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

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

Foreword

Dear user,

We are really grateful for your patronage and purchase of this GSK980MDi Plus Milling CNC system made by GSK CNC Equipment Co., Ltd.

The user manual describes the programming, operation, installation and connection of this GSK980MDi Plus Milling CNC system. Please read it carefully before operation in order to get the safe and effective working.

Warning



This system can only be operated by authorized and qualified personnel as improper operations may cause accidents.

Please carefully read this user manual before use!

Note: The power supply installed on (in) the cabinet is exclusive to GSK'S CNC systems.

The power supply form is forbidden to be used for other purposes. Otherwise, there may be extreme danger!

This user manual shall be kept by final user.

Notes

■ Delivery and storage

- **Packing box over 6 layers in pile is unallowed.**
- **Never climb the packing box, neither stand on it, nor place heavy objects on it.**
- **Do not move or drag the product by the cables connected with it.**
- **Forbid collision or scratch to the panel and displayer.**
- **Packing box should be protected from damping, insolation and raining.**

■ Open packing box to check

- **Ensure things in packing box are the required ones.**
- **Ensure the product is not damaged in delivery.**
- **Ensure the parts in packing box are in accordance to the order.**
- **Contact us in time if the product type is inconsistent with the order, there is short of accessories, or product damage in delivery.**

■ Connection

- **Only qualified persons can connect the system or check the connection.**
- **The system must be earthed, its resistance must be less than 4 Ω and the ground wire cannot be replaced by zero wire.**
- **Connection must be correct and firm to avoid the product to be damaged or other unexpected result.**
- **Connect with surge diode in the specified direction to avoid the damage to the system.**
- **Switch off power supply before pulling out plug or opening electric cabinet.**

■ Troubleshooting

- **Switch off power supply before troubleshooting or changing components.**
- **Troubleshoot and then startup the system when there is short circuit or overload.**
- **Do not switch on or off it frequently and an interval is 1 minute at least after the system is powered on again.**

Announcement !

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

Warning !

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

Cautions !

- **Functions, technical indexes described in this user manual are only for the system. Actual functions and technical performance of machine tool with this CNC system are determined by machine builder's design, so refer to its user manual.**
- **The system is employed with integrated machine control panel and the keys on machine control panel are defined by PLC program. Functions of keys in this user manual are for standard PLC program. Please notice it!**
- **Refer to user manual from machine manufacturer about functions and meanings of keys on machine control panel.**

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

I Programming

GSK980MDi Plus CNC Technical Specification, Product Type, Command and Program Format

II Operation

GSK980MDi Plus CNC Operation Use

III Installation

GSK980MDi Plus CNC Installation, Connection and Setting

Appendix

CNC Ladder Function Allocation, Alarm Message Table

Safety Responsibility

Manufacturer's safety responsibility

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

——The manufacturer should be responsible for the CNC system and the accessories.

——The manufacturer should be responsible for the message and the suggestion for the user.

User's safety responsibility

——The user should study and train the system safety operation, master the safety operation content.

——The user should be responsible for the danger caused by increasing, changing or modifying the CNC system, the accessories by itself.

——The user should be responsible for the danger because of the mistaken operation, regulation, maintenance, installation and storage.

Contents

I Programming

Chapter 1 Programming Fundamentals 3
1.1 Introduction 3
Chapter 2 MSTF Code 7
2.1 M Codes (Miscellaneous Function) 7
2.1.1 End of Program (M02) 7
2.1.2 Rigid Tapping Designation M29 7
2.1.3 End of Run (M30) 7
2.1.4 Subprogram Call (M98) 7
2.1.5 Return from Subprogram (M99) 8
2.1.6 Macro Program Call (M9000~M9999) 9
2.1.7 Program Stop M00 9
2.2 Spindle Function 9
2.3 Tool Function 9
2.4 Feeding Function 10
2.4.1 Cutting Feed (G94/G95, F Command) 10
Chapter 3 G Command 13
3.1 G Command Brief 13
3.1.1 Modal, Non-modal and Initial State 15
3.1.2 Examples 15
3.2 Rapid Positioning G00 16
3.3 Linear Interpolation G01 17
3.4 Arc and Helical Interpolation G02, G03 18
3.5 Dwell G04 21
3.6 Forward Command G5.1 21
3.7 Cylindrical Interpolation G07.1 22
3.8 Programmable Data Input G10 25
3.8.1 Modifying Tool Compensation Data 26
3.8.2 Modyfing a Workingpiece Coordinate System 26
3.8.3 Modifying an Additional Workpiece Cooridnate System 26
3.9 Polar Coordinate Command (G15, G16) 26
3.10 Plane Selection Command G17, G18 and G19 29
3.11 Conversion of Inch and Metric G20 and G21 30
3.12 Reference Point Return G28 30
3.13 Return from Reference Point G29 32
3.14 The 2nd, 3rd and 4th Reference Point Return G30 33
3.15 Skip Function G31 34
3.16 Tool Nose Radius Compensation C (G40, G41 and G42) 35
3.17 Tool Length Compensation (G43, G44, G49) 38
3.18 Scaling G50, G51 41
3.19 Programmable Mirror Image G50.1, G51.1 43
3.20 Setting Local Coordinate System G52 44
3.21 Select Machine Coordinate System G53 47
3.22 Workpiece Coordinate System G54~G59 49
3.23 Additional Workpiece Coordinate System G54.1 50
3.24 Coordinate System Rotation G68, G69 51
3.25 Compound Cycle Command 55
3.25.1 High-speed Peck Drilling Cycle G73 55
3.25.2 Left-handed Tapping Cycle G74 56

3.25.3 Finish Boring Cycle G76 57
3.25.4 Drilling Cycle, Spot Drilling Cycle G81 58
3.25.5 Drilling Cycle, Counter Boring Cycle G82 59
3.25.6 Peck Drilling Cycle G83 59
3.25.7 Tapping Cycle G84 60
3.25.8 Boring Cycle G85 61
3.25.9 Boring Cycle G86 62
3.25.10 Back Boring Cycle G87 63
3.25.11 Boring Cycle G88 64
3.25.12 Boring Cycle G89 65
3.25.13 Groove Rough Milling Inside the Round G110/G111 65
3.25.14 Fine-milling Cycle Inside Full Circle G112/G113 68
3.25.15 Fine-milling Cycle Outside Circle G114/G115 69
3.25.16 Roughing Cycle Outside Circle G116/G117 71
3.25.17 Plane milling G126/G127 73
3.25.18 Rectangular Roughing G132/G133 74
3.25.19 Rectangle Groove Rough-milling G134/G135 76
3.25.20 Rectangle Groove Inner Fine-milling Cycle G136/G137 79
3.25.21 Finish-milling Cycle Outside the Rectangle G138/G139 80
3.26 Continous Drilling 81
3.26.1 Line Series Punch (L function) 82
3.26.2 Line Series Punch (K function) 83
3.26.3 Rectangle Series Punch (G140/G141) 84
3.26.4 Arc Serial Punching (G142/G143) 85
3.26.5 Chessboard continuous drilling (G144) 86
3.27 Absolute and Incremental Commands G90 and G91 87
3.28 Workpiece Coordinate System Setting G92 88
3.29 Feed per min. G94, Feed per rev. G95 88
3.30 G98, G99 89
3.31 Chamfering Function 90
3.31.1 Linear Chamfering 90
3.31.2 Arc Chamfering 90
3.31.3 Limit 91
3.32 Rigid Tapping 91
3.32.1 Rigid Tapping 92
3.32.2 Peck Rigid Tapping 93
3.32.3 Address Explanation 94
3.32.4 Technic Specification 95
3.32.5 Specify a Rigid Tapping Mode 96
3.32.6 The Cancellation of Rigid Tapping Mode 96
3.32.7 F and G Signals 97
3.32.8 Alarm Message 98
3.32.9 Program Example 98

Chapter 4 Macro Programs 99

4.1 Macro Call 99
4.2 Variables 103
4.2.1 Null Variables 106
4.2.2 Local Variables 106
4.2.3 System Variables 107
4.3 Arithmetic and Logic Operation 110
4.3.1 Traditional Format 111
4.3.2 Macro Statement 114
4.3.3 Priority of Operations 116
4.3.4 Bracket Nesting 116
4.4 Branch and Repetition 116
4.4.1 Unconditional Branch (GO TO statement) 117
4.4.2 Conditional Branch (IF statement) 117
4.4.3 Conditional Expression 117

4.4.4 Logical Expression....118
4.4.5 Repetition（WHILE Statement）....119
Chapter 5 Small Line Segment Foresight Function....121
5.1 Overview of Small Line Segment Foresight Function....121
5.2 Enabling Method for Small Line Segment Foresight....121
5.2.1 G5.1 Command Enable....121
5.2.2 Enable Parameter Settings....122
5.3 Forward Looking Parameters for Small Line Segments....122

II Operation

Chapter 1 Operation Mode And Display....125
1.1 Panel Division....125
1.2 Summary of Operation Mode....126
Chapter 2 Manual Operation....127
2.1 Coordinate Axis Moving....127
2.1.1 Manual Feed....127
2.1.2 Manual Rapid Traverse....127
2.2 Feedrate Override Adjustment....128
2.2.1 Manual Feedrate Override Adjustment....128
2.2.2 Manual Rapid Override Adjustment....128
2.2.3 Spindle Override Adjustment....128
Chapter 3 MDI Operation....129
3.1 Blocks Input....129
3.2 Blocks Execution....130
Chapter 4 Auto Operation....131
4.1 Auto Run....131
4.1.1 Selection of a Program To Be Run....131
4.1.2 Program Start....132
4.1.3 Stop of Auto Run....132
4.1.4 Auto Run From an Arbitrary Block....133
4.1.5 Adjustment of Feedrate Override, Rapid Override....133
4.1.6 Spindle Override Adjustment....134
4.2 DNC Running....134
4.3 Running State....134
4.3.1 A Single Block Execution....134
4.3.2 Dry Run....134
4.3.3 Machine lock....135
4.3.4 MST Lock....135
4.3.5 Block Skip....135
4.3.6 Optional Stop....135
Chapter 5 Machine zero Return Operation....137
5.1 Machine Zero....137
5.2 Machine Zero Return Steps....137
Chapter 6 Find The Workpiece Centre....139
6.1 Operation of Finding the Center Point of the Workpiece....139
6.1.1 Scope of Application....139
6.1.2 How to Find Part's Center Point....139
6.1.3 Find Center Point of Rectangle Part....140
6.1.4 Find Center Point of Arc Part....143
6.2 Z-axis Tool Alignment Operation....146
6.2.1 Basic Methods....146

6.2.2 Measurement Input Tool Alignment 147
Chapter 7 Drilling And Tapping Current Detection 149
7.1 Functional Description 149
7.2 Specify Current Detection Through Programming 149
7.3 Set the Current Limit Value Through Parameters 150
7.3.1 Drilling Set Current Exceeds the Limit 150
7.3.2 Tapping Set Current Exceeding Limit Value 150
Chapter 8 DXF Drill And Tap 151
8.1 Transfer and Read DXF File 151
8.2 File Drill Parameter Setting 152
8.2.1 Drill Setting 153
8.2.2 Slot Settings 154
8.2.3 DXF File Process Modify 155
8.2.4 Coordinate System Setting 158
8.3 Program Generating and Parameter Control 159
Chapter 9 Data Setting,Backup And Restore 163
9.1 Data Setting 163
9.1.1 Switch Setting 163
9.2 The Password Setting and Alteration 163
9.2.1 Entry of Operation Level 164
9.3 Macro Variable Note 165
9.3.1 Macro Variable Note Page 166
9.3.2 Note List 166
9.3.3 Macro Variable Note File 167
Chapter 10 Document Management 169
10.1 Document Management Page 169
10.2 Often Used File Operation Function Introduction 170
10.2.1 File Extension and Return 170
10.2.2 File Copy (Current Directory in C Disk ←→ Current Directory in U Disk) 171
10.2.3 System Update 172
Chapter 11 Resume And Backup 173
11.1 Backup Operation Example 173
11.2 Resume Operation Example 174
Chapter 12 Waveform Analysis 177
12.1 Rigid Tapping Test 177
12.1.1 Generate Test Program 177
12.1.2 Sampling 178
12.1.3 Velocity Waveform 179
12.1.4 Error Waveform 179
12.2 Circularity Test 180
12.2.1 Generate Test Program 180
12.2.2 Sampling 181
12.2.3 Velocity Waveform 181
12.2.4 Roundness Error 182
Chapter 13 Communication 185
13.1 Brief About GSK980MDi Plus Communication Software GSKComm 185
13.1.1 PC System Required 185
13.1.2 Software Interface 185
13.1.3 Transfer File (CNC-PC) 186
13.1.4 Send File PC-CNC 188
13.1.5 Part Program Managing 190
13.2 Third Party Communication Software Serial Port Transmission Program 190
13.2.1 Serial Port Settings 190

13.2.2 Transmission Program Operation 191
13.2.2.1 Input Program 191
13.2.2.2 Output Programme 191
13.2.3 Precautions 192

III Installation

Chapter 1 Installation Layout 195
1.1 GSK980MDi Plus Connection 195
1.2 GSK980MDi Plus Installation 195
1.2.1 GSK980MDi Plus External Dimensions 195
1.2.2 Measures Against Interference 195
Chapter 2 Interface Signals 197
2.1 Connection to Drive Unit 197
2.2 Being Connected with Spindle Encoder 197
2.2.1 Spindle Encoder Interface Definition 197
2.2.2 Signal Explanation 198
2.2.3 Spindle Encoder Interface Connection 198
2.3 MPG Connection 199
2.3.1 MPG Interface Definition 199
2.3.2 Signal Explanation 199
2.4 Analog Spindle Interface 201
2.4.1 Analog Spindle Interface Definition 201
2.4.2 Inverter Connection 201
2.5 GSK980MDi Plus being Connected with PC 202
2.5.1 Communication Interface Definition 202
2.5.2 Communication Interface Connection 202
2.6 Network Interface 203
2.7 I/O Interface Definition 203
2.8 System Specific Input/Output Interface 206
2.8.1 CN78 I/O Interface 206
2.8.2 CN66 External Spindle Magnification Interface 207
2.8.3 CN67 External Input Magnification Interface 207
2.9 GSK980MDi Plus Operation Panel Interface 207
2.9.1 Host Communication 208
2.9.2 Hand Pulse Interface 208
2.9.3 Expand I/O 209
Chapter 3 Parameter 211
3.1 Parameter Description (Sorted in Order) 211
Chapter 4 Machine Debugging 227
4.1 ESP & Limit 227
4.1.1 ESP & Limit Series-Connection 227
4.1.2 Separately Connect of ESP & Limit 227
4.2 Drive Unit Configuration 228
4.3 Backlash Offset 228
Chapter 5 Diagnosis Message 231
5.1 CNC Axis Motion Status and Data Diagnostic Information 231
Chapter 6 Screw-Pitch Compensation 233
6.1 Screw-Pitch Compensation 233
6.2 Compensation Value 233
6.3 Cautions for Offset Setting 233
6.4 Examples of Offset Parameters Setting 233
6.5 Import REN Format Screw File 237

6.5.1 Import Page....237
6.5.2 Import Operation....238
6.5.3 REN Example of Screw Repair File....239

Appendix

Appendix 1 GSK980MDi Plus Series External Dimensions....243
1.1 GSK980MDi Plus-H External Dimensions....243
1.2 GSK980MDi Plus-V External Dimensions....244
Appendix 2 Diagram of I/O Deconcentrator....245
2.1 IOR-04T / IOR-44T....245
2.2 IOR-44F....246
2.3 MCT01B....246
2.4 MCT06-R....248
2.5 MCT07....248
2.6 IOR-20TP(Bus to Pulse I/O Unit)....249
Appendix 3 Explanations of Rigid tapping....253
3.1 Setting of Spindle Electronic Gear Ratio....253
3.2 Related Parameter Setting....254
Appendix 4 Analog Spindle Function....255
4.1 T Type Gear Shift....255
4.2 M Type Gear Shift....257
Appendix 5 Explanations for Standard Ladder Function....261
5.1 Address Definition....261
5.2 Function Configuration....264
5.2.1 Spindle Rotation Control....264
5.2.2 The 2nd Spindle Rotation Control....266
5.2.3 The 3nd Spindle Rotation Control....268
5.2.4 T- type Spindle Auto Gear Shift....269
5.2.5 M-type Spindle Auto Gear Shift....271
5.2.6 Spindle Switching Volume Control....273
5.2.7 Cycle Start and Feed Hold....274
5.2.8 Coolant Control....274
5.2.9 Lubrication Control....275
5.2.10 Block Skip....276
5.2.11 Machine Lock....277
5.2.12 Auxiliary Lock....277
5.2.13 Single Block....277
5.2.14 Dry Run....277
5.2.15 Optional Stop....278
5.2.16 Stroke Limit and Emergency Stop....278
5.2.17 Tri-colour Indicator....280
5.2.18 Reset and Cursor Return....280
5.2.19 Rigid Tapping....281
5.2.20 Spindle Exact Stop....281
5.2.21 External MPG Control....282
5.2.22 Cs Axis Switching....282
5.2.23 Index Table Tightness Control....283
5.2.24 Safe Gate Function....285
5.2.25 Holding Brake Control....285
5.2.26 Spindle Releasing/Clamping Tool....286
5.2.27 Pot Tool Magazine....286
5.2.28 Turret Tool Magazine....294
5.2.29 Turret Knife Magazine....300

5.2.30 M10/M11 Turn Control 303
5.2.31 Spindle Ratio 303
5.2.32 Work Light 304
5.2.33 Chip Removal 304
5.2.34 Addition Panel Signal 305
5.2.35 Handwheel Trial Cutting 306
5.2.36 Workpiece Blowing 306

I Programming

Chapter 1 Programming Fundamentals

1.1 Introduction

Being upgraded product of GSK980MDi Plus, GSK980MDi Plus CNC turning is the better choice.

The updated development GSK980MDi Plus Turning Machine CNC System made by GSK CNC EQUIPMENT CO., LTD. is the new product based upon the GSK980MDi Plus updated software/hardware, which owns the vertical and horizontal structures. GSK980MDi Plus uses the GSK-Link 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.

Characteristics：

- Five axes control (X, Y, Z ,4th and 5th); 3 axes linkage; optional interpolation precision (1μm/0.1μm); maximum speed 60m/min; optional axis types (linear axis or revolving axis) for the 4th and 5th axes; CS axis control available for the 4th and 5th axes.
- Electronic gear ratio: (1～32767):(1～32767)
- Screw-pitch error compensation, backlash compensation, tool length compensation, tool abrasion compensation and tool nose radius compensation.
- Embedded with PLC can be downloaded to CNC from PC.
- DNC function supports for real-time program transmission for machining.
- Be compatible with G commands of GSK980MC, GSK928MA, GSK980MD, GSK980MDc, executing the cycle machining including drilling/boring, circle grooving/rectangular grooving, full-circle/rectangular finishing, circular truncated cone/rectangular roughing, linear/rectangular continuous drilling, supporting spiral interpolation, cylinder interpolation and polar coordinate command performing.
- Spindle encoder tapping and rigid tapping can be detected during tapping cycle, so that high precision machining can be performed.
- Supporting metric/inch programming with scaling on/off, programmable image and automatic chamfering function.
- Support statement macro command program, macro program call with parameters.
- Supporting multi-additional coordinate system and local coordinate system.
- Supporting PLC axis control function.
- Chinese, English, Russian and Spanish display selected by the parameters.
- Full screen program editing; 40MB program capacity for storing up to 40000 of part programs.
- USB data communication; CNC system upgrading, machining programs reading through U disk and bidirectional transfer between CNC and U disk.
- 2-channel 0V～10V analog voltage output.
- 41 input interfaces/36 output interface meeting logical control requirements for disc tool magazine and bamboo tool magazine.
- Alarm log; multi-level passwords for equipment maintenance and management.
- Bidirectional transfer between CNC and CNC, CNC and PC; upgrade of CNC software and PLC programs.
- The installation dimensions and the electric ports are compatible with GSK980MD，GSK980MDc、GSK980MC.

Specifications:

Motion control	Controlled axes: five axes (X,Y,Z,4th and 5th); (for the 4th and 5th axes) optional axis types (linear axis or revolving axis) and CS contouring control available.
	Interpolation functions: linear interpolation (for X, Y, Z, 4th and 5th axes); helical interpolation (for X, Y and Z axes); circular interpolation (for arbitrary 2 axes).
	Position command range: -99999999～99999999; least command increment: 1μm/0.1μm. (selected via parameters)
	Electronic gear ratio: command multiplier 1～32767，command frequency divisor 1～32767.
	Rapid traverse speed: maximum 60000mm/min. Rapid traverse override: F0, 25%, 50%, 100% four levels real-time tuning.
	Cutting feedrate: maximum 15000mm/min (feed per min.) or 500mm/r. (feed per rotation) Feedrate override: 0～150% sixteen-level real-time tuning.
	Manual feedrate: 0～1260mm/min sixteen-level real-time tuning.
	MPG feed: 0.001, 0.010, 0.100,1.000mm four gears.
	Acceleration/deceleration type: S-type for rapid traverse; exponential-type for cutting feed.
	Automatic chamfering.
G Code	82 kinds of G codes：G00, G01, G02, G03, G04, G10, G11, G17, G18, G19, G20, G21, G28, G29, G30, G31, G40, G41, G42, G43, G44, G49, G50,G51,G50.1,G51.1,G52,G53,G54.1,G54, G55, G56, G57, G58, G59, G65, G66, G67, G73, G74, G80, G81, G82, G83, G84, G85, G86, G88, G89, G90, G91, G92, G94, G95, G98, G99, G110, G111, G112, G113, G114, G115, G116,G117, G132,G133,G134, G135, G136, G137, G138, G139, G140, G141, G142, G143.
Macro command	31 kinds of arithmetic, logical operations and skip can be achieved by macro command G65.
	Macro statement command. eg: IF, WHILE, GOTO.
Operation mode	Seven operation modes: EDIT, AUTO, MDI, DNC, MACHINE ZERO, MPG/STEP and MANUAL.
Tapping	Tapping function: lead 0.001～500mm or 0.06～25400 pitch/inch
	Encoder tapping: settable line number of encoder（0 or100p/r～5000p/r）; no detect for spindle encoder (when the line number is set to 0)
	Rigid tapping: by rotary axis
	Drive ratio between encoder and spindle:（1～255）:（1～255）.
Precision compensation	Backlash compensation: 0～2.000mm.
	Pitch error compensation: 255 compensation points per axis; compensation amount of each point: ±0.255mm.
	Tool compensation: 32 groups tool length compensation, tool wear compensation, cutter compensation C.
M command	Special M commands (redefinition unallowed): M02,M29, M30, M98, M99,M9000～M9999. Other M □□ commands are defined or disposed by PLC program.
	M commands defined by standard PLC program: M00, M03, M04, M05 M08, M09, M10, M11, M32, M33
T command	tool number T01～T32 (32 numbers at most); manual tool change or auto-tool change selected by the parameters; auto tool change sequence set by PLC program.
	Tool life management; 32 groups, 8 kinds/groups of tool life management data.
Spindle speed control	Speed switching value control: S □□ command is defined or disposed by PLC program; the standard PLC programs S1, S2, S3 and S4 directly output; The output of S1,S2, S3, and S4 are closed by S0.
	Speed analog voltage control: the spindle speed per minute commanded by S codes; output 0～10V voltage to spindle converter; spindle stepless speed changing supports 4 spindle mechanical gears.
PLC function	9 kinds of basic commands; 23 kinds of function commands; 2-level PLC program involving up to 5000 steps (2μs processing time for each step). 8ms refresh cycle for the first level program; Ladder diagram edit software and communication software downloadable.
	Integrated machine panel: 44 points input (key), 44 points output (LED) . Basic I/O: 41 points input/ 36 points output.

Display interface	Displayer: 800×600 lattice, 8.4" wide-screen multi-color LCD.
	Display modes: Chinese, English, Russian, Spanish display selected by parameters; machining path displayable.
Program edit	Capacity: 40MB for up to 40000 part programs; custom macro program call; 4 nesting-levels of subprogram
	Edit modes: full-screen editing; absolute/incremental programming.
USB	CNC system upgrade.
	Part programs reading in USB.
	Bidirectional files transfer between CNC and USB (including programs, parameters, PLC backup and recovery) .
Clock display	Clock, date and week display.
Serial Communication	Bidirectional transfer between CNC and PC, CNC and CNC (involving programs, parameters, tool compensation data); download and upgrade of system software and PLC program serial ports.
Matching drive unit	AC servo or step drive device by using the pulse+direction signal input. (DA98 or DY3 series)

G Code Table

Code	Function	Code	Function	Code	Function
G00	Positioning (rapid traverse)	*G54	Workpiece coordinate system 1	*G94	Feed per minute
*G01	Linear interpolation	G55	Workpiece coordinate system 2	G95	Feed per rotation
G02	Circular/helical interpolation (CW)	G56	Workpiece coordinate system 3	*G98	Return to initial plane in canned cycle
G03	Circular/helical interpolation (CCW)	G57	Workpiece coordinate system 4	G99	Return to R point in canned cycle
G04	Dwell, exact stop	G58	Workpiece coordinate system 5	G110	Inner circle groove roughing (CCW)
G10	Tool life management	G59	Workpiece coordinate system 6	G111	Inner circle groove roughing (CW)
G11	Tool life management end	G65	Macro program/ macro code	G112	Inner circle finishing (CCW)
*G17	XY plane selection	G66	Macro program modal call	G113	Inner circle finishing (CW)
G18	ZX plane selection	*G67	Macro program modal call cancel	G114	Circular outer finish milling (CW)
G19	YZ plane selection	*G69	Coordinates rotation cancel	G115	Outer circle finishing (CCW)
G20	Inch input	G73	High-speed peck drilling	G116	Outer roughing (CCW)
G21	Metric input	G74	Counter tapping cycle	G117	Outer roughing (CW)
G28	Reference position return	G76	Finishing boring cycle	G132	Rectangular roughing (CCW)
G29	Return from reference position	*G80	Canned cycle cancel	G133	Rectangular roughing (CW)
G30	2nd, 3rd, 4th, reference position return	G81	Drilling cycle (spot drilling cycle)	G134	Rectangular groove roughing (CCW)
G31	Skip function	G82	Drilling cycle (stepped hole boring cycle)	G135	Rectangular groove roughing (CW)
*G40	Cutter Compensation cancel	G83	Peck drilling cycle	G136	Rectangular groove inner finishing (CCW)
G41	Cutter compensation left	G84	Tapping cycle	G137	Rectangular groove inner finishing (CW)
G42	Cutter compensation right	G85	Boring cycle	G138	Rectangular outer finishing (CCW)
G43	Tool length compensation + direction	G86	Drilling cycle	G139	Rectangular outer finishing (CW)
G44	Tool length compensation - direction	G89	Boring cycle	G140	Rectangular continuous drilling (CW)
G49	Tool length compensation cancel	G89	Boring cycle	G141	Rectangular continuous drilling (CCW)

Code	Function	Code	Function	Code	Function
*G50	Scaling cancel	*G90	Absolute programming	G142	Arc continuous drilling (CW)
G51	Scaling ON	G91	Incremental programming	G143	Arc continuous drilling (CCW)
*G50.1	Programming image cancel	G92	Coordinate system setting		

Note: mark " * " means initial state.

PLC Codes List

Code	Function	Code	Function	Code	Function
LD	Normal open contact read	SET	Setting	SPE	Subprogram end
LDI	Normal closed contact read	RST	Resetting	ADDB	Binary addition
OUT	Output coil	CMP	Comparison setting	SUBB	Binary subtraction
AND	Normal open contact in series	CTRC	Counter	ALT	Alternative output
ANI	Normal closed contact in series	TMRB	Timer	DIFU	Differential up
OR	Normal open contact in parallel	CODB	Binary code transformation	DIFD	Differential down
ORI	Normal closed contact in parallel	ROTB	Binary rotational control	MOVE	Logical AND
ORB	Serial block in parallel	MOVN	Data copy	PARI	Parity check
ANB	Parallel block in series	DECB	Binary decode	LBL	Program skip numbering
END1	First level program end	JMPB	Jump	CALL	Subprogram call
END2	Second level program end	SP	Subprogram numbering		

Chapter 2 MSTF Code

2.1 M Codes (Miscellaneous Function)

2.1.1 End of Program (M02)

Format: M02

Function: The M02 code is executed in the Auto mode. The automatic run is ended after the other codes of current block are executed; the cursor stops in the block in which the M02 is located and does not return to the head of the program. If the program is to be executed again, the cursor should return to the beginning of the program.

Besides the above-mentioned functions processed by CNC, the functions of code M02 also can be defined by the PLC ladder diagram. The function defined by standard ladder diagram can be: the current input state of CNC is not change after the code M02 is executed.

2.1.2 Rigid Tapping Designation M29

Format: M29

Function: In auto mode, after the execution of M29, the G74, G84 that followed is processed as rigid tapping codes.

2.1.3 End of Run (M30)

Format: M30

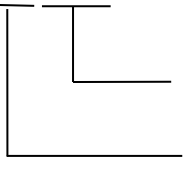
Function: If M30 command is executed in the Auto mode, the automatic run is ended after the other commands of current block are executed; the system cancels the tool nose radius compensation and the cursor returns to the beginning of the program when the workpieces number is added by one (whether the cursor returns to the head of the program is determined by parameters).

The cursor does not return to the beginning of the program when the BIT4 of parameter №810 is set to 0; when it is set to 1, the cursor returns to the beginning of the program as soon as the program execution is finished.

Besides the above-mentioned functions processed by CNC, the functions of code M30 also can be defined by the PLC ladder diagram. The function defined by standard ladder diagram can be: turn OFF the M03, M04 or M08 output signal after the M30 command is executed, and meanwhile output M05 signal.

2.1.4 Subprogram Call (M98)

Format: M98 L○○○○P□□□□



The called subprogram No.（0000～9999）.The leading zero of subprogram can be omitted when the called times are not input; the subprogram No. should be 4 digits when the called times is input;

Called times（1-9999，calling for once, the input can be omitted

Function: In Auto mode, when the M98 is executed, the subprogram specified by P is called after the

execution of other codes in the current block. The subprogram can be performed 9999 times at most. M98 cannot be performed in MDI, or an alarm will occur.

Note: U command specifying arc chamfering cannot be with M98 in the same block. The linear chamfering is invalid when it and M98 are in the same block, and it is taken as the cycle times of calling subprograms.

2.1.5 Return from Subprogram (M99)

Format: M99 P○○○○

The block No. (0000～9999) when return to main program is executed, the leading zero can be omitted.

Function: (in subprogram) as the other commands of current block are executed, the block specified by P is performed continuously when the main program is returned. The next block is performed continuously by calling current subprogram of M98 command when returning to the main program; because of the P is not given. If the main program is ended by using the M99 (namely, the current program is not called by other programs for execution), the current program will be run circularly. So, the M99 command is disabled in MDI.

Example: Fig. 2-1 shows that the execution route of the subprogram is called (the P command within M99). Fig. 2-2 shows that the execution route of the subprogram is called (the P command is not in M99.

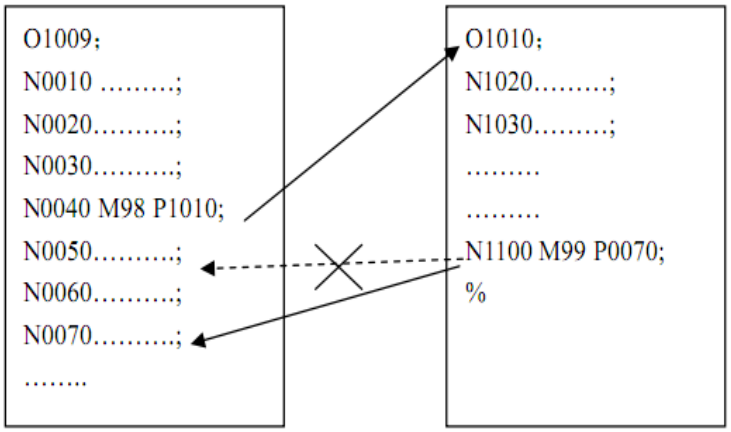


Fig.2-1

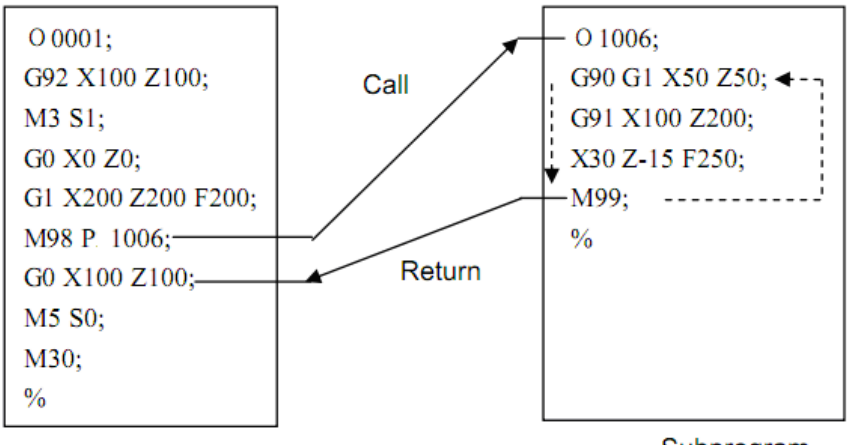


Fig. 2-2

This GSK980MDi Plus can calls quadruple subprogram, namely, the other subprogram can be called from the subprogram. (See Fig. 2-3)

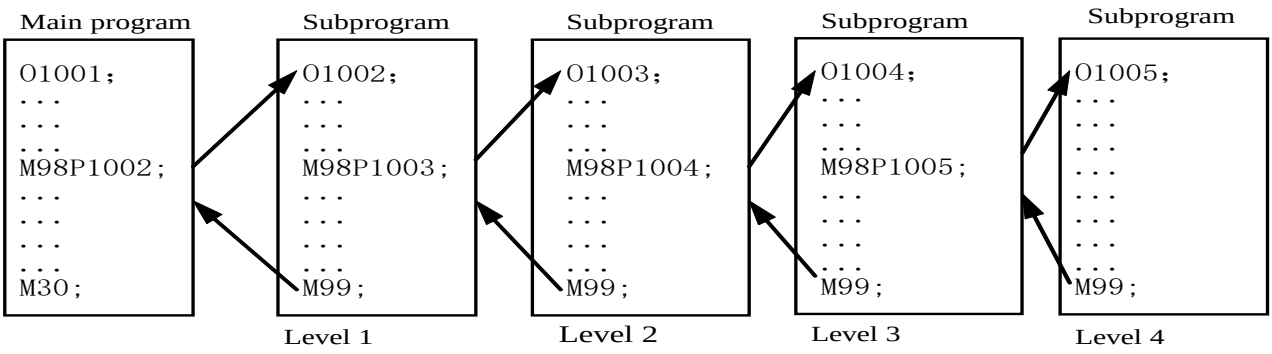


Fig. 2-3 subprogram nestifications

2.1.6 Macro Program Call (M9000~M9999)

Format : M □□□□

9000～9999

Command function: Call the macro program which is corresponded by the command value （O9000～O9999）.

Macro program: Program 09000~09999 is special space obligated for the machine tool manufacturer for using editing and achieving special function subprogram, which is called macro program. Two-level operation authority is needed when editing the program 09000~09999, the user can not modify or run the

2.1.7 Program Stop M00

Format: M00

Command function: After M00 is executed, program run stops, “Pause” appears, the cycle start key is pressed to continuously run the program.

2.2 Spindle Function

The spindle speed is analog voltage control when the BIT4 of current bit parameter is set to 1

Format: S oooo

01～32（leading zero can be omitted）

spindle speed analog voltage control command address

Command function: The CNC outputs 0~10V analog voltage to control the spindle servo or inverter for achieving the stepless speed regulating of the spindle when the spindle speed is set. The S command value is not memorized when the power is turned off, and then the parameter recovers to 0 when the power is turned on.

2.3 Tool Function

GSK980MDi Plus tool function is controlled by ladder, and compiling ladder can logic controlling dial

tool magazine and turret tool magazine.

Format: TOO
└── 01～32（leading zero can be omitted）

Command function: Set the tool number for use。

2.4 Feeding Function

2.4.1 Cutting Feed (G94/G95, F Command)

Format:G94F_; (F0001~F8000, leading zero can be omitted, for feedrate per minute, mm/min).

Command function: The cutting federate is specified by mm/min, G94 is modal G command. If the current mode is G94 that it needs no G94 any more.

Format: G95F_; (F0.0001~F500, leading zero can be omitted)

Command function: The cutting feedrate is offered by the unit of mm/rev., G95 is modal G command. The G95 command can be omitted if the current mode is G95. When the CNC performs G95 F_, the cutting federate is controlled by federate command based on the multiplication of F command value (mm/rev) and current spindle speed (rev/min). The actual feedrate varies with the spindle speed. The spindle cutting feedrate per revolution is specified by G95 F_, the even cutting line can be formed on the face of workpiece. It is necessary to install spindle encoder when the G95 mode is operated.

The G94 and G95 are modal G commands at the same group, one of them is available only. The G94 is initial state G command, so, it defaults the G94 when the CNC is switched on. The following below shows the conversion formula of feed value per rev. and feed value per min:

$$F_m = F_r \times S$$

There into: F_m: feed value per minute (mm/min);

F_X: feed value per revolution (mm/r);

S: spindle speed (r/min).

Cutting feed: The CNC makes tool movement path and the path (linear or circular arc) defined by command into consistency (The circular interpolation can be performed by two axis in selected plane when it is circular arc, the helical interpolation is formed by the third axis linear interpolation linkage), by which, the CNC controls three directions movement for X axis, Y axis, Z axis ,4th axis and 5th axis at the same time. The instantaneous speed of movement path in a tangential direction is consistent with the F command value, so this is called CUTTING FEED or INTERPOLATION. The cutting feedrate is supplied by F command, which it is disassembled to each interpolation axis according to the programming path when the CNC performs the interpolation command (cutting feed).

Linear interpolation: The CNC can control the instantaneous speed in the directions of X axis, Y axis , Z axis ,4th axis and 5th axis, so the vector resultant speed in these five directions are equal to the F command value.

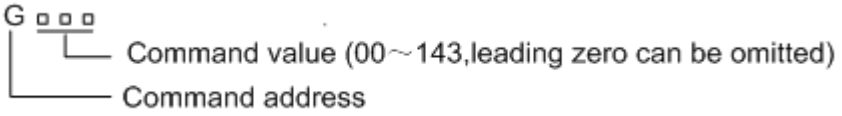
Circular interpolation (helical interpolation): Performing the arc interpolation in selected plane, the third axis performs linear interpolation, so the F value is circular interpolation speed. An interpolation of linear and circular arc has the following relation when the linear interpolation speed is f:

$$f = F \times \frac{\text{linear axis length}}{\text{circular arc length}}$$

Chapter 3 G Command

3.1 G Command Brief

The G command is composed by the command address G and the 1 to 3 digits command value after the command G. Many kinds of operations are specified such as tool movement relative to workpiece, coordinate set, etc. See Table 3-1 for G commands.



The G command words can be classified into 16 groups such as 00, 01, 02, 03, 05, 06, 07, 08, 09, 10 ,12，14，16，17 and 22. They share the same block except for 01 and 00 groups, different groups G commands can be defined at the same block. The last G command is valid when two or more same group G commands are introduced at the same block. Different G command groups without common parameter (command word) can be defined at the same block, and their functions are simultaneously valid regardless of sequence. If the G command or the optional G command other than Table 3-1 is employed, alarm occurs.

Table 3-1 G command word list

Command word	Group	Function	Remark
G04	00	Dwell, exact stop	Non-modal G command
G10		Programmable data input ON	
G11		Programmable data input OFF	
G07.1		Cylinder interpolation	
G28		Machine zero return	
G29		Return from reference point	
G30		2^{nd}, 3^{rd} and 4^{th} reference point return	
G31		Skip function	
G52		Local coordination system setting	
G53		Machine coordinate system selection	
G92		Coordinate system set	
G65		Macro	
G00 (initial G command)	01	Rapid traverse	Modal G command
G01		Linear interpolation	
G02		Circular interpolation (CW)	
G03		Circular interpolation (CCW)	
G73		Peck drilling cycle	
G74		Left-hand (counter) tapping cycle	
G80 (initial G command)		Canned cycle cancellation	
G81		Drilling cycle (spot drill cycle)	
G82		Drilling cycle (counter bore cycle)	
G83		Peck drilling cycle	
G84		Tapping cycle	

Command word	Group	Function	Remark
G85		Boring cycle	
G86		Drilling cycle	
G87		Back boring cycle	
G88		Boring cycle	
G89		Boring cycle	
G110		Circular groove inner roughing CW	
G111		Circular groove inner roughing CCW	
G112		Circular groove inner finishing CW	
G113		Circular groove inner finishing CCW	
G114		Outer finishing CW	
G115		outer finishing CCW	
G116		Outer roughing CCW	
G117		Outer roughing CW	
G132		Rectangular roughing CCW	
G133		Rectangular roughing CW	
G134		Rectangle groove roughing CW	
G135		Rectangle groove roughing CCW	
G136		Rectangle groove inner finishing CW	
G137		Rectangle groove inner finishing CCW	
G138		Rectangle outer finishing CW	
G139		Rectangle outer finishing CCW	
G17 (initial G command)	02	XY plane selection	Modal G command
G18		ZX plane selection	
G19		YZ plane selection	
G90 (initial G command)	03	Absolute programming	Modal G command
G91		Relative programming	
G94 (initial G command)	05	Feed per minute	Modal G command
G95		Feed per revolution	
G20	06	Data inch input	Modal power down memorize
G21		Data metric input	
G40 (initial G command)	07	Tool nose radius compensation cancellation	Modal G command
G41		Tool nose radius compensation left	
G42		Tool nose radius compensation right	
G43	08	Tool length offset in + direction	Modal G command
G44		Tool length offset in - direction	
G49		Tool length offset cancellation	
G140	09	Rectangle path serially punch CW	Non-modal G command
G141		Rectangle path serially punch CCW	
G142		Arc path serially punch CW	
G143		Arc path serially punch CCW	

Command word	Group	Function	Remark
G98 (initial G command)	10	Return to initial plane in canned cycle	Modal G command
G99		Return to R plane in canned cycle	
G50（initial G command）	11	Scaling OFF	Modal G command
G51		Scaling ON	
G67 (initial G command)	12	Macro program call	Modal G command
G66		Cancel macro program call	
G54 (initial G command)	14	Workpiece coordinate system 1	Modal G command
G54.1		Additional workpiece coordinate system	
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	
G68	16	Coordinate system rotation ON	Modal G command
G69（initial G command）		Coordinate system rotation OFF	
G15（initial G command）	17	Polar coordinate command OFF	Modal G command
G16		Polar coordinate command ON	
G50.1（initial G command）	22	Programmable image OFF	Modal G command
G51.1		Programmable image ON	

3.1.1 Modal, Non-modal and Initial State

The G commands can be set to 16 groups such as 00, 01, 02, 03, 05, 06, 07, 08, 09, 10 ,12 ,14,16,17and 22. Thereinto, G commands of 00 group are non-modal G commands, that of other G group are modal commands. G00, G80, G40, G49 ,G67 and G94 are initial G commands.

After the G command is executed, the function defined or status is valid until it is changed by other G command where in the same group, this kind of command is called **modal G command.** After this G command is performed and before the function defined or status is changed, this G command need not be input again when the next block performs this G command.

After the G command is performed, the function defined or status is valid for once, The G command word should be input again while every time the G command is performed, this kind of command is called **non-modal G command.**

The modal G command is valid without performing its function or state after the system is powered on, this is called **initial G command**. If the G command is not introduced after the power is turned on, then the initial G command is executed. The initial commands of GSK980MDi Plus are G00, G15, G17, G40, G50, G50.1, G54, G67, G69, G80, G90, G94, and G98.

3.1.2 Examples

Example 1

O0001;

G17 G0 X100 Y100;（Move to G17 plane X100 Y100 at the rapid traverse rate; modal command

G0 and G17 valid）

X20 Y30; （Move to X20 Y30 at the rapid traverse rate; modal command G0 can be omitted）

G1 X50 Y50 F300;（Linear interpolation to X50 Y50, feedrate is 300mm/min, modal command G1 valid)

X100; （Linear interpolation to X100 Y50, feedrate is 300mm/min; the Y coordinate is not input, use current value Y50; keep F300, the modal command G01 can be omitted）

G0 X0 Y0； （Move to X0 Y0 at the rapid traverse rate, modal G command G0 valid）

M30；

Example 2

O0002；

G0 X50 Y5；（Move to X50 Y5 at the rapid traverse rate）

G04 X4；（Time delay for 4 seconds）

G04 X5；（Time delay again for 5 seconds non-modal command G04 should be input again）

M30；

Example 3: (the first operation after the power is turned on)

O0003；

G90 G94 G01 X100 Y100 F500; （G94 feed per minute，feedrate is 500mm/min）

G91 G95 G01 X10 F0.01; （G95 feed per revolution, input the F value again）

G90 G00 X80 Y50；

M30；

3.2 Rapid Positioning G00

Format: G00 X____ Y____ Z____;

Function: X, Y and Z axes simultaneously move to end points from start at their rapid traverse rates. See Fig.3-1. Two axes move at their respective speeds, the short axis arrives at the end firstly, the long axis moves the rest of distance independently, and their resultant paths are possibly not linear.

Explanation: G00, which is initial G command;

X, Y, Z：-99999999～99999999×least input increment；

X, Y and Z axes, one of them can be omitted or all of them can be omitted. When one of them is omitted, it means that the coordinate value of start and end points are same. The start and end points share the same position when they are omitted at the same time.

Command path figure:

Tool positions at the rapid traverse rate independently for each axis. Usually, the tool path is not linear.

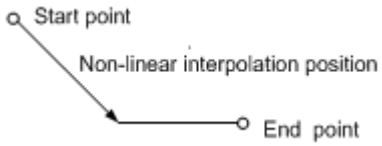


Fig. 3-1

X, Y and Z axes are separately set by the system data parameter No. 022 at their rapid traverse rate, the actual traverse rate can be modified by the rapid override keys on the machine panel.

Speed in dry run mode is limited by values set each axis's max. rapid traverse speed parameter (No.022).

The rapid traverse acceleration or deceleration time constant of X, Y and Z axes are set by the system data parameter No. 023.

Example: tool traverses from point A to point B. See Fig.3-2.

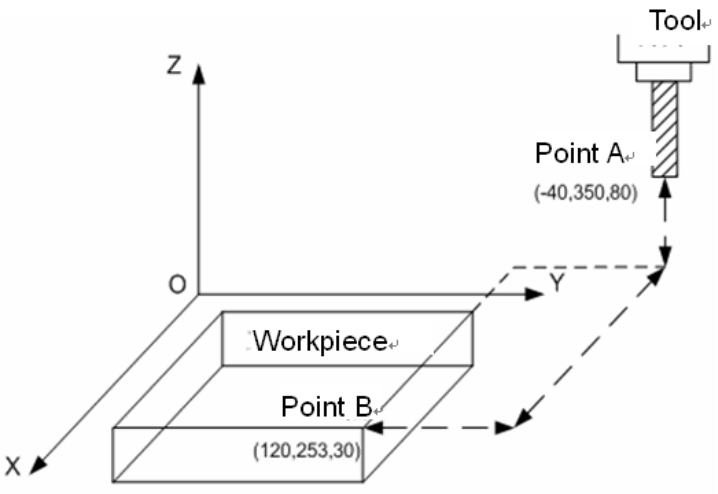


Fig.3-2

G90 G0 X120 Y253 Z30; (absolute coordinate programming)

G91 G0 X160 Y-97 Z-50; （relative coordinate programming）

3.3 Linear Interpolation G01

Format: G01 X_Y_Z_F_;

Function: Movement path is a straight line from start to end points.

Explanation: G01, which is modal G command;

X, Y, Z：-99999999～99999999×least input increment;

X, Y and Z axes which one of them can be omitted or all of them can be omitted. When one of them is omitted, it means that the coordinate value of start and end points are consistent. The start and end points share the same position when they are omitted at the same time.

F command value is vector resultant speed of instantaneous rates in X, Y and Z axes directions, the actual feedrate is the product of override and F command value;

F command value is invariable after it is performed till the new one is executed. The following G command with F command word uses the same function.

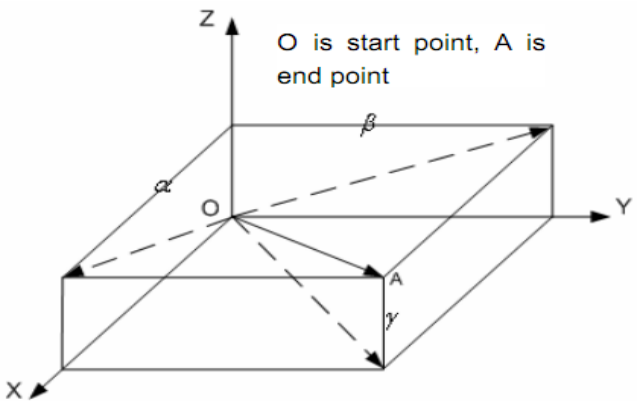
The value range is indicated as follows:

Command function	G94 (mm/min)	G95 (mm/rev)
Value range	1~15000	0~500

Command path figure:

The linear interpolation is performed from point O to point A:

G01 X α Y β Z γ F f ;



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

Fig. 3-3

The feedrate specified by F is the tool movement speed along the line. The speed of each axis is as follows:

$$\text{Speed in X axis direction}: \quad F_X = \frac{\alpha}{L} \times f$$

$$\text{Speed in Y axis direction}: \quad F_Y = \frac{\beta}{L} \times f$$

$$\text{Speed in Z axis direction}: \quad F_Z = \frac{\gamma}{L} \times f$$

Note: The F initial default value is set by data parameter No.026 when the power is turned on.

