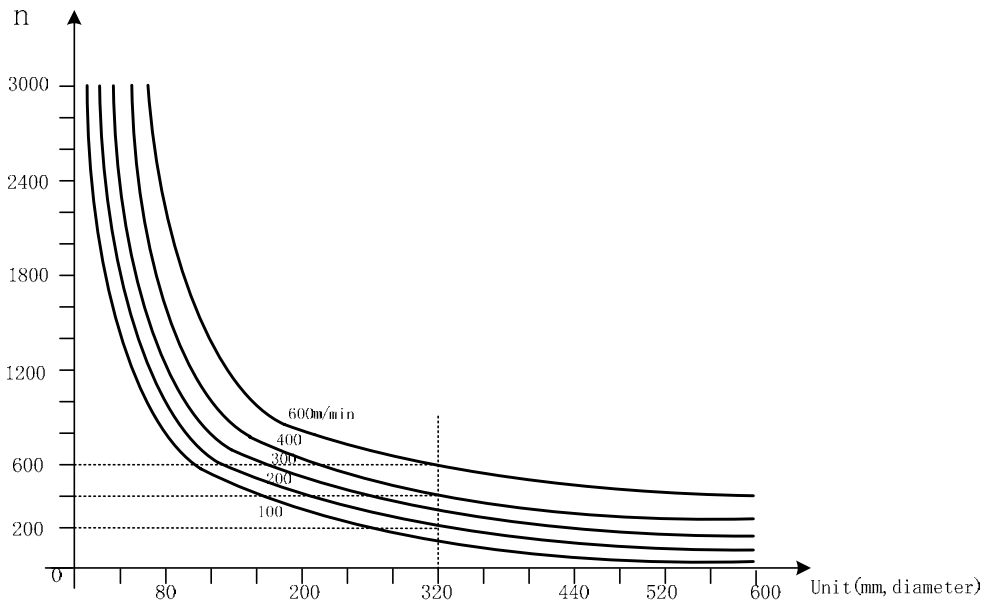


Fig. 2-4

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:

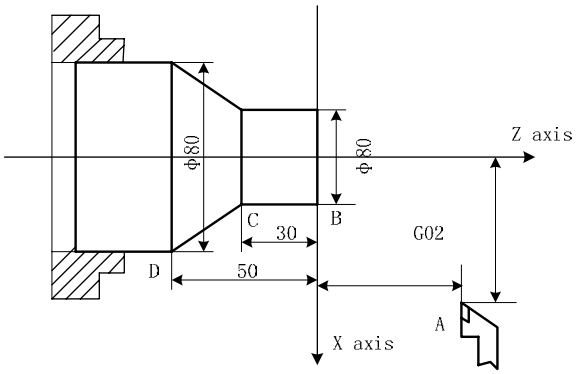


Fig. 2-5

```
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.

2.2.5 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 restricted 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 GSK980TDi standard PLC ladder.

The spindle actual speed specified by GSK980TDi 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**.

2.2.6 Multiple Spindle Control Function

GSK980TDi 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 GSK980TDi 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.

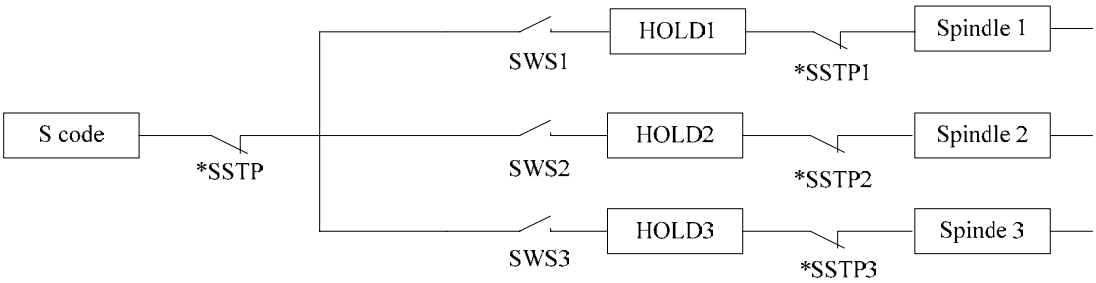
When №001＃4 is set to 1, setting MSEN（№196＃4）to 1 can start the multiple spindle control function.

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.

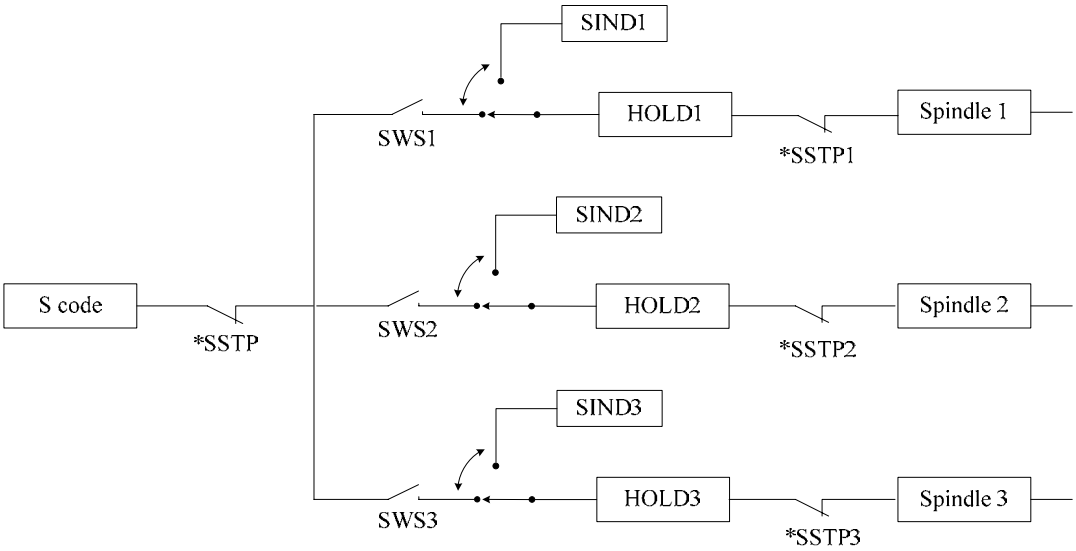


- Multiple spindle control method B

Each spindle has separate SIND signal.

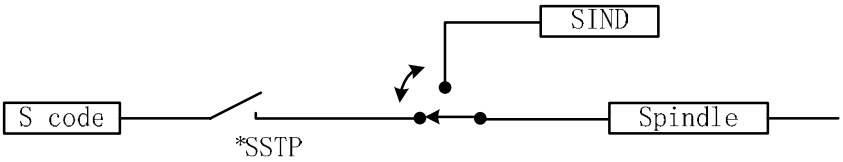
When the spindle selection signal, the 1st spindle or the 2nd spindle SIND signal is set to 1, SIND signals separately determine each spindle to be controlled by PLC or CNC.

Multiple spindle control method B is shown below.



- Multiple spindle control function being invalid

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



2.2.7 Cs Contour Control Funciton

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.

2.2.8 Spindle Positioning Function

In the turning machining, the spindle rotates at one certain speed, therefore, the workpiece installed on the spindle revolved and machined. In this case, the spindle control state is called “Spindle Rotation Mode”. Compare to the others, moves one angle for the spindle, so that the workpiece installed on the spindle positions at one angle, which is regarded as “Spindle Positioning Function”. In this case, the controllable state of the spindle is called “Spindle Positioning Mode”. Generally, the spindle positioning axis is mechanically clamped to use after the positioning operation is performed, and therefore, the relation operations between the spindle clamping and releasing may be performed before/after the positioning.

The spindle positioning function can be devided into 3 steps based upon the operation:

- Release the spindle rotation method, shift to the spindle positioning mode

  Shift the spindle to its positioning method by executing the specific M code (Parameter setting), and then set up a reference position.

- The positioning spindle operation in the spindle positioning method.

  Spindle positioning can be performed the position of arbitrary angle by axis address, which can be used the absolute and relative commands. It is better not to perform with the shared block other than the G00 or G50 command.

  For example, suppose the controllable axis is C axis:

  G00 C45.000

  G00 H-60.000

- Release the spindle positioning method, shift to the operation of the spindle rotation method.

  Shift the spindle to the spindle rotation method by performing the specific M code (Parameter Setting).

Compare the spindle positioning function with the Cs axis contour controllable function, regardless of the linkage interpolation with other axes; generally, the mechanical equipment clamping can be used when the position is completed.

Especially note, the spindle positioning function and Cs axis contour controllable function can not be simultaneously used.

2.3 Tool Function

2.3.1 Tool Control

T functions of GSK980TDi: 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.

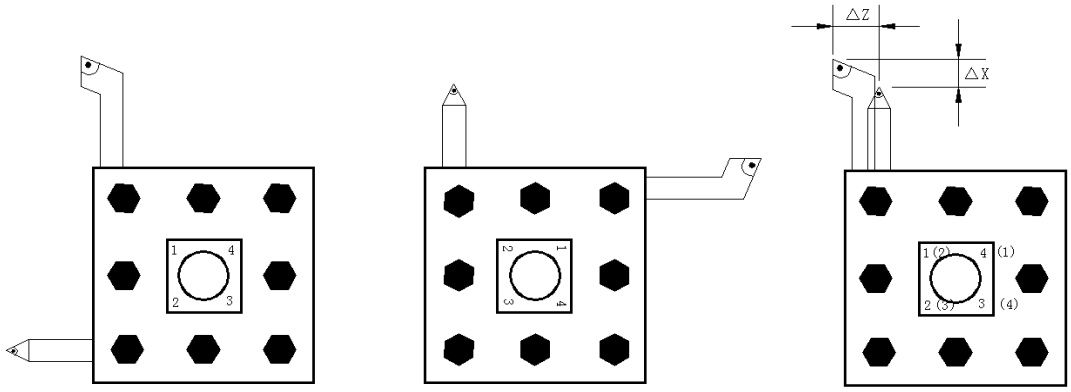


Fig.2-6 tool offset

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.

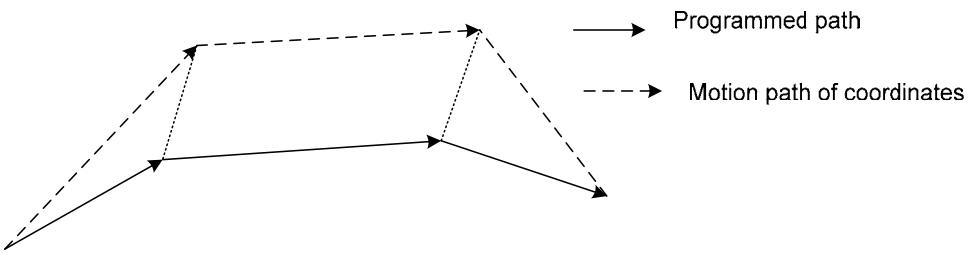


Fig. 2-7 Creation, execution and cancellation of tool length compensation

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

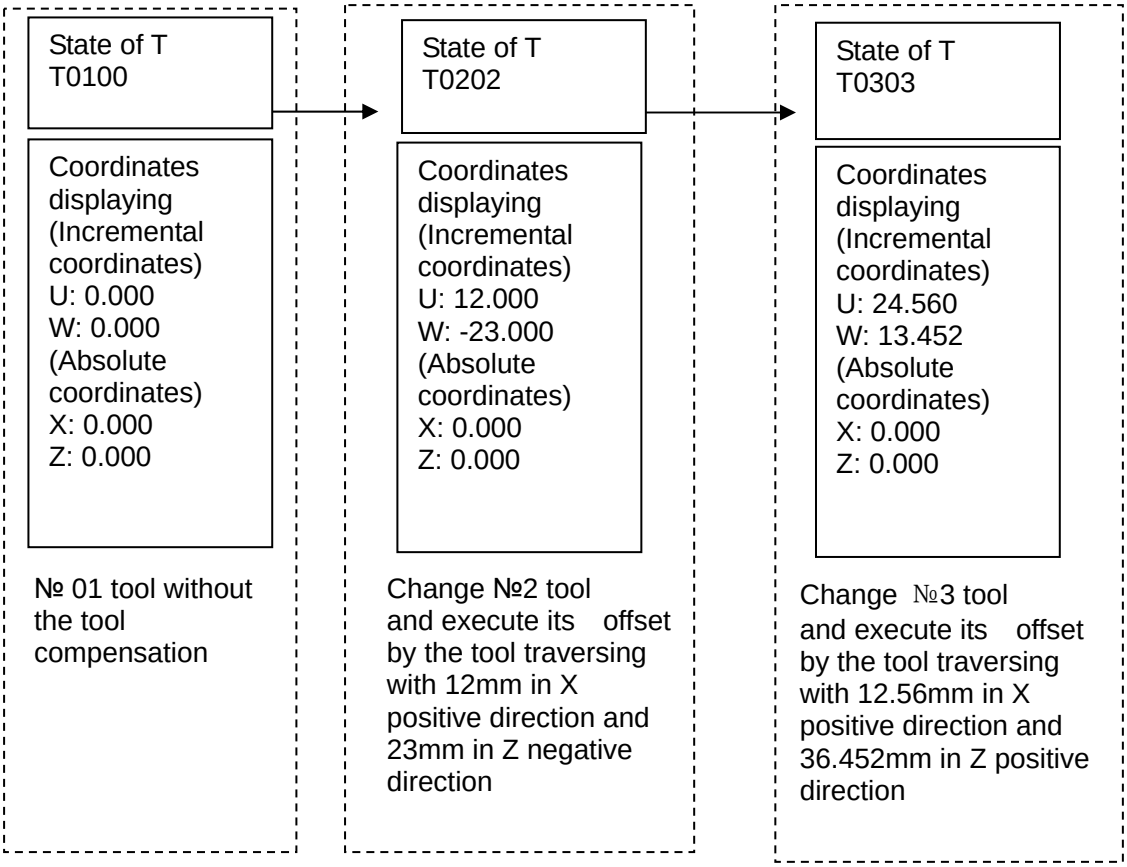


Fig. 2-8 Tool traversing mode

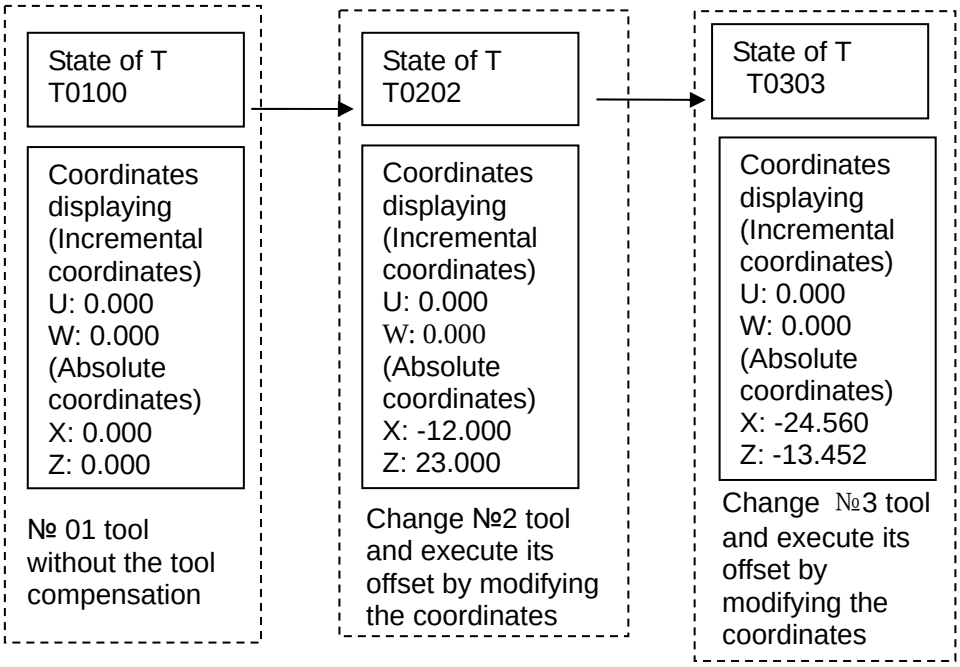
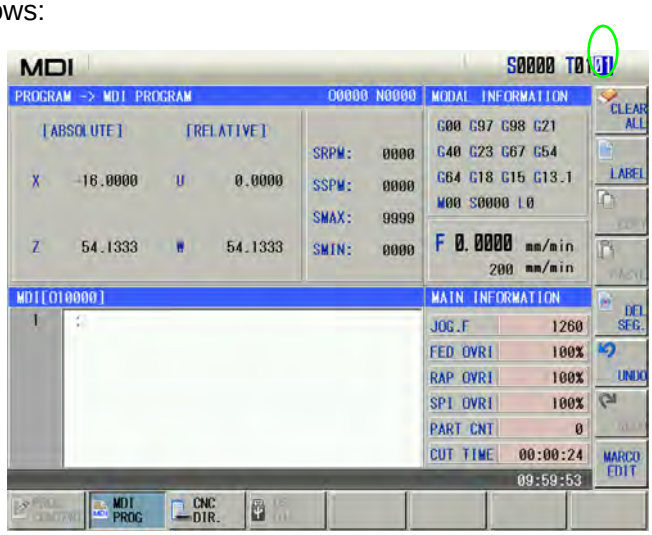


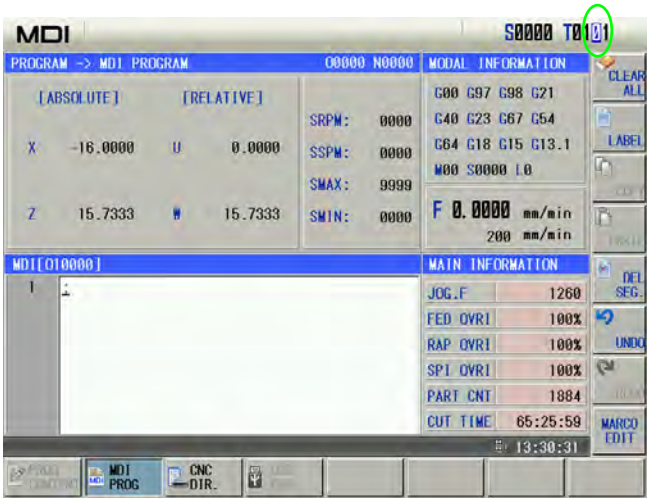
Fig. 2-9 Modifying the coordinates mode

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.2-8 and Fig.2-9). 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.

Example: When No.003 Bit4 is 1 and a sole T0101 is executed, the system displays after 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:

1. Execute T□□00 command;
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. GSK980TDi 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

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

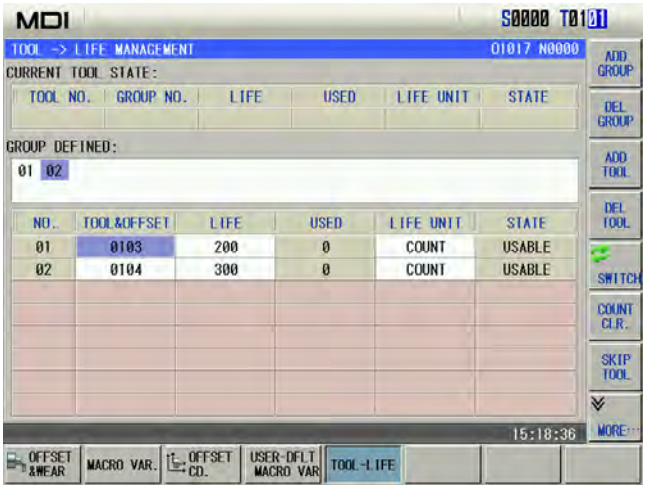
Bit 0 Tool life management function is not valid.
1 Tool life management function is valid.

2. Tool life management display window

Press repetitively [OFFSET] into the tool life management display window [TOOL-LIFE].

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 [COUNT CLR.] 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) . Press [UNIT: COUNT], [UNIT: MIN] to change the count unit of the tool.

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

Press [SKIP TOOL] to change the tool state to skip when the tool is not used in the current window.

Defined group: It only display defined group number and the undefined group number is not displayed. The displayed group back lighted means that all tool life in this group is over.

2) Creating and displaying tool group number

A. In tool group state display window, press [ADD GROUP], group number, [DATA INPUT] 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, GSK980TDi 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;		

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 doest 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).

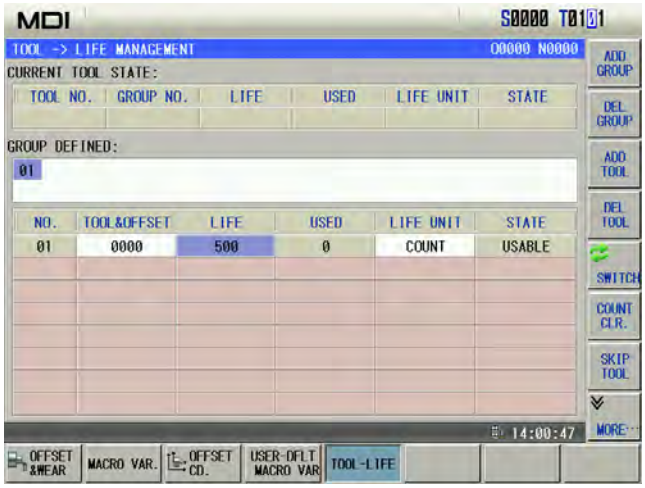
Note 6: All tool life data is stored when power-down.

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 SWITCH to move the tool to the life position, input directly data, such as 500, press DATA INPUT to confirm the input. Press ← → ↓ ↑ to complete data input (tool offset, life) .

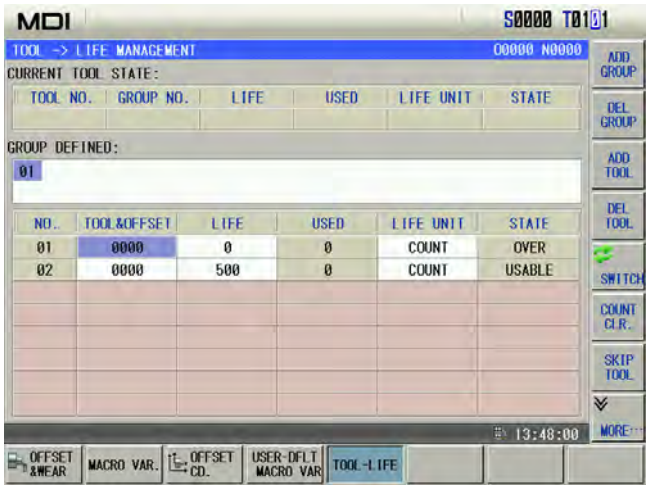


B. Inserting data:

Insert any serial number in the current page, press ADD TOOL——>[01～08]——> DATA INPUT, 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

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



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

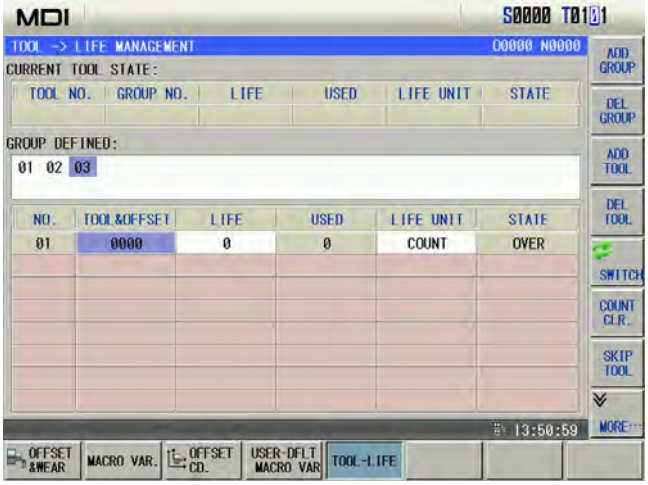
c) Insert behind

C. Delete data:

a) Delete data in all groups.

In tool compensation-> tool life window, press

to delete all defined data.

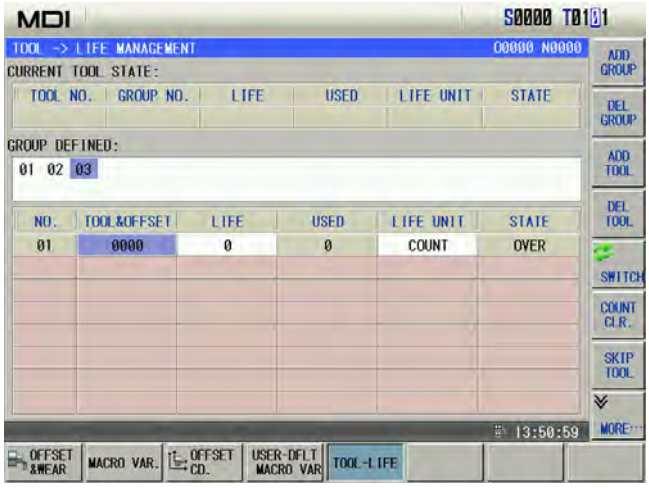


Before operation

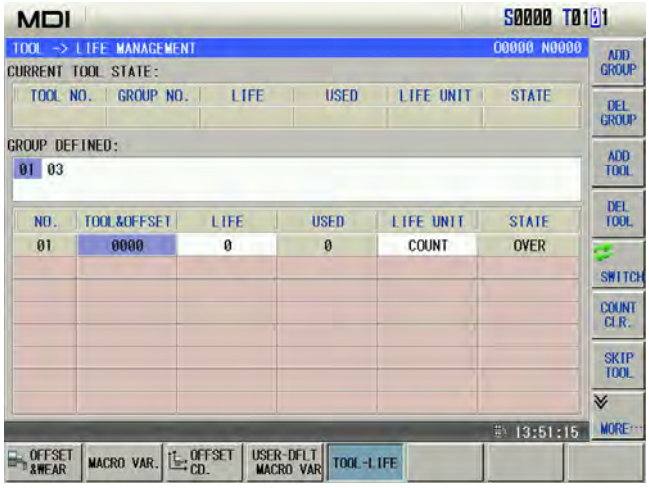
After operation

b) Delete data in any group

Press DEL GROUP ——>input serial number［02］——> DATA INPUT;



Before operation



After deletion

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

Press DEL TOOL ——>input serial number［04］——> DATA INPUT

Before deletion

After deletion

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						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.

1). 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.

2). Number of times count

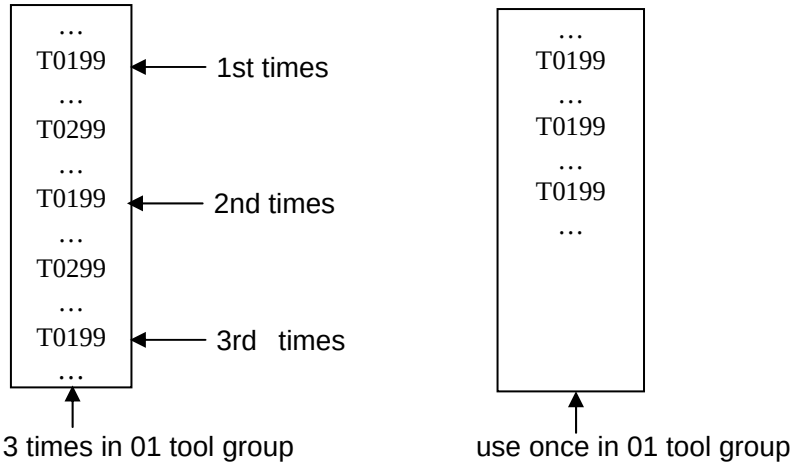
There are two methods about the number of times count which is determined by No.002 Bit2 (LIFC).

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

1)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.

2)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.

3)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.

4)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:

a） 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.

b）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.

5)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 □□

Command value（00～99，the leading zero can be omitted）

Command address G

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	01	Rapid traverse movement	Initial modal G command
G01		Linear interpolation	Modal G commands
G02		Circular interpolation(CW)	
G03		Circular interpolation(CCW)	
G05(G5.1)		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	
G83		axle drilling cycle	
G84		End rigid tapping	
G87		radial drilling cycle	
G88		Side rigid tapping	

G94		Radial cutting cycle	
G04	00	Dwell time preset	Non-modal G commands
G7.1		Cylinder interpolation	
G10		Data input	
G11		Data input cancel	
G28		Machine reference point automatic return	
G30		Machine 2nd, 3rd , 4th reference point return	
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	
G78		Enhancement 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	07	Cancel cutter radius compensation	Initial mode G command
G41		Tool nose radius compensation left contour (option)	Modal G command
G42		Tool nose radius compensation right contour(option)	
G22	09	Open the stored stroke limit detection 2	Non-modal G command
G23		Close the stored stroke limit detection 2	
G66	12	Modal call	Modal G command
G67		Modal call cancel	
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	
G09		Exact stop between blocks	Non-modal G command

G61		Exact stop begins between blocks	
G64		Cancel the exact stop between blocks	
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, One-Shot & 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 one-shot 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.

Table 3-2

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

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: ±99999999× 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:

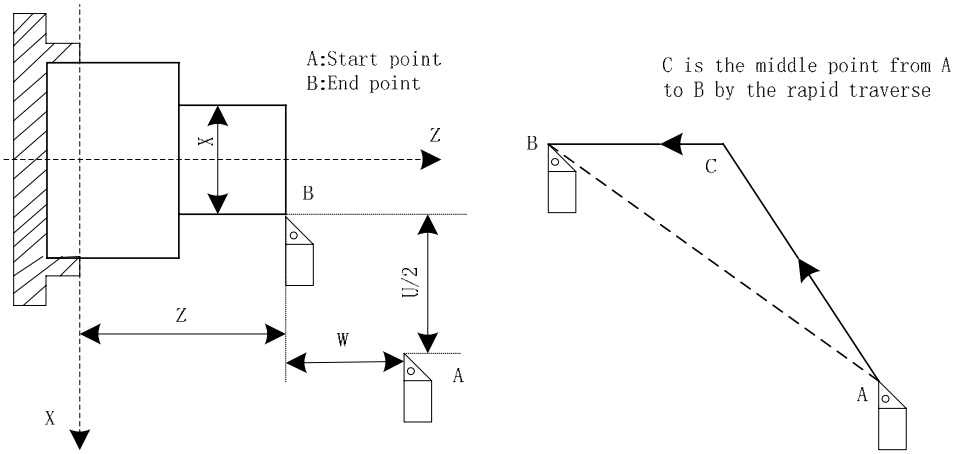


Fig. 3-1

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

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

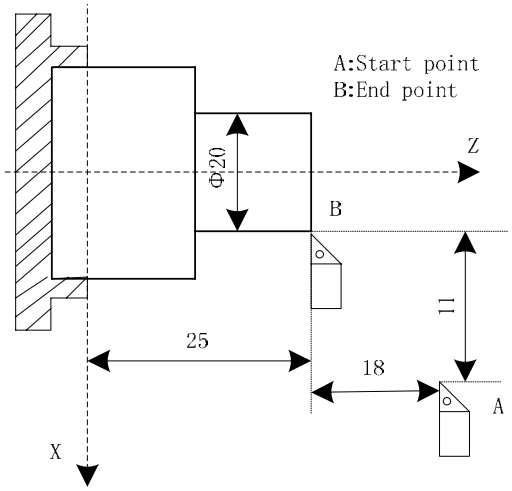


Fig. 3-2

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 rang 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 of Section 1.4.1.

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:

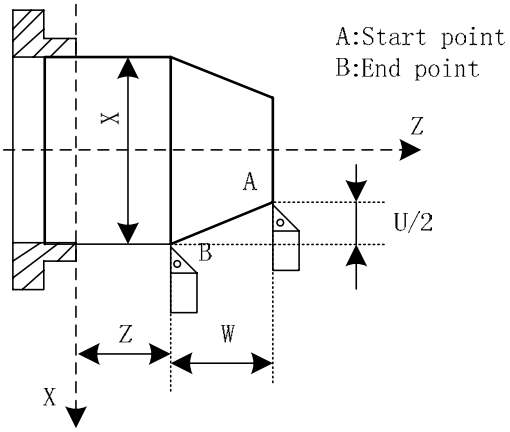


Fig. 3-3

Example: Cutting path from Φ40 to Φ60 as Fig.3-4:

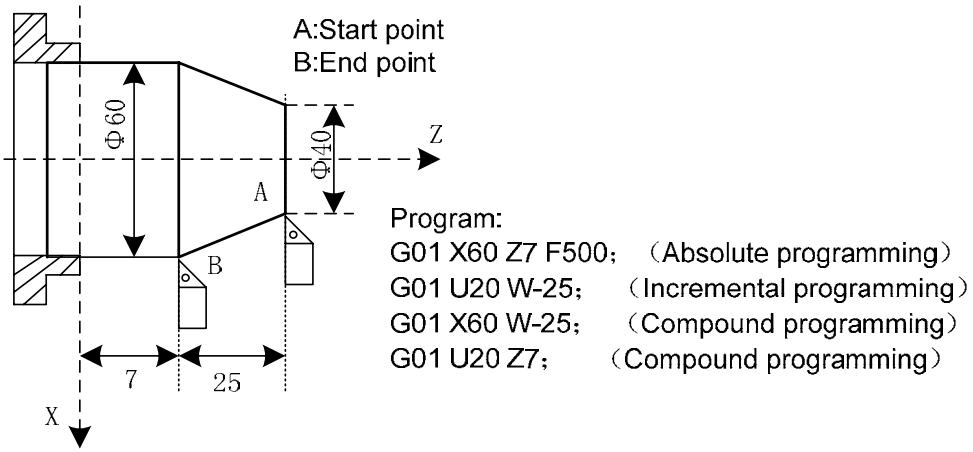


Fig. 3-4

3.4 Circular Interpolation G02, G03

Command format:

$$\left.\begin{matrix}G02\\G03\end{matrix}\right\}X(U)__\quad Z(W)__\quad\left\{\begin{matrix}R__\\I__\ K__\end{matrix}\right.$$

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:

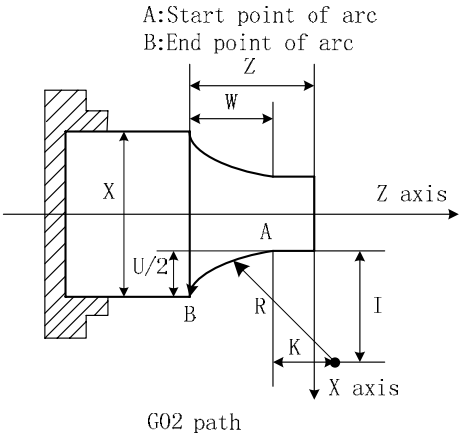


Fig. 3-5 G02 path

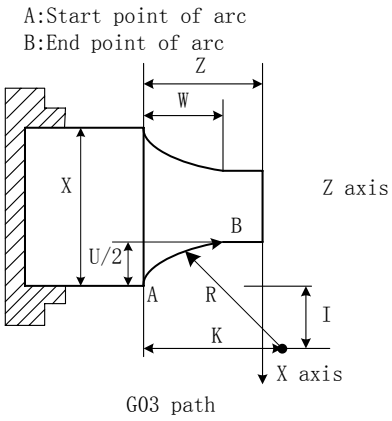


Fig.3-6 G03 path

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 in radius;

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.

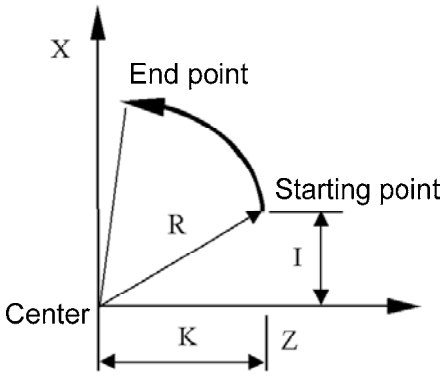


Fig. 3-7

Arc direction: G02/G03 direction (clockwise/counterclockwise) is opposite on the front tool post coordinate system and the rear one as Fig.3-8:

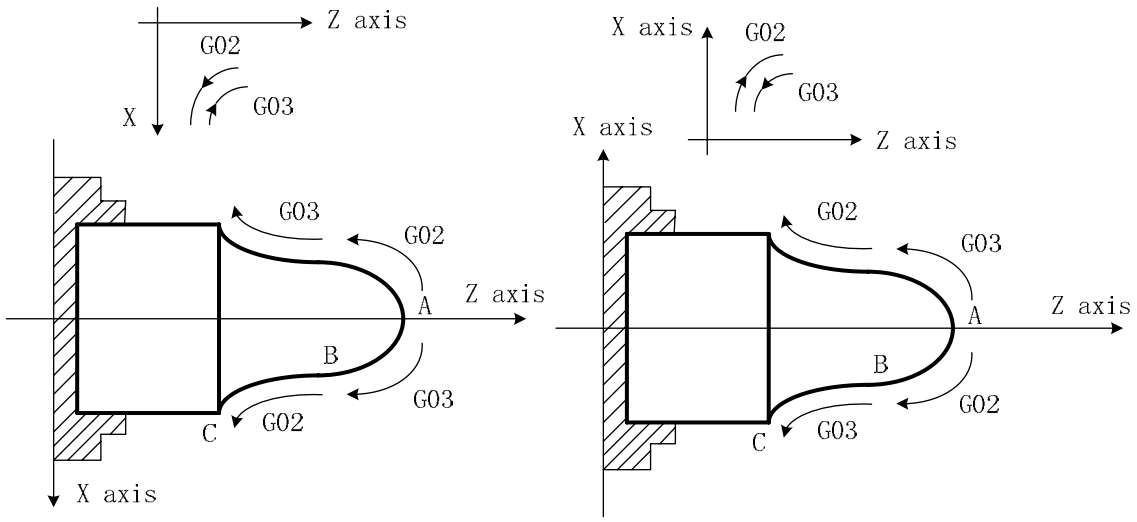
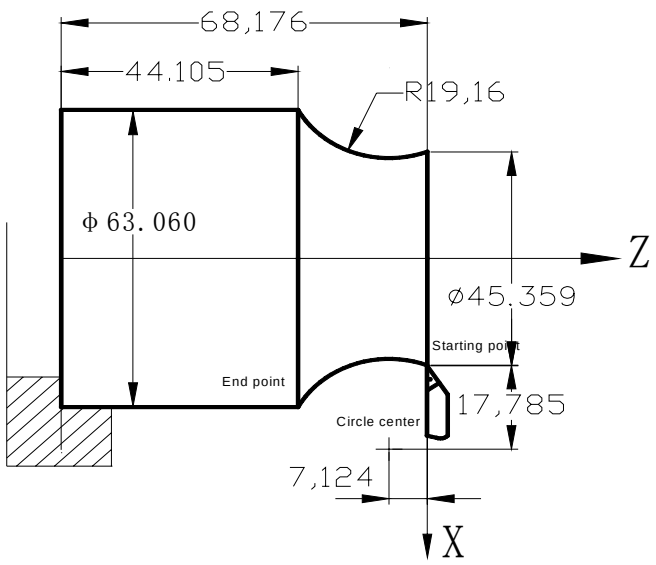


Fig. 3-8

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 Φ45.25 to Φ63.06 shown in Fig. 3-9.



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. 3-9

Compound programming in G02/G03:

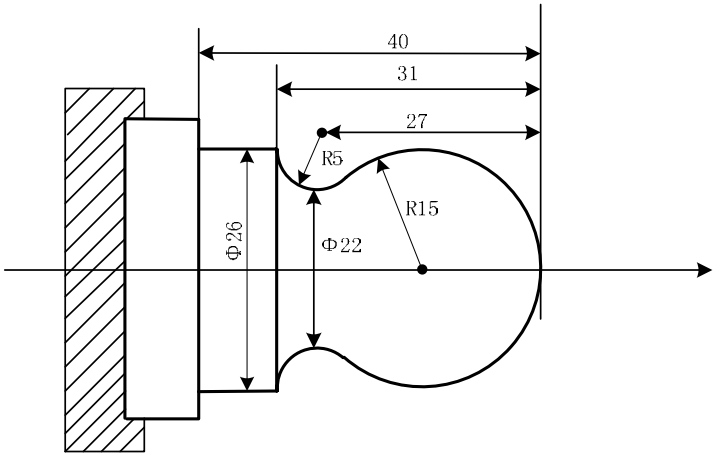


Fig. 3-10 Circular programming example

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)

3.5 Three-point Circular Interpolation G05, G05.1

Command format: G05(G05.1) X(U)__ Z(W)__ I__ K__ F__

Command function: when you do not know positions of the circular center of arc and the radius but coordinates of three points on the arc, you can use G5 to confirm the arc direction through the middle position between the initial point and end point.

Command specification: G05(G05.1) is modal;

I: incremental coordinate value(X) (radius value, direction) of the middle point where the circular passes corresponding to the starting point, range: ±99999999× least input increment;

K: incremental coordinate value(Z, direction) of the middle point where the circular passes corresponding to the starting point, range: ±99999999× least input increment;

X, U, Z, W, R, I, K range is referred to Table 1-2 of Section 1.4.1, unit: mm/inch.

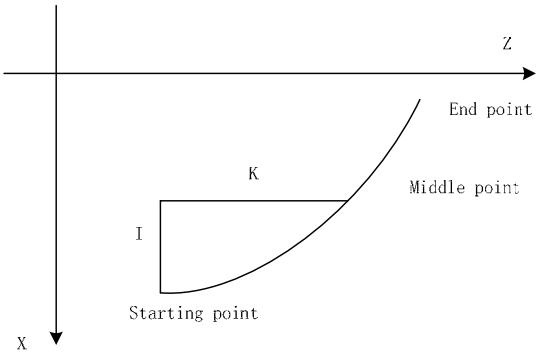


Fig. 3-11 G05 code Middle point

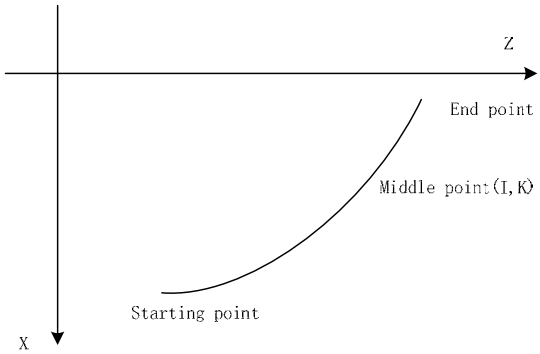
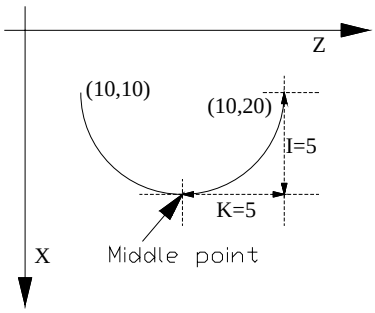


Fig.3-12 G05.1 code Middle point

Notes:

1) Middle point: any one point except for the starting point and end point on the arc;
2) The system alarms when the three points are in the same line;
3) I=0 when it is omitted, K=0 when it is omitted; the system alarm when they are omitted simultaneously;
4) The meanings of I, K I are similar to their displacement values of the circle center coordinates corresponding to starting point ones in G02/G03;
5) G05 does not execute the whole circle machining;

Example: (machining semi-circle)



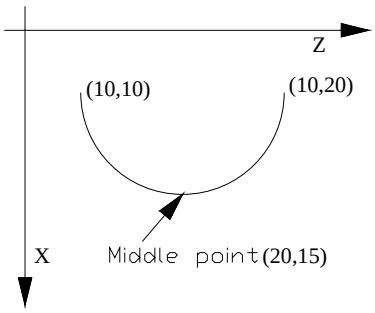


Fig.3-13 G05

Program 1

G00 X10 Z20;

G5.1 X10 Z10 I5 K-5 F150;

Fig.3-14 G05.1

Program 2

G00 X10 Z20;

G05.1 X10 Z10 I20 K15;

3.6 Ellipse Interpolation G6.2, G6.3

Command format: G6.2 / G6.3 } X(U)__ Z(W)__ A__ B__ Q__

Command function: G6.2 motion path is the CW(rear tool post coordinates)/CCW(front tool post coordinates) ellipse.
G6.3 motion path is the CCW(rear tool post coordinates)/CW(front tool post coordinates) ellipse.

Command path:

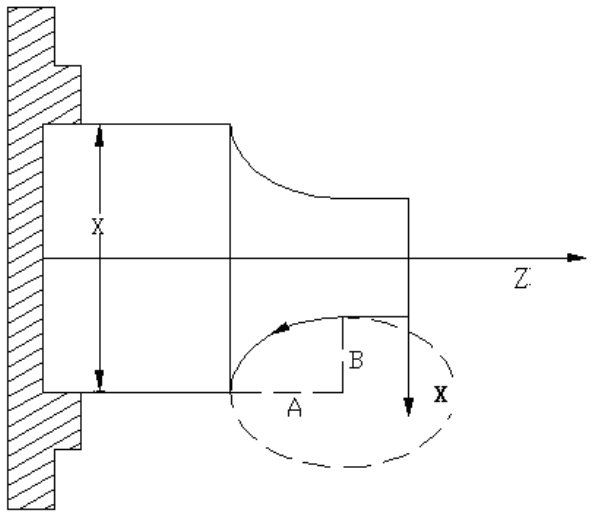


Fig. 3-15 G6.2 path sketch map

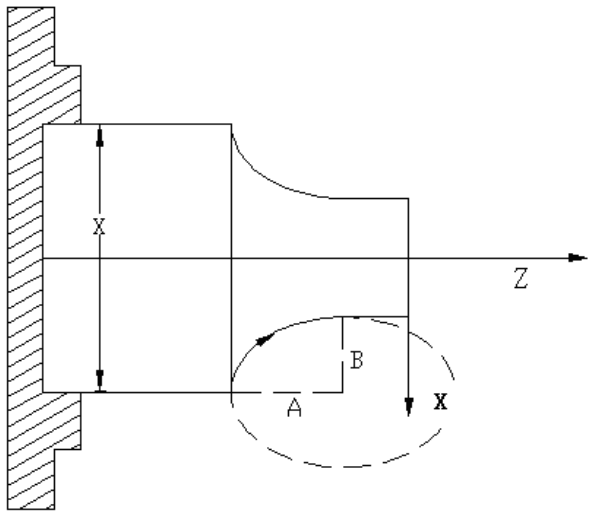


Fig. 3-16 G6.3 path sketch map

Explanation:

A: Length of ellipse's long radius (0<A<= 99999.999mm, no sign)

B: Length of ellipse's short radius (0<B<= 99999.999mm, no sign)

Q: angle between the long axis and Z coordinate system of ellipse (CCW 0-99999999, unit: 0.001°)

Ellipse direction: defined by that of G6.2/G6.3, they are the reverse in the front and rear tool post coordinates.

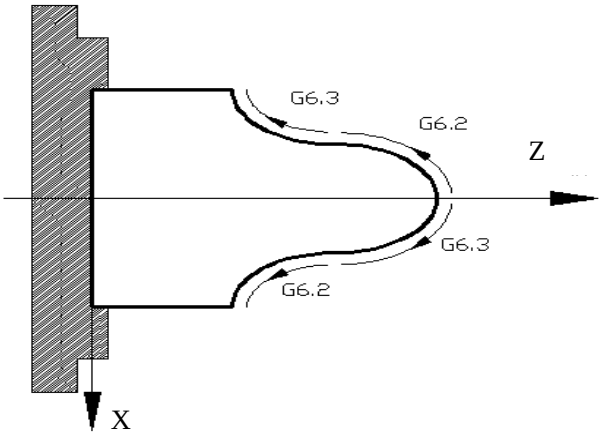


Fig. 3-17 Front tool post coordinate system

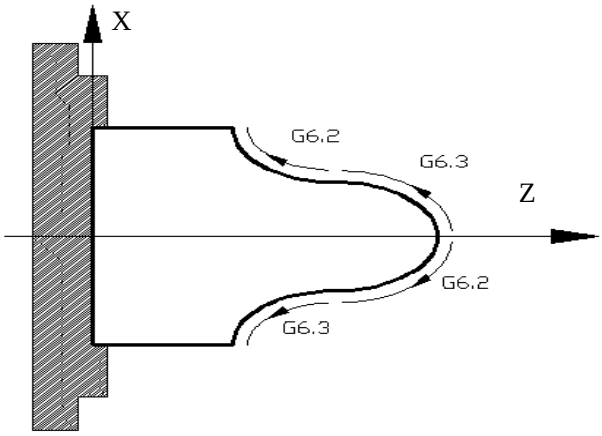
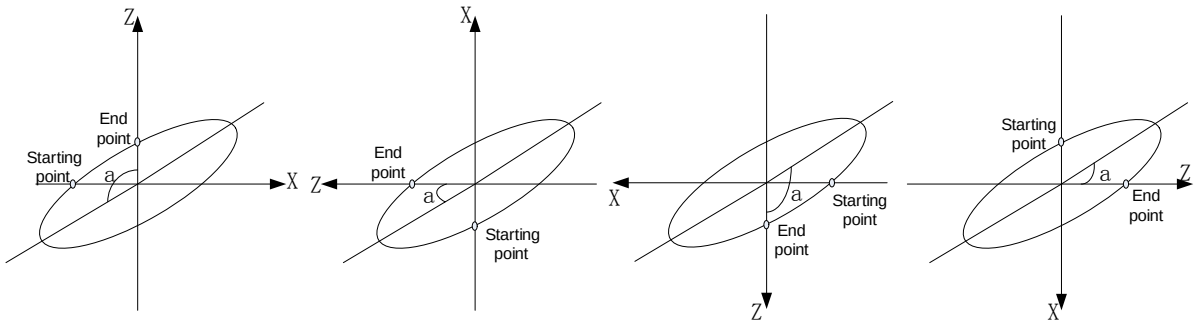


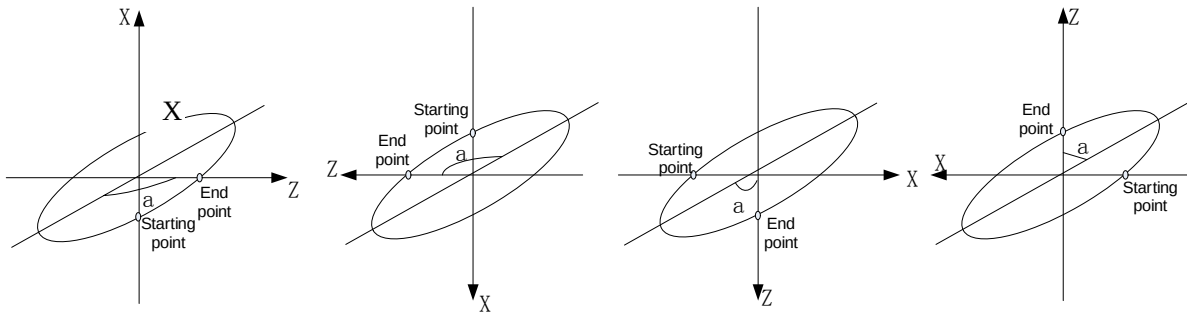
Fig. 3-18 Rear tool post coordinate system

Q value:

1) In the G18 plane, Q value is the gained viewing XZ plane along with the positive direction of Y axis in the Cartesian right-hand coordinate system, as well, it is an angle when the Z axis positively rotates to the the overlapping with the ellispse long axis along with the CW; refer to the following figure:



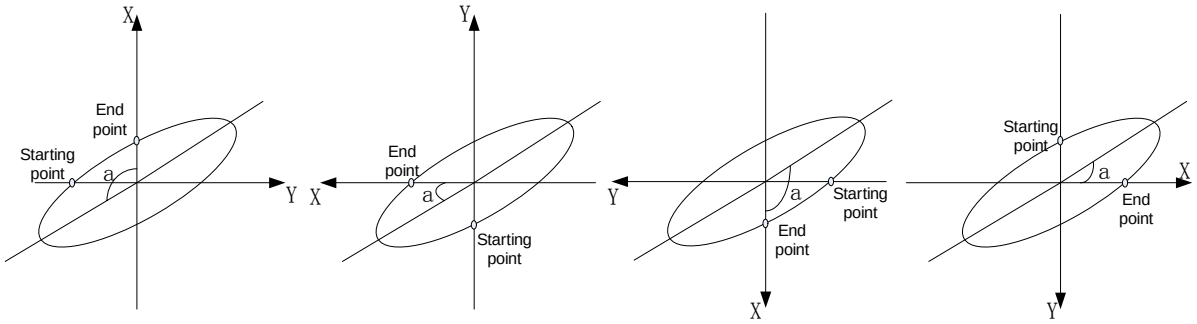
As for the above-mentioned four coordinates (Front tool-post), angle a is the symmetry axis based upon that it rotates to ellipse along with the Z axis CCW.



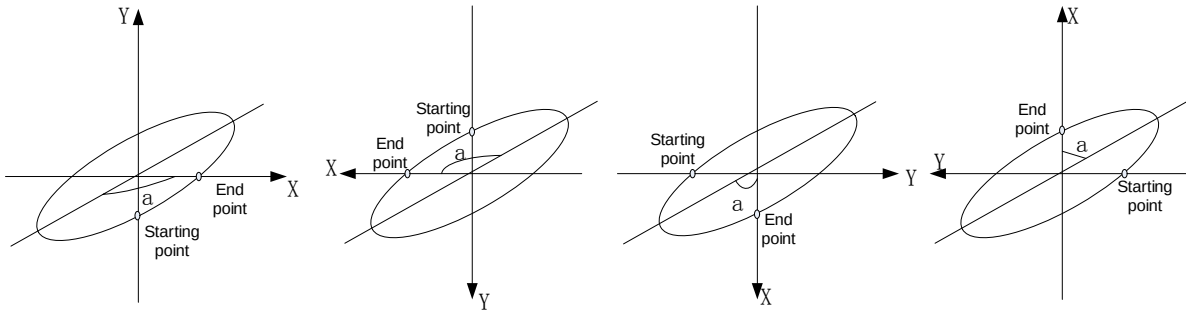
As for the above-mentioned four coordinates (Back tool-post), angle a is the symmetry axis based upon that it rotates to ellipse along with the Z axis CW.

Fig. 3-19

2) In the G17 panel, refer to the following figure:



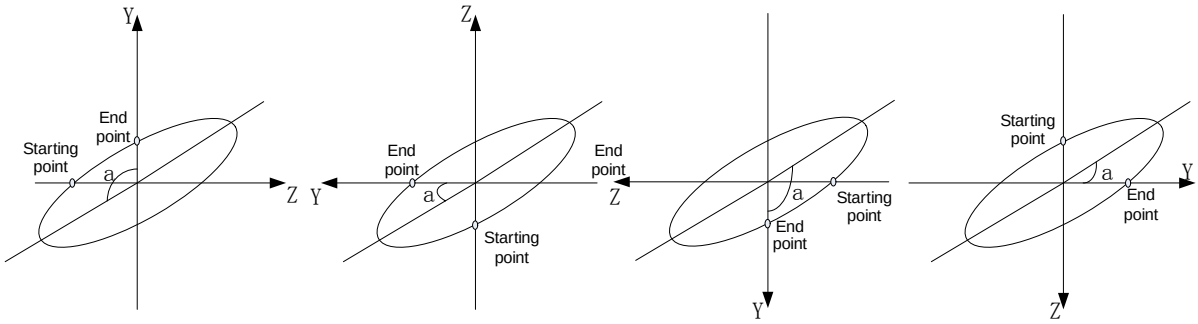
As for the above-mentioned four coordinates (Front tool-post), angle a is the symmetry axis based upon that it rotates to ellipse along with the X axis CCW.



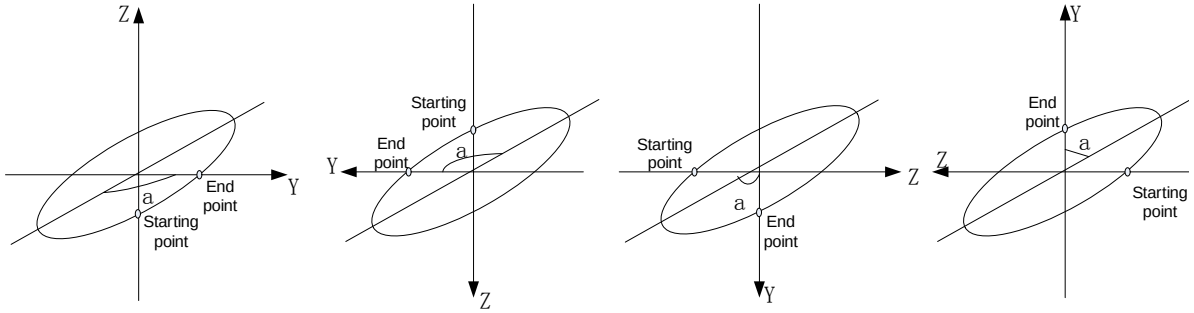
As for the above-mentioned four coordinates (Back tool-post), angle a is the symmetry axis based upon that it rotates to ellipse along with the X axis CW.

Fig. 3-20

3) In the G19 plane, refer to the following figure:



As for the above-mentioned four coordinates (Front tool-post), angle a is the symmetry axis based upon that it rotates to ellipse along with the Y axis CCW.



As for the above-mentioned four coordinates (Back tool-post), angle a is the symmetry axis based upon that it rotates to ellipse along with the Y axis CW.

Fig. 3-21

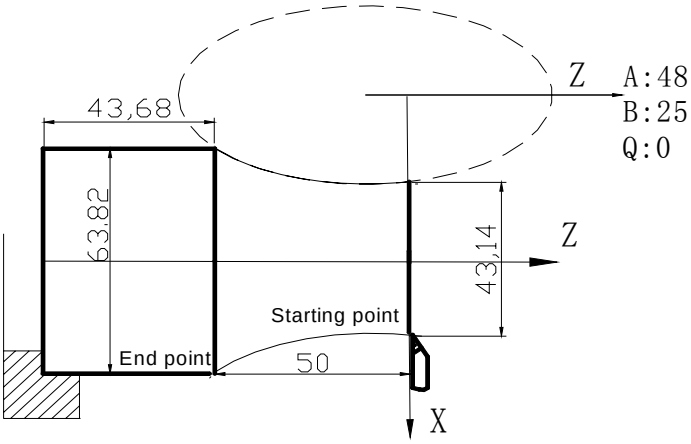
Notes:

1) A, B are not modal parameter. They are 0 when they are not input. When A=0 or B=0, the system

alarms; when A=B, G02/G03 is executed to machine the circular;

2) Q value is a non-modal parameter, and it must be specified when it is used. When it is omitted, it is 0°, the long and the short axis are parallel or coincident ;
3) Q unit is 0.001°, the angle between it and Z is 180°. When Q 180000 is input, Q180 or Q180.0 is input, they are 0.18°;
4) The angle defined by Q value is an angle along Z axis in G18 panel, it is an angle along Y axis in G19, an angle along X axis in G17.
5) When the distance between the starting point and end point is longer than the long axis, the system alarms;
6) One or both of X(U), Z(W) can be omitted; omitting one of them indicates that the starting point and the end point of the axis are consistent; omitting the both indicates they are the same one;
7) Only machines the ellipse which is less than 180°(including 180°);
8) G6.2/G6.3 are used to the compound cycle G70—G73, and their notes are the same those of G02, G03;
9) G6.2, G6.3 are used to C tool compensation.

Example: machine from Φ43.14 to Φ63.82:



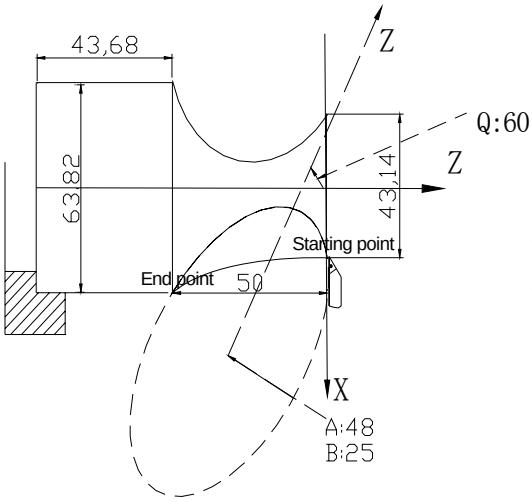
Program:

G6.2 X63.82 Z-50.0 A48 B25 Q0 ； or

G6.2 U20.68 W-50.0 A48 B25 ；

Fig. 3-22

Example: machine from Φ43.14 to Φ63.82:



Program:

G6.2 X63.82 Z-50.0 A48 B25 Q60000； or

G6.2 U20.68 W-50.0 A48 B25 Q60000；

Fig. 3-23

G6.2/G6.3 compound program example :

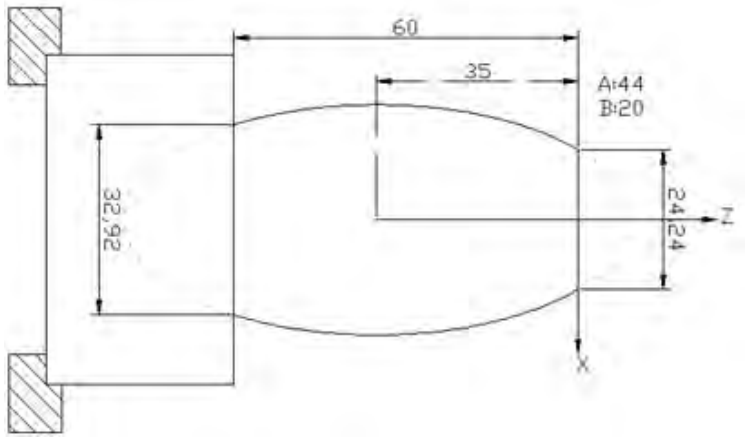


Fig. 3-24

➢ Program: O0001

```
N001 G0 X60 Z5;                  (rapidly position)
N002 M03 S200;                   (spindle ON)
N003 G01 X24.24 Z0 F100;         (approach the workpiece)
N005 G6.3 X40 W-35 A44 B20;      (cutting A44 B20 ellipse block)
N006 G01 X32 Z-60;
N007 Z-79;
N008 G0 X60
N009 Z5;                         (return to the starting point)
N0010 M30;                       (end of program)
```

3.7 Parabola Interpolation G7.2, G7.3

Command format: G7.2 / G7.3 } X(U)__ Z(W)__ P__ Q__

Command function: G7.2 motion path is the CW (rear tool post coordinates)/CCW(front tool post coordinates) parabola.

G7.3 motion path is the CCW (rear tool post coordinates)/CW(front tool post coordinates) parabola.

Command path:

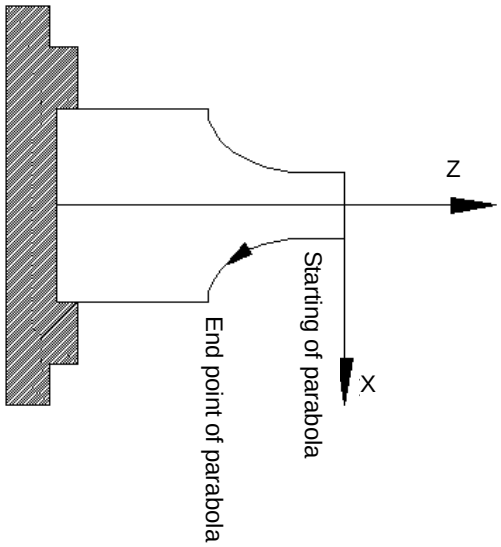


Fig. 3-25 G7.2 path sketch map

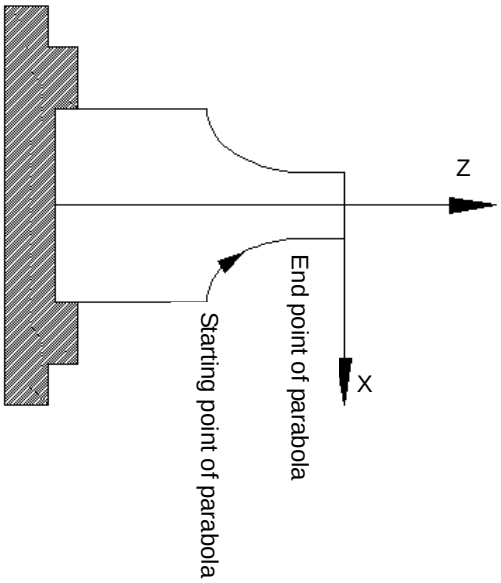


Fig. 3-26 G7.3 path sketch map

Explanation:

G7.2, G7.3 are modal;

P is in the parabola standard equation $Y^2=2PX$, its range: 1～9999999(unit: 0.001mm, without sign);

Q is the angle between the symmetrical axis of parabola and Z, and its range: 0～99999999 (unit: 0.001°) .

Parabola direction: G7.2/7.3 interpolation directions in the front tool post coordinate system and the rear are reverse.

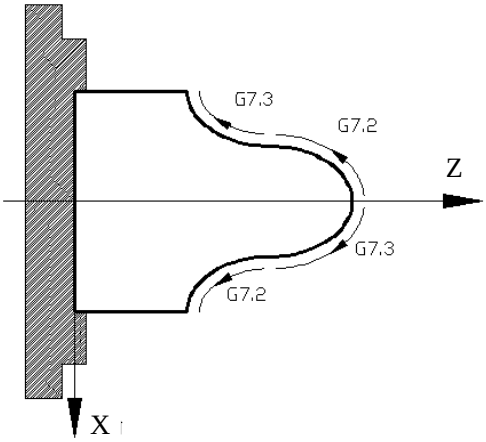


Fig. 3-27 Front tool post coordinate system

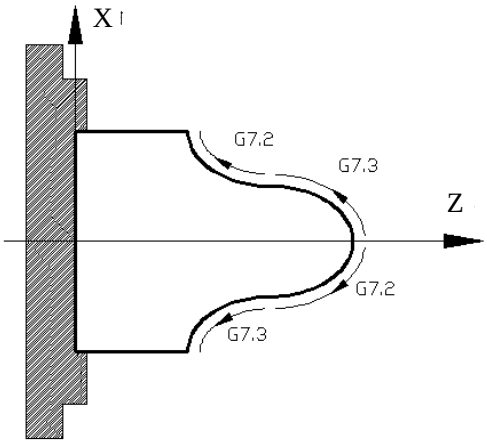
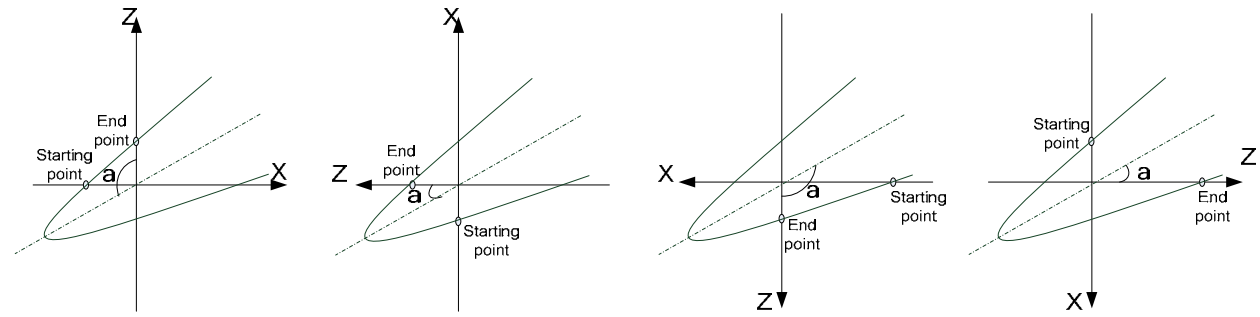


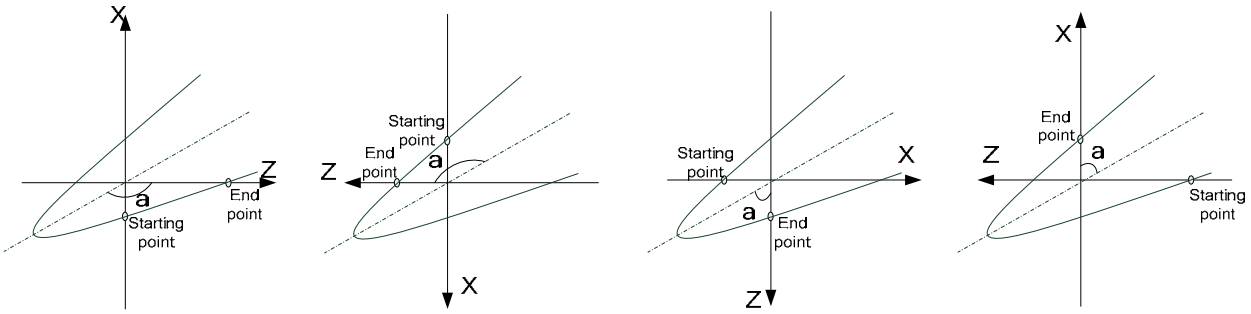
Fig. 3-28 Rear tool post coordinate system

Q value:

1) In the G18 plane, Q value is the gained viewing XZ plane along with the positive direction of Y axis in the Cartesian right-hand coordinate system, as well, it is an angle when the Z axis positively rotates to the the parabola with the ellispse long axis along with the CW; refer to the following figure:



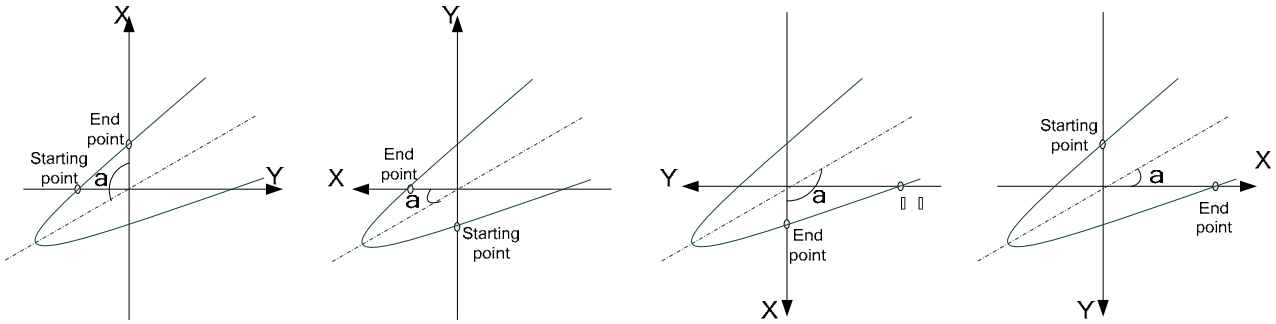
As for the above-mentioned four coordinates (Front tool-post), angle a is the symmetry axis based upon that it rotates to parabola along with the Z axis CCW.



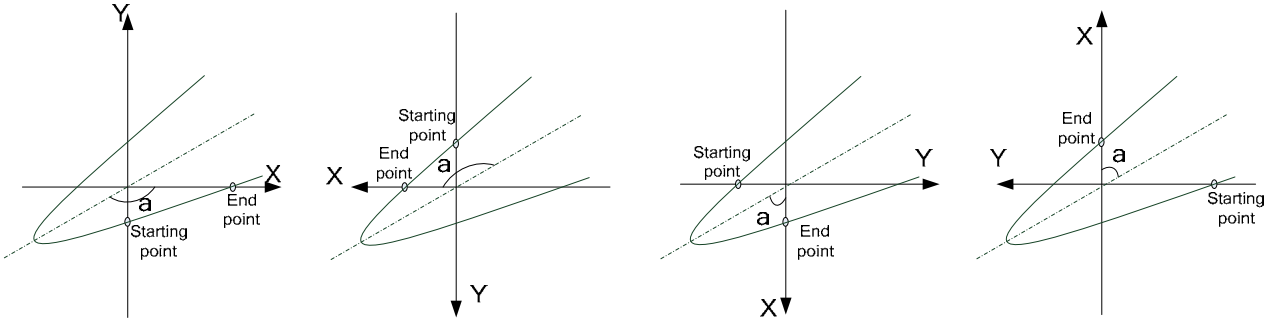
As for the above-mentioned four coordinates (Back tool-post), angle a is the symmetry axis based upon that it rotates to parabola along with the Z axis CW.

Fig. 3-29

2) In the G17 plane, Q value is the gained viewing XY plane along with the positive direction of Z axis in the Cartesian right-hand coordinate system, as well, it is an angle when the X axis positively rotates to the the parabola with the ellispse long axis along with the CW; refer to the following figure:



As for the above-mentioned four coordinates (Front tool-post), angle a is the symmetry axis based upon that it rotates to parabola along with the X axis CCW.



As for the above-mentioned four coordinates (Back tool-post), angle a is the symmetry axis based upon that it rotates to parabola along with the X axis CW.

Fig. 3-30

3) In the G17 plane, Q value is the gained viewing ZY plane along with the positive direction

of X axis in the Cartesian right-hand coordinate system, as well, it is an angle when the Y axis positively rotates to the the parabola with the ellispse long axis along with the CW; refer to the following figure:

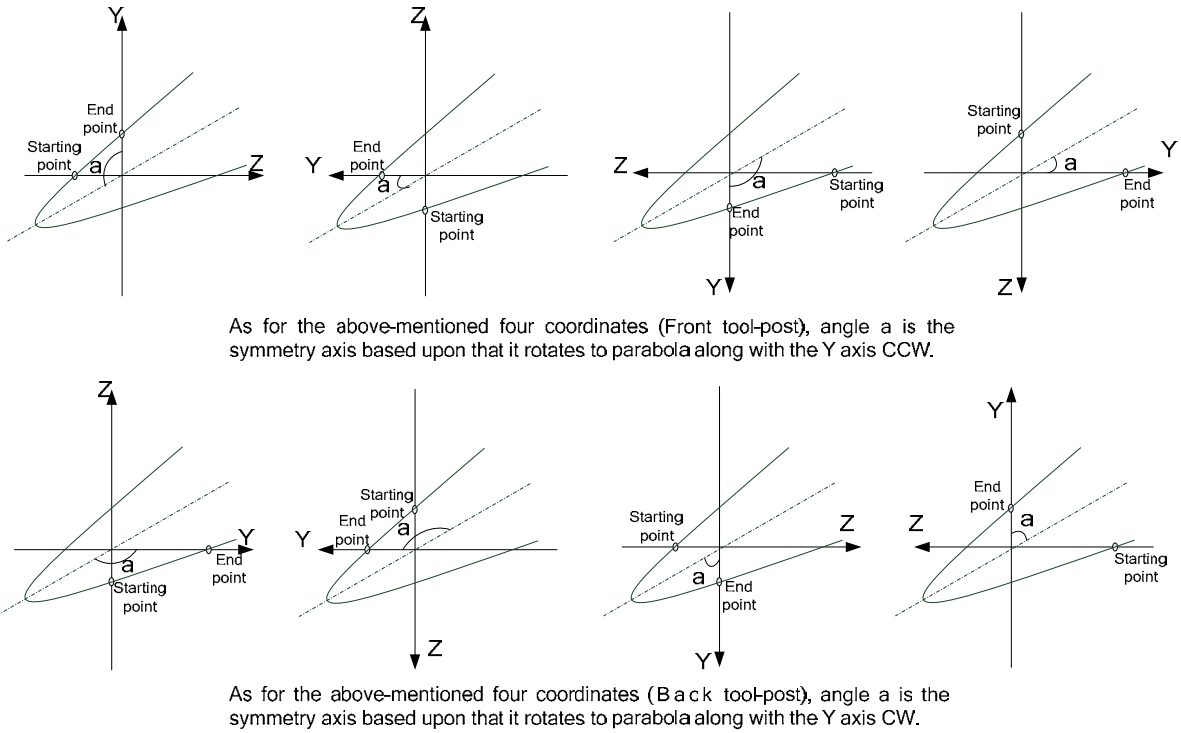


Fig. 3-31

Notes:

1) P cannot be 0 or omitted, otherwise the system alarms;

2) P has no sign. If it has the negative sign, its absolute value is executed;

3) Q can be omitted, at the time; the symmetrical axis of the parabola is parallel with or coincident with Z;

4) The system alarms when the straight line on which the starting point and the end point is parallel with the symmetrical axis of the parabola;

5) G7.2, G7.3 can be used to the compound cycle G70～G73, and their notes are the same those of G02, G03;

Example: when the parabola P=100(the least increment is 0.0001mm), its symmetrical axis is parallel with Z. Its machining sketch map and programming are as follows:

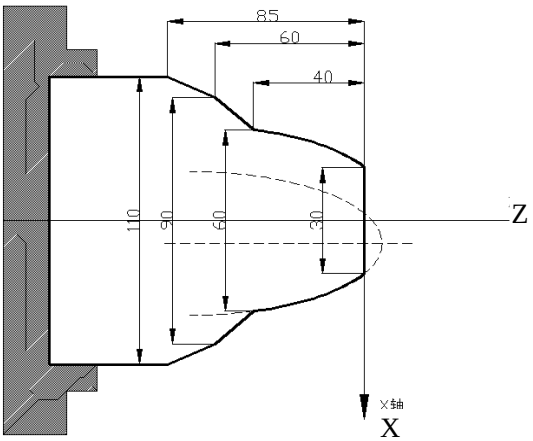


Fig.3-32

```
Program:
        O0001(O0001)
        G00 X120 Z100 T0101 M03 S800;
        G00 X10 Z10;
        G00 X0;
        G01 Z0 F120 M08;
        X30;
        G7.3 X60 Z-40 P10000 Q0;
        G01 X90 Z-60;
        X110 Z-85;
        X120;
        M09;
        G00 X120 Z100 M05 S0;
        M30;
```

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:

1) Firstly set the basic axis Y when the system selects G17, G19 plane;
2) Cannot switch the planes in C tool compensation;
3) G71~G76，G90，G92，G94 can be used in G18 plane;
4) The plane selection code can be in the same block with G codes in the other groups;
5) The movement command is not relevant to the plane selection;
6) 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;
7) The tool nose direction of C tool compensation is 0 in G17, G19.
8) Angel Q in the oval and parabola command, in G18 panel, it is an angle along Y axis in G19, an angle along X axis in G17.

3.9 Polar Coordinate Interpolation G12.1, G13.1

Command format: G12.1----start polar coordinate interpolation method-（can be written to G112）---(1)

G98
G01 X__ C__
G04 X__
G41/G42 G1 X__ C__
G6.2/G6.3 X__ C__ A__ B__ Q__
G7.2/G7.3 X__ C__ A__ B__ Q__
G05 X__ C__ I__
G02/G03 X__ C__ R__
G40 G1 X__ C__
G65/G66/G67

} --- (2) available command

G13.1----cancel polar coordinate interpolation method-（can be written to G113）--- (3)

Command function: Polar coordinate interpolation is a function that exercises contour control in converting a command programmed in a Cartesian coordinate system to the movement of a linear axis (movement of a tool) and the movement of a rotary axis (rotation of a workpiece). This method is useful in cutting a front surface and grinding a cam shaft on the machine.

Command explanation: G12.1, G13.1 are non-modal

Linear axis: X or Z, Y or 4th or 5th axis

Rotary axis: axes except for feed axis(Y or 4th or 5th axis)

Before starting the polar coordinate, set the linear axis and the rotary axis by No.235 and No.236.

Taking example of the linear axis X and the rotary axis 5th .

Polar coordinate interpolation plane: G12.1 starts the polar coordinate interpolation mode and selects a polar coordinate interpolation plane(below), and the polar coordinates are completed in the plane.

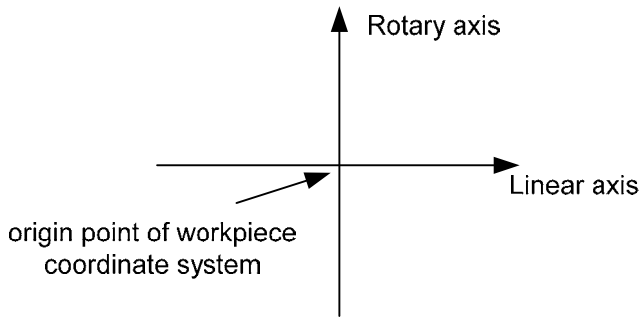
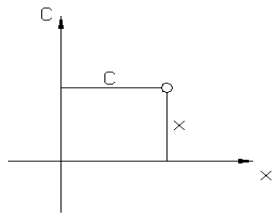


Fig.3-33 Polar coordinate interpolation plane

Note1: After G12.1 is used, the previous plane is cancelled and the system enters the polar coordinate interpolation plane, and after G13.1 is used, the plane is cancelled and the previous is recovered;

Note 2：After the system resets, it cancels the polar coordinate interpolation and recovers the previous plane, and the cursor returns to the beginning of the program.

Programming format: create the rectangular coordinate system in the polar coordinate interpolation plane below:

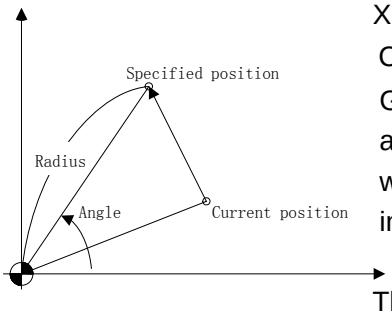


X：distance of linear axis, unit: mm/inch;

C：distance of rotary axis, unit: mm/inch;

The linear axis can use the diameter programming and the rotary axis uses only the radius;

The polar coordinates in the polar coordinate interpolation plane is prepared below:



X: length between current tool and the origin, its unit: mm/inch;

C：angle unit of current rotary axis: deg;

G16 is the polar coordinates of current coordinates being complied, and G15 is to cancel it. The Cartesian coordinate system is used without G16. G16/G15 is valid in the only polar coordinate interpolation.

The radius programming is applied to the linear axis and the rotary axis below:

●**Length compensation**: the length compensation is not applied to the rotary axis, and the length offset should be command before G12.1 mode, and cannot be changed in the polar coordinate interpolation.

●**Tool radius compensation:** the tool nose direction is 0.

●**Machine motion**：the linear axis is vertical to the rotary axis.

●**Circular interpolation in the interpolation plane:** the address of the arc radius is determined by the 1st axis(linear axis) in the interpolation plane.

- When the linear axis is X or its parallel axis, I and J are used in Xp-Yp plane
- When the linear axis is Y or its parallel axis, J and K are used in Xp-Zp plane
- When the linear axis is Z or its parallel axis, K and I are used in Zp-Xp plane

The arc radius can also use R command.

●**Command speed:** tangential speed in the polar coordinate plane.

When the tool approaches the center of the workpiece, the speed component of C axis exceeds its maximum cutting federate (set by No.27). F value is calculated by the followings:

L：the distance (mm) between the center of the tool and that of the workpiece

R：maximum cutting feedrate of C axis（deg/min）

Conclusion：F＜L×R×π /180（mm/min）

It is suggested that the workpiece should not be machined near the polar because the feedate override must be changed in some occasion to avoid the overload of the rotary axis.

● **Movement along the axis in the non polar coordinate interpolation plane in the polar coordinate interpolation mode:** The tool traverses normally along these axes and is not relevant to the polar coordinate interpolation, the parameter specifies one axis to execute the polar coordinate interpolation before executing the polar coordinate interpolation, the axis which does not specify the polar coordinate can moves along the normal path in G00 or G01, but the axis which polar coordinate is not specified is disabled (i.e. the block is ignored and the axis does not move) in arc or ellipse command.

● **Coordinates display:** After G12.1 is executed, the absolute coordinates, the machine coordinates and the incremental coordinates display the actual position of the tool, the remaining distance to move in a block is displayed based on the coordinates in the polar coordinate interpolation plane, and after G13.1 is executed or the reset is done, the coordinates in the current system plane is displayed.

Note 1: G12.1，G13.1 are in Group 21, G12.1，G13.1，G16，G15 are in a separate line.

Note 2: The tool change cannot be executed in G12.1-G13.1, the tool change operation and the positioning followed by the tool change must be performed before G12.1.

Note 3: The system cannot start the polar coordinate interpolation during C tool compensation or in G99, otherwise, it alarms.

Note 4: When G12.1 is commanded, the tool position of the polar coordinate interpolation is at the angle of 0.

5) It is necessary to firstly perform the length offset value before starting it; otherwise, the alarm may occur.

Example:

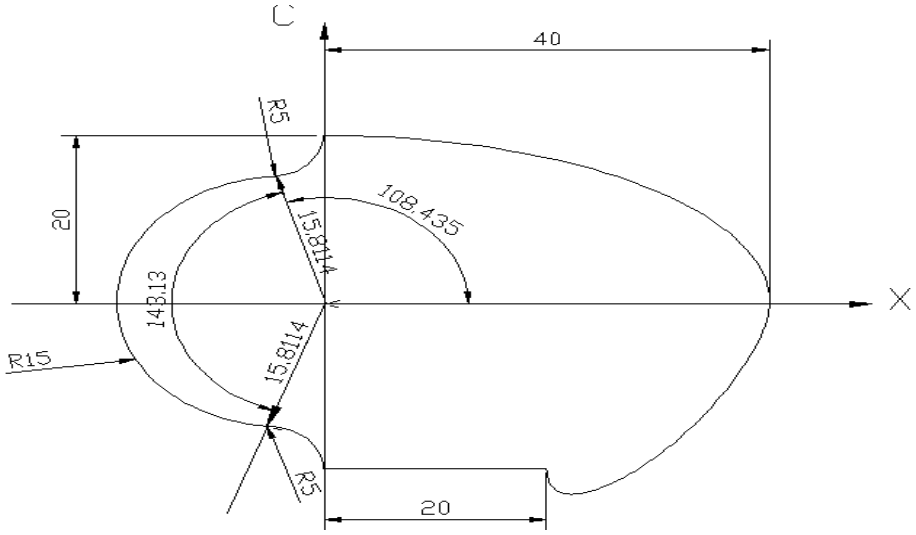


Fig.3-34

```
O0000 (O0000)
T0101
G0 X80 C0 W0
G12.1
G6.3 X0 C20 A40 B20 F1000
```

G2 X-10 C15 R5
G3 X-10 C-15 R15
} replace to →
G16----the followings are the length and the angle programming
G2 X15.8114 C108.435 R5
G3 X15.8114 C251.565 R15
G15-----cancel the above programming mode and the followings are Cartesian coordinate programming

```
G2 X0 C-20   R5
G1 X40 C-20
G7.3 X80 C0 P10000 Q60000
G13.1
M30
```

3.10 Cylindrical Interpolation G7.1

Command function: The amount of travel of a rotary axis specified by an angle is once internally converted to a distance of a linear axis along the outer surface so that linear interpolation or circular interpolation can be performed with another axis. After interpolation, such a distance is converted back to the amount of travel of the rotary axis. The cylindrical interpolation function allows the side of a cylinder to be developed for programming (below).

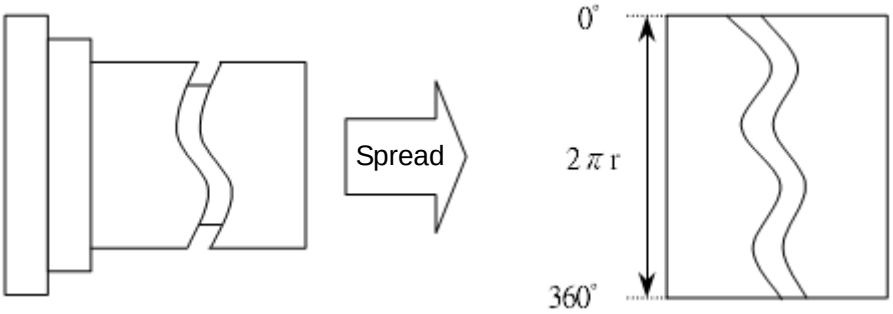


Fig.3-35

Command format:

```
Type I
G07.1 C(c); (1) starting the cylindrical interpolation
…
…              (2)
…
G7.1 C0;     (3) cancel the cylindrical interpolation
```

```
Type II
G07.1 C(c) J(Δi) L(d) R(Δd) K(Δu); (1) starting the cylindrical interpolation
…
…              (2)
…
G7.1 C0;     (3) cancel the cylindrical interpolation
```

Code meaning: Code devides into 3 sections:

(1): Provide a column radius, the column interpolation begins;

(2): Provide the block of the its section of the defined finish-turning path or the finish-turning residual;

(3): Cancel the column interpolation

There are two programming types of cylinder interpolation, J(Δi), L(d) , R(Δd) and K(Δu) settings are not desired for the type I; the system operates based upon the block of the finish-turning path. Type II is the circular interpolation cycle; System is automatically calculated the rough-turning offset value, single feed value and path based upon the data such as the finih-turning allowance, total cutting value and cutting times, etc. The path of each cutting is the offset of the finishing-turning path, the cutting path is gradually closed to the path of the finish-turning, the last cutting path is the finish-turning one.

Command explanation:

Cc: the radius of the cylinder；

Radius ≠0：starts the cylindrical interpolation mode

Radius＝0：cancels the cylindrical interpolation mode

its range: 0～99999999×least input increment (unit：mm/inch)

J(Δi)：Total cutting value along X axis in the rough-turning,its range:±99999999×least input increment(unit：mm/inch，diameter，with sign, when the symbol is positive, X axis infeeds along with the positive direction; when the symbol is negative, X axis infeeds along with the negative direction), type I setting does not desire；

R(Δd)：Tool-retraction value along X axis in the rough-turning，its range:±99999999×least input increment(unit：mm/inch，diameter，without sign symbol) ，type I setting does not desire；

K(Δu)：Finish-turning allowance along X axis, its range:-99999.999~99999.999(IS_B) /-9999.9999~9999.9999(IS_C) (unit：mm/inch，diameter，without sign symbol), the last rough-turning path offsets related to the X axis coordinate of the finish-turning, type I setting does not desire；

L(d)：Cutting times（without including the finish-turning path）,its range:1~9999 (unit：times), L5 means that the rough cutting cycle is completed for 5 times, the 6th times is finish turning cycle. If the cutting times is 1, the system will complete the rough cutting cycle based upon 2 times. Type I setting does not desire.

Code path explanation: (Type II)

1) X axis rapidly moves to cycle start G from start point F；
2) X axis is performed the feedrate to An point, in-feed value is Δd＋n×Δu, which is performed the circular interpolation along with the path of the An－>Bn－>Cn－>Dn (Note: n is the nth times cycle, the resolution n=1, 2, 3…)
3) X axis rapidly retreats to point E along Dn－>E path
4) Tool returns to the cycle start G along E－>G path (Note: X, Z and C axes are returned to the cycle start);
5) Repeated steps 2), till to the machining allowance along X axis is 0;

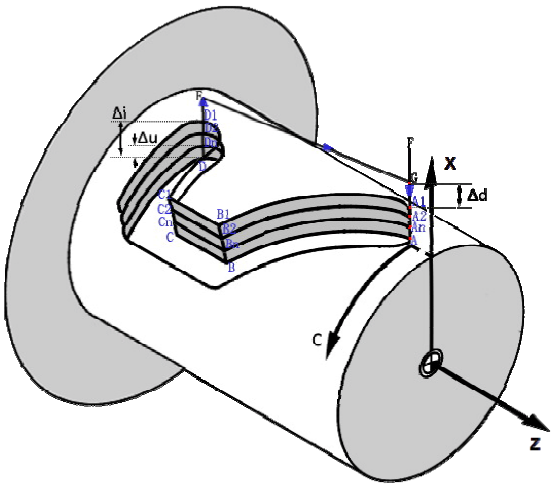


Fig.3-36

Note：

1) G7.1 is non-modal；

2) The cylindrical interpolation mode is cancelled in power-on and reset;
3) The rotary axis executes a program by an angle, a rolling function of a rotary axis in the cylindrical interpolation mode will be automatically invalid, and a programming command value must be more than 360^0 when the interpolation range is more than one circle;
4) The tool nose radius compensation G41, G42 can be executed and their tool nose directions are 0;
5) A feedrate specified in the cylindrical interpolation mode is a speed on the developed cylindrical surface, its unit: mm/min or inch/min
6) In the cylindrical interpolation mode, the linear G1, the arc G2, G3（arc radius is specified by R, its unit is mm or inch）, ellipse G6.2, G6.3, parabola G7.2, G7.3 interpolation;
7) G00 positioning operation cannot be executed in a cylindrical interpolation mode;
8) Before executing the cylindrical interpolation mode, the tool nose radius compensation mode which is being executed is cancelled, the tool compensation is started or completed in a cylindrical interpolation mode;
9) Miscellaneous function T cannot be used in the cylindrical interpolation mode;
10) The feedrate in the cylindrical interpolation mode specifies G98(feed per minute);
11) G50 setting workpiece coordinate system in the cylindrical interpolation mode cannot be used;
12) A rotary axis and a linear axis of only current cylinder in the cylindrical interpolation is specified;

Plane selection: Before a cylindrical interpolation, the plane where the interpolation is should be selected firstly, one axis in the plane is a linear axis in the cylindrical interpolation and another is a corresponding linear axis of the developed rotary axis (below).

① Basic coordinate system:

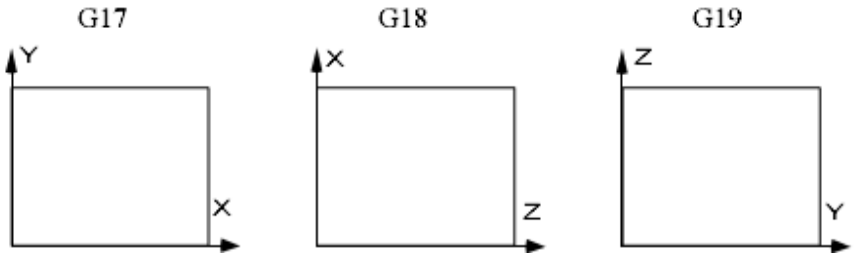


Fig.3-37

② The rotary axis is set to X or its parallel axis in the cylindrical interpolation mode:

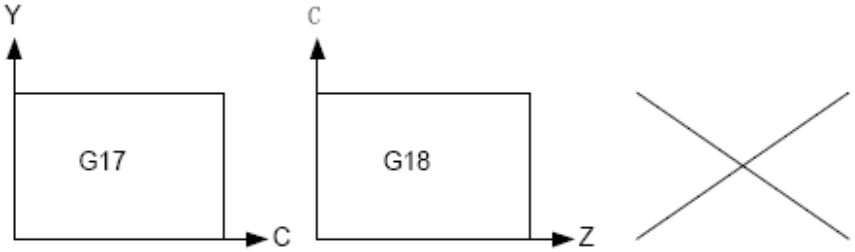


Fig.3-38

③ The rotary axis is set to Y or its parallel axis in the cylindrical interpolation mode:

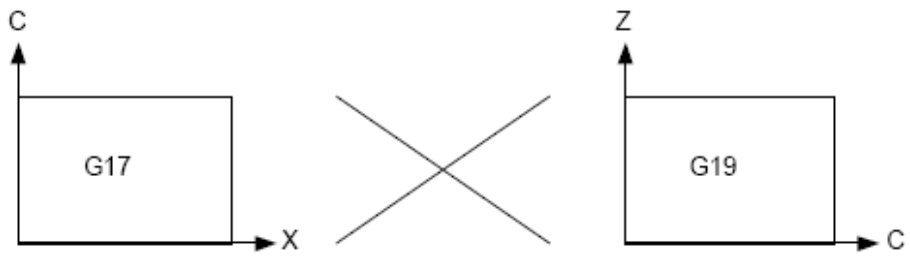


Fig.3-39

④The rotary axis is set to Z or its parallel axis in the cylindrical interpolation mode:

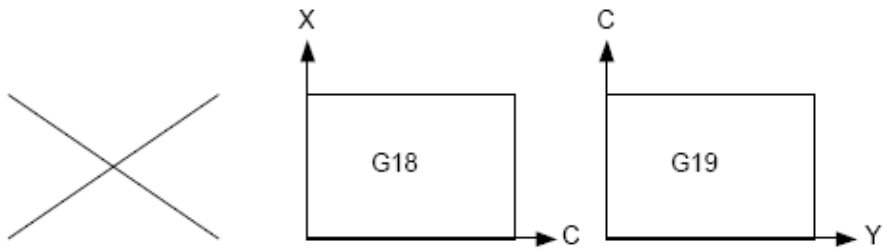


Fig.3-40

Relative parameters:

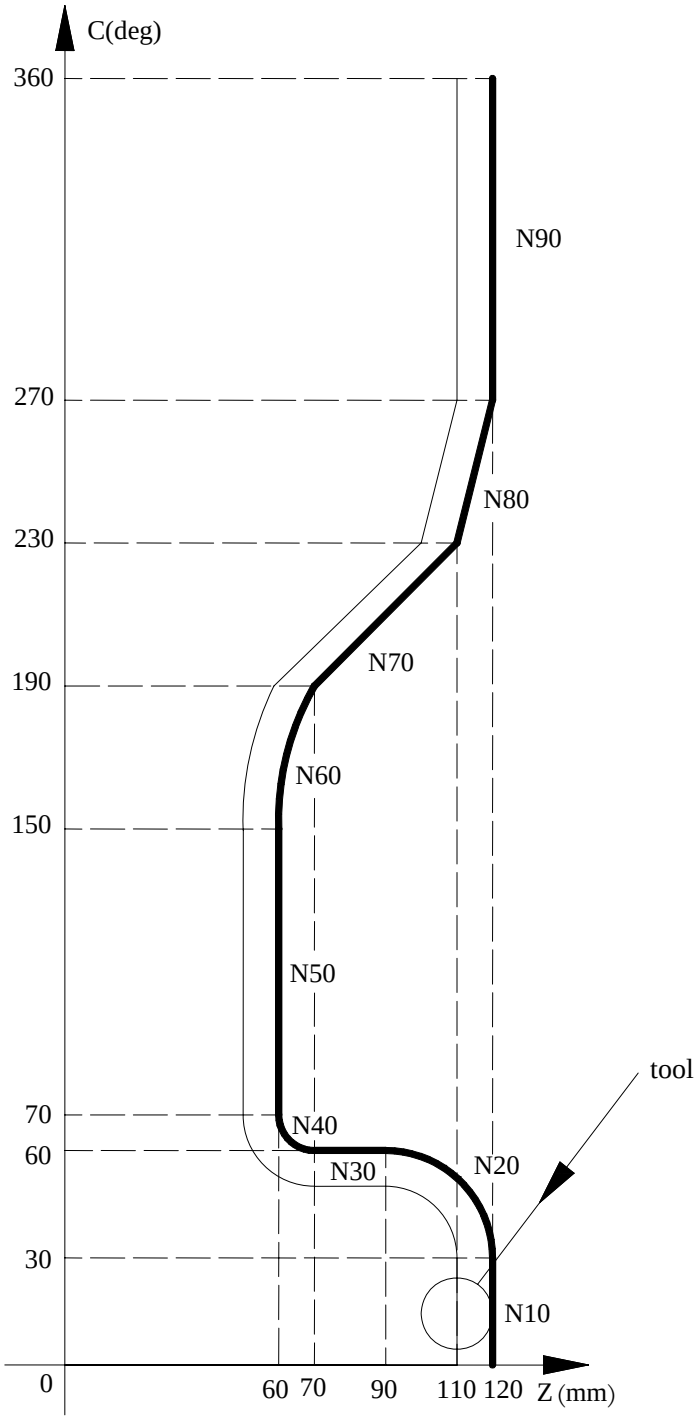
Only rotary axis is specified in the cylindrical interpolation, and it can be a basic axis or an axis parallel to the basic axis. The axis name of three additional axes are set by NO.225, NO.226, NO.227 (Y：89，A：65，B：66，C：67）, the attribute of an axis is set by NO.230, NO.231, NO.232 (see the following table).

Setting value	Meaning
0	It is not three basic axes and their parallel axis
1	X of three basic axes
2	Y of three basic axes
3	Z of three basic axes
5	An axis parallel to X axis
6	An axis parallel to Y axis
7	An axis parallel to Z axis

Example：

No. 224 is set to 5, the cylindrical interpolation is executed in G18 plane, and the rotary axis of the cylindrical interpolation is set to the 5th axis (state parameter P187). The axis name of the 5th is C (data parameter No. 225), the axis is one parallel to X axis (NO.230）. The radius of the cylinder is 57.299mm, and the developed cylindrical surface is shown below:

O0071（taking example of the cylindrical interpolation G7.1）




```
Type I:
O0071 (Cylinder interpolation G7.1
 application example)
G18;
G98;
G00 X150 Z105 C0;
G01 X114.598 Z105 F200;
G07.1 C57.299;
G41 G01 Z120;
N10 G01 C30.0;
N20 G03 Z90 C60 R30;
N30 G01 Z70;
N40 G02 Z60 C70 R10;
N50 G01 C150;
N60 G02 Z70 C190 R75;
N70 G01 Z110 C230;
N80 G03 Z120 C270 R75;
N90 G01 C360;
G40 G01 Z105;
G07.1 C0;
M30;

Type II:
O0071 (Cylinder interpolation G7.1
application example)
G18;
G98;
G00 X150 Z105 C0;
T0101
G01 X114.598 Z105 F200;
G07.1 C57.299 J-10 K0.25 R2 L4;
G41 G01 Z120;
N10 G01 C30.0;
N20 G03 Z90 C60 R30;
N30 G01 Z70;
N40 G02 Z60 C70 R10;
N50 G01 C150;
N60 G02 Z70 C190 R75;
N70 G01 Z110 C230;
N80 G03 Z120 C270 R75;
N90 G01 C360;
G40 G01 Z105;
G07.1 C0;
M30;
```

Fig.3-41

Note: In the progrmming example of the type II, the overall cutting value is 10mm, the finish machining residual is 0.25mm, tool is 2mm distant from the workpiece along with X direction, the rough-turning cycle times is 4, finish-turning cycle times is 1; tool retreats from the start position (116.598,105,0) after the rough-tuning cutting is performed each time.

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. GSK980TDi 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.

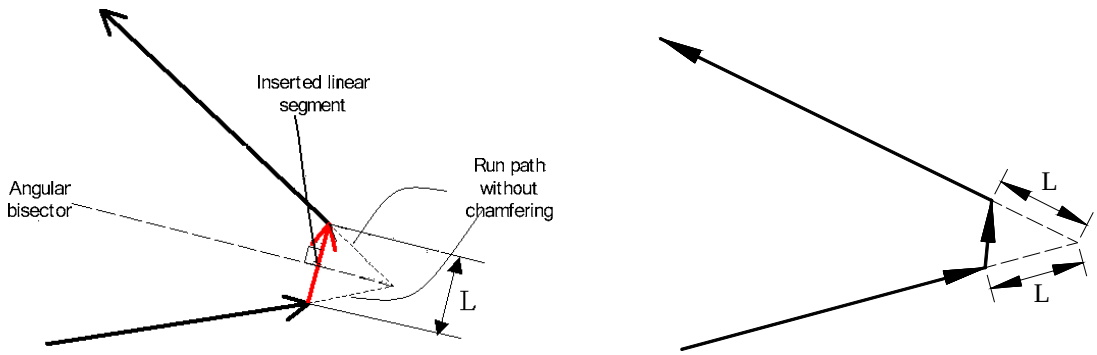
Linear chamfering divides into two types, two types are mainly distinguished from the different definitions of the command value L in the program, one is L chamfering length, the other is length between the chamfering point and intersection point, which can be selected by state parameter No.181.0. In the following explanation, type I represents that L is chamfering length; type II is the length between the chamfering point and intersection point.

1). Linear to linear

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

Command format: Insert a section of linear block between two straight line interpolation.

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

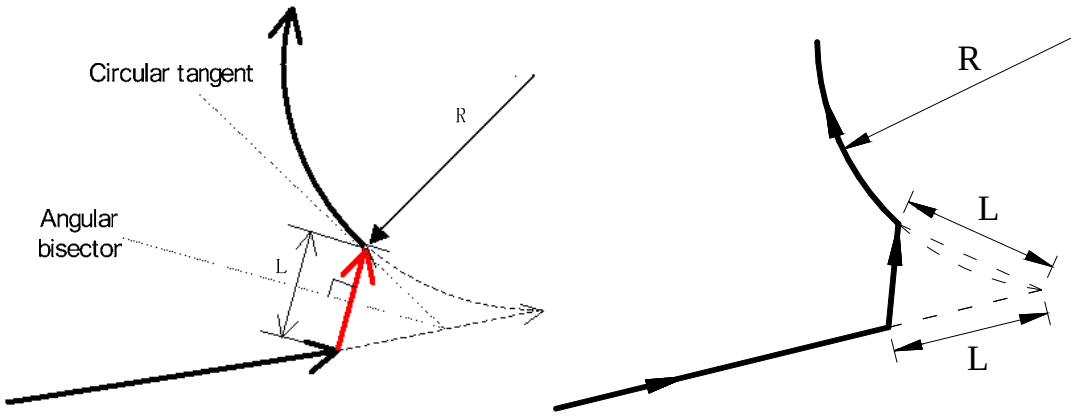


type Ⅰ type Ⅱ

2). 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.

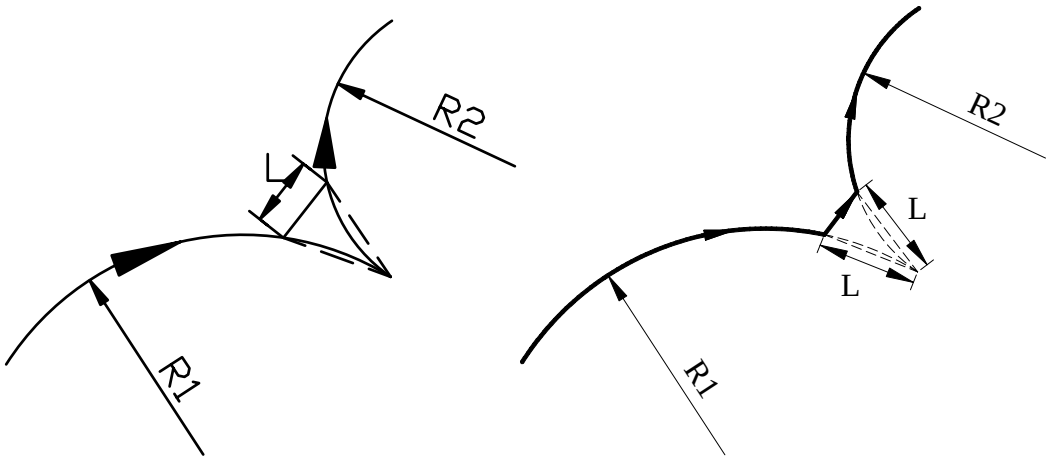


type Ⅰ type Ⅱ

3). 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.

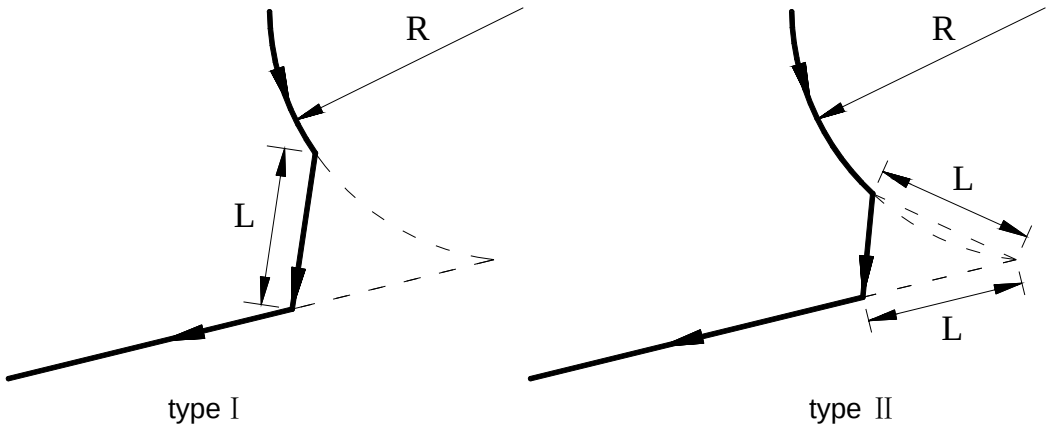


type Ⅰ type Ⅱ

4). 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.



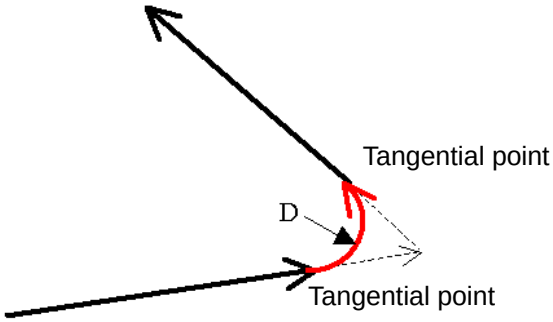
3.11.2 Circular Chamfering

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.

1). Linear to linear

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

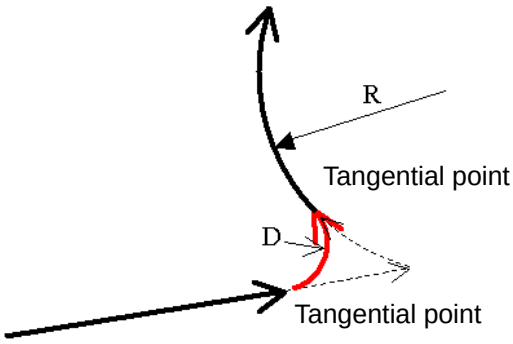
Command function: insert one circular between two straight lines, the inserted circular block and two straight lines are tangent, the radius is the data behind the command address D.



2). 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_;

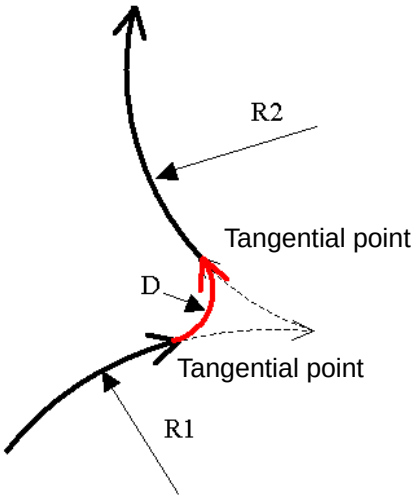
Command function: insert one circular between linear and circular, the inserted circular is tangent to the linear and the circular, and the radius is the data behind the command address D.



3). 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.

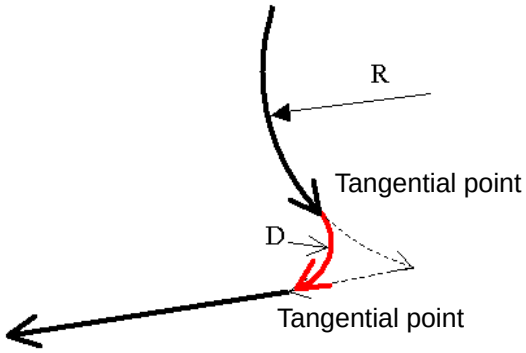


4). Circular to linear

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

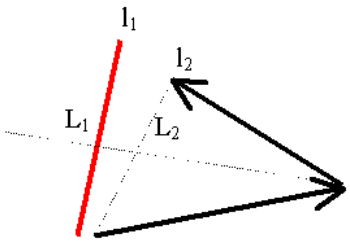
The chamfering function is invalid or alarms as follows:

1) Linear chamfering

Case 1. The chamfering function is invalid when two interpolation straight lines are in the same linear.

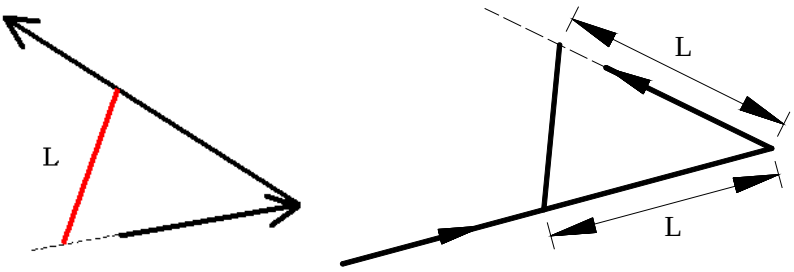
Case 2. CNC alarms when the chamfering linear is too long.

L1 i 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 than L_2 as follows:



Case 3. Some linear block is too short

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



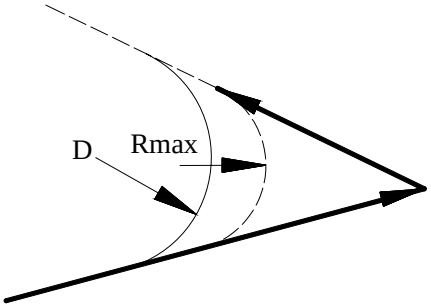
type Ⅰ type Ⅱ

2) Circular chamfering

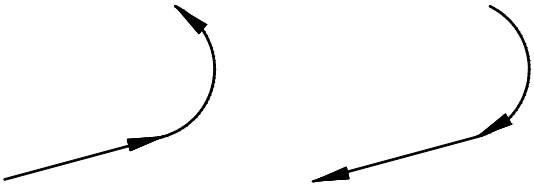
Case 1. The circular chamfering function is invalid when two interpolation straight lines are in the same block.

Case 2. CNC alarms when the chamfering circular radius is too big.

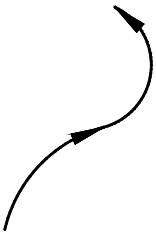
CNC alarms when the chamfering circular radius is D, max. circular radius of the tangential linear lines is R_{max} which is less than D as follows.



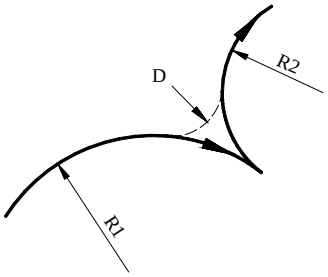
Case 3. The circular chamfering function is invalid when the linear and the circular, or the circular and the linear are tangential.



Case 4. 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 Exact Stop Function Between Blocks

Command format: G09 / G61 / G64 } ……

Command	Command specification
G09	the programmed axis of a block must exactly stop at the end point of the block, and the system continuously executes a next block., Non-modal G commands

G61	The blocks included the overall motions between the current block and the G64 are exactly performed, and then execute the next block, modal command.
G64	Cancel the G61 modal.

Note:

1) G09, G61/G64 are disabled in the G71, G72 and G73 cycle procedure;
2) G09 is only enabled to the current block, if the cutting command generates in the block of the G09, execute the next block after this block is exactly stopped; if the cutting command does not generate in this block, the next block will not perform the exact stop;
3) When the current parameter №0007.5 sets to "1", modal information column is fixedly displayed as G61, and the G64 shows disabling;
4) When the current parameter №0007.5 sets to "1", the G61/G64 position in modal information column is immediately displayed as G61; when the current parameter №0007.5 sets to "0", the G61/G64 position in modal information column is immediately displayed as G64, and then enters to the smooth state between the blocks.
5) When G64 are shared a block with cutting command, the G64 locates at the block which is cancelled the exact-stop in-position;
6) When G09, G64 and G61 are shared with a same block, the last G code is enabled;
7) Perform G64 code or power-on again, G61 modal is being cancelled.

G09 code application example:

After the motion command of the block of the G09 command is performed, the next block command is then performed. The N2 block of the program 1 does not add the G09 command, smoothly transit between N2 block and N3 block; refer to the path in the Fig. 3-42. The N2 block of the program 2 adds G09 command. The command of N3 can be performed till the motion command of the N2 block is performed; refer to the path in Fig. 3-43.

Program 1: Program path is shown in the Fig. 3-42.

N1 G00 X0 Z0
N2 G01 Z50
N3 X100
N4 Z100
N5 M30

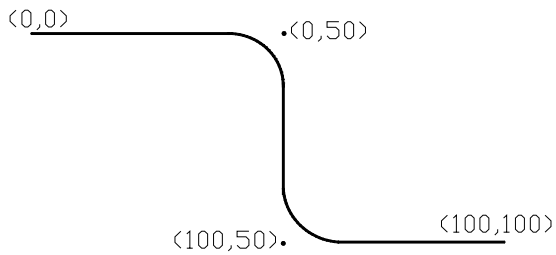


Fig. 3-42

Program 2: Program path is shown in the Fig. 3-43.

N1 G00 X0 Z0
N2 G09 G01 Z50; G09 command is only enabled to the current line.
N3 X100
N4 Z100
N5 M30

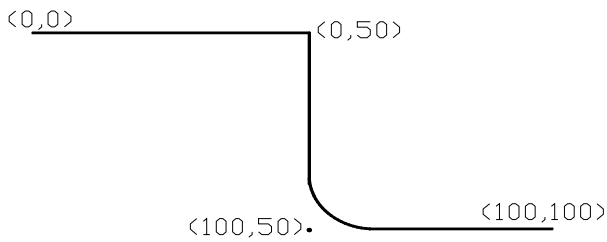


Fig. 3-43

G61, G64 code application example:

G61 is modal command, the next block can be performed after the overall motion blocks between G61 and G64 are executed. There is G61 command in the program 3, smoothly transit between two blocks, and refer to the Fig. 3-44. The N2 block of the program 4 adds G61 command, N7 block adds G64 command, and therefore, after the overall motion blocks between N2 block and N7 block are performed, the next block is then executed; refer to the Fig. 3-45.

Program 3: Program path is shown as Fig. 3-44.

N1 G00 X0 Z0

N2 G01 Z50

N3 X100

N4 Z120

N5 X200

N6 M30

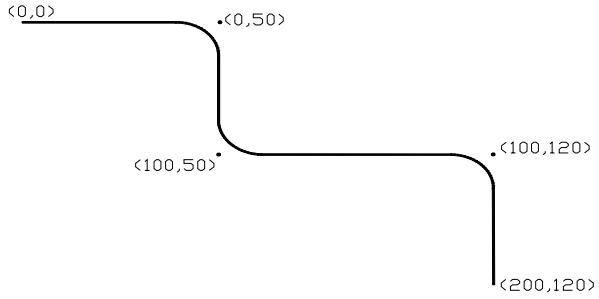


Fig. 3-44

Program 4: Program path is shown as Fig. 3-45.

N1 G00 X0 Z0

N2 G61;

N3 G01 Z50

N4 X100

N5 Z120

N6 X200

N7 G64

N8M30

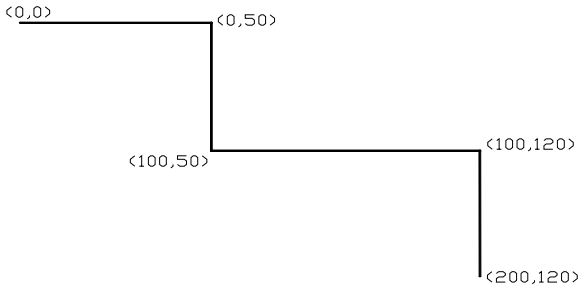


Fig. 3-45

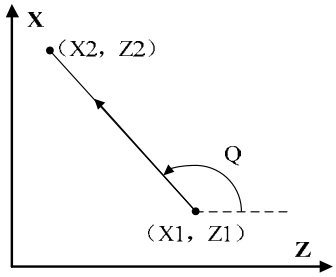
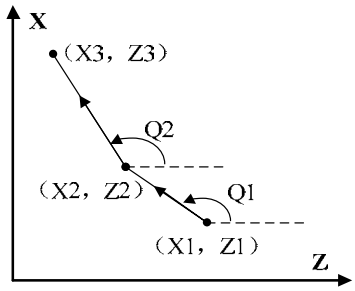
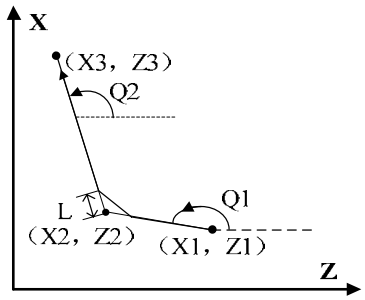
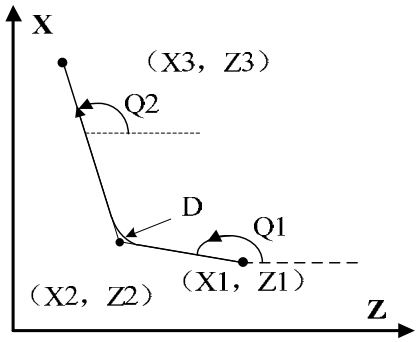
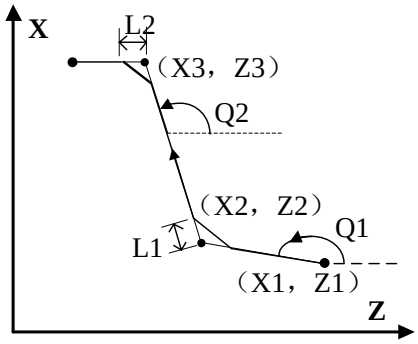
3.13 Function of Directly Inputting Graphic Dimension

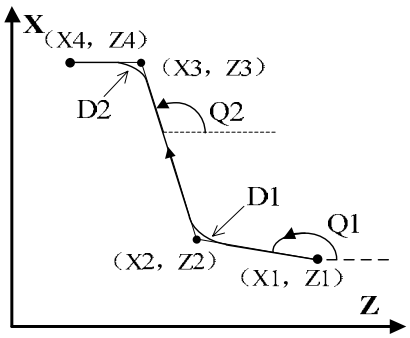
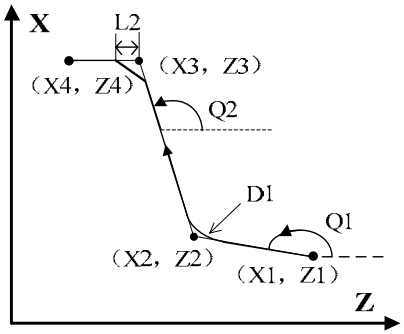
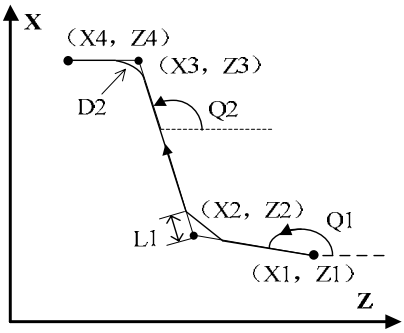
The function of directly inputting graphic dimension can make the user directly use the linear angle, chamfering value in the machining drawing to program.

Command format: Drawing dimension direct input function is only used the linear interpolation (G01), can specify the plane in G17 plane（XY plane）, G18 plane（XZ plane）, G19 plane(ZY plane). Taking example of G18 plane(XZ plane), the format changes when G17/G19 plane command is used:

G17 plane: “Z”→“X”, “X”→“Y”

G19 plane: “Z”→“Y”, “X”→“Z”

	Command format	Command path
1	G01 X2_ (Z2_) Q_	
2	G01 Q1_ G01 X3_ Z3_ Q2_	
3	G01 Q1_ L_ G01 X3_ Z3_ Q2_	
4	G01 Q1_ D_; G01 X3_ Z3_ Q2_;	
5	G01 Q1_ L1_ G01 X3_ Z3_ Q2_ L2_ G01 X4_ Z4_	

	Command format	Command path
6	G01 Q1_ D1_ G01 X3_ Z3_ Q2_ D2_ G01 X4_ Z4_	
7	G01 Q1_ D1_ G01 X3_ Z3_ Q2_ L2_ G01 X4_ Z4_	
8	G01 Q1_ L1_ G01 X3_ Z3_ Q2_ D2_ G01 X4_ Z4_	

Command explanation: When the command format is treated as 1, X or Z direction coordinate only can be specified one; otherwise, it is performed (Angle Q is ignored) based upon the normal straight line. Wherein, Q_ is an angle between straight line and Z axis,its range:-99999999~99999999（unit：0.001degree），its angle direction is shown below:

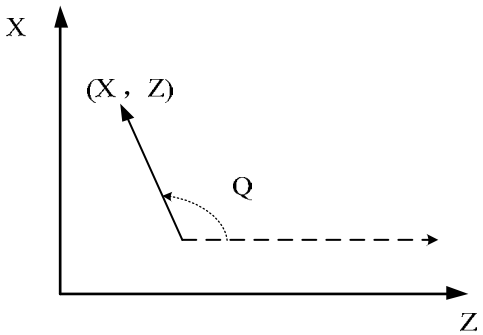


Fig. 3-46 Rear tool post coordinate system

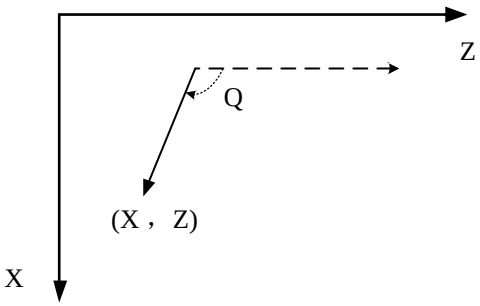


Fig. 3-47 Front tool post coordinate system

Note：

1) **G00 and G01 commands are used for the drawing dimension direct input function; however, G00 only performs the positioning when using drawing dimension direction input function of the G00;**

2) Prevent the excessive distance from calculating, in the command format 1, the angle tolerance value of the calculation intersection is ±1° (Q value converts into 0°～360°).

3) X_ Q_ (If the angle Q (0° or 180°) value is within (0° or 180°) ±1°, the alarm may occur.)

4) Z_ Q_ (If the angle Q (90° or 270°) value is within (0° or 180°) ±1°, the alarm may occur.)

5) **When the intersection is calculated, the alarm may generate if the angle difference composes of two straight lines between ±1.**

6) **In a movement block following with the block which only specifies an angle command of which the coordinate (absolute command) and angle command values should be specified.**

（Example）:

N1 X_ Q_
N2 Q_
N3 X_ Z_ Q_

An absolute command value and angle command Q along the X or Z axis should be specified in N3 block; otherwsie, the alarm may occur.

7) **In the consecutive drawing dimension direct input command, the G04 and the M, S and T blocks can be inserted, however, if there are 2 or more blocks are inserted, the alarm may occur.**

8) **In the copound canned cycle, use the program of the drawing dimension direct input between the sequence numbers specified by P or Q; however, the block specified by Q sequence number can not be stopped in the middle of consecutive drawing dimension direct input command.**

3.14 Dwell G04

Command format: G04 P__ Q_ ; or
G04 X__ Q_ ; or
G04 U__ Q_; 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. Within the G04 dwell time, receive the skip signal (it is determined by Q value), intermit the dwell and then perform the next block.

Command specification: G04 is non-modal.

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

P Range is -99999999～99999999 （unit: ms）.

X, U range is -9999.999～9999.999 (unit: s)

Q: Skip signal address selection, solution range 1～4,

Notes:

- The system exactly stop a block when P, X, U, Q are not input.
- When P, X or U specifies a negative value, it means that the dwell time is 0.
- P is valid when P, X, U are in the same block; X is valid when X, U are in the same block.
- In the G04 code execution, perform the feed hold operation, the dwell shows at the upper left corner of the current page; however, the G04 timing does not stop, when the timing is executed, the cursor stops at the next program.

3.15 Machine Zero Function

3.15.1 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. 3-48) :

(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.

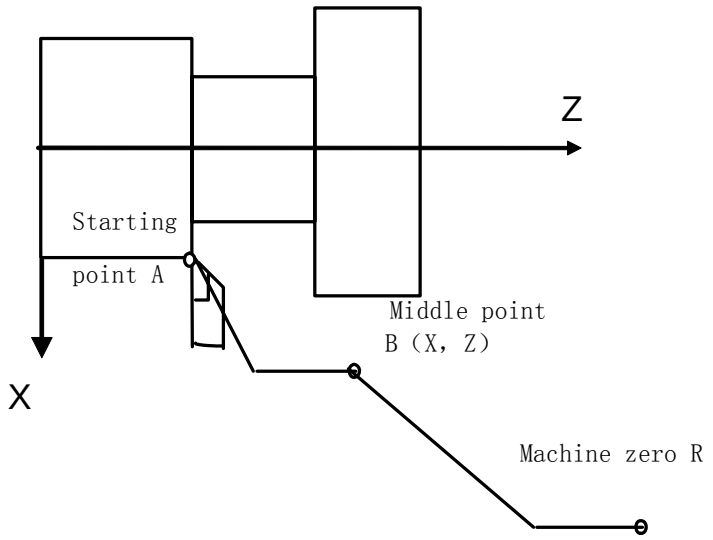


Fig.3-48

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;

3.15.2 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.

GSK980TDi has machine 2nd, 3rd, 4th reference point functions. Use separately No.122～No.127 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:

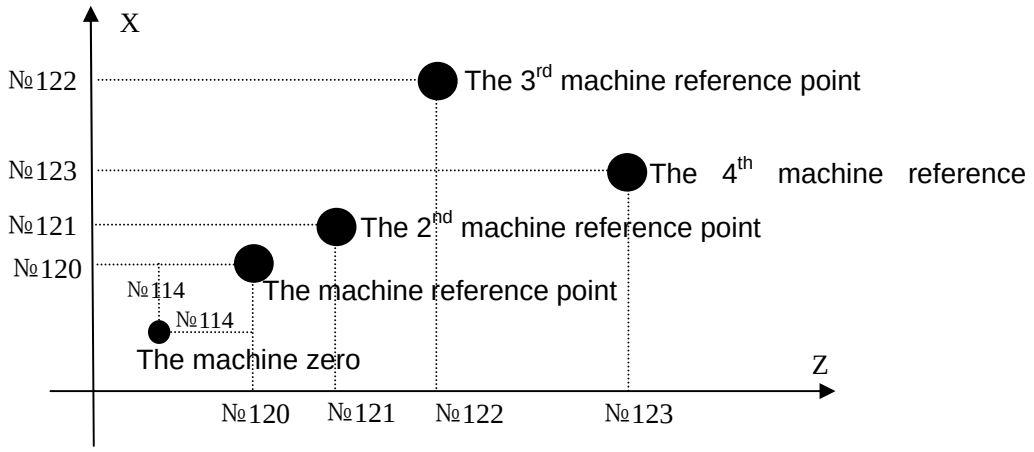


Fig.3-49

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

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

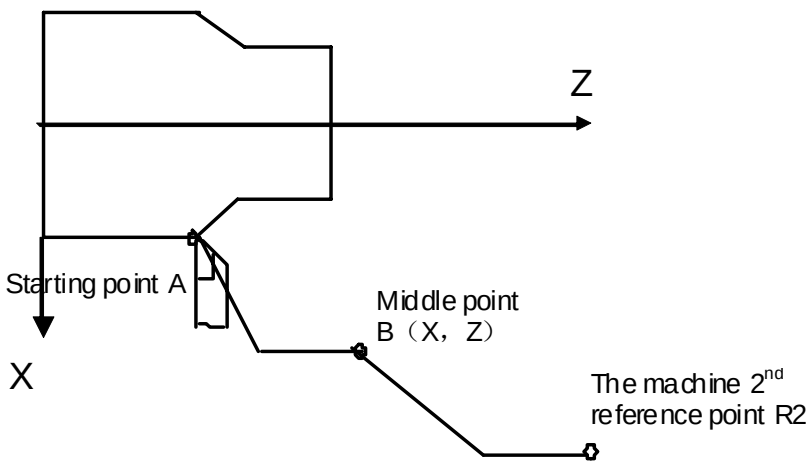
Note 2: Do not check the deceleration, zero signal when you execute the machine 2nd, 3rd, 4th reference point.

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 B0→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.

3.16 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 specifications: non-modal G command (00 group);

X(U)__ Z(W)__ ：End point

F__ ：Cutting feedrate

P__ ：Skip signal address selection，its range:1～4，The skip signal address from the corresponding P1～P4 are set by parameters №148～№151, it is regarded as 1 by default when the P value does not input.

Note:

1) Cancel the tool nose radius compensation before using it;

2) Feedrate should not be set to too big to get the precise stop position;

3.16.1 Following Block Execution After Skip

1. The next block of G31 is the incremental coordinate programming shown in Fig. 3-51:

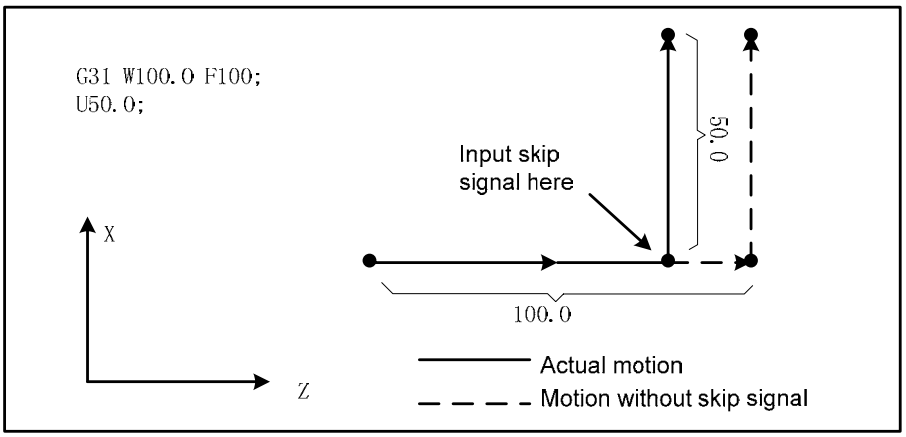


Fig. 3-51

2. The next block of G31 is the absolute coordinate programming of one axis as Fig. 3-52:

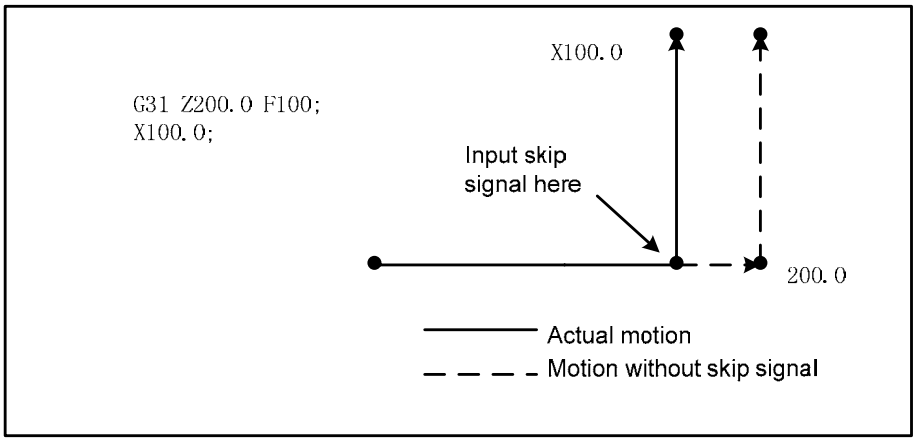


Fig. 3-52

3. The next block of G31 is the absolute coordinate programming of two axes shown in Fig. 3-53:

Program: G31 Z200 F100
G01 X100 Z300

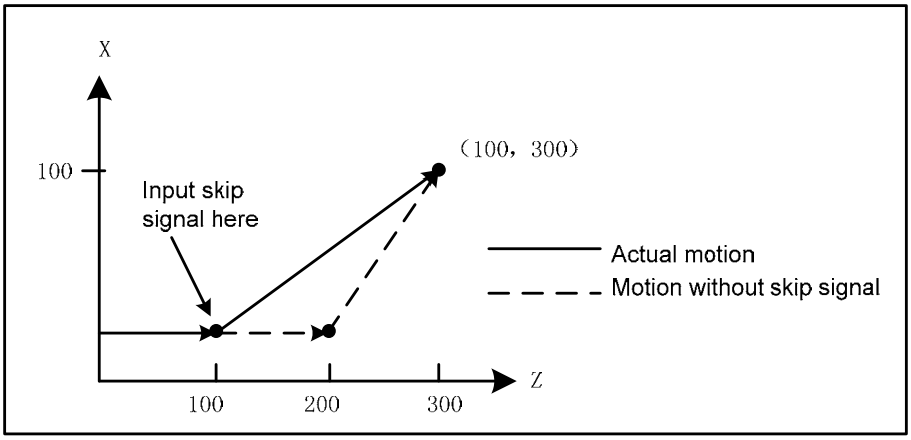


Fig. 3-53

3.16.2 Signals Relevant to G31

Skip signal:

SKIP: X3.5

Type: input signal

Function: X3.5 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 (#997～#999 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: If G31 is not used, X3.5 input interface is used to the common input interface. The skip signal is valid, 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.

3.16.3 Parameters Relevant to G31: Refer to Ⅲ INSTALLATION & CONNECTION, Section 3.2.15.

3.17 Automatic Tool Offset G36, G37

Command format: G36 X__;
G37 Z__;

Command function: when the command is executed to make the tool move to the measured position, the CNC automatically measures the difference between the current actual coordinates and the command coordinates to be the tool offset value. The function is used to the automatic toolsetting.

Explanations: X absolute coordinate (only used to G36), Z absolute coordinate (only used to G37);
Non-modal G command (00 group);
Cancel the tool nose radius compensation before using it;
Only use the absolute programming;
Define the workpiece coordinate system before using the command;
Specify the tool number and tool compensation number before using the command;

3.17.1 Signals Relevant to G36, G37 Automatic Tool Offset:

Measured position arrival signal:

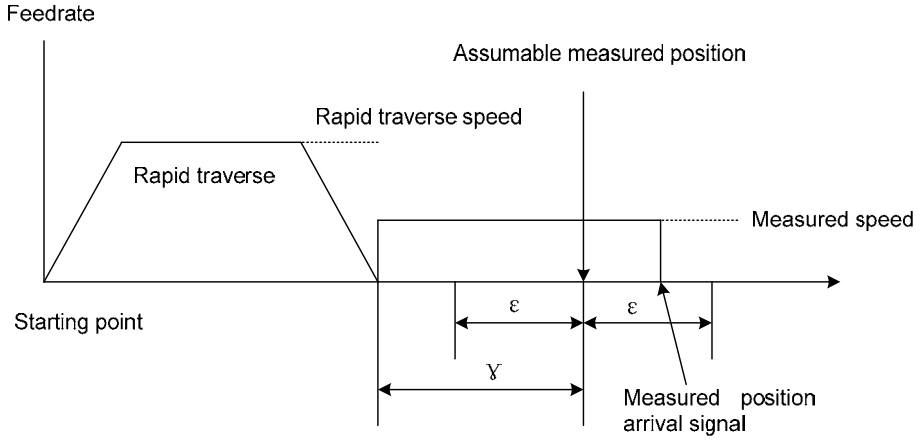
XAE(X3.6) ————corresponding to G36

ZAE(X3.7) ————corresponding to G37

Type: input signal

Function: when the position measured by the program command is different from that where the tool actually reaches (i.e. at the time, the measured position arrival signal becomes “1”), the difference of the coordinates is added to the current tool compensation value to update the compensation value. When G36X_(or G37Z_) is executed, the tool firstly rapidly traverses to the

position measured by the command, and decelerates and temporarily stop the position before the measured position, and then, reaches to the measured position at the speed set by No.141. When the measured position arrival signal corresponding to G command becomes “1”, and the tool is in the measured position range ±ε, CNC updates the offset compensation value and ends the block. When the measured position arrival signal does not become “1”, and after the tool reaches the measured position distance ε, the CNC alarms, ends the block and does not update the offset compensation value.

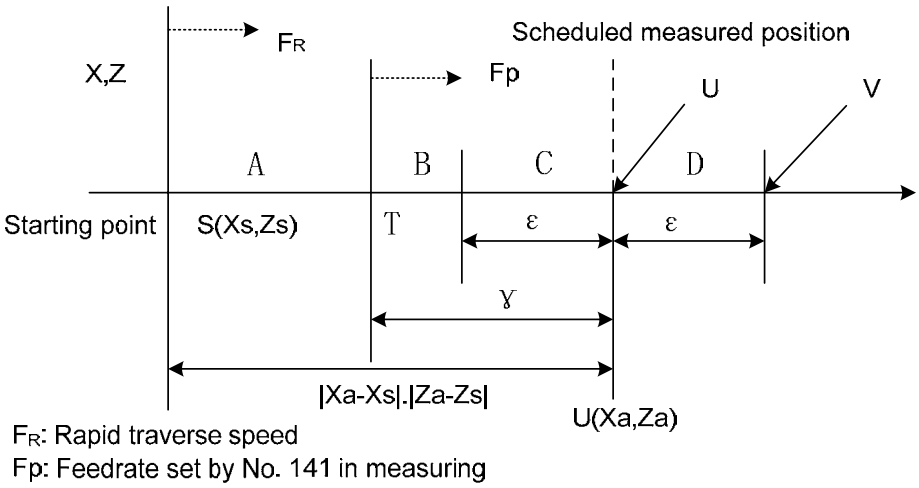


3.17.2 Parameters Relevant to G36, G37: Refer to INSTALLATION, Section 3.2.16.

3.17.3 G36, G37 Automatic Tool Offset Command Use

Feedrate and alarm

From the initial position to the measured position specified by Xa or Za in G36 or G37, the tool rapidly traverses to A zone and stops at T point (Xa–γx or Za–γz) , and then traverses to B, C and D at the feedrate set by No.141. The system alarms when the tool traverses in B zone and the measured point arrival signal of the end point is set to. CNC alarms when the tool stops at V point.

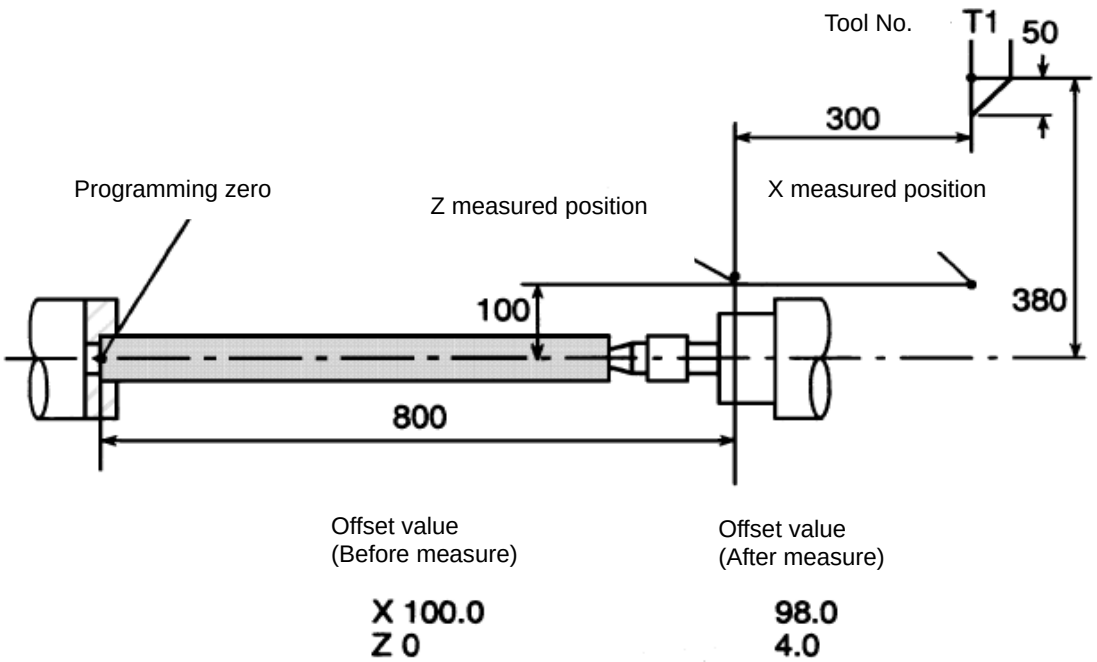


Example:

G50 X760 Z1100; create the workpiece coordinate system

T0101; define No. 1 tool and execute its tool compensation

G36 X200;	traverse to X toolsetting point (X toolsetting point coordinate: 200)
T0101;	execute X tool compensation again
G00 X204;	retract a little
G37 Z800;	traverse to Z toolsetting point (Z toolsetting point coordinate: 800)
T0101;	execute Z tool compensation again and the toolsetting is completed
M30;	



Note: Measure position arrival signal is enabled, the feed of CNC stop axis is immediately stopped (regardless of the acceleration or deceleration treatment). Therefore, in order to guarantee the accuracy of the stop position, the measure speed of the G36 or G37 should be set as low as possible; it is clamped at the speed when the federate exceeds 200mm/min.

3.18 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;

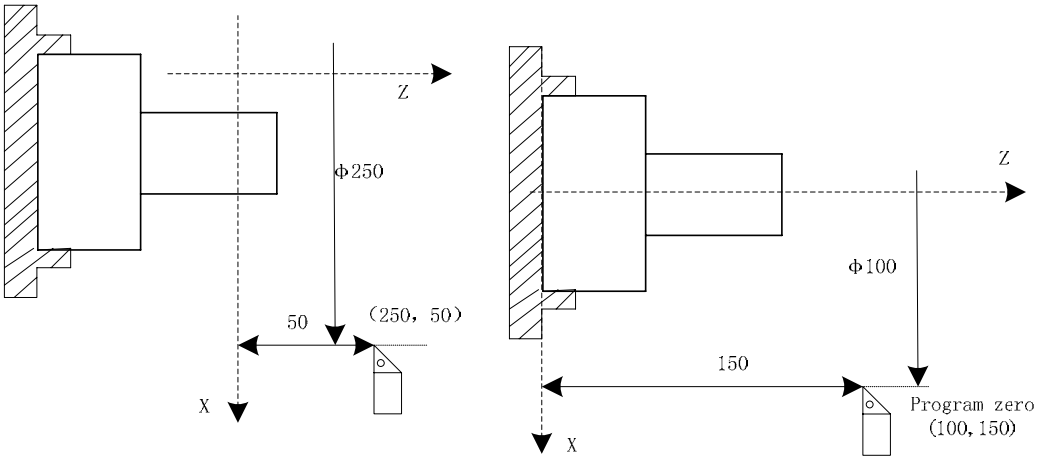
Z: New absolute coordinates of current position in Z direction;

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:



Before setting coordinate system with G50 After setting coordinate system with G50

Fig.3-54

As Fig.3-54, 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.

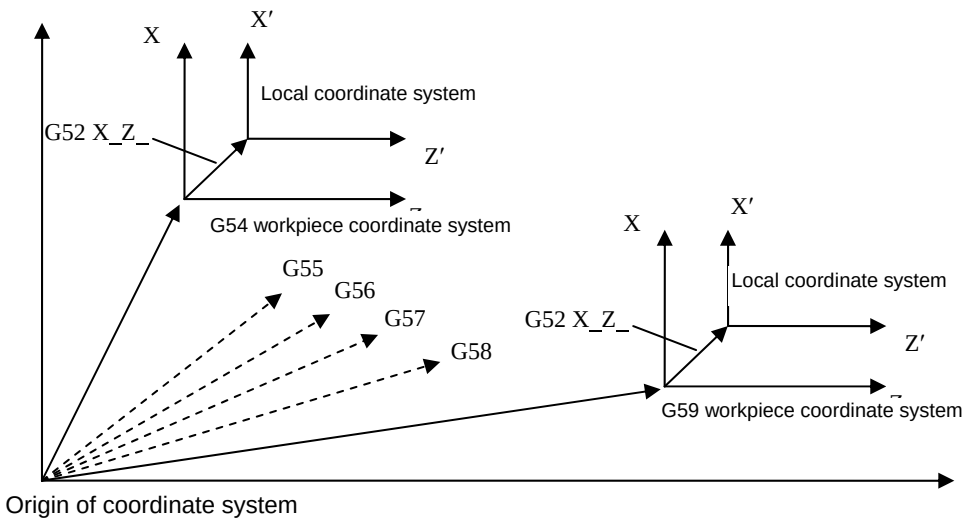
Current tool compensation state	Executing motion command	Coordinate value after executing G50 X20 Z20	No. 01 tool compensation value
T0100 or T0101	G0 X_ Z	X: 20 Z: 20	X: 12 Z: 23
	Motion command not executed	Coordinate value after executing G50 X20 Z20	
	※※※	X: 8 Z: -3 or X: 32 Z: 43	

3.19 Local Coordinate System

When the programming is executed in a workpiece coordinate system, a sub workpiece coordinate can be created in the workpiece coordinate system, which is called as a local coordinate system.

Command format: G52 X(U)__ Z(W)__

Command function: when G52 is executed, all workpiece coordinate system can (G54～G59) set a local coordinate system. Origin of each local coordinate system is the specified position X(U)__ Z(W)__ of each workpiece coordinate system. Corresponding relation of workpiece coordinate systems is shown below:



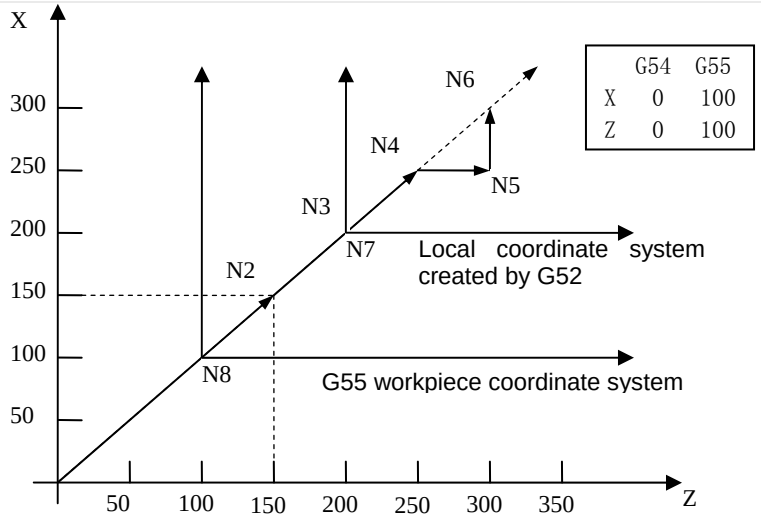
Explanation: G52 is in Group 00, and is simple. X(U)__ Z(W)___is the position of origin of the specified local coordinate system in the current workpiece coordinate system. Results in absolute or incremental execution are the same.

Notes:

1. **Setting of local coordinate system does not change a workpiece coordinate system and a machine coordinate system.**
2. **Temporarily cancel tool nose radius compensation when G52 is executed.**
3. **After G52 is specified, the local coordinate system does not remain valid until the next G52 is specified. G52 does not perform movement when it is specified.**
4. **When a local coordinate system is cancelled, its zero and that of workpiece coordinate system are consistent, i.e., command G52 X0 Z0 or G52 U0 W0.**
5. **When G50 sets a workpiece coordinate system, local coordinate systems of all workpiece coordinate systems which axes are specified are cancelled. The local coordinate systems which coordinate axes are not specified are not cancelled when not all axes' coordinate value are specified.**
6. **Whether the local coordinate system is cancelled is determined by a parameter when reset, machine zero return or end of program.**

Example:

```
N1 G28 X0 Z0;
N2 G55 G00 X50 Z50;
N3 G52 X100 Z100;
N4 G00 X50 Z50;
N5 G01 Z100 F100;
N6 X100;
N7 G52 X0 Z0;
N8 G00 X0 Z0;
N9 M30;
```



In N3 block, a local coordinate system is created according to G55 workpiece coordinate system, and it is cancelled in the block N7.

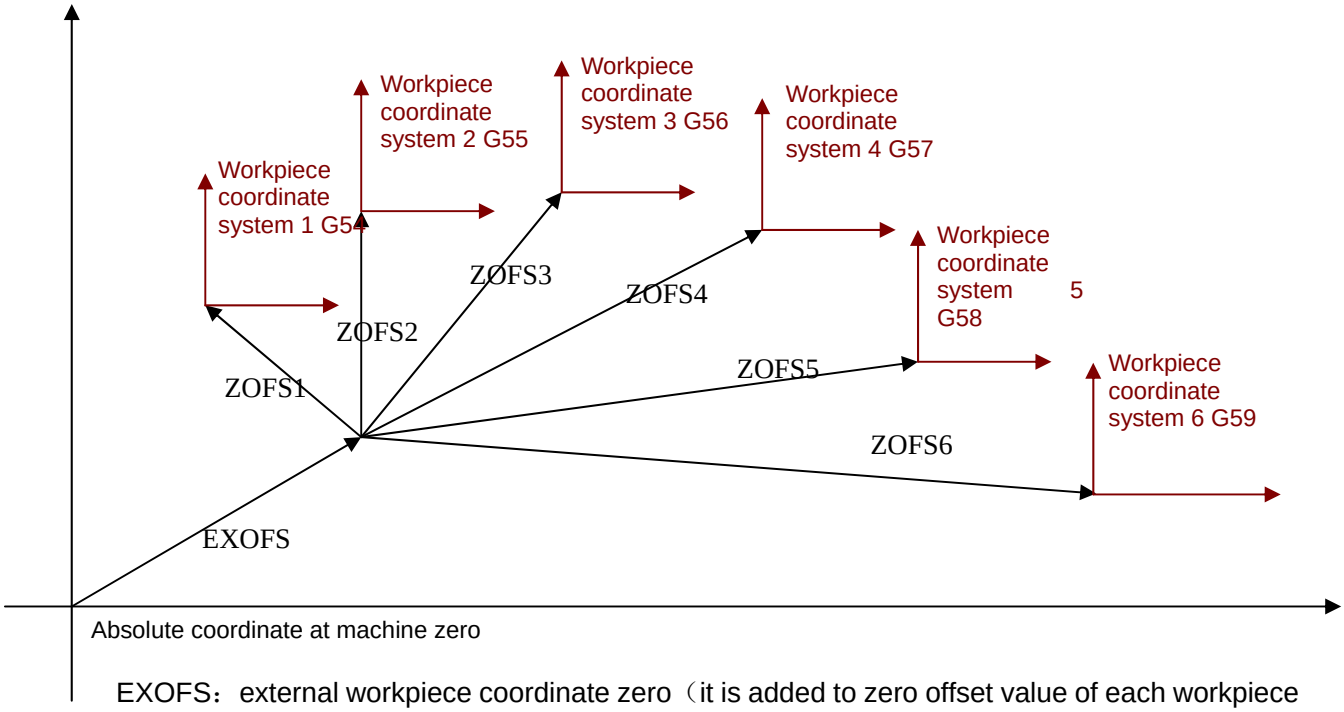
3.20 Workpiece Coordinate System G54～G59

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:



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)

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);

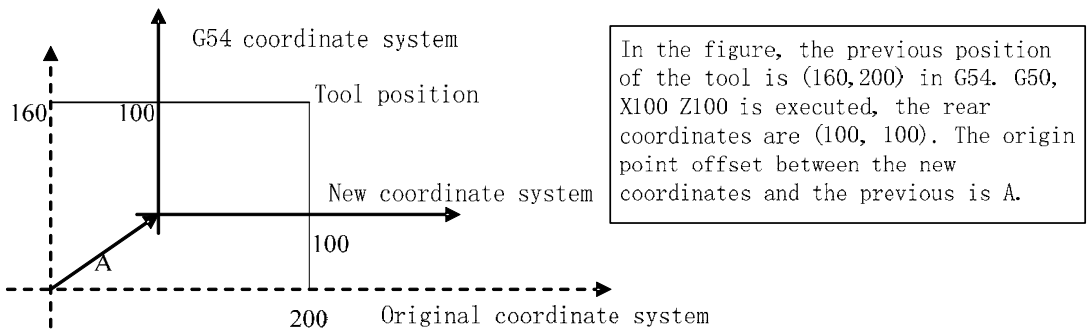
Absolute coordinate values after zero return in G55: 20 - (10 + 5) = 5;

Example 3：

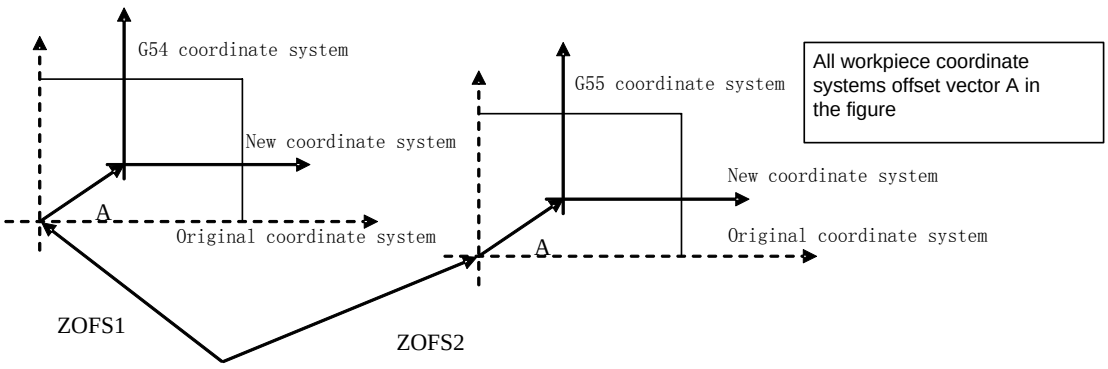
Current absolute coordinates (10,10); EXOF=(5,5) ; ZOFS1=(-10,-10); ZOFS2 = (-30,-30); absolute coordinate values from G54 to G55: 10-（-30-（-10））= 30

Notes：

1. **When №12.7（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.**
2. **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.**
3. **The workpiece coordinate system offset is immediately valid after values of EXOFS and ZOFSn are modified.**
4. **Whether the incremental coordinates include offset values of the coordinate system is set by№05.1（PPD）.**
5. **In tool nose radius compensation mode, compensation is cancelled temporarily in G54~G59, and is recovered in the next movement command.**
6. **When G54~G59 and G50 are in the same block, G54~G59 are invalid, but G50 is valid.**
7. **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:



3.21 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.

3.21.1 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, $R \le |U/2|$; 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 traverses 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 ①), 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.

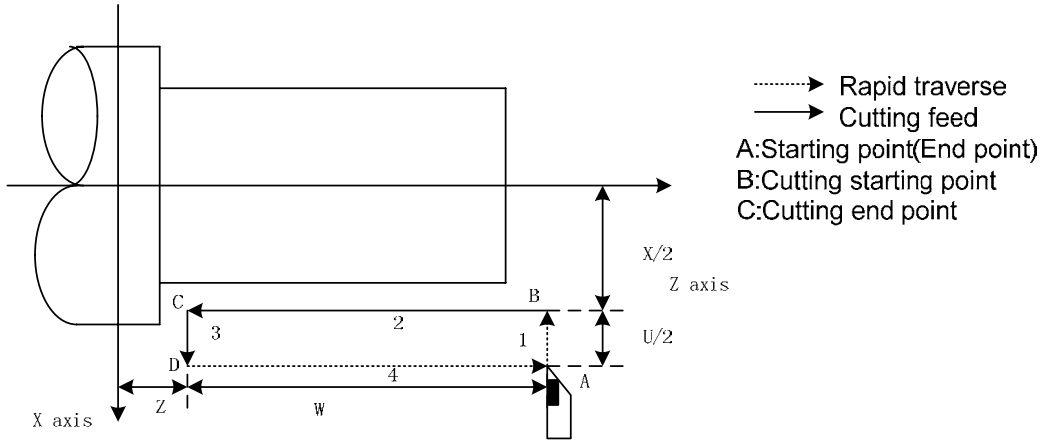


Fig. 3-55

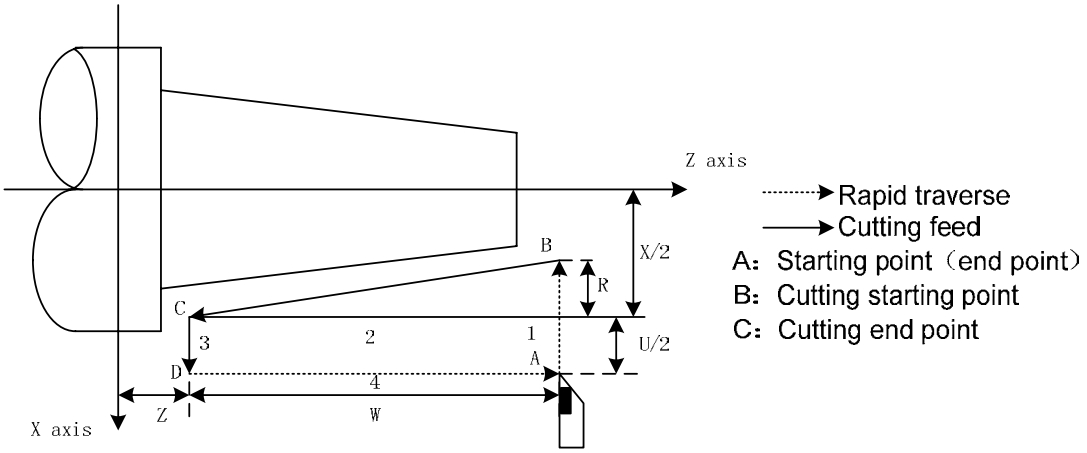
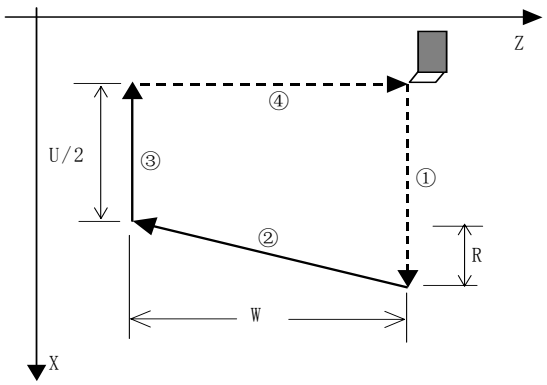


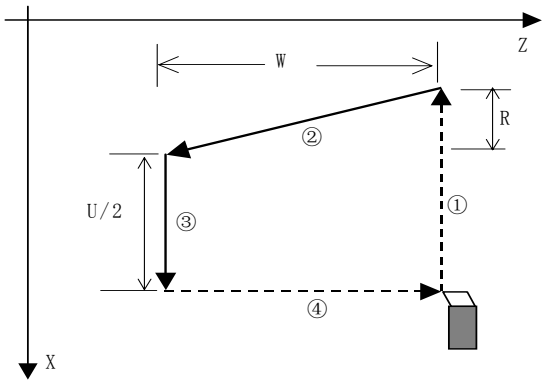
Fig. 3-56

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



3） U>0，W>0，R<0 ,|R|≤|U/2|

4） U<0，W>0，R>0,|R|≤|U/2|

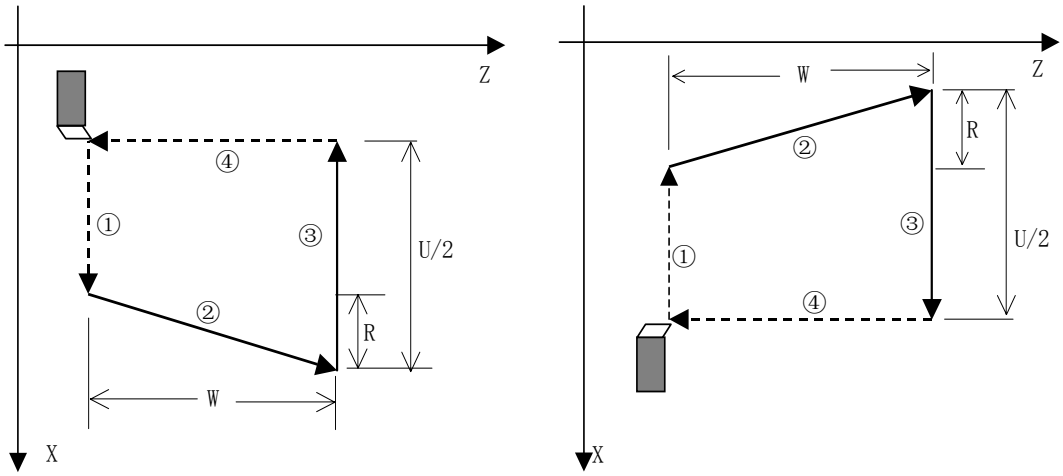


Fig. 3-57

Example: Fig. 3-58, rod Φ125×110

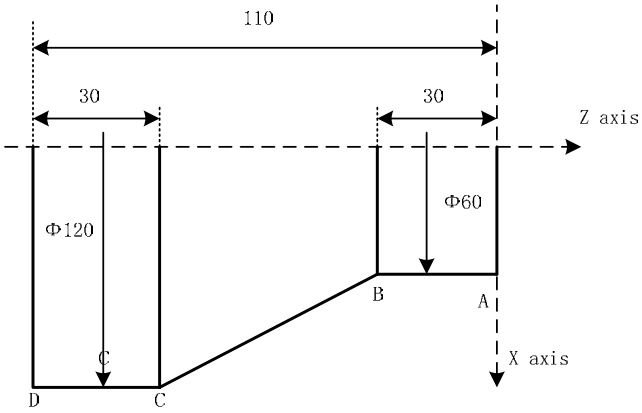


Fig. 3-58

```
Program : O0002;
M3 S300 G0 X130 Z3;
G90 X120 Z-110 F200;          (A→D, cut Φ120)
X110 Z-30;
X100;
X90;                          （A→B，6 times cutting cycle Φ60, increment of 10mm）
X80;
X70;
X60;
G0 X120 Z-30;
G90 X120 Z-44 R-7.5 F150;
Z-56 R-15
Z-68 R-22.5                   （B→C，4 times taper cutting）
Z-80 R-30
M30;
```

3.21.2 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, $|R| \le |W|$.

Radial linear cutting is shown in Fig. 3-21, radial taper cutting is as Fig. 3-60.

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.

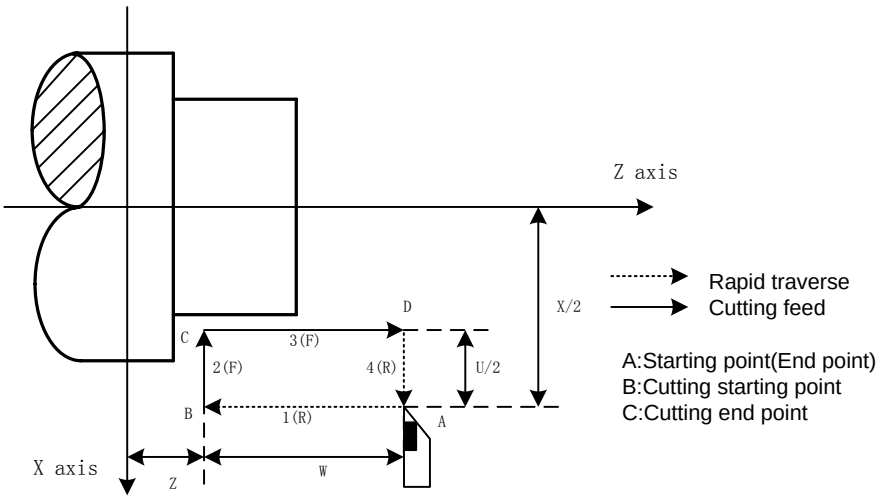


Fig. 3-59

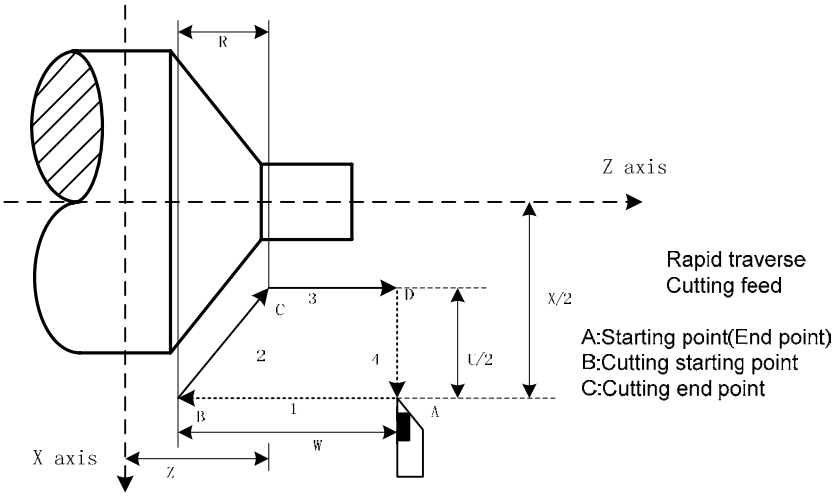
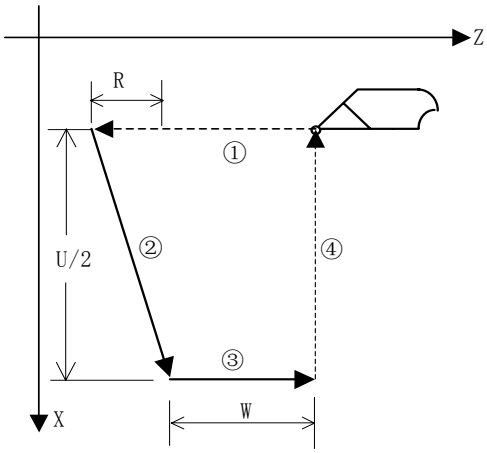


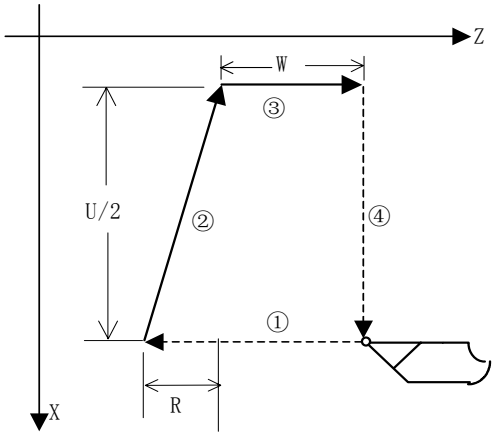
Fig. 3-60

Cutting path: Relative position between cutting end point and starting point with U, W is shown in Fig.3-61:

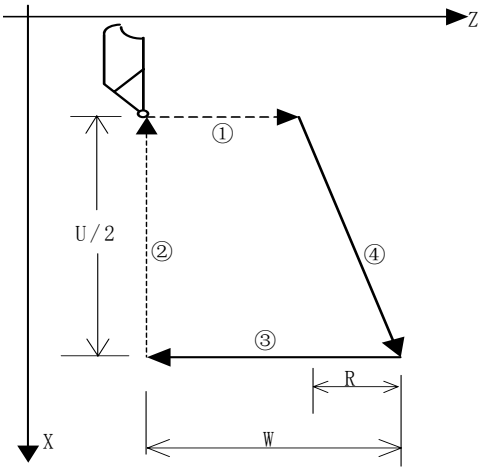
1) U>0 W<0 R<0



2) U<0 W<0 R<0



3）U>0 W>0 R<0 （|R|≤|W|）



4）U<0 W>0 R<0 （|R|≤|W|）

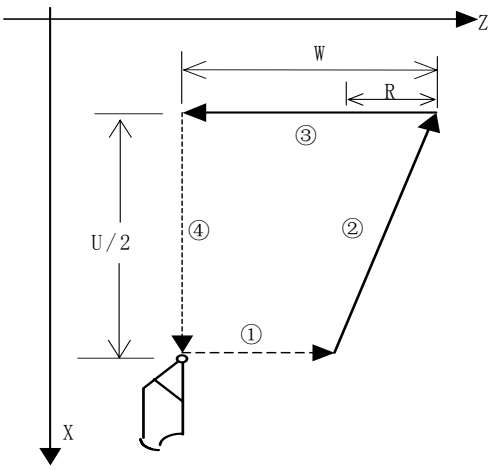


Fig. 3-61

Example: Fig. 3-62, rod Φ125×112

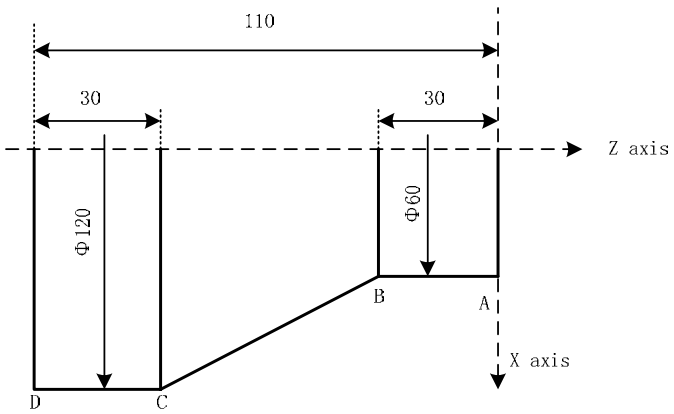


Fig. 3-62

```
Program: O0003;
        G00 X130 Z5 M3 S1;
        G94 X0 Z0 F200          End face cutting
        X120 Z-110 F300;        （Outer cutting Φ120）
        G00 X120 Z0
        G94 X108 Z-30 R-10
        X96 R-20
        X84 R-30                （C→B→A，cutting Φ60）
        X72 R-40
        X60 R-50;
        M30;
```

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

3.22 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.

The G76 multi-thread cutting cycle code will describe in the Section of the Thread Function.

3.22.1 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)

```
N(ns) G0/G1 X(U)..;                N(ns) G0/G1 X(U) Z(W)...;
 . . . . . . . . ;                  . . . . . . . . ;
  . . . . F;                          . . . . F;
  . . . . S;        (3)Type I            . . . . S;        (3)Type II
  . . . .                              . .
N(nf). . . . . ;                   N(nf). . . . . ;
```

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 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 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 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'→C' point.

Δd: It is each travel of X tool infeed in roughing, its value: 0.001~99.999（IS_B）/0.0001~99.9999（IS_C）(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). 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 of X tool retraction in roughing its value: 0~99.999（IS_B）/0~99.9999（IS_C）(unit: mm/inch,radius value) without sign, and the direction of tool retraction is opposite to that of tool infeed, the command value e is reserved 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 -99999.999~99999.999(IS_B) /-9999.9999~9999.9999(IS_C) (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 Δu=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 -99999.999~99999.999(IS_B) /-9999.9999~9999.9999(IS_C) (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 Δw=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. 3-63)

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→B point;

③ Z executes the cutting feeds to the roughing path, and its direction is the same that of Z coordinate A→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 (Δd+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 (Δd+e)again, execute ③; after executing the tool infeed (Δd+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.

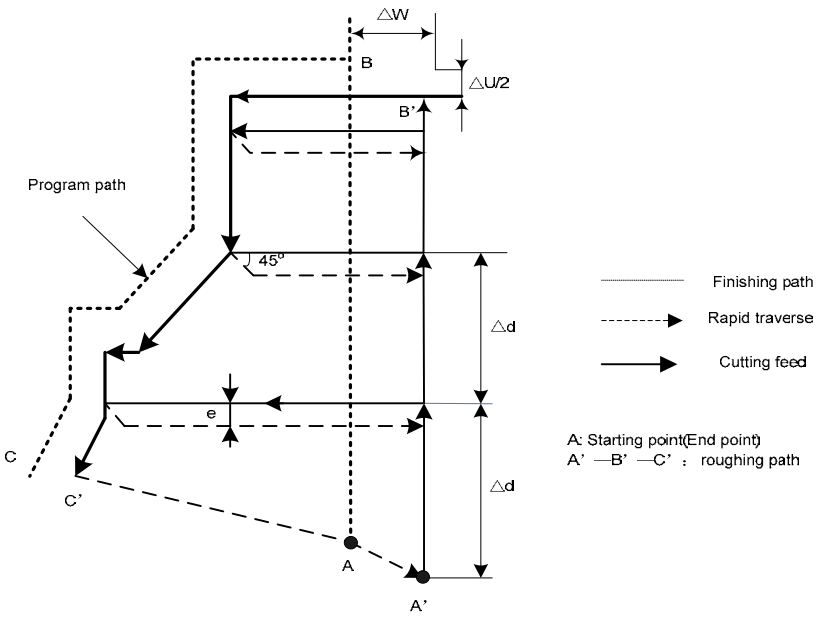


Fig. 3-63 G71 cycle path

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. 3-64: B→C for finishing path, B'→C' for roughing path and A is the tool start-up point.

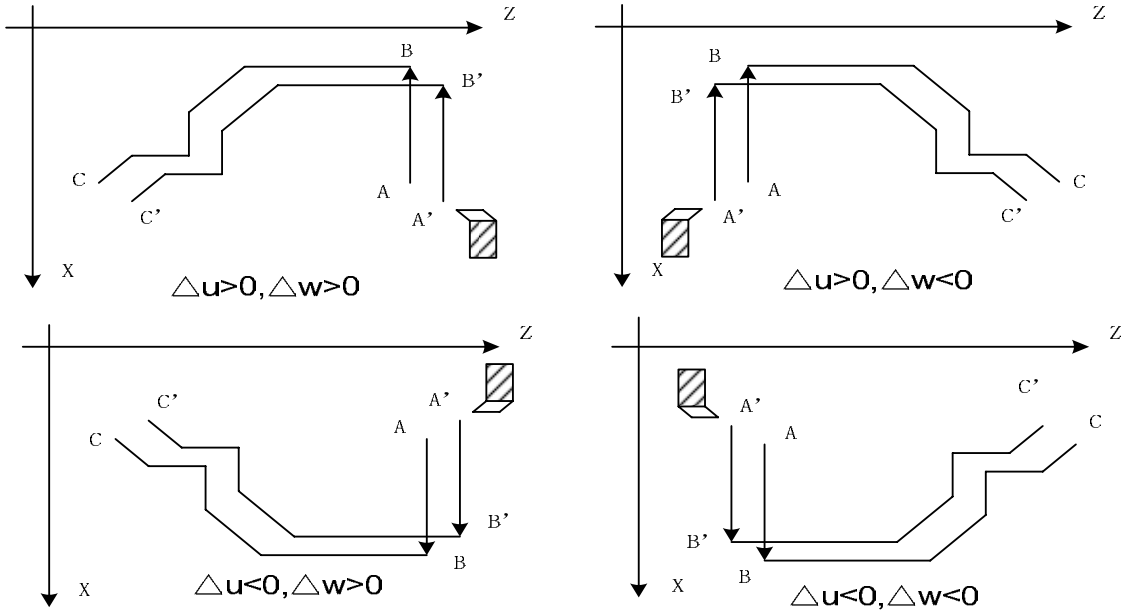


Fig.3-64

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:

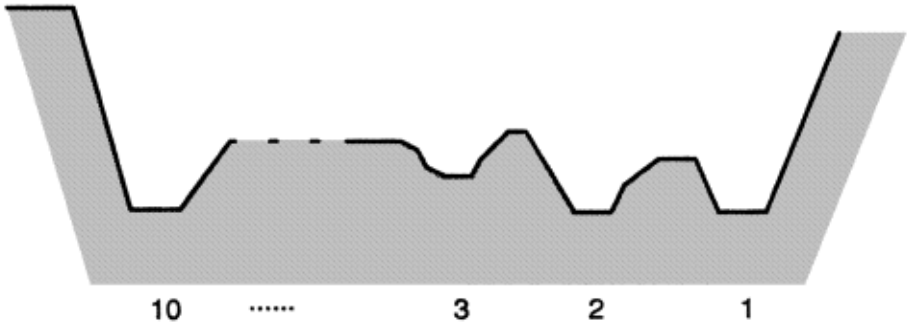
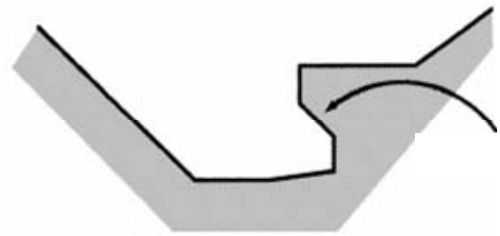


Fig. 3-65（type Ⅱ）

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



Z axis is not the monotonous change

Fig. 3-66（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:

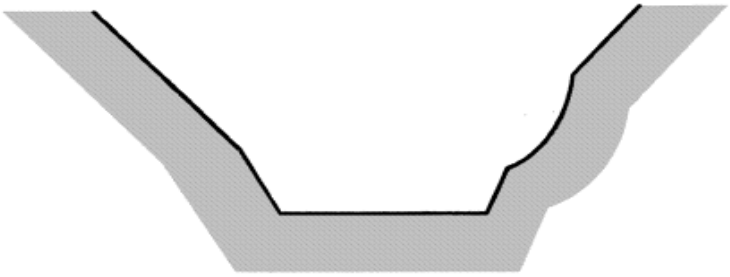


Fig. 3-67

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

e is set by parameter

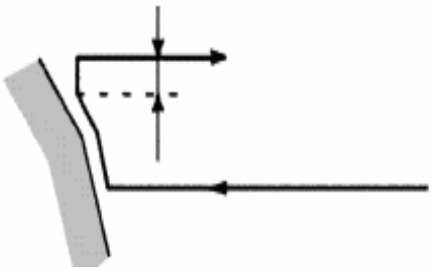


Fig. 3-68（type Ⅱ）

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

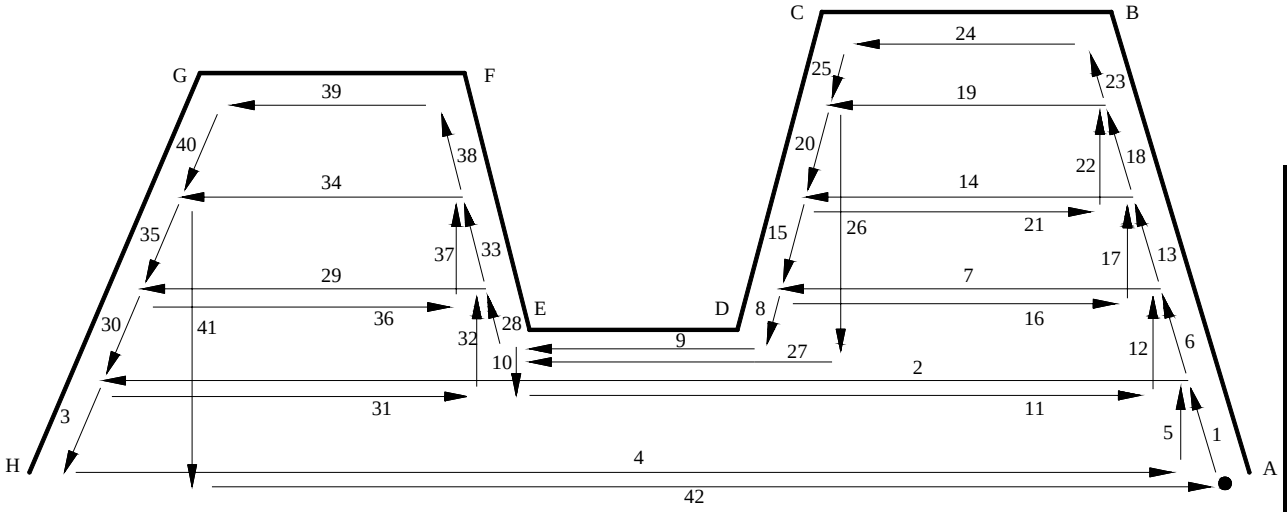


Fig. 3-69（typeⅡ）

Notes：

1. 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;
2. 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;
3. 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;
4. Some workpiece without remarking the type I or the type II adapts the both;
5. 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;
6. 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;
7. 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;
8. 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);
9. In ns～nf ,the program block quantity cannot exceed 100;
10. G96, G97, G98, G99, G40, G41, G42 are invalid when G71 is executed, and are valid when G70 is executed;
11. 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;

12. When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path;
13. △d,△u are specified by the same U and different with or without being specified P, Q commands;
14. There are no the same block number in ns~nf when compound cycle commands are executed repetitively in one program;
15. The tool retraction point should be high or low as possible to avoid crashing the workpiece.

Cutter radius compensation:

When the cutter radius compensation should be used in the compound cycle code (G70, G71, G72 or G73), the G41/G42 code should be specified at the 1st block (ns block) of the finish-turning path; otherwise, the cutter compensation will not be used in the compound cycle; it is not necessary to specify the G40 in the finish-turning path. The cutter radius compensation will authomatically cancel when the compound cycle is completed. Actually, when the cutter radius compensation is used in the compound cycle code, the CNC system is performed the cutter nose radius offset for the finish-turning path instead of entering the state of the cutter radius compensation. Therefore, the cutter radius compensation state on the position interface will not change.

The cutter radius compensation genterates before entering the compound cycle, temporarily cancel the cutter radius compensation; restore the cutter radius compensation when the cycle end tool leaves from the cycle start; refer to the following schematic:

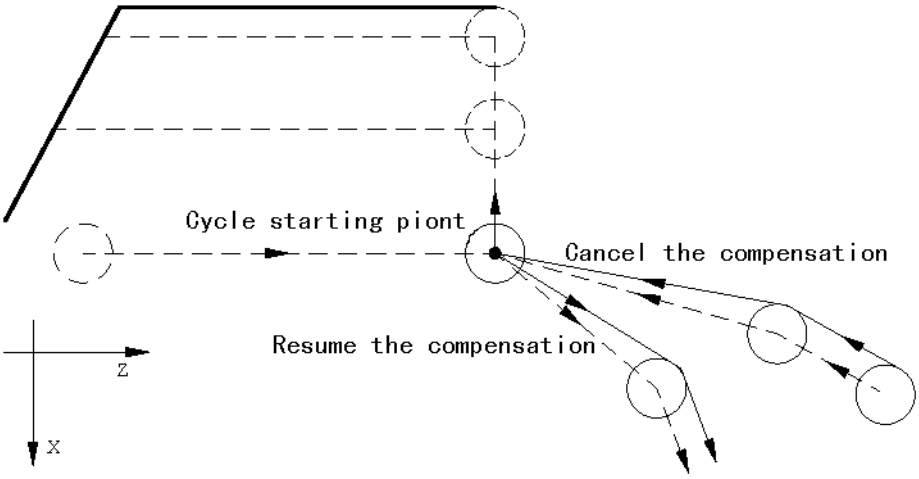


Fig. 3-70

When the compound the cycle is performed in the rough machining, perform the infeed along with the cycle start and 1st point offset vector point of the finish-turing path, and its feed value is △d; refer to the following figure:

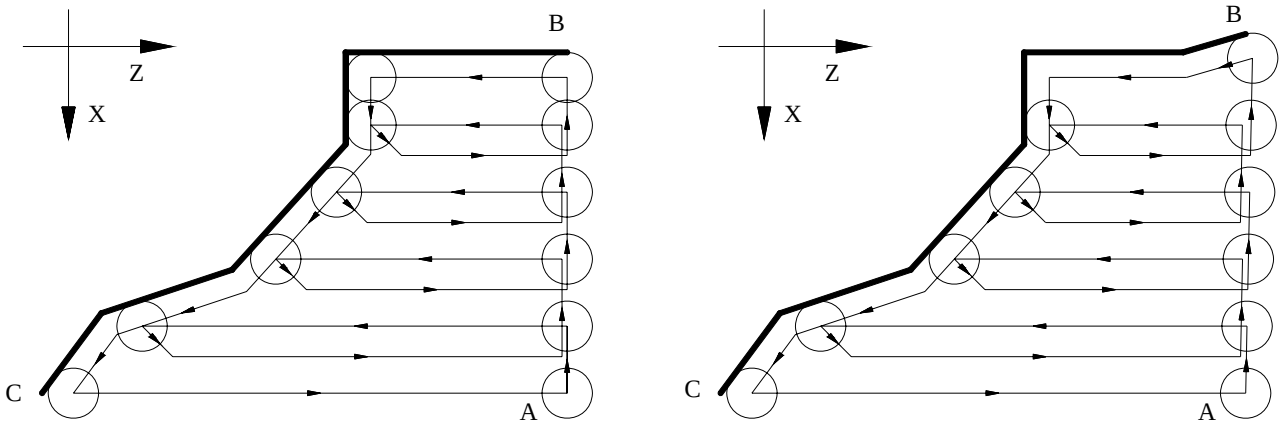


Fig. 3-71 G71 cycle I start-up

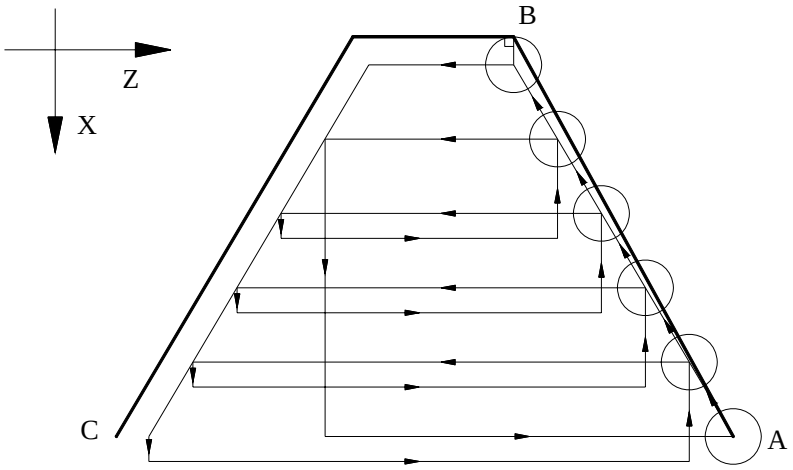


Fig 3-72 G71 cycle II start-up

Example：Fig. 3-73 (Type I)

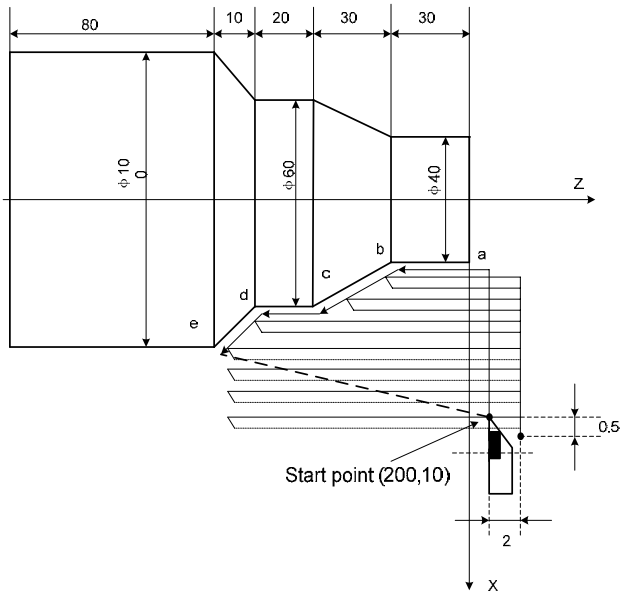


Fig. 3-73

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)
G01 Z-30 F100 ;              (a→b)   ⎫
X60 W-30;                    (b→c)   ⎬ a→b→c→d→e blocks for finishing path
W-20;                        (c→d)   ⎪
N120 X100 W-10;              (d→e)   ⎭
G70 P80 Q120;                (a---e blocks for finishing path)
M30;                         (End of block)
```

3.22.2 Radial Roughing Cycle G72

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

G72 P(ns) Q(nf) U(Δu) W(Δw) K0/1 J0/1; (2)

```
N(ns) G00(G01) X(U)_;   ⎫                N(ns) G00(G01) X(U)_ Z(W)_ ⎫
. . . F;                ⎪                . . . F;                   ⎪
. . . S;                ⎬ (3)type I      . . . S;                   ⎬ (3)type II
. . .                   ⎪                . . .                      ⎪
N(nf). . . . . ;        ⎭                N(nf). . . . . ;           ⎭
```

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（IS_B）/0.0001~99.9999（IS_C）(unit:

mm/inch) without sign symbol, and the direction of tool infeed is determined by ns block traverse direction. The value of system parameter No.051 is regarded as the tool infeed clearance when W(Δd) is not input.

e: it is Z tool retraction clearance in roughing, its value: 0~99.999（IS_B）/0~99.9999（IS_C）(unit: mm) without sign symbol, and the direction of tool retraction is opposite to that of tool infeed, the specified value e is reserved 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: -99999.999~99999.999(IS_B）/-9999.9999~9999.9999(IS_C) (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: -99999.999~99999.999(IS_B）/-9999.9999~9999.9999(IS_C) (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).

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: 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:

①X rapidly traverses to A' from A point, X travel is Δu, and Z travel is Δw;

②X 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;

③X executes the cutting feeds to the roughing path, and its direction is the same that of X coordinate B→C point;

④X, Z execute the tool retraction e (45°straight line)at feedrate, the directions of tool retraction is opposite to that of tool infeed ;

⑤X rapidly retracts at rapid traverse speed to the position which is the same that of Z coordinate;

⑥After Z tool infeed (Δd+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 (Δd+e)again, ③ is executed; after the tool infeed (Δd+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 ⑦ is executed;

⑦Cutting feed from B' to C' point along the roughing path;

⑧Rapidly traverse to A from C' point and the program jumps to the next clock following nf block after G71 cycle is completed.

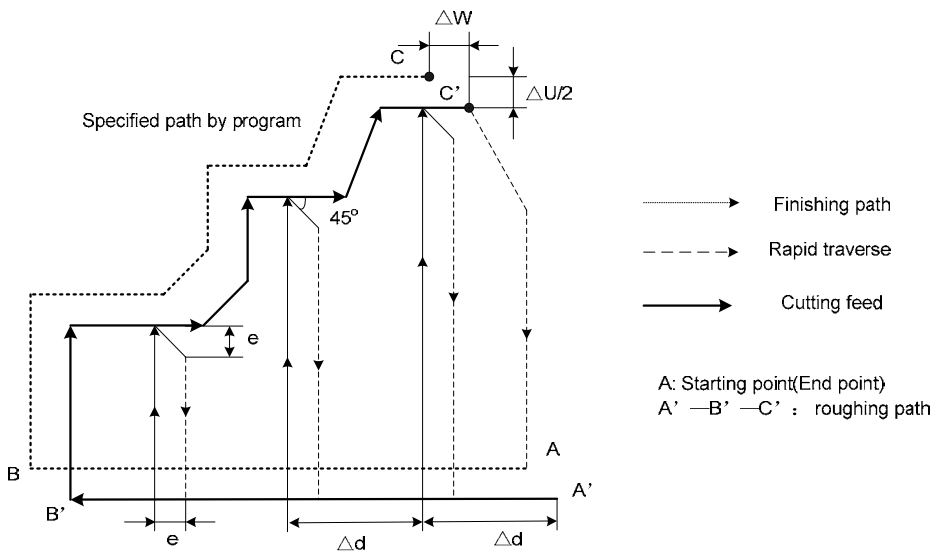


Fig. 3-74

Command specifications:

1. 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;
2. 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;
3. The dimensions in X, Z direction must be changed monotonously (always increasing or reducing) for the finishing path;
4. 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);
5. G96, G97, G98, G99, G40, G41, G42 are invalid when G72 is executed, and are valid when G70 is done;
6. In the ns～nf, the program block quantity cannot exceed 100;
7. 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;
8. When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path;
9. △d, △w are specified by the same W and different with or without being specified P, Q commands;
10. There are no the same block number in ns~nf when compound cycle commands are executed repetitively in one program;
11. 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.

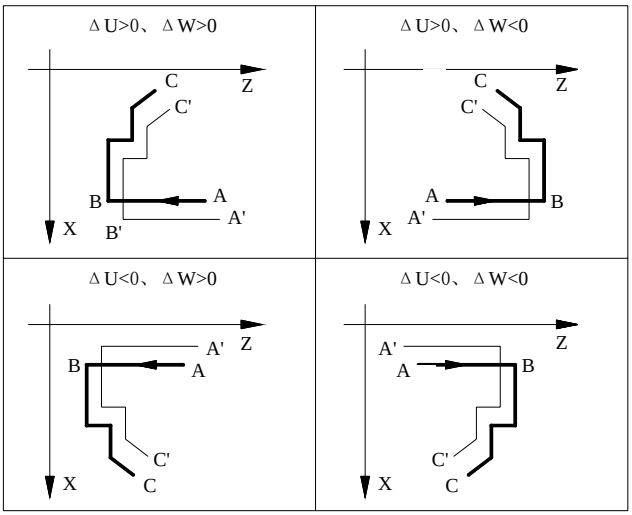


Fig.3-75

The selection both type I and II

G72 owns type I and II. It is necessary to use the type II when the slot in the fir -turining shape.

The tool-retraction operation of the type I and II machining are different after the rough-turning is performed towards to the 2nd axis of the panel (It is the X axis on the ZX panel). Type I executes the tool-retraction along with the 45°, and the type II is performed the cutting along with the finish-turning. When there is no slot in the finish-turing shape, flexibly use based upon the tool-retraction method.

Select the type I or II in the 1st block (Block number is ns) of the finish-turning path.

1) Type I selection

Only specify the command of the 1st axis of the panel (It is the Z axis on the ZX panel). There is no command of the 2nd axis (It is the X axis on the ZX panel) of the panel.

2) Type II selection

Specify the command of the 2nd axis (It is X axis on the ZX panel) and 1st axis (It is Z axis on the ZX panel) on the panel. If the movement does not on the 2nd axis of the panel instead of using the type II, the increment code specified the movement value is 0, which is programmed (It is U0 on the ZX panel).

The distinction between the G72 type II and G71 type II:

The differences from the G71 are shown below:

1) G72 moves the cutting workpiece by the 2nd axis paralleled with the panel (It is the X axis on the ZX panel).

2) It is not necessary that the shape is simply added or reduced of the 1st axis (It is the Z axis on the ZX panel) on the panel; the sunken part (slot) can be set. However, there is not alternative other than the simply change along with the 2nd axis (It is X axis on the ZX panel) direction on the panel.

3) In the finish-turning, the block parallels with the 2nd axis on the panel which is regarded as botton of the slot.

4) Tool is temporarily returned to the start after the overall rough machining cuttings of the 2nd axis on the panel are performed, and then the rough machining cutting is executed.

G72 code cutter radius compensation function:

Refer to the partial description of the G71 code cutter radius compensation function.

G72 code machining example (Type I):

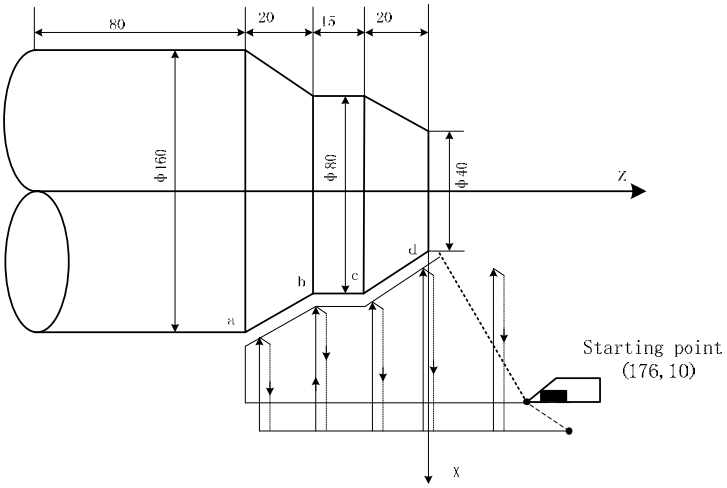


Fig.3-76

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 Q020 M30;         (Finishing a—d)
```

3.22.3 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;                                (3)
. . . . . ;
.
N__(nf) . . . . . . ;
```

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.3-77.

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 (Δi×2+Δu, Δw+Δk) (diameter programming) , the coordinates offset value of the last cutting compared to finishing path is(Δu, Δw) , the coordinates offset value of each cutting compared to the previous one is as follows:

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

Δi: It is X tool retraction clearance in roughing, and its range is ±99999999× 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 |Δi| in roughing, and X cutting direction is opposite to the sign of Δi: Δi ＞0, the system executes X negative cutting in roughing.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 -99999.999～99999.999(IS_B）/-9999.9999～9999.9999(IS_C) (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 |Δk| in roughing, and Z cutting direction is opposite to the sign of Δk: Δk＞0, the system executes Z negative cutting in roughing.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.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 -99999.999~99999.999(IS_B）/-9999.9999~9999.9999(IS_C) (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 A_1 compared to A. Δu＞0,it is the offset of the last X positive roughing path compared to finishing path. The system defaults Δu=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-99999.999~99999.999(IS_B) /-9999.9999~9999.9999(IS_C) (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 A_1 compared to A. Δw＞0,it is the offset of the last X positive roughing path compared to finishing path. The system defaults Δw=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. 3-77)

① $A \rightarrow A_1$: Rapid traverse;

② First roughing $A_1 \rightarrow B_1 \rightarrow C_1$:

$A_1 \rightarrow B_1$: Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;

$B_1 \rightarrow C_1$: Cutting feed.

③ $C_1 \rightarrow A_2$: Rapid traverse.

④ Second roughing $A_2 \rightarrow B_2 \rightarrow C_2$:

$A_2 \rightarrow B_2$: Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;

$B_2 \rightarrow C_2$: Cutting feed.

⑤ $C_2 \rightarrow A_3$: Rapid traverse:

……

No. n times roughing, $A_n \rightarrow B_n \rightarrow C_n$:

$A_n \rightarrow B_n$: ns Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;

$B_n \rightarrow C_n$: Cutting feed.

$C_n \rightarrow A_{n+1}$: Rapid traverse;

……

Last roughing, $A_d \rightarrow B_d \rightarrow C_d$:

$A_d \rightarrow B_d$: Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;

$B_d \rightarrow C_d$: Cutting feed.

$C_d \rightarrow A$: Rapid traverse to starting point;

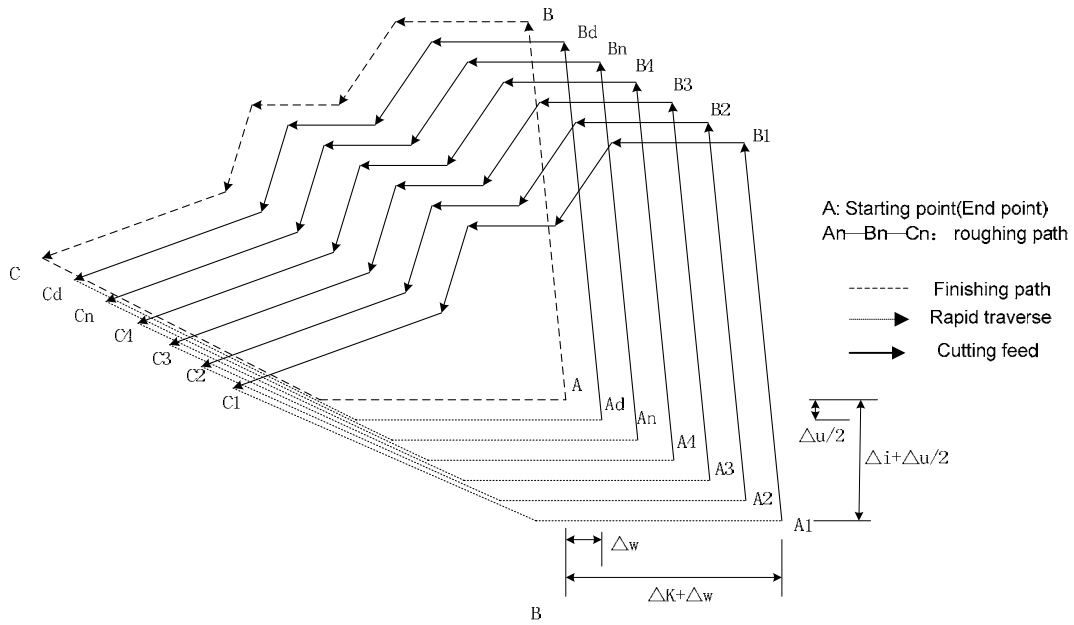


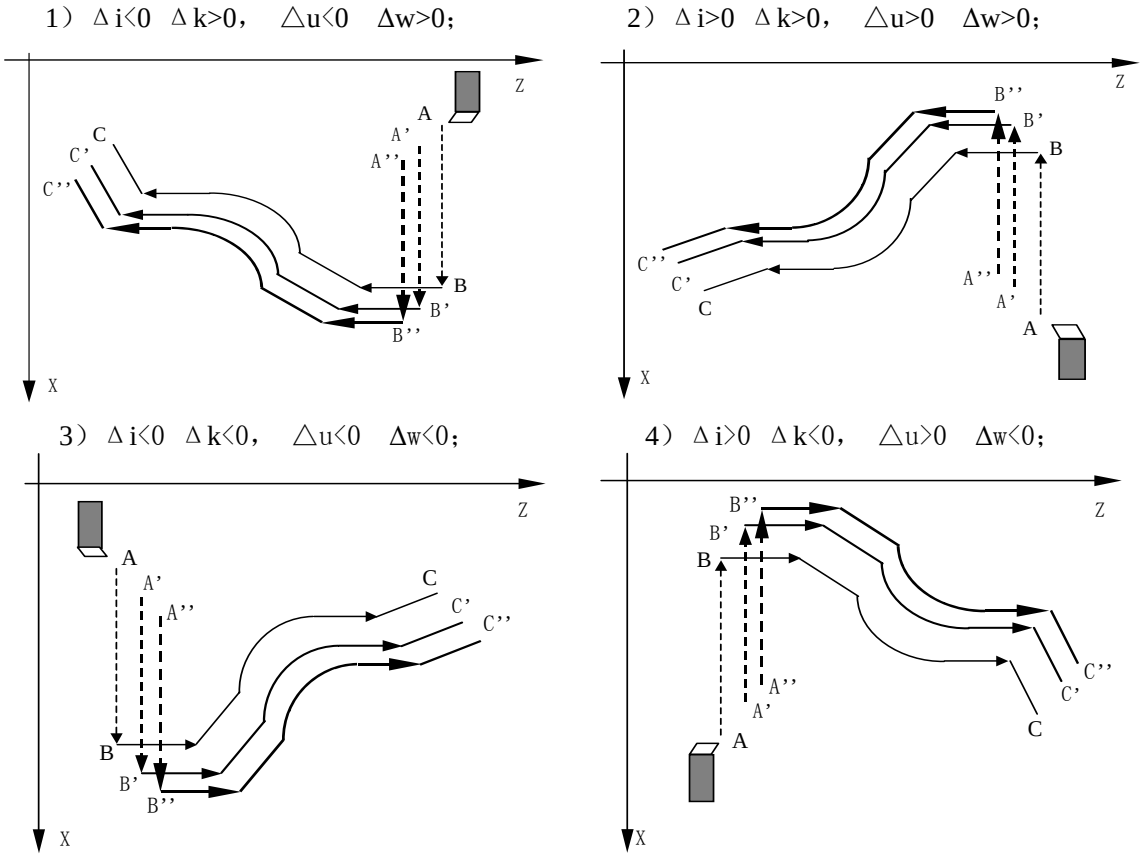
Fig. 3-77 G73 path

Command specifications:

1. 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.
2. 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.
3. There are only G00, G01 in ns block.
4. 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) .
5. In ns～nf, the program block quantity cannot exceed 100;
6. G96, G97, G98, G99, G40, G41, G42 are invalid when G73 is executed, and are valid when G70 is executed.
7. 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.
8. When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path.
9. △i, △u 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.
10. G73 cannot be executed in MDI, otherwise, the system alarms.
11. There are no the same block number in ns~nf when compound cycle commands are executed repetitively in one program.
12. 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.



G73 code cutter radius compensation function:

Refer to the parital description of the G71 code cutter radius compensation.

G73 code machining example:

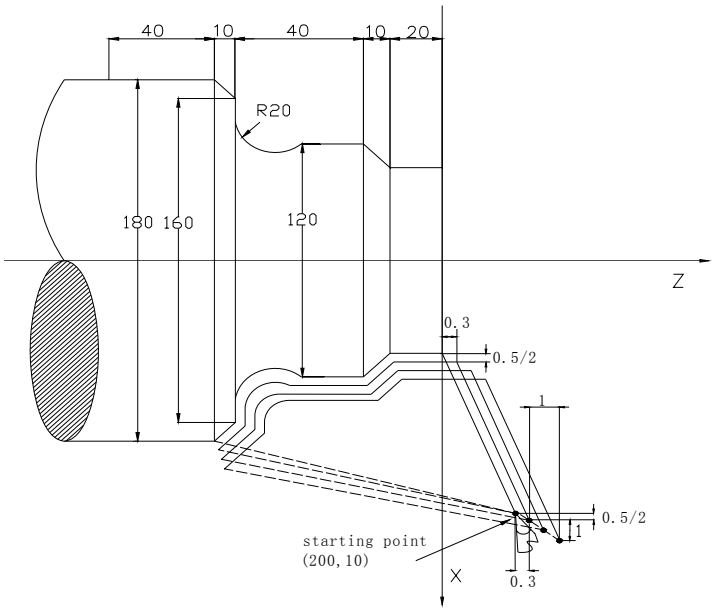


Fig.3-78

```
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 ;                          Blocks for finishing
G02 X160 W-20 R20 ;
N19 G01 X180 W-10 ;
G70 P14 Q19 M30;                (Finishing)
```


3.22.4 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                Blocks for finishing path
          .
          .
    N__(nf)……
          . . .
    G70 P(ns) Q(nf);
```

Command specifications:

1. ns～nf blocks in programming must be followed G70 blocks.
2. F, S, T in ns～nf blocks are valid when executing ns～nf to command G70 finishing cycle.
3. G96, G97, G98, G99, G40, G41, G42 are valid in G70;
4. 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.
5. When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path.

6. There are no the same block number in ns~nf when compound cycle commands are executed repetitively in one program.
7. In ns～nf, the program block quantity cannot exceed 100;
8. The tool retraction point should be high or low as possible to avoid crashing the workpiece.

3.22.5 Axial Grooving Multiple Cycle G74

Command format: G74 R(e) ;

G74 X(U) __ Z(W) __ P(Δi) Q(Δk) R(Δd) 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……), 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……), 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……), 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……), 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(IS-B)/0~99.9999(IS-C) (unit:mm,radius) without sign symbols. The specified

value is reserved validly after R(e) is executed. 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(Δi) : radial(X) cutting for each axial cutting cycle , range: 0＜Δi≤9999999(IS_B)/0＜Δi≤99999999（IS_C） (unit: least input increment, diameter value, without sign symbol).

Q(Δk): radial(Z) cutting for each axial cutting cycle, range: 0＜Δk≤ 9999999(IS_B)/0＜Δk≤99999999（IS_C）(unit: least input increment, diameter value, without sign symbol).

R(Δd) : radial (X) tool retraction after cutting to end point of axial cutting, range: 0~99999999× 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(Δi) are omitted.

Execution process: (Fig. 3-79)

① Axial (Z) cutting feed △k 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 (Δk+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 (Δk+e)again and execute ②; after Z executing the cutting feed (Δk+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 △d/2 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 Dn, 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 (△d/2+△i/2) , i.e. $D_n \rightarrow A_{n+1}$ 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 (△d/2+△i/2), 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.

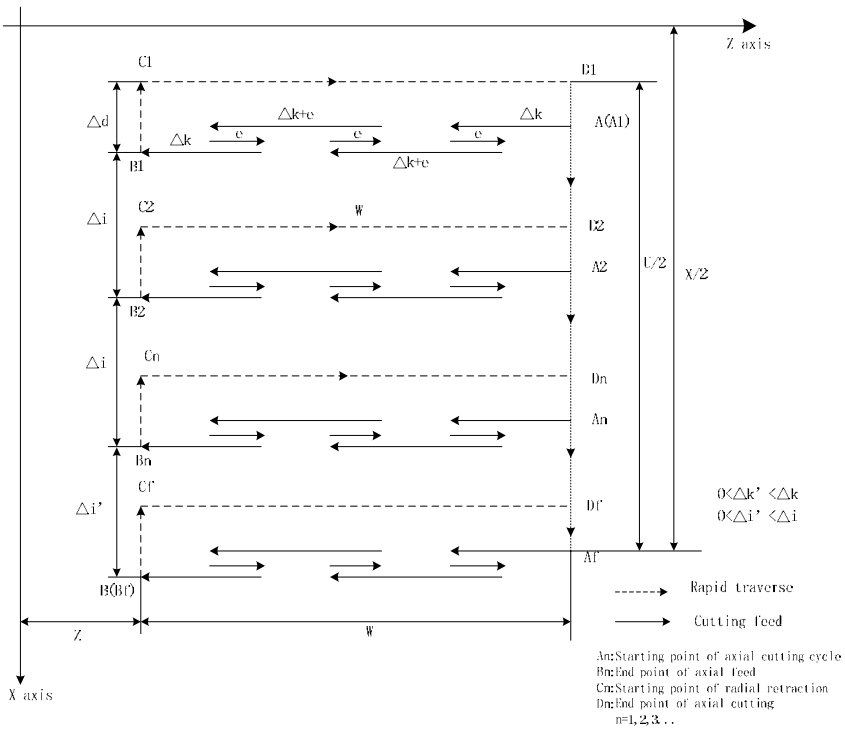


Fig. 3-79 G74 path

Command specifications:

1. 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;
2. 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;
3. 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.
4. When the single block is running, programs dwell after each axial cutting cycle is completed.
5. 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. 3-80

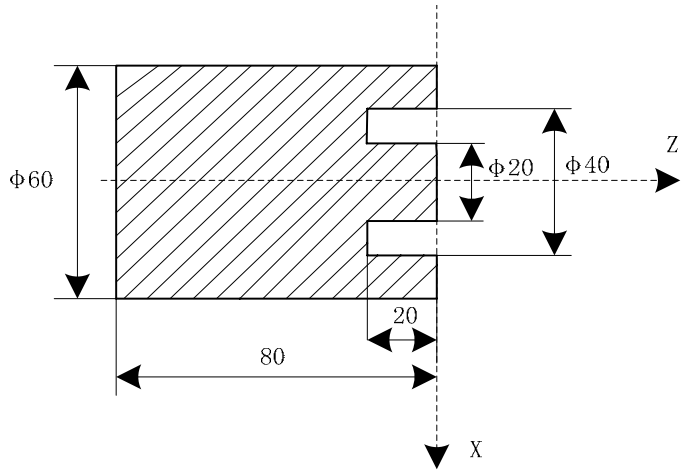


Fig.3-80

Program (suppose that the grooving tool width is 4mm, system least increment is 0.0001mm):

```
O0007;
G0 X36 Z5 M3 S500;          (Start spindle and position to starting point of machining)
G74 R0.5 ;                  (Machining cycle)
G74 X20 Z-20 P30000 Q50000 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)
```


3.22.6 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……), 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……), 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……), 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……), 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(Δi) : Radial(X) discontinuous tool infeed of each axial cutting cycle, its range: 0＜Δi≤ 9999999 (IS_B)/ 0＜Δk≤ 99999999（IS_C） (unit: least input increment, without sign symbol).

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

R(Δd)：Axial (Z) tool retraction clearance after cutting to end point of radial cutting, its range: 0~99999999×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 (Δd) is omitted.

The system defaults it executes the positive tool retraction when Z(W) and Q(Δk) are omitted.

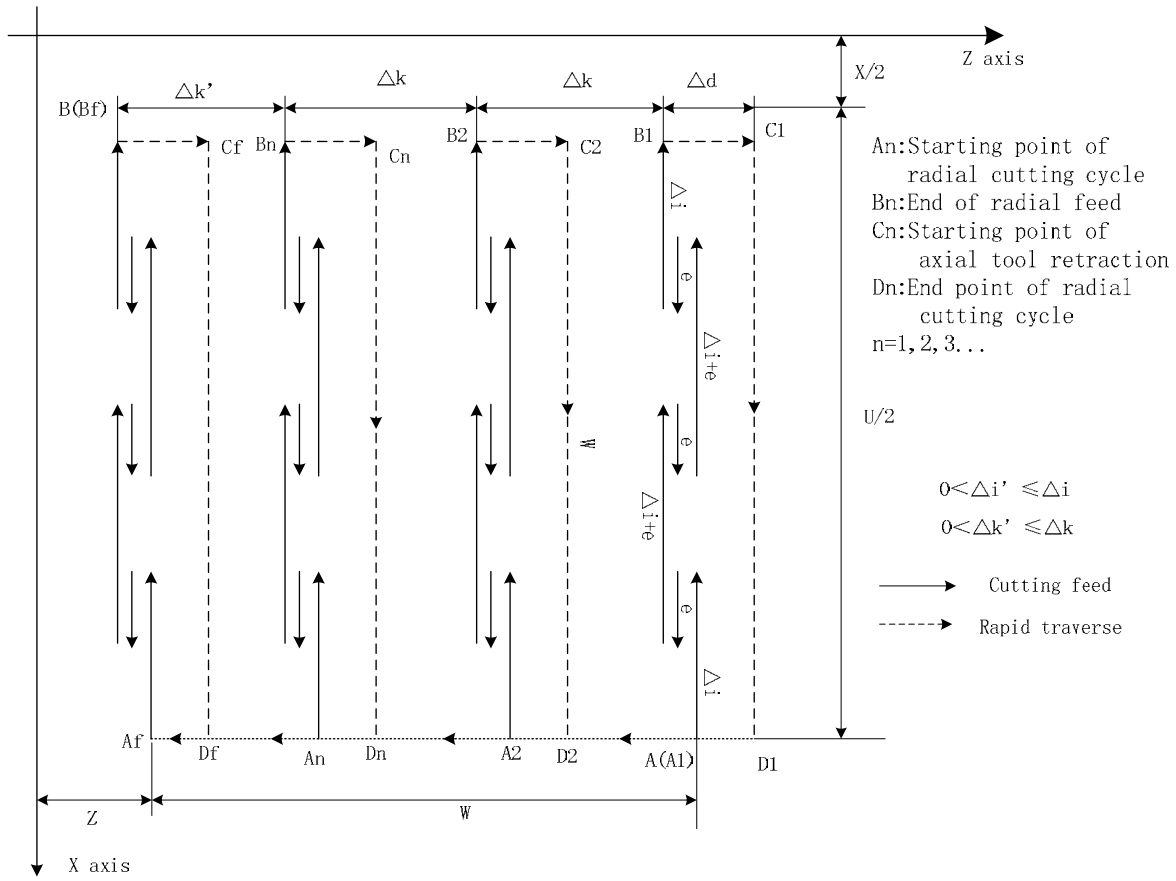


Fig. 3-81 G75 path

Execution process: (Fig. 3-81)

① Radial (X) cutting feed △i 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 (Δi+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 (Δi+e) again and executes ②; after X cutting feed (Δi+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 execute④;

④ Axial(Z) rapid tool retraction △d to C_n , when Z coordinate of B_f (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 (X) rapid retract tool to Dn, 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;

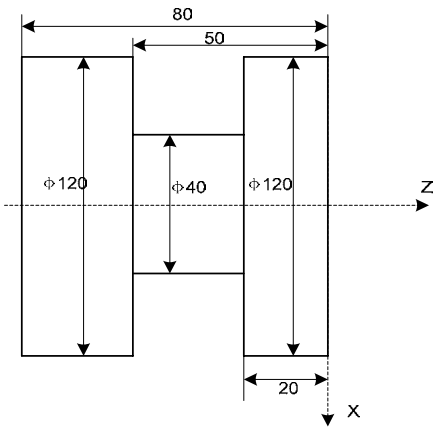
⑥ Axial(X) rapid tool infeed, and it direction is opposite to ④ retract tool. If the end point of tool infeed is still on it between A and A_f (starting point of last radial cutting cycle) after Z tool infeed (△d+△k), i.e. $D_n \rightarrow A_{n+1}$ and then execute ① (start the next radial cutting cycle); if the end point of tool infeed is not on it between D_n and A_f after Z tool infeed (△d+△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.

Explanation:

1. 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;
2. 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;
3. 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;
4. When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path;
5. 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:



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-24;                  (Position to starting point of machining)
G75 R0.5 F150;                 (Machining cycle)
G75 X40 Z-50 P60000 Q30000;    (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)
```

3.22.7 Drilling Cycle G83 or G87

Command format: axle drilling G83 X(U)__ C(H)__ Z(W)__ P__ Q__ R__ F__ K__M__ J0/1;
radial drilling G87 Z(W)__ C(H)__ X(U)__ P__ Q__ R__ F__ K__ M__ J0/1;

Code meaning: Drilling Cycle: Rapid traverse to the drilling position from start, then perform it along with the drilling hole direction) feed, retraction, re-feed… till to the botton of a hole, and then retreact to the drilling position, a drilling cycle is then completed; Position to the drilling position again, and then perform the next drilling cycle; stop at the drilling position after the last drilling operation is performed, the drilling cycle is then completed. The drilling cycle divides into axial drilling G83 and radial drilling G87, which is used in the axial drilling (Z axis) and radial drilling (X axis) separately.

Relevant definition: G83, G87 is modle G code

G83: Axial drilling cycle G code, axial (Z axis) drilling;

G87: Radial drilling cycle G code, axial (X axis) drilling;

(X, C): Drilling position; ------G83

Z: at the bottom of a hole; ------G83

(Z, C); Drilling position; ------G87

X: at the bottom of a hole; ------G87

P: It is pause time at the bottom, modal data, initial default value is 0 (Unit: ms), its resolution is shown in the table 1-2 of Section 1.4.1.

Q: In-feed value each time, its range:-9999999~9999999 (IS_B) / -99999999~99999999 (IS_C) (unit：least input increment, radius value); Evaluate its absolute value when inputting the negative value, modal data; the initial default value is 0. When the Q value is 0, directly feed to the botton of the hole.

R: In-feed value after the in-feed for each time, its range:0~9999.999 (IS_B)/ 0~999.9999 (IS_C) (unit：mm/inch，radius value); Evaluate its absolute value when inputting the negative value, modal data; the initial default value is 1.

F: Cutting feddrate, refer to the table 1-2 in the Section 1.4.1 for its resolution.

K: Drilling repeated times can be omitted; it is regarded as 1 by default when ignores; refer to the table 1-2 in the Section 1.4.4 for its resolution. If the the drilling position is regarded as the relevant coordinate programming, drilling is performed at difference positions.

M: The M code is used for clamping the indexing spindle; its code value is set by data parameter No.170, which can be omitted; it does not use the spindle clamping by default when it ignores.

J: 0：High speed peck drilling cycle;1：Peck drilling cycle, modal data. Its initial value regardes as 0 by default.

Code performance procedure:

I：High speed deep hole drilling cycle（**J＝0**）：

① Rapidly position to the drilling start position（G83：X，C； G87：Z，C；）,if the M code with clamping spindle exists, output Mα after the positioning is completed;

② Cutting feed Q distance: P specification time dwells and then perform ⑥ if it reaches to the bottom of a hole;

③ Rapid tool-retraction R distance: If it reaches to the drilling start position during the tool-retraction, it is then retreated to the drilling start position;

④ Cutting feed (Q+R) distance: If it retracts to the drilling start position at the last time, and then cut the feed distance (Q+tool-retraction amount at last time);

⑤ Cycle ③④ till to the bottom of a hole, dwell the P specification time;

⑥ Rapid return to the drilling start position, if it executes the M code of spindle clamping,output the Mβ after the positioning is completed;

⑦ If the cycle does not end instead of returning to the ①, the next machining cycle is then started;

II：Deep hole drilling cycle（**J＝1**）：

① Rapidly position to the drilling start position,（G83：X，C； G87：Z，C；）,if the M code with clamping spindle exists, output the Mα after the positioning is completed;

② Cutting feed Q distance: P specification time dwells and then perform ⑦ if it reaches to the bottom of a hole;

③ Rapid perform the tool-retraction to the drilling start position;

④ Rapid positioning to the R position distance from the last feeding depth (Point C in the drawing);

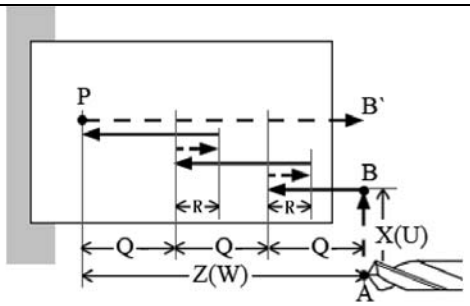
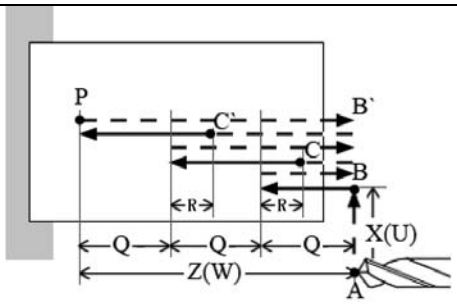
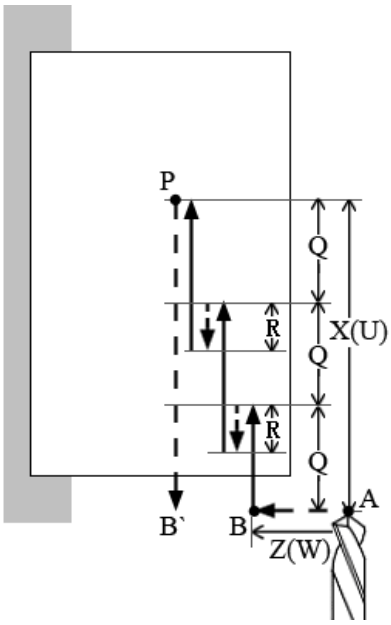
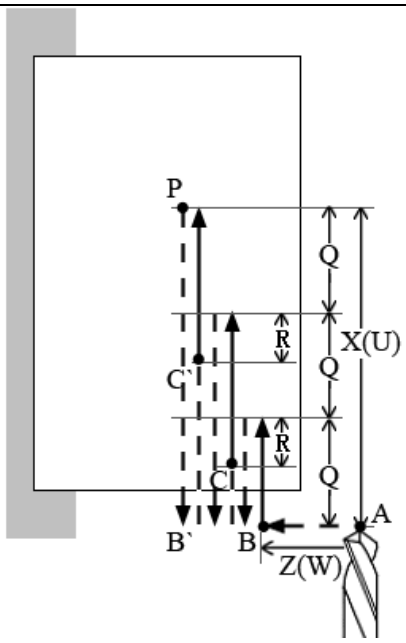
⑤ Cutting feed (Q+R) distance

⑥ Cycle ③⑤ till to the bottom of a hole

⑦ Rapid return to the drilling start position, if it executes the M code of spindle clamping, output the Mβ after the positioning is completed;

⑧ If the cycle does not end instead of returning to the ①, the next machining cycle is then started;

Note: α value is set in the data parameter No. 170, β = α + 1; therefore, the M codes in the PLC should be treated.

	Common drilling	Peck drilling
G83		
G87		

Precautions:

1) G83 or G87 is the modal G command of the group 01, which can be cancelled by the G command of group 01. The rest of the command words are the modal data other than the positioning place, cycle times and the M code of the clamping index spindle.

2) Do not use the cutter compensation C in G83 or G87 command, the cutter compensation may automatically retract when entering the drilling cycle, it may automatically recover after drilling.

3) It is necessary to select the G18 panel when performing the G83 or G87 command.

4) Fail to perform the G83, G87 command in the G71～G73 commands or polar coordinate

mode.

5) The drilling of G83, G87 command should be set as linear axis. (G83 drilling axis is regarded as the 2nd axis, G87 treats as the 1st one).

6) When the in-feed amount Q of each time is 0, perform the common drilling cycle. The drillingmachining will directly feed to the botton of a hole, and then moves to the start position at the rapid traverse rate.

3.23 Thread Cutting Commands

GSK980TDi 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.

3.23.1 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.3-82; 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 by 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～500 mm. 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:（-99999999~99999999）× 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: 0~99999999× 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 0^0.

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 : 0.001^0 . Q180000 is input in program if it offsets 180^0 with spindle one-turn; if Q180 or Q180.0, it is 0.18°.

Difference between long axis and short axis is shown in Fig. 3-82.

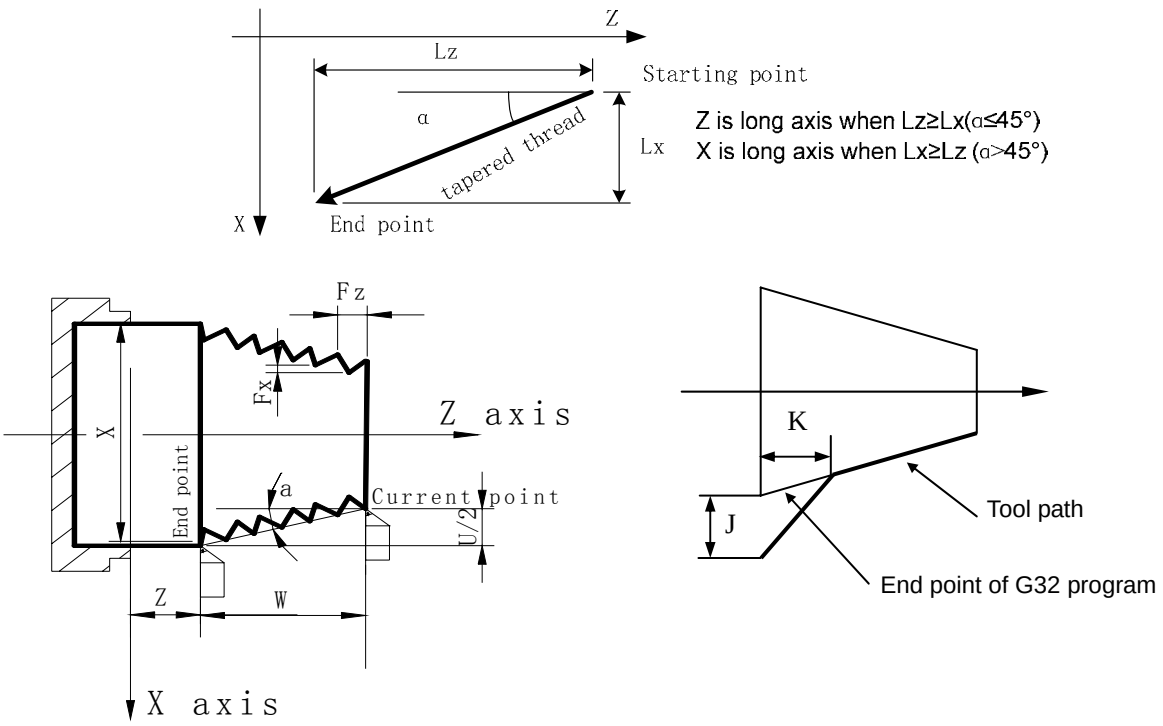


Fig. 3-82 G32 path

Notes:

1. 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;

2. There is no thread run-out when J, or K are omitted; K=J is the thread run-out value when K is omitted;
3. There is no thread run-out when J=0 or J=0, K=0;
4. The thread run-out value J=K when J≠0,K=0;
5. There is no thread run-out when J=0 or K≠0;

6. When parameter No.107 set to 0, its tail-retraction angle is matched with the J or K ratio. When the parameter No.107 does not set to 0, the tail-retraction speed of the short axis is set by parameter value, and the tail-retraction angle is determined by thread cutting speed and short axis tail-retraction speed;

6. 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;
7. 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;
8. 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;
9. The thread cutting decelerates to stop when the system resets, emergently stop or its drive unit alarms.

Example: Pitch: 2mm. δ1 = 3mm,δ2 = 2mm,total cutting depth 2mm divided into two times cut-in.

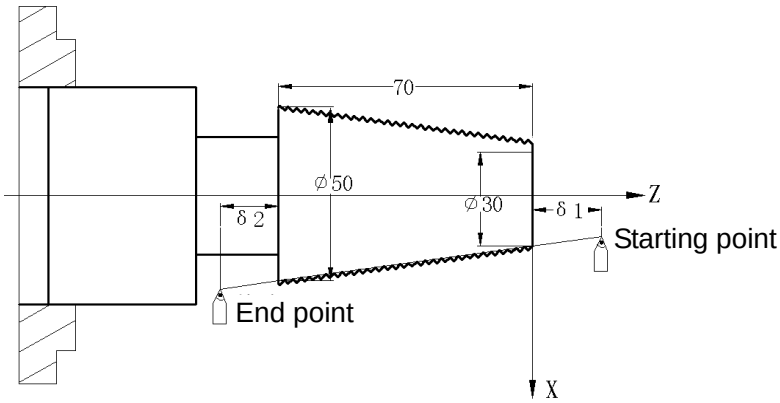


Fig.3-83

Program:

```
O0009;
G00 X27.4286 Z3;              (First cut-in 1mm)
G32 X48.8571 W-75 F2.0;       (First taper cutting)
G00 X55;                      (Tool retraction)
W75;                          (Z returns to the starting point)
X26.4286;                     (Second tool infeed 0.5mm)
G32 X47.4286 W-75 F2.0;       (Second taper thread cutting )
G00 X55;                      (Tool retraction)
W75 ;                         (Z returns to the starting point)
M30;
```

3.23.2 Rigid Thread Cutting G32.1

Command format: G32.1 X(U)__ Z(W)__ C(H)__F(I)__ S__;

Command function: The traditional thread interpolation calculates the current distance of the movement of the feed axis based on the feedback pulse count of the position encoder installed on the spindle, which can realize the feed axis to follow the thread interpolation mode of the spindle. Its disadvantages are that there is a big error in the thread lead in acceleration/deceleration. In the rigid thread interpolation mode, the working of the spindle motor is the same that of the servo motor, and the interpolations between the feed axis and the spindle execute the thread interpolation to get the thread with high precision.

Command explanation:

G32.1 : Rigid thread interpolation command;

C : Start angle of thread interpolation;

（X,Z） : End point coordinates of thread interpolation;

F（I） : Thread lead, F（I）> 0 right-hand thread, F（I）< 0 left-hand thread;

J: Movement in the short axis in thread run-out, its range:（-99999999~99999999）× 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: 0~99999999× least input increment. If the long axis is X, its value is in radius without direction; K is modal parameter.

S: spindle speed;

Explanations:

1) G32.1 is in Group 01;
2) When there is the C movement command in a block, the spindle firstly positions the start point of C axis before turning the thread;
3) Start points of X, Z, C axis must be the same one in the repetitive machining;
4) G32.1 cannot specify the thread run-out, and the feed axis and the spindle stops simultaneously at the end point of the thread;
5) The input range of the programmed address value in G32.1 is the same that of the general thread turning command（G32）;
6) When G32.1 is executed, the pause signal is disabled temporarily and the spindle override is fixed to be 100%.

Command path is shown below:

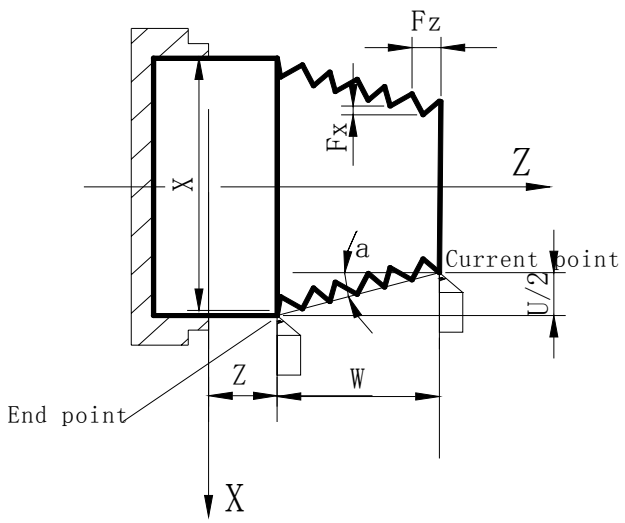


Fig. 3-84 G32.1 path

Notes：

1) The spindle must be in the position control mode when it works;
2) In multiple spindle control function, G32.1 is used only between the 1st spindle and the feed axis, and their uses and the related parameter settings are the same those of the rigid tapping;
3) In G32.1, CNC does not check whether the current spindle is the position control mode or the speed control mode. So, the servo control axis of the 1st spindle is set to Cs working mode to avoid the coordinate overflow;
4) Absolute coordinates of C axis are set to the cycle mode to avoid the coordinate overflow;
5) The speed control mode is switched to the position control mode in the spindle control mode. Please execute that the spindle returns to the reference point or use G50 to set the start point of the current C axis;

Example: Supposing M14: the spindle is switched to the position control mode; M15: the spindle is switched to the speed control mode.

The thread is the right-hand, its lead is 2mm, the spindle speed is 500 r/m in thread cutting, thread cutting length is 20mm, and its programming format is shown below:

```
O0132（0132）;
G00 X100 Z100；//position to the safe position to change the tool
T0101；          //change a thread tool（supposing No. 01 is the thread tool）
G00 X25 Z2；     //position to the thread starting point（supposing the shape has been made at
                   the last tool）
M14；            //switch the spindle from speed control mode to position control mode（it is 0°
                   position after switch）
G50 C0；         // set the zero of the rotary axis(it is important to set it because the operation is
                    relevant to the start angle of the following thread turning）
G32.1 Z-20 F2 S500 M08；//thread turning; the speeds of the spindle and the feed axis are0 in
                         end point
G00 X100；            //tool retraction
Z100；                //return to tool change position
```

```
M15;                    //switch the spindle from position control mode to speed control mode
. . . . ;               //execution the 2nd machining procedure
. . . . ;
M30;                    //end of program
```

3.23.3 Arc Threading Cutting Code G32.2 or G32.3

Code format: G32.2 / G32.3 } X(U)__ Z(W)__ R__ (I__ K__) F__ Q__

Code function: The movement path of arc thread is a section of arc from start to end; The coordinate axis with bigger shifting value where from the start to end is regarded as long axis; the other one is called short axis. The long axis moves a lead while the spindle rotates one cycle during the operation. A constant pitch helical slot generates on the workpiece surface when the tool cuts the workpiece, so that the constant pitch thread machining is carried out. The F command is used for the provided thread pitch, the arc thread machining of the constant pitch can be performed by executing the G32.2 / G32.3 command.

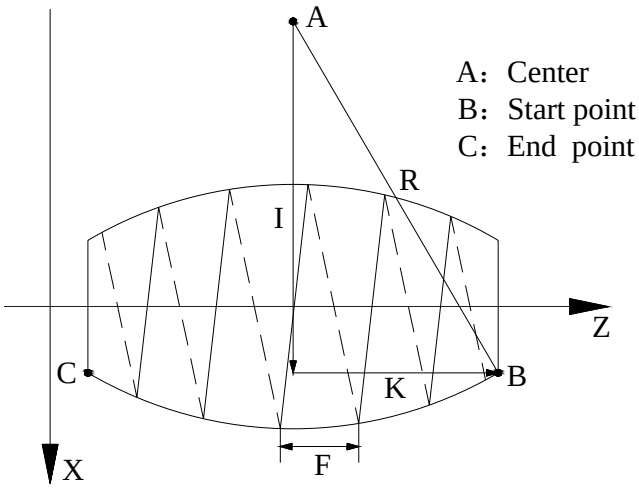
Code explanations:

1) G32.2/G32.3 is modal;
2) The leading of the thread is the displacement value (The displacement value along X axis is based upon the radius value) of one cycle for the spindle.
3) When bit 7 of parameter No.202 is set to 0, arc thread follows with the feedback signal of spindle encoder; when the bit 7 of the state parameter No.202 is 1, arc thread cutting feed is performed the linkage interpolation with the servo spindle; in this case, the spindle, there is not alternative other than the servo spindle;
4) G32.2 CW arc thread machining, G32.3 CCW arc thread machining; the judgement methods for the CW, CCW and G02, G03 arc maching command are identical;

Q use rules:

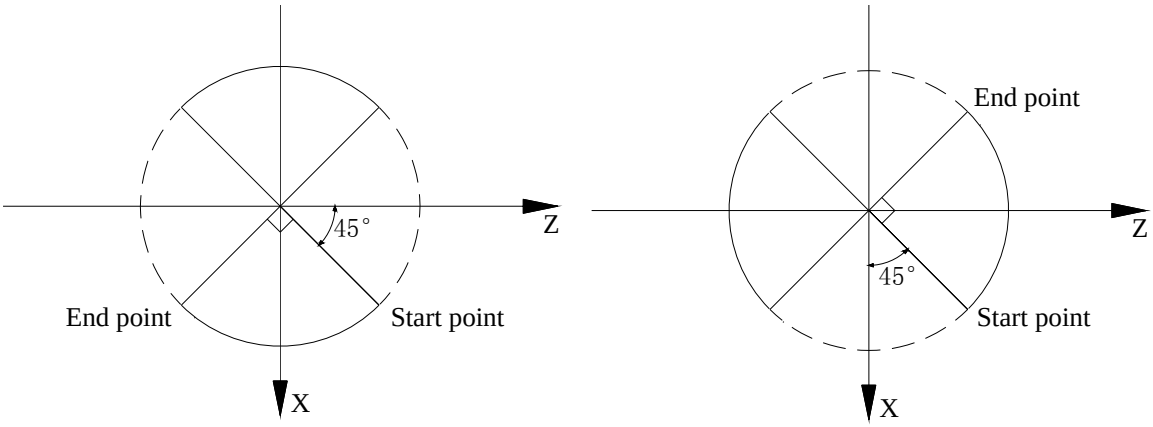
1) When the bit 7 of state parameter No. 202 sets to 1, Q value is disabled; in this case, the start angle of the thread can be directly specified by C address;
2) If the Q does not specify, namely, it is regarded as start angle 0 by default;
3) The overall threads of the mult-pitch formed by start angle indexing should be less than 65535.
4) Q's unit is 0.001 degree, if it offsets 180 degree to the spindle one-turn signal, input Q180000 to the program; if the Q180 or Q180.0 is input, it separately regards as 0.18 degree.

The schematic of the command execution path is shown below:

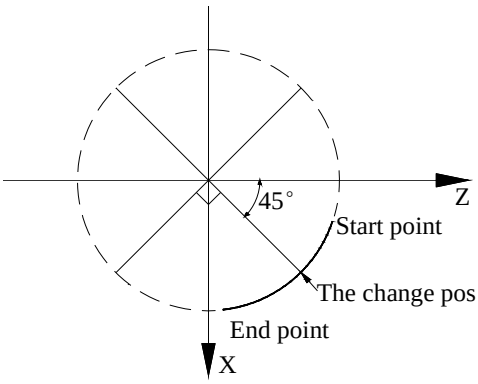


Precautions:

1) R value is enabled when R is simultaneously specified with I or K;
2) The specified arc block by arc thread must be performed within the certain area (The area of which the X or Z axis is separately regarded as long axis); refer to the following figure:



3) The arc specified by arc thread, if its long axis changes, the alarm may occur; refer to the following figure:



3.23.4 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: ±0.001~±500.000 mm/pitch (metric thread);

±0.060~±2540 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.

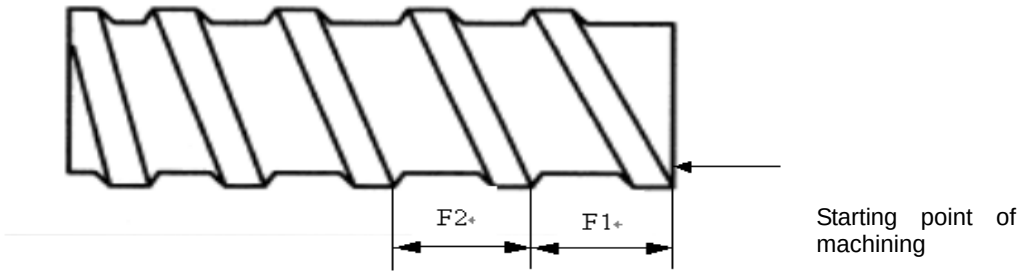


Fig. 3-85 Variable pitch thread

Note: It is the same as that of G32.

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

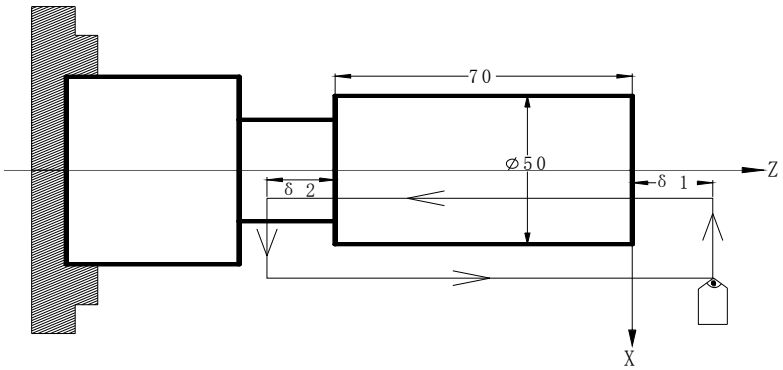


Fig. 3-86 Variable pitch thread machining

Use macro variables to simplify programming when G34 is used many times. δ1 = 4mm,δ2 = 4mm, 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;          First tool infeed: assignment #102=0.8mm
G65 H01 P#103 Q0;            Cycle count: assignment #103=0
N10 G65 H02 P#104 Q#103      Cycle count starting: #104=#103+1
R1;                          #103=#104
```

G65 H01 P#103 Q#104;	Total cutting cycle times: #104=15,jump to block N30
G65 H81 P30 Q#104 R15;	
G00 U-10;	Tool infeed to Φ50
G65 H01 P#100 Q#102;	Cutting infeed: #100=#102
G00 U-#100;	Tool infeed
G34 W-78 F3.8 J5 K2 R0.2;	Variable pitch cutting
G00 U10;	Tool retraction
Z4;	Z returns to starting point
G00 U-10;	
G65 H03 P#101 Q#100 R0.2;	Decreasing of cutting feed again: #101=#100-0.2
G65 H01 P#102 Q#101;	Assignment again #102=#101
G65 H86 P20 Q#102 R0.2;	Infeed: Jump to block N20 when #102≦0.2mm
G65 H80 P10;	Unconditionally jump to block N10
N20 G65 H01 P#102 R0.2;	Min. infeed: #102=0.2
G65 H80 P10;	Unconditionally jump to block N10
N30 ;	

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

L: Multi threads: 1～99 and it is modal parameter. (The system defaults 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 ①～⑥ if multi threads are machined.

Example: Fig. 3-87, thread M10×1.5

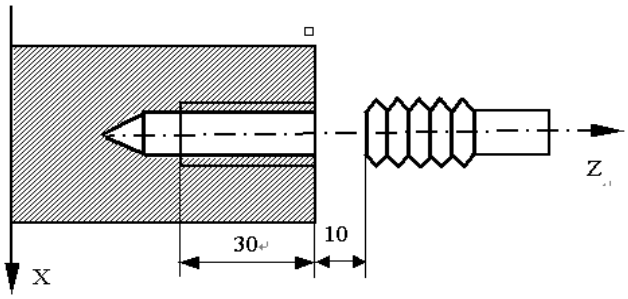


Fig.3-87

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.

3.23.6 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, and its range is referred to Table 1-2

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, and its range is referred to Table 1-2

M: Used to clamp the graduation spindle.

Rigid tapping method:

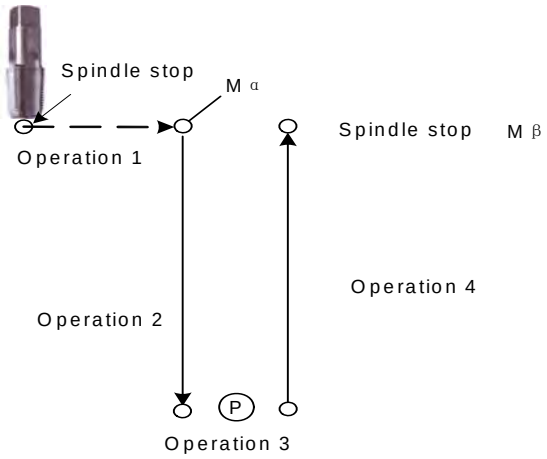
1. M29 S_ before G84/G88 is specified below
   M29 S_;
   G84(G88) X_C_(Z_C_) Z_(X_) P_ F_ K_ M_;
2. M29 S_ in the same block is commanded below
   G84(G88) X_C_(Z_C_) Z_(X_) R_ P_ F_ K_ M29 S_;

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：

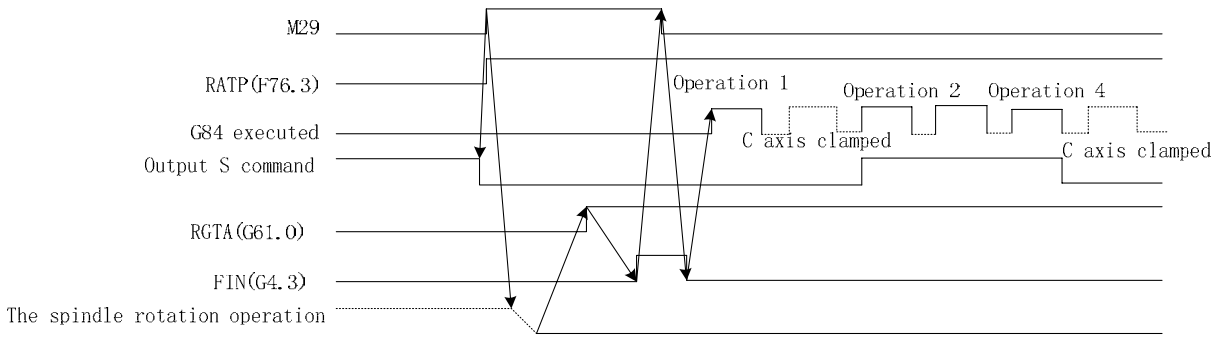
1) Operation 1：positioning to the hole position（starting point of rigid tapping）;
2) Before starting operation 2, the system outputs Mα when it has M code used to clamp the spindle;
3) Operation 2：start the rigid tapping；
4) Operation 3：pause time P in rigid tapping in hole bottom;
5) Operation 4：the rigid tapping returns to the hole bottom（starting point of rigid tapping;
6) When the block for rigid tapping specifies M code used to clamp the spindle, Mβ is output.

Note: α value is set in №170, β = α + 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;

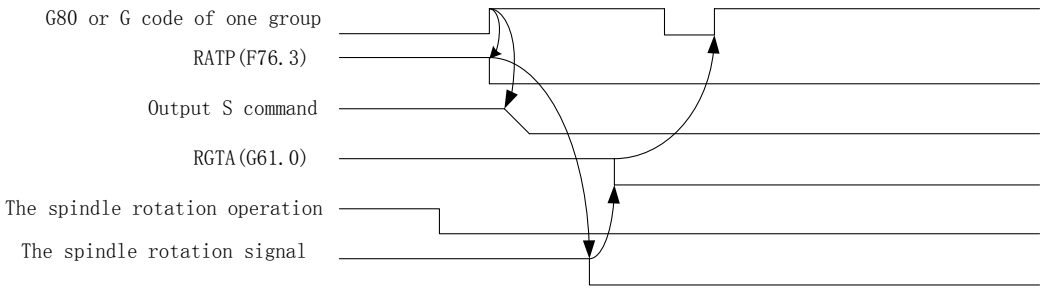
4） CNC resets.

The falling edge of F76.3 signal cancels the rigid tapping mode signal.

1) RTCRG(№186＃2) is set to ‘1’ when:

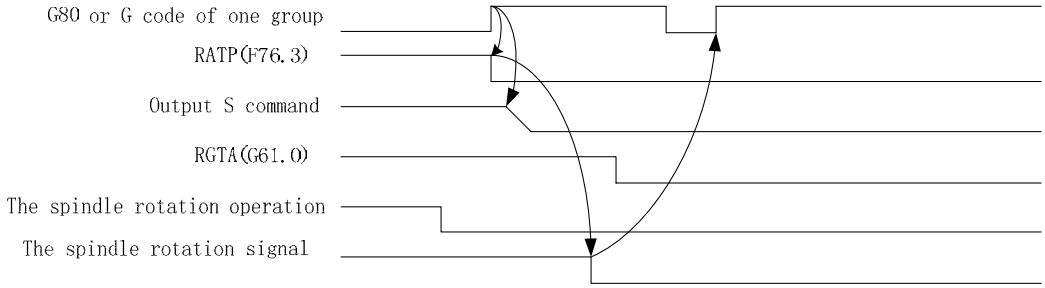
The system executes directly the next block instead of waiting the rigid tapping mode signal RGTAP<G61＃0> to become ‘0’.

Rigid tapping cancellation is shown below:



2) 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:

1) 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;
2) 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;
3) The dry run function is disabled in rigid tapping;
4) 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;
5) 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;
6) 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;
7) 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 №34. The backlash compensation along the tapping axis is performed in general method;
8) 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;

9) Setting RTORI（state parameter №186＃7）to 1 can execute the reference point return of the spindle before the system starts the rigid tapping.

3.23.7 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. 3-88.

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, $R \le |U/2|$, unit: mm;

F: Thread lead, its range: 0＜ F ≤500 mm. After F value is executed, it is reserved and can be omitted;

I: Thread teeth per inch, its range: 0.06tooth/inch～25400tooth/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 0~99999999× 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: 0~99999999× 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: 1～99 and it is modal parameter. (The system defaults it is single thread when L is omitted).

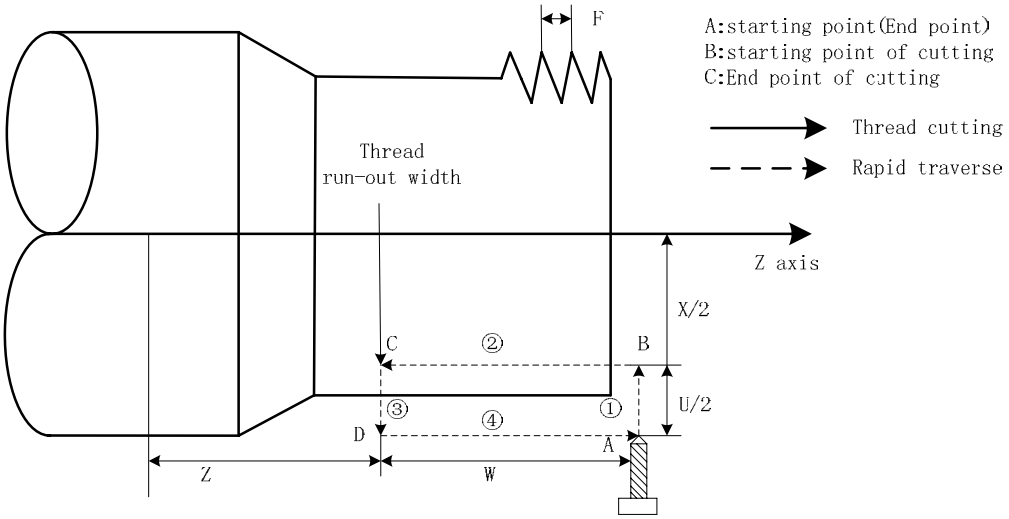


Fig. 3-88

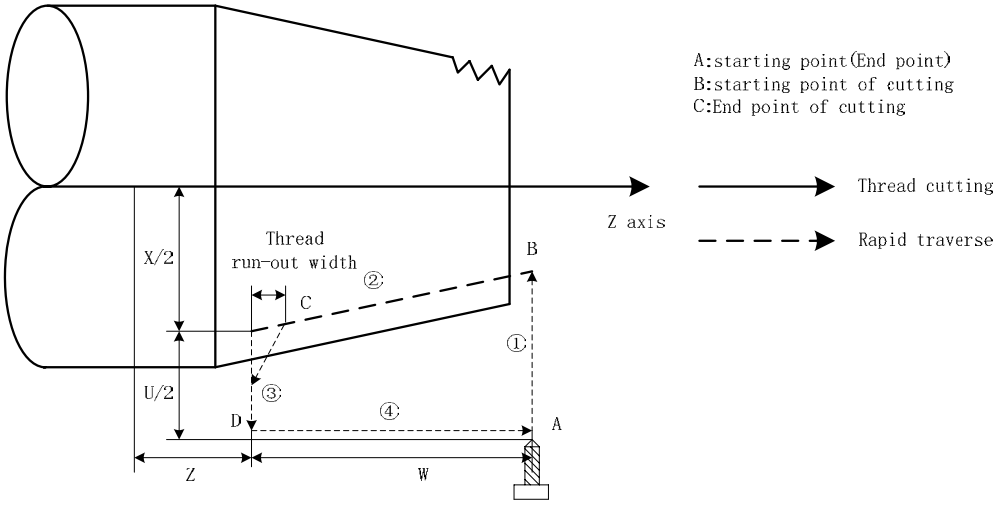


Fig. 3-89

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.3-88 and taper thread as Fig.3-89.

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 ①), 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.

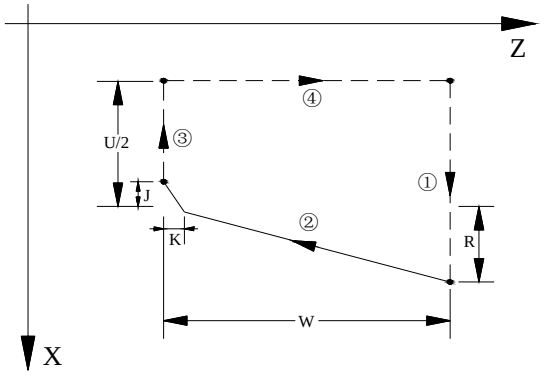
Notes:

1) Length of thread run-out is specified by №019 when J, K are omitted;
2) Length of thread run-out is K in the long direction and is specified by №019 when J is omitted;
3) Length of thread run-out is J=K when K is omitted;

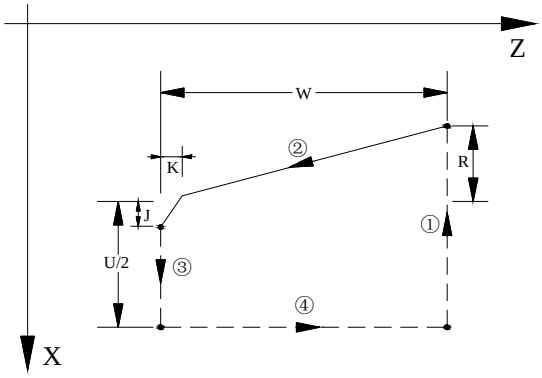
4) There is no thread run-out when J=0 or J=0, K=0;
5) Length of thread run-out is J=K when J≠0,K=0;
6) There is no thread run-out when J=0,K≠0;
7) When parameter No. 107 sets to 0, the tail-retraction angle is matched with the J or K ratio. When the No. 107 does not set to 0, the tail-retraction speed of the short axis is regarded as parameter setting value, and the tail-retraction angle is determined by thread cutting speed and short axis tail-retraction speed;
8) After executing the feed hold in thread cutting, the system does not stop cutting until the thread cutting is completed with ***Pause*** on screen;
9) After executing single block in thread cutting, the program run stops after the system returns to starting point(one thread cutting cycle is completed);
10) They are executed as the positive values when J, K negative values are input;
11) 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:

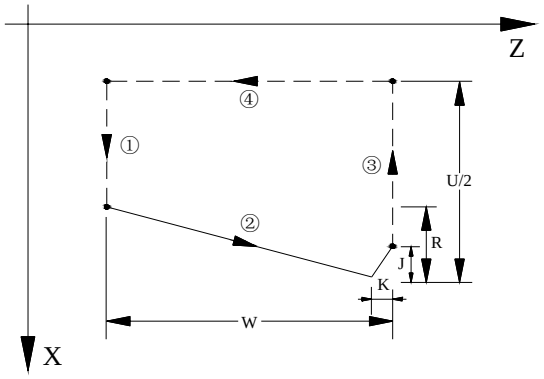
1） U>0，W<0，R>0



2） U<0，W<0，R<0



3） U>0，W>0，R<0 、|R|≤|U/2|



4） U<0，W>0，R>0、|R|≤|U/2|

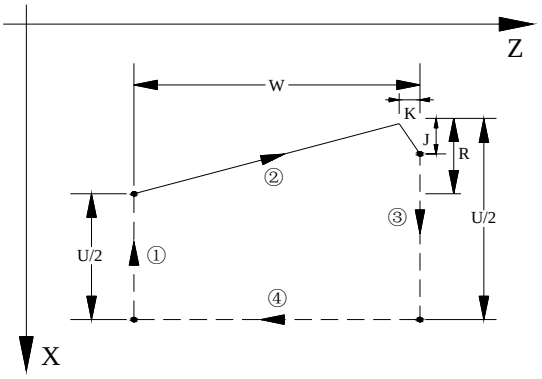


Fig.3-90

Example：Fig.3-91

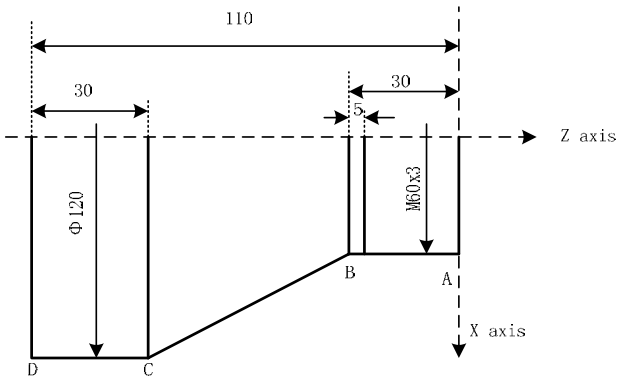


Fig.3-91

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;
```

3.23.8 Rigid Thread Cutting Cycle G92.1

Command format: G92.1 X(U)_ Z(W)_ F_ L_ S_; (Metric straight thread cutting cycle)

G92.1 X(U)_ Z(W)_ I_ L_ S_; (Inch straight thread cutting cycle)

G92.1 X(U)_ Z(W)_ R_ F_ L S_; (Metric taper thread cutting cycle)

G92.1 X(U)_ Z(W)_ R_ I_ L S_; (Metric taper thread cutting cycle)

Command function: Perform the radial (X axis) in-feed or axial (Z axis or X and Z axes) cutting from the cutting start, simultaneously, the work of the spindle motor is identical with the feed servo motor, as well the thread interpolation is performed by interpolation between the feed axis and spindle; therefore, the higher accuracy thread is then gained to achieve the cutting cycle of the straight thread of the constant pitch or the cone thread.

Command specification:

G92.1 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, $|R| \le |U/2|$, unit: mm;

F: Thread lead,. After F value is executed, it is reserved and can be omitted; F（I）> 0 right-hand thread,F（I）< 0 left-hand thread；

L: Multi threads: 1～99 and it is modal parameter. (The system defaults it is single thread when L is omitted)

S: The speed of thread cutting and modal value can be specified the spindle speed before this block;

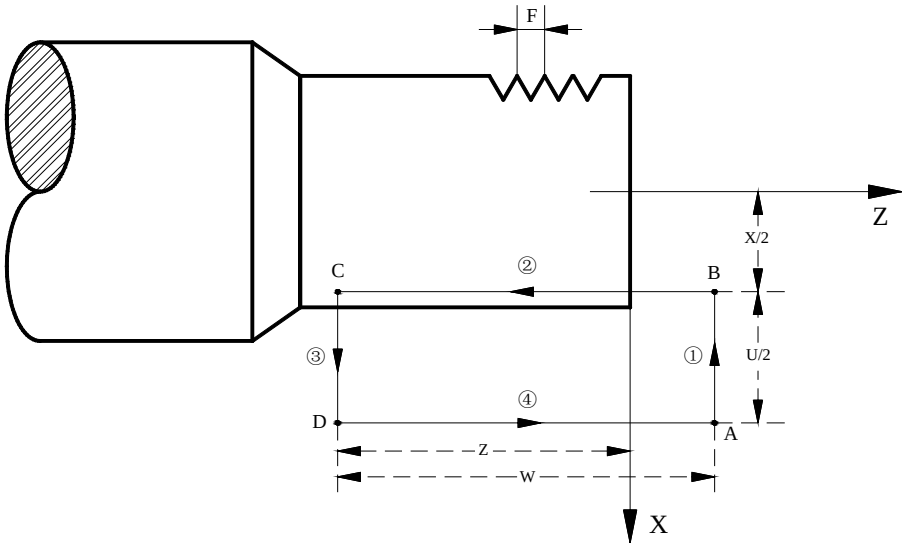


Fig. 3-92

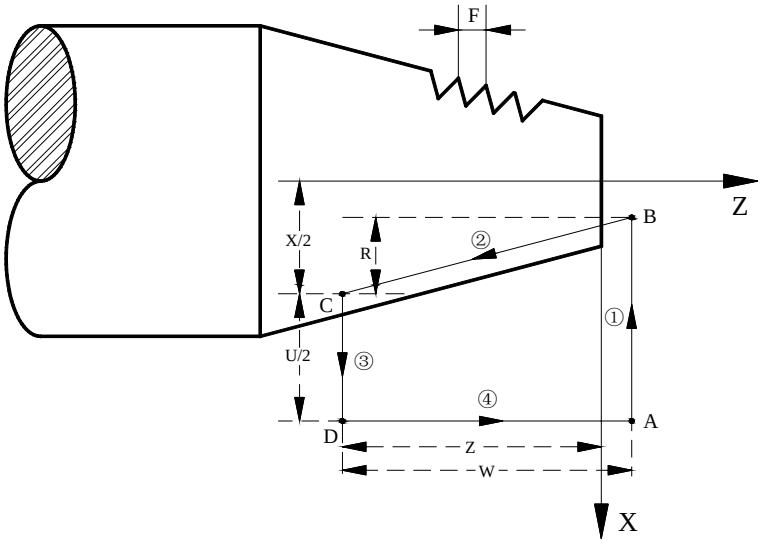


Fig. 3-93

The system can machine one thread with many tool infeed in G92.1, but cannot do continuous two thread and end face thread. Definition of thread pitch in G92.1 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.3-92 and taper thread as Fig.3-93.

① X axis rapidly moves to the cutting start B from the cycle start A. Spindle positions to the start angle if it is the multi-head thread. Refer to the motion ① in the Fig. 3-92 or 3-93;

② The feed axis is interpolated to end C with the spindle thread from the cutting start B,

the feed axis and spindle are entirely stopped at the end C.Refer to the motion ② in the Fig. 3-92 or 3-93;

③ X axis retracts to the point D from C at the rapid traverse rate, refer to motion ③ in the Fig. 1 or 3-93;

④ Z axis retreats to A from point D at the rapid traverse rate, simultaneously, the spindle positions to the initial angle of the cycle machining, and then the cycle is ended, refer to the motion ④ in the Fig. 3-92 or 3-93.