3.4 Arc and Helical Interpolation G02, G03

Format:

Circular interpolation:

Arc in the XY plane:

$$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X__ Y__ \begin{Bmatrix} R__ \\ I__ J__ \end{Bmatrix} F__$$

Arc in the XZ plane:

$$G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X__ Z__ \begin{Bmatrix} R__ \\ I__ K__ \end{Bmatrix} F__$$

Arc in the YZ plane:

$$G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Y__ Z__ \begin{Bmatrix} R__ \\ J__ K__ \end{Bmatrix} F__$$

Helical interpolation

Arc interpolation in XY plane, Z axis linear interpolation linkage;

$$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X__ Y__ Z__ \begin{Bmatrix} R__ \\ I__ J__ \end{Bmatrix} F__$$

Arc interpolation in XZ plane, Y axis linear interpolation linkage;

$$G18\begin{Bmatrix}G02\\G03\end{Bmatrix}X_\ Z_\ Y_\ \begin{Bmatrix}R_\\I_K_\end{Bmatrix}F_$$

Arc interpolation in YZ plane, X axis linear interpolation linkage;

$$G19\begin{Bmatrix}G02\\G03\end{Bmatrix}Y_\ Z_\ X_\ \begin{Bmatrix}R_\\J_K_\end{Bmatrix}F_$$

Function: Only two axes of circular interpolation can be linked for controlling tool movement along with the arc on the selected plane in any time. If the 3rd axis is specified simultaneously in linear interpolation mode, it will be linked by linear interpolation type to constitute helical interpolation. G02 movement path is CW from start to end points. G03 movement path is CCW from start to end points.

Explanation:

G02 and G03 are modal G commands.

R is an arc radius, its value：-99999999～99999999×least input increment.

When the circle center is specified by address I, J and K, they are corresponding with the X, Y and Z axes separately.

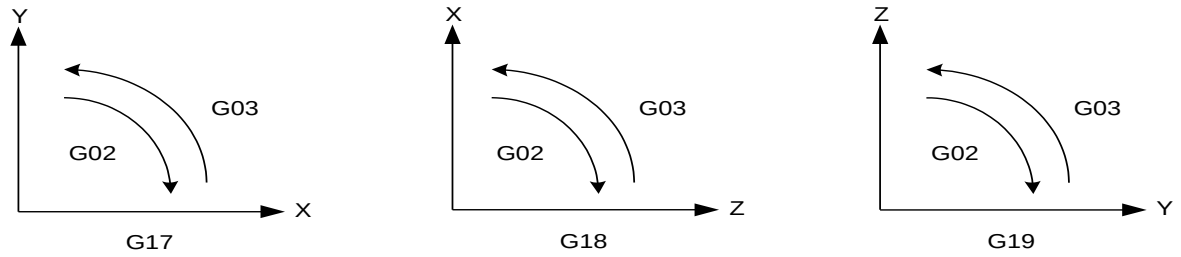
I is the difference between the center point and the arc start point in the X axis direction, I= center point coordinate X- X coordinate of arc start point; its value：-99999999～99999999×least input increment.

J is the difference between the center point and the arc start point in the Y axis direction, J=center point coordinate Y- Y coordinate of circle arc start point; its value：-99999999～99999999×least input increment .

K is the difference between the center point and circle start point in the Z axis direction, K=center point coordinate Z- Z coordinate of circle start point; its value：-99999999～99999999×least input increment.

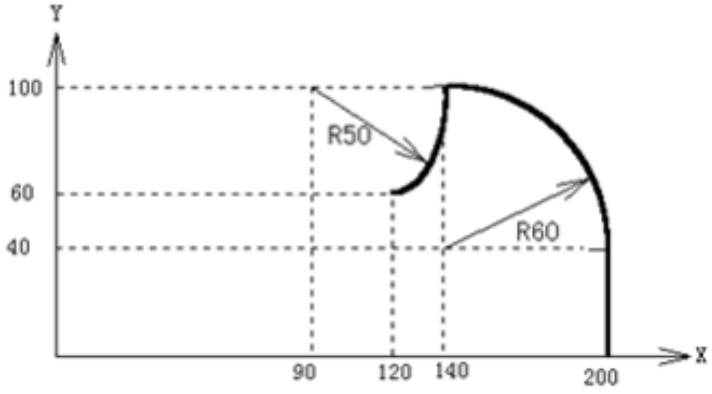
Note：When I, J and K are for whole-circle that they have signs according to the direction. And they are positive values when I, J and K share the same directions with X, Y and Z axes; otherwise they are negative ones.

"Clockwise" and "Counterclockwise" are defined when XY plane(ZX plane, YZ plane) is viewed in the positive-to-negative direction of the Z axis (Y axis, X axis) in the Cartesian coordinate system, see the following figure:



CW or CCW

(Example for the programming)



To program the above paths using the absolute mode and incremental mode respectively:

(1) Absolute mode

G92 X200.0 Y40.0 Z0 ;

G90 G03 X140.0 Y100.0 I-60.0 F300.0 ;

G02 X120.0 Y60.0 I-50.0 ;

Or G92 X200.0 Y40.0 Z0 ;

G90 G03 X140.0 Y100.0 R60.0 F300.0 ;

G02 X120.0 Y60.0 R50.0 ;

(2) Incremental mode

G91 G03 X-60.0 Y60.0 I-60.0 F300.0 ;

G02 X-20.0 Y-40.0 I-50.0 ;

Or G91 G03 X-60.0 Y60.0 R60.0 F300.0 ;

G02 X-20.0 Y-40.0 R50.0 ;

The feedrate of circular interpolation is specified by F command; it is the speed of the tool along the arc tangent direction.

Note 1: I0, J0 and K0 can be omitted; but, it is very necessary to input one of the addresses I, J, K or R, or the system alarm is generated.

Note 2: The X, Y and Z can be omitted simultaneously when the end and start points share same position. When the center point is specified by address I, J and K, it is a 360° arc.

G02 I_; (Full circle)

The circle is 0° when using R.

G02 R_; (not move)

It is recommended that programming uses R. In order to guarantee the start and end points of the arc are consistent with the specified value, the system will move by counting R again according to the selected plane, when programming using the I, J and K. After calculation, the radius difference cannot exceed the permissive value set by No.3410.

Plane selection	Count the radius R value again
G17	$R=\sqrt{I^2+J^2}$
G18	$R=\sqrt{I^2+K^2}$
G19	$R=\sqrt{J^2+K^2}$

Note 3: The error between the actual tool feedrate and the specified feedrate is ±2% or less. The command speed is movement speed after tool radius offset along the arc.

Note 4: The R is effective when address I, J and K are commanded with the R, but the I, J and K are disabled at one

time.

Note 5: The axis not exists is specified on the set plane, the alarm occurs.

Note 6: If the radius difference between start and end points exceeds the permitted value by parameter (№.3410), a P/S alarm occurs.

3.5 Dwell G04

Format: G04 P_ ; or
G04 X_ ;

Function: Axes stop, the current G command mode and the data, status are invariable, after delaying time specified, the next block will be executed.

Explanation: G04, which is a non-modal G-command;
G04 delay time is specified by command words P_, X_;
See the following figure table for time unit of P_ and X_ command value:

Address	P	X
Unit	0.001 s	s
Valid range	0～9999999	0~9999.999

Note 1: X can be specified by the decimal but P not, or the alarm will be generated.

Note 2: When the P and X are not introduced or they are negative value, it means exact stop between the programs to ignore the delay.

Note 3: The P is effective when the P and X are in the same block.

Note 4: The operation is held on when feeding during the G04 execution. Only the delay time execution is finished, can the dwell be done.

3.6 Forward Command G5.1

Format: G5.1 Q__

Function: Enable the forward-looking function to adapt to the processing of mold parts made by software such as UG and Mastercam, and is also commonly used in the manufacturing of workpieces with high contour accuracy requirements.

Explanation : G5.1 Specify that the Q value is valid;

Note:

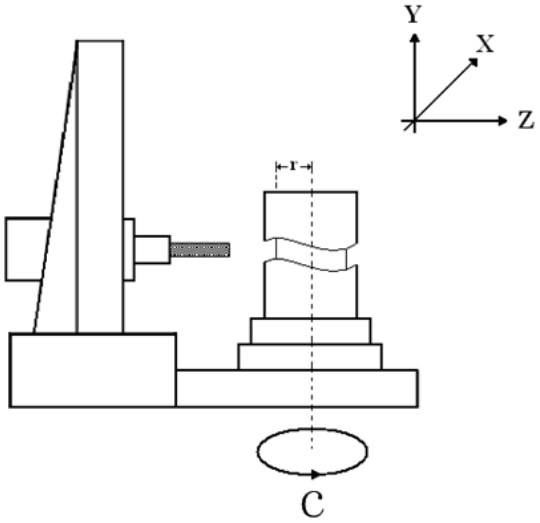
- The Q value can only be selected as 0 or 1, and entering other values will prompt an alarm;
- When Q1 is specified, the small line segment foresight function is effective;
- When Q0 is specified, the small line segment forward function is invalid;
- After the end of M30 operation, the small line segment foresight function is invalid;
- When powered on, the default small line segment forward function is invalid.

Example:

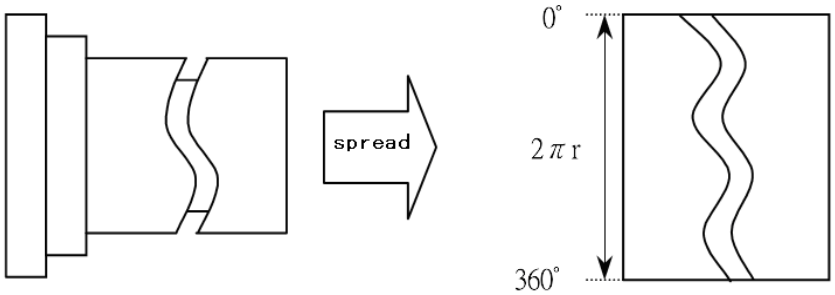
```
G0 X0 Y0 Z0
G5.1 Q1          (Enter the small line segment foresight)
G01 X0.1 Y0.1
X0.3 Y0.2
.....
G5.1 Q0          (Cancel small line segment foresight)
```

3.7 Cylindrical Interpolation G07.1

In the cylindrical interpolation, the travel amount of rotary axis specified by an angle is converted to a distance of a linear axis on the outer surface in CNC, so that linear interpolation or circular interpolation can be performed with another axis. After interpolation, convert this distance to the travel amount of the rotary axis.



Because the side of a cylinder is allowed to use in programming, programs for cylindrical cam grooving can be created very easily.



Format: G07.1 IPr; —— Starts the cylindrical interpolation mode

… —— (cylindrical interpolation is valid)

G07.1 IP0; —— The cylindrical interpolation mode is cancelled

Thereinto, IP is the address of rotary axis; r is the radius of the cylinder. Namely, when r≠0 interpolation starts, r=0 interpolation stops.

G07.1is G code of 00. G107 can be used instead of G07.1.

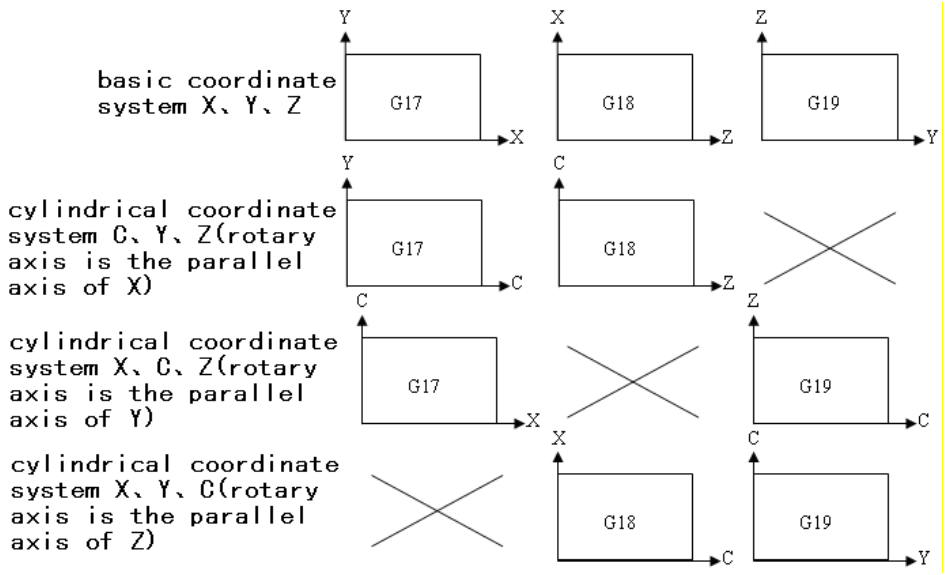
Explanations for plane selection: Plane selection is needed in circular interpolation, tool nose radius compensation and automatic chamfer. The following table shows the planes of G code selection.

G code	Selected plane	X_P	Y_P	Z_P
G17	X_P-Y_P plane	X axis or its parallel axis	Y axis or its parallel axis	Z axis or its parallel axis
G18	Z_P-X_P plane			
G19	Y_P-Z_P plane			

Specify rotary axis as a parallel axis of X, Y, Z by parameter No.1022. Specify G code selection plane, and now the rotary axis is regarded as a specified linear axis for the plane. For example, when

rotary axis is parallel to the axis of X, X_P-Y plane should be specified by G17. This plane is determined by rotary axis and Y axis.

For cylindrical interpolation, only a rotary axis can be set. Parallel axes for different planes are listed as follows.



Note: The above are the plane of "X" displaying, which can not be specified. If it is specified, P/S alarm occurs.

Related explanations for circular interpolation

In cylindrical interpolation mode, circular interpolation is possible with the rotary axis and another linear axis. The unit for rotary axis is not degrees but millimeters (for metric input) or inches (for inch input).

- Circular interpolation between the Z axis and C axis

  When the C axis of parameter No.1022. is set to 5 (parallel axis of X axis), in this case, the command for circular interpolation is:

  G18 Z_ C_; → G18 Z_ X_P_; （X_P is the parallel axis for X axis）

  G02/G03 Z_ C_ R_; → G02/G03 Z_ X_P_ R_;

  When the C axis of parameter No.1022. is set to 6 (parallel axis of Y axis), in this case, the command for circular interpolation is:

  G19 C_ Z_; → G19 Y_P_ Z_; （Y_P is the parallel axis for Y axis）

  G02/G03 Z_ C_ R_; → G02/G03 Z_ Y_P_ R_;

- Circular interpolation between the X axis (or Y axis) and C axis is similar to the above.

Feedrate: The specified federate F (mm/min) in the cylindrical interpolation is the speed of the unfolded cylinder surface, which is called linear speed. The linear interpolation and circular interpolation is performed with speed F (mm/min).

Thereinto: $F = \sqrt{F_c^2 + F_{x/y/z}^2}$

F_c ： In linear and circular interpolation, linear velocity (mm/min) of C axis,

$F_{x/y/z}$ ： In linear and circular interpolation, linear velocity (mm/min) of X/Y/Z axes.

After interpolation, The C axis output linear distance by converting it to the angle of rotary axis. That is, the relation between output speed ω (deg/min) of C axis and F_c (mm/min) are as follows:

$$\omega = \frac{180 F_c}{R\pi} \text{ (deg/min)}$$

Speed restriction: The increase of the speed F will make the output speed ω of C axis exceeds the upper speed (parameter No.027.) that is specified by the system, so the program speed F should be

restricted.

$$F < R \times \omega_{max} \times \frac{\pi}{180} \text{ (mm/min)}$$

Thereinto:

R：Indicates the cylinder radius of specified cylindrical interpolation (unit: mm)

ω_{max}：Upper cutting feedrate of each axis (parameter No.27.，it of rotary axis: deg/min)

Note: Speed command F should be specified in the mode G94. If it is specified in G95, P/S alarm occurs.

Auxiliary function

- Command can be performed correctly in cylindrical interpolation mode with auxiliary function (M).
- Please command tool T, H commands before cylindrical interpolation. If it is commanded in cylindrical interpolation mode, P/S alarm occurs.

Tool nose radius compensation

To perform tool nose radius compensation in the cylindrical interpolation mode, cancel any ongoing tool nose radius compensation before entering the cylindrical interpolation mode. Then, start and stop tool nose radius compensation in the cylindrical interpolation mode. The tool offset value can not be changed in cylindrical interpolation mode.

Cylindrical interpolation accuracy

In the cylindrical interpolation mode, the travel amount of rotary axis specified by an angle is converted to a distance of a linear axis on the outer surface in the system, so that linear interpolation or circular interpolation can be performed with another axis. After interpolation, this travel amount is changed into angle.

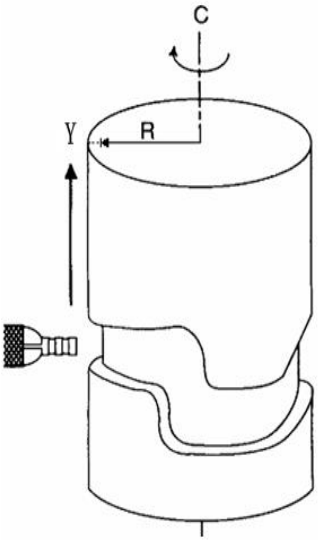
Therefore, when the radius of a cylinder is small, the actual amount of travel amount may differ from the specified one after the travel amount is rounded to the minimum input increment unit. However, such error is not accumulative.

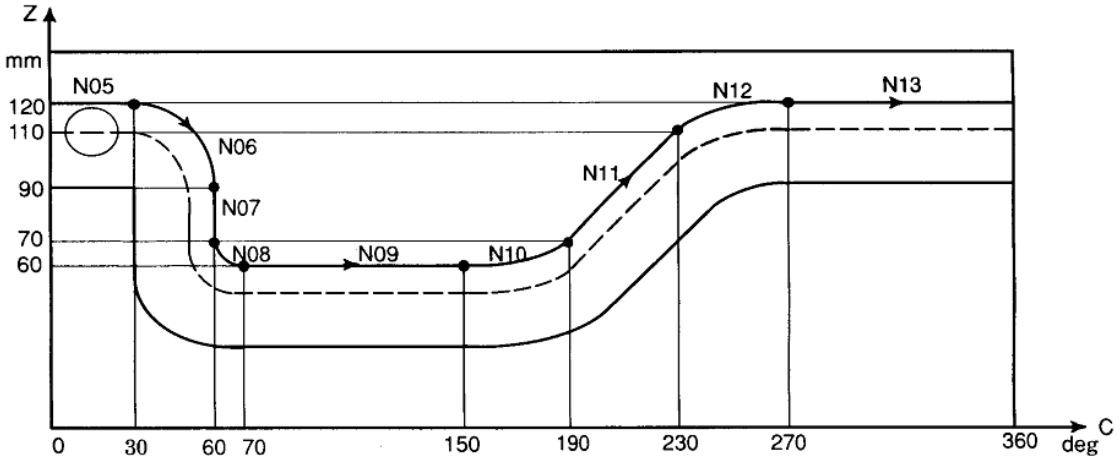
Restrictions

- The cylindrical interpolation command can not be specified in MDI mode, otherwise, alarm occurs.
- In the cylindrical interpolation mode, arc radius is specified by the address R but not specified by I, J, K, otherwise, alarm occurs.
- In the cylindrical interpolation mode, positioning operation G00 cannot be specified (including the commands that produce rapid traverse such as G28, G53 and canned cycle G73~G89). Before positioning is specified, the cylindrical interpolation mode must be cancelled. Cylindrical interpolation (G07.1) can not be performed in the positioning mode (G00).
- To determine rotary axis of interpolation, a plane must be selected before entering the cylindrical interpolation mode. The plane can not be switched after entering the cylindrical interpolation mode.
- Tool length compensation must be specified before cylindrical interpolation. The function of tool length compensation can not be performed in cylindrical interpolation. The cylindrical interpolation command can not be specified simultaneously in the block where the length compensation command is specified, otherwise, alarm occurs.
- The validity of the cylindrical interpolation can not be specified repeatedly in the cylindrical interpolation mode and only a rotary axis can be set in this mode.
- The workpiece coordinate system (G54~G59,G92) and local coordinate system (G52) can not be specified in the cylindrical interpolation mode.
- Programmable mirror image, scaling and coordinate system rotary function can not be specified in the cylindrical interpolation mode.
- Cancel cylindrical interpolation mode in the following modes:
  1) G07.1 IP0 cancel
  2) Reset

3) Program executed

Example:

O0001 (G07.1) N01 G90 G54 G49 G40 G17 G0 X0 Y30 Z0 C0 N02 G01 Z5 F800 N03 G19 G07.1 C57.299 N04 G41 D1 G01 Y120 N05 C30 N06 G02 Y90 C60 R30 N07 G01 Y70 N08 G03 Y60 C70 R10 N09 G01 C150 N10 G03 Y70 C190 R75 N11 G01 Y110 C230 N12 G02 Y120 C270 R75 N13 G01 C360 N14 G40 Y100 N15 G07.1 C0 N16 G90 G00 X-10 Y30 Z-10 C90 N17 M30	C axis is the parallel axis of Z axis 



The above figure is side stretched-out drawing of the cylinder in the above example. It can be seen from the figure that: when travel amount of rotary axis (C axis) specified by angle is converted to a distance of a linear axis on the outer surface, the interpolation formed by it and another linear axis (Y axis) can be seen as an interpolation in the plane coordinate system Y-Z on plane G19.

3.8 Programmable Data Input G10

G10 can modify some data value when a program is executed.

Command format ：G10 Lm Pn Rx;

Command explanation：m：modified data type；

n：data serial number

x：input value is determined to absolute or increment by G90/G91.

3.8.1 Modifying Tool Compensation Data

Tool compensation type	Command format	Explanation
Geometry compensation value of H command	G10 L10 P_ R_	P：tool compensation number
Geometry compensation value of D command	G10 L12 P_ R_	R：Tool compensation value is sum of it adding the value of specified tool compensation number in absolute value command（G90） or incremental value command（G91）.
Wear compensation value of H command	G10 L11 P_ R_	
Wear compensation value of D command	G10 L13 P_ R_	

Note: Geometry compensation value of tool radius compensation D cannot be negative, otherwise, an alarm occurs.

3.8.2 Modyfing a Workingpiece Coordinate System

Command format：G10 L2 P n IP_;

n=0: modify workpiece zero float.

n=1~6: modify workpiece zero floats of workpiece coordinate system 1~6.

IP： coordinate setting value.

IP value is a setting value of corresponding coordinate system;

When G91 is executed, the specified value of setting value +IP of current coordinate system is a setting value of a new coordinate system.

Corresponding coordinate system's value of default axis does not change.

When P is defaulted, a coordinate system is not set, which is taken an alarm.

An alarm occurs when the specified P is beyond 0~6;

An alarm occurs when the specified coordinate data is beyond the above range;

When a program is executed and value of current coordinate system is changed, and the absolute coordinate system is immediately change.

3.8.3 Modifying an Additional Workpiece Cooridnate System

Command format：G10 L20 Pn IP_; modify an additional workpiece coordinate system.

n：1~48

IP_: setting values of axis address and workpiece origin offset distance;

When G09 is executed, IP value is a setting value of corresponding coordinate system;

When G91 is executed, the specified value of setting value +IP of current coordinate system is a setting value of a new coordinate system.

Corresponding coordinate system's value of default axis does not change.

When P is defaulted, a coordinate system is not set, which is taken an alarm.

An alarm occurs when the specified P is beyond 0~48;

An alarm occurs when the specified coordinate data is beyond the above range;

When a program is executed and value of current coordinate system is changed, and the absolute coordinate system is immediately change.

3.9 Polar Coordinate Command (G15, G16)

The coordinate value can be input in polar coordinates (radius and angle).

Format： G9□ G1□ G16； ……Start the polar coordinate mode

G00 IP_ ；
…　} Polar coordinate command

G15； …… Cancel the polar coordinate mode

Command descriptions：G16：Start the polar coordinate command

G15：Cancel the polar coordinate command

G1□：Plane selection of the polar coordinate command (G17, G18 or G19)

G9□：G90 specifies the zero point of the workpiece coordinate system as the origin of the polar coordinate system, from which a radius is measured.
G91 specifies the current position as the origin of the polar coordinate system, from which a radius is measured.

IP_： Specify the addresses and values of selected plane for the polar coordinate system.
The first axis: Radius of the polar coordinate
The second axis: Angle of the polar coordinate

Corresponding axis for radius and angle of the polar coordinate on each plane

G code	The first axis（radius）	The second axis（angle）
G17	X axis	Y axis
G18	Z axis	X axis
G19	Y axis	Z axis

(1) The polar coordinate commands G15, G16 are G code of No.17

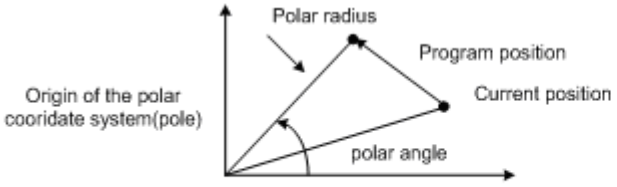
(2) G15 is the initial state when the system power on. Cancel the polar coordinate command after the program is finished or reset. （parameter N0.528 RSTP=1）

(3) Absolute command or incremental command (G90, G91) can be used in polar coordinate radius and angle.

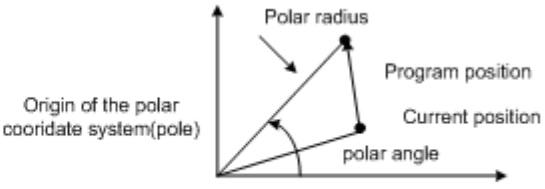
(4) when the polar coordinate radius is specified with the negative value, it is taken as the positive to execute; when the specified angle is positive, the polar coordinate rotates counterclockwise of the 1st axis's positive direction in the currently selected plane, and when it is negative, the polar coordinate rotates clockwise.

Set the zero point of the workpiece coordinate system as the origin of the polar coordinate system

When the polar coordinate command mode is set to start by G90, the origin point of the current workpiece coordinate system is set to be the origin of the polar coordinate system. When a local coordinate system (G52) is used, the origin of the local coordinate system is set to be the origin of the polar coordinate system.



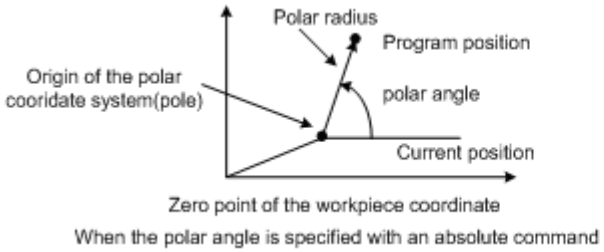
When the polar angle is specified with an absolute command



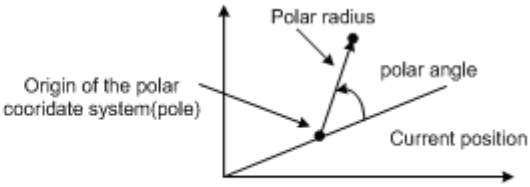
When the polar angle is specified with an incremental command

Set the current position as the origin of the polar coordinate system

If the polar coordinate command is set to start by G91, the current position is set as the origin of the polar coordinate system.



When the polar angle is specified with an absolute command



When the polar angle is specified with an incremental command

Specify radius of polar coordinate system

In absolute mode, the specified radius is the distance between the program point and the origin of the polar coordinate system.

In relative mode, the specified radius is the increment of the current radius of the polar coordinate system.

Specify angle of polar coordinate system

In the absolute mode, the specified polar angle is the rotated programmed angle taking the vector which is parallel to the 1st axis as the rotary side, and taking the polar point as the rotary center.

In the relative mode, the specified angle is incremental value of the current angle (the vector angle from the origin of the workpiece coordinate to the current position).

The origin of the altered polar coordinate system

In the program, once the origin of the polar coordinate system is specified, it is valid in the polar command. If the origin of the polar coordinate system is to be changed, the polar coordinate command must be specified again.

See the following program:

Polar coordinate command

```
G90 G17 G16; …… Polar coordinate command starts. Set the zero point of the workpiece coordinate
                system as the origin of the polar coordinate system
G00 X50 Y30; …… Specify a distance of 50mm and an angle of 30 degrees
  …
G91 G16; …… Change the origin of the polar coordinate system, and the current position is taken as the
            origin of the polar coordinate system
  …
G90 G16; ……Change the origin of the polar coordinate system, and zero point of the workpiece
            coordinate system is taken as the origin of the polar coordinate system
  …
G15; …… Cancel the polar coordinate command
```

Of course, the polar coordinate command mode is cancelled by G15, and then is specified again by G16 and the new polar position is set.

Axes commands that are not considered as polar coordinate commands

In the polar coordinate mode, the following specified axes are not considered as the polar coordinate command. Moreover, the axes other than the first and the second axes on the selected plane are not considered as the polar coordinate command.

G command	Function	G command	Function
G04	Dwell	G52	Set the local coordinate system
G53	Select the machine coordinate system	G92	Set the workpiece coordinate system
G28,G30	Return to the reference point	G31	Skip
G65,G66	Call macro program	G51.1,G50.1	Programmable mirror image
G51,G50	Scaling	G68,G69	Coordinate system rotation

Limitations

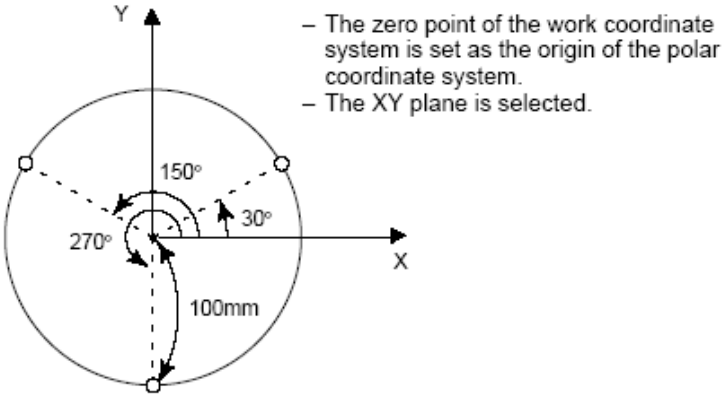
- In the polar coordinate mode, specify a radius for circular interpolation or helical cutting (G02, G03) with R.
- In the polar coordinate mode, no chamfer function can be specified.
- Cylinder interpolation mode cannot be specified in polar coordinate mode.

Cancel the polar coordinate mode

- Specify G15
- Reset
- Program execution finished

Example

Bolt hole circle machining



- Specify angles and radius with absolute commands

N1 G17 G54 G90 G16; ……Specify the polar coordinate command and select XY plane. Set the zero point of the workpiece coordinate system G54 as the origin of the polar coordinate command

N2 G81 X100 Y30 Z-20 R-5 F200; ……Specify a distance of 100mm and an angle of 30 degrees

N3 Y150; ……Specify a distance of 100mm and an angle of 150 degrees

N4 Y270; ……Specify a distance of 100mm and an angle of 270 degrees

N5 G15 G80; ……Cancel the polar coordinate system command

- Specify angles with relative values

N1 G17 G54 G90 G16; ……Specify the polar coordinate command and select XY plane. Set the zero point of the workpiece coordinate system G54 as the origin of the polar coordinate command.

N2 G81 X100 Y30 Z-20 R-5 F200; ……Specify a distance of 100mm and an angle of 30 degrees

N3 G91 Y120; ……Specify a distance of 100mm and an angle of +120 degrees

N4 Y120; ……Specify a distance of 100mm and an angle of +120 degrees

N5 G15 G80; ……Cancel the polar coordinate system command

3.10 Plane Selection Command G17, G18 and G19

Format:

G17 **……XY** plane

G18 **……ZX** plane

G19 **……YZ** plane

Function: The plane of arc interpolation and tool radius compensation are chosen by using the G code.

Explanation: G17, G18 and G19 are modal G commands, the plane will not be changed when a block

without any command inside.

Command example:

G18 X_ Z_ ; ZX plane

X_ Y_ ; Invariable plane (ZX plane)

Note 1: The plane selection command can share the same block with other group G commands.

Note 2: The move command is regardless of the plane selection. For example, the Z axis is not On XY plane, the Z axis movement is regardless of the XY plane in command G17 Z_ .

3.11 Conversion of Inch and Metric G20 and G21

Format:

G20/G21;

Function: The input unit either inch or metric is chosen by G code.

Explanation:

Unit system	G codes	least setting unit
Metric	G20	0.0001 inch
Inch	G21	0.001 mm

The G code should be placed in front of the program when inch and metric is switched each other. Before the coordinate system is set, it is specified by a single block command.

The following unit systems vary according to the G code for inch or metric conversion.

(1) Feedrate command value by F.

(2) Command value related to the position.

(3) Offset.

(4) 1 scale value for MPG.

(5) Step amount value.

(6) current coordinate value.

Note 1: The G code for inch or metric conversion when the power is turned on is the same as that at the power off.

Note 2: Changing G20 and G21 are unallowed during programming. Or the alarm occurs.

Note 3: When the unit systems between the machine and input are different, the max. error is 0.5 of the min. move unit; and the error is not be cumulated.

Note 4: As the inch input (G20) and the metric input (G21) switches each other, the offset should be suited to the reset of the input unit.

3.12 Reference Point Return G28

Format: G28 X_ Y_ Z_;

Function: The middle point position specified by X, Y and Z is reached from the start point at the rapid traverse rate, then it returns to the reference point.

Explanation:G28 is a non-modal G-command;

X: The absolute coordinate of middle point in X axis is indicated by G90, the middle point increment against current point in X axis is indicated by G91;

Y: The absolute coordinate of middle point in Y axis is indicated by G90, the middle point increment against current point in Y axis is indicated by G91;

Z: The absolute coordinate of middle point in Z axis is indicated by G90, the middle point increment against current point in Z axis is indicated by G91.

Command	Function
G28	3 axes hold on at the initial position, the next block continued.
G28 X	X axis reference point return, Y and Z axes still in the original position
G28 Y	Y axis reference point return, X and Z axes still in the original position
G28 Z	Z axis reference point return, X and Y axes still in the original position

Command	Function
G28 X___Z	X and Z axes reference point return simultaneously, Y axis in the original position
G28 X___Y	X and Y axes reference point return simultaneously, Y axis in the original position
G28 Y___Z	Y and Z axes reference point return simultaneously, X axis in the original position
G28 X___Y___Z	X, Y and Z reference point return simultaneously

One of the command addresses X, Y and Z or all of them can be omitted, as follows:
Process for command action (See the figure 3-4):

(1) Positioning from current position to intermediate point of command axis at the rapid traverse rate (From point A to B).

(2) Positioning to the reference point from intermediate point at the rapid traverse rate (From point B to R).

(3) If the machine tool is unlocked, the zero return indicator lights up when the reference point return is finished.

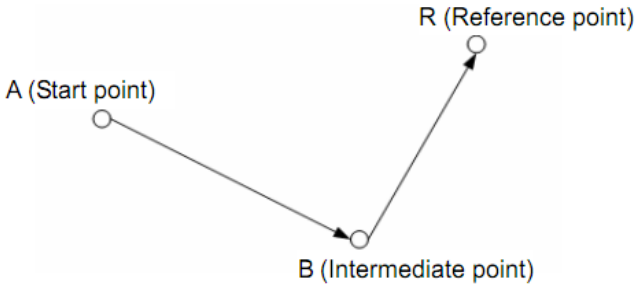


Fig. 3-4

Note:

- After power-on, if G28 is executed prior to the manual machine zero return, the process of G28 machine zero return should be consistent with manual machine zero return, and the deceleration signal and one-rotation signal should be detected. The G28 machine zero return hereafter will not detect the deceleration signal and one-rotation signal, but directly position to zero point.
- During the process of point A→B and B→R, the two axes move at two independent speeds, therefore, the paths may not be linear.
- After the execution of G28 machine zero return, the bit 7 of parameter No.13 decides whether cancel cutter compensation or not.
- In compensation mode, if command G28 is specified, the compensation will be cancelled in the intermediate point. The compensation mode is cancelled automatically after reference point return.
- If zero point switch is not equipped on the machine tool, G28 command and machine zero return are disabled.

- The intermediate point can only be established during the movement from the intermediate point to the reference point which is followed the movement from the start point to the intermediate point.
- After the modification of parameters which set the zero return point, manual reference point return is necessary; G28 command can be executed later.

3.13 Return from Reference Point G29

Format: G29 X_ Y_ Z_;

Function: When a rapid traverse is performed from the current point to mid point, it positions to the specified position by X, Y and Z at the rapid traverse rate.

Explanation:

X: The absolute coordinate of aim point in X axis is indicated by G90; the aim point increment against the mid point in X axis is indicated by G91;

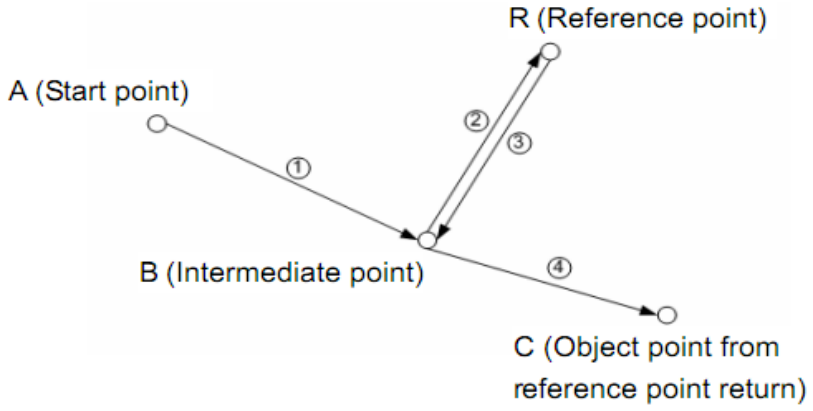
Y: The absolute coordinate of aim point in Y axis is indicated by G90; the aim point increment against the mid point in Y axis is indicated by G91;

Z: The absolute coordinate of aim point in Z axis is indicated by G90; the aim point increment against the mid point in Z axis is indicated by G91;

One of the command address X, Y and Z or all of them can be omitted, see the following figure:

Command	Function
G29	X,Y and Z axes are in the original position, the next block continued
G29 X	Only X axis performs the command returning from the reference point
G29 Y	Only Y axis performs the command returning from the reference point
G29 Z	Only Z axis performs the command returning from the reference point
G29 X_ Z	Only X and Z axes perform the command returning from the reference point
G29 X__ Y	Only X and Y axes perform the command returning from the reference point
G29 Y___Z	Only Y and Z axes perform the command returning from the reference point
G29 X___Y__Z___	X, Y and Z perform the command returning from the reference point

Process for command action:



(1) The command axis direction performs positioning at the intermediate point specified by G28 (from point R to B), the action is ①→②.

(2) The positioning is performed from intermediate point to specified point (from point B to C), moving

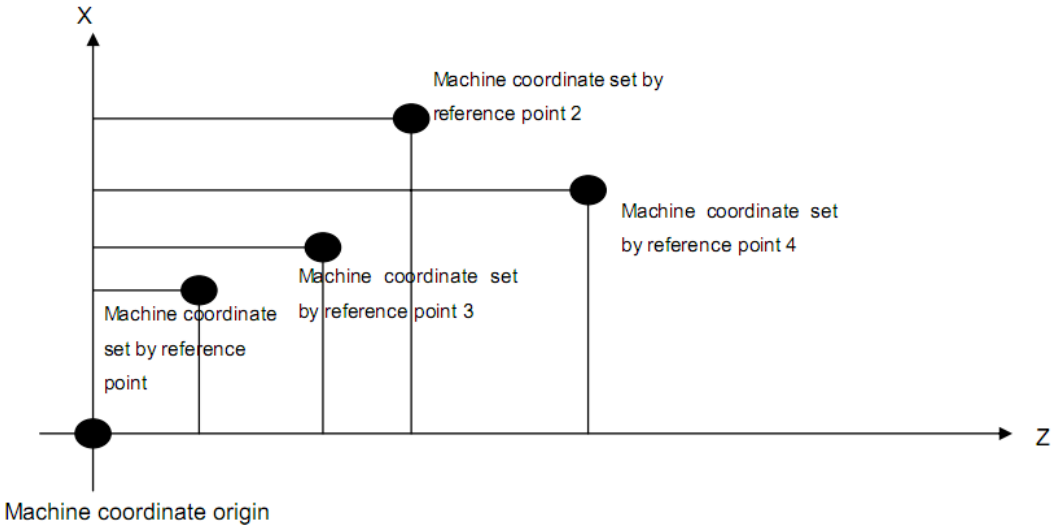
to the intermediate and command point at a rapid feedrate, the action is ③→④.

Note 1:G29 is specified after G28, if an intermediate point is not specified by any of axes, the system alarm will be generated.

Note 2: It is incremental distance against the intermediate point in G91 coordinate programming.

3.14 The 2nd, 3rd and 4th Reference Point Return G30

Reference point is a fixed point on the machine. By parameters (No.1240~No.1243) it can set four reference points in the machine coordinate system.



Format:

G30 P2 X_ Y_ Z_ ; the machine 2nd reference point return (P2 can be omitted)

G30 P3 X_ Y_ Z_ ; the machine 3rd reference point return

G30 P4 X_ Y_ Z_ ; the machine 4th reference point return

Function: From the start point, after the intermediate point by X, Y and Z is reached at a rapid traverse rate, the machine 2nd, 3rd and 4th reference points are returned. The command word P2 can be omitted when the machine 2nd reference point is returned.

Explanation: G30, which is a non-modal G-command;

X: X axis coordinate for intermediate point;

Y: Y axis coordinate for intermediate point;

Z: Z axis coordinate for intermediate point;

One of the command address X, Y and Z or all of them can be omitted, see the following figure:

Command	Function
G30 Pn X	Machine nth reference point return for X axis, Y and Z axes in the original position
G30 Pn Y_ Z_	Machine nth reference point return for Y and Z axes, X axis in the original position
G30	3 axes in the original position, the next block continued
G30 Pn X_ Y_ Z _	X, Y and Z axes return to the machine nth reference point simultaneously.

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

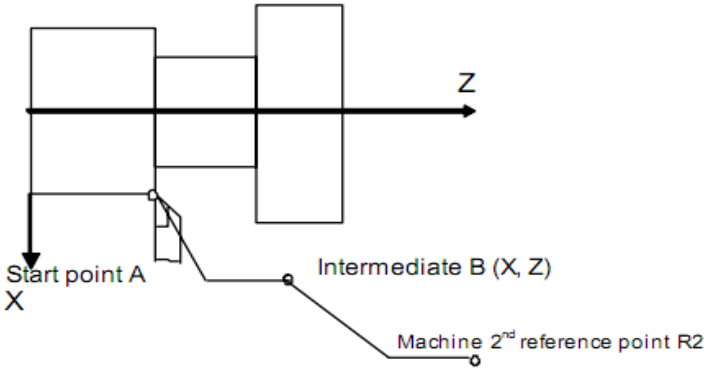
Note 2: Deceleration and zero signals check are not needed when 2nd, 3rd and 4threference points return is performed.

Command action process (see the following figure, an instance of machine 2nd reference point return):

(1) Positioning to intermediate point of the specified axis from current position at a rapid traverse rate (from point A to point B);

(2) Positioning to the 2nd reference position set by data parameter No.1241 at the setting speed by data parameter No.31 (from point B to point R2)

(3) When the reference point returns if the machine is unlocked, the Bit 0 and Bit 1 of the reference point returning end signal F96(ZPn) are HIGH.



Note 1: After returning the machine reference point by manual or the G28 command is performed, the machine 2nd, 3rd and 4th reference point return function can be employed only, or the 2nd, 3rd and 4th reference point operation of G30 command , the system alarm will be generated.

Note 2: From point A to B or from point B to R2, the 2 axes are moved at their separately rate, so the path is not straight line possibly.

Note 3: After machine 2nd, 3rd and 4th reference point returned by the G30 command, the system tool length compensation cancellation is defined by bit 7 of the parameter No.13

Note 4: The 2nd, 3rd and 4th reference point operation of G30 command can not be executed if the **zero switch is not installed on the machine tool.**

Note 5: The workpiece coordinate system is set after the machine 2nd, 3rd and 4th reference point are returned.

3.15 Skip Function G31

As G01 linear interpolation is performed, if an external SKIP signal is valid during execution of this command, execution of this command is interrupted and the next block is executed. The skip function is used when the end of machining is not programmed but specified with a signal from the machine, for example, in grinding. It is used also for measuring the dimensions of a workpiece.

Format:

G31 X__ Y__ Z__

Explanation:

1. G31, which is a non-modal G-code, it is effective only in the block in which it is specified.
2. G31 can not be specified in the tool compensation C and chamfering, or the alarm will be generated. It is very necessary to cancel the tool compensation C and chamfering firstly before the G31 command is specified.
3. Error is allowed in the position of the tool when a skip signal is input.

Signal: The SKIP signal input is on the fixed address X1.0 (CN61-42).

Parameter:

0	5	5	1

						SKIP	G31P

SKIP 1: High level skip signal is valid;

0: Low level skip signal is valid.

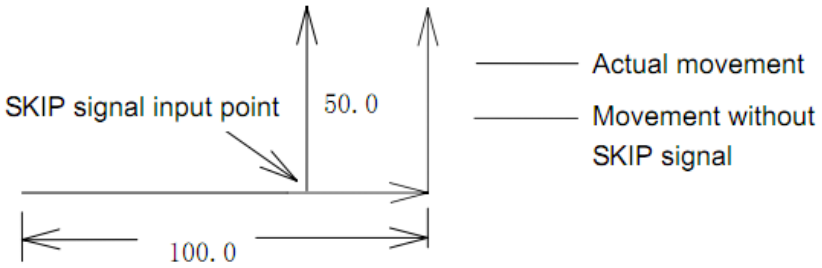
G31P 1: G31 is for immediate stop as the skip signal is valid;

0: G31 is for decelerating stop as the skip signal is valid.

1. **The next block to G31 is incremental command 1**: it moves with incremental value from the position interrupted by the skip signal.

   Example: G31 G91 X100.0 F100;

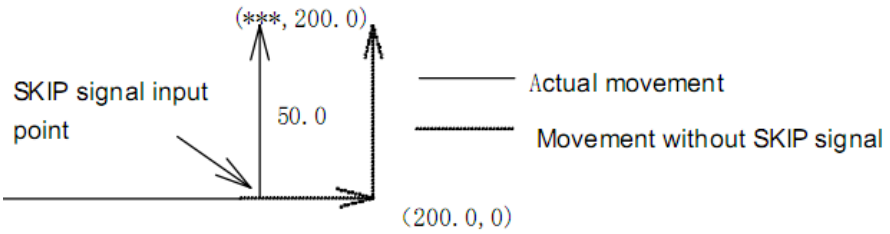
   Y50.0;



2. **The next block to G31 is absolute command for one axis:** The command axis moves to the specified position, and the axis not specified keeps at the skip signal input position.

   Example: G31 G90 X200.0 F100;

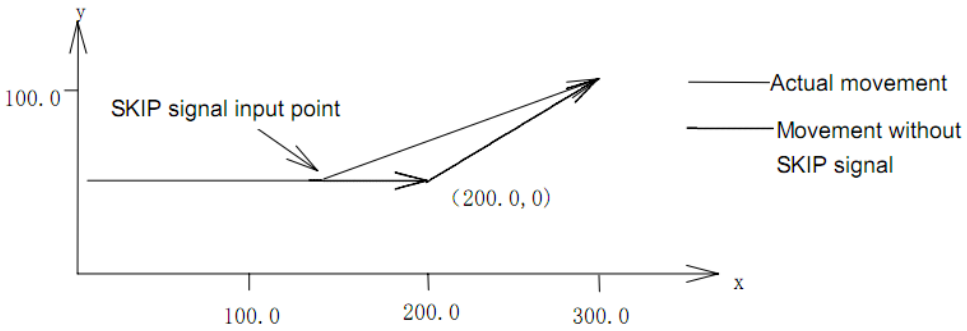
   Y100.0;



3. **The next block to G31 is absolute command for 2 axes:** Wherever the skip signal input is, the tool moves to specified position of next block.

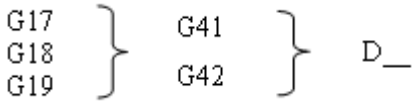
   Example: G31 G90 X200.0 F100;

   X300.0 Y100.0;



3.16 Tool Nose Radius Compensation C (G40, G41 and G42)

Format:



Functions:

Tool nose radius compensation function.

To cancel or perform the tool radius compensation vector by using the commands G40, G41 and G42. They are combined with the commands G00, G01, G02 and G03 for specifying a mode which can be confirmed the compensation vector value, direction and the direction of tool movement.

G codes	Functions
G40	Tool radius compensation cancellation
G41	Tool radius left compensation
G42	Tool radius right compensation

G41 or G42 drives the system into compensation mode; G40 cancels the system compensation mode.

Explanation:

- Compensation plane

  The compensation plane can be confirmed based upon plane selection command; the tool compensation C is calculated in this plane.

Plane selection	Plane compensation
G17	X－Y plane
G18	Z－X plane
G19	Y－Z plane

- Compensation value (D code)

  This system can be set for 32 compensation values at most. Two digits specified by D code in the program, is called serial number of compensation value, the compensation value should be set by MDI/LCD unit.

  D code determines the compensation value in tool offset page according to the bit 1 of parameter No.002, it is very important to notice that the value applied is diameter or radius.

  Setting range of compensation value is as follows:

  Compensation value: 0～9999999×least input increment.

- Compensation vector

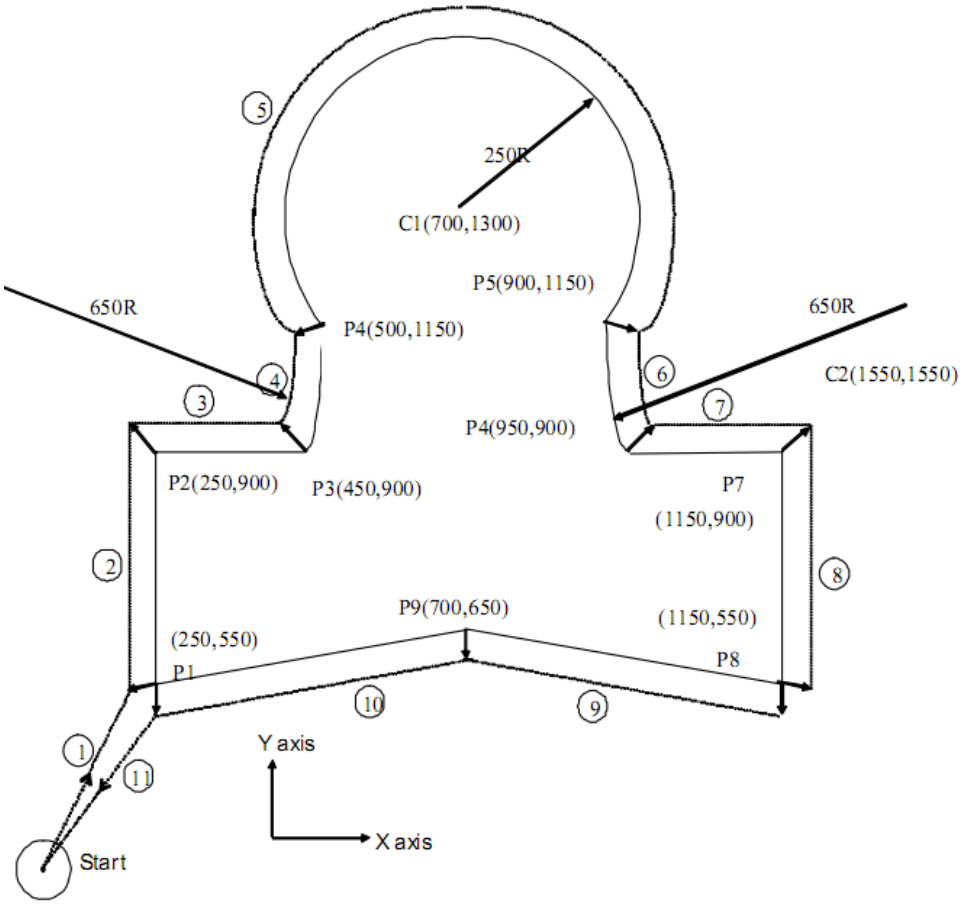
  The compensation vector is two-dimensional vector; it is equal to the compensation value specified with D code. The compensation vector is calculated in control unit, its direction is real-time modified along with the tool path in each block. You can calculate how much compensation is needed for tool movement when the compensation value is applied in control unit. Compensation path (tool center path) = programmed path tool radius (or diameter) (determined by compensation direction).

Note:

- Compensation operation is executed in the plane selected by G17, G18, G19. For example, when XY plane is selected, (X,Y) or (I, J) is used to carry out compensation operation and vector operation. The coordinate value whose axis in not in the compensation plane is not affected by the compensation.
- In 3-axis linkage control, compensation only performed for the tool path projected on the compensation plane.
- The alteration of compensation plane should be executed posterior to the compensation mode cancelled. Otherwise, the system will give an alarm and machine stops.
- When the cutter compensation is cancelled by G40, movement amount should be specified,

otherwise, an alarm will occur.

- In the canned cycle G codes, G40, G41, G42 codes are disabled.



Example :

Block (1) is named start; the compensation cancellation mode becomes compensation mode by G41 in this block. At the end of this block, tool center is compensated in the direction that tool radius is vertical to next program path (From P1 to P2). Tool compensation value is specified with D07, so set the compensation number to 7, then the G41is indicated with tool path compensation left.

After the compensation begins, tool path compensation performs automatically when creating the workpiece as P1→P2……P8→P9→P1.

N00 G92 X0 Y0 Z0;

N01 G90 G17 G00 G41 D7 X250.0 Y550.0 ； （The compensation value should be pre-set with compensation number）

N02 G01 Y900.0 F150 ；

N03 X450.0 ；

N04 G03 X500.0 Y1150.0 R650.0 ；

N05 G02 X900.0 R-250.0 ；

N06 G03 X950.0 Y900.0 R650.0 ；

N07 G01 X1150.0 ；

N08 Y550.0 ；

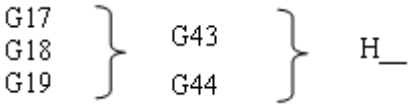
N09 X700.0 Y650.0 ；

N10 X250.0 Y550.0 ；

N11 G00 G40 X0 Y0 ；

3.17 Tool Length Compensation (G43, G44, G49)

Format:

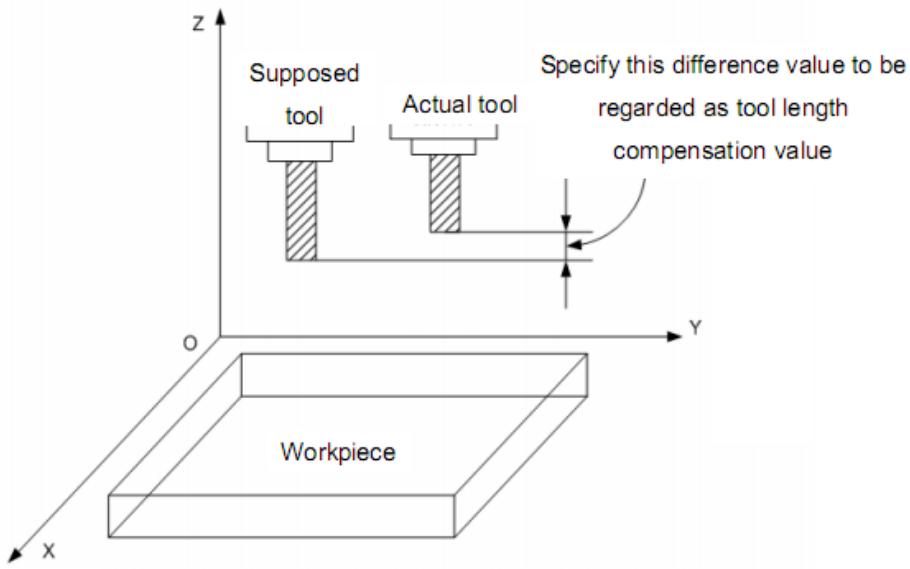


Functions:

Tool length compensation function.

Explanation:

G43 and G44 are modal G codes; they are effective before meeting other G codes in the same group.



The end point specified by Z axis moves an offset value, as above figure G17 plane is selected. Difference between supposed and actual machined tool length value is pre-set at the offset storage when the program is applied. Different length tool can be employed by changing tool length compensation value, so, program change is not needed.

Different offset directions were specified by G43 and G44, the offset number is specified by H code.

Offset axis

The offset axes are vertical to the specified planes (G17, G18 and G19)

Specifying plane	Offset axes
G17	Z axis
G18	Y axis
G19	X axis

Tool position offset for two or more axes can be used to specify the offset axis and the offset axis changed by 2~3 blocks.

(Example) X and Y axes compensation

G19 G43 H_ ; …X axis offset

G18 G43 H_ ; …Y axis offset, composed with the previous block, X and Y axes are compensated.

Offset direction

G43: Positive offset

G44: Negative offset

Compensation axes can be regarded as Z, Y and X. Either absolute or incremental command, the end

point coordinate value specified by Z axis movement command in program adds the offset specified by H codes in G43 (set in the offset storage), or subtracts the offset specified by H code in G44, finally, the value calculated is regarded as the end point coordinate.

The following command is indicated for Z axis move omitting: When the offset is positive, G43 is for an offset in the positive direction; G44 is for an offset in the negative direction.

It reversely moves when the offset is negative value.

Specifying the offset

An offset number is specified by H code and its corresponding offset adds or subtracts Z axis movement command value in program to get a new Z axis movement command value. The offset number is H00~H32.

Offset value corresponded with offset number is pre-set in the offset storage by using the panel of LCD/MDI. Setting range for offset is as follows:

Offset value: -9999999～9999999×least input increment

Offset number 00, i.e. H00 corresponds to the 0 offset. It is disabled to set offset value to H00.

Tool length compensation cancellation

G49 or H00 can be specified when the tool length compensation is cancelled. When two or more axes compensations are cancelled, all of the axes compensation will be cancelled if the G49 is applied. Compensation value of the vertical axis for currently specified plane is cancelled with H00. After G49 or H00 is specified, the system immediately cancels the compensation value.

Note 1: In the block that tool length compensation is specified, G02，G03, G04, G92 ，G31，G142 and G143 cannot be specified at the same time, otherwise, an alarm will occur.

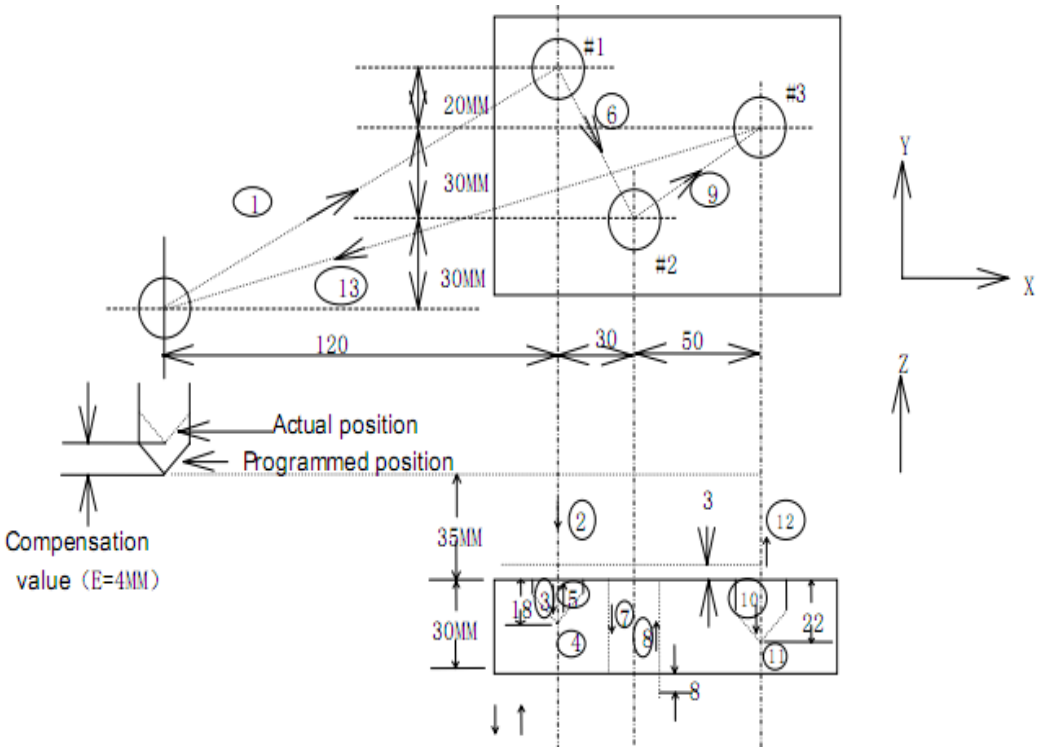
Note 2:Tool length compensation command can be specified in the block in which canned cycle is specified. But after the canned cycle is executed, the tool length compensation is disabled and is not modal.

Example：

Normal	Modal	Explanation (H1=10.0mm，H2=20.0mm)
G43 H1 G44 G01 X50 Y50 Z50 H2 G90 G00 X100 Y100 Z100	G43 H1 G44 H2 G44 H2	Setting H1, tool length compensation in the positive direction. Linear interpolation, setting H2 tool length compensation in negative direction Position to X100 Y100 Z100(Z80) with H2 compensation offset.
In the same block with G02,G03,G04,G31,G92		
G43 H1 G49 G02 X50 R25 H2	G43 H1 G43 H1	Setting H1 tool length compensation in the positive direction. Alarm occurs.
In the same block with canned cycle code		
G43 H1 G44 G81 X50 R5 Z-70 H2 G90 G00 X100 Y100 Z100	G43 H1 G44 H2 G44 H2	Setting H1 tool length compensation in the positive direction. Setting H2 tool length compensation in the negative direction. Starts the canned cycle from H2.
Specified in the canned cycle		
G43 H1 G90 G81 X50 R5 Z-70 G49 H2 G49 G0 X75 Y75 Z75 H0	G43 H1 G43 H1 G43 H1 G49 H0	Setting H1 tool length compensation in the positive direction. Compensation offset with H1; enters into canned cycle mode. The tool length compensation (G49,H2) in the canned cycle is ineffective, and the previous block remains modal. Cancel all the axis compensations, and set H0 modal. Position to X75 Y75 Z75(Z75).

Command Example:

Tool length compensation (#1, #2 and #3 hole machining)



offset H01 = 4.0

N1 G91 G00 X120.0 Y80.0 ； ..... (1)

N2 G43 Z-32.0 H01 ； ..... (2)

N3 G01 Z-21.0 ； ..... (3)

N4 G04 P2000 ； ..... (4)

N5 G00 Z21.0 ； ..... (5)

N6 X30.0 Y-50.0 ； ..... (6)

N7 G01 Z-41.0 ； ..... (7)

N8 G00 Z41.0 ； ..... (8)

N9 X50.0 Y30.0 ； ..... (9)

N10 G01 Z-25.0 ； ..... (10)

N11 G04 P2000 ； ..... (11)

N12 G00 Z57.0 H00 ； ..... (12)

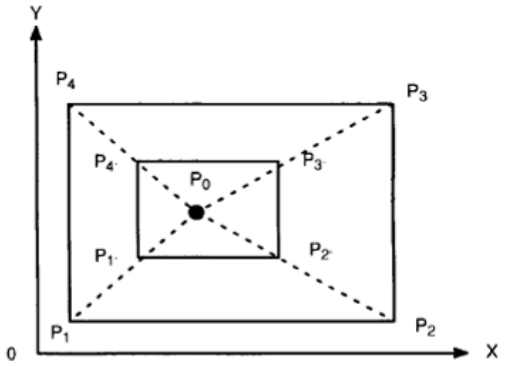
N13 X-200.0 Y-60.0 ； ..... (13)

N14 M30 ；

Z, X or Y axis offsets a value at offset storage positively or negatively from the original end position according to the above command. Offset axes can be specified with G17, G18 and G19, offset direction can be specified with G43 and G44. Offset No. corresponding to the offset is specified by H code.

3.18 Scaling G50, G51

Scaling means programmed figure can be magnified or reduced. The dimension specified by X, Y, Z can be scaled up or down with the same or different rates of magnification. The magnification rate can be specified by the program or parameter.



As the above figure, P0 is the scaling center. P1P2P3P4 scales to P1'P2'P3'P4'.

Format:

Scale up or down along all axes at the same magnification rate	
Format	**Significance of command**
G51 X_Y_Z_P_; Scaling start : } Scaling is valid : } (Scaling mode) **G50**; Scaling cancel	**X_Y_Z_** : Absolute command for center coordinate value of scaling **P_** : Scaling rate

Scale up or down along all axes at a different magnification rate	
Format	**Significance of command**
G51 X_Y_Z_I_J_K_;Scaling start : } Scaling is valid : } (Scaling mode) **G50**; Scaling cancel	**X_Y_Z_** : Absolute command for center coordinate value of scaling **I_J_K_** : Scaling rate for X axis, Y axis, Z axis respectively

G50，G51 are G code of No.11, it is a modal G code.

The unit of scaling is 0.001.

State parameter SCLX, SXLY, SCLZ(No.562) are used to set whether the scaling function of each axis is valid.

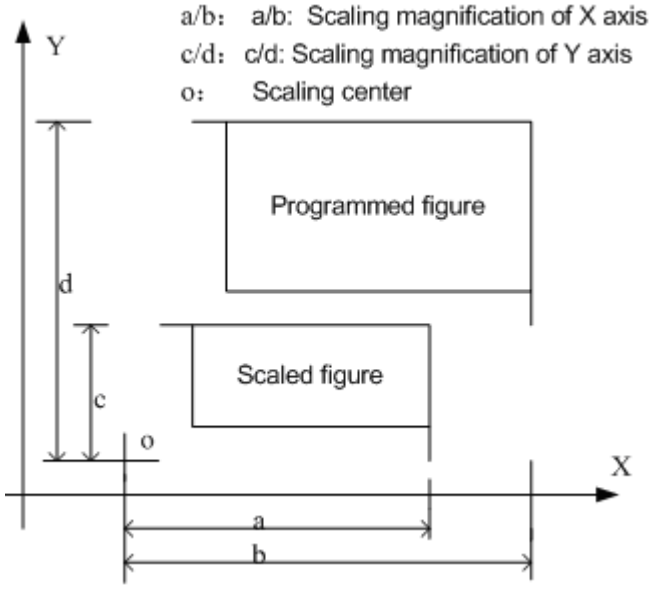
Explanations:

- **Scale up or down along all axes at the same magnification rate**
  When state parameter XSC (No.562) is set to 0, if P is specified on the block G51, the scaling is specified by P, otherwise, the value set by data parameter will be taken as the magnification rate.
- **Scale up or down along all axes at a separate scaling.**
  When state parameter XSC (No.562) is set to 1, and each axis is scaled up or down at a different **scale**, the rate is specified by I, J, K on the block G51. If I, J, K are not specified on the block, the rate is determined by data parameter SCLVX, SCLVY and SCLVZ(No.5421).
  When a negative scale is specified, mirror image is applied.
- **Negative magnification rate**

When a negative scale is specified, mirror image is formed (see related explanations of programmable mirror image)

- **Scale of different figure**

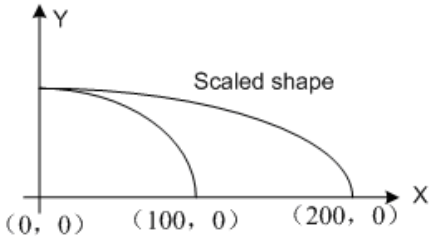
1. Magnification rate of linear scaling



2. Scaling of circular interpolation

Even different magnifications are specified to circular interpolation, tool will not trace ellipse.

When magnifications for axes are different, and the circular interpolation is programmed with radius R, its figure is as follows, (magnification 2 is applied to X-axis and magnification 1 is applied to Y axis)



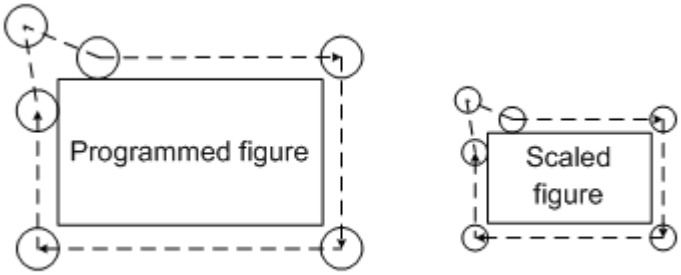
G90 G00 X0.0 Y100.0;
G51 X0.0 Y0.0 Z0.0 I2000 J1000;
G02 X100.0 Y0.0 R100.0 F500;
Above commands are equivalent to the following commands

G90 G00 X0.0 Y100.0 Z0.0;
G02 X200.0 Y0.0 R200.0 F500;
Magnification of radius R is depends on I, or J whichever is larger

When different magnifications are applied to axes, and circular interpolation is specified with I, J, K, alarm occurs after scaled if a circular is not formed.

3. Tool compensation

The scaling is invalid in tool radius compensation values, tool length compensation values and tool offset values. Only the figure before scaling are proceeded, namely, scaling is done before the calculation of tool compensation, see the following figure:



Cutter compensation C values are not scaled

- **Invalid scaling**

  1. In canned cycle, moving scaling of cut-in value Q, Z and retraction value d are invalid.
  2. In manual operation, the travel distance can not be increased by using scaling function.

- **Commands related to reference position return and coordinate system**

  In scaling mode, the G codes (G28-G30 etc.) returned to the reference point and G codes (G92, G54-G59 etc.) of command coordinate system can not be specified. If these G codes must be specified, specify them after the scaling function is cancelled.

- **Position display**

  Position display indicates the coordinate value after scaling.

Related parameters

0	5	6	2

	XSC	***	***	***	SCLZ	SCLY	SCLX

SCLX,SCLY,SCLZ=1：X,Y,Z Scaling is valid.

0：X,Y,Z Scaling is invalid.

XSC=1： Axes are scaled up or down at different magnification rate. If the rate is a negative value, mirror image is formed.

0：Axes are scaled up or down at the same magnification rate.

5	4	1	1

SCLVSAME: Axes are scaled up or down at the same magnification rate

SCLVSAME: If P is not specified, set values with defaulted magnification rate, setting range is 1~99999999.

5	4	2	1

SCLVX: Magnification rate of X axis

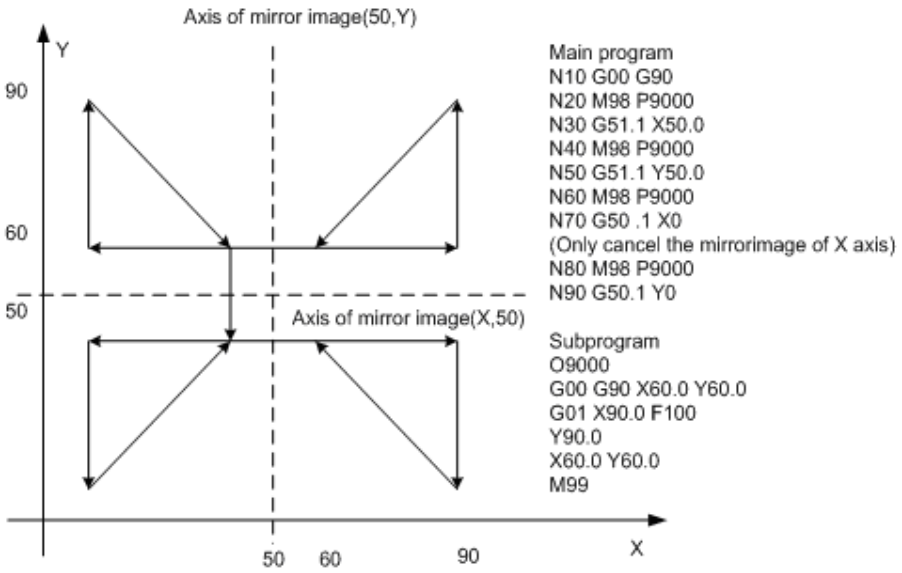
SCLVY: Magnification rate of Y axis

SCLVZ: Magnification rate of Z axis

SCLVX, SCLVY, SCLVZ: Set magnification rate for X, Y, Z axis. Setting range is -99999999~99999999, which can not be 0.

3.19 Programmable Mirror Image G50.1, G51.1

If the shape of a workpiece is symmetrical on an axis, a part of the workpiece can be programmed. Then machining program of a whole part can be obtained by using mirror image (or scaling) and subprogram.



Format： G51.1 X__ Y__ Z__; Set programmable mirror image

: ;
: ; } According to G51.1 X__ Y__ Z__, specified mirror image of these blocks are generated from specified symmetry axis
: ;

G50.1 X__ Y__ Z__; Specify corresponding axis to cancel the mirror image of axes

G50.1,G51.1are G codes of No.22, which are modal G codes.

Command function： Specify mirror image function for axes.

G51.1 X__ Y__ Z__：Specify mirror image function for axes. （put on the axis as a mirror）. Thereinto, X__ Y__ Z__is an absolute command.

G50.1 X__ Y__ Z__：Cancel mirror image function for axes. If the address of the axis is not specified, which indicates no mirror image function is cancelled.

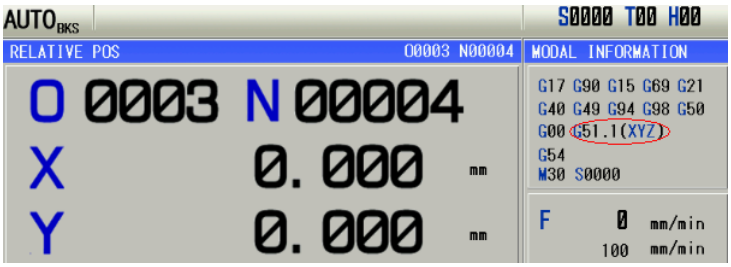
Explanations:

1. Some commands are interchanged when a mirror image to the plane is specified.
   - G02/G03 of circular commands are interchanged.
   - G41/G42 of cutter compensation commands are interchanged.
   - CW and CCW（directions of rotation）are interchanged.
2. This function is not effective for 4th and 5th axes.
3. In canned cycle, the depth of Z are not proceeded with the mirror image.

Limitations

- In programmable mirror image, G codes related to reference return（G27/G28/G29/G30, etc.）and those for changing the coordinate system（G52~G59,G92, etc）can not be specified. If any of these G codes is necessary, specify it only after canceling the programmable mirror image mode.
- Processing proceeds from program mirror image to scaling and coordinate rotation. The commands should be specified in order, for cancellation, in the reverse order. G50.1 and G51.1 can not be specified in scaling and rotation mode.

Modal display of scaling



3.20 Setting Local Coordinate System G52

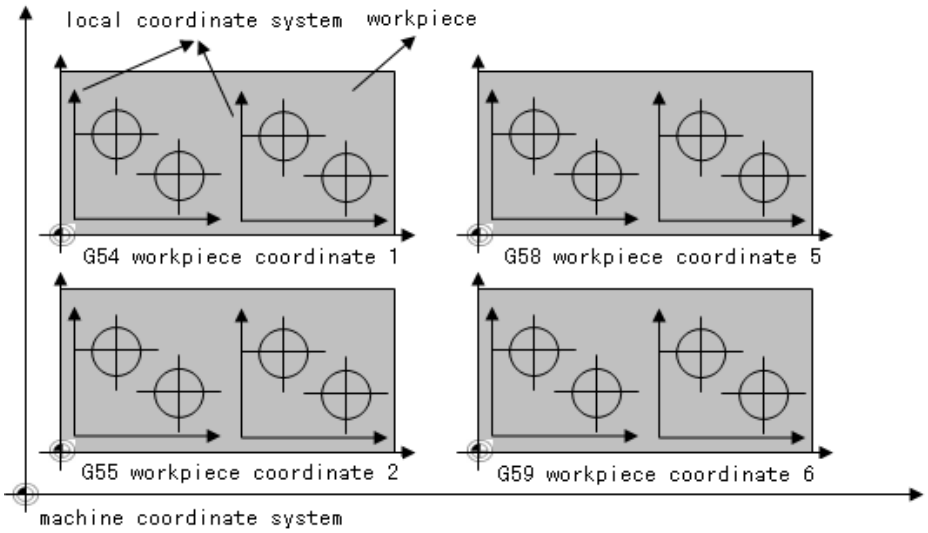
When a program is created in a workpiece coordinate system, the subprogram of the workpiece coordinate system (G54-G59) can be set for easy program. Sub coordinate system is called local coordinate system.

Machine coordinate system, workpiece coordinate system and local coordinate system

The machine coordinate system is a fixed coordinate system on the machine, it indicates a fixed position of the machine.

The workpiece coordinate system is a system facilitates workpiece machining, in which the reference point on the workpiece is taken as the origin point.

Local coordinate system is set on workpiece coordinate system to facilitate the programming of some machining programs.

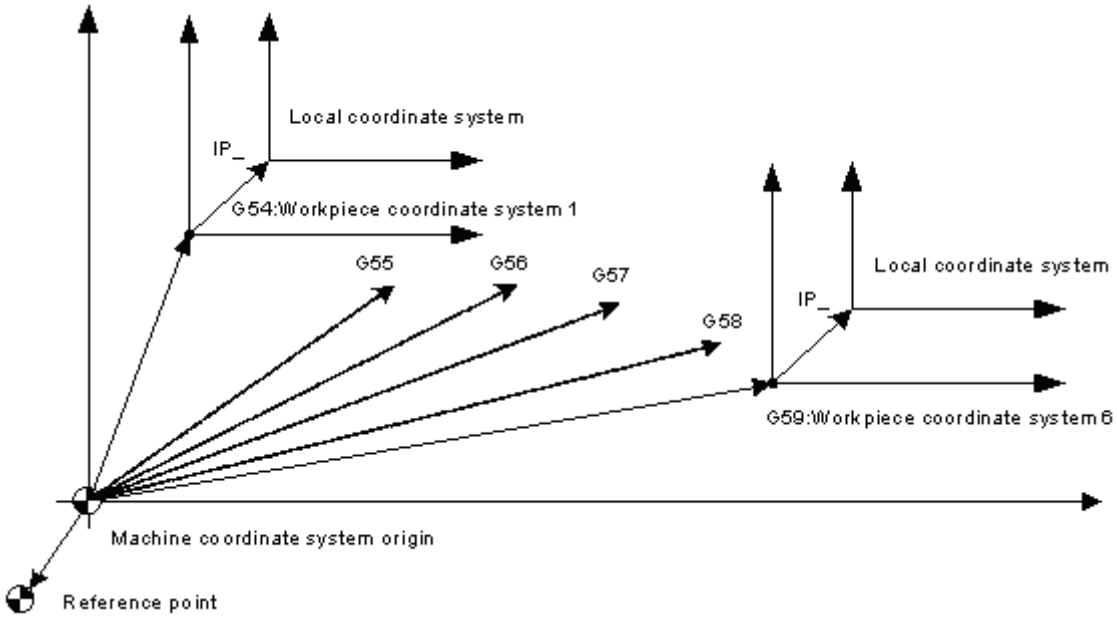


Format：G52 IP；…… Set local coordinate system

G52 IP0；……Cancel local coordinate system

IP：Specify the position of the origin point of the local coordinate system in the current workpiece coordinate system

G52 is G code of the group, which is a non-modal G code.



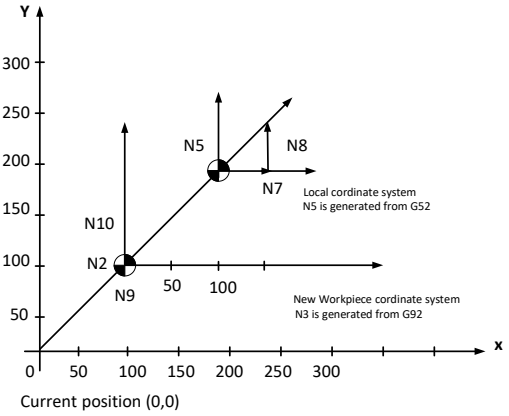
Explanations

- When the local coordinate is set, the following movement specified by absolute mode (G90) is coordinate value in local coordinate system. The position of the local coordinate system can be changed by specifying new origin point with G52.
- In order to specify the origin of the machining program and the offset value of the workpiece origin, replace command G92 by specifying command G52.
- Make the origin of the local coordinate consistent with the zero point of the workpiece coordinate system to cancel the local coordinate system and specify the value in workpiece coordinate system.
- When a G52 is specified, local coordinate system is valid before another G52 command is specified. It is not move when G52 command is being specified.

Example

Set local coordinate system in a single workpiece coordinate system.

```
N1 G28 X0 Y0 Z0;
N2 G90 G54 G00 X100 Y100;
N3 G92 X0 Y0;
N4 G00 X50 Y50;
N5 G52 X100 Y100;
N6 G00 X0 Y0;
N7 G01 X50 F100;
N8 Y50;
N9 G52 X0 Y0;
N10 G00 X0 Y0;
N11 M30;
```

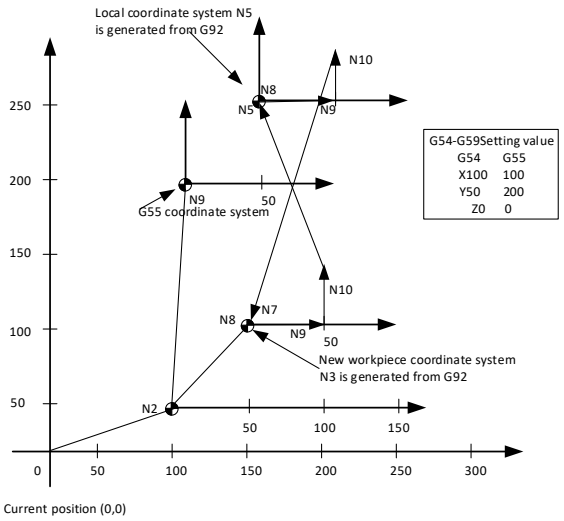


The local coordinate system is set by G54 coordinate system in the block N5. It is cancelled in the block N9, whose cancelled coordinate system is consistent with that set by G92 of block N3.

Set local coordinate system in multiple workpiece coordinate systems

```
N1 G28 X0 Y0 Z0;
N2 G90 G54 G00 X0 Y0;
N3 G52 X50 Y50;
N4 M98 P1234;
N5 G90 G55 G00 X0 Y0;
N6 M98 P1234;
N7 G90 G54 G00 X0 Y0;

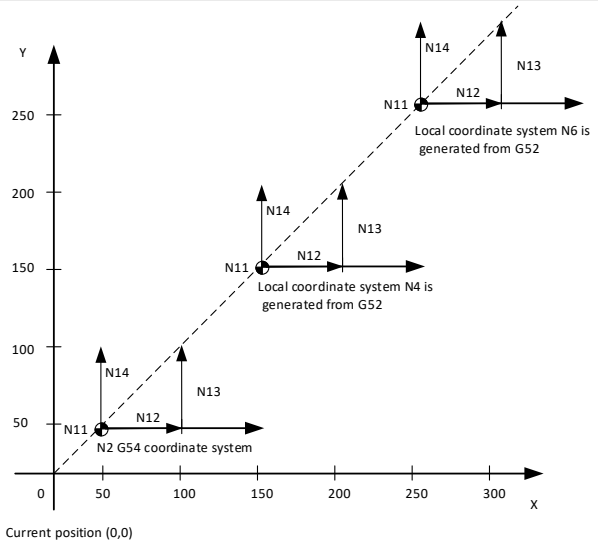
O1234 (Subprogram)
N8 G00 X0 Y0;
N9 G01 X50;
N10 Y50 M99;
```



Multiple local coordinate systems in the workpiece coordinate system

```
N1 G28 X0 Y0 Z0;
N2 G90 G54 G00 X0 Y0;
N3 M98 P1234;
N4 G52 X100 Y100;
N5 M98 P1234;
N6 G52 X200 Y200;
N7 M98 P1234;
N8 G52 X0 Y0;
(N8 G91 G52 X-200 Y-200;)

O1234 (Subprogram)
N11 G00 X0 Y0;
N12 G01 X50;
N13 Y50;
N14 X0 Y0 M99;
```



The local coordinate system generated from the N8 block in the brackets is consistent with the workpiece coordinate G54, which is the same with the result after (N8 block) is cancelled in the local coordinate system.

Notes

- When the parameter No.542 ZCL is set to 1, the local coordinate system of the axis is cancelled in the process of reference return.
  G52 a0；(a is the axis return to the reference point)
- The workpiece coordinate system and the machine coordinate system are not changed by setting the local coordinate system.
- Parameter setting determines whether cancel the local coordinate system after reset. When the parameter No.542 RLC is set to 1, all local coordinate systems of the workpiece coordinate system are cancelled.
- When the workpiece coordinate system is set by specifying command G92, the local coordinate system of all workpiece coordinate systems of the specified axis is cancelled. If the coordinate values of the axes not all specified, the local coordinate systems of the unspecified axes are not cancelled, that is, keep unchanged.
- Command G52 can not be specified at the same block with the length compensation command, otherwise, alarm occurs. Cancel the offset of the tool radius compensation temporarily when the G52 command is being specified.
- After block G52, specify travel command immediately by absolute value mode.

Related parameters

0	5	4	2

RLC	MRC	ZCL					

RLC = 1：Local coordinate system is cancelled after reset

RLC = 0：Local coordinate system is not cancelled after reset

MRC = 1：Local coordinate system is cancelled after the execution of M02, M30

MRC = 0：Local coordinate system is not cancelled after the execution of M02, M30

ZCL = 1：Local coordinate system is cancelled after returning to the reference point

ZCL = 0：Local coordinate system is not cancelled after returning to the reference point

3.21 Select Machine Coordinate System G53

A specific point that serves as the reference point is referred to the machine zero point. The coordinate system with a machine zero point set as its origin is referred to a machine coordinate system. A machine coordinate system is set by performing manual reference position return after power on. Once the machine coordinate system is set, it remains unchanged until the power is turned off.

Format：（G90）G53 IP_；

Explanations: IP_：Tool travel amount is specified by absolute value.

When a position on a machine coordinate system is specified, the tool moves to the position by rapid traverse. G53, which is used to select a machine coordinate system, is a non-modal G code. It is valid only in the block that specifies the machine coordinate system. The absolute value specifies X, Y, Z. When the tool moves to the specific position, for example: tool exchange position, travel program on the machine coordinate system should be edited by G53.

Restrictions:

- Cancel of the compensation function

  When the G53 command is specified, cancel the radius compensation. However, the parameter NO.013 determines whether the tool length compensation is cancelled or not.

- G53 Specify G53 immediately after power on

  Since the machine coordinate system must be set before the G53 command is specified, manual reference position return or automatic reference position return by the G28 command must be performed after the power on. Otherwise, P/S alarm occurs: G53 can not be performed before reference position return.

- In the same block with other G commands

  1. In the same block with group 03 (G90, G91)

     G53, G90 and G91 are performed normally, and related modal is updated. The position command specified by G53 is treated as absolute position.

  2. In the same block with tool length compensation and tool radius compensation

     G53 is performed normally, and cancel the tool length compensation and tool radius compensation.

  3. G53 and the group 01 in the same block

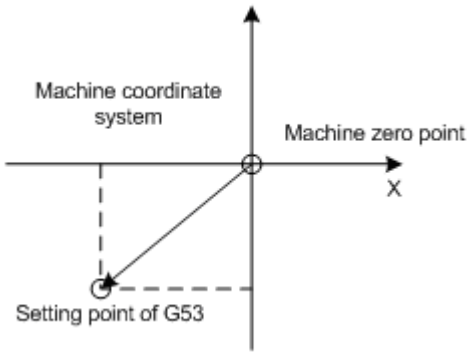
     When it is in the same block with group 01 G command, P/S alarm occurs.

- Processing in the canned cycle

  When canned cycle is valid, P/S alarm will occur after G53 is specified: G53 can not be specified if the canned cycle is valid.

Coordinate display:

After the CNC system is powered on and returned to the reference point, a machine coordinate system is set immediately, whose coordinate values are set by parameter NO.1240



Example

	Relative coordinate (X,Y,Z)	**Absolute coordinate (X,Y,Z)**	**Machine coordinate (X,Y,Z)**
Initial position	-110,-110,-110	40,40,40	-120,-120,-120
G53 X25 Y25 Z25;	35,35,35	185,185,185	25,25,25
G0 X0 Y0 Z0;	-150,-150,-150	0,0,0	-160,-160,-160
G1 X40 Y40 Z40;	-110,-110,-110	40,40, 40	-120,-120,-120

3.22 Workpiece Coordinate System G54~G59

Format:

G54 X__ Y__ Z__; Workpiece coordinate system 1

G55 X__ Y__ Z__; Workpiece coordinate system 2

G56 X__ Y__ Z__; Workpiece coordinate system 3

G57 X__ Y__ Z__; Workpiece coordinate system 4

G58 X__ Y__ Z__; Workpiece coordinate system 5

G59 X__ Y__ Z__; Workpiece coordinate system 6

Function:

There are 6 workpiece coordinate systems for machine tool regardless of the G92, any of coordinate system can be selected by G54~G59.

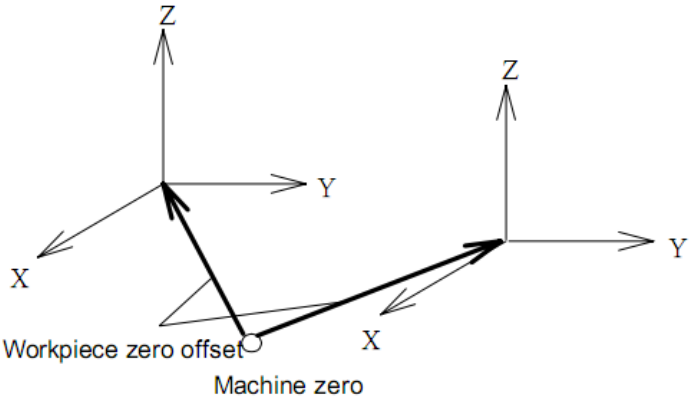
Explanation:

X: New X axis absolute coordinate in current position;

Y: New Y axis absolute coordinate in current position;

Z: New Z axis absolute coordinate in current position.

These six workpiece coordinates are set by the distances (workpiece zero offset) from machine zero to each coordinate system origin.

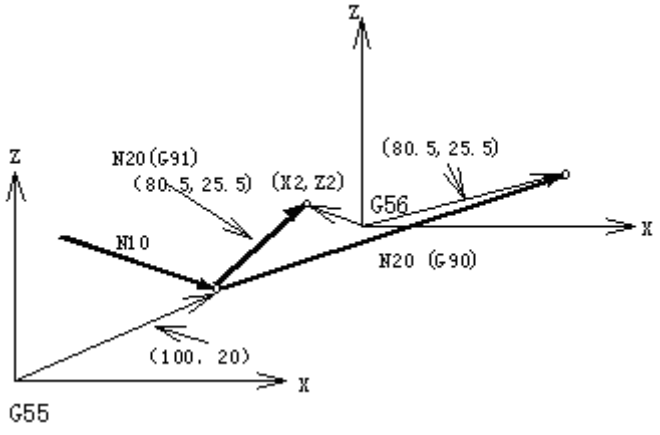


Examples:

N10 G55 G90 G00 X100.0 Z20.0;

N20 G56 X80.5 Z25.5;

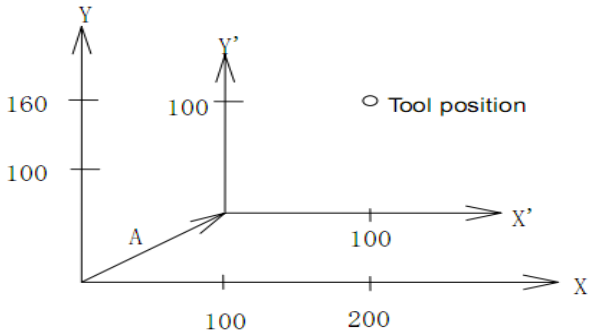
Rapidly positioning to workpiece coordinate system 3 (X=80.5, Z=25.5) from workpiece coordinate system 2 (X=100.0, Z=20.0). For example, if N20 block is G91, it is incremental movement. The absolute coordinates automatically become the coordinates in coordinate system G56.



The absolute position for the figure is coordinate value under the current coordinate system.

Note:

- Workpiece coordinate systems 1~6 is set up as soon as machine zero return is executed after power-on. When the system is restarted, the coordinate system is the one set by parameter No. 540 SCRD.
- Whether the relative position varies with coordinate system depends on status parameter №540 PPD. when PPD=0, it changes; when PPD=1, it does not change.
- When the workpiece coordinate system function is determined, usually, G92 is not needed to set coordinate system. if G92 is used, coordinate system 1~6 will be moved. Do not confuse with G92 and G54~G59, unless workpiece coordinate systems G54~G59 are to be moved. When G54~G59 are in the same block with G92, G54~G59 are disabled.
- Workpiece coordinate system can be modified in the program run. The new coordinate system is effective till the system is restarted.



If it performs G92 X100 Y100 commands when the tool is positioned a（t 200，160）in the G54 coordinate system; the offset vector A for workpiece coordinate system 1 is (X', Y'). And the other workpiece coordinate systems offset for vector A.

3.23 Additional Workpiece Coordinate System G54.1

The system supports sixes standard workpiece coordinate systems (G54~G59), and also uses 48 additional workpiece coordinates.

Command format: G54.1 Pn or G54 Pn

Pn: specify commands of workpiece coordinate system

n: 1~48

Example:

G54.1 P1----------additional workpiece system 1

G54.1 P2----------additional workpiece system 2

⋮

G54.1 P48---------additional workpiece system 48

When P command and G54.1(G54) are executed in the same block, a corresponding workpiece coordinate system in additional workpiece coordinates is selected. The selected workpiece coordinate system is valid till it is replaced by another one.

When the system is started again, the defaulted workpiece coordinate system is controlled by SCRD of No.540.

Restrictions:

After G54.1, P must be specified, the workpiece coordinate system 1 （G54.1 P1）is defaulted when P following G54.1 is not specified in the same block.

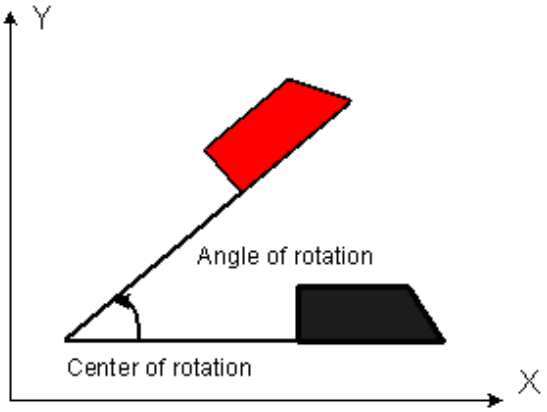
P/S alarm occurs when the specified command value in P command exceeds 1~48.

When other G commands containing P command in G54.1(G54) is specified, P command is shared when they are executed. It is shown below: G54.1 G04 P1000 or G54 M98 P48

Other notes are the same those of G54~G59.

3.24 Coordinate System Rotation G68, G69

The programmed shape can be rotated. By using this function (rotation command), a workpiece can be rotated with a specific angle. If the pattern of the workpiece comprising some identical shapes, the time required for programming and the length of the program can be reduced by editing a subprogram and calling it with the rotation command of the main program. The function is as follows:



Format:

```
G17
G18 G68 α_ β_ R_;    //Start rotation of a coordinate system
G19
...
...          //Coordinate system rotation mode (The coordinate system is rotated)
G69;  // Cancel rotation of a coordinate system
```

G68, G69 are G codes of the group 16, which is a modal G code.

Explanations:

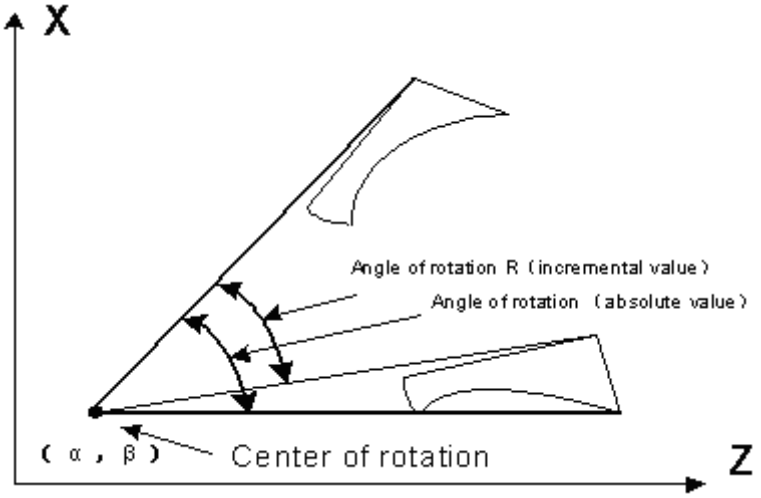
G17 (G18 or G19): Because they only support the rotation on two-dimension plane, select related plane and perform rotation on it.

α, β: The center of rotation. Absolute coordinate commands of two axes of X, Y and Z axes that correspond to the specified coordinate plane.

R: The positive value of angular displacement indicates CCW rotation. The state parameter NO.5410（RIN）determines whether the rotary angle is always an absolute value or specified by a specific G code (G90/G91). When R is not defined, the value specified by the data parameter NO.5410 is taken as the angular displacement value.

The minimum input unit of the angular displacement: 0.001deg（IS-B）

Effective data range of angular displacement: -360.000~360.000deg（IS-B）



Absolute angle：

If the default initial absolute angle is 0 degree, the first specified absolute angle is equivalent to the relative angle specified the same value. If the figure is rotated to the position of 90 degrees, it can be specified by absolute angle or relative angle. Because the position that rotates to 90 degrees (absolute) for the first time and where rotates 90 degrees (relative) from 0 degree are the same. When the figure has been rotated 90 degrees and then 30 degrees to be rotated, rotary angle of 120 degrees should be edited if absolute angle specifying is used, that is, rotate the figure to 120 degrees from 0 degree. If relative specifying is used, angle of rotation should be edited as 30 degrees, that is, rotates 30 degrees from 90 degrees to achieve the same effect with the other way.

Explanations

- Plane selection
  Plane selecting code (G17-gG19) can not be specified in the coordinate system rotation mode.
- Center of rotation
  When α, β are not programmed, the tool position (current position) of block G68 is assumed as the center of rotation.
- Angle of rotation
  When angle of rotation (R command) is not specified, the value specified in parameter NO.187 is assumed as the angle.
- Rotation cancellation
  The G code (G69) used to cancel coordinate system rotation mode can be specified in a block where another command is specified.

Limitations:

- Commands related to the coordinate system
  In coordinate system rotation mode, G codes (G28, G29, G30, etc.) related to reference position return and those related to coordinate system (G52 to G59，G92) can not be specified. These G codes should not be specified in coordinate system rotation mode, otherwise, alarm occurs.
- Radius compensation C mode
  In radius compensation C mode, coordinate rotation mode can not set to be valid. First cancel the radius compensation if it is necessary to specify.
- Rotation center command
  Rotation center must be specified by absolute value in coordinate system rotation mode. The rotation center of the relative command is assumed as the absolute command.