Precautions:

1) The spindle, there is not alternative other than to work at the position controllable mode;

2) In the case of the multi-spindle controllable function, G92.1 is only used between the 1st spindle simultaneous feed, the usage and relevant parameter setting is identical with the rigid tapping;

3) When performing the G92.1 command, CNC system will not detect whether the current spindle is on the position or speed controllable method, therefore, when using this command, it is better to set the 1st spindle servo controllable axis as the Cs axis working mode to guarateen the safety;

4) The absolute coordinate value of the spindle controllable axis (C axis) should be set as cycle method to avoid the coordinate value flowing;

5) When the spindle controllable method shifts to the position method from speed controllable method, it is better to perform the spindle reference point return operation or set the start position of the current controllable axis (C axis) by G50.

6) Both the feed axis and spindle are performed the interpolation from the start of the cutting to the end. The spindle and feed axis are stopped at the thread interpolation end position, instead of performing the tail-retraction operation;

7) Spindle returns to the initial angle when the cycle is completed, for example, the current position of the spindle is 30° before the cycle, the single cycle is then ended; spindle returns to 30° in the motion ④; the positioning direction is identical with the spindle rotation direction of which when the thread interpolation is performed.

8) During the multi-head thread, spindle positions to the outside of its corresponding start angle by the current thread head when the motion ① is performed. For example: the current position of the spindle is 30°, machine the 6-head thread, when machining at the 2nd theard head, the spindle positions at the 90° when performing the motion ①, as well it returns to 30° again when performing the motion ④;

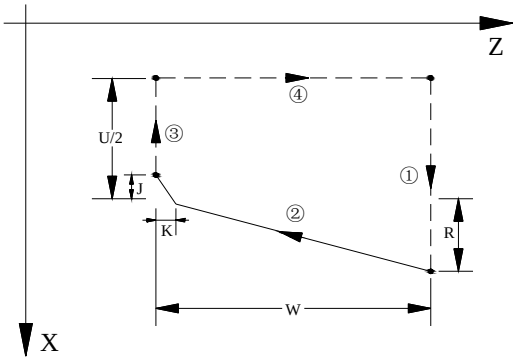
9) After executing the feed hold in thread cutting, the system does not stop cutting until the thread cutting is completed with Pause on screen;

10) After executing single block in thread cutting, the program run stops after the system returns to starting point(one thread cutting cycle is completed);

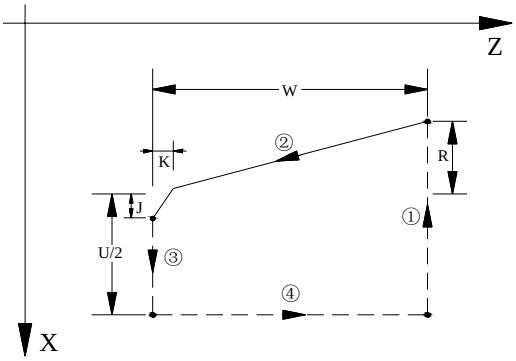
11) Thread cutting decelerates to stop when the system resets, emergently stops or its driver alarms.

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:

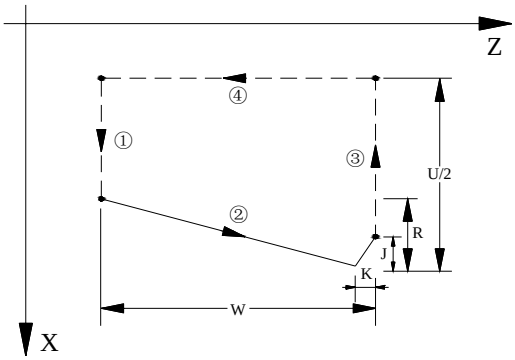
1）U>0，W<0，R>0 2）U<0，W<0，R<0

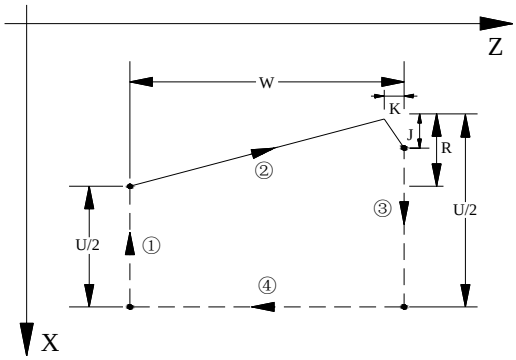


3）U>0，W>0，R<0，|R|≤|U/2|

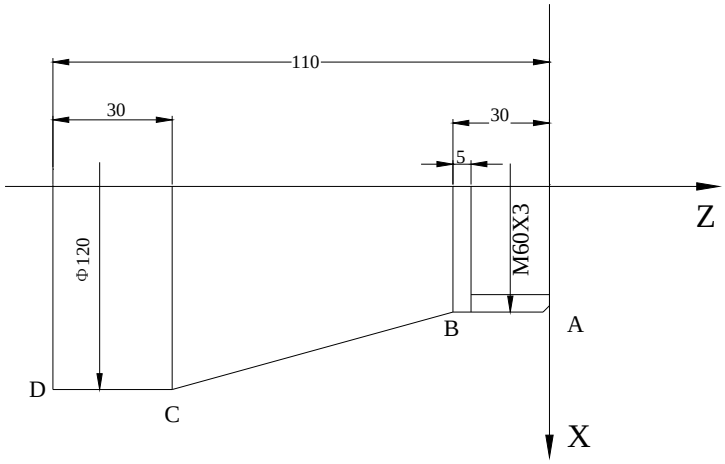


4）U<0，W>0，R>0，|R|≤|U/2|





Example：



Program：

O0012；

G0 X150 Z50； (Tool-change performs after positioning at a safety place)

T0101； (Thread tool)

M14； (Shift the spindle to the position controllable method)

G0 X65 Z5； (Rapid traverse)

G92.1 X58.7 Z-28 F3 S600； (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)

M15； (Shift the spindle to the speed controllable method)

M30；

3.23.9 Thread Reparation Function G92 P1 or G92.1 P1

Code format: G92 X(U)_ Z(W)_ R_ F(I)_ L_ P_ S_; (Follow thread reparation)

G92.1 X(U)_ Z(W)_ R_ F(I)_ L_ P_ S_; (Interpolation thread reparation)

Code function: When the machine tool matches with the frequency-conversion spindle, the damaged thread can be recovered using the G92 P1 code; the long axis moves a lead while the spindle rotates one cycle, its motion procedure is identical with the G92. When the machine tool matches with the servo spindle, the damaged thread can be recovered using the G92.1 P1 code, and the thread interpolation can be performed by the interpolation between the feed axis and spindle, its motion procedure is identical with the G92.1.

Code explanations:

Modal command;

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, $R \le |U/2|$, unit: mm;

F: Thread lead. After F value is executed, it is reserved and can be omitted; F（I）> 0 right-hand thread,F（I）< 0 eft-hand thread;

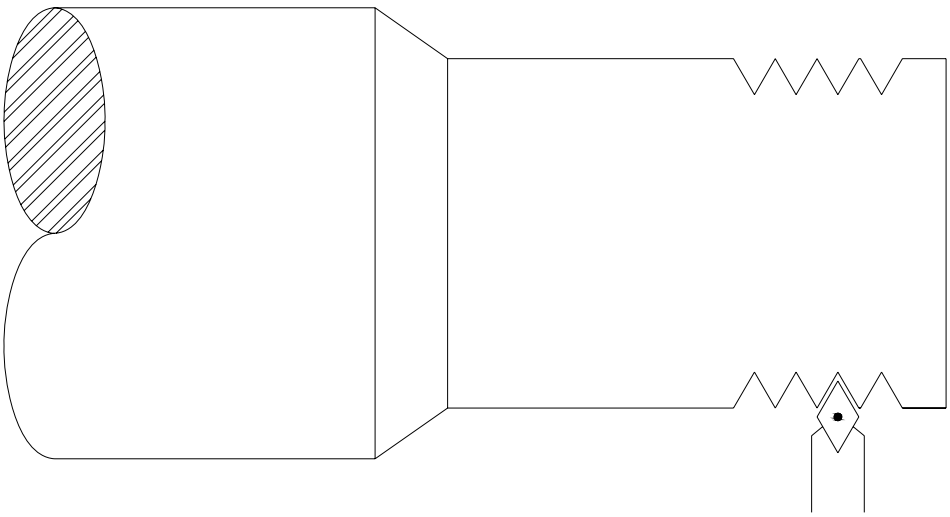
L: Multi threads: 1～99 and it is modal parameter. (The system defaults it is single thread when L is omitted)

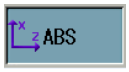
P: Thread recovery mark: 0：Do not perform the reparation function, 1：Perform the reparation function;

S: The revolving speed of the thread cutting and the modal value can be specified the spindle speed before this block;

Operation methods:

1) Thread reparation function usage: Before using the thread recovery, set the Bit 4 of state parameter No.174 to "1", the function is then performed; the setting of Bit 5 of state parameter No.174 can be recovered the mode of the thread.
2) Thread reparation position record: In the Manual, MPG or single-step mode, shift the spindle into the position controllable method (Do not use this controllable method when using follow thread reparation), move the tool to the measure position by the X, Z axis or spindle; cutter nose should be coincided with the tooth cavity as much as possible by moving the tool, refer to the following figure:



In this case, enter to the position interface by , to the absolute coordinate page by 

soft function button; press the

soft function button, the system is then automatically recorded the coordinate information of this position and the angle of the spindle. The recorded position information is performed the M30 code in the program, alternatively, when the main program is ended with M99 code; here, the recorded position is regarded as already used by the system, the recorded value is then cleared; if the thread needs recover again, it is necessary to record the position again. This operation will directly affect the accuracy of the thread recovery, therefore, it is better to overlap the cutter nose of the tool and the thread slot as much as possible.

3) Thread recovery performance: Perform the thread recovery operation by using the G92.1 or G92 code; the system may automatically perform the thread recovery operation based upon the recorded position previously.

Precautions:

1) When the thread reparation function is disabled, P1 parameter is specified in G92, G92.1 code, the system is then automatically ignored;
2) When the thread reparation function is enabled, P1 does not specify in the G92, G92.1 code or the value of other Non-1, the system will not perform the reparation function;
3) When the thread recovery function is enabled, as well the P1 parameter is specified in G92 or G92.1 code; the system alarm may occur when the thread recovery position record does not prform in the Manual, MPG or Single-step mode;
4) When the multi-block G92, G92.1 code is used to repair the thread, the P1 parameter only should be specified in the G92, G92.1 code of the 1st block, the thread machining start angle calculated by system may be succeeded to the subsequent G92, G92.1 thread block;
5) When the coordinate of the measure position long axis does not between the start and end points of which the thread block long axis is specified by P1, the alarm is then generated.
6) When the thread is recovered by G92 P1, the Bit 5 of state parameter No.174 should be set to 1; when the thread is recovered by G92.1 P1, the Bit 5 of state parameter No.174 should be set to 0.
7) When the thread reparation is performed, the numerical value of the data parameter No.191 should be calibrated (This value is the tolerance value (EPO) at the stable

position where the axis operation speed of servo drive along each axis is 1m/min); otherwise, the recovered thread may be generated an offset;

3.23.10 Multiple Thread Cutting Cycle G76

Command format: G76 P(m) (r) (a) Q(△dmin) R(d) ;
G76 X(U) __ Z(W) __ R(i) P(k) Q(△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. 3-94:

Command specifications:

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). 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×L,L is the thread pitch). 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(°). 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;

△Q(△dmin): Minimum cutting travel of thread roughing, range: 0～999999(IS-C)/ 0～99999(IS-B),(unit: least input increment, radius value). When ($\sqrt{n}-\sqrt{n-1}$) ×△d＜△dmin, △dmin is regarded as the cutting travel of current roughing and the subsequent rough-machining cutting value and the in-feed depth will not calculate based upon the formula any more. Setting △dmin is to avoid the too small of roughing amount and too many roughing times caused by the cutting amount deceleration in thread roughing. When Q (△dmin) 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(IS_B）/00～99.9999(IS_C) (unit: mm/inch, radius value without sign symbols), the radius value is equal to X absolute coordinates between cut-in point Be of thread finishing and Bf of thread roughing.. 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(△d): Depth of the 1st thread cutting, range: 1～99999999 unit: Least input increment, radius value, without sign symbols). The system alarms when △d 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.

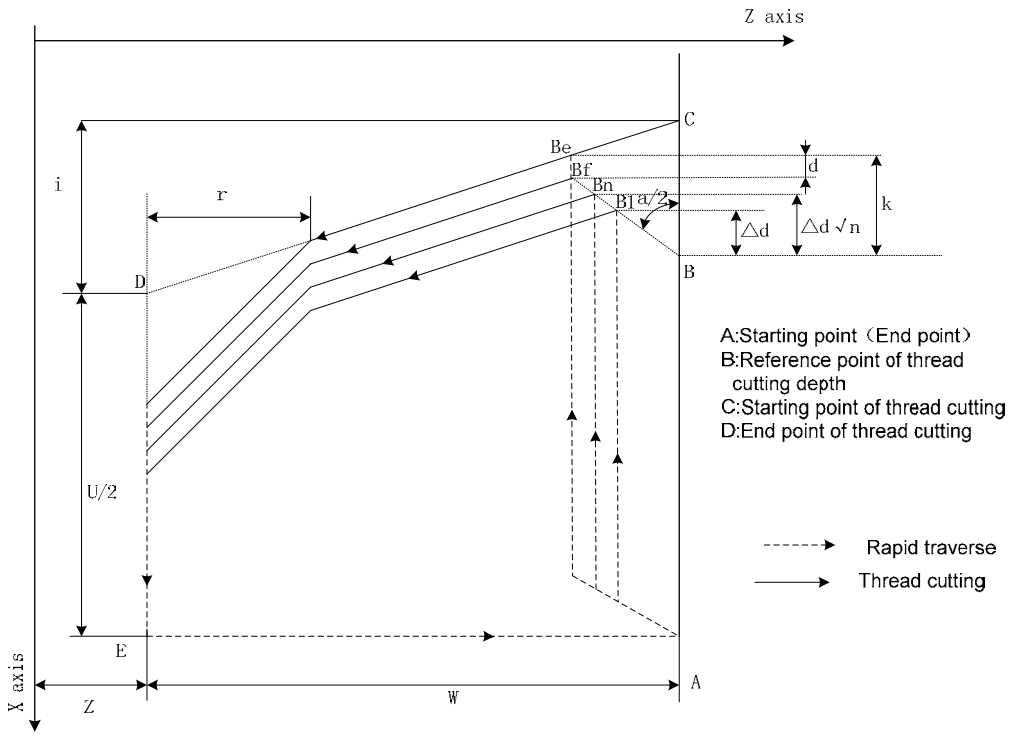


Fig. 3-94

Cut-in method as follows: Fig. 3-95：

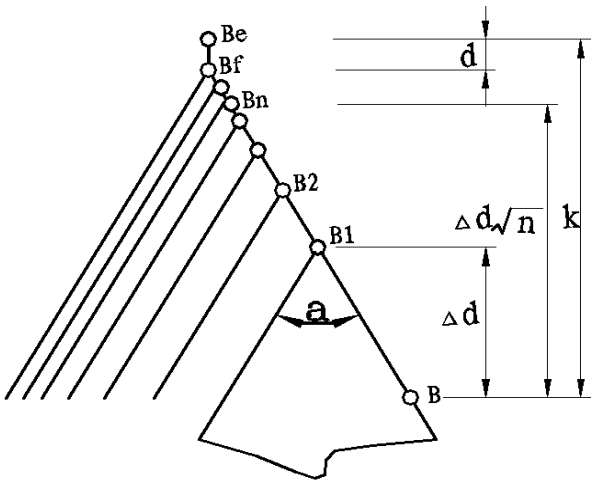


Fig. 3-95

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 B_1, and the thread cutting depth is △d. 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≠0;

② The tool cuts threads paralleling with C→D to the intersection of D→E (r≠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 ($\sqrt{n}$ ×△d) ,($\sqrt{n-1}$×△d+△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_f, 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≠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:

1) In thread cutting, execute the feed hold, the system displays ***Pause*** after the thread cutting is executed completely, and then the program run pauses;
2) The thread cutting decelerates to stop when the system resets and emergently stop or the driver alarms;
3) Omit all or some of G76 P(m) (r) (a) Q(△d_{min}) R(d) . The omitted address runs according to setting value of parameters;
4) 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;
5) 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. 3-96.

Example: Fig. 3-96, thread M68×6.

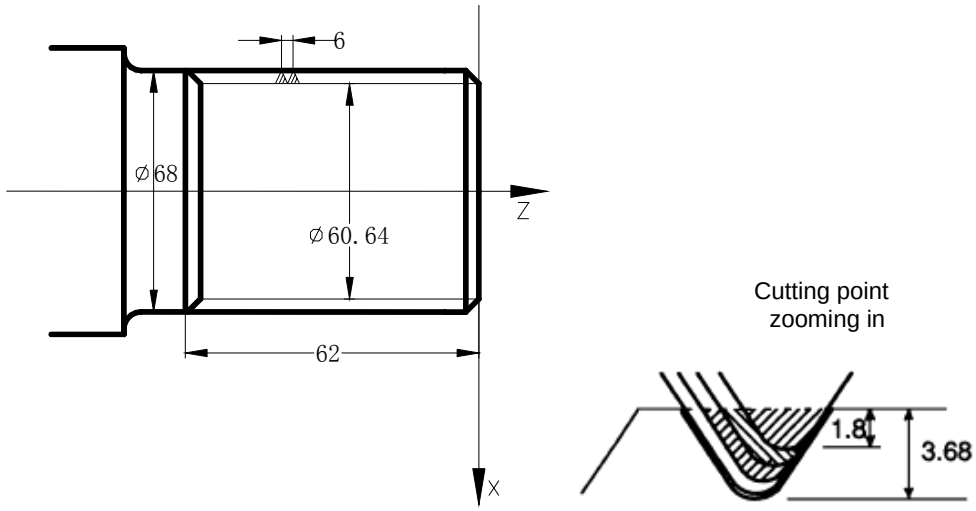


Fig.3-96

Program:

O0013; （**system least input increment is 0.0001mm**）

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 Q1500 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 P36800 Q18000 F6; (Tooth height 3.68, the first cutting depth 1.8)

G00 X100 Z50 ; (Return to starting point of program)

M30; (End of program)

3.23.11 Enhancement Thread Cutting Cycle G78

Command format:G78 P(m)(r)(a) Q(△dmin) R(d) J_ K_ D_ L_ E_;

G78 X(U)__ Z(W)__ R(i) P(k) Q(△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. G78 owns that the thread procession and tail-retraction function can be selected the single-side blade thread cutting or dual-side blade alternative cutting, which is contributed to thread cutting with single tool edge to reduce the wear of tool and to improve the precision of machining thread. G78 can be used for machining the straight and taper thread, But G78 cannot be used for machining the face thread.

Command specifications:

X: X absolute coordinate of thread end point;

U: Different value of X absolute coordinate between thread end point and start point;

Z: Z absolute coordinate of thread end point;

W: Different value of Z absolute coordinate between thread end point and start point;

P (m): Cutting depth selection 0: Equidistance in-feed; 1: Diminishing in-feed (The decreasing in-feed is identical with the G76 in-feed method).

The cutting value of each time of the equidistant in-feed is Q(△d), the cutting value of

each time of the decreasing in-feed is ($\sqrt{n}$ - $\sqrt{n-1}$)×△d (n is the cutting times). The specified value m reservation is enabled after the P(m) is executed, and then write the m value to the data parameter No. 061.

P(r): Cutting method 0: Blade cuts in along with the netrual line of the thread theeth; 1: Blade cuts in along with the left of the thread theeth;

2: Blade cuts in along with the right of the thread theeth; 3: Blade cuts in along thread teeth with left/right in turn;

The specified value R reservation is enabled after the P(r) is executed; and then write the r value to the data parameter No. 062.

P(a): Angles at taper of neighboring two tooth，range: 00～99，unit：deg (°). 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;

△Q(△dmin): Minimum cutting travel of thread roughing, range: 0～999999(IS-C)/ 0～99999(IS-B),(unit: least input increment, radius value). When ($\sqrt{n}-\sqrt{n-1}$) ×△d＜△dmin, △dmin is regarded as the cutting travel of current roughing and the subsequent rough-turning cutting value and the in-feed depth will be not calculated based upon the formula any more. Setting △dmin is to avoid the too small of roughing amount and too many roughing times caused by the cutting amount deceleration in thread roughing. When Q (△dmin) 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(IS_B）/00～99.9999(IS_C) (unit: mm/inch, radius value without sign symbols), the radius value is equal to X absolute coordinates between cut-in point Be of thread finishing and Bf of thread roughing.. The value of system parameter №060 is regarded as the cutting travel of thread finishing when R(d) is not input;

J：Movement amount (run-out amount) (its value range: 0~99999999×least input increment, 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) (its value range: 0~99999999×least input increment 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.

D：Finish-turning times, its range:0~99；

L：Head quantity of multi thread, its value range: 1～99 and it is modal parameter. (The system defaults it is single thread when L is omitted)；

E：The rotation distance value along with X, its range:-9999.9999~9999.9999（unit：mm，radius value、without sign symbol)

R(i)：It is thread taper and is the different value of X absolute coordinate between thread start 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(△d): Depth of the 1st thread cutting, range: 1～99999999 (unit: least input increment,

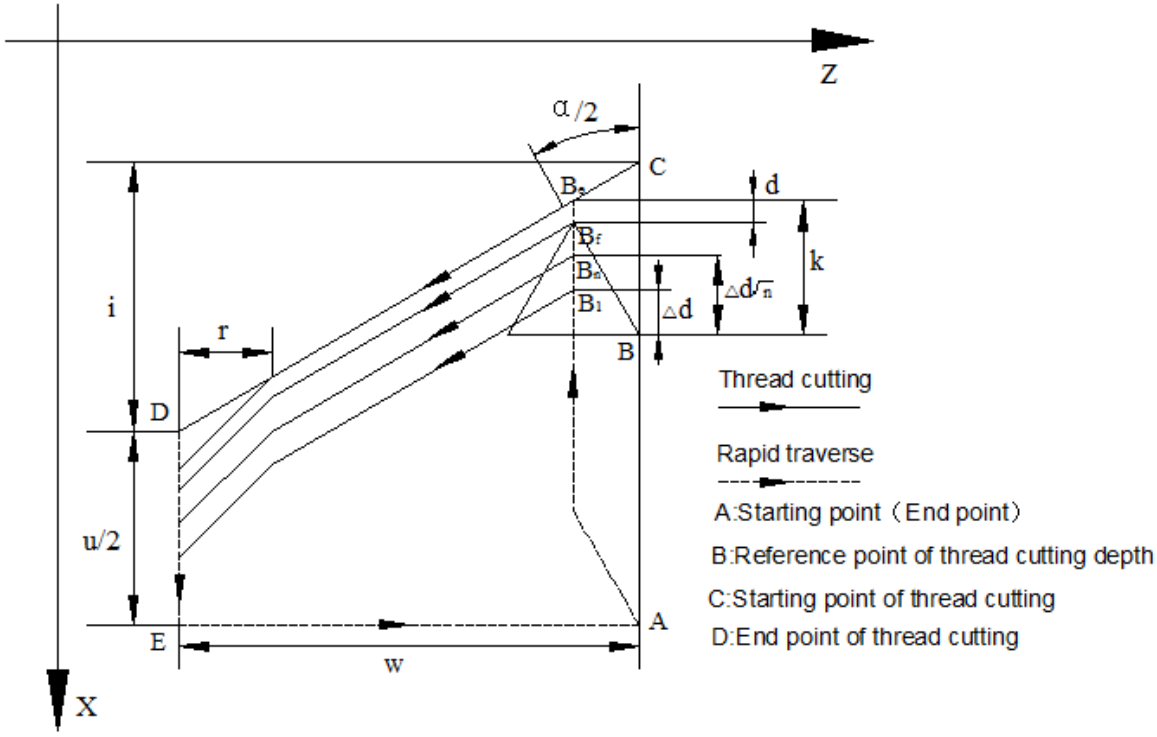
radius value, without sign symbols). The system alarms when △d is not input;

F: metric thread pitch, its range is referred to 1.4.1 Table 1-2;

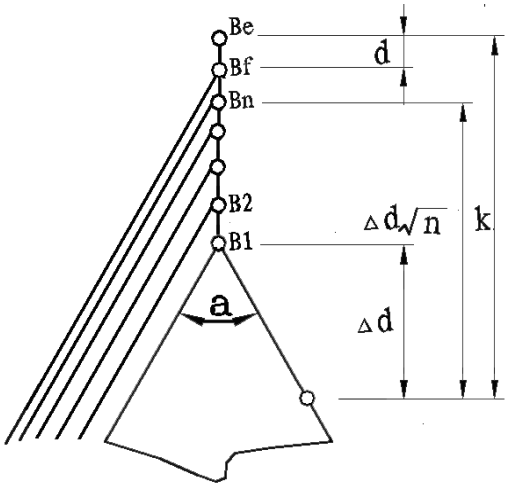
I: thread teeth per inch for inch thread, its range is referred to 1.4.1 Table 1-2.

Command path:

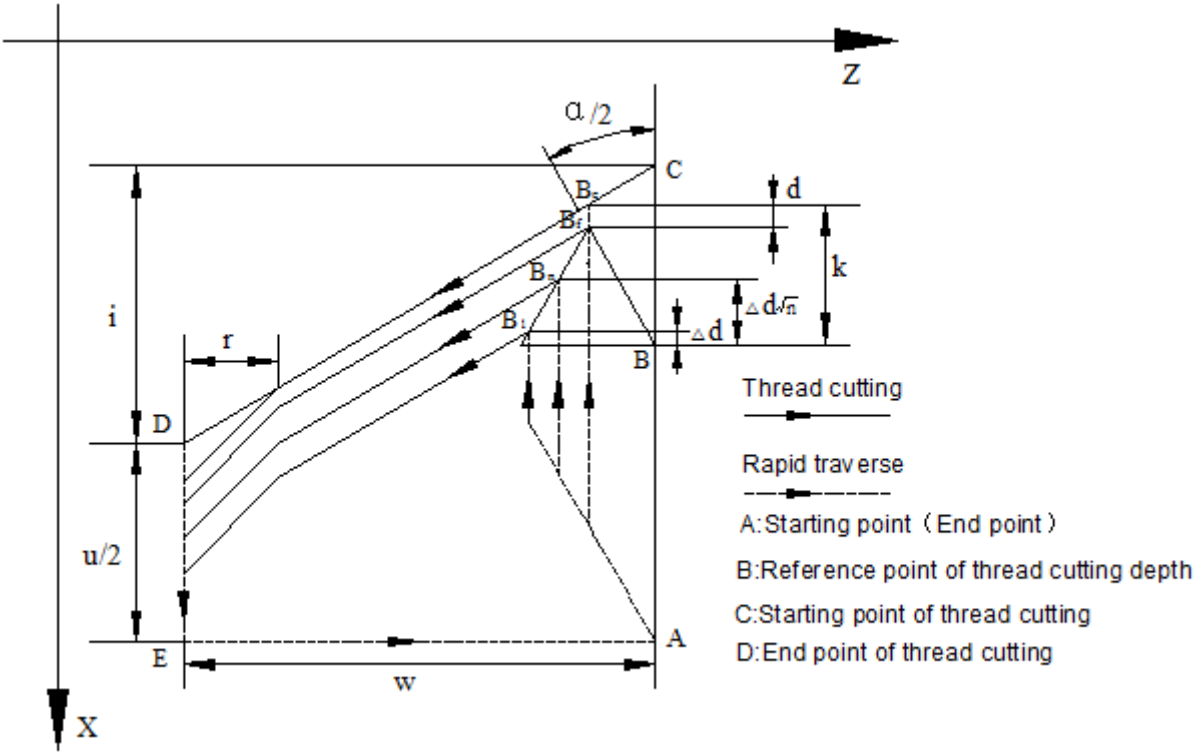
P(r)=0: Cut in from the middle of the teeth



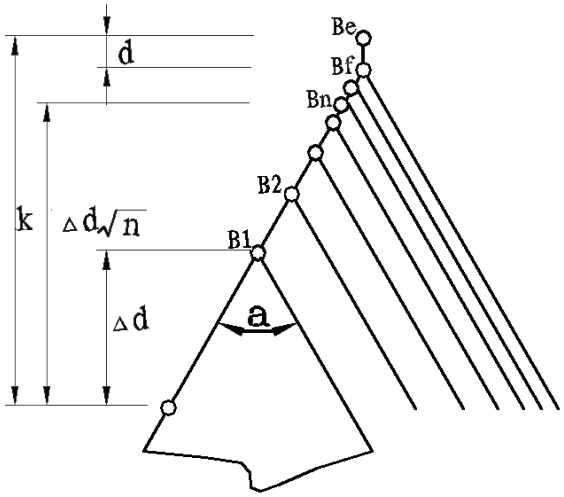
Cut-in method as follows:



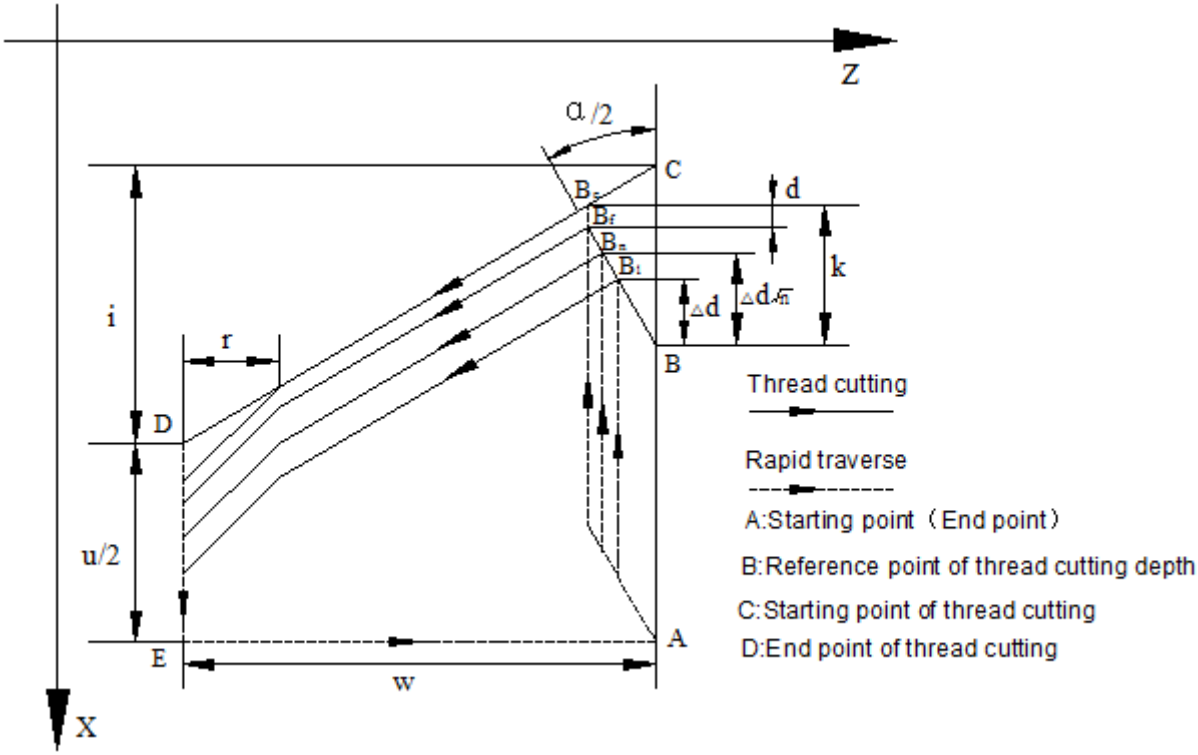
P(r)=1：Left cut-in method



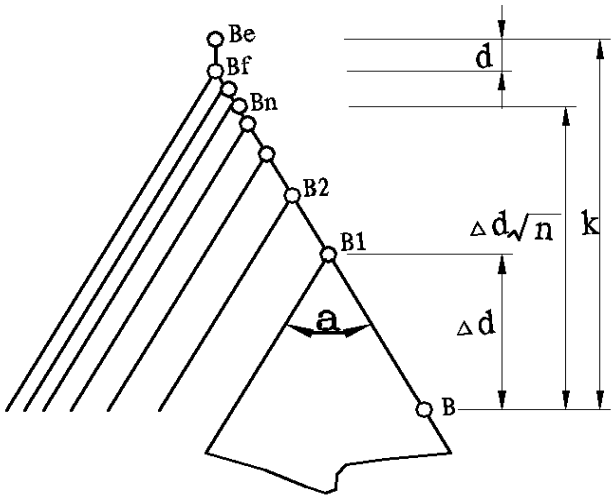
Cut-in method as follows:



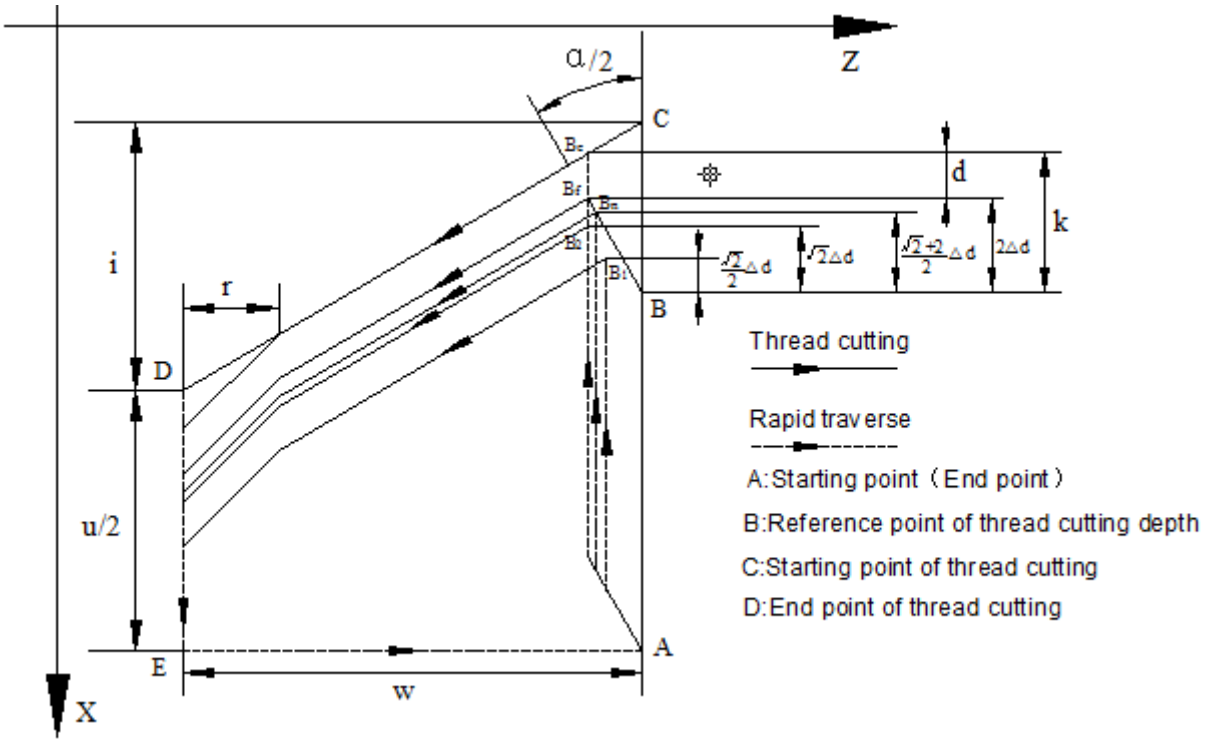
P(r)=2：Right cut-in method



Cut-in method as follows:

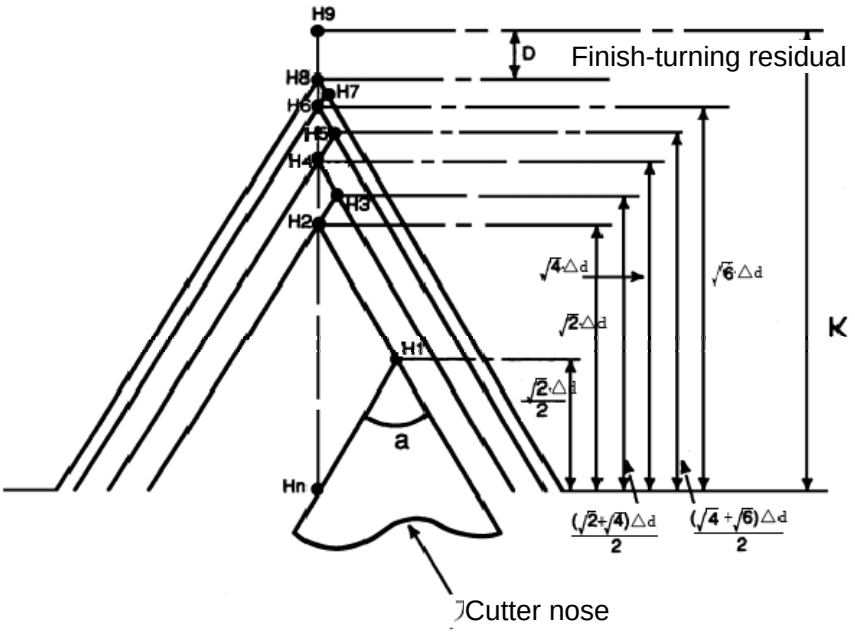


P(r)=3：Shifting cut-in method between left and right

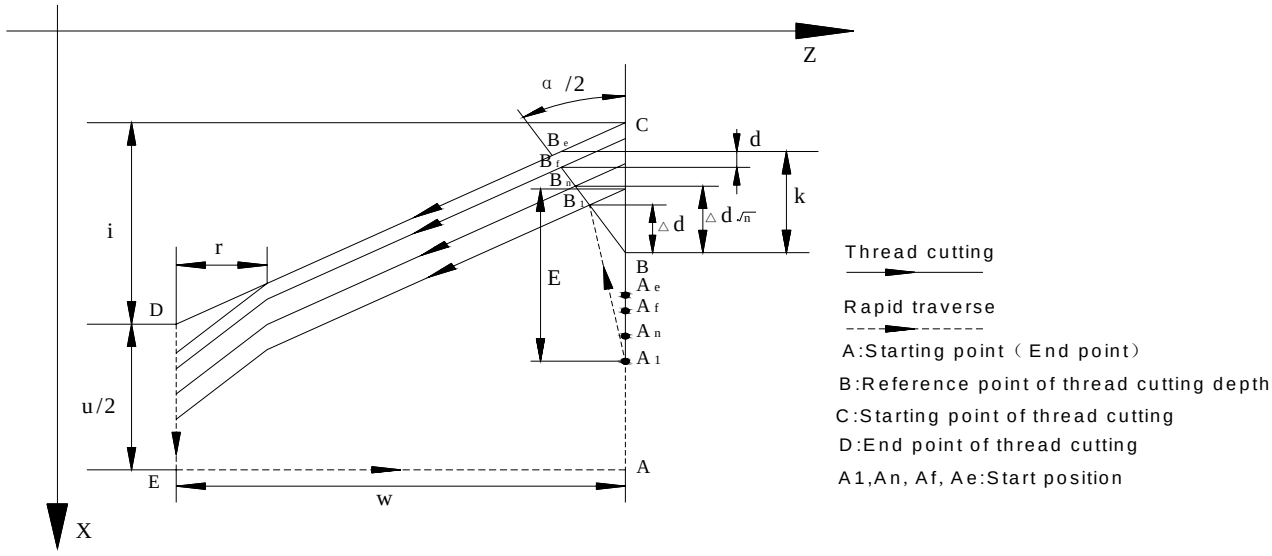


Cut-in method as follows:

Interlaced thread cutting



the rotation function:



3.24 Constant Surface Speed Control G96, Constant Rotational Speed Control G97

The detailed is referred to Section 2.2.3.

3.25 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 №030 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.

Cutting feed: The system can control the motions in X, Z direction contributed that the motion path of tool and the defined path by commands (line straight, arc) is consistent, and also instantaneous speed on the tangent of motion path and F word is consistent, which motion control is called cutting feed or interpolation. The cutting feedrate is specified by F, the system divides the cutting feedrate specified by F according to the programming path into vector in X, Z direction, also controls the instantaneous speed in X, Z direction to contribute that the combined speed of vector in X, Z direction is equal to F command value.

$$f_x = \frac{d_x}{\sqrt{d_x^{\ 2} + d_z^{\ 2}}} \bullet F$$

$$f_z = \frac{d_z}{\sqrt{d_x^{\ 2} + d_z^{\ 2}}} \bullet F$$

F is the combined speed of vector of X/Y-axis instantaneous speed;
d_x is the X-axis instantaneous(d_t) increment,
f_x is the X-aixs instantaneous speed in X direction;
d_z is the Z-axis instantaneous(d_t) increment ,
f_z is the Z-axis instantaneous speed.

Example: In Fig. 2-97, the data in the brackets are the coordinates for each point (it is the diameter in X axis), the system parameter No.022 is 3800, the system parameter No.023 is 7600, the rapid override and feedrate override are 100%.

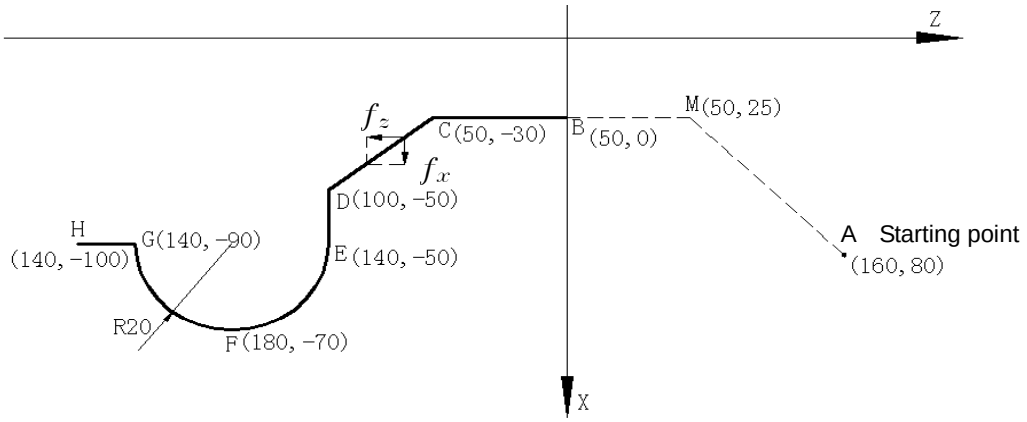


Fig. 3-97

Program as follows:

G50 X160 Z80; (Create a workpiece coordinate system)

G0 G98 X50 Z0; (Rapid traverse from A to B through M point. A→M: X-axis rapid traverse speed 7600mm/min, Z-axis 7600mm/min in Z direction, M→B: X-axis rapid traverse
speed 0mm/min, Z-axis 7600mm/min in Z direction

G1 W-30 F100; (B→C, X-axis rapid traverse speed 0mm/min, Z-axis 100mm/min)

X100 W-20; (C→D, X-axis rapid traverse speed 156mm/min, Z-axis 62mm/min)

X140; (D→E, X-axis rapid traverse speed 200mm/min, Z-axis 0mm/min)

G3 W-40 R20; (EFG circular interpolation, E point: X axis instantaneous speed 200mm/min, Z axis 0mm/min
F point: X-axis instantaneous speed 0mm/min, Z axis 100mm/min)

W-10; (G→H, X axis rapid traverse speed 0 mm/min, Z axis 100mm/min)
M30;

The system supplies 16 steps for spindle override (0%～150%, increment of 10%).PLC ladder defines tune ways of spindle override and whether the actual feedrate override steps is reserved or not after the system is switched off, which is referred by user manual from machine manufacturer when using the system. Refer to the following functions of GSK980TDi standard PLC ladder.

The cutting feedrate can be tuned real time by the feedrate override key on the operator panel or the external override switch, and the actual cutting feedrate is tuned at 16 steps in 0～150% (increment of 10%) but it is invalid for thread cutting to tune the feedrate override. Refer to Ⅱ **OPERATION** about cutting feedrate override.

Related parameters:

System parameter No.027: the upper limit value of cutting feedrate(they are the same in X, Z direction, diameter/min in X direction);

System parameter No.029: exponential function for time constant of acceleration/deceleration when cutting feed and manual feed;

System parameter No.030: initial (ultimate) speed of acceleration/deceleration in exponential function when cutting feed and manual feed.

3.26. Additional Axis Function

3.26.1 Additional Axis Start

Additional axis: Y, 4th, 5th. They can be set to the linear axis or rotary axis. Whether the selected additional axis is valid is determined by the state bit parameter 187, and the axis name is changed by data parameter 225; taking example of Y is as follows:

3.26.2 Motion of Additional Axis

1) rapidly traverse: G00 Y(V)__
2) feed motion: (G98/G99) G01 Y(V)__ F__
3) tapping: G33 Y(V)__ F(I)__
4) machine zero return: G28 Y(V)__
5) machine 2nd, 3rd, 4th reference point return: G30 P2(3,4) Y(V)__
6) G50 setting a coordinate system: G50 Y(V)__
7) Manual/Step/MPG feed, program zero return, manual machine zero return.

Note 1: Axis name is Y, absolute coordinate programming axis is Y, relative coordinate programming axis is V. Axis name is C, absolute coordinate axis name is C, relative coordinate axis name is H. When axis name is A or B, the relative coordinate programming axis name and absolute coordinate programming axis name are the same.

Note 2: The additional axis Y does not execute X/Z interpolation motion;

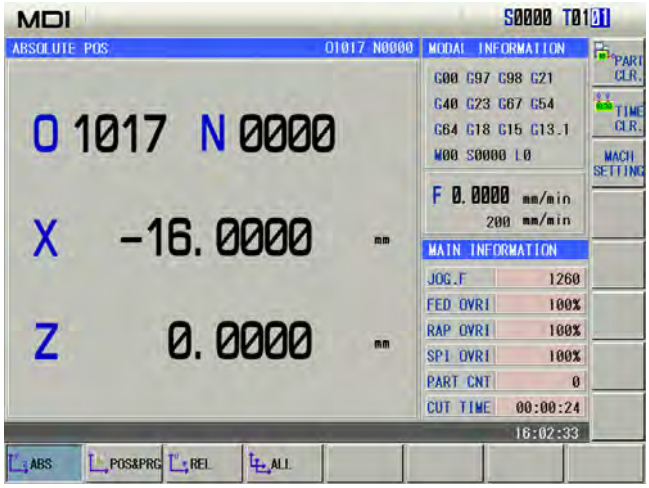
Note 3: Y(V) in G00, G28 , X(U) , Z(W) are in the same block, and each rapidly traverses with their separately specified speed;

Note 4: Y(V) in G50 , X(U) , Z(W) are in the same block;

Note 5: Y(V) in G01, X(U) , Z(W) are not in the same block, otherwise, the system alarms;

Note 6: Use the modal F of X/Z when G01 traverse speed of Y is not specified; the time constant is set by №29.

3.26.3 Additional Axis Coordinates Display



3.27 Macro Commands

GSK980TDi 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.

3.27.1 Macro Variables

- **Presentation of macro variables**

  Present with “#”＋ macro variables number.;

  Format: # i(i=100,102,103,……) ;

  Alternatively, the variable is specified by “#”+[Expression]

  Format:# [**i**] (i = #100 + #102 + #103）

  Example:#105，#109，# [#125 + #100 * #111]。

- **Variable Type**

  The variable is divided into four types according to the variable number:

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**

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: 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.

- **Null variable**

When the variable value is not defined, it is null, the variable #0 is always null and only is read instead of writing.

1). Reference

When an undefined variable (null variable) is referred, the address is ignored.

#1=<null>	#1=0
G00 X100 Z#1 is equal to G00 X100	G00 X100 Z#1 is equal to G00 X100 Z0

2). Operation

Except for using <null variable > to value, the <null variable> used to operation in other conditions is the same as that of “0”.

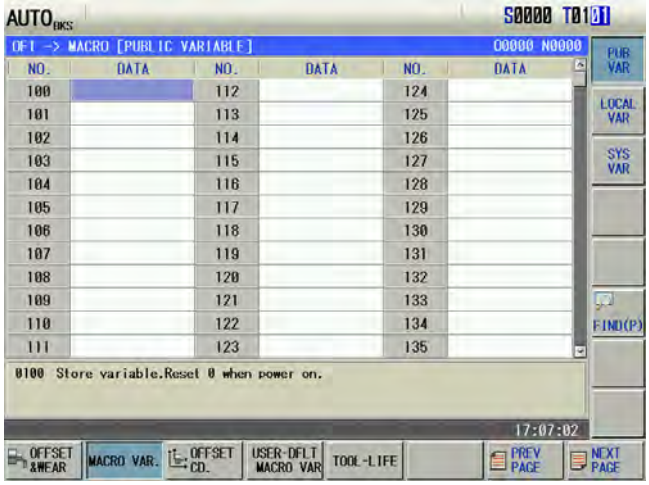
#1=<null>	#1=0
#2=#1 ↓(execution result) #2=<null>	#2=#1 ↓(execution result) #2=0
#2=#1 * 5 ↓(execution result) #2=0	#2=#1 * 5 ↓(execution result) #2=0
#2=#1+#1 ↓(execution result) #2=0	#2=#1+#1 ↓(execution result) #2=0

3). Condition expression

<null>s in EQ(=) & NE(≠) are different to “0”.

#1=<null>	#1=0
#1 EQ #0 ↓ Valid	#1 EQ #0 ↓ Invalid
#1 NE #0 ↓ Invalid	#1 NE #0 ↓ Invalid
#1 GE #0 ↓ Valid	#1 GE #0 ↓ Invalid
#1 GT #0 ↓ Invalid	#1 GT #0 ↓ Invalid

- **Variable display**



(1) In macro window, the variable being displayed to the null means it is null, i.e. it is not defined.

(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.

- **System variable**

（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
#1000~#1015 #1032	Correspond G54.0~G54.7, G55.0~G55.7 signal states
	Correspond G54, G55 signal states
#1100~#1115 #1132	Correspond F54.0~G54.7, F55.0~F55.7 signal states
	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 … 32	#2001 … #2032	#2701 … #2732	#2101 … #2132	#2801 … #2832	#2201 … #2232	#2901 … #2932	#2301 … #2332	#2401 … #2432	#2451 … #2482

(3) Machined workpiece number:

Variable No.	Function
#3901	Machined workpiece number(completion)
#3902	The Max. machining numbers

(4) System modal information variable

Variable No.	Function	
#4001	G00, G01, G02, G03, G05, G32, G33, G34, G80,	No. 1 group

	G84, G88, G90, G92, G94, G124(G06.2), G126(G06.3),G132(G32.1),G144(G07.2),G146(G07.3)	
#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) The system variable of MPG interruption value: Readable variable

Variable No.	Function
#5121	The 1st axis MPG interruption value
#5122	The 2nd axis MPG interruption value
#5123	The 3rd axis MPG interruption value
#5124	The 4th axis MPG interruption value
#5125	The 5th axis MPG interruption value

（7）Compensation values of workpiece coordinate systems

Variable No.	Function
#5201~#5205	External workpiece zero offset value of 1st ~5th axis
#5221~#5225	G54 workpiece zero offset value of 1st ~5th axis
#5241~#5245	G55 workpiece zero offset value of 1st ~5th axis
#5261~#5265	G56 workpiece zero offset value of 1st ~5th axis
#5281~#5285	G57 workpiece zero offset value of 1st ~5th axis

#5301~#5305	G58 workpiece zero offset value of 1st ~5th axis
#5321~#5325	G59 workpiece zero offset value of 1st ~5th axis

Note: Message described in the above table separately corresponds to X, Y, Z, the 4th, 5th axis. For example, #5201 means the offset message of X axis, #5202 means the one of Y axis，#5203 means the one of Z axis,#5204 means the one of the 4th axis, #5205 means the one of the 5th axis.

(8) Application:

```
O0100; (#100 is the machined workpiece number, its initial value=0)
  G00 X100 Z100;
  T0101;
  IF[#100<100]GOTO10; (has continuously machined 100？)
  G65 P9580 U-0.01 W-0.01; (call the macro program compensation tool wear)
  N10 G00 X50 Z50;
  ……;
  ……;
  ……;
  Machine program;
  ……;
  T0202;
  IF[#100<100]GOTO20; (has continuously machined 100？)
  G65 P9580 U-0.01 W-0.01; (call the macro program compensation tool wear)
  N20 ……;
  ……;
  ……;
  #100=#100+1; (machined workpiece number +1)
  IF[#100==101]THEN #100=0;
  M30;
```

```
O9580;
G65 H23 P#101 Q#4120 R100; (obtain the tool offset No.)
IF[#101==0]GOTO10; (Do not modify the wear value when the tool offset is regarded as 0)
#102=2000+#101; (X wear macro variable)
#103=2100+#101; (Z wear macro variable)
#9102=#9102+#21; (modify X wear)
#9103=#9103+#23; (modify Z wear)
T#4120; (call a new tool offset value)
N10 M99;
```

3.27.2 Operation & Jump Command G65

Command format:

G65 Hm P# i Q# j R# k;

Command significance:

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.

i = #j O # k

Operation sign specified by Hm

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
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

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 = $\sqrt{\#J}$

G65 H21 P#I Q#J

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

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 = $\sqrt{\#J^2+\#K^2}$

G65 H27 P#I Q#J R#K

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

16) 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))

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 1: The alarm content (/*) can be omitted. When this alarm content is omitted, only the alarm occurs, and the alarm content is null.**

Note 2: 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.

3.27.3 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.

3.27.3.1 One-Shot Call G65

Command format: G65 P_ L_ <argument>_;

Macro program specified by P is called, the argument (data) is sent to the user macro program body.

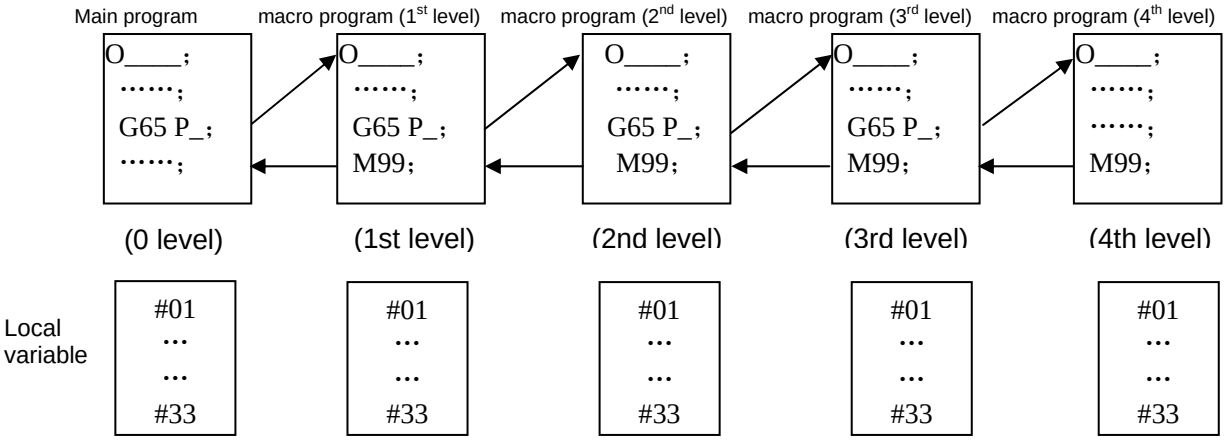
Command explanation:

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.



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 Ii, 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.

Note 3: CNC internal can automatically identify the argument specifying mode I and II. When the two modes are specified, the later specified is valid.

3.27.3.2 Modal Call G66

Command format: G66 P_ L_ <Argument>_;

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: G66 call has four-level nest.

Note:

1) **Macro program is called firstly after G66 block is executed.**
2) **The macro program is called again after G00, G01, G02, G03, G05 are executed (after G66 is executed and before the modal call is cancelled).**
3) **Call the value which is updated from the argument to the local variable.**
4) **G65 call will automatically cancel G66 modal call.**

3.27.3.3 Modal Call Cancel G67

Command format: G67;

Command explanation: cancel G66 modal macro program call

Application:

O2005(O2005) ;

G00 X100 Z50;

G66 P0100 L2 A2 B20 C20 I30 J20 K20; call P0100 two times when the system executes the block

G01 X80 Z50; call P0100 two times (update the local variable according to the argument) after

the system has executed the block
G67; cancel G66 modal call
G01 X20 Z50; the system does not call P0100 after it executes the block
M30;

3.28 Statement Macro Command

3.28.1 Arithmetic & Logic Operation

Arithmetic and logic operation

Function	Expression format	Remark
Definition or assignment	#i = #j	
addition subtraction multiplication division	#i = #j + #k #i = #j - #k #i = #j * #k #i = #j / #k	
Or And Exclusive Or	#i = #j OR #k #i = #j AND #K #i = #j XOR #K	Logic operation is executed by the binary system
Square root Absolute value Rounding-off FUP FIX Natural logarithm Exponential function	#i = SQRT[#j] #i = ABS[#j] #i = ROUND[#j] #i = FUP [#j] #i = FIX [#j] #i = LN[#j] #i = EXP[#j]	
Sine Arc sine Cosine Arc cosine Tangent Arc tangent	#i = SIN[#j] #i = ASIN[#j] #i = COS[#j] #i = ACOS[#j] #i = TAN[#j] #i = ATAN[#i]/ [#j]	Angle unit is specified by degree. For example: 90°30' is expressed by 90.5°
BCD to BIN BIN to BCD	#i = BIN[#j] #i = BCD[#j]	Used for switching with PMC

Relative explanation:

1. Angle unit

   Angle units of SIN, COS, ASIN, ACOS, TAN and ATAN are degree (°). For example: 90°30 ´ means to be 90.5° (degree).

2. Arc sine # i=ASIN[#j]

   i. result output range:

   No.180#7 NAT is set to 1: 90°~ 270°;

   No.180#7 NAT is set to 0: -90°~ 90°;

   ii. when #j exceeds the range from -1 to 1, the system alarms P/S.

   iii. the constant replaces the variables #j.

3. Arccosine # i =ACOS[#j]

   i. Result output range 180°~ 0°.

ii. When #j exceeds the range from -1 to 1, the system alarms P/S.

iii. The constant replaces the variables #j.

4. Arc tangent #i=ATAN[#j]/[#k]

Specify the lengths of two sides and separate them with a slash "/".

i. Result output range:

When No.180#7 NAT is set to 1: 90°~ 270°;

[For example] #1=ATAN[-1]/[-1]: #1=225°;

When No.180#7 NAT is set to 0 -90°~ 90°;

[For example]#1=ATAN[-1]/[-1]: #1=45.0°;

ii. The constant replaces the variables #j.

5. Natural logarithm #i=LN[#j]

The constant replaces the variables #j

6. Exponential function #i=EXP[#j]

The constant replaces the variables #j

7. ROUND function

When arithmetical operation or logic operation IF or WHILE includes ROUND, ROUND rounds in the first decimal place.

For example: #1=ROUND[#2]: #2=1.2345, the variables 1 is 1.0.

8. FUP FIX

After CNC executes the operation, the result integer absolute value is bigger the previous absolute value, which is called FUP; the result integer absolute value is less than the one, which is call FIX. Pay more attention to the negative execution.

Example:

Hypothetically, #1=1.2, #2= -1.2

When #3=FUP[#1] is executed, 2.0 is assigned to #3.

When #3=FIX[#1] is executed, 1.0 is assigned to #3.

When #3=FUP[#2] is executed, -2.0 is assigned to #3.

When #3=FIX[#2] is executed, -1.0 is assigned to #3.

3.28.2 Transfer & Cycle

In the program, the system uses GOTO and IF statement to change the control flow. There are three types of transfer and cycle operation.

1. GOTO statement (unconditional transfer).
2. Condition control IF statement.
3. WHILE cycle statement.

1) Unconditional transfer (GOTO statement)

Transfer to the block which serial number is n. The system alarms when others exceeds the range from 1 to 99999, and it specifies the serial number with the statement.

Format: GOTO n; n: serial number(1~99999)

Example: GOTO 1;

GOTO #101;

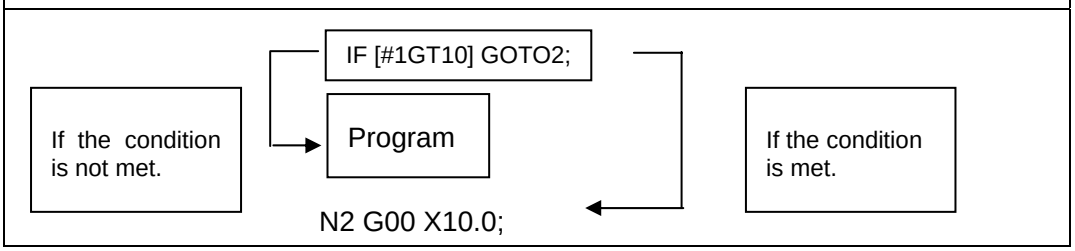
2) Conditional control (IF statement)

GOTO format: IF[conditional statement]GOTO n;

When the specified conditional statement is valid, the system transfers to the block which serial number is n; When the specified conditional statement is valid, the system executes the next block.

Example:

When the variable #1 is more than 10, the system transfers to the block which serial number is N2.



THEN format: IF[conditional expression]THEN<macro program statement >;

When the condition expression is valid, the system executes only one statement following THEN.

Example: IF[#1 EQ #2] THEN #3=0;

When #1 value is equal to the #2, 0 is assigned to the variable #3; when they are not equal, the system orderly executes the followings instead of the assignment statement after THEN.

Conditional expression: the conditional expression must include the conditional operator, two sides of conditional operator can be variable, constant or expression, and it must be closed with the brackets '[' ']'.

Conditional operator: the system uses the conditional operators listed in the following table.

Conditional operator	Meaning
EQ or = =	Equal to (=)
NE or <>	Not equal to (≠)
GT or >	More than (>)
GE or >=	More than or equal to (≥)
LT or <	Less than (<)
LE or <=	Less than or equal to (≤)

Example: IF[3<>2]GOTO 2; its meaning: when 3 is not equal 2, the system skips to N2 block;

IF[#101>=7.22]THEN #101=SIN30; its meaning: when #101 is more than or equal to 7.22, the system executes the assignment after THEN. i.e. the sine value of 30 degree is assigned to the variable #101.

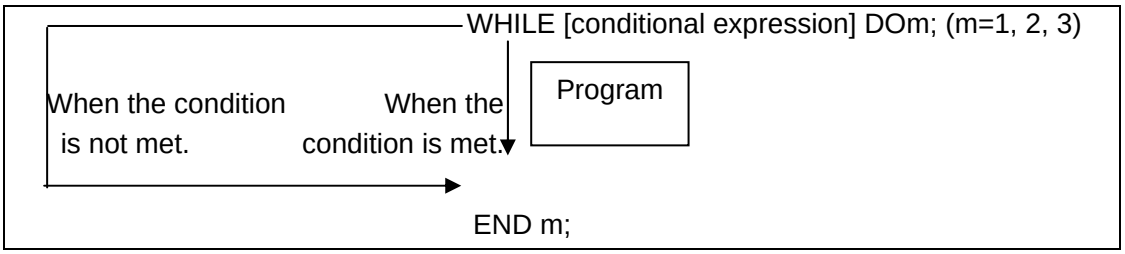
Typical program: the following program counts the sum of the integer 1~10.

```
O9500
#1=0;  … …the sum is initialized to be 0
#2=1;  … …the summand number is initialized to be 1
N1 IF[#2 GT 10]GOTO2;  … … the system skips to N2 when the summand is more than 10
#1= #1+#2;     … …count the sum of two numbers
#2= #2+1;     … … the summand adds 1
GOTO1;     … … unconditionally skip to the block N1
N2 M30;     … … end of program
```

3) Cycle (WHILE statement)

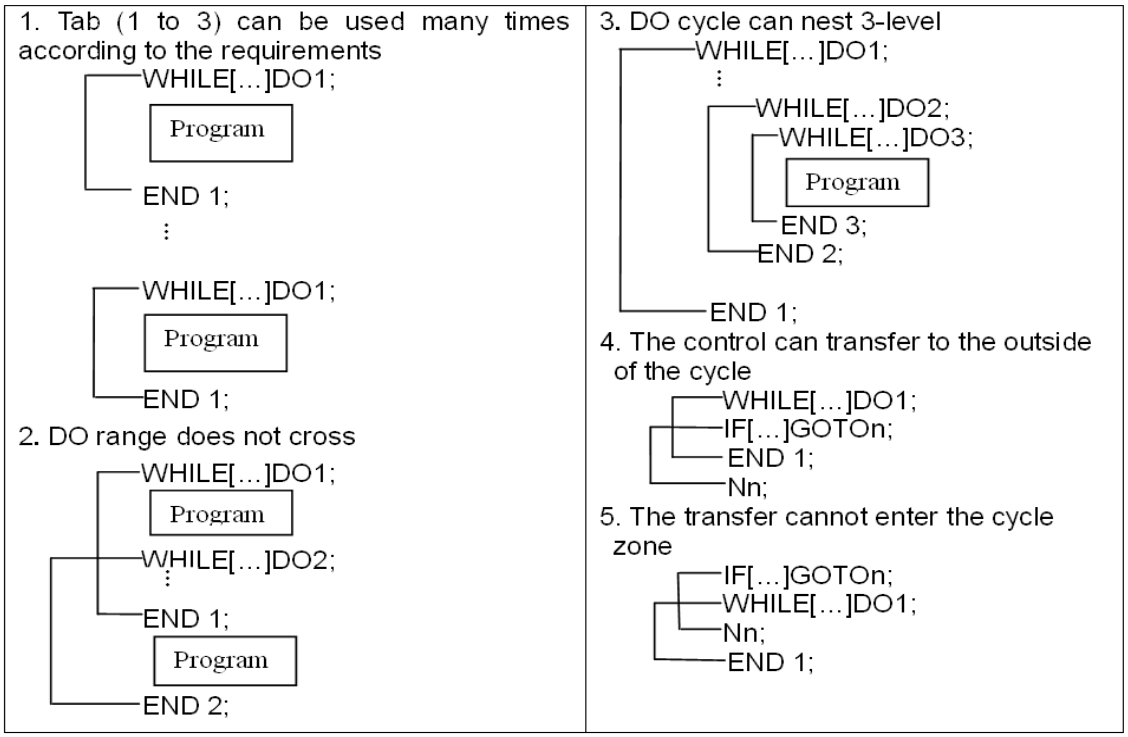
Specify one conditional expression after WHILE. When the specified conditional is valid, the system executes the blocks between DO and END; otherwise, the system skips to the block after END.

Example:



Explanation: when the specified condition is valid, the system executes the block between DO and END; otherwise, executes the block after END. The two tabs after DO and END are consistent, and the tab value can be 1, 2 or 3, otherwise, the system alarms.

Nest: the tab (1~3) in DO, END can be used many times. But the system alarms when there is the intercross repetitive cycle in the program.



3.29 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# 0 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**

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 №.45~ №.46、№.56、№.59、№.60、№.114、№.120~ №.123、№.140、№.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.47 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:

(3) 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:

- 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: №.22、№.27、№.28、№.30～№.33、№.41、№.107、№.113;
- 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: №.34、№.45~ №.47、№.99 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.

3.30 Stored Stroke Limit 2 Detection G22 or G23

Code format: G22 X__ Y__ Z__ I__J__ K__;
G23;

Code function: Set the stored stroke limit 2 detection. Make out a fixed area, and specify the inner or outer side (Bit 2 of parameter No.172 setting) of this area as the limit area. An alarm issues when the machine tool coordinate enters into the limit area.

Code explanation: G code of the group 9

G22： Open the stored stroke limit detection 2

G23： Close the stored stroke limit detection 2

X_Y_Z_： The limit coordinate value along positive direction, after the command is performed, this data is then converted into the corresponding value to save in the data parameter No.100. When the data does not input, the value of the data parameter No.100 is regarded as the limit coordinate value along the positive direction based upon the stored stroke limit detection.

I_J_K_： The limit coordinate value along negative direction, after the command is performed, this data is then converted into the corresponding value to save in the data parameter No.101. When the data does not input, the value of the data parameter No.101 is regarded as the limit coordinate value along the negative direction based upon the stored stroke limit detection.

Precautions:

1) The limit coordinate values of the stored stroke limit detection 2 are the coordinate values of the machine tool coordinates;
2) When the Bit 4 of state parameter No. 172 sets to 0, the limit function is disabled, the modal is fixed as G23; the alarm may issue by using the G22/G23 command;
3) The initial modal is G23 when the system power is turned on;
4) The inner/outer prohibition of the setting range is set by bit 2 of parameter No. 172;
5) When the bit 1 of state parameter No. 172 sets to 1, if the stored stroke limit detection 2 sets as inner limit, the limit function is then enabled after the overall axes are returned to zero following with the system power is turned on; if it sets as the outer limit, after the system ON, the limit function of an axis is enabled after this axis returns to zero;
6) G22/G23 coe can not be shared a same block with other G and MST codes; otherwise, the alarm issues.
7) G22 code only can be changed the limit area of the X, Z or Y axis, if the limit area of the rest of the axes should be altered, which can be set by the data parameter No. 100 or No.101;
8) When the prohibition area of the stored stroke limit detection 2 is set, as well when the positive stroke is less than or equals to the negative one and the external prohibition is selected, the overall areas are the forbidden areas;
9) When the prohibition area of the stored stroke limit detection 2 is set, as well when the positive stroke is less than or equals to the negative one and the internal prohibition is selected, the overall areas are the movable areas;
10) Set whether an axis is performed the stored stroke detection 2 by the bit 2 of parameter No.190.

3.31 G Code Macro Program Call

Code format: G□□□□ L_ <Agrument >_

Code value（0~9999）

Code explanation: Code value: It is set by data parameter №.86~№.95;

L: The called times (It defaults to 1 if it omits, which can be specified the repeated times from 1 to 9999);

<Argument>: The data converted to the macro program, its value is assigned to the corresponding local variable, it is same that the G65 calls macro program.

Code value and the setting of program number call:

Data parameter No.86~No.95 set the used G code in the macro command call, use the set G code, that is, the macro program call using G code can be defined. If the data parameters No.86~No.95 are set to 0, this function will be then disabled. The callable macro programs are O9010~O9019, the corresponding relationships between parameter number and program number are as follows:

Para. No.	Program No.	Para. No.	Program No.
86	O9010	91	O9015
87	O9011	92	O9016
88	O9012	93	O9017
89	O9013	94	O9018
90	O9014	95	O9019

For example: №.86 ＝ 100, №.87 ＝ 103, then:

```
G00 X0 Z0;
G100;        //Call the No. O9010 macro program
G103;        // Call the No. O9011 macro program
…;
M30;
```

Precaution:

The allowable range of the data parameters No. 86~No. 95 setting value is 0～9999, when the setting value is identical with the G code, this function is automatically disabled.

3.32 Editable Parameter Read/Write G10 L52

Code format: G10 L52; Set the parameter read or write method

N_/K_/D_/DT_/DC_ R(W)_; Read/write program of the system/K/D/DT/DC parameters

N_ P_ R(W)_; Read/write program of axis, spindle or channel parameter

…

G11; Cancel the parameter read/write

Code explanation: N_ ：Parameter number；

K_ ：K parameter number of PLC；

D_ ：D parameter number of PLC；

DT_ ：DT parameter number of PLC；

DC_ ：DC parameter number of PLC；

R(W) _：R is the read address from parameter, which only can be regarded as the macro variable number；W is the write-in value from parameter, which can be treated as the constant or marco variable；When the read/write parameter is the state one, its value from reading/writing is binary, and the others are decimal system;；

P_: The axis number 1 to the top controllable axis number (from 1 to the

Max. channel number) can be used for reading the axis, spindle or channel parameter;

Precautions:

1) In the parameter write/read method, simultaneously, when two or more command words (including the repeated display for a same command word) of N_/K_/D_/DT_/DC_ are displayed at the same line, system alarm is then shown: 178 alarm, parameter (N_/K_/D_/DT_/DC_) (including the inexistent specified parameter) does not specify in the parametr read/write mode or the parameter is repeatedly specified;
2) In the parameter write/read method, when block specifies the operation command (R/W), but it does not exist any one command word of the N_/K_/D_/DT_/DC_, alternatively, there is the command word without data, system alarm is then shown: 178 alarm, parameter (N_/K_/D_/DT_/DC_) (including the inexistent specified parameter) does not specify in the parametr read/write mode or the parameter is repeatedly specified;
3) In the parameter write/read method, when the block specifies the parameter number, but it does not exist in R/W operation command or R, W exist at the same time, system alam is then shown: 177 alarm; the read/write method does not specify or the read/write method is repeatedly specified in the parameter read/write method;
4) In the parameter write/read method, if there are command words other than the _/P_/K_/D_/DT_/DC_ R_/W_, ignore it, the command operations except from the parameter reading/writing will not be performed;
5) Specify P value in the block, if the axis corresponding set parameter does not exist, igore it;
6) When reading or writing the axis parameter or spindle parameter, if the P value does not occur, the system alarm is then displayed: 178 alarm; the axis number does not specify when reading/writing the axis parameter;
7) The system can not set the parameter with 2 authority levels or above if it is under the 3 or less authories. If the program exists the block to be modified the parameter with 2 or more authority levels, the system alarm is then displayed: 174 alarm, the current authority prohibits to be modified the parameter;
8) In the parameter write/read method, R_ can only be accompanied with macro variable, otherwise, the alarm occurs: 180 alarm, as well, R only can be specified the marco variable in this method;
9) When writing the bit parameter, if the W_ is followed with the operation number other than the binary constant and macro variable, the system alarm is then displayed: 176 alarm. The W value or the macro variable value followed with the W are not the enabled binary value;
10) When writing the bit parameter, when the value followed with the W exceeds the corresponding parameter range, the system alarm is then displayed: 175 alarm; The W value or the macro variable value followed with the W are not the enabled binary value;
11) The dual-channel system forbids other channels to perform the machining procedure in the parameter read/writing state; otherwise, the system alarm is then displayed: 186 alarm; the other channels can not performed the machining procedure during the parameter is read or written;
12) The parameter read/write function+n can not be used together with the tool life span administration function.

Programming example:

1) State parameter read, write example

Program	Explanation
＃101 ＝ 10000000；	Macro variable No. 101 is the setting value of the state parameter No.2.
G10 L52；	Set the parameter read or write method
N1 R#100；	Read the state parameter No.1 to the macro variable No.100.
N1 W00000010；	Set the state parameter No.1 as 00000010.
N2 W#101；	Set the state parameter No.2 as the value of the macro variable No.101.
G11；	Cacnel the read or write method of the parameter.

2) Data parameter read/write example

Program	Explanation
＃101 ＝ 50；	Marco variable ＃101 is the setting value of the data parameter No.31.
G10 L52；	Set the parameter read or write method
N30 R#100；	Read data parameter No.30 to the macro variable ＃100.
N30 W200；	Set the data parameter No.30 as 200.
N31 W#101；	Set the data parameter No. to the value of the macro variable ＃101.
G11；	Cacnel the read or write method of the parameter.

3) Axle type parameter read or write example

Program	Explanation
＃103＝00000101；	Macro variable ＃103 is the setting value of the data parameter No.188 along with the 1st axis (X axis).
＃104＝100；	Macro variable ＃104 is the setting value of the data parameter No.23 along with the 5th axis (C axis).
＃105＝6000；	Macro variable ＃105 is the setting value of the data parameter No.38 along with the 2nd axis (S2).
G10 L52；	Set The read or write method of parameter.
N187 P1 R#100；	Read the 1st axis parameter value of the axle-type state parameter No, 187 to the macro variable ＃100.
N187 P5 W01100001；	Set the axle type state parameter No.187 along with the 5th axis is 01100001.
N188 P1 W#103；	Set the 1st axis of the axle type state parameter No.188 to the value of the macro variable ＃103
N22 P1 R#101；	Read the 1st axis parameter value of the axle type data parameter No.22 to the macro variable ＃101.
N22 P2 W8000；	Set the 2nd axis parameter of the axle type data parameter

	No.22 as 8000.
N23 P5 W#104；	Set the 5th axis of the axle type data parameter No.23 to the value of the macro variable ＃104
N37 P1 R#102；	Read the 1st axis parameter value of the axle type data parameter No.37 to the macro variable ＃102.
N37 P3 W8000；	Set the 3rd axis parameter of the spindle data parameter No.37 as 8000.
N38 P2 W#105；	Set the 5th axis of the spindle data parameter No.38 to the value of the macro variable ＃105
G11；	Cacnel the read or write method of the parameter.

Note: The read/write programming method of the K parameter is consistent with the one of the system state parameter; The read/write programming method of the D/DT/DC parameters are coincident with the one of the system data parameter.

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.

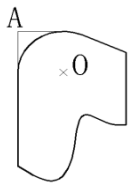


Fig. 4-1 Tool

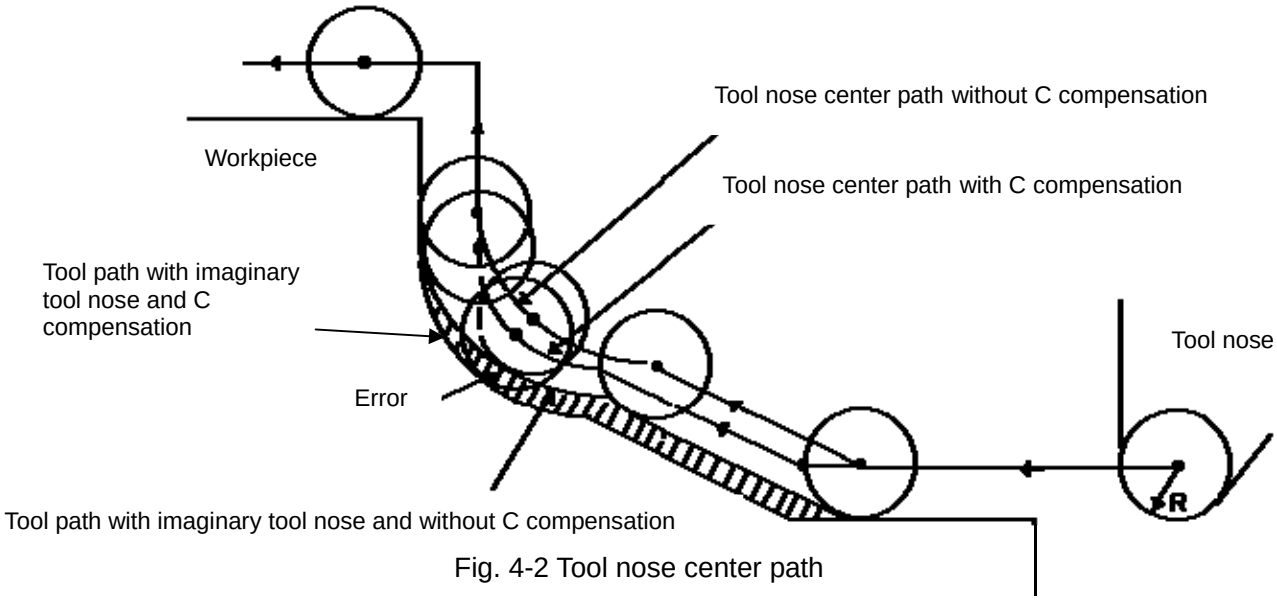


Fig. 4-2 Tool nose center path

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.

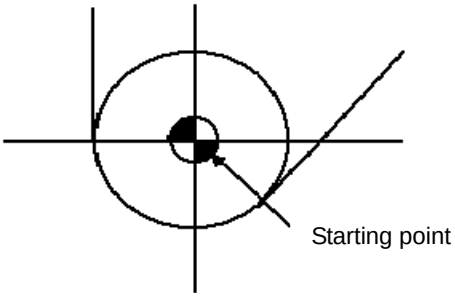


Fig. 4-3 Programming with tool nose

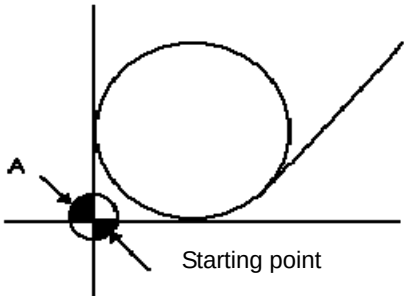


Fig. 4-4 Programming with imaginary tool nose

Tool path is shown in Fig.4-5 when the tool nose center programming is executed.

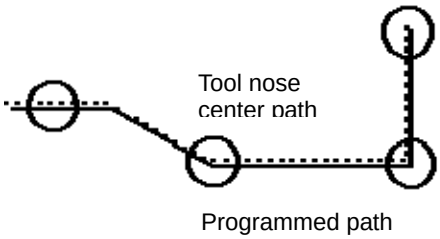


Fig. 4-5-1 the tool nose center path is the same that of programmed path without tool radius compensation

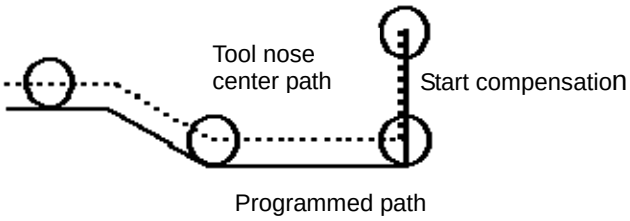


Fig. 4-5-2 precise cutting with tool radius compensation

The tool path is shown in Fig. 4-6 when imaginary tool nose programming is executed

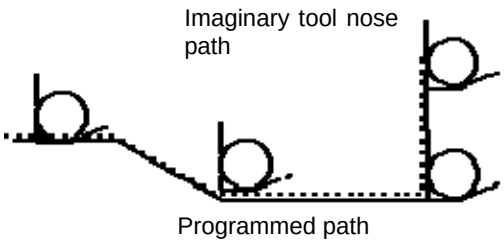


Fig. 4-6-1 the imaginary tool nose path is the same that of programmed path without tool radius compensation

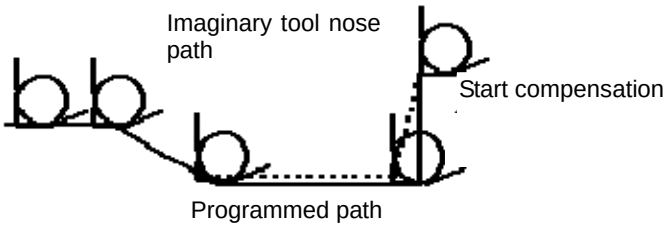


Fig. 4-6-2 precise cutting with tool radius compensation

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.

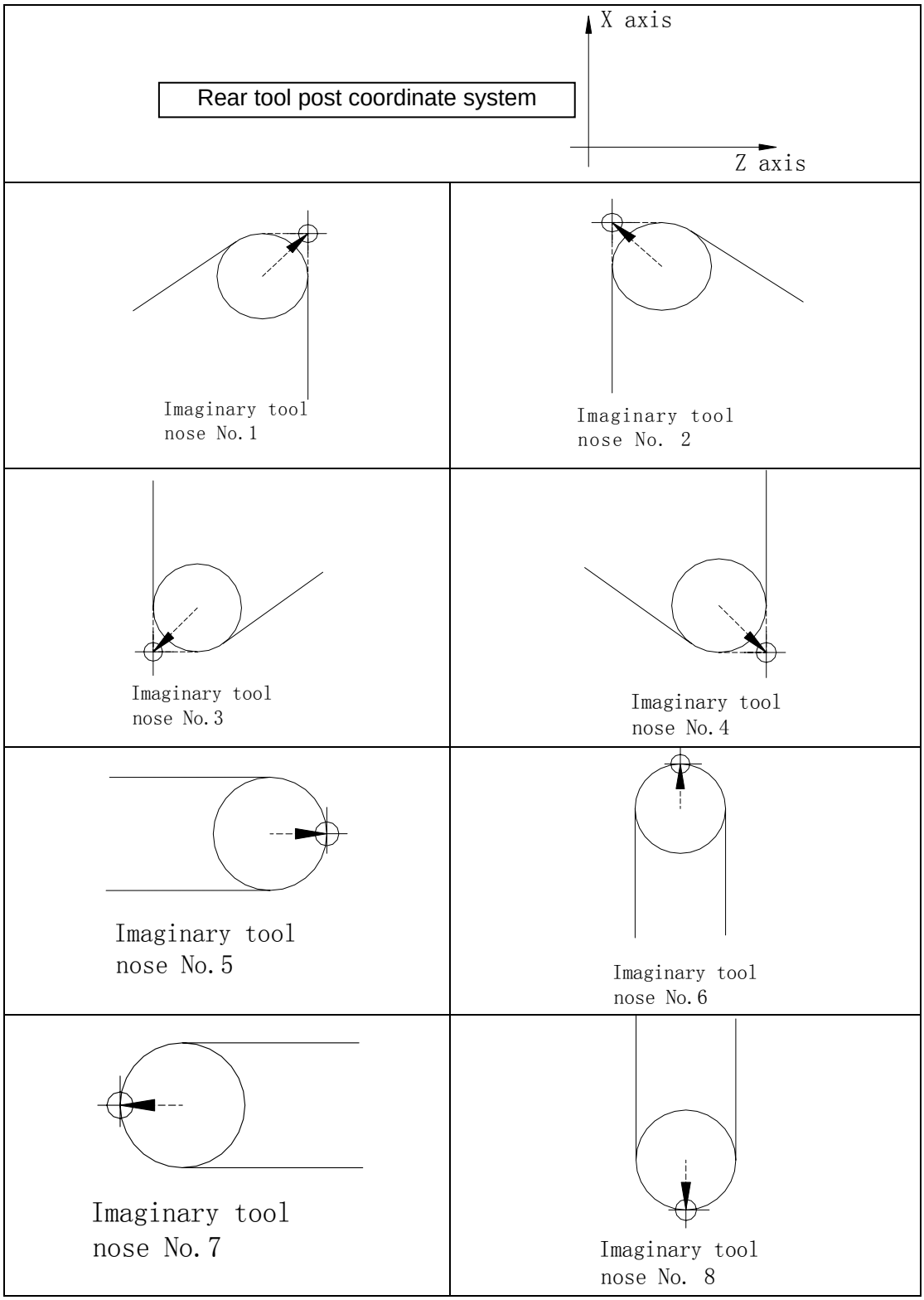


Fig. 4-7 Imaginary tool nose number in rear tool post coordinate system

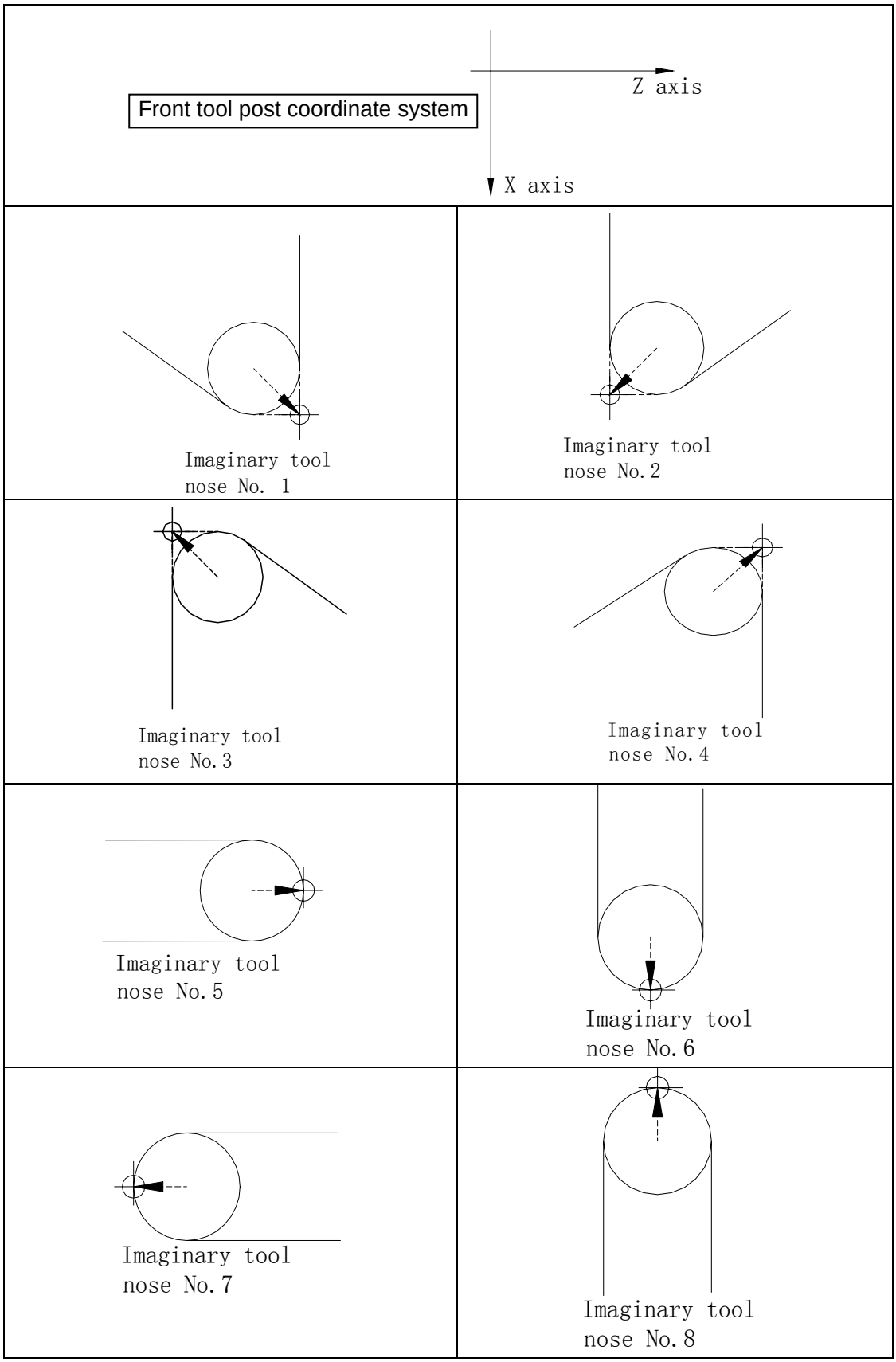


Fig. 4-8 Imaginary tool nose number in front tool post coordinate system

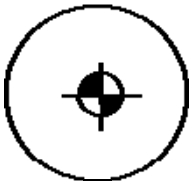


Fig. 4-9 Tool nose center on starting point

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.

Table 4-1 CNC tool nose radius compensation value display window

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

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 Tn (n=0~9) when taking Tn(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).

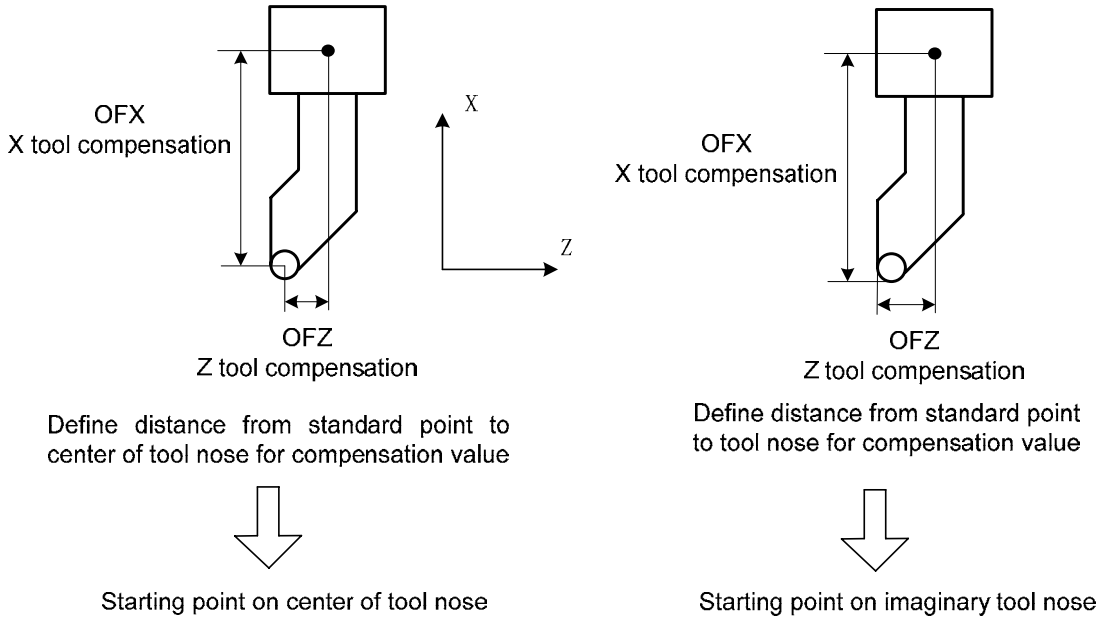


Fig. 4-10 Tool offset value of tool post center as benchmark

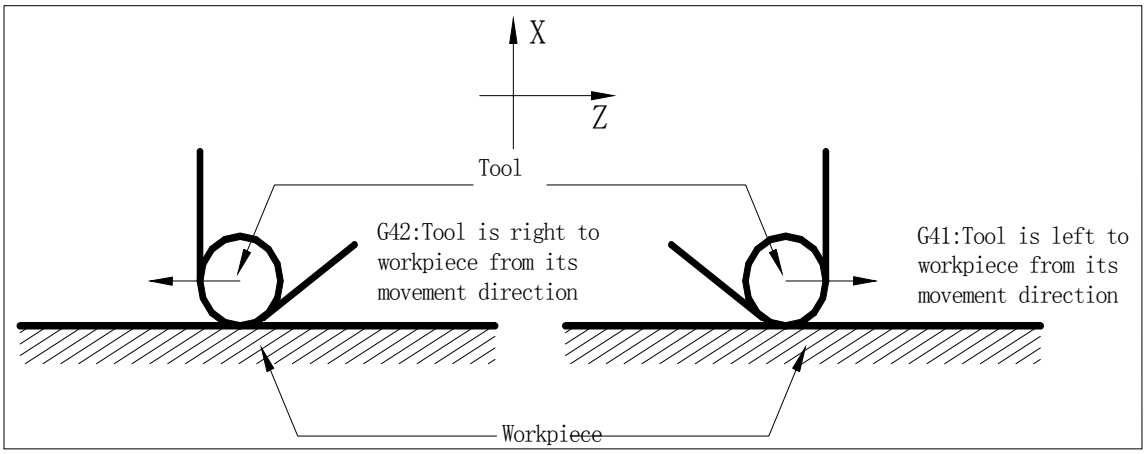
4.1.4 Command Format

$$\left\{\begin{matrix} G40 \\ G41 \\ G42 \end{matrix}\right\} \left\{\begin{matrix} G00 \\ G01 \end{matrix}\right\} \quad X__ \quad Z__ \quad T__ \ ;$$

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.



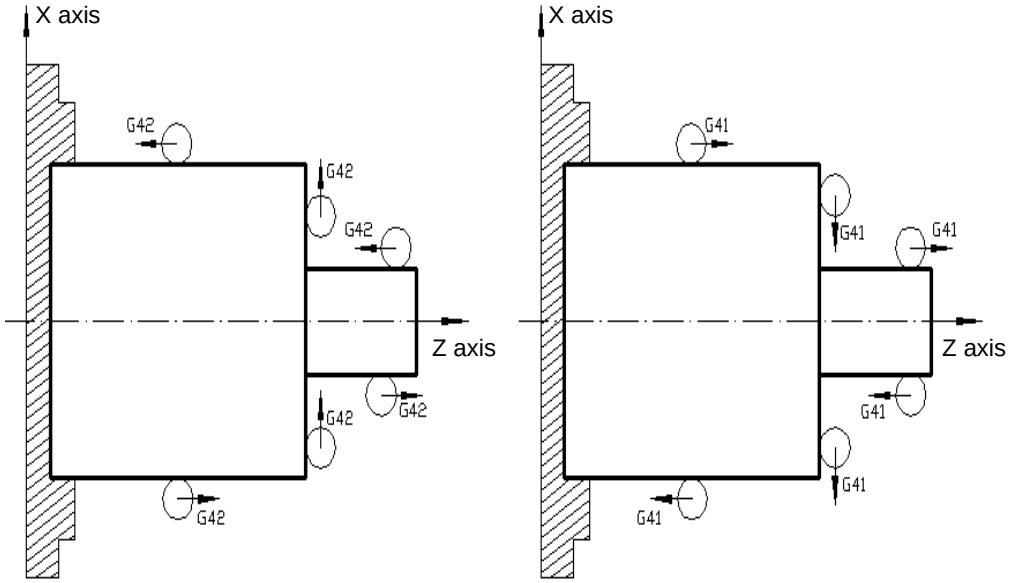


Fig. 4-11 Compensation direction of rear coordinate system

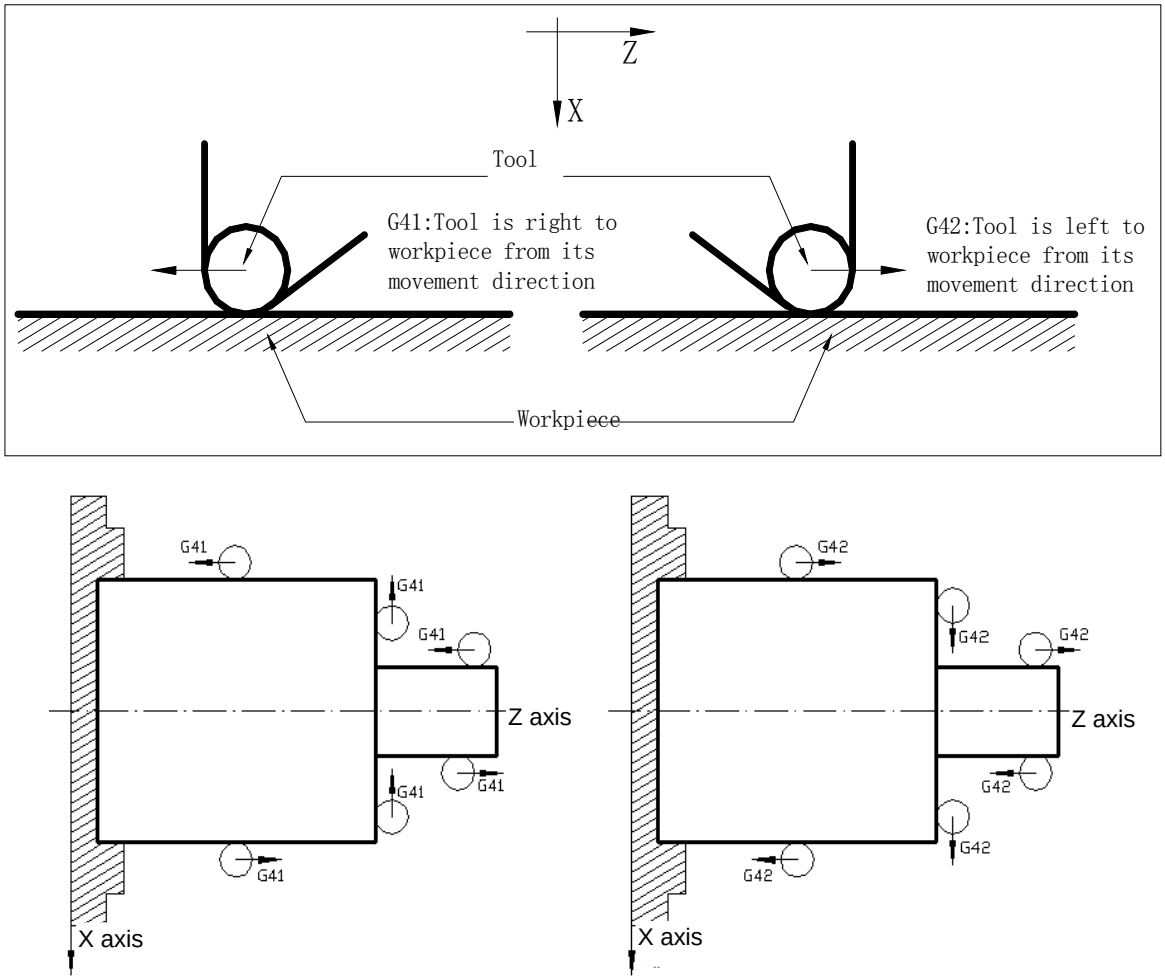


Fig. 4-12 Compensation direction of front coordinate system

4.1.6 Notes

- In tool nose radius compensation mode, the system will alarm when it executes 7 block or more than blocks without motion Command.
- 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 key or execute M30
- 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 (before execute M98) and then create it again in the subprogram.
- The system does not execute the tool nose radius compensation in G74, G75, G76 and cancel it temporarily.
- The system executes the tool nose radius compensation in G90, G94, it offsets one tool nose radius (based upon the imagine cutter nose No.0) for G41 or G42.

- The compensation mode will be automatically retreated by performing the G71, G72 or G73, if the tool nose compensation should be used in the G71, G72 or G73 code, it can be rebuilt at the 1st part of the finish machining block.

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.

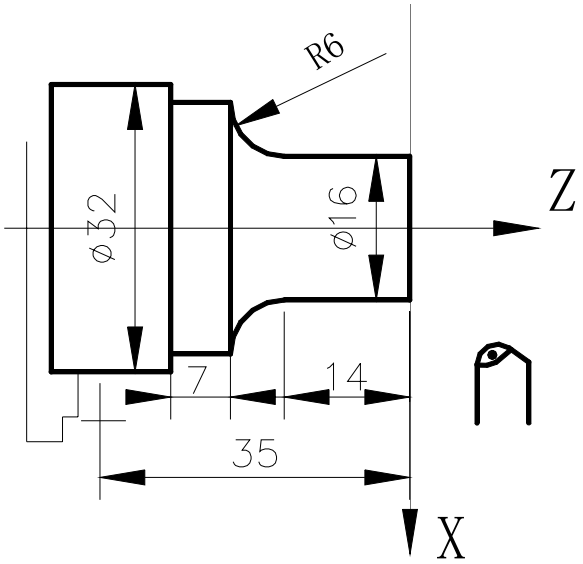


Fig. 4-13

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:

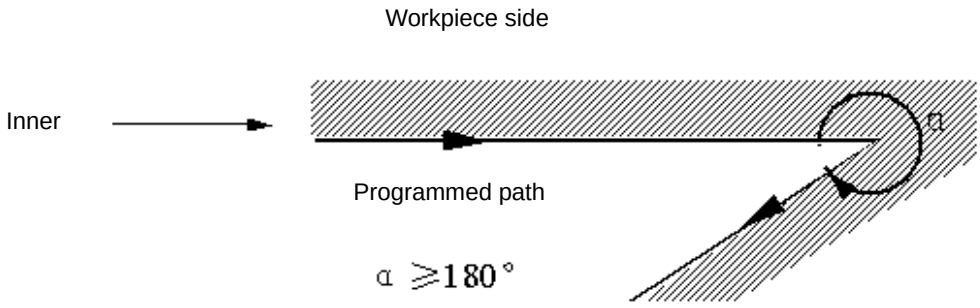
```
G00 X100 Z50 M3 T0101 S600;     (Position, start spindle, tool change and execute
                                 tool compensation)
G42 G00 X0 Z3;                  (Set tool nose radius compensation)
G01 Z0 F300;                    (Start cutting)
X16;
Z-14 F200;
G02 X28 W-6 R6;
G01 W-7;
```

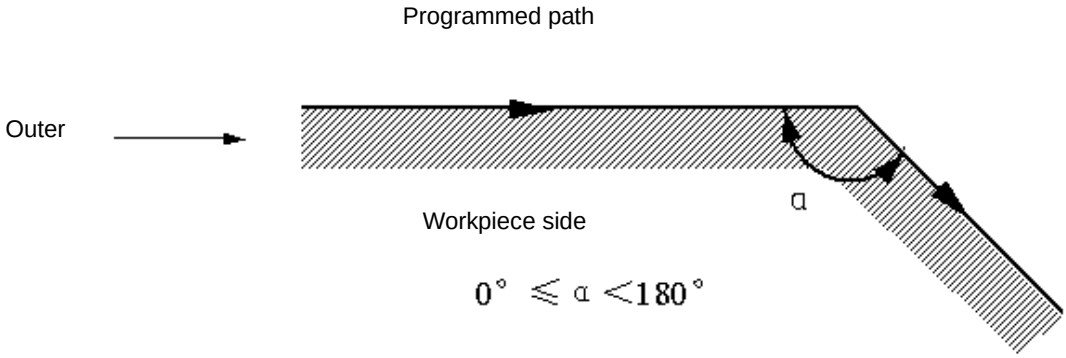
```
X32;
Z-35;
G40 G00 X90 Z40;                    (Cancel tool nose radius compensation)
G00 X100 Z50 T0100;
M30;
```

4.2 Tool Nose Radius Compensation Offset Path

4.2.1 Inner & Outer Side

Inside is defined that an angle at intersection of two motion blocks is more than or equal to 180°; **Outside** is 0~180°.





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.

Note1: Meanings of S, L, C in the following figures are as follows: S——Stop point of single block; L——linear; C——circular.

Note2: Tool movement path is marked with the tool nose direction 0 in the following figures.

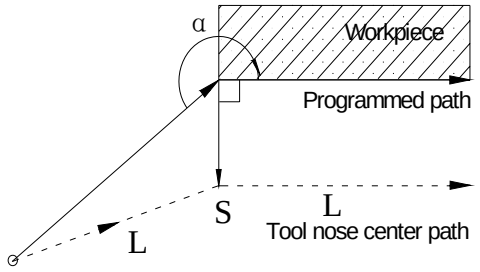
Note3: Tool path is described in rear tool post coordinate system and tool compensation direction G41 in the following figures.

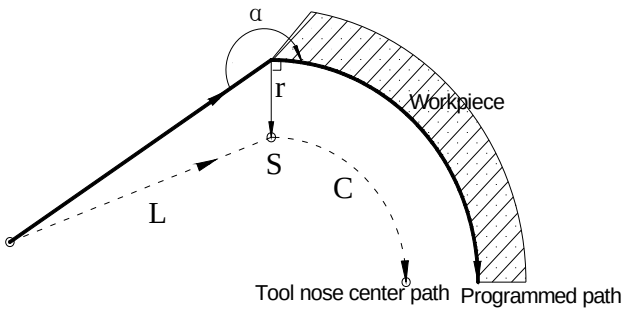
4.2.2 Tool Traversing when Starting Tool

(a) Tool traversing inside along corner(α≥180°)

1) Linear → linear

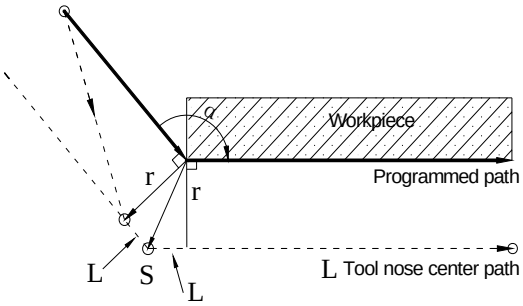
2) linear → circular



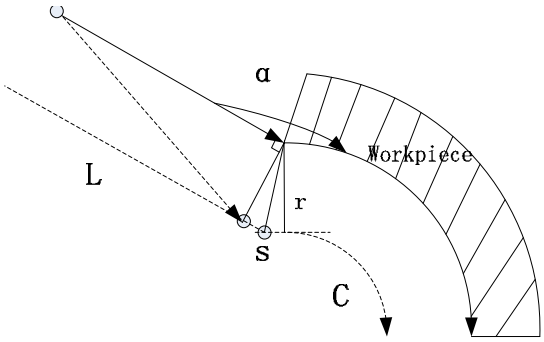


(b) Tool traversing inside along corner(180°＞α≥90°)

1) Linear→ linear

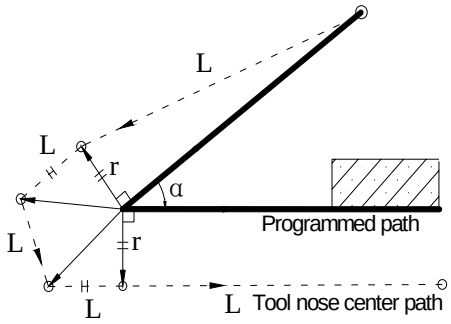


2) linear → circular

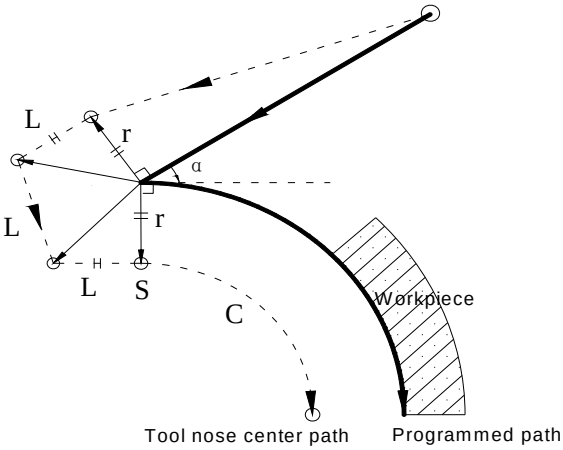


(c) Tool traversing inside along corner (α＜90°)

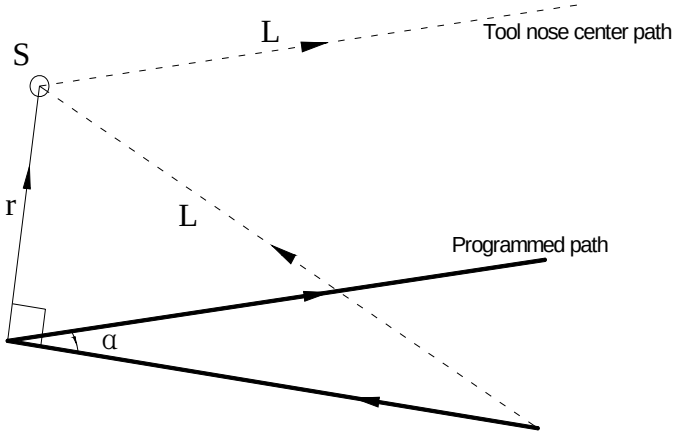
1）Linear→linear



2）Linear→circular



(d) Tool traversing outside along corner , linear →linear (α≤ angle set by No. 237)



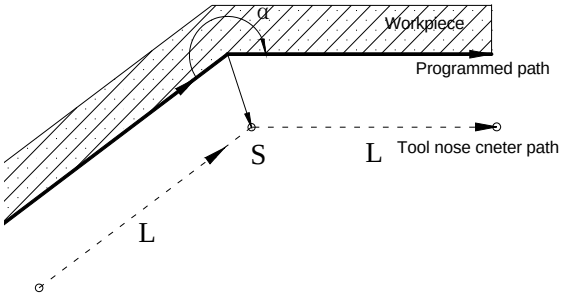
4.2.3 Tool Traversing in Offset Mode

Offset mode is called to ones after creating tool nose radius compensation and before canceling it.

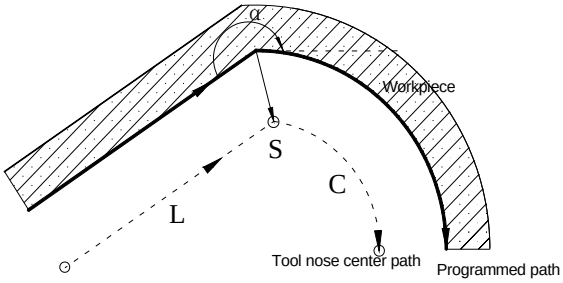
- **Offset path without changing compensation direction in compensation mode**

(a) Tool traversing inside along corner(α≥180°)

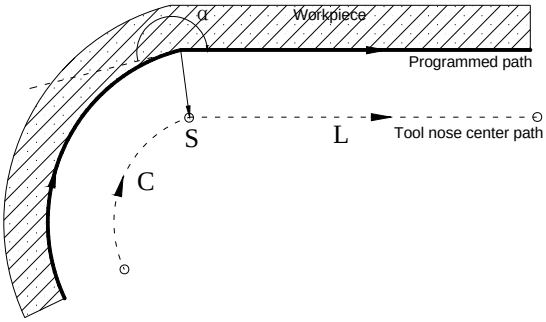
1）Linear→ linear



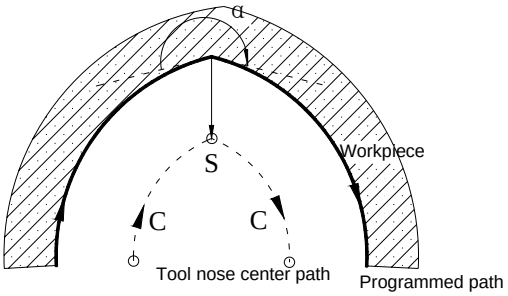
2) Linear→circular



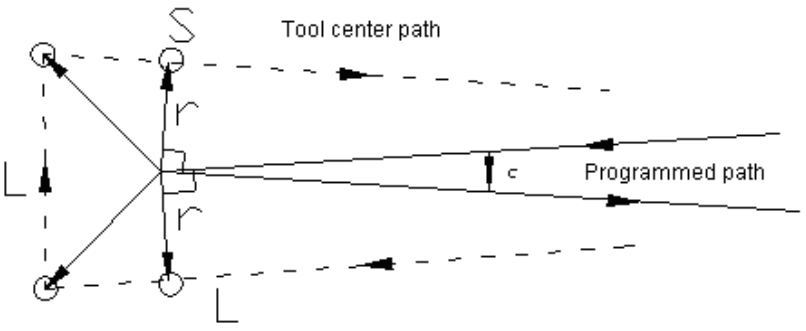
3) Circular→ linear



4) Circular→ circular

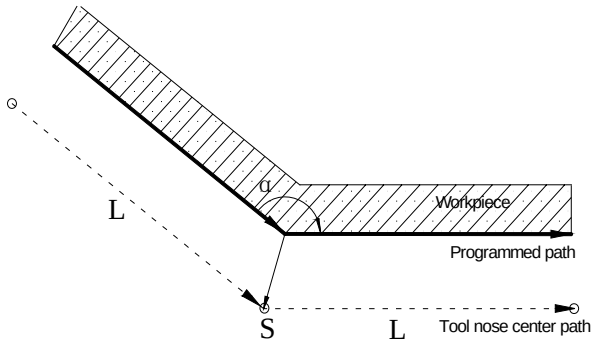


5）Inner side machining and scaling up compensation valid for acute angled being less than α（α: set by No. 237）.

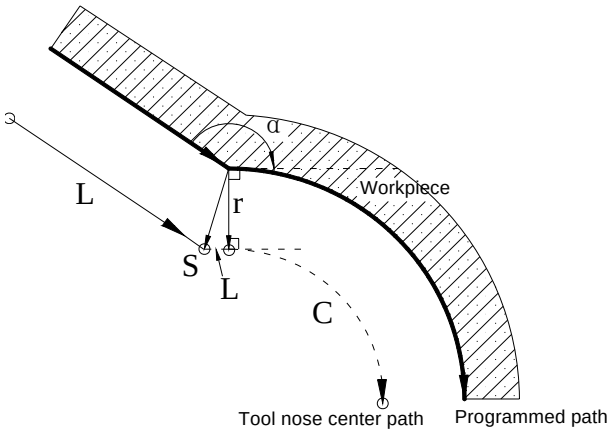


(b) Tool traversing outside along corner(180°＞α≥90°)

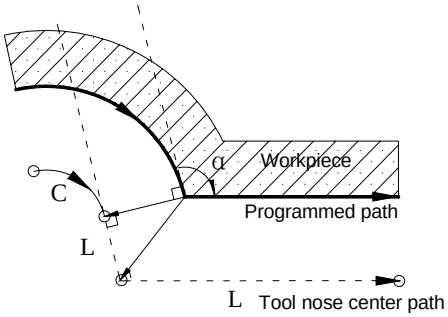
1）Linear→ linear



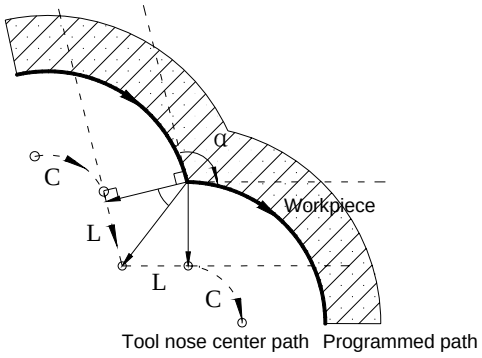
2) Linear→circular



3）Circular→Linear



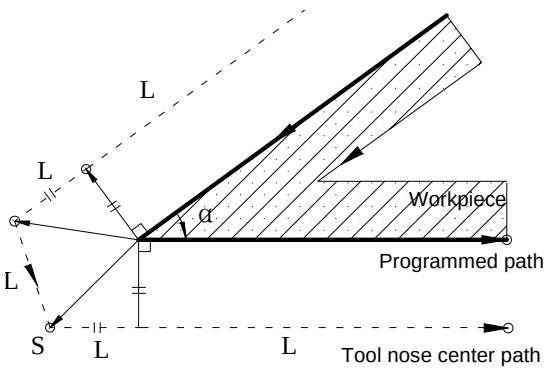
4）Circular→Circular

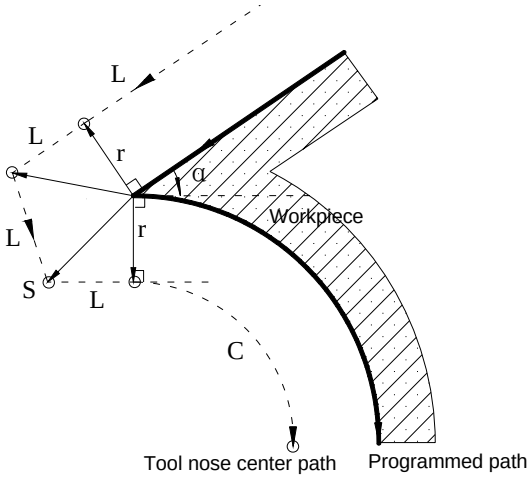


(c) Tool traversing outside along corner(α＜90°)

1）Linear→linear

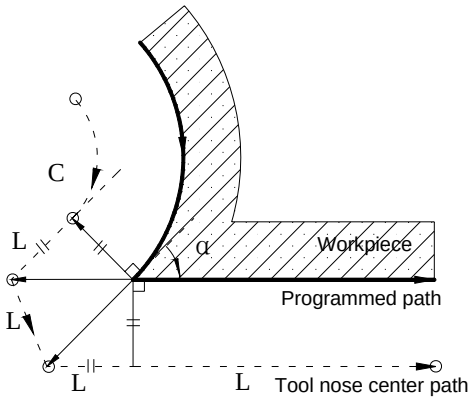
2）linear→circular

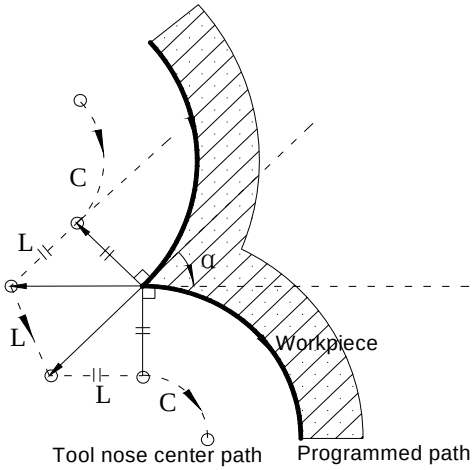




3）Circular→linear

4）Circular→circular



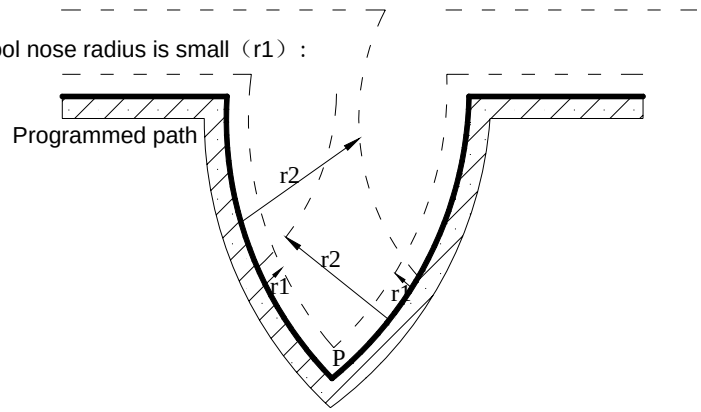


(d) Special cutting

1) Without intersection

Tool nose radius is big (r2):

Tool nose radius is small（r1）:



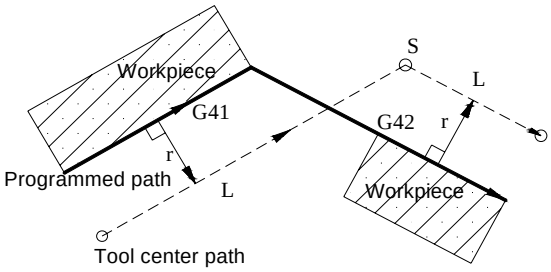
There is an intersection P between arc and arc compensation path when the tool radius is small; none when the radius is big, and the system alarms.

Fig. 4-14 Paths without intersection after offset

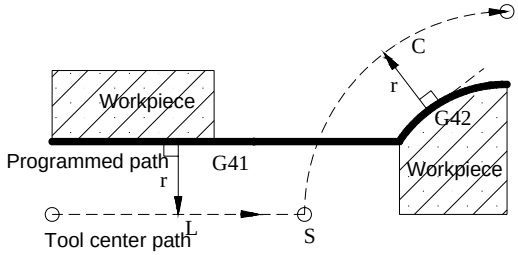
- **Offset path of changing compensation direction in compensation mode**

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:

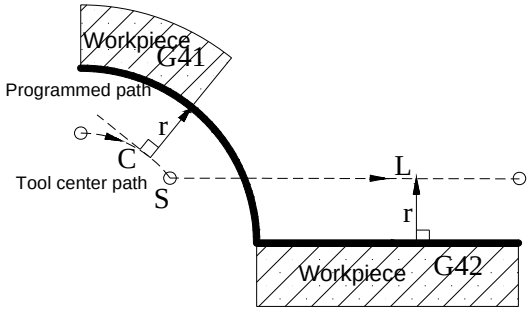
1）Linear→linear



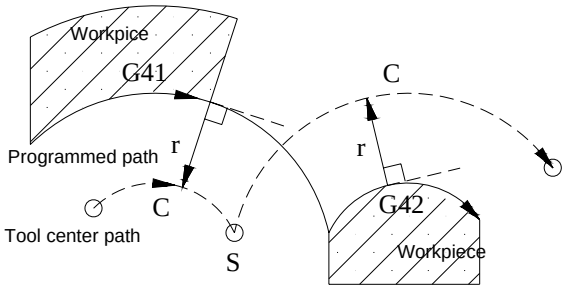
2）Linear→circular



3）Circular→linear



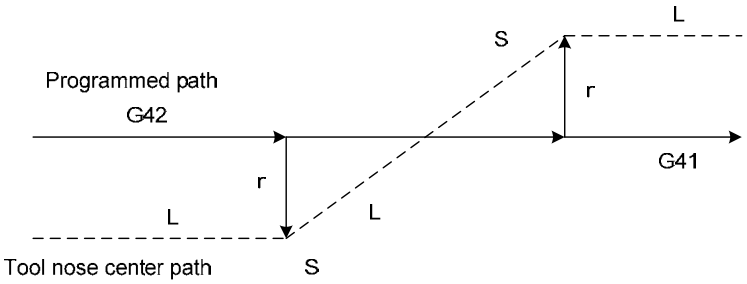
4）Circular→circular



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



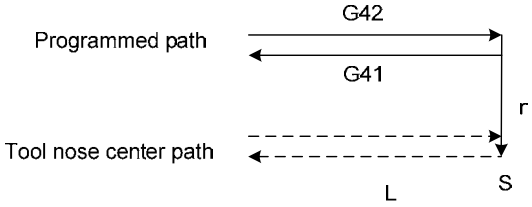


Fig. 4-15a Linear—linear, no intersection（changing compensation direction）

ii) Linear ---circular

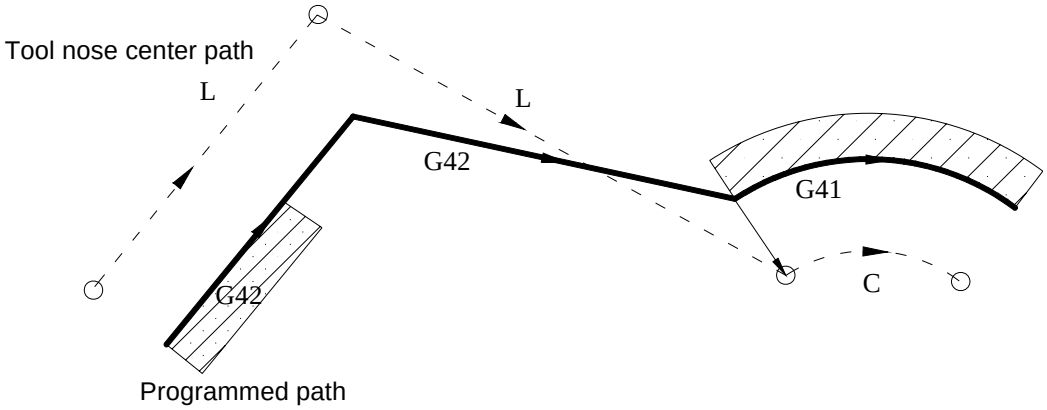


Fig. 4-15b Linear—circular without intersection （changing compensation direction）

iii) Circular-----circular

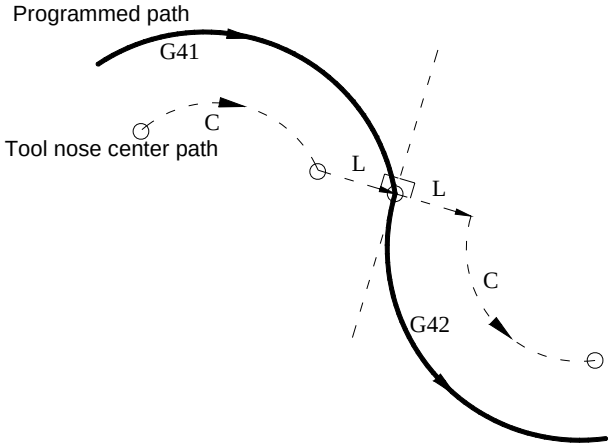


Fig. 4-15c Circular—circular without intersection (changing compensation direction)

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.

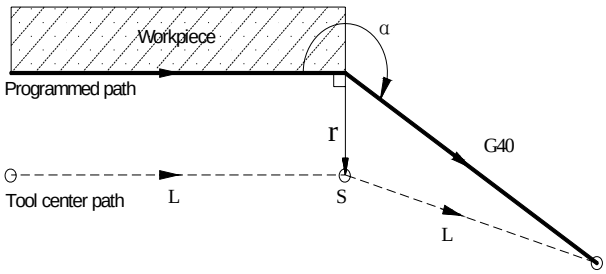
1. Execute G40 in a program;
2. Execute M30.

The system cannot execute G02 and G03 when canceling C tool compensation (tool nose radius compensation), otherwise the system alarms and stops run.

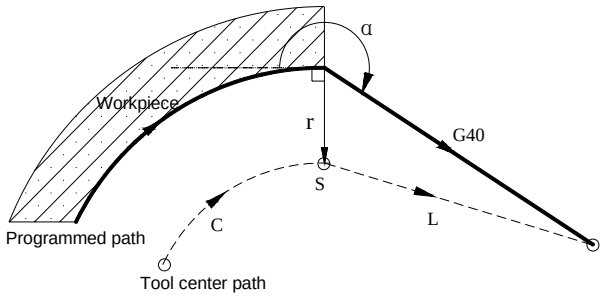
In compensation canceling mode, the system executes the block and ones in the register for tool nose radius compensation. At the moment, the run stops after one block is executed when single block is ON. The system executes the next one but does not read its following one when pressing CYCLE START **button again.**

(a) Tool traversing inside along corner(α≥180°)

1）Linear→linear

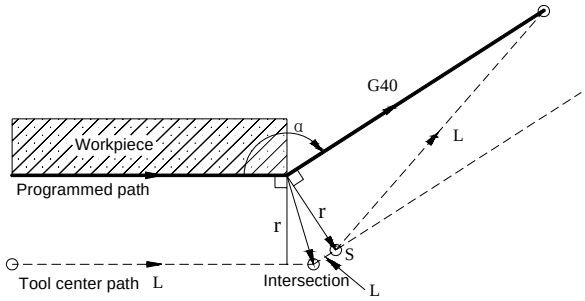


2）Circular→linear

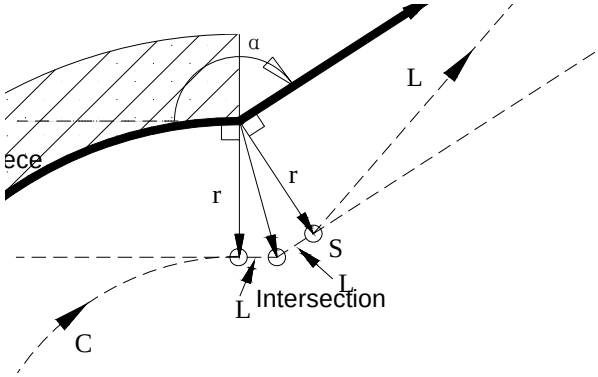


(b) Tool traversing outside along corner(180°＞α≥90°)

1）Linear→linear

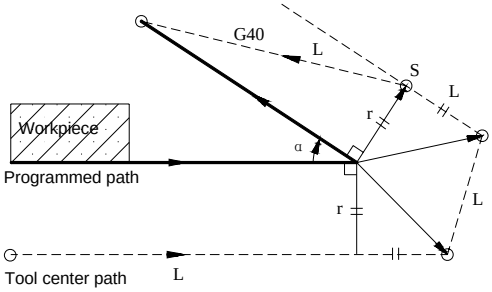


2）Circular→linear

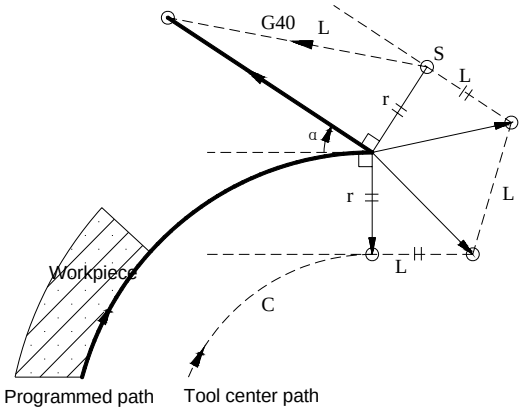


(c) Tool traversing outside along corner(α＜90°)

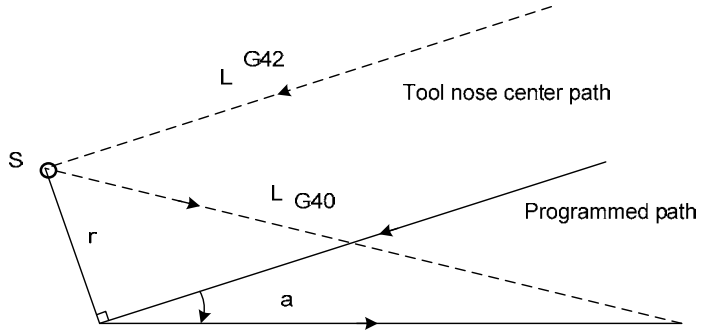
1）Linear→linear



2）Circular→linear



（d）Move along outer side of acute angle being less than α, linear→linear（α≦angle set by No. 237）



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°~270°).