Related parameter

0	5	6	2

RIN	***	***	***	***	***	***	***

RIN = 1：The angle of coordinate system rotation is specified by command G90 or G91

RIN = 0：The angle of coordinate system rotation is always an absolute command

5	4	1	0

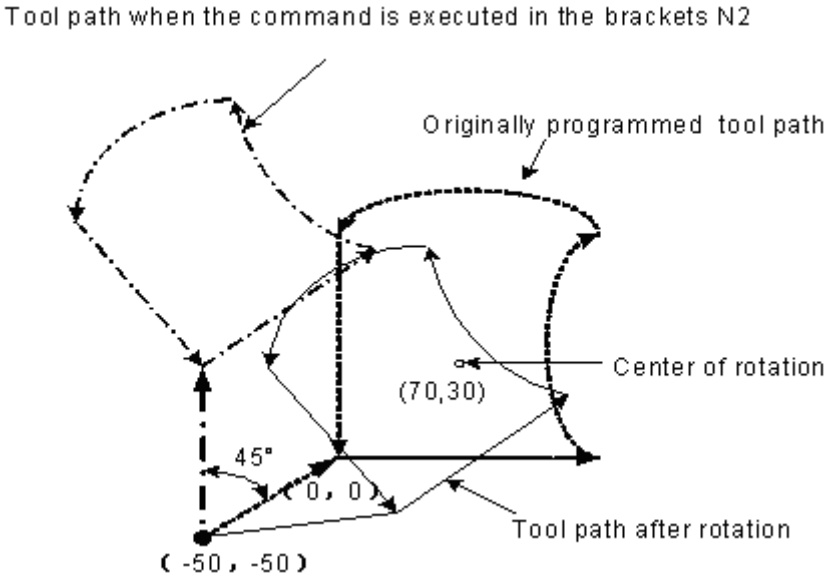
When the angle of rotation is not specified, the angle in coordinate system rotation

Setting range：-360000 ~ 360000（unit:0.001deg）（IS-B）

Example:

```
N1 G90 G69 G54 G49 G40 G17 X-50 Y-50 Z30
N2 G68 X70 Y30 R45 (N2 G68 R45)
N3 G90 G01 X0 Y0 F800
N4 G91 X100
N5 G02 Y100 R100
```

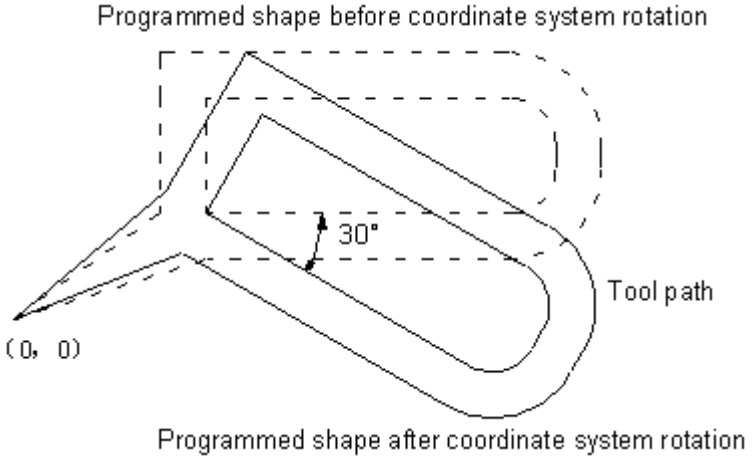
N6 G03 X-100 I-50 J-50
N7 G01 Y-100
N8 G90 G69 X-50 Y-50
N9 M30



Coordinate rotation and cutter compensation C

Cutter compensation C can be specified in G68 and G69 mode. The rotation plane must consistent with the plane of tool compensation.

N1 G90 G69 G54 G17 G00 X0 Y0	N5 G03 Y10 R10 J5
N2 G90 G68 X10 Y10 R-30	N6 G01 X-20
N3 G42 G01 X10 Y10 F800 D1	N7 Y-10
N4 G91 X20	N8 G90 G69 G40 X0 Y0



Scaling and coordinate system rotation

If a coordinate system rotation command is executed in the scaling mode, the rotation center will also be scaled, but not the rotation angle. When a move command is issued, the scaling is executed first and then the coordinate are rotated. The command G68 can not be issued in scaling mode (G51) and cutter compensation C mode. The coordinate system rotation command should always be specified prior to setting the cutter compensation C mode.

- When the system is not in cutter compensation C mode, specify the command in the following order:

  G51； //Scaling mode start

  G68； //Coordinate system rotation mode start

  ;

  G69； // Coordinate system rotation mode cancel

  G50； // Scaling mode cancel

- When the system is in cutter compensation C mode, specify the command in the following order: (Cutter compensation C cancel (G40))

  G51； // Scaling mode start

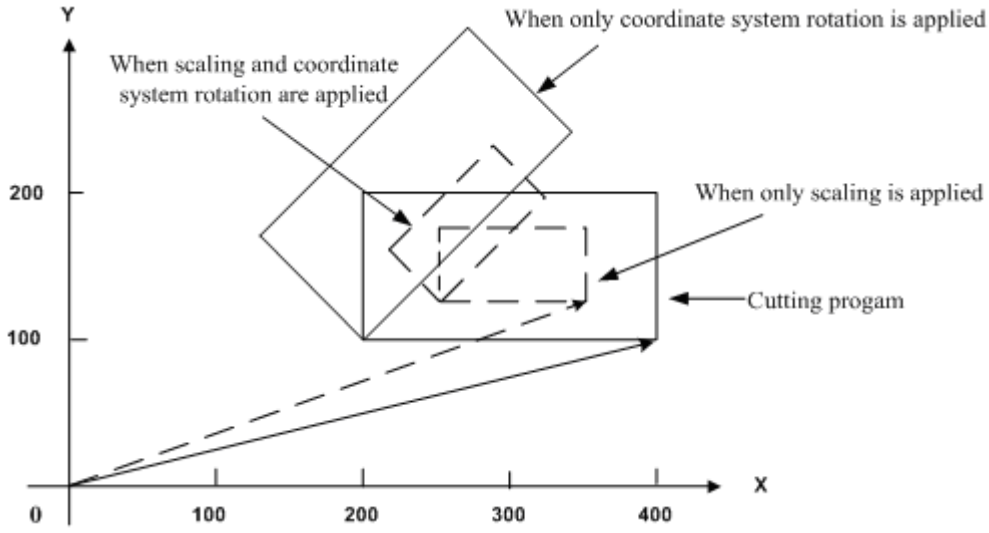
  G68； // Coordinate system rotation mode start

  ;

  G41； // Cutter compensation C mode start

- Scaling and coordinate system rotation

N1 G90 G69 G17 G00 X0 Y0	N5 Y100
N2 G51 X300 Y150 P500	N6 X-200
N3 G68 X200 Y100 R45	N7 Y-100
N4 G91 G01 X400 Y100 F800	N8 X200



Repetitive commands for coordinate system rotation

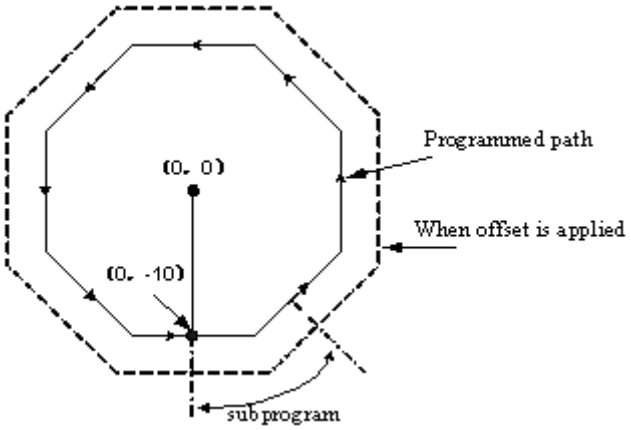
It is possible to store a program as a subprogram and call subprogram several times by changing the angle.

The program when RIN (parameter NO. 562) is set to 1. The specified angular displacement is treated as an absolute or incremental value depending on the specified G code (G90 or G91).

```
Main program
G92 G69 G17 X0 Y0;
G01 F200 H01;
M98 P2100;
M98 P072200;
G90 G00 X0 Y0;
M30;
```

```
O2200
G91 G68 X0 Y0 R45;
G90 M98 P2100;
M99;
```

```
O2100
G90 G42 G01 X0 Y-10;
X4.142;
X7.071 Y-7.071
G40;
M99;
```



Note: Because the radius compensation setting and canceling of the above program are done in the subprogram, the whole workpiece will be over cut if B-type tool starting and retraction of radius compensation C mode is used. In order to compensate the whole rotated workpiece figure with radius compensation function when the above program is in execution, please set state parameter NO.002 SUP to 0, otherwise, above mentioned effects will not be achieved.

3.25 Compound Cycle Command

3.25.1 High-speed Peck Drilling Cycle G73

Format: G98/G99 G73 X_ Y_ R_ Z_ Q_ F_ L_;

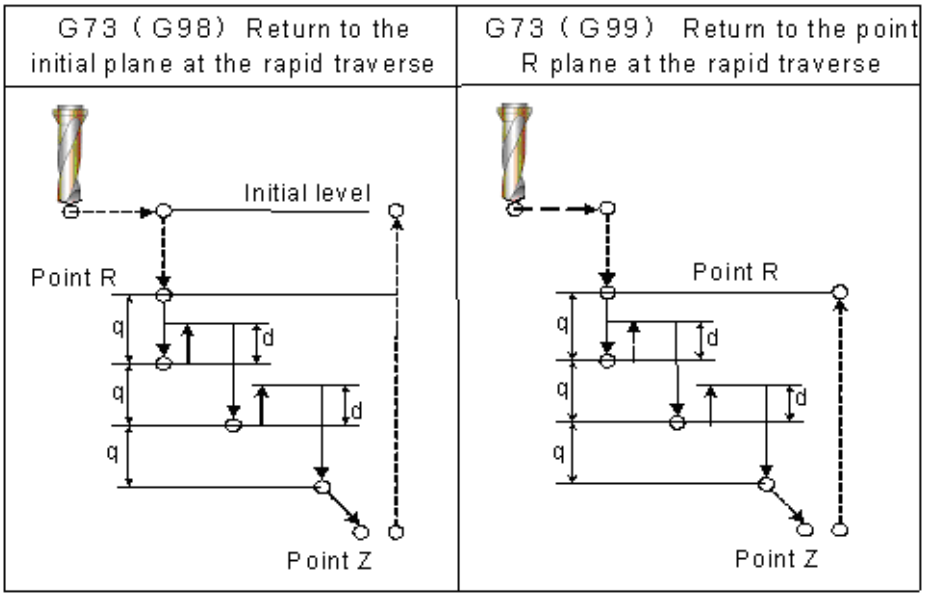
Function: This kind of cycle performs high-speed peck drilling, it performs intermittent cutting feed to the bottom of a hole, and eliminating the chips from the hole simultaneously.

Explanation: Refer to the command explanation of canned cycle in Table 3-2.

Cycle process:

(1) Positioning to XY plane level at the rapid traverse;
(2) Down to the point R plane at the rapid traverse rate;
(3) Cutting feed for Q distance;
(4) Retract *d* distance in rapid traverse;
(5) Cutting feed for (Q+d) distance
(6) Machine to the Z axis hole bottom by cycling the (4) and (5);
(7) Return to the start point level or point R plane according to G98 or G99 at the rapid traverse.

Command Path:



Related Explanation:

(1) This kind of cycle is peck drilling for Q value intermittent feeding along the Z-axis direction. The Q value should be positive, the sign is ineffective even if the negative value is specified. If the Q value is not specified, then it defaults 0.1mm. If a depth to be cut is less than the Q value, then cut to the bottom of the hole without tool retraction at the rapid traverse for the first time.

(2) To remove chips from the hole easily, a small value can be set for retraction. This allows drilling to be performed efficiently. The tool is retracted in rapid feed, the retraction amount d is set by parameter No.5114, the default is 1000, unit: 0.001mm.

(3) The command P is disabled, but its value is reserved as canned cycle modal value.

3.25.2 Left-handed Tapping Cycle G74

Format: G98/G99 G74 X_ Y_ R_ Z_ P_ F_ L

Function: This cycle performs left-handed tapping. In the left-handed tapping cycle, the spindle rotates clockwise for tapping till the bottom of the hole has been reached, then retracts by counter-clockwise after dwell.

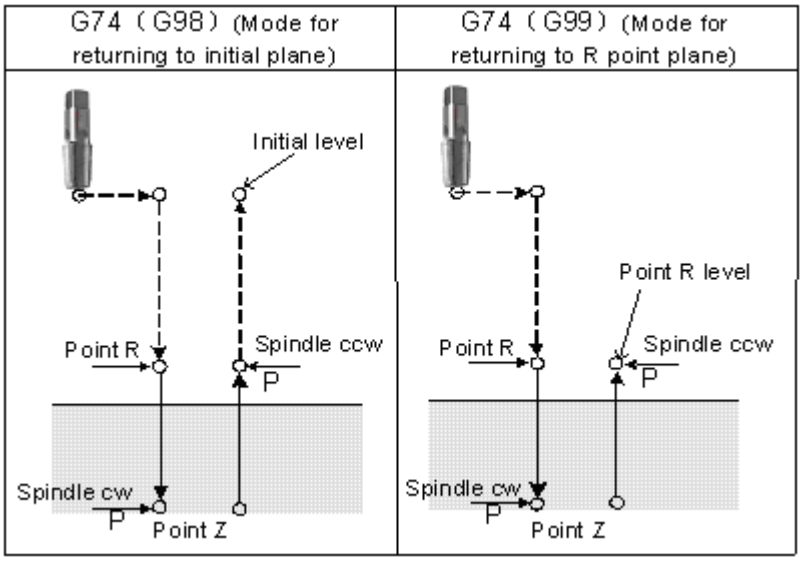
Explanation: For canned cycle explanation, see the Table 3-2

Thereinto, the F is indicated for tooth pitch. The value range are indicated as 0.001 mm ~500.00mm (metric), 0.06~25400 teeth/inch (inch)

Cycle process:

(1) Positioning to XY plane level at the rapid traverse;
(2) Down to the point R plane at the rapid traverse;
(3) Tapping to the bottom of a hole;
(4) The spindle stops;
(5) Pause for time P if dwell is specified;
(6) The spindle rotates CCW, and then retracts to point R plane;
(7) The spindle is stopped; pause for time P if dwell is specified;
(8) Spindle rotates CW;
(9) Return to the initial plane if it is G98.

Command Path:



Related Explanation:

(1) The F is tapping modal value, the last tapping F value is taken when it is omitted, or alarm will be generated if it does not exist.

(2) The metric or inch of the F value is determined by G20 (metric) or G21 (inch).

(3) The command Q is disabled in this cycle, but its value will be reserved as canned cycle modal value.

Note: When No. 3720 (spindle encoder's line or resolution) is set to 0, the drilling hole is done by G74 without checking the encoder.

3.25.3 Finish Boring Cycle G76

Command format

G76 X_ Y_ Z_ R_ Q_ P_ F_ L_

X_ Y_: Positioning data of hole

Z: Hole depth. Incremental value specifies distance between point R plane to the hole or absolute value commands coordinate values of hole bottom.

R: Incremental value specifies distance between point R plane to the hole or absolute value commands coordinate values of hole bottom.

Q: Offset value of hole bottom.

P: Pause time of hole bottom.

F: Cutting feedrate.

L: L means the quantity of hole from starting to the point set by G76. L means the times of drilling hole cycle at the current position.

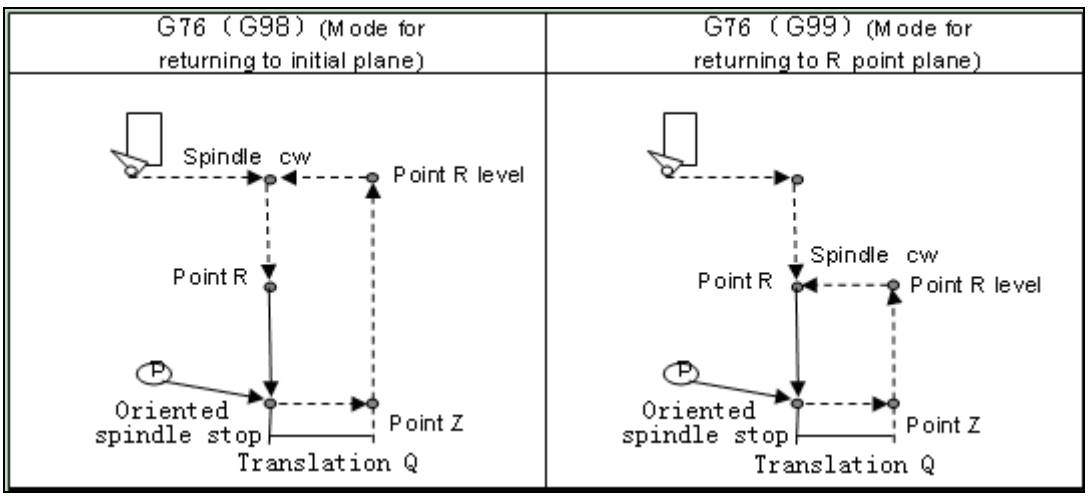
Command function

Fine boring cycle is used to bore precise holes. The tool leaves the workpiece when arriving at the hole bottom, which avoid smooth of workpiece surface influenced by tool trace, and reduces to damage the tool.

Cycle process

(1) Rapidly traverse to XY plane;
(2) Rapidly approach point R plane;
(3) Cut to hole bottom;
(4) When P is commanded, the system pauses within the time P;
(5) When the spindle orientation is executed, the spindle stops at a fixed position;
(6) The tool reversely traverses the distance Q at the hole bottom;
(7) Rapidly return to the initial point plane or point R plane with G98 or G99;
(8) Offset Q distance to XY initial position;
(9) The spindle rotates clockwise;

Command path



➢ **Related explanations**

（1） Q value must be positive, i.e. the negative value is commanded, the sign is invalid; when Q value is not commanded or Q0 is commanded, Q value is defaulted to 0.1mm; Q value is modal, it can be used in other fixed cycle commands, and the Q value cannot be big, otherwise, the tool retraction operation can hit the workpiece, so Q value must be specified to the small.

（2） Offset direction and axis selection are set by No. 584:

0 5 8 4	***	***	***	***	***	***	RD2	RD1

RD1 =1：negative tool retraction

=0：positive tool retraction

RD2 =1：Y-axis executing tool retraction

=0：X-axis executing tool retraction

（3） Direction of tool retraction in final boring cycle is determined by positive/negative X-axis or Y-axis.

（4） M commands selection of spindle orientation is determined by No. 4960, M commands are set to complete the spindle orientation according to the current ladder.

3.25.4 Drilling Cycle, Spot Drilling Cycle G81

Format: G98/G99 G81 X- Y_ R_ Z_ F_ L_ ;

Function: This cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole, the tool is then retracted from the bottom of the hole in rapid traverse.

Explanation: For the command explanation of canned cycle, see the Table 3-2.

Cycle Process:

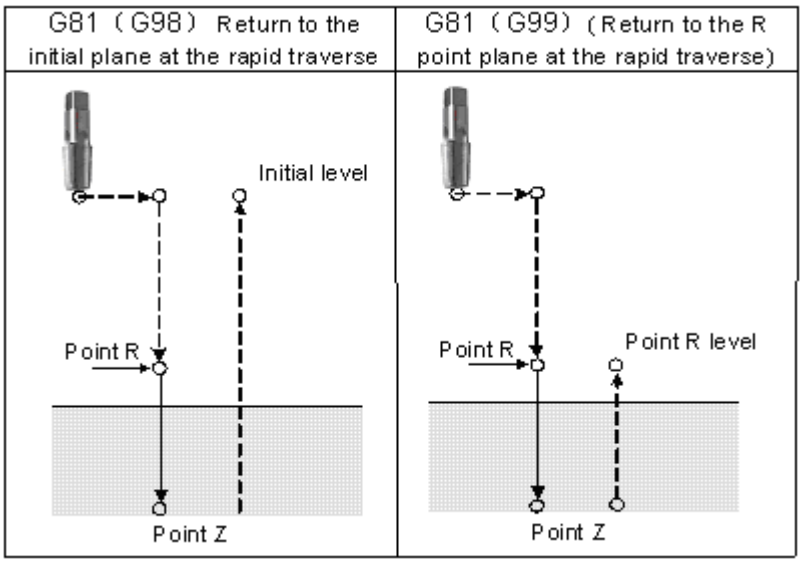
(1) Positioning to the XY plane level position at the rapid traverse;

(2) Down to the point R plane at the rapid traverse;

(3) Cutting feed to the bottom of the hole;

(4) Returning to the initial point or point R plane at rapid traverse according to the G98 or G99;

Command Path:



Related Explanation:

The command Q or P is disabled in this cycle, but its value will be saved as canned cycle modal value.

3.25.5 Drilling Cycle, Counter Boring Cycle G82

Format: G98/G99 G82 X_ Y_ R_ Z_ P_ F_ L_ ;

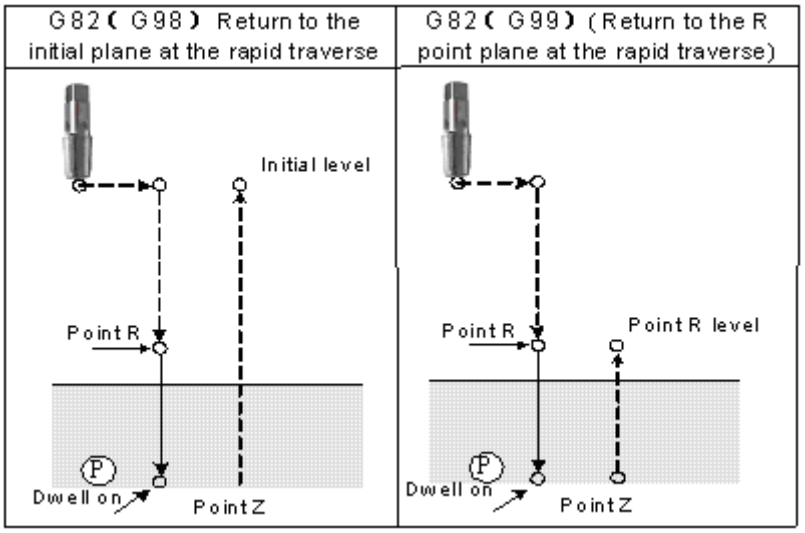
Function: Cutting feed is performed to the bottom of the hole. Hole depth precision is added when the dwell is performed, and then the tool is retracted from the bottom of the hole at rapid traverse.

Explanation: For the command explanation of these canned cycles, see the Table 3-2

Cycle process:

(1) Positioning to the XY plane level at the rapid traverse;
(2) Down to the point R plane at the rapid traverse;
(3) Cutting feed to the bottom of a hole
(4) Dwell for P time if it is commanded.
(5) Returning to the initial point or point R plane according to G98 or G99 at the rapid traverse;

Command Path:



Related Explanation:

(1) They are basically the same as G81 (drilling and spot-drilling machining), it is up after dwell at the bottom of a hole only (the dwell time is specified by P, the dwell will not be executed if it is not specified, and the command action is same as that of G81). In the blind hole, the accuracy of hole can be improved by the dwell.
(2) The command Q is disabled in this cycle, but its value will be reserved as the canned cycle modal value.

3.25.6 Peck Drilling Cycle G83

Format: G98/G99 G83 X_ Y_ R_ Z_ Q_ F_ L_ ;

Function: This cycle performs high-speed peck drilling; it performs intermittent cutting feed to the bottom of a hole while removing chips from the hole.

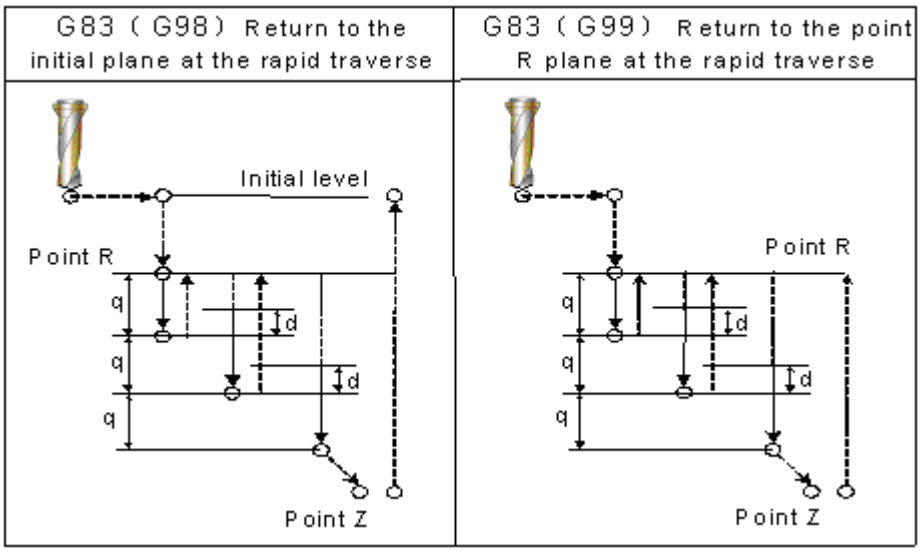
Explanation: The command explanation for canned cycle, see the table 3-2.

Cycle Process:

(1) Positioning to the XY plane level at the rapid traverse;
(2) Down to the point R plane at the rapid traverse;
(3) Cutting feed for Q distance;
(4) Retract to the point R plane at the rapid traverse;
(5) Rapid feed to d distance to the end surface;

(6) Cutting feed for (Q+d) distance;
(7) Cycling (4) (5) and (6) to the bottom of a hole along Z-axis;
(8) Return to the initial point or point R plane according to the G98 or G99 at the rapid traverse.

Command Path:



Related Explanation:

(1) Same as G73, after feeding for Q, it returns to the point R plane at the rapid traverse firstly, and then rapid feeds to d mm to the end surface, then cutting feed is applied and the cycle is performed in turn. The Q value should be positive, even if the negative value is specified, and the sign is also disabled. Q value 0.001mm is defaulted if Q value is not specified; d, is set by the parameter No.5115, its default value is 1000, and the unit is 0.001mm. If the cutting depth is less than the Q value, then cutting to the bottom of a hole at the first time, and rapid traverse retraction is not performed.
(2) The command P is disabled in this cycle, but its value will be reserved as canned cycle modal value.

3.25.7 Tapping Cycle G84

Format: G98/G99 G84 X_ Y_ R_ Z_ P_ F_ L_ ;

Function: This cycle is used to machine a thread. The tapping is performed by spindle rotating positively, when the bottom of a hole has been reached, the spindle is retracted in the reverse direction.

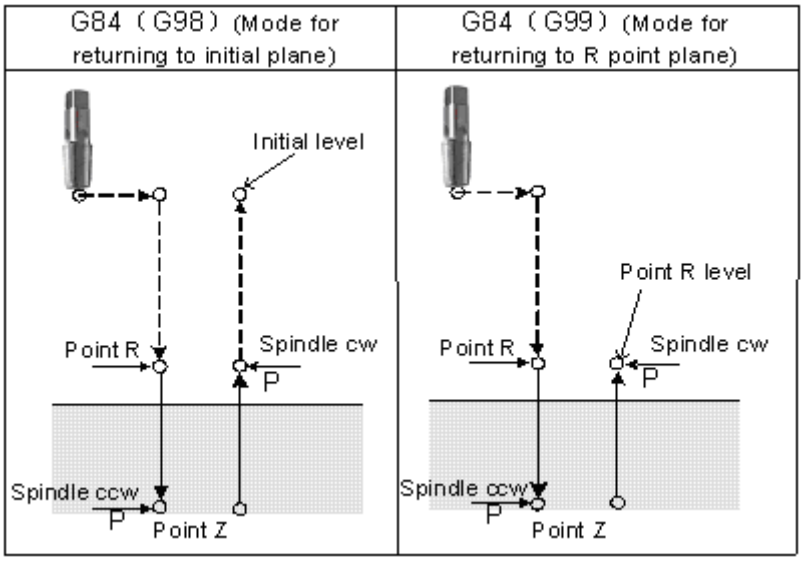
Explanation: For command explanation of canned cycle, see the Table 3-2.

There into, the F is tooth-pitch. The value range is 0.001~500.00mm (metric), 0.06~25400 tooth/inch (inch).

Cycle Process:

(1) Positioning to the XY plane level at the rapid traverse;
(2) Down to the point R plane at the rapid traverse;
(3) Tapping to the bottom of a hole;
(4) Spindle stops;
(5) For dwell time P if it is commanded
(6) Spindle returns to the point R plane in reverse direction;
(7) Spindle stops; for dwell time P if the P is commanded;
(8) The spindle is rotated in the positive direction;
(9) Returning to the initial point level if it is G98.

Command Path:



Related Explanation:

Please refer to the related explanation for G74 (Counter tapping cycle) .

Note: No. 3720 (spindle encoder lines) is set to 0, the encoder is not checked to execute holing when G84 is executed.

3.25.8 Boring Cycle G85

Format: G98/G99 G85 X_ Y_ R_ Z_ F_ L_ ;

Function: After positioning along X and Y axes, rapid traverse is performed to point R; the boring is performed from point R to point Z thereafter. Cutting feed is performed to return point R plane when the Z point has been reached the bottom of a hole.

Explanation: Command explanation for the canned cycle, see the table 3-2.

Cycle process:

(1) Positioning to the XY plane level at the rapid traverse;

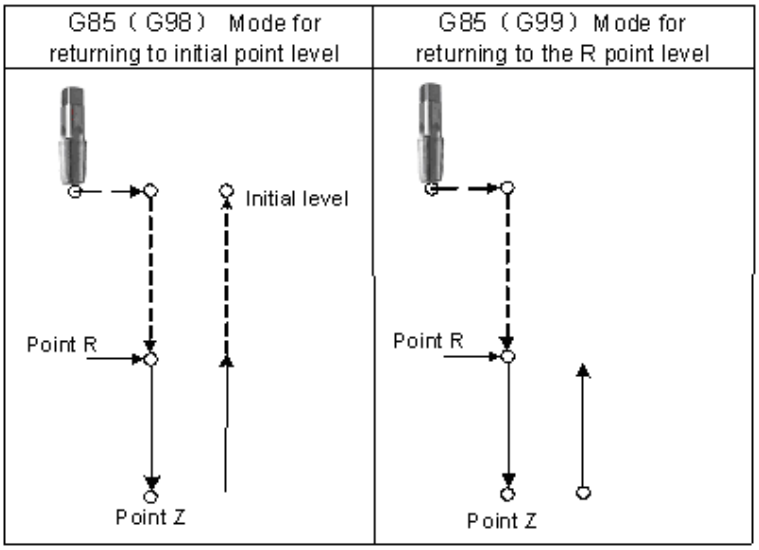
(2) Down to the point R plane at the rapid traverse;

(3) Cutting feed to the bottom of a hole;

(4) Cutting feed to the point R plane;

(5) Returning to the initial point level if it is G98.

Command Path:



Related explanation

(1) This cycle is used to bore a hole. The command motion is basically same as the G81 (Drilling, Spot-drilling cycle), the difference is that by the G81 it returns to the point R plane in rapid traverse rate, while by the G85 it returns to the point R plane in feedrate when the cutting feed reaches the bottom of a hole.

(2) The Q and P commands are disabled in this cycle, but its value is reserved as the canned cycle modal value.

3.25.9 Boring Cycle G86

Format: G98/G99 G86 X_ Y_ R_ Z_ F_ L_ ;

Function: After positioning along X and Y axes, rapid traverse is performed to R point, and the boring is performed from point R to point Z. The tool is retracted in rapid traverse and spindle is rotated positively when the spindle is stopped at the bottom of the hole.

Explanation: For command explanation for canned cycle, see the table 3-2.

Cycle process:

(1) Positioning to the XY plane level at the rapid traverse;

(2) Down to the point R plane at the rapid traverse;

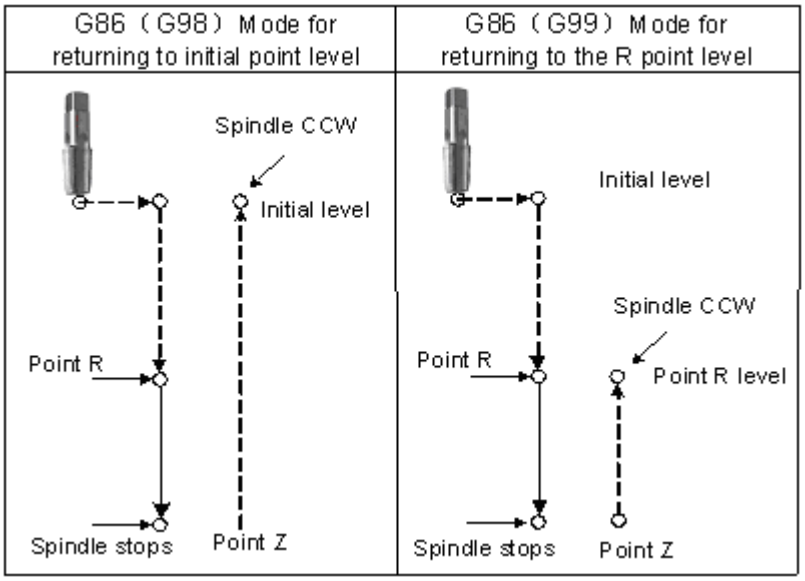
(3) Cutting feed to the bottom of a hole;

(4) The spindle stops;

(5) Returning to the initial point or point R plane at rapid traverse according to the G98 or G99;

(6) The spindle is rotated in the positive direction.

Command Path:



Related explanation:

(1) This cycle is used to be bore a hole. The command operation is basically same with G81, only spindle rotation status is different. After cut feeds to the bottom of a hole, the M05 is executed (spindle stops), then the point R plane is retracted at the rapid traverse, the M03 is then performed (spindle rotates positively) regardless of the currently spindle rotation status and the positive or negative rotation are specified before the canned cycle.

(2) The command Q and P are disabled in this cycle, but its value is reserved as canned cycle modal value.

3.25.10 Back Boring Cycle G87

Command format

G87 X_ Y_ Z_ R_ Q_ P_ F_ L_

X_ Y_：hole's positioning data.

Z：hole depth. Incremental specifies distance between point R plane and hole bottom or absolute value specifies coordinates of hole bottom.

R：Incremental specifies distance between point R plane and hole bottom or absolute value specifies coordinates of point R plane.

Q：Offset amount of hole bottom.

P：Pause time of hole bottom.

F：Cutting feedrate.

L：L means quantity of holing from starting point and point set by G87. L means times of currently circular holing when G87 does not set holing point.

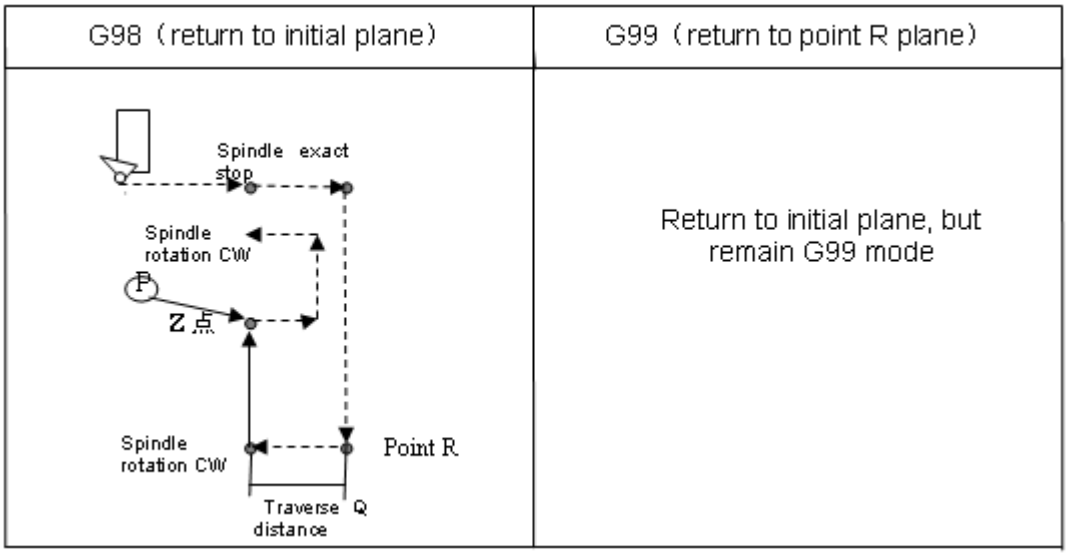
Command function

Back boring cycle is used to boring precise hole, and the tool safely retracts to the initial plane after machining completion.

Cycle process

(1) Rapidly position to XY plane;
(2) The spindle orientation is done and stops at the fixed position after XY plane positions;
(3) Traverse Q distance;
(4) Rapidly traverse to point R;
(5) Traverse Q distance;
(6) Spindle rotates CW;
(7) Execute boring cycle to point Z;
(8) When P is commanded, the system pauses at P time;
(9) The spindle orientation is executed, it stops at the fixed position;
(10) Traverse Q distance;
(11) Return to initial plane;
(12) Offset Q distance to XY initial position;
(13) The spindle rotates CW.

Command path



Related explanation：

(1) Q value must be positive, i.e. the negative value is commanded, the sign is invalid; when Q value is

not commanded or Q0 is commanded, Q value is defaulted to 0.1mm; Q value is modal, it can be used in other fixed cycle commands, and the Q value cannot be big, otherwise, the tool retraction operation can hit the workpiece, so Q value must be specified to the small.

(2) Offset direction and axis selection are set by No. 584:

0 5 8 4								
	***	***	***	***	***	***	RD2	RD1

RD1 =1：negative tool retraction

=0：positive tool retraction

RD2 =1：Y-axis executing tool retraction

=0：X-axis executing tool retraction

(3) Direction of tool retraction in final boring cycle is determined by positive/negative X-axis or Y-axis.

(4) G87 executes only G98 returning initial point plane, its mode remains after it is specified even if specifying G99 is invalid.

(5) M commands selection of spindle orientation is determined by No. 4960, M commands are set to complete the spindle orientation according to the current ladder.

3.25.11 Boring Cycle G88

Format: G98/G99 G88 X_ Y_ R_ Z_ P_ F_ L_ ;

Function: A dwell is performed at the bottom of a hole, the spindle is stopping. If the manual operation is applied now, tool can be removed manually. It is better to retract the tool safely from the hole regardless of any kind of manual operation. It is rapidly retracted to point R or initial plane when the automatic operation is performed again, the spindle is stopped and G88 is finished.

Explanation: For the command explanation of the canned cycle, see the table 3-2.

Cycle process:

(1) Positioning to the XY plane at the rapid traverse rate;

(2) Down to the point R plane at the rapid traverse rate;

(3) Cutting feed to the bottom of hole;

(4) The spindle is stopped;

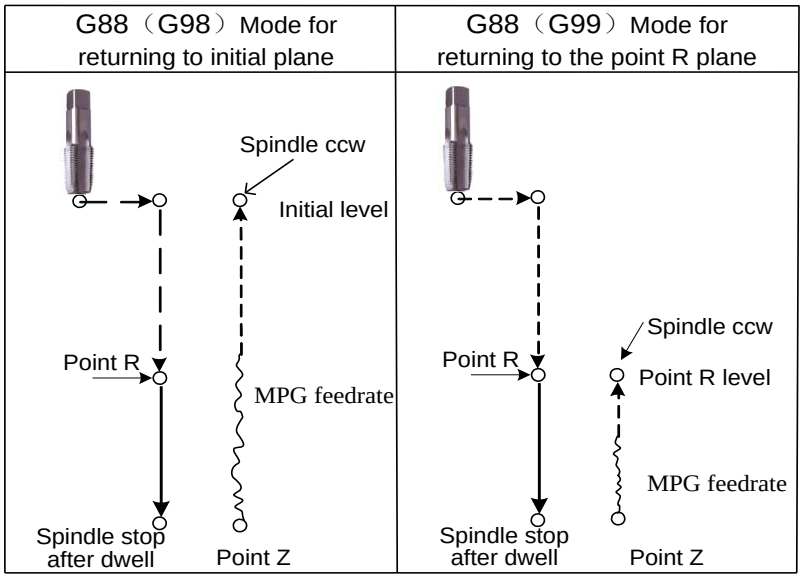
(5) P time is delayed if it is specified.

(6) Manual operation will be performed if the dwell is executed.

(7) Restoring the automatic mode, retracting to initial point or point R plane according to the G98 or G99 at the rapid traverse rate.

(8) The spindle rotates positively;

Command Path:



Related Explanation:

The command Q is disabled in this cycle, but its value is reserved as the canned cycle modal value.

3.25.12 Boring Cycle G89

Format: G98/G99 G89 X_ Y_ R_ Z_ P_ F_ L_ ;

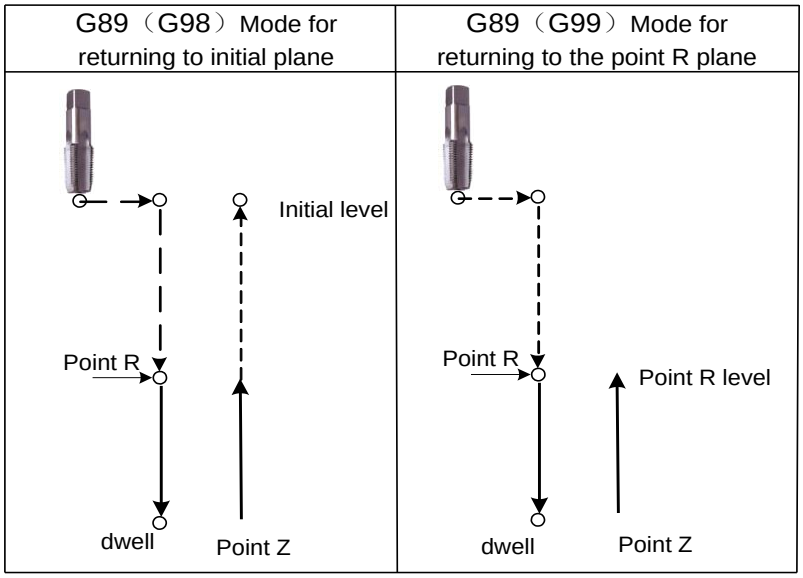
Function: This cycle is used to bore a hole normally. This cycle performs a dwell at the bottom of the hole; the tool is then retracted from the bottom of the hole at the rapid traverse rate.

Explanation: For the command explanation of the canned cycle, see the table 3-2.

Cycle process:

(1) Positioning to XY plane at the rapid traverse rate;
(2) Down to the point R plane at the rapid traverse rate;
(3) Cutting feed to the bottom of a hole;
(4) For dwell time P if the P is specified;
(5) Cutting feed to the point R plane;
(6) Returning to the initial point level if it is G98;

Command Path:



Related Explanation:

(1) G89 (Boring cycle) is basically same as the G85, a dwell is applied at the bottom of a hole (Dwell time is specified by P, if it is not specified, the dwell is not applied, the command operation is same to the G85)
(2) The command Q is disabled in this cycle, but its value is reserved as canned cycle modal value.

3.25.13 Groove Rough Milling Inside the Round G110/G111

Format:

G98/G99 G110/G111 X_ Y_ R_ Z_ I_ W_ Q_ K_ V_ E_ D_ F_

Function: From the beginning of the center point, arc interpolations are performed helically till the round groove of programming dimension has been machined.

Explanation: For command explanation of the canned cycle, see the table 3-2.

G110：Groove rough-milling inside the round in CCW;

G111: Groove rough-milling inside the round in CW;

I: I is radius inside the round groove, it should be more than the radius of current tool.

W: The firstly cutting depth is from the R reference level to the undersurface along the Z axis direction, it should be more than 0 (The first cutting position is over the bottom of the groove, then bottom position is regarded as machining position);

Q: The cutting incremental value each time along Z axis direction;

K: The width increment of cut inside XY plane, it should be less than the tool radius, and more than 0;

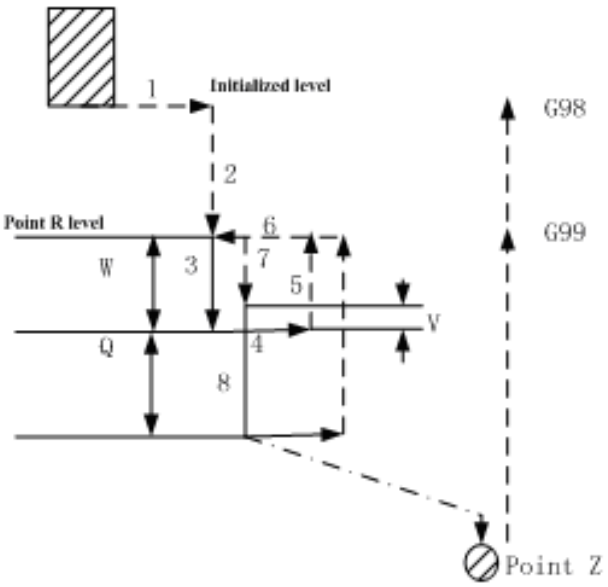
V: The distance to the end machining plane at the rapid traverse, it should be more than 0 when cutting;

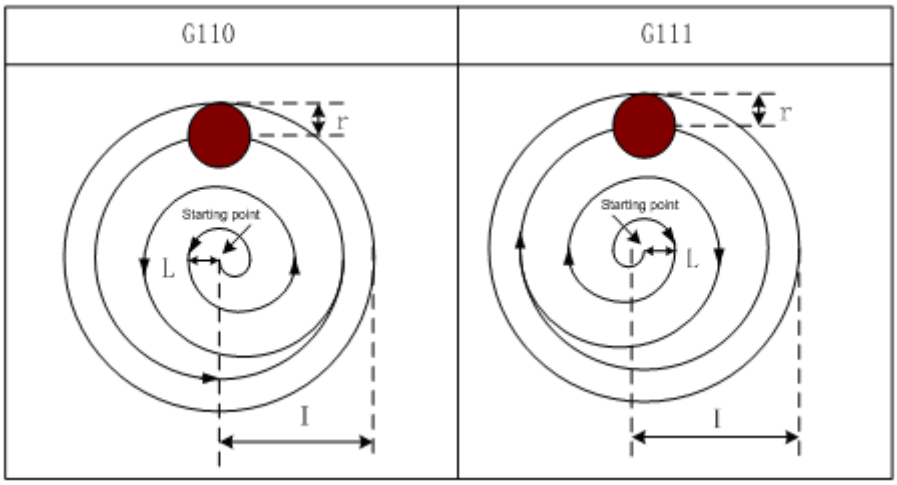
D: Tool radius serial number, the value range is 0~32, 0 is the default of D0. The current tool radius is determined by the specified serial number.

Cycle process:

(1) Positioning to the XY plane level at the rapid traverse rate;
(2) Down to the point R plane at the rapid traverse rate;
(3) Cut W depth downwards in cutting feedrate;
(4) Mill a round face with radius I helically by K increment each time from center point to outside;
(5) The Z axis is retracted to the R reference surface at the rapid traverse rate;
(6) X and Y axes are positioned to the center at the rapid traverse rate;
(7) Down to distance V to the end machining surface along Z axis at the rapid traverse rate;
(8) Cut along Z axis for (Q+V) depth;
(9) Cycling the operations from (4) ~ (8) till the round surface of total depth is finished;
(10) Return to the initial plane or point R plane according to G98 or G99;
(11) Return to XY starting point.

Command Path:



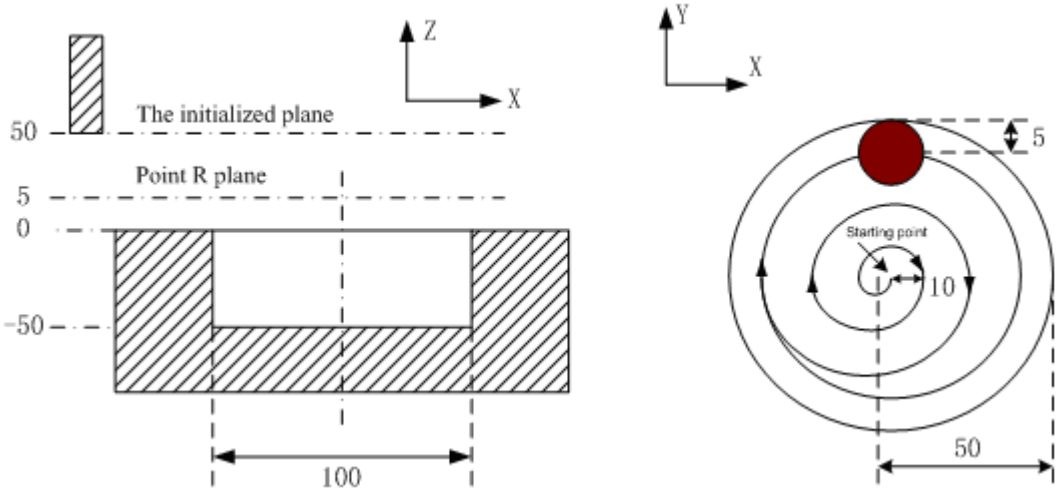


Related Explanation:

The P and L are disabled in this cycle, but the P value will be reserved as canned cycle modal value.

For example:

A round inside groove rough-milling is specified in canned cycle G111, see the following Figure.



G90 G00 X50 Y50 Z50; (G00 positioning at the rapid traverse rate)

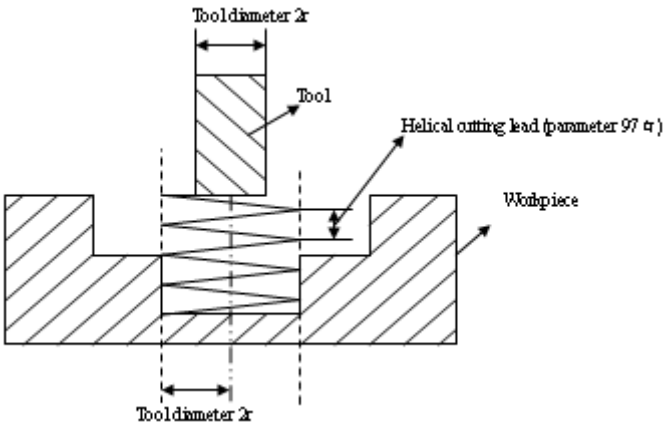
G99 G111 X25 Y25 R5 Z-50 150 W20 Q10 K10V10 F800 D1; (Rough-milling cycle inside the round groove D1=5)

G80 X50 Y50 Z50; (Canceling canned cycle, returning from the point R plane)

M30;

Note: Set the 5122# parameter value to one which is more than 10, by G110and G111 it feeds helically along Z axis;The value of D must be greater than or equal to minimu increment of 2000 times and less than or equal to 2/3 I; If increment is IS-C,the value of D must be greater than or equal to 0.2 and less than 2/3I.Rough-milling machining can be directly performed for non-groove workpiece.

See the following figure for helical cutting path:



3.25.14 Fine-milling Cycle Inside Full Circle G112/G113

Format:

G112

G98/G99 X_ Y_ R_ Z_ I_ J_ D_ F_

G113

Function: A fine-milling inside the full circle is finished with the specified radius value I and direction, the tool is retracted after the fine-milling.

Explanation: For command explanation of canned cycle, see the table 3-2.

G112: Fine-milling cycle inside the full circle in CCW.

G113: Fine-milling cycle inside the full circle in CW

I: Fine-milling circle radius, the value range is indicated as 0~9999.999mm, the absolute value is taken when it is negative.

J: Fine-milling distance from start point to the center point, the value range is indicated as 0~9999.999mm, the absolute value is taken when it is negative

D: Sequence number of tool radius, the value range is indicated as 0~32, the 0 is default of D0. The current tool radius value is taken according to the specified sequence number.

Cycle process:

(1) Positioning to the XY plane level at the rapid traverse rate;

(2) Down to the point P level at the rapid traverse rate;

(3) Feed to the bottom of a hole;

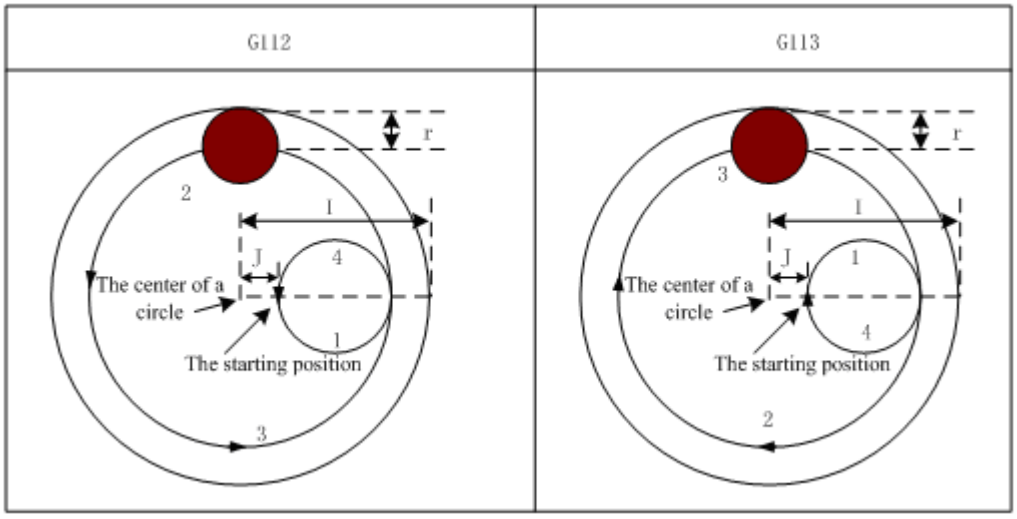
(4) Perform the circle interpolation by the path of transit arc 1;

(5) Perform the full circle interpolation by the path of arc 2 and arc 3;

(6) Perform circular interpolation by the path of transit arc 4 and return to the start point;

(7) Return to the initial point level or point R plane according to G98 or G99.

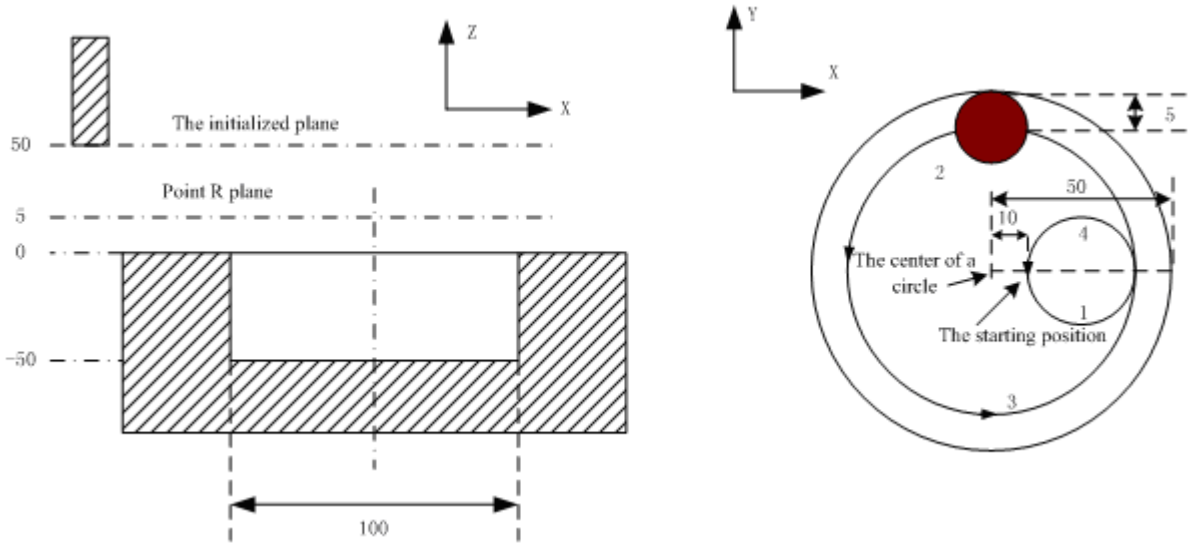
Command Path:



Related Explanation:

The commands Q, P and L are disabled in this cycle, but the Q and P value will be reserved as the canned cycle modal value.

For example: Fine-mill a finished rough-milling round groove by the canned cycle G112 command, see the following figure:



G90 G00 X50 Y50 Z50; (G00 rapid positioning)

G99 G112 X25 Y25 R5 Z-50 150 J10 F800 D1; (Start canned cycle, fine-milling cycle inside the circle at the bottom of a hole D1=5)

G80 X50 Y50 Z50; (The canned cycle is cancelled, returning from the point P level)

M30;

3.25.15 Fine-milling Cycle Outside Circle G114/G115

Format:

G114

G98/G99 X_ Y_ R_ Z_ I_ J_ D_ F_;

G115

Function: A fine-milling outside the full circle is performed by the specified radius value and the direction, and the tool is retracted after the fine-milling is finished.

Explanation: For command explanation of canned cycle, see the table 3-2.

G114: Finish-milling cycle for outside circle in CCW.

G115: Finish-milling cycle for outside circle in CW.

I: A fine-milling circle radius, value range: -99999999～99999999×least command increment, the absolute value is taken when it is negative.

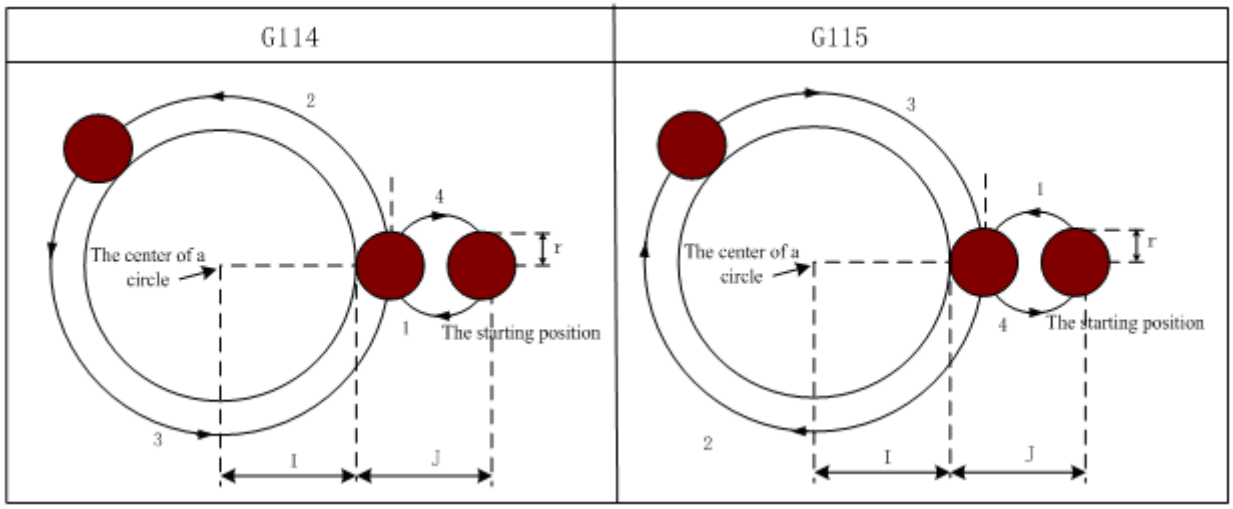
J: Distance of fine-milling between the start point and the circle, 取值范围-99999999～99999999×least command increment，the absolute value is taken when it is negative; the absolute value is taken when it is negative.

D: The sequence number of tool radius, the value range is 0~32, 0 is the default of D0. The current tool radius value is taken according to the specified sequence number.

Cycle process:

(1) Positioning to the XY plane level at the rapid traverse rate;
(2) Down to the point R plane at the rapid traverse rate;
(3) Cutting feed to the bottom of a hole;
(4) Perform the circle interpolation by the path of transit arc 1;
(5) Perform the full circle interpolation by the path of arc 2 and arc 3;
(6) Perform circular interpolation by the path of transit arc 4 and return to the start point;
(7) Return to the initial point level or point R plane according to G98 or G99.

Command path:

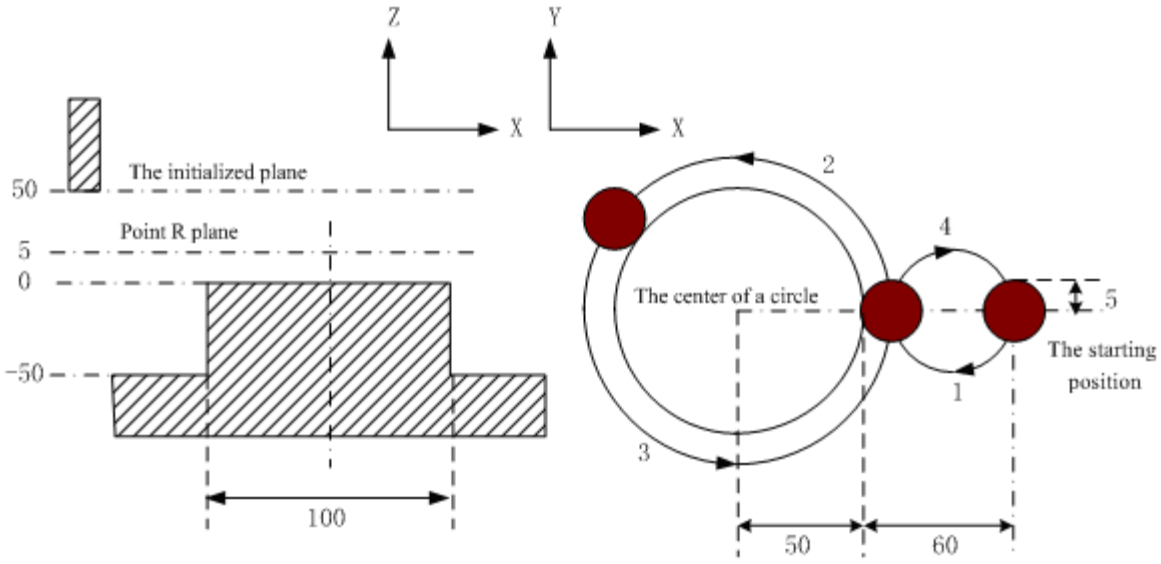


Related Explanation:

(1) The interpolation direction of between transit arc and fine-milling arc are different when the fine-milling outside circle is performed, the interpolation direction in command explanation is the interpolation direction of fine-milling arc.

(2) The command Q, P and L are disabled in this cycle, but the Q and P value are reserved as canned cycle modal value.

For example: A finished rough-milling round groove is performed by fine-milling with the canned cycle G114 command, see the following figure :




```
G90 G00 X50 Y50 Z50;        (G00 rapid positioning)
G99 G114 X25 Y25 R5 Z-50 150 J60 F800 D1;    (Start canned cycle, the fine-milling cycle is
                                              performed outside the circle at the bottom of
                                              a hole D1=5)
G80 X50 Y50 Z50;        (The canned cycle is cancelled, returning from the point R plane)
M30；
```

3.25.16 Roughing Cycle Outside Circle G116/G117

Command format: ：

G116

G98/G99 X_ Y_ Z_ R_ I_ J_ W_ Q_ K_ C_ E_ D_ F_

G117

Command function: the tool executes the full-circle interpolation at the specified radius value and direction from the starting point till the programmed circle convex plate is machined completely.

Command explanation: related commands are referred to Table 3-2.

G116：roughing cycle for outside circle in CCW.

G117：roughing cycle for outside circle in CW.

I：outer roughing circle radius (it should be more than 0, and its absolute value is done when it is negative) .

J：blank workpiece radius (it should be more than 0, and its absolute value is done when it is negative）.

E：allowance of excircle roughing in XY plane(it should be more than 0, and its absolute value is done when it is negative) .

W：Z-axis cutting depth of first starting downward from R reference plane should be more than 0, its absolute value is done when it is negative (when the first cutting depth exceeds the groove bottom, the groove bottom is the reference to machine) .

Q：Z-axis cutting depth every time, its absolute value is done when it is negative.

K：cutting width increment in XY plane (it should be less than tool diameter, its absolute value is done when it is negative; when K is not specified, K is defaulted to tool radius D) .

C：X-axis cutting amount of first tool infeed(it should be more than or equal to tool radius +2.0. when C value is more than 0, the tool executes infeed in positive X direction, the workpiece is at positive starting point. When it is less than 0, the tool executes infeed in negative X direction, the workpiece is at negative starting point.）.

D: serial number of tool radius, range: 0~32，D0 is defaulted to be 0. The current tool radius value is taken out according to the given serial number.

Cycle process：

(1) Rapidly position to starting point XY plane.

(2) Rapidly approach downward point R plane.

(3) Rapidly approach downward the distance W.

(4) X-axis firstly executes tool infeed amount C，linear 1 is the path to execute the linear interpolation infeed.

(5) Execute full-circle interpolation based on the path arc 2.

(6) Mill the circle surface(its radius is I+E) from outer to inner in an increment of K value every time.

(7) Z-axis rapidly positions to R reference plane.

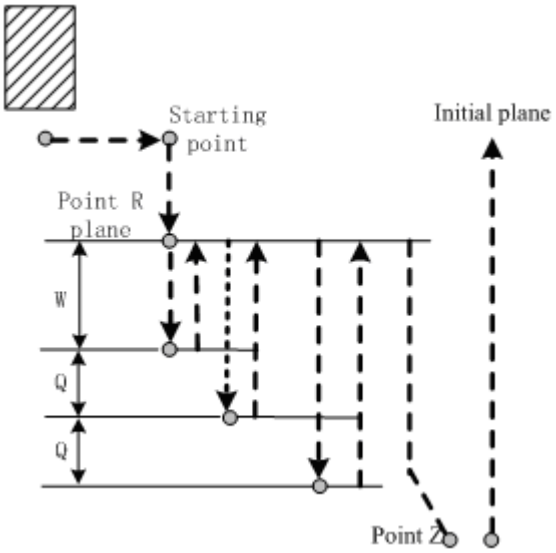
(8) Rapidly position to starting point XY plane.

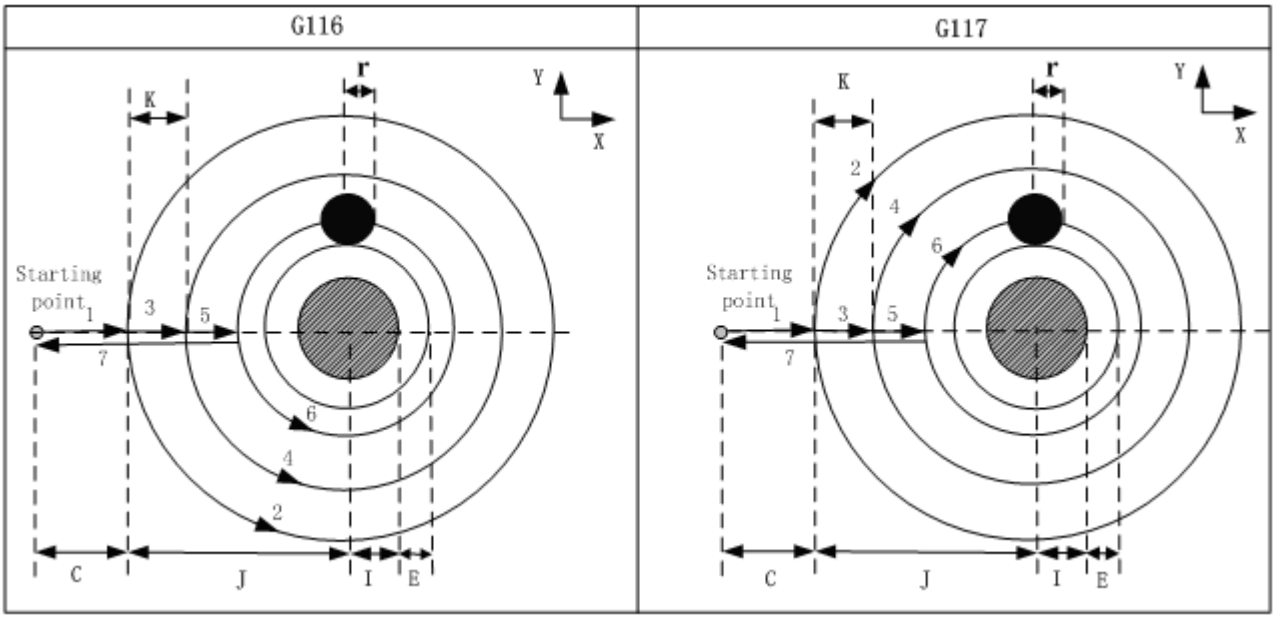
(9) Z-axis rapidly approaches the distance (unmachined distance +Q) .

(10) Execute the above operation step 5~9 till the circle surface (its total cutting depth is Z) is machined completed.

(11) Return to initial point plane or point R plane according to G98 or G99.

(12) Return to XY starting point.

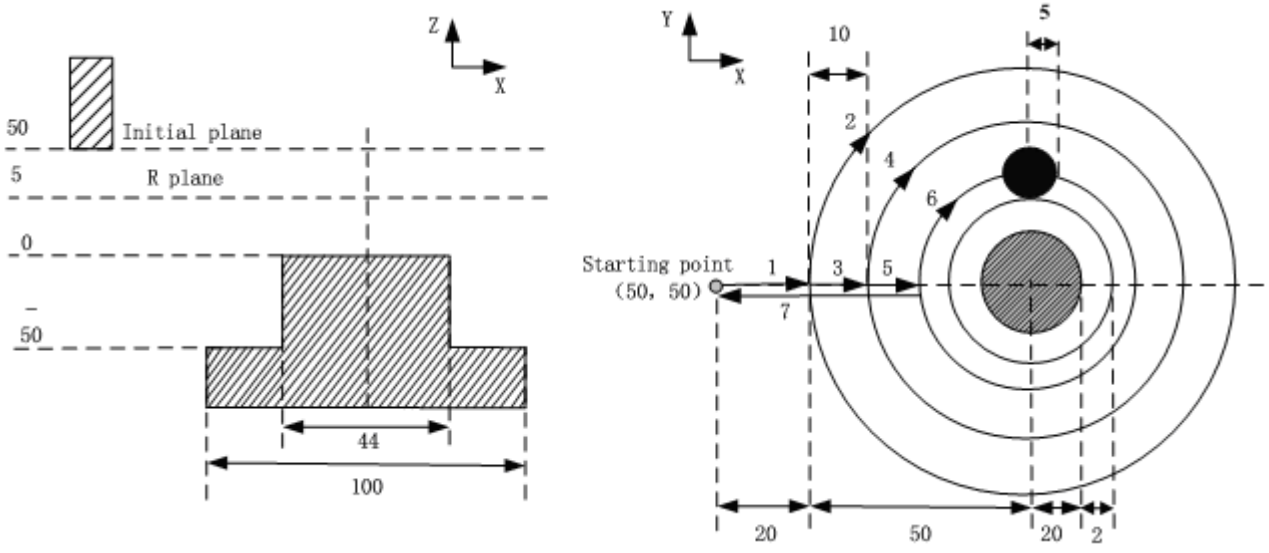




Related explanation:

Commanding P, L in the cycle is invalid, but P value is saved as modal value of the fixed cycle.

Example: G117 commands to rough mill a circle convex plate, which is shown below:



```
G90 G00 X0 Y0 Z50;    （G00 rapidly position）
G99 G117 X50 Y50 R5 Z-50 I20 J50 W20 Q10 K10 C20 E2 F800 D1;    （ execute the rough milling cycle of outer convex plate D1=5）
G80 X50 Y50 Z50;    （cancel fixed cycle, return from point R plane）
M30;
```

3.25.17 Plane milling G126/G127

Command format:

G98/G99 G126/G127 X_ Y_ Z_ R_ I_ J_ K_ D_ F_

Command function: According to the specified length and width of each side, mill into a plane, and finally return to the XY specified position.

Command explanation:

G126: Round trip milling; G127: Unidirectional milling;

X. Y: Starting point coordinates, incremental value or absolute value;

Z: Depth cutting position in the Z-axis direction, incremental or absolute value;

R: R plane position, incremental or absolute value;

I: The range in the X-axis direction is directional, with positive values indicating incremental values in the positive direction of the starting point and negative values indicating incremental values in the negative direction of the starting point;

J: The range in the Y-axis direction is directional, with positive values representing incremental values in the positive direction of the starting point and negative values representing incremental values in the negative direction of the starting point;

K: The step distance for cutting in the Y direction within the XY plane, with negative values taken as absolute values, should be less than or equal to the tool diameter and greater than 0; When K value is not specified but D value is specified, the cutting tool radius specifies the step distance

F: Cutting feed rate.

G98/G99: After cutting the XY plane, return to the corresponding Z-axis position according to the specifications of G98 and G99. Then return to the plane specified by XY.

D: The tool radius serial number has a value range of 0~32, with D0 set to 0 by default. Retrieve the current tool radius value based on the given serial number.

Loop process:

(1) Quickly locate the starting position of the XY plane.

(2) Quickly descend to the R-point plane.

(3) Cut at cutting speed to the depth cutting position in the Z-axis direction.

(4) If it is a back and forth milling G126, mill I along the X-axis direction, step K in the Y-axis direction in the XY plane, mill I along the opposite direction, and continue milling K along the Y-axis direction.

(5) If milling G127 in one direction, milling I horizontally along the X-axis direction, moving I quickly in the opposite direction, and milling K horizontally along the Y-axis direction in the XY plane.

(6) Repeat (4) until milling J in the Y-axis direction.

(7) According to G98/G99, the Z-axis direction can quickly move to the starting point plane or the R-point plane.

(8) According to parameter 0582, quickly move to the starting position in the XY plane, or do not return and maintain the processed XY position.

Related parameters:

0 5 8 2

BKST							

BKST: 0- After completing the plane processing, return to the XY initial position;

1-After completing the plane processing, do not return and maintain the XY position after processing.

Note: G126 and G127 are modal instructions for group 01, and I J K is not saved.

3.25.18 Rectangular Roughing G132/G133

Command format:

G132

G98/G99 X_ Y_ Z_ R_ I_ J_ A_ B_ W_ Q_ K_ C_ U_ E_ D_ F_

G133

Command function: the tool executes the linear cutting cycle from starting at the specified parameter data till the programmed rectangular convex plane is machined completed.

Command explanation: related commands are referred to Table 3-2.

G132：outer rectangle roughing CCW

G133：outer rectangle roughing CW

XY：start-up coordinates in XY plane

I：X-axis width of outer rectangle roughing(it is more than 0, and its absolute value is done when it is negative）.

J：Y-axis width of outer rectangle roughing（it is more than 0, and its absolute value is done when it is negative）.

A: X-axis width of blank（it is more than 0, and its absolute value is done when it is negative）.

B：Y-axis width of blank（it is more than 0, and its absolute value is done when it is negative）.

E：Allowance in XY plane of outer rectangle roughing（it is more than 0, and its absolute value is done when it is negative.）.

W：Z-axis depth of first cutting is the distance downward from R reference plane, and should be more than 0, its absolute value is done when it is negative (when the first cutting depth exceeds groove bottom position, directly machining the groove position is done）.

Q：Z-axis cutting depth increment every time. its absolute value is done when it is negative.

K：Cutting width increment in XY plane（it should be less than tool diameter, its absolute value is done when it is negative; when K is not specified, K is defaulted to tool radius D）.

C：X-axis cutting amount of first tool infeed (it should be more than or equal to tool radius +2.0. when C value is more than 0, the tool executes infeed in positive X direction, the workpiece is at positive starting point. When it is less than 0, the tool executes infeed in negative X direction, the workpiece is at negative starting point.）.

U：Coner arc radius. Omitting it means no corner arc transition. its absolute value is done when it is negative.

D：serial number of tool radius, range: 0~32，D0 is defaulted to be 0. The current tool radius value is taken out according to the given serial number.

Cycle process:

(1) Rapidly position to starting point XY plane.

(2) Rapidly approach downward point R plane.

(3) Rapidly approach downward the distance W

(4) X-axis firstly executes tool infeed amount C， linear 1 is the path to execute the linear interpolation infeed.

(5) Execute full-circle interpolation based on the path arc 2.

(6) Mill the rectangle(its length is I+2E, width J+2E) from outer to inner in an increment of K value

every time.

(7) Mill arc angle of the rectangle

(8) Z-axis rapidly positions to R reference plane.

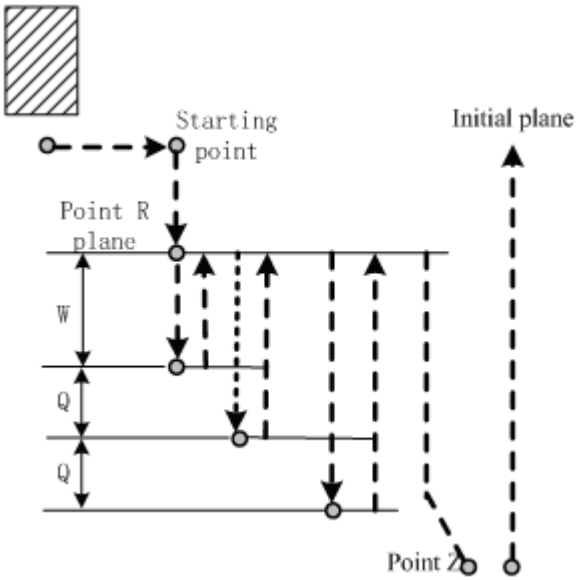
(9) Rapidly position to starting point XY plane.

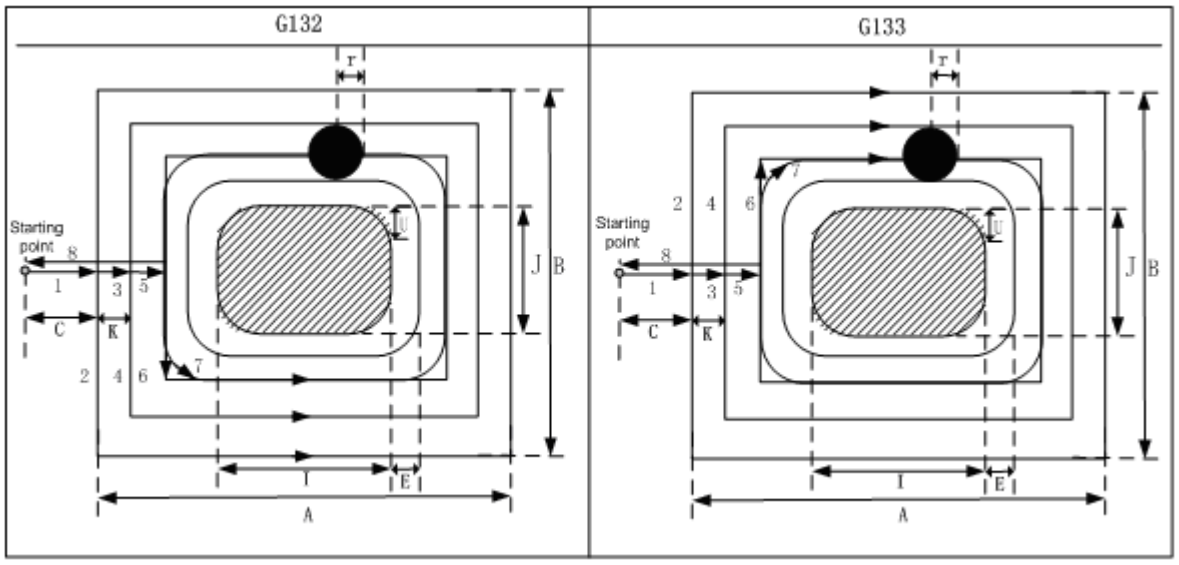
(10) Z-axis rapidly approaches the distance (unmachined distance +Q) .

(11) Execute the above operation step 4~9 till the circle surface (its total cutting depth is Z) is machined completed.

(12) Return to initial point plane or point R plane according to G98 or G99.

(13) Return to XY starting point.

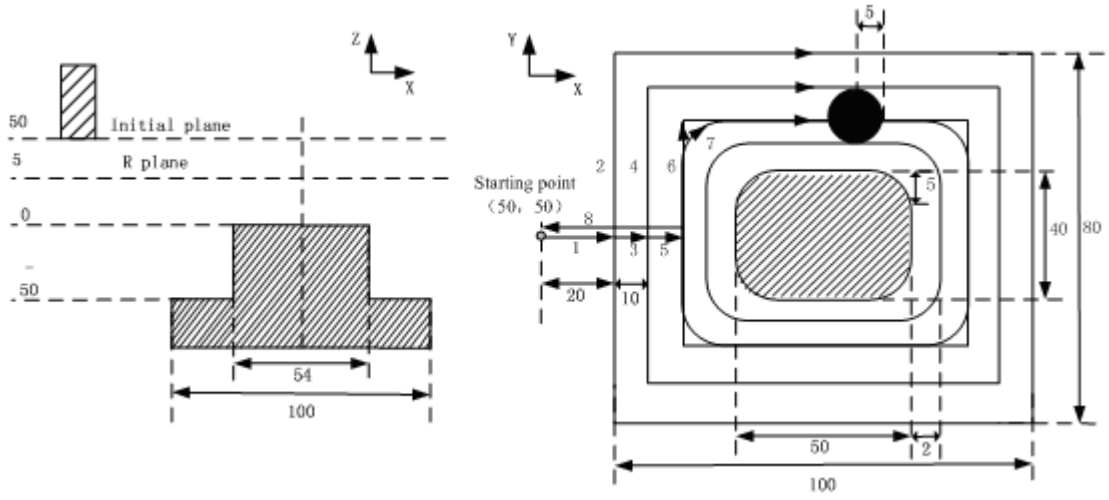




Related explanation:

Commanding P, L in the cycle is invalid, but P value is saved as modal value of the fixed cycle.

Example: G133 commands to rough mill a convex plate of outer rectangle, which is shown below:



G90 G00 X0 Y0 Z50； （G00 rapidly position）

G99 G133 X50 Y50 R5 Z-50 I50 J40 A100 B80 W20 Q10 K10 C20 E2 U5 F800 D1；（execute the rough milling cycle of inner groove cycle D1=5）

G80 X50 Y50 Z50； （cancel fixed cycle, return from point R plane）

M30；

3.25.19 Rectangle Groove Rough-milling G134/G135

Format: G134 G98/G99 X_ Y_ Z_ R_ I_ J_ K_ W_ Q_ V_ U_ D_ F_
G135

Function: From the center of the rectangle, the linear cutting cycle is applied by the specified parameter data, till the rectangle groove with programmed dimension is made out.

Explanation: For command explanation of canned cycle, see the table 3-2.

G134: Rectangle groove rough-milling in CCW.

G135: Rectangle groove rough-milling in CW.

I: The width of rectangle groove along the X axis direction.

J: The width of rectangle groove along the Y axis direction.

K: The cut width increment inside XY plane, it is less than the tool radius, but, more than 0.

W: For the first cutting along the Z axis direction, the distance is downward to the R reference surface, it is more than 0 (if the first cutting is over the position of the bottom of the groove, then the bottom of the groove is taken as the machining position) .

Q: The cutting incremental value each time along Z axis.

V: Distance to the end machining surface, which is more than 0, when the rapid traverse is executed.

U: Corner arc radius, if it is omitted, that is no corner arc transition is not shown.

E: Allowance of rectangle groove rough in XY plane（it should be more than 0, its absolute value is done when it is negative.）.

D: Sequence number of tool radius, its value range is indicated as 0 ~ 32, thereunto, the 0 is default of D0. The current tool radius value is taken out according to the specified sequence number.

Cycle process:

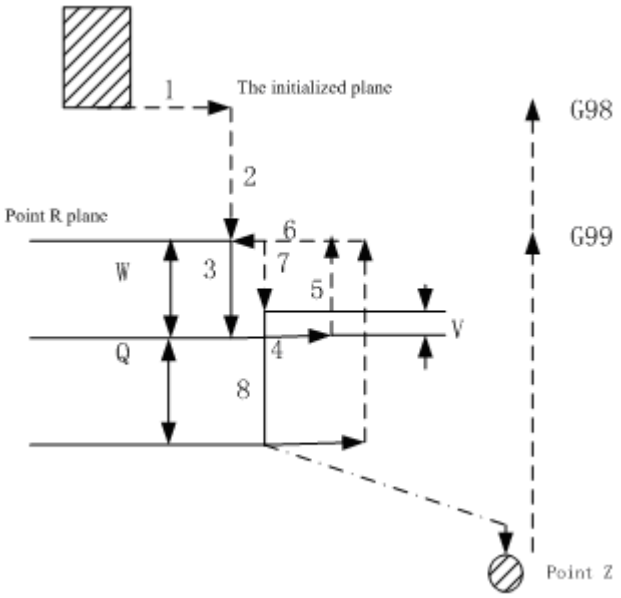
(1) Positioning to the XY plane at the rapid traverse rate.

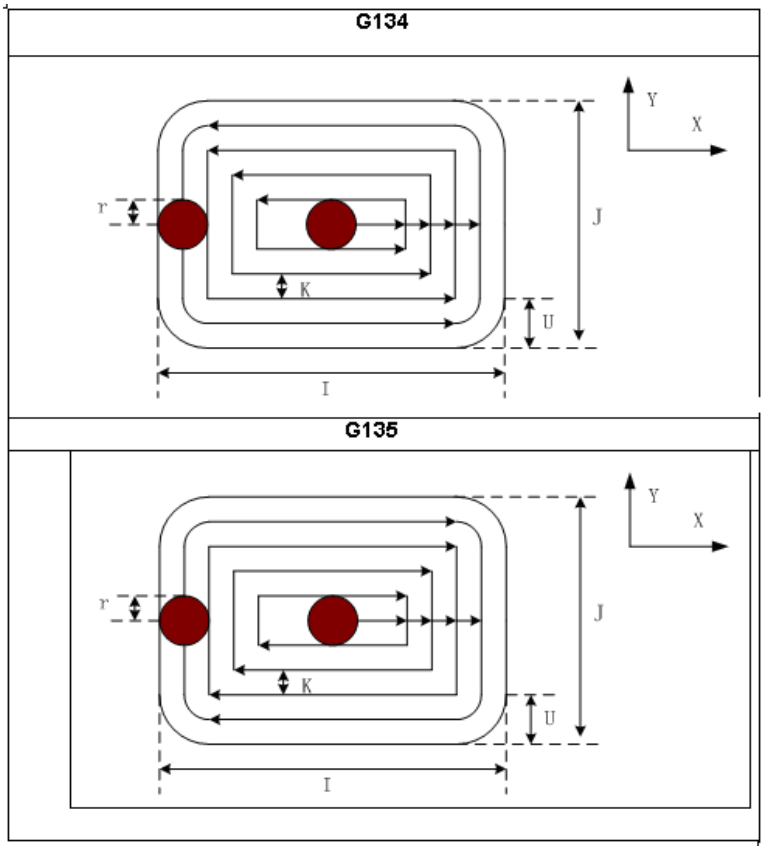
(2) Down to the point R plane at the rapid traverse rate.

(3) W distance depth is cut downwards by cutting feedrate.

(4) Mill a rectangle face helically by K increment each time from center point to outside.
(5) R reference surface is retracted along the Z axis at the rapid traverse rate.
(6) The center of rectangle is positioned along the X and Y axes at the rapid traverse rate.
(7) Down to distance V to the end machining surface along Z axis at the rapid traverse rate;
(8) Cut along Z axis for (Q+V) depth;
(9) Cycling the operation from (4) ~ (8) till the surface of total cutting is performed.
(10) Return to the initial plane or point R plane according to G98 or G99.
(11) Return to XY hole positioning position.

Command Path:

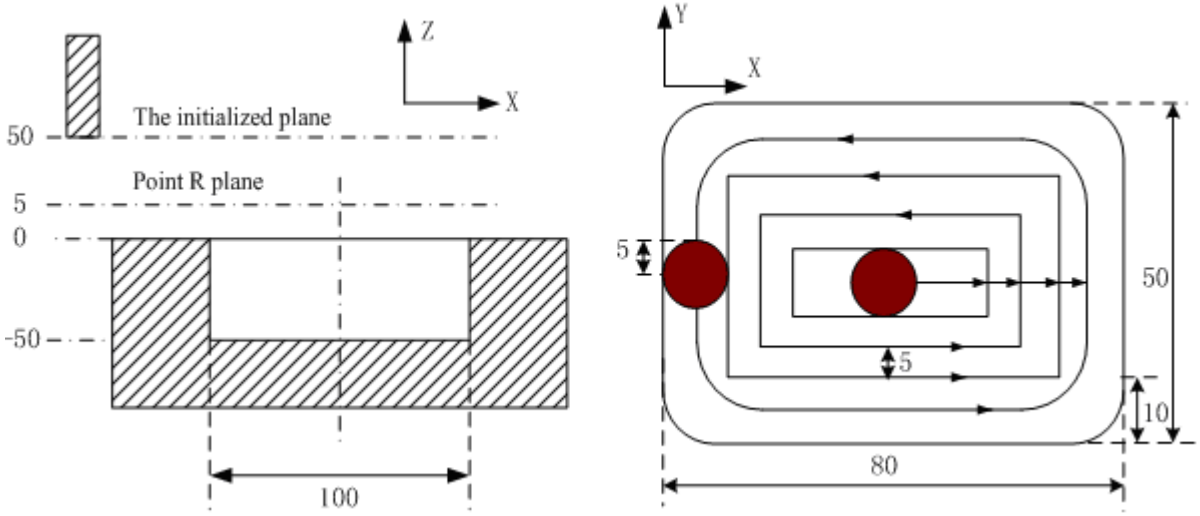




Related Explanation:

The commands P and L are disabled in this cycle, but the P value is reserved as canned cycle modal value.

For example: An inside rectangle groove rough-milling is specified by G134 in canned cycle, see the following figure.



G90 G00 X50 Y50 Z50;(G00 rapid positioning)

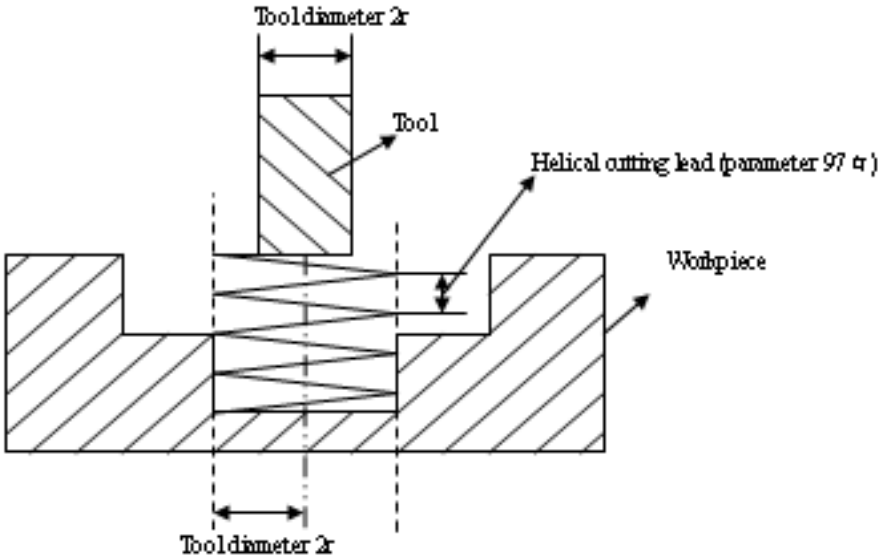
G99 G134 X25 Y25 R5 Z-50 I70 J50 W20 Q10 K5 V10 U10 F800 D1;(Groove rough-milling cycle inside rectangle is performed D1=5)

G80 X50 Y50 Z50;(The canned cycle is cancelled, returning from the point R plane)

M30;

Note: Set the 5122# parameter value to one which is more than 10, by G134 and G135 it feeds helically along Z axis;The value of D must be greater than or equal to minimu increment of 2000 times and less than or equal to 2/3 I; If increment is IS-C,the value of D must be greater than or equal to 0.2 and less than 2/3I.Rough-milling machining can be directly performed for non-groove workpiece.

The helical feeding path is as follows:



3.25.20 Rectangle Groove Inner Fine-milling Cycle G136/G137

Format:

G136

G98/G99 X_ Y_ R_ Z_ I_ J_ D_ K_ U_ F_;

G137

Function: The tool performs fine-milling inside the rectangle with the specified width and direction, it is returned after finishing the fine-milling.

Explanation: For command explanation of canned cycle, see the table 3-2.

G136: Finish-milling cycle inside groove of rectangle in CCW.

G137: Finish-milling cycle inside groove of rectangle in CW.

I: The rectangle width along the X axis, range: -99999999～99999999×least input increment.

J: The rectangle width along the Y axis, range: -99999999～99999999×least input increment.

D: Sequence number of tool radius, the value range is 0~32, the 0 is default value of D0. The current tool radius value is taken out according to the specified sequence number.

K: The distance between the finish-milling start point and the rectangle side in X axis direction, ra

U: Corner arc radius; no corner arc transition if it is omitted. When the U is omitted or it is equal to 0 and the tool radius is more than 0, the alarm is generated.

Cycle process:

(1) Positioning to XY plane at the rapid traverse rate;

(2) Down to point R plane at the rapid traverse rate;

(3) Cutting feed to the bottom of a hole;

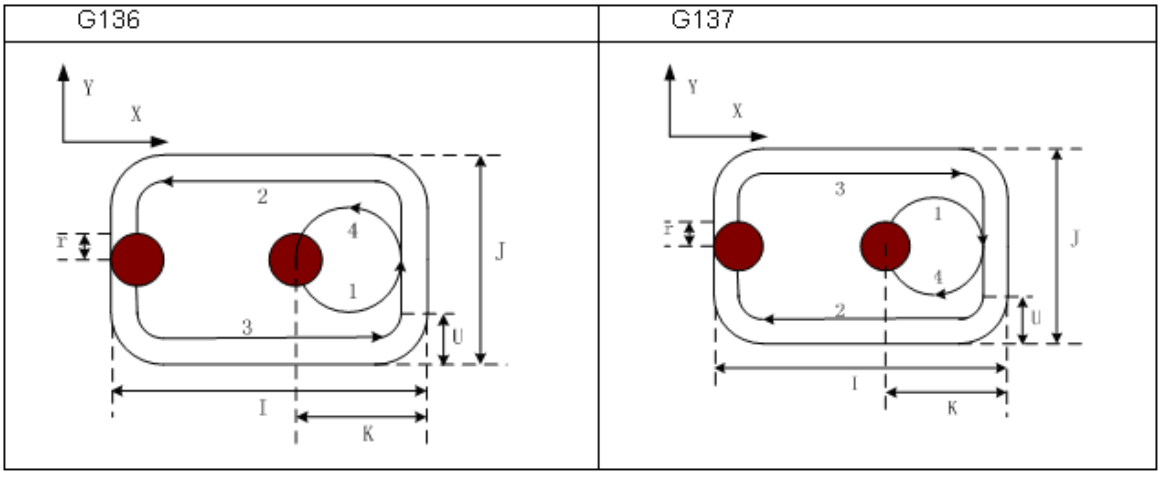
(4) Perform the circle interpolation by the path of transit arc 1;

(5) Perform the circular and linear interpolation by the path of 2-3-4-5-6;

(6) Perform circular interpolation by the path of transit arc 7 and return to the start point;

(7) Returning to the initial plane or point R plane according to G98 or G99.

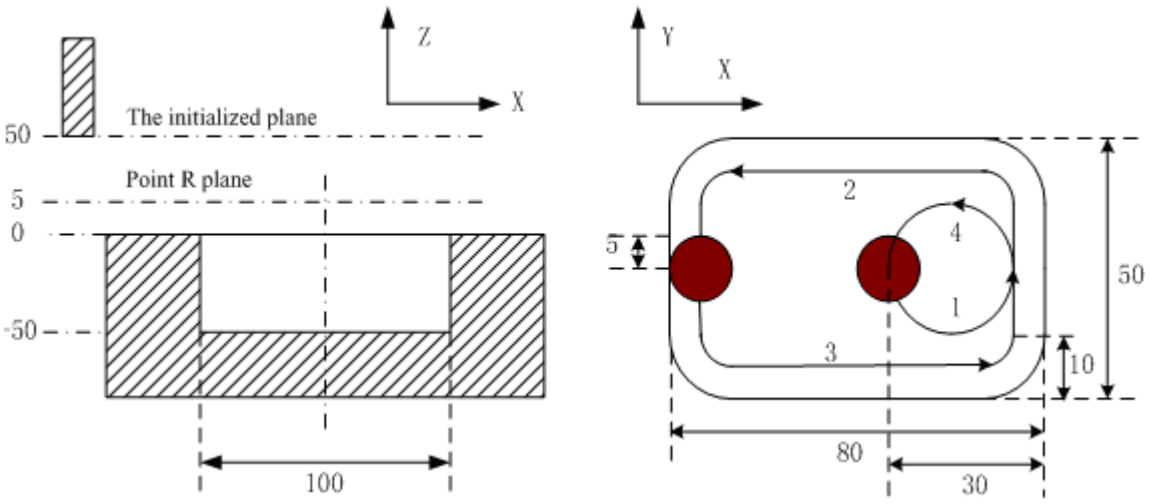
Command Path:



Related Explanation:

The commands Q, P and L are disabled in this cycle, but the Q and P values are reserved as the canned cycle modal value.

For example: To perform a fine-milling for the finished rough-milling rectangle groove with the canned cycle G136 command, see the following figure:



G90 G00 X50 Y50 Z50; (G00 rapid positioning)

G136 X25 Y25 R5 Z-50 I80 J50 K30 U10 F800 D1; (Perform finish-milling inside the rectangle groove at the bottom of a hole in the canned cycle D1=5)

G80 X50 Y50 Z50; (The canned cycle is cancelled, returning from the point R plane)

M30;

3.25.21 Finish-milling Cycle Outside the Rectangle G138/G139

Format:

G138

G98/G99 **X_ Y_ R_ Z_ I_ J_ D_ K_ U_ F_**

G139

Function: The tool performs fine-milling outside the rectangle by the specified width and direction, it is returned after finishing the fine-milling.

Explanation:

G138: Finish-milling cycle outside the rectangle in CCW.

G139: Finish-milling cycle outside the rectangle in CW.

I: The width of rectangle along the X axis, range: -99999999～99999999×least input increment.

J: The width of the rectangle along the Y axis, range:-99999999～99999999×least input increment.

D: Sequence number of tool radius, its value range is indicated as 0 ~ 32, thereinto, the 0 is default of D0. The current tool radius value is taken out according to the specified sequence number.

K: The distance between the finish-milling start point and the side of rectangle along the X axis, range: -99999999～99999999×least input increment.

U: Corner arc radius, if it is omitted, no corner arc transition.

Cycle process:

(1) Positioning to the XY plane at the rapid traverse rate;

(2) Down to the point R plane at the rapid traverse rate;

(3) Cutting feed to the bottom of a hole;

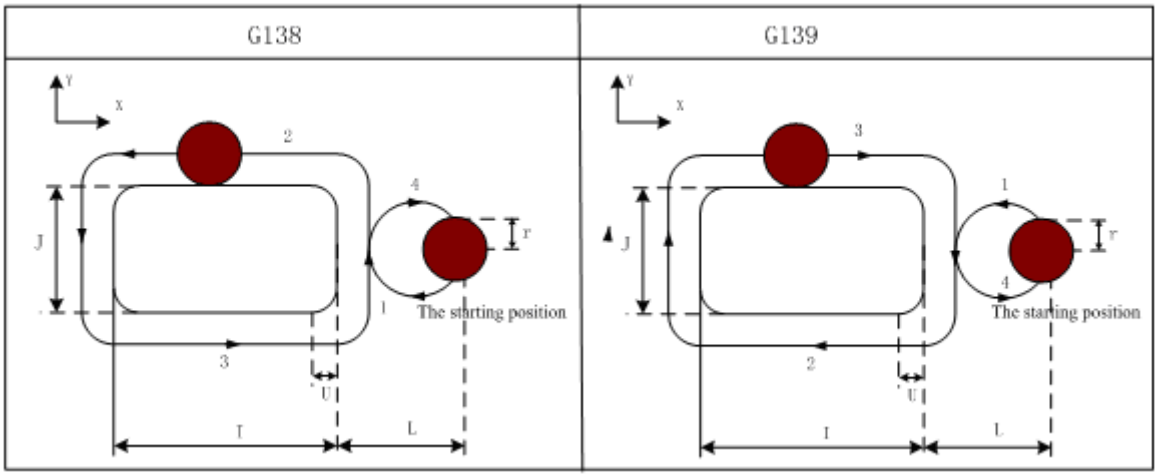
(4) Perform the circle interpolation by the path of transit arc 1;

(5) Perform the circular and linear interpolation by the path of 2-3-4-5-6;

(6) Perform circular interpolation by the path of transit arc 7 and return to the start point;

(7) Returning to the initial plane or point R plane according to G98 or G99.

Command Path:

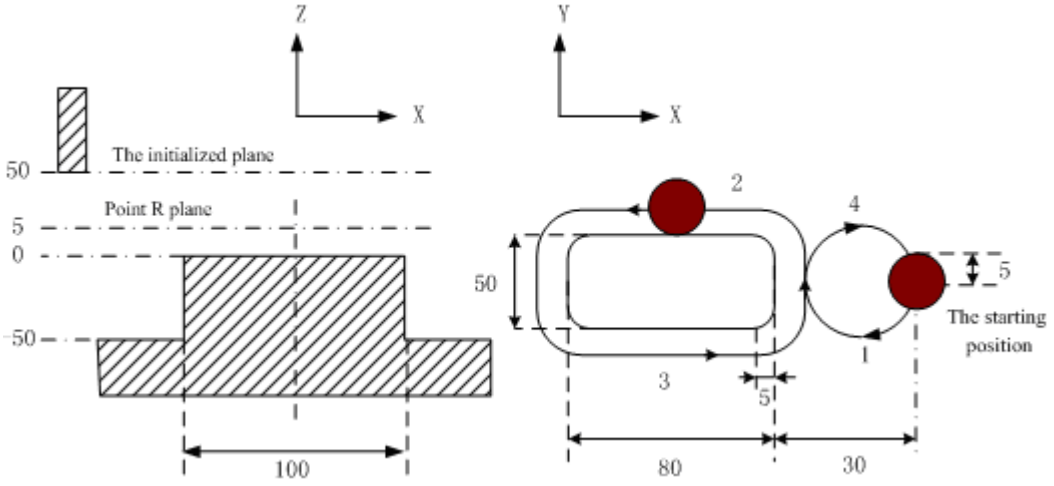


Related Explanation:

(1) The interpolation direction of transition arc is inconsistent to that of the fine-milling arc when a fine-milling is performed outside the rectangle. The interpolation direction is the one for the fine-milling arc in the command explanation.