2) There is a big difference (α＞180°) 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

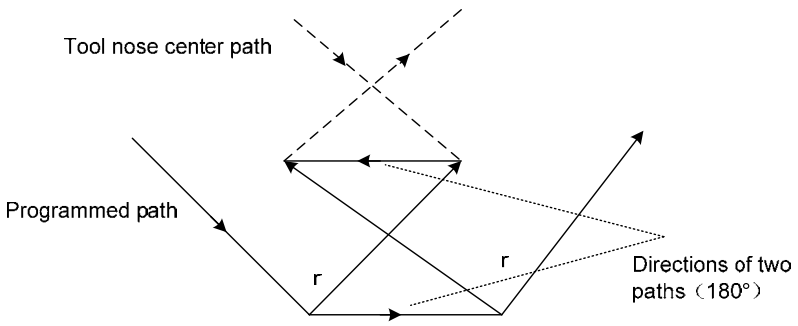


Fig. 4-16a Machining interference (1)

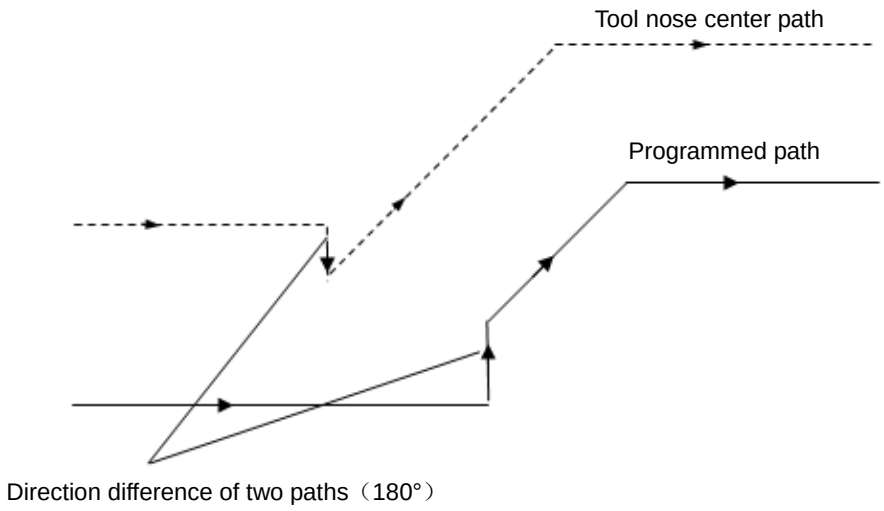


Fig. 4-16b Machining interference (2)

(2) Executing it without actual interference

1) Concave groove less than compensation value

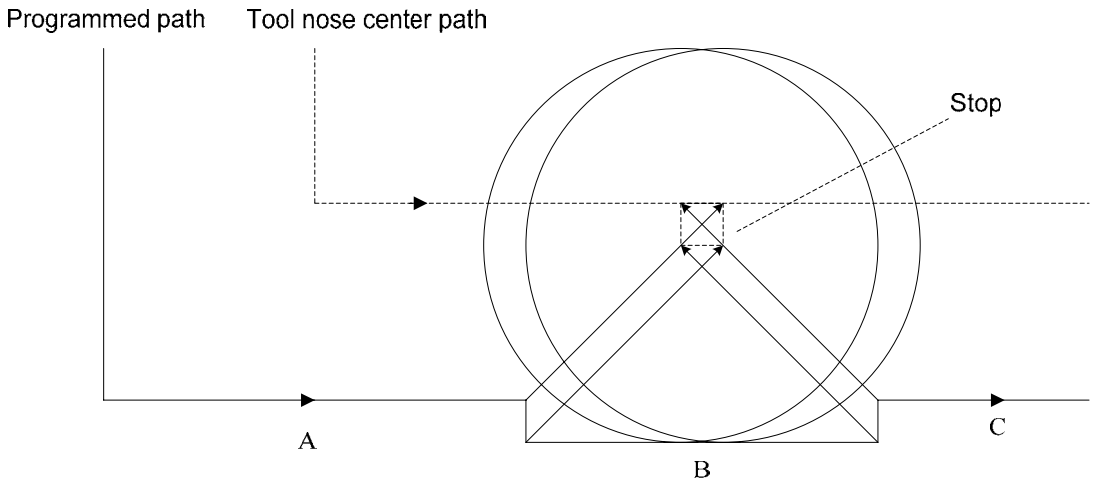


Fig. 4-17 Executing interference (1)

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

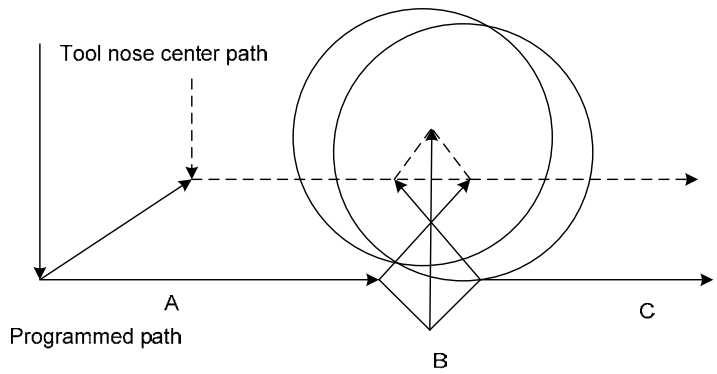


Fig. 4-18 Executing interference (2)

Directions of block B and tool nose radius compensation path are opposite without interference, the tools 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**

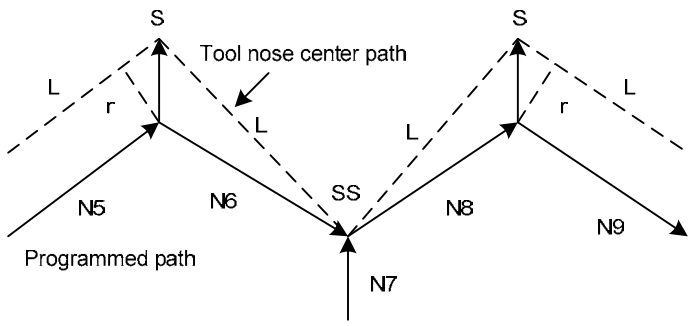


Fig. 4-19 Temporary compensation vector in G50

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.

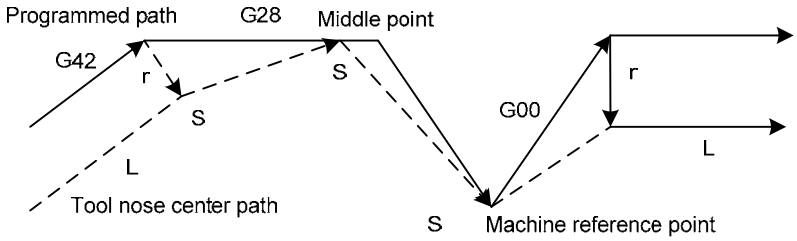


Fig. 4-20 Cancel compensation vector temporarily in G28

- **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".

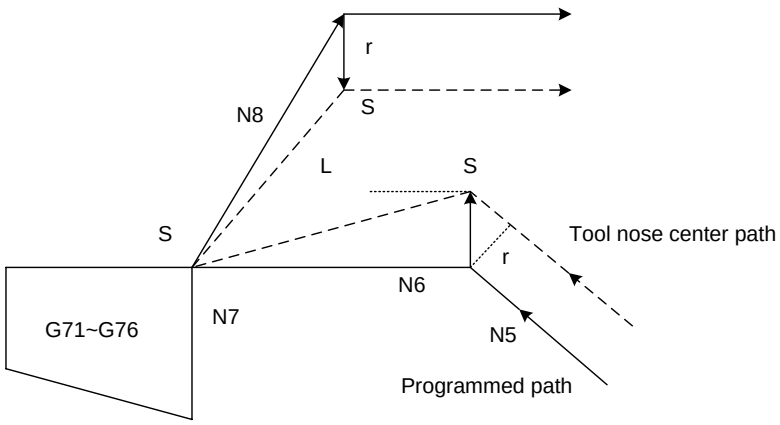


Fig. 4-21 Cancel compensation vector temporarily in G71～G76

- **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:

  A. Cancel the previous tool nose radius compensation;

  B. Create the previous C compensation before cutting, and the path ① in the following figure creates the previous radius compensation mode;

  C. The paths 2, 3 in the following figure are the radius compensation cutting;

  D. 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.

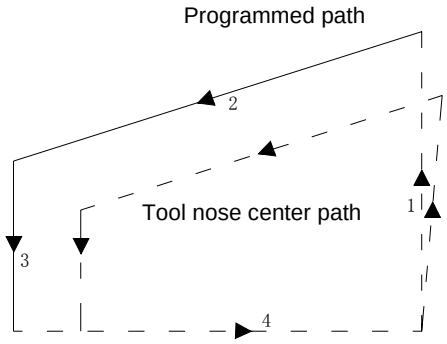


Fig. 4-22 Offset direction of G90 tool nose radius compensation

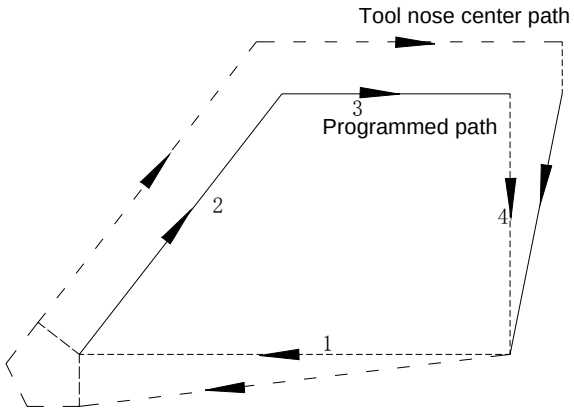


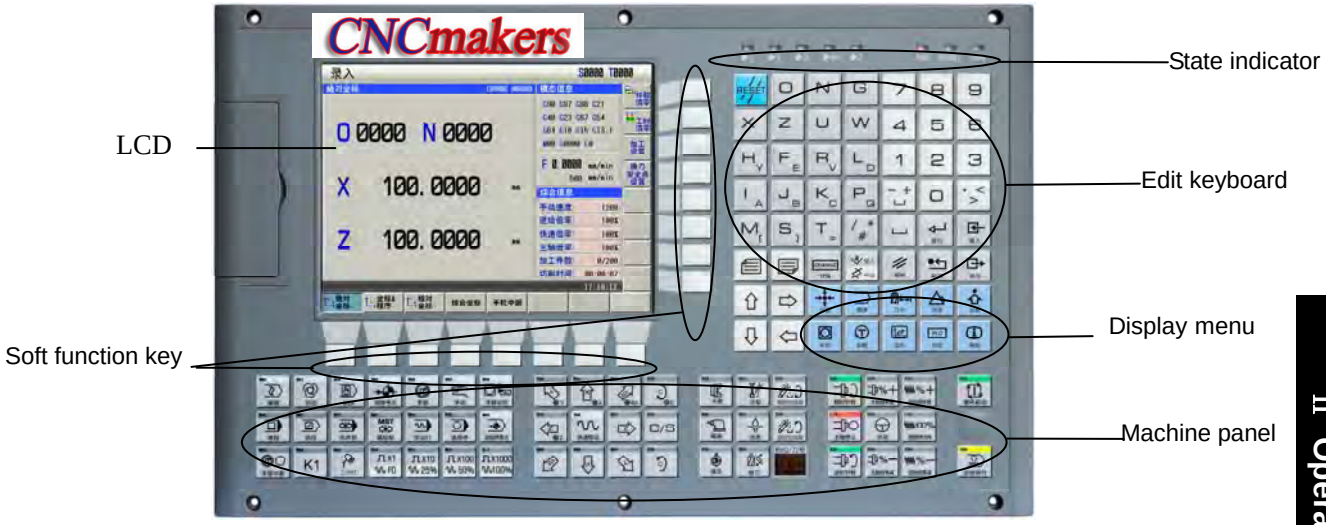
Fig. 4-23 Offset direction of G94 tool nose radius compensation

II Operation

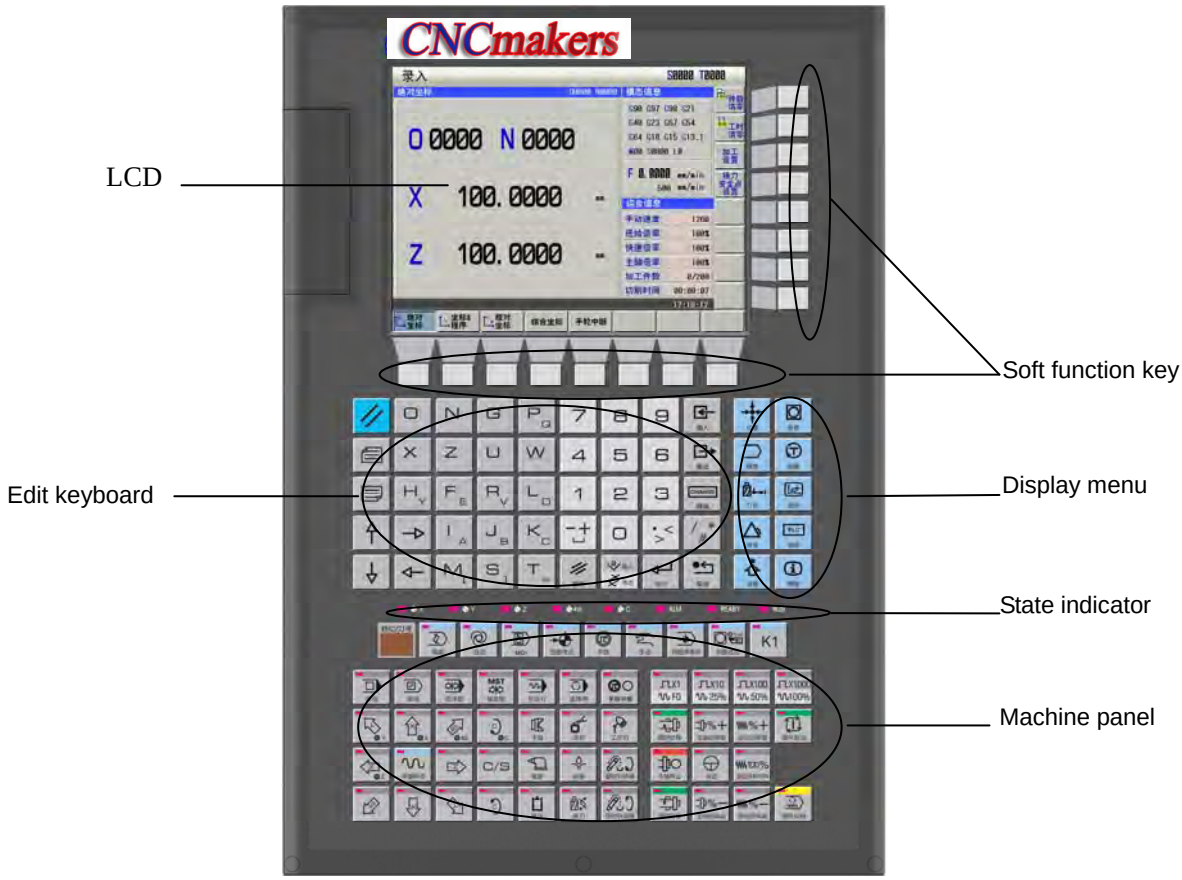
CHAPTER 1 OPERATION MODE AND DISPLAY INTERFACE

1.1 Panel Division

GSK980TDi, GSK980TDi-V CNC system uses an integrated panel, which is divided as follows:

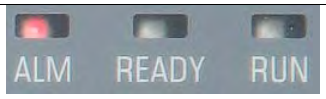


GSK980TDi panel division



GSK980TDi-V panel division

1.1.1 State Indication

	Indicator for axis zero return completion
	Three-color light

1.1.2 Edit Keypad

Press key	Name	Function
	RESET key	CNC reset, feed, output stop etc.
O N G X Z U W M S T	Address key	Address input
H F R L I J K P		Double address key, switching them by pressing it repetitively
- + / * ␣ #	symbol	Key to three addresses,, switching them by pressing it repetitively
7 8 9 4 5 6 1 2 3 0	Number key	Number input
. < >	Decimal point	Decimal point input
	Input key	Parameter, compensation value and other data input
	Output key	Communication output
CHANGE	Change key	Switching message, display
	Edit key	Inserting, altering, deleting programs, fields in EDIT working mode(compound key, switching them by pressing it repetitively)
	EOB key	Inputting the end character of block
	Cursor move keys	controlling cursor move

Press key	Name	Function
	Window key	Switch the page in the same display page

1.1.3 Menu Display

Menu key	Remark
	To enter Position interface including RELATIVE POS, ABSOLUTE POS, POS & PRG, INTEGRATED POS, MPG interrupted page. ● "ABSOLUTE POS" page can display the current absolute coordinates, clear out machining count, machining time, as well, it can be set the Max. workpiece number, current number, and the tool-change safety position; ● "POS & PRG" page can display current coordinate information and current run program; ● "RELATIVE POS" page can display current relative coordinates, clear relative coordinates; ● "INTEGRATED POS" can display current coordinate information; the reference position along each axis can be set under the 2nd level authority; ● The input/output equivelant amount along each axis can be displayed based upon the MPG interruption on its MPG interruption page, which can be eliminated the interruption amount along each axis.
	To enter Program interface including program content, program list, Marco variable note, program state page ● Edit/search current open programs in program content page; ● Edit MDI programs, display coordinates, override and modal state in MDI page; ● Display, preview, open, copy machining programs in CNC in local catalog page; ● The noted macro variable note name can be viewed on the macro variable note page; ● Display, preview, open and copy machining programs in U disk in U disk catalog page
	Enter tool offset interface including offset setting, macro variables, workpiece coordinate system and tool life page ● Set/search tool offset value and wear value in tool offset setting page; ● Set/search CNC macro variables in macro variable page; ● Set/search G54～G59 coordinate system and zero offset value in workpiece coordinate system page; ● Set/search use information of current tool life
	Enter alarm interface including alarm information, alarm log page ● Search CNC alarm, CNC warning, PLC alarm, PLC warning, servo alrm, in alarm information page; the caused reason and troubleshootings can be viewed by selecting some one alarm or precaution; ● Display alarm/warning history record in alarm log page

	Enter Setting interface including CNC setting, system clock, file management, machine tool debugging, GSKLink, setting IP page ● Set switch, level and parameters in CNC setting page; ● Set system clock, data in system clock page; ● Search/management files in CNC/U disk, copy/resume files in file management page ● Rapid grouping can be performed in the machine tool debugging page, so that the each functions for the machine tool can be easily debugged; ● Each controllable axis of the system corresponding the drive and motor can be set in GSKLink page; ● Set the IP page for using the relevant parameter setting of the RJ45 internet access communication
	Enter Parameter interface including state parameters, data parameter, often-used parameters, servo parameters, and pitch compensation page. ● Search/set state parameters in the state parameter page; ● Search/set data parameters in the data parameter page; ● Search/set user's customized parameters in the often used parameter page; ● Search/set pitch error compensation data of each axis in the pitch compensation page. ● View/set the drive parameter along each axis on the servo parameter page.
	● Enter Diagnosis interface including the system diagnosis, system information, Machine tool diagnosis、Servo diagnosis page ● Search the current diagnosis information of the CNC in the diagnosis page. ● Search the product information, ladder information and ladder state. ● Machine diagnosis can be directly inspected the working state of machine tool's each component; ● Servo diagnosis can be viewed servo operation state of each axis.
	PLC page includes three subpages: PLC state, PLC monitor, PLC data, Program list. ● PLC state page can search states of X, Y, F, G, R, A, C,T; ● PLC monitor page can monitors the execution state of current ladder on-line; ● PLC data page can search/set K, D, DT, DC value. ● The registered PLC program can be selected in the program list page, and then start the operation.
	Enter Graph page to display the movement path of X, Z axis
	Entering to the system help page can be captured the helf information, such as the programming, operation and installation.

1.1.4 Machine Panel

The key functions on GSK980TDi machine panel are defined by PLC program (ladder), the detailed function meanings are referred to machine manufacturer manual.

The functions of this GSK980TDi machine panel keys 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
X1 F0 X10 25% X100 50% X1000 100%	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
	Manual tool change key	manual tool change	Machine zero return, Manual, Step, MPG, Program zero return mode
	JOG key	spindle jog on/off	Machine zero return, Manual, Step, MPG, Program zero return mode
C/S	C/S Switch	switch spindle speed/ position control	
	Lubricating key	For lubricating ON/OFF	
	Cooling key	For cooling ON/OFF	Auto, Edit, MDI, Machine zero return, Manual, Step, MPG, Program zero return mode
	Chuck key	Chuck clamping/releasing	Auto, Edit, MDI, Machine zero return, Manual, Step, MPG, Program zero return mode
	Hydraulic key	Hydraulic output ON/OFF	Auto, Edit, MDI, Machine zero return, Manual, Step, MPG, Program zero return mode
	Spindle control keys	For spindle CCW For spindle stop For spindle CW	Machine zero return, Manual, Step, MPG, Program zero return mode
	Rapid traverse key	For rapid traverse /feedrate switching	Auto, MDI, Manual mode

Key	Name	Function explanation	Operation mode
	X feed key	Positive/negative movement of each axis in Manual, Step mode	Machine zero return, Step, Manual, Program zero return mode
	Z feed key		
	Y feed key		
	The 4th feed key		
	Cs feed key		
	MPG axis selection key	Each axis selection in MPG mode	MPG mode
	MPG/Step increment and Rapid override selection key	Move amount per MPG scale 0.001/0.01/0.1 mm Move amount per step 0.001/0.01/0.1 mm Rapid override F0, F50%, F100%	Auto, MDI, Machine zero return, Manual, Step, MPG, Program zero return mode
	Optional stop	Execute M01 to pause when the optional stop is enabled	Auto, MDI, and MPG trial-cut 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

Key	Name	Function explanation	Operation 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 GSK980TDi, 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.

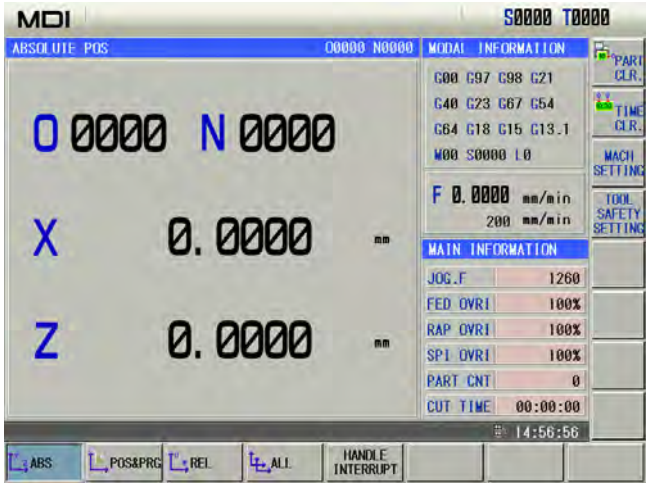
1.3 Display Interface

The chapter introduces page switch, relationship between operation input and soft key, and concrete operation methods.

GSK980TDi has 9 function keys including POS, PRG, SET, etc. on its edit keyboard. Each function key corresponds to one interface which has many pages and operation soft keys, which are shown below:

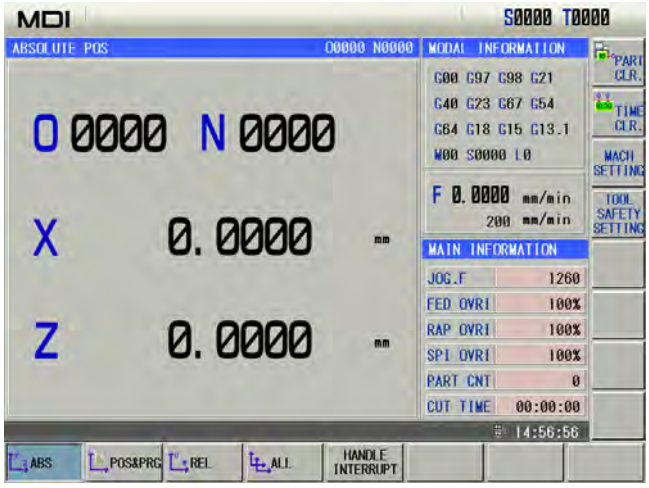
1.3.1 POS Interface

Press POSITION to enter POS interface which includes pages ABS, REL, ALL, HANDLE INTERRUPT and POS&PRG, and these pages can be viewed by corresponding soft function key or repetitively press POSITION.



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.

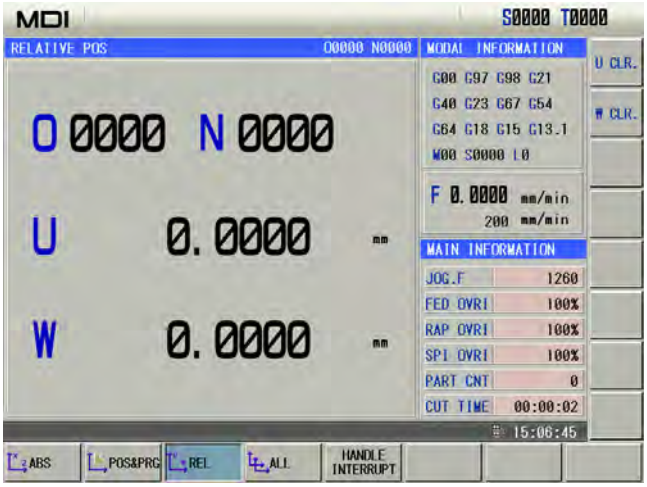


Basic information in POSITION interface (taking an example of the above figure):

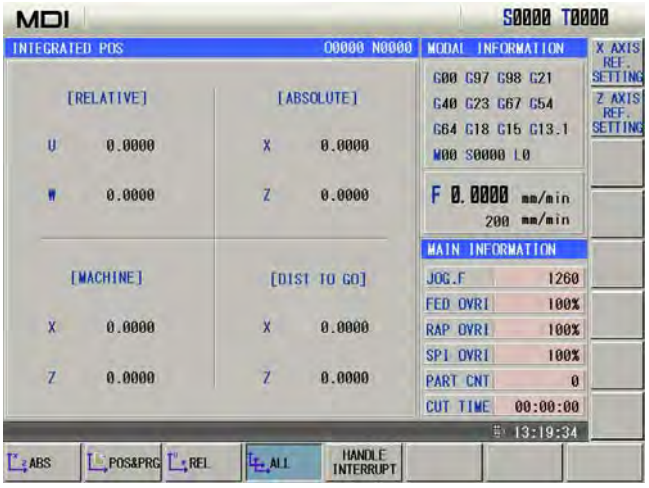
- ✓ G code : information of each G code;
- ✓ M : last executed M code;
- ✓ S :it is input spindle speed when analog spindle is executed, and input spindle's gear when the spindle is executed;
- ✓ L : call times of subprogram;
- ✓ F :the first half (0.0000mm/min) is the actual speed, and the second half (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 program executes the M code set by data parameter No.173;
- ✓ CUT TIME: the system counts the time when the automatic run is started, time unit is hour, minute and second in order.

2) **RELATIVE POS display interface**

Displayed U, W coordinates are the current position relative to the 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.

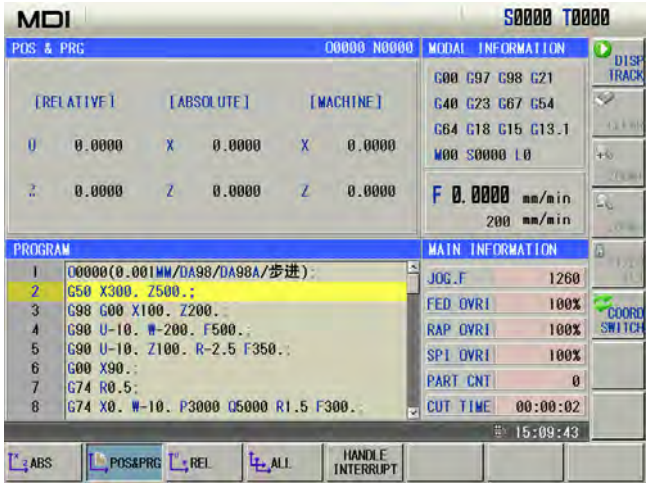


3) INTEGRATED POS display interface



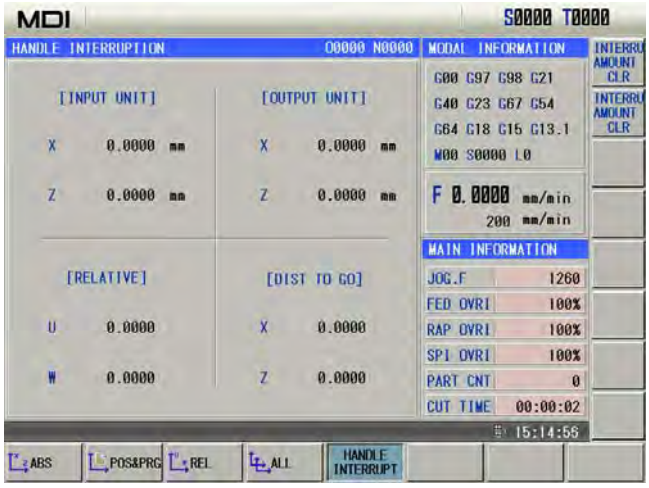
4) POS&PRG display interface

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.



5) MPG interrupted page

When the Bit 1 of state parameter No.0012 sets to "1", the MPG interruption sub-page is then displayed on the position page; refer to the following figure:



1.3.2 PRG Interface

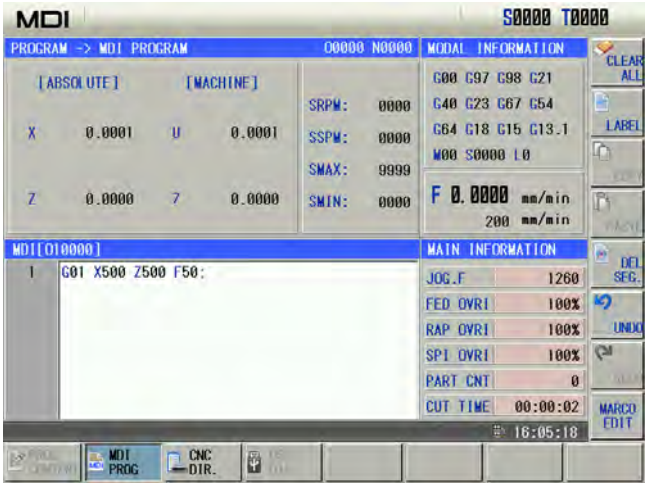
Press PROGRAM to enter PRG interface, which includes PROG CONTENT MDI PROG CNC DIR. (the soft function menu can be displayed when USB device is connected to the system). Press PROGRAM repetitively or press the corresponding soft function keys to switch interfaces. The following descriptions are aimed to program page set of the connected USB equipment of the GSK980TDi system.

1) PRG CONTENT page

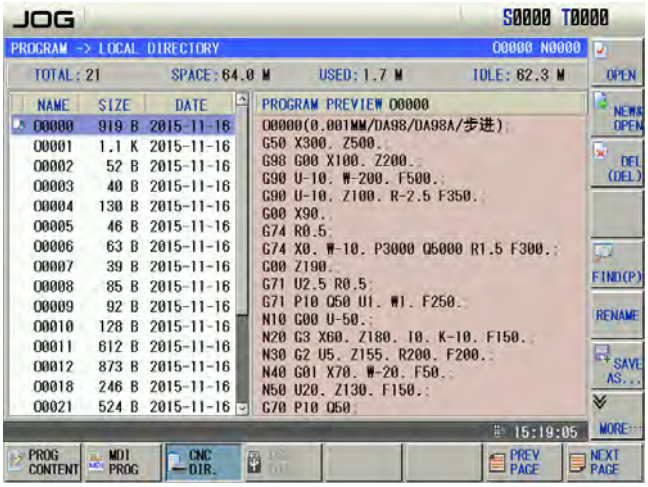
In the page, the program content including current block can be displayed. In Edit mode, the program content can be viewed forward or backward by pressing or key.

2) MDI program page

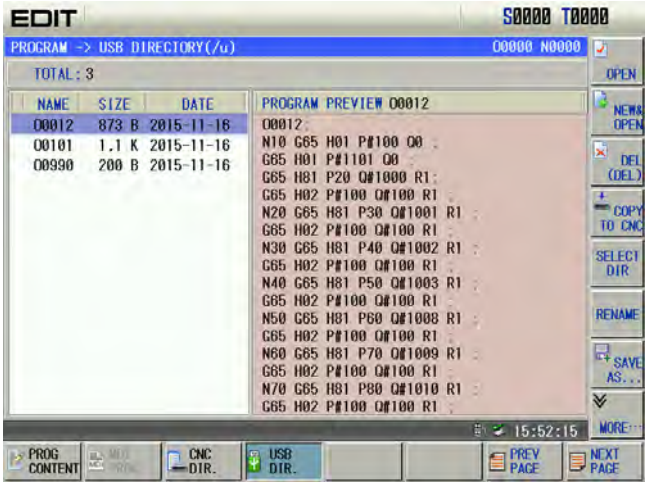
In MDI mode, the page can be displayed, and is value when the corresponding soft function key is pressed as follows:



3) Local directory page



4) U disk directory page

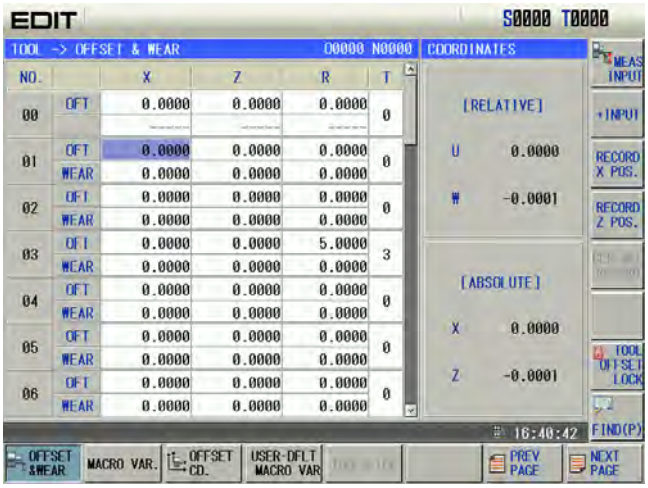


1.3.3 TOOL OFFSET Interface

Press OFFSET to enter the TOOL OFFSET interface which includes OFFSET &WEAR, MACRO VAR., OFFSET CD., TOOL-LIFE, USER-DFLT MACRO VAR (it enters the interface when No. 002 Bit0 is set to 1). Press OFFSET repetitively or corresponding keys to switch pages.

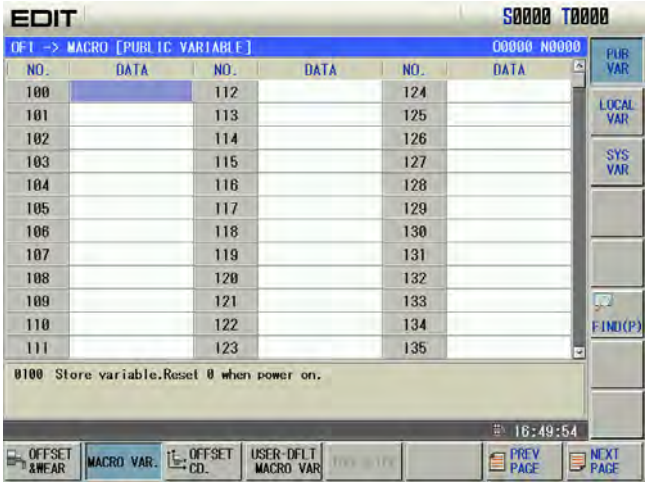
1） TOOL OFFSET&WEAR interface

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.



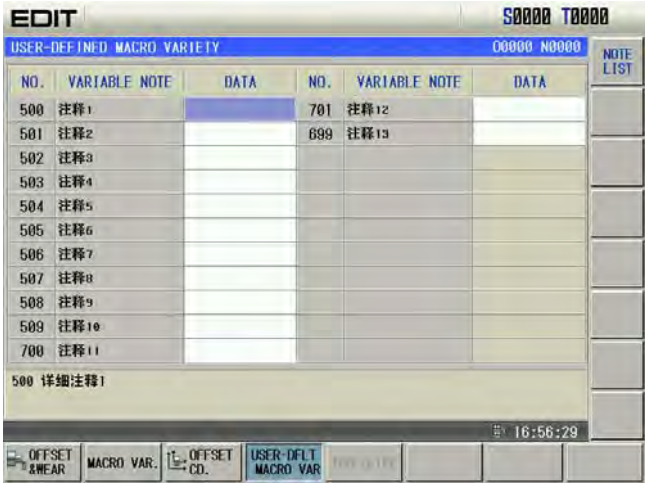
2） MACRO interface

The macro variable divides into 5 sub-pages: Common variable, Local variable, System variable, Note list and Customized macro variable. There are 19 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:



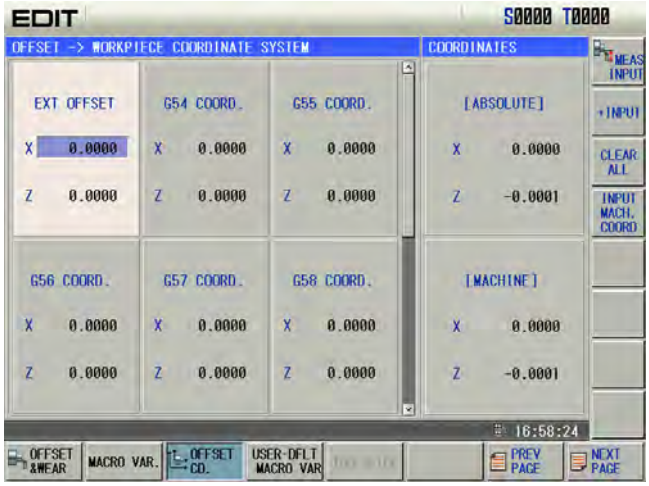
The local variable page is shown below:

The system variable page is shown below:



Note: The local variable and the system local are specified by macro codes instead of directly set by the keyboard in the page.

3） Workpiec coordinate system setting



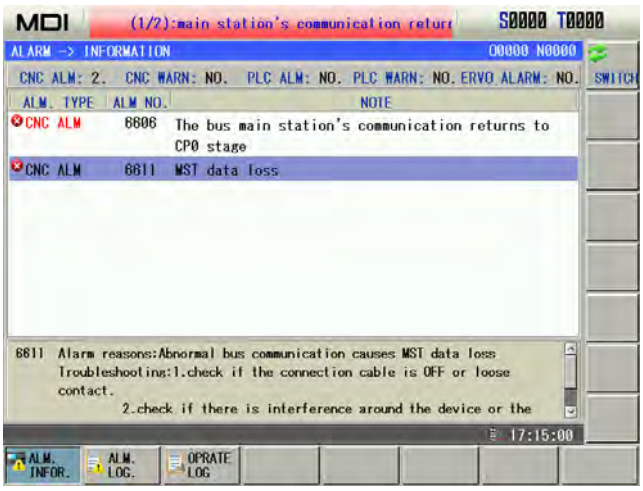
4）Tool life management

1.3.4 ALARM Interface

1) Alarm:

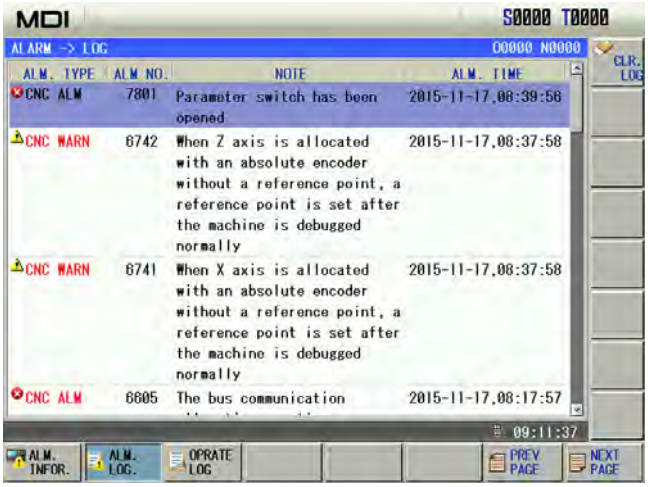
Press key to enter alarm interface including 、 which can be switched by pressing repetitively , or the corresponding soft function.

1） Alarm information page



Note: Alarm clearing: It can clear alarms by pressing .

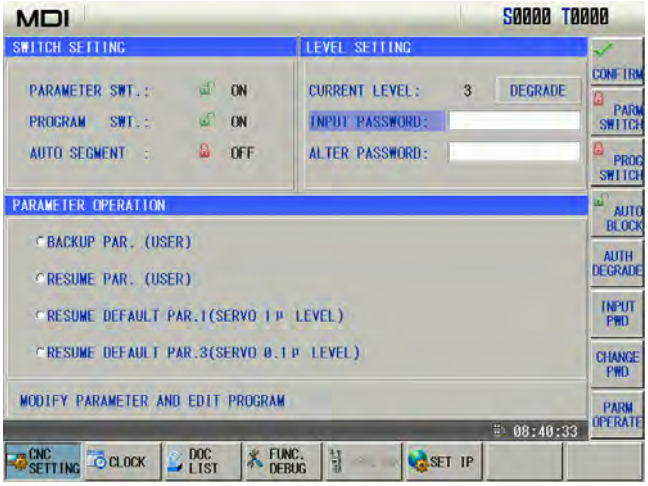
2） Alarm log page



1.3.5 Setting Interface

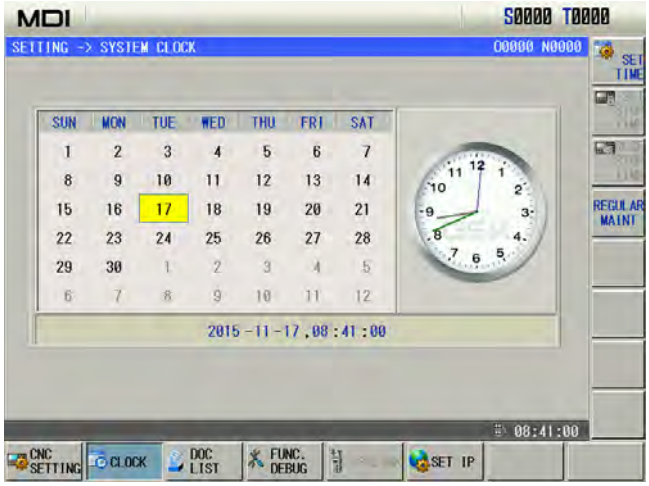
Press to enter the SETTING interface including 、 、 、 、 、 which can be switched by pressing repetitively or corresponding soft keys.

1) CNC setting page

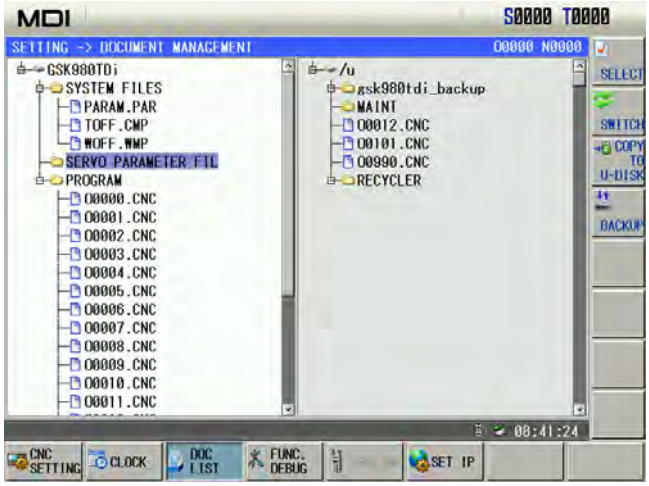


2) System clock page

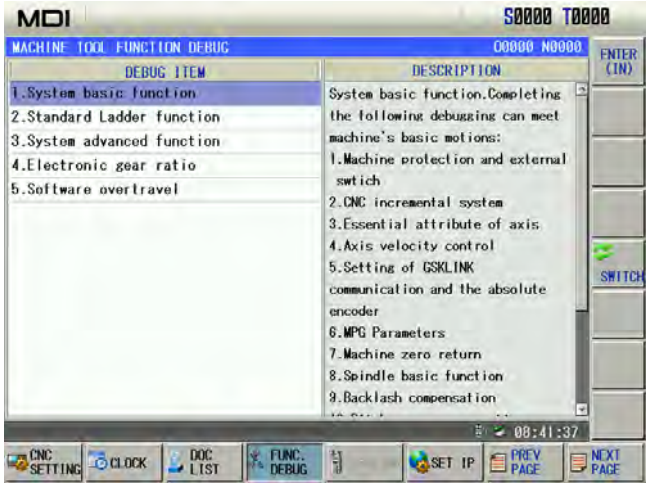
II Operation



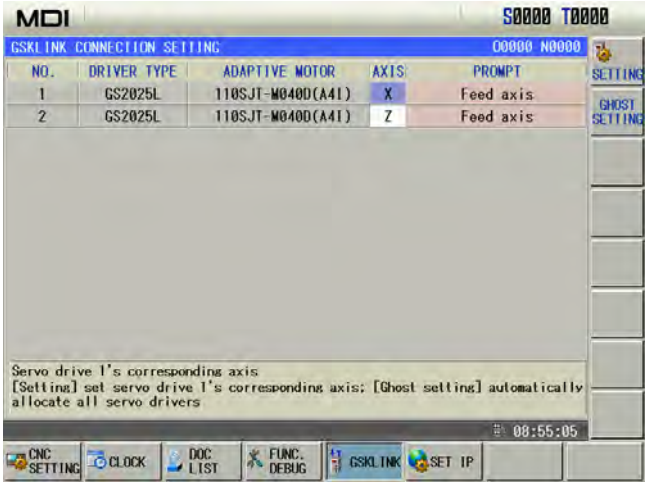
3) **Document management page**



4) **Machine function debugging page**



5) **GSKLink setting page**

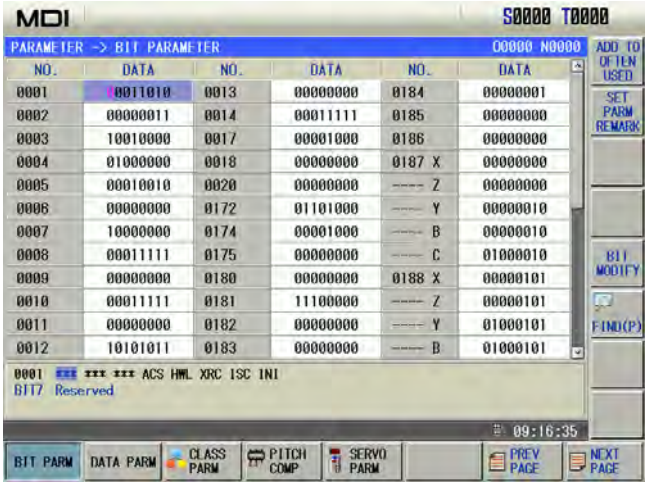


6) IP page setting

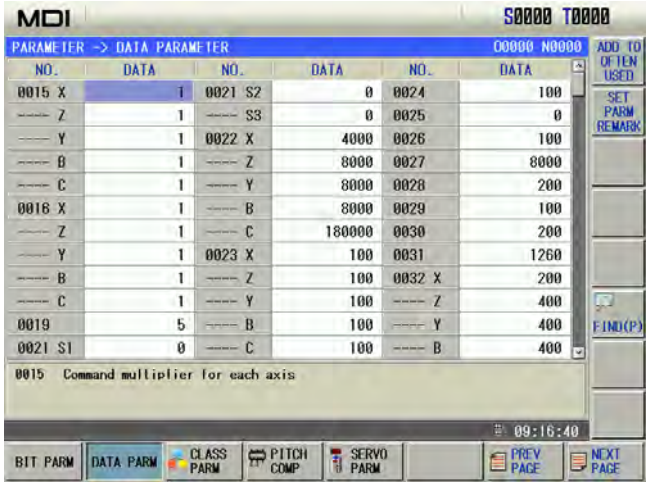
1.3.6 Parameter Interfaces

Press PARAMETER to enters PARAMETER interface including pages BIT PARM、DATA PARM、CLASS PARM、PITCH COMP、SERVO PARM which can be switched by pressing repetitively PARAMETER or the corresponding soft keys.

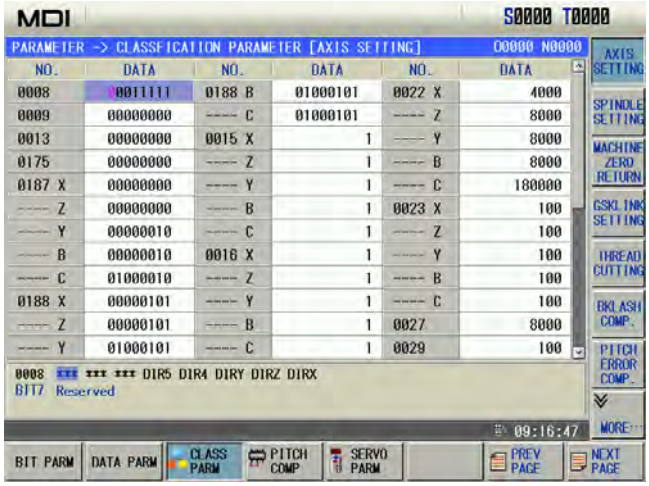
1) Bit parameter page



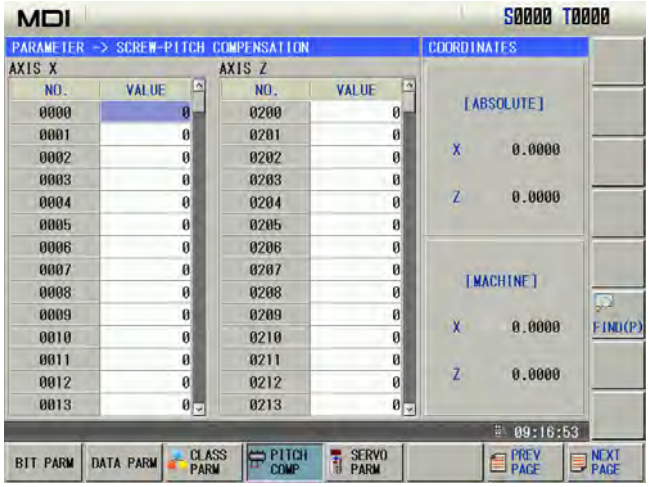
2) Data parameter page



3) **Common used parameter page**

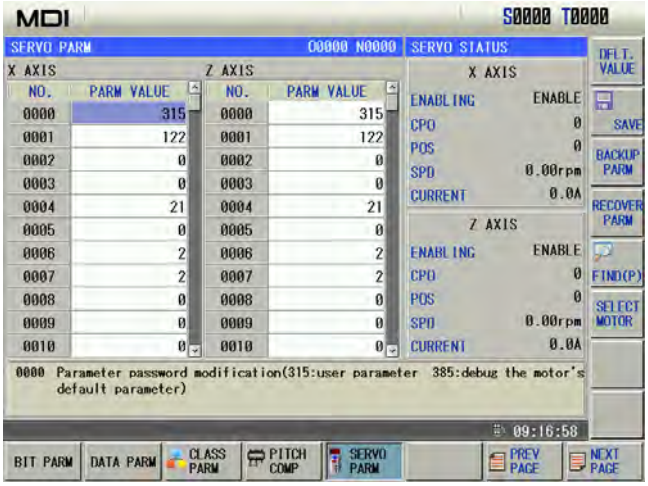


4) **Pitch compensation parameter page**



Note: When the system axis number is more than 2, pitch compensation parameter page will not display the "Coordinate information" window.

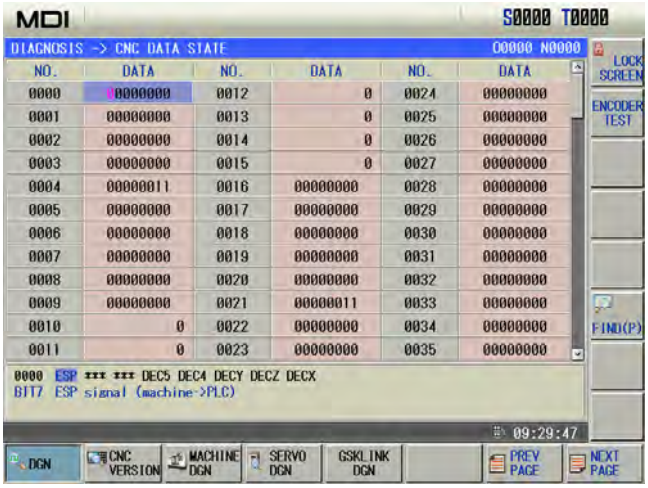
5) **Servo parameter interface**



1.3.7 Diagnosis Interface

Press DIAGNOSIS to enter diagnosis interface including pages DGN、CNC VERSION、MACHINE DGN、SERVO DGN which can be switched by pressing repetitively DIAGNOSIS or the corresponding soft keys.

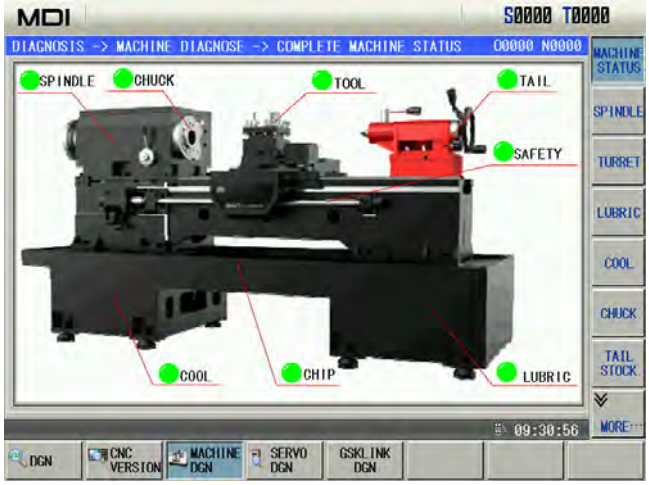
1) System diagnosis page



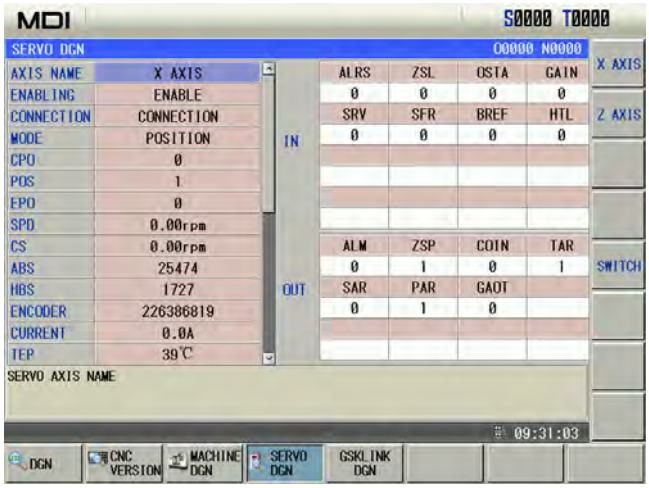
The page is used to detect CNC interface signal and its inner run state.

2) System information page

3) Machine diagnosis page



4) Servo diagnosis page



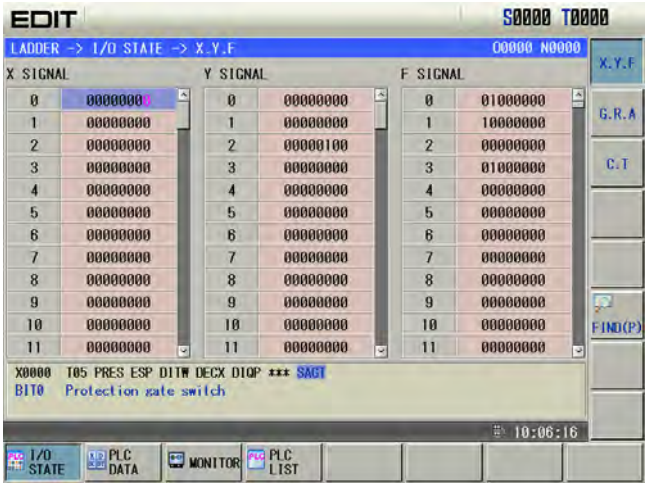
1.3.8 Graph Interface

Press to enter GRAPH interface.Graph page is shown below:

1.3.9 Ladder Interface

Press PLC to enter LADDER interface including pages I/O STATE、PLC DATA、MONITOR、PLC LIST which can be switched by pressing repetitively PLC or corresponding soft keys.

1) PLC I/O state page

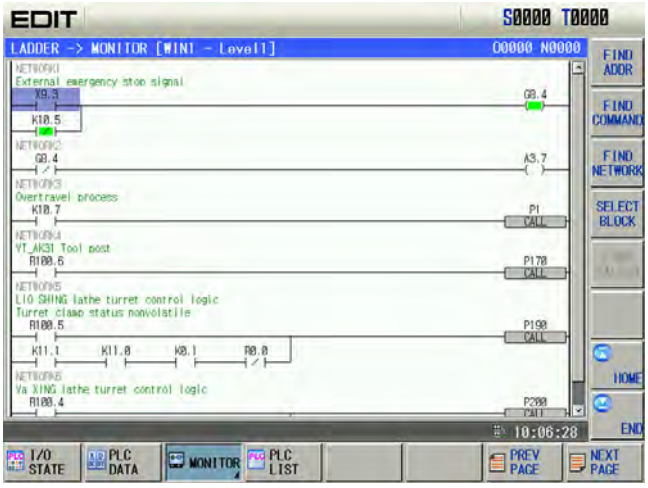


The page can display PLC I/O state information.

2) PLC data page

PLC data includes K, D, DT, DC, which can be set in the page.

3) 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. X0.5 ⊣■⊢ means the contact X0.5 is ON, Y25.2 ⊣( )⊢means Y25.2 is OFF.

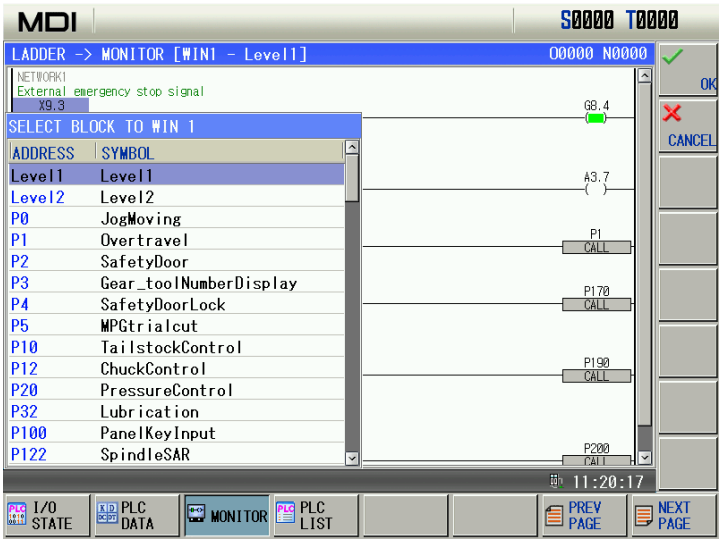
1. Page program view

In MONITOR page, the system can monitor simultaneously three windows, the ladder blocks corresponded to each window can be viewed by separately pressing [WIN1 Level1]、[WIN2 Level2]、[WIN3 P0], 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 [WIN1 Level1]、[WIN2 Level2]、[WIN3 P0] to select the windows.

（2）Press [SELECT BLOCK] 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 [OK], and then return the previous menu, press [CANCEL] 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 [FIND ADDR]、[FIND COMMAND]、[FIND NETWORK] to find the corresponding parameter, functional command and network in the current displayed block, and move the cursor to the corresponding position.

（3）Press [HOME]、[END] to locate the cursor to the home or end of the current block.

4) File list page

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

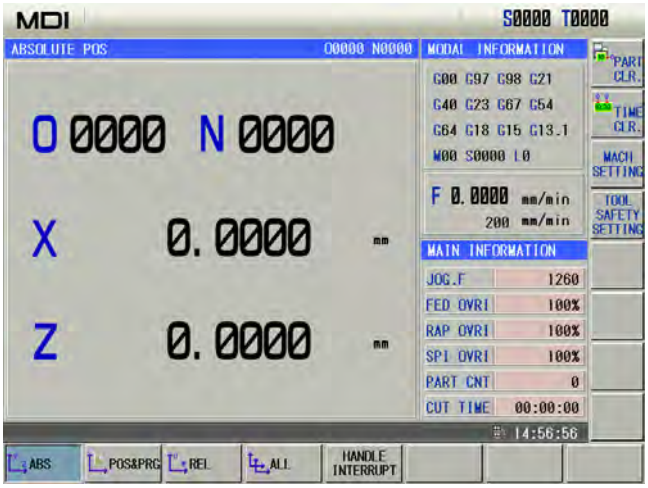
Before GSK980TDi power on, the following items should be confirmed:

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 GSK980TDi is turned on:

CNCmakers
www.CNCmakers.com

The current position (ABSOLUTE POS) page is displayed after GSK980TDi automatic detection and initiation are finished.



2.2 System Power-off

Before power is off, ensure that:

1. The feed 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.

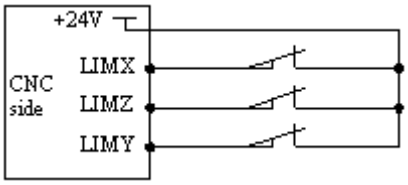
Note: Please refer to the machine manufacturer's manual about turn-off the machine's power supply.

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

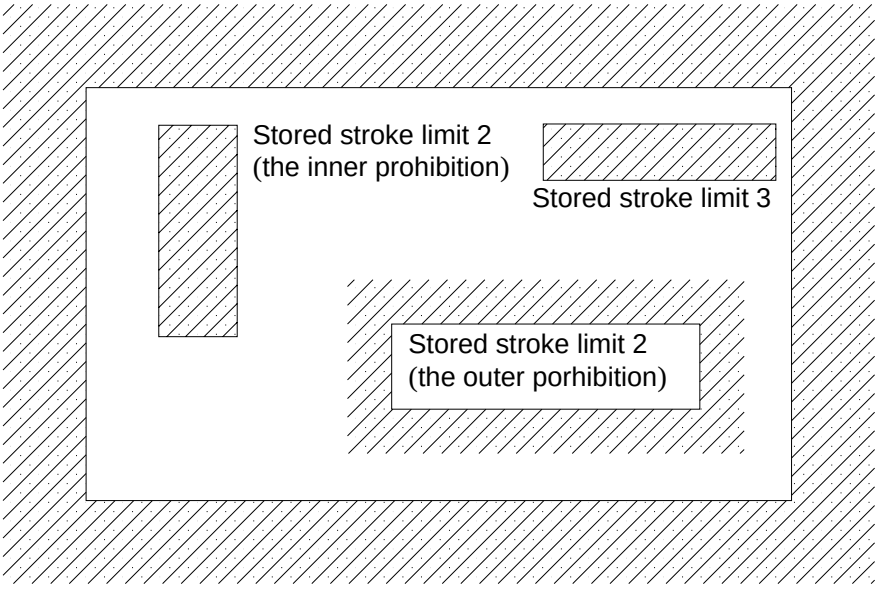
The stroke switches are fixed at the positive and negative maximum travel of the machine X, Z, Y axis respectively, they are connected by the following figure. Standard PLC program defines that the Bit7 of parameter K10 is whether the overtravel along each axis is enabled; Bit 6 is the overtravel signal LEVEL selection. If the overtravel occurs, the stroke switch acts to make GSK980TDi stop, and the emergency alarm issues.



When the hardware overtravel occurs, there will be an “overtravel stop” alarm in GSK980TDi. The steps to eliminate this alarm are: In the Manual or MPG mode, move the table reversely to detach the stroke switch (for positive overtravel, move negatively; vice versa).

2.3.2 Software Overtravel Protection

GSK980TDi sets the software overtravel protection of the machine tool by the stored stroke limit function.There are 3 stored stroke limit areas can be set in GSK980TDi. The oblique area is the forbidden one.



: the forbidden area

1) Open the limit function: Set the state parameter Bit 4 of No.172 to 0, and then open the limit detection function.
2) Stored stroke limit detection 1: The limit area boundary is set by parameter (No.045, No.046), the external side of the setting range is limit area. Usually, this area is regarded as the Max. stroke range of the machine tool.
3) Stored stroke limit detection 2 (G22 G23): The limit area boundary is set by parameter

(No.100, No.101) or G22; the inner prohibition/outer forhibition of the setting range are set by Bit 2 of parameter No.172. This limit is enabled when the G22 is at the current modal, and the limit is disabled when the modal is G23. Refer to the program help G22/G23 command.

4) Stored stroke limit detection 3: The limit area boundary is set by parameter (No.102, No.103); the inner side of the setting range is limit area.
5) Whether the axis is performed the limit inspection: Bit 2 of parameter No.190 and Bit 3 of parameter No.190 can be set an axis whether it performs the stored stroke inspection 2 or 3. The stored stroke inspectin 2 or 3 is only judged the axis from the inspection. For example: When X or Z axis performs the judgement, and the Y axis does not, only detect whether the X or Z axis enters to the limit area.
6) Zero return enabled: When the state parameter Bit 1 of No. 172 sets to 1, it needs to return the reference position by manual or G28 after the power is turned on, the function is then enabled accordingly. After an axis is performed the zero return for the external limit area, and the limit area of this axis is immediately enabled; it is disabled of the axis that does not perform the zero return. For the inner limit area, the limit function is enabled after the overall axes executed the detection are performed the zero return.
7) Limit deceleration: State parameter Bit 0 of No.172 can be set a stop method when it reaches to the limit boundary. Deceleration stop near the forbidden area or immediately stop after entering to the forbidden area can be selected.

Note 1 :If the positive stroke is less than or equals to the negative one in the setting forbidden area, the area is shown below:

1) When the forbidden area is the stored stroke inspection 1 or the 2 from the external prohibition, the overall areas are treated as the forbidden area.

2) When the forbidden area is the stored stroke inspection 3 or the 2 from the internal prohibition, the overall areas are regarded as the moveable areas.

Note 2 : An alarm may occur if the reference position locates within the limit area when returning the reference position.

2.4 Emergency Operation

During the machining, some unexpected incidents may occur because of the user programming, operation and product fault etc. So the GSK980TDi 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 GSK980TDi 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 ESP

During machine running, if the ESP button 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 button is released, the emergency alarm is cancelled and the CNC resets.

Note 1: The ESP alarm generates during the operation of the machine tool, 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 2: 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 tool-setting operation should be performed again.

CHAPTER 3 MANUAL OPERATION

Note!

The key functions of this GSK980TDi machine panel are defined by PLC program (ladders), please refer to the manual from the machine builder for their function significance.

Please note that the following functions for the machine panel keys are described based on the GSK980TDi standard PLC programs!

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 in and X feeds negatively or positively, and its feeding stops if the key is released; press or and Z axis feeds negatively or positively, and its feeding stops if the key is released; press or , and Y feeds negatively or positively, and its feeding stops if the key is released; press or and the 4th axis feeds negatively or positively, and its feeding stops if the key is released; press or and the 5th axis feeds negatively or positively, and its feeding stops if the key is released.

In Manual mode, press key to make the indicator enter the manual rapid traverse mode.

3.1.2 Manual Rapid Traverse

Press in and the key indicator is ON; press or and X

feeds negatively or positively, and its feeding stops if the key is released; press or and Z axis feeds negatively or positively, and its feeding stops if the key is released; press or , and Y feeds negatively or positively, and its feeding stops if the key is released; press or and the 4th axis feeds negatively or positively, and its feeding stops if the key is released; press or and the 5th axis feeds negatively or positively, and its feeding stops if the key is released. The rapid override time-tuning is enabled.

In Manual mode, press RAPID key to make the indicator OFF, and the rapid traverse is disabled and manual feed 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 GSK980TDi system.

Note 2: In Edit/MPG mode, key is inactive.

3.1.3 Speed Regulation

In Manual mode, press %+ F. OVERRIDE, 100% F. OVERRIDE, %− F. OVERRIDE to alter the manual feed override which is divided into 16 steps. The relation of the override and the feedrate are as follows table when data parameter No.031 is set to 1260:

Feedrate override (%)	Feedrate (mm/min)
0	0
10	126
20	252
30	378
40	504
50	630
60	756
70	882
80	1008
90	1134
100	1260
110	1386
120	1512
130	1638
140	1764
150	1890

Note : There is about 2% error for the data in the above table.

In the manual rapid traverse, it can press X1 F0 X10 25% X100 50% X1000 100% 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

: In Manual mode, the spindle rotates counterclockwise if pressing this key;

: In Manual mode, the spindle stops if pressing this key;

: In Manual mode, the spindle rotates clockwise if pressing this key.

3.2.2 Spindle 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 the 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 the 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 the stops the output. 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 Tailstock Control

: In any mode, press it to switch the tailstock forward/backward.

Function description:

When the tailstock control is invalid, the tailstock control M command alarms;

When the tailstock and the spindle control is interlocked, before the spindle is started, the system check whether the tailstock forward is valid. After the spindle is started, the tailstock backward cannot be executed.

Parameter setting:

PLC parameter：K13.0 1/0：tailstock control valid/invalid

PLC parameter：K13.1 1/0：the spindle rotation and the tailstock forward/backward are/not interlocked

PLC parameter K13.2: 1/0: External tailstock control is enabled/disabled when the program is being operated.

3.2.7 Hydraulic Control

: press it to switch the hydraulic motor ON/OFF in any mode.

Function description:

When the hydraulic control function is valid,

is pressed to control HPST output to start the hydraulic motor.

When the system is in non-run, the spindle stops and the speed is zero,

is pressed to close HPST output.

Parameter setting:

PLC parameter: K14.7 1/0: hydraulic control function valid/invalid

3.2.8 Manual Tool Change

: In Manual mode, pressing the key can manually change tools in sequence (if current tool is No.1, it is changed for No.2 tool by pressing it; if current tool is No.4, it is changed for No.1 tool by pressing it.

3.2.9 Spindle Override

In Manual mode, if the spindle speed is controlled by analog voltage output, the spindle speed may be overrided.

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.

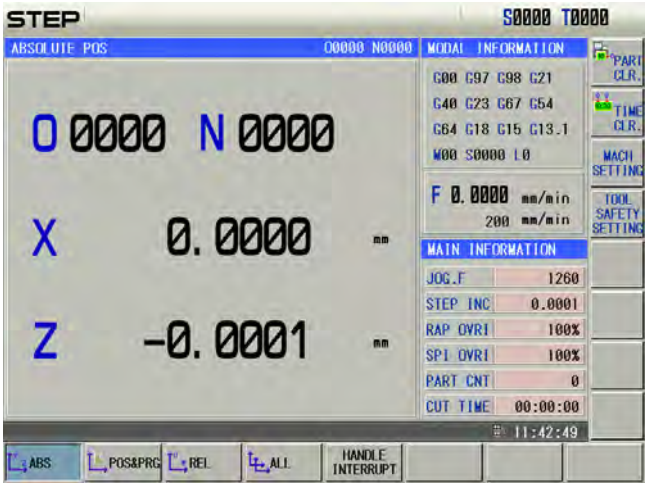
Note!

The key functions of this GSK980TDi 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 GSK980TDi standard PLC program!

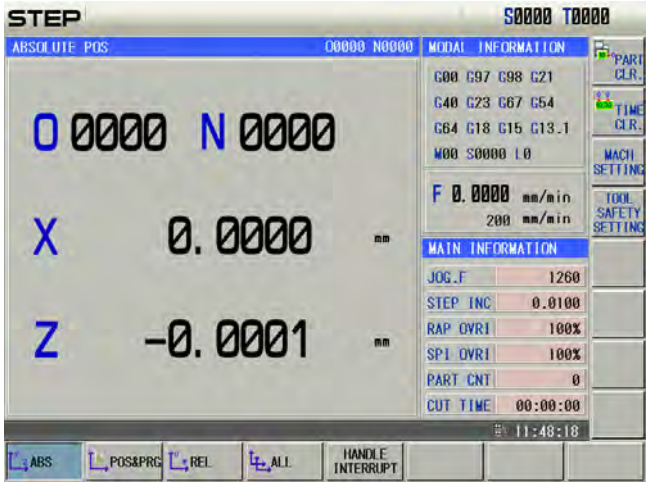
4.1 Step Feed

Set the system parameter No.001 Bit3 to 0, and press key to enter the STEP working mode, it displays as follows:



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, are all active. For example, to press key, the window is shown as follows:

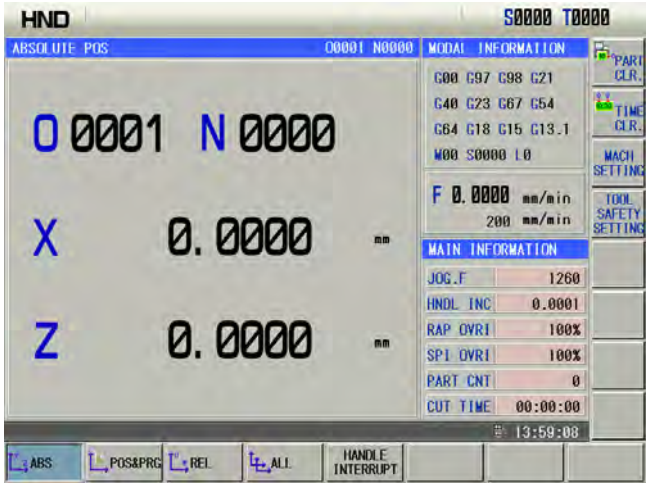


4.1.2 Moving Direction Selection

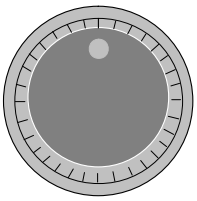
Pressing or key once can move X negatively or positively by a step increment; pressing or key once can move Z negatively or positively by a step increment; pressing or key once can move Y negatively or positively by a step increment.

4.2 MPG Feed

Set the BIT3 of the system parameter No.001 to 1, and press key to enter the MPG mode, it displays as follows:



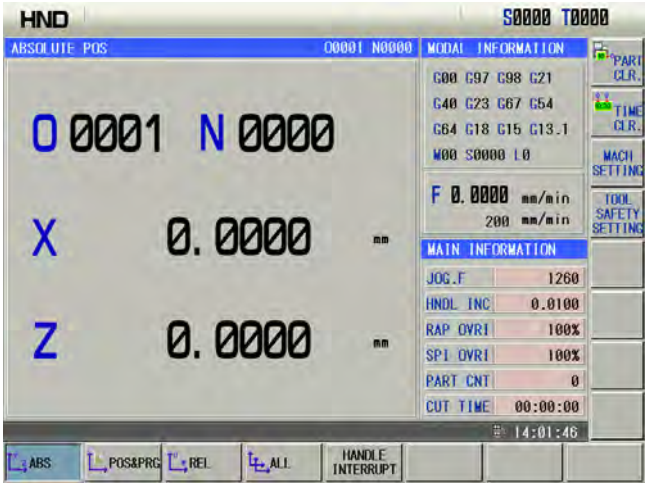
The MPG figure is shown as follows:



The handwheel (MPG) figure

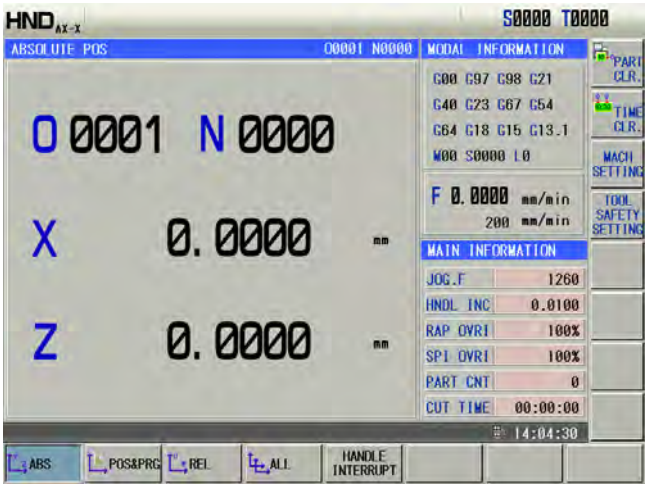
4.2.1 Increment Selection

Press X1 F0 X10 25% X100 50% X1000 100% key to select the move increment, the increment will be shown in the window. When the BIT7(SINC) of PLC K016 is 1, step X1000 100% is inactive; when BIT7 is 0, X1 F0 X10 25% X100 50% X1000 100% are all active. For example, to press X100 50% key, the window is shown as follows:



4.2.2 Moving Axis & Direction Selection

In MPG mode, press X, Y, Z, 4th, C key, the corresponding axis will be selected. For example, press X key and the window is shown below:



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 MPG Feedrate

The upper limit of the MPG feedrate is the rapid traverse rate of each axis, which can be set by data parameter No. 022.

When the state parameter No.013.6 = 0, and the MPG feedrate exceeds the rapid traverse rate, the exceeded pulse is ignored; the MPG scale is inconsistent with movement amount。

When the state parameter No.013.6 = 1, and the MPG feedrate exceeds the rapid traverse rate, the exceeded pulse does not ignore; the MPG scale is consistent with movement amount, but the axis may consecutively move when MPG stops.

4.2.4 Other Operations

1） Spindle CCW, CW, stop control

: In Manual mode, the spindle rotates counterclockwise if pressing this key;

: In Manual mode, the spindle stops if pressing this key;

: In Manual mode, the spindle rotates clockwise if pressing this key.

2） Spindle JOG

: at the moment, the spindle is in JOG working mode.

In spindle JOG mode, pressing can rotate counterclockwise for jogging; pressing can rotate clockwise for jog. The jogging time and speed are set by DT12 and No.109 respectively. Referred to Section 3.2.2.

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） Tailstock control

Refer to OPERATION, Section 3.2.6

7）Hydraulic control

Refer to II OPERATION, Section 3.2.7

8) Manual tool change

: In MPG/Step mode, press it to change tools orderly.

9) Spindle override tune

In MPG/Step mode, if the spindle speed is controlled by analog voltage output, the spindle speed may be overrided.

Pressing , can change the spindle speed in real-time to adjust 8 steps of 50％～120％ of the spindle override

4.2.5 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

（Taking example of the least input increment 0.001mm）

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.

Note!

The key functions of this GSK980TDi machine panel are defined by PLC program (ladders), please refer to the manuals by the machine builder for their significance.

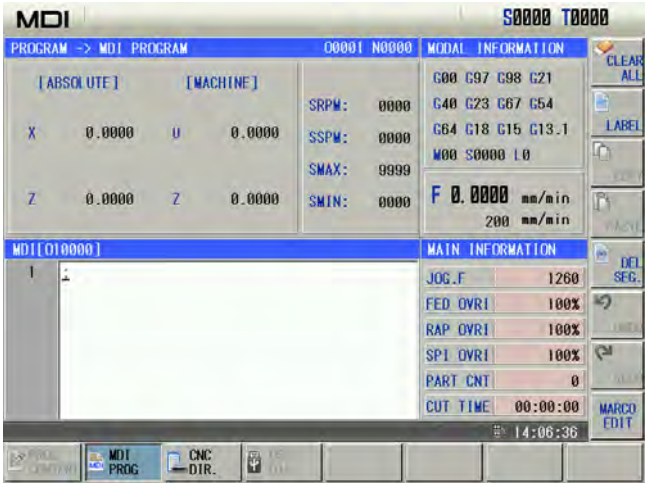
Please note that the following description for the key functions in this chapter is based on the GSK980TDi standard PLC program!

5.1 Block Input

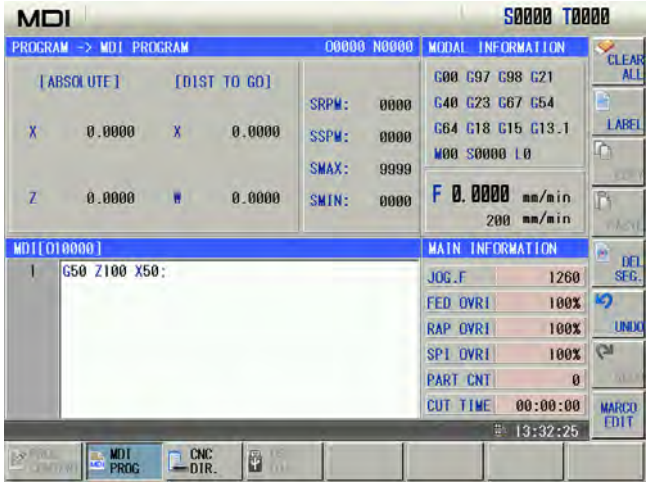
Select MDI mode to enter PROGRAM->MDI page, to input an block G50 X50 Z100, the steps are shown below:

1) Press key to enter MDI mode;

2) Press , and then press to enter MDI program page below:



The page is shown as follows after above operations are completed:



Note: Up to 10-block programs can be input of the MDI program.

5.2 Block Execution

After the word is input, is pressed and the display is as follows:

After the code words are input and is pressed to display the block, key is pressed to execute the input block. During the execution, , and Emergency Stop button may be pressed to terminate these code words execution.

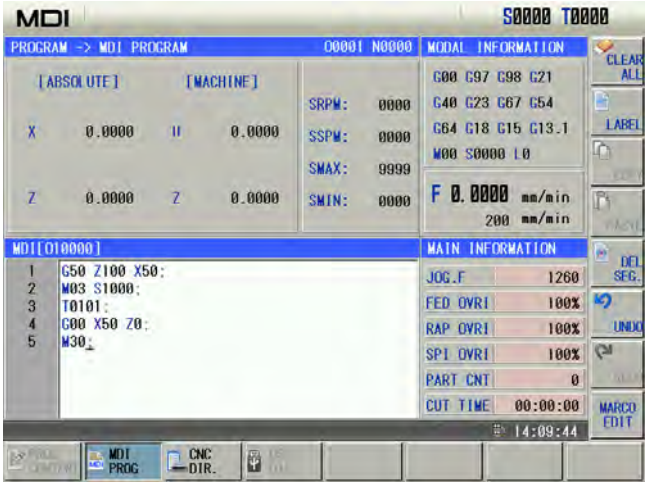
5.3 Parameter Setting

In MDI mode, the parameter value can be altered after entering the parameter interface. The detailed is referred to **OPERATION, Chapter 11**.

5.4 Data Alteration

There are , , , , , , and program edit operation items on the MDI program page.

In the MDI mode, press the PROGRAM, then MDI PROG soft key, and then enter to the MDI program page.



CLEAR ALL soft button: Clear the overall contents of the MDI program input frame;

LABEL soft button: The movement cursor can be selected the block for the program after pressing this button, which uses matching with other function soft buttons;

COPY soft button: Copy the block that locates at the cursor; it can be used matching with the LABEL soft button;

PASTE: Paste the copied block to the next line of the cursor;

DEL SEG.: Delete the block of which the cursor locates, which can be used matching with the LABEL soft button;

UNDO soft button: Retract the previous edit operation of the MDI program, up to 10 times can be performed;

REDO soft button, Cancel the retract operation happed just last step;

MARCO EDIT soft button: Spring out the macro command list, insert the selected macro command at the cursor position;

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.

Pressing S. OVERRIDE, S. OVERRIDE 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 federate override to change the federate, which can realize 16 steps adjustment from 0～150％.

4. Feedrate override is available.

CHAPTER 6 PROGRAM EDIT & 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 GSK980TDi system. And it must be turned on before program editing. Please see details in Section 10.1.1 of the part.

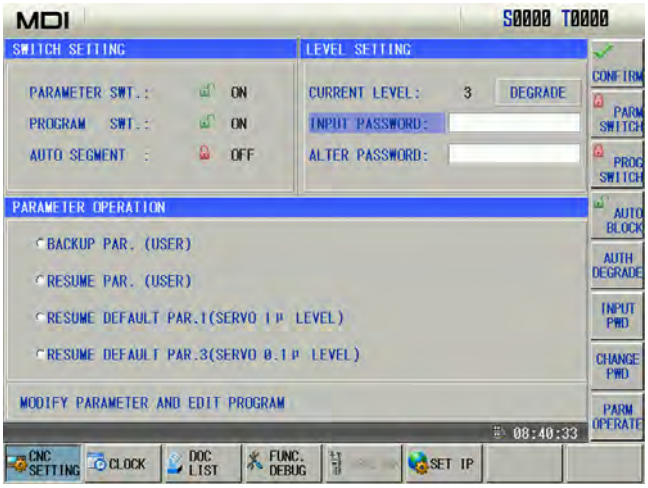
The system provides multi-level user permission to facilitate the management. More than 4 levels (4 level, 3 level) can open the program switch to edit programs. See **OPERATION, Section 10.1.2 .**

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 SEG” 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 SEG” switch in SWITCH SETTING window is ON, CNC generates the block number automatically, it automatically generates the next block number by pressing EOB or INSERT 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 EDIT to enter the Edit mode. Press PROGRAM to enter the Program interface, create a machining program before inputting a machining program. The creating method is shown below:

Method 1:

1） Press address key O, number key 0, 0, 0 and 1 key by sequence (e.g. taking an example of creating program O0001, the leading 0 can be omitted when its name is input). Its display is shown below:

2） Press (or) to create the new program. The display is shown below:

Method 2:

1) Pres to enter the local directory page. The display is shown below:

2) Press , input orderly , , , in the pop-up dialog box（taking example of creating O0001, the leading zero can be omitted when it is input）. It display is shown below:

3) Press (or) to create the new program. The current page is automatically switched into the program content page. 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.

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;

2. Finding: To search for the specified character upward or downward from the cursor current location

The steps of finding is as follows:

1) Press EDIT to enter Edit mode, and press PROGRAM key to enter the PRG CONTENT window;

2) Press SEARCH, and input the required characters to be searched(can input one block). For example, finding "G00" page is shown below:

4) Press key (or is determined by the location of the character searched to the character where the cursor locates), it displays as follows:

4) After the finding, the CNC system is still in FIND state, press ↑ or ↓ key again, the next character can be found. Or press CHANGE key to exit the FIND state.

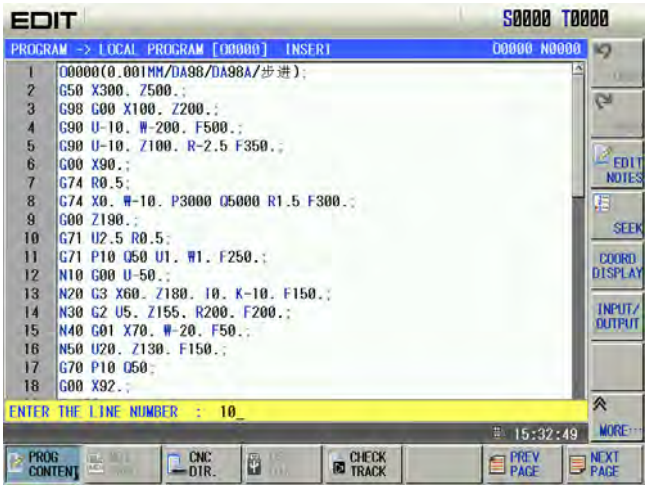
5) If the character is not found, the prompt of “Search fail” will be displayed.

Note : During the searching, it doesn’t search the characters in the called subprogram, and the character in subprogram is searched in subprogram.

3. Positioning the cursor to the specified line

1) Press EDIT to enter Edit mode, and press PROGRAM to enter the PRG CONTENT page.

2) Press SEEK, and input the line number (the line number of block is the left of one row label). For example, the cursor positions to the 10th line, the display is shown below:



3) Press INPUT (or EOB), and the display page is shown below:

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, and the “Insertion” shows at the title column), the window is as follows:

3) Input the character to be inserted (to insert G98 code before G50 in the above figure, input G, 9, 8); it displays as follows:

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 "space" will be added automatically following the decimal point.

6.1.5 Deleting a Character

Steps:

1) Press to enter PRG CONTENT page in Edit mode;

2) Press key to delete the character before the cursor; press key to delete the character where the cursor locates.

6.1.6 Altering a Character

There are 2 methods for the character alteration:

Insertion: First delete the character altered by the methods in Section 6.1.5, then insert the character required by the methods in Section 6.1.5 of this part.

Direct alteration:

1) Select the PRG CONTENT page in Edit mode;

2) Press key to enter the ALT state (the cursor is a backlight rectangle, and the "Modification" shows at the tile column), the display page is as follows:

3) Input the modified character(as the above figure, alter X300 to X250, input 2, 5, 0), it displays as follows:

Note : In ALTER state, the current character where the cursor locates is altered for the input one, and the cursor moves a byte forward;

6.1.7 Deleting a Single Block

Single block deletion:

1) Press PROGRAM in Edit mode to enter program interface, and switch program content page.

2) Move the cursor to the head of the block to be deleted, then press DEL SEG. key.

Multi-block deletion:

1) Select the edit operation mode, enter to the program page set by PROGRAM button, and then shift to the program content page;

2) Move the cursor to the beginning of the block to be deleted, press the LABEL soft button, and then move the selected block by cursor, the page is as follows:

4) Delete the selected block by .

6.1.8 Copying & Pasting a Block

Single block copy/paste

Steps:

1) Select Edit mode, enter to the program page set by and then shift to the program content page;

2) Move the cursor to the required copy block, press and , and the current block is copied;

3) Move the cursor to the required paste position, press , and the copied block is pasted.

Single block copy/paste

Steps:

1) Select the Edit mode, enter to the program page set by and then shift to the program content page;

2) Move the cursor to the beginning of the block to be copied, press the soft button, and then move the selected block by cursor;

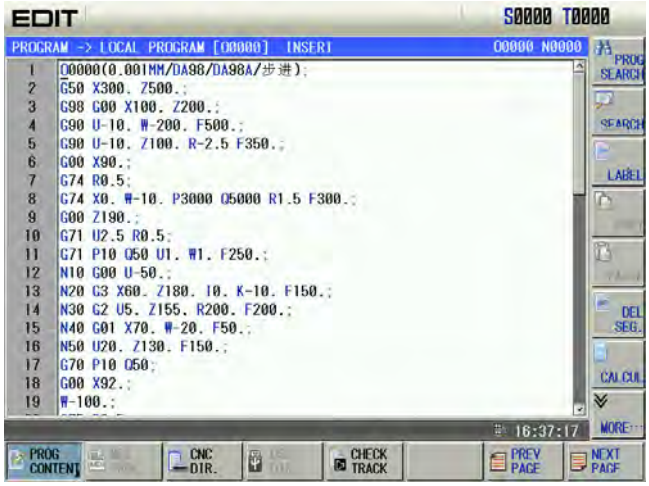
3) Copy the selected block by soft button;

4) Move the cursor to the required paste position, press , and the copied block is pasted.

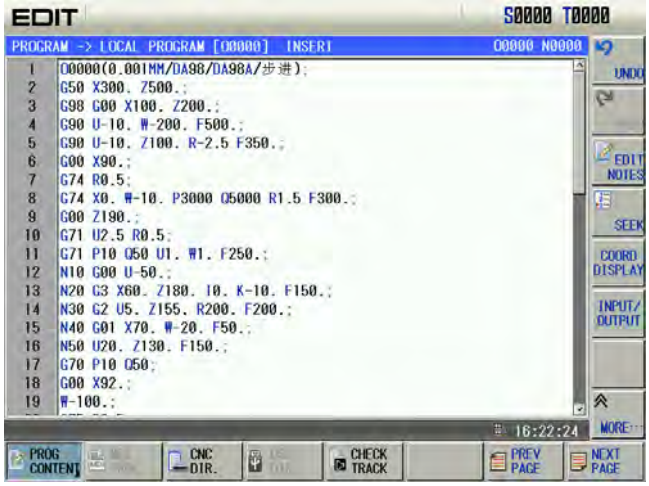
6.1.9 Canceling & Recovering a Program

Steps:

1) Select Edit mode, enter to the program page set by and then shift to the program content page, which is shown below::



2) Press MORE to enter the next menu, which is shown below:



3) When the current content is changed(after modification, copy or deletion), UNDO is pressed to recover the previous content before the copy;

4) Recover and cancel are matched. Press UNDO, REDO to cancel the previous operation.

Note: Up to 10 previous steps can be retreated by using the retraction function;

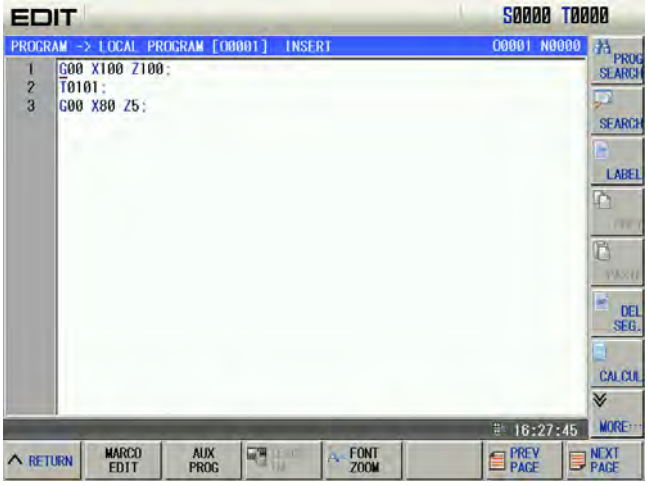
6.1.10 Circle Command Aided Programming

The cycle G command should be input more parameter commands. This function is the programming guiding function for each cycle command; Operator can easily finish the programming operation for the cycle command by the miscellaneous programming function. There are two methods to enter the miscellaneous function, refer to the following steps:

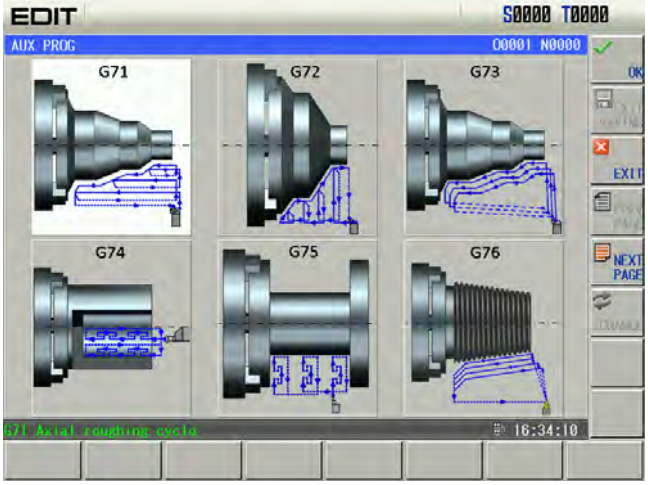
Method 1:

1）Press EDIT soft key enter to edit mode，press PROGRAM soft key, enter to the program page set and then shift to the program content page, so that the cursor does not the line of that the cycle command locates, which is shown below:

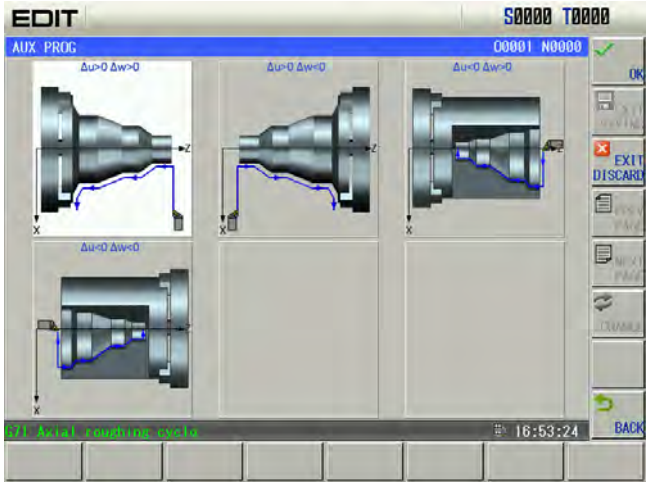
2）Open the program content sub-page by PROG CONTENT soft button, which is shown below:



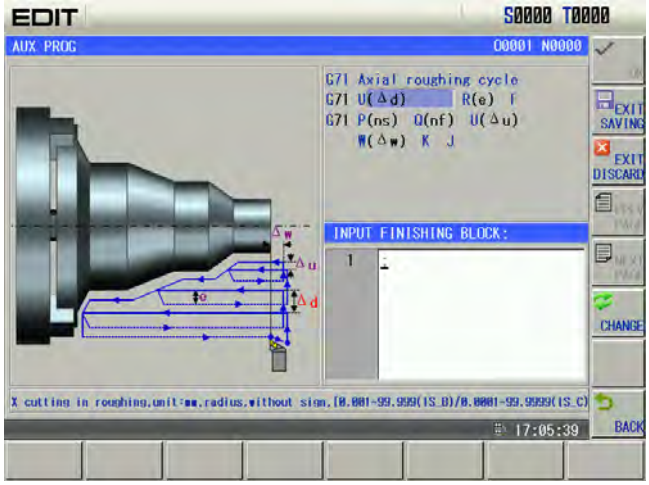
3）Enter to the miscellaneous programming page by controlling the AUX PROG soft function button.，which is shown below:



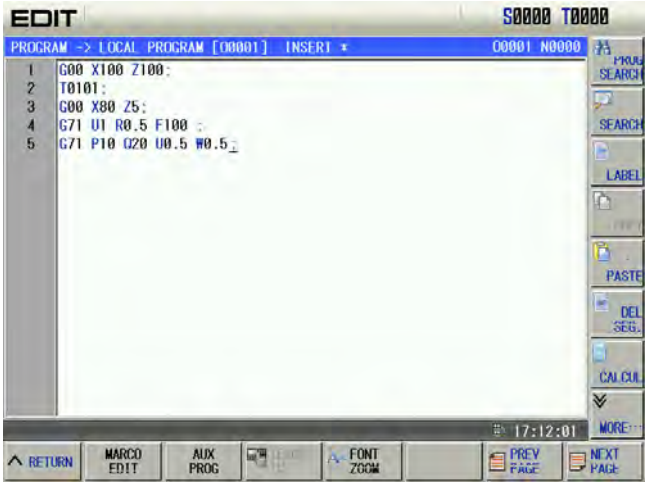
4) Select the cycle command to be compiled by ↑, ↓, →, or ← button, the selected cycle commands will show with highlight color; next page can be displayed by shifting the , , (PREV PAGE or NEXT PAGE), and then enter to the next page by OK soft button; refer to the following figure:



4) Perform the cutting-in direction selection on this page (Partial commands are regardless of the cutting-in direction selection, but enter to the next page directly); enter to the next page by OK soft button; refer to the following figure:

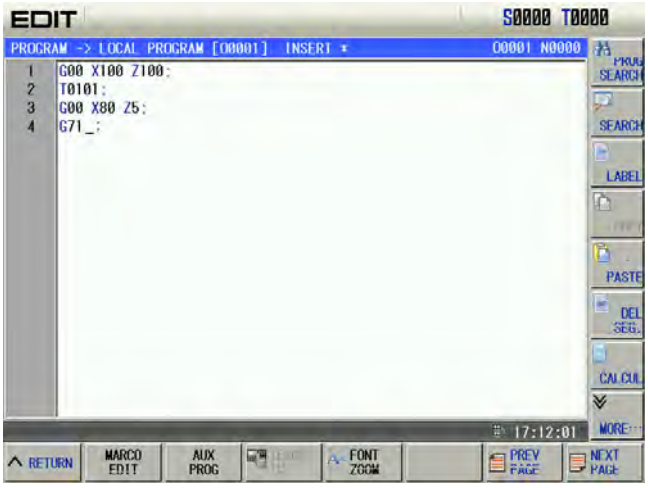


5) Input the parameter numerical value in this page, the figure at the left is the machining schematic; the command parameter input frame is at the upper right coner; the command parameter to be input can be selected by ↑, ↓, → or ← button. When the command parameter is selected the left machining schematic, the corresponding address may display with red color; the numerical value then can be written into the input frame by its corresponding number buttons; write the numerical value to the input frame by . The input frame (Only the G71, G72 or G73 owns the accuracy-machining block input frame) at the lower right corner is the one for inputting the accuracy-machining block. The cursor can be shifted between the command parameter input frame and the one of the accuracy-machining block by CHANGE or CHANGE soft button. A new program is then created by EXIT SAVING soft button after inputting data; refer to the following figure:

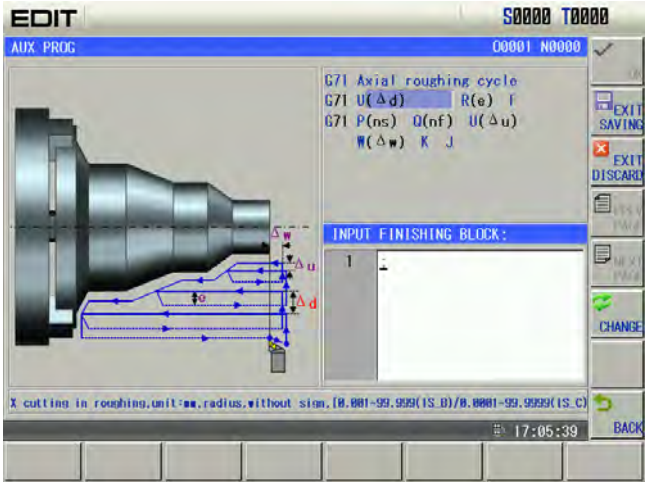


Method 2:

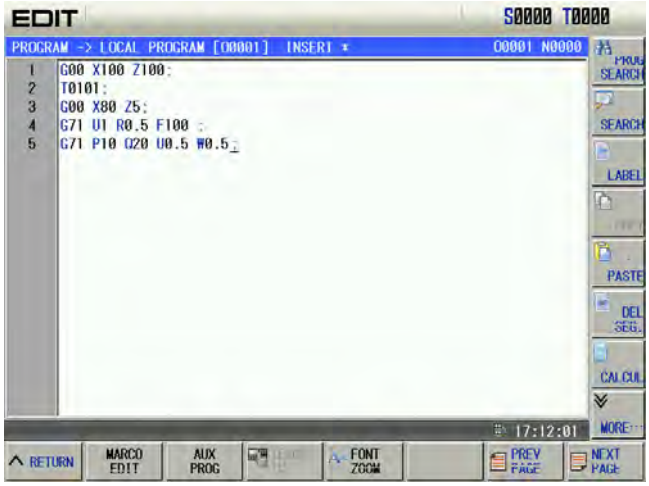
1) Enter to the Edit operation mode by ; enter to the program page set by and the shift to the program content page, and then open the program content sub-page by , put the cursor on the cycle command line, which is shown below:



2) System may automatically identify that the cycle command type enters to the corresponding programming help interface by , and then record the stored data, which is shown below:



3) A new program is created by soft button after inputting the data, which is shown below:



6.1.11 Program Save

In Edit mode, after the program is edited or modified, although the system automatically save programs at regular time, shift to other working mode from edit or switch the current display page and the CNC will automatically save the current program.

Note: "*" in the title bar is displayed when the program is not saved.

6.1.12 Macro Program Edit

Method 1:

Select the edit operation mode, enter to the program page set by and then shift to the program content page; enter to the macro edit state consecutively pressing the ; refer to the following figure:

Then it may input some special characters ([,] , =, >, <, +, *).

For example: To input character " [", it may press key repeatedly.

Method 2:

Edit operation method selection, enter to the program page set by and then shift to the

program content page，press PROG CONTENT soft key，enter to the program content page，press MARCO EDIT soft button to open the macro edit function menu, which is shown below:

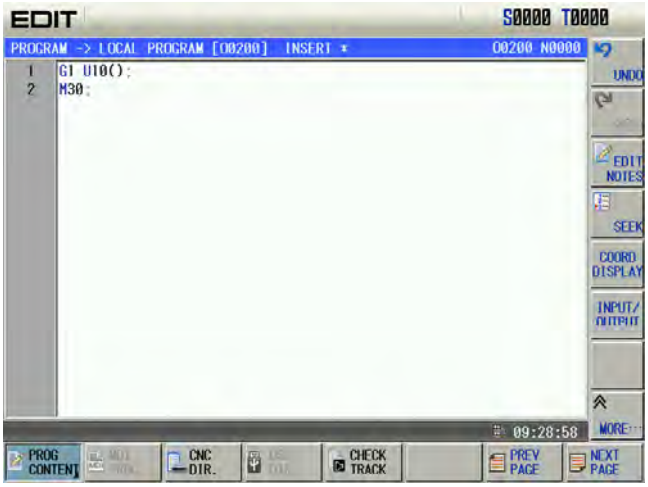
In this case, automatically enter to the macro edit state, some special characters, such as ([,], ＝, ＞,＜, ＋ and ＊) can be input. The macro program command can be inserted by the function button at the right side, so that the macro edit is more simple and enabled

Note: In the macro program edit state, program index or scan operation is unavailable.

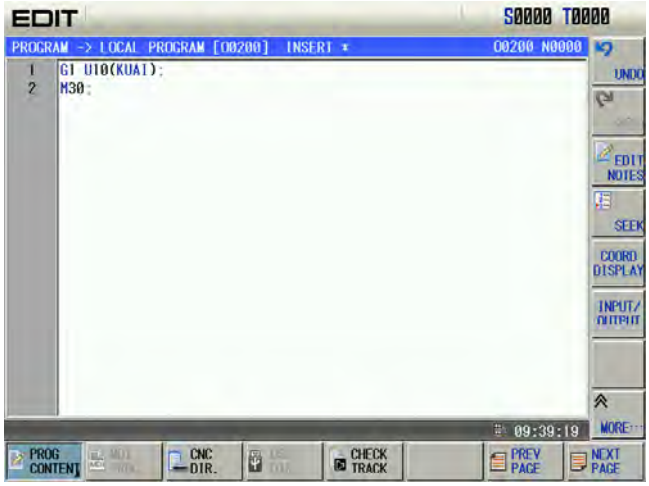
6.1.13 Creating & Modifying a Program Annotation

GSK980TDi can add annotation to each line, and its steps are shown below:

1) Select Edit mode, press PROGRAM to enter program interface and switch enter the program content page;
2) Press EDIT NOTES and the CNC automatically insert “( )” into the end of line where the cursor is, and the display page is shown below:



3) Input annotation character into “( ). For example, input “KUAI” character string, and the display page is shown below:



4) The modification of annotation is the same that of program content.

Note : The annotation added to the CNC could only be character. Actually, the CNC takes all content in "()" as annotation, so, adding other annotation can be executed by using PC editing, and then be transmitted to the CNC.

6.2 Deleting Programs

6.2.1 Deleting a Program

Steps:

1) Select Edit mode, press PROGRAM to enter program interface and switch enter the program content page;

2) Press address key O, number key 0, 0, 0, 1 by sequence (taking example of O0001);

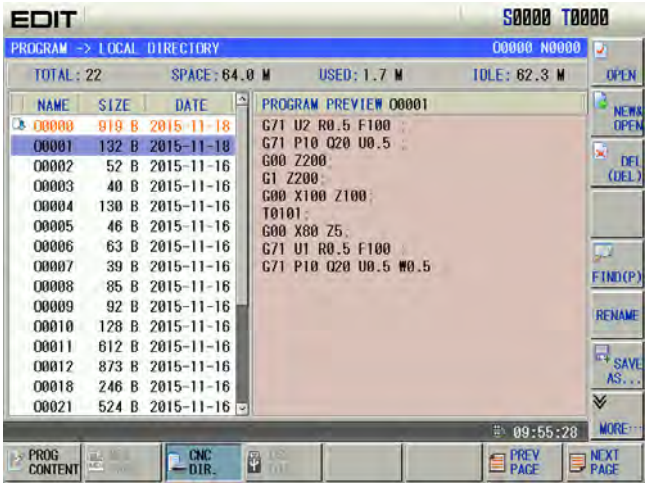
3) Press DELETE key, pop-up the dialog box as follows:

4) Press and O0001 program is deleted; press and the deletion is cancelled.

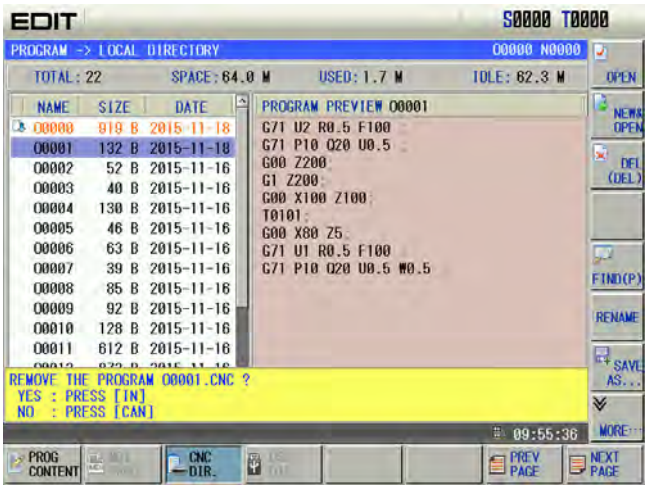
Method 2:

1) Select Edit mode, press to enter program interface, press to enter the local program page;

2) Press or to select the required deleted program. For example, select O0001 program and the display is shown below:



3) Press key, pop-up the dialog box as follows:



4) Press [DATA INPUT] and O0001 program is deleted; press [CANCEL] and the deletion is cancelled.

6.2.2 Deleting All Programs

Steps:

1) Refer to Section 6.2.1, Method 2 to enter the local program page;

2) Press [MORE…] to enter the next menu and the display is shown below:

3) Press [DELETE ALL], and then press [DATA INPUT], the all program are deleted.

6.3 Selecting a Program

When there are multiple programs in CNC system, a program can be selected by searching, scanning and direct confirmation.

6.3.1 Searching

1) Select Edit or Auto mode，enter to the program page set by [PROGRAM] and then shift to the program content page;

2) Press [PROG SEARCH], (the short key is [O]) and the display is shown below:

3) Input the program number in the pop-up dialogue box. For example, search O0001 program (the leading zero can be omitted when the program name is input) and the display is shown below:

4) Press ↓ (or ↑, EOB, INSERT), the searched program will be displayed. If the program does not exist, the CNC prompts “File does not exist”.

Note: Press EOB or INSERT to search programs in Edit mode. When there is no program, the CNC automatically creates programs; press ↓ or ↑ to execute the search, and the CNC prompts “The file does not exist” when there is not the program.

6.3.2 Scanning

1) Select Edit or Auto mode;

2) Press PROGRAM to enter the program display page;

3) Press O;

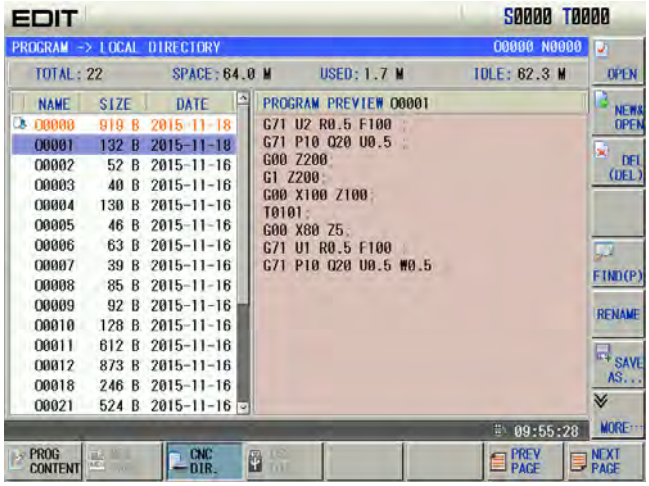
4) Press ↓ or ↑ to display the next or previous program;

5) Repeat step 3 and 4 to display the saved programs one by one.

6.3.3 Direct Confirmation

1) Select Edit or Auto mode (must be in non-run state);

2) Press PROGRAM to enter the program interface. When the system is not in “Local Directory” , CNC DIR. is pressed, which displays is shown below:



3) Press ↑, ↓ to move the cursor to the program name which is to be selected.

4) Press EOB (or INSERT), and the selected program is opened and the CNC skips to “PRG. CONTENT” page.

Note: when the CNC is connected with the U disk, the above similar method is referred to open the program in the U disk.

6.4 Executing a Program

After the program to be executed is selected by the method in OPERATION, Section 6.3, select the Auto mode, then press CYCLE START key (or external cycle start key), the program will be executed automatically.

6.5 Renaming a Program

Methods of renaming a program is divided into two:

Method 1:

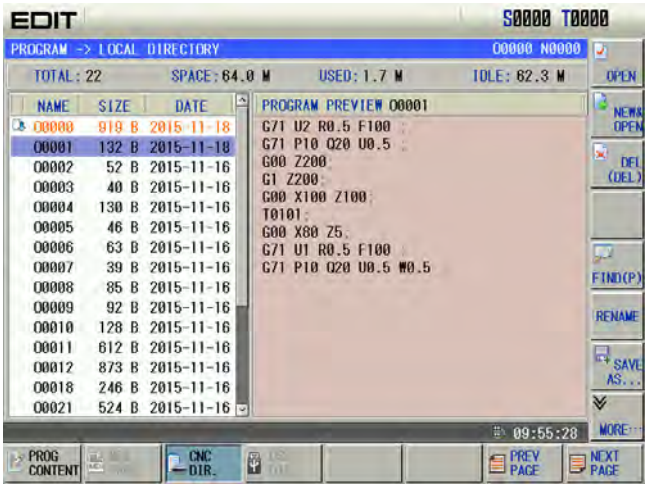
1) Select Edit mode, press PROGRAM to enter the program interface, and in necessity press PROG CONTENT to enter PRG CONTENT page (must be in Insert, Alter state):

2) Press to input a new program name;

3) Press .

Method 2:

1) Select Edit mode, press to enter the program interface, and in necessity press CNC DIR. to enter the local directory page:



2) Move the cursor to the program which is to be modified, press RENAME and input a new program name in the pop-up diaglog box, and at last press .

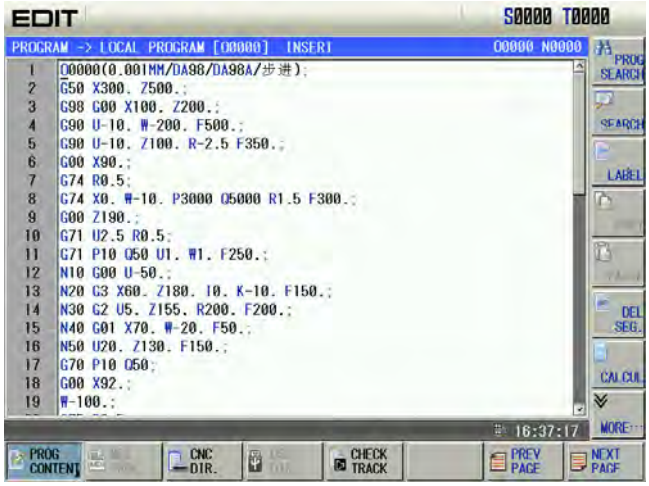
Note: the method 1 is only used to the current program, and the method 2 is used to any programs.

6.6 Copying a Program

Methods of copying a program is divided into two:

Method 1:

1) Select Edit mode, press to enter the program interface, and in necessity press PROG CONTENT to enter PRG CONTENT page (must be in Insert, Alter state):

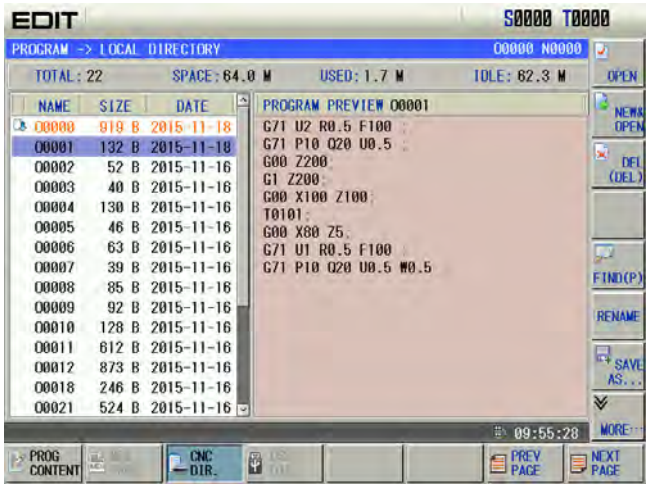


2) Press to input a new program name;

3) Press CHANGE.

Method 2:

1) Select Edit mode, press PROGRAM to enter the program interface, and in necessity press CNC DIR. to enter the local directory page:

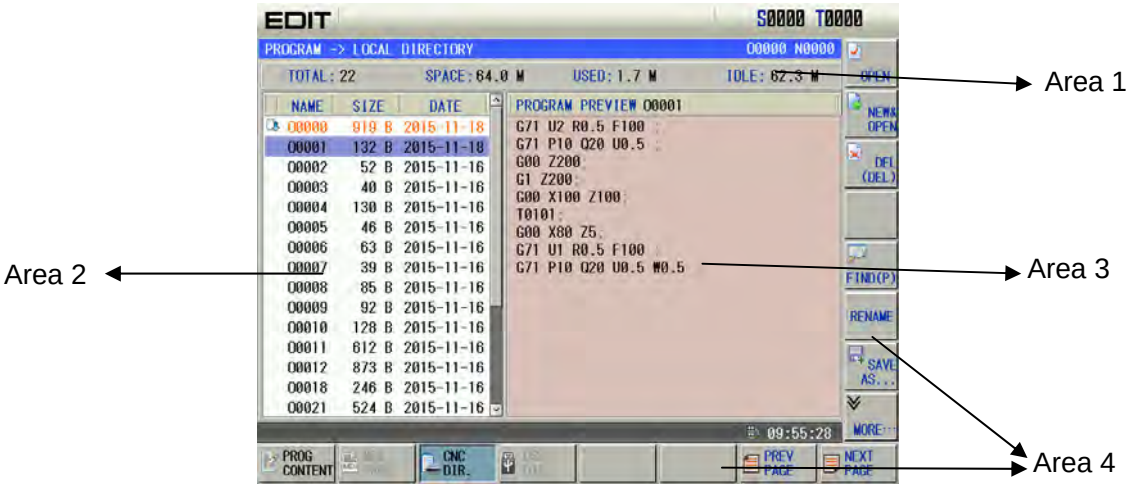


2) Move the cursor to the program which is to be modified, press SAVE AS... and input a new program name in the pop-up diaglog box, and at last press .

Note: the method 1 is only used to the current program, and the method 2 is used to any programs.

6.7 Program Management

Press PROGRAM ,and then press CNC DIR. to enter the local program display page. In the page, it lists the program names saved in CNC system, and it can display up to 15 names in a page. When the saved exceed 15 programs, it may press or to display the next page.



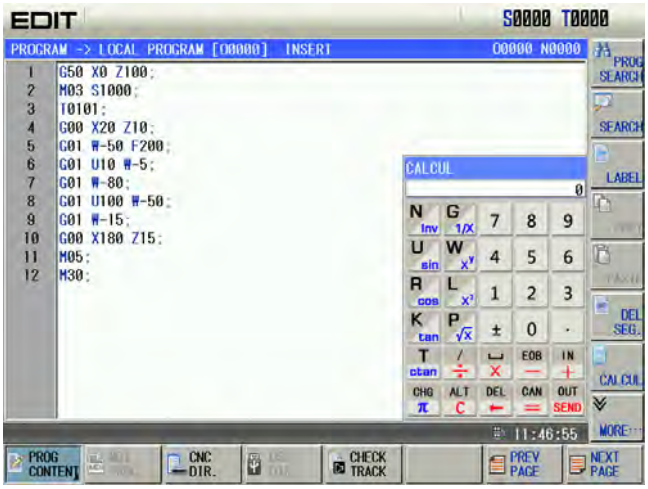
Explanations:

1) The area 1 is used to display the saved programs, the total memory capacity, the used capacity and the left;
2) The area is used to display the current program list;
3) The area 3 is used to preview the content of program where the current cursor is in the area 2;
4) The area 4 belongs to the soft function menu position. Pressing NAME SORT, TIME SORT can display in order programs located at the area 2.

6.8 Counter Function

1、press PROGRAM to enter the program content page。

2、Spring out the counter window by CALCUL soft button; Alternatively, directly press the at the program content page, the calculator may appear, refer to the following figure:



3. Control the corresponding button function on the MDI panel to perform the number input and calculation based upon the content of the calculator. The calculated result can be transmitted to the program cursor on the MDI panel based upon the displayed OUT SEND from calculator, and then retreat from the calculation function.

a) Press the to retreat from the calculation function, alternatively, it can be carried out by

soft button.

6.9 Font Scaling

Enter to the program page set by

; enter to the program content page by

soft button (It can not be entered in the MDI mode) if necessary, refer to the following figure:

Open the next level optional menu by pressing the

soft button again; refer to the following figure:

Open the font scaling function optional menu by

soft button; refer to the following figure:

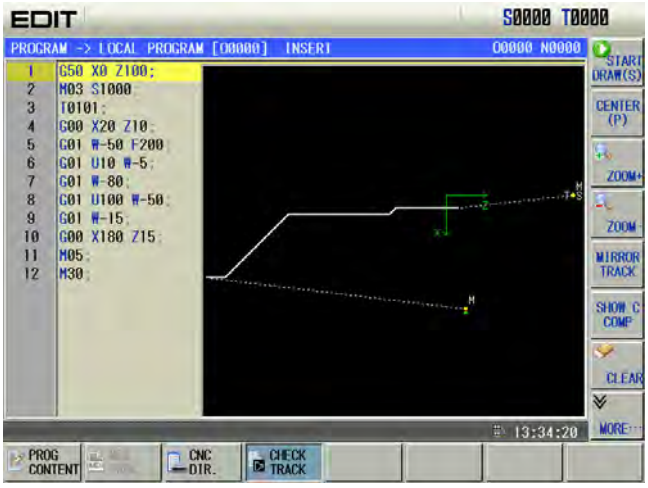
The font size can be adjusted by FONT ENLARGE、FONT NARROW,this function is only enabled to the displayed machining program on the program content page.

6.10 Program Preview Function

In order to reduce the alarm error in the operation program and improve the safety of component program performance, GSK980TDi adds the [Path Preview] function, also displayed the direction of the programming path, so that it is convenient to confirm whether the program is correct.

6.10.1 Interface Display of Path Preview

Select the Edit mode, on the program -> local program interface; press the CHECK TRACK soft button after the program is compiled; refer to the following figure:



6.10.2 Operation of Path Preview

1) The beginning or dwell of path preview

Enter to the path preview page; the programming path is then automatically displayed. Alternatively, the program path can be shown by START DRAW(S) soft button (or S button) on the path preview page, refer to the following figure:

The drawing can be temporarily stopped by STOP(T) soft button (or T button) during the diagram;

In the process of the path preview, when the path exceeds its range, the diagram can be displayed at the middle of the screen by CENTER (P) soft button (or P).

The current diagram path can be eliminated by CLEAR soft button (or R).

2) Drawing speed selection

Enter to the next soft function menu by MORE···; the drawing speed can be controlled by FULL SPEED, RAPID SPEED, COMMON or LOW SPEED (The corresponding shortcut buttons are 0, 1, 2 or 3); the speed can be overridden with real-time during the drawing.

3) Scaling up/down of path

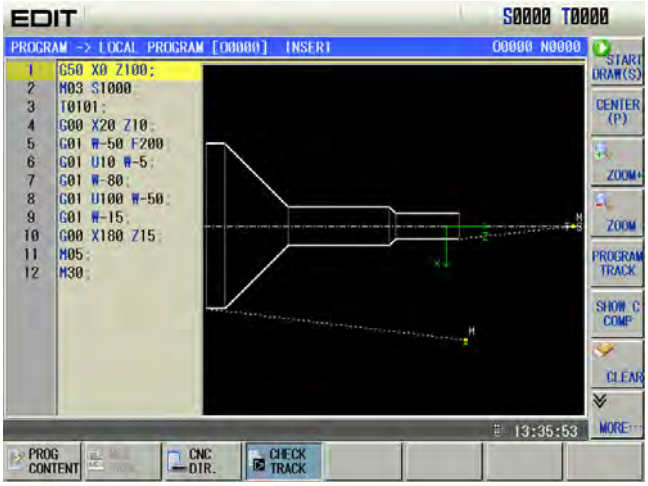
In the path preview page, the scaling up/down of the diagram path with real-time can be performed by ZOOM+ soft button (or I), ZOOM- soft button (or M). The diagram path can be zoomed in $\sqrt{2}$ times by the ZOOM+; the diagram path can be zoomed out $1/\sqrt{2}$ times by ZOOM-, as well the current scaling shows at its corresponding prompt column.

4) The displayed movement of diagram path

The diagram path movement can be carried out by cursor movement button in the path display page. The movement interval can be adjusted by MOVEDIS soft button (or J); the current movement interval is then displayed at its corresponding prompt column.

5) Mirror image path

In the path preview page, the programming path and symmetric line of the programming path (cutting command) will display by MIRROR TRACK; refer to the following figure:



The MIRROR TRACK will be cancelled by PROGRAM TRACK soft button, only the programming path is then drawn.

6) Display the cutter compensation C

If the program in the cutter compensation C, and press the SHOW C COMP soft button; the programming path and the cutter compensation C path will display, and the cutter compensation C shows with red, refer to the following figure:

The SHOW C COMP will be cancelled by CANCEL C COMP soft button; only the programming path is drawn.

6.10.3 Command Display of Path Preview

(Single dotted-line)：G00

(Double full-line)：G01/G02/G03/G05/G122/G124/G144/G146

(Thread line)：G32

(Symmetric line): Mirror image path symmetric axis

(Single full-line): The connection cable of the cutting path and symmetric cables in the mirror image path.

Explanations:

(1) G28, G30: This section will not show if there is no intermediate point to be specified; if the intermediate point is specified in command, the drawing will be performed to the intermediate point, and part of the figure where from intermediate point to the mechanical zero will not be drawn;

(2) G50, G52-59 are denoted with coordinate systems, color: Green;

(3) M, S and T commands are shown by Yellow dot; the M30, M99, M00, M01 and M02 are not displayed near the note M, S, T and M codes;

(4) G31/G36/G37 are treated as G01;

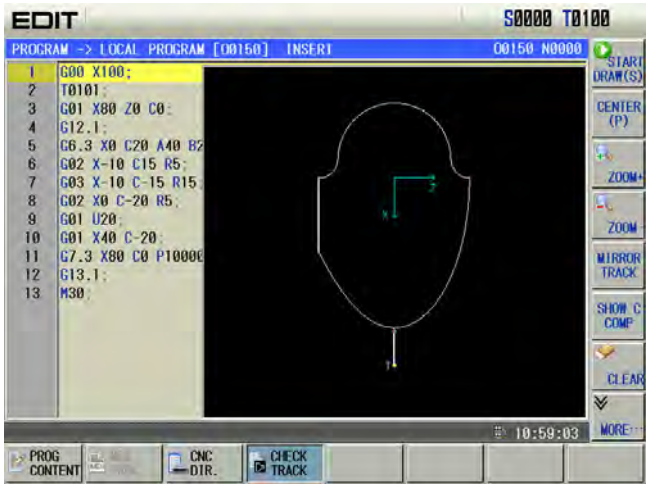
(5) The path at the G17, G19 plane can not be regarded as the path preview;

(6) The writ of the system variable in the macro program does not register to the cutter compensation file; the read for the system variable, is the state, before entering to the program preview;

(7) Enter to the polar coordinate, cylinder interpolation, the graph is shown be Red;

If the polar coordinate and cylinder command are programmed in the G17 and G19 planes, at last, the graph is shown in the X, Z planes。

The difference between the displayed graph and the actual one of the programmed path are 90 degrees (The coordinate system of drawing can be consistent with the one of the programming by adjusting the data parameter No. 175 and pressing the [SHIFT] soft button). The example program (G17 panel) of the polar coordinate interpolation in this Manual can be illustrated as follows:



If the polar coordinate and the cylinder command are programmed in the G18 plane, but the eventual graph displays in the X, Y plane。

The displayed graph and the programmed one are symmetric (The coordinate system and programming system based upon drawing can be consistent by adjusting the data parameter 175) to Z axis to describe the example program (G18 plane) in the cylinder interpolation; refer to the following figure:

6.10.4 Precautions

(1) The figure in the path preview is a schematic of the programming path, which is not absolutely equal that of actual workpiece;

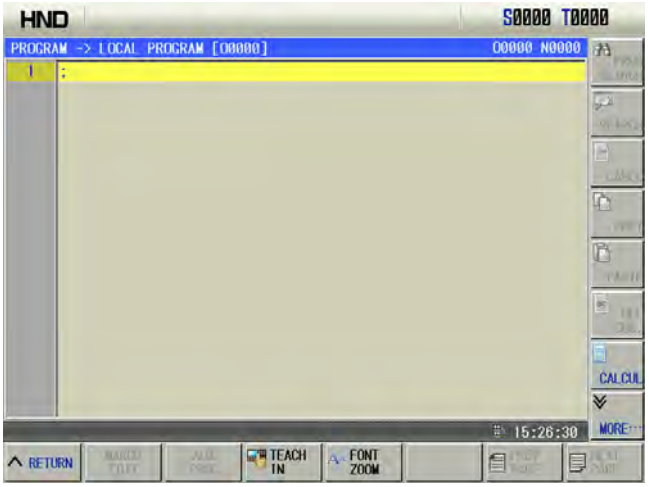
(2) Path preview is mainly inspected whether the programming path is correct, regardless of the correction of the miscellaneous commands M, S and T.

6.11 Teaching function

Teaching is a programming method by the artificial guiding machining operation, GSK980TDi adds a teaching function, so that the teaching programming is more simply and efficient.

Select the Manual or MPG operation method and press the PROGRAM to enter the program page set, and then enter to the program content page by pressing PROG CONTENT if necessary; the display is shown below:

Open the next optional menu by pressing the PROG CONTENT again, refer to the following figure:



Enter to the Teaching function page by TEACH IN, refer to the following figure:

INS X COORD、INS Z COORD: Insert the current absolute coordinate (original value input state) or the sum (offset input state) of the absolute coordinate and offset value input for the corresponding axis at the cursor position.

CHG TO OFFSET VAL.: Shift the system to the retyping offset value when the absolute coordinate value along each axis is inserted; simultaneously, it becomes CHG TO CURRENT VAL. after controlling this button.

CHG TO CURRENT VAL.: Shift the system to state that only the absolute coordinate along each axis can be inserted; simultaneously, it becomes CHG TO OFFSET VAL. after controlling this button.

EXIT: Retreat from Teaching mode.

6.12 Other Operations Available in Edit Mode

The operations by this GSK980TDi machine panel allowed in Edit mode are defined by ladders, please refer to the materials by the machine builder.

Please note that the following functions are described based on the 980TDi standard PLC programs!

1) Spindle override can be adjusted by pressing %+, %−;
2) Feedrate override can be adjusted by pressing %+, 100%, %−;
3) CNC can be reset by pressing key;
4) The operation mode is switched by pressing any of the keys among AUTO, MDI, MACHINE ZERO, MPG, MANUAL or PROGRAM ZERO;
5) The data transmission
6) Refer to II OPERATION, Chapter 12
7) Cooling control
   Refer to II Operation, Section 3.2.3
8) Lubricating control

Refer to II Operation, Section 3.2.4

9) Chuck control

Refer to II Operation, Section 3.2.5

10) Tailstock control

11) Refer to II Operation, Section 3.2.6

12) Hydraulic control

Refer to II Operation, Section 3.2.7

CHAPTER 7 TOOL OFFSET AND SETTING

The actual location of tool can be overlooked in programming for simplifying programming. Three methods including positioning tool-setting, trial tool-setting and machine zero tool-setting are available in this GSK980TDi system. The tool offset data are obtained from this tool-setting operation.