(2) The commands Q, P and L are disabled in this cycle, but, the value of Q and P are reserved as canned cycle modal value.

For example: A finished rough-milling rectangle groove is performed by the fine-milling by the command G138 in canned cycle. See the following figure.



G90 G00 X50 Y50 Z50; (G00 rapid positioning)

G99 G138 X25 Y25 R5 Z-50 180 J50 K30 U5 F800 D1; (The rectangle outside finish milling is performed under the canned cycle at the bottom of a hole D1=5)

G80 X50 Y50 Z50; (The canned cycle is cancelled, it returns from the point R plane)

M30;

3.26 Continous Drilling

Continuous equal interval drilling cycle is performed in the way that canned cycle is called according to the specified linear, rectangular or arc path.

Parameters related to continuous drilling

0 5 8 2	***	***	***	***	***	LPTK	RPTK	BRCH

LPTK =1：Locating with G01 in line interval drill；

=0：Locating with G00 in line interval drill；

RPTH =1: Locating with G01 in circle and rectangle interval drill；

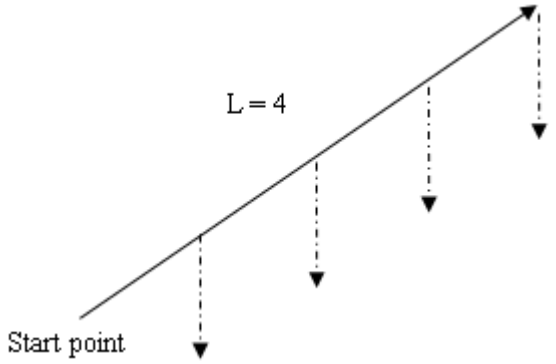
=0：Locating with G00 in circle and rectangle interval drill；

BRCH =1：the return plane when continuous drilling is selected by G98, G99.

=0：the return plane when continuous drilling is selected by G99.

3.26.1 Line Series Punch (L function)

L holes machining cycle should be performed from current plane position to end point specified by X and Y are indicated if the L word is specified in canned cycle, so the current position (block start and end) will not be drilled, the end point position is regarded as the last hole, holes are equal-spaced, as follows:



L value setting	System execution result
Value is negative	Ineffective, the value should be positive
The value is unspecified or equals to 1	Normal drilling cycle 1 time
The value is 0	No change of axes, the system reserves relevant cycle modal data
The value is decimal	When L＞1,using round number When L＜1, it is processed as L=0, not moving but reserving its modal data and relevant cycle parameter values.

Note 1：Input range of command L: -99999999～99999999×least input increment; Decimals is ignored and absolute value is used instead of negative value. L code is effective only in current block.

Note 2：In continuous drilling, the return planes are R point plan. After the last hole is processed, the return plane is specified by G98/G99.

Note 3：When there is no axis position command in the specified L block, it means drilling cycle is performed L times in the original place.

Note 4：Fixed cycle G110,G111,G112,G113,G114,G115,G134,G135,G136,G137,G138,G139 have no consecutive drilling function.

Note 5：When L0 is specified, no drilling will be performed.

Related parameters: status parameter №.582 LPTK

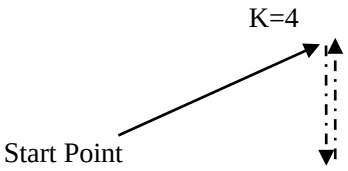
1: hole positioning of linear continuous drilling is executed by cutting path (G01).

0: hole positioning of linear continuous drilling is executed by rapid traverse path (G00).

3.26.2 Line Series Punch (K function)

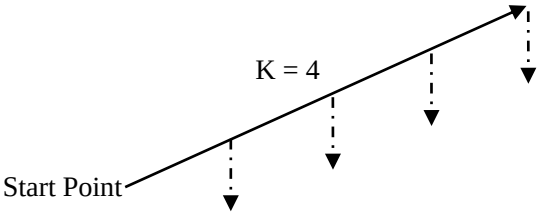
When instructing the K value in the fixed loop punching instruction, if the current mode is G90, perform K repeated punching operations at the current position. If the current mode is G91, perform K incremental punching operations on a straight line, as shown below:

G90：(Repeat drilling K times at this position)

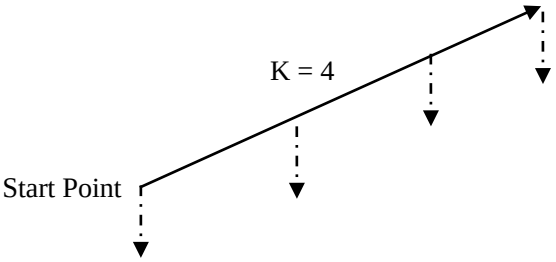


G91：（Make K holes in increments of X and Y）

When STWP=0, continuous drilling of K values



When STWP=1, continuous drilling of K values



K value setting	System execution result
Value is negative	Ineffective, the value should be positive
The value is unspecified or equals to 1	Normal drilling cycle 1 time
The value is 0	No change of axes, the system reserves relevant cycle modal data
The value is decimal	When K＞1,using round number When K＜1, it is processed as L=0, not moving but reserving its modal data and relevant cycle parameter values.

Related parameters: State parameter No. 582.2

1: The hole positioning of linear continuous drilling is based on the cutting trajectory.

0: The hole positioning of linear continuous drilling is carried out with a fast trajectory.

3.26.3 Rectangle Series Punch (G140/G141)

Format:

G140

G98/G99 Gxx X_ Y_ R_ Z_ A_ B_ J_ F_

G141

Function: Performing series punch on each side of the rectangle according to the punch number specified.

Explanation:

G140 – Punching in CW

G141 – Punching in CCW

Gxx – Punching type (G73, G74, G81, G83, G84, G85, G86, G88, G89)

X, Y – End coordinate of the first rectangle side

R – R plane position

Z – Hole depth

A – The punching number on the 1st and 3rd side

B – The punching number on the 2nd and 4th side

J- The length of the 2nd side

F – Cutting feedrate

Related parameter：status parameter №.0582：

RPTK=1: hole positioning of continuous drilling is executed by cutting path (G01).

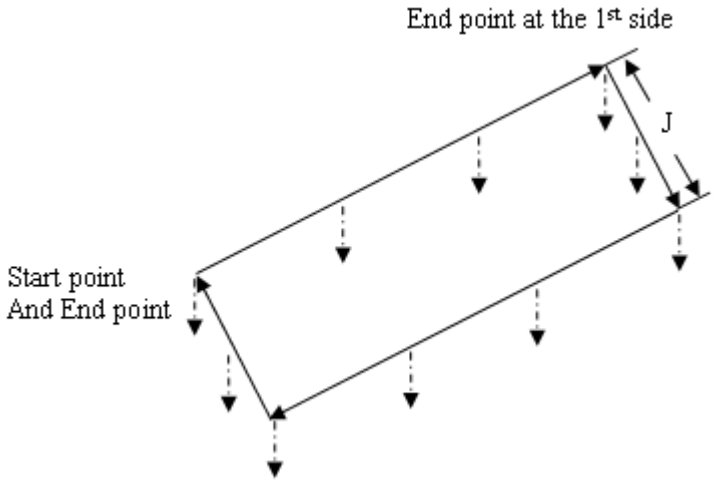
=0: hole positioning of continuous drilling is executed by rapid traverse path (G00).

BRCH =1：return plane of continuous drilling is selected by G98,G99

=0：return plane of continuous drilling is selected by G99

For example:

The end point coordinate of the rectangle first side is X90, Y40; the length of the 2nd side is 20mm as for the rectangle path punching. The punching holes are machined by G81, to punch 3 holes at 1st and 3rd side each other; punch 2 holes at 2nd and 4th side each other, the hole depth is 25mm;



Its programming is as follows:

G90 G17 G0 X0 Y0 Z25；

M03；

G140 G81 X90 Y40 R5 Z-25 A3 B2 J20 F800；

G80 G0 X100 Y100 M05；

M30

There are 10 holes such as A1~A3, B4, B5, A6~A8, B9 and B10 to be machined as in above figure.

Note 1: If the G140 or G141 is specified in the canned cycle, it is indicated that the rectangle serial punching will be performed. The rectangle data are defined according to specified X, Y coordinates and J value in a program, and the serial punching cycle is performed according to the punch mode (canned cycle command).

Note 2: The command value of maximum punching number A and B at each side is 9999; the command is disabled when it is negative. The decimal part will be rounded off if the command is decimal; if the A or B is not specified, then 0 is a default.

Note 3: The rectangle is defined by the current start point, the end of the 1st side and the length of the 2nd side; the default is current start point if the end of 1st side is not specified; the alarm will be generated if the length (namely, the J is not specified) of 2nd side is not specified.

Note 4: The returned levels are all R point plane in serial punching, the corresponding plane will be retracted according to G98/G99 specified in a block when the last hole is performed.

Note 5: Canned cycles, such as G110, G111, G112, G113, G114, G115, G134, G136, G137, G138 and G139 have no serial punching functions.

Note 6: The command words G140, G141, A, B and J are only effective in current block. The alarm will be generated if the G140 and G141 are specified without the canned cycle (punching). The A, B and K will be ignored if A, B and K are specified instead of the G140 or G141.

3.26.4 Arc Serial Punching (G142/G143)

Format:

G142

G98/G99 Gxx X_ Y_ R_ Z_ B_ (I_ J_) C_ F_

G143

Function: Serial punching is performed according to the specified punching number on specified arc.

Explanation:

G142 – Punching in CW

G143 – Punching in CCW

Gxx – Punching type（G73, G74,G81,G82, G83, G84,G85, G86, G88,G89）

X,Y – End point coordinate for the arc, it is fixed for G17 plane.

R – R plane position

Z – Hole depth

B – Radius of arc, when a negative value is specified, it is major arc.

(I_ J_) – The circle center and radius are calculated by I or J when the R value is not specified.

C – Number of punching

F – Cutting feedrate

Related parameter：status parameter №.0582：

RPTK=1: hole positioning of continuous drilling is executed by cutting path (G01).

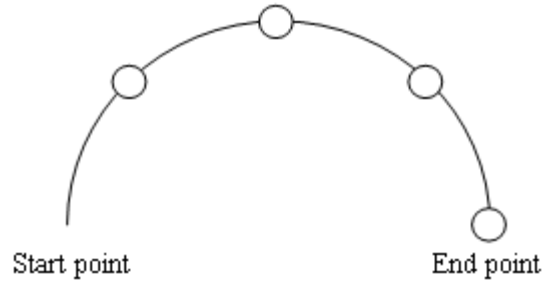
0: hole positioning of continuous drilling is executed by rapid traverse path (G00).

BRCH =1：return plane of continuous drilling is selected by G98,G99

=0：return plane of continuous drilling is selected by G99

For example:

G91 G142 G81 X100 R50 Z-50 C4



Example 2： when drilling 7 holes in full circle, the start points and end points are coordinate origins, and the radius is 50, hole depth is 50.

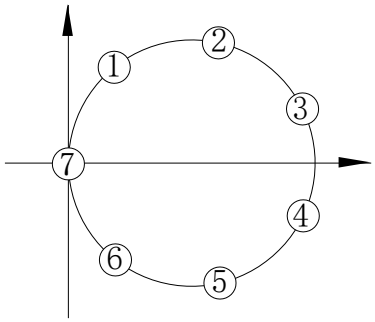
O0001；

G00 G90 X0 Y0 Z0 G17；

G98 G142 G82 I50 J0 R-10 Z-50 C7 F3000；

M30；

%



Note 1:When starting point and end point are the same one in the continuous drilling and I, J are used to program, the full-circle drilling is executed.

Note 2: Canned cycle G110, G111, G112, G113, G114, G115, G134, G135, G136, G137, G138, G139 has no continuous drilling function.

Note 3: The maximum drilling number C is 9999; the negative value is processed as absolute value; the decimals are rounded.

Note 4： When C is not specified or equals to 0, it reaches the end point directly and no drilling will be performed.

3.26.5 Chessboard Continuous Drilling (G144)

Format:

G98/G99 G144 Gxx X_ Y_ I_ J_ A_ B_ F_

Function:

Starting from the starting point (X, Y), drill holes at fixed intervals parallel to the X-axis according to the specified direction, and the machining trajectory is like a grid on a chessboard.

Instruction Description: Gxx - Drilling Method (G73, G74, G76, G81, G82, G83, G84, G85, G86, G87, G88, G89).

X. Y - The starting and ending coordinates of the continuous drilling process on the chessboard.

I - Specify the interval distance between drilling holes in the X-axis direction. If a negative value is specified, it represents the negative direction of the X-axis.

The number of holes machined in the A-X-axis direction.

J - Specify the spacing distance for drilling in the Y-axis direction. If a negative value is specified, it represents the negative direction of the Y-axis.

The number of holes machined in the B-Y-axis direction.

F - Cutting feed rate.

Related parameters: State parameter No. 582:

RPTK=1: The hole positioning of continuous drilling is carried out based on the cutting trajectory (G01).

=0:The positioning of continuous drilling holes is carried out using a fast trajectory (G00).

BRCH=1: The return plane during continuous drilling is selected by G98 and G99.

=0: The return plane for continuous drilling is selected by G99.

BKST=1: Whether to return to the initial XY position after the completion of flat milling and checkerboard hole processing;

=0: After the completion of flat milling and checkerboard hole processing, it does not return to the initial XY position.

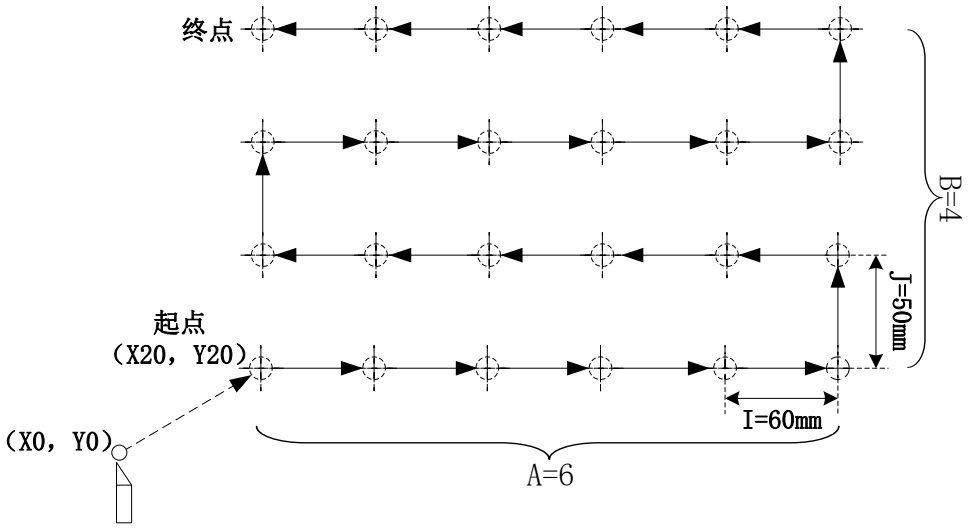
Example: It is necessary to machine 6 * 4 holes in the positive direction of X and Y on a workpiece with a starting position of (X20, Y20) using a point drill. The spacing between the X axis squares should be 60mm, and the spacing in the Y axis direction should be 50mm. The starting plane for processing is -2mm, and the depth is 8mm;

The programming is as follows:

G17 G54 G91 G00 X0 Y0 Z0

G144 G81 X20 Y20 I60 A6 J50 B8 R-2 Z-10

M30



3.27 Absolute and Incremental Commands G90 and G91

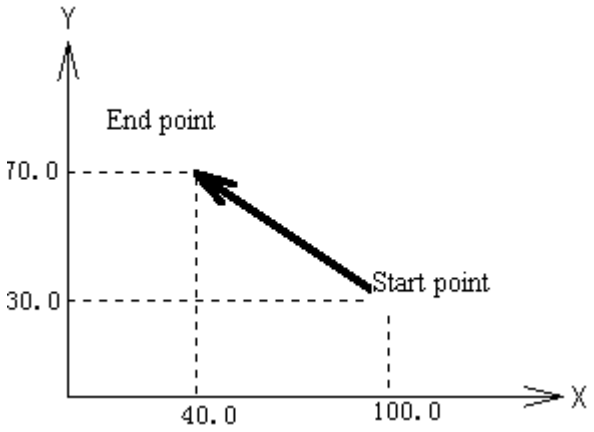
Format:

G90;	Absolute command
G91;	Incremental command

Function:

There are two kinds of modes for commanding axis offset, one is absolute command the other is incremental command. The absolute command is programmed by coordinate value of the terminal position by the axis movement. The incremental command is directly programmed by the movement value of the axis. They are separately specified by G90 and G91 commands.

Example:



The above movement is programmed by absolute and incremental commands, which is as follows:
G90 X40.0 Y70.0 ; or G91 X－60.0 Y40.0;

3.28 Workpiece Coordinate System Setting G92

Function: The workpiece coordinate system is set by setting the absolute coordinate in current position in the system (It is also called floating coordinate system). After the workpiece coordinate is set, the coordinate value is input in absolute programming in this coordinate system till the new workpiece coordinate system is set by G92.

Command explanation: G92, which is a non-modal G-command;

X: The new X axis absolute coordinate of current position;

Y: The new Y axis absolute coordinate of current position;

Z: The new Z axis absolute coordinate of current position;

Note: In G92 command, current coordinate value will be not changed if the X, Y and Z are not input, the program zero is set by the current coordinate value. When the X, Y or Z is not input, the coordinate axis not input keeps on the original set value.

3.29 Feed per min. G94, Feed per rev. G95

Format: G94 Fxxxx; (F0001~ F8000, the leading zero can be omitted, the feedrate per min. is offered, mm/min.)

Function: The cutting feedrate is offered in mm/min unit when the G94 is modal G command. The G94 can be omitted if the current mode is G94.

Format: G95 Fxxxx; (F0.0001～F500, The leading zero can be omitted）

Command Function: The cutting feedrate is offered in mm/rev unit when the G95 is modal G command. The G95 can be omitted if the current mode is G95. The product of F command value (mm/r) and current spindle speed(r/min) is regarded as the command cutting feedrate to control the actual feedrate when the G95 Fxxxx is performed by system. The actual cutting feedrate varies with the spindle speed. The spindle cutting feed value per rev is specified by G95 Fxxxx, it can form even cutting grain on the surface of the workpiece. The machine should be installed spindle encoder when the G95 mode is used.

G94 and G95 are modal G commands in same group, one of them is effective in one time. G94 is initial modal G command, it is defaulted effective when the power is turned on.

The conversion formula for feed value per rev and per min is as following:

$$F_m = F_r \times S$$

F_m: Feed value per min (mm/min);

F_r: Feed value per rev per rev (mm/r);

S: Spindle speed (r/min).

The feedrate value is set by system data parameter No.030 when the power is turned on for the system; an F value is invariable after the F command is performed. The feedrate is 0 after the F0 is executed. The F value is invariable when the system is reset or emergency stop. **The feed override is memorized when the power is turned off.**

Related parameters:

Data parameter N0.027：upper speed of each axis' cutting feed（X, Y, Z）;

Data parameter N0.029：acceleration/deceleration time constant of cutting feed;

Data parameter N0.030：initial speed of cutting feed's acceleration/deceleration;

Data parameter N0.041：initial speed in MPG/Step feed mode;

Data parameter N0.042：acceleration/deceleration time constant in MPG/Step/Manual feed mode;

Note: The cutting feedrate becomes uneven when the spindle speed is less than 1 rev/min in G95 mode; the actual feedrate has following error when the spindle speed fluctuates. In order to guarantee the machining quality, it is recommended that the spindle speed can not be lower than spindle servo or the lowest speed of effective torque introduced by inverter during machining.

3.30 G98, G99

Format:

G98;

G99;

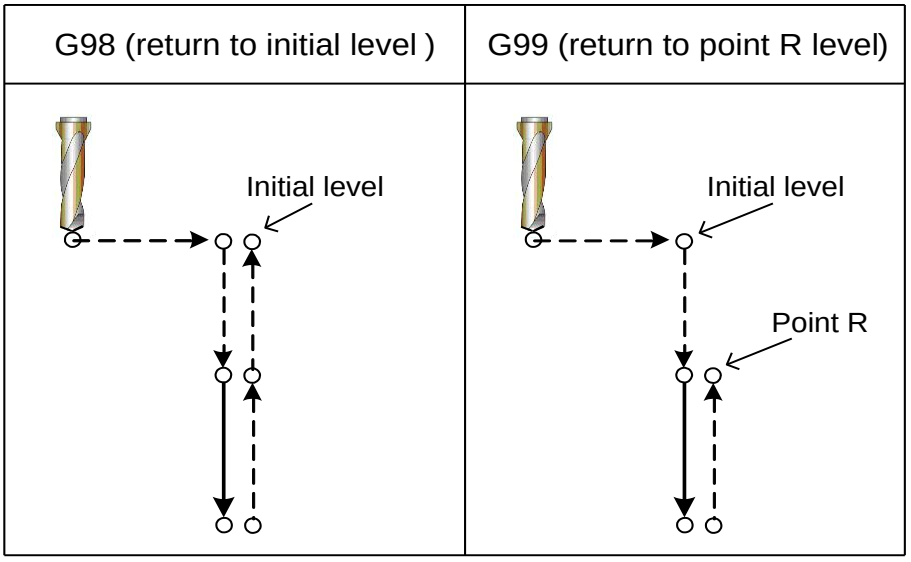
Function:

G98; Tool returns to the initial plane when the hole machining is returning.

G99; Tool returns to the point R plane when the hole machining is returning.

Explanation:

Modal G command

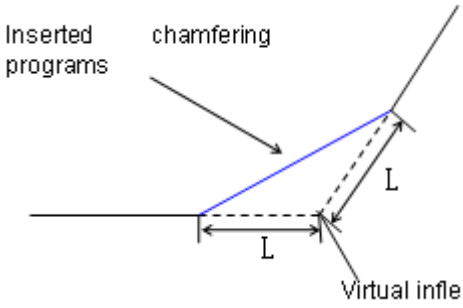


Refer to the explanation for canned cycle command.

3.31 Chamfering Function

C chamfering function is to insert a linearity or an arc between two contours to ensure the tool smoothly transits from a contour to another. A block for chamfering transition can insert the following blocks:

- Blocks for linear interpolation and linear interpolation
- Blocks for linear interpolation and circular interpolation
- Blocks for circular interpolation and linear interpolation
- Blocks for circular interpolation and circular interpolation



Note: virtual inflection is defined to a subsistent inflection when the chamfering function is not executed.

3.31.1 Linear Chamfering

Data following the command L is distance from virtual inflection to starting point or end point of chamfering; the distance in an arc is a chord length corresponded to an arc segment.

Command format ：G_ IP_ L_;（_：G01~G03；IP：axis movement command）
G□ IP_;

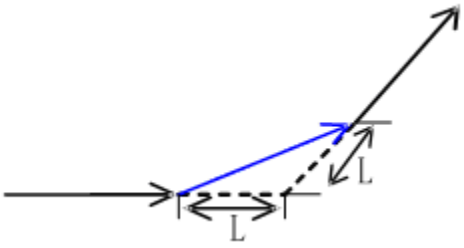


Fig. 3-5 linear chamfering of linear to linear

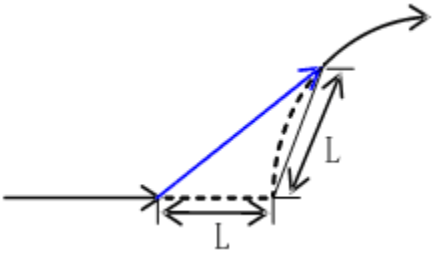


Fig. 3-6 linear chamfering of linear to arc

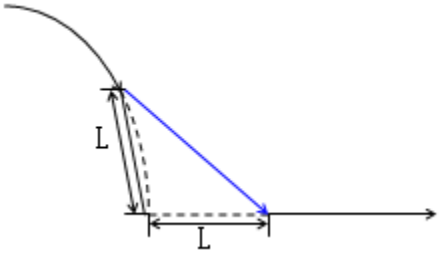


Fig. 3-7 linear chamfering of arc to linear

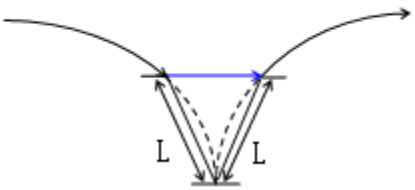


Fig. 3-8 linear chamfering of arc to arc

3.31.2 Arc Chamfering

Data following the command U is a radius of chamfering arc.

Command format：G□ IP_ U_;（□：G01~G03；IP：axis movement command ）
G□ IP_;

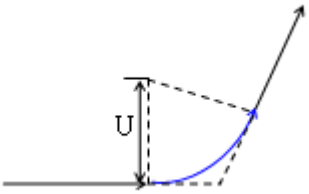


Fig. 3-9 arc chamfering of linear to linear

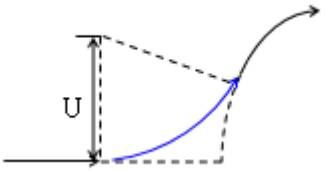


Fig. 3-10 arc chamfering of linear to arc

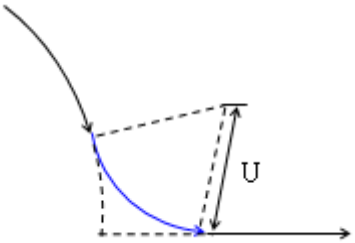


Fig. 3-11 arc chamfering of arc to linear

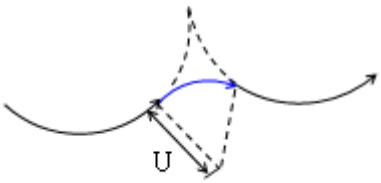


Fig. 3-12 arc chamfering of arc to arc

3.31.3 Limit

Plane selection

Linear chamfering and arc chamfering are executed in the specified planes (G17, G18, G19), and the planes cannot be switched between their chamfering blocks.

Next block

A block with a specified chamfering can follow an interpolation block (G01, G02, G03), and can also follow some non-movement commands (including G04, G90, G91, G94, G95, G98, G99, FMST and non-movement commands which block numbers are N, it can command up to 10 blocks, among which M commands do not include M02, M30, M98, M99, M9000~M9999). P/S alarm occurs when the next block includes other commands except for the above commands.

Excessive motion range

P/S alarm occurs when the inserted chamfering length or arc radius is too big, or exceeds the specified length in a program.

Polar coordinate command

Chamfering function cannot be specified when the system is in polar coordinate command mode.

Others: the blocks following coordinate change G92, G54~G59 or reference point return G28~G30 cannot execute the chamfering.

Note: When a chamfering is specified, an alarm occurs when L and U are specified in the same block.

3.32 Rigid Tapping

The right-handed tapping cycle (G84) and left-handed tapping cycle (G74) may be performed in standard mode or rigid tapping mode. In standard mode, the spindle is rotated and stopped along with a movement along the tapping axis using miscellaneous functions M03 (rotating the spindle clockwise), M04 (rotating the spindle counterclockwise), and M05 (stopping the spindle) to perform tapping.

In rigid mode, tapping is performed by controlling the spindle motor as if it were a servo motor and by interpolating between the tapping axis and spindle. When tapping is performed in rigid mode, the spindle rotates one turn every time a certain feed (thread lead) which takes place along the tapping axis. This operation does not vary even during acceleration or deceleration.

3.32.1 Rigid Tapping

Code format:

Left-handed rigid tapping: G74 X_ Y_ Z_ R_ P_ F（I）_ L_ C_

Right-handed rigid tapping: G84 X_ Y_ Z_ R_ P_ F（I）_ L_ C_

Code function: In rigid mode, tapping is performed by controlling the spindle motor as if it were a servo motor and by interpolating between the tapping axis and spindle. When tapping is performed in rigid mode, the spindle rotates one turn every time a certain feed (thread lead) which takes place along the tapping axis. This operation does not vary even during acceleration or deceleration.

Cycle process: (1) Position to the XY plane at the rapid traverse rate;

(2) Reduce to the point R plane rapidly, then to the position where the C is specified at the rapid traverse rate;

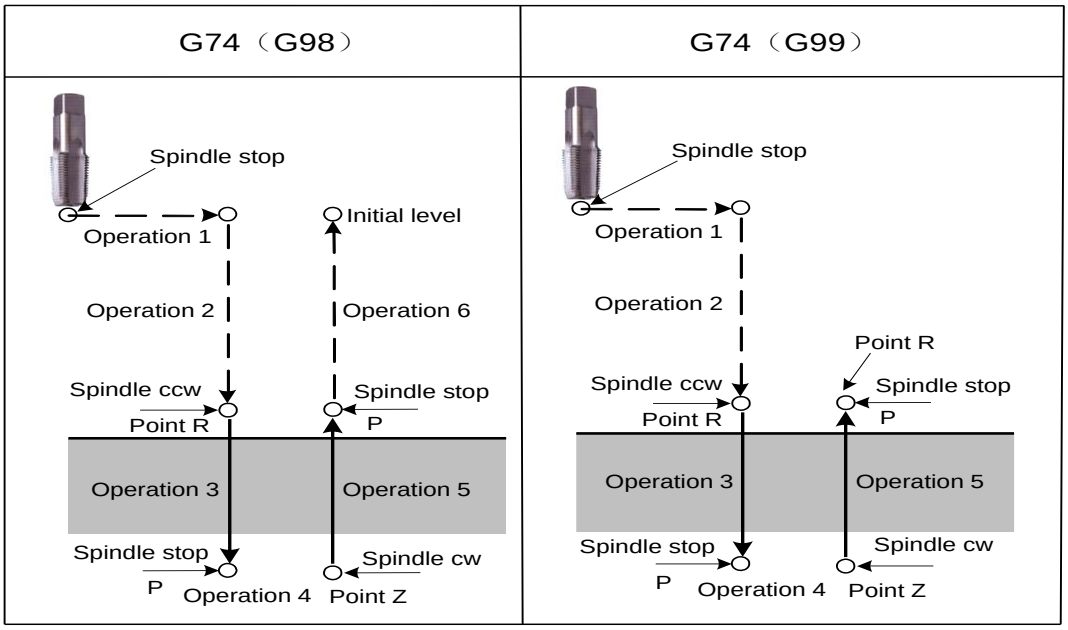
(3) Tapping is performed to the bottom of the hole, then the spindle stops;

(4) Dwell time P is performed if the P is specified;

(5) Spindle rotates reversely returns to the point R plane, the spindle then stops; dwell time P is performed if the P is specified;

(6) Return to the origin plane if the command is G98.

Code path: (G74 shows a sample)



Explanations:

When the tapping operation 3 is being performed, the feedrate override can not be adjusted; when the operation 5 is performing, the speed override value is set by the data parameterNo.5211, when the data parameter No.5211 is set to 0, the override value is fixed as 100%

When the tapping operation 3 is being performed, the linear acceleration or deceleration constant value is set by the data parameter No.5261; when the tapping operation 5 is performed, the linear acceleration constant value is set by data parameter No.5271, if the data parameter No.5271 is se to 0, the linear acceleration/deceleration time constant in operation 5 is set by the data parameter No.5261.

3.32.2 Peck Rigid Tapping

Code format:

(High-speed/standard) peck left-handed rigid tapping: G74 X_ Y_ Z_ R_ P_ F（I）_ L_ Q_ C_

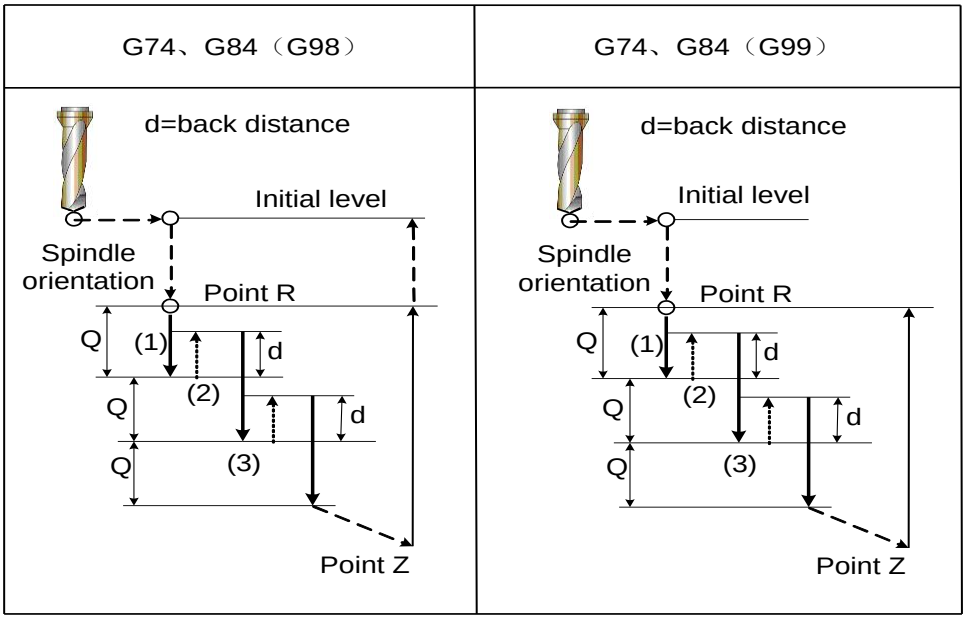
(High-speed/standard) peck right-handed rigid tapping: G84 X_ Y_ Z_ R_ P_ F（I）_ L_ Q_ C_

Code function: When the peck tapping is performed in rigid tapping, due to chips sticking to the tool or increased cutting resistance, in such cases, the preferable tapping can be performed by the peck rigid tapping.

High-speed peck rigid tapping:

When the RTPCP of state parameter No.588 is set to 1, the high-speed peck rigid tapping cycle is selected.

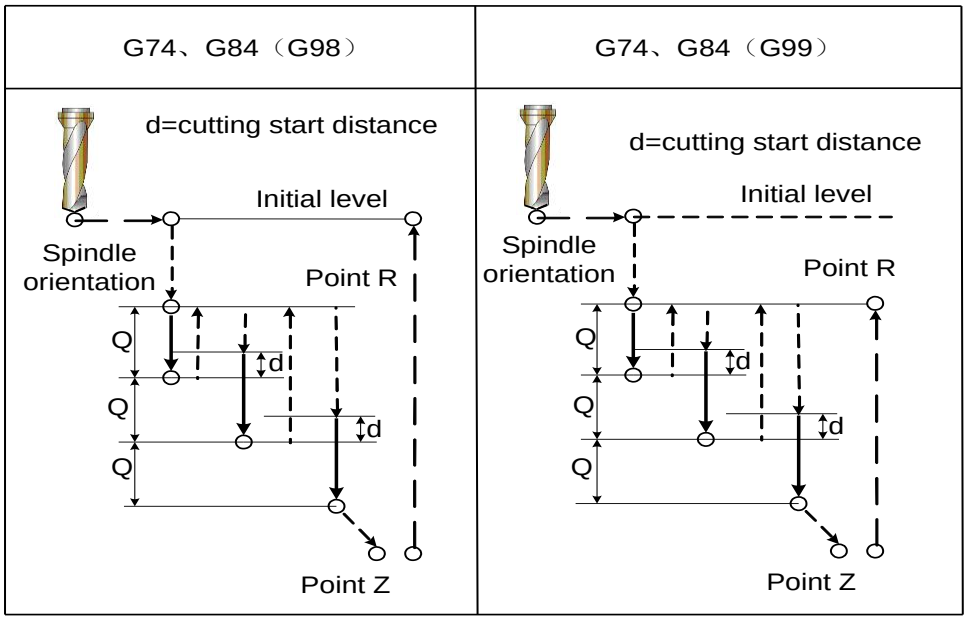
After positioning along the X- and Y-axes, rapid traverse is performed to point R, then position to the place where specifies by C. From point R, cutting is performed with depth Q (depth of cut for each cutting feed), then the tool is retracted by distance d, the retraction speed can be overridden. When point Z has been reached, the spindle is stopped, and then rotated in the reverse direction for retraction. The tool retracts to the point R, the spindle stops. If it is G98 state, rapidly move to the initial position, the Figure is shown below:



Standard peck rigid tapping:

When the RTPCP of state parameter No.588 is set to 1, the standard peck rigid tapping cycle is selected.

After positioning along the X- and Y-axes, rapid traverse is performed to point R, then position to the place where specifies by C. From point R, cutting is performed with depth Q (depth of cut for each cutting feed), then the tool is retracted by distance d, the retraction speed can be overridden. The position is performed from point R to a distance d from the end of the last cutting, which is where cutting is restarted, and the cutting feed is performed. When point Z has been reached, the spindle is stopped, then rotated in the reverse direction for retraction. The tool retracts to the point R, the spindle stops. If it is G98 state, rapidly move to the initial position, the Figure is shown below:



Explanations:

When tapping feed is performing, the speed override can not be adjusted; when the retraction is performed, the speed override value is set by data parameter No.5211, when the data parameter No.5211 is set to 0, the override value is fixed as 100%.

The linear acceleration or deceleration constant value in tapping feed is set by data parameter No.5261, the linear acceleration or deceleration constant in retraction is set by data parameter No.5271, if the No.5271 is set to 0, the acceleration or deceleration constant in retraction is then set by data parameter No.5261. and the retraction distance d is set by data parameter No.5213.

3.32.3 Address Explanation

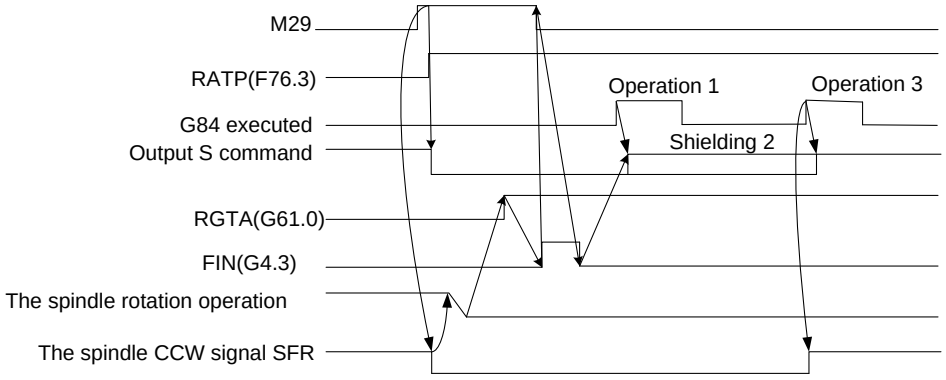
Specified content	Address	Command address explanation
Hole position data	X,Y	Specify the hole position by the absolute value or incremental
Hole machining data	R	From the initial plane to the point distance
	Z	Depth of a hole, the distance from point R to the bottom of the hole
	P	Specify the dwell time at the bottom of the hole or at point R when a return is made. The dwell does not perform when it is not input or the value is 0.
	Q	Tool infeed value of peck tapping
	L	It indicates that the consecutive machining cycle of L holes are performed on this line segment from start (the start position of block) to XY coordinate position. The continued drilling may not perform if it is not input or the value is 0.
	F	Metric thread leading, the solution range: 0.001~500mm. The alarm 201 may alarm if it is not input.
	I	The number of the thread head per/inch, the solution range is 0.06~25400 gear/inch
	C	Start angle

3.32.4 Technic Specification

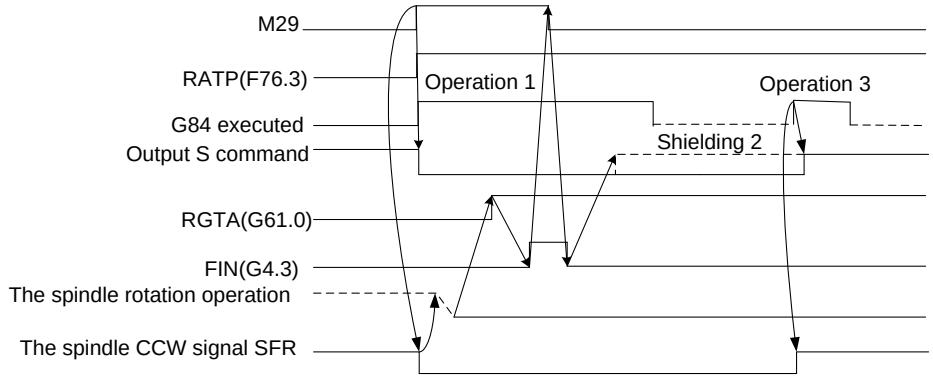
- Acceleration/deceleration
  Rigid tapping adopts the acceleration or deceleration before a straight line to control.
- Override
  The override regulation is invalid for rigid tapping infeed, but the override value can be adjusted or not which is determined by data parameter.
- Dry run
  G84/G74 can be used a dry run, the dry run equals to the feedrate along Z axis. The override adjustment is invalid in dry run.
- Machine lock
  G84/G74 can be used a machine lock, the tapping axis and spindle axis are not moved when the machine lock is enabled.
- Resetting
  The resetting can be reset the tapping when the rigid tapping is performed, but the G74/G84 can be not be reset.
- Dwell
  The dwell is disabled.
- Working
  G84/G74 is only valid in Auto or MDI mode.
- Manual feed
  The rigid tapping can not used for manual feed.
- Tool length compensation
  If the tool length compensation (G43, G44 or G49) is specified in canned cycle, the offset value is added till position to the point R.
- Cutter compensation
  Cutter compensation is ignored in canned cycle.
- Axis switching
  The Z axis tapping can only be performed in rigid mode.
- S code
  If the command speed is more than the maximum speed, the alarm may occur.
- M29
  Specify an axis movement code between M29 and G84/G74 causes alarm.
- P/Q
  If they are specified in non-drilling block (If they are specified in a block that does not perform drilling), they are not stored as modal data. When Q0 is specified, the peck rigid tapping cycle is not performed.
  Specify them in tapping block, they are stored as modal data, when the tapping command is retracted, either Q modal (did it).
- Cancellation
  Do not specify a group 01 G code and G84/G74 in the same block.
- A Cs contour control is used with rigid tapping at the same time.
  CS axis selects a speed mode or position mode which is determined by CON (G27.7), but, the system is rigid tapping mode, regardless of the value of CON. After the rigid tapping is cancelled, the rotation axis is either CS axis or common one which is determined by state parameter. The C axis can not be moved in manual mode when the rigid tapping is not cancelled.

3.32.5 Specify a Rigid Tapping Mode

- Specify M29 before G74/G84
  G84 shows a sample for the following time-sequence



- Specify M29 and G74/G84 at the same block
  G84 shows a sample for the following time-sequence



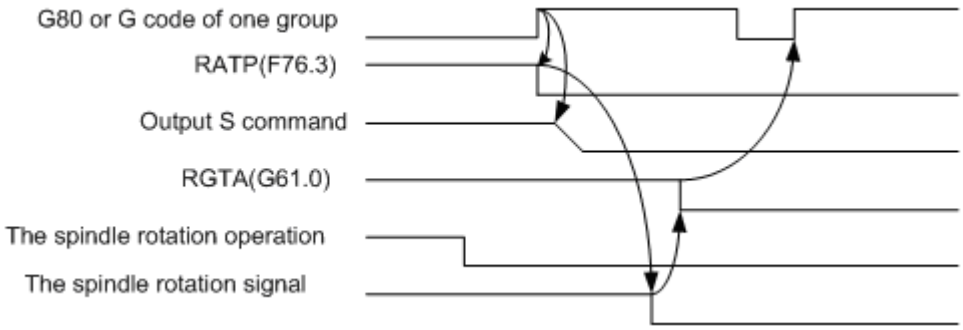
- The explanation of time sequence
  The spindle rotation operation means that the rotation axis is shifted to the position control mode (namely, the servo spindle is needed to send a switch signal in position mode), and check the position mode arrival signal of servo spindle.

3.32.6 The Cancellation of Rigid Tapping Mode

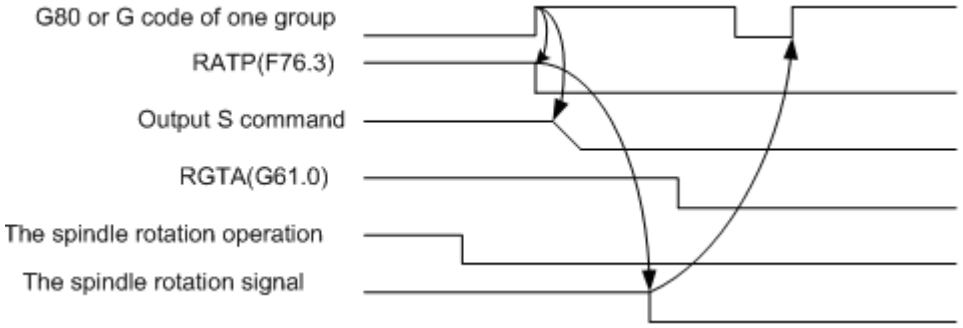
- The rigid tapping mode is canceled by G80
- Specify other canned cycles by G codes
- The other G codes of group 1.
- CNC resetting

The signal descending of F76.3 along the signal with canceling the rigid tapping of PLC, if the state RTCRG of parameter No.588 is equal to 1, the system is then performed the next block without waiting for the rigid tapping mode signal which G61.0 is set to 0;

When the state parameter No.588 (CRG) =0, the time sequence is as follows:



When the state parameter No.588 (CRG) =1, the time sequence is as follows:



3.32.7 F and G Signals

RGTAP (G61.0): Rigid tapping signal

When the M 29 is commanded, PMC enters the rigid tapping mode, and the signal is then set to 1 to inform the CNC

1: PMC enters the rigid tapping mode

0: PMC does not enter the rigid tapping mode

If this signal does not set to 1, after the M29 has been commanded, the alarm may occur in the block of G74/G84.

RGSPM, RGSPP (F65.1, 0) spindle turning signal

When the rigid tapping is performed, the signal is informed to the PMC whether the current spindle is CCW (positive) or CW (negative).

RGSPM: 1 spindle CW (negative) RGSPP: 1 spindle CCW (positive)

In rigid tapping, these signals are output when the spindle is rotated. In the mode of rigid tapping, when the spindle is positioned at the hole or stopped at the bottom of the hole or R position, these signals are not output.

In the mode of rigid tapping, when the spindle is positioned at the inter-locked stop, machine lock or Z axis ignorance states, the spindle does not regard as a stop state, in this case, these signals are output. These signals are only enabled in rigid tapping, and they are all set to 0 in the normal spindle control mode.

RTAP (F76.3): Rigid tapping process signal

This signal informs PMC which has been in the mode of rigid tapping or not. The CNC is in the mode of rigid tapping currently when the signal is set to 1.

This signal can be locked M29, PLC has been commanded the rigid tapping mode, the PMC is then treated with the corresponding logic, and this signal can be replaced the lock of M29, even so, the FIN signal of M29 is not ignored still.

3.32.8 Alarm Message

Alarm No.	Display Content	Explanation
218	Fail to specify the tool pitch F value in G74 or G84.	Fail to specify F value.
230	The spindle feed can not be performed due to the S value is 0.	S value is 0, or S code does not specify.
231	S value exceeds the maximum spindle speed allowed with rigid tapping.	S value exceeds the setting value of data parameter 086.
232	Other axis movement codes are specified between M29 and G74/G84.	Specify a axis movement between M29 and G74/G84.
233	G61.0 signal is abnormal in rigid tapping mode.	Rigid tapping signal G61.0 is not 1 during performing in G74/G84.
234	Specify M29 repeatedly.	Specify M29 or it is consecutively specified more than twice in rigid tapping.

3.32.9 Program Example

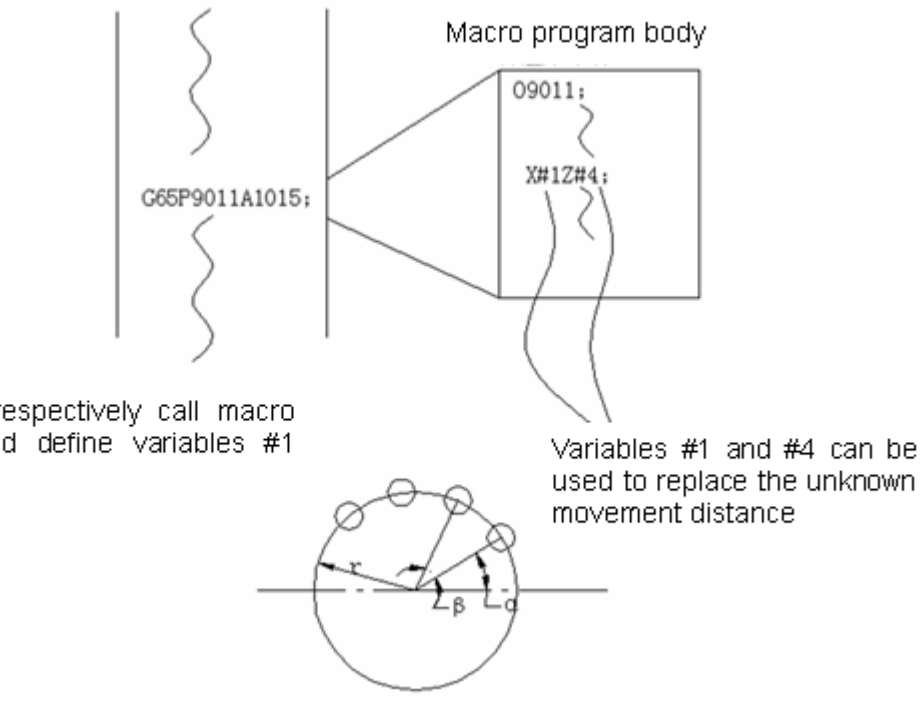
G84 shows an example for the following program

```
O1000 (Rigid tapping example);
G0 X0 Y0 Z0；
M29 S200;
G84 X10 Y10 Z-10 R-5 P2000 F2 C20;
X20 C40
G80;
M30;
```

Chapter 4 Macro Programs

GSK980MDi Plus provides macro programs which is similar to high level language. Variable assignment, arithmetic operation, logical judgment and conditional branch can be realized through custom macro program. It is in favor of the programming for special parts, lessens the complex operation and simplifies the custom program.