7.1 Tool Positioning Setting

Steps:

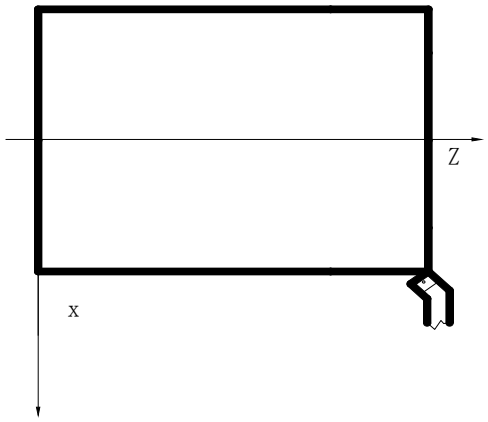


Fig. A

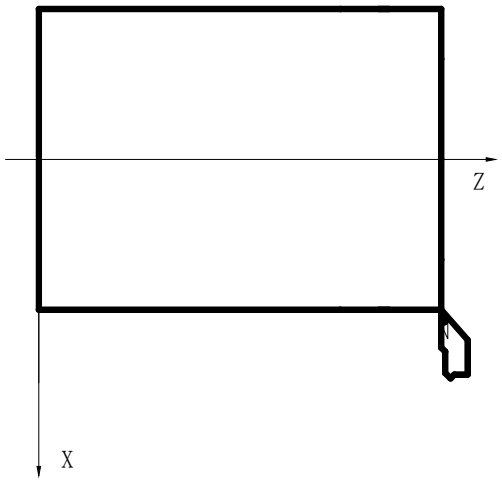


Fig. B

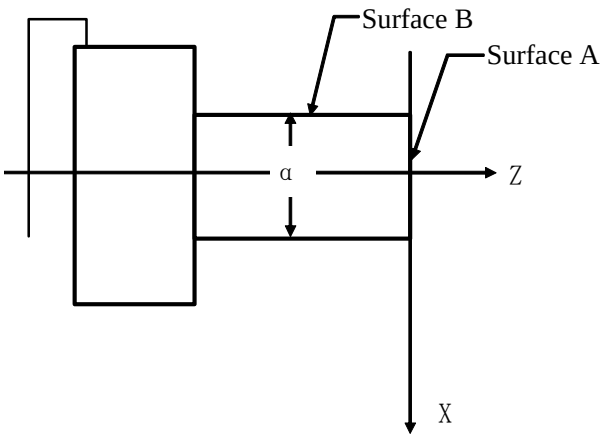
1. Firstly determine if the offset values are zero in X, Z, if not, clear all the tool number offset values;
2. Set the offset No. for 00 (i.e. T0100,T0300), as for the offset value: (method: execute a move code or perform the machine zero return in T0100 state, then clear the offset value automatically 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 (tool-setting point), as shown in Fig. 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 Fig. B;
8. Press OFFSET key and move the cursor by ↓, ↑ key to select the corresponding offset number of that tool;
9. Press address key U, then press INPUT key to input the tool offset value of X axis into the corresponding offset number;
10. Press address key W, then press INPUT 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 tool-setting operation for other tools.

Note: For the fixed tool-setting, the original system tool offset should be cleared, input the new offset one time instead of many times by pressing U, W. Refer to II OPERATION, Section 7.5.4 about the tool compensation clearing method.

7.2 Trial Tool-setting

Whether the method of trial tool-setting is inactive is defined by the system parameter No.012 Bit5.

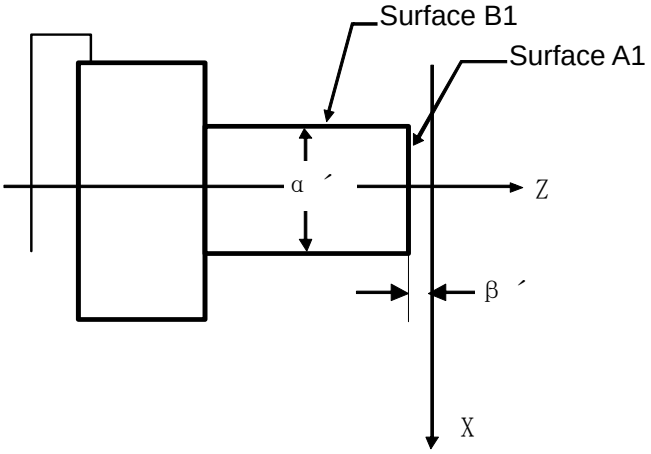
Steps (workpiece coordinate system by part end surface):



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; directly press RECORD X POS. and RECORD Z POS., and the CNC records the absolute values of the position, at the moment, directly move the tool;

Press OFFSET key to enter the Offset interface, select the TOOL OFFSET page, then move

3) the cursor by pressing ↑, ↓ key to select the corresponding offset number;
4) Key in by sequence the address key Z, number key 0 and INPUT 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; directly press RECORD X POS. and RECORD Z POS., and the CNC records the absolute values of the position, at the moment, directly move the tool;
7) Measure the diameter "α" (supposing α=15) ;
8) Press OFFSET 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 in the address key X by sequence, number key 1, 5 and INPUT key;
10) Move the tool to a safety height to change for another tool;



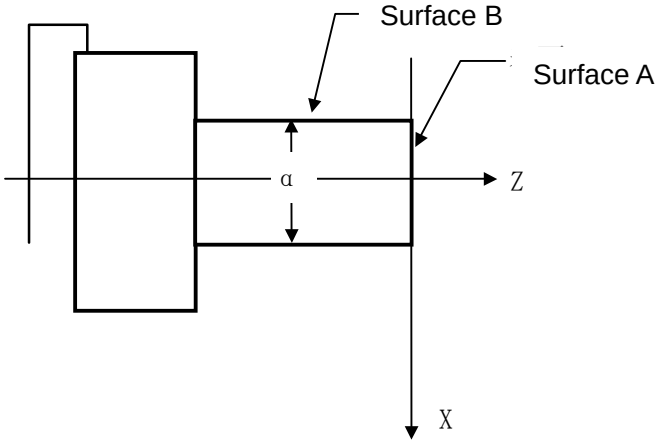
11) Make the tool to cut on Surface A1;
12) Retract the tool along X axis without Z axis moving and stop the spindle;
13) Measure the distance "β´" between the Surface A1 and the workpiece coordinate origin(supposing β´=1) ;
14) Press key to enter the Offset interface, select the TOOL OFFSET page, then move the cursor by pressing , to select the corresponding offset number;
15) Key in by sequence the address key Z, sign key, number key 1, and 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 α'=10);
19) Press 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, 0 and key;
21) Repeat the execution from Step 10 to Step 20 to perform the tool-setting operation for other tools.

Note: The offset value may be large by this tool-setting 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.

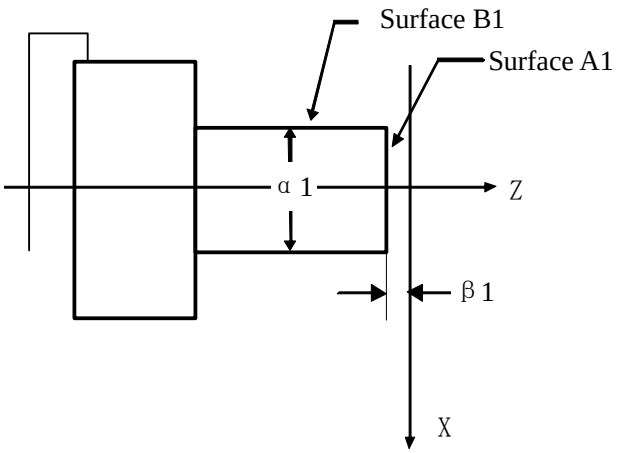
7.3 Tool-Setting by Machine Zero Return

There is no reference tool in this tool-setting 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 tool-setting. 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 RECORD X POS. and RECORD Z POS., 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 Z, number key 0 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 α=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 X, number key 1, 5 and key to set 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 "β1" between the Surface A1 and the workpiece coordinate system origin (supposing β1=1) ;
16) Press 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, sign key, number key 1, and key to set Z offset value;
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 "α1" (supposing α1=10);
21) 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;
22) Key in by sequence the address key X, number key 1, 0 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 tool-setting operation for other tools.

Note 1: Machine zero switch must be fixed for the tool-setting operation by machine zero return.

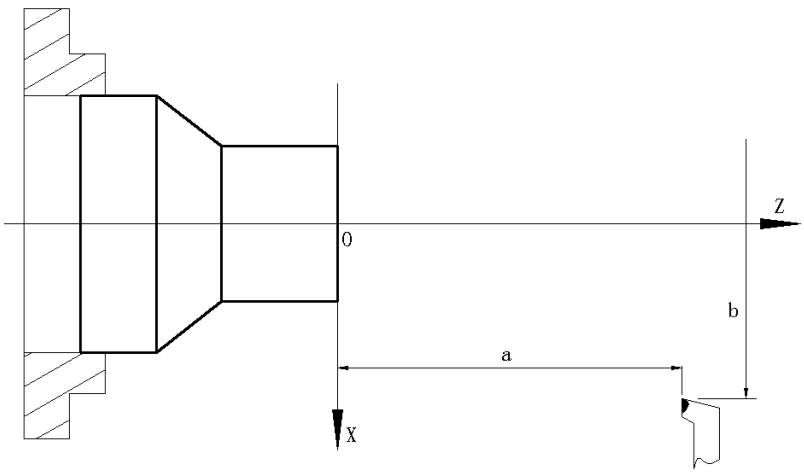
Note 2: The workpiece coordinate system setting can't be done by G50 code after tool-setting by machine zero return.

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 tool length compensation.

Note 4: The corresponding parameters should be set as follows:
Bit7=0 of the system parameter No.004;
Bit5=1 of the system parameter No.012;
Bit7=1 of the system parameter No.012.

Note 5: The setting values of the system parameter No.047 should be close to the absolute coordinate values of machine zero in workpiece coordinate system XOZ, as is shown in the following figure:

As shown below:



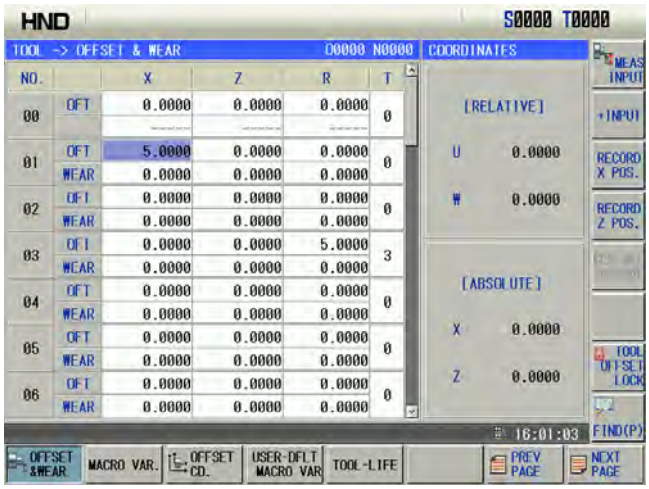
Example: After machine zero returning, when the absolute coordinate of the tool in workpiece coordinate system is (a,b), the setting value of system parameter No.047 should be close to a,b.

7.4 Coordinates Record

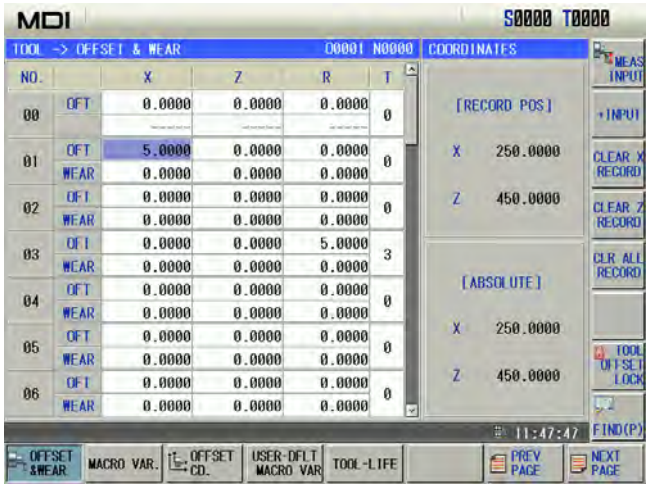
When the trail-cut tool-setting or machine zero return tool-setting is executed, the coordinates record function can be used to get the convenient tool-setting operation.

When the tool-setting is executed and after the tool cuts along X (or Z), the following steps are executed to directly move the tool to the safety position as follows:

1) Press [OFFSET] to enter the tool offset interface, and then press [OFFSET &WEAR] to enter the tool offset page as follows:



2) Press [RECORD X POS.] and [RECORD Z POS.], absolute corrdinates of X and Z（without including current tool offset values）are recorded in the CNC. After the absolute values are recorded correctly, there is a flash “Record coordinates” in the upper of the screen, and there is a prompt “Current X、Z Absolute coordinates have been recorded” in the bottom of the screen as follows:



3) Retract the tool to the safety position, and stop the spindle. When the diameter is measured for the outer circle and the distance from the reference plane to it is done, the measured

value is input in the tool offset page; the system counts the new tool offset value and sets it to the selected tool offset number according to the input value and the previous recorded absolute coordinates.

Use the following methods to clear the recorded coordinates when the above methods are used to execute the tool-setting:

1) In the tool offset page, after the absolute coordinates are input in some tool offset(for example: Z0 is input in the No.01 tool offset number) and then the axis moves, the recorded coordinates are cleared;
2) Press CLEAR X RECORD or CLEAR Z RECORD or CLR ALL RECORD in the tool offset page and the recorded coordinates are cleared.
3)

7.5 Setting & Altering an Offset Value

Refer to Section 1.3.3 about methods of entering the offset setting page.

7.5.1 Offset Setting

1) Press OFFSET key to enter the offset setting page, select 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 keys to directly move the cursor to the input position:

Press P + offset number + INPUT or FIND(P) and input the offset number in the pop-up dialogue box, press INPUT and the cursor directly positions the target offset number.

3) Numerical value input.

Method 1: Input number after pressing the address button (the corresponding axis name button for the X, Z, etc.);

Method 2: Input number after pressing the MEAS INPUT soft button;

3) Press INPUT and the CNC calculates the offset value automatically and displays the result in the window. Its range:-9999.999~9999.999（IS_B）/-9999.9999~9999.9999（IS_C）(unit：mm)

7.5.2 Offset Alteration

1) Refer to OPERATION, Section 7.5.1, 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; alternatively, select the data position to be changed by +INPUT soft button;
3) Then key in 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 keyed in is U 1.5
Then the new offset value is 7.178 (=5.678+1.5)

7.5.3 Offset Alteration in Communication Mode

For the alteration and setting of the offset value in the communication, please see Chapter 12 of this part for its operations.

Note 1: While changing the offset value, the new offset value takes effect after the T code is executed.

Note 2: If the actual workpiece dimensions doesn't conforms to that of the part drawing, subtract the error from the original offset value for the oversize workpiece, add the error to the original offset value for the undersize workpiece.

Example: The external diameter of the workpiece to be machined is Φ55.382, and the No.01 offset value is applied in the machining. Before machining, the values in No.01 are shown as follows table:

No.	X	Z	T	R
00	0	0	0	0
01	16.380	-24.562	0	0

In machining, the actual external diameter measured of workpiece is Φ55.561, so the offset value of No.01 can be altered as follows table:

No.	X	Z	T	R
00	0	0	0	0
01	16.201	-24.562	0	0

16.380- (55.561-55.382)

Note: To backup and restore the tool offset values, the relevant operations may be performed on PC via communication.

7.5.4 Offset Values Clearing

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, which is shown as follows:

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.

7.5.5 Setting & Altering a 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 GSK980TDi 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 input value. The wearing value resolution is -9999.999~9999.999 (IS_B) /-9999.9999~9999.9999 (IS_C) (unit：mm); The upmost incremental value of each input value can be restricted to prevent the incorrect numerical value from inputting. The wear input range is defined by the data parameter No.140, and they are saved even at power down.

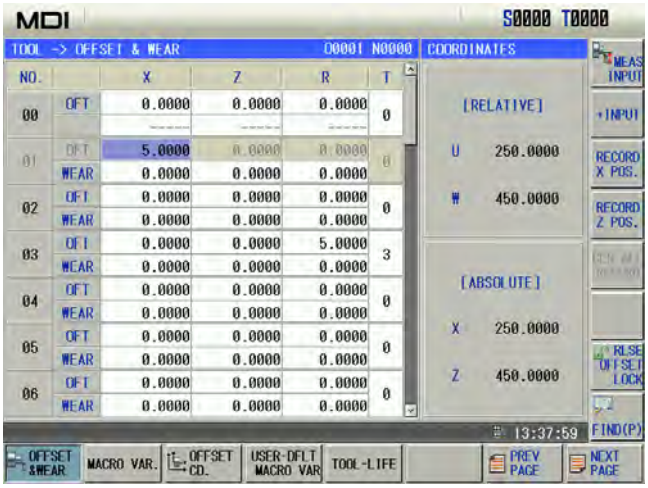
The setting and alteration methods for the tool wear are approximately identical to that of the tool offset, and the wear value is input by U(X axis), W(Z axis), V(Y axis).

7.5.6 Locking & Unlocking an Offset Value

In order to protect the offset value to be operated by mistaken, the offset values can be put into lock, the operation steps are as follows:

1) Move the cursor to the location of the offset number to be locked;

2) Press the CHANGE or TOOL OFFSET LOCK key, the current offset value will be backlighted for locking, which is forbidden to alter, press the CHANGE or RLSE OFFSET LOCK key again, the locking may be cancelled. No.1 tool offset is locked as follows:

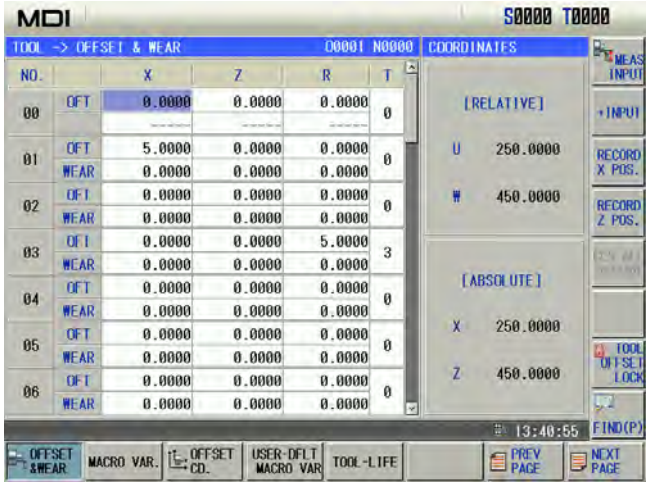


Note: The tool wear values can't be locked.

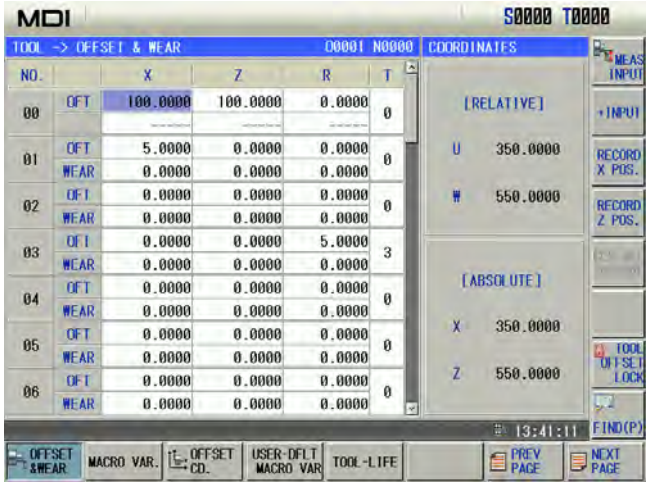
7.5.7 No.0 Tool Offset Moving Workpiece Coordinate System

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.

7.6 High Efficiency Tool-Change

7.6.1 Breif

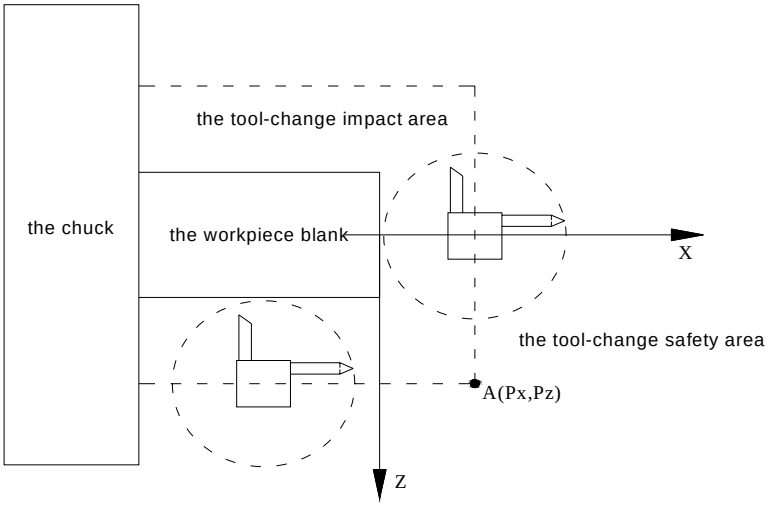
When the tool operation is performed during the program machining; usually, the steps are shown below:

1) Return to the specified safety tool-change position by program from the current position;
2) Perform the tool-change operation and waiting for the completion of the tool-change;
3) Return to the cutting position from safety tool-change position;

The tool-change safety area is set by machine operator before the machining is performed (that is, the area, without any impact to the workpiece, when the tool-post rotates a circle); Tool returns to the safety tool-change position specified by program from the current position. Tool is immediately performed the tool-change operation as long as it enters to the tool-change safety area when the tool-change operation is performed; that is, the step 1 and 2 in the above-mentioned combines a completion motion, so that it can be reached to the high-efficiency tool-change. It is better to set the Bit 7 of the state parameter to "1".

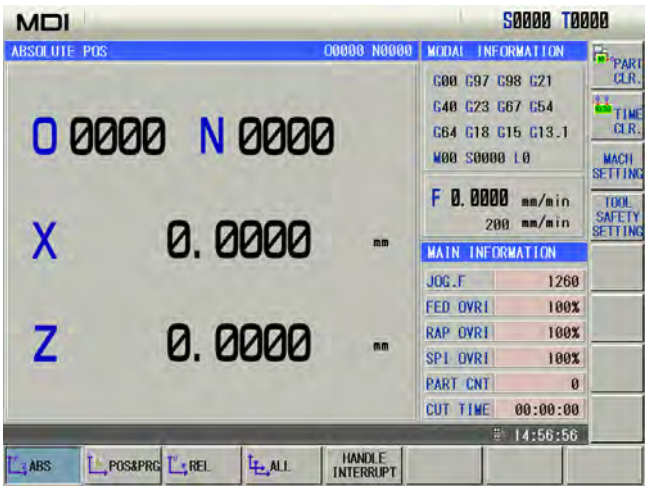
7.6.2 Tool-Change Safety Area Setting

When the desired tools for program machining are totally installed, in the case that the spindle clamps the workpiece blank, the area, without any impact for one cycle of the tool-post rotation, is called as the tool-change safety area; refer to the following figure:

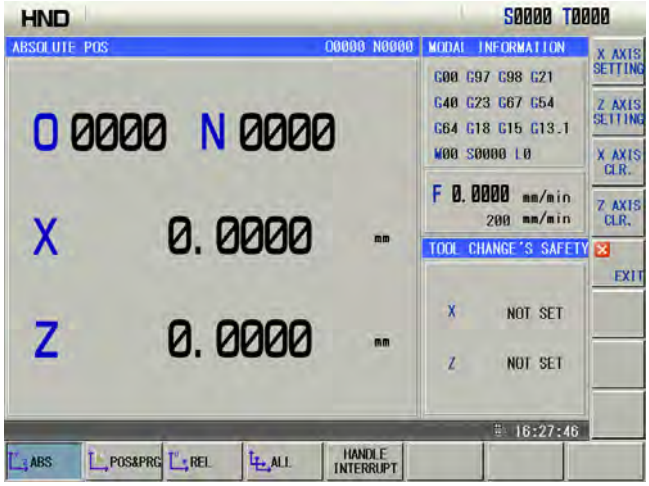


The coordinate of point A in the figure is (Px, Pz), the tool-change safety area is X > Px or Z > Pz; the tool will not collide with workpiece as long as the tool-change is performed in this safety area. To carry out the high-efficiency tool-change in the actual machining, the machine operator should be performed within the safety area setting based upon the following steps:

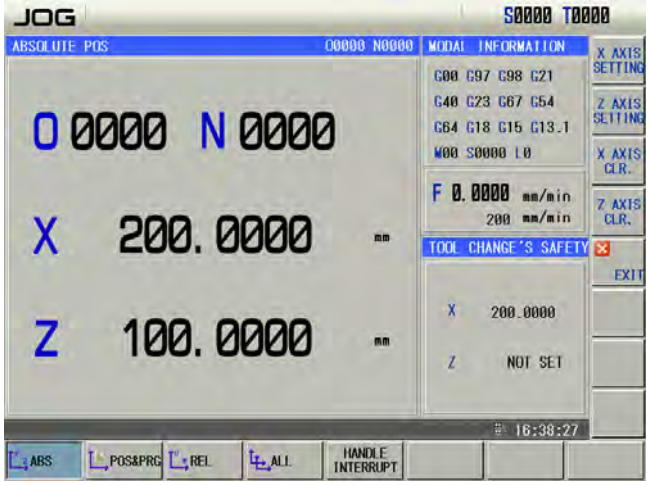
1) In the absolute coordinate page, press TOOL SAFETY SETTING soft key, refer to the following figure:



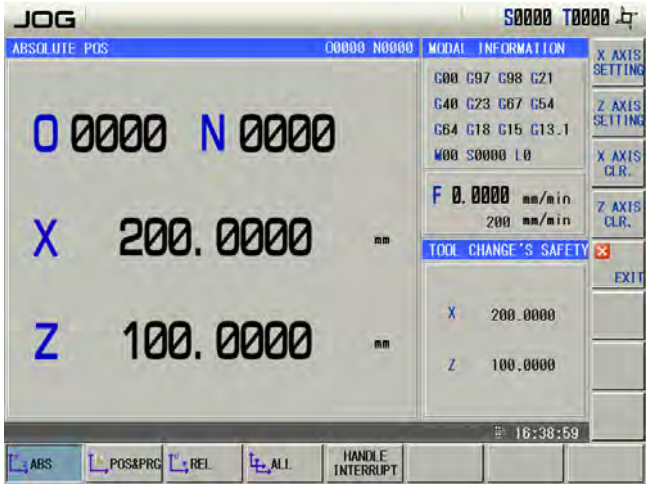
2) Press the TOOL SAFETY SETTING soft key, the tool-change safety point setting information window is shown and covered the integrated information window. It will show a unset state when X or Z axis does not set; it will show the coordinate value of setting when X or Z is set. The initial state is as follows:



3) Manually move the tool-post at the side of + X of the workpiece, and the optimized position of the impact will not occur when the tool-change executes at this side; record the X axis coordinate at this position by pressing the X AXIS SETTING; refer to the following figure:



4) Manually move the tool-post at the side of + Z on the surface of the workpiece, and the optimized position of the impact will not occur when the tool-change executes at this side; record the Z axis coordinate at this position by pressing the Z AXIS SETTING; refer to the following figure:



In the above-mentioned interface, shift the interface or press the Z AXIS SETTING, automatically conceal the small window of “Tool-change safety point setting”, this area recovers and displays the small

window of the “Integral information”

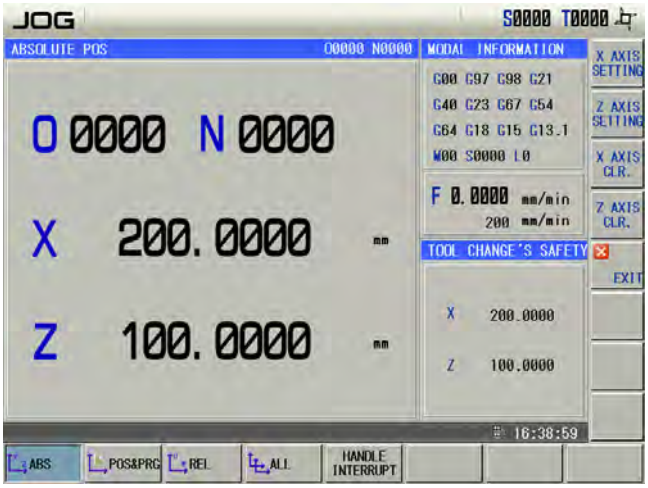
The high-efficiency tool-change function can be enabled after the X and Z are set; the setting value at the tool-change safety area is memorized after the power is turned off. After the high efficiency tool-change is enabled, the tool mark is then shown at the right corner of the page “ ”.

Note: The record in the setting of the tool-change safety point is the machine coordinate.

7.6.3 Setting Clear in Tool-Change Safety Area

The setting value of the set tool-change safety area can be eliminated by the following methods:

1) The set data along with the X or Z can be eliminated by soft button X AXIS CLR. or Z AXIS CLR. after the tool-change safety point setting window is shown;



2) The setting data set by X or Z axis on the tool-change safety area will be automatically eliminated after the G50 command sets the coordinate system;

3) The setting data set by X or Z axis on the tool-change safety area will be automatically eliminated after the machine zero return is operated;

4) On the tool offset page, if the tool-setting operation is performed by the measure input method or trial cutting tool-setting method; the current tool is regarded as to be changed; the setting data set by its corresponding data on the tool-change safety area will be automatically eliminated.

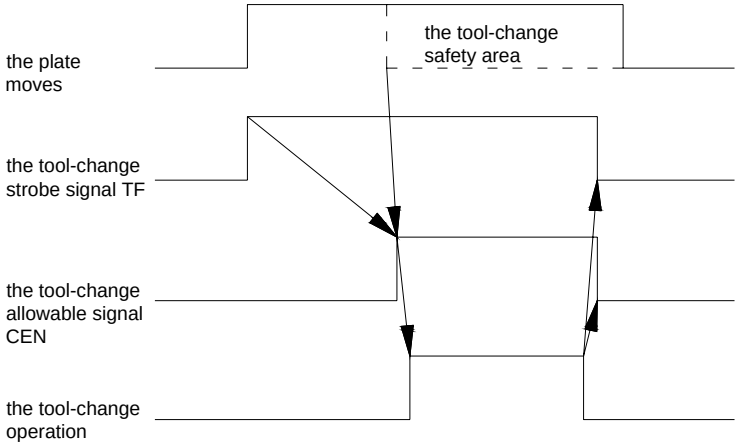
7.6.4 High-Efficiency Too-Change Programming Format

In order to realize the high-efficiency tool-change, user should use the shared programming format both the positioning G code and Tool-change T code when the machining procedure is programmed.

G00 X_ Z_ T_ or

G01 X_ Z_ T_

T code is performed sharing a same block with G00 or G01 code, simultaneously, the T code is performed while the positioning code executes; and then deliver the tool-change strobe signal TF to PLC. When the plate moves to the tool-change safety area, CNC delivers the tool-change allowable signal TCEN. PLC program triggers the tool-change operation by identifying the tool strobe signal TF and tool-change allowable signal TCEN. Refer to the following signal output time of high-efficiency tool-change:



When T code does not share a same block with the movement code, simultaneously output tool strobe signal TF and tool-change allowable signal TCEN when T code is performed.

7.6.5 High-Efficiency Tool-Change Procedure Explanation

Firstly, set the tool-change safety area before using this function. If the tool is installed again or the raw-material dimension is changed during using this function, the safety area should be set again.

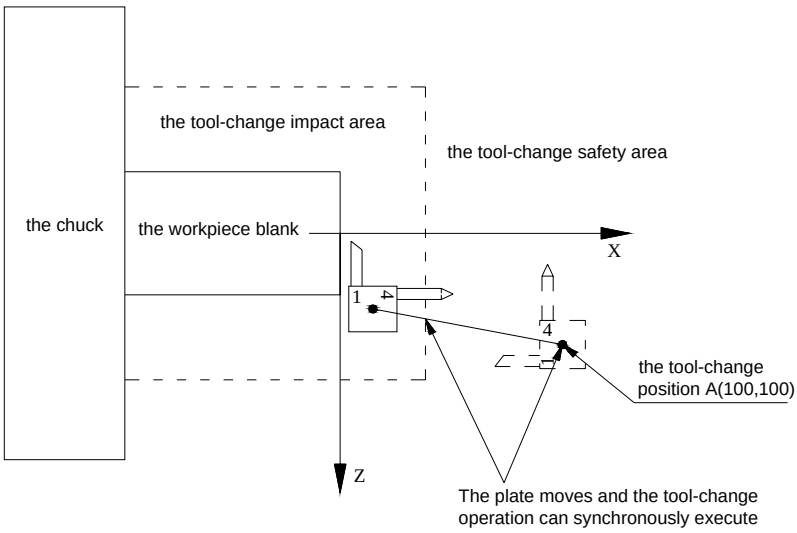
For example:

…

G00 X100 Z100 T0404;

…

The tool-change motion and carriage plate positioning are performed as follows:



7.6.6 Precaution of High-Efficiency Tool-Change

1) When the tool-change code does not performed with a same block to the axis positioning code; the tool-change allowable signal will be output before performing the tool-change;
2) When the tool-change code and axis positioning code (G00 or G01) are performed based upon a same block; the tool-change allowable signal TCEN output after the carriage plate enters to the safety area. When the tool-change code and other movement codes are performed with a same block; the tool-change allowable signal TCEN outputs after the movement ends;

3) When the high-efficiency tool-change is performed, It is necessary to prevent the tool rotation from impacting to other machine components during that the plate moves to the tool-change point within the tool-change safety area;；
4) High-efficiency tool-change function is suitable for the tool-change safety detection within the XZ panel, if additional axes are placed at the machine tool, it is necessary to confirm that tool-change will not interfere to other additional axes when programming.

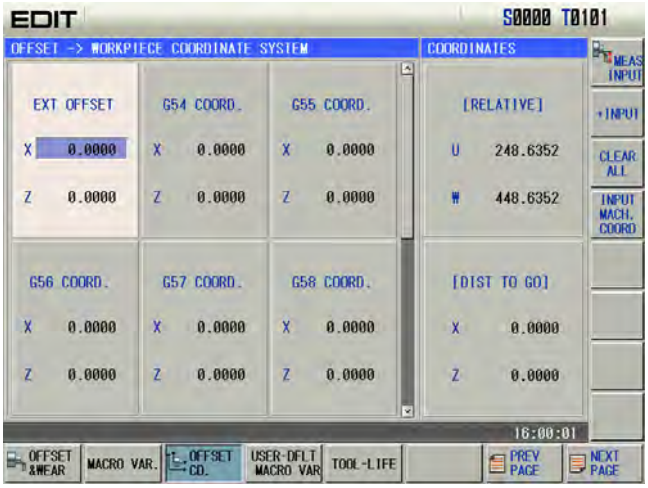
Note: The high-efficiency tool-change function will disable after performing G50 command or mechanical zero point return, as well the tool-setting operation in measure input method is performed; the high-efficiency tool-change function can be enabled after resetting the tool-change safety area operation.

CHAPTER 8 COORDINATE SYSTEM

GSK980TDi owns workpiece coordinate system selection function, which can be contented several workpiece coordinate system during the machining; specifying one of the G54~G59 code in the block can be selected in the workpiece coordinate systems 1~6.

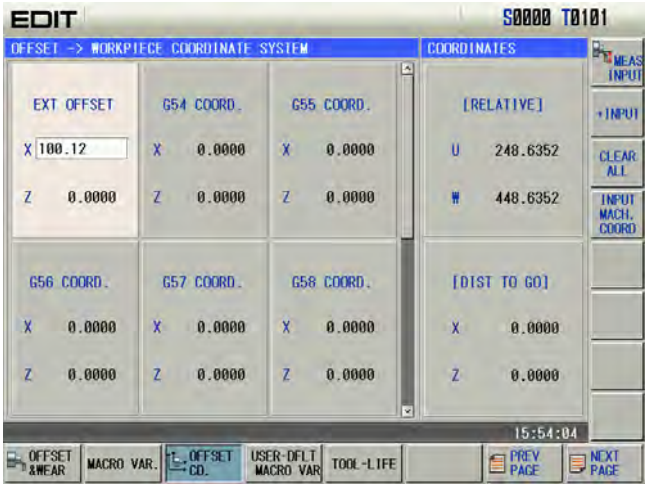
8.1 Coordinate System Setting

Press OFFSET, OFFSET CD. to enter its corresponding page:



8.1.1 Directly Input

Enter to the workpiece coordinate system page, press ↑、↓ Move the cursor to the inputted position, input the numerical value by the number buttons, and then directly input the numerical value by the button.

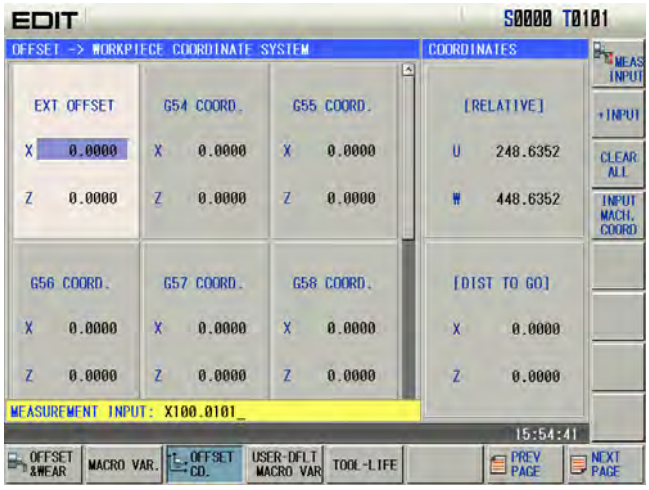


8.1.2 Measure Input

CNC automatically calculates the data such as the cutter compensation or zero offset, etc. when using the measure input, so that the input data becomes the absolute coordinate of the new coordinate system at the machine's current position.

Method 1: Press ↑ or ↓ Move the cursor to the inputted position, input the coordinate value after pressing the MEAS INPUT soft button, the CNC will automatically calculate to input the data by controlling the button;

Method 2: Move the cursor to the selected coordinate system by pressing ↑ or ↓, key the coordinate value by address button X or Z; CNC automatically calculates and then inputs the data by ;



Note: The input data from measure will become the absolute coordinate of which the current point locates at the setting coordinate system, for example:

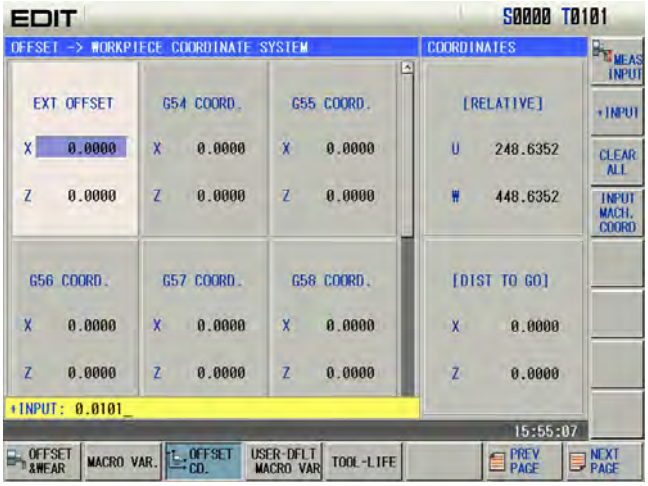
System executes G54 command, input X100; Z100 in G54 coordinate system by measure input method. CNC automatically calculates the G54 offset value, so that the absolute coordinate at the current position will become X100; Z100.

8.1.3 Machine Tool Coordinate Record

Enter to the workpiece coordinate system setting page, move the cursor to the inputted position by ↑ or ↓; directly input the machine tool coordinate by INPUT MACH. COORD soft button.

8.2 Coordinate System Modification

Enter to the workpiece coordinate system setting page, press ↑、↓,move the cursor to the data position to be modified; input the incremental value by the +INPUT soft button +INPUT, press , add the current workpiece coordinate system offset value and the inputted incremental value, and its result will show as a new workpiece coordinate system offset value.



8.3 Coordinate System Clearing

Enter to the workpiece coordinate system setting page; the overall workpiece coordinate offset values will clear after the confirmation by pressing the CLEAR ALL soft button; alternatively, Use the directly input method of the workpiece coordinate system in this chapter to set the axis offset value of any coordinate system as "0".

CHAPTER 9 AUTO OPERATION

Note!

The key functions of GSK980TDi machine panel are defined by PLC program (ladders), please refer to the materials by the machine builder for their significance.

Please note that the following description for the keys function in this chapter is based on GSK980TDi standard PLC program!

9.1 Automatic Run

9.1.1 Selection of a Program Running

Refer to Section 6.3.

9.1.2 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.

9.1.3 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 button

If the external emergency button (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 button is released, the alarm is cancelled and CNC system enters 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 Auto mode, 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 button before power on and off.

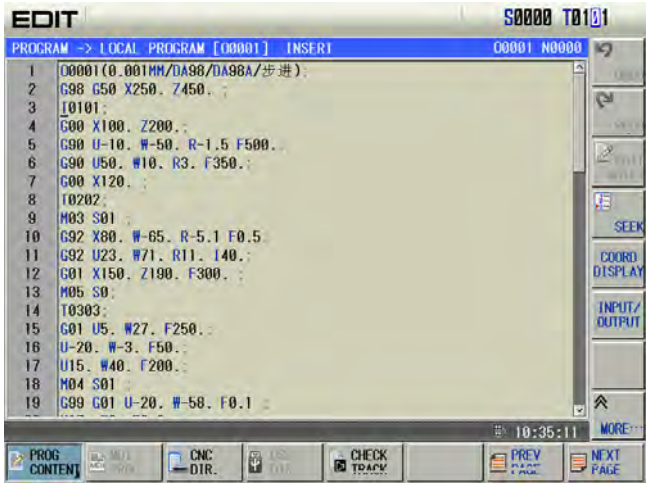
Note 3: The Machine zero return operation should be performed again after the emergency alarm is cancelled to ensure the correctness of the position coordinates (but this operation is forbidden if there is no machine zero in the machine).

Note 4: Only the BIT3 (MESP) of the bit parameter No.172 is set to 0, could the external emergency stop be active.

9.1.4 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.

9.1.5 Adjustment of 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 , , , .

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.022Z×rapid override
Note 2: When the rapid override is F0, the min. rapid traverse rate is set by bit parameter No.032.

9.1.6 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.

9.2 Running State

9.2.1 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.

9.2.2 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 GSK980TDi is defined to be in auto running state (Auto, MDI mode).

9.2.3 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 the 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.
3. Press the on the system operation panel to close the machine lock; the system is then automatically recovered the absolute coordinate value along each axis; that is, the coordinate system automatically restores.

Precautions:

1) The machine lock open/close system automatically restores the absolute/relative coordinate along each axis;
2) The tool state is regardless of the automatic recovery of the workpiece coordinate system before/after the machine lock. For example, the tool state before the machine lock is T0101; after is T0202;
3) The state of the cutter compensation C is regardless of the automatic recovery of the workpiece coordinate system before/after the machine lock. For example, the state of the cutter compensation C before the machine lock is G40; after is G42;
4) The system ON-OFF state is regardless of the automatic recovery of the workpiece coordinate before/after the machine lock. For example, the machine is locked before the power-off; the system can be automatically restored the workpiece coordinate system after the power is turned on again.

5) To avoid the tool interval motion during the automatic operation, it is better not to change the state of the machine tool after starting the program;

9.2.4 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.

9.2.5 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 (panel key active or external skip input active), this block is skipped without execution in Auto mode. In Auto mode, press to enter the state that the program skip is valid.

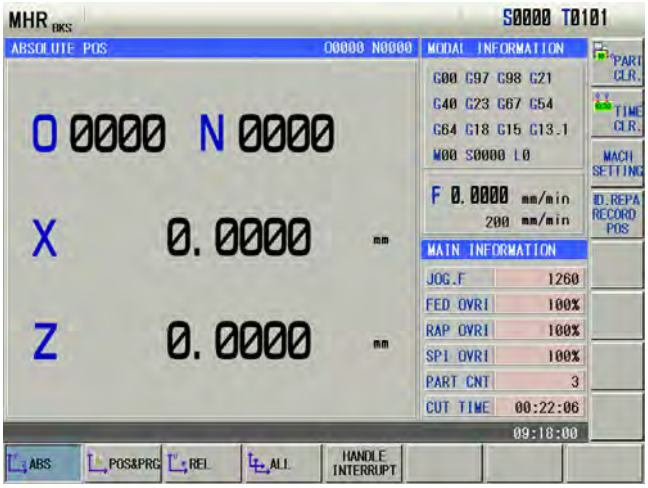
Note: While the block skip switch is off, the blocks headed with "/" signs are executed normally in Auto mode.

9.3 MPG Trial-Cut

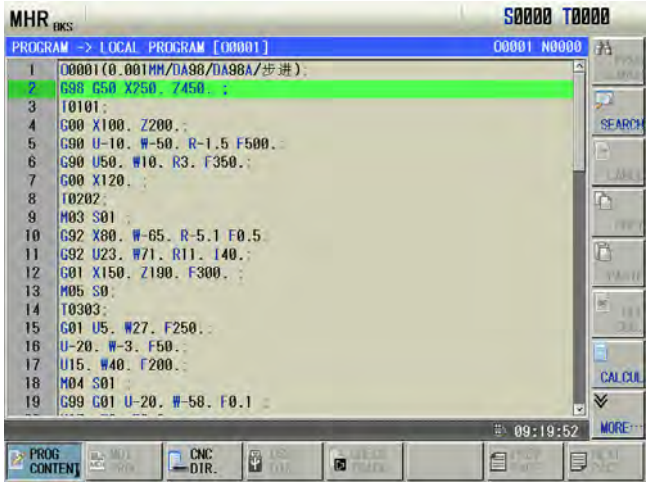
The user can use MPG trial-cut function after editing part programs, check the run path of program. In MPG trial-cut function, rotating MPG controls the execution speed of program, which can easily check the program error conveniently.

9.3.1 Switching MPG Trial-cut Mode

Press to enter the MPG trial-cut mode after a machinig program is selected as follows:



At the moment, press and the display is shown below:



At the moment, rotating MPG makes the program start running. When the execution speed of program is proportional to MPG speed, execution speed of program fastens as soon as the MPG rapidly rotates; the execution speed slows down as soon as the MPG slowly rotates. Movement amount of one pulse can be adjusted by rapid override.

When the system is in the MPG trial-cut mode, it returns to Auto mode after is pressed. All operations in MPG trial-cut mode are the same those of Auto mode.

9.3.2 Command Speed in MPG Trial-Cut Mode

1) Cutting feedrate

Cutting command speed in MPG trial-cut mode is determined by No. 241 and rapid override. The command speed during actually rotating MPG is executed by the following:

[Command speed]×[MPG pulse amount per 1 second]×λ×([the parameter setting value] /100) × (8/1000) (mm/min or inch/min)。

λ value is determined by the current rapid override, and their relationship is:

Rapid override	λ value	Rapid override	λ value
F0	1	25%	10
50%	100	100%	100

Rapid traverse speed is clamped at 100% speed when it exceeds 100%, i.e.:

[MPG pulse amount per 1 second]×λ×([No.241setting value]/100)×(8/1000)≤1

2) Rapid traverse speed

Rapid traverse speed is clamped at the speed set by No.240, i.e.:

[Rapid traverse speed]＝[rapid traverse speed of each axis]×([No.240 setting value]/100)× [MPG pulse amount per 1 second]×λ×([No.241 setting value]/100)×(8/1000)

At the moment, when the speed by rapidly rotating MPG exceeds the clamped, the rapid traverse speed is clamped at the setting value by parameter, i.e.:

[MPG pulse amount per 1 second]×λ×([No.241 setting value]/100)×(8/1000) ≤1

9.3.3 Notes in MPG Trial-Cut Mode

Pay more attention to the followings when the system is in MPG trial-cut mode:

1) Press in Auto mode and the CNC does not switch to MPG trial-cut mode; when the CNC is in MPG trial-cut mode, it escapes the MPG trial-cut mode from the next block by

pressing the mode switch key;

2) When the system executes the MPG trial-cut control, the single block signal and the feed hold signal are valid. When the single block or feed hold stops, the program execution state is recovered to the MPG trial-cut control after the cycle start key is pressed.
3) Execution speed of blocks for movement and pause can be controlled by rotating MPG. Their speed of blocks with M, S, T, F, i.e. without movement or pause cannot be controlled(except for executing the tool offset in traverse mode) by rotating MPG which only controls whether they are executed.
4) The spindle speed is not related to the MPG pulse. The spindle rotates with the commanded speed even if the system is in the MPG trial-cut mode. For feed per rotation, reading the current spindle speed is switched to the execution after feed per minute.

9.3.4 Temporarily Invalid in MPG Trial-cut Mode

1) Executing screw cutting commands

In the course of executing screw cutting blocks (G32, G32.1, G33, G34, G76, G84, G88, G92，G92.1), the MPG trial-cut function is temporarily invalid with the speed under the override 100%, and is valid in the next block again. The MPG control is invalid when the screw cutting is executed actually, and it is valid in others.

2) Executing from the middle point to machine zero

In the course of executing the block (G28, G30) from the middle point to machine, the MPG trial cutting function is enabled from the start to intermediate point, and the MPG trial cutting is temporarily disabled from intermediate point to mechanical zero. The block is executed with the speed under the override 100%, and the next block is valid again.

3) Measuring related G commands execution with the override 100%

When the system measures the related G commands, the MPG trial-cut is invalid temporarily, is executed with the speed under the override 100%, at the moment, execution of the next block is valid again. The related G commands include:

G31, G36, G37

9.4 MPG Interrupted Function

In the Auto (Auto, MDI) and Edit mode, In the Auto (Auto, MDI) and Edit mode, the movement value from MPG can be overlapped to the Auto movement by rotating the MPG. The axis performed the MPG interruption can be selected by the MPG interruption axis selection signal. The movement distance of the MPG interruption is determined by the rotation value of the MPG and hand-wheel federate.

Note : It is necessary to set the Bit 1 of parameter No.0012 to "1" when using the MPG interruption function, it is necessary to support by dedicated PLC program.

9.4.1 MPG Interrupted Operation

In the Auto, MDI and Edit mode, enter to the MPG interruption state to perform the MPG based upon the MPG axle number on the selection panel after controlling the [MPG interruption switch] (External switch) once; retreat from the MPG interruption by pressing the [MPG interruption switch] again.

Perform the axle selection by , , , or button when entering to the MPG state. When the axle selection signal of MPG interruption of one axis is “1”, then execute the MPG interruption to this axis by rotating the MPG. The equivalent is identical between the MPG interruption and MPG feed.

The velocity during the MPG interruption is the sum both in the Auto operation and MPG interruption movement, and its overlapping speed is controlled within the upper limit speed of the cutting feed. For example, cutting upper limit is 8000mm/min, if the current feedrate is 2000mm/min, the MPG interruption speed range is -10000~6000mm/min (“-“ means that the MPG interruption movement direction is reverse to the feed one).

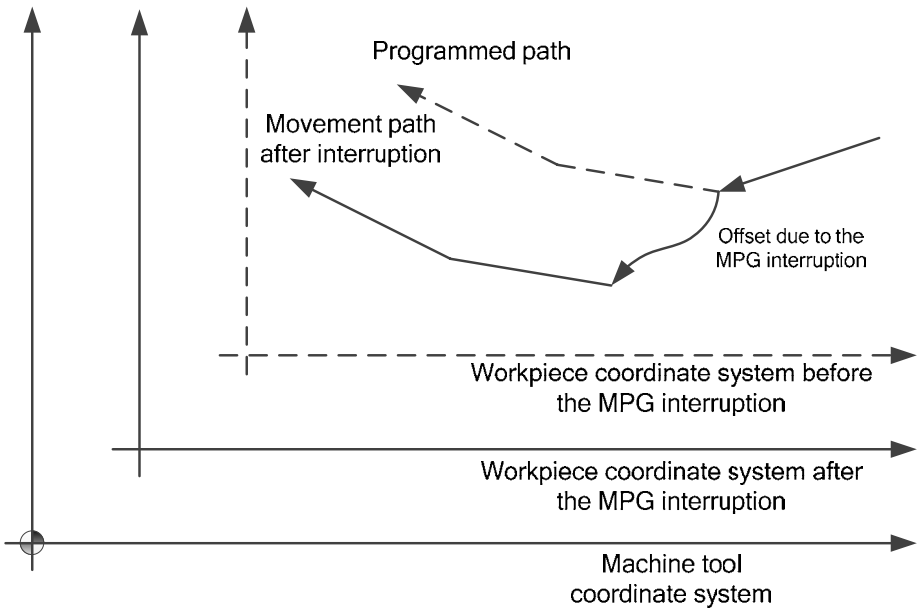
Related parameter

K0019.4	MPG interruption function 0: Disabled 1: Enabled
K0019.5	MPG interruption operation 0: Disabled 1: Enabled

9.4.2 Affection of Coordinate System from MPG Interruption

Axis moves by MPG interruption, and the actual position of the movement axis changes, the machine coordinate is renewed instead of the absolute coordinate. Therefore, machine moves after the MPG interrupts, and the machine coordinate system holds, but the workpiece coordinate system will offset.

- Absolute coordinate → Absolute coordinate value will not change due to the MPG interruption.
- Relative coordinate → Due to the MPG interruption, Interrupted amount change.
- Mechanical coordinate → Due to the MPG interruption, interrupted amount change.



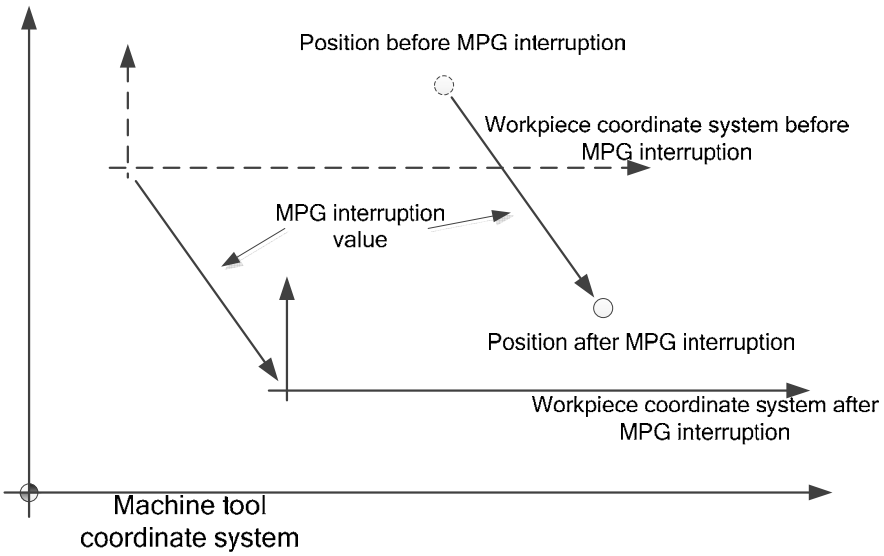
When automatically returning the reference point (G28), although the end (reference point) does not affect by MPG interruption, the intermediate point may generate the offset of interruption value by the MPG interruption at the workpiece coordinate system

9.4.3 MPG Interruption Value Cancel

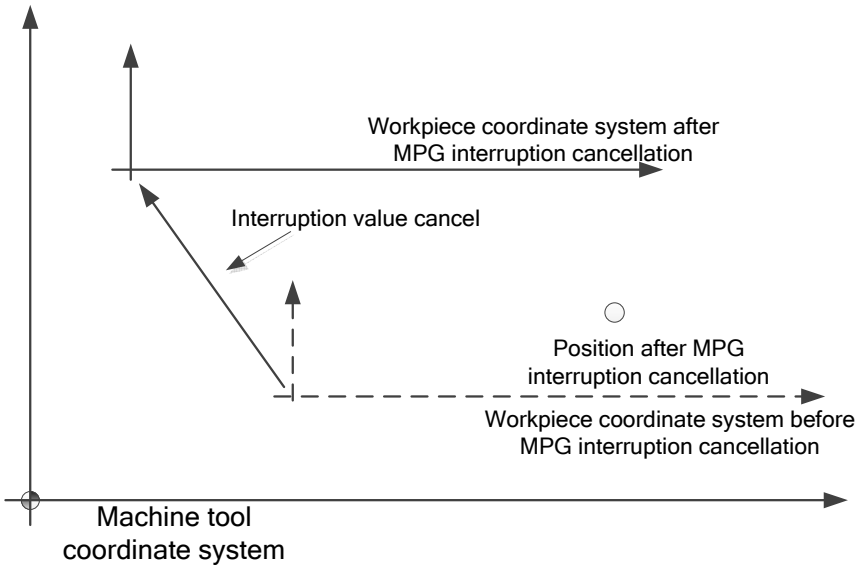
The cancellation of the MPG interruption value can be returned the workpiece coordinate system after offset to the one before offset.

When the cancellation operation of the interruption value is performed, the MPG interruption value reflects to the absolute coordinate value.

The workpiece coordinate system offset generated from MPG interruption is shown below:



The workpiece coordinate system recovers to the state before interruption after the MPG interruption is cancelled, however, the actual position is invariable, refer to the following figure:



The interruption value can be cancelled base upon the following situations:

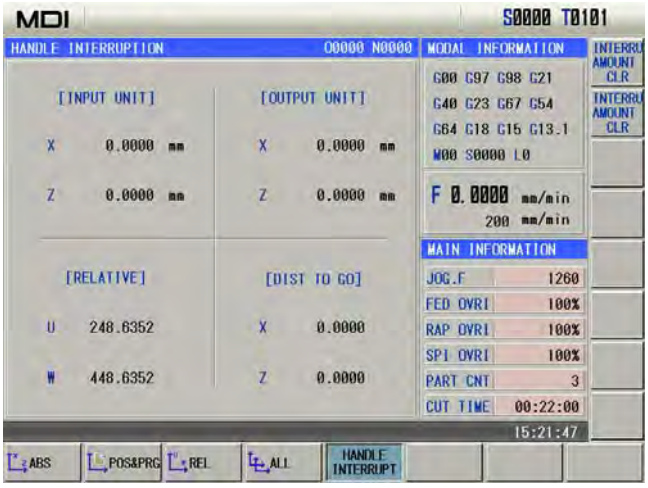
- Reset, EMS (Set whether it is cancelled by parameter).
- Perform G28 or Manual return reference point operation. (When the absolute coordinate system is set after returning to the reference position, the interruption value will be directly eliminated regardless of the new-created absolute coordinate system. When the absolute coordinate system does not set after returning to the reference position, the workpiece system will recover to the previous state of the interruption, simultaneously, the interruption value is then eliminated).

- When the non-block reference point set is performed. (The treatment is consistent with the manual reference point return)
- When performing the G50 workpiece coordinate system setting. (In this case, the workpiece coordinate is one by G50, simultaneously, the interruption value is eliminated)
- System restarts. The MPG interruption value will be also cleared after the system is power off, but the workpiece coordinate system holds, and therefore, the system is turned off after the MPG interrupt, the workpiece coordinate offset generated in the MPG interruption will always enabled, and it can not recover to the previous state of interruption.)

9.4.4 MPG Interruption Display Page

The Bit 1 of state parameter No.0012 should be set to “1” before using the MPG interruption function, and then lead in the dedicated PLC program;

Press the soft function button to enter its corresponding page after controlling the HANDLE INTERRUPT button, refer to the following figure:



The following 4 data can be simultaneously displayed at the MPG interruption display page:

1) Input unit:

The MPG interruption movement value in the input unit means the specified movement value of MPG interruption based upon its least input command unit.

2) Output unit:

The MPG interruption movement value in the output unit means the specified movement value of MPG interruption based upon its least output movement unit.

3) Relative coordinate:

The position at the relative coordinate system and the position of the current relative coordinate system may generate the change of the interruption value due to the interruption of the MPG.

4) Residual movement value:

The residual movement value at the current block is irrelevant to the MPG interruption.

The MPG interruption movement value is cleared when the manual reference point end return is performed along each axis.

Note 1: In the MPG interruption page, the letters of the coordinate axis name should be marked by in the MPG interruption axis.

Note 2 : Manually clear the MPG interruption value by the soft button of the MPG interruption page; the

workpiece coordinate system will not renew; that is, the coordinate system position before the MPG interruption will not recover, the workpiece coordinate offset generated in the MPG interruption will always enabled, and then can not be recover again.

9.4.5 Precaution of MPG Interruption

1) MPG interruption is disabled when the MPG feed (Including the Manual, MPG, Single-step, Machine tool zero return or program zero return) is performed;
2) The axis is controlled by PLC axis; MPG interruption is disabled.
3) MPG interruption is disabled when MPG trial-cutting is performed.
4) The axis does not move when the machine is locked, and the MPG interruption is disabled.
5) MPG interruption is disabled when returning the reference position.
6) MPG interruption is disabled when the system alarm occurs.

Note: The MPG interruption disabling are on the states 1)～3), as the above-mentioned, the system will not enter to the MPG interruption state. If it on the states 4)～6), the system can be entered to the MPG interruption state, instead of generating the enabled MPG interruption value.

9.5 Other Operations

1. In Auto mode, press key to switch on/off the cooling;

2. Press any of the , , , , , , keys to switch the operation modes;

3. Press the key to reset this CNC system.
4. Automatic lubricating operation (Refer to **OPERATION, Chapter 3**).

CHAPTER 10 ZERO RETURN OPERATION

Note!

The key functions of this GSK980TDi 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 panel key functions in this chapter is based on the GSK980TDi standard PLC program!

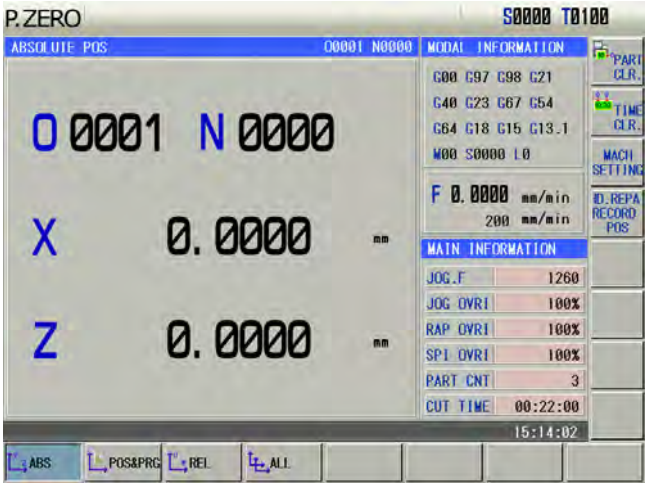
10.1 Program Zero Return

10.1.1 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.

10.1.2 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).

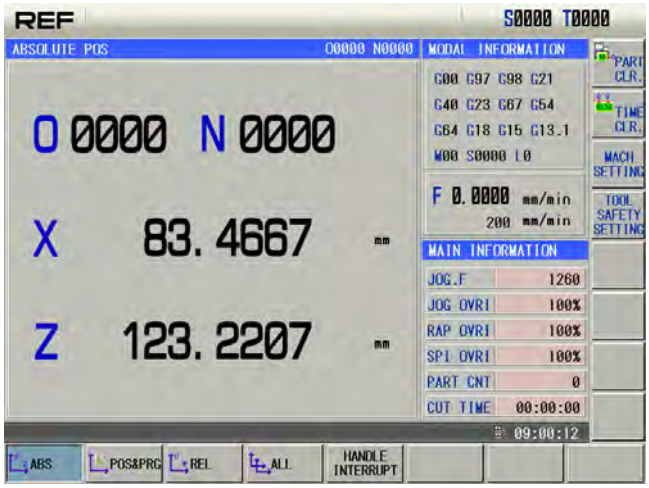
10.2 Machine Zero Return

10.2.1 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.

10.2.2 Machine Zero Return Steps

1. Press key, it enters the Machine zero mode, the bottom line of the window displays "REF":



2. Press , , , , or key to return to the machine zero of X, Z, Y, 4th or C axis;

3. Carrier moves along with the machine tool zero; when it matches with the incremental encoder motor, return to the machine tool zero after passing the deceleration signal or zero signal detection. When it matches with the absolute encoder motor, directly position the set mechanical zero based upon the velocity of the mechanical zero turn; in this case, the axis stops, and the zero return indicator is ON.

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.

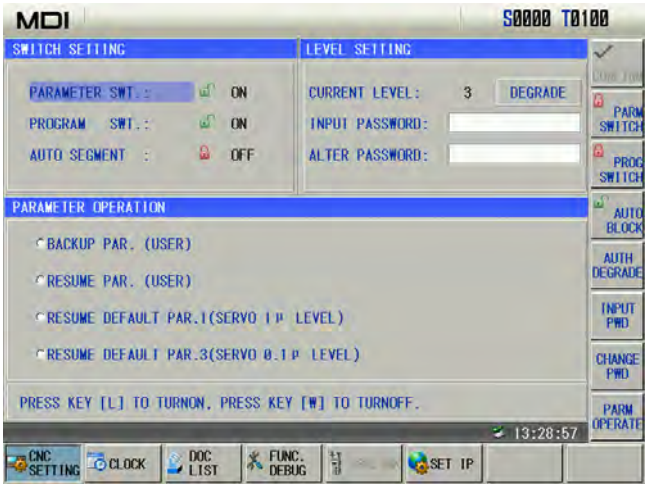
10.3 Other Operations in Zero Return

1) The spindle rotates (CW, CCW), stops.
2) The cooling is ON or OFF.
3) The lubricating control.
4) Manual relative tool change.
5) The spindle override tuning.
6) Rapid override tuning.
7) Feedrate override tuning.
8) Hydraulic chuck control;
9) Hydraulic tailstock control.

CHAPTER 11 DATA SETTING

11.1 Data Setting

Press and then to enter the CNC setting page which includes the switch setting, level setting and parameter operation function as follows:



11.1.1 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) Directly press the , or soft button to open the switch, if it is ON, press the corresponding , or soft button to close it; simultaneously, the cursor will automatically move to the switch item;

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 . 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.

11.1.2 Level Setting

To prevent programs, CNC parameters from being modified at will, GSK980TDi 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 (tool-setting 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.

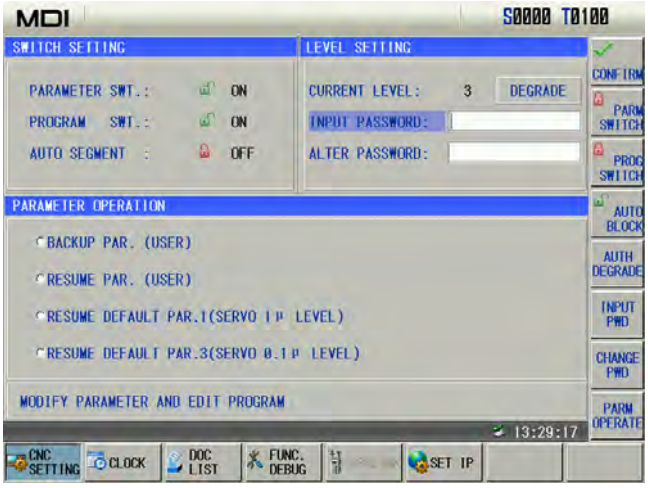
1) Operation grade entry

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) Press the CHANGE PWD soft button or manually move the cursor; Move the cursor to the line “ALATER PASSWORD”;

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.

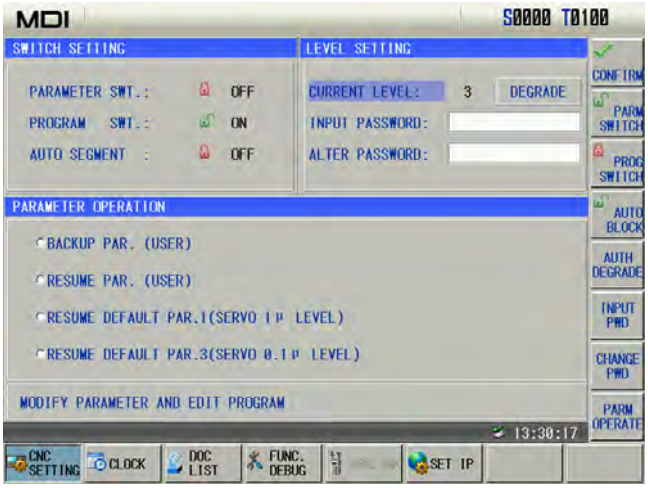
Note: The system prompts, “The new password is not identical with the confirmation” when the two input passwords are not identical. At the moment, input the new password again.

3) Operation grade degradation

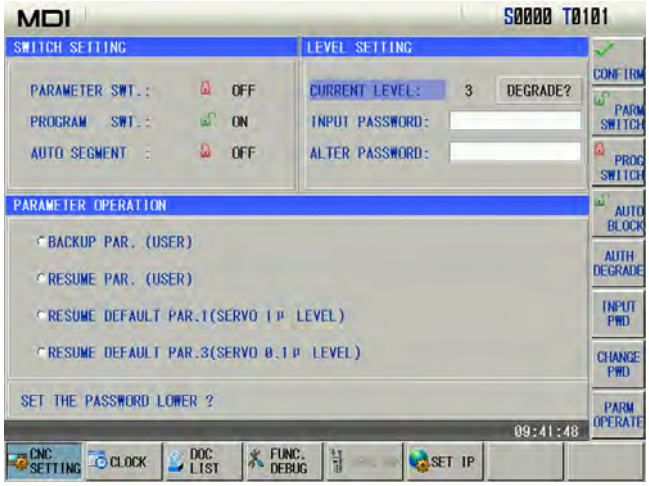
The degradation is to degrade from the high to the low, and its steps are shown below:

a) Input the password according the method of “Operation grade entry” after the system enters the setting page;

b) Press AUTH DEGRADE soft key, Move the cursor to the line “CURRENT LEVEL” and the system pops up DEGRADE as follows:



c) Press CONFIRM（or ）, and the CNC prompts “Password degrades one grade” as follows:

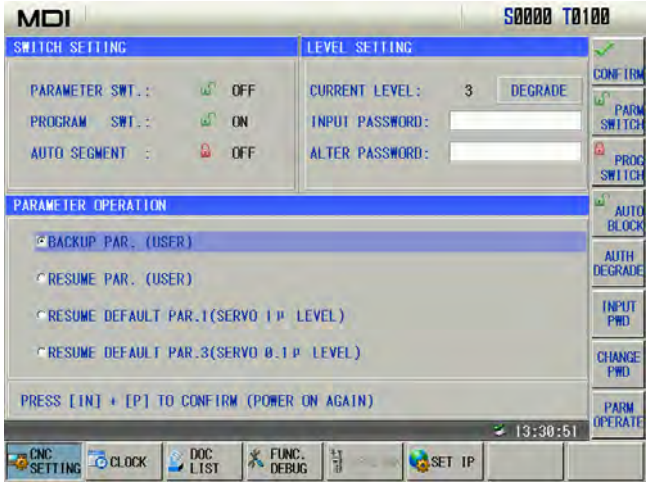


Note 1: The degradation operation cannot be executed when the current level is the 6th grade.

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.

11.1.3 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:



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 (servo 1u level): the user reads the original parameter data matched with the servo drive with the precision 1um

Resume parameter (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.

2) Operation methods:

a) Open the parameter switch;

b) Press to enter the MDI mode, move the cursor the required item;

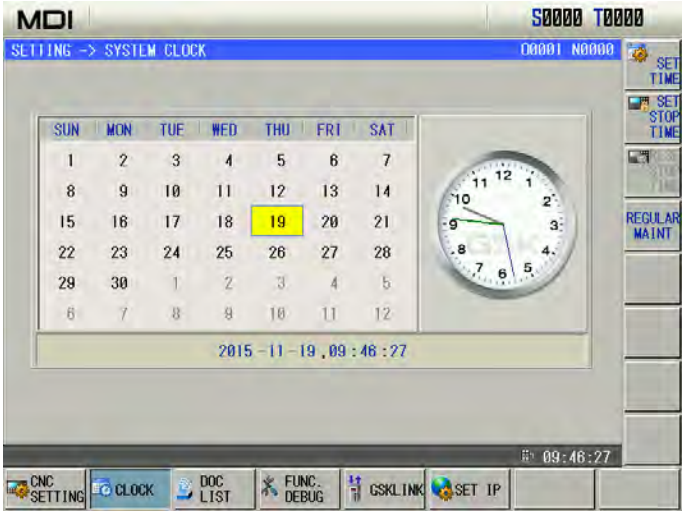
c) Press CONFIRM, and the system prompts whether to confirm the current operation.

d) Press , and the system prompts the backup or resume is done successfully. When the recovery is done, the system prompts power-on again.

Note: Don't turn off the system and execute other operations when the backup or resuming is executed.

11.2 Clock Setting

Press SETTING and then CLOCK to enter the system clock page. The operator can alter the current data and time as follows:



Press SET TIME or CHANGE to enter the alteration mode, press ←, → to move the cursor to the required modification (Year/Month/Date/Minute/Second), input directly numbers, and press , then SET TIME or CHANGE to escape from the alteration mode.

11.3 Graphic Display

Press GRAPH key to enter the path page as follows:

1) The START, STOP and CLEAR of the graphic path

In the path display page, press the START (S) (or S) and the system starts the drawing; press STOP(T) (or T) , it stops drawing; press CLEAR (or R), 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 FIXED (L) (or L) and the display area is locked, the system does not regulate the graphic center on when the path exceeds the display area. Press FLOAT (L) (or L) and the system releases the lock.

2) Scaling up and down

In the path display page, press ZOOM+ (or I), ZOOM- (or M), the system scales up/down the path in real time. Press ZOOM+ (or I) each time, the graphic path is scaled up $\sqrt{2}$ fold, press ZOOM- (or M) each time, the graphic path is scaled down $\sqrt{2}$ fold.

3) Path display moving

In the graph display page, press the cursor movement key to move the graphic path. Press MOVEDIS (or J)to adjust mobile intervals.

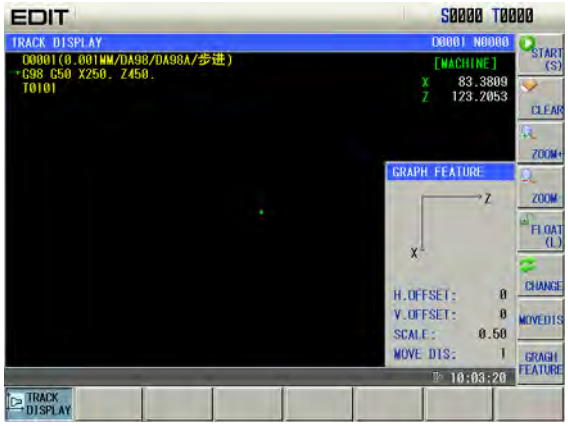
4) Graphic attribution display

In the path display page, press GRAGH FEATURE (or G) to display the graphic attribution window, including the current coordinate system, translation position, scaling and mobile interval. It is shown below:

Press GRAGH FEATURE (or G) again to close the graph attribution window.

5) Graphic parameter meaning

Coordinate system setting: 8 types of graphic paths can be displayed in this GSK980TDi CNC system according to the front or rear tool post coordinate system, and there is 4 kinds of path as follows: (switch X, Z axis by pressing CHANGE or K C)

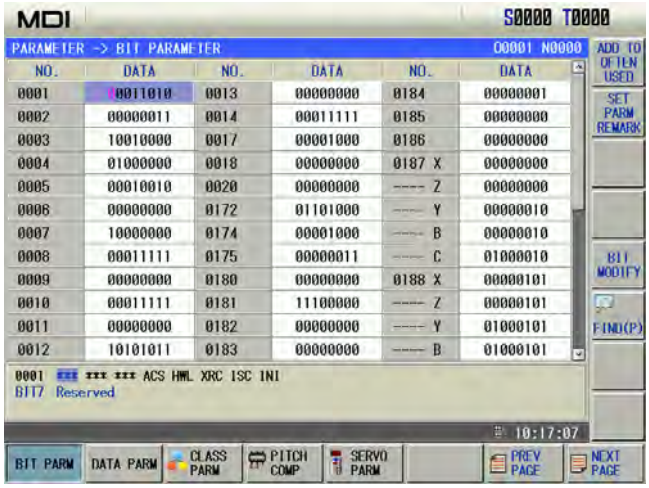
Bit parameter No.175		Graphic path coordinate display
Bit1	Bit0	
0	0	
0	1	

Bit parameter No.175		Graphic path coordinate display
Bit1	Bit0	
1	0	
1	1	

11.4 Parameter Setting

Parameter setting can adjust related characteristics of the drive unit and machine.

Press PARAMETER to enter Parameter interface which includes BIT PARM, DATA PARM, CLASS PARM, PITCH COMP, SERVO PARM as follows:



11.4.1 State Parameter

1) Byte alteration:

a) Turn on the parameter switch;

b) Enter the MDI mode;

c) 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 FIND(P) (or) , and input the parameter number in the pop-up dialogue box, at last, press .

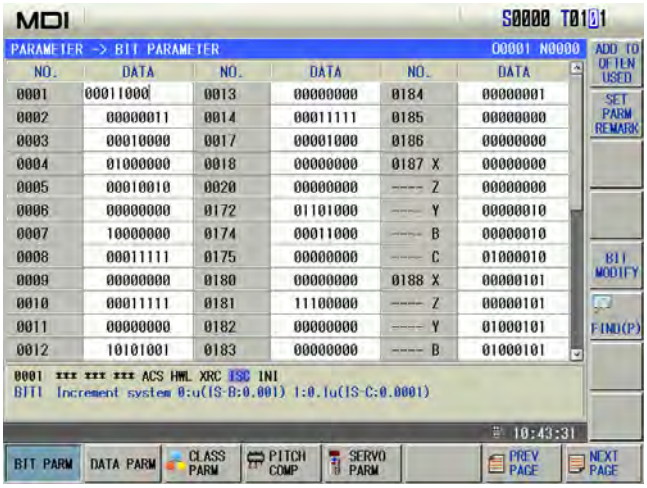
4) Key in a new parameter value;

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.

Example:

Set the bit parameter No.001 Bit 3(STEP/MPG) to 1, and the other bits remain unchanged. Move the cursor to No.001, input 00011000 by sequence according to the above steps. The display is shown below:



Press to finish the parameter alteration. The page is shown below:

2) Alteration by bit:

a) Turn on the parameter switch;

b) Enter the MDI mode;

c) Move the cursor to the parameter number to be set（the method is referred to Section 10.4.1 Step c）;

d) Press CHANGE to enter the bit alteration mode, at the moment, some bit of the parameter is backlighted. Press → or ← to move the cursor to the bit to be altered, then key in 0 or 1;

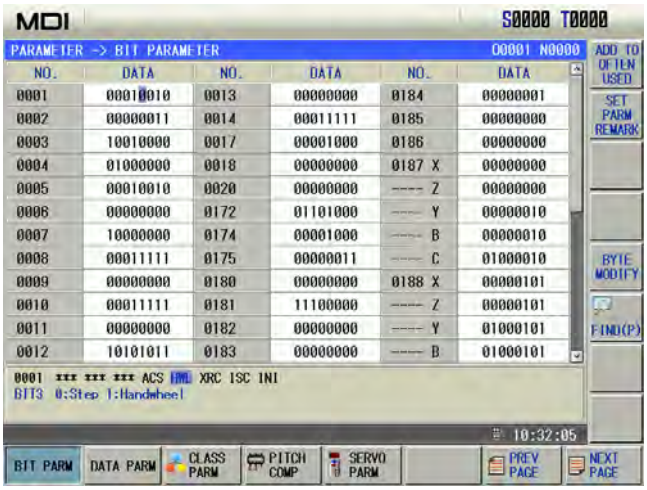
e) After all parameters setting is finished, the PARM SWT needs to be set to OFF for security.

Note: After the system enters some bit of the parameter, BYTE MODIFY or CHANGE is pressed to escape from the alteration mode, and the cursor stops the parameter number .

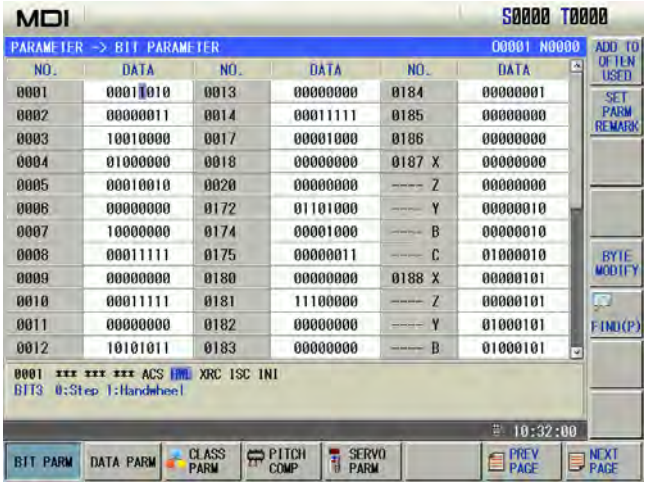
Example:

Set the Bit3 of the bit parameter No.001 to 1, and the other bits remains unchanged.

Move the cursor to No.001 according to the above steps, press CHANGE to skip to some bit of the parameter, and move the cursor to Bit3. The display is shown below:



Input 1, and the parameter alteration is done.



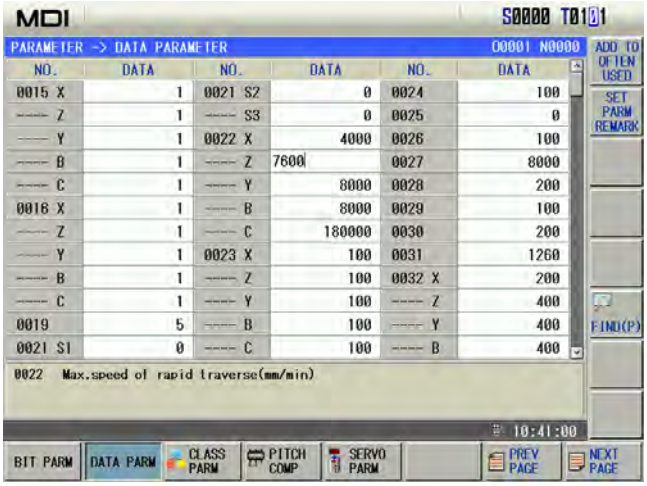
11.4.2 Data Parameter, Pitch Compensation

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.

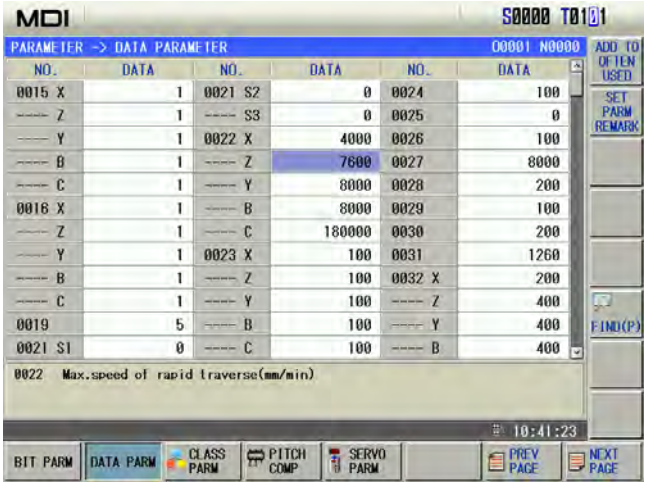
Explanation: The screw-pitch parameter can only be altered under the 2 level password authority.

Example 1: set the data parameter No.027 to 7600.

Move the cursor to No.027 by the steps above, key in 7600 by sequence as follows:

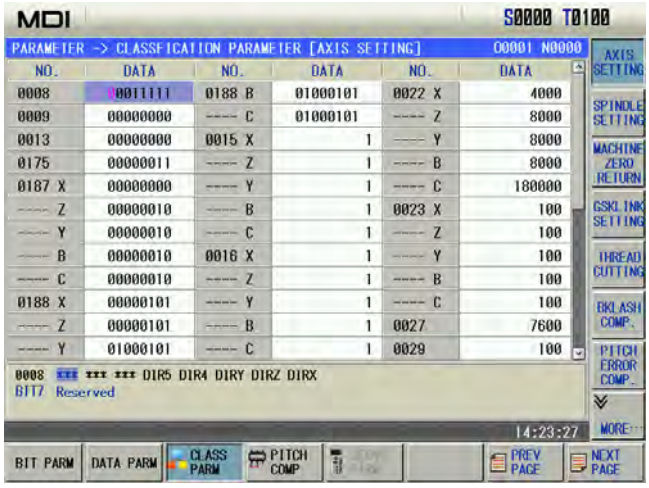


Press to finish the alteration. The display is shown below:

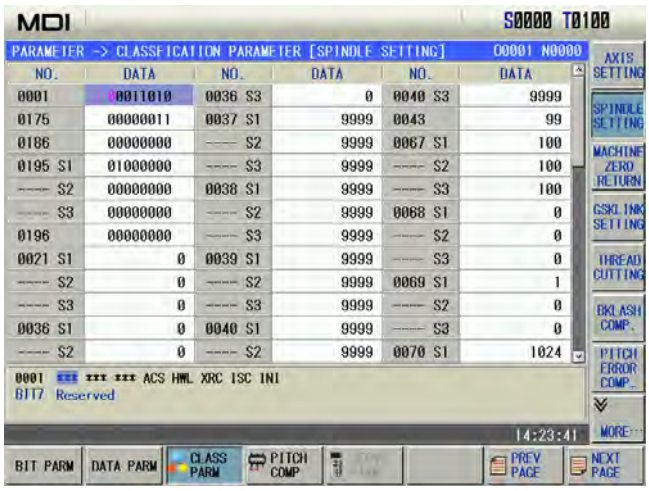


11.4.3 Classification Parameter

Press soft button to enter its page after controlling the button ,The display is shown below :

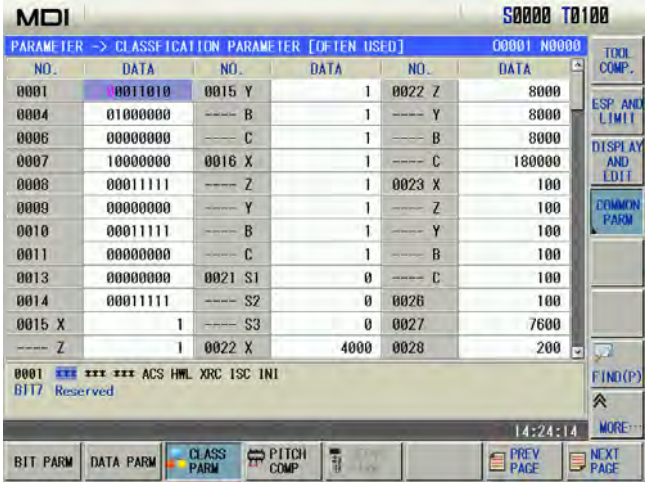


In the classification parameter, the parameter can be displayed based upon its function, it is convenient to check or modification. The current displayed classification parameter can be shifted by its corresponding soft button; for example, press the SPINDLE SETTING soft button, the following figure is shown below:



11.4.4 Often Used Parameters

Press [PARAMETER] to enter parameter interface, and press [COMMON PARM] to enter the often used parameter page. The page is shown below:



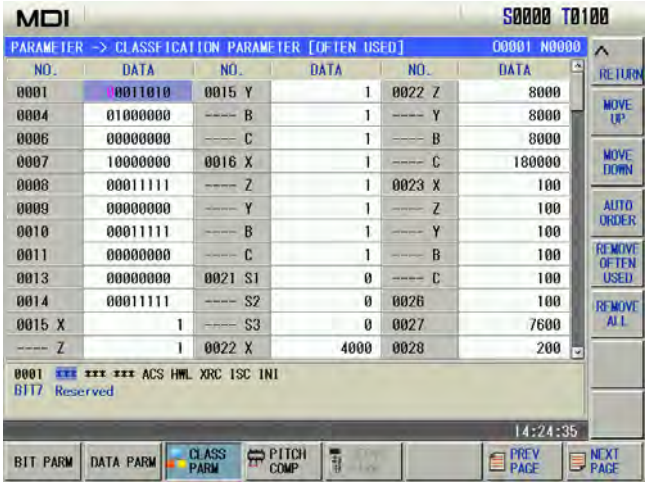
The user can add the often used state parameter, data parameters to the often used parameters to get the convenient search and alteration.

1) Adding often used parameters

In the state, data or classification parameter page, move the cursor to the parameter number where is desired to add as the common parameter; press the [ADD TO OFTEN USED] soft button, the current parameter is then add as the common one.

2) Sorting often used parameters

In the common parameter page, enter to the next menu of the common parameter by pressing the [COMMON PARM] soft button again; refer to the following figure:



Pressing [MOVE UP], [MOVE DOWN] can move the parameter where the current cursor is. Pressing [AUTO ORDER] sorts all often used parameter in sequence number.

3) Removing often used parameters

Move the cursor to the required parameter in the often used parameter page. Press [REMOVE OFTEN USED] and the current often used parameter is removed. Press [REMOVE ALL] **and all often used parameters are removed.**

4) Altering often used parameter

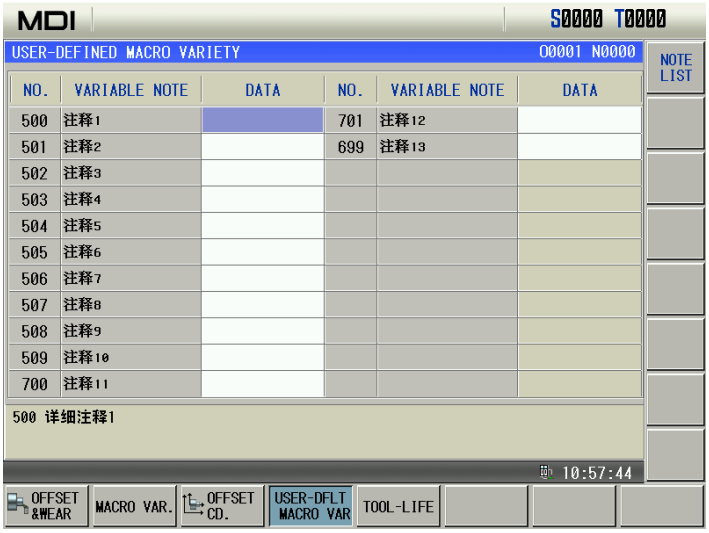
Parameters can be altered directly in the often used parameter page. Its alteration method is the same that of state parameter and data parameter. Its value is also refreshed in the state parameter or data parameter page.

11.5 Macro Variable Note

Macro variable note display is mainly administrated for the macro note and macro note file. The administration of the macro note file includes the leading-in, leading-out, rename, overloading and remark, etc. of the macro note file.

11.5.1 Macro Variable Note Page

Press the soft button to enter its corresponding customization page ,after entering the cutter compensation page by controlling the USER-DFLT MACRO VAR button, The page is shown below:



The overloaded file data can be set on the “Custom macro variable” page, only 4 and above operation authorities can be performed.

11.5.2 Note List

Press the NOTE LIST button to enter the macro variable note list after entering the macro variable note page.

Function soft button explanations:

SELECT **soft button:** Select the note file to enable the file;

RENAME **soft button:** The file can be renamed for the current file, the note file at current loading can not bealtered;

REMARK **soft button:** Modify or add the remark information, the note file at current loading can not bealtered;

DELETE **soft button:** Delete the file at the cursor's position; (2 or above authorities are desired)

DELETE ALL **soft button**: Clear the overall annotated files and contents;

LEAD ALL **soft button**: The "EXT files" under the MAROEXT list of the U-disk are led to the system note list;

EXPORT ALL **soft button:** The overall files in the note list are led out to the MACROEXT list of the U-disk; if there are files are shared with a same name, the same files will be replaced;

Note 1: The SELECT can be performed in any authority, the DELETE ALL or LEAD ALL should be operated with 2 or above levels authorities; other functions should be performed with 3 or more authorities.

Note 2: The RENAME can be performed in any method, other functions should be operated in the Edit method.

11.5.3 Macro Variable Note File

1. File content format

1）Create a TXT file，the surfix is renamed as .EXT。

2）The file content format is as follows：

0, Remark the display content

Variable number, custom content \t Note content.

For example: Set up a O0009.EXT file, its content is as follows:

0, Note file example

500, Note 1\t Detailed note 1

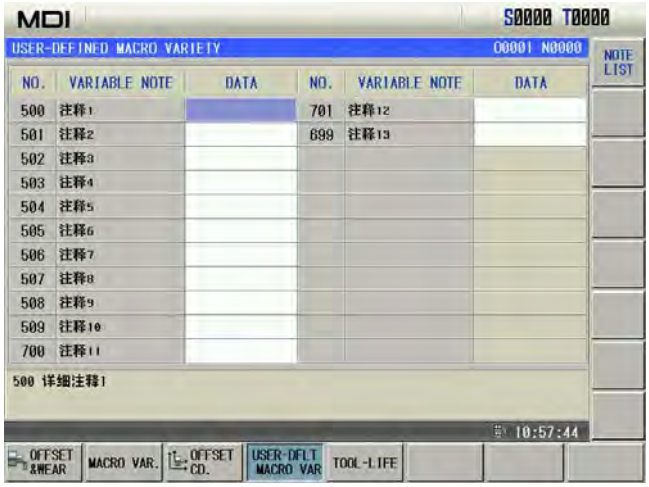
501, Note 2\t Detailed note 2

……

701, Note 12\t Detailed note 12

699, Note 13\t Detailed note 13

The displayed result of the above-mentioned note file is as follows:



2.File leading-in/out operation

Leading-in operation

1）Create a new file MACROEXT under the root list of the U-disk;；

2) The performed system custom file is delivered to the folder;

3) Set the system authority as 2-level or above, and then insert to U-disk;

4) The *.EXT file under the U disk MACROEXT list can be entirely lead out to the system note list by LEAD ALL soft button.

Leading-out operation

1) Insert an U-disk;

2) The overall files in the system note list are lead out to the U disk MACROEXT folder by EXPORT ALL soft button; if the note file name is same as the system list under the U disk MACROEXT; the same name note file will be covered under the systme root list.

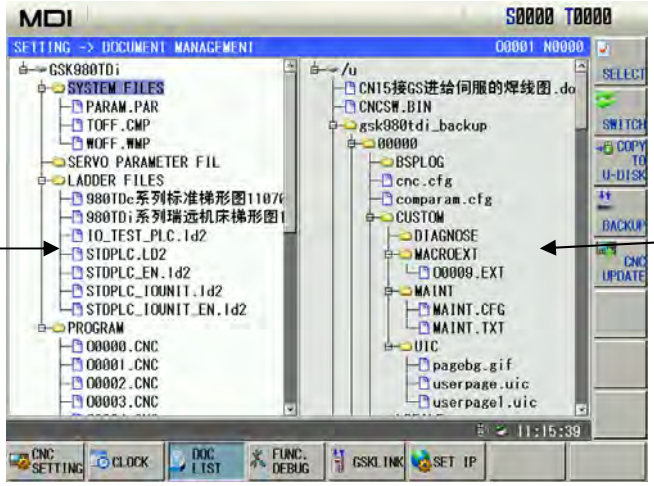
CHAPTER 12 U OPERATION FUNCTION

12.1 File Management Function Introduction

GSK980TDi has the file management function to execute the file copy, data backup, data resume, the system upgrading (2nd grade) and so on.

In any states, press to select the setting interface, press to enter the file management page. Insert U disk and the system automatically identifies it. The system displays the icon at the lower right bottom of the system after it sucessfully identifies the icon as follows:

Display the user parameters, tool compensation, pitch compensation, PLC programs, part program and so on in the CNC catalog window



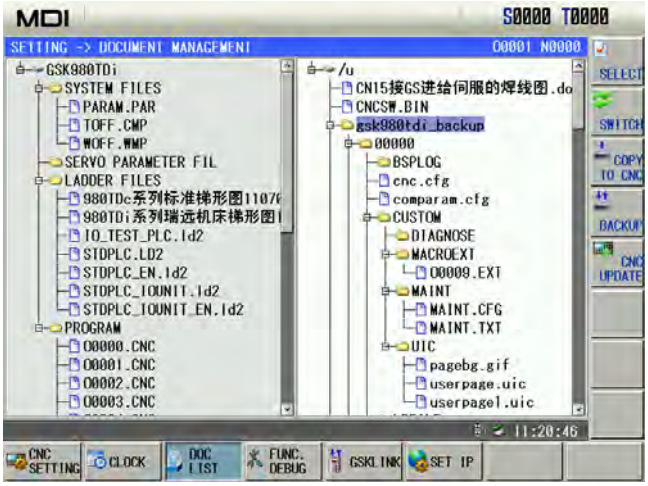
Display all files in the U disk catalog window

Press the soft key (or) to select the CNC catalog or the U disk catalog, and press , to move the line where the current cursor is.

12.2 Often Used File Operation Function Introduction

12.2.1 File Extension & Return

1) Move the cursor to the required extension file, press to extend it as follows:

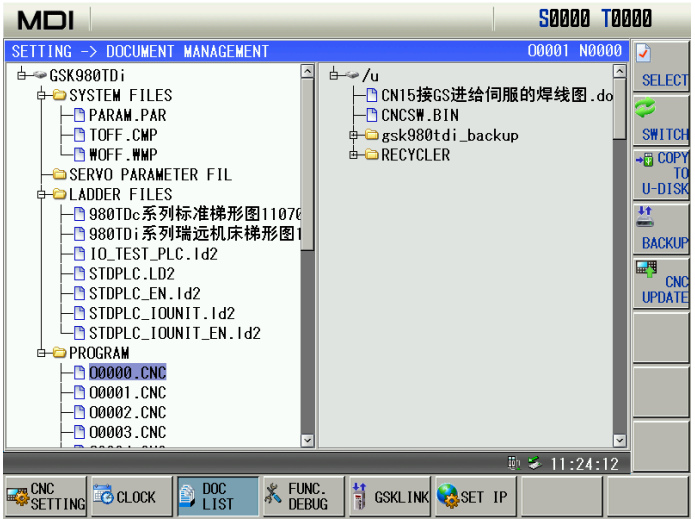


2) Move the cursor to the target file, press ← to return the previous catalog of the current file as follow:

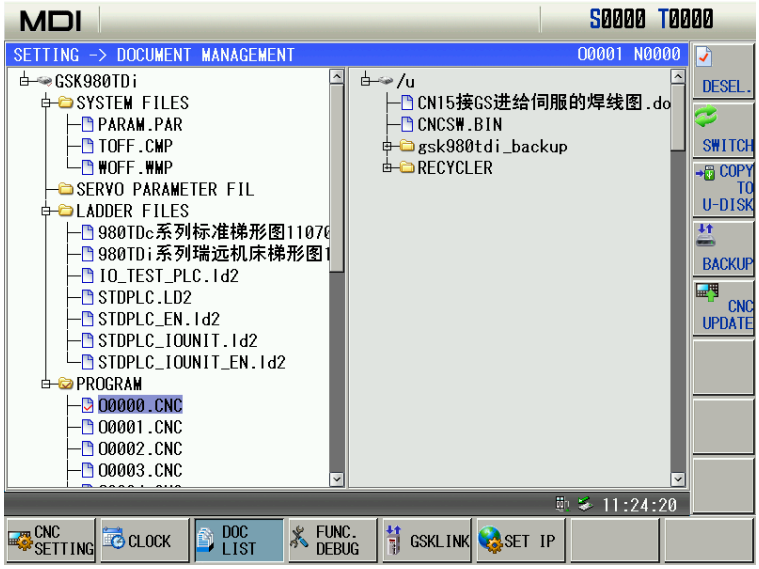
12.2.2 File Selection & Cancellation of File Selection

Take an example of U:\gsk980TDi_backup\user\NCPROG\O0000.CNC to introduce the selection and cancellation.

1) After entering the file administration page, move the cursor above the O0000.CNC by ↑ or ↓, refer to the following figure:



2) Press SELECT (or) to select O0000.CNC as follows:



1) Press DESEL. (or again) to cancel the selection of O0000.CNC.

12.2.3 File Copy

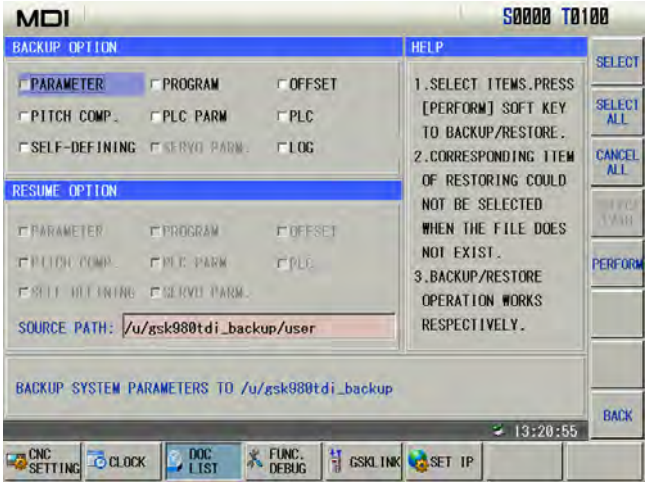
Refer to Section 11.2.2 to select the file. When the cursor is in the U disk catalog window, COPY TO CNC is pressed and the the files in the U disk is copied to the CNC; when the cursor is in the CNC catalog window, COPY TO CNC is pressed and the file in the CNC is copied to the U disk.

12.3 Data Copy & Resume

In the file managemetn page, backup, resume the CNC data (including the system parameters, part parameter, tool compensation, pitch compensation, ladder parameters, ladder, self-defined data, servo parameter and daily record, etc.) via the U disk. The CNC can be operated to the file in the MDI working mode and keeps at the stopping state.

12.3.1 Entering Backup/Resume Page

After the system identifies the U disk, it enters the file management page. Press BACKUP to enter the backup/resume page as follows:



12.3.2 Backup/Resume Operation

1) Press ↑, ↓, ←, → to move the cursor to the required option;

2) Press SELECT (or ⏎) to select the required option to be backup/resumed;

3) Press DESEL. (or ⏎) to cancel the selection when the cursor has been in the selected option;

4) Press SELECT ALL or CANCEL ALL to select or cancel all selection;

5) Press PERFORM (or ⏎) to execute the current selected.

Note 1: Data which has been backup or has been done to store in the U disk can be resumed.

Note 2: When the cursor is in the "BACKUP OPTION" column, all options in the the "RESUME OPTION" column cannot be selected; when the cursor is in the "RESUME OPTOIN", all options in the "BACKUP OPTION" column cannot be selected;

Note 3: Pressing PERFORM executes operations only in the column where the cursor is. When the cursor is in "BACKUP OPTION" column, the resume operation is executed. The backup, resume cannot be executed simultaneously.

12.3.3 Operation Path Selection

When the user uses the U disk to execute the CNC data backup, the CNC creates a backup file "gsk980TDi_backup" in the root catalog of the U disk. The backup file includes two backup data "U:\ gsk980TDi_backup\user" and "U:\ gsk980TDi_backup\serial number" (it is created by the system's serial number). When the user uses one U disk to execute data backup of many CNC systems, the data in "U:\ gsk980TDi_backup\user" is replaced by the new but data in "U:\ gsk980TDi_backup\serial number" remains unchanged.

For the above reason, the user should execute the operation path selection during resuming data:

Default path ：U:\ gsk980TDi_backup\user

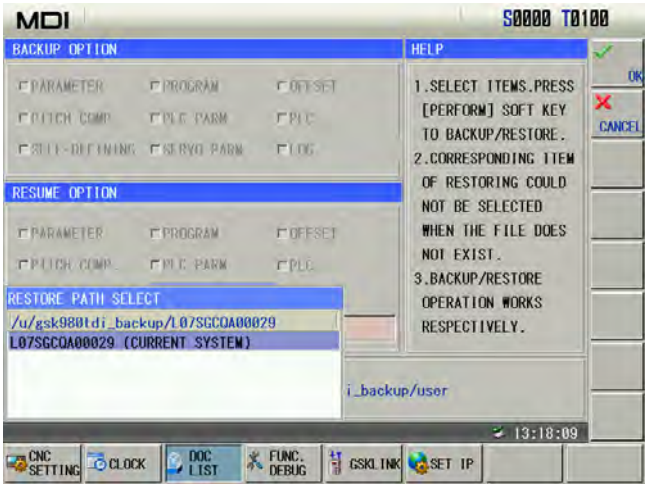
Serial number path：U:\ gsk980TDi_backup\serial number

Example: backup/resume data of many CNC systems

When the user backups the system A (serial number CT0000001), the generated file by data backup is “U:\gsk980TDi_backup\CT0000001” and “U:\gsk980TDi_backup\user”; When the user backups the system B（serial number CT0000002）, the generated file by data backup is “U:\gsk980TDi_backup\CT0000002”. The “U:\gsk980TDi_backup\user” generated by A is replaced automatically by the same name of the system B. At the moment, the user can execute the operation path selection when it executes the data resume operation of the system A:

1) When the previous data of the system A is resumed, the operation path is : “U:\gsk980TDi_backup\CT0000001”;
2) When the backup data of the system B is copied to the system A, the operation path is: “U:\gsk980TDi_backup\CT0000002” or default path;

Press SELECT PATH and appear the dialog box of operation path selection, and then select the operation path by ↑,↓ when the cursor is in the resume column. Taking an example of selecting the default path, the display is shown below:



12.3.4 Format of Data Backup/Resume File

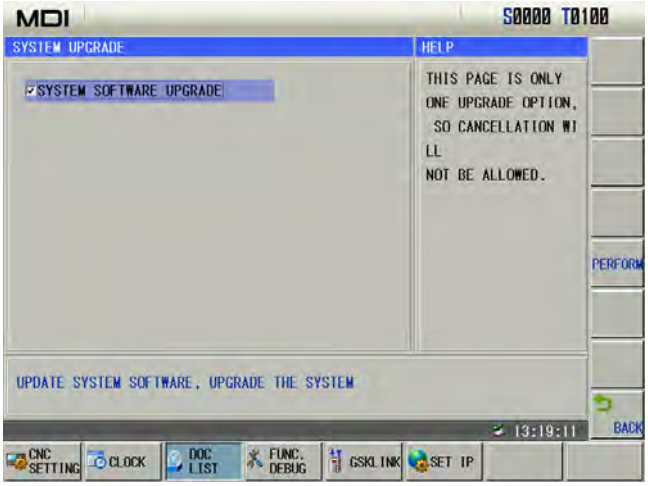
	File	Extension name	Content
Parameter file	PARAM	.PAR	State parameters, data parameters
Tool compensation data	TOFF	.CMP	Tool offset, tool wear
Pitch compensation	WOFF	.WMP	Pitch compensation
Alarm log	Alarm.his	.his	Alarm log
PLC program	No requirements	.ld2	PLC programs
PLC parameter	plcpar	.dat	K, D, DT, DC parameters
Part program	O0000~O9999	.CNC	Part programs

12.3.5 Level Explanation of Data Backup/Resume Operation

Backup operation	System parameters	More than the 5th grade password
	Tool compensation data	More than 5th grade(including the 5th grade) password
	Pitch compensation	More than 5th grade(including the 5th grade) password
	Alarm log	More than 5th grade(including the 5th grade) password
	PLC programs	More than 5th grade(including the 5th grade) password
	PLC programs	More than 5th grade(including the 5th grade) password
	Part programs	More than 5th grade(including the 5th grade) password
Resume operation	System parameters	More than 3rd grade(including the 3rd grade) password
	Tool compensation data	More than 5th grade(including the 5th grade) password
	Pitch compensation	More than 2nd grade(including the 2nd grade) password
	PLC programs	More than 5th grade(including the 5th grade) password
	PLC parameters	More than 2nd grade(including the 2nd grade) password
	Part programs	More than 4th grade(including the 4th grade) password

12.4 System Update

In the system update page, the CNC update operation can be performed by the U-disk. The updated file is registered under the root-directory of the U-disk when the operation is performed; so that the CNC can be operated holding on the MDI working method and keeping on the stop state.



Select the update system by PERFORM soft button, and confirm the update state by .

Note: Page update can be entered above the 2-level authorities.

12.5 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) It is necessary to connect the GSKLINK bus normally when the back up/recovery of the servo parameter is performed;
3) Backup/resume operation must be in MDI mode;
4) When the user executes the data backup operation, the system automatically replaces all files in “U:\gsk980TDi_backup\user”, the user should save them as when they cannot be replaced;
5) Stop part programs running before data backup/resume;
6) They must not stop unless the data backup/resume operation is completed;
7) The system is turned on again after the data resume is done.

CHAPTER 13 COMMUNICATION

13.1 GSK980TDi Communication Software GSKComm

GSKComm is an allocated project administrator. It can execute the upload/download between PC and CNC, conveniently transmit batch with easy operation, high communication efficiency and reliability.

The followings introduce communication transmission and setting between GSK980TDi and GSKComm. Refer to GSKComm explanations about GSKComm communication software functions.

13.1.1 GSKComm's System (PC) Requirements

Hardware: general PC with RS232 serial port, serial communication cable (3-wire)

Operation system: Microsoft Windows 98/2000/XP/2003

13.1.2 Software Interface

Interface after GSKComm's initial run:

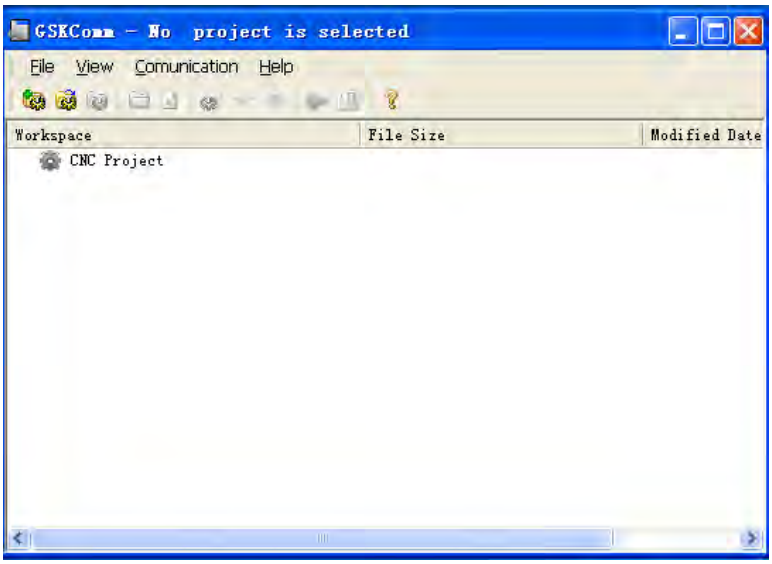
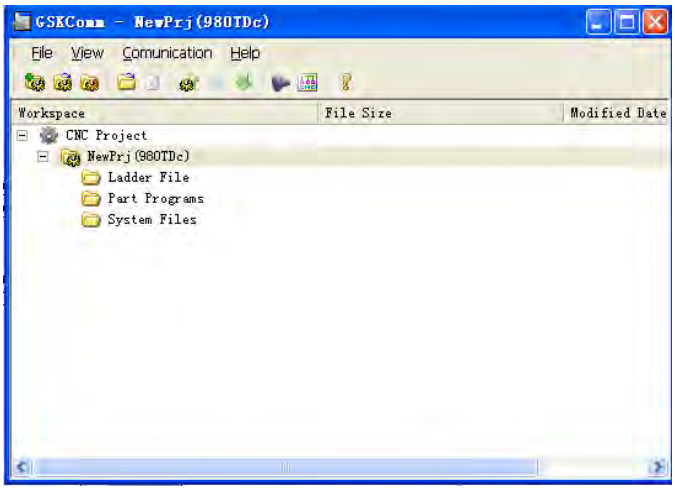


Fig. 12-1 file download (PC→CNC) interface

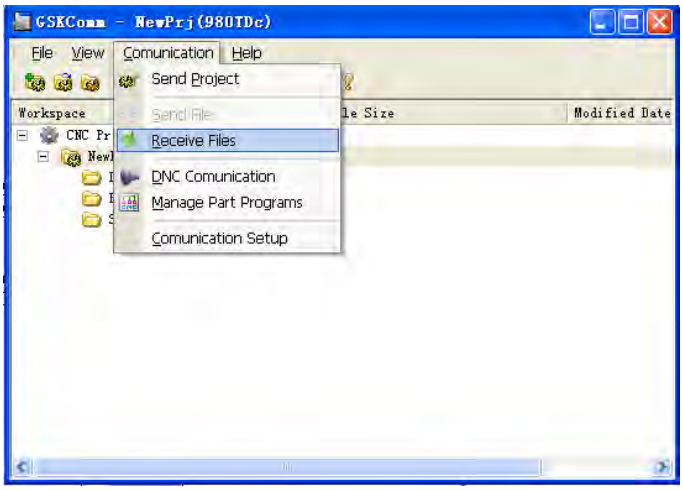
GSKComm software takes an administrator as a unit to complete management. When it run firstly, it does not upload project files, at the moment, the user should create an project program or upload project to execute the communication operation. The display is shown below after the an project is created:



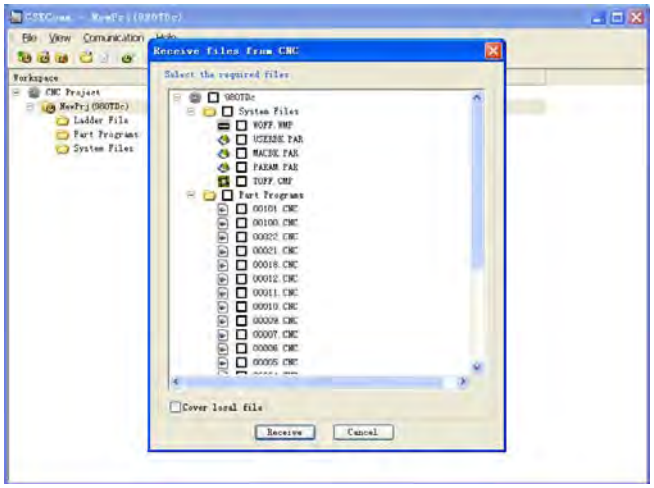
At the mode, the user can receive the files from the CNC to the current project or send files in the current project to the CNC.

13.1.3 Receiving Files (CNC→PC)

Click the option [Receive files from CNC] in [Communication] and GSKComm can receive files from the CNC after the current project is selected as follows:

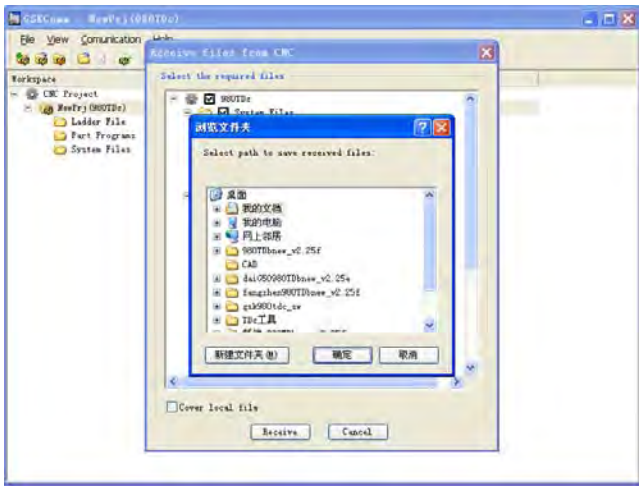


After [Receive Files] is clicked, a dialog box about receiving files from the CNC, the required files from the CNC including part programs, tool compensation (TOFF. CMP), pitch compensation (WOFF.WMP), parameters (PARAM.PAR).

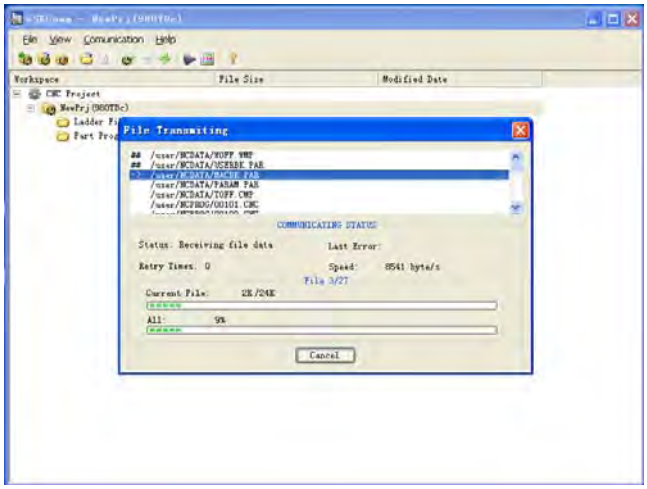


After the user selects the required file, [Start Receiving] is pressed and the system pops up a

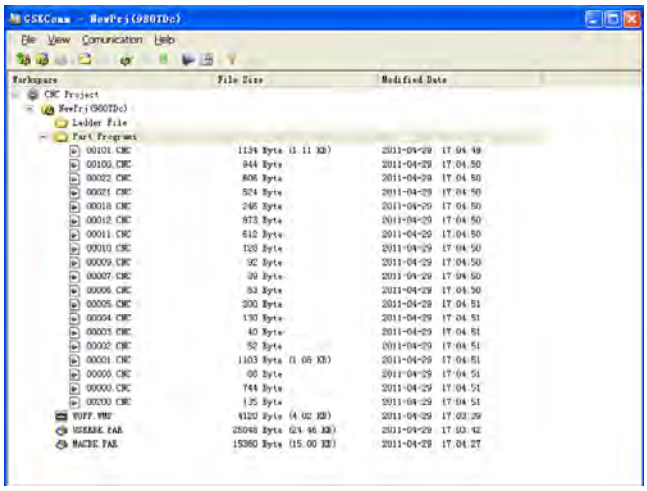
path selection dialog box to save the received files as follows:



Select the save path, start to receive the file from CNC to PC by pressing [i.e. Confirm] as follows:

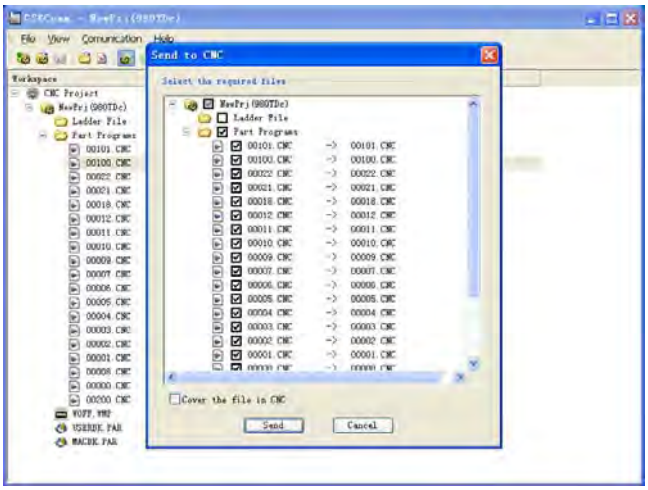


After the receiving is done, the system displays below:



13.1.4 Sending Files (CNC→PC)

After the current project file is selected, the user clicks [Send projects to CNC] in the menu [Communication], and so the receiving file from CNC is done. A dialog box to send files to CNC is popped up as follows:



Hereby, the user can select the required receiving files from CNC including part programs, tool compensation (TOFF.CMP), pitch compensation (WOFF.WMP), parameters (PARA.PAR). The part programs which can be altered are saved to the file name of the CNC. Move the cursor to the part program required to alter its program name, double-click the left key to pop up a dialog box as Fig. A, and the file which can be altered is saved the file name of the CNC (as Fig. B).

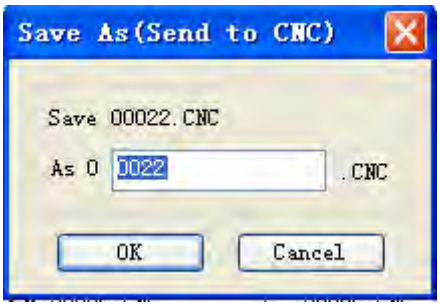


Fig. A

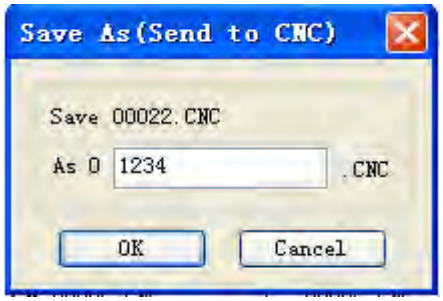
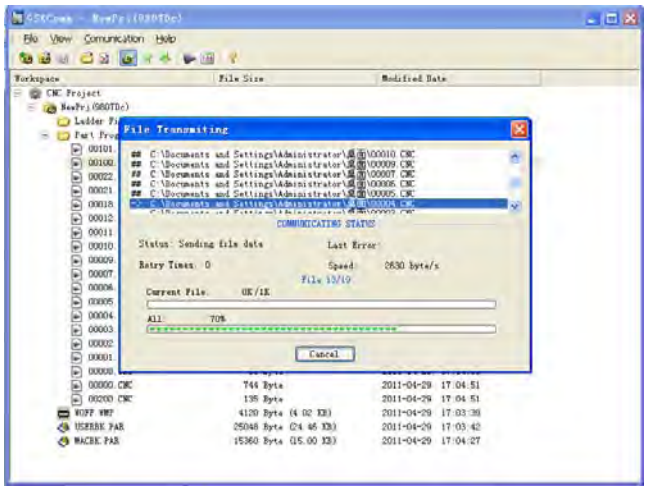
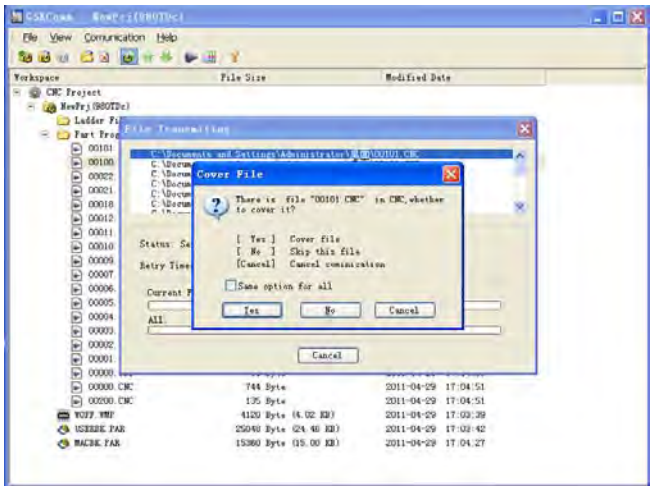


Fig. B

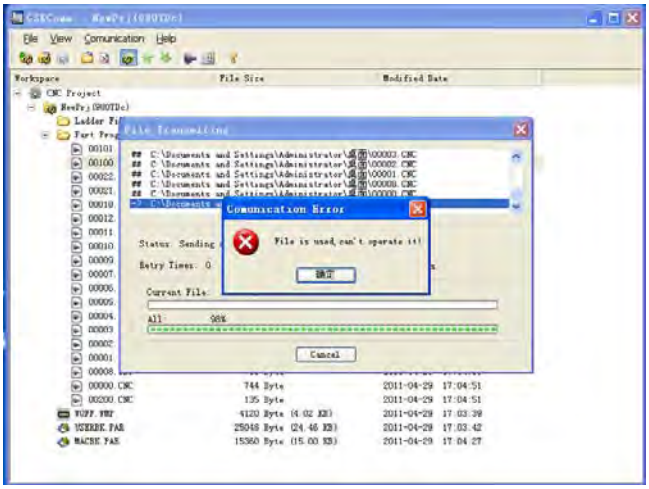
Select the required file to the CNC and confirm the saved file name, then press [start sending], and the file is started to send to the CNC as follows:



When a file name in the CNC is the same that of the transmitting, a dialog box is popped up to select the covering, skipping the file or cancelling transmission as follows:

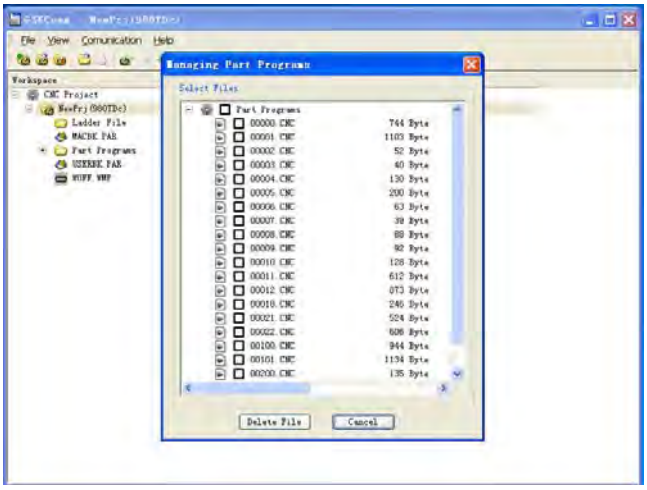


When the part program name is the same that of the transmitted file, the covering cannot be done, otherwise, the system pops up the wrong prompt as follows:



13.1.5 Part Program Management

Select the current project file, then click [Manage Part Programs] in the menu [Communication] , and the part programs in the CNC are managed. The system pops up a dialog box to manage part programs in the CNC as follows:



Press [Delete File] and the selected part programs are deleted in the CNC after the required programs are selected.

Note: The program being used in the CNC cannot be deleted.

13.2 Preparatory Before Communication

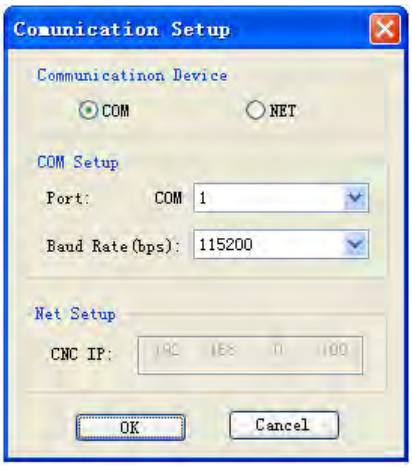
13.2.1 Communication Cable Connection

DB9 male plug is inserted into CN51 communication interface of the CNC, DB9 female plug is inserted in to DB9-male serial port (COM0 or COM1) of PC. Connect the communication cable when PC and CNC are turned off.

13.2.2 Communication Setting Baud Rate

Setting the baud rate of communication ensures that the baud rate of PC is the same that of the CNC communication.

The communication baud of the serial port is set by No.044; click [Communication Setup] in the menu [Communication] and the system pops up a dialog box as follows:



Set the serial port and baud rate. The baud rate should be not less than 4800 when the data is transmitted between CNC and PC, the factory setting is :115200.

13.2.2.1 Network Cable Connection

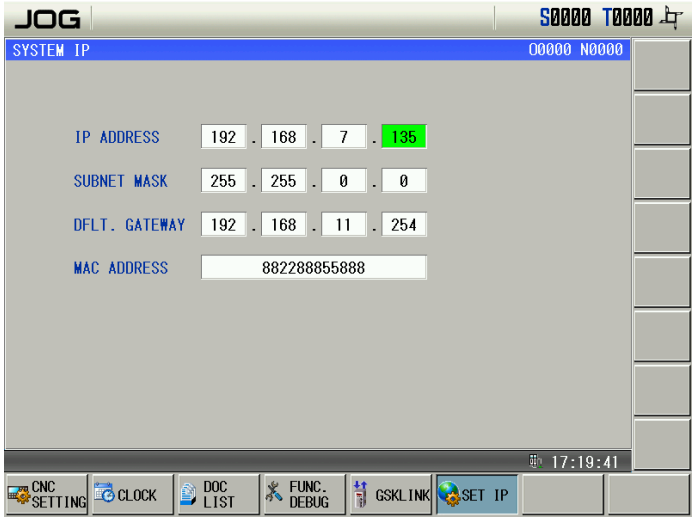
Use the common network cable, one terminal connects with the internet access of GSK980TDi, the other one directly connects the Computer or Router access.

13.2.2.2 Communication IP Setting

Set the IP address of communication, so that the IP address between PC and CNC communication are consistant.

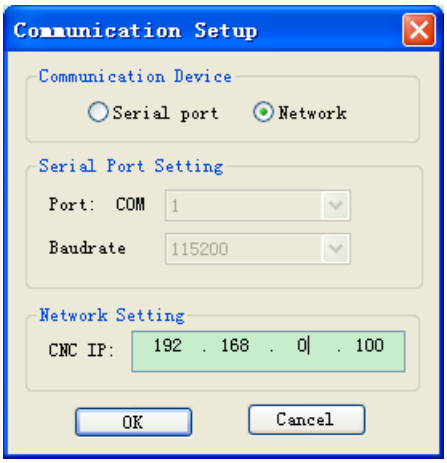
The IP setting of the GSK980TDi is as follows: —>

and then set the IP address and sub-net mask, etc. after entering the system IP setting page, refer to the following figure:



Wherein, the IP address can not be identical with the other PC computer in the network or CNC. The subnet mask should be same with the PC computer or the subnet mask of the router; it is not necessary to alter the default subnet.

In the GSKComm, click the [Communication Setting] in the [Communication] menu, its corresponding dialog frame is shown below, refer to the following figure:



Network selection of communication equipment, write the IP address set to CNC in the network setting.

Note 1: Stop machining when the machining is being done to get the stable communication.

Note 2: Must not turn off during the data transmission, otherwise, it causes the mistaken data transmission.

CHAPTER 14 MACHINING EXAMPLES

Machine a part by a bar stock with dimension Φ136mm×180 mm, as follows:

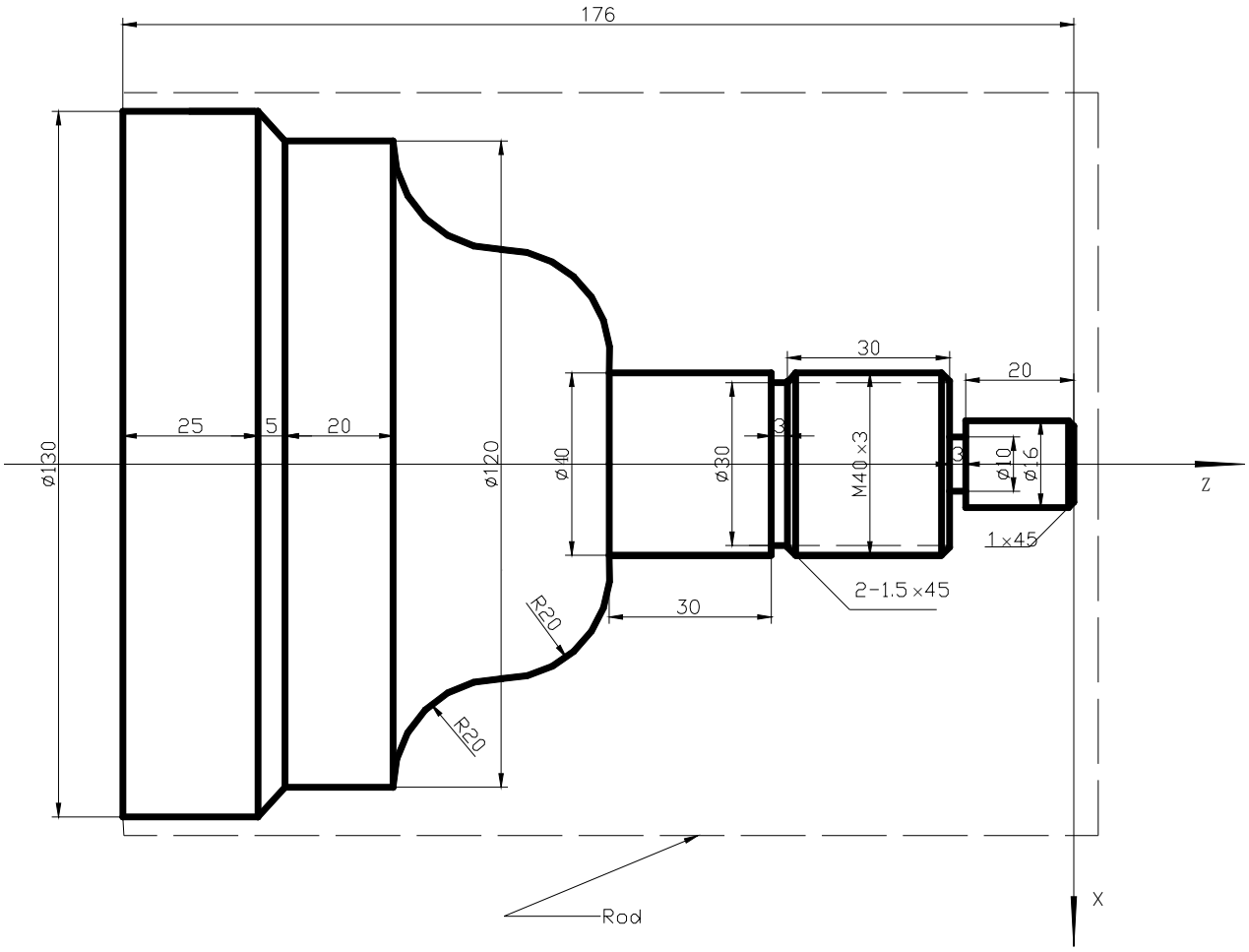
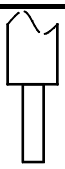


Fig. 14-1

Machine it with 4 tools as follows:

Tool number	Tool shape	Explanation
No. 1		Outer circle rough turning tool
No. 2		Outer circle finish turning tool
No. 3		Grooving tool, tool width 3mm
No. 4		Threading tool, tool nose angle 60°

14.1 Programming

Set up the workpiece coordinate system as Fig.14-1 according to the machining process and the codes introduced in this manual. The programming steps are as follows:

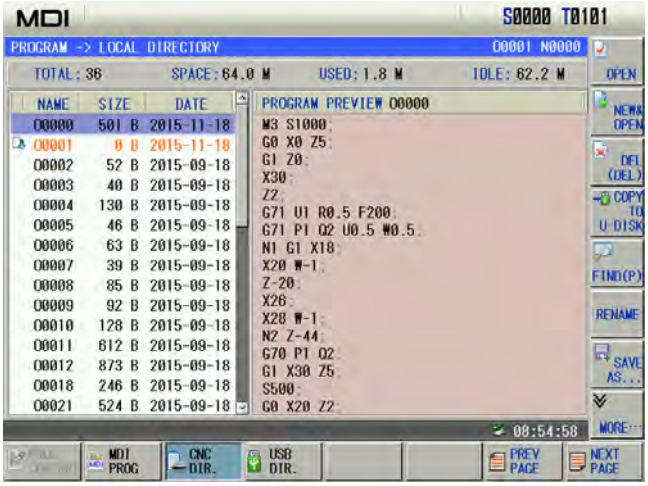
O 0 0 0 1 ;		Name of the part program
N 0 0 0 0	G0 X150 Z50 ;	Position to the safety height for tool change
N 0 0 0 5	M12 ;	Clamp the chuck
N 0 0 1 0	M3 S800 ;	Start the spindle with speed 800
N 0 0 2 0	M8 ;	Turn on the cooling
N 0 0 3 0	T0101 ;	Change for the No. 1 tool
N 0 0 4 0	G0 X136 Z2 ;	Approach the part
N 0 0 5 0	G71 U0.5 R0.5 F200 ;	Cut depth 2mm and retract 1mm
N 0 0 5 5	G71 P0060 Q0150 U0.25 W0.5 ;	0.25mm pre-reserved in X axis, 0.5mm machining allowance in Z axis
N 0 0 6 0	G0 X16 ;	Approach to the end face of the part
N 0 0 7 0	G1 Z-23 ;	Cut the Φ16 outer circle
N 0 0 8 0	X39.98 ;	Cut the end face
N 0 0 9 0	W-33 ;	Cut the Φ39.98 outer circle
N 0 1 0 0	X40 ;	Cut the end face
N 0 1 0 5	W-30 ;	Cut the Φ40 outer circle
N 0 1 1 0	G3 X80 W-20 R20 ;	Cut the convex arc
N 0 1 2 0	G2 X120 W-20 R20 ;	Cut the concave arc
N 0 1 3 0	G1 W-20 ;	Cut the Φ120 outer circle
N 0 1 4 0	G1 X130 W-5 ;	Cut the cone
N 0 1 5 0	G1 W-25 ;	Cut the Φ130 outer circle
N 0 1 6 0	G0 X150 Z185 ;	Rough cut end and back to the tool change point
N 0 1 7 0	T0202 ;	Change for the No.2 tool and execute its offset
N 0 1 8 0	G70 P0060 Q0150 ;	Fine cut cycle
N 0 1 9 0	G0 X150 Z185 ;	Rough cut end and back to the tool change point
N 0 2 0 0	T0303 ;	Change for the No.3 tool and execute its offset
N 0 2 1 0	G0 Z-56 X42 ;	Approach to the part

N 0 2 2 0	G1 X30 F100 ;	Cut the Φ30 groove
N 0 2 3 0	G1 X37 F300 ;	Return
N 0 2 4 0	G1 X40 W1.5 ;	Chamfering
N 0 2 5 0	G0 X42 W30 ;	Keep the width of the grooving
N 0 2 6 0	G1 X40 ;	
N 0 2 6 2	G1 X37 W1.5 ;	Chamfering
N 0 2 6 4	G1 X10 ;	Cut the Φ10 groove
N 0 2 6 6	G0 X17 Z-1 ;	
N 0 2 6 8	G1 X16 ;	
N 0 2 7 0	G1 X14 Z0 F200 ;	Chamfering
N 0 2 8 0	G0 X150 Z50 ;	Return to the tool change point
N 0 2 9 0	T0404 S100 ;	Changing for the No. 4 tool and set the spindle speed for 100
N 0 3 0 0	G0 X42 Z-20 ;	Approach the part
N 0 3 1 0	G92 X39 W-34 F3 ;	Thread-cutting cycle
N 0 3 2 0	X38 ;	Feed 1mm for the 2nd cutting
N 0 3 2 0	X37 ;	Feed 1mm for the 3rd cutting
N 0 3 3 0	X36.4 ;	Feed 0.6mm for the 4th cutting
N 0 3 3 2	X36 ;	Feed 0.4mm for the 5th cutting
N 0 3 4 0	G0 X150 Z50 ;	Return to the tool change point
N 0 3 5 0	T0100 U0 W0 ;	Change for the No.1 tool and execute its offset
N 0 3 6 0	M5 ;	Turn off the spindle
N 0 3 7 0	M9 ;	Turn off cooling
N 0 3 8 0	M13 ;	Unclamp the chuck
N 0 3 9 0	M30 ;	Program ends

14.2 Program Input

14.2.1 View a Saved Program

Press to enter program interface , and press to enter the local catalog page as follows:



In above window the names of the programs saved can be viewed for renaming the new program.

14.2.2 Creating a New Program

Enter the local catalog page in Edit mode, press address key O, choose a name that is not same with the ones in this window (i.e. 0001), key in the number key O, O, O, 1 and INSERT to create a new program as follows:

Input the characters of the above edit program one by one and the editing program is done.

14.3 Checkout a Program

14.3.1 Graphic Setting

Press GRAPH to enter graph interface as follows:

14.3.2 Program Check

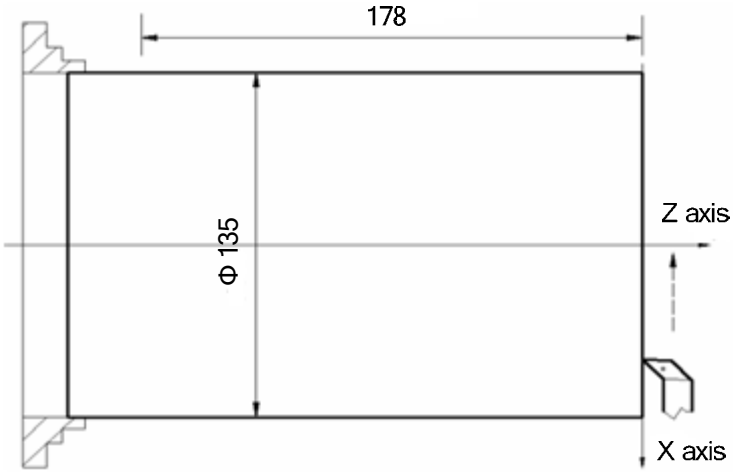
Press to enter graph display page. Press to enter Auto mode, press , , to make the auxiliary function lock indicator , machine lock indicator , and dry run indicator to enter the corresponding state. Press to start the drawing, press to automatically run programs, check the program accuracy by displaying the tool motion path, and the display page is the following after the run is completed:

When there is error in the program path, analysize the error and alter the part program, make another checkout for the program by the method above till the error is eliminated.

14.4 Tool-setting & Running

1) Move the tool to a safe position, execute T0100 U0 W0 command in PROGRAM->MDI page in MDI mode, and cancel the tool offset;

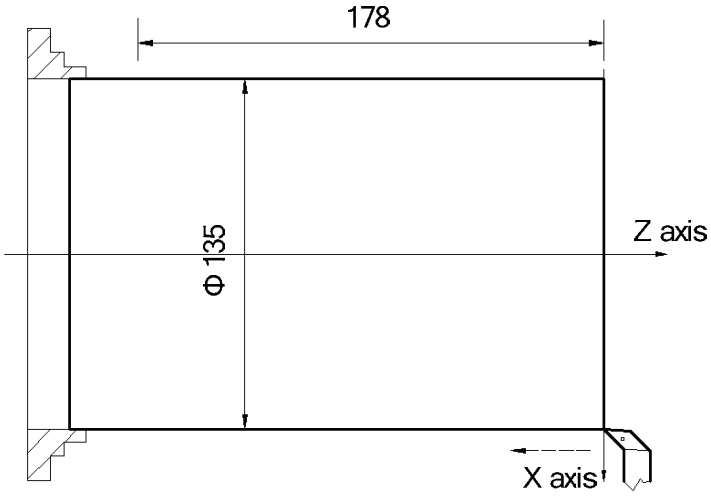
2) Move the tool to cut in the part end surface;



3) Release the tool along X when Z does not move, and stop the spindle, execute G50 Z0 in PROGRAM->MDI page in MDI mode to set the coordinate of Z axis;

4) Switch to tool offset page and input Z0 to No.001 offset;

5) Move the tool and make it to cut along the outer circle of the part;



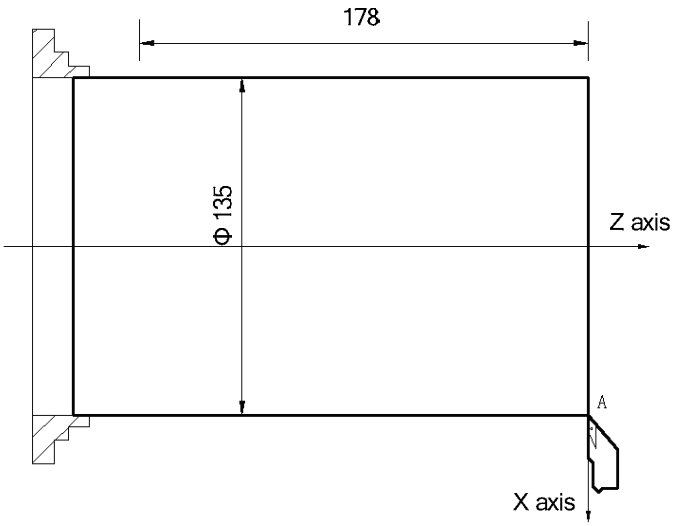
6) Release the tool along Z when X does not move, and stop the spindle, measure the dimensions of the outer circle of the part (e.g. The measuring value is 135mm);

7) Execute G50 X135 command in PROGRAM>MDI page in MDI mode to set the coordinate of X axis;

8) Switch to tool offset page, and input X135 to No.001 offset;

9) Move the tool to a safe position, and change No. 002 tool;

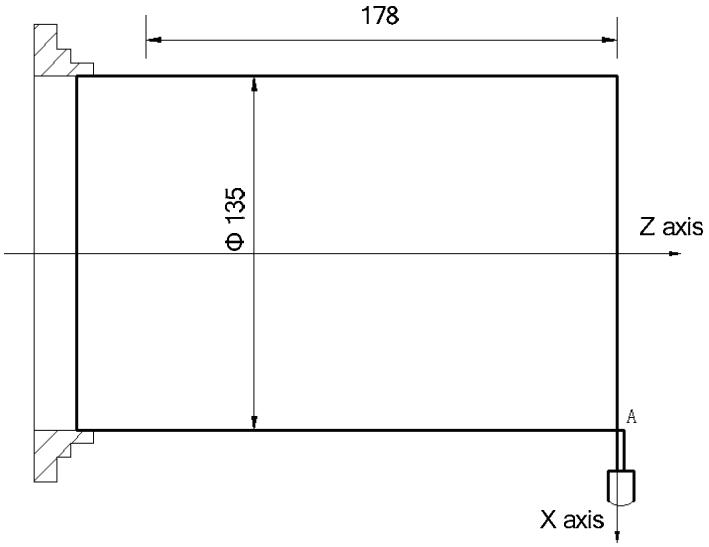
10) Start the spindle and move the tool to the tool-setting point, as A point in the following figure;



11) Switch to TOOL OFFSET window, move the cursor to No.002 offset and input X135 Z0;

12) Move the tool to a safe position, and change the No. 003 tool ;

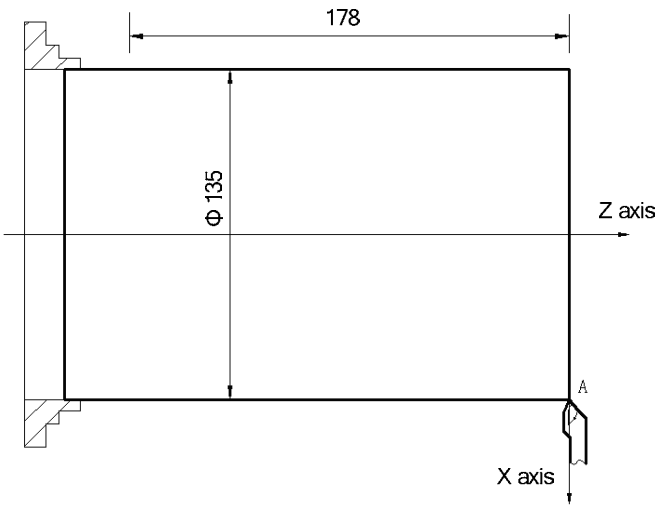
13) Start the spindle and move the tool to the tool-setting point, as A point in the following figure;



14) Switch to TOOL OFFSET window, move the cursor to No.003 offset and input X135 Z0;

15) Move the tool to a safe position, and to change the No. 004 tool;

16) Move the tool to the tool-setting point, as point A in the following figure;



17) Switch to tool offset, move the cursor to No.004 offset and input X135 Z0;

18) Move the tool to a safe position after the tool-setting is finished;

19) Start the machining in Auto mode;

20) If there is any error between the designed and the actual dimensions, the tool offset may be altered till the part dimensions are within the tolerance.

Note: Press key to make the auto running to pause if dwell is needed during the machining. Also if emergency occurs, the user presses the key, Emergency stop button to cut off the power to terminate the program running.

III Connection

CHAPTER 1 INSTALLATION LAYOUT

1.1 GK980TDi System Connection

1.1.1 GK980TDi Rear Cover Interface Layout

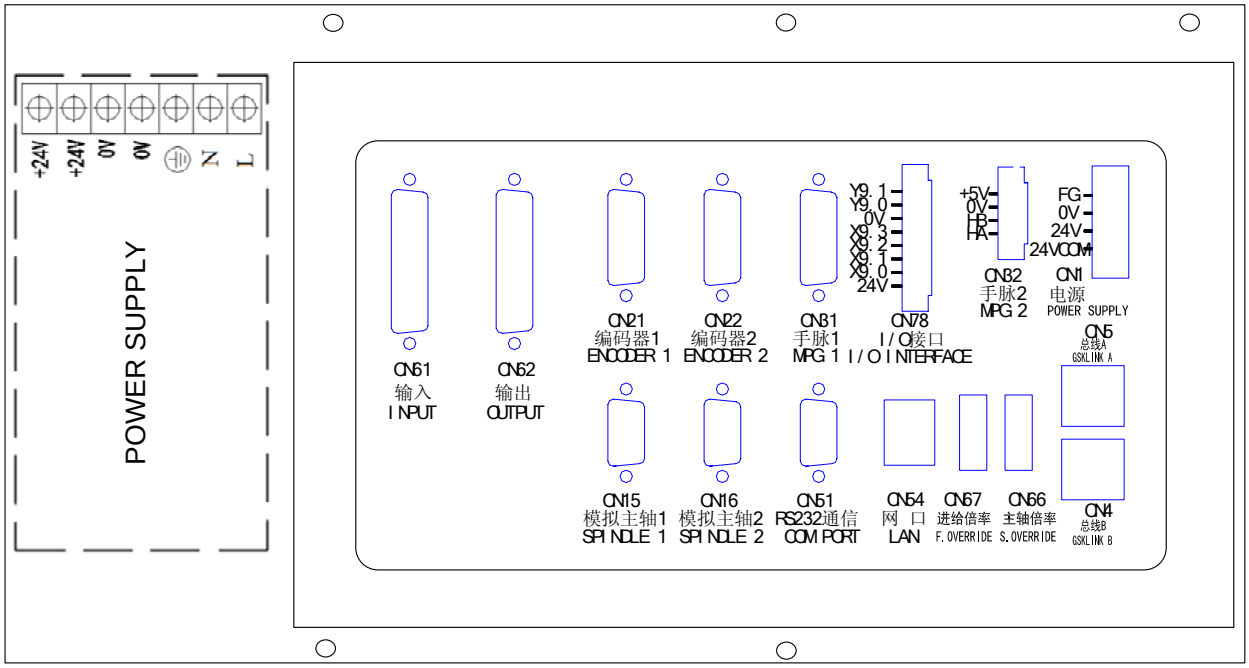


Fig. 1-1 GSK980TDi rear cover interface layout

1.1.2 Interface Explanation

- CN1: power supply interface
- CN4, CN5:GSKLINK bus connection interface B,A are connected with the servo unit and IO unit
- CN15:Analogy spindle 1,male socket of 9-cord in type D，connect with analogy spindle 1
- CN16:Analogy spindle 2,male socket of 9-cord in type D，connect with analogy spindle 2
- CN21: encoder, pin15 D male,connect with spindle encoder 1
- CN22: encoder, pin15 D male,connect with spindle encoder 2
- CN31: MPG, pin26 D male, connect with MPG
- CN51: communication, pin9 D female, connect PC RS232 interface
- CN54:Ethernet interface, plug are connected with the PC to transmit data
- CN61: input, pin44 D male, connect with machine input
- CN62: output, pin44 D female, connect with machine output
- CN66:the spindle override input signal, 4 input interfaces are connected with the additional panel
- CN67:the feedrate override input signal, 5 input interfaces are connected with the additional panel
- CN78:4 input/2 output interfaces, 8 connection terminals, machine input,output

1.2 GSK980TDi Installation

1.2.1 GSK980TDi External Dimensions

See Appendix Ⅰ, Ⅱ.

1.2.2 Preconditions of 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℃ 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. 1-2), RC circuit should approach to inductive loading as close as possible; reversely connect parallel freewheeling diode at both ends of DC winding (Fig. 1-3); connect parallel surge absorber at the ends of AC motor winding (Fig. 1-4);

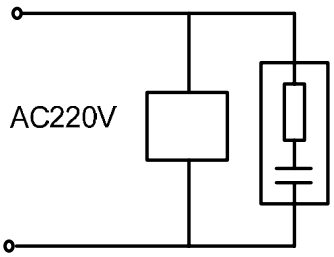


Fig.1-2

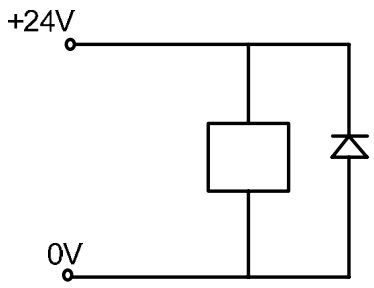


Fig. 1-3

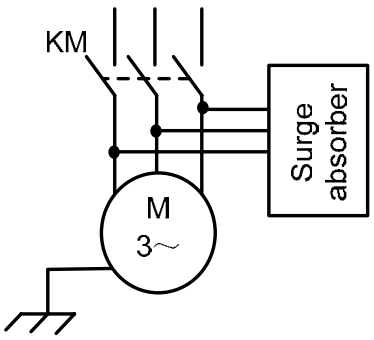


Fig. 1-4

4. The CNC leadout cables use the twisted shield cable or shield cable, the cable shield tier is grounded by an terminal at CNC side, signal cable should be as short as possible;
5. To reduce the mutual interference among the CNC signal cables, and among the strong current, the wiring should follow the following:

Table 1-1 The Wiring requirement

Group	Cable type	Wiring requirement
A	AC power cable	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 and they are twisted pair cables.
	Position feedback cable	
	Position encoder cable	
	Handwheel（MPG）cable	
	Other cables for shield	

CHAPTER 2 DEFINITION & CONNECTION OF INTERFACE SIGNALS

2.1 GSKLink Bus Interface

Bus interfaces of GSK980TDi are CN4 and CN5 (GSKLinkB and GSKLinkA), which is connected with the GSKLink bus communication function, the feedrate servo drive unit, spindle drive unit and extended I/O communication. GSKLink bus communication uses a loop control, and therefore, CN4 AND CN5 interfaces should be used together.

GSKLINK bus connection is shown below:

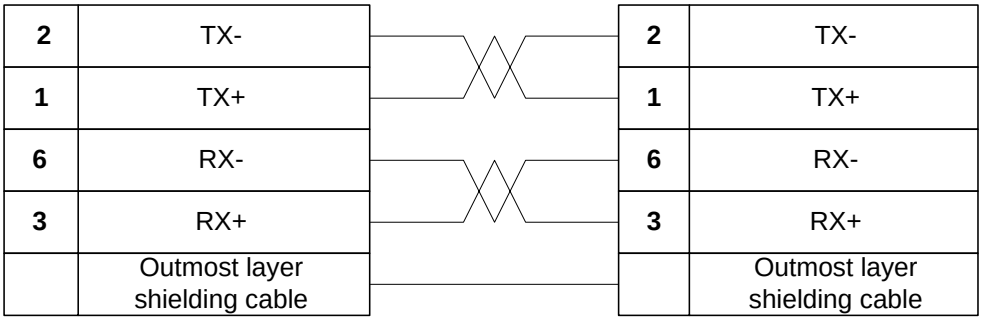


Fig. 2-1 GSKLink communication connection

2.2 Being Connected with Spindle Encoder

2.2.1 Spindle Encoder Interface Definition

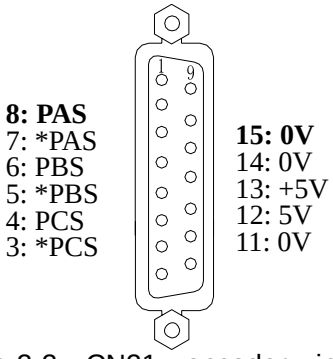


Fig.2-2 CN21 encoder interface (15-core D type male socket)

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 the encoder C, B, A phases differential input signals respectively, which are received by 26LS32; *PAS/PAS,*PBS/PBS are orthogonal square wave with phase shift 90°and their maximum signal frequency is less than 1MHz; the encoder pulses for GSK980TDi are set at will by parameter, the setting range is from 100 to 5000.

Its interior circuit is shown in Fig. 2-13: (n=A, B, C)

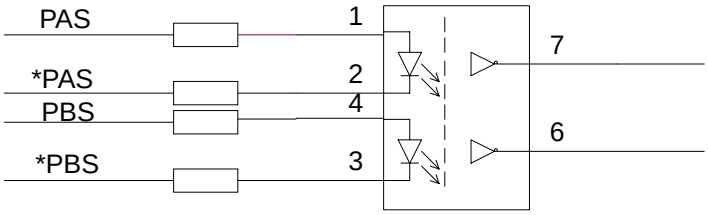


Fig. 2-3 Encoder signal circuit

2.2.3 Spindle Encoder Interface Connection

GSK980TDi is connected with spindle encoder shown in Fig. 2-4, and it uses twisted pair cables. (exemplified by CHANGCHUN YIGUANG ZLF-12-102.4BM-C05D encoder):

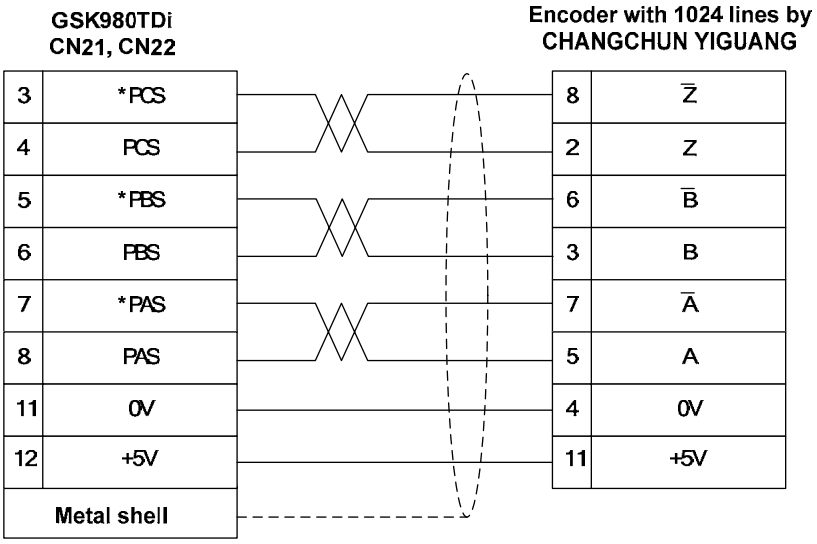


Fig. 2-4 GSK980TDi is connected with the encoder

2.3 MPG Connection

2.3.1 MPG Interface Definition

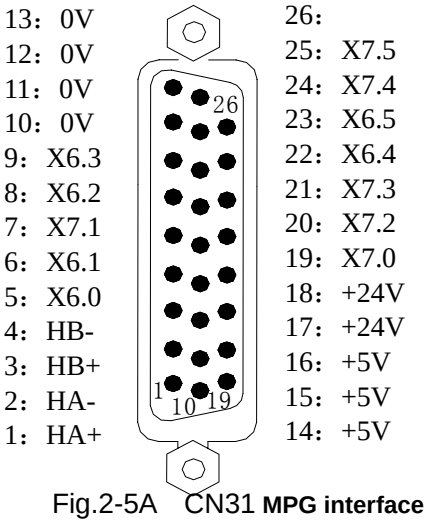


Fig.2-5A CN31 **MPG interface (26-core DB type male socket)**

Signal	Explanation
HA+,HA-	MPG A phase signal
HB+,HB-	MPG B phase signal
+24V	DC power supply
+5V,0V	
Standard PLC Definition	
Address	**Function**
X6.0	X MPG axis selection
X6.1	Y MPG axis selection
X6.2	Z MPG axis selection
X7.0	4TH_axis selection signal
X7.1	5TH_axis selection signal
X6.3	Increment0.001
X6.4	Increment0.01
X6.5	Increment0.1

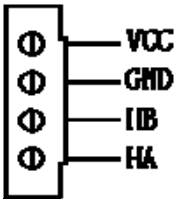


Fig. 2-5B CN32MPG interface (4-point wiring terminal)

Signal	Explanation
HA	MPG A phase signal input
HB	MPG B phase signal input
GND	0V
VCC	5V

2.3.2 Signal Explanation

HA,HB are the MPG A, B phase input signals respectively. Their interior circuit is shown in Fig. 2-6:

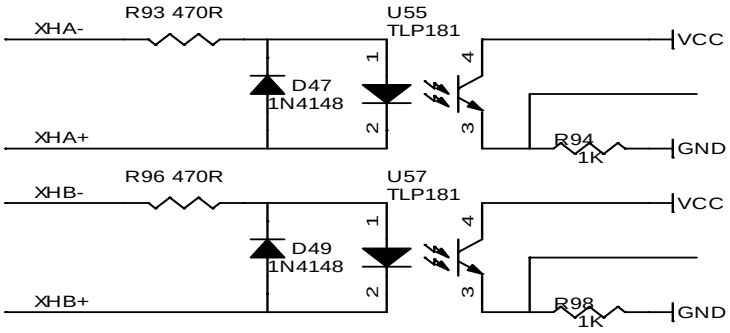


Fig. 2-6 MPG signal circuit

GSK980TDi is connected with MPG shown in Fig. 2-7:

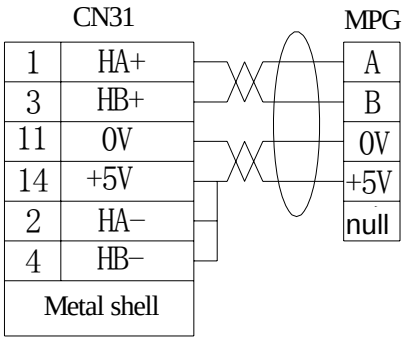


Fig. 2-7A CN31 connects with the single-port MPG input

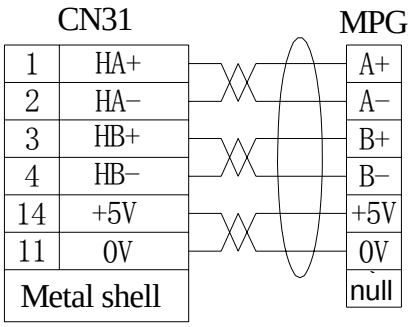


Fig. 2-7B CN31 connects with the difference MPG input

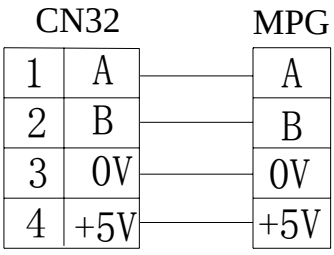


Fig. 2-8 CN32 connects with MPG

2.4 Analog Spindle Interface

2.4.1 Analog Spindle Interface Definition

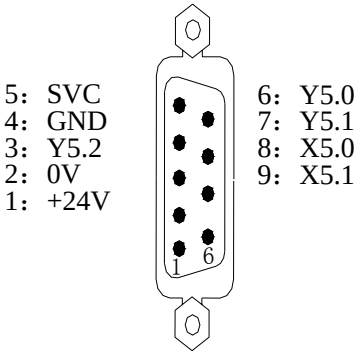


Fig.2-9 CN15 analogy spindle interface（male socket of 9-cord in type D）

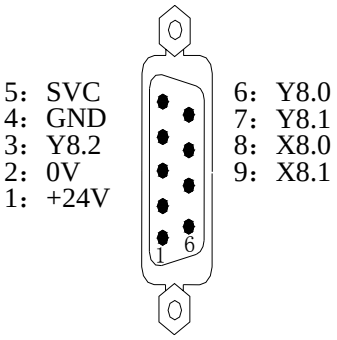


Fig.2-10 CN16 analogy spindle interface（male socket of 9-cord in type D）

Signal	Explanation
SVC	0~10V Analog voltage
GND	Signal grounding

Standard PLC Definition	
Address	**Function**
X5.0	Spindle 1 alarm signal
X5.1	
Y5.0	series spindle 1 rotation signal CW(positive)
Y5.1	series spindle 1 rotation signal CCW(negative)
Y5.2	
X8.0	Spindle 2 alarm signal
X8.1	
Y8.0	series spindle 2 rotation signal CW(positive)
Y8.1	series spindle 2 rotation signal CCW(negative)
Y8.2	

2.4.2 Inverter Connection

The analog spindle interface SVC may output 0~10V voltage, its interior signal circuit is shown in Fig. 2-11:

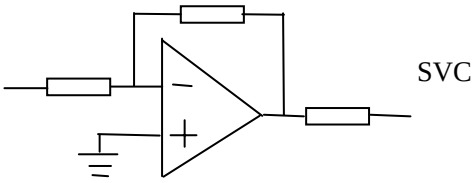


Fig. 2-11 SVC signal circuit

GSK980TDi is connected with the inverter shown in Fig. 2-12:

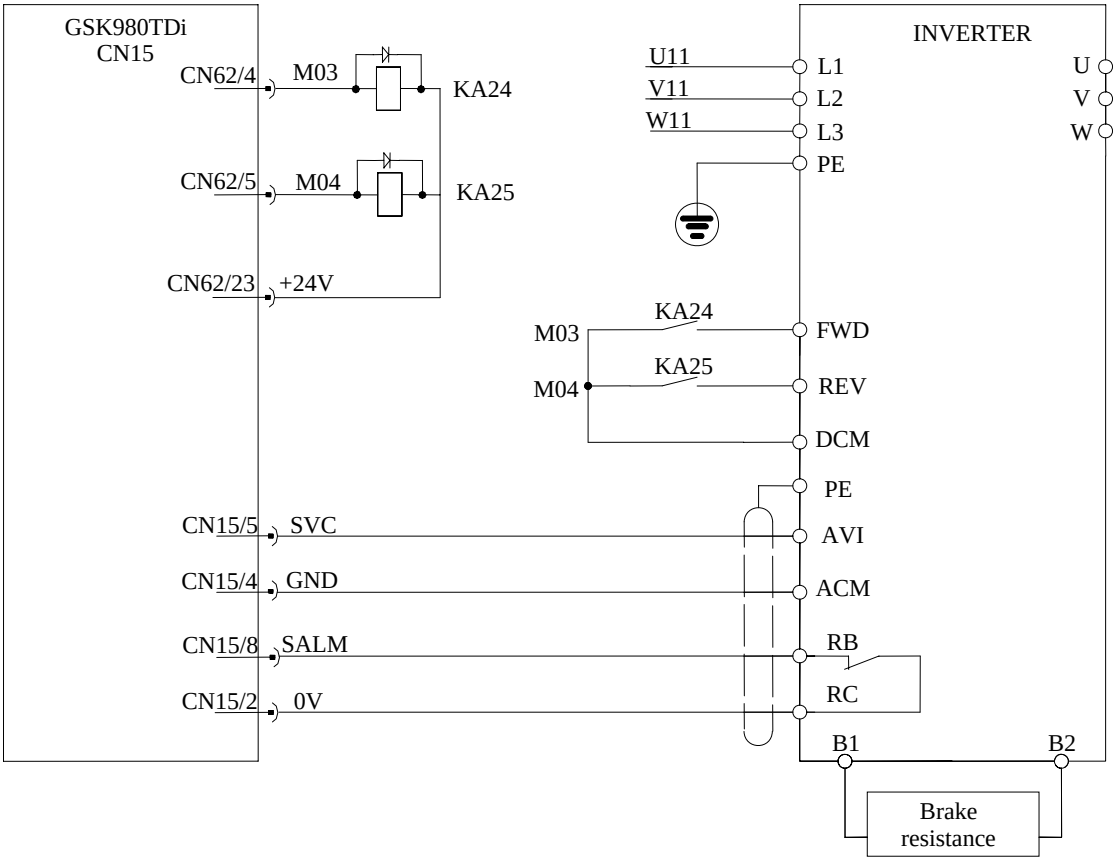
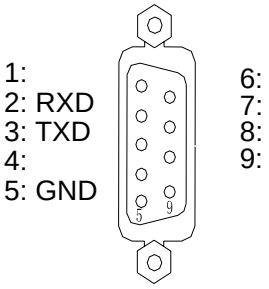


Fig. 2-12 GSK980TDi is connected with the inverter

2.5 GSK980TDi / GSK980TDi-V being Connected with PC

2.5.1 Communication Interface Definition



Signal	Explanation
RXD	Receiving data
TXD	Transmitting data
GND	Signal grounding

Fig. 2-13 CN51 communication interface (DB9-female)

2.5.2 Communication Interface Connection

GSK980TDi can perform the communication by CN51 and PC(optional communication software). GSK980TDi is connected with PC shown in Fig 2-14:

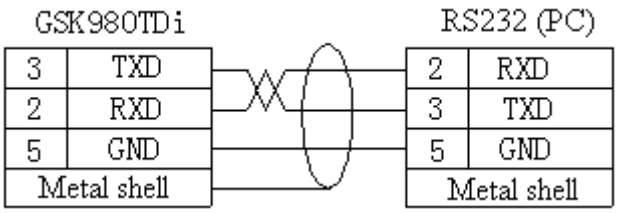


Fig. 2-14 GSK980TDi is connected with PC

2.6 Network Interface

Network interfaces of GSKTDi CN54 are standard and their definitions are shown below:

PIN NO.	Signal	PIN NO.	Signal
1	TXDLAN+	9	LINK_LED
2	TXDLAN-	11	LAN_LED
3	RXDLAN+	10, 12	VDD33
6	RXDLAN-	13, 14	Shell grounding

Note: TXD+ and TXD- are difference signal, as well the **RXD+ and RXD-**, which are required to connect with twisted-pair.

2.7 Power Interface Connection

The power box is applied in GSK980TDi system, which involves 2 groups of voltage: +24V. The connection of power box to GSK980TDi CN1 interface has been done for its delivery from factory, and the user only need to connect it to a 220V AC power in using.

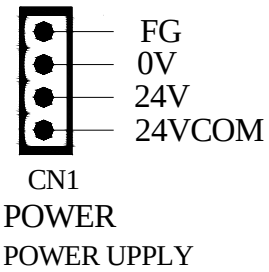


Fig. 2-15 system power interface CN1

24VOM is connected with the output interface CN62 24V. When GSK980TDi's power supply box supplies power supply for an external electron device, the supply box's +V is connected with 24VCOM. When an external power supply is used, its 24V is connected with 24VCOM and its 0V must be connected with the system's supply box's-V.

2.8 I/O Interface Definition

Note!

The I/O function significances of the unlabelled fixed addresses of this GSK980TDi/GSK980TDi-V turning machine CNC system are defined by PLC programs (ladders), and they are defined by the machine builder when matching with a machine, please refer to the manual by the machine builder.

The fixed address I/O function not be marked are described for GSK980TDi PLC. The described contents without other special explanation are also applied to GSK980TDi-V.

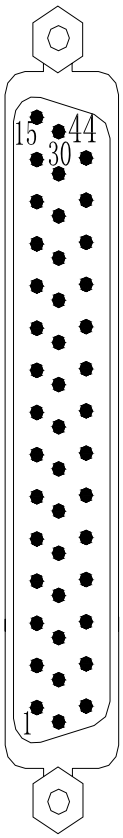


Fig.2-16 CN61 inputinterface (male socket of 44-cord in type D)

Pin	Address	Function	Explanation
21～24	0V	Power supply interface	
17～20 25～28	Floating	Floating	Floating
1	X0.0	SAGT	Guard door check signal
2	X0.1	Reserved	Reserved
3	X0.2	DIQP	Chuck input signal
4	X0.3	DECX(DEC1)	X deceleration signal(fixed address)
5	X0.4	DITW	Tailstock control signal
6	X0.5	ESP	External emergency stop signal (fixed address)
7	X0.6	PRES	Pressure check signal
8	X0.7	T05	Tool signal 5 /Sensor E (LIO_SHING Tool Post)/YA_XING locking signal
9	X1.0	T06/ strobe	Tool signa l 6/ strobe signal (YantaiAK31)/Sensor F(LIO_SHING Tool Post)
10	X1.1	T07/ pregraduation	Tool signal 7 /OV3/pregraducation proximity switch(YantaiAK31)/YA_XING releasing
11	X1.2	T08/ tool post worktable overheat	Tool signal 8//tool post worktable overheat check (YantaiAK31)
12	X1.3	DECZ(DEC3)	Z deceleration signal(fixed address)
13	X1.4	Reserved	Reserved
14	X1.5	M41I	Shifting gear to 1st gear in-position
15	X1.6	M42I	Shifting gear to 2nd gear in-position
16	X1.7	T01	Tool signal / Sensor A(LIO_SHING Tool Post)
29	X2.0	T02	Tool signal / Sensor B (LIO_SHING Tool Post)
30	X2.1	T03	Tool signal / Sensor C (LIO_SHING Tool Post)
31	X2.2	T04	Tool signal / Sensor D (LIO_SHING Tool Post)
32	X2.3	DECY(DEC2)	Y deceleration signal(fixed address)
33	X2.4	DEC4/LUBS	4th deceleration signal(fixed address)/lubricating oil detection
34	X2.5	DEC5	5th deceleration signal(fixed address)/
35	X2.6	TCP	Tool post clamping signal
36	X2.7	EXHP	external hydraulic control input
37	X3.0	LMIX	X overtravel
38	X3.1	LMIY	Y overtravel
39	X3.2	LMIZ	Z overtravel
40	X3.3	WQPJ/VPO2	Chuck releasing in-position signal
41	X3.4	NQPJ/SALM2	Chuck clamping in-position signal
42	X3.5	SKIP	G31 skip signal
43	X3.6	AEX	X tool measure position arrival signal (G36)
44	X3.7	AEZ	Z tool measure position arrival signal (G37)

Note: Partial input interface is the fixed address for some functions, for example: the signal of ESP, DECX, DECZ or SKIP, etc. If this function does not use, it will be redefined in PLC.

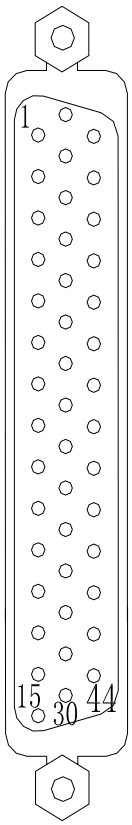


Fig.2-17 CN62output interface(female socket of 44-cord in type D)

Pin	Address	Function	Explanation
17,18,19, 26,27,28	0V	Power supply interface	Power supply 0V terminal
20～25	+24V	Power supply interface	Power supply ＋24V terminal
1	Y0.0	M08	Cooling output
2	Y0.1	M32	Lubricating output
3	Y0.2	HOLD	HOLD release output
4	Y0.3	M03	Spindle rotation (CCW)
5	Y0.4	M04	Spindle rotation (CW)
6	Y0.5	M05	Spindle stop
7	Y0.6	SCLP	spindle clamped output
8	Y0.7	SPZD	Spindle brake
9	Y1.0	S1/M41	Spindle machine gear output 1
10	Y1.1	S2/M42	Spindle machine gear output 2
11	Y1.2	S3/M43	Spindle machine gear output 3
12	Y1.3	S4/M44	Spindle machine gear output 4
13	Y1.4	DOQPJ(M12)	Chuck clamping output
14	Y1.5	DOQPS(M13)	Chuck releasing output
15	Y1.6	TL+	Tool post CCW rotation
16	Y1.7	TL-	Tool post CW rotation
29	Y2.0	TZD/TLS	Tool post worktable brake (YantaiAK31) / tool pot releasing (LIO_SHING Tool Post)
30	Y2.1	INDXS/TCLP	Tool post worktable pregraduation coil (YantaiAK31) / tool pot lock (LIO_SHING Tool Post)
31	Y2.2	YLAMP	Three-color lamp-yellow
32	Y2.3	GLAMP	Three-color lamp -green
33	Y2.4	RLAMP	Three-color lamp -red
34	Y2.5	DOTWJ(M10)	Tailstock forward
35	Y2.6	DOTWS(M11)	Tailstock backward
36	Y2.7	SLK	Protective door lock
37	Y3.0		reserved
38	Y3.1	HPST	Hydraulic control output
39	Y3.2	UO0/WKLP	User macro output 0/ indicator
40	Y3.3	UO1/EMOT1	User macro output 1/ Extended M function output signal 1
41	Y3.4	UO2/EMOT2	User macro output 2/ Extended M function output signal 2
42	Y3.5	UO3	User macro output 3
43	Y3.6	UO4	User macro output 4/Chip-cleaner forward
44	Y3.7	UO5	User macro output 5/ chip-cleaner backward

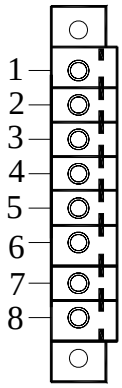


Fig.2-18 CN78

Pin.No	Address	Function	Explanation
1	Y9.1	SPLAMP	Extended pause lamp
2	Y9.0	STLAMP	Extended start lamp
3	0V	0V	Power supply 0 V
4	X9.3	EXESP	Extended emergency stop signal
5	X9.2	reserved	reserved
6	X9.1	SP	Extended pause
7	X9.0	ST	Extended cycle start
8	+24V	24V	Power supply + 24V

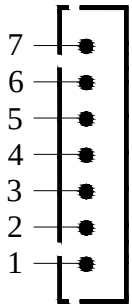
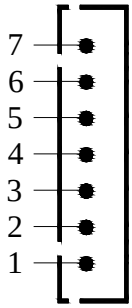


Fig.2-19 CN66

Pin.No	Address	Function	Explanation
1	X29.4	BAUD1_A	Spindle override wave-band switch signal 0
2	X29.6	BAUD1_B	Spindle override wave-band switch signal 2
3	Floating	Floating	Floating
4	+24V	24V	Power supply ＋24V
5	X29.7	BAUD1_E	Spindle override wave-band switch signal 3
6	X29.5	BAUD1_F	Spindle override wave-band switch signal 1
7	INH	BAUD1_INH	INH



Pin.No	Address	Function	Explanation
1	X29.0	BAUD2_A	Feedrate wave-band switch signal 0
2	X29.2	BAUD2_B	Feedrate wave-band switch signal 2
3	Floating	Floating	Floating
4	+24V	24V	Power supply 24V
5	X29.3	BAUD2_E	Feedrate wave-band switch signal 3
6	X29.1	BAUD2_F	Feedrate wave-band switch signal 1
7	INH	BAUD2_INH	INH

Fig.2-20 CN67

Note 1: Various functions can be defined to some of the input and output interfaces, and they are indicated by "/"sign in the table above.

Note 2: If output function is active, the output signal is through on to 0V. If output function is inactive, the output signal is cut off by high impedance.

Note 3: If input function is active, the input signal is through on to +24V. If input function is inactive, the input signal is cut off with it.

Note 4: The effectiveness of +24V, COM terminals are equivalent to those of the GSK980TDi/GSK980TDi power box terminals that have the same names.

2.8.1 Input Signal

Input signal means the signal from machine to CNC, when this signal is through on with +24V, the input is active; when it is off with +24V, the input is inactive. The contact of input signal at machine side should meet the following conditions:

Capacity of the contact: DC30V, 16mA above

Leakage current between contacts in open circuit: 1mA below

Voltage drop between contacts 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 contact switch whose signals are from keys, stroke switch and contacts of relay at machine side, as shown in Fig. 2-21A:

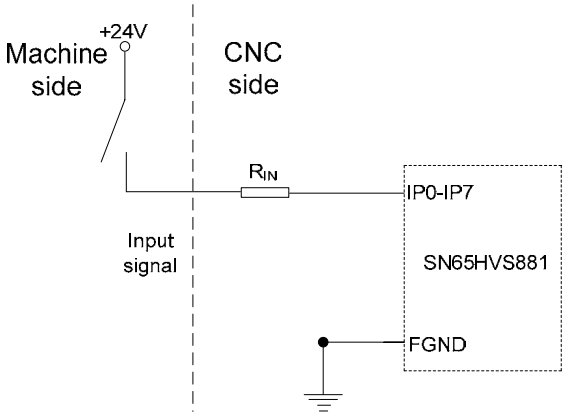


Fig. 2-21A

The other type is input by switch with no contacts (transistor) as shown in Fig. 2-21B, 2-21C:

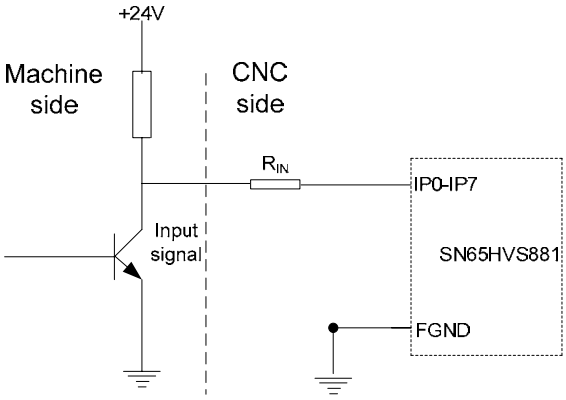


Fig. 2-21B Connection of NPN type

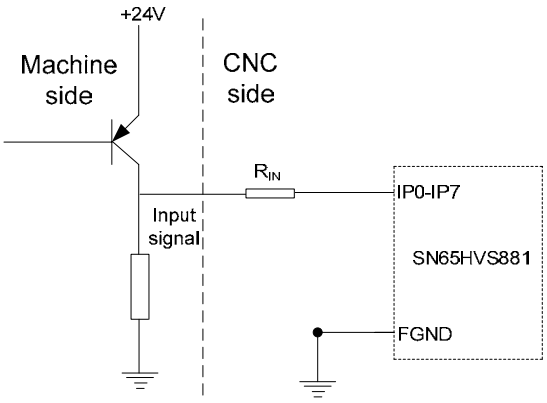


Fig. 2-21C Connection of PNP type

The input interface signals defined by PLC of GSK980TDi / GSK980TDi-V system involve XDEC, ZDEC, ESP, ST, SP/SAGT, PRES, DITW, DIQP, T01～T08, TCP and so on.

2.8.2 Output Signal

The output signal is used for the machine relay and indicator. When the output is enabled, the output state of its corresponding Y address is 1, and its output signal potential is 0V. When the output is disabled, the output state of its corresponding Y address is 0, this output signal represents as high-impedance state. The digital value output of I/O interfaces are owned the same structure, refer to the following figure:

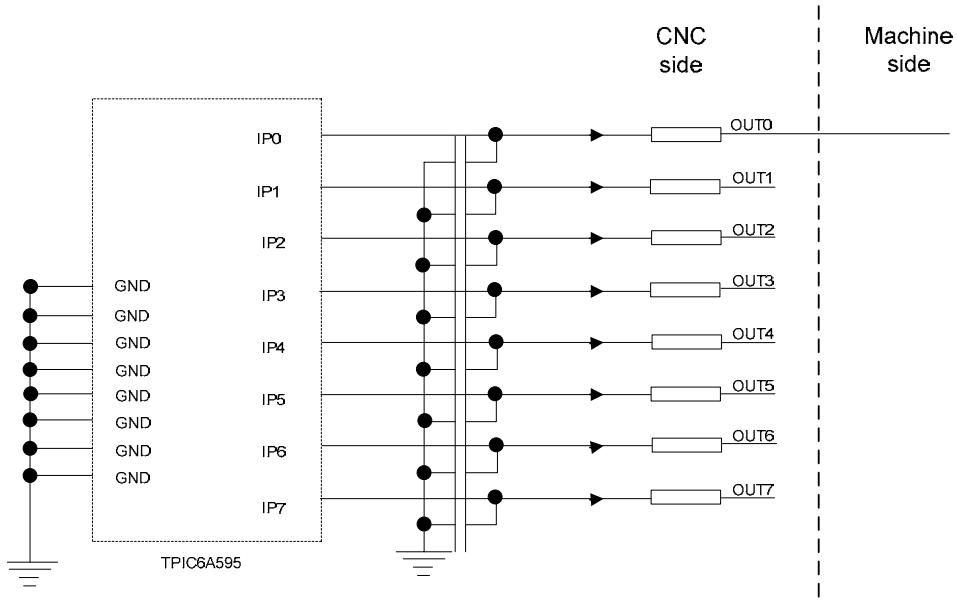


Fig. 2-22 Circuit for digital volume output module

Its typical application is as follows:

- To drive LED

A serial resistance is needed to limit the current (usually 10mA) that goes through the LED by using output to drive LED, which is shown in Fig. 2-23:

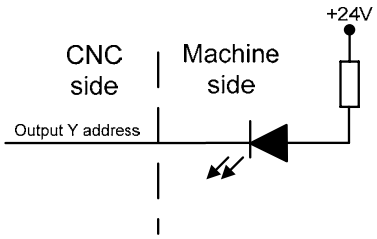


Fig. 2-23

- To drive filament indicator

An external preheat resistance is needed to decrease the current impact at power on by using output to drive filament indicator, and this resistance value should be within a range that the indicator cannot be lighted up as shown in Fig. 2-24:

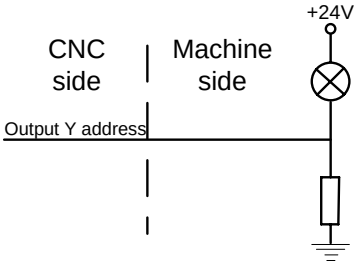


Fig. 2-24

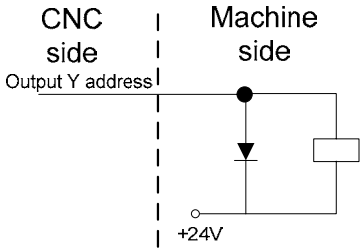


Fig.2-25

- To drive inductive load (such as relay)

To use output to drive an inductive load, it requires connecting a freewheeling diode near the coil to protect output circuit and reduce interference as shown in Fig. 2-25:

The significances of the output signal of I/O interface are defined by PLC program, these signals include S1～S4 (M41～M44), M3, M4, M5, M8, M10, M11, M32, TL-, TL+, U00～U05, DOQPJ, DOQPS, SPZD signals etc.. And their common terminal is +24V.

2.9 I/O Function & Connection

Note！

The I/O function significance of this GSK980TDi / GSK980TDi-V turning machine CNC system is defined by PLC programs (ladders), and they are defined by the machine builder when matching with a machine, please refer to the manual by the machine builder.

The fixed address I/O function not be marked are described for GSK980TDi PLC. The described contents without other special explanation are also applied to GSK980TDi-V.

2.9.1 Stroke Limit & ESP

- **Relevant signal**

ESP: emergency stop signal, alarm issued if the system is not connected with +24V
EXESP: External emergency stop signa, when not be connected with +24V emergency alarm
LMIX: X overtravel limit check input
LMIY: Y overtravel limit check input
LMIZ: Z overtravel limit check input

- **Diagnosis data**

0	**0**	**0**
Pin		

ESP							
CN61.6							

- **Signal diagnosis**

Signal	ESP	EXESP	LMIX	LMIY	LMIZ
Diagnosis address	X0.5	X9.3	X3.0	X3.1	X3.2
Interface pin	CN61.6	CN78.4	CN61.37	CN61.38	CN61.39

- **Control parameter**

Bit parameter

1	7	2						ESP			

ESP =0: Check ESP signal

=1: Do not check ESP signal

- **PLC bit parameter**

K	1	0	LMIT	LMIS	EXESP					

LMIT =1: Travel limit check function of each axis is valid.

=0: Travel limit check function of each axis is invalid

LMIS =1: The system alarms for overtravel when the travel limit check signal is not connected with +24V.

=0: The system alarms for overtravel when the travel limit check signal is connected with +24V

EXESP =1:External emergency stop signa is valid。

=0:External emergency stop signa is invalid。

- **Signal connection**

The ESP signal circuit is shown in Fig.2-26:

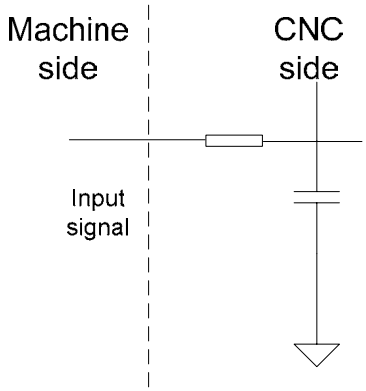


Fig. 2-26 ESP signal circuit

- **Machine external connection**

(1) The series connection between the emergency stop and travel switch is shown in Fig. 2-27:

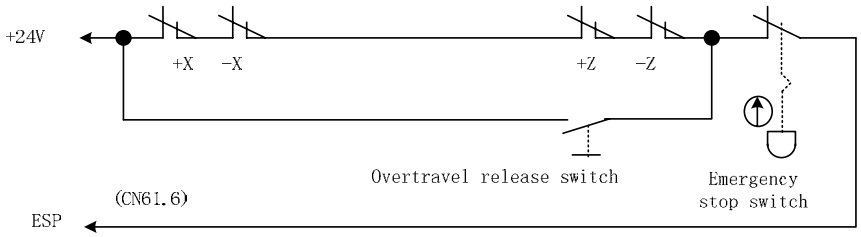


Fig.2-27 Series connection between emergency stop and travel switch

(2) The separate connection between the emergency stop and travel switch is shown in Fig. 2-34B:

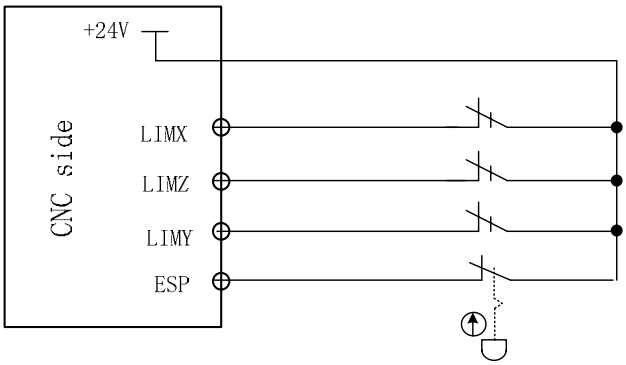


Fig. 2-28 Separate connection between the emergency stop and travel switch

- **Control logic**

(1) Series connection between the emergency stop and travel switch

When the contact of the emergency stop switch is cut off, the ESP signal is off to +24V, and CNC makes an emergency alarm. Meanwhile the CNC turns off the enable (EN) signal to stop the pulse output. Except the functions processed by NC, other functions can also be defined by PLC program when the emergency alarm is issued. The function defined by standard PLC program is: when emergency alarm is issued, it turns off the signal output of M03, M04, M08, and outputs M05 signal at the same time.

(2) Separate connection between the emergency stop and travel switch

1. Each axis has only one overtravel contact, and the system judges the overtravel alarm based on the axis movement direction.
2. When the system alarms for the overtravel, the axis moves reversely, the reset key is pressed to clear the alarm after the axis exceeds the limit position.

Note: Before the overtravel limit function is enabled, the slider must be between the positive and the negative limit; otherwise, the prompted alarm will not be consistent with actual situation.

2.9.2 Tool Change Control

- **Relevant signals (by standard PLC program)**

Ladder diagram supports 4 kinds of tool-post controllable logic; refer to the tool-change logic for the Signal Specification.

- **Control parameter**

Bit parameter

K	1	1

CHOT		CHET	TCPS	CTCP	TSGN	CHTB	CHTA

CHTA: tool change mode selection bit 0

CHTB: tool change mode selection bit 1（see following table）

CHTB	CHTA	Tool post type
0	0	Standard tool change mode B
0	1	Standard tool change mode A
1	0	Yantai AK31
1	1	Unused

TSGN =0 : tool signal HIGH(turn on +24V) is valid

=1 : tool signal LOW(turn off +24V) is valid

CTCP =0 : do not check tool post locking signal

=1 : check tool post locking signal

TCPS =0 : tool post locking signal LOW(turn off +24V) is valid
=1 : tool post locking signal HIGH(turn on +24V) is valid
CHET=0 : do not check tool signal after the tool change is completed
=1 : check tool signal after the tool change is completed
CHOT=0 : do not check tool post overheat
=1 : check tool post worktable overheat

D	T	0	4	TLMAXT

The maximum time limit for tool change

D	T	0	7	T1TIME

Tool change time T1: Tool post delay time from CCW stop to CW output (ms)

D	T	0	8	TCPCKT

Tool post lock signal detection time (ms)

D	T	0	9	TCPTIME

Tool change time T2: tool post CW rotation locking time

0	0	8	4	TMAX

Total tools selection

- **Signal connection**

1. The T01～T08, TCP signals input are employed with photocoupler, its interior circuit is shown in Fig. 2-29:

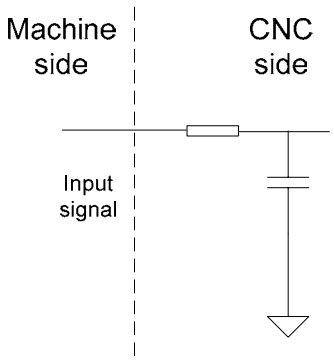


Fig. 2-29

2. TL+, TL- are tool post CCW/CW output signal, its interior circuit is shown in Fig.2-30:

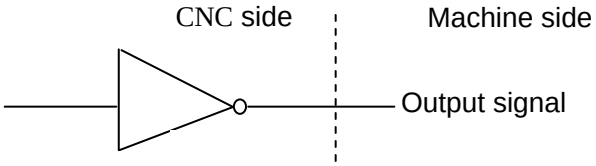


Fig. 2-30

3. The external circuit of the tool number signal is shown in Fig. 2-31, when the tool number signal is low level active, it requires an external pull-up resistor.

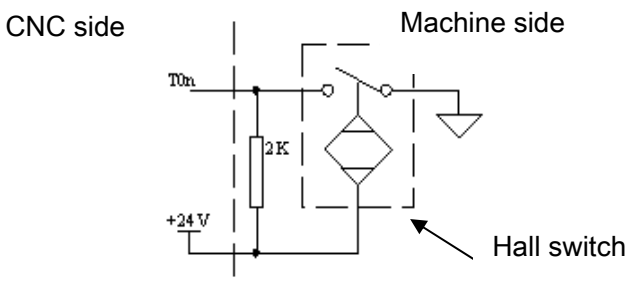


Fig. 2-31

- **Function description (defined by standard PLC program)**

The control sequence and control logic of the tool change are defined by PLC program. There are 4 tool change modes defined as follows by standard PLC program:

1. CHTB (K11.1)=0, CHTA (K11.0)=0: tool change mode B

1.1 **Signal specification**

Signal type	Code	Interface	Address	Function	Remark
Intput signal	CN61.16	X1.7	T01	tool signal	
	CN61.29	X2.0	T02	tool signal	
	CN61.30	X2.1	T03	tool signal	
	CN61.31	X2.2	T04	tool signal	
	CN61.8	X0.7	T05	tool signal	
	CN61.9	X1.0	T06	tool signal	
	CN61.10	X1.1	T07	tool signal	
	CN61.11	X1.2	T08	tool signal	
	CN61.35	X2.6	TCP	tool post locked signal	
Output signal	CN62.15	Y1.6	TL+	tool post CW rotation	
	CN62.16	Y1.7	TL-	tool post CCW rotation	

1.2 Tool-Change schedule

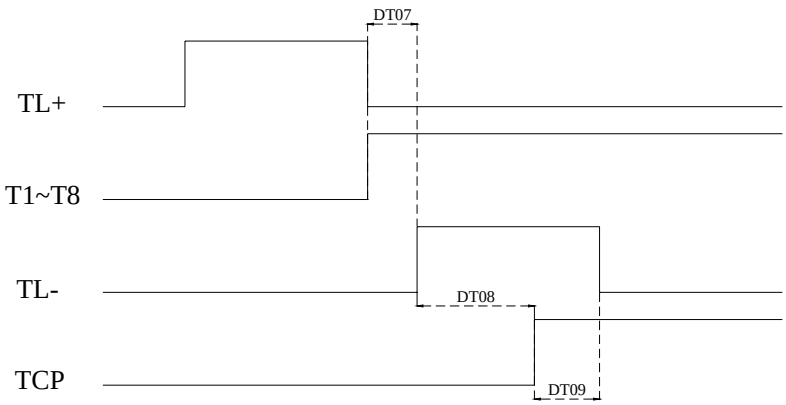
①During the tool change process, CNC outputs TL+ signal until the tool in-position signal is detected, then CNC turns off TL+ signal output and outputs TL- signal after a delay time specified by data parameter No.082. Then CNC detects TCP signal till it is detected, the CNC turns off TL- signal after a delay time specified by the data parameter No.085. So the tool change is over.

②When CHET(K0011.5) is set to 1(check tool signal after the tool change ends) and the tool post (CCW) rotation time ends to confirm whether the current tool input signal is consistent with the current tool No., if not, the system alarms.

③ The tool change process ends.

④ After the system outputs the tool post rotation (CCW) signal, if the CNC doesn't receive the TCP signal within the time set by DT08, an alarm will be issued and the TL- signal will be turned off.

⑤ When the tool post has no tool post locking signal, CTCP(K0011.3) is set to 0, at the time, the system does not check the tool post locking signal.



Sequence of tool change mode B

2. CHTB (K11.1)=0，CHTA (K11.0)=1: YA_XING hydraulic tool post HLT series:

2.1 Signal explanation

Signal type	Code	Interface	Add.	Function	Remark
Input signal	CN61.16	X1.7	T01	tool signal	Cutter position 8421 code
	CN61.29	X2.0	T02	tool signal	
	CN61.30	X2.1	T03	tool signal	
	CN61.31	X2.2	T04	tool signal	
	CN61.8	X0.7	TCP	tool post locked signal	Normally-closed contactor (Low LEVEL enabled)
	CN61.9	X1.0	SELE	tool post strobe signal	
	CN61.10	X1.1	TUCP	tool post released signal	Normally-closed contactor (Low LEVEL enabled)
Output signal	CN62.15	Y1.6	TL+	tool post CW rotation	
	CN62.16	Y1.7	TL-	tool post CCW rotation	
	CN62.29	Y2.0	TRLS	tool post released outout	
	CN62.30	Y2.1	TCLP	tool post locked outout	

Notice: The cutter-head clamping signal and releasing signal are normally-closed contactor, which are enabled in low LEVEL.

2.2 Tool-change schedule

① Cutter-head relases and waits for that the cutter-head releasing signal is low LEVEL.

② The system judges the shortest path based upon the destination tool number and current tool number; select the output rotation direction in terms of the principle of the approximated tool-selection regardless of the positive signal (TL+) or negative one (TL-), the tool starts to rotate and perform the tool-selection.

③ In the rotation process, the system performs the coding based upon the cutter position encoder signals T1~T4 to identify the current tool number. When rotating to the previous

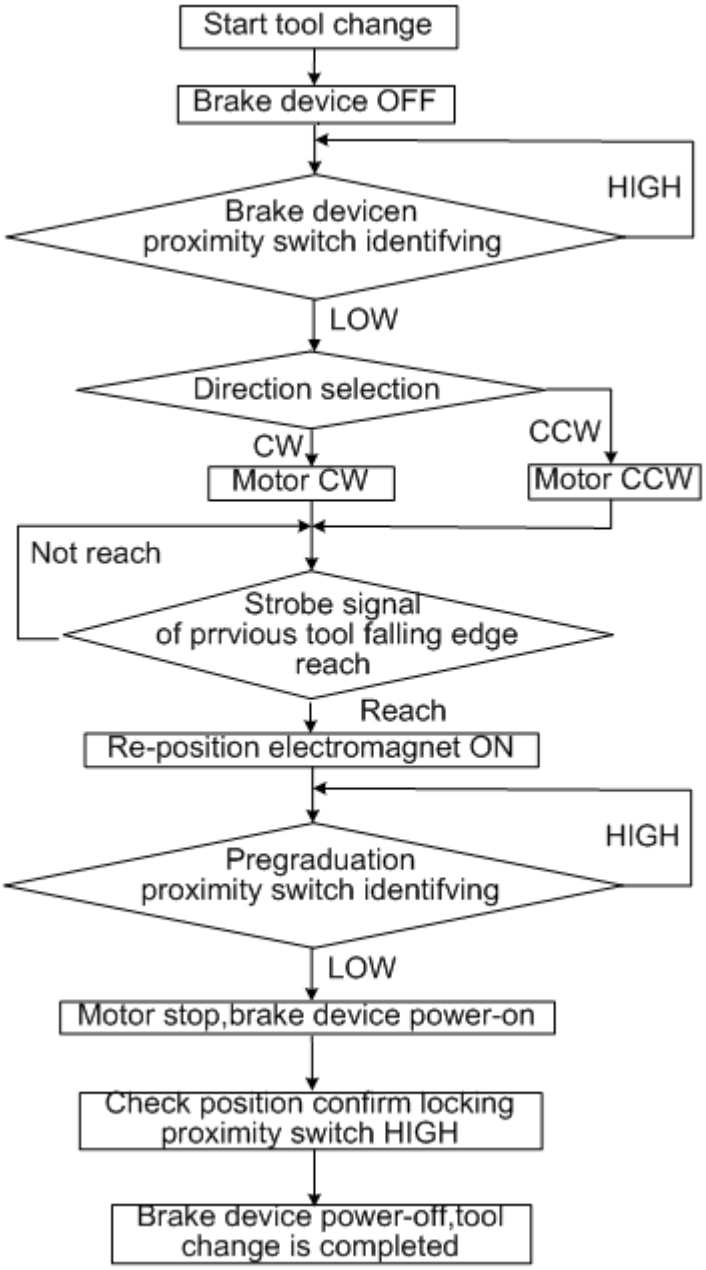
station of the destination tool number, detect the skip of the cutter-head strobe signal. System closes the cutter-head releasing signal and outputs the cutter-head clamping signal at the falling-edge of the previous state strobe signal of the destination tool number.

④ When the cutter-head clamping input signal in detecting is low LEVEL, close the cutter-head rotation output signal (TL+ or TL-), the motor is then stopped.

⑤ Confirm whether the current tool position code signal is consistent with the detstination tool number, confirm whether the cutter-head clamping signal is Low LEVEL again.

⑥ The tool-change is absolutely performed if the above-mentioned steps are correct.

2.3 Tool-Change Schedule



2. CHTB（K11.1）=1，CHTA（K11.0）=0 Yantai AK31 series (6, 8, 10, 12 tools) Chanzhou SBWD-80

3.1. Signal specification

Signal type	Code	Interface	Address	Function	Remark
Input signal	CN61.16	X1.7	T01	tool signal	Tool position is 8421 encoder
	CN61.29	X2.0	T02	tool signal	
	CN61.30	X2.1	T03	tool signal	
	CN61.31	X2.2	T04	tool signal	
	CN61.35	X2.6	TCP	Tool post locked signal	
	CN61.9	X1.0	SELE	strobe signal	
	CN61.10	X1.1	INDX	Tool post worktable pregraduation coil	
	CN61.11	X1.2	CHOT	tool post overheat check	
Output signal	CN62.15	Y1.6	TL+	Tool post CW rotation	
	CN62.16	Y1.7	TL-	ool post CCW rotation	
	CN62.29	Y2.0	TZD	Tool post worktable brake	
	CN62.30	Y2.1	INDXS	Tool post worktable pregraduation coil	

3.2. Tool change process

⑴. Ensure the tool post brake TZD signal is turned on.

⑵.The system executes the shortest path judge based on the target tool no. and the current tool no., and selects the rotation direction of the output according to the nearby tool selection, outputs the positive signal(TL+) or negative signal (TL-), and the tool post rotates to select the tool.

⑶.During the rotation, the system decodes to identify the current tool no. based on the tool encoding signal T1~T4 input. When it rotates to the previous tool before the target tool no., it checks the skip of the tool post strobe signal. The strobe signal of the previous signal before the target tool no. executes the falling edge, the system outputs pregraduation electromagnetic signal of the tool post, and the electromagnet is turned on.

⑷.When the input signal of the pregraduation proximity input signal of the tool post is HIGH, the system closes the tool post rotation output signal (TL+ or TL-), and the motor stops.

⑸.After the system delays 50ms, it outputs the signal(TL- or TL+) which is reverse to the previous rotation direction and the tool post rotates reversely.

⑹.When the proximity input signal of the tool post locked is HIGH, the system closes the rotation signal (TL-, TL+), and the motor stops. Then the system outputs the tool post brake signal TZD, and the motor brake is turned on.

⑺.When the system delays 200ms，it closes the pregraduation electromagnetic output signal, and then the electromagnet is turned off.

⑻.The system checks the current tool no. again and confirms whether the encoding signal of the current tool is consistent with the target tool no..

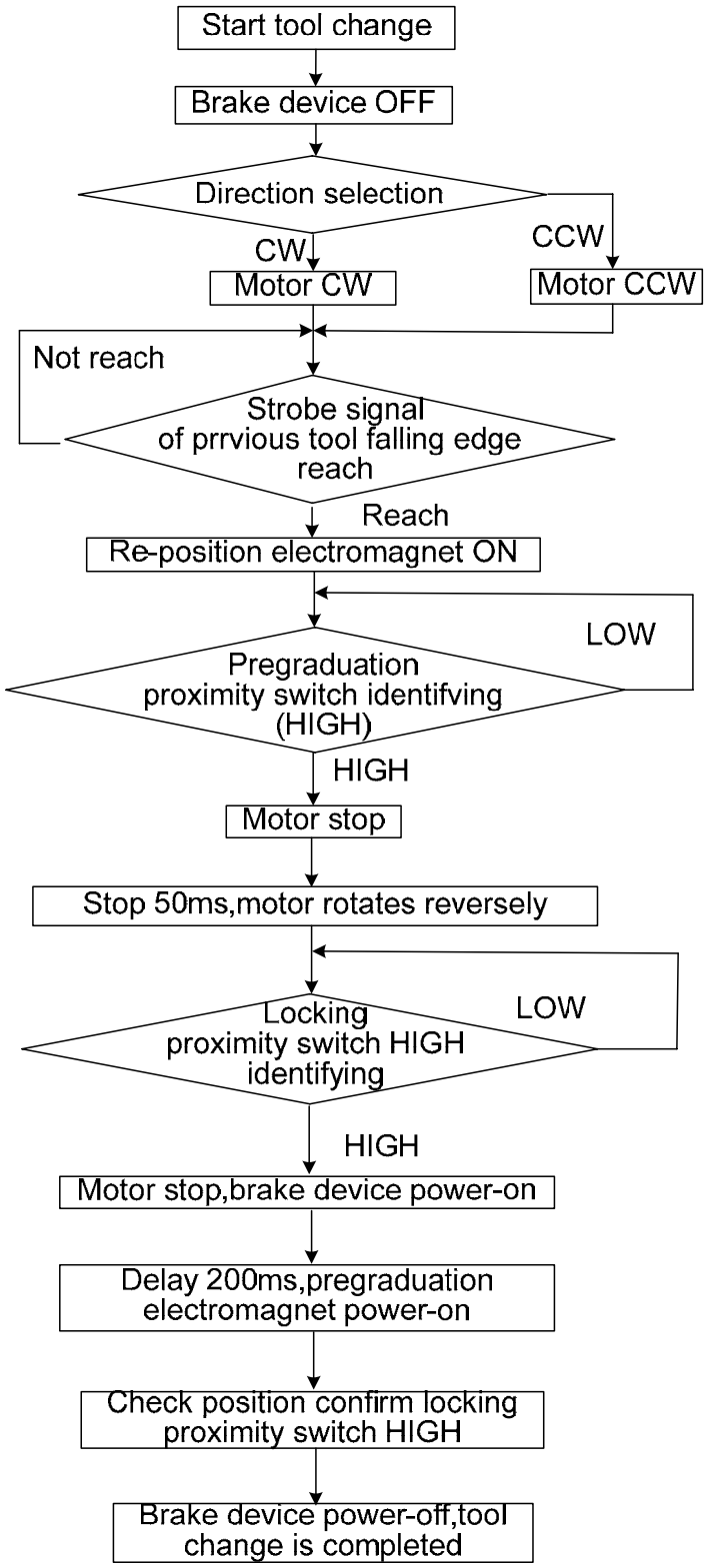
⑼.The system confirms again whether the locked proximity switch signal is HIGH.

⑽.When the above steps are correct, the system closes the tool post brake signal TZD, and so the tool change is completed.

⑾.In the course of the tool change, when the system has found the motor overheat signal, it

alarms and closes output of all signals.

3.3. Tool change flow



4. CHTB (K11.1)=1，CHTA(K11.0)=1: LIO_SHING hydraulic tool post

4.1. Signal specification

Signal type	Code	Interface	Address	Function	Remark
Input signal	CN61.16	X1.7	T01	tool signal	Sensor A
	CN61.29	X2.0	T02	tool signal	Sensor B
	CN61.30	X2.1	T03	tool signal	Sensor C
	CN61.31	X2.2	T04	tool signal	Sensor D
	CN61.8	X0.7	SSE	the tool pot stop and the start signal be locked	Sensor E
	CN61.9	X1.0	SSF	tool post releasing/lock signal	Sensor F
Output signal	CN62.15	Y1.6	TL+	Tool post CCW rotation	Sol B
	CN62.16	Y1.7	TL-	Tool post CW rotation	
	CN62.29	Y2.0	TRLS	tool post released outout	Sol A
	CN62.30	Y2.1	TCLP	tool post locked outout	

Sensor A, B, C, D: only check the tool, and are not any start signals

Sensor E: it executes the inductive once when one tool change is executed, which makes the tool pot stop and the start signal be locked. When the tool pot rotates to the required position, Sensor is inductive, i.e. the electromagnet to control the tool pot rotation is turned off, which makes the tool stop rotation and start the electromagnet to lock the tool pot to ensure that the tool pot is locked.

Sensor F: releasing/lock confirmation signal. When it is not inductive, the tool pot is released, at the time, the tool pot is started to rotate. When it is not inductive, the tool pot is locked, i.e. the tool change is completed.

Sol A: control the tool pot releasing/locking

Sol B: control the tool pot rotation(CW/CCW)

4.2 . Corresponding table between position and signal

8-tool:

	1	2	3	4	5	6	7	8
A			●		●	●	●	
B	●				●		●	●
C				●	●	●		●
D		●				●	●	●
E	●	●	●	●	●	●	●	●

10-tool:

	1	2	3	4	5	6	7	8	9	10
A			●		●	●	●		●	
B	●				●		●	●	●	
C				●	●	●		●		●
D		●				●	●	●		●

E	●	●	●	●	●	●	●	●	●	●

12-tool:

	1	2	3	4	5	6	7	8	9	10	11	12
A	●			●	●		●	●	●			
B			●			●	●		●	●	●	
C		●			●	●		●	●	●		
D				●			●	●		●	●	●
E	●	●	●	●	●	●	●	●	●	●	●	●

4.3 Tool change process description

Example: No. 1 tool is changed into No. 4 tool

Step 1: Sol A is turned on(tool pot is released)

Step 2: Ensure Sensor F is not inductive, Sol B is turned and the oil hydraulic motor rotates

Step 3: check the tool signal(Note: Sensor E is inductive for No. 1, 2, 3 tool, does not execute the locking operation when the tool pot does not reach No. 4 tool). When No. 3 tool signal is confirmed, Sensor E preparatory operation should be set to make the tool pot rotate to No.4 tool, at the time, Sensor E is inductive, i.e. Sol B is turned on, the tool pot stops rotation and Sol A makes the tool pot be locked.

2.9.3 Machine Zero Return

Notice

When the feed axis matches with the absolute encoder, set the arbitrary point as the machine reference point by the [n axis reference point setting] on the page of the “Position - > Integral Coordinate”. It will directly position to the reference point when zero return after the reference position is set.

Mainly, the following relevant description is about the adaptive incremental encoder and the zero return procedure of the rotation axis.

- Relative signal

DECX: X deceleration signal;

DECY: Y deceleration signal;

DECZ: Z deceleration signal;

DEC4: 4th deceleration signal;

DEC5: 5th deceleration signal;

PCX: X zero signal;

PCY: Y zero signal;

PCZ: Z zero signal;

PC4: 4th zero signal;

PC5: 5th zero signal;

DECX	X deceleration signal		PCX	X zero signal
DECZ	Z deceleration signal		PCZ	Z zero signal
DECY	Y deceleration signal		PCY	Y zero signal
DEC4	4th deceleration signal		PC4	4th zero signal
DEC5	5th deceleration signal		PC5	5th zero signal

- Diagnosis data

0	0	0				DEC5	DEC4	DECZ	DECY	DECX
Interface pin						CN61.34	CN61.33	CN61.12	CN61.32	CN61.4
0	0	8				PC5	PC4	PCY	PCZ	PCX
Interface pin						CN15.10	CN14.3	CN13.3	CN12.3	CN11.3

- Controlling parameter

  Status parameters

0	**0**	**4**			DECI						

Bit 5 1: During machine zero return, the deceleration signal is at high level;

0: During machine zero return, the deceleration signal is at low level;

0	**0**	**5**								**PPD**	

Bit 1: 1: G50 sets relative coordinate system value;

0: G50 does not set relative coordinate system value;

0	**0**	**6**				**ZM5**	**ZM4**	**ZMY**	**ZMZ**	**ZMX**

Bit4 1: 5th axis zero point return method C

0: 5th axis zero point return method B

Bit3 1: 4th axis zero point return method C

0: 4th axis zero point return method B

Bit2 1:Y axis zero point return method C

0:Y axis zero point return method B

Bit1 1:Y axis zero point return method C

0:Y axis zero point return method B

Bit0 1:X axis zero point return method C

0:X axis zero point return method B

0	**0**	**7**				**ZC5**	**ZC4**	**ZCY**	**ZCZ**	**ZCX**

Bit 4 1: During machine zero return, the deceleration signal of the 5th axis (DEC5) and one-rotation signal (PC5) are connected in parallel (use a proximity switch as a deceleration signal and zero-point signal at the same time).

0: During machine zero return, the deceleration signal of the 5th axis (DEC5) and one-rotation signal (PC5) are connected separately (use a proximity switch as a deceleration signal and zero-point signal at the same time).

Bit 3 1: During machine zero return, the deceleration signal of the 4th axis (DEC4) and one-rotation signal (PC4) are connected in parallel (use a proximity switch as a deceleration signal and zero-point signal at the same time).

0: During machine zero return, the deceleration signal of the 4th axis (DEC4) and one-rotation signal (PC4) are connected separately (use a proximity switch as a

deceleration signal and zero-point signal at the same time).

Bit 2 1: During machine zero return, the deceleration signal of the Y axis (DECY) and one-rotation signal (PCY) are connected in parallel (use a proximity switch as a deceleration signal and zero-point signal at the same time).

0: During machine zero return, the deceleration signal of the Y axis (DECY) and one-rotation signal (PCY) are connected separately (use a proximity switch as a deceleration signal and zero-point signal at the same time).

Bit 1 1: During machine zero return, the deceleration signal of the Z axis (DECZ) and one-rotation signal (PCZ) are connected in parallel (use a proximity switch as a deceleration signal and zero-point signal at the same time).

0: During machine zero return, the deceleration signal of the Z axis (DECZ) and one-rotation signal (PCZ) are connected separately (use a proximity switch as a deceleration signal and zero-point signal at the same time).

Bit 0 1: During machine zero return, the deceleration signal of the X axis (DECX) and one-rotation signal (PCX) are connected in parallel (use a proximity switch as a deceleration signal and zero-point signal at the same time).

0: During machine zero return, the deceleration signal of the X axis (DECX) and one-rotation signal (PCX) are connected separately (use a proximity switch as a deceleration signal and zero-point signal at the same time).

Note: In the zero return method B, it is set to 0; the zero return can be detected the PC signal after releasing from the block when it independently connects. If it is set to 1, PC detection signal begins after pressing the block to decelerate to zero when it is connected in parallel, and the block then can be reserved.

0	1	1

				NORF	ZNLK	REFC2	

Bit3 1: Manual machine zero return is invalid;

0: Manual machine zero return is valid;

Bit 2: 1: The direction key is self-locked during machine zero return; press the direction key once till the machine zero return is finished;

0: The direction key is not self-locked during machine zero return; press down the direction key;

Bit1 1:Ref.C mode,the feeding slowdown when machine press down switch, it is reversed after decelerating to zero.

0:Ref.C mode,the feeding slowdown when machine release switch, it is reversed after decelerating to zero.

0	1	2

APRS							ISOT

Bit 7 1: CNC sets the absolute coordinate system automatically after reference point return;

0: CNC does not set absolute coordinate system automatically after reference point return;

Bit 0 1: Manual rapid traverse is valid after system power-on and before machine zero return;

0: Manual rapid traverse is invalid after system power-on and before machine zero return;

0	1	4

			ZRS5	ZRS4	ZRSY	ZRSZ	ZRSX

Bit4, Bit3, Bit2, Bit1, Bit0

1: Machine zero point is set on the 5th, 4th, Y, Z, X axes; deceleration signal and zero point signal are needed to be detected during machine zero return;

0: Machine zero point is not set on the 5th, 4th, Y, Z, X axes; deceleration signal and zero point signal are not detected during machine zero return;

1	8	3

			MZR5	MZR4	MZRY	MZRZ	MZRX

Bit4/Bit3/Bit2/Bit1/Bit0 1: The direction of machine zero return is negative;
0: The direction of machine zero return is positive;

1	8	8

RFCK	RRT						

Bit7 1:G28 running every time to check the PC signal
0:G28 running only first to check the PC signal
Bit6 1:Zero return D mode is used when the additional axis is a rotary axis;
0:Zero return A, B, C mode is used when the additional axis is a rotary axis (power-on again).

Data parameters

0	3	3	ZRNFLn

The low speed of machine zero return;

1	1	3	REF_SPDn

The high speed of machine zero return;

0	4	7	PRSn

Setting value of absolute coordinate after machine zero return;

- **Signal connection**

The interior circuit of deceleration signal is shown in Fig. 2-32:

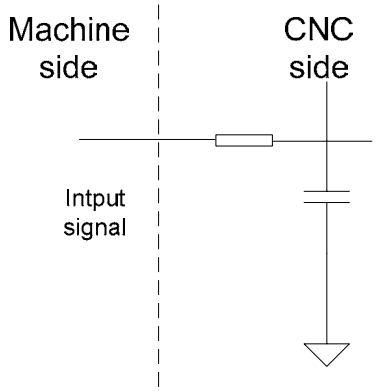


Fig. 2-32

- **Machine zero return by regarding servo motor one-turn signal as the zero signal**

①Its sketch map is shown below:

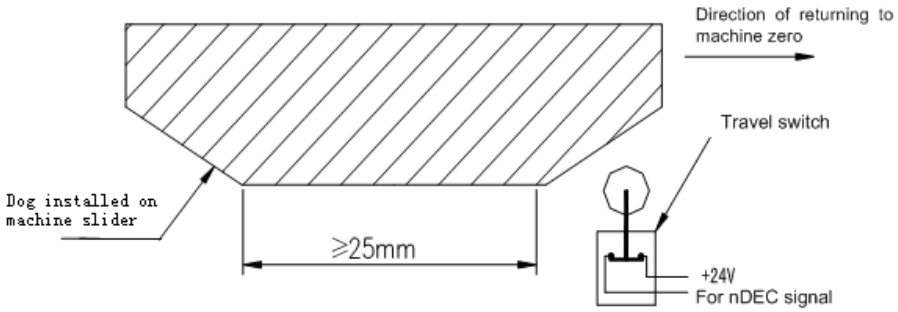


Fig. 2-33

② The circuit of deceleration signal

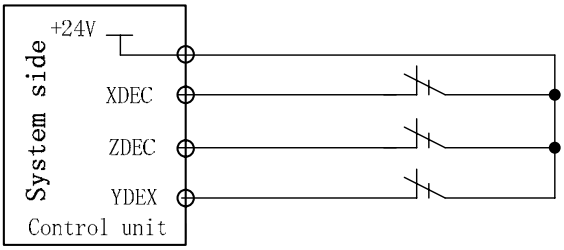


Fig. 2-34

③ Sequence of machine zero return

When the BIT0 (ZMX) of the bit parameter No.006 is set to 0, and the BIT5(DECI) of the bit parameter No.004 is set to 0, the system chooses the machine zero return mode B, and the deceleration signal low level is active.

So the sequence of machine zero return mode B is shown as follows:

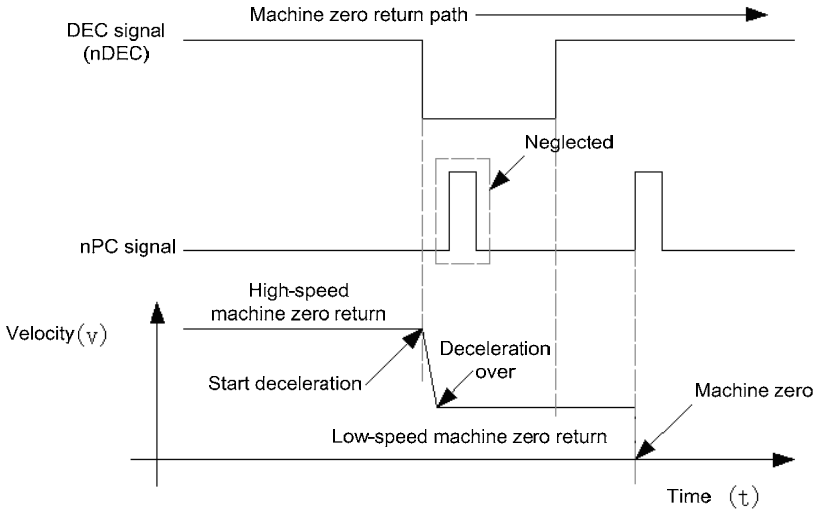


Fig. 2-35

Return process of machine zero mode B

A: Select Machine zero mode, press the manual positive or negative feed key (machine zero return direction set by bit parameter No.183), the corresponding axis moves to the machine zero by a rapid traverse speed (No.113). As the axis press down the deceleration switch to cut off deceleration signal, the feeding slows down immediately, and it continues to run in a fixed low speed (No.33).

B: When the deceleration switch is released, the deceleration signal contact is closed again. And CNC begins to detect the encoder one-turn signal (PC), if this signal level skips, the motion will be halted. And the corresponding zero return indicator on the operator panel lights up for machine zero return completion.

When the BIT0 (ZMX) of the bit parameter and No.006 are both set to 1, and the BIT5 (DECI) of the bit parameter No.004 is set to 0, it chooses the machine zero return mode C, and the deceleration signal low level is active. The Bit 1 of status parameter No.011 can be set the deceleration time.

So the sequence of machine zero return mode C is shown as follows:

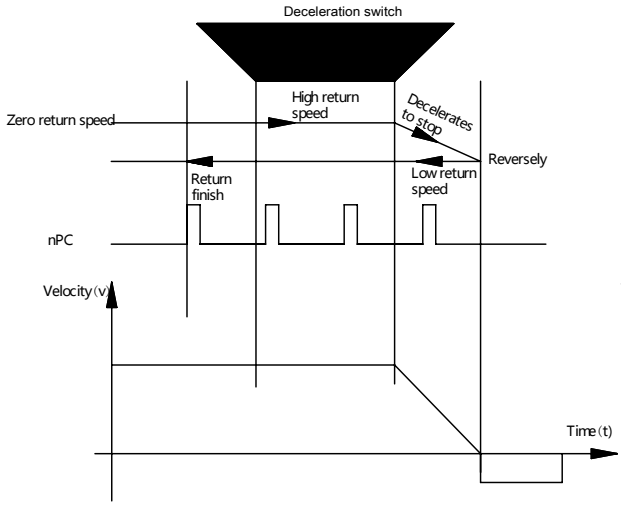


Fig. 2-36 (Out-of-block deceleration)

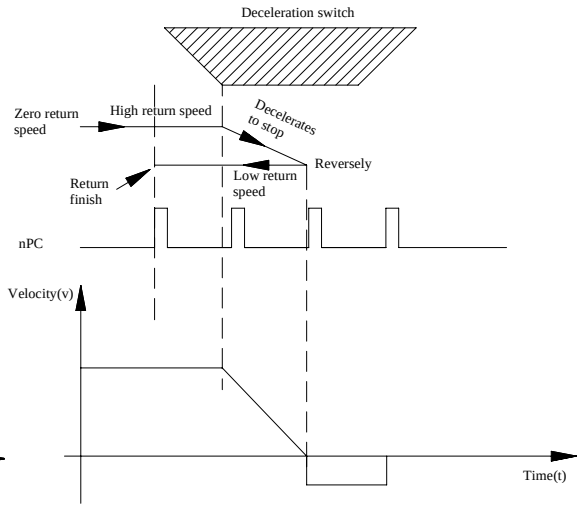


Fig. 2-37 (Push the block deceleration)

Return process of machine zero mode C

A: Select Machine zero mode, press the manual positive or negative feed key (return direction set by bit parameter No.183), the corresponding axis moves to the machine zero by a rapid traverse speed (No.113). As the axis press down the deceleration switch to cut off deceleration signal, if the pushing block begins deceleration (Bit 1 of status parameter No.011), the operation speed is then decelerated to zero. The feeding keeps rapid rate and depart from the deceleration switch, when the DEC signal contact is closed, the feeding slows down to zero, then run reversely to return to machine zero in a low speed.

B: In the reverse running, it presses down the deceleration switch to cut off the DEC signal contact and continues returning; as it departs from the deceleration switch, the deceleration signal contact is closed again. And CNC begins to detect the encoder one-turn signal (PC), if this signal level skips, the motion will be halted. And the corresponding axis zero return indicator on the operation panel lights up for zero return completion.

- **Machine zero return by a proximity switch taken as both deceleration and zero signals**

① Its sketch map is as follows:

Sketch map of zero tongue for machine zero return mode B:

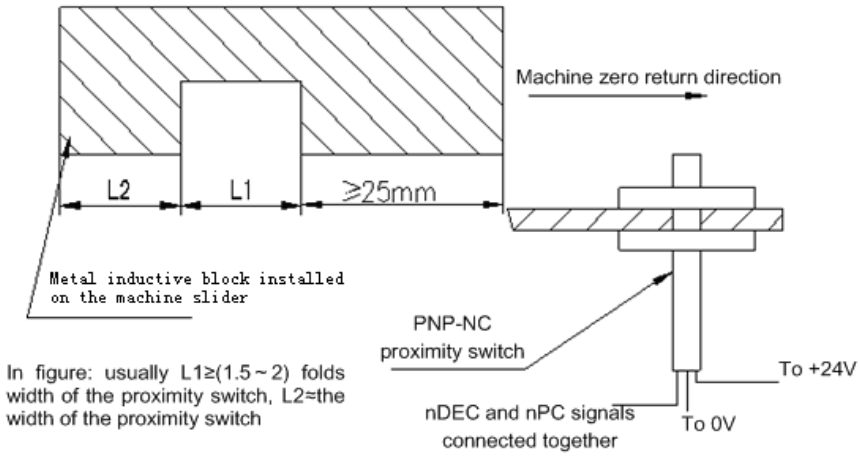


Fig. 2-38

Sketch map of zero tongue for machine zero return mode C:

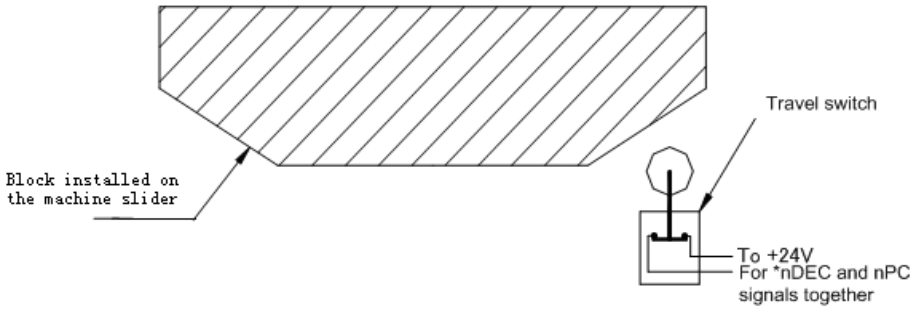


Fig. 2-39

② Circuit of the deceleration signal

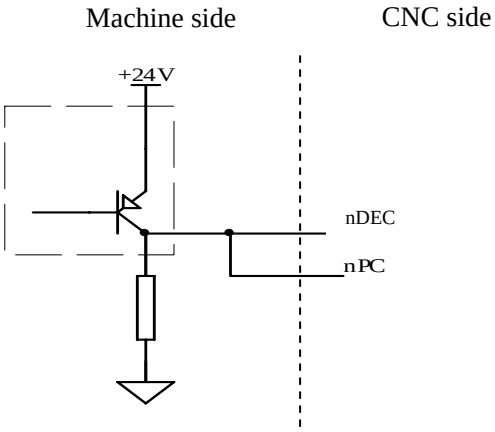


Fig. 2-40

③Sequence of machine zero return (taking example of X)

When the BIT0 (ZMX) of the bit parameter No.006 are all set to 0, and the BIT5 (DECI) of the bit parameter No.004 is 0, it chooses the machine zero return mode B.

The sequence of machine zero return is as follows:

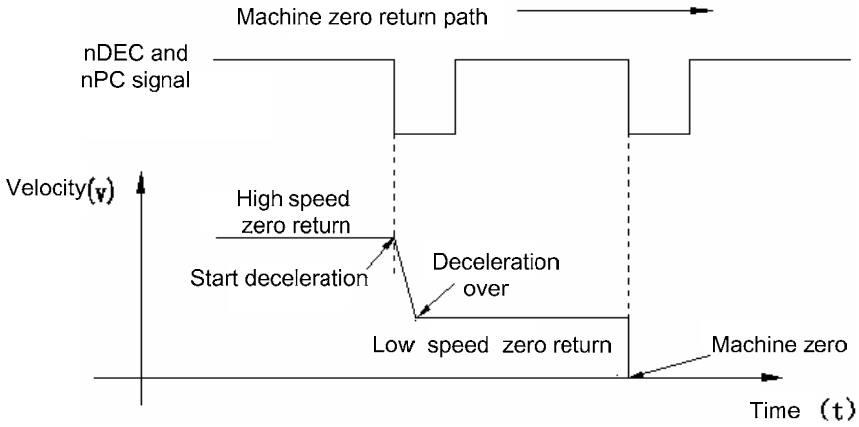


Fig. 2-41

B type machine zero return process

A: Select the Machine Zero mode, press manual positive or negative (zero return direction set by bit parameter No.183) feed key, the corresponding axis will move to the zero at a high speed(set by parameter No.113).

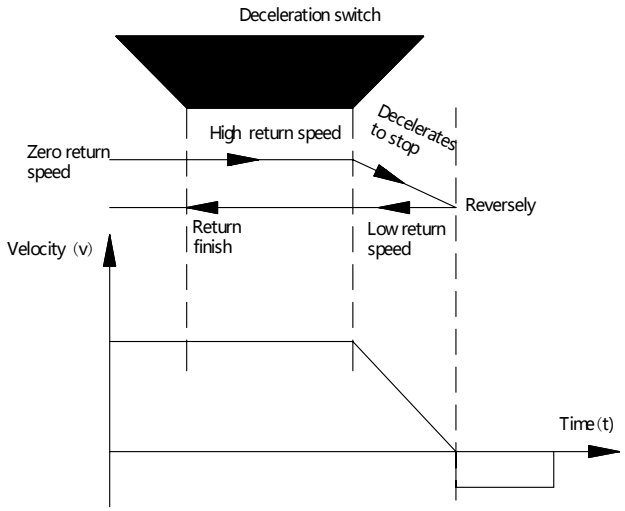
B: As the proximity switch touches the tongue for the first time, the deceleration signal is active and speed immediately slows down and it runs in a fixed low speed (set by parameter No.33).

C: As the proximity switch detaches the tongue, the deceleration signal is inactive; it moves at a fixed low speed after deceleration and starts to detect zero signal (PC).

D: As the proximity switch touches the tongue for the second time, the zero signal is active and the movement stops. The zero return indicator on the panel lights up for machine zero return completion.

When the BIT0 (ZMX) of the bit parameter No.006 are all set to 1, and the BIT5(DECI) of the bit parameter No.004 is 0, it chooses the machine zero return mode C.

The sequence of machine zero return mode C is shown as follows:



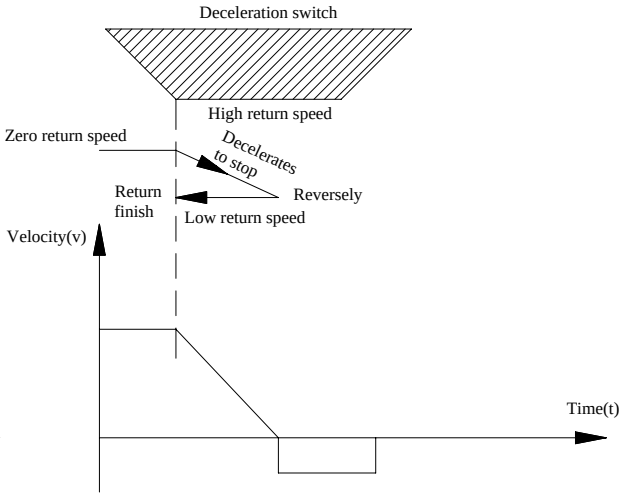


Fig.2-42 (the feeding slowdown release switch) Fig.2-43 (the feeding slowdown press down switch)

Ref.C mode,the feeding slowdown when 0:release switch 1:press down switch

C type machine zero return process

A: Select the Machine Zero mode, press manual positive or negative (zero return direction set by bit parameter No.183) feed key, the corresponding axis will move to the zero at a high speed(set by parameter No.113).

B: As the proximity switch touches the tongue for the first time, the deceleration signal is active. If the pushing block deceleration begins (The Bit 1of the status parameter equals to 1), the feed speed decelerates to zero, then it runs reversely by low return speed.

C: If the setting block begins to decelerate (The Bit 1 of the status parameter No.011 sets to 0), it is still operated with same speed when its speed descends. When the deceleration signal contactor is closed, the operation speed decelerates to zero till leaving the deceleration switch, and then it reversely operates with low speed based upon the machine tool zero.

D: As the proximity switch touches the deceleration switch for the second time in reverse running, deceleration signal contact is cut off, it still runs by low return speed; as it detaches the deceleration switch, deceleration signal contact is closed again, so the movement stops. The zero return indicator for the corresponding axis on the panel lights up for zero return completion.

- **The machine zero return (zero-return method D) of deceleration signal or zero signal by using the servo motor one-turn signal**

There are four zero return methods for rotation axis, which are separately divided into: A, B, C and D. Wherein, zero return method A, B and C are identical with the one of the linear axis. Only the D is a special zero return method of the rotation axis.

When the controllable axis is rotation one, and the Bit 6 of status parameter No.188 sets to 1, select the machine zero return method D.

The time-sequency figure of the machine zero operation in method D is as follows:

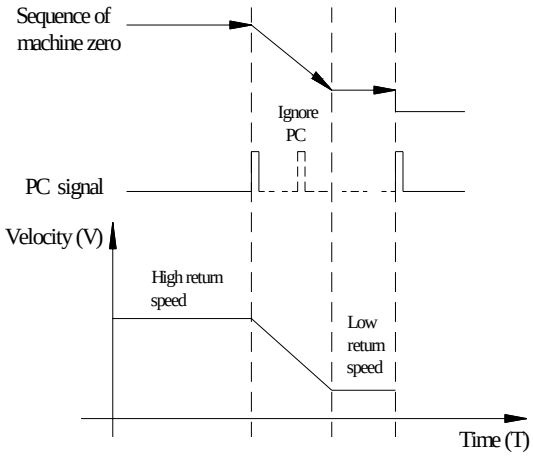


Fig. 2-44

The process of machine zero return in method D:

A: Select the mechanical zero return method, positively feed manually, the corresponding axis moves to the zero direction at the rapid traverse rate.

B: When met the one-turn signal (PC) of the servo axis at the first time, the system decelerates to the zero return.

C: When the system decelerates to the zero return, the system is then detected the one-turn signal again (PC).

D: The system operation stops when mets the one-turn signal (PC) of the servo axis; simultaneously, the zero return indicator of if its corresponding axis on the operation panel is turned on, the mechanical zero return is then ended.

2.9.4 Spindle Control

- **Relevant signal (by standard PLC program)**

Type	Sign	Interface	Address	Function	Remark
Input signal	SALM	CN15.8	X5.0	Spindle abnormity alarm input	
Output signal	M03	CN62.4	Y0.3	Spindle rotation(CCW)	
	M04	CN62.5	Y0.4	Spindle rotation(CW)	
	M05	CN62.6	Y0.5	Spindle stop	
	SPZD	CN62.8	Y0.7	Spindle brake	
	SFR	CN15.6	Y5.0	Spindle rotation(CW)	Its function is consistent with that of M03
	SRV	CN15.7	Y5.1	Spindle rotation(CCW)	Its function is consistent with that of M04
Command format	M03			Spindle rotation(CCW)	
	M04			Spindle rotation(CW)	
	M05			Spindle stop	

Note: When the spindle is serial-connected, system delivers the rotation signal to servo by

bus regardless of the above-mentioned IO signals.

- **Control parameter**

Bit parameter

K	**1**	**0**							**RSJG**	

RSJG =1: CNC not turn off M03, M04, M08, M32 output signals when pressing key;

=0: CNC turns off M03, M04, M08, M32 output signals when pressing key.

K	**1**	**7**					**SALM**	**SSTP**			

SALM =1: The system alarms when the spindle abnormal detection input signal is LOW (it is turned off with 0V);

=0: The system alarms when the spindle abnormal detection input signal is HIGH (it is turned on with 0V) .

SSTP =1: The analog voltage is closed when the spindle stops;

=0: The analog voltage is not closed when the spindle stops.

1	**7**	**5**		**SAR**							

Bit6 1: The spindle SAR signal is checked before cutting;

0: The spindle SAR signal is not checked before cutting.

Data parameter

0	**7**	**2**	**SAR_DELEY**

Delay check time (ms) of the spindle speed arrival.

D	**T**	**0**	**5**	**MTIME**

Execution time of M code (ms)

D	**T**	**1**	**0**	**SPDDLT**

Spindle braking delay output time after spindle stop (M05) is output

D	**T**	**1**	**1**	**SPZDTIME**

Spindle braking output time (ms)

D	**T**	**2**	**3**	**SVF_DELEY**

Delay time before spindle motor torque decrease to 0 when spindle starts clamping;

D	**T**	**2**	**0**	**SCLPTIME**

Spindle clamping/releasing command performance time (ms).

D	**C**	**0**	**0**	**SPZERO**

Spindle zero speed output range (m/min);

- **Signal connection**

M03, M04, M05, SPZD signal output circuit is shown in Fig. 2-45A:

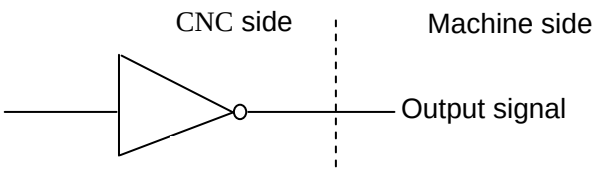


Fig. 2-45A

SAR, SALM signal input circuit is shown in Fig.2-45B:

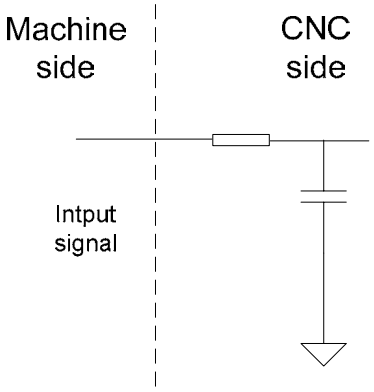


Fig.2-45B

- **Operation sequence （defined by standard PLC program）**

Spindle operation sequence is show in Fig 2-46:

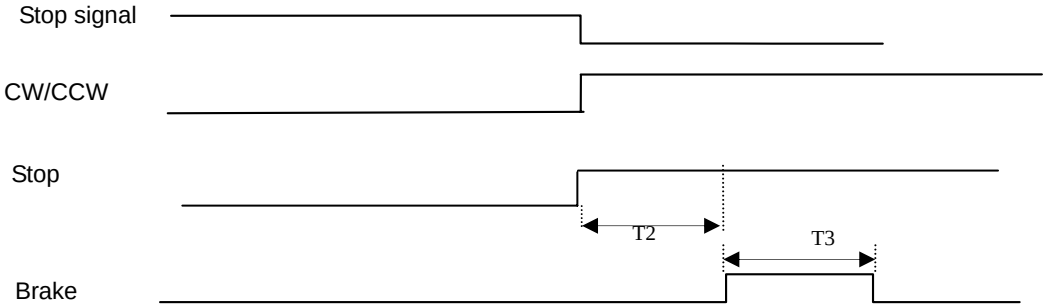


Fig. 2-46 spindle rotation(CW/CCW) sequence

Note: T2 is the delay time from sending the spindle stop signal to sending the spindle brake signal; T3 is the spindle brake hold time.

- **Function description（defined by standard PLC program）**

① After the CNC is turned on, when M05 output is valid, M03 or M04 is executed, M03 or M04 output is valid and remains, at the time, M05 output is closed; when M03 or M04 output is valid, M05 is executed, M03 or M04 is closed, M05 output is valid and remains; the spindle brake SPZD signal output delay is set by No.087（delay time from the spindle stop signal output to the spindle brake SPZD signal output）, the hold time of the brake signal is set by No. 089（the spindle brake output time）.

② When M03（M04）output is valid, M04（M03）is executed, the alarm occurs.

③ When №175.6 is set to 1, the system checks whether the speed arrival signal SAR is valid, if it is valid, the system normally runs, if not, “Check speed arrival……” is displayed.

④ When the spindle speed command and the cutting feed command are in the same block, the system executes the delay check for SAR signal to avoid that the CNC starts the cutting based on the last spindle speed arrival signal SAR, the delay time is set by No.072.

⑤ DC00 is the spindle zero-speed output range. When the actual spindle speed is not more

than DC00 setting value, the system defaults the spindle speed to be zero, and can release the chuck, close the hydraulic and other operations.

⑥ SALM（X5.0）is spindle abnormity alarm input。

Note 1: In the emergency stop, it turns off M03, M04, M08 signals, and outputs M05 signal;

Note 2: Whether M03, M04 is cancelled is set by BIT3 of the bit parameter No.009 when CNC is reset.

If Bit 1=0, CNC turns off M03, M04 at reset;

If Bit 1=1, M03, M04 is kept at reset.

2.9.5 Spindle Wwitching Volume Control

- **Relevant signal(defined by standard PLC program)**

S01～S04: Control signal for spindle speed switching volume, they are compound interfaces defined by standard PLC program, and they share common interfaces with M41～M44, U00～U03.

- **Signal diagnosis**

Signal	S4	S3	S2	S1
Diagnosis address	Y1.3	Y1.2	Y1.1	Y1.0
Interface pin	CN62.12	CN62.11	CN62.10	CN62.09

- **Control parameters**

Bit parameter

0	**0**	**1**					**ACS**				

Bit4 =1: Analog voltage control of spindle speed

=0: Switching volume control of spindle speed

- **Control logic (defined by standard PLC program)**

S1～S4 output are inactive at power on. If any code of them is executed, the corresponding S signal output is active and held on, and the other S signal outputs are cancelled. S1～S4 outputs are cancelled when executing S00 code, and only one of them is active at a time.

2.9.6 Spindle Automatic Gearing Control

- **Relevant signal (defined by standard PLC program)**

M41～M44: Spindle automatic gear shifting output signals. It supports 4-gear spindle automatic gear shifting control when the system selects the spindle analog value control(0～10V analog voltage output)

M41I, M42I: spindle automatic gear shifting No.1, 2 gear in-position signals to support gear shifting in-position check function

- **Signal diagnosis**

Signal	M42I	M41I	M44	M43	M42	M41
Diagnosis address	X1.6	X1.5	Y1.3	Y1.2	Y1.1	Y1.0
Interface pin	CN61.15	CN61.14	CN62.12	CN62.11	CN62.10	CN62.09

- **Signal connection**

The circuit for M41~M44 is shown in Fig.2-47:

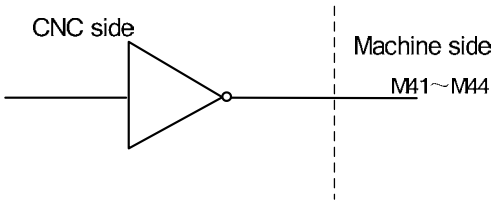


Fig. 2-47

- **Control parameter**

Bit parameter

0	**0**	**1**					**ACS**				

Bit4 =1: Spindle analog volume control, set to 1 if using spindle automatic gearing
=0: Spindle switching volume control

1	**9**	**5**		**PLCS**							

Bit6 =1: PLC spindle rotation speed function of each axis enabled;
=0: PLC spindle rotation speed function of each axis disabled。

K	**1**	**5**						**SHT**	**AGIM**	**AGIN**	**AGER**

AGER =1: Spindle automatic gearing active
=0: Spindle automatic gearing inactive
AGIN =1: Detect M41I, M42I signal when shifting to gear 1, 2
=0: Not detect M41I, M42I signal when shifting to gear 1, 2
AGIM =1: Active when M41I, M42I signals disconnecting to +24V
=0: Active when M41I, M42I signals connecting to +24V
SHT =1: spindle gear power-down executes the memory
=0: spindle gear power-down does not execute the memory

Data parameter

0	**3**	**7**	**GRMAX1**
0	**3**	**8**	**GRMAX2**
0	**3**	**9**	**GRMAX3**
0	**4**	**0**	**GRMAX4**

GRMAX1, GRMAX2, GRMAX3, GRMAX4: The respective max. speeds of spindle gear 1, 2, 3, 4 when analog voltage output is 10V. Spindle speeds for M41, M42, M43, M44 when spindle automatic gearing is active.

0	**6**	**7**	**SFTREV**

The output voltage during spindle gear shift（0～10000, Unit: mV）

D	**T**	**0**	**0**	**SFT1TME**

Time for original gear disable during spindle gear shift (ms)，refer to function description for details.

D	**T**	**0**	**1**	**SFT2TME**

Delay time after spindle gear shift (ms), refer to function description for details.

- **Function description (defined by standard PLC program)**

The spindle automatic gearing is active only under the spindle analog voltage control (BIT4 of the bit parameter No.001 set to 1) and the BIT 7 of the bit parameter No.164 is set to 1; if the spindle auto

gearing is inactive, alarm will be issued when M41～M44 is being executed and only one of them is active at a time.

When spindle auto gearing is used to control automatic spindle mechanical gear switching, as CNC executes S□□□□ code, it calculates the analog voltage output to spindle servo or frequency inverter based on the parameter of the current gear by M4n (M41～M44 to data parameters No.037~No.040 respectively) to make the actual speed to be consistent with the S code.

When CNC is powered on, the spindle gear memorizing is set by the BIT1 of bit parameter No.168.

If the BIT4 of bit parameter No.001 is 0, the spindle gear is not memorized at repowering after power down, and the gear 1 will be defaulted, M41~M44 are not output. If BIT4 of bit parameter No.001 is 1, the spindle gear is memorized at repowering after power down.

No gearing is done if the specified gear is consistent with the current gear. If not, gearing will be performed, and the process defined by standard ladders is shown in the following:

①Execute any of M41, M42, M43, M44 codes, output analog voltage to spindle servo or frequency inverter according to a value set by data parameter No.067 (Unit: mV);

②After a delay (gearing time 1) by the DT00, turn off the original gear output signal and output the new gearing signal;

③If the gear is 1 or 2, and the BIT6 of the bit parameter No.164 is 1, it jumps to ④, or else it jumps to ⑤;

④Check the gear in-position input signal M41I, M42I, it jumps to ⑤ if the gear in-position is done; if not, the CNC waits the gear in-position signal;

⑤After a delay (gearing time 2) by DT01, output spindle analog voltage by the current gear according to a value set by data parameter No.037~No.040 (gear 1~4) and finish the gearing.

Note: The output of M41～M44 is held on when CNC is reset or i emergency stop, which is defined by standard PLC ladder.

2.9.7 Spindle 8-point Orientation Function

- **Command format**

M51: spindle orientation command

M50: cancel spindle orientation command

Note: When the system is matched with GS-L Spindle Servo Unit, it can realize one-point orientation.

- **Control parameter**

K	1	5				SPOR					

SPOR =1: The spindle eight-orientation function is valid.

=0: The spindle eight-orientation function is invalid.

- **Functional description(standard PLC program definition)**

1. After the positioning function commands M51 are executed, PLC→CNC executes the orientation signal.

2. CNC issues an orientation command after reciving the orientation signal; (after the system delays 100ms, the spindle rotation(CW/CCW) is disabled, at the same time, the system starts to check the orientation completion signal (COIN）.

3. Drive unit starts the orientation.

4. After the drive unit orientation is completed, Drive →CNC→PLC outputs the spindle completion signal COIN;

5. After PLC receives the spindle orientation completion signal COIN, the spindle orientation is completed, but the spindle orientation signal STAO keeps the output.

6. When M51 is input again, the spindle oriented to another position.

7. After the orientation is completed and when M50 is executed or the spindle orientation is operated, PLC closes the spindle orientation signal STAO output and cancels the positioning function.

8. When the system has not received the orientation completion signal after the PLC sends the orientation completion signal for 10000ms, it alarms “the spindle orientation time being long”.

9. Before the orientation, the spindle can rotate or stop; the spindle stops after the orientation.

Note 1: The drive is for the spindle servo drive unit.

Note 2: The spindle should be the serial-connected when selecting this function; simultaneously, adapt the GS-L series spindle servo drive unit.

- **Connected diagram**

When GSK980TDi matches with the GS-L series servo drive, it is connected with bus regardless of the special wiring.

2.9.8 Spindle Cs Axis Control Function

- **Command format**

M14: The spindle is switched from the speed control mode into the position

M15: The spindle is switched from the position control mode into the speed

Note: When the spindle is rotary, Cs function is valid and the corresponding spindle servo drive unit is speed/position switch working mode, the spindle speed/position control switch is valid.

- **Control parameters**

State parameters

0	**0**	**1**					**Analog spindle**				

Bit4 =1: Spindle speed analog control. When the spindle automatic gear change function is used, it must be set to 1;

=0: Spindle speed switch control.

1	**8**	**7**								**ROS**	**ROT**

Bit1, Bit0 : 00: set the additional axis to the linear axis; 01: set the additional axis to the rotary axis (A-type); 11: set the additional axis to the rotary axis (B-type); 10: set the additional to be invalid.

1	**9**	**5**	**SSPD**							**CSREF**	**RCS**

Bit0 =1:Cs function of spindle valid ,power on;

=0:Cs function of spindle is:invalid,power on.

Bit1 1:Change to position mode,the axis do return REF;

0:Change to position mode,the axis don't return REF.

Bit7 1:series spindle controllable function enabled (series spindle The slave ID number of the spindle 1~3 is 11~13) , it is necessary to restart;

0:series spindle controllable function disabled (analogy spindle), it is necessary to restart.

2	**2**	**1**	Feed axis of the corresponding spindle

The feed axis number related with the spindle should be set when matches with the CS axis function.

1	**6**	**4**	Spindle shifting M code

Spindle CS shifts the M code [Setting value +1 is the M code of the function releasing].

Note 1: When the spindle positioning function is enabled, it is regarded as the M code shifted by spindle positioning.

Note 2: The setting value should be shifted the M code by the compiled CS in the ladder diagram.

Note 3: The setting is enabled when its setting is less than the 6. The coordinate system will not be cleared as zero by shifting the CS axis when its value is disabled.

K	1	5		RCS	RSCS						

RCS =1: Cs function of the spindle is valid.

=0: Cs function of the spindle is invalid.

RSCS =1: The spindle contour control is disabled in emergency stop, reset.

=0: The spindle contour control is enabled in emergency stop, reset.

Note: K15.7 sets whether all Cs functions are valid, №o187.5, №o189.5, №o191.5 set whether Cs functions of the special axis is valid. When the multiple spindle function is valid, Cs function of only one axis is valid.

- **Function description（defined by standard PLC program）**

1. Control logic

1.1. The PLC sets the spindle contour controllable shifting signal CON (G27.7) to 1 after the M14 is executed;

1.2. CNC controls the drive to shift the controllable method, after the switch is completed, CNC→PLC outputs the spindle contour controllable shifting end signal FSCSL (F44.1);

1.3. Aafter the PLC detects the spindle contour controllable shifting end signal FSCSL (F44.1), end the performance of the M14. The spindle controllable shifting method is ended.

2. M15 execute processes are the same those of M14.

3. Only, the M15 cancels the speed/position shifting performed by M14; if the spindle shifts into the Cs contour control by executing the M29, the performance of the M15 will not be converted into the speed controllable method.

4. C/S on the panel is equal to the spindle speed/position control mode switch in Manual mode. In Manual/MPG mode, press C/S and the CNC starts the spindle speed/position control mode switch. After the CNC is switched into the spindle position control mode, the spindle orientation indicator is ON, at the time, Cs position movement is performed. Press C/S again and the CNC is switched into the spindle speed control mode and the spindle orientation indicator is OFF.

5. Before switch, the spindle can rotate or stop. After switch, the spindle stops.

6. When the spindle is in Cs contour control mode, the spindle rotation(CW/CCW) output is invalid.

7. When the spindle is in the external speed control mode, the manual operation of Cs axis is invalid.

8. When the alarm occurs in the servo or because of the too long switch time or the reset or emergency stop appears in switch, PLC switches the spindle into the speed control state.

9. When PLC K15.6 is set to 1, the spindle contour control is disabled in emergency stop, reset.

PLC switches the spindle into the speed control state.

10. There is the CS shifting M code (Standard ladder diagram defines as M14) in the setting of the No.164; the coordinate system can be cleared after the feed CS shifts.

- **Wiring diagram**

When the GSK980TDi matches with the GS-L series servo drive; the bus then can be connected regardless of the special wiring.

2.9.8 Multiple Spindle Function

When the multiple spindle function is valid, related signals, parameters and commands are consistent with related explanations of the spindle control, the 1st, the 2nd spindle explanations are shown below.

- **Related signals（defined by PLC program）**

Type	Sign	Interface	Address	Function	Remark
Input signal	SALM2	CN16.8	X8.0	The 2nd spindle abnormality alarm signal	
Output signal	SFR2	CN16.6	Y8.0	The 2nd spindle rotation(CW)	
	SRV2	CN16.7	Y8.1	The 2nd spindle rotation(CCW)	

Note: It is unnecessary to to connect the above-mentioned signals after the spindle is serial-connected.

- **Command format**

M63 the 2nd spindle rotates (CW)
M64 the 2nd spindle rotates (CCW)
M65 the 2nd spindle stops
M24 the 2nd spindle is switched from the speed control mode into the position control mode
M25 the 2nd spindle is switched from the position control mode into the speed control mode. The speed control mode is defaulted after the system is turned on.

- **Control parameters**

1	8	7				RCS				ROS	ROT

Bit5 =1: Cs function of the additional axis is valid (power-on again);
=0: Cs function of the additional axis is invalid(power-on again).

Bit 1, Bit0: 00 sets Y to be the linear axis; 01 sets Y to be the rotary axis (A type); 11 sets Y to be the rotary axis(B type); 10 sets Y to be invalid.

Bit1, Bit0 : 00: set the additional axis to the linear axis; 01: set the additional axis to the rotary axis (A-type); 11: set the additional axis to the rotary axis (B-type); 10: set the additional to be invalid.

1	9	5		SSPD							RCS

Bit0 =1:Cs function of spindle valid ,power on;
=0:Cs function of spindle is:invalid,power on.

Bit7 =1:series spindle controllable function enabled (series spindle The slave ID number of the spindle 1~3 is 11~13) , it is necessary to restart;
=0:series spindle controllable function disabled (analogy spindle), it is necessary to restart.

0	2	2	1	The feed axis related with the spindle

The feed axis number related with the spindle should be set when matches with the CS axis function.

Setting value meaning: 0: Without 1~5: the 1st axis ~ the 5th axis

1	9	6				MSEN				

Bit4 1: The multiple spindle control function is valid;

0: The multiple spindle control function is invalid;

K	1	5	RCS	RSCS						

RCS =1: The spindle Cs function is valid;

=0: The spindle Cs function is invalid.

RSCS =1: The spindle contour control is disabled in emergency stop, reset;

=0: The spindle contour control is enabled in emergency stop, reset.

K	1	7	MSEN		SALM2					

MSEN =1: The multiple spindle control function is valid;

=0: The multiple spindle control function is invalid.

SALM2 =1: The system alarms when the check input signal of the 2nd spindle abnormality is LOW;

=0: The system alarms when the check input signal of the 2nd spindle abnormality is HIGH;

Function description(defined by the standard PLC program）

1.1. After the multiple spindle control function setting is valid, the system defaults to the 1st spindle control after power-on;

1.2. When M03, M04, M05 is executed, the corresponding signal of the 1st spindle outputs, at that time, the system sets the 1st spindle control, hereafter, the S command being executed will change the 1st spindle speed. In a similar way, M63, M64, M65 is executed, the corresponding signal of the 2nd spindle outputs, at the time, the corresponding signal of the 2nd spindle outputs, and is switched into the 2nd spindle control, after that, the executed S command will change the 2nd spindle speed.

1.3. The spindle switch is invalid in rigid tapping.

1.4. The manual CW, CCW, STOP, SPINDLE OVRRRIDE keys on the panel are valid only to the 1st spindle. When the current system is controlled by the 2nd spindle, the manual control key of the spindle on the panel still controls the 1st spindle.

1.5. The spindle override of the 2nd spindle is fixed to 100%.

1.6. Cs axis control of two spindles can be separately switched and the current used spindle has no requirements, that is, when the current is the 1st(2nd) spindle, the 2nd (1st) spindle executes the Cs switch.

- **Cs control logic of the 2nd spindle**

1.1. After M24 is executed, PLC sets the spindle contour controllable signal CON (G127.7) to 1;

1.2. CNC controls the drive to shift the controllable method, after the shifting is performed, CNC→PLC outputs the spindle contour controllable shifting end signal FSCSL (F144.1);

1.3. After the PLC detects the spindle contour controllable shifting end signal FSCSL (F144.1), end the performance of the M24. The spindle controllable method is then completed.

2. M25 execute processes are the same those of M24.

3. M25 only cancels M24 executing the speed/position switch. Executing M25 cannot switch the

system into the speed control mode when executing M29 switches the spindle into Cs contour control.

4. on the panel is only used to Cs switch of the 1st spindle.
5. Before switch, the spindle can rotate or stop. After switch, the spindle stops.
6. When the spindle is in Cs contour control mode, the spindle rotation(CW/CCW) output is invalid.
7. When the spindle is in the external speed control mode, the manual operation of Cs axis is invalid.
8. When the alarm occurs in the servo or because of the too long switch time or the reset or emergency stop appears in switch, PLC switches the spindle into the speed control state.
9. When PLC K15.6 is set to 1, the spindle contour control is disabled in emergency stop, reset. PLC switches the spindle into the speed control state.

- **Wiring**

When the GSK980TDi matches with the GS-L series servo drive; the bus then can be connected regardless of the special cable.

2.9.9 Rigid Tapping Function

- **Related commands**

M20 spindle clamped output

M21 spindle clamped/released output

M29 entering rigid tapping state

Note: When only the selected spindle used to the rigid tapping interpolation is the rotary axis, the rigid tapping function can be executed.

- **Function description（defined by the standard PLC program）**

① When the Cs function of the tapping spindle is invalid, the system is switched into the rigid tapping state instead of switching the spindle servo speed/position in executing M29, defaults the spindle servo to be the position control state.

② When Cs function of the tapping spindle is enabled, the spindle servo for the tapping is switched into the position control state and the system is switched into Cs contour control state.

③ Before M29 switch, the tapping spindle is permitted to the spindle mode or Cs contour control mode. After the rigid tapping return, the tapping spindle returns to the previous control mode.

④ When the system is in the rigid tapping state, the manual movement tapping spindle cannot be performed.

⑤ The rigid tapping state is cancelled in reset or emergency stop.

⑥ In M29, when the 1st or the 2nd spindle is currently used, it is switched into the rigid tapping state. In rigid tapping, a spindle should be selected firstly to execute the rigid tapping interpolation. After the spindle is switched into the rigid tapping state, the spindle switch is invalid.

⑦ After M29 is execute, M20 and the tapping command in the same block are executed to clamp the spindle before drilling or tapping.

2.9.10 Spindle Clamped Function

- **Relative signals（Standard PLC program definition）**

Signal type	Code	Interface	Address	Function	Remark

Output signal	SCLP	CN62.07	Y0.6	Spindle clamped output	

- **Relative commands**

  M20 Spindle clamping output

  M21 Spindle clamping output cancel

  Note: The spindle clamping equipment should be matched with the machine tool.

- **Control parameters**

  Status parameters

K	1	7								
				SCLP						

SCLP =1:Spindle clamped function is valid。

=0:Spindle clamped function is invalid

D	T	2	0	SCLPTIME

Spindle clamping/releasing command execution time (ms).

D	T	2	3	SVF_DELEY

spindle is started to clamp, delay to reduce the spindle servo exciting time (ms).

- **Function description（Standard PLC program definition）**

  ① M20 and M21 are a pair of M commands for controlling the spindle clamp and release, and they are for clamping the spindle after positioning, so that the spindle won't rotate due to the force during drilling or tapping.

  ② During executing M20, to avoid the clamped spindle servo motor current too big, CNC will control the spindle and cut off the servo after the delay time set by DT23, and then, the spindle servo reduces the motor excitation, while the position control can't be executed. However the position detection can still functions and the position won't get lost.

  ③ The spindle can't be clamped by M20 during the spindle moving or rotating. After the spindle is clamped, the spindle can neither be rotated nor moved; otherwise, PLC issues the alarm.

2.9.11 Spindle Positioning Function

- **Relevant command**

  M14 Spindle shifts to the positioning control mode

  M15 Spindle shifts back to speed control mode

  Note: It is necessary to set the CS function of the spindle, and the spindle clamping equipment is matched with the machine tool

- **Control parameters**

  Status parameters

1	9	5								
			SSPD				RPOS		CSREF	RCS

Bit7 1:series spindle controllable function enabled (series spindle The slave ID number of the spindle 1~3 is 11~13) , it is necessary to restart；

0: Series spindle controllable function disabled (analogy spindle), it is necessary to restart.

Bit3 1: Spindle position function is valid (When it is enabled, the mechinial releasing/clamping interlocking operaton is performed before/after the spindle is positioned);

0: Spindle position function is invalid.

Bit1 1: Change to position mode, the axis do return REF.

0: Change to position mode, the axis does not return REF.

Bit0 1: Cs function of spindle valid, power on

0: Cs function of spindle is: invalid, power on

1	6	4	Spindle shifting M code

Spindle CS shifts the M code [Setting value +1 is the M code of the function releasing].

Note 1: When the spindle positioning function is enabled that is regarded as the M code of the spindle positioning method shifting.

Note 2: The setting value should be performed based upon compiled CS shifting M code in the ladder diagram.

Note 3: The setting value is disabled when it is less than 6, in this case, the CS axis shifting will not perform the coordinate system clearing.

- **Function description（Standard PLC program definition）**

① The system stops the detection spindle when executing the M14, simultaneously, output the spindle releasing signal; the spindle is then converted into positioning mode after confirming the spindle is released; the spindle is clamped after shifted.

② When the spindle is at the positioning mode, the arbitrary angle positioning based upon the axis address command can be performed by G00 command. Firstly output the spindle releasing signal when the position is performed, confirm that the spindle positions after releasing; output the spindle clamping signal after moves to the specified position, the positioning procedure is completed after confirming the spindle clamping.

③ The system will output the spindle releasing signal when executing M15; shift the spindle to the controllable mode after confirming that the spindle is released, the shifting is then performed.

④ Firstly set the Cs axis function before setting the spindle positioning function. When the positioning function setting is enabled, only the positioning operation can be performed instead of using the relevant function to the Cs contour control.

⑤ When the spindle positioning function is enabled, there are relevant operations between the spindle clamping and releasing before/after positioning. Therefore, the spindle clamping equipment should be matched with the machine tool.

2.9.12 External Cycle Start & Feed Hold

- **Relevant signal (defined by standard PLC program)**

ST: External cycle start signal, whose function is the same with the CYCLE START key on the machine panel;

SP: External feed hold signal, whose function is the same with the FEED HOLD key on the machine panel, and it shares an interface with SAGT(safety door detect) signal.

- **Signal diagnosis**

Signal	SP	ST
Diagnosis address	X9.1	X9.0
Interface pin	CN78.6	CN78.7

- **Signal connection**

The interior circuit of SP/ST signal is shown in Fig. 2-48:

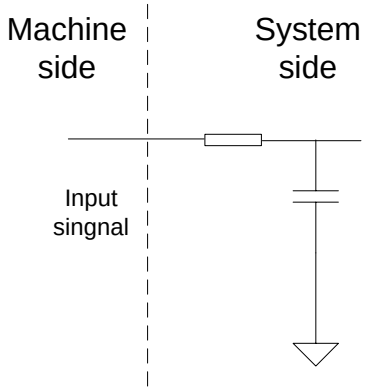


Fig. 2-48

- **Control parameter**

Bit parameter

1	7	2			MST	MSP					

Bit 6 =1: External cycle start signal (ST) is invalid;
=0: External cycle start signal (ST) is valid;
Bit 5 =1: External stop signal (SP) is invalid;
=0: External stop signal (ST) is valid; the external feed hold switch is needed, otherwise, “Feed hold” is displayed.

- **External connection circuit**

The external connection circuit of SP, ST signals is shown in Fig. 2-49:

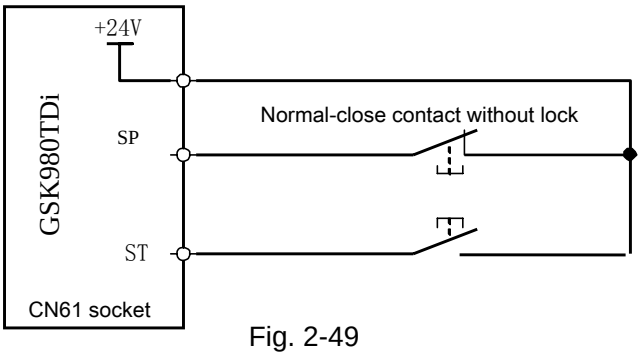


Fig. 2-49

2.9.13 Cooling Control

- **Relevant signal (defined by standard PLC program)**

Type	Sign	Interface	Address	Function	Remark
Output signal	M08	CN62.1	Y0.0	Cooling control output	
Command format	M08			Cooing ON	
	M09			Cooling OFF	

- **Signal connection**

Its internal circuit is shown in Fig. 2-50:

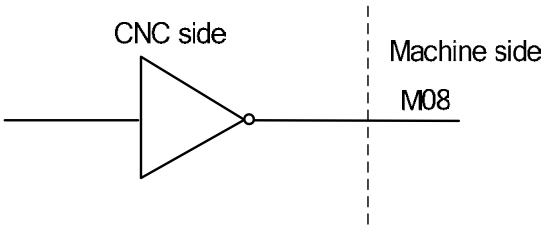


Fig. 2-50 M08 internal circuit

- **Function description (defined by standard PLC program)**

M09 is active, i.e. M08 is inactive, after CNC power on. To execute M08, M08 output is active and cooling is turned on; to execute M09, M08 output is cancelled and cooling is turned off.