Custom macro programs are similar to subprograms. However, macro program allows variable assignment, arithmetic operation, logical judgment and conditional branch, which makes it easier to program the same machining process.



It is easy to machine the screw holes distributed in circles (shown in the figure above).

After a macro program used in circular holes is programmed and edited, it can be performed if the NC system has circular hole machining function.

By the following command, programming personnel can use circular holes function.

G65 P p R r A a B b K k ;

p：Macro program number of circular holes

r：Radius

a：Start angle of the hole

b：Angle of holes intervals

k：Holes number

In this way, users can improve the NC performance on their own. Macro programs can be either provided by machine tool builder or defined by users.

4.1 Macro Call

Macro call (G65, G66) differs from subprogram call (M98) as described below:

1. With G65 or G66, an argument (data passed to a macro) can be specified. M98 does not have this

capability.

2. When an M98 block contains another NC command (for example, G01 X100.0 M98 P), the macro program P_ is called after the command G01 is executed. On the other hand G65 unconditionally calls a macro P_.
3. When an M98 block contains another NC command (for example,G01 X100.0 M98 P_), the machine stops in the single block mode. On the other hand, G65 does not stop the machine.
4. With G65 or G66, the level of local variables changes. With M98, the level of local variables does not change.

- **Non-modal call（G65）**

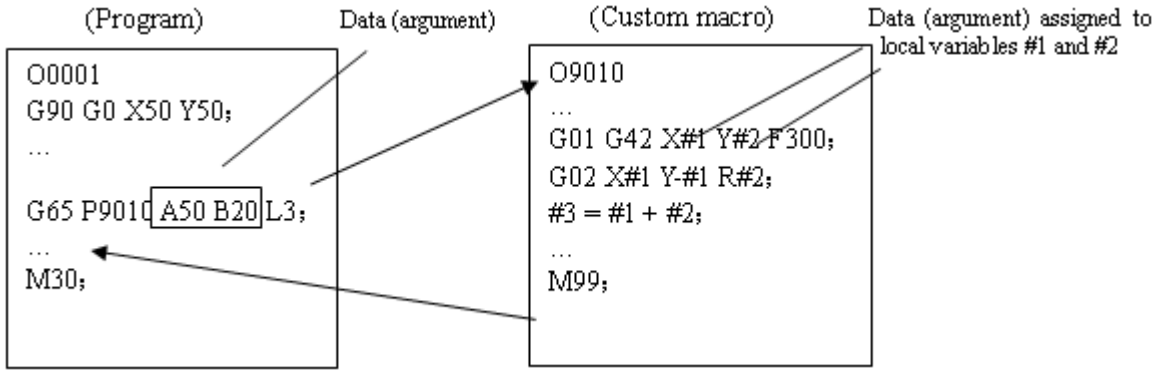
  When G65 is specified, the macro program specified at address P is called. Argument (data) can be passed to the custom macro program.

  Format： G65 P_ L_ <argument>_;

  Explanation： P —— number of the program to be called

  L —— repetition count（1 by default, 1 to 9999 can be specified）

  <Argument> —— Data passed to the macro. Its value is assigned to the corresponding local variables.



Argument specification: two types of argument specification are available.

Argument specification I：it uses letter other than G, L, O, N and P once each. In repeated specification, the last one prevails.

Argument specification I

Address	Variable number
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
H	#11

Address	Variable number
I	#4
J	#5
K	#6
M	#13
Q	#17
R	#18
S	#19

Address	Variable number
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

Note: Addresses that need not to be specified can be omitted. Local variables corresponding to an omitted address are set to null.

Argument specification II：Uses A, B, C and Ii, Ji, Ki (i is 1~10) and automatically decides the argument specification type according to the letters and the sequence. Uses A, B, C once each and uses I, J, and K up to ten times.

Argument specification II

Address	Variable number
A	#1
B	#2
C	#3
I_1	#4
J_1	#5
K_1	#6
I_2	#7
J_2	#8
K_2	#9
I_3	#10
J_3	#11

Address	Variable number
K_3	#12
I_4	#13
J_4	#14
K_4	#15
I_5	#16
J_5	#17
K_5	#18
I_6	#19
J_6	#20
K_6	#21
I_7	#22

Address	Variable number
J_7	#23
K_7	#24
I_8	#25
J_8	#26
K_8	#27
I_9	#28
J_9	#29
K_9	#30
I_{10}	#31
J_{10}	#32
K_{10}	#33

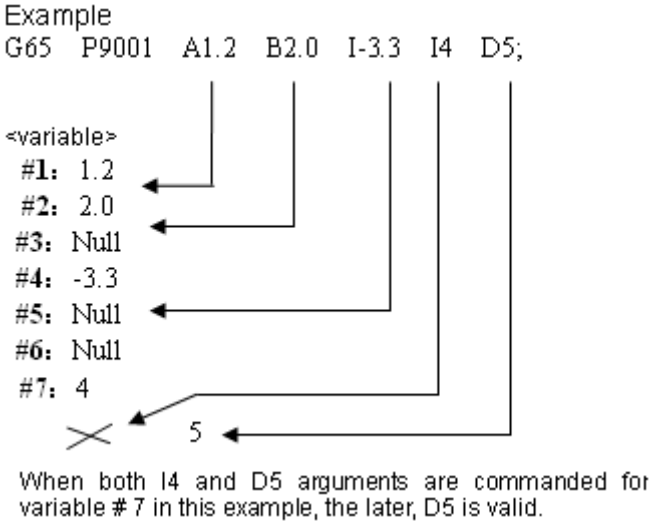
Note 1: Subscripts of I, J and K for indicating the order of argument specification are not written in the actual program.

Note 2: Argument I, J, K do not need to be written in orders. They will be identified according to the present sequence. For example: G65 P9010 A1 B2 C3 I14 J15 I6 J7 K9 K11 K12 J30; The variables are passed as follows:

I14→#4，J15→#5，I6→#7，J7→#8，K9→#6，K11→#9，K12→#12，J30→#11;

Format： G65 must be specified before any argument.

Mixture of argument specifications I and II: The CNC internally identifies argument specification I and II. If a mixture of argument specification I and II is specified, the type of argument specification specified later take precedence.



- **Modal call（G66）**

Once G66 is issued to specify a modal call, a macro is called after a block specifying movement along axes is executed. This continues until G67 is issued to cancel a modal call.

Note: The format, functions and argument specification of G65 are identical with that of the G65 (non-modal call). (Refer to the introduction of G65 for detailed description).

Modal call nesting：Modal calls can be nested by specifying another G66 code during a modal call.

Explanation： 1. In the specified G66 block, only argument is passed, and macro modal call will not be executed.

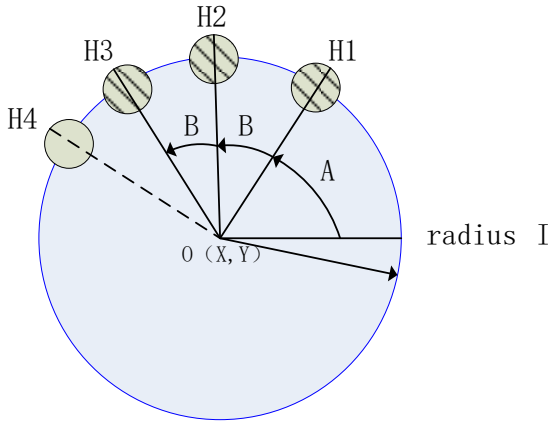
2. Macro modal call can only be executed in the blocks with G00, G01, G02, and G03.

3. No macro program can be called in a block which contains a code such as miscellaneous function that does not involve movement along an axis.

4. G65 and G66 should not be specified at the same time.
5. Multiple macro programs cannot be called in G66 block.
6. As with G65, G66 should be specified prior to arguments and P.

- **Sample program**
  - **G65 call (bolt hole circle)**

Create a macro program for machining holes on a circle. The radius is I; start angle is A; holes interval is B, holes number is H; the center of the circle is (X,Y). Commands can be specified in either the absolute or incremental mode. To drill in the clockwise direction, specify a negative value for B.



Format：G65 P9100 Xx Yy Zz Rr Ii Aa Bb Hh;

X：X coordinate of center point (absolute or incremental) (#24)

Y：Y coordinate of center point (absolute or incremental) (#25)

Z：Hole depth（#26）

R：Coordinates of an rapid approaching point（#18）

F：Cutting feedrate（#9）

I：Circle radius（#4）

A：Drilling start angle（#1）

B：Incremental angle (clockwise when negative value is specified). (#2)

H：Number of holes （#11）

Macro call ：O0002

G90 G00 X0 Y0 Z100;
G65 P9100 X100 Y50 R30 Z-50 F500 I100 A45 B30 H5;
M30;

Macro program（the called program）：O9100

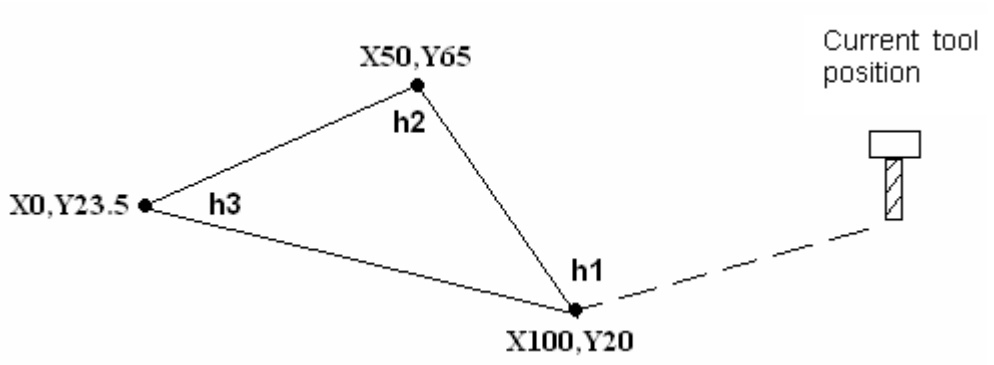
```
#3=#4003 … …….. Stores G codes of 03 group
IF[#3 EQ 90]GOTO 1; … ……Branches to N1 in the G90 mode
#24=#5001+#24; … ……Calculates the X coordinate of the center point
#25=#5002+#25; … ……... Calculates the Y coordinate of the center point
N1 WHILE [#11 GT 0] DO 1; … ……Until the number of remaining holes reaches 0
#5=#24+#4*COS[#1];……Calculates.the.hole.position.onX.axis
#6=#25+#4*SIN[#1]; … ……..Calculates the hole position on X axis
G90 G81 X#5 Y#6 Z#26 R#18 F#9; …Drilling after moving to the target position
#1=#1+#2; … ……Updates the angles
```

#11=#11-1; … ……Decrements the number of holes
END 1;
G#3 G80; … ……Returns the G codes to the original state.
M99;

Argument meanings： #3 store G codes of 03 group
#5 X coordinate of the next hole to drill
#6 Y coordinate of the next hole to drill

- **G66 modal call**

Shown as follows: machine 3 holes (h1,h2,h3)



Call format： G66 P9201 Aa Bb Cc; （the argument in this example is assumed）

Macro program： O0001

G90 G17 G00 X0 Y0 Z0；
G00 X150 Y20； ----------position
G66 P9201 A-10 B-40 C2000；-----pass the argument, be ready for machining
G00 X100 Y20；----------position to h1, call macro program (hole machining)
G00 X50 Y65；----------position to h1, call macro program (hole machining)
M09； ----------non-movement code, does not call macro program
G00 X0 Y23.5；---------position to h1, call macro program (hole machining)
G67；----------cancel macro program modal call
G00 X150 Y20；----------positioning return
M30;

Called macro program： O9201（machining process）
G81 G98 R#1 Z#2 F#3；
M99；

4.2 Variables

An ordinary machining program specifies a G code and the travel distance directly with a numeric value, for example, G01 and X100.0. With a custom macro program, numerical value can be specified directly or using variables, for example, G#101 X#102. When variables are used, the variable value can be changed by programs or using operation on the MDI panel.

- **Representation and using methods of variables**

  Differ from argument (data), variables are considered as the carrier of data, for example, #1, #101 …are variables; A100, B200 …are arguments. Data of arguments A100, B200 should be transferred to variable #1 and #2. When using or programming macro programs, numerical value can

be specified directly (such as G01, X100) or using variables (such as G#01, X#07). When variables are used, the variable value can be changed by programs or using operation on the panel.

The address value of a macro body can be specified by variables. The variable value can be set by the main program or be assigned the calculated value when executing the macro body. Multiple variables can be identified by numbers.

(1) Variable representation

A number sign # followed by a variable number is shown as follows:

#i (i = 1, 2, 3, 4 ……). For example: #5, #109, #1005

(2) Omission of decimal point

When a variable value is defined in a program, the decimal point can be omitted. For example: when defining #1=123, the actual value of variable #1 is 123.000.

(3) Referencing variables

To reference the value of a variable in a program, specify a word address followed by the variable number. A program with an expression <address>#i or <address>-#i indicates that the variable value or negative value is used as address value.

For example: Z-#110…when #110 = 250, it is equals to Z-250.

G#130…when #130 = 3, it is equals to G3

(4) Replace variable numbers with variables

When replace variable numbers with variables, #9100 rather than ##100 is used, the 9 followed # means the replacement. For example: when #100 = 105, #105 = 500,

X#9100 and X500 are equal. i.e. X#9100 → X##100，X#105 → X500

X-#9100 and X-500 are equal.

Note: Program number o, sequence number N and optional block skip number '/' cannot be followed with variables. For example: O#1, /#2, N#3 .

- **Types of variables**

Variables are classified into four types by variable number:

Variable number	Type of variable	Function	Range	Remark
#0	Null variable	This variable is always null. No value can be assigned to this variable.	NULL	
#1~#33	Local variable	Local variable can only be used within a macro to hold data such as the results of operations. When the power is turned off, local variables are initialized to null. When a macro is called, arguments are assigned to local variables.		
#100~#199	Public variable	Public variables can be shared among different macro programs.	When the power is turned off, variables are initialized to null.	read/ write/ display
#500~#999			When the power is turned off, data is stored	
#1000~#1015	System variable （234）	G54, G55 output	0,1 processed by PLC	Read only
#1032		Store G54, G55, read all 16 bits of a signal at one time		
#1100~#1115		G54, G55 input		Read/write
#1132		Store G54, G55,write all 16 bits of a signal at one time		
#1133		Store G56~G59, write all 32 bits of a signal at one time		

Variable number	Type of variable	Function	Range	Remark
#2001~#2032	System variable	Tool length compensation wear	-9999.999~9999.999	Read/write
#2201~#2232		Tool length compensation	-9999.999~9999.999	Read/write
#2401~#2432		Cutter compensation wear	-9999.999~9999.999	Read/write
#2601~#2632		Cutter compensation wear	-9999.999~9999.999	Read/write
#3003~#3004		Automatic operation control—#3003	0，1，2，3	Read/Write
		Automatic operation control—#3004	0~7	
#3901		The number of machined parts	0~99999999	Read/write
#4001		G00, G01, G02, G03, G73, G74, G80, G81, G82, G83, G84, G85, G86, G88, G89, G110, G111, G112, G113, G114, G115, G134, G135, G136, G137, G138, G139	modal G code group1	Read only
#4002~#4003		G17, G18, G19—#4002	modal G code group 2	Read only
		G90, G91—#4003	modal G code group 3	Read only
#4005~#4007		G94, G95—#4005	modal G code group 5	Read only
		G20, G21—#4006	modal G code group 6	Read only
		G40, G41, G42—#4007	modal G code group 7	Read only
#4008		G43, G44, G49	modal G code group 8	Read only
#4010		G98, G99	modal G code group 10	Read only
#4014		G54~G59	modal G code group 14	Read only
#4107		D code	0~32	Read only
#4109		F code	0~15000	Read only
#4111		H code	0~32	Read only
#4113~#4115		M code—#4113	0~99	Read only
		Sequence number—#4114	0~99999	Read only
		Program number —#4115	0~9999	Read only
#4119~#4120		S code—#4119	0~9999	Read only
		T code—#4120	0~32	Read only
#5001~5005	System variable	1~5 axes; block end point; workpiece coordinate system; tool compensation value not included	-9999.999~9999.999	Read only
#5021~5025		1~5 axes; current position; machine coordinate system; tool compensation value included	-9999.999~9999.999	Read only
#5041~5045		1~5 axes, the current position, workpiece coordinate system contain tool compensation value	-9999.999~9999.999	Read only
#5061~5065		1~5 axes, skip signal position; workpiece coordinate system; tool compensation value included	-9999.999~9999.999	Read only
#5081~5085		1~5 axes; tool length compensation value; current execution value.	-9999.999~9999.999	Read only
#5201~5205		1~5 axes; external workpiece zero point offset value	-9999.999~9999.999	Read/write
#5221~5225		1~5 axes, G54 workpiece zero point offset value	-9999.999~9999.999	Read/write
#5241~5245		1~5 axes, G55 workpiece zero point offset value	-9999.999~9999.999	Read/write
#5261~5265		1~5 axes, G56 workpiece zero point offset value	-9999.999~9999.999	Read/write
#5281~5285		1~5 axes, G57 workpiece zero point offset value	-9999.999~9999.999	Read/write
#5301~5305		1~5 axes, G58 workpiece zero point offset value	-9999.999~9999.999	Read/write
#5321~5325		1~5 axes, G59 workpiece zero point offset value	-9999.999~9999.999	Read/write

4.2.1 Null Variables

When the variable value is undefined, the variable is null. Variable #0 is always null, and can be read only.

a, referencing

The address itself is ignored when an undefined variable (null variable) is quoted.

When #1=< Null>,	**When #1=0**
G90 X100 Y#1 equals to G90 X100	G90 X100 Y#1 equals to G90 X100 Y0

b, Arithmetic operation

<Null> equals to 0 in any case except when assigned by <Null>.

When #1=< Null >	**When #1=0**
#2=#1 （assignment） The arithmetic operation result #2 equals to< Null>	#2=#1 The arithmetic operation result #2 equals to 0
#2=#1 * 5 The arithmetic operation result #2 equals to 0	#2=#1 * 5 The arithmetic operation result #2 equals to 0
#2=#1+#1 The arithmetic operation result #2 equals to 0	#2=#1+#1 The arithmetic operation result #2 equals to 0

c. Conditional expression

<Null> differs from 0 only for EQ and NE.

When #1= Null	**When #1=0**
#1 EQ #0 ↓ True	#1 EQ #0 ↓ False
#1 NE #0 ↓ False	#1 NE #0 ↓ False
#1 GE #0 ↓ False	#1 GE #0 ↓ False
#1 GT #0 ↓ False	#1 GT #0 ↓ False

4.2.2 Local Variables

Local variables are the variables internally defined in a program. They are effective only within the program, i.e., it is only can be used within the program.

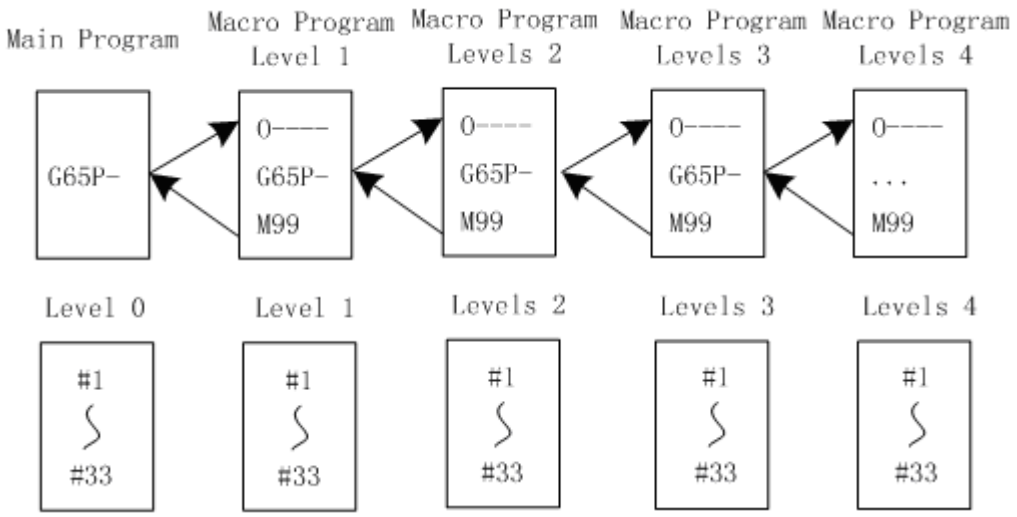
A local variable #1 that calls macro programs at a certain moment is different from the #1 at another moment. (No matter the macro programs are identical or not). Therefore, when macro program B is called from macro program A, like nesting, the local variables used in macro A will not be misused in macro B, and will not disable the value in macro B.

Usually, the local variables are used to accept the value passed from argument. Please refer to" Argument Specification" for the relationship between arguments and addresses. Pay attention that, the initial state of local variable is Null, before the local variable is defined (assigned).

- **Custom macro program nesting and local variable**

  When calling a macro program, its nesting level increases by one, and correspondingly, the level of local variable increases by one as well.

The relationship between macro program call and local variable is shown as follows:



- **Explanations**

1. #1～#33 local variables (0 level) are provided in the main program.
2. When a macro program (1 level) is called by G65, the local variable (0 level) is stored, and local variables #1~#33 of the new macro program is prepared.
3. Each time a macro program (2, 3, 4 levels) are called, local variables (1, 2,3 levels) in each group are stored, and new local variables (2,3,4, levels) are prepared.

4.2.3 System Variables

System variables are used to read and write CNC internal data, such as tool length compensation value, tool nose radius compensation value. Some system variables can only be read. System variables are the basis of automatic control and general-purpose machining program development.

- **Interface signal** The macro variable corresponding to interface signal is the exchange signal between PLC and custom macro program.

Variable No.	Function
#1000~#1015	A 16-bit signal can be sent from the PLC to a custom macro. Used to read signal bit by bit.
#1032	A 16-bit signal can be sent from the PLC to a custom macro. Used to read al 16 bits of a signal at one time.
#1100~#1115	A 16-bit signal can be sent from the PLC to a custom macro. Used to read and write signal bit by bit.
#1132	A 16-bit signal can be sent from the PLC to a custom macro. Used to read and write all 16 bits of a signal at one time.
#1133	A 32-bit signal can be sent from the PLC to a custom macro. Used to read all 32 bits of a signal at one time.

Note1: Please refer to the *GSK980TD PLC User Manual* for the relationships between variables and F, G signals.

- **Tool compensation value** tool compensation value can be read and written

Compensation No.	Tool length compensation		Cutter compensation	
	Geometric（H）	Wear （H）	Geometric（D）	Wear （D）
01	#2201	#2001	#2601	#2401
02	#2202	#2002	#2602	#2402
03	#2203	#2003	#2603	#2403
…….				
31	#2231	#2031	#2631	#2431
32	#2232	#2032	#2632	#2432

- **Automatic operation control** The control state of automatic operation can be changed

Variable No.	Variable value	Single block	Completion of an auxiliary function
#3003	0	Enabled	To be awaited
	1	Disabled	To be awaited
	2	Enabled	Not to be awaited
	3	Disabled	Not to be awaited

Note 2: When the power is turned on, the value of this variable is 0.

Note 3: When single block stop is enabled (G46.1 is 1), the state of #3003 can change the execution of single block stop.

Note 4: When single block stop is disabled (G46.1 is 0), single block stop operation is not performed even if the single block switch is set to ON.

Note 5: When a wait for the completion of auxiliary function (M, S and T functions) is not specified, program execution proceeds to the next block before completion of auxiliary functions. Also distribution completion signal DEN is not output.

Variable No.	Variable value	Feed hold	Feedrate override	Exact stop
#3004	0	Enabled	Enabled	Enabled
	1	Disabled	Enabled	Enabled
	2	Enabled	Disabled	Enabled
	3	Disabled	Disabled	Enabled
	4	Enabled	Enabled	Disabled
	5	Disabled	Enabled	Disabled
	6	Enabled	Disabled	Disabled
	7	Disabled	Disabled	Disabled

Note 6: When the power is turned on, the value of this variable is 0.

Note 7: When feed hold is disabled, if the feed hold button is held down, the machine stops in the single block stop mode. However, single block stop operation is not performed when the single block mode is disabled with variable #3003.

Note8: When the feed hold is disabled, if the feed hold button is pressed then released, the machine does not stop; program execution continues and the machine stops at the first block where feed hold is enabled; the feed hold lamp is ON.

Note 9: When feedrate override is disabled, an override of 100% is always applied regardless of the setting of the feedrate override.

Note 10: When exact stop check is disabled, no exact stop check is made even in blocks including those which do not perform cutting.

- **Number of machined parts** The number of machined parts can be read and written.

Variable No.	Function
#3901	Number of machined parts

- **Modal information**

Modal information specified in blocks up to the immediately preceding block can be read.

Variable No.	Function
#4001	Group 1 （G00, G01, G02, G03, G73, G74, G80, G81, G82, G83, G84, G85, G86, G88, G89, G110, G111, G112, G113, G114, G115, G134, G135, G136, G137, G138, G139）
#4002	Group 2（G17, G18, G19）
#4003	Group 3（G90, G91）
#4005	Group 5（G94, G95）
#4006	Group 6（G20, G21）
#4007	Group 7（G40, G41, G42）
#4008	Group 8（G43, G44, G49）
#4010	Group 10（G98, G99）
#4014	Group 14（G54, G55, G56, G57, G58, G59）
#4107	D code
#4109	F code
#4111	H code
#4113	M code
#4114	Block sequence number
#4115	Program name
#4119	S code
#4120	T code

- **Current position** Position information can be read.

Variable No.	Function	Read during movement
#5001~#5005	Workpiece coordinate system block end point (tool compensation value not included)	Enabled
#5021~#5025	Machine coordinate system current position(tool compensation value included)	Disabled
#5041~#5045	Workpiece coordinate system current position (tool compensation value included)	Disabled
#5061~#5065	Workpiece coordinate system skip signal position (tool compensation value included)	Enabled
#5081~#5085	Tool length compensation value	Disabled

Note 11: The first digit (from 1 to 5) represents an axis number.

Note 12: The tool length compensation value currently used for execution rather than the immediately preceding tool compensation value is held in variables #5081~#5085.

- **Workpiece coordinate system compensation value**

Workpiece coordinate system compensation value can be read and written.

Variable No.	Function
#5201~#5205	The first to the fifth axes external workpiece zero point offset value
#5221~#5225	The first to the fifth axes G54 workpiece zero point offset value
#5241~#5245	The first to the fifth axes G55 workpiece zero point offset value
#5261~#5265	The first to the fifth axes G56 workpiece zero point offset value
#5281~#5285	The first to the fifth axes G57 workpiece zero point offset value
#5301~#5305	The first to the fifth axes G58 workpiece zero point offset value
#5321~#5325	The first to the fifth axes G59 workpiece zero point offset value

4.3 Arithmetic and Logic Operation

- ***Macro programs in both traditional G65 H format and statement format are compatible with GSK980MDi Plus.Users can alternatively select one of them for programming. This makes programming more convenient and flexible.***
- ***Please strictly observe the formats and specifications in the following "Arithmetic and Logic Operation" table.***

Arithmetic and Logic Operation

Function	Statement format	Traditional G65H format	Remark
Definition, assignment	#i = #j	G65 H1 P#i Q#j	
Sum Subtraction Multiplication Division	#i = #j + #k #i = #j - #k #i = #j * #k #i = #j / #k	G65 H2 P#i Q#j R#k G65 H3 P#i Q#j R#k G65 H4 P#i Q#j R#k G65 H5 P#i Q#j R#k	Logic operation is performed on binary numbers bit by bit.
OR AND XOR	#i = #j OR #k #i = #j AND #k #i = #j XOR #k	G65 H11 P#i Q#j R#k G65 H12 P#i Q#j R#k G65 H13 P#i Q#j R#k	
Square root Absolute value Rounding off Rounding up Rounding down Nature logarithm Exponential function	#I = SQRT ［#j］ #I = ABS ［#j］ #I = ROUND ［#j］ #I = FUP ［#j］ #I = FIX ［#j］ #I = LN ［#j］ #I = EXP ［#j］	G65 H21 P#i Q#j G65 H22 P#i Q#j G65 H23 P#i Q#j G65 H24P#i Q#j G65 H25 P#i Q#j G65 H26 P#i Q#j G65 H27 P#i Q#j	
Sine Arcsine Cosine Arccosine Tangent Arctangent	#i = SIN ［#j］ #i = ASIN ［#j］/［#k］ #i = COS ［#j］ #i = ACOS ［#j］ #i =TAN ［#j］ #i = ATAN［#j］/［#k］	G65 H31 P#i Q#j G65 H32 P#i Q#j G65 H33 P#i Q#j G65 H34 P#i Q#j G65 H35 P#i Q#j G65 H36 P#i Q#j R#k	An angle is specified in degree. 90 degrees and 30 minutes is represented as 90.5 degree.
Conversion from BCD to BIN Conversion from BIN to BCD	#i = BIN ［#j］ #i = BCD ［#j］	G65 H41 P#i Q#j G65 H42 P#i Q#j	Used for the signal exchange to and from PLC.

Unconditional branch
Equals to branch
Not equals to branch
Greater than branch
	GOTO #i
	IF (#i EQ #j) GOTO #k
	IF (#i NE #j) GOTO #k
	IF (#i GT #j) GOTO #k
		G65 H80 P#i Q#j R#k
		G65 H81 P#i Q#j R#k
		G65 H82 P#i Q#j R#k
		G65 H83 P#i Q#j R#k
			Please note that #K is the skip signal in macro statement and P#i is the skip signal in traditional G65H

Function	Statement format	Traditional G65H format	Remark

Smaller than branch
Greater than or equals to branch
Smaller than or equals to branch
	IF (#i LT #j) GOTO #k
	IF (#i GE #j) GOTO #k
	IF (#i LE #j) GOTO #k
		G65 H84 P#i Q#j R#k
		G65 H85 P#i Q#j R#k
		G65 H86 P#i Q#j R#k
			format.

User alarm	None	G65 H99 P#i	0≤P≤100

4.3.1 Traditional Format

If traditional G65 H format is used for programming, only limited operations and jump command can be specified by it. The currently used H operation needs at most 3 operands, so the corresponding operation can be completed when the needed variables (or constants) are obtained in a block.

- **General format**

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

m: 01～99 means operation command or jump command function

#i: the name of variable that stored the operation result

#j: operand 1; it can be constant.

#k: operand 2; it can be constant.

Meaning: #i = #j ○ #k

└———— Operational sign, designated by Hm

(Example) G65 Hm P#100 Q#101 R#102……#100 = #101 ○ #102 ;

G65 Hm R#100 P#101 Q15 ……#101 = 15 ○ #100 ;

G65 Hm Q#100 R-100 P#102……#102 = #100 ○ -100 ;

Note 1: G65 H should be commanded prior to operation or jump command.

Note 2: when P code is commanded in G65 block, G65 P means macro program call. H means argument. No operation or jump command is performed.

Note 3: At most 4 decimal numbers of the constant decimal part can be obtained for rounding. 3 digit numbers can be displayed in the window.

- **Code function explanation**

(1) Variable value assignment, #I ＝ #J

G65 H01 P#I Q#J;

(example) G65 H01 P#101 Q125; (#101 ＝ 125)

G65 H01 P#101 Q#110; (#101 ＝ #110)

G65 H01 P#101 Q-#102; (#101 ＝ -#102)

(2) Addition operation #I ＝ #J ＋ #K

G65 H02 P#I Q#J R#K;

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

G65 H02 P#101 Q#110 R#102; (#101 ＝ #110 ＋ #102)

(3) Subtraction 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) 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) 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)

Note 4: The divisor #k cannot be 0, otherwise an alarm occurs.

(6) OR operation #I = #J OR #K

G65 H11 P#I Q#J R#K;

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

(7) AND operation #I = #J AND #K

G65 H12 P#I Q#J R#K;

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

(8) XOR operation #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) Square root $\#I = \sqrt{\#J}$

G65 H21 P#I Q#J;

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

Note 5: the radicand #J cannot be negative, otherwise, an alarm occurs.

(10) Absolute value #I = |#J|

G65 H22 P#I Q#J;

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

(11) Rounding off #I = ROUND[#J]（ROUND off the first decimal）

G65 H23 P#I Q#J;

(example) G65 H23 P#101 Q1.2359; (#101 = 1.2359 #101=1)

(12) Rounding up #I = FUP[#J]

G65 H24 P#I Q#J;

(13) Rounding down #I = FIX [#J]

G65 H25 P#I Q#J;

With CNC, when the absolute value of the integer produced by an operation on a number is greater than the absolute value of the original number, such an operation is referred to as rounding up to an integer. Conversely, when the absolute value of the integer produced by an operation on a number is less than the absolute value of the original number, such an operation is referred to as rounding down to an integer. Be particular careful when handling negative numbers.

(Example) suppose that #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

(14) Natural logarithm #I = LN [#J]

G65 H26 P#I Q#J;

(example) G65 H26 P#101 Q#102;（#101 = LN[#102]）

Note 6: when the antilogarithm #j is zero or smaller, otherwise, an alarm is issued.

(15) Exponential function #I = EXP[#J]

G65 H27 P#I Q#J;

(example) G65 H27 P#101 Q#102; （#101 = EXP [#102]）

(16) Sine #I = SIN[#J] (unit: deg)

G65 H31 P#I Q#J;

(example) G65 H31 P#101 Q#103; (#101=SIN[#103])

(17) Arcsine #I = ASIN[#J]

G65 H32 P#I Q#J;

(example) G65 H32 P#101 Q#103; (#101=ASIN[#103])

Note 7: When the NAT bit of parameter No.520 is set to 0, the output range is 270° ~ 90°
When the NAT bit of parameter No.520 is set to 1, the output range is -90° ~ 90°

Note 8: Arcsine operand J cannot exceed the range -1~1, otherwise, an alarm is issued.

(18) Arccosine #I = COS[#J] (unit：deg)

G65 H33 P#I Q#J;

(example) G65 H33 P#101 Q#103; (#101=COS [#103])

(19) Arccosine #I = ACOS[#J]

G65 H34 P#I Q#J;

(example) G65 H34 P#101 Q#103; (#101=ACOS [#103])

Note 9: Arccosine operand J cannot exceed the range -1~1, otherwise, an alarm is issued.

(20) Tangent #I = TAN[#J] (deg)

G65 H35 P#I Q#J;

(example) G65 H35 P#101 Q#103; (#101=TAN [#103])

Note 10: #J cannot be equal to Kπ+π/2（K=0, ±1, ±2, ±3 …）, otherwise the result is wrong.

(21) Arctangent #I = ATAN [#J] / [#K] (unit：deg)

G65 H36 P#I Q#J R#K;

(example) G65 H36 P#101 Q#103 R3; (#101=ATAN [#103] /[3])

Note 11: When the NAT bit of parameter No.520is set to 0, the output range is 0° ~ 360°
When the NAT bit of parameter No.520 is set to 1, the output range is -180° ~ 180°

(22) Conversion from BCD to BIN #I = BIN[#J]

G65 H41 P#I Q#J;

(example) G65 H41 P#101 Q#102; (#101 = BIN[#102])

(23) Conversion from BIN to BCD #I = BCD[#J]

G65 H42 P#I Q#J;

(example) G65 H42 P#101 Q#102; (#101 = BCD[#102])

(24) Unconditional branch

G65 H80 Pn; Pn: sequence number

(example) G65 H80 P120; (Go to N120 block)

(25) Equal to conditional branch

G65 H81 Q#I R#J Pn; Pn: sequence number, can be variable

(example) G65 H81 Q#101 R#102 P1000;

When #101 equals to #102, branch to N1000 block; or execut in order.

(26) Not equal to conditional branch

G65 H82 Q#I R#J Pn; Pn: sequence number, can be variable

(example) G65 H82 #101 #102 C1000;

When #101 does not equal to #102, branch to N1000 block; or execute in order.

(27) Greater than conditional branch

G65 H83 Q#I R#J Pn; Pn: sequence number, variable

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

When #101 is greater than #102, branch to N1000 block; when #101≤#102, execute in order.

(28) Smaller than conditional branch

G65 H84 Q#I R#J Pn; Pn: sequence number, variable

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

When #101 is smaller than #102, branch to N1000 block, or execute in order.

(29) Greater than or equals to conditional branch

G65 H85 Q#I R#J Pn; Pn: sequence number, variable

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

When #101 is greater than or equals to #102, branch to N1000 block, or execute in order.

(30) Smaller than or equals to conditional branch

G65 H86 Q#I R#J Pn; Pn: sequence number, variable

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

When #101 is smaller than or equals to #102, branch to N1000 block, or execute in order.

(31) P/S alarm issued

G65 H99 Pn; Pn: sequence number, variable（alarm No.=n +600）

(example) G65 H99 P15;

P/S custom alarm 615 is issued.

4.3.2 Macro Statement

The operations listed in "Arithmetic and Logic Operation" table can be executed in program. The expressions right to the operator contain constants and (or) variables that consisting of functions and operators. The variables #j and #k in the expression can be assigned as constants. The left variable (the first variable) can be assigned by expression. The macro statement is more intuitive, convienent and flexible. It can perform compound operation and multinesting. Sometimes, a macro statement is equal to several tranditional G65H macro programs.

- **General format**

  Please refer the statement format in the "Arithmetic and Logic Operation" table for editing macro statement.

- **Macro program editing**

  In program editing mode or MID mode, by pressing 

key, macro editing state can be switched or inserted.

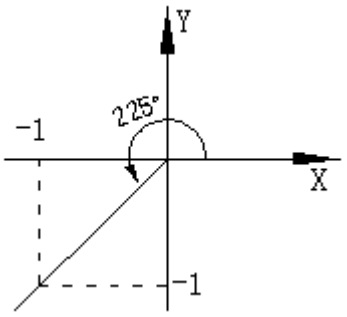
Differences of two states	Automatic space	Processing of letter O	Input of special signs
Insert state	When editing, spaces are automatically added to identify the words.	Press O to switch, copy, delete programs	Special signs cannot be input
Macro editing state	space are not automatically added	Input as a letter "O"	Special signs can be input

- **Explanations**

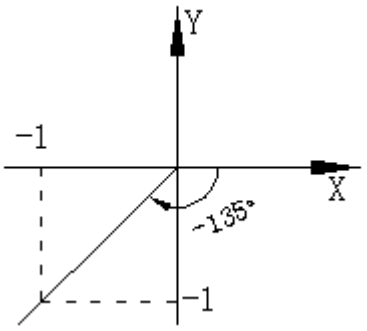
  1. Angular unit

     The angular units of function SIN, COS, ASIN, ACOS, TAN and ATAN are degree. For example, 90°30′means 90.5 degree.

2. ARCSIN # i=ASIN[#j]
   i. the solution ranges are as indicated below
      when the NAT bit of parameter No.520 is set to 0: 270°~ 90°
      when the NAT bit of parameter No.520 is set to 1: -90°~ 90°
   ii. when the #j is beyond the range of -1 to 1, P/S alarm is issued.
   iii. a constant can be used instead of the #j variable.
3. ARCCOS # i =ACOS[#j]
   i. the solution ranges from 180°~ 0°
   ii. when the #j is beyond the range of -1 to 1, P/S alarm is issued.
   iii. a constant can be used instead of the #j variable.
4. ARCTAN #i=ATAN[#j]/[#k]
   i. Specify the lengths of two sides and separate them by a slash /.
      The solution ranges are as follows:
      When the NAT bit of parameter No.520 is set to 0: 0°~ 360°
      [Example] when #1=ATAN[-1]/[-1] is specified, #1=225°



      When the NAT bit of parameter No.520 is set to 1: -180°~ 180°
      [Example] when #1=ATAN[-1]/[-1] is specified, #1=-135°



   ii. A constant can be used instead of the # j variable.
5. Natural logarithm #i=LN[#j]
   i. Note that the relative error may be greater than $10^{-8.}$
   ii. When the antilogarithm #j is zero or smaller, P/S alarm is issued.
   iii . A constant can be used instead of the #j variable.
6. Exponential function #i=EXP[#j]
   i. Note that the relative error may be greater than 10^{-8} .
   ii. When the result of the operation exceeds 3.65×10^{47} (j is about 110), an overflow occurs and P/S alarm is issued.
   iii. A constant can be used instead of the # j variable.

7. ROUND function

When the ROUND function is included in an arithmetic or logic operation command, IF statement, or WHILE statement, the ROUND function rounds off at the first decimal place.

Example:

When #1=ROUND[#2] is executed where #2=1.2345 the value of variable #1 is 1.0.

When the ROUND function is used in NC statement address, the ROUND function rounds off the specified value according to the least input increment of the address.

8. Rounding up and down to an integer

With CNC, when the absolute value of the integer produced by an operation on a number is greater than the absolute value of the original number, such an operation is referred to as rounding up to an integer. Conversely, when the absolute value of the integer produced by an operation on a number is less than the absolute value of the original number, such an operation is referred to as rounding down to an integer. Be particular careful when handling negative numbers.

Example:

Suppose that #1=1.2, #2= -1.2

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

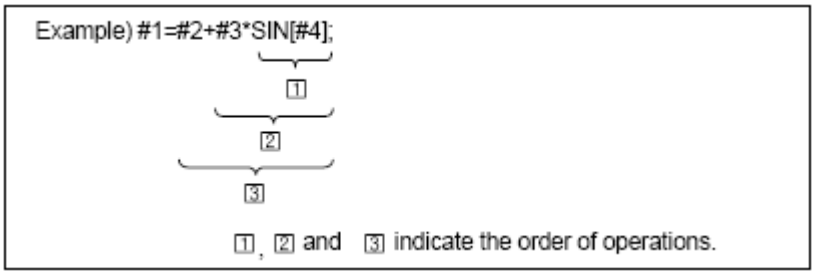
When #3=FIX[#1] is executed, 1.0 is assigned to #3.

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

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

4.3.3 Priority of Operations

1. Function
2. Operation such as multiplication and division（*, /, AND）
3. Operation such as addition and subtraction （+, -, OR, XOR）



4.3.4 Bracket Nesting

Brackets are used to change the order of operations. Brackets can be used to multinesting.

Note that the square bracket [,] is used to enclose an expression; the round bracket（，）is used in notes. When the priority is not defined, it is advised to use square bracket to enclose.

4.4 Branch and Repetition

In a program, the flow of control can be changed using the GOTO statement and IF statement. Three types of branch and repetition operations are used:

1. GOTO statement (unconditional branch)
2. IF statement (conditional branch: IF…THEN…)
3. WHILE statement (repetition WHILE…)

4.4.1 Unconditional Branch (GO TO statement)

Go to the block with sequence number n. when a sequence number out the range of 1~99999 is specified, an alarm is raised. A sequence number can also be specified using an expression.

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

Example： GOTO 1；GOTO #101；

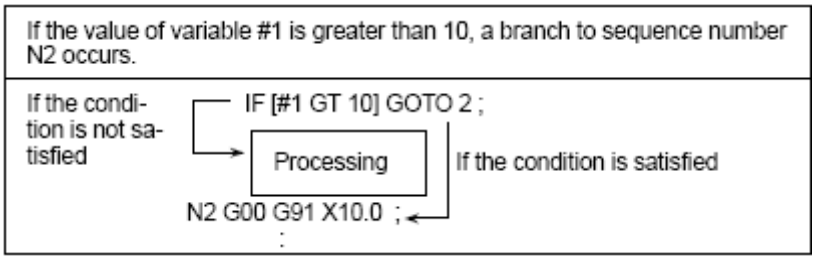
4.4.2 Conditional Branch (IF statement)

Specify a conditional expression after IF.

GOTO format: IF [conditional expression] GOTO n；

If the specified conditional expression is satisfied, a branch to sequence number n occurs. If the specified condition is not satisfied, the next block is executed.

Example:



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

If the specified conditional expression is satisfied, a predetermined macro statement is executed. Only a single macro statement is executed.

Example:

IF[#1 EQ #2] THEN #3=0；

If the value of #1 and #2 are the same, 0 is assigned to #3; if not, no execution will be performed.

4.4.3 Conditional Expression

Conditional expression: A conditional expression must include an operator between two variables or between a variable and constant, and must be enclosed in brackets [,]. An expression can be used instead of a variable.

Operators: In GSK980MDc Plus, operators in the following table are used to compare two values to determine whether they are equal or one value is smaller or greater than the other value.

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

Example： IF [3<>2] GOTO 2; it means if 3 is not equal to 2, branch to N2 block

IF [#101>=7.22] THEN #101=SIN30; it means, if #101 is greater than 7.22, the expression after THEN is

executed, i.e., assign Sin 30°to #101.

Sample program The sample program below finds the sum of number 1 to 10.

```
O9500
 #101=0                  Initial value of the variable to hold the sum
#102=1                   initial value of the variable as an addend
N1 IF[#102 GT 10]GOTO 2 … … Branch to N2 when the addend is
                                    greater than 10
#101= #101+#102   … … calculation to find the sum
#102= #102+1   … … Next addend
GOTO 1   … … Branch to N1
N2 M30   … … End of program; Sum of number 1 to 10
```

4.4.4 Logical Expression

A macro program logical expression is composed of conditional expression or a single expression, and the expression obeys its operation rules. Logical express outputs 0 or 1 after operation. Outputting 1 means the logic expression is truth, 0 means is not truth. An alarm occurs when logic operation is not 0 or 1.

Operator: GSK980MDc Plus logical operation uses AND（&&）, OR（||）, logical operation outputting result is 0 or 1, which means the logical expression to be truth or not. The following explains their uses of the two logic operators:

Logical operator	***AND operation（&&）***				***OR operation（||）***			
Expression 1	***1***	***1***	***0***	***0***	***1***	***1***	***0***	***0***
Expression 2	***1***	***0***	***1***	***0***	***1***	***0***	***1***	***0***
Output result	***1***	***0***	***0***	***0***	***1***	***1***	***1***	***0***

Example 1: IF[#101>1&& #102>3] GOTO 12, while #101>1 and #102>3 are truth, the system skips to N12 block, otherwise, it does not execute the skip. Set #101 to 2 and #102 to 5, and the program executes skip;

Example 2: only one of IF[#101>1|| #102>3] GOTO 12，#101>1 and #102>3 is truth, skipping to N12 is executed, otherwise the skip is not executed. #101 is set to 3, the system executes the skip even if #102 value is anyone.

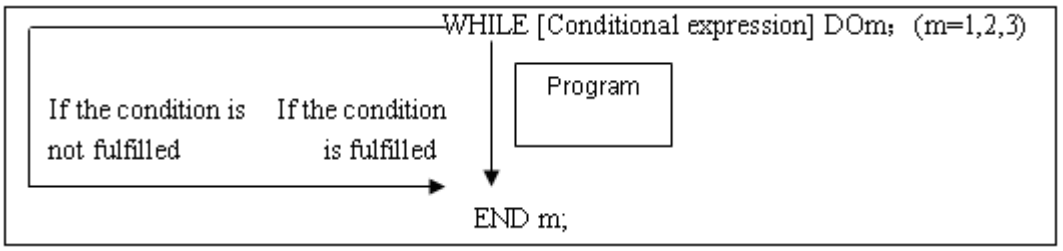
Note:

- Macro program logical expression can execute the logical for many expressions, but their output result must be 0 or 1;
- Output result 1 of logical expression means the condition to be true, 0 means not be true;
- Logical expression must be with IF or while statement, otherwise, an alarm occurs;
- Logical expression supports judgement statement of a single expression. For example, IF[#101] or IF[1] or IF[A>0&&#101], #101 or the constant reading data is 0 or 1, the expression can run, an alarm occurs when the data is 1.0.

4.4.5 Repetition（WHILE Statement）

Specify a conditional expression after WHILE. While the specified condition is satisfied, the program from DO to END is executed. If not, program execution proceeds to the block after END.

Example：



Explanations: While the specified condition is fulfilled, the program from DO to END after WHILE is executed. If the specified condition is not fulfilled, program execution proceeds to the block after END. The same format as the IF statement applies. A number after DO and a number after END are identification numbers for specifying the range of execution. The number 1, 2, and 3 can be used. When a number other than 1, 2, and 3 is used, P/S alarm occurs.

Nesting: The identification number (1 to 3) in a DO, END loop can be used as many times as desired. Note, however, when a program includes crossing repetition loops (overlapped DO ranges), P/S alarm occurs.

1. The identification numbers (1 to 3) can be used as many times as required.

```
WHILE [ ... ] DO 1 ;
    Processing
END 1 ;
:
WHILE [ ... ] DO 1 ;
    Processing
END 1 ;
```

2. DO ranges cannot overlap.

```
WHILE [ ... ] DO 1 ;
    Processing
    WHILE [ ... ] DO 2 ;
:
END 1 ;
    Processing
END 2 ;
```

3. DO loops can be nested to a maximum depth of three levels.

```
WHILE [ ... ] DO 1 ;
:
    WHILE [ ... ] DO 2 ;
    :
        WHILE [ ... ] DO 3 ;
            Processing
        END 3 ;
        :
    END 2 ;
    :
END 1 ;
```

4. Control can be transferred to the outside of a loop.

```
WHILE [ ... ] DO 1 ;
    IF [ ... ] GOTO n ;
END 1 ;
Nn
```

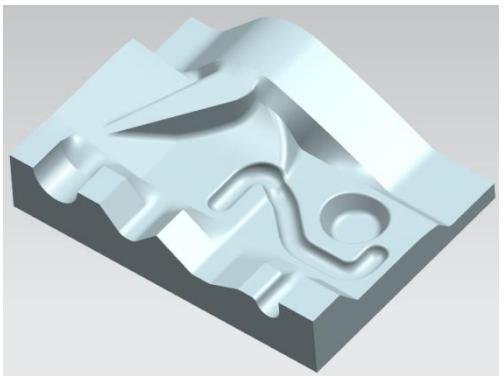
5. Branches cannot be made to a location within a loop.

```
IF [ ... ] GOTO n ;
:
WHILE [ ... ] DO 1 ;
Nn ... ;
END 1 ;
```

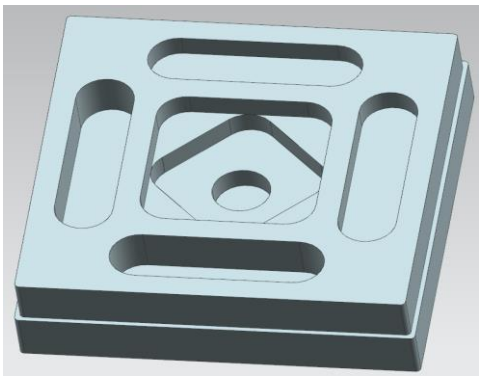
Chapter 5 Small Line Segment Foresight Function

5.1 Overview of Small Line Segment Foresight Function

The small line segment forward-looking function is mainly used for mold part processing in software such as UG and Mastercam, and is also commonly used in the manufacturing of workpieces with high contour accuracy requirements.