Note 1: M08 output is cancelled at CNC emergency stop.

Note 2: Whether M08 is cancelled is set by BIT3 of the bit parameter No.009 when CNC is reset.
When Bit1=0, M08 output is cancelled as CNC is reset;
When Bit1=1, M08 output is not cancelled as CNC is reset;

Note 3: There is no corresponding output signal for M09, and M08 output is cancelled if M09 is executed.

Note 4: The cooling can be controlled by the **key on operation panel, see details in OPERATION.**

2.9.14 Lubricating Control

- **Relevant signal (defined by standard PLC program)**

Type	Sign	Interface	Address	Function	Remark
Output signal	M32	CN62.2	Y0.1	Lubricating control output	
Input signal	LUBS	CN61.33	X2.4	Lubricating position detection signal	
Command format	M32			Lubricating ON	
	M33			Lubricating OFF	

- **Signal connection**

The internal circuit is shown in Fig. 2-51:

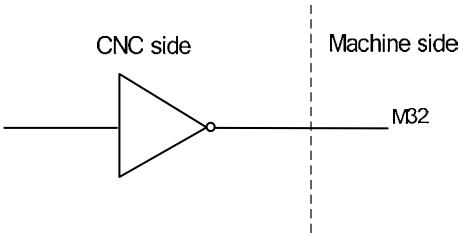


Fig. 2-51 M32 circuit

- **Control parameter**

PLC state parameter

K	1	0								RSJG	

RSJG =1: when is pressed, CNC does not close M03, M04, M08, M32 output signal.

=0: when is pressed, CNC closes M03, M04, M08, M32 output signal.

K	1	6					LUBS	LUBCK	M32A		

M32A =1: output lubricating in starting the system when the automatic lubricating is valid
=0: do not output lubricating in starting the system when the automatic lubricating is valid
PLC value
LUBCK =1: Detection with low lubricating position
=0: Without detection with low lubricating position
LUBS =1: Alarm with low lubricating position and low level
=0: Alarm with low lubricating position and high signal

D	T	1	3	MANLUBT

Time for manual lubricating ON (ms) (0: lubricating time is not limited);

D	T	1	6	AUTOLUBMT

Automatic lubricating time (0～2147483647ms)

D	T	1	7	AUTOLUBT

Automatic lubricating interval time (0～2147483647ms)

- **Function description**

The lubricating defined by standard PLC program for this GSK980TDi system has two types: Non-automatic and automatic lubricating, which are set by parameter:

DT17 =0: Non-automatic lubricating (same as version before)

DT17>0: Automatic lubricating, lubricating time DT17 and lubricating interval time DT16 available

1. Manual lubricating function

Press on the machine operator panel and the system executes the lubricating output, pressing it again and the system cancels the output. Execute M32 and the system executes the output; execute M33 and the system stops it.

DT13>1: the system executes the timed lubricating output, and the output is executed by pressing , and is cancelled when the output exceeds the one set by DT13. After the system executes M32, the lubricating output is executed within the time set by DT13 and it is cancelled. Execute M33 in the time set by No. 112 and the output is cancelled.

2. Automatic lubricating

When K16.2 is set to 1, the system executes the lubricating in the time set by DT17, and then stops the output. After it keeps the stop in the time set by DT16, it executes the lubricating again repetitively, and executes the cycle in turn. In automatic lubricating, M32, M33, on the machine panel are valid, and the lubricating time is the one set by DT17.

Note 1: The lubricating output is disabled in emergency stop;

Note 2: K No.0.10 Bit1 sets whether the lubricating output is cancelled in reset:

Bit1 = 0: the lubricating output is disabled in reset;

Bit1 = 1: the lubricating output remains unchanged in reset.

Note 3: The lubricating position detectiong signal is duplicatedly usable with DEC4, therefore, when the system uses the DEC4 signal, it is better to set the disabled lubricating position detection.

2.9.15 Chuck Control

- **Relevant signal (defined by standard PLC program)**

DIQP: Chuck control input signal

DOQPJ: Inner chuck clamping output/outer chuck releasing output signal

DOQPS: Inner chuck releasing output/outer chuck clamping output signal

NQPJ: Inner chuck clamping in-position/outer chuck releasing in-position signal, sharing a common interface with T08, M42I

WQPJ: Inner chuck releasing in-position/outer chuck clamping in-position signal, sharing a common interface with T07, M41I

Note 1: The key on the operation panel of GSK980TDi can replace DIQP input signal, and the diagnosis address is X0026.6.

- **Signal diagnosis**

Signal	DIQP	NQPJ	WQPJ	DOQPJ	DOQPS
Diagnosis address	X0.2	X3.4	X3.3	Y1.4	Y1.5
Interface pin	CN61.3	CN61.40	CN61.41	CN62.13	CN62.14

- **Control parameter**

K	**1**	**2**				**RCKC**	**CCHU**	**NYQP**	**SLSP**	**SLQP**

SLQP =1: Chuck control function is active;

= 0: Chuck control function is inactive;

SLSP =1: Check chuck clamping when the chuck function is active;

0: Does not check chuck clamping when the chuck function is active; if chuck is not clamping, the spindle is disabled and an alarm is issued.

NYQP =1: Outer chuck mode; NQPJ is outer chuck releasing signal, WQPJ is outer chuck clamping signal

=0: Inner chuck mode; NQPJ is inner chuck clamping signal, WQPJ is inner chuck releasing signal

CCHU =1: Check chuck in-position signal.

=0: Does not check chuck in-position signal.

RCKC =1: External chuck control is active during program running;

=0: External chuck control is inactive during program running;

D	**T**	**1**	**4**	

Chuck clamping command (M12) execution time (ms)

D	**T**	**1**	**5**	

Chuck released command (M13) execution time (ms)

DT0018	

DT18 >0: Chuck clamping and releasing signals are pulse output, the pulse width is set by DT18

=0: Chuck clamping and releasing signals are level output

- **Signal connection**

The circuit of DOQPJ/DOQPS is shown in Fig. 2-52:

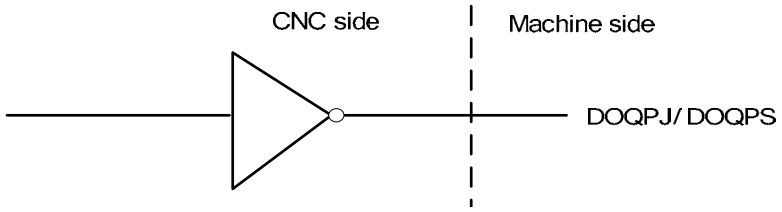


Fig. 2-52

- **Sequence**

①When SLQP=1, SLSP=0, NYQP=0, CCHU=1, CNC chooses inner chuck mode, and chuck in-position signal detecting is active:

DOQPS: chuck releasing output; WQPJ: releasing in-position signal;

DOQPJ: chuck clamping output; NQPJ: clamping in-position signal.

DOQPJ and DOQPS output high resistance at power on, when CNC detects that the chuck input signal DIQP is active for the 1st time, DOQPJ is connected to 0V and chuck is clamped.

After M12 is executed, DOQPS (pin 14 of CN62) outputs high resistance, DOQPJ(pin 13 of CN62) outputs 0V, chuck is clamped and CNC waits for NQPJ signal to be in-position.

After M13 is executed, DOQPJ (pin 13 of CN62) outputs high resistance, DOQPS(pin 14 of CN62) outputs 0V, chuck is released and CNC waits for WQPJ signal to be in-position.

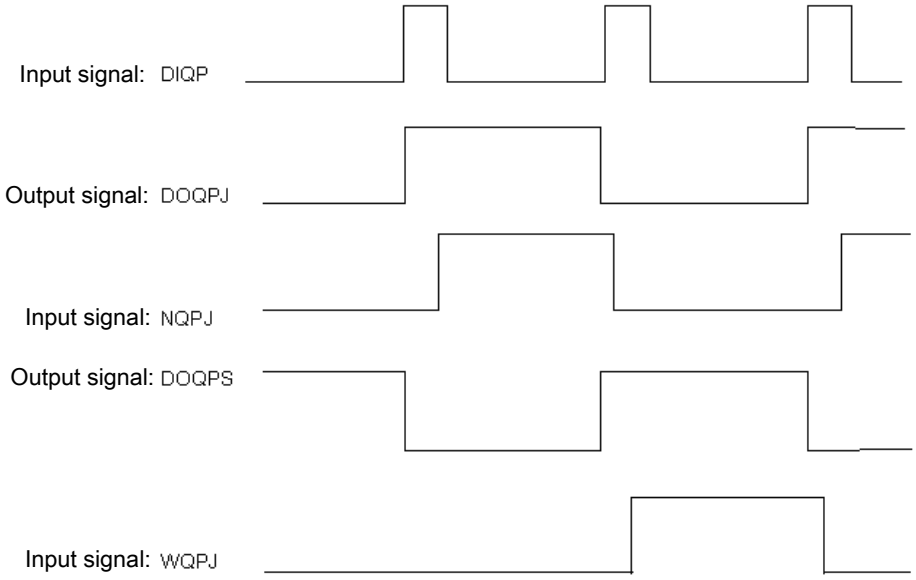


Fig. 2-53 (Chuck clamping, releasing signals are level output)

②When SLQP=1, SLSP=0, NYQP=1, CCHU=1, CNC chooses outer chuck mode, and chuck in-position signal detecting is active:

DOQPS: chuck clamping output; WQPJ: clamping in-position signal;

DOQPJ: chuck releasing output; NQPJ: releasing in-position signal.

DOQPJ and DOQPS output high resistance at power on, when CNC detects that the chuck input signal DIQP is active for the 1st time, DOQPS is connected to 0V and chuck is clamped.

After M12 is executed, DOQPS (CN62.14) outputs 0V, DOQPJ(CN62.13) outputs high resistance, chuck is clamped and CNC waits for WQPJ signal to be in-position.

After M13 is executed, DOQPJ (CN62.13) outputs 0V, DOQPS(CN62.14) outputs high resistance, chuck is released and CNC waits for NQPJ signal to be in-position.

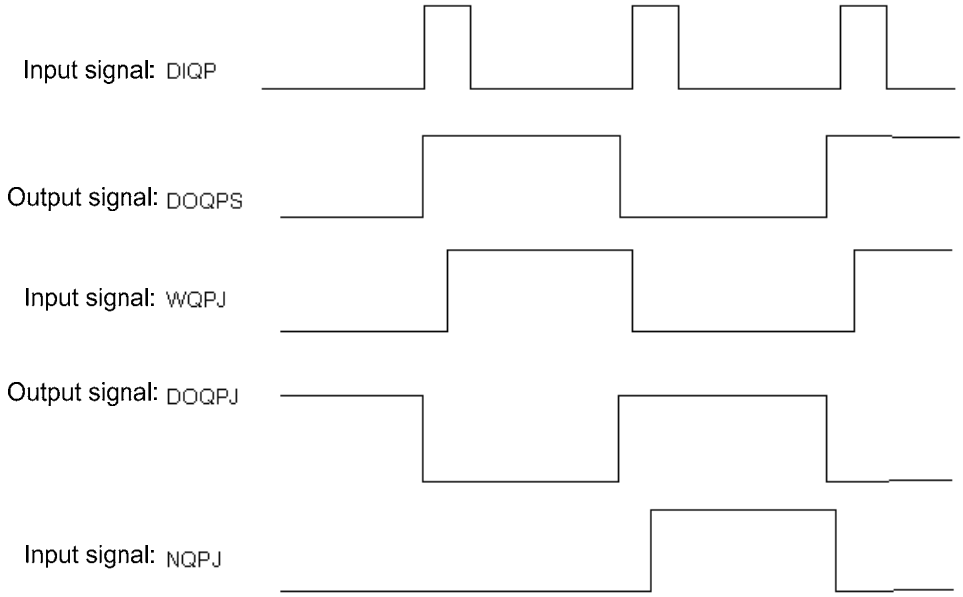


Fig. 2-54 Chuck clamping, releasing signals are level output

As the 2nd chuck input is active, DOQPS outputs 0V, chuck is released. The chuck clamping/releasing signal is output alternatively, i.e. the output is changed each chuck input signal is active.

③The interlock between the chuck and the spindle

When SLQP=1, SLSP=0, M3 or M4 is active, the alarm is issued if M13 is executed and the output is unchanged.

When SLQP=1, SLSP=0, CCHU=1, if M12 is executed in MDI or Auto mode, CNC does not execute next code till it detects the chuck clamping in-position signal is active. When the chuck input signal DIQP is active in Manual mode, the panel spindle CW, CCW key are inactive till it detects the chuck clamping in-position signal is active. In spindle running or auto cycle processing, DIQP input signal is inactive. And DOQPS, DOQPJ is held on at CNC reset and emergency stop.

2.9.16 Tailstock Control

- **Relevant signal (defined by standard PLC program)**

  DOTWJ: Tailstock forward output signal

  DOTWS: Tailstock backward output signal

  DITW: Tailstock input signal, DITW and BDT share a common interface.

Note: For GSK980TDi, can replace DITW input signal, and the diagnosis address is X0026.5.

- **Signal diagnosis**

Signal	DITW	DOTWJ	DOTWS
Diagnosis address	X0.4	Y2.5	Y2.6
Interface pin	CN61.5	CN61.34	CN61.35

- **Control parameter**

State parameter

K	1	3							RTSC	SPTW	SLTW

SLTW =1: Tailstock function active.

=0: Tailstock function inactive.

SPTW =1: No interlock between spindle rotation and tailstock advancing and retracting, tailstock may be moved regardless of the spindle, or spindle may run regardless of tailstock;

=0: Interlock between spindle rotation and tailstock advancing and retracting, tailstock retraction disabled as spindle is running, spindle disabled if tailstock does not advance.

RTSC =1: External tailstock control is active when a program runs.

=0: External tailstock control is inactive when a program runs

- **Signal connection**

The tailstock circuit is shown in Fig. 2-55:

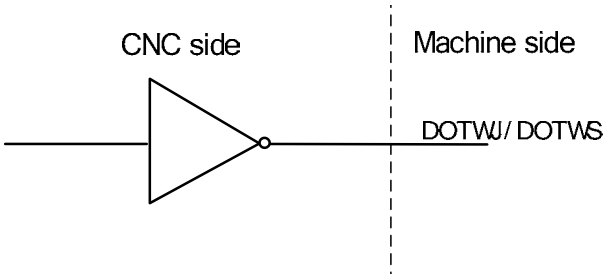


Fig. 2-55

- **Sequence (defined by standard PLC program)**

The sequence of tailstock is shown in Fig. 2-56:

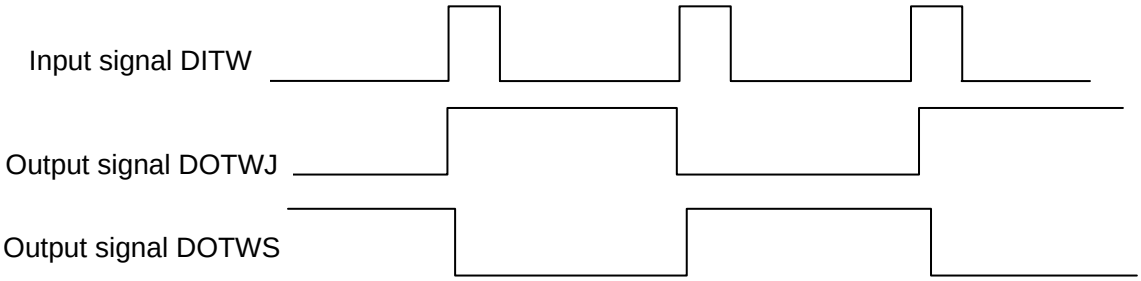


Fig. 2-56 Tailstock sequence

Tailstock advancing (DOTWJ) and retracting(DOTWS) are both inactive when power on; when the tailstock input (DITW) is active for the 1st time, tailstock advancing is active; when it is active for the 2nd time, tailstock retracting is active, so the DOTWJ/ DOTWS signal interlock is output alternatively, i.e. The output changes each time the DITW signal is active. If M10 is executed, DOTWJ (CN62.34) outputs 0V and tailstock advances; if M11 is executed, DOTWS (CN62.35) outputs 0V and tailstock retracts.

DITW signal is inactive as spindle is running. If M11 is executed, alarm will be issued, and its

output are held on. And DOTWS, DOTWJ outputs are held on at CNC reset or emergency stop.

2.9.17 Low Pressure Detection

- **Relevant signal (defined by standard PLC program)**

  PRES: Low pressure alarm detection signal, sharing a common interface with TCP.

- **Signal diagnosis**

Signal	PRES
Diagnosis address	X0.6
Interface pin	CN61.7

- **Control parameters**

  State parameter

K	**1**	**4**								**SPB3**	**PB3**

PB3 =0: Low pressure detection inactive.

=1: Low pressure detection active.

SPB3 =0: Low pressure alarm issued when PRES is connected with +24V.

=1: Low pressure alarm issued when PRES is connected with 0V.

PLC data parameter

D	**T**	**0**	**2**	**PEALMTIM**

PEALMTIM: Waiting time before low pressure alarm(0~60000ms)

- **Function description**

①When PB3=1 and SPB3=0, PRES is connected to +24V, which is confirmed to be low pressure alarm by CNC;

②When PB3=1 and SPB3=1, PRES is connected to 0V, which is confirmed to be low pressure alarm by CNC;

③When low pressure alarm detection function is selected, once CNC detects the low pressure alarm signal PRES to be active, and the signal holding time is over the setting by the data parameter No.069 alarm is issued by CNC. And the axis feed is held, spindle is stopped, cycle start is disabled, the alarm may be cancelled by pressing key or cutting off power;

2.9.18 Hydraulic Control

- **Related signal（defined by the standard PLC program）**

  HPST: hydraulic control output signal

- **Signal diagnosis**

Signal	**HPST**
Diagnosis address	Y3.1
Interface pin	CN62.38

- **Control parameter**

PLC state parameter

K	1	4		HPST							

PB3 =0: the hydraulic control function is disabled.
=1: the hydraulic control function is enabled.

- **Function description**

① When the hydraulic control function is enabled, pressing can control HPST output to start the hydraulic motor.

② In non-run, the spindle stop with the speed 0, pressing can disable HPST output.

③ When the hydraulic control function enables and the hydraulic motor is started, the system alarm, at the time, the spindle cannot rotate.

2.9.19 Safety Door Detection

- **Relevant signal**

SAGT: For safety door detection, sharing a common interface with SP signal
SLK: Defense door lock control output. Only GSK980TDi-H is enabled.

Note: SLK defense door controllable function. Only GSK980TDi-H is enabled.

- **Signal diagnosis**

Signal	SAGT	SLK
Diagnosis address	X0.0	Y2.7
Interface pin	CN61.1	CN62.36

- **Control parameter**

- PLC status parameters

K	**1**	**4**			**SLK**	**SGSP**	**SPB4**	**PB4**		

PB4 =0: Protection door detection function is invalid;
=1: Protection door detection function is valid;
SPB4 =0: When SAGT is at low level (disconnect with +24V), the protection door is closed;
=1: When SAGT is at high level (connected with +24V), the protection door is closed;
SGSP =0: Spindle is disabled when the protection door is open;
=1: Spindle is not disabled when the protection door is open;
SLK =0: (980TDi-H) defense door disabled.
=1: (980TDi-H) defense door enabled.

PLC data parameter

D	**T**	**2**	**8**	

Delay the controllable time (ms) of the defense door close

- **Function description (defined by standard PLC program)**

① When PB4＝1, SPB4＝0, CNC confirms that the safety door is closed as SAGT is connected to 0V;

② When PB4＝1, SPB4＝1, CNC confirms that the safety door is closed as SAGT is connected to +24V;

③ In Auto mode, if CNC detects the safety door is open, alarm is issued as cycle starts;

④ In Auto running, if CNC detects the safety door is open, the axis feed is held, and alarm is issued by CNC;

⑤ The safety door detection function is only active in Auto mode;

⑥ When the defense door lock function is enabled, in the Auto/MDI mode, the program should be at the dwell state by opening the door lock pressing the “safety door” button on the panel. Firstly close the spindle and cooling pressing the defense door lock button on the panel; the defense door lock output when detecting that the spindle rotation speed is zero; in this case, the defense door can be opened. The feed dwells when opening the lock by M18, the others are consistent with the buttons on the button.

⑦ The spindle or cooling will not be closed in Manual/MPG mode.

⑧ Defense door lock controllable output, close the door lock controllable output after delaying the DT28 setting time. When the defense door becomes close state from the open state, the door lock controllable outputs are entirely closed regardless of the DT28 timing is performed or not.

2.9.20 CNC Macro Variables

- **Relevant signal**

Macro output signal: standard PLC defines 5 macro output interfaces #1100～#1105；

Macro input signal: standard PLC defines 16 macro output interfaces #1000～#1015

- **Signal diagnosis**

Macro variable number	#1105	#1104	#1103	#1102	#1101	#1100
Diagnosis address	Y3.7	Y3.6	Y3.5	Y3.4	Y3.3	Y3.2

Macro variable number	#1007	#1006	#1005	#1004	#1003	#1002	#1001
Diagnosis address	X0.7	X0.6	X0.5	X0.4	X0.3	X0.2	X0.1

Macro variable number	#1015	#1014	#1013	#1012	#1011	#1010	#1009
Diagnosis address	X1.7	X1.6	X1.5	X1.4	X1.3	X1.2	X1.1

- **Function description (defined by standard PLC program)**

U00～U05 signal output may be changed if macro variable ＃1100～＃1105 are assigned. If they are assigned for “1”, it outputs 0V, if they are assigned for “0”, it turns off their output signals.

Detect the macro variable ＃1000～＃1015 values (input signal state), they may be used for various processing if combined with other transfer judgment macro code.

2.9.21 External Override

- **Related signals**

Type	Sign	Interface	Address	Function	Remark
Input signal	CN67.1	FOV0	X29.0	Feed override wave-band switch 0	
	CN67.6	FOV1	X29.1	Feed override wave-band switch 1	
	CN67.2	FOV2	X29.2	Feed override wave-band switch 2	
	CN67.5	FOV3	X29.3	Feed override wave-band switch 3	
	CN66.1	SOV0	X29.4	Spindle override wave-band switch 0	
	CN66.6	SOV1	X29.5	Spindle override wave-band switch 1	
	CN66.2	SOV2	X29.6	Spindle override wave-band switch 2	
	CN66.5	SOV3	X29.7	Spindle override wave-band switch 3	

- **Related parameters**

PLC state parameter

K	1	6								ROVI	SOVI

ROVI =0: External feedrate override does not inverse.
=1: External feedrate override inverses.

SOVI =0: External feedrate override switch is disabled.
=1: External feedrate override switch is enabled.

- **Function description**

1. When the external feedrate override switch is selected, the button on the panel is disabled.
2. When the external feedrate override inverses, the system changes the direction of the override regulation.

2.9.22 External MPG

- **Related signals**

CN31(MPG)	PLC address	Address character	Function	Remark
5	X6.0	EHDX	X MPG	Applied toPSG-100-05E/L, ZSSY2080 MPG
6	X6.1	EHDY	Y MPG	
8	X6.2	EHDZ	Z MPG	
19	X7.0	EHD4	4th MPG	
7	X7.1	EHD5	5th MPG	
9	X6.3	EMP0	Increment ×1	
22	X6.4	EMP1	Increment ×10	
23	X6.5	EMP2	Increment ×100	
11, 12, 13	GND			
14,15	+5V			
17,18	+24V			

- **Related parameters**

State parameter

0	0	1						MPG			

Bit3 =0: Step working mode.

=1: MPG working mode.

PLC status parameters

K	1	6	SINC								

SINC =0: When the system increment is 0.1μ, gear ×1000 in MPG and STEP mode is valid;

=1: When the system increment is 0.1μ, gear ×1000 in MPG and STEP mode is invalid;

K	2	0								DBMPG1	DBMPG0

DBMPG 0 Dual-MPG function selection bit 0; 00: Disabled 01: Enabled, as well the MPG 1 fixes the X axis and the MPG 2 fixes the Z axis, the shared step-length can be selected;

DBMPG 1 Dual-MPG function selection bit 1; 10: It is enabled and its panel button is only enabled by MPG 1; external MPG input is only enabled by MPG 2.

- **Function description**

① MPG/STEP mode ×1000-gear selection is disabled. When x1000-gearis selected before modifying the parameter, the system automatically changes into ×100-gear;

② When the external MPG, its axis selection does not lock, that is, the axis selection of MPG is disabled, the system changes to the non-axis selection state.

③ When the external MPG axis selection and gear selection input are enabled, the axis selection on the panel and the gear selection keys are disabled; when the external MPG axis selection and gear selection input are disabled, the axis selection on the panel and the gear selection keys are enabled and self-locked.

④ When the dual-MPG is enabled, which can be selected the dual-MPG controllable mode by parameters K20.1 and K20.0.

⑤ When K20.1=0 K20.0=1, MPG 1 fixes the controllable X axis, MPG 2 fixes the controllable Z axis; the step-length values of two MPGs are identical, which can be adjusted by step-length on the panel or external step-length input signal.

⑥ When K20.1=1 K20.0=0, the panel MPG axis selection or step-length button only can be controlled the MPG 1, and the external MPG input signal can be controlled the MPG 2.

2.9.23 Tri-color Lamp

- **Relevant signals and function definition (standard PLC program)**

Signal type	Sign	Signal interface	Address	Function
Output signal	CLPY	CN62.31	Y2.2	Yellow— normal state (non-running, non-alarm state)
	CLPG	CN62.32	Y2.3	Green—running state
	CLPR	CN62.33	Y2.4	Red—alarm state

2.9.24 Extended M Function

- **Relevant signals（standard PLC program）**

Signal interface	Address	Address characters	Function
CN62.40	Y3.3	EMOT1	Extended M function signal 1
CN62.41	Y3.4	EMOT2	Extended M function signal 2

- Relevant commands

M34 Extended M function signal 1（EMOT1）outputs

M35 Extended M function signal 1（EMOT1）closes output

M36 Extended M function signal 2（EMOT2）outputs

M37 Extended M function signal 2（EMOT2）closes output

- Control parameters

PLC parameters

K	1	8		EXTM	RSTM						

EXTM =0:extended M function is invalid.

=1:extended M function is valid.

RSTM =0:extended M function closes output when emergency stop, reset.

=1:extended M function remains output when emergency stop, reset.

- Function explanation

1. When M34 is executed, extended M function 1 (EMOT1) outputs; when M35 is done, its output is closed.
2. When M36 is executed, extended M function 2 (EMOT2) outputs; when M37 is done, its output is closed.
3. Whether extended M function output is closed is set by K18.6 when emergency stop, reset.

2.9.25 HOLD Control

- **Relevant signals（standard PLC program）**

Signal interface	Address	Address characters	Function
CN62.03	Y0.2	HOLD	HOLD release output

- Control parameter

PLC state parameter

K	1	8				HOLD					

HOLD =0:HOLD release control is invalid.

=1:HOLD release control is valid.

- Function

1. After power-on, there is no emergency stop alarm or servo spindle alarm and the system delays 4S, HOLD signal is output to release HOLD.
2. HOLD signal is closed when an emergency stop alarm or servo spindle alarm occurs.

2.9.26 Handle Interrupt

- **Control parameters**

PLC Status parameters

K	1	9				HDFD					

HDFD =0: MPG interruption function is disabled in the process of the operation.
=1: MPG interruption function is enabled in the process of the operation.

0	**1**	**2**								**MPG interruption**	

Bit1 1:Handle interrupt is valid;
0:Handle interrupt is invalid。

- **Function description**

1. When the MPG interruption function is enabled, in the Edit/Auto/MDI mode, enter the MPG interruption state by the MPG axis number on the selection panel after pressing the once, the MPG interruption operation is then can be performed. The MPG interruption can be retracted by pressing this button again.
2. In the Non-Edit/Auto/MDI mode, the MPG interruption state will be retreated.
3. Whether the MPG interruption is enabled that can be set by K19.5 during the operation of the program.
4. The panel rapid traverse override can not be adjusted at the state of MPG interruption; the indicator displays the MPG override state when the MPG override adjustment is enabled.

2.9.27 Chip-Removal Control

- Relative signals

Signal type	Symbol	Signal interface	Address	Function description	Remark
Signal output	CHRO+	CN62.43	Y3.6	Chip-removal CW (positive)	
	CHRO-	CN62.44	Y3.7	Chip-removal CCW (negative)	
Command format	M38			Chip-removal CW (positive)	
	M39			Chip-removal stop	

- **Relevant parameters**

PLCStatus parameters

K	**1**	**3**	**CHIPR**							

CHIPR =1: Chip-removal control enabled.
=0: Chip-removal control disabled.

- Function description

1. The reverse rotation of the chip-removal only can be performed the JOG by hand. If the chip-removal is being rotated, firstly stop the positive rotation when the JOG reverse is performed.
2. When the chip-removal is positively rotated, press the chip-removal positive rotation by hand again, and then it is closed.
3. Close the chip-removal rotation in the ESP resetting.

2.9.28 Working Indicator

- Relative signals

Signal type	Symbol	Signal interface	Address	Function description	Remark
Output signal	WKLP	CN62.39	Y3.2	Working indicator output	

- Relevant parameters

PLCStatus parameters

K	1	3				WKLP					

WKLP =1: Working indicator control enabled.

=0: Working indicator control disabled.

- Function description

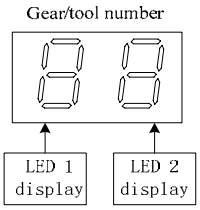
The working indicator outputs pressing once; it closes by pressing it again.

2.9.29 Gear/tool Number Display

- **Signal diagnosis**

Y0025	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0
Corresponding BCD code	LED1 2^3	LED1 2^2	LED1 2^1	LED1 2^0	LED0 2^3	LED0 2^2	LED0 2^1	LED0 2^0

LED 0 displays the current tool number, BIT0~BIT3 of PLC diagnosis number Y25 are BCD code of the current tool number; LED 1 displays the gear of the current spindle, BIT4~BIT7 of PLC diagnosis number Y25 are BCD code of the current spindle gear and is shown below:



- **Function description**

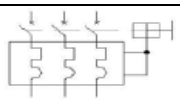
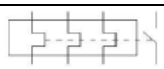
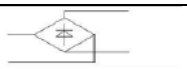
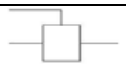
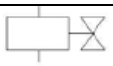
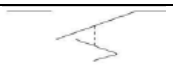
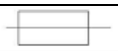
① When the tool count is less than 10, the gear and the tool number separately use one LED to display.

② When the tool count is more than 10, LED 1 displays 10-digit numbers and the LED 0 displays the single digits.

③ When the gear and the tool number are displayed in time-division, that is, the gear instead of the tool number is firstly display, the tool number instead of the gear is displayed after 1.5S, and the system executes them like the above cycle.

2.10 Commonly Use Sign of Electricity Drawing

GSK980TDi DC24V power supply and the electromagnetic valve with power working current separately use DC24V, and the electronic component explanations are as follows:

Name	Sign	Graph	Name	Sign	Graph
Air breaker	QF		Contactor coil contact and auxiliary contact	KM	
transformer	TC		Heat relay and contact	FR	
Rectifier	VC		Capacity	C	
Motor	M		Resistant	R	
Diode	VD		Hall switch		
Electromagnetic coil	YV		Travel switch	SQ	
Relay coil and contact	KA		Veneer socket		
			Pedal switch	SA	
			Fuse	FU	

CHAPTER 3 PARAMETERS

The CNC bit and data parameters are described in this chapter, various functions can be set by these parameters. The major parameter data type are listed as follows:

Data type		Range
(1) Bit	0001 00011000	8 bit, 0 or 1
(2) Bit axis	0187 X 00000010 ---- Z 00000010	
(3) Bit spindle		
(4) Word	0030 200	See relevant parameters for details
(5) Word axis	0015 X 1 ---- Z 1	
(6) Word spindle	0021 S1 0 ---- S2 0	

Note: if the data type is not specified, the type is bit or word.

3.1 Parameter Description (by Sequence)

3.1.1 Bit Parameter

The state parameter is expressed as follows:

Parameter No.	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0
0 0 1	***	***	***	ACS	MPG	PROG	ISC	INI

Bit4 1: Spindle analog voltage control
0: Spindle switching volume control
Bit3 1: MPG mode
0: Step mode
Bit2 1: Programming by radius
0: Programming by diameter
Bit1 1: IS-C incremental system
0: IS-B incremental system

Increment system	Least input increment, least command increment	Abbreviation
0	0.001mm,0.0001inch	**IS-B**
1	0.0001mm, 0.00001inch	**IS-C**

Bit0 1: Inch input
0: Metric input

0 0 2	CLK	***	***	LIFJ	MDITL	LIFC	TNRC	TLIF

Bit7 1: Show clock
0: Conceal clock
Bit4 1: Tool life management group skip active
0: Tool life management group skip inactive
Bit3 1: Tool life management active in MDI mode
0: Tool life management inactive in MDI mode
Bit2 1: Tool life counting type 2, by times
0: Tool life counting type 1, by times
Bit1 1: Tool nose radius offset active
0: Tool nose radius offset inactive
Bit0 1: Tool life management active
0: Tool life management inactive

0	0	3

HTCHG	***	PCOMP	TCOMP	CDCOMP	***	***	OIM

Bit7 1: High efficiency tool-change function enabled
0: High efficiency tool-change function disabled
Bit5 1: Pitch error offset active
0: Pitch error offset inactive
Bit4 1: Tool offset by coordinate offset
0: Tool offset by move
Bit3 1: Tool compensated by the shift of coordinate system do not memorize the unexecuted tool compensation
0: Tool compensated by the shift of coordinate system memorize the unexecuted tool compensation
Bit0 1: Offset automatically change in metric and inch conversion
0: Offset not change in metric and inch conversion

0	0	4

ABOT	RDRN	DECI	ORC	***	***	PROD	SCW

Bit7 1: do not memorize absolute coordinate in power-down
0: memorize the absolute coordinate in power-down
Bit6 1: G00 is rapid traverse speed in dry run mode
0: G00 is manual feedrate in dry run mode
Bit5 1: DEC signal is high level for machine zero return
0: DEC signal is low level for machine zero return
Bit4 1: Tool offset by radius
0: Tool offset by diameter
Bit1 1: Relative position for programming in POSITION page
0: Relative position involving offset in POSITION page
Bit0 1: Inch system for min. code unit, active after repowering
0: Metric system for min. code unit, active after repowering

0	0	5

***	***	SMAL	M30	***	POSRE	PPD	PCMD

Bit 5 1: Manual spindle gear change during the execution of S code;
0: Automatic spindle gear change during the execution of S code;
Bit 4 1: Cursor returns to the head of the program after the execution of M30;
0: Cursor does not return to the head of the program after the execution of M30;

Bit 2: 1: Position recording signal PRC is used during tool setting;
0: Position recording signal PRC is not used during tool setting;
Bit 1: 1: G50 sets relative coordinate value;
0: G50 does not set relative coordinate value;
Bit 0: 1: Axial output wave is pulse;
0: Axial output wave is square;

0	0	6

***	***	***	ZM5	ZM4	ZMY	ZMZ	ZMX

Bit4~Bit0 1: 5th, 4th, Y, Z, X zero return type C
0: 5th, 4th, Y, Z, X zero return type B

0	0	7

DISP	SMZ	***	ZC5	ZC4	ZCY	ZCZ	ZCX

Bit7 1: Enter ABSOLUTE POS page after power on
0: Enter RELATIVE POS page after power on
Bit6 1: Execute the next block after all motion block exactly are executed to the in-position
0: Smooth transition between two blocks
Bit4~Bit0 1: In machine zero return, 5th, 4th, Y, Z, X deceleration signal (DEC5~DEC1) and one-revolution signal (PC5~PC1) are parallel(use one proximity switch as the deceleration signal and zero signal)
0: In machine zero return, 5th, 4th, Y, Z, X deceleration (DEC5~DEC1) and one-revolution signal (PC5~PC1) is separately connected (need the separate deceleration signal and zero signal)

Note: In the zero-return B method; when it is set to 0 and connected independently, it is necessary to detect the PC signal after releasing the block during the zero-return; when it is set to 1 and connected in parallel, the PC signal detects with block after the pressing block decelerates to zero during the zero-return method.

0	0	8

***	***	***	DIR5	DIR4	DIRY	DIRZ	DIRX

Bit4~Bit0 1: The motor's rotation diretion of the 5th, 4th, Y, Z and X axes are positive;
0: The motor's rotation diretion of the 5th, 4th, Y, Z and X axes are negative;

Note: The movement direction of the machine tool is inconsistent with the actual direction, which can be modified by adjusting this parameter.

0	0	9

***	***	***	5ALM	4ALM	YALM	ZALM	XALM

Bit4~Bit0 1: Servo not ready alarm without detection of the 5th, 4th, Y, Z or X axis.
0: Servo not ready alarm detection of the 5th, 4th, Y, Z or X axis.

0	1	0

***	***	***	CPF5	CPF4	CPF3	CPF2	CPF1

Bit0~ Bit4: Pulse frequency setting values of backlash offset (by BCD code)

The set frequency =(setting value+1) Kpps

CPF5	CPF4	CPF3	CPF2	CPF1	Set frequency (Kpps)
0	0	0	0	0	1
0	0	0	0	1	2
0	0	0	1	0	3
0	0	0	1	1	4
0	0	1	0	0	5

0	0	1	0	1	6
0	0	1	1	0	7
0	0	1	1	1	8
0	1	0	0	0	9
0	1	0	0	1	10
0	1	0	1	0	11
0	1	0	1	1	12
0	1	1	0	0	13
0	1	1	0	1	14
0	1	1	1	0	15
0	1	1	1	1	16
1	0	0	0	0	17
1	0	0	0	1	18
1	0	0	1	0	19
1	0	0	1	1	20
1	0	1	0	0	21
1	0	1	0	1	22
1	0	1	1	0	23
1	0	1	1	1	24
1	1	0	0	0	25
1	1	0	0	1	26
1	1	0	1	0	27
1	1	0	1	1	28
1	1	1	0	0	29
1	1	1	0	1	30
1	1	1	1	0	31
1	1	1	1	1	32

0	**1**	**1**

BDEC	**BD8**	***	***	**NORF**	**ZNIK**	**REFC2**	***

Bit7 1: Backlash offset type B, the offset data are output by ascending or descending type and the set frequency is inactive;

0: Backlash offset type A, the offset data are output by the set frequency (by bit parameter No.010) or 1/8 of it.

Bit6 1: Backlash offset is done by the 1/8 of the set frequency

0: Backlash offset is done by the set frequency

Bit3 1: Manual machine zero return inactive

0: Manual machine zero return active

Bit2 1: Direction key locked during zero return, homing continues to end by pressing direction key once;

0: Direction key unlocked, which is held on during zero return

Bit1 1: Ref.C mode, the feeding slowdown when pressing down switch, reverse after decelerating to zero.

0: Ref.C mode, the feeding slowdown when releasing the switch, reverse after decelerating to zero.

0	1	2		APRS	WSFT	DOFSI	***	EAL	***	HDINT	ISOT

Bit7 1: Automatic ABSOLUTE set after reference return,;
0: ABSOLUTE not automatically set after reference return
Bit6 1: Workpiece coordinate offset active, defined by offset No.000
0: Workpiece coordinate offset inactive
Bit5 1: Trial tool setting active
0: Trial tool setting inactive
Bit3 1: Program editing allowed during CNC alarming
0: Program editing unallowed during CNC alarming
Bit1 1:Handle interrupt is valid (power on)
0:Handle interrupt is invalid (power on)
Bit0 1: Prior to machine zero return after power on, manual rapid traverse active
0: Prior to machine zero return after power on, manual rapid traverse inactive

0	1	3		RTH	RPH	***	HW5	HW4	HWY	HWZ	HWX

Bit7 1: When system reset, the handle interrupt move amount is cancelled
0: When system reset, the handle interrupt move amount is not cancelled
Bit6 1: When MPG feedrate exceeds the rapid movement speed,the exceeding part is not ignored and saved in CNC
0: When MPG feedrate exceeds the rapid movement speed,the exceeding part is ignored and saved in CNC
Bit4~Bit0 1: Coordinates increase in 5th, 4th, Y, Z or X MPG (CCW) rotation
0: Coordinates increase in 5th, 4th, Y, Z or X MPG (CW) rotation

0	1	4		***	***	***	ZRS5	ZRS4	ZRSY	ZRSZ	ZRSX

Bit 4, Bit3, Bit2, Bit1, Bit0
1: 5th, 4th, Y, Z, X have the machine zeroes. The system needs to check the deceleration signal and zero signal in executing the machine zero return
0: 5th, 4th, Y, Z, X have no machine zeroes. The system does not check the deceleration signal and zero signal till it returns to the zero of machine coordinate in executing the machine zero return

0	1	7		SCRD	***	RSCD	***	TSD	QNI	TSL	TS1

Bit7 1:Coordinate system is stored before power-off;
0: Coordinate system is not stored before power-off; the default system is set by G54 after power-on;
Bit5 1: Coordinate system is set by G54 after reset;
0: Coordinate system is not changed after reset;
Bit3 1: Movement direction discrimination is valid during offset measurement 4 point input;
0: Movement direction discrimination is invalid during offset measurement 4 point input;
Bit2 1: Tool offset number is set through G signal;
0: Tool offset number is set through the current cursor;
Bit1 1 Tool offset measurement detection signal is valid at low level;
0: Tool offset measurement detection signal is valid at high level;
Bit0 1:Tool offset automatic measurement detection signal 1 point input;
0: Tool offset automatic measurement detection signal 4-point input;

1	7	2

***	MST	MSP	MOT	ESP	***	LZR	MOTDEC

Bit6 1: External cycle start signal (ST) inactive)
0: External cycle start signal (ST) active
Bit5 1: External SP signal inactive
0: External SP signal active with external dwell switch needed, otherwise CNC shows "Pause".
Bit4 1: Not detect software stroke limit
0: Detect software stroke limit
Bit3 1: Emergency stop inactive
0: Emergency stop active
Bit1 1: Software limit is inactive after power-on and before machine zero return;
0: Software limit is active after power-on and before machine zero return;
Bit0 1: When Arriving stored travel position, CNC is stopped slowly
0: When Arriving stored travel position, CNC is stopped immediately

1	7	4

PARTSET	***	TRPMOD	TRP	KEY1	THDTLMOD	THDA	CKSPD

Bit7 1:Continue to run part program while current machining workpiece counts reach to max. counts.and machining workpiece counts accumulate
0:Continue to run part program while current machining workpiece counts reach to max. counts.and machining workpiece counts restart
Bit5 1:Screw repair mode use follow the spindle with encoder
0:Screw repair mode use interpolation with servo spindle
Bit4 1:Thread repair function is valid
0:Thread repair function is invalid
Bit3 1: Program switch ON at power on
0: Program switch OFF at power on
Bit2 1:Thread tailing mode tailling according angle;
0:Thread tailing mode rapid tailing.
Bit1 1:Thread cut is exp. acc&dec
0:Thread cut is lin. acc&dec
Bit0 1:Whether detect the spindle speed is too slow in thread cutting don't detect\
0:Whether detect the spindle speed is too slow in thread cutting do detect

1	7	5

SPFD	SAR	THDA	VAL5	VAL4	VALY	VALZ	VALX

Bit7 1: In cutting feed, do not permit the spindle stopping rotation; the spindle stops, the feed also stops when the system appears No.404 alarm cutting
0: In cutting feed, permit the spindle stops rotation; the spindle stops rotation, but the feed does not stop
Bit6 1: Detect spindle SAR signal prior to cutting
0: Not detect spindle SAR signal prior to cutting
Bit5 1: Thread machining is the exponential acceleration/deceleration
0: Thread machining is the linear acceleration/deceleration
Bit4 1: 5th movement key is positive, is negative
0: 5th movement key is positive, is negative

Bit3 1: 4th movement key is positive, is negative

0: 4th movement key is positive, is negative

Bit2 1: Y movement key is positive, is negative

0: Y movement key is positive, is negative

Bit1 1: Z movement key is positive, is negative

0: Z movement key is positive, is negative

Bit0 1: X movement key is positive, is negative

0: X movement key is positive, is negative

1	8	0

NAT	***	***	***	DISPC	CUSTOM	MLKEY	SPOS

Bit7 1: Function ATAN, ASIN range is 90.0～270.0;

0: Function ATAN, ASIN range is -90.0～90.0

Bit3 1:Boot dispaly if custom page setted to display custom page

0:Boot dispaly if custom page setted to display fault page

Bit2 1:Display custom page display

0:Display custom page hide

Bit1 1:Multi-letter key input mode shift mode only\

0:Multi-letter key input mode repeat keystroke or shift mode

Bit0 1: DIS TO GO displayed in POS&PRG page

0: RELATIVE POS displayed in POS&PRG page

1	8	1

RLC	MRC	ZCL	CLMDI	MCL	M30&MDI	DDP	LCNMD

Bit7 1: G52 local coordinate system is cancelled after reset;

0: G52 local coordinate system is not cancelled after reset;

Bit6 1: G52 local coordinate system is cancelled after the execution of M30, M02;

0: G52 local coordinate system is not cancelled after the execution of M30, M02;

Bit5 1: G52 local coordinate system is cancelled after reference point return;

0: G52 local coordinate system is not cancelled after reference point return;

Bit4 1:When system reset,the program block of MDI are deleted

0:When system reset,the program block of MDI are not deleted

Bit3 1: Blocks edited in MDI mode are deleted after reset;

0: Block edited in MDI mode are not deleted after reset;

Bit2 1:After the execution of M02, M30,the program block of MDI are not deleted

0:After the execution of M02, M30,the program block of MDI are deleted

Bit1 1:The angle in CHF/CNR measurement-programe-inputing function was specified by Supplementary one

0:The angle in CHF/CNR measurement-programe-inputing function was specified by Nornal format

Bit0 1: In linear chamfering, L means the length between chamfering point and cross point;

0: In linear chamfering, L means the length of chamfering;

1	8	3	***	***	***	MZR5	MZR4	MZRY	MZRZ	MARX

Bit4 1: for 5th, press key to execute the machine zero return

0:for 5th, press key to execute the machine zero return

Bit3 1: for 4th, press to execute the machine zero return

0: for 4th, press to execute the machine zero return

Bit2 1: for Y, press to execute the machine zero return

0: for Y, press to execute the machine zero return

Bit1 1:for Z, press to execute the machine zero return

0: for Z, press to execute the machine zero return

Bit0 1: for X, press to execute the machine zero return

0: for X, press to execute the machine zero return

1	8	4	***	***	***	***	***	L2	L1	L0

Bit0, Bit1, Bit2: Language selection:

Language	Bit2	Bit1	Bit0
Chinese	**0**	**0**	**0**
English	**0**	**0**	**1**
Portuguese	**0**	**1**	**0**
Spanish	**0**	**1**	**1**
Russian	1	1	0

1	8	5	SK0	SKF	AEO	***	***	***	PRPD	PLA

Bit7 1: Skip input active as G31 signal is 0

0: Skip input active as G31 signal is 1

Bit6 1: Feedrate override per minute or dry run active for G31

0: Feedrate override per minute or dry run inactive for G31

Bit5 1: Input active if G36, G37 (XAE, ZAE) signal is 0

0: Input active if G36, G37 (XAE, ZAE) signal is 1

Bit1 1: Axis rapid traverse rate of PLC by input value

0: Axis rapid traverse rate of PLC by parameter value (X axis: No.022；Z axis:No.023；Y axis:No.134)

Bit0 1: PLC axis control active, active after repowering

0: PLC axis control inactive, active after repowering

1	8	6	RTORI	SRS	***	***	***	RTCRG	***	***

Bit7 1: In executing M29, the spindle executes the machine zero return

0: In executing M29, the spindle does not execute the machine zero return

Bit6 1: The spindle selection signal is RGTSPn in the multiple spindle rigid tapping

0: The spindle selection signal is SWSn in the multiple spindle rigid tapping

Bit2 1: In rigid tapping cancel, do not wait for G61.0 to be 0 in executing the next block
0: In rigid tapping cancel, wait for G61.0 to be 0 in executing the next block

1	8	7

IS1	IS0		***	***	***	ROS	ROT

[Data type]:Bit axis

BIT7, BIT6:set increment system of each axis.00:accordance with increment system of the system.
01:IS-A，10:IS-B；11:IS-C。

Bit1, Bit0 : 00: set the additional axis to the linear axis; 01: set the additional axis to the rotary axis（A type）;11: set the additional axis to the rotary axis (B type); 10: set the additional axis to be invalid.

Note: The parameter is invalid to setting of X, Z axis.

1	8	8

RFCK	RRT	***	***	***	RRL	RAB	ROA

[Data type]:bit axis

Bit7 1:G28 running every time to check the PC signal
0:G28 running only first to check the PC signal

Bit6 1:Zero return D mode is used when the additional axis is a rotary axis;
0:Zero return A, B, C mode is used when the additional axis is a rotary axis (power-on again).

Bit2 1:The relative coordinate cycle function is valid when the additional axis is rotary;
0:The relative coordinate cycle function is invalid when the additional axis is rotary(power-on again).

Bit1 1:The rotation direction is based on that of sign when the additional axis is rotary;
0:The rotation direction is based on the nearby principle when the additional axis is rotary.

Bit0 1:The absolute coordinate cycle function is valid when the additional axis is rotary;
0:The absolute coordinate cycle function is invalid when the additional axis is rotary (power-on again).

Note: The parameter is invalid to setting of X, Z axis.

1	8	9

CTS	***	APC	APZ	***	***	***	***

[Data type]:Bit axis

Bit7 1:Each servo axis didn't connect servo drive , it is necessary to restart;
0:Each servo axis connected servo drive , it is necessary to restart.

Bit5 1:The used encoder type of each axis is absolute encoder；
0:The used encoder type of each axis is incremental encoder。

Bit4 1:When each axis matches with an absolute encoder, the mechanical position and absolute one are shown consistent;
0:When each axis matches with an absolute encoder, the mechanical position and absolute one are shown inconsistent.

1	9	0

***	***	***	***	LT3	LT2	***	***

[Data type]:Bit axis

Bit3 1:Whether each axis detects the stroke 3 in memory type detect;
0:Whether each axis detects the stroke 3 in memory type is not detect.

Bit2 1:Whether each axis detects the stroke 2 in memory type detect;
0:Whether each axis detects the stroke 2 in memory type detect.

1	9	5		SSPD	PLCS	***	***	RPOS	SRV	CSREF	RCS

Bit7 1:series spindle controllable function enabled (series spindle The slave ID number of the spindle 1~3 is 11~13) , it is necessary to restart;
0: Series spindle controllable function disabled (analogy spindle), it is necessary to restart.

Bit6 1:PLC spindle rotation speed function of each axis enabled;
0:PLC spindle rotation speed function of each axis disabled。

Bit3 1:Spindle position function is valid (When it is enabled, the spindle will perform mechanical releasing/clamping interlocking operation before/after positioning);
0:Spindle position function is invalid.

Bit2 1:Servo motor spindle function is valid ,power on;
0:Servo motor spindle function is invalid ,power on。

Bit1 1: Change to position mode,the axis do return REF.
0: Change to position mode, the axis doesn't return REF.

Bit0 1: Cs function of spindle valid ,power on
0: Cs function of spindle is: invalid, power on

1	9	6		***	***	***	MSEN	***	***	***	***

Bit4 1: The multiple spindle control is enabled
0: The multiple spindle control is disabled

2	0	1		SNTP	***	***	ECN	HBR	ERV	***	***

Bit7 1:Synchronization mode follow
0:Synchronization mode interpolation

Bi4 1:Whether the G81 synchronization command can be specified in the synchronization process allow
0:Whether the G81 synchronization command can be specified in the synchronization process prohibition

Bit3 1:The system resets in the synchronization process reserve the synchronized control
0:The system resets in the synchronization process cancel the synchronization control

Bit2 1:In the synchronization procedure,in the G99 feed per revolution use the theoretical revolving speed of workpiece\
0:In the synchronization procedure,in the G99 feed per revolution use the encoder feedback revolving speed

2	0	2		CTHDN OENC	AAC	***	***	AOT	***	***	EXTRA IO

Bit7 1:Circular screw interpolation adopts C axis interpolation mode.
0:Circular screw interpolation adopts follow spindle mode..

Bit6 1:Slope axis control enabled
0:Slope axis control disabled

Bit3 1:The soft-limit parameter of the slope axis sets to 0/1 when its axis control is enabled the coordinate value under the rectangular coordinate system\
0:The soft-limit parameter of the slope axis sets to 0/1 when its axis control is enabled the coordinate value under the slope coordinate system.

Bit0 1:Extended I/O unit linked is valid
0:Extended I/O unit linked is invalid\

2	5	0

***	***	***	***	***	***	VERI	GSKLINK

Bit1 1:System GSKLINK bus communication CRC verification enabled
0:System GSKLINK bus communication CRC verification disabled
Bit0 1: Whether the system GSKLINK bus connection is enabled
0: Whether the system GSKLINK bus connection is disabled

3.1.2 Data Parameter

0	1	5	CMRn

［Data type］ Word axis
［Data meaning］ Pulse frequency multiplication coefficient
［Data range］ 1～99999999

0	1	6	CMDn

［Data type］ Word axis
［Data meaning］ Pulse frequency division coefficient;
［Data range］ 1～99999999

Explanation: Formula of electric gear ratio: $\frac{CMR}{CMD} = \frac{4*C*\delta}{L} * \frac{Z_M}{Z_D}$

δ: Least command increment;
Z_M: Teeth number of pulley on leading screw side;
C: Motor encoder line number;
Z_D Teeth number of pulley on motor side;
L: Screw lead

0	1	9	THDCH

［Data meaning］ Run-out length in threading
［Data range］ 0～225
Thread run-out width= THDCH×0.1×screw lead

0	2	1	PSANGNT

［Data type］ Word spindle
［Data meaning］ Voltage offset value when spindle max. speed analog voltage output is 10V
［Data unit］ mV
［Data range］ -2000～2000(Unit: mV)

0	2	2	RPDFn

［Data type］ Word axis
［Data meaning］ Maximum rapid traverse rate of all axes
［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］ 10～99999999

0	2	3	LINTn

［Data type］ Word axis

［Data meaning］ Acceleration/deceleration time constant of each axis during rapid traverse

［Data unit］ ms

［Data range］ 0~4000

0	2	6	THRDT

［Data meaning］ ACC&DEC time constant of short axis in thread run-out

［Data unit］ ms

［Data range］ 0～4000

0	2	7	FEDMX

Data meaning］ Upper limits of axis cutting feed

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］

	IS-B	IS-C
Metric machine	10～30000	10～30000
Inch machine	5～11600	10～11600

0	2	8	THDFL

［Data meaning］ X, Z initial speed in threading (mm/min)

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］

	IS-B	IS-C
Metric machine	6～8000	6～8000
Inch machine	3～3000	3～3000

0	2	9	FEEDT

［Data meaning］ Exponential ACC&DEC time constant of cutting and manual feed

［Data unit］ ms

［Data range］ 0～4000

0	3	0	FEDFL

［Data meaning］ Exponential acceleration start speed and deceleration end speed in cutting feed

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］

	IS-B	IS-C
Metric machine	0～8000	0～8000
Inch machine	0～3000	0～3000

0	3	1	JOGSPEED

［Data meaning］ the set speed when the manual feedrate override is 100%

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］ 10～30000

0	3	2	RPDFLn

［Data type］ Word axis

［Data meaning］ Rapid traverse rate as axis rapid override is F0

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］

	IS-B	IS-C
Metric machine	0～4000	0～4000
Inch machine	0～1500	0～1500

0	3	3	ZRNFLn

［Data type］ Word axis

［Data meaning］ Low rate as axes return to machine zero

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］

	IS-B	IS-C
Metric machine	6～4000	6～4000
Inch machine	3～1500	3～1500

0	3	4	BKLn

［Data type］ Word axis

［Data meaning］ backlash compensation of each axis

［Data unit］

Setting unit	IS-B	IS-C
Metric machine	0.001 mm	0.0001 mm
Inch machine	0.0001 inch	0.00001 inch

［Data range］

Setting unit	IS-B	IS-C
Metric machine	0～2000	0～20000
Inch machine	0～2000	0～20000

Note : X is the diameter value.

0	3	6	SPDLC

［Data type］ Word spindle

［Data meaning］ Voltage offset for 0V analog voltage output of spindle min. speed code

［Data unit］ mV

［Data range］ -1000～1000

0	3	7	GRMAX1
0	3	8	GRMAX2
0	3	9	GRMAX3
0	4	0	GRMAX4

［Data meaning］ GRMAX1, GRMAX2, GRMAX3, GRMAX4: Max. speeds of gear 1, 2, 3, 4 respectively when spindle analog voltage output is 10V (the function defined by GSK980TDi PLC is: if the spindle automatic gearing is active, it uses the speeds of M41, M42, M43, M44; if the automatic gearing is inactive or CNC is powered on, the gear is memorized by parameter or the gear 1 is defaulted.)

［Data unit］ r/min

［Data range］ 10～9999

0	4	1	JOGFL

［Data meaning］ Exponential ac-deceleration start speed and deceleration final speed in manual feed

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］ 0～8000

0	4	2	SEQINC

［Data meaning］ Block No. increment for auto block No. insertion

［Data range］ 1～100

0	4	3	LOWSP

［Data meaning］ (G96) Spindle min. speed under the constant surface speed control

［Data unit］ r/min

［Data range］ 0～9999

0	4	4	BRATE0

［Data meaning］ Serial communication baud rate,

［Data unit］ bit/s

［Data range］ 1200,2400,4800,9600,19200,38400 57600 115200

0	4	5	LT1X1
0	4	6	LT1Z1

［Data type］ Word axis

［Data meaning］ LT1X1（X）,LT1Z1（Z）positive max. travel，LT1X2（X）,LT1Z2（Z）negative max. travel

［Data unit］

Setting unit	IS-B	IS-C
Metric machine	0.001 mm	0.0001 mm
Inch machine	0.0001 inch	0.00001 inch

［Data range］ -99999999～99999999

Note: If the BIT2 of the parameter No.001 is set for diameter, the X axis value is specified by diameter; if for radius, the X axis value is specified by radius.

0	4	7	PRSn

［Data type］ Word axis

［Data meaning］ Setting value of absolute coordinate after machine zero return

［Data unit］

Setting unit	IS-B	IS-C
Metric input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

［Data range］ -99999999～99999999

0	5	1	MRCCD

［Data meaning］ Each feeding for G71, G72 rough turning cycle, which may also be changed by program code in auto running.

［Data unit］

Setting unit	IS-B	IS-C
Metric input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

［Data range］

	IS-B	IS-C
Metric input	1～99999	1～999999
Inch input	1～99999	1～999999

0	5	2	MRCDT

［Data meaning］ Each retraction for G71, G72 rough turning cycle, which may also be changed by program code in auto running.

［Data unit］

Setting unit	IS-B	IS-C
Metric input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

［Data range］ 0～99999

0	5	3	PECSCX

［Data meaning］ X retraction amount in G73, which may be also changed by program code in auto running.

［Data unit］

Setting unit	IS-B	IS-C
Metric input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

［Data range］ -9999999～9999999

0	5	4	PECSCZ

［Data meaning］ Z axis rough turning retraction in G73, which may also be changed by program code in auto running

［Data unit］

Setting unit	IS-B	IS-C
Metric input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

［Data range］ -9999999～9999999

0	5	5	PATIM

［Data meaning］ Cycle times of G73, which may also be changed by program code in auto running

［Data unit］ times

［Data range］ 1～9999

0	5	6	GROVE

［Data meaning］ Retraction value of Z axis in G74; retraction value of X axis in G75; a specified value in program during automatic running can also change the setting of this parameter;

［Data unit］

Setting unit	IS-B	IS-C
Metric input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

［Data range］ 0～99999

0	5	7	THRPT

［Data meaning］ Repetitions of G76 finish machining, which may also be changed by program

code in auto running

［Data unit］ times

［Data range］ 1～99

0	5	8	THANG

［Data meaning］ Tool nose angle of G76 cycle, which may also be changed by program code in auto running

［Data unit］ deg

［Data range］ 0～99

0	5	9	THCLM

［Data meaning］ Min. cut depth of G76 cycle, which may also be changed by program code in auto running

［Data unit］

Setting unit	IS-B	IS-C
Metric input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

［Data range］

	IS-B	IS-C
Metric input	0～99999	0～999999
Inch input	0～99999	0～999999

0	6	0	THDFN

［Data meaning］ Finish allowance of G76, which may also be changed by program code in auto running

［Data unit］

Setting unit	IS-B	IS-C
Metric input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

［Data range］

	IS-B	IS-C
Metric input	0～99999	0～999999
Inch input	0～99999	0～999999

0	6	1	THDCUT

［Data type］ Word

［Data meaning］ The default cut-in selection in G78 cycle, when the program does not specify, the parameter value is used for the cut-in selection.

0: Equidistant in-feed; 1: Degressive in-feed.

［Data range］ 0~1

0	6	2	THDMOD

［Data type］ Word

0: Cutting edge is cut in along with the neutral cable of the thread teeth;
1: Cutting edge is cut in along the left of the thread teeth;
2: Cutting edge is cut in along the right of the thread teeth;
3: Cutting edge is cut in alternatively with left and right of the thread teeth.

［Data range］ 0~3

0	6	7	SFTREV

［Data type］ Word spindle
［Data meaning］ Output voltage of spindle gearing
［Data unit］ mV
［Data range］ 0～10000

0	6	8	SJOGREV

［Data type］ Word spindle
［Data meaning］ spindle speed in manual (MPG) mode
［Data unit］ r/min
［Data range］ 0～3000

0	6	9	ENCODER_CSE

［Data type］ Word spindle
［Data meaning］ The corresponding spindle encoder number of each spindle.
0: without installing the encoder;
1: Spindle revolving signal comes from the encoder interface 1;
2: Spindle revolving signal comes from the encoder interface 2;
3: Spindle revolving signal comes from the GSKLINK bus
［Data range］ 0～3

0	7	0	ENCODER_CNT

［Data type］ Word spindle
［Data meaning］ Spindle encoder pulses
［Data unit］ line/revolution
［Data range］ 100～10000

0	7	1	RESET_TIME

［Data meaning］ Output time of reset signal
［Data unit］ ms
［Data range］ 16～4080

0	7	2	SAR_DELEY

［Data meaning］ Delay detecting time of SAR signal
［Data unit］ ms

［Data range］ 0～4080

0	7	3	SPMOTORMAX

［Data type］ Word spindle

［Data meaning］ Max. spindle motor clamping speed, setting value=(max. spindle motor clamping speed/max. spindle motor speed) ×4095

［Data unit］ r/min

［Data range］ 0～4095

Setting value =（max. clamp speed of spindle/max. speed of spindle motor）×4095.

0	7	4	SPMOTORMIN

［Data meaning］ Min. spindle motor clamping speed, setting value=(min. spindle motor clamping speed/max. spindle motor speed) ×4095

［Data unit］ r/min

［Data range］ 0～4095

Setting value =（min. clamping speed of spindle motor/max. speed of spindle motor）×4095

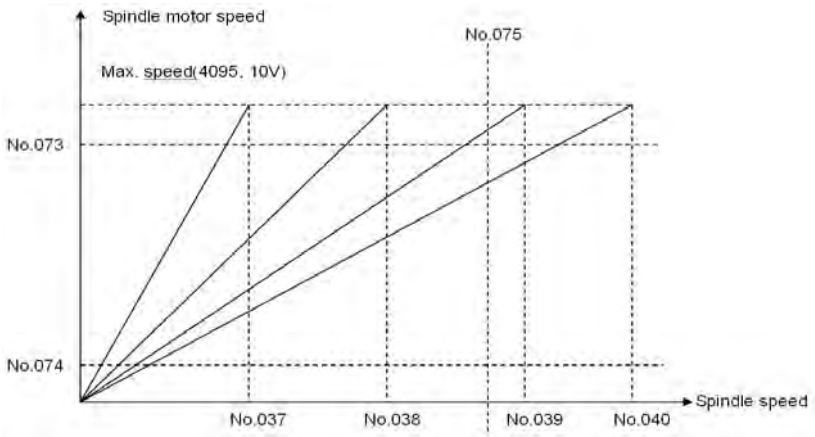
0	7	5	SPSPEEDLIMIT

［Data type］ Word spindle

［Data meaning］ Maximum spindle rev speed(0~6000r/min)[0:unlimited]

［Data unit］ r/min

［Data range］ 0～6000



0	7	6	SPDLIMITEXT

［Data type］ Word spindle

［Data meaning］ The spindle speed limit when speed limit signal is active [Without restriction when it is set to 0]

［Data unit］ r/min

［Data range］ 0～9999

0	7	7	FEEDLIMITEXT

［Data meaning］ The feedrate limit when speed limit signal is active [Without restriction when it is set to 0].

［Data unit］

Setting unit	Data unit
Machine in metric system	mm/min
Machine in inch system	0.1inch/min

［Data range］ 0～15000

	IS-B	IS-C
Machine in metric system	0～15000	0～15000
Machine in inch system	0～5800	0～5800

0	8	4	TMAX

［Data meaning］ Total tools number selection

［Data unit］ piece

［Data range］ 1～32

0	8	6	MACALL_GVAL1
0	8	7	MACALL_GVAL2
0	8	8	MACALL_GVAL3
0	8	9	MACALL_GVAL4
0	9	0	MACALL_GVAL5
0	9	1	MACALL_GVAL6
0	9	2	MACALL_GVAL7
0	9	3	MACALL_GVAL8
0	9	4	MACALL_GVAL9
0	9	5	MACALL_GVAL10

［Data meaning］ User macro program G code of calling program No. 9010～9019

［Data range］ 0～9999

0	9	6	PECMINn

［Data type］ Word axis

［Data meaning］ Minimum pitch error compensation position number

［Data range］ 0～1199

0	9	7	PECMAXn

［Data type］ Word axis

［Data meaning］ Maximum pitch error compensation position number

［Data range］ 0～1199

0	9	8	PECORGn

［Data type］ Word axis

［Data meaning］ Pitch error compensation position numbers corresponding to machine zero points

［Data range］ 0～1199

0	9	9	PECINTn

［Data type］ Word axis

［Data meaning］ Pitch error compensation intervals

［Data unit］

Setting unit	IS-B	IS-C
Metric machine	0.001 mm	0.0001 mm
Inch machine	0.0001 inch	0.00001 inch

［Data range］

	IS-B	IS-C
Metric machine	1000～999999	10000～9999999
Inch machine	400～999999	4000～9999999

1	0	0	LTMAX_2
1	0	1	LTMIN_2

［Data type］ Word axis

［Data meaning］ Stored stroke limit 2 of each axis, LTMAX_2 positive maximum stroke position, LTMIN_2 negative maximum stroke position.

［Data unit］

Setting unit	IS-B	IS-C
Metric machine	0.001 mm	0.0001 mm
Inch machine	0.0001 inch	0.00001 inch

［Data range］ -99999999～99999999

1	0	2	LTMAX_3
1	0	3	LTMIN_3

［Data type］ Word axis

［Data meaning］ Stored stroke limit 3 of each axis, LTMAX_3 positive maximum stroke position, LTMIN_3 negative maximum stroke position.

［Data unit］

Setting unit	IS-B	IS-C
Metric machine	0.001 mm	0.0001 mm
Inch machine	0.0001 inch	0.00001 inch

［Data range］ -99999999～99999999

1	0	6	THD_SPD_VAR

［Data meaning］ Spindle fluctuation alarm limit in threading (not detect spindle fluctuation alarm if set to 0)

［Data unit］ %

［Data range］ 0～100

Formula=|current speed － previous period speed|×200/(current speed＋previous period speed)

1	0	7	THD_TAIL_SPD

［Data meaning］ Short axis speed in threading run-out (run-out by threading feedrate if set to 0)

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］ 0～8000

1	0	9	SPL_REV_SPD

［Data meaning］ Spindle jog speed (r/min)

［Data unit］ r/min

［Data range］ 1～8000

1	1	0	MGR

［Data type］ Word spindle

［Data meaning］ Gear ratio of encoder to spindle: spindle gear teeth number

［Data range］ 1～255

1	1	1	SGR

［Data type］ Word axis

［Data meaning］ Gear ratio of encoder to spindle: encoder gear teeth number

［Data range］ 1～255

1	1	3	REF_SPDn

［Data type］ Word axis

［Data meaning］ Axes rapid traverse rates in machine zero return

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］ 10～99999999

1	1	4	REF_OFFSETn

［Data type］ Word axis

［Data meaning］ Offset of each axis machine zero

［Data unit］

Setting unit	IS-B	IS-C
Metric input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

［Data range］

	IS-B	IS-C
Metric input	-99999～99999	-999999～999999
Inch input	-99999～99999	-999999～999999

1	1	9	KEY_P_NUM

［Data meaning］ Number of active keys at a time

［Data unit］ piece

［Data range］ 2～5

1	2	0	REF1_COORDn
1	2	1	REF1_COORDn
1	2	2	REF2_COORDn
1	2	3	REF2_COORDn

［Data type］ Word axis

［Data meaning］ REF1_COORDn 1st reference point machine coordinates；
REF2_COORDn 2nd reference point machine coordinates；
REF3_COORDn 3rd reference point machine coordinates；
REF4_COORDn 4th reference point machine coordinates

［Data unit］

Setting unit	IS-B	IS-C
Metric input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

［Data range］ -99999999～99999999

1	3	6	FPGA_JMP_XPL

［Data type］ Word

［Data meaning］ Tool offset testing input in method B, X plus direction test signal add (0:invalid;20~27,30~37:add X2.0~X3.7)

［Data range］ 0, 20~27, 30~37

1	3	7	FPGA_JMP_XML

［Data type］ Word

［Data meaning］ Tool offset testing input in method B, X minus direction test signal add (0:invalid;20~27,30~37:add X2.0~X3.7)

［Data range］ 0, 20~27, 30~37

1	3	8	FPGA_JMP_ZPL

［Data type］ Word

［Data meaning］ Tool offset testing input in method B, Z plus direction test signal add (0:invalid;20~27,30~37:add X2.0~X3.7)

［Data range］ 0, 20~27, 30~37

1	3	9	FPGA_JMP_ZML

［Data type］ Word

［Data meaning］ Tool offset testing input in method B, Z minus direction test signal add (0:invalid;20~27,30~37:add X2.0~X3.7)

［Data range］ 0, 20~27, 30~37

1	4	0	TWEAR_MAX

［Data meaning］ + or – limit of each input wear in TOOL OFFSET&WEAR interface

［Data unit］

Setting unit	IS-B	IS-C
Metric input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

［Data range］

	IS-B	IS-C
Metric input	1～99999	1～999999
Inch input	1～99999	1～999999

1	4	1	AUTO_OFFSET_FEED

［Data meaning］ Feedrate in automatic tool offset measure

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］

	IS-B	IS-C
Metric machine	15～1000	15～1000
Inch machine	7～375	7～375

1	4	2	AUTO_OFFSET_Y_X
1	4	3	AUTO_OFFSET_Y_Z

［Data meaning］ X(AUTO_OFFSET_Y_X) ,Z(AUTO_OFFSET_Y_Z)γ values in automatic tool offset

［Data unit］

Setting unit	IS-B	IS-C
Metric machine	0.001 mm	0.0001 mm
Inch machine	0.0001 inch	0.00001 inch

［Data range］ 1～9999999

Note: X setting value is the radius in diameter or radius programming

1	4	4	AUTO_OFFSET_E_X
1	4	5	AUTO_OFFSET_E_Z

［Data meaning］ AUTO_OFFSET_E_X（X）,AUTO_OFFSET_E_Z（Z）ε values in automatic tool offset

［Data unit］

Setting unit	IS-B	IS-C
Metric machine	0.001 mm	0.0001 mm
Inch machine	0.0001 inch	0.00001 inch

［Data range］ 1～9999999

Note: X setting value is the radius in diameter or radius programming

1	4	8	MULTISTAGE_SKIP1
1	4	9	MULTISTAGE_SKIP2
1	5	0	MULTISTAGE_SKIP3
1	5	1	MULTISTAGE_SKIP4

［Data type］ Word

［Data meaning］ Multilevel skip G31 P1～P4 or G04 Q1～Q4 signal detect address setting(0:no effect; 20~27,30~37: address X2.0~X3.7)

［Data range］ 0, 20~27, 30~37

1	5	4	RADIUS_ERR_RANGE

［Data meaning］ Max. arc radius error

［Data unit］

Setting unit	IS-B	IS-C
Metric machine	0.001 mm	0.0001 mm
Inch machine	0.0001 inch	0.00001 inch

［Data range］

	IS-B	IS-C
Metric machine	0～1000	0～10000
Inch machine	0～50	0～500

1	6	2	CSACCFL

［Data type］ Word spindle

［Data meaning］ initial speed of CS acceleration/deceleration

［Data unit］ deg/min

［Data range］ 0～4000

1	6	3	CSACCTIME

［Data type］ Word spindle

［Data meaning］ acceleration/deceleration time constant of CS

［Data unit］ ms

［Data range］ 0～4000

1	6	4	CSMCODE

［Data type］ Word spindle

［Data meaning］ Spindle CS shifts the M code [Setting value +1 is the M code of the function releasing].

Note 1: When the spindle positioning function is enabled, the M code is regarded as the spindle positioning method shifint.

Note 2: The setting value should be set by the M code shifted based upon the CS compiled in the ladder diagram.

Note 3: The setting value is disabled when it is less than 6, the coordinate system will not be cleared when the CS axis is shifted in the disabled state.

［Data range］ 0～96

1	6	6	CAACCFL

［Data type］ Word spindle

［Data meaning］ initial speed of linear acceleration/deceleration in rigid tapping

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］ 0～4000

1	6	7	CAACCTIME

［Data meaning］ linear acceleration/deceleration time constant in rigid tapping tool in-feed

［Data unit］ ms

［Data range］ 0～4000

1	6	9	RTAPBACKRATE

［Data meaning］ override value in rigid tapping tool retraction (override is 100% when it is set to 0)

［Data unit］ %

［Data range］ 0～200

1	7	0	CMCODE

［Data meaning］ M code used for spindle clamping during rigid tapping

［Data unit］

［Data range］ 3～97

1	7	3	CALCODE

［Data meaning］ Take count of machining piece M code (0:defined M99 & M30)

［Data unit］

［Data range］ 0～97

1	9	1	POSERR_SERVO

［Data type］ Word axis

［Data meaning］ Servo axes position error value when running speed 1m/min

［Data unit］

［Data range］ 0～99999999

1	9	7	EGB_MASTERAX

［Data meaning］ In the synchronized control, the corresponding system axis number of the master axis

［Data unit］

［Data range］ 0～5

Setting value	meaning
0	disabled
1~5	the 1st feed axis ~ the 5th axis

1 9 8 **EGB_SLAVEAX**

［Data meaning］ In the sychronized control, the corresponding axis number of the slave axis

［Data unit］

［Data range］ 0～5

Setting value	meaning
0	disabled
1~5	the 1st feed axis ~ the 5th axis

2 1 9 **PLCDIDOn**

［Data type］ Word axis

［Data meaning］ Select DI/DO according to PLC controlled axes (0 means PLC axis control is not applied)

［Data unit］

［Data range］ 0～4

2 2 0 **SP_SERVOn**

［Data type］ Word spindle

［Data meaning］ The corresponding interface number of each spindle output

［Data unit］

［Data range］ 0～14

Setting value	meaning
2	analog spindle interface 2
1	analog spindle interface 1
0	Disabled analog spindle output
11～14	extension IO unit analog output interface; 1~4

2 2 1 **SP_SP_CSAXIS**

［Data type］ Word spindle

［Data meaning］ The functions such as the feed axis number and CS axis, etc. related with each spindle should be set.

［Data unit］

［Data range］ 0～5

Setting value	meaning	Setting value	meaning	Setting value	meaning
0	Nothing	2	Z axis	4	4th axis
1	X axis	3	3rd axis	5	5th axis

2 2 5 **NAME_n**

［Data type］ Word axis

［Data meaning］ Axis name definition of each axis

［Data range］ 65, 66, 67, 88, 89, 90

Axis name	Setting value	Axis name	Setting value	Axis name	Setting value
X	88	Y	89	B	66
Z	90	A	65	C	67

Note: The axis name of X or Z axis is fixed, which can not be modified.

2 3 0 **ATTRIB_n**

［Data meaning］ ATTRIB_n is the additional attribution setting

［Data range］ 0~7

Setting value	Meaning
0	It is neither the three basic axes and an axis parallel to them
1	X axis of the three basic axes
2	Y axis of the three basic axes
3	Z axis of the three basic axes
5	An axis parallel to X
6	An axis parallel to Y
7	An axis parallel to Z

2 3 5 **POLARLINEAXIS**

［Data meaning］ Linear axis setting of the polar coordinate interpolation

［Data range］ 0~4

Setting value	Meaning
0	X axis
1	Z axis
2	Y axis
3	4th axis
4	5th axis

2 3 6 **POLARTOSAXIS**

［Data meaning］ Rotary axis setting of the polar coordinate interpolation

［Data range］ 2~4

Setting value	Meaning
2	Y axis
3	4th axis
4	5th axis

2 3 7 **ACCEPTANGLEERR**

［Data meaning］ Maximum fluctuating value when the angle is 0°or 180°for C tool offset C

［Data range］

IS-B	IS-C
0～1000	0～10000

2	4	0	HWRPDFRATE

［Data meaning］ Rapid traverse override of MPG trial cut function

［Data range］ 0～100

2	4	1	HWPULSEINC

［Data meaning］ Movement amount per pulse during trial cut function

［Data range］ 0～100

2	4	3	CNC_MPG_NUM

［Data meaning］ Quantity of MPG

［Data range］ 1～2

2	4	4	MPG1_INTERFACE
2	4	5	MPG2_INTERFACE

［Data meaning］ MPG interface of MPG 1,2

［Data range］ 0～1

Setting value	Meaning
0	Rear cover MPG interface 1 (CN31)
1	Rear cover MPG interface 2(CN32)

2	4	9	ABSPOSERR

［Data type］ Word axis

［Data meaning］When an absolute encoder is equipped with each axis, the Max. position error value allows when the coordinate system is set up with power-on.

［Data unit］

Setting unit	IS-B	IS-C
Metric input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

［Data range］ 0～999999

3.1.3 PLC Parameters（Standard PLC Definition）

3.1.3.1 K Parameters

K	1	0	LMIT	LMIS	EXESP	JSPD	OVRI	***	RSJG	RRW

Bit7 1: overtravel of each axis is enabled;
0: overtravel of each axis is disabled.

Bit6 1: alarm for overtravel signal LOW of each axis;
0: alarm for overtravel signal LOW of each axis

Bit5 1: External ESP signal is enabled;
0: External ESP signal is disabled.

Bit4 1: spindle JOG is enabled in any modes;
0: spindle JOG is enabled in Manual, MPG, Zero Return mode.

Bit3 1: feedrate override is fixed to 100%;

0: feedrate override can be adjusted.

Bit1 1: spindle lubricating, cooling output remain in reset;

0: spindle lubricating, cooling output are disabled in reset.

Bit0 1: the cursor returning to the beginning of the program is enabled in any working mode in reset;

0: the cursor returning to the beginning of the program is enabled in Edit working mode.

K	1	1		CHOT		CHET	TCPS	CTCP	TSGN	CHTB	CHTA

Bit7 1: checks the tool post worktable overheating;

0: does not check the tool post worktable overheating.

Bit5 1: checks the tool signal when the tool change is completed;

0: does not check the tool signal when the tool change is completed.

Bit4 1: tool post locking signal HIGH (be connected with +24V) is enabled;

0: tool post locking signal LOW (not be connected with +24V) is enabled

Bit3 1: checks the tool post locking signal;

0: does not check the tool post locking signal.

Bit2 1: tool signal LOW(not be connected with +24V) is enabled;

0: tool signal HIGH(be connected with +24V)is enabled.

Bit1 Bit0 : tool change mode selects Bit1 tool change mode selects Bit 0

CHTB	CHTA	Tool post type
0	0	Standard tool change mode B
0	1	Standard tool change mode A
1	0	Yantai AK31 series(6, 8, 10, 12 tools)
1	1	LIO_SHING hydraulic tool post（8, 10, 12 tools）

K	1	2				RCKC	CCHU	NYQP	SLSP	SLQP

Bit4 1: External chuck control is enabled during program running;

0: External chuck control is disabled during program running

Bit3 1: the system checks the chuck in-position signal;

0: the system does not check the chuck in-position signal.

Bit2 1: in outer mode, NQPJ is the outer chuck released signal, and WQPJ is the outer chuck clamped signal;

0: in inner mode, NQPJ is the inner chuck clamped signal, and WQPJ is the inner chuck released signal.

Bit1 1: the system does not check whether the chuck is clamped when the chuck function is enabled;

0: the system checks whether the chuck is clamped when the chuck function is enabled; when the chuck is not clamped, the system cannot start the spindle and alarms.

Bit0 1: the chuck control function is enabled;

0: the chuck control function is disabled.

K	1	3		CHIPR	***	WKLP	***	***	RTSC	SPTW	SLTW

Bit7 1: The chip-removal controller is enabled;

0: The chip-removal controller is disabled

Bit5 1: Indicator control is enabled;

0: Indicator control is disabled.

Bit2 1: External tailstock control is enabled during program running;

0: External tailstock control is disabled during program running;

Bit1 1: the spindle rotation and the tailstock forward/backward does not interlock, the tailstock executes the forward no matter what the spindle is in any states; the spindle rotates no matter what the tailstock is in any states;

0: the spindle rotation and the tailstock forward/backward interlock, the tailstock cannot execute the forward when the spindle rotates; the spindle cannot be started when the tailstock does not execute the forward.

Bit0 1: the tailstock control function is enabled;

0: the tailstock control function is disabled.

K	1	4	HPST			SGSP	SPB4	PB4	SPB3	PB3

Bit7 1: the hydraulic control function is enabled;

0: the hydraulic control function is disabled.

Bit4 1: Spindle is enabled when the protection door is open

0: Spindle is disabled when the protection door is open;

Bit3 1: when SAGT is connected with +24V, the safety door is disabled;

0: when SAGT is not connected with +24V, the safety door is disabled.

Bit2 1: the safety door check function is enabled;

0: the safety door check function is disabled.

Bit1 1: when PRES is not connected with +24V，the system alarms for pressure LOW;

0: when PRES is connected with +24V，the system alarms for pressure LOW.

Bit0 1: the pressure LOW check function is enabled;

0: the pressure LOW check function is disabled.

K	1	5	RCS	RSCS	SPOR		SHT	AGIM	AGIN	AGER

Bit7 1: Cs function is enabled;

0: Cs function is disabled.

Bit6 1: the spindle contour control is disabled in emergency stop, reset;

0: the spindle contour control is enabled in emergency stop, reset.

Bit5 1: the spindle eight-point orientation function is enabled;

0: the spindle eight-point orientation function is disabled.

Bit3 1: the spindle gear power-down memory;

0: the spindle gear power-down does not execute the memory.

Bit2 1: the gear in-position signal M41I, M42I which are not connected with +24V is enabled;

0: the gear in-position signal M41I, M42I which are connected with +24V is enabled.

Bit1 1: when the spindle automatic gear change is to Gear 1, 2, the system checks the gear change in-position signal M41I, M42I;

0: when the spindle automatic gear change is to Gear 1, 2, the system does not check the gear change in-position signal M41I, M42I.

Bit0 1: the spindle automatic gear change function is enabled;

0: the spindle automatic gear change function is disabled

K	1	6	SINC	***	SCWM	LUBS	LUBCK	M32A	ROVI	SOVI

Bit7 1: Gear “×1000” in STEP (MPG) mode is disabled;

0: Gear “×1000” in STEP (MPG) mode is enabled;

Bit5 1: When spindle stop command is in the same block with movement command, these commands are executed at the same time;

0: When spindle stop command is in the same block with movement command, movement command comes first to spindle stop command;

Bit4 1: When cooling OFF command and movement command are in the same block, they are executed at the same time;

0: When cooling OFF command and movement command are in the same block, movement command comes first to the cooling OFF command;

Bit2 1: the lubricating outputs in power-on when the automatic lubricating is enabled;

0: the lubricating does not output in power-on when the automatic lubricating is enabled.

Bit1 1: the external feedrate override inverses;

0: the external feedrate override does not inverse.

Bit0 1: the external feedrate override switch is enabled

0: the external feedrate override switch is disabled.

K	1	7	MSEN	SCLP	SALM2	SALM	SSTP	***	***	SPDIR

Bit7 1: the multiple spindle control function is enabled;

0: the multiple spindle control function is disabled.

Bit6 1: the spindle clamped command clamps the 2nd spindle when the multiple spindle is enabled;

0: the spindle clamped command clamps the 1st spindle when the multiple spindle is enabled

Bit5 1: the system alarms when the 2nd spindle is LOW;

0: the system alarms when the 2nd spindle is HIGH.

Bit4 1: the system alarms when the spindle is LOW;

0: the system alarms when the spindle is HIGH.

Bit3 1: the analog voltage is disabled when the spindle stops;

0: the analog voltage is enabled when the spindle stops.

Bit0 1: The spindle button rotation direction on the panel is reversed;

0: The spindle button rotation direction on the panel is positive.

K	1	8	EXTM	RSTM	HOLD	***	MDOK	MD4	MD2	MD1

Bit7 1: extended M function is valid;

0: extended M function is invalid

Bit6 1: extended M function remains output when emergency stop, reset;

0: extended M function output closes when emergency stop, reset.

Bit5 1: HOLD release control is valid;

0: HOLD release control is invalid.

Bit3 1: the operation mode is specified by MD4, MD2, MD1 state when the system is turned on;

0: the operation mode is the state which is that of power-off when the system is turned on.

MD4	MD2	MD1	Operation mode in power-on
0	0	0	MDI mode
0	0	1	Auto mode
0	1	0	Program zero return
0	1	1	Edit mode
1	0	0	MPG mode
1	0	1	Manual mode
1	1	0	Machine zero return

K	1	9

***	MACH	HDFD	***	***	***	***	***

Bit6 1: The machine tool panel is the vertical one;
0: The machine tool panel is the horizontal one
Bit5 1: MPG interruption is enabled in operation;
0: MPG interruption is disabled in operation.

K	2	0

***	***	***	***	***	***	DBMPG 1	DBMPG 0

Bit 1 Dual-MPG function selection bit 1: 10: It is enabled and only the MPG 1 on the panel button is enabled, as well only the MPG 2 is enabled of the external MPG input.
Bit 0 Dual-MPG function selection bit 0: 00 Disabled 01: It is enabled and the MPG 1 fixes the X axis, and the MPG2 fixes the Z axis, and the step-length can be selectecd;

3.1.3.2 DT Parameter

Address	Meaning
DT000	Time for original gear disabling during spindle automatic gear changing (ms)
DT001	Delay time after the changed spindle gear is in position (ms)
DT002	Detection time for low pressure alarm (ms)
DT004	Permit time for tool changing (ms)
DT005	M command execution time (ms)
DT006	S command execution time (ms)
DT007	Delay time for tool post from CW to CCW output (ms)
DT008	Tool post lock signal detection time (ms)
DT009	Tool post CCW rotation lock time (ms)
DT010	Spindle braking delay output time (ms)
DT011	Spindle braking output time (ms)
DT012	Spindle JOG time (ms)
DT013	Time for manual lubricating ON (0: time is not limited) (ms)
DT014	Chuck clamping command execution time (ms)
DT015	Chuck unclamping command execution time (ms)
DT016	Interval of automatic lubricating (ms) (can be modified directly)
DT017	Time for automatic lubrication output (ms) (can be modified directly)
DT018	Chuck pulse output width (ms) (can be modified directly)
DT020	Spindle clamping/unclamping time (ms)
DT021	Spindle stop, chuck enable delay time (ms)
DT023	Delay time before spindle motor torque decreases to 0 after spindle starts clamping
DT024	Spindle CS shifting or positioning shifting allowable time (ms)

3.1.3.3 DC Parameters

Address	Meaning
DC000	Spindle zero speed output range (r/min)
DC001	Spindle revolving-speed arrival output range (r/min)

3.2 Parameter Description (by Function Sequence)

3.2.1 X, Z, Y, 4th,5th Axis Control Logic

0	0	1

***	***	***				ISC	

Bit1 1: IS-C incremental system;

0: IS-B incremental system.

ISC	Least input increment, least command increment	Abbreviation
0	0.001mm,0.0001inch	**IS-B**
1	0.0001mm,0.00001inch	**IS-C**

0	0	8

			DIR5	DIR4	DIRY	DIRZ	DIRX

Bit4~Bit0 1: Direction signal (DIR) is HIGH when 5th, 4th, Ym Z or X positively moves ;

0: Direction signal (DIR) is HIGH when 5th, 4th, Ym Z or X negatively moves.

If machine moving direction is inconsistent with the actually required direction, it may be adjusted by this parameter.

0	0	9

			5ALM	4ALM	YALM	ZALM	XALM

Bit4~Bit0 1: Alarm without detection when servo does not ready of the 5th, 4th, Y, Z or X axis;

0: Alarm detection when servo does not ready of the 5th, 4th, Y, Z or X axis.

1	7	5

SPFD							

SPFD =1: Feeding stops if spindle stops in cutting

=0: Feeding not stop if spindle stops in cutting

1	8	7

IS1	IS0					Axis type 1	Axis type 0

[Data type]: Bit axis

BIT7, BIT6:set increment system of each axis. 00:accordance with increment system of the system; 01:IS-A，10:IS-B；11:IS-C.

Bit1, Bit0:00: set the additional axis to the linear axis; 01: set the additional axis to the rotary axis（A type）;11: set the additional axis to the rotary axis (B type); 10: set the additional axis to be invalid.

Note: The parameter is invalid to setting of X, Z axis.

3.2.2 Acceleration & Deceleration Control

0	2	2

RPDFn

［Data type］ Word axis

［Data meaning］ max. speed of X rapid traverse (radius value)

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］ 10～99999999

0	2	3	LINTn

［Data type］ Word axis

［Data meaning］ Acceleration/deceleration time constant value

［Data unit］ ms

［Data range］ 10～4000

0	2	7	FEDM

［Data meaning］ Upper limit of cutting feedrate

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］

	IS-B	IS-C
Metric machine	10～30000	10～30000
Inch machine	5～11600	10～11600

0	2	9	FEEDT

［Data meaning］ Acceleration/deceleration time constant during cutting feed and manual feed

［Data unit］ ms

［Data range］ 0～4000

0	3	0	FEDFL

［Data meaning］ The starting speed of cutting feed

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］

	IS-B	IS-C
Metric machine	0～8000	0～8000
Inch machine	0～3000	0～3000

0	3	1	JOGSPEED

［Data meaning］ Set speed when manual feedrate override is 100%

［Data unit］

Setting unit	Data unit

Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］ 10～30000

0	3	2	RPDFLn

［Data type］ Word axis

［Data meaning］ The rapid traverse rate when the override is F0

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］

	IS-B	IS-C
Metric machine	0～4000	0～4000
Inch machine	0～1500	0～1500

0	4	1	JOGFL

［Data meaning］Exponential ac-deceleration start speed and deceleration final speed in manual feed

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］ 0～8000

0	7	7	FEEDLIMITEXT

［Data meaning］ The feedrate limit when speed limit signal is active [It is not restricted when setting to 0.]

［Data unit］

Setting unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］ 0～15000

	IS-B	IS-C
Metric machine	0～15000	0～15000
Inch machine	0～5800	0～5800

3.2.3 Precision Compensation

0	0	3			PEC					

Bit5 1: Pitch error offset active

0: Pitch error offset inactive

0	1	0	***	***	***	CPF5	CPF4	CPF3	CPF2	CPF1

CPF6~CPF1: Setting values of backlash offset pulse frequency (by BCD code)
The set frequency =(setting value+1) Kpps

CPF5	CPF4	CPF3	CPF2	CPF1	Set frequency(Kpps)
0	0	0	0	0	1
0	0	0	0	1	2
0	0	0	1	0	3
0	0	0	1	1	4
0	0	1	0	0	5
0	0	1	0	1	6
0	0	1	1	0	7
0	0	1	1	1	8
0	1	0	0	0	9
0	1	0	0	1	10
0	1	0	1	0	11
0	1	0	1	1	12
0	1	1	0	0	13
0	1	1	0	1	14
0	1	1	1	0	15
0	1	1	1	1	16
1	0	0	0	0	17
1	0	0	0	1	18
1	0	0	1	0	19
1	0	0	1	1	20
1	0	1	0	0	21
1	0	1	0	1	22
1	0	1	1	0	23
1	0	1	1	1	24
1	1	0	0	0	25
1	1	0	0	1	26
1	1	0	1	0	27
1	1	0	1	1	28
1	1	1	0	0	29
1	1	1	0	1	30
1	1	1	1	0	31
1	1	1	1	1	32

0	1	1

BDEC	BD8						

BDEC 1: Backlash offset type B, the offset data are output by ascending or descending type and the set frequency is inactive;

0: Backlash offset type A, the offset data are output by the set frequency (by bit parameter No.010) or 1/8 of it;

BD8 1: Backlash offset is done by the 1/8 of the set frequency；

0: Backlash offset is done by the set frequency；

0	3	4	BKLn

［Data type］ Word axis

［Data meaning］ BKLX（X axis）,BKLZ（Z axis）backlash compensation.

［Data unit］

Set unit	IS-B	IS-C
Metric machine	0.001 mm	0.0001 mm
Inch machine	0.0001 inch	0.00001 inch
Data range	2000	20000

［Data range］ 0～2000

Note: X is diameter value.

0	9	6	PECMINn

［Data type］ Word axis

［Data meaning］ Minimum pitch error compensation number

［Data range］ 0～1023

0	9	7	PECMAXn

［Data type］ Word axis

［Data meaning］ Maximum pitch error compensation number

［Data range］ 0～1023

0	9	8	PECORGn

［Data type］ Word axis

［Data meaning］ Pitch error compensation number corresponding to machine zero

［Data range］ 0～1023

0	9	9	PECINTn

［Data type］ Word axis

［Data meaning］ Pitch error compensation intervals

［Data unit］

Setting unit	IS-B	IS-C
Metric machine	0.001 mm	0.0001 mm
Inch machine	0.0001 inch	0.00001 inch

［Data range］

	IS-B	IS-C
Metric machine	1000～999999	10000～9999999
Inch machine	400～999999	4000～9999999

3.2.4 Machine Protection

1	7	2	***	MST	MSP	MOT	ESP	***	LZR	MOTDEC

Bit6 1: External cycle start signal (ST) inactive)
0: External cycle start signal (ST) active
Bit5 1: External SP signal inactive
0: External SP signal active with external dwell switch needed, otherwise CNC shows “Pause”.
Bit4 1: Not detect software stroke limit
0: Detect software stroke limit
Bit3 1: Emergency stop inactive
0: Emergency stop active
Bit1 1: Software limit is inactive after power-on and before machine zero return;
0: Software limit is active after power-on and before machine zero return;
Bit0 1: When Arriving stored travel position, CNC is stopped slowly
0: When Arriving stored travel position, CNC is stopped immediately

0	**4**	**5**	**LT1n1**
0	**4**	**6**	**LT1n2**

［Data type］ Word axis

［Data meaning］ LT1n1: Maximum stroke on positive direction; LT1n2: Maximum stroke on negative direction

［Data Unit］

Setting unit	IS-B	IS-C
Metric machine	0.001 mm	0.0001 mm
Inch machine	0.0001 inch	0.00001 inch

［Data range］ -99999999～99999999

Note: When bit 2 of parameter No. 001 set diameter programming, X axis is set by diameter system; when bit 2 set radius programming, X axis is set by radius system.

1	**0**	**0**	**LTMAX_2**
1	**0**	**1**	**LTMIN_2**

［Data type］ Word axis

［Data meaning］Stored stroke limit 2 of each axis, LTMAX_2 top stroke position along with the positive direction, LTMIN_2 top stroke position along with the negative direction.

［Data unit］

Setting unit	IS-B	IS-C
Metric machine	0.001 mm	0.0001 mm
Inch machine	0.0001 inch	0.00001 inch

［Data range］ -99999999～99999999

1	**0**	**2**	**LTMAX_3**
1	**0**	**3**	**LTMIN_3**

［Data type］ Word axis

［Data meaning］Stored stroke limit 3 of each axis, LTMAX_3 top stroke position along with the positive direction, LTMIN_3 top stroke position along with the negative direction.

［Data unit］

Setting unit	IS-B	IS-C
Metric machine	0.001 mm	0.0001 mm
Inch machine	0.0001 inch	0.00001 inch

［Data range］ -99999999～99999999

PLC parameter

D	T	0	2	PEALMTIM

［Data meaning］ Under-pressure alarm detection time

［Data unit］ ms

［Data range］ 0～2147483647

D	T	2	1	PSANGNT

［Data meaning］ Chuck operation delay time after spindle stop;

［Data unit］ ms

［Data range］ 0～2147483647

3.2.5 Machine Zero Return

0	0	4	***		DECI		***			

Bit5 1: Deceleration begins as DEC signal is connected to +24V in machine zero return

0: Deceleration begins as DEC signal is disconnected to +24V in machine zero return

0	0	5							PPD	

Bit1 1: Relative and absolute coordinate set together for absolute setting and reference return by G50

0: Only absolute coordinate set for absolute setting and reference point by G50

0	0	6				ZM5	ZM4	ZMY	ZMZ	ZMX

ZMX, ZMZ, ZMY, ZM4, ZM5: Return and initial backlash direction of X, Y, Z axes machine zeroes at power on

0: Zero return type C, machine zero return and backlash direction is positive;

1: Zero return type B, machine zero return and backlash direction is negative.

0	0	7				ZC5	ZC4	ZCY	ZCZ	ZCX

ZCX, ZCZ, ZCY, ZC4, ZC5 =0: Separate DEC and zero signals needed during machine zero return

=1: DEC and zero signals together by a proximity switch during machine zero return

Note: This parameter should be set to 1 if a proximity switch is taken as both deceleration and zero signal.

0	1	1			***	***	NORF	ZNIK	REFC2	***

Bit3 1: Manual machine zero return inactivity

0: Manual machine zero return active

Bit2 1: Stop at zero by pressing direction key once in zero return with direction key locked;

0: Direction key should be held on in zero return with direction key unlocked.

Bit1 1:Ref.C mode,the feeding slowdown when press down switch, it moves negatively after decelerating.

0:Ref.C mode,the feeding slowdown when release switch, it moves negatively after decelerating.

0	1	2

APRS			***		***		ISOT

Bit7 1: ABSOLUTE automatically set after reference return, the coordinate set by parameter No.049 and No.050;

0: ABSOLUTE not automatically set after reference return

Bit 0 1: Prior to machine zero return after power-on or E-stop, manual rapid traverse active

0: Prior to machine zero return after power-on or E-stop, manual rapid traverse inactive

0	1	4

***	***	***	ZRS5	ZRS4	ZRSY	ZRSZ	ZRSX

Bit4~0 1: 5th, 4th, Y, Z, X has machine zero, the system detects DEC signal and zero signal in executing machine zero return.

0: 5th, 4th, Y, Z, X has no machine zero, the system does not detect DEC signal and zero signal and directly executes machine zero return

1	8	3

***	***	***	MZR5	MZR4	MZRY	MZRZ	MZRX

Bit4 1: For the 5th axis, press to execute machine zero return;

0: For the 5th axis, press to execute machine zero return（has not defined the press key in the standard PLC）

Bit3 1: For the 4th axis, press to execute the machine zero return;

0: For the 4th axis, press to execute the machine zero return.

Bit2 1: For Y axis, press to execute the machine zero return;

0: For Y axis, press to execute the machine zero return.

Bit1 1: For Z axis, press to execute the machine zero return;

0: For Z axis, press to execute the machine zero return.

Bit0 1: For X axis, press to execute the machine zero return;

0: For X axis, press to execute the machine zero return.

1	8	8

***	RRTn	***	***	***			

［Data type］ Bit axis

RRTn 1: when Y, 4th, 5th are rotary axis, the zero return use D mode;

0: when Y, 4th, 5th are rotary axis, the zero return use A, B,C mode(need to power-on again).

0	3	3	ZRNFLn

［Data type］ Word axis

［Data meaning］ Lowest speed rate of X,Z machine zero return.

［Data unit］

Set unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］

	IS-B	IS-C
Metric machine	6～4000	6～4000
Inch machine	3～1500	3～1500

0	4	7	PRSn

［Data type］ Word axis

［Data meaning］ Setting value of absolute coordinate after machine zero return is performed

［Data unit］

Setting unit	IS-B	IS-C
Metric input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

［Data range］ -99999999～99999999

1	1	3	REF_SPDn

［Data meaning］ high-speed of each axis machine zero return.

［Data unit］

Set unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］ 10～99999999

1	1	4	REF_OFFSETn

［Data type］ Word axis

［Data meaning］ Offset value of each axis machine zero point

［Data unit］

Set unit	IS-B	IS-C
Metric input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

［Data range］

	IS-B	IS-C
Metric input	-99999～99999	-999999～999999
Inch input	-99999～99999	-999999～999999

1	2	0	REF1_COORDn
1	2	1	REF2_COORDn

1	2	2	REF3_COORDn
1	2	3	REF4_COORDn

［Data type］ Word axis

［Data meaning］ REF1_COORDn machine coordinates of 1st reference point of each axis;
REF2_COORDn machine coordinates of 2nd reference point of each axis;
REF3_COORDn machine coordinates of 3rd reference point of each axis;
REF4_COORDn machine coordinates of 4th reference point of each axis.

［Data unit］

Set unit	IS-B	IS-C
Metric input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

［Data range］ -99999999～99999999

3.2.6 Thread Function

1	7	4						THDTLMOD	THDA	CKSPD

Bit2 1:Thread tailing mode tailling according angle;
0:Thread tailing mode rapid tailing.
Bit1 1:Thread cut is exp. acc&dec
0:Thread cut is lin. acc&dec
Bit0 1:Whether detect the spindle speed is too slow in thread cutting don't detect\
0:Whether detect the spindle speed is too slow in thread cutting do detect

0	1	9	THDCH

［Data meaning］ thread run-out length in thread cutting.

［Data range］ 0～225
Thread run-out width=THDCH×0.1×thread lead

0	2	6	THRDT

［Data meaning］ short acceleration/deceleration time constant in thread run-out.

［Data unit］ ms

［Data range］ 0～4000

0	2	8	THDFL

［Data meaning］ X, Z initial speed in thread cutting.

［Data unit］

Set unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］ 6～8000

1	0	6	THD_SPD_VAR

［Data meaning］ Spindle fluctuation alarm limit in threading (not detect spindle fluctuation alarm if set to 0)

［Data unit］ %

［Data range］ 0～100

Formula=|current speed － previous period speed|×200/(current speed＋previous period speed)

1	0	7	THD_TAIL_SPD

［Data meaning］ Short axis speed in threading run-out (run-out by threading feedrate if set to 0)

［Data unit］

Set unit	Data unit
Metric machine	mm/min
Inch machine	0.1inch/min

［Data range］ 0～8000

3.2.7 Spindle Control

0	0	1				SPIN				

Bit4 1: Analog voltage control of spindle speed

0: Switching volume control of spindle speed

K	1	0							RSJG	

RSJG 1: when the reset is pressed, CNC does not close M03,M04,M08,M32 output signal;

0: when the reset is pressed, CNC closes M03, M04, M08, M32 output signal

0	2	1	PSANGNT

［Data meaning］ voltage offset offset value when the analog voltage outputs 10V.

［Data unit］ mV

［Data range］ -2000～2000

0	3	6	SPDLC

［Data meaning］ voltage offset offset value when the analog voltage outputs 10V.

［Data unit］ mV

［Data range］ -1000～1000

0	3	7	GRMAX1
0	3	8	GRMAX2
0	3	9	GRMAX3
0	4	0	GRMAX4

［Data meaning］ GRMAX1, GRMAX2, GRMAX3, GRMAX4: Max. speeds of spindle gear 1, 2, 3, 4 respectively when analog voltage output is 10V (function defined by GSK980TDi PLC: if the spindle automatic gearing is active, they are the speeds of M41, M42, M43, M44; if the automatic gearing is inactive or CNC is powered on, the gears are memorized by parameter setting or the gear 1 is defaulted.)

［Data unit］ r/min

［Data range］ 10～9999

0	6	7	SFTREV

［Data meaning］ output voltage in spindle gear shifting.

［Data unit］ mV

［Data range］ 0～10000

0	6	8	SJOGREV

［Data meaning］ spindle rotary axis speed in Manual (MPG) mode.

［Data unit］ r/min

［Data range］ 0～3000

0	6	9	ENCODER_CSE

［Data type］ Word spindle

［Data meaning］ The corresponding spindle encoder number of each spindle。

0: without installing the encoder;

1: Spindle revolving signal comes from the encoder interface 1;

2: Spindle revolving signal comes from the encoder interface 2;

3: Spindle revolving signal comes from the GSKLINK bus

［Data range］ 0～3

0	7	0	ENCODER_CNT

［Data meaning］ spindle encoder lines.

［Data unit］ line/r

［Data range］ 100～10000

0	7	2	SAR_DELEY

［Data meaning］ spindle speed arrival signal delay check time.

［Data unit］ ms

［Data range］ 0～4080

0	7	3	SPMOTORMAX

［Data meaning］ max. clamp speed of spindle motor speed.

［Data unit］ r/min

［Data range］ 0～4095

Setting value=（max. clamp speed of spindle motor/its max. speed）×4095.

0	7	4	SPMOTORMIN

［Data meaning］ min. clamp speed of spindle motor speed.

［Data unit］ r/min

［Data range］ 0～4095

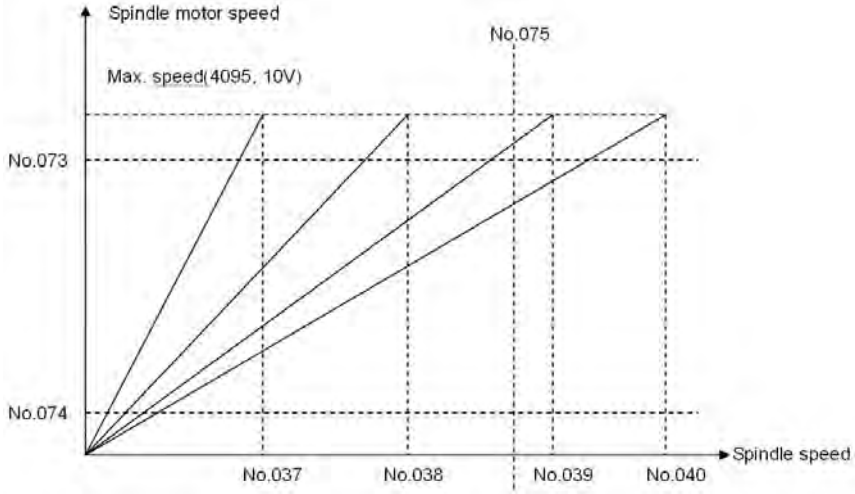
Setting value=（min. clamp speed of spindle motor/its min. speed）×4095.

0	7	5	SPSPEEDLIMIT

［Data meaning］ max. spindle speed ［0: not limit speed］.

［Data unit］ r/min

［Data range］ 0～6000



0	7	6	SPDLIMITEXT

［Data type］ Word spindle

［Data meaning］ The spindle speed limit when speed limit signal is active [It is not restricted when setting to 0]

［Data unit］ r/min

［Data range］ 0～9999

1	0	9	SPL_REV_SPD

［Data meaning］ rotary speed in spindle jog.

［Data unit］ r/min

［Data range］ 1～8000

1	1	0	MGR

［Data type］ Word spindle

［Data meaning］ Gear ratio of encoder to spindle: spindle gear teeth number

［Data range］ 1～255

1	1	1	SGR

［Data type］ Word axis

［Data meaning］ Gear ratio of encoder to spindle: encoder gear teeth number

［Data range］ 1～255

1	9	5	SSPD	PLCS	***	***	RPOS	SRV	CSREF	RCS

Bit7 1:series spindle controllable function enabled (series spindle The slave ID number of the spindle 1~3 is 11~13) , it is necessary to restart;

0:series spindle controllable function disabled (analogy spindle), it is necessary to restart.

Bit6 1:PLC spindle rotation speed function of each axis enabled;

0:PLC spindle rotation speed function of each axis disabled。

Bit3 1:Spindle position function is valid (The spindle will perform the mechanical releasing/clamping interlocking operation before/after positioning when it is enabled);

0:Spindle position function is invalid.

Bit2 1:Servo motor spindle function is valid ,power on；

0:Servo motor spindle function is invalid ,power on。

Bit1 1:Change to position mode,the axis do return REF.

0:Change to position mode,the axis don't return REF.

Bit0 1:Cs function of spindle valid ,power on

0:Cs function of spindle is:invalid,power on

1	9	6

***	***	***	MSEN	***	***	***	***

Bit4 1: The multiple spindle control is enabled

0: The multiple spindle control is disabled

2	0	1

SNTP	***	***	ECN	HBR	ERV	***	***

Bit7 1:Synchronization mode follow

0:Synchronization mode interpolation

Bi4 1:Whether the G81 synchronization command can be specified in the synchronization process allow

0:Whether the G81 synchronization command can be specified in the synchronization process prohibition

Bit3 1:The system resets in the synchronization process reserve the synchronized control

0:The system resets in the synchronization process cancel the synchronization control

Bit2 1:In the synchronization procedure,in the G99 feed per revolution use the theoretical revolving speed of workpiece\

0:In the synchronization procedure,in the G99 feed per revolution use the encoder feedback revolving speed

2	2	0

SP_SERVOn

［Data type］ Word spindle

［Data meaning］The corresponding interface number of each spindle output

［Data unit］

［Data range］ 0～14

Setting value	meaning
2	analog spindle interface 2
1	analog spindle interface 1
0	Disabled analog spindle output
11～14	extension IO unit analog output interface; 1~4

2	2	1

SP_SP_CSAXIS

［Data type］ Word spindle

［Data meaning］The functions, such as the feed axis number, CS axis related to each axis must be set.

［Data unit］

［Data range］ 0～5

Setting value	meaning	Setting value	meaning	Setting value	meaning
0	Nothing	2	Z axis	4	4th axis
1	X axis	3	3rd axis	5	5th axis

3.2.8 Tool Compensation

0	0	2								
									NRC	

Bit1 1: Tool nose radius offset active

0: Tool nose radius offset inactive

0	0	3								
						TCOMP	CDCOMP			OIM

Bit4 1: Tool offset by coordinate offset

0: Tool offset by move

Bit3 1: Tool compensated by the shift of coordinate system do not memorize the unexecuted tool compensation

0: Tool compensated by the shift of coordinate system memorize the unexecuted tool compensation

Bit0 1: Offset automatically change in metric and inch conversion

0: Offset not change in metric and inch conversion

0	0	4								
						ORC			PROD	

ORC 1: Tool offset by radius

0: Tool offset by diameter

PROD 1: Programming position for RELATIVE display

0: Position involving offset for RELATIVE display

0	1	2								
					DOFSI					

DOFSI 1: Trial tool setting active

0: Trial tool setting inactive

3.2.9 Tool Life Management Function

0	0	2								
				***	***	LIFJ	MDITL	LIFC		TLIF

Bit4 1: Tool life management group skip active

0: Tool life management group skip inactive

Bit3 1: Tool life management active in MDI mode

0: Tool life management inactive in MDI mode

Bit2 1: Tool life counting type 2, by times

0: Tool life counting type 1, by times

Bit0 1: Tool life management active

0: Tool life management inactive

3.2.10 Tool Wear Parameter

1	4	0	TWEAR_MAX

［Data meaning］ positive/negative limit of tool wear value in tool offset & wear window.

［Data unit］

Set unit	IS-B	IS-C
Metric input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

［Data range］

	IS-B	IS-C
Metric input	1～99999	1～999999
Inch input	1～99999	1～999999

3.2.11 Edit & Display

0	0	5				M30				

M30 1: Cursor to beginning after M30
0: Cursor to beginning by pressing CYCLE START key after M30

0	1	2					ALM& PRG			

Bit3 1: Program editing allowed during CNC alarming
0: Program editing unallowed during CNC alarming

1	8	0					DISPC	CUSTOM		SPOS

Bit3 1:Boot dispaly if custom page setted to display custom page
0:Boot dispaly if custom page setted to display fault page
Bit2 1:Display custom page display
0:Display custom page hide
Bit0 1: DIS TO GO displayed in POS&PRG page
0: RELATIVE POS displayed in POS&PRG page

1	8	1				CLMDI	MCL	M30&MDI		

Bit4 1:When system reset,the program block of MDI are deleted
0:When system reset,the program block of MDI are not deleted
Bit3 1: Blocks edited in MDI mode are deleted after reset;
0: Block edited in MDI mode are not deleted after reset;
Bit2 1:After the execution of M02, M30,the program block of MDI are not deleted
0:After the execution of M02, M30,the program block of MDI are deleted

1	8	4	***	PTEST	***	***	***	L2	L1	L0

Bit0, Bit1, Bit2: Interface language selection

Language	Bit2	Bit1	Bit0
Chinese	0	0	0
English	0	0	1

Portugal	0	1	0
Spanish	0	1	1
Russian	1	1	0

3.2.12 Communication Setting

0	4	4	BRATE0

［Data meaning］ baud rate of serial communication.

［Data unit］ bit/s

［Data range］ 1200,2400,4800,9600,19200,38400,57600 ,115200

3.2.13 MPG Parameters

0	0	1		***	***		MPG			

Bit3 1: MPG mode

0: Step mode

0	1	3	RTH	RPH	***	HW5	HW4	HWY	HWZ	HWX

Bit7 1: When system reset,the handle interrupt move amount is cancelled

0: When system reset,the handle interrupt move amount is not cancelled

Bit6 1:When MPG feedrate exceeds the rapid movement speed,the exceeding part is not ignored and saved in CNC

0:When MPG feedrate exceeds the rapid movement speed,the exceeding part is ignored and saved in CNC

Bit4~Bit0 1: coordinates increase when 5th, 4th, Y, Z or X MPG rotates CCW;

0: coordinates increase when 5th, 4th, Y, Z or X MPG rotates CW.

K	1	6	SINC		***	***	***	***	***	***

Bit7 1: In STEP(MPG) mode, ×1000 gear is invalid

0: In STEP(MPG), ×1000 gear is valid

2	4	3	CNC_MPG_NUM

［Data meaning］ Quantity of MPG

［Data range］ 1～2

2	4	4	MPG1_INTERFACE
2	4	5	MPG2_INTERFACE

［Data meaning］ MPG interface of MPG 1,2

［Data range］ 0～1

Setting value	Meaning
0	Rear cover MPG interface 1 (CN31)
1	Rear cover MPG interface 2(CN32)

3.2.14 PLC Axis Control Function

1	8	5

			***	***	***	PRPD	PLA

PRPD 1: Rapid traverse rate of PLC axis by input value

0: Rapid traverse rate of PLC axis by parameter value (X axis: No.022; Z axis:No.023; Y axis:No.155;)

PLA 1: PLC axis control active, and active after repowering

0: PLC axis control inactive, and active after repowering

Note: The PLC axis control function is only provided by machine builder.

2	1	9

PLCDIDOn

［Data type］ Word axis

［Data meaning］ Select DI/DO in PLC axis control PLC

［Data unit］

［Data range］ 0～4

3.2.15 Skip Function

1	8	5

SK0	SKF		***	***	***		

SK0 1: Skip input active as G31 signal is 0

0: Skip input active as G31 signal is 1

SKF 1: Feedrate override per minute or dry run active for G31

0: Feedrate override per minute or dry run inactive for G31

Note: The skip function is only provided by machine builder.

1	4	8	MULTISTAGE_SKIP1
1	4	9	MULTISTAGE_SKIP2
1	5	0	MULTISTAGE_SKIP3
1	5	1	MULTISTAGE_SKIP4

［Data type］ Word

［Data meaning］ Multilevel skip G31 P1～P4 or G04 Q1～Q4 signal detect address setting(0:no effect; 20~27,30~37: address X2.0~X3.7)

［Data range］ 0、20~27, 30~37

3.2.16 Automatic Tool Setting Function

1	8	5

		AEO	***	***	***		

AWO 1: when G36, G37 signal（XAE，ZAE）is 0, the input is valid;

0: G36, G37 signal（XAE，ZAE）is 1, the input is valid.

1	4	1

AUTO_OFFSET_FEED

［Data meaning］ feedrate in automatic tool compensation measure.

［Data unit］

Set unit	Data unit
Metric machine	mm/min

Inch machine	0.1inch/min

［Data range］

	IS-B	IS-C
Metric machine	15～1000	15～1000
Inch machine	7～375	7～375

1	4	2	AUTO_OFFSET_Y_X
1	4	3	AUTO_OFFSET_Y_Z

［Data meaning］ X（AUTO_OFFSET_Y_X）, Z（AUTO_OFFSET_Y_Z）γ value in automatic tool offset.

［Data unit］

Set unit	IS-B	IS-C
Metric machine	0.001 mm	0.0001 mm
Inch machine	0.0001 inch	0.00001 inch

［Data range］ 1～9999999

Note: X setting value is the radius in diameter programming and radius programming.

1	4	4	AUTO_OFFSET_E_X
1	4	5	AUTO_OFFSET_E_Z

［Data meaning］ AUTO_OFFSET_E_X（X）,AUTO_OFFSET_E_Z（Z）ε value in automatic tool offset.

Note: The function needs the automatic tool setting device.

3.2.17 Input & Output Function in Metric/Inch System

0	0	1		***	***					INI

INI 1: Inch input
0: Metric input

0	0	3	***	***			***	***	***	OIM

OIM 1: Offset automatically change in metric and inch conversion
0: Offset not change in metric and inch conversion

0	0	4	***							SCW

SCW 1: Inch system for min. code unit, active after repowering
0: Metric system for min. code unit, active after repowering

3.2.18 Parameters Related to Arc Turning

1	5	4	RADIUS_ERR_RANGE

［Data meaning］ max. value of arc radius error.

［Data unit］

Setting unit	IS-B	IS-C
Metric machine	0.001 mm	0.0001 mm
Inch machine	0.0001 inch	0.00001 inch

［Data range］

	IS-B	IS-C
Metric machine	0～1000	0～10000
Inch machine	0～50	0～500

3.2.19 Parameters Related to Addition

1	8	7

IS1	IS0	RCS	***	***	***	ROS	ROT

[Data type]: Bit axis

BIT7,BIT6: set incremental system of each axis，00: consistent with the incremental system of the system，01: IS-A，10: IS-B；11: IS-C.

Bit5 1: Cs function of additional axis is valid/invalid (power-on again);
0: Cs function of additional axis is valid/invalid (power-on again).

Bit1,Bit0: 00 sets the additional axis to be the linear, 01 sets the additional axis to be the rotary(A type)，11 sets the additional axis to be the rotary (B type)，10 sets the additional axis to be invalid.

Note: This parameter is invalid to axes X and Z.

1	8	8

***	RRT	***	***	***	RRL	RAB	ROA

[Data type]: Bit axis

Bit6 1: When the additional axis is rotary, the zero return uses D mode;
0: When the additional axis is rotary, the zero return uses A, B, C (power-on again).

Bit2 1: When the additional axis is rotary, cycle function of the relative coordinate is valid;
0: When the additional axis is rotary, cycle function of the relative coordinate is invalid (power-on again).

Bit1 1: When the additional axis is rotary, its rotation direction is based on the sign direction;
0: When the additional axis is rotary, its rotation direction is contiguous.

Bit0 1: When the additional axis is rotary, cycle function of the absolute coordinate is valid;
0: When the additional axis is rotary, cycle function of the absolute coordinate is invalid (power-on again).

Note: This parameter is invalid to axes X and Z.

2	2	5

NAME_n

［Data type］ Word axis

［Data meaning］ axis name definitions

［Data range］ 65, 66, 67, 88, 89, 90

Axis name	Setting value	Axis name	Setting value	Axis name	Setting value
X	88	A	65	C	67
Z	90	B	66	Y	89

2	3	0

ATTRIB_n

［Data type］ Word axis

［Data meaning］ The properties of each axis in coordinate system

［Data range］ 0～7

Setting value	Meaning
0	Neither the basic three axes nor the parallel axis

1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Parallel axis of X axis
6	Parallel axis of Y axis
7	Parallel axis of Z axis

3.2.20 GSKLink Bus Related to Absolute Encoder

1	8	9	CTS	***	APC	APZ	***	***	***	***

[Data type]:Bit axis

Bit7 1:Each servo axis didn't connect servo drive , it is necessary to restart;
0:Each servo axis connected servo drive , it is necessary to restart.

Bit5 1:The used encoder type of each axis is absolute encoder;
0:The used encoder type of each axis is incremental encoder。

Bit4 1:When each axis matches with an absolute encoder, the mechanical position and absolute one are shown consistent;
0:When each axis matches with an absolute encoder, the mechanical position and absolute one are shown inconsistent.

1	9	5	SSPD							

Bit7 1:series spindle controllable function enabled (series spindle The slave ID number of the spindle 1~3 is 11~13) , it is necessary to restart;
0:series spindle controllable function disabled (analogy spindle), it is necessary to restart.

2	4	9	ABSPOSERR

［Data type］ Word axis

［Data meaning］When an absolute encoder is equipped with each axis, the Max. position error value allows when the coordinate system is set up with power-on.

［Data unit］

Setting unit	IS-B	IS-C
Metric input	0.001 mm	0.0001 mm
Inch input	0.0001 inch	0.00001 inch

［Data range］ 0～999999

2	5	0	***	***	***	***	***	***	VERI	GSKLINK

Bit1 1:System GSKLINK bus communication CRC verification enabled
0:System GSKLINK bus communication CRC verification disabled

Bit0 1: Whether the system GSKLINK bus connection is enabled
0: Whether the system GSKLINK bus connection is disabled

CHAPTER 4 MACHINE DEBUGGING METHODS AND MODES

The trial run methods and steps at initial power on for this GSK980TDi/GSK980TDi-V are described in this chapter. The corresponding operation can be performed after the debugging by the following steps.

4.1 ESP & Limit

This GSK980TDi/GSK980TDi-V system has software limit function, it is suggested that hardware limit is employed by fixing the stroke limit switches in the positive or negative axes.

4.1.1 ESP & Limit Series-Connection

The connection is as follows (2 axes):

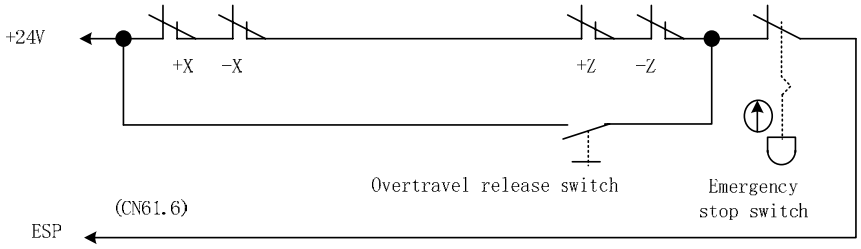


Fig. 4-1

So the BIT3 (ESP) of bit parameter No.172 should be set to 0.

The diagnostic message DGN000.7 monitors the emergency stop input signal.

In Manual or MPG mode, slowly move the axes to testify the validity of stroke limit switch, correctness of alarm display, validity of overtravel release button. When the overtravel occurs or Emergency Stop button is pressed, “ESP” alarm will be issued by CNC system. The alarm can be cancelled by pressing down the OVERTRAVEL key for reverse moving.

4.1.2 Separately Connect of ESP & Limit

The connection is shown below; refer to the Fig. 4-2 (3-axis is regarded as an example):

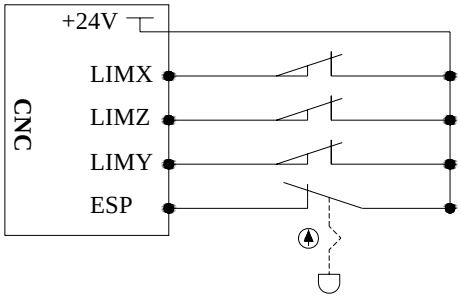


Fig. 4-2

The Bit 3 of status parameters №172 equals to 0: ESP function is enabled;

The Bit 7 of PLC parameter K10 equals to 1: The overtravel function along each axis is enabled;

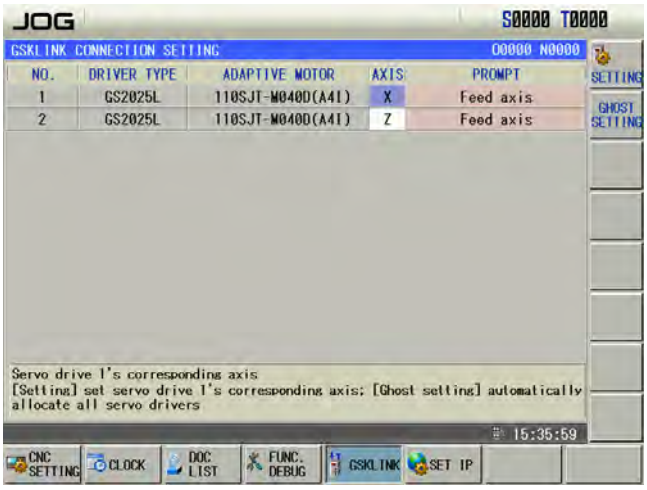
The Bit6 of PLC parameter K10: High/Low LEVEL selection of overtravel signal

In the Manual or MPG mode, slowly move the effectiveness of the overtravel limit swith of each axis verification or the correction of alarm display; the CNC ESP alarm may occur when pressing the ESP button is controlled; when the overtravel alarm occurs, it will moves towards to the negative direction; after moving out the limit position, the resetting clear alarm then can be pressed.

4.2 Drive Unit Configuration

The corresponding axis of each servo drive unit should be set based upon the valid configuration of the system, which can be directly set on the page of “Setting->GSKLink setting” based upon the 2-level authorities.

Under the 2-level or more authorities, enter to the setting page by SETTING, and then to the GSKLink setting page by GSKLink, the display is shown below:



Select the servo drive by ↑ or ↓ button, set the selected corresponding axis of the servo drive by SETTING: When the connection sequence of the servo drive is: X axis, Z axis, Y axis, the 4th axis, the 5th axis, spindle 1, spindle 2 or spindle 3 (regardless of the unavailable axis), the overall servo drives can be set by GHOST SETTING.

If the connection sequence of the servo axis does not correct, we would not definitely know the corresponding servo drive of each axis on the page of the “Setting->GSKLink setting”, it can be directly set at the side of the servo side. The corresponding setting parameter of the servo is PA156; the comparison table of the parameter setting is shown below:

System axis	Servo parameter	System axis	Servo parameter	System axis	Servo parameter
X	1	Y	3	5TH	5
Z	2	4TH	4		
Spindle 1	11	Spindle 2	12	Spindle 3	13

The Bit 4, Bit 3, Bit 2, Bit 1 and Bit 0 (DIR4, DIR5, DIRY, DIRZ or DIRX separately corresponds to the 5th, the 4th, Y, Z or X axis) of the state parameter No.008 can be modified if the machine tool

movement direction is inconsistent with the one of command required.

The manual movement direction can be changed by the Bit 4, Bit 3, Bit 2, Bit 1 and Bit 0 (5VAL, 4VAL, YVAL, ZVAL or XVAL separately corresponds to the 5th, 4th, Y, Z or X axis movement button) of parameter No.175.

4.3 Gear Ratio Adjustment

№015～№016 can be modified for electronic gear ratio adjustment to meet the various mechanical transmission ratios when the machine travel distance is not consistent with the displacement distance displayed by the CNC.

Formula:

$$\frac{CMR}{CMD}=\frac{4*C*\delta}{L}*\frac{Z_M}{Z_D}$$

CMR: Code multiplier coefficient (data parameter No.015)

CMD: Code frequency division coefficient (data parameter No.016)

C: Motor' s encoder lines

α : Pulse volume, motor rotation angle for a pulse

L: Screw lead

δ: Current min. input code unit of CNC

ZM : gear teeth number of lead screw

ZD: gear teeth number of motor

For example, the screw lead and motor are directly connected by linkage axis, the motor's encoder is 17-bit absolute one, screw lead is 4mm, the current increment of CNC is 0.1μ;

X electronic gear ratio:

$$\frac{CMR}{CMD}=\frac{C*\delta}{L}*\frac{Z_M}{Z_D}=\frac{131072*0.00005}{4}*\frac{1}{1}=\frac{1024}{625}$$

Z electronic gear ratio:

$$\frac{CMR}{CMD}=\frac{C*\delta}{L}*\frac{Z_M}{Z_D}=\frac{131072*0.0001}{4}*\frac{1}{1}=\frac{2048}{625}$$

Then, the parameter data №015X（CMRX）=1024， №016X（CMDX）=625；№015Z（CMRZ）=2048， №016Z（CMDZ）=625。

If the electronic gear ratio numerator is more than the denominator, the allowed CNC max. speed will decrease. For example: the data parameter No.015Z (CMRZ) =2, No.016Z (CMDZ) =1, so the allowed Z axis max. speed is 30000mm/min.

If the electronic gear ratio numerator is not equal to the denominator, the allowed CNC positioning precision may decrease. For example: the data parameter No.015Z(CMRZ) =1, No.016Z(CMDZ) =5, so the pulse is not output as the input increment is 0.0004, but a pulse is output when the input increment is 0.0005.

In order to ensure the CNC positioning precision and speed index, it is suggested that the CNC electronic gear ratio is set to 1:1 and the electronic gear ratio calculated is set to the digital servo when matching with digital servo.

When matching with the step drive, choose the drive unit with step division function as possible as it can, and properly select mechanical transmission ratio. The 1:1 electronic gear ratio should be ensured to avoid the excessive difference between the numerator and the denominator of this CNC electronic gear ratio.

4.4 ACC&DEC Characteristic Adjustment

Adjust the relative CNC parameters according to the factors such as the drive unit, motor characteristics and machine load:

Data parameter No.022: rapid traverse rate of each axis;

Data parameter No.023: linear ACC&DEC time constant of each axis rapid traverse rate;

Data parameter No.026: X axis exponential ACC&DEC time constant in threading;

Data parameter No.028: exponential ACC&DEC start/termination speed in threading;

Data parameter No.029: exponential ACC&DEC time constant in cutting and manual feeding;

Data parameter No.030: exponential ACC&DEC start/end speed in cutting feeding;

Data parameter No.041: ACC&DEC start/termination speed in manual feeding;

BIT5 (SMZ) of bit parameter No.007: for smooth transition between cutting feedrates of adjacent blocks

The larger the ACC&DEC time constant is, the slower the ACC&DEC is, the smaller the machine movement impact and the lower the machining efficiency is, and vice versa.

If ACC&DEC time constants are equal, the higher the ACC&DEC start/end speed is, the faster the ACC&DEC is, the bigger the machine movement impact and the higher the machining efficiency is, and vice versa.

The principle for ACC&DEC characteristic adjustment is to properly reduce the ACC&DEC time constant and increase the ACC&DEC start/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 ACC&DEC time constant is set too small, and the start/end speed is set too large, it is easily to cause faults such as drive unit alarm, motor out-of-step or machine vibration.

When the bit parameter No.007 BIT5(SMZ) =1, the feedrate drops to the start speed of the ACC&DEC at the cutting path intersection, then it accelerates to the specified speed of the next block to obtain an accurate positioning at the path intersection, but this will reduce the machining efficiency. When BIT5=0, the adjacent cutting path transits smoothly by the ACC&DEC. The feedrate does not always drop to the start speed when the previous path is finished and a circular transition (non-accurate positioning) will be formed at the path intersection. The machining surface by this path transition has a good finish and a higher machining efficiency. When the stepper motor drive unit is applied, the BIT5 of the bit parameter No.007 should be set to 1 to avoid the out-of-step.

If AC servo drive unit is applied to this system, the machining efficiency can be improved by a larger start speed and a smaller ACC&DEC time constant setting. If optimum acc/dec characteristics are required, the ACC&DEC time constant may be set to 0 which can be gotten by adjusting the AC servo acc/dec parameters. The suggested settings for these parameters are as follows(electronic gear ratio is 1:1):

Data parameter No.022 X＝5000	Data parameter No.022 Z＝10000
Data parameter No.023 X≤60	Data parameter No.023 Z≤60
Data parameter No.029≤50	Data parameter No.028≤500
Data parameter No.026≤50	Data parameter No.030≤400

The parameter settings above are recommended for use, please refer to the actual conditions of the drive unit, motor characteristic and motor load for their proper setting.

4.5 Mechanical (Machine) Zero Adjustment

The debugging methods of the machine zero point are different based upon the encoder type of the adapted motor.

1. When the adapated absolute encoder and the servo axis is linear one.

When the bus connection is normal, as well the debugging of the electric gear ratio of servo axis or the movement direction is normal, the position from movement axis to preset value on machine tool is regarded as the machine zero, directly set the current position on the machine as the machine reference point by the [n axis reference point setting] soft button on the page of the “Position—> Integrated coordinate”. The machine will directly position to the reference point when it returns to zero after the reference point is set.

2. When the adapted incremental encoder or servo axis is rotation axis.

Adjust the relevant parameters based on the active level of the connection signal, zero return type and direction applied:

BIT5 (DECI) of the bit parameter No.004: active level of deceleration signal in machine zero return.

BIT0, BIT1, BIT2，BIT3，BIT4 (ZMX, ZMZ, ZMY，ZM4, ZM5) of the bit parameter No.006: X, Z, Y, 4th, 5th axis machine zero return mode B or C selection.

BIT0, BIT1, BIT2, BIT3, BIT4 (ZCX, ZCZ, ZCY, ZC4, ZC5) of the bit parameter No.007: whether a proximity switch is taken as both deceleration and zero signals.

BIT2(ZNLK) of the bit parameter No.011: whether the direction key in zero return is self-locked.

BIT0, BIT1, BIT2, BIT3, BIT4 (ZRSCX, ZRSCZ, ZRSCY, ZRSC4, ZRSC5) of the bit parameter No.014: have/not the machine zero(zero turn mode BC/A)setting bit.

Data parameter No.033: low deceleration speeds of each axis in machine zero return.

Data parameter No.113: high speed of each axis in machine zero return.

BIT0, BIT1, BIT2, BIT3, BIT4 (MZRX, MZRZ, MZRY, MZR4, MZR5) of the bit parameter No.183: each axis zero return direction: negative or positive.

Only the stroke limit switch validity is confirmed, can the machine zero return be performed.

The machine zero is usually fixed at the max. travel point, and the effective stroke of the zero return touch block should be more than 25mm to ensure a sufficient deceleration distance for accurate zero return. The more rapid the machine zero return is, the longer the zero return touch block should be. Or the moving carriage will rush over the block and it may affect the zero return precision because of the insufficient deceleration distance.

The connection to AC servo motor: using a travel switch and servo motor one-turn signal separately

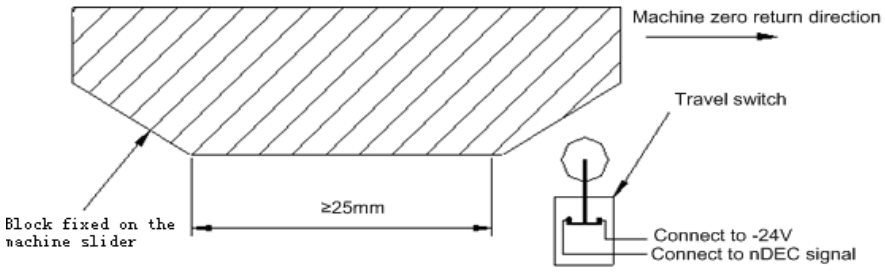


Fig. 4-2

By this connection, when the deceleration switch is released in machine zero return, the one-turn signal of encoder should be avoided to be at a critical point after the travel switch is released. In order to improve the zero return precision, and it should ensure the motor reaches the one-turn signal of encoder after it rotates half circle.

The parameter setting is as follows:

Bit parameter No.004 BIT5(DECI) =0

Bit parameter No.006 BIT0(ZMX) , BIT1(ZMZ) , BIT2(ZMY), BIT3(ZM4), BIT4(ZM5) =0

Bit parameter No.007 BIT0(ZCX) , BIT1(ZCZ) , BIT2(ZCY) , BIT3(ZC4), BIT4(ZC5)=0

Bit parameter No.011 BIT2(ZNLK) =1

Bit parameter No.014 BIT0(ZRSCX) , BIT1(ZRSCZ) , BIT2(ZRSCY), BIT3(ZRSC4), BIT4(ZRSC5) =1

Data parameter No.033=200

Data parameter No.183 BIT0(MZRX) , BIT1(MZRZ) , BIT2(MZRY), BIT3(MZR4) , BIT4(MZR5) =0

4.6 Spindle Adjustment

4.6.1 Spindle Encoder

Encoder with the pulses 100~10000p/r is needed to be installed on the machine for threading. The pulses are set by data parameter No.70. 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. The encoder number used by spindle is set in the data parameter No.069.

Synchronous belt transmission should be applied for it (no sliding transmission).

The DGN.011 diagnosis message is used to check the validity of thread signal of the spindle encoder.

4.6.2 Spindle Brake

After M05 code is executed, proper spindle brake time should be set to stop the spindle promptly in order to enhance the machining efficiency. If the brake is employed with energy consumption type, too long braking time may burn out the motor.

Data parameter No.087: delay from spindle stop(M05) to spindle brake output

Data parameter No.089: spindle braking time

4.6.3 Switch Volume Control of Spindle Speed

When the machine is controlled by a multi-speed motor, the motor speed codes are S01～S04. The relevant parameters are as follows:

State parameter No.001 Bit4=0: select spindle speed switch control;

4.6.4 Analog Voltage Control of Spindle Speed

This function can be obtained by the parameter setting of CNC. By interface outputting 0V～10V analog voltage to control frequency inverter, the stepless shift can be obtained. And the related parameters needed to be adjusted are:

Bit parameter No.001 Bit4=1: for spindle speed analog voltage control;

The Bit 7 of the status parameters №195 equals to 0: Select the spindle revolving speed is analog voltage control;

Data parameter No.021: offset value as spindle speed code voltage is 10V;

Data parameter No.036: offset value as spindle speed code voltage is 0V;

Data parameter No.037～ No.040: for max. speed clamping of spindle gear 1～4; it defaults the spindle gear 1 when CNC power on.

Basic parameters are needed to adjust the inverter:

CW or CCW code mode selection: it is determined by terminal VF;

Frequency setting mode selection: it is determined by 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 No.037～No.040.

Speed adjustment method: select the corresponding spindle gear, determine the data parameter is 9999 as for this system gear, set the spindle override for 100%. Input spindle run command in MDI mode to run the spindle: M03/M04 S9999, view the spindle speed shown on the right bottom of the screen, then input the speed value displayed into the corresponding system parameter.

When entering S9999 code, the voltage should be 10V, S0 for 0V. If there is a voltage error, adjust bit parameter No.021 and No.036 to correct the voltage offset value (corrected by manufacturer, usually not needed).

For the current max. speed gear, if the analog voltage output by CNC is not 10V, set it for 10V by adjusting the data parameter No.021; when the input speed is 0, if the spindle still slowly rotates, it means the analog voltage output by CNC is higher than 0V, so set a smaller value for data parameter No.036.

If the machine is not fixed with an encoder, the spindle speed can be detected by a speed sensing instrument, input S9999 in MDI mode to set the speed value displayed by the instrument into the data parameter No.037～ No.040.

4.6.5 Spindle Speed Series Control

The spindle speed series control can be carried out by CNC parameter, directly control the spindle servo drive unit to achieve the stepless speed by delivering the seires data from bus; the relevant parameters to be adjusted are shown below:

The Bit 4 of the status parameter №001 equals to =1: Select the spindle speed analog voltage or series control;

The Bit 7 of the status parameter №195 equals to 1: Select the spindle speed as the series control;

Data parameters №037～№040: The top speed of each step;

When the revolving speed specified by programming is inconsistent with the one of the encoder detection, the specified speed becomes consistent with the actual one by adjusting the data parameters №037～№040.

Revolving speed adjustment method: Firstly, shift the spindle to the corresponding gears and ensure that the corresponding gear data parameter for the system is 9999, adjust the spindle override as 100%; Input the spindle operation command on the MDI interface and operate the M03/M04 S9999; secondly, observe the spindle speed on the right corner of the screen; lastly, input the displayed revolving speed value to the corresponding data parameter of the gear.

When the machine tool does not install the encoder, it can be detected the spindle speed by revolving speed sensor; MDI code inputs the S9999; Set the displayed revolving speed on its sensor to the corresponding gear data parameters №037～№040.

4.7 Backlash Offset

The X axis backlash offset value is input by diameter, Z axis backlash offset value is input by the actual backlash which 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 (taking example of Z):

```
O0001；
N10 G01 W10 F800 ；
N20 W15；
N30 W1；
N40 W-1；
N50 M30.
```

- Set the backlash error offset to 0 before measuring:
- Run the program by single blocks, search the measuring benchmark after 2 positioning operations, record the current data, move 1mm in the same direction, then move 1mm to point B reversely, read the current data.

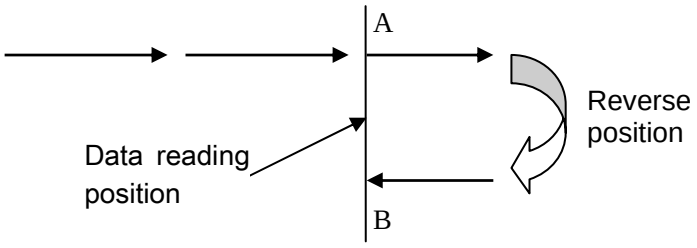


Fig. 4-4 Schematic map of backlash measuring method

- Backlash error offset value= | data of point A –data of point B |; then input its outcome to the data parameter No.034（BKLn） (for X axis, multiply 2 and then input the outcome to data parameter No.034).

  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 Bit7 of CNC parameter No.011; the backlash frequency can be set by Bit6 of parameter No.011 and Bit4, Bit3, Bit2, Bit1, Bit0 of bit parameter No.010.

Note 2: Check the machine backlash every 3 months.

4.8 Tool Post Debugging

GSK980TDi supports various kinds of tool post, and the parameter settings are based on the machine manual. The parameter settings for the tool post running are as follows:

BIT2(TSGN) of K parameter No.011: high/low level selection of tool post in-position signal, when the signal is low level active, a parallel pull-up resistor is needed.

Bit3 (CTCP) of K parameter No.011: check/do not check tool post lock signal in tool change;

Bit4 (TCPS)of K parameter No.011: tool post lock signal HIGH/LOW selection;

Bit5 (CHET) of K parameter No.11: check/do not check tool signal;

Combinations and functions of tool change mode selection Bit1(CHTB), Bit0(CHTA) of K parameter No.11 are referred to **Tool Change Control**.

Data parameter No.076: Duration for moving a tool in tool changing

Data parameter No.078: Upper limit time for changing one tool

Data parameter No.082: Delay time from tool post CCW stop to CW clamping start

Data parameter No.084: Total tools number

Data parameter No.085: Delay of tool post CW clamping

If the tool post doesn't rotate at first power on for tool change, the phase connection of the 3-phase power of the tool post motor may be incorrect, it needs to press the RESET key immediately and cutoff the power, then check the wiring; if the fault is caused by this, exchange two phases of the 3-phase power.

The CW clamping duration setting should be proper, it should be neither longer nor shorter, longer delay may damage the motor, shorter delay may cause the tool post not to be completely clamped. The method to check the tool post clamping is: approach the dial-indicator to the tool post, turn the tool post manually, and the pointer floating of the dial-indicator should not be over 0.01mm.

The BIT7(TL-) and BIT6(TL+) of diagnosis message DGN.005 are used to check the validity of tool post CCW/CW output signal.

The BIT0～BIT3 (T01～T04) of diagnosis message DGN.000 are used to check the validity of tool number T01～T04 signal.

During debugging, every tool, max. tools change should be performed to check the correctness of the tool change, time parameter setting.

4.9 Step/MPG Adjustment

The key on the panel can be used to select the Step mode or MPG mode, which is set by the BIT3 of bit parameter No.001.

Bit3 =1: MPG mode active, Step mode inactive;

=0: Step mode active, MPG mode inactive;

CHAPTER 5 DIAGNOSIS MESSAGE

Diagnosis messages for GSK980TDi system are described in this chapter.

5.1 CNC Diagnosis

The part is used to check the CNC interface signals and internal running and it can't be modified.

5.1.1 I/O Status & Data Diagnosis Message

0	0	0
Pin		
PLC fixed address		

ESP	***	***	DEC5	DEC4	DECZ	DECY	DECX
CN61.6			CN61.34	CN61.33	CN61.12	CN61.32	CN61.4
X0.5			X2.5	X2.4	X1.3	X2.3	X0.3

DECX, DECY, DECZ, DEC4, DEC5: machine zero return signal of X, Y, Z, 4th, 5th
ESP: emergency stop signal

0	0	1
Pin		
PLC fixed address		

***	***	***	***	***	***	***	SKIP
							CN61.42
							X3.5

SKIP: skip signal

5.1.2 CNC Motion State & Data Diagnosis Message

0	0	4

***	***	***	EN5	EN4	ENZ	ENY	ENX

EN5～ENX: X, Z, Y, 4th , 5th enabling signal

0	0	6

***	***	***	DRO5	DRO4	DROZ	DROY	DROX

DRO5～DROX: X, Y, Z, 4th, 5th motion direction output

0	0	9

***	***	***	ALM5	ALM4	ALMZ	ALMY	ALMX

ALM5～ALMX: X, Y, Z, 4th, 5th alarm signal

0	1	0	MPG rotation data
0	1	1	Spindle speed 1 feedback data
0	1	2	Spindle speed 2 feedback data
0	1	3	Spindle 1 analog output value
0	1	4	Spindle 2 analog output value
0	1	5	Spindle 3 analog output value
0	4	8	X axis output pulse number
0	4	9	X axis motor feedback position

0	5	0	Y axis output pulse number
0	5	1	Y axis motor feedback position
0	5	2	Z axis output pulse number
0	5	3	Z axis motor feedback position
0	5	4	4th axis output pulse number
0	5	5	4th axis motor feedback position
0	5	6	5th axis output pulse number
0	5	7	5th axis motor feedback position
0	6	1	MPG 1 counting（Rear cover MPG interface CN31）
0	6	2	MPG 2 counting（Rear cover MPG interface CN32）
0	6	5	Spindle 1 encoder counting（Rear cover CN21）
0	6	6	Spindle 2 encoder counting（Rear cover CN22）
0	6	9	The pulse number from the time that X axis starts detecting PC signal to PC signal is detected
0	7	0	The pulse number from the time that Y axis starts detecting PC signal to PC signal is detected
0	7	1	The pulse number from the time that Z axis starts detecting PC signal to PC signal is detected
0	7	2	The pulse number from the time that 4th axis starts detecting PC signal to PC signal is detected
0	7	3	The pulse number from the time that 5th axis starts detecting PC signal to PC signal is detected
0	7	5	The total times of system history underpressure
0	7	6	The counts that power is low(Power is on)
0	7	7	When power is low,times of not finishing save data (unit:ms)
0	9	0	Nvram data accumulation error times
1	0	0	Spindle encoder 1 detection linear number (Rear cover encoder interface CN21)
1	0	1	Spindle encoder 2 detection linear number (Rear cover encoder interface CN22)
1	0	4	The accumulation times of GDT data verification error along X axis
1	0	5	The accumulation times of GDT data verification error along Z axis
1	0	6	The accumulation times of GDT data verification error along Y axis
1	0	7	The accumulation times of GDT data verification error along 4TH axis
1	0	8	The accumulation times of GDT data verification error along 5TH axis
1	1	0	The accumulation times of GDT data verification error of spindle 1
1	1	1	The accumulation times of GDT data verification error of spindle 3
1	1	2	The accumulation times of GDT data verification error of spindle 4
1	1	3	The accumulation times of GDT data verification error of IO unit
1	1	4	The accumulation times of GDT data verification error of gateway

5.1.3 Diagnosis Keys

DGN.016～DGN.024 are the diagnosis messages of edit keypad keys; DGN.024～DGN.029 are the diagnosis messages of machine operation panel keys. 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.

0	1	6	VF1	RST	O	N	G	7	8	9
key				//	O	N	G	7	8	9

0	1	7	VF2	X	Z	U	W	4	5	6
key				X	Z	U	W	4	5	6

0	1	8	VF3	H/Y	F/E	R/V	D/L	1	2	3
key				H Y	F E	R V	L D	1	2	3

0	1	9	VF4	I/A	J/B	K/C	P/Q	-	0	.
key				I A	J B	K C	P Q	- +	0	. < >

0	2	0	VF5	M	S	T	/、#	BLK	EOB	IN
key				M [	S]	T =	/ * #	␣	EOB	DATA INPUT

0	2	1	VF6	PGU	PGD	CHG	INS/ALT	DEL	CAN	OUT
key						CHANGE	ALTER	DELETE	CANCEL	DATA OUTPUT

0	2	2	VF7	CRU	RIGHT	POS	RPG	OFT	ALM	SET
key						POSITION	PROGRAM	OFFSET	ALARM	SETTING
0	2	3	VF8	CRD	LEFT	PAR	DGN	GRA	PLC	HELP
key						PARAMETER	DIAGNOSIS	GRAPH	PLC	HELP

0	2	4	F1	F2	F3	F4	F5	F6	F7	F8
key										

5.2 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 states.

5.2.1 X Address (Machine→PLC , Defined by Standard PLC Ladders)

X0000	T05	PRES	ESP	DITW	DECX	DIQP	SP	SAGT

T05: tool signal T05

PRES: pressure check input signal

ESP: emergency stop signal

DITW: tailstock control input

DECX: X deceleration signal

DIQP: chuck control input

SP: external pause

SAGT: safety door check signal

X0001	T01	M42I	M41I	ST	DECZ	T08/CHOT	T07/INDX	T06/SELE

T01: tool signal T01

M42I: spindle automatic gear shifting 2-gear in-position signal

M41I: spindle automatic gear shifting 1-gear in-position signal

ST: external cycle start

DECZ: Z deceleration signal

T08/CHOT: tool signal T08/tool post worktable overheat

T07/INDX: tool signal T07/Yantai tool post pregraduation input

T06/SELE: tool signal T06/Yantai tool post strobe signal

X0002		TCP	DEC5	DEC4	DECY	T04	T03	T02

TCP: tool post clamp signal

DEC5: 5th deceleration signal

DEC4: 4th deceleration signal

DECY: Y deceleration signal

T04: tool signal T04

T03: tool signal T03

T02: tool signal T02

X0003	AEZ	AEX	SKIP	NQPJ	WQPJ	LMIZ	LMIY	LMIX

AEZ: G37 skip signal

AEX: G36 skip signal

SKIP: G31 skip signal

NQPJ: inner chuck clamp/outer chuck clamp in-position signal

WQPJ: inner chuck release/outer chuck clamp in-position signal

LMIZ: Z overtravel input

LMIY: Y overtravel input

LMIX: X overtravel input

X0005	***	***	***	***	***	***	***	SALM

SALM: spindle 1 abnormality alarm input

X0006	***	***	×100	×10	×1	ZHAN	YHAN	XHAN

×100: increment ×100（external）

×10: increment×10（external）

×1: increment×1（external）

ZHAN: Z selection（external）

YHAN: Y selection（external）

XHAN: X selection（external）

X0008	***	***	***	***	***	***	***	SALM2

SALM2: the alarm input signal of second spindle

X0009	***	***	***	***	EXESP	***	SP	ST

EXESP: External emergency stop input signal

SP: External pause

ST: External cycle start

5.2.2 Y Address (PLC→Machine, Defined by Standard PLC Ladders)

Y0000	SPZD	SCLP	M05	M04	M03	***	M32	COOL

SPZD: spindle brake

SCLP: spindle clamped

M05: spindle stop

M04: spindle rotation CCW

M03: spindle rotation CW

M32: lubricating output

COOL: cooling output

Y0001	TL-	TL+	DOQPS	DOQPJ	S4/M44	S3/M43	S42/M42	S41/M41

TL-: TL-tool post rotation (CW)

TL+: TL+tool post rotation(CCW)

DOQPS: chuck release

DOQPJ: chuck clamp

S4/M44: spindle mechanical 4-gear

S3/M43: spindle mechanical 3-gear

S2/M42: spindle mechanical 2-gear

S1/M41: spindle mechanical 1-gear

Y0002	***	VP2	DOTWS	DOTWJ	CLPR	CLPG	CLPY	INDXS/TCLP	TZD/TLS

DOTWS: tailstock going backward

DOTWJ: tailstock going forward

CLPR: three-color lamp-red

CLPG: three-color lamp-green

CLPY: three-color lamp-yellow

INDXS/TCLP: Yantai Tool post worktable pregraduation coil /LIO_SHING Tool Post locked

TZD/TLS: Yantai Tool post worktable brake /LIO_SHING Tool Post releasing

Y0003	UO5	UO4	UO3	UO2	UO1	UO0	HPST	***

UO5: user macro output 5

UO4: user macro output 4

UO3: user macro output 3

U2: user macro output 2

U1: user macro output 1

U0: user macro output 0

HPST: hydraulic control output

Y0005	***	***	***	***	***	***	SRV	SFR

SRV: spindle rotation(CW)

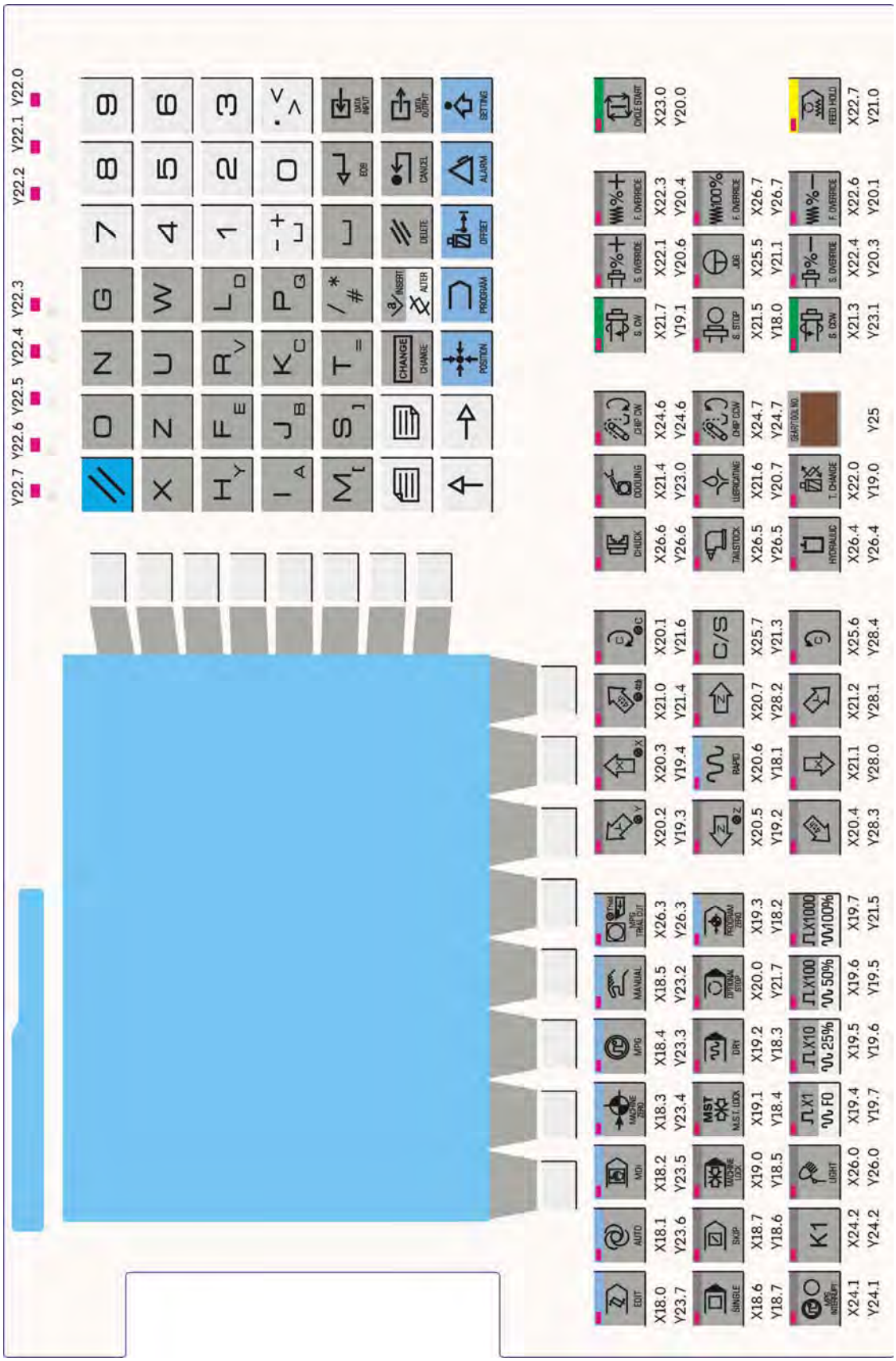
SFR: spindle rotation(CCW)

Y0008	***	***	***	***	***	***	SRV2	SFR2

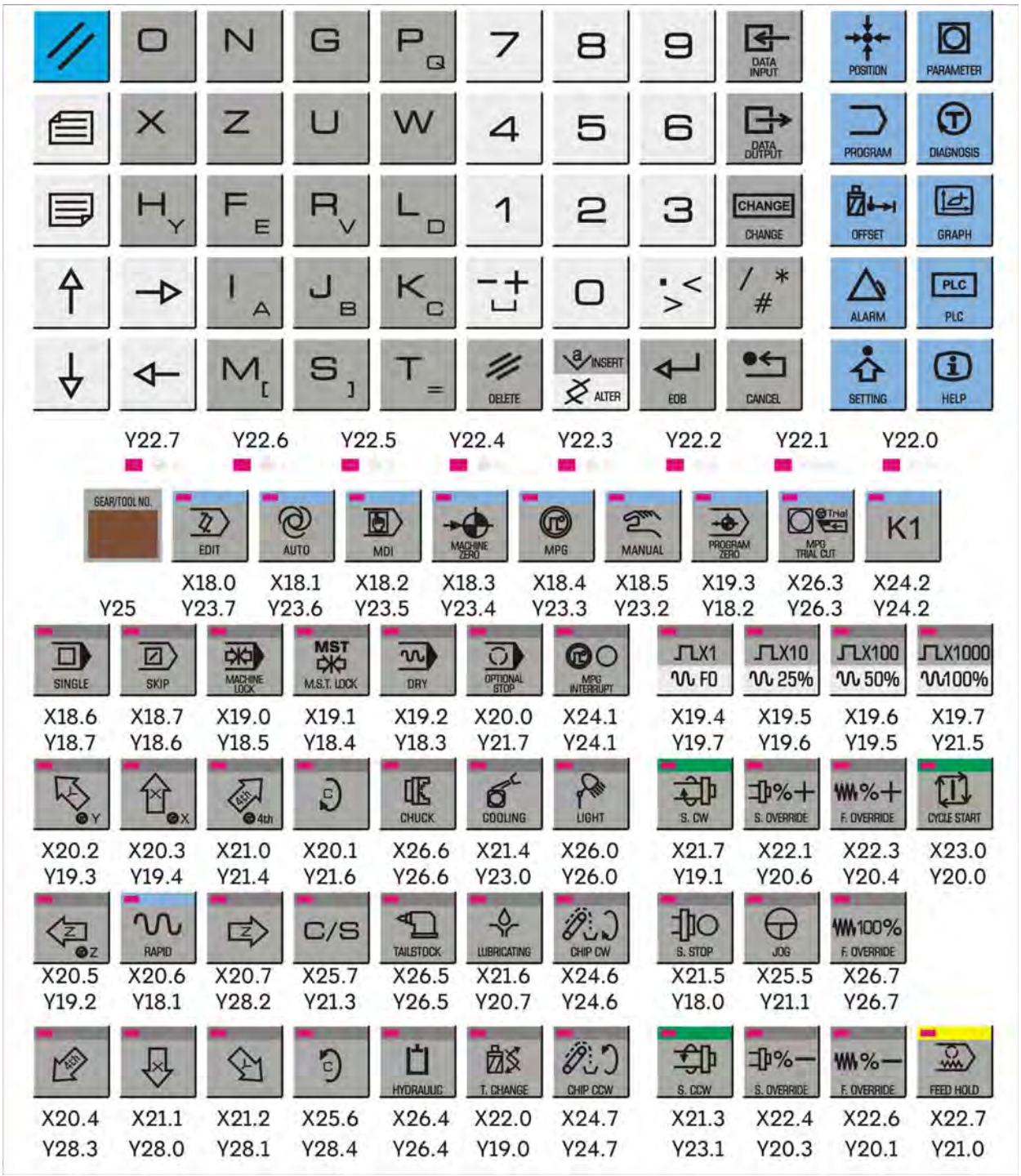
SRV2: second spindle CCW

SFR2: second spindle CW

5.2.3 Machine Panel



GSK980TDi machine panel



GSK980TDi-V machine panel

5.2.4 F Address(CNC→PLC)

F0000	OP	SA	STL	SPL	***	***	***	RWD

OP: Auto run signal

SA: Servo ready signal

STL: Cycle start indicator signal

SPL: Feed hold indicator signal

F0001	MA	TCEN	TAP	ENB	DEN	***	RST	AL

MA: CNC ready signal
TCEN: Tool-change allowable signal
TAP: Tapping signal
ENB: Spindle enable signal
DEN: Designation end signal
RST: Reset signal
AL: Alarm signal

F0002	MDRN	CUT	***	SRNMV	THRD	CSS	RPDO	INCH

MDRN: Dry run detection signal
CUT: Cutting feed signal
SRNMV: Program start signal
THRD: Threading signal
CSS: Constant surface speed signal
RPDO: Rapid feed signal
INCH: Inch input signal

F0003	***	MEDT	MMEM		MMDI	MJ	MH	MINC

MEDT: Memory edit selection detection signal
MMEM: Auto run selection detection signal
MMDI: MDI selection detection signal
MJ: JOG selection detection signal
MH: MPG selection detection signal
MINC: Increment feed detection signal

F0004	***	MPST	MREF	MAFL	MSBK	MABSM	MMLK	MBDT1

MPST: Program beginning return detection signal
MREF: Manual reference return detection signal
MAFL: MST lock detection signal
MSBK: Single block detection signal
MABSM: JOG absolute detection signal
MMLK: All machine axes lock detection signal
MBDT1: Optional block skip detection signal

F0007	***	***	***	***	TF	SF	***	MF

TF: Tool function strobe signal
SF: Spindle speed strobe signal
MF: MST function strobe signal

F0009	DM00	DM01	DM02	DM30	***	***	***	***

DM00: M decoding signal
DM01: M decoding signal
DM02: M decoding signal
DM30: M decoding signal

F0010	MB07	MB06	MB05	MB04	MB03	MB02	MB01	MB00

F0011	MB15	MB14	MB13	MB12	MB11	MB10	MB09	MB08
F0012	MB23	MB22	MB21	MB20	MB19	MB18	MB17	MB16
F0013	MB31	MB30	MB29	MB28	MB27	MB26	MB25	MB24

Miscellaneous function codes MB00~MB31

F0022	SB07	SB06	SB05	SB04	SB03	SB02	SB01	SB00
F0023	SB15	SB14	SB13	SB12	SB11	SB10	SB09	SB08
F0024	SB23	SB22	SB21	SB20	SB19	SB18	SB17	SB16
F0025	SB31	SB30	SB29	SB28	SB27	SB26	SB25	SB24

Spindle speed code signal SB00~SB31

F0026	TB07	TB06	TB05	TB04	TB03	TB02	TB01	TB00
F0027	TB15	TB14	TB13	TB12	TB11	TB10	TB09	TB08
F0028	TB23	TB22	TB21	TB20	TB19	TB18	TB17	TB16
F0029	TB31	TB30	TB29	TB28	TB27	TB26	TB25	TB24

Tool function code signal TB00~TB31

F0036	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
F0037	***	***	***	***	R12O	R11O	R10O	R09O

S12 bit code signal R01O~R12O

F0040	AR7	AR6	AR5	AR4	AR3	AR2	AR1	AR0
F0041	AR15	AR14	AR13	AR12	AR11	AR10	AR9	AR8

Spindle actual speed AR0~AR15

F0044	***	***	***	***	***	***	FSCSL	***

FSCSL: The 1st spindle outline control shifting end signal

F0045	***	RTAPA	PSRA	SARA	TLMA	ORARA	SSTA	ALMA

RTAPA: series spindle 1 rigid tapping signal
PSRA: Series spindle1 position arrival signal
SARA: Series spindle1 speed arrival signal
TLMA: Series spindle1torque arrival signal
ORARA: Series spindle1orientation completion signal
SSTA: Series spindle1zero-speed signal
ALMA: Series spindle1alarm signal

F0049	***	RTAPB	PSRB	SARB	TLMB	ORARB	SSTB	ALMB

RTAPB: series spindle 2 rigid tapping signal
PSRB: Series spindle2position arrival signal
SARB: Series spindle2 speed arrival signal
TLMB: Series spindle2orque arrival signal
ORARB: Series spindle2orientation completion signal
SSTB: Series spindle2zero-speed signal
ALMB: Series spindle2alarm signal

F0053	***	RTAPC	PSRC	SARC	TLMC	ORARC	SSTC	ALMC

RTAPC: series spindle 3 rigid tapping signal
PSRC: Series spindle3 position arrival signal
SARC: Series spindle3 speed arrival signal
TLMC: Series spindle3torque arrival signal
ORARC: Series spindle3orientation completion signal
SSTC: Series spindle3zero-speed signal
ALMC: Series spindle3alarm signal

F0054	UO07	UO06	UO05	UO04	UO03	UO02	UO01	UO00
F0055	UO15	UO14	UO13	UO12	UO11	UO10	UO09	UO08

Macro output signal UO00~UO15

F0056	U107	U106	U105	U104	U103	U102	U101	U100
F0057	U115	U114	U113	U112	U111	U110	U109	U108
F0058	U123	U122	U121	U120	U119	U118	U117	U116
F0059	U131	U130	U129	U128	U127	U126	U125	U124

Macro output signal U100~131

F0064	***	***	***	***	***	***	TLNW	TLCH

TLNW: New tool selection signal (for tool life management)
TLCH: Tool change signal(for tool life management)

F0065	***	***	***	***	***	***	RGSPM	RGSPP

RGSPM: Spindle CCW rotation (in rigid tapping)
RGSPP: Spindle CW rotation (in rigid tapping)

F0076	***	***	***	***	RTAP	***	***	***

RTAP: rigid tapping method signal

F0091	***	***	***	***	MMMOD	***	***	***

MMMOD: Mode signal; it informs system of the manual trial cut mode;

F0094	***	***	***	ZP5	ZP4	ZP3	ZP2	ZP1

ZP5~ZP1: Reference point return end signal ZP5~ZP1

F0096	***	***	***	ZP25	ZP24	ZP23	ZP22	ZP21

ZP25~ZP21: The 2nd Reference point return end signal ZP25~ZP21

F0098	***	***	***	ZP35	ZP34	ZP33	ZP32	ZP31

ZP35~ZP31: The 3rd Reference point return end signal ZP35~ZP31

F0100	***	***	***	ZP45	ZP44	ZP43	ZP42	ZP41

ZP45~ZP41: The 4th Reference point return end signal ZP45~ZP41

F0102	***	***	***	MV5	MV4	MV3	MV2	MV1

MV5~MV1: Axis move signal MV5~MV1

F0106	***	***	***	MVD5	MVD4	MVD3	MVD2	MVD1

MVD5~MVD1 Axis move direction signal MVD5~MVD1

F0120	***	***	***	ZRF5	ZRF4	ZRF3	ZRF2	ZRF1

ZRF5~ZRF1: Reference point creation signal ZRF5~ZRF1

F0127	***	***	***	ABLE5	ABLE4	ABLE3	ABLE2	ABLE1

ABLE5~ ABLE1: axis active signal ABLE5~ ABLE1

F0129	***	***	***	***	***	***	***	KEAXSL

KEAXSL: PLC axis control status selection signal

F0130	EBSYA	***	***	EGENA	EDENA	***	***	***

EBSYA: PLC axis control group A command read completed signal
EGENA: PLC axis control group A axis movement signal
EDENA: PLC axis control group A auxiliary function implement signal

F0131	***	***	***	***	***	***	EABUFA	EMFA

EABUFA: PLC axis control group A buffer full signal
EMFA: PLC axis control group A auxiliary function strobe signal

F0132	EM28A	EM24A	EM22A	EM21A	EM18A	EM14A	EM12A	EM11A

PLC axis control group A auxiliary function code signals EM11A~EM28A

F0133	EBSYB	***	***	EGENB	EDENB	***	***	***

EBSYB: PLC axis control group B command read completed signal
EGENB: PLC axis control group B axis movement signal
EDENB: PLC axis control group B auxiliary function implement signal

F0134	***	***	***	***	***	***	EABUFB	EMFB

EABUFB: PLC axis control group B buffer full signal
EMFB: PLC axis control group B auxiliary function strobe signal

F0135	EM28B	EM24B	EM22B	EM21B	EM18B	EM14B	EM12B	EM11B

PLC axis control group A auxiliary function code signals EM11B~EM28B

F0136	EBSYC	***	***	EGENC	EDENC	***	***	***

EBSYC: PLC axis control group C command read completed signal
EGENC: PLC axis control group C axis movement signal
EDENC: PLC axis control group C auxiliary function implement signal

F0137	***	***	***	***	***	***	EABUFC	EMFC

EABUFC: PLC axis control group C buffer full signal
EMFC: PLC axis control group C auxiliary function strobe signal

F0138	EM28C	EM24C	EM22C	EM21C	EM18C	EM14C	EM12C	EM11C

PLC axis control group A auxiliary function code signals EM11C~EM28C

F0139	EBSYD	***	***	EGEND	EDEND	***	***	***

EBSYD: PLC axis control group D command read completed signal
EGEND: PLC axis control group D axis movement signal
EDEND: PLC axis control group D auxiliary implement signal

F0140	***	***	***	***	***	***	EABUFD	EMFD

EABUFD: PLC axis control group D buffer full signal
EMFD: PLC axis control group D auxiliary function strobe signal

F0141	EM28D	EM24D	EM22D	EM21D	EM18D	EM14D	EM12D	EM11D

PLC axis control group A auxiliary function code signals EM11D~EM28D

F0142	EM48A	EM44A	EM42A	EM41A	EM38A	EM34A	EM32A	EM31A

PLC axis control group A auxiliary function code signals EM31A~EM48A

Fn144	***	***	***	***	***	***	FSCSL2	***

FSCSL2: The 2nd spindle outline control shifting end signal

F0145	EM48B	EM44B	EM42B	EM41B	EM38B	EM34B	EM32B	EM31B

PLC axis control group B auxiliary function code signals EM3BA~EM48B

F0148	EM48C	EM44C	EM42C	EM41C	EM38C	EM34C	EM32C	EM31C

PLC axis control group C auxiliary function code signals EM31C~EM48C

F0151	EM48D	EM44D	EM42D	EM41D	EM38D	EM34D	EM32D	EM31D

PLC axis control group D auxiliary function code EM31D~EM48D

F0182	***	***	***	EACNT5	EACNT4	EACNT3	EACNT2	EACNT1

EACNT5: PLC axis control 5th axis control signal
EACNT4: PLC axis control 4th axis control signal
EACNT3: PLC axis control Y axis control signal
EACNT2: PLC axis control Z axis control signal
EACNT1: PLC axis control X axis control signal

F0198	***	***	***	PRO5	PRO4	PRO3	PRO2	PRO1

PRO4: Program zero return end signal PRO5
PRO4: Program zero return end signal PRO4
PRO3: Program zero return end signal PRO3
PRO2: Program zero return end signal PRO2
PRO1: Program zero return end signal PRO1

F0199	***	***	***	***	***	***	***	MSPHD

MSPHD: Spindle JOG detection signal

F0200	***	***	***	SIMSPL	***	***	***	***

SIMSPL: Analog spindle active

F0201	***	***	DECI	***	***	DCS	***	***

DECI: DEC signal level selection in reference return

DCS: If OUT key may start program in MDI mode

F0205	***	***	***	***	***	ZNIK	***	***

ZNIK: Axis move key hold active in zero return

F0207	TN7	TN6	TN5	TN4	TN3	TN2	TN1	TN0

total tools number

F0210	***	MST	MSP	***	MESP	***	SINC	SOVI

MST: Shield external cycle start signal

MSP: Shield external feed hold signal

MESP: Shield external emergency stop signal

F0212	***	***	***	COVL	CINP	CDWL	***	***

COVL: Bit check being executed

CINP: G04 code being executed

CDWL: RS232 interface being receiving

F0213	***	***	***	***	***	***	CTRD	CTPU

CTRD: RS232 interface being transmitting

CTPU: Spindle jog/lubricating selection

F0214	***	***	***	VAL5	VAL4	VALY	ZVAL	XVAL

VAL5: 5th axis direction selection

VAL4: 4th axis direction selection

VALY: Y direction selection

VALZ: Z direction selection

VALX: X direction selection

F0215	SGR07	SGR06	SGR05	SGR04	SGR03	SGR02	SGR01	SGR00
F0216	***	***	***	***	SGR11	SGR10	SGR09	SGR08

Speed command signal for gear shift SGR00~ SGR11

F0240	INP07	INP06	INP05	INP04	INP03	INP02	INP01	INP00
F0241	INP15	INP14	INP13	INP12	INP11	INP10	INP09	INP08

Input speed command of spindle INP00~ INP15

F0250	SYS7	SYS6	SYS5	SYS4	SYS3	SYS2	SYS1	SYS0

system type 0:980TDi 1:980TDi-V

5.2.5 G Address (PLC→CNC)

G0004	***	***	***	***	**FIN**	***	***	***

FIN: MST function end signal

G0005	***	**AFL**	***	***	***	***	***	***

AFL: MST lock signal

SFIN: Spindle S function end signal

G0006	***	**SKIPP**	***	**OVC**	***	**ABSM**	***	**SRN**

OVC: Feedrate override cancel signal

SRN: Program restart signal

G0007	**RLSOT**	**EXLM**	***	***	***	**ST**	**STLK**	***

ST: Cycle start signal

G0008	**ERS**	**RRW**	**SP**	**ESP**	***	***	***	**IT**

ERS: External reset signal

RRW: Reset and cursor return signal

SP: Feed hold signal

ESP: Emergency stop signal

G0010	JV07	JV06	JV05	JV04	JV03	JV02	JV01	JV00
G0011	JV15	JV14	JV13	JV12	JV11	JV10	JV09	JV08

JOG override signal JV00~JV15

G0012	FV07	FV06	FV05	FV04	FV03	FV02	FV01	FV00

Feedrate override signal FV00~FV07

G0014	***	***	***	***	***	***	**RV2**	**RV1**

RV2: Rapid feedrate override signal RV2

RV1: Rapid feedrate override signal RV1

G0018	H2D	H2C	H2B	H2A	H1D	H1C	H1B	H1A

H2D: MPG 2 feed selection signal (01015th)

H2C MPG 2 feed selection signal (01004th)

H2B: MPG 2 feed selection signal (0011Y)

H2A: MPG 2 feed selection signal (0001X. 0010Z)

H1D: MPG 1 feed selection signal (01015th)

H1C MPG 1 feed selection signal (01014th)

H1B: MPG 1 feed selection signal (0011Y)

H1A: MPG 1 feed selection signal (0001X. 0010Z)

G0019	**RT**	***	**MP2**	**MP1**	***	***	***	***

RT: Manual rapid feed selection signal

MP2: MPG override signal MP2

MP1: MPG override signal MP1

G0027	CON	***	SSTP3	SSTP2	SSTP1	***	SWS2	SWS1

CON: Cs contouring control switch signal
SSTP3: 3rd spindle stop signal (for multi-spindle function)
SSTP2: 2nd spindle stop signal (for multi-spindle function)
SSTP1: 1st spindle stop signal (for multi-spindle function)
SWS2: Multi-spindle selection signal 2
SWS1: Multi-spindle selection signal 1

G0028	***	***	***	***	***	GR2	GR1	***

GR2: Gear selection signal
GR1: Gear selection signal

G0029	***	KSSTP	***	SAR	GR32	GR31	GR22	GR21

KSSTP: spindle stop signal
SAR: spindle speed arrival
GR32: 3rd spindle gear selection signal 2 (for multi-spindle function)
GR31: 3rd spindle gear selection signal 1 (for multi-spindle function)
GR22: 2nd spindle gear selection signal 2 (for multi-spindle function)
GR21: 2nd spindle gear selection signal 1 (for multi-spindle function)

G0030	SOV7	SOV6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0

SOV7: 2nd Spindle override signal SOV7
SOV6: 2nd Spindle override signal SOV6
SOV5: 2nd Spindle override signal SOV5
SOV4: 2nd Spindle override signal SOV4
SOV3: 1st Spindle override signal SOV3
SOV2: 1st Spindle override signal SOV2
SOV1: 1st Spindle override signal SOV1
SOV0: 1st Spindle override signal SOV0

G0032	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

G0033	SIND	***	***	***	R12I	R11I	R10I	R09I

SIND: Spindle motor speed code selection signal
R12I: Spindle motor speed code signal R12I
R11I: Spindle motor speed code signal R11I
R10I: Spindle motor speed code signal R10I

R09I: Spindle motor speed code signal R09I

G0039	GOQSM	WOQSM	***	***	***	***	***	***

GOQSM: Tool offset writing mode select signal

WOQSM: Workpiece coordinate system offset writing mode select signal

G0040	WOSET	PRC	***	***	***	***	***	***

WOSET: Workpiece coordinate system offset writing signal

PRC: Absolute coordinate postion record signal

G0043	**ZRN**	***	**DNC1**	***	***	**MD4**	**MD2**	**MD1**

ZRN: Current mode selection 4

DNC1: DNC selection signal

MD4: Current mode selection 3

MD2: Current mode selection 2

MD1: Current mode selection 1

G0044	**HDT**	***	***	***	***	***	**MLK**	**BDT**

HDT: Manual tool change signal

MLK: Machine lock signal(PLC→CNC)

BDT: Block skip signal (PLC→CNC)

G0046	**DRN**	***	***	***	**KEY1**	***	**SBK**	***

DRN: Dry run signal

KEY1: Memory protection signal

SBK: Single block signal (PLC→CNC)

G0047	TL128	TL64	TL32	TL16	TL08	TL04	TL02	TL01

Tool group number select signal TL01~TL128

G0048	TLRST	***	TLSKP	***	***	***	***	***

TLRST: tool change reset signal

TLSKP: tool skip signal

G0054	**UI07**	**UI06**	**UI05**	**UI04**	**UI03**	**UI02**	**UI01**	**UI00**
G0055	**UI15**	**UI14**	**UI13**	**UI12**	**UI11**	**UI10**	**UI09**	**UI08**

UI00~UI15: Macro input signal UI00~UI15

G0061	*****	*****	**RGTSP2**	**RGTSP1**	*****	*****	*****	**RGTAP**

RGTAP: Rigid tapping signal

RGTSP1: Spindle selection in rigid tapping RGTSP1

RGTSP2: Spindle selection in rigid tapping RGTSP2

G0070	HLVA	BREFA	SFRA	SRVA	TAPA	ORCMA	ZSLA	ARSTA

HLVA: series spindle 1 high/low speed shifting signal

BREFA: series spindle 1 mechanical locking signal

SFRA: series spindle 1 rotation signal CW(positive)

SRVA: series spindle 1 rotation signal CCW(negative)
TAPA: series spindle 1 rigid tapping signal
ORCMA: series spindle 1 orientation start signal
ZSLA: series spindle 1 zero clamping signal
ARSTA: series spindle 1 alarm cleaning signal

G0074	HLVB	BREFB	SFRB	SRVB	TAPB	ORCMB	ZSLB	ARSTB

HLVB: series spindle 1 high/low speed shifting signal
BREFB: series spindle 1 mechanical locking signal
SFRB: series spindle 1 rotation signal CW(positive)
SRVBA: series spindle 1 rotation signal CCW(negative)
TAPB: series spindle 1 rigid tapping signal
ORCMB: series spindle 1 orientation start signal
ZSLB: series spindle 1 zero clamping signal
ARSTB: series spindle 1 alarm cleaning signal

G0078	HLVC	BREFC	SFRC	SRVC	TAPC	ORCMC	ZSLC	ARSTC

HLVC: series spindle 1 high/low speed shifting signal
BREFC: series spindle 1 mechanical locking signal
SFRC: series spindle 1 rotation signal CW (positive)
SRVC: series spindle 1 rotation signal CCW (negative)
TAPC: series spindle 1 rigid tapping signal
ORCMC: series spindle 1 orientation start signal
ZSLC: series spindle 1 zero clamping signal
ARSTC: series spindle 1 alarm cleaning signal

G0100	***	***	***	**+J5**	**+J4**	**+J3**	**+J2**	**+J1**

+J1: Feed axis and direction selection signal +J1
+J2: Feed axis and direction selection signal +J2
+J3: Feed axis and direction selection signal +J3
+J4: Feed axis and direction selection signal +J4
+J5: Feed axis and direction selection signal +J5

G0102	***	***	***	**-J5**	**-J4**	**-J3**	**-J2**	**-J1**

-J1: Feed axis and direction selection signal -J1
-J2: Feed axis and direction selection signal -J2
-J3: Feed axis and direction selection signal -J3
-J4: Feed axis and direction selection signal -J4
-J5: Feed axis and direction selection signal –J5

G0114	***	***	***	**+L5**	**+L4**	**+L3**	**+L2**	**+L1**

+L1: Axis overtravel signal +L1
+L2: Axis overtravel signal +L2
+L3: Axis overtravel signal +L3
+L4: Axis overtravel signal +L4
+L5: Axis overtravel signal +L5

G0116	***	***	***	-L5	-L4	-L3	-L2	-L1

-L1: Axis overtravel signal -L1
-L2: Axis overtravel signal -L2
-L3: Axis overtravel signal -L3
-L4: Axis overtravel signal -L4
-L5: Axis overtravel signal –L5

G0136	***	***	***	EAX5	EAX4	EAX3	EAX2	EAX1

EAX5: Axis 5th select signal of PLC control
EAX4: Axis 4th select signal of PLC control
EAX3: Axis Y select signal of PLC control
EAX2: Axis Z select signal of PLC control
EAX1: Axis X select signal of PLC control

G0141	***	***	***	***	EABSD	EABSC	EABSB	EABSA

EABSD: PLC control group D is absolute (1)/relative (0) coordinate
EABSC: PLC control group C is absolute (1)/relative (0) coordinate
EABSB: PLC control group B is absolute (1)/relative (0) coordinate
EABSA: PLC control group A is absolute (1)/relative (0) coordinate

G0142	EBUFA	ECLRA	ESTPA	ESOFA	ESBKA	EMBUFA	***	EFINA

EBUFA: Instruction reading signal of PLC control group A
ECLRA: Reset signal of PLC control group A
ESTPA: Axis pause signal of PLC control group A
ESOFA: Servo off signal of PLC control group A
ESBKA: Single block stop signal of PLC control group A
EMBUFA: Buffer prohibition signal of PLC control group A
EFINA: Auxiliary function finish signal of PLC control group A

G0143	EC7A	EC6A	EC5A	EC4A	EC3A	EC2A	EC1A	EC0A

Control instruction signal of PLC control group A

G0144	EIF7A	EIF6A	EIF5A	EIF4A	EIF3A	EIF2A	EIF1A	EIF0A
G0145	EIF15A	EIF14A	EIF13A	EIF12A	EIF11A	EIF10A	EIF9A	EIF8A

Feed speed signal of PLC control group A

G0146	EID7A	EID6A	EID5A	EID4A	EID3A	EID2A	EID1A	EID0A
G0147	EID15A	EID14A	EID13A	EID12A	EID11A	EID10A	EID9A	EID8A
G0148	EID23A	EID22A	EID21A	EID20A	EID19A	EID18A	EID17A	EID16A
G0149	EID31A	EID30A	EID29A	EID28A	EID27A	EID26A	EID25A	EID24A

Data signal of PLC control group A

G0150	***	***	***	***	***	***	ROV2E	ROV1E

ROV2E: Rapid ratio signal of PLC control
ROV1E: Rapid ratio signal of PLC control

G0151	FV7E	FV6E	FV5E	FV4E	FV3E	FV2E	FV1E	FV0E

Feed ratio signal of PLC control

G0154	EBUFB	ECLRB	ESTPB	ESOFB	ESBKB	EMBUFB	***	EFINB

EBUFB: Instruction reading signal of PLC control group B
ECLRB: Reset signal of PLC control group B
ESTPB: Axis pause signal of PLC control group B
ESOFB: Servo off signal of PLC control group B
ESBKB: Single block stop signal of PLC control group B
EMBUFB: Buffer prohibition signal of PLC control group B
EFINB: Auxiliary function finish signal of PLC control group B

G0155	EC7B	EC6B	EC5B	EC4B	EC3B	EC2B	EC1B	EC0B

Control instruction signal of PLC control group B

G0156	EIF7B	EIF6B	EIF5B	EIF4B	EIF3B	EIF2B	EIF1B	EIF0B
G0157	EIF15B	EIF14B	EIF13B	EIF12B	EIF11B	EIF10B	EIF9B	EIF8B

Feed speed signal of PLC control group B

G0158	EID7B	EID6B	EID5B	EID4B	EID3B	EID2B	EID1B	EID0B
G0159	EID15B	EID14B	EID13B	EID12B	EID11B	EID10B	EID9B	EID8B
G0160	EID23B	EID22B	EID21B	EID20B	EID19B	EID18B	EID17B	EID16B
G0161	EID31B	EID30B	EID29B	EID28B	EID27B	EID26B	EID25B	EID24B

Data signal of PLC control group B

G0166	EBUFC	ECLRC	ESTPC	ESOFC	ESBKC	EMBUFC	***	EFINC

EBUFC: Instruction reading signal of PLC control group C
ECLRC: Reset signal of PLC control group C
ESTPC: Axis pause signal of PLC control group C
ESOFC: Servo off signal of PLC control group C
ESBKC: Single block stop signal of PLC control group C
EMBUFC: Buffer prohibition signal of PLC control group C
EFINC: Auxiliary function finish signal of PLC control group C

G0167	EC7C	EC6C	EC5C	EC4C	EC3C	EC2C	EC1C	EC0C

Control instruction signal of PLC control group C

G0168	EIF7C	EIF6C	EIF5C	EIF4C	EIF3C	EIF2C	EIF1C	EIF0C
G0169	EIF15C	EIF14C	EIF13C	EIF12C	EIF11C	EIF10C	EIF9C	EIF8C

Feed speed signal of PLC control group C

G0170	EID7C	EID6C	EID5C	EID4C	EID3C	EID2C	EID1C	EID0C
G0171	EID15C	EID14C	EID13C	EID12C	EID11C	EID10C	EID9C	EID8C
G0172	EID23C	EID22C	EID21C	EID20C	EID19C	EID18C	EID17C	EID16C
G0173	EID31C	EID30C	EID29C	EID28C	EID27C	EID26C	EID25C	EID24C

Data signal of PLC control group C

G0178	EBUFD	ECLRD	ESTPD	ESOFD	ESBKD	EMBUFD	***	EFIND

EBUFD: Instruction reading signal of PLC control group D
ECLRD: Reset signal of PLC control group D
ESTPD: Axis pause signal of PLC control group D
ESOFD: Servo off signal of PLC control group D
ESBKD: Single block stop signal of PLC control group D
EMBUFD: Buffer prohibition signal of PLC control group D
EFIND: Auxiliary function finish signal of PLC control group D

G0179	EC7D	EC6D	EC5D	EC4D	EC3D	EC2D	EC1D	EC0D

Control instruction signal of PLC control group D

G0180	EIF7D	EIF6D	EIF5D	EIF4D	EIF3D	EIF2D	EIF1D	EIF0D
G0181	EIF15D	EIF14D	EIF13D	EIF12D	EIF11D	EIF10D	EIF9D	EIF8D

Feed speed signal of PLC control group D

G0182	EID7D	EID6D	EID5D	EID4D	EID3D	EID2D	EID1D	EID0D
G0183	EID15D	EID14D	EID13D	EID12D	EID11D	EID10D	EID9D	EID8D
G0184	EID23D	EID22D	EID21D	EID20D	EID19D	EID18D	EID17D	EID16D
G0185	EID31D	EID30D	EID29D	EID28D	EID27D	EID26D	EID25D	EID24D

Data signal of PLC control group D

G0200	***	***	***	***	***	***	***	**SPD**

SPD Spindle jog signal

G0201	**NT07**	**NT06**	**NT05**	**NT04**	**NT03**	**NT02**	**NT01**	**NT00**

NT00: Current tool No. NT00
NT01: Current tool No. NT01
NT02: Current tool No. NT02
NT03: Current tool No. NT03
NT04: Current tool No. NT04
NT05: Current tool No. NT05
NT06: Current tool No. NT06
NT07: Current tool No. NT07

G0202	**NT15**	**NT14**	**NT13**	**NT12**	**NT11**	**NT10**	**NT09**	**NT08**

NT08: Current tool No. NT08
NT09: Current tool No. NT09
NT10: Current tool No. NT10
NT11: Current tool No. NT11
NT12: Current tool No. NT12
NT13: Current tool No. NT13
NT14: Current tool No. NT14
NT15: Current tool No. NT15

G0203	**NT23**	**NT22**	**NT21**	**NT20**	**NT19**	**NT18**	**NT17**	**NT16**

NT16: Current tool No. NT16
NT17: Current tool No. NT17

NT18: Current tool No. NT18
NT19: Current tool No. NT19
NT20: Current tool No. NT20
NT21: Current tool No. NT21
NT22: Current tool No. NT22
NT23: Current tool No. NT23

G0204		NT31	NT30	NT29	NT28	NT27	NT26	NT25	NT24

NT24: Current tool No. NT24
NT25: Current tool No. NT25
NT26: Current tool No. NT26
NT27: Current tool No. NT27
NT28: Current tool No. NT28
NT29: Current tool No. NT29
NT30: Current tool No. NT30
NT31: Current tool No. NT31

5.2.6 Address A (Message Display Requiery Signal, Defined by Standard PLC Ladders)

Address	Alarm No.	Content
A0000.0	1000	Tool change too long
A0000.1	1001	Tool post not in-position alarm as tool change ends
A0000.2	1002	Tool change unfinished alarm
A0000.3	1003	Tool post clamping signal not received
A0000.4	1004	Recheck clamping signal, and clamping signal inactive as tool change ends
A0000.5	1005	Tool change execution is mistaken before power off
A0000.6	1006	Pre-indexing proximity switch not arrived
A0000.7	1007	Tool post overheat alarm
A0001.0	1008	M10 and M11 codes disabled for tailstock function inactive
A0001.1	1009	Run-out disabled in spindle running
A0001.3	1011	Spindle start unallowed as tailstock advancing not detected
A0001.4	1012	Cycle unallowed for cycle start disabled
A0001.5	1013	Spindle start unallowed for spindle enable off
A0001.6	1014	Confirm the tools of tool post (8, 10, 12 tools)
A0002.0	1014	Please reset BIT1, BIT0 of parameter 168 for tool post model
A0002.1	1016	Alarm of safe door not closed
A0002.3	1017	Alarm of chuck low pressure
A0002.4	1019	Chuck released unallowed in spindle running
A0002.5	1020	Clamping in-position signal inactive alarm in spindle running
A0002.6	1021	Spindle start unallowed if chuck clamping in-position signal inactive
A0003.0	1022	Spindle start unallowed for chuck releasing
A0003.0	1024	M12/M13 code disabled as chuck inactive
A0003.1	1025	Has not checked the chuck clamped/released start signal
A0003.3	1027	The tool pot is not released

Address	Alarm No.	Content
A0003.4	1028	Has not found the target tool
A0003.5	1029	Has not received tool pot stop and locking start signal
A0003.7	1031	Total tools is more than 4, and the external override cannot be connected
A0004.1	1032	Illegal M code
A0004.2	1033	Spindle jog disabled in non-analog spindle mode
A0004.3	1035	M63, M64 designation error
A0004.4	1036	Spindle gear change time is too long
A0004.5	1037	Spindle speed/position control switch time is too long
A0004.6	1038	The 2nd spindle speed/position control switch time is too long
A0004.7	1039	Spindle position control does not permit the orientation
A0005.0	1040	Spindle orientation time is too long
A0005.1	1041	Alarm for the abnormal spindle servo or frequency converter for abnormality
A0005.2	1042	Alarm for the abnormal 2nd spindle servo or frequency converter
A0005.3	1043	Forbid clamping the spindle when the spindle rotating or feeding
A0005.4	1044	Spindle is clamped not to rotate or feed
A0007.0	2000	Hydraulic motor is not started
A0007.1	2001	Safety door has been opened
A0007.3	2003	Alarm for the tool pot unclocked
A0007.5	2005	Feedrate override is fixed to 100％,can't be adjusted
A0007.6	2006	The panel override is disabled when the external override is enabled.
A0007.7	2007	The program can not be operated due to the external dwell switch is cut off.
A0008.0	2008	The axis can not be moved due to the manual override is 0.
A0008.3	2011	Machining numbers are reached to the setting one.

CHAPTER 6 MEMORIZING PITCH ERROR COMPENSATION

6.1 Function Description

There are more or less precision errors in the pitch of machine axes lead screw, and it will definitely affect the parts machining precision. This GSK980TDi CNC system has the memorizing pitch error offset function that it can accurately compensate the pitch error of the lead screw.

6.2 Specification

1) The set offset amount is concerned with the offset origin, offset intervals etc.;
2) Pitch error offset value is get by searching the table about machine coordinates and pitch error compensation origin;
3) Points to be compensated: 1200 points for 5 axes, each axis confirms point range by parameter setting;
4) Axis compensated: X, Y, Z ,4th,5th axis
5) Compensation range: -2550～2550×least input increment;
6) Compensation interval: 10000～9999999××least input increment;
7) Compensation amount of compensation point N(N=0,1,2,3,…1023) is determined by the mechanical error between point N and point N-1;
8) The setting is the same as the CNC parameters input, see Ⅱ **Operation.**

6.3 Parameter Setting

6.3.1 Pitch Compensation

Bit parameter

0	0	3				PCOMP					

Bit5=1: Pitch error offset active;

Bit5=0: Pitch error offset inactive;

6.3.2 Pitch Error Compensation Number

Set the maximum pitch error compensation number and the minimum pitch error compensation number of each axis through parameters No.096 and No.097 so that the compensation range can be determined. In an offset value list, the number that corresponds to machine zero point is called pitch error compensation origin (reference point). The pitch error compensation origin is set by parameter No. 098. It can be any value within the compensation range according to mechanical requirement.

Data parameter

0	9	6	PECMINn

The maximum pitch error compensation number

0	9	7	PECMAXn

The minimum pitch error compensation number

0	9	8	PECORGn

The pitch error compensation number that corresponds to machine zero point

6.3.3 Pitch error compensation interval

0	9	9	PECINTn

［Meaning］ Interval between pitch error compensation point

［Unit］

Setting Unit	IS-B	IS-C
Metric machine	0.001 mm	0.0001 mm
Inch machine	0.0001 inch	0.00001 inch

［Range］

	IS-B	IS-C
Metric machine	1000～999999	10000～9999999
Inch machine	400～999999	4000～9999999

6.3.4 Offset Value

Pitch error offset amount of each axis is set according to the following table, and offset amount is input in diameter value without relating to diameter programming or radius programming, and the input value unit is the least input increment.

Offset No.	Offset value	Z	Offset value
000	0	0200	0
001	5	0201	-2
002	-3	0202	4
…	…	…	…
…	…	…	…

6.4 Notes of Offset Setting

①The setting and alteration of pitch offset can only be done at the authority of password level 2.

②After the parameter of pitch offset is set, only the machine zero is returned could the offset be done.

6.5 Setting Examples of Offset Parameters

Assume that the pitch error compensation range is 0~255.

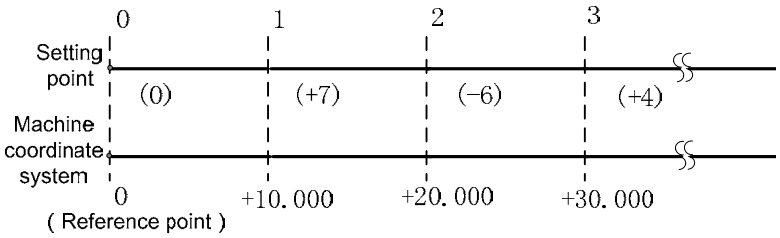
① Data parameter No.098 (pitch error origin) =0, Data parameter No.099 (offset interval)=10.000

When the pitch error origin is set to 0:

The offset value for the 1st segment is set by the position No.001 in the offset table, the offset value for the 2nd segment is set by the position No.002 in the offset table, and the offset value for the Nth segment is set by the position No.000+N in the offset table.

The machine zero is regarded as the reference point of pitch error origin, it begins to compensate

the position No.001 in the offset table from the machine zero. So the pitch error offset can only be done in the positive moving direction of the machine coordinate system.



The position No.000 in the offset table corresponds to the reference point (i.e. pitch error origin 0), the offset point 1 corresponds to a point of positive 10.000 moving from this reference point, and So there is a offset point every 10.000 mm. l. At offset point N, set an offset value which is the compensation amount by moving from (N-1) × (offset interval) to N×(offset interval).

The following table content is resulted from the above:

Machine coordinate system	Offset parameter No.	Offset value	Drive unit current code pulses before compensating	Drive unit current command pulses after compensating
Reference point 0	000	000	00000	00000
10.000	001	7	10000	10007
20.000	002	-6	20000	20001
30.000	003	4	30000	30005
……	004	…		

Actually the machine moves from the reference point to the point of +30.000, the pitch error offset is: (+7) + (-6) + (+4) =+5

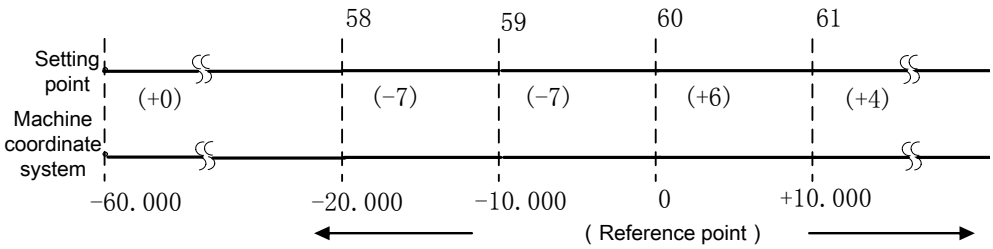
②Data parameter No.098 (pitch error origin) =60, No.099 (offset interval) =10.000

When the pitch error origin is set to 60:

For the positive moving, the 1st segment error offset is set by position No.061 in the offset table, the 2nd segment by position No.062. The Nth segment error offset is set by position No.060+N in the offset table.

For the negative moving, the 1st segment error offset is set by position No.060 in the offset table, the 2nd segment by position No.059. The Nth segment error offset is set by position No.060-N in the offset table.

The machine zero is taken as the reference point of the pitch error origin, when moving from the positive of machine coordinate system, it begins to compensate by the corresponding position No.061 in the offset table; when moving from the negative of the machine coordinate system, it begins to compensate by the position No.060. Therefore the pitch offset can be done when moving in the positive or the negative axis of machine coordinate system.



The position No.060 in the offset table corresponds to the reference point, No.061value in the

compensation table is a compensation amount by executing a positive movement 10.000mm from origin. So there is an offset point every 10.000. No.061value in the compensation table is a compensation amount by executing a negative movement 10.000mm from origin. So there is an offset point every -10.000. No.59 is a compensation amount at -10.000~-20.000mm. Therefore the point N is set by an offset of move from (N-61) × (offset interval) to (N-60) × (offset interval).

The following table content is resulted from the above:

Machine coordinate system	Offset parameter No.	Offset value	Drive unit current code pulses before compensating	Drive unit current code pulses after compensating
-30.000	058	-7	-30000	-29992
-20.000	059	-7	-20000	-19999
-10.000	060	+6	-10000	-10006
Reference point 0			0	0
10.000	061	+4	10000	10004
……	062	…		

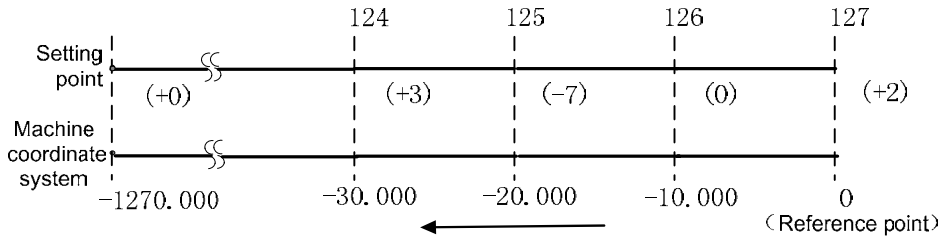
Actually as the machine moves from point -30.000 to the point +10.000, the pitch error offset is: (-7)+(-7)+(+6)+(+4)=(-4)

③Data parameter No.098 (pitch error origin) =127, No.099 (offset interval) =10.000

When the pitch error origin is set to 255:

The offset value for the 1st segment is set by the position No.255 in the offset table, the offset value for the 2nd segment is set by the position No.254 in the offset table, and the offset value for the Nth segment is set by the position No.256-N in the offset table.

The machine zero is regarded as the reference point of pitch error origin, the system compensates the compensate amount set by No.255 in the offset table from the machine zero to the negative direction. So the pitch error compensation can be executed when the machine coordinates are less than 0.



The position No.255 is an offset value from reference point to -10.000mm, No.254 value in the table is the one from -10.000mm to -20.000mm, so there is an offset point every -10.000. Therefore the point N is set by an offset of move from (N-256)×(offset interval) to (N-255)×(offset interval).

Above is the example of the following offset interval error.

Machine coordinate system	Offset parameter number	Offset value	Drive unit current code pulses before compensating	Drive unit current code pulses after compensating
Reference point 0			0	0
-10.000	127	2	10000	10002
-20.000	126	0	20000	20002
-30.000	125	-7	30000	29995
-40.000	124	3	40000	39998

Actually the machine moves from the point -40.000 to the reference point, the pitch error offset is: (+3) + (-7) + (0) + (+2) = (-2).

6.5.1 Rotation Axis Pitch Error Compensation

Addition axis is the linear or rotation axis (Type B), the pitch error compensation method is identical with the common linear axis. The pitch error compensation function in which the addition axis is regarded as the rotation axis (Type A); refer to the following explanations:

- Movement value per revoltion: 360°
- Pitch error position interval: 45°
- The compensation position number of reference point: 60

After the above-mentioned parameters are set, the farthest position number along the negative direction of the roation axis equals to the compensation position number of the reference point;

The farthest compensation number along with the positive direction is as follows:

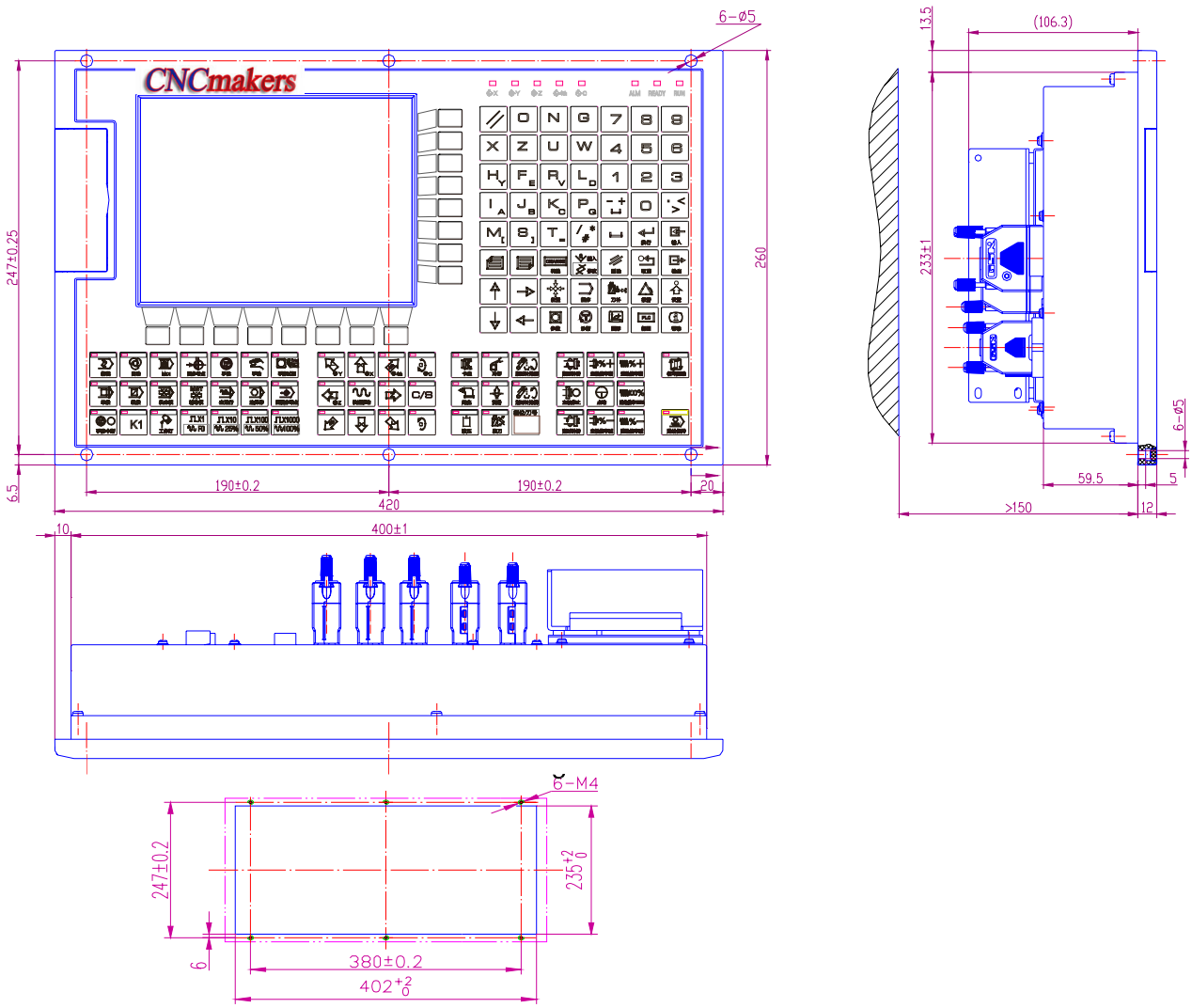
Compensation position number of reference point + (Movement value per revolution/Compensation position interval) = 60 + 360/45 = 68;

The corresponding relationships between the machine coordinate and compensation position number are shown below:

IV Appendix

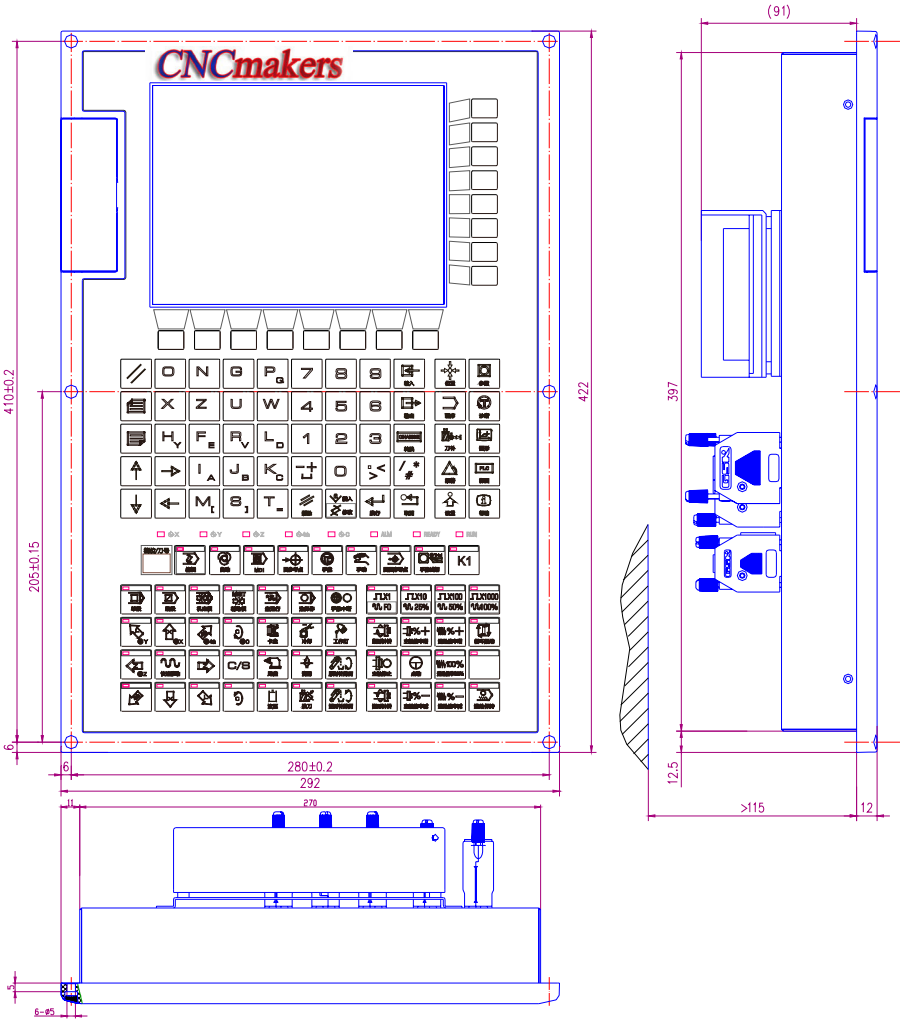
APPENDIX I GSK980TDi,GSK980TDi-V APPEARANCE DIMENSION

1.1 GSK980TDi Appearance Dimension

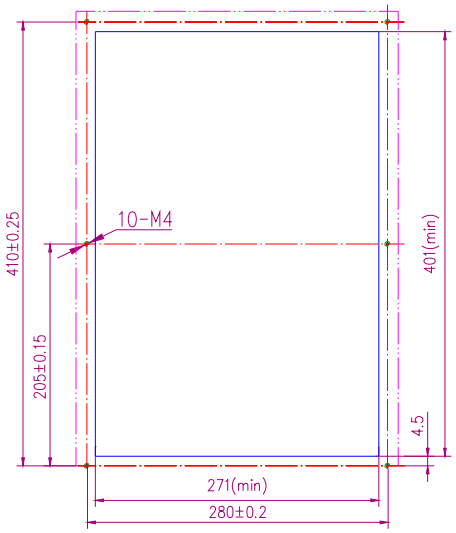


GSK980TDi appearance dimension

1.2 GSK980TDi-V Appearance Dimension



User istallation drawing

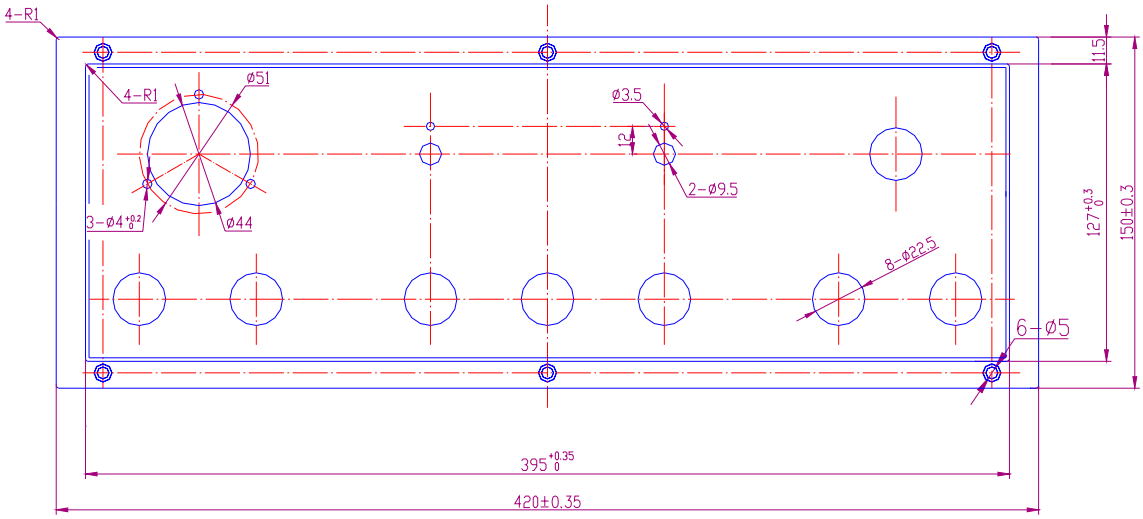


GSK980TDi-V appearance dimension

APPENDIX Ⅱ OUTLINE DIMENSION of ACCESSIONAL PANEL

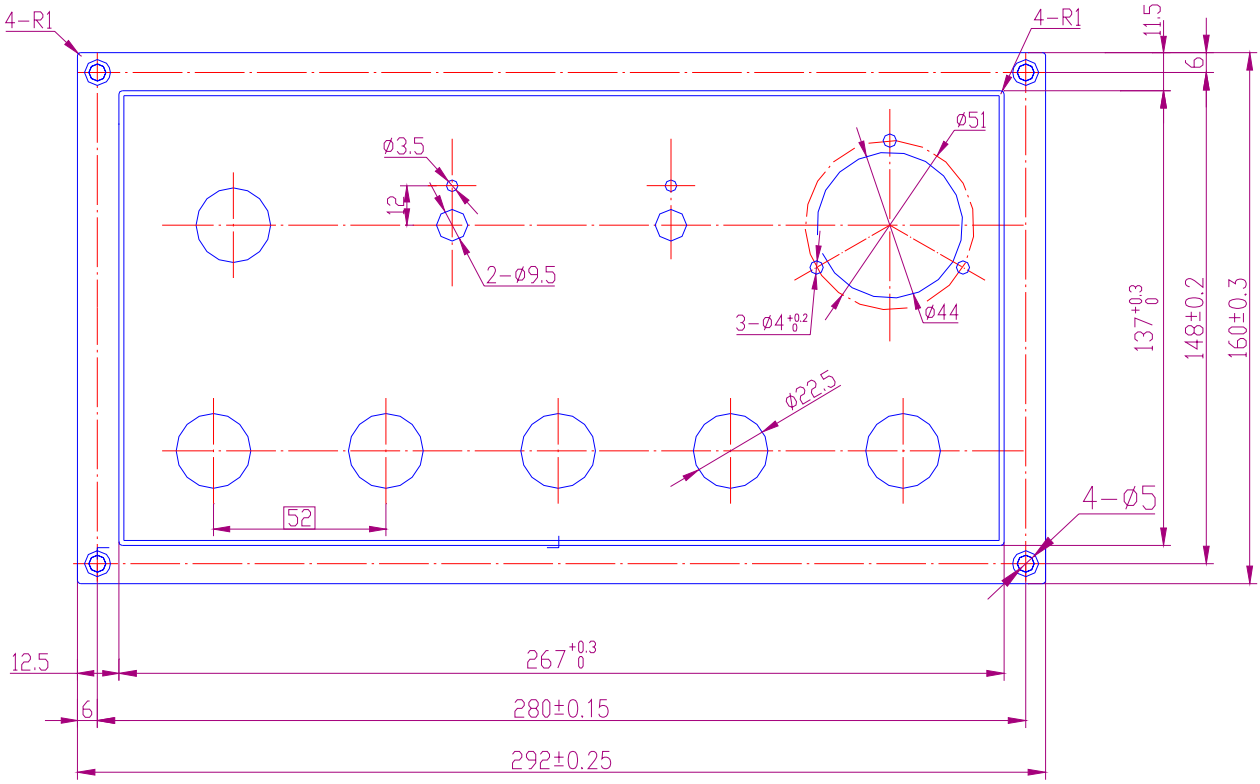
2.1 Outline Dimension of Accessional Panel AP01A

AP01: Accessional Panel of GSK980TDi, can be fixed below the panel and the outline dimension is as follow:

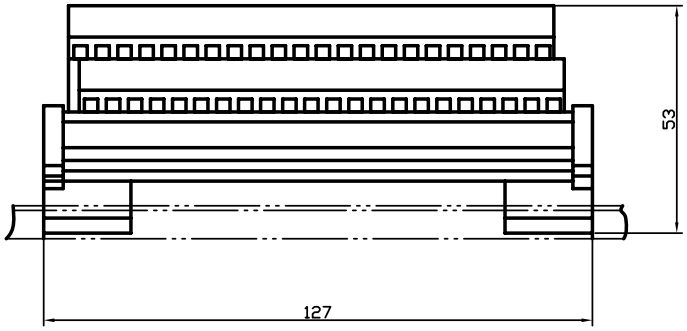


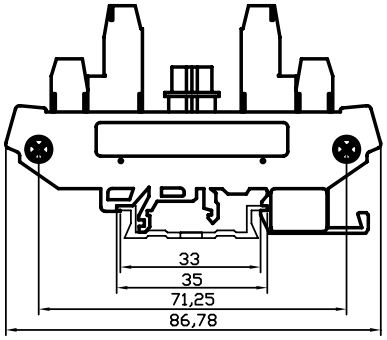
2.2 Outline Dimension of Accessional Panel AP03A

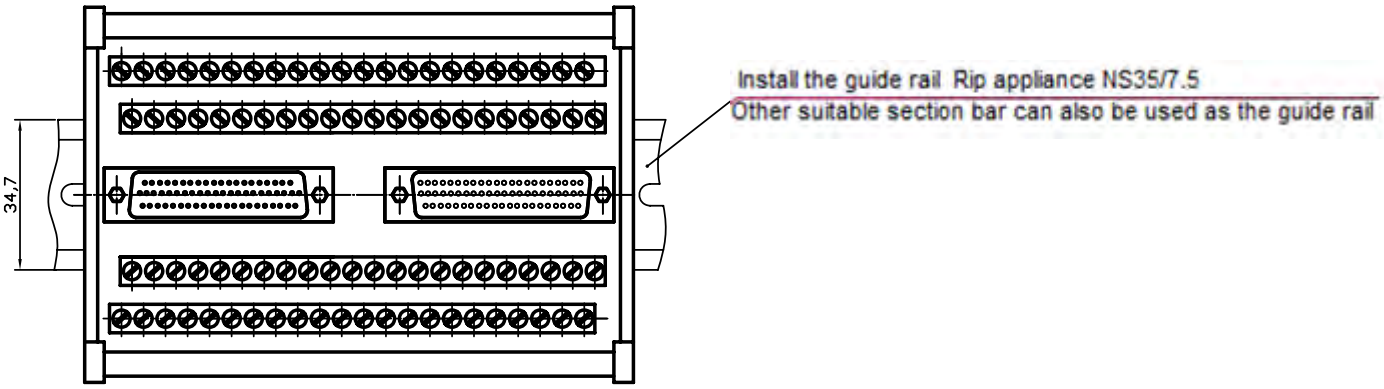
AP01B: Accessional Panel of GSK980TDi-V, can be fixed at the side of panel and the outline dimension is as follows:



APPENDIX Ⅲ OUTLINE DIMENSION of I/O Deconcentrator MCT01B

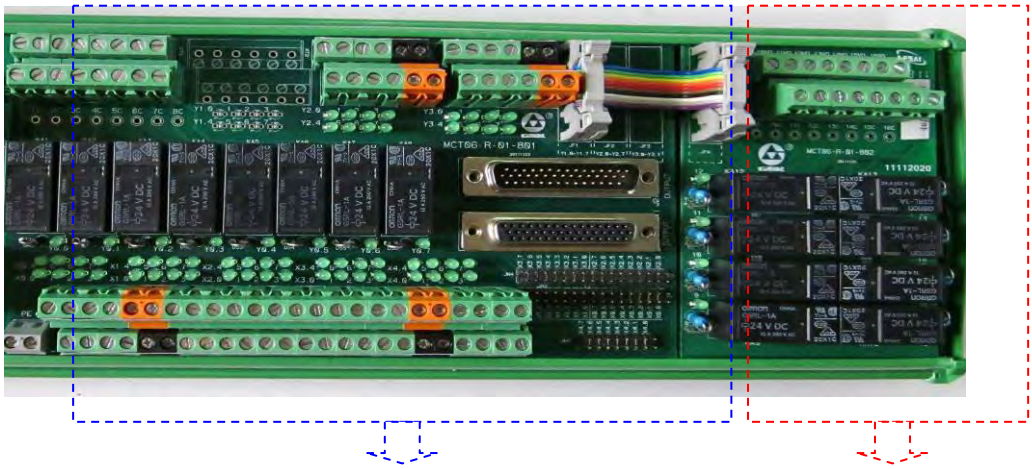






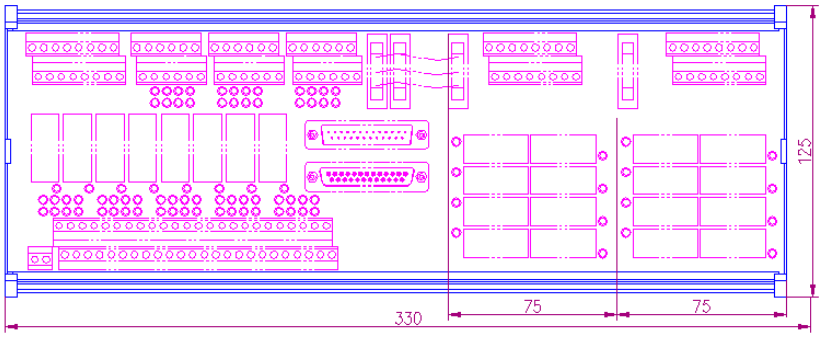
APPENDIX Ⅳ OUTLINE DIMENSION of I/O Deconcentrator MCT06R

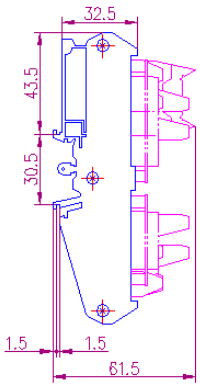
For installation, debugging and maintenance of the product, our company has already developed the brand-new I/O deconcentrator MCT06-R with the relay to reduce the faults, such as the product running abnormal and the component damage, etc due to the incorrect connection during installation and connection. MCT06-R deconcentrator is composed of the deconcentrator basic module, the relay extension module and the spindle servo transfer module or some of them and it is configurated eight specifications based on the user requirements.



The deconcentrator basic module

The relay extension module (maximum 3)





TYPE		MCT06-R1	MCT06-R2	MCT06-R3	MCT06-R4
OVERALL DIMENSION （mm*mm）		180*125	255*125	330*125	405*125
Form	The deconcentrator basic module	1 piece	1 piece	1 piece	1 piece
	The relay extension module	0pc（0 relay ）	1pc（8 relays）	2pcs（16 relays）	3pcs（24 relays）

APPENDIX Ⅴ DELIVERY STANDARD PARAMETER

State parameter	Standard parameter (servo 1u-level)	Standard parameter (servo 0.1u-level)	User parameter (backup)
001	00000000	00011010	
002	00000010	00000010	
003	00010000	00010000	
004	01000000	01000000	
005	00010010	00010010	
006	00000000	00000000	
*007	10000000	10000000	
008	00011111	00011111	
009	00000000	00000000	
010	00001111	00011111	
011	00000000	00000000	
012	10101011	10101011	
013	00000000	00000000	
014	00011111	00011111	
017	00001000	00001000	
018	00000000	00000000	
020	00000000	00000000	
172	01100000	01100000	
174	00001000	00001000	
*175	00000000	00000000	
180	00000000	00000000	
181	11100000	11100000	
183	00000000	00000000	
184	00000000	00000000	
185	00000000	00000000	
186	00000000	00000000	
187 X	00000010	00000010	
---- Z	00000010	00000010	
188 X	01000101	01000101	
---- Z	01000101	01000101	
189 X	00100000	00100000	
---- Z	00100000	00100000	
190 X	00000000	00000000	
---- Z	00000000	00000000	
195 S1	01000000	01000000	
---- S2	00000000	00000000	
---- S3	00000000	00000000	
196	00000000	00000000	
202	00000000	00000000	
201	00000001	00000001	
203	00000000	00000000	
250	00000001	00000001	

State parameter	Standard parameter 1 (servo 1u-level)	Standard parameter 3 (servo 0.1u-level)	User parameter (backup)
015 X	1	1	
---- Z	1	1	
016 X	1	1	
---- Z	1	1	
019	5	5	
021 S1	0	0	
---- S2	0	0	
022 X	4000	4000	
---- Z	8000	8000	
*023 X	100	100	
---- Z	100	100	
024	0	0	
025	0	0	
*026	100	100	
*027	8000	8000	
*028	200	200	
*029	100	100	
*030	200	200	
031	1260	1260	
032 X	200	200	
---- Z	400	400	
033 X	100	100	
---- Z	200	200	
034 X	0	0	
---- Z	0	0	
035	0	0	
036 S1	0	0	
---- S2	0	0	
---- S3	0	0	
037 S1	9999	9999	
---- S2	9999	9999	
---- S3	9999	9999	
038 S1	9999	9999	
---- S2	9999	9999	
---- S3	9999	9999	
039 S1	9999	9999	
---- S2	9999	9999	
---- S3	9999	9999	
040 S1	9999	9999	
---- S2	9999	9999	
---- S3	9999	9999	
*041	100	100	
042	10	10	
043	99	99	
044	115200	115200	
045 X	99999999	99999999	
---- Z	99999999	99999999	

State parameter	Standard parameter 1 (servo 1u-level)	Standard parameter 3 (servo 0.1u-level)	User parameter (backup)
046 X	-99999999	-99999999	
---- Z	-99999999	-99999999	
047 X	0	0	
---- Z	0	0	
048	0	0	
049	0	0	
050	0	0	
051	1	1	
052	0	0	
053	0	0	
054	0	0	
055	1	1	
056	0	0	
057	1	1	
058	0	0	
059	0	0	
060	0	0	
061	0	0	
062	0	0	
063	0	0	
064	0	0	
065	0	0	
066	0	0	
067 S1	100	100	
---- S2	100	100	
---- S3	100	100	
068 S1	0	0	
---- S2	0	0	
---- S3	0	0	
069 S1	1	1	
---- S2	0	0	
---- S3	0	0	
070 S1	1024	1024	
---- S2	1024	1024	
---- S3	1024	1024	
071	32	32	
072	50	50	
073 S1	65535	65535	
---- S2	65535	65535	
---- S3	65535	65535	
074 S1	0	0	
---- S2	0	0	
---- S3	0	0	
075 S1	0	0	
---- S2	0	0	
---- S3	0	0	
076 S1	0	0	

State parameter	Standard parameter 1 (servo 1u-level)	Standard parameter 3 (servo 0.1u-level)	User parameter (backup)
---- S2	0	0	
---- S3	0	0	
077	0	0	
078	0	0	
079	0	0	
080	0	0	
081	0	0	
082	0	0	
083	0	0	
084	4	4	
085	0	0	
086	0	0	
087	0	0	
088	0	0	
089	0	0	
090	0	0	
091	0	0	
092	0	0	
093	0	0	
094	0	0	
095	0	0	
096 X	0	0	
---- Z	200	200	
097 X	199	199	
---- Z	399	399	
098 X	100	100	
---- Z	300	300	
099 X	10000	100000	
---- Z	10000	100000	
100 X	0	0	
---- Z	0	0	
101 X	0	0	
---- Z	0	0	
102 X	0	0	
---- Z	0	0	
103 X	0	0	
---- Z	0	0	
104	0	0	
105	0	0	
106	0	0	
107	0	0	
108	0	0	
109	40	40	
110 S1	1	1	
---- S2	1	1	
---- S3	1	1	
111 S1	1	1	

State parameter	Standard parameter 1 （servo 1u-level）	Standard parameter 3 （servo 0.1u-level）	User parameter （backup）
---- S2	1	1	
---- S3	1	1	
112	0	0	
113 X	3800	3800	
---- Z	7600	7600	
114 X	0	0	
---- Z	0	0	
115	0	0	
116	0	0	
117	0	0	
118	0	0	
119	3	3	
120 X	0	0	
---- Z	0	0	
121 X	0	0	
---- Z	0	0	
122 X	0	0	
---- Z	0	0	
123 X	0	0	
---- Z	0	0	
124	0	0	
125	0	0	
126	0	0	
127	0	0	
128	0	0	
129	0	0	
130	0	0	
131	0	0	
132	0	0	
133	0	0	
134	1	1	
135	0	0	
136	0	0	
137	0	0	
138	0	0	
139	0	0	
140	1000	1000	
141	100	100	
142	10000	100000	
143	20000	200000	
144	7500	75000	
145	15000	150000	
146	0	0	
147	0	0	
148	35	35	
149	0	0	
150	0	0	

State parameter	Standard parameter 1 (servo 1u-level)	Standard parameter 3 (servo 0.1u-level)	User parameter (backup)
151	0	0	
152	0	0	
153	0	0	
154	10	100	
155	0	0	
156	0	0	
157	0	0	
158	0	0	
159	0	0	
160	0	0	
161	0	0	
162 S1	10	10	
---- S2	10	10	
---- S3	10	10	
163 S1	100	100	
---- S2	100	100	
---- S3	100	100	
164 S1	14	14	
---- S2	0	0	
---- S3	0	0	
165	0	0	
166 S1	10	10	
---- S2	10	10	
---- S3	10	10	
167 S1	200	200	
---- S2	200	200	
---- S3	200	200	
168 S1	0	0	
---- S2	0	0	
---- S3	0	0	
169 S1	0	0	
---- S2	0	0	
---- S3	0	0	
170	20	20	
171	0	0	
176	0	0	
177	0	0	
178	0	0	
179	0	0	
182	0	0	
189	0	0	
190	0	0	
191 X	0	0	
---- Z	0	0	
192	0	0	
193	0	0	
194	0	0	

State parameter	Standard parameter 1 （servo 1u-level）	Standard parameter 3 （servo 0.1u-level）	User parameter （backup）
197	0	0	
198	0	0	
199	0	0	
200	0	0	
204	0	0	
205	0	0	
206	0	0	
207	0	0	
208	0	0	
209	0	0	
210	0	0	
211	0	0	
212	0	0	
213	0	0	
214	0	0	
215	0	0	
216	0	0	
217	0	0	
218	0	0	
219 X	1	1	
---- Z	2	2	
220 S1	1	1	
---- S2	2	2	
---- S3	0	0	
221 S1	5	5	
---- S2	4	4	
---- S3	3	3	
222	0	0	
223	0	0	
224	2	2	
225 X	88	88	
---- Z	90	90	
226 X	0	0	
---- Z	0	0	
227	0	0	
228	0	0	
229	0	0	
230 X	1	1	
---- Z	3	3	
231	0	0	
232	0	0	
233	0	0	
234	0	0	
235	0	0	
236	4	4	
237	0	0	
238	0	0	
239	0	0	

State parameter	Standard parameter 1 (servo 1u-level)	Standard parameter 3 (servo 0.1u-level)	User parameter (backup)
240	10	10	
241	1	1	
242	0	0	
243	1	1	
244	0	0	
245	1	1	
246	0	0	
247	0	0	
248	0	0	
249 X	1000	10000	
---- Z	1000	10000	
251	0	0	
252	0	0	
253	0	0	
254	0	0	

Note: The parameters of its number with "*" influence the acceleration/deceleration characteristics and the user can properly adjust them according to the system allocation and load characteristics.

APPENDIX VI ALARM LIST

1.CNC alarm

No.	Content	Remark
000	Emergent stop alarm and ESP open circuit	
001	Called programs do not exist or cannot be opened	
003	Characters are less than 2 or more than 11 for one word	
004	Address error (address is A~Z)	
005	Illegal command value	
006	Block numbers are negative or with decimals	
007	Illegal G commands	
008	Execute G96 when the spindle analog voltage control is invalid	
009	Command movement distance when G commands in group 00 and 01 are not input and there are invalid commands in group 01	
010	There are the same addresses in one block	
011	There are more than 40 words in one block	
012	Command values exceed their valid range or command addresses which cannot be omitted is omitted	
013	Input S commands except for S00~S99 when the spindle analog voltage control is invalid	
014	G commands in group 00, 01 and 12 cannot be in the same one block	
015	Execute M commands for spindle automatic gear shifting when the spindle analog voltage control is invalid	
016	Tool offset numbers exceed their valid range(0~32)	
017	Tool number exceeds the range of No.084 parameter	
018	The data specified by the interpolation command cannot form a correct curve	
019	The tool group number exceeds the range (1~32) in the tool life management	
020	Cannot execute T command in C tool nose radius compensation, cancel it	
0021	G71~G78,G90,G92,G94 can only be used in G18 plane	
0022	Forbid to use plane exchange code G17~G19 here	
0023	The programming radius of G05 code is out of allowable range	
024	There is no G11 or G13.1 in the program	
025	There is no tool in the current tool group in the tool life management	
026	The current tool group has not been defined in the tool life management	
027	The tool quantity exceeds 8 in the current tool group in tool life management	
028	The tool life management function is invalid and G10 L3 cannot be used	
030	G17~G19 codes Cann't be executed in C tool compensation or cylindrical interpolation	
032	Absolute value of R is more than that of U/2 in G90, G92,G92.1	
033	Absolute value of R is more than that of W in G94	
040	There are more than 100 blocks in G70~G73	
041	Ns and Nf are reversed each other in G70~G73	
042	There is no Ns or Nf or they exceed their allowed range in G70~G73	

No.	Content	Remark
043	There is no Ns or Nf in G70~G73	
044	Call subprograms in G70~G73	
045	Ns does not command G00 or G01 in G70~G73	
046	Initial blocks are employed with the forbidden G commands in G70~G73	
047	Unusable G code is in the block specified by Q in G70,G71,G72,G73	
050	G71~G72 command can't satisfy the condition of single increase or decrease	
051	Single infeed exceeds its allowed range in G71 or G72	
052	Single tool retraction exceeds its allowed range in G71 or G72	
053	The cycle starting point is in the closed area formed by the starting point and end point of the finishing path in G71 or G72	
054	Flute quantity is more than 10 in G71 or G72	
055	X/ Z value is not specified in the first block or the movement is 0 in G71	
056	X/ Z value is not specified in the first block or the movement is 0 in G72	
057	Cycle times is less than 1 or more than 99999 in G73	
060	Single tool retraction R(e) exceeds its allowed range in G74 or G75	
061	The tool retraction is negative at the end of cutting in G74 or G75	
0062	Z value is 0 or not input in G74	
0063	Q is in default or out of range in G74	
0064	X single cutting value(P) exceeds its allowed range in G74	
0065	X value is 0 or not input in G75	
0066	P is in default or out of range in G75	
0067	Z single cutting value(Q) exceeds its allowed range in G75	
0069	Starting point coordinates are abnormal when cycle ends and returns to starting point	
0070	The tool starts from the midway in taper screw thread cutting in G76/G78	
0071	Min. cutting value exceeds its allowed range in G76 or G78	
0072	Finishing allowance exceeds its allowed range in G76 or G78	
0073	Tooth height is less than finishing allowance or 0 in G76 or G78	
0074	Cycle times exceeds its allowed range in G76 or G78	
0075	X/Z movement distance in G76 is zero	
0076	There is no specified tooth height P in G76 or G78	
0077	There is no the first cutting depth Q or Q is 0 or Q is not input in G76 or G78	
0078	Tool nose angle is more than or equal to thread angle in G76 or G78	
0079	Cutting value of G78 is out of range(0,1)	
0080	Cutting mode of G78 is out of range(0,1,2,3)	
0082	Have no the measured position arrival signal in parameter specifying area in automatic tool offset(G36,G37)	
0083	Specifying an invalid axis or a command is incremental in automatic tool offset (G36,G37). Modify the program	
0084	Have the measured position arrival signal in area set by the parameter in automatic tool offset(G36,G37)	
0085	The distance between the starting point and the target is less than the setting (142,143) in automatic tool offset(G36,G37), have no set the rapid motion area	

No.	Content	Remark
0086	The parameter (142,144) or (143,145) area setting are wrong in automatic tool offset(G36,G37)	
0087	Firstly create or set the coordinate system before executing G36,G37	
0088	Firstly set the correct tool number and the compensation number before executingG36,G37	
0089	The tool offset is in protection state, and cannot be written	
0090	When tool offset measure executes the direction input B operation, axis movement directions are interlocked	
0092	Can't find the thread repair position. An soft key [THD.REPAIR RECORD POS] displayed on the ABSOLUTE screen. In JOG,HND or STEP mode, press the key record the thread repair position.	
0093	The thread repair position beyond the thread repair program, please check the program and the thread repair position	
0095	Subprogram numbers are not input or are illegal when M98 calls them	
0096	Layers of nested subprograms are more than 4	
0097	Calling programs in M98 is the current one(main program)	
0099	Use M98 or M99 in C tool radius compensation	
0103	Denominator is 0 for division operation in G65	
0104	G65 commands illegal H command	
0105	Macro variable number of G65 is illegal(error)	
0109	P command value isn’t variable in G65	
0111	H99 user alarm number in G65 exceeds its range	
0112	Block number of macro command (G65)skip or M99 program return exceeds their range	
0113	There is no block number for block skip or M99 program return	
0114	G65, G66 command format are mistaken	
0115	G65 H operand is mistaken	
0116	G code format of calling macro program is err	
0118	Number of argument I, J, K cannot exceed 10	
0120	P value of G30 code is out of range (2~4)	
0122	G22,G23 command code can not be executed with other G,M,S,T command simultaneously	
0123	Stored travel check function has been closed,Can't use G22,G23 command code.\nmodify parameter No.172 please.	
0129	Programmed axis name is invalid	
0130	axis address is not normative	
0131	G31/G32/G32.1/G33/G34/G36/G37/G84/G88/auxiliary axis code can't be used in C offset type	
0133	Specified axis is a linearity of cylinder or rotary axis	
0134	Cannot specify the cylinder radius again	
0135	The linear or rotary axis setting used to the cylinder interpolation is mistaken	
0136	Only currently use the tool nose direction T0 or T9 to create C tool compensation	
0140	The macro format is specified in error	
0141	DO,END tabs are not 1,2,3 in macro statement	
0142	DO,END formats are specified in error	

No.	Content	Remark
0143	The brackets are not matched in macro statement or the format is specified in error	
0144	The divisor in macro statement cannot be 0	
0145	The specified anti-tangent ATAN format is mistaken in macro statement	
0146	The anti logarithm of LN in macro statement is 0 or less than 0	
0147	The square root in macro statement cannot be negative	
0148	The tangent TAN result in macro statement is endless	
0149	The operands of anti sine or anti cosine in macro statement exceed the range between -1 and 1	
0150	The macro variable number or the variable value is illegal(error) in macro statement	
0151	The local variable is empty	
0152	The variable #0 is always empty and cannot be operated	
0153	The operand in macro statement is not binary	
0154	P value don't want to be defined while calling macro program of using G code	
0155	Format of program note is mistaken (mismatching)	
0156	G20,G21 command code can not be executed with other command simultaneously	
0157	Metric/Inch system has been changed,and will be effective after restart	
0174	Some parameters cann't be modified for current operation level	
0175	W value or macro variety value of W is out of range while writing parameters	
0176	W value or macro variety value of W is not a valid binary value while writing parameters	
0177	Not specify writing or reading operand under the state of writing or reading parameters	
0178	Not specify a parameter or the parameter don't exist under the state of writing or reading parameters	
0179	Not specify axis NO. when writing or reading axis parameters	
0180	R only specify macro variety under the state of writing or reading parameters	
0181	The axis NO. of setting is out of range while writing or reading parameters	
0182	Specify simultaneously a operand of writing and reading in one program segment under the state of writing or reading parameters	
0183	Operand cann't be a decimal fraction under the state of writing or reading parameter	
0184	Specify two or above parameters in one program segment under the state of writing or reading parameters	
0185	Parameter NO. cann't be a decimal fraction under the state of writing or reading parameters	
0190	Not specify screw pitch or out of range in machinning screw or tap	
0191	Head quantity of multi-thread is out of range	
0192	Initial angle of screw (Q value) is out of range	
0193	Nonzero move value of X axis has been commanded in G33	
0194	Y axis can't interpolate with X/Z axis in G33	
0195	R value of G34 is out of range	

No.	Content	Remark
0200	Circle radius is zero or undefined	
0201	The distance of circle start and end point is too small	
0202	Circle start point and center of the circle are the same as one point	
0203	The error of circle radius is out of range of the parameter NO.154	
0204	Not specify middle point in three point circle instruction	
0205	Three points are a line in three point circle instruction	
0206	Helical interpolation is invalid if defined plane hasn't movement	
0208	Long radius or short radius of ellipse is zero or undefined	
0209	P value of parabola is zero or undefined	
0227	Spindle positioning axis isn't allowed to execute the function	
0228	Spindle positioning axis and some code cann't be in the same block	
0229	Unusable code or command in synchro control	
0232	Specify the axis movement command in M29 or G84/G88	
0233	Rigid tapping signal (G61.0) in rigid tapping mode is abnormal	
0234	Specify M29 repetitively or not input M29 before rigid tapping	
0235	The spindle must be oriented, M29 cannot be with G84/G88 in the same block when rigid tapping	
0236	Set the axis used to the spindle control to the rotary axis before the rigid tapping/rigid thread machining	
0237	Have used unnecessary commands in rigid tapping or rigid screw	
0238	Have not selected the spindle used to the rigid tapping for the multi-spindle function device when rigid tapping	
0239	Cannot switch the spindle used to the tapping when rigid tapping	
0251	Mistake in programming causes an error of C tool nose radius compensation	
0252	Mistake in programming causes an end point of arc machining is not on the arc in the course of C tool nose radius compensation	
0253	Mistake in programming causes there are the same coordinates for two neighboring points not to execute C tool nose radius compensation	
0254	Mistake in programming causes there are the same coordinates between center point and starting point of arc not to execute C tool nose radius compensation	
0255	Mistake in programming causes there are the same coordinates between center point and end point of arc not to execute C tool nose radius compensation	
0256	Arc radius is less than that of tool nose to cause not to execute C tool nose radius compensation	
0257	Mistake in programming causes there is not intersection of two arcs of current tool radius in the course of C tool nose radius compensation	
0258	Specify one arc command as executing C tool nose radius compensation	
0259	Specify one arc command as canceling C tool nose radius compensation	
0260	There is excessive cutting as checking C tool nose radius compensation	
0261	Mistake in programming causes there is not intersection between straight line and arc of current tool radius in the course of C tool nose radius compensation	

No.	Content	Remark
0262	Mistake in programming causes there is not intersection between arc and straight line of current tool radius in the course of C tool nose radius compensation	
0263	There are too non movement commands in C too radius compensation, the buffer overflows	
0270	The circle centre angle of circular screw interpolation is not in the range of 45°~135°or 135°~225°or 225°~315°or -45°~45.modify the program please	
0276	The end point and angle value using direct input of drawing dimensions are not applicable or normal	
0277	The instruction isn't allowed in direct input of drawing dimensions or non-motion blocks are too much	
0278	The middle block using direct input of drawing dimensions cann't be the last block of a loop	
0279	Angle is not normal or the end point cann't be calculated in direct input of drawing dimensions	
0281	The linear chamfer length is too length	
0282	The arc chamfer radius is too big	
0283	The linear chamfer length is too big or the arc data is mistaken	
0284	The arc chamfer radius is too big or the arc data is mistaken	
0285	The linear chamfer length is too big or the arc data is mistaken	
0286	The arc chamfer radius is too big or the arc data is mistaken	
0287	The linear chamfer length is too big or the intersection is not on the arc	
0288	The arc chamfer radius is too big or the intersection is not on the arc	
0289	The chamfer cannot be executed in end position	
0290	Cannot start the polar coordinate and the cylinder interpolation in C tool compensation state or G99	
0291	The linear axis or rotary axis of the polar coordinates does not correspond to No.189 or the linear axis attribution setting is mistaken	
0292	Execute the length compensation of all axes before starting the polar coordinates	
0293	X cannot be negative or 0 in G16/G15	
0294	Have used the command which cannot be used to the polar coordinates and the cylinder interpolation	
0295	C tool compensation is not cancelled in the polar coordinates and the cylinder interpolation	
0296	Finish machining counts of cylindrical interpolation loop are over 9999	
0297	Cann't cut the workpiece in cylindrical interpolation loop because X orientation and cutting direction are not same	
6000	CNC emergent stop is failure(restart the CNC)	
6001	No program zero is defined	
6003	Feed speed is too high	
6004	Spindle stopped accidentally, please stop feeding and check the spindle	
6005	Not define the highest speed of some gear, please check the parameter No.37~No.40	
6010	Spindle doesn't rotate,or speed is too slow in thread cutting	

No.	Content	Remark
6011	feed speed exceeds limit speed in cutting(Parameter No.027)	
6012	Spindle speed fluctuation has exceeded the alarm valve in thread cutting	
6013	The screw pitch occurs 0	
6014	Spindle encoder pulse is out of 100~10000	
6020	reference point hadn't found,Don't go back 2nd or 3rd or 4th reference point	
6050	PLCAxis is working,Don't change Ctrlmode	
6051	CNCAxis is working,Don't change Ctrlmode	
6052	PLCAxis is working,CNC can't Ctrl this axis	
6053	The movement distance of PLC axis is out of range	
6054	In Ref mode A, the move is fail	
6300	Stored stroke limit 1 has been exceeded	
6301	Stored stroke limit 2 has been exceeded	
6302	Stored stroke limit 3 has been exceeded	
6304	Reference point is in the limit area of stored stroke limit 1, Please modify parameter No.045, No.046	
6305	Reference point is in the limit area of stored stroke limit 2.Please modify parameter No.100, No.101	
6306	Reference point is in the limit area of stored stroke limit 3.Please modify parameter No.102, No.103	
6311	overtravel in setting positive location of axis X	
6312	overtravel in setting positive location of axis Z	
6313	overtravel in setting positive location of axis Y	
6314	overtravel in setting positive location of axis 4th	
6315	overtravel in setting positive location of axis 5th	
6316	overtravel in setting positive location of axis 6th	
6321	overtravel in setting negative location of axis X	
6322	overtravel in setting negative location of axis Z	
6323	overtravel in setting negative location of axis Y	
6324	overtravel in setting negative location of axis 4th	
6325	overtravel in setting negative location of axis 5th	
6326	overtravel in setting negative location of axis 6th	
6331	The X axis positive location of stored stroke limit 2 has been exceeded	
6332	The Z axis positive location of stored stroke limit 2 has been exceeded	
6333	The Y axis positive location of stored stroke limit 2 has been exceeded	
6334	The 4th axis positive location of stored stroke limit 2 has been exceeded	
6335	The 5th axis positive location of stored stroke limit 2 has been exceeded	
6336	The 6th axis positive location of stored stroke limit 2 has been exceeded	
6341	The X axis negative location of stored stroke limit 2 has been exceeded	
6342	The Z axis negative location of stored stroke limit 2 has been exceeded	
6343	The Yaxis negative location of stored stroke limit 2 has been exceeded	
6344	The 4th axis negative location of stored stroke limit 2 has been exceeded	
6345	The 5th axis negative location of stored stroke limit 2 has been exceeded	
6346	The 6th axis negative location of stored stroke limit 2 has been exceeded	
6351	The X axis positive location of stored stroke limit 3 has been exceeded	
6352	The Z axis positive location of stored stroke limit 3 has been exceeded	

No.	Content	Remark
6353	The Y axis positive location of stored stroke limit 3 has been exceeded	
6354	The 4th axis positive location of stored stroke limit 3 has been exceeded	
6355	The 5th axis positive location of stored stroke limit 3 has been exceeded	
6356	The 6th axis positive location of stored stroke limit 3 has been exceeded	
6361	The X axis negative location of stored stroke limit 3 has been exceeded	
6362	The Z axis negative location of stored stroke limit 3 has been exceeded	
6363	The Y axis negative location of stored stroke limit 3 has been exceeded	
6364	The 4th axis negative location of stored stroke limit 3 has been exceeded	
6365	The 5th axis negative location of stored stroke limit 3 has been exceeded	
6366	The 6th axis negative location of stored stroke limit 3 has been exceeded	
6411	axis X+ overtravel	
6412	axis Z+ overtravel	
6413	axis Y+ overtravel	
6414	axis 4th+ overtravel	
6415	axis 5th+ overtravel	
6416	axis 6th+ overtravel	
6421	axis X- overtravel	
6422	axis Z- overtravel	
6423	axis Y- overtravel	
6424	axis 4th- overtravel	
6425	axis 5th- overtravel	
6426	axis 6th- overtravel	
6431	The axis X driver is not prepared	
6432	The axis Z driver is not prepared	
6433	The axis Y driver is not prepared	
6434	The axis 4th driver is not prepared	
6435	The axis 5th driver is not prepared	
6436	The axis 6th driver is not prepared	
6441	Spindle servo 1 is not ready	
6442	Spindle servo 2 is not ready	
6443	Spindle servo 3 is not ready	
6500	Slave device number conflict in GSKLINK communication	
6600	The bus connection is OFF	
6601	Enumberation overtime	
6602	The bus connection creating communication fails	
6603	The bus connection delaying causes the test device to fail	
6604	The bus communication's parameterr allocation fails	
6605	The bus communication allocation overtime	
6606	The bus main station's communication returns to CP0 stage	
6607	The slave station number set by the system is inconsistent with the actual	
6608	The slave device's axis name (logic address) is not matched with the system's valid axis	
6610	MDT data loss	
6611	MST data loss	
6612	MDT data check error	

No.	Content	Remark
6613	GDT data check error	
6620	IDN16, 24 allocation fails	
6621	IDN32,35 allocation fails	
6622	IDN5030,5031,5033 allocation fails	
6630	I/O unit allocation fails	
6702	Setup machine zero point again please. Because the parameter file of zero point failed to save as using absolute encoder motor	
6703	Setup machine zero point again please. Because the parameter file of zero point failed to read as using absolute encoder motor.	
6711	The error of actual position and memory position oversteps the range permitted as axis X equipped absolute encoder. Return machine zero point manually and set zero point again.	
6712	The error of actual position and memory position oversteps the range permitted as axis Z equipped absolute encoder. Return machine zero point manually and set zero point again	
6713	The error of actual position and memory position oversteps the range permitted as axis Y equipped absolute encoder. Return machine zero point manually and set zero point again	
6714	The error of actual position and memory position oversteps the range permitted as axis 4TH equipped absolute encoder. Return machine zero point manually and set zero point again.	
6715	The error of actual position and memory position oversteps the range permitted as axis 5TH equipped absolute encoder. Return machine zero point manually and set zero point again.	
6716	The error of actual position and memory position oversteps the range permitted as axis 6TH equipped absolute encoder. Return machine zero point manually and set zero point again.	
6721	Initialization of machine coordinates makes a mistake while starting as axis X equipped absolute encoder. Reset machine zero point again.	
6722	Initialization of machine coordinates makes a mistake while starting as axis Z equipped absolute encoder. Reset machine zero point again.	
6723	Initialization of machine coordinates makes a mistake while starting as axis Y equipped absolute encoder. Reset machine zero point again.	
6724	Initialization of machine coordinates makes a mistake while starting as axis 4TH equipped absolute encoder. Reset machine zero point again	
6725	Initialization of machine coordinates makes a mistake while starting as axis 5TH equipped absolute encoder. Reset machine zero point again.	
6726	Initialization of machine coordinates makes a mistake while starting as axis 6TH equipped absolute encoder. Reset machine zero point again.	
6741	When X axis is allocated with an absolute encoder without a reference point, a reference point is set after the machine is debugged normally	
6742	When Z axis is allocated with an absolute encoder without a reference point, a reference point is set after the machine is debugged normally	
6743	When Y axis is allocated with an absolute encoder without a reference point, a reference point is set after the machine is debugged normally	

No.	Content	Remark
6744	When 4th axis is allocated with an absolute encoder without a reference point, a reference point is set after the machine is debugged normally	
6745	When 5th axis is allocated with an absolute encoder without a reference point, a reference point is set after the machine is debugged normally	
6746	When 6th axis is allocated with an absolute encoder without a reference point, a reference point is set after the machine is debugged normally	
6751	Gear ratio of CNC or servo ,the direction of axis moving and the machine zero points are not as different as data established successfully while axis X equipped absolute encoder. Check up parameters and renew to set machine zero points	
6752	Gear ratio of CNC or servo ,the direction of axis moving and the machine zero points are not as different as data established successfully while axis Z equipped absolute encoder. Check up parameters and renew to set machine zero points.	
6753	Gear ratio of CNC or servo ,the direction of axis moving and the machine zero points are not as different as data established successfully while axis Y equipped absolute encoder. Check up parameters and renew to set machine zero points.	
6754	Gear ratio of CNC or servo ,the direction of axis moving and the machine zero points are not as different as data established successfully while axis 4TH equipped absolute encoder. Check up parameters and renew to set machine zero points.	
6755	Gear ratio of CNC or servo ,the direction of axis moving and the machine zero points are not as different as data established successfully while axis 5TH equipped absolute encoder. Check up parameters and renew to set machine zero points.	
6756	Gear ratio of CNC or servo ,the direction of axis moving and the machine zero points are not as different as data established successfully while axis 6TH equipped absolute encoder. Check up parameters and renew to set machine zero points.	
6761	The batteries of absolute encoder are too low as axis X equipped absolute encoder. Renew batteries and set machine zero point again.	
6762	The batteries of absolute encoder are too low as axis Z equipped absolute encoder. Renew batteries and set machine zero point again.	
6763	The batteries of absolute encoder are too low as axis Y equipped absolute encoder. Renew batteries and set machine zero point again.	
6764	The batteries of absolute encoder are too low as axis 4th equipped absolute encoder. Renew batteries and set machine zero point again.	
6765	The batteries of absolute encoder are too low as axis 5th equipped absolute encoder. Renew batteries and set machine zero point again.	
6766	The batteries of absolute encoder are too low as axis 6th equipped absolute encoder. Renew batteries and set machine zero point again.	
6771	Machine zero point Can't be returned because no zero point position is established as axis X of equipping absolute encoder	
6772	Machine zero point Can't be returned because no zero point position is established as axis Z of equipping absolute encoder	

No.	Content	Remark
6773	Machine zero point Can't be returned because no zero point position is established as axis Y of equipping absolute encoder	
6774	Machine zero point Can't be returned because no zero point position is established as axis 4TH of equipping absolute encoder	
6775	Machine zero point Can't be returned because no zero point position is established as axis 5TH of equipping absolute encoder	
6776	Machine zero point Can't be returned because no zero point position is established as axis 6TH of equipping absolute encoder	
6781	The position of movining is being out of range as axis X of equipping absolute encoder	
6782	The position of movining is being out of range as axis Z of equipping absolute encoder	
6783	The position of movining is being out of range as axis Y of equipping absolute encoder	
6784	The position of movining is being out of range as axis 4TH of equipping absolute encoder	
6785	The position of movining is being out of range as axis 5TH of equipping absolute encoder	
6786	The position of movining is being out of range as axis 6TH of equipping absolute encoder	
7001	The name of axis isn't in the usable range or exists same name with others,set parameter #225	
7002	increment system of additional axis(Y,4th,5th) should not less than current increment system(IS-B,IS-C).setting again	
7003	basic axis X/Y/Z can only set one,set parameter #230	
7008	parameters of electric-gear set wrong	
7009	data errors！restart and return reference please! adjust tool offsets，check all parameters.	
7010	can't set two or more CS axes effectively if muilt-spindles function are not effect or if no using second CS axis	
7011	Can't use CS spindle because the spindle don't bind a valid feed axis.\n Please check the parameter No.221	
7012	Set rotary axis function firstly before using CS axis or rigid tapping.	
7013	Can't use axis CS because it isn't a serial spindle.	
7014	Fail to switch CS mode	
7015	don't switch spindle working mode while using Cs axis	
7016	Set rotary axis function firstly before using CS axis or rigid tapping	
7017	Can't use axis CS because it isn't a serial spindle.	
7020	Spindle No. definition error or repeated\n please set parameter No.220	
7021	The feed axis which binds to spindle is set by error or repeated.\n Please check the parameter No.221	
7022	The feed axis which binds to spindle is invalid or CS axis at first. Then the function of servo motor spindle can be used.	

No.	Content	Remark
7023	There isn't a feed axis which binds to spindle. Can't use spindle function of servo motor. Please check parameter No.221	
7024	Spindle positioning axis can't move in manual	
7030	Synchronous control master axis and slave axis cannot return to reference	
7031	Synchronous axis no.incorrect,duplicated or axis is invalid.\nModify parameter No.197~No.199 please.	
7035	Function is invalid due to detecting signal add settting incorrect.\nmodify parameter No.136~No.139 please	
7037	The coordinates switched is out of range when G54~G59 coordinates system switchs.Return machine zero point and set tool offset again please.	
7100	Nvram data error(CurMachCoord,CurAbsCoord,CurRelCoord)\n Restart machine and set zero point and set tool offset again please.	
7101	Nvram data error(MlkAbsCoord,MlkRelCoord,PoffMlkState)\n Restart machine and return zero point and set tool offset again please.	
7102	Nvram data error(LocalCoordOFT)\n Restart machine and return zero point and set tool offset again please.	
7103	Nvram data error(MoveDirIsPlus)\n Restart machine and return zero point and set tool offset again please.	
7104	Nvram data error(OffsUnmoved)\n Restart machine and return zero point and set tool offset again please.	
7105	Nvram data error(Last_Toffs)\n Restart machine and return zero point and set tool offset again please.	
7106	Nvram data error(ToffsUnmoved)\n Restart machine and return zero point and set tool offset again please.	
7107	Nvram data error(OffsRcd)\n Restart machine and return zero point and set tool offset again please.	
7108	Nvram data error(OffsRcdts)\n Restart machine and return zero point and set tool offset again please.	
7109	Nvram data error(LOffsDiff)\n Restart machine and return zero point and set tool offset again please.	
7110	Nvram data error(CompPos)\n Restart machine and return zero point and set tool offset again please.	
7111	Nvram data error(CurCoordSys)\n Restart machine and return zero point and set tool offset again please.	
7112	Nvram data error(CNC_CurrentToolNo, CNC_CurrentTComp, CurrentRoffsmode)\n Restart machine and return zero point and set tool offset again please.	
7113	Nvram data error(ToffLock)\n Restart machine and return zero point and set tool offset again please.	
7114	Nvram data error(PubMacroBuf_S)\n Restart machine and return zero point and set tool offset again please.	
7115	Nvram data error(APCRefPos)\n Restart machine and set zero point and set tool offset again please.	
7116	Nvram data error(RefCurMachCoord)\n Restart machine and set zero point and set tool offset again please.	

No.	Content	Remark
7117	Nvram data error(BLashSum)\n Restart machine and set zero point and set tool offset again please.	
7118	Nvram data error(PitchSum)\n Restart machine and set zero point and set tool offset again please.	
7120	Nvram data error(TSafetyPos)\n Restart machine and set safe position of switching tool please.	
7801	Parameter switch has been opened	
7803	Fail to open or delete the part program	
7804	Set position number of screw compensation error.\nmodify parameter No.96 ~ 98 please.。	
7805	USB disk connectted again. please reload the program if the file has been modified!	
7806	turn on power again when parameter is modified	
7807	Success to renew the working ladder which has been stopped.rerun it or power on again please.	
7808	CNC is failed to communicate with keyboard	
7810	Fail to open PLC file	
7811	The software version of ladder_chart is not suited	
7812	The first grade program of ladder_chart is too long	
7815	The voltage is low or unsteady.	
7816	NVRAM data verifies error!	
7817	The time is too short to save NVRAM data for last time when lost power.	
7830	Machine diagnosis configuration file is modified, it is better to restart again.	
7900	Loaded system allocation file is mistaken. Confirm whether cnc.cfg. exists	
7901	Loaded system allocation file is mistaken. Confirm whether param.cfg. exists	
7902	Loaded parameter value file is mistaken	
7903	Tool offset error!Refresh tool offset please. Clear alarm for pressing keys [CAN],[RESET] simultaneously	
7904	Loaded pitch compensation file is mistaken	
7905	Fail to load file COORD.CRD	
7906	Fail to start PLC ladder\nCheck ladder!	
7908	Fail to load toollife file!	
7909	Limited-time stop time is over and the system cannot run normally. Please contact with salesmen	
7910	CNC level degrade to lowest beause the time of stopping has been arrived,contact your supplier please.	
7911	CNC is going to stop work in several days.\nContact with supplier in advance please.	
7912	The password of stopping operation is default.Suggest to change the password please.	
7998	System software error! please contact with GSK CNC equipment co.,ltd.!	
7999	There is an unknown alarm. Please contact with developers。	

2.PLC alarm（standard PLC ladder definition）

Address	Alarm No.	Content
A0000.0	1000	Tool change time is too long
A0000.1	1001	Alarm for the tool is not in-position when tool change ends
A0000.2	1002	Alarm for the tool change is not completed
A0000.3	1003	Alarm for the system has not received the lock signal
A0000.4	1004	The system repetitively checks the lock signal and the lock signal is invalid when tool change is done
A0000.5	1005	Tool change makes mistakes before the system is turned off
A0000.6	1006	Pre-indexing proximity switch does not reach
A0000.7	1007	Alarm for tool post overheat
A0001.0	1008	Tailstock function is invalid, and M10 and M11 cannot be executed
A0001.1	1009	The thread run-out cannot be executed when the spindle rotating
A0001.3	1011	Have not checked the tailstock forward and cannot start the spindle
A0001.4	1012	Tool change mode A and B can have up to 8 tools
A0001.5	1013	Tool use life ends
A0001.6	1014	Confirm tool number of tool post (8, 10, 12)
A0001.7	1015	Confirm tool number of tool post (6, 8, 10, 12)
A0002.0	1016	Alarm for protection door not closed
A0002.1	1017	Alarm for pressure low
A0002.3	1019	Can’t release chuck when spindle rotating
A0002.4	1020	Alarm for disabled clamping in-position signal when spindle rotating
A0002.5	1021	Can’t start spindle when disabled clamping in-position signal
A0002.6	1022	Can’t start spindle when chuck is released
A0003.0	1024	Can’t execute M12 or M13 with disabled chuck function
A0003.1	1025	Have not checked chuck clamping/releasing in-position signal
A0003.3	1027	Tool pot is not released or lock signal is not cancelled
A0003.4	1028	Has not found the target tool number
A0003.5	1029	Has not received tool pot stop and locking start signal
A0004.0	1032	Illegal M code
A0004.1	1033	The current spindle is not analog, and Jog function cannot be executed
A0004.2	1034	M03,M04 specification are mistaken
A0004.3	1035	M63,M64 specification are mistaken
A0004.4	1036	Spindle gear shift time is too long
A0004.5	1037	Spindle speed/position control switch time is too long
A0004.6	1038	The 2nd spindle speed/position control switch time is too long
A0004.7	1039	Spindle position control does not permit orientation
A0005.0	1040	Spindle orientation time is too long
A0005.1	1041	Alarm for the abnormal spindle servo or frequency converter for abnormality
A0005.2	1042	Alarm for the abnormal 2nd spindle servo or frequency converter
A0005.3	1043	Forbid clamping the spindle when the spindle rotating or feeding
A0005.4	1044	Spindle is clamped not to rotate or feed
A0007.0	2000	Hydraulic motor is not started
A0007.1	2001	Safety door has been opened
A0007.3	2003	Alarm for the tool pot unclocked
A0007.5	2005	Regulation is invalid when feedrate override is fixed to 100%
A0007.6	2006	Override on the panel is disabled when external override is enabled
A0007.7	2007	Program does not run when external pause switch is OFF
A0008.0	2008	Axis does not move when manual override is 0
A0008.1	2009	Disabled feed enabling knob input, fail to operate the program
A0008.2	2010	Disabled spindle enabling knob input, fail to rotate the spindle
A0008.3	2011	Workpiece number has been reached to the setting numbers
A0008.4	2012	Spindle is on the position control, which can not be rotated the spindle.