Mold processing,



high-precision part processing

5.2 Enabling Method for Small Line Segment Foresight

The preview function of small line segments can be enabled in two ways through G5.1 commands and parameter settings.

5.2.1 G5.1 Command Enable

Format: G5.1 Q_

Explanation: 1. The Q value can only be selected as 0 or 1. Entering other values will prompt an alarm

2. When Q1 is specified, the small line segment foresight function is effective;
3. When Q0 is specified, the small line segment forward function is invalid;
4. After the end of M30 operation, the small line segment foresight function is invalid;
5. When powered on, the default small line segment forward function is invalid.

Example:

```
G0 X0 Y0 Z0
G5.1 Q1 (Enter small segment foresight)
G01 X0.1 Y0.1
X0.3 Y0.2
.....
G5.1 Q0 (Cancel small line segment foresight)
```

5.2.2 Enable Parameter Settings

No.20	Cutting control selection, 0: Efficiency priority, 1: Improve contour accuracy under efficiency priority, 2: Forward contour priority

When No.20 #=2, the small line segment foresight function is often effective. No need to use G5.1 command to enable this function.

5.3 Forward Looking Parameters for Small Line Segments

No.80	When the small line segment foresight is effective, the maximum cutting feed rate for all axes is 5000 (standard value)

Explanation: The upper limit speed of cutting, if the programming speed exceeds this speed, it will run at this speed.

No.82	When the small line segment foresight is effective, the maximum composite acceleration of all axes (standard value 50)

Explanation: The upper limit of cutting acceleration and deceleration. If it exceeds this acceleration during operation, it will operate at this acceleration.

No.83	When the small line segment foresight is effective, the maximum allowable acceleration for each axis (standard value 120)

Explanation: If the upper limit of single axis acceleration and deceleration for cutting exceeds the minimum acceleration during operation, it will pass through the minimum acceleration,limit synthetic acceleration.

No.84	When the small line segment foresight is effective, the maximum speed step value of all axes (standard value 120)

Explanation: The larger the step value, the faster the transition speed between segments, the greater the vibration of the machine tool, and the smaller the step value,The slower the speed between segments, the less vibration the machine tool will have, and the smoothness of the machining will decrease.

No.89	When the small line segment foresight is effective, the accuracy of reverse clearance determination is 0.001mm (Z-axis standard value 3)

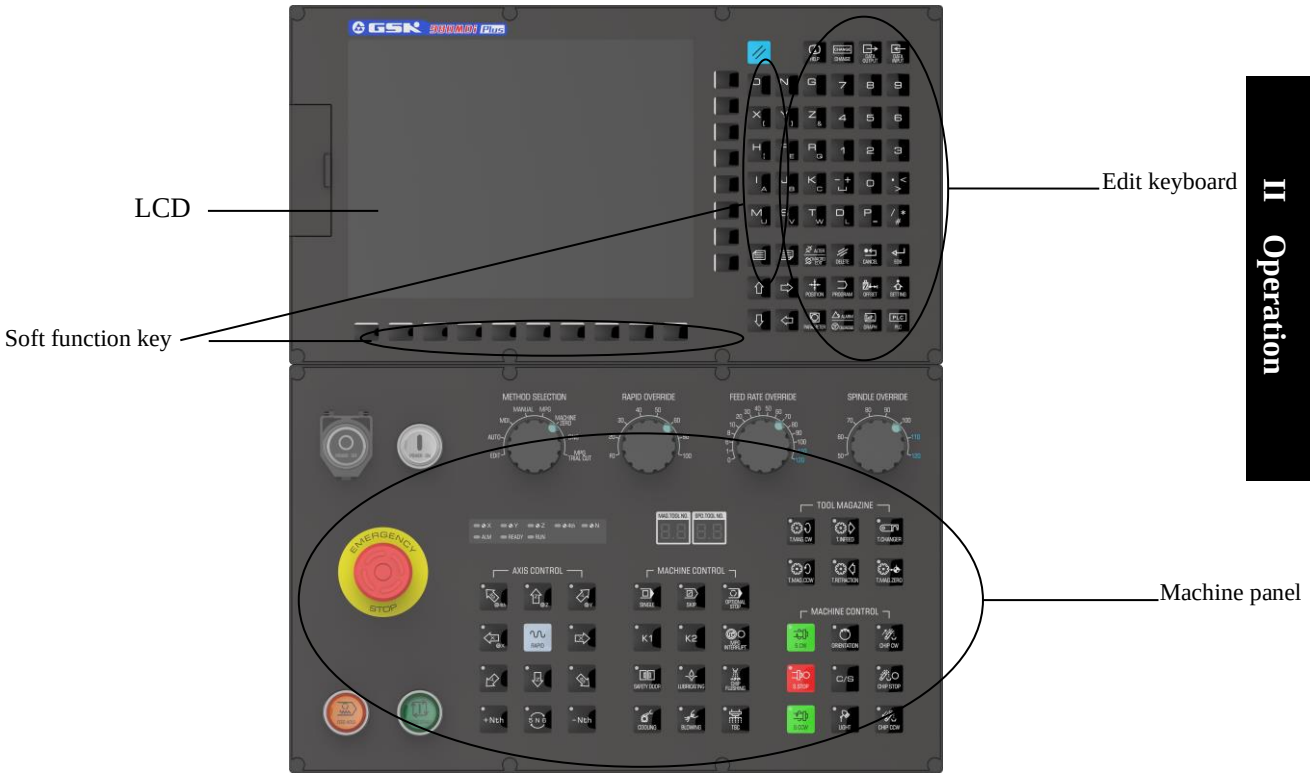
Explanation: Determining the reverse accuracy is to avoid repeated reverse operations when programming small line segment programs with extremely small reverse data,Causing scratches on the surface of the processing mold.

Ⅱ Operation

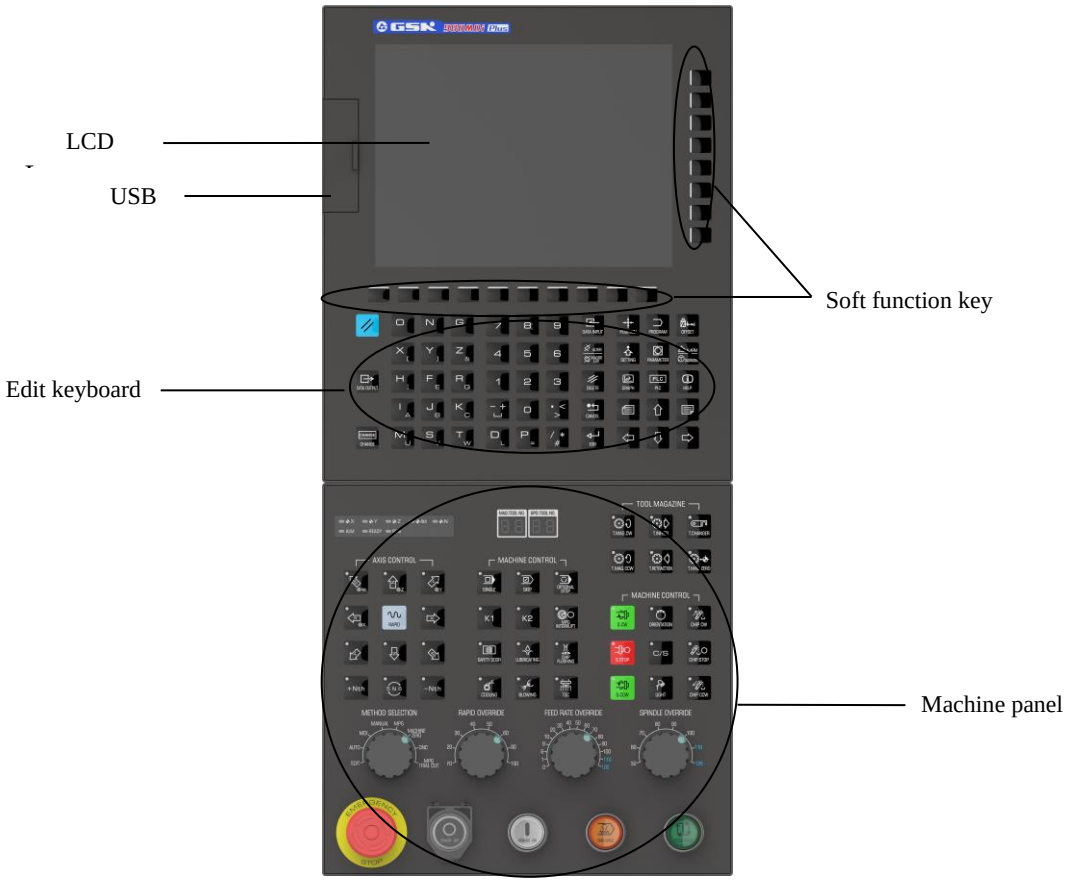
Chapter 1 Operation Mode And Display

1.1 Panel Division

This GSK980MDi Plus adopts an integrated panel, which division is as follows:



GSK980MDi Plus-H panel division



GSK980MDi Plus-V panel division

1.2 Summary of Operation Mode

There are 8 modes that include Edit, Auto, DNC,MDI, Machine zero, Step/MPG, Manual, MPG test cutting, modes in this GSK980MDi Plus.

- **Edit mode**
  In this mode, the operation of part program setting-up, deletion and modification 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, Y, Z, 4th, 5th axis 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, Lubrication ON/OFF, spindle jog, manual tool change can be performed.
- **DNC mode**
  In this mode, the program is run by DNC mode.
- **MPG test cutting mode**
  In the manual pulse cutting method, the speed of program execution can be controlled by rotating the manual pulse, thereby achieving the goal of detecting whether the processing program is correct.

Chapter 2 Manual Operation

Note!

The keys functions of this GSK980MDi Plus machine panel are defined by Ladder Diagram; please refer to the respective materials by the machine builder for the function significance.

Please note that the following function introduction is described based on the GSK980MDi Plus standard PLC programs!

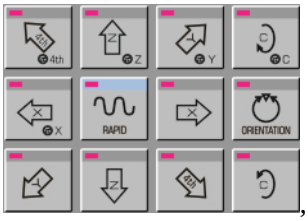
Press 

key, it enters Manual mode. In this mode, the manual feed, spindle control, override adjustment operations can be performed.

2.1 Coordinate Axis Moving

In Manual mode, the coordinate axis can be moved manually for feeding and rapid traverse.

2.1.1 Manual Feed

Press feed axis and axis direction key in the direction selection area 

, the corresponding axis may be moved positively or negatively, and the axis stops moving if releasing these two keys; and the direction selection keys of X. Y. Z. 4th. 5th axes can be hold on at a time to make the 5 axes to move simultaneously.

2.1.2 Manual Rapid Traverse

First press 

key in the feed axis and direction selection area 

till the rapid traverse indicator in the State area lights up. The corresponding axis can be rapidly moved positively or negatively by pressing direction selection key, and the axis stops moving if releasing the key; and the direction selection keys of X. Y. Z. 4th. 5th axes can be hold on at a time to make the 5 axes to move simultaneously.

In Manual rapid mode, press 

key to make the indicator go out, and the rapid traverse is invalid, it enters the Manual feed mode.

Note 1: Before machine zero return, the validity of manual rapid traverse is set by the ISOT of the bit parameter No.012.

Note 2: In Edit or MPG mode,  **key is invalid.**

2.2 Feedrate Override Adjustment

2.2.1 Manual Feedrate Override Adjustment

In Manual mode, the addition or reduction key  in can be pressed to modify the Manual feedrate override, and the override has 16 levels. The speed at a feed rate of 100% is determined by data parameter No.044.

2.2.2 Manual Rapid Override Adjustment

In the manual rapid traverse,  can be pressed (also by  key with the respective override F0, 25%,50%，100%)to modify the Manual rapid override, and there are 4 gears of F0, 25%, 50%，100% for the override.(F0 is set by data parameter No.024)

2.2.3 Spindle Override Adjustment

In Manual mode, if the spindle speed is controlled by analog voltage output, the spindle speed may be adjusted.

By pressing the  key in Spindle Override keys, the spindle speed can be changed by real-time adjusting of the spindle override that has 8 levels of 50％～120％.

Chapter 3 MDI Operation

Note!

The keys functions of this GSK980MDi Plus machine panel are defined by Ladder; please refer to the respective materials by the machine builder for the function significance.

Please note that the following function introduction is described based on the 980MDi Plus standard PLC programs!

In MDI mode, the operations of parameter setting, words input and execution can be performed. In MDI mode, edited programs can be gained by editing program file O0000, and MDI runs programs used for simple test operations.

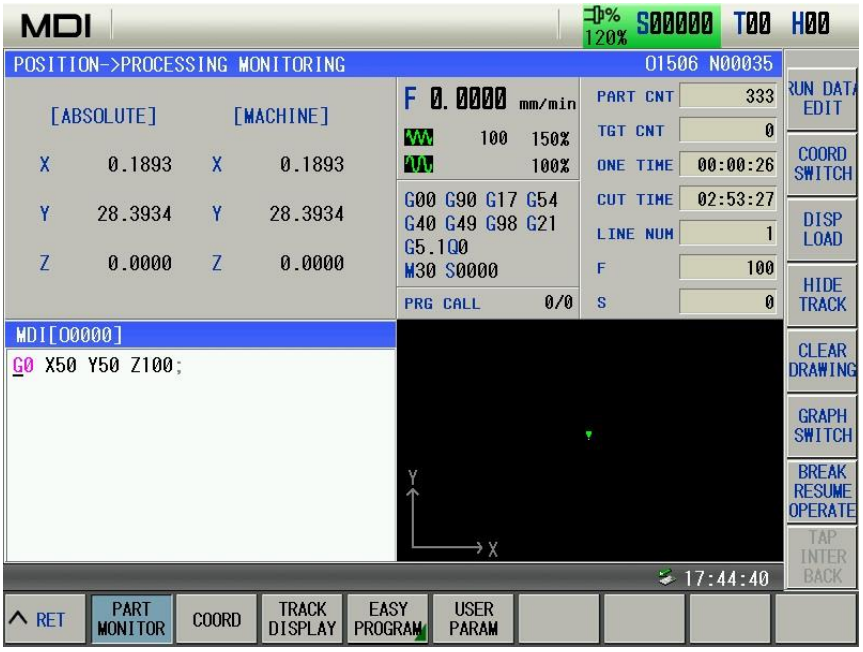
3.1 Blocks Input

Select MDI mode to enter the PRG STATE page, to input an block “G00 X50 Z100”，the steps are as follows:

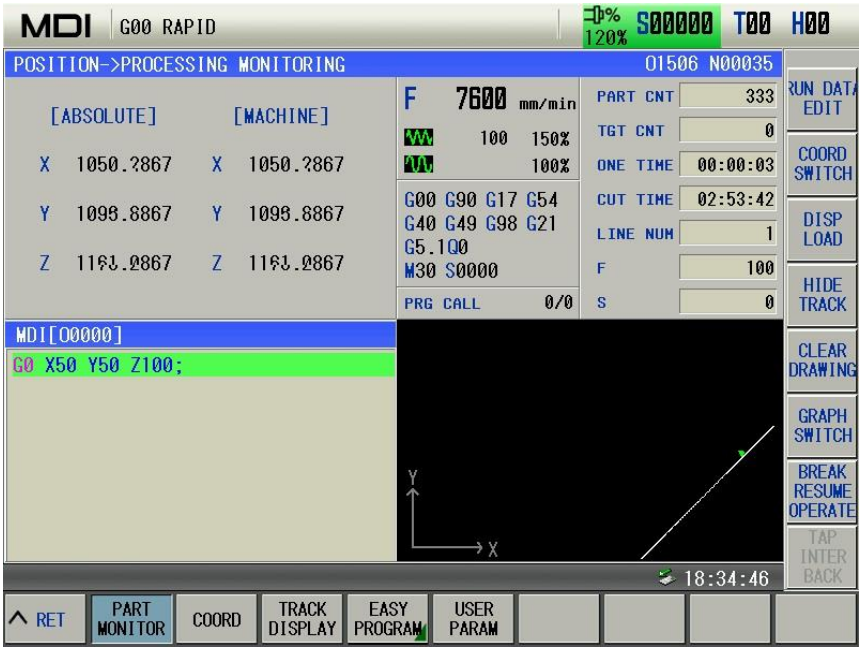
1. Rotating working mode button to enter MDI mode;
2. Press MDI key to enter MDI PROGRAM page:



3. Input G. O. O, X. 5. O, Y& 5. O, Z; 1. O. O by sequence, the page is as follows:



4. Press start，the page is as follows:



3.2 Blocks Execution

After the words are input, and press , the background color of program segment becomes white, these MDI words are executed after the  key is pressed. During the execution, press ,  and Emergency Stop button may be pressed to terminate the MDI words execution. If  key is pressed, the background color of program segment will becomes black, then words can be input again.

Note: The subprogram call command (M98 P, etc.) is invalid in MDI mode.

Chapter 4 Auto Operation

Note!

The keys functions of this GSK980MDi Plus machine panel are defined by Ladder; please refer to the respective materials by the machine builder for the function significance.

Please note that the following function introduction is described based on the GSK980MDi Plus standard PLC programs!

4.1 Auto Run

4.1.1 Selection of a Program To Be Run

1. Search method

1）Select the Edit or Auto mode;

2）Press  key to enter the PRG CONTENT page;

3）Press the address key  and key in the program No.

4）Press  or  key, the program retrieved will be shown on the screen, if the program doesn't exist, the system prompts the required file does not exist at the bottom of the CNC.

5) Press , the program retrieved will be shown on the screen, if the program doesn't exist, the system prompts "PROGRAM DOES NOT EXIST".

Note: In step 4, if the program to be retrieved does not exist, a new program will be setup by CNC system after pressing  key.

2. Scanning method

1）Select the Edit or Auto mode;

2） Press  key to enter the PRG display page;

3）Press the address key ;

4）Press the  or  key to display the next or previous program;

5）Repeat the step 3, 4 above to display the saved program one by one.

3. File open method

Select the edit or operation mode:

1）Press  key twice to enter the page of file list;

2）Press ,  keys to move the cursor to the file will be selected.

3）Press  key to select a file.

4）Press  key to open the selected file.

Note: The file can not be opened if the expanded name is not ".CNC".

4.1.2 Program Start

1. Press  key to select the Auto mode.

2. Press  key to start the program, and the program execution begins.

Note: Since the program execution begins from the block where the cursor locates, before pressing the  key, make a check whether the cursor is located at the block to be executed. If begins from the start line, but the cursor is not in this line, move the cursor to the line.

4.1.3 Stop of Auto Run

- **Stop by command (M00)**

  A block containing M00 is executed, the auto run is stopped. So the modal function and state are all reserved. Press the key  or the external Run key, the program execution continues.

- **Stop by a relevant key**

  1. In Auto run, by pressing key  or external dwell key, the machine remains at the following state:

  （1）The machine feed decelerate to stop.

  （2）During the execution of the dwell command (G04), it pauses after G04 command execution is finished.

  （3）The modal function and state are saved.

  （4）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 invalid (the automatic cut-off of signals such as spindle CCW/CW, lubrication, 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

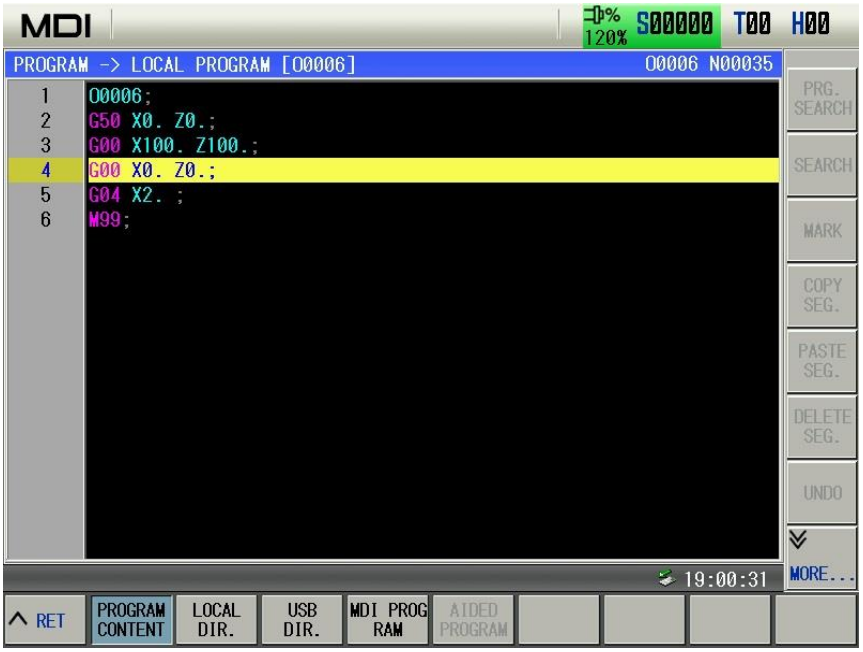
  If the external emergency button (external emergency signal valid) is pressed under the dangerous or emergent situation during the machine running, the CNC system enters into emergency state, and the machine moving is stopped immediately, all the output (such as spindle rotation, coolant) are cut off. If the Emergency button is released, the alarm is cancelled and CNC system enters into reset mode.

  4. By Mode switching

  When the Auto mode is switched to the Machine zero, MPG/Step, the current block "dwells" immediately; when the Auto mode is switched to the Edit, MDI mode, the "dwell" is not displayed till the current block is executed.

4.1.4 Auto Run From an Arbitrary Block

1. Press  key to enter the Edit mode, press  key to enter the Program interface, or press  key several times to select the PRG CONTENT page:
2. Move the cursor to the block to be executed (for example, move the cursor to the 4th line head if it executes from the 4th line);



3. If the mode （G, M, T, F command）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 the next step.
4. Press  key to enter the Auto mode, then press  key to start the program.

4.1.5 Adjustment of Feedrate Override, Rapid Override

In Auto mode, the running speed can be altered by adjusting the feedrate override, rapid override with no need to change the settings of the program and parameter.

- Adjustment of the feedrate override

  Rotating working mode button 

, it can realize 16-level real time feedrate adjustment.
- Adjustment of rapid override

  It can realize the 4-level real time rapid override F0. 25%. 50%. 100% adjustment by pressing 

.

4.1.6 Spindle Override Adjustment

While the spindle speed is controlled by the analog voltage output in Auto mode, it can be adjusted by spindle override.

Rotating working mode button 

to adjust the spindle override for the spindle speed, it can realize 8-level real-time override adjustment between 50%～120%.

4.2 DNC Running

The CNC system has a DNC function, by the connection of the DNC communication software with this system, the high speed, high capacity program can be performed in this system.

In Auto mode, press the 

key, it enters the DNC mode. Then press the 

key to start the program DNC machining under the condition that the PC is get ready

When DNC runs, subprograms can be executed, the max. built-in subprogram is 1-layer.

When DNC runs, macro program does not skip in the main program, otherwise, an alarm occurs: Cannot execute program skip in DNC mode.

Please refer to the DNC communication software for details.

4.3 Running State

4.3.1 A Single Block Execution

When the system first executes a program, it may select Single block mode to execute the program to avoid the programming errors.

In Auto mode, the methods for turning on single are as follows.

Press the 

key to make the single block indicator in State area to light up, it means that the single block function has been selected

In Single block mode, when the current block execution is finished , the CNC system stops; if next block is to be executed, it needs to press the 

key.

Note : Even at the mid point, the single block stops in G28,G29, G30 commands.

4.3.2 Dry Run

Before the program is to be executed, in order to avoid the programming errors, it may select the Dry run mode to check the program. And the machine runs by a constant speed other than the speed specified by the program.

In Auto mode, the method for turning on the Dry run switch are as follows.

Press 

key to make the dry run indicator in State area to light up, it means that the dry run function is selected.

The speed specified by the program is invalid in dry run, and actual feedrate is set by the data parameter No.25.

4.3.3 Machine lock

In Auto mode, the ways to make machine lock function valid are as follows.

Press the  key to make the machine lock indicator in State area to light up, it means that it has entered the machine lock state.

While in the machine lock mode:

1. The machine carriage doesn't move, the "MACHINE" in the INTEGRATED POS page of the POSITION interface does not vary too. The RELATIVE POS and ABSOLUTE POS, DIST TO GO are refreshed normally
2. M, S, T commands can be executed normally.

4.3.4 MST Lock

In Auto mode, the ways to make MST lock function valid are as follows.

Press the  key to make the MST lock indicator in State area to light up, it means that it has entered the MST lock state. And the carriage move is not performed by M, S, T commands

Note: When the MST lock is valid, it has no effect on the execution of M00, M30, M98, M99.

4.3.5 Block Skip

If 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 function is valid, this block is skipped without execution in Auto mode

In Auto mode, the way to make block skip function valid is as follows.

Press the  key to make the block skip indicator in State area to light up, it means that the block skip function is valid.

Note: While the block skip function is invalid, the blocks headed with "/" signs are executed normally in Auto mode.

4.3.6 Optional Stop

In AUTO mode, the valid optional stop function is as follows:

Press  key to enter optional stop and the indicator lights up.

The program will be "stopped" at command M01. Press  key again to continue program execution.

Chapter 5 Machine Zero Return Operation

5.1 Machine Zero

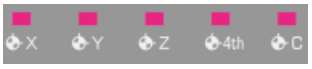
The machine coordinate system is a basic coordinate system for CNC coordinate calculation. It is an inherent coordinate system of the machine. The origin of the machine coordinate system is called **machine zero** (or machine reference point). It is defined by the zero return switches fixed on the machine. Usually the switch is fixed on the positive max. Strokes of X, Y, Z axes.

Note: the mechanical zero is called "machine zero".

5.2 Machine Zero Return Steps

1. Press 

key, it enters the Machine zero mode, the bottom line of the screen page shows "REF", the figure is shown below.
2. Press  or  or  key to select the machine zero of X, Y or Z axis.
3. The machine moves along the machine zero direction, and returns to the machine zero via the deceleration signal, zero signal detection. And the axis stops with the machine zero finish indicator lighting up.



Machine zero finish indicators

Note:When machine zero return, bit parameter №011 ZNIK determines whether axis movement is locked automatically.

Chapter 6 Find The Workpiece Centre

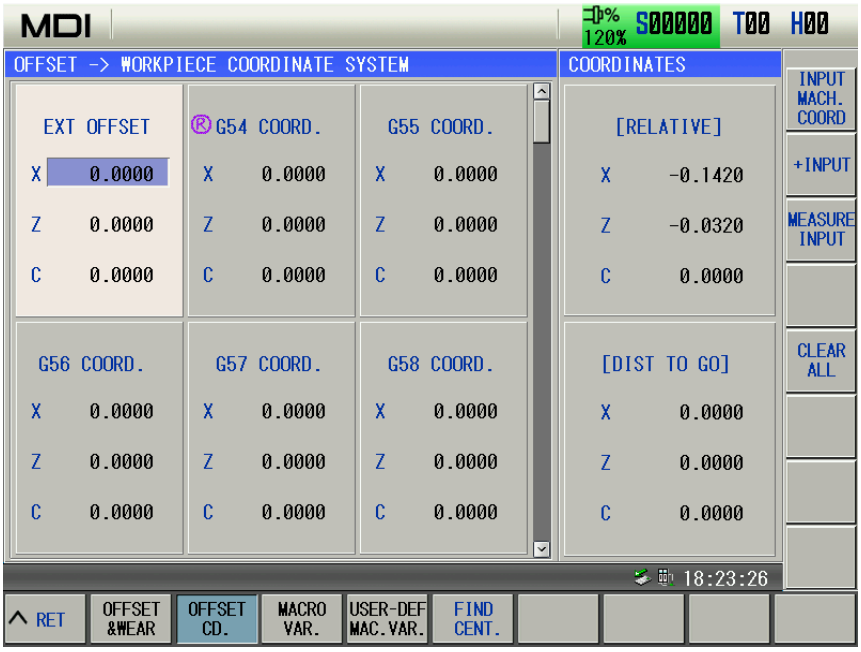
6.1 Operation of Finding the Center Point of the Workpiece

6.1.1 Scope of Application

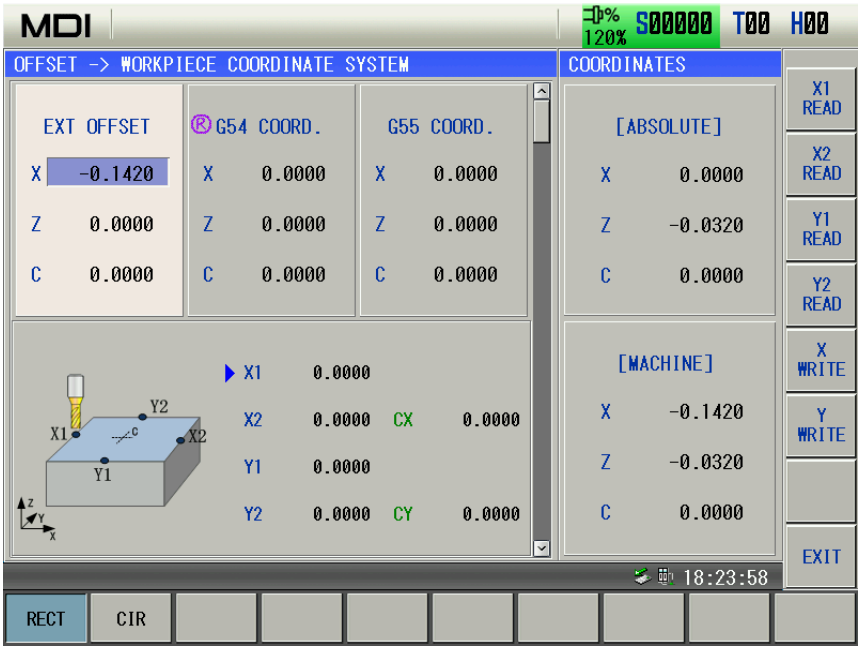
For the machining of rectangular and circular arc workpieces, by using the function of finding the center point of the workpiece, only a few necessary measurement point coordinates need to be input, and the system can automatically calculate the machine tool coordinates of the center point of the rectangular or circular arc workpiece, and then write them into the coordinate system.

6.1.2 How to Find Part's Center Point

Press button ,then press soft key to select coordinate page, press again  to enter sub-directories. As shown in below screen-shot.

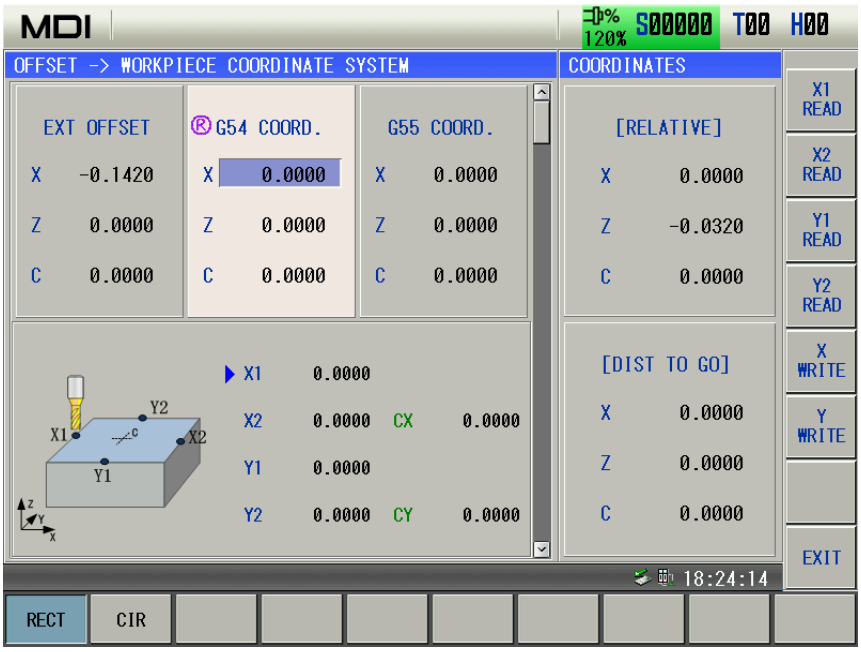


Pre softkey FIND CENT. to enter the interface of finding part center point.As shown in below screen-shot.

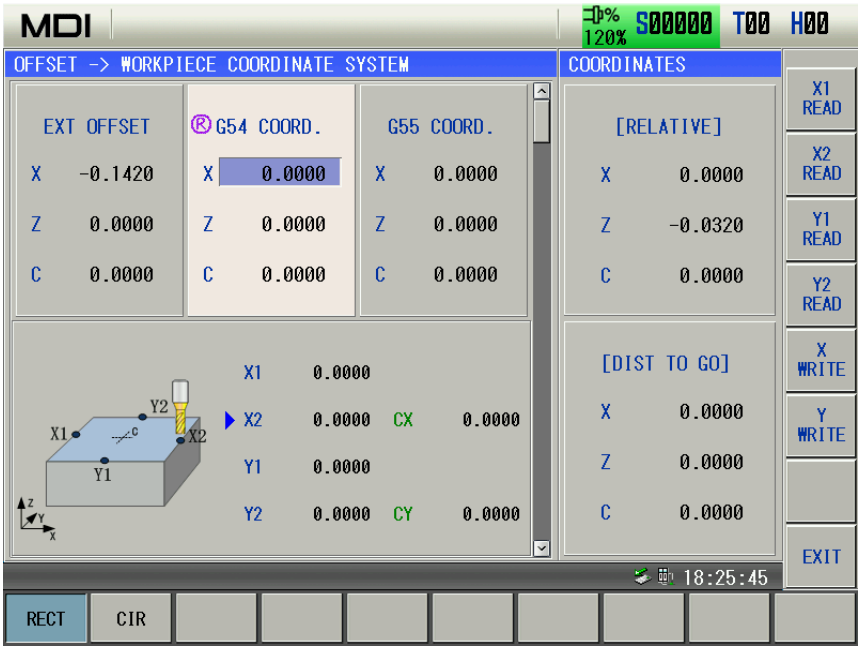


6.1.3 Find Center Point of Rectangle Part

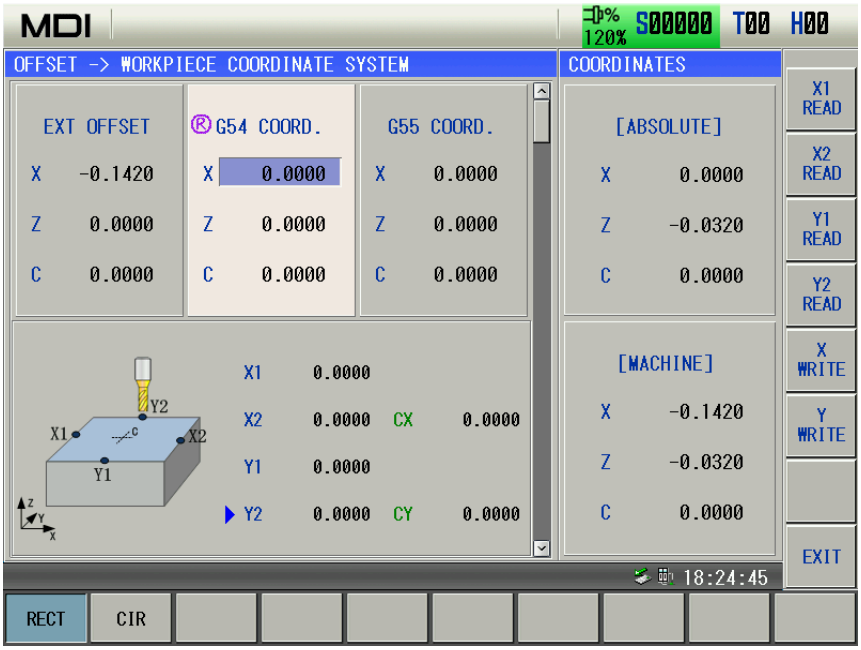
Select axis X or Y in coordinate, press softkey FIND CENT. to enter the interface of finding center point of rectangle part.As shown in below screen-shot.



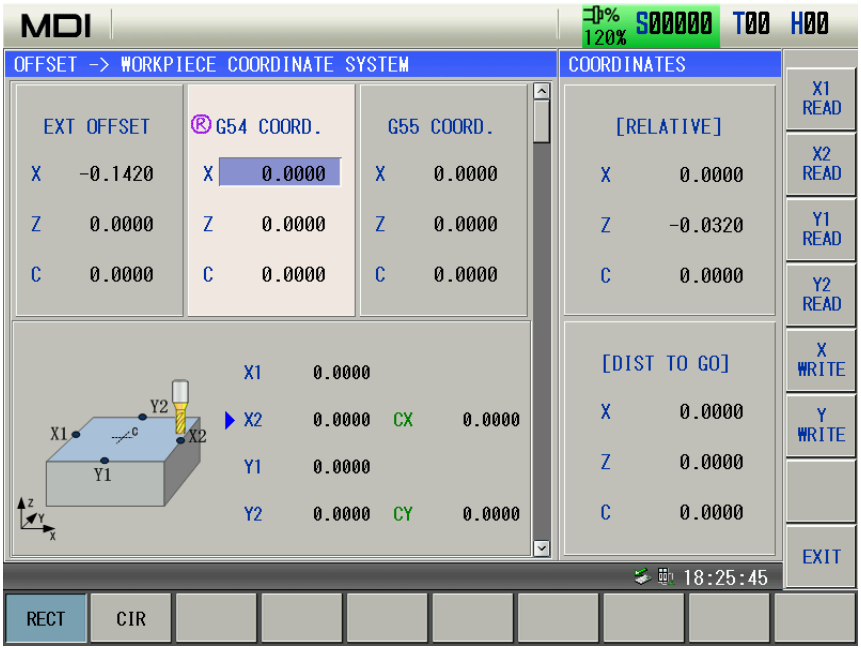
Move X axis to the first measure point, then press RD MACH COORD. to read the measure point No.1 machine coordinate X1.



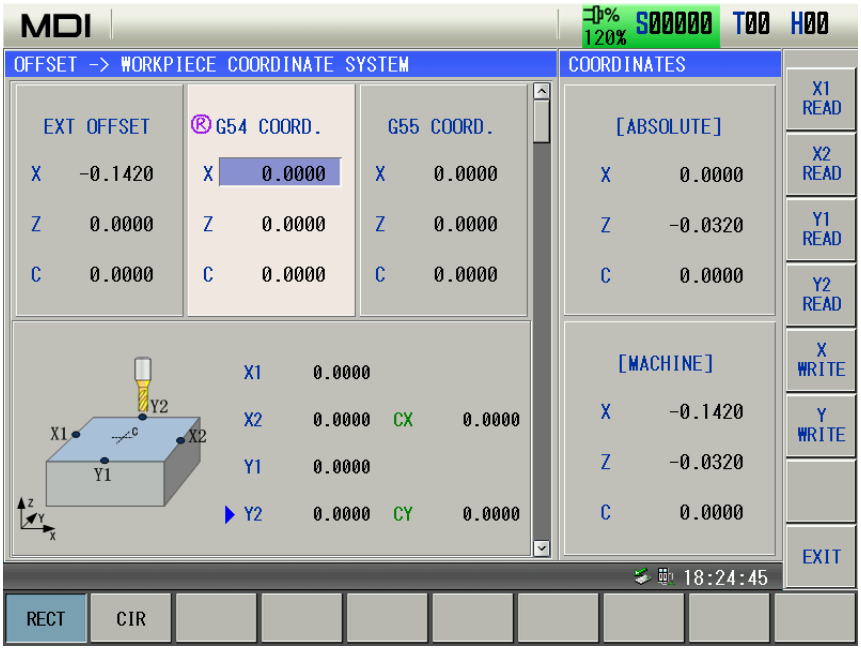
In the same way as above, read machine coordinate of measure point No.2,No.3,No.4 (X2,Y1,Y2),then CNC calculate the machine coordinate of part center CX,CY.



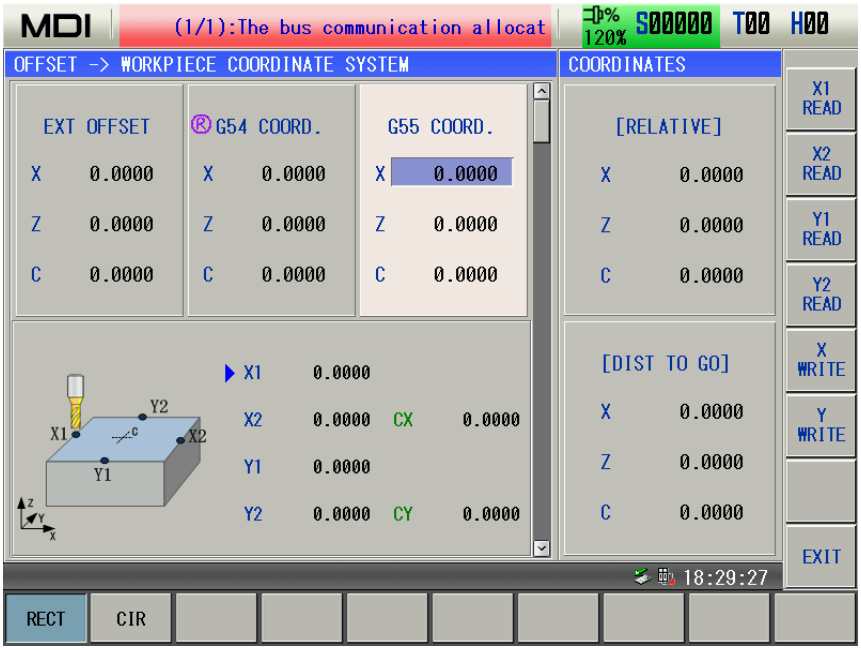
Use ,  to read or re-read the coordinate of target measure point.As shown in below screen-shot,to re-read coordinate of X1.



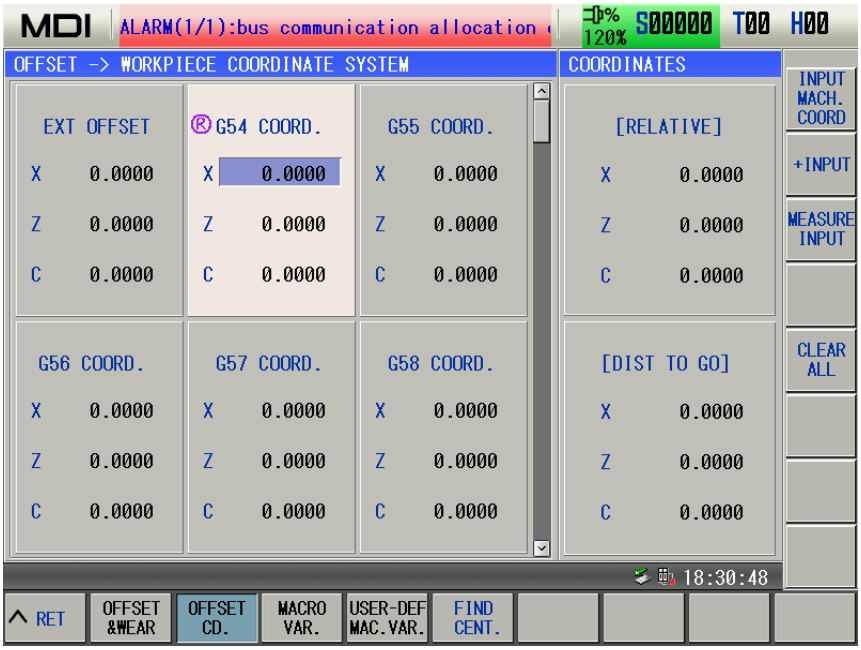
After reading all points coordinate,press softkey CENT SET COORD.,write machine coordinate of center point CX,CY to the selected coordinate system.As shown in below screen-shot.



When writting the center point coordinate, use  for page turning, and  to select the previous or next coordinate system. As shown in below scree-shot, write center point coordinate to G54 and G55 coordinate system.

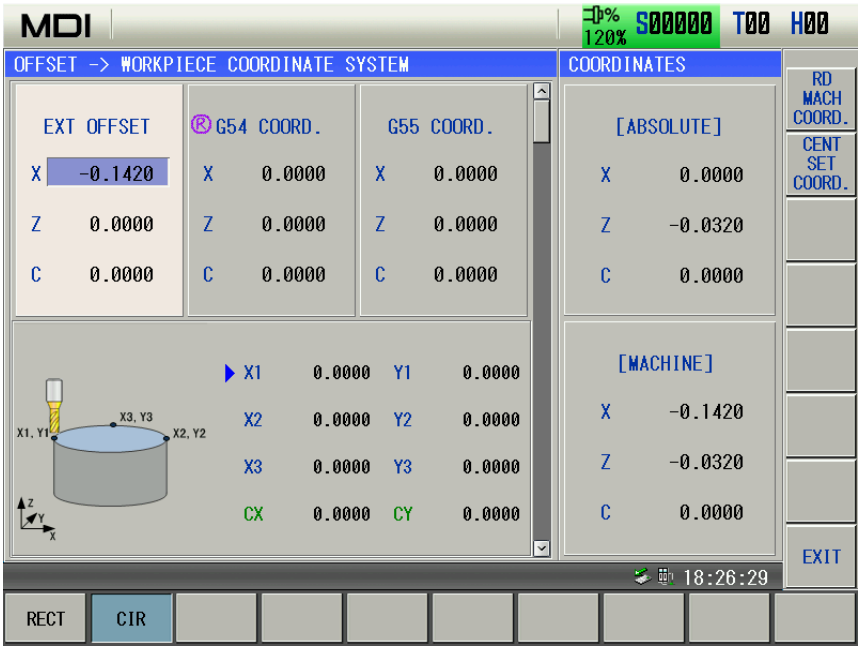


After writing center point coordinate, press softkey EXIT to quit current interface and to return to coordinate interface.

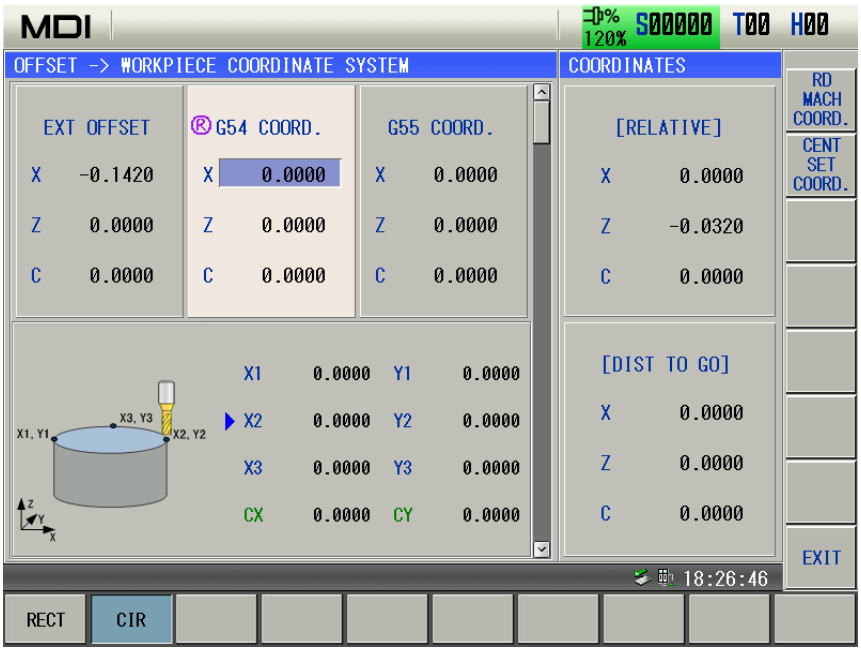


6.1.4 Find Center Point of Arc Part

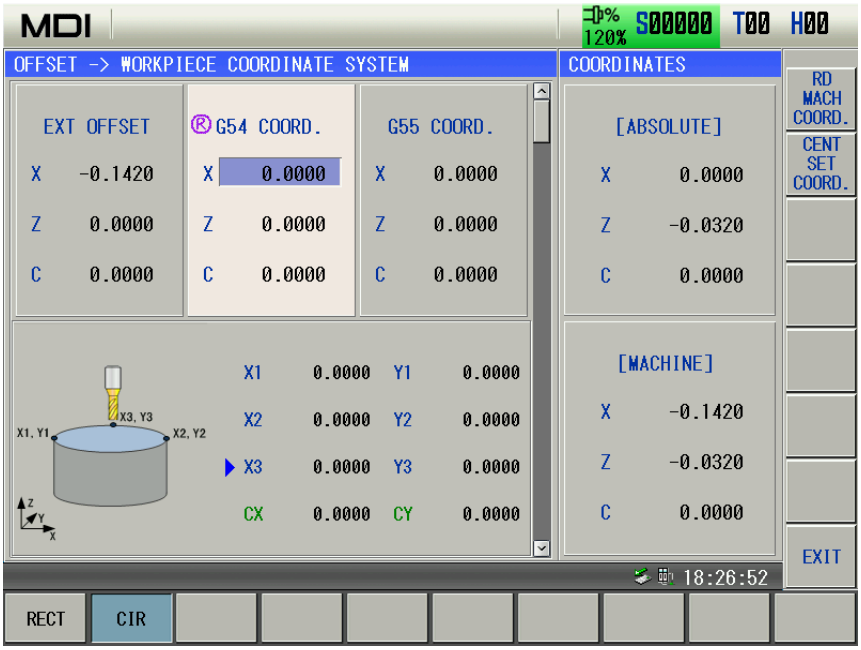
Select axis X or Y in coordinate system, press softkey FIND CENT. to enter interface of Find center point of rectangle part,then press softkey CIR to switch to interface for arc part.As shown in below screen-shot.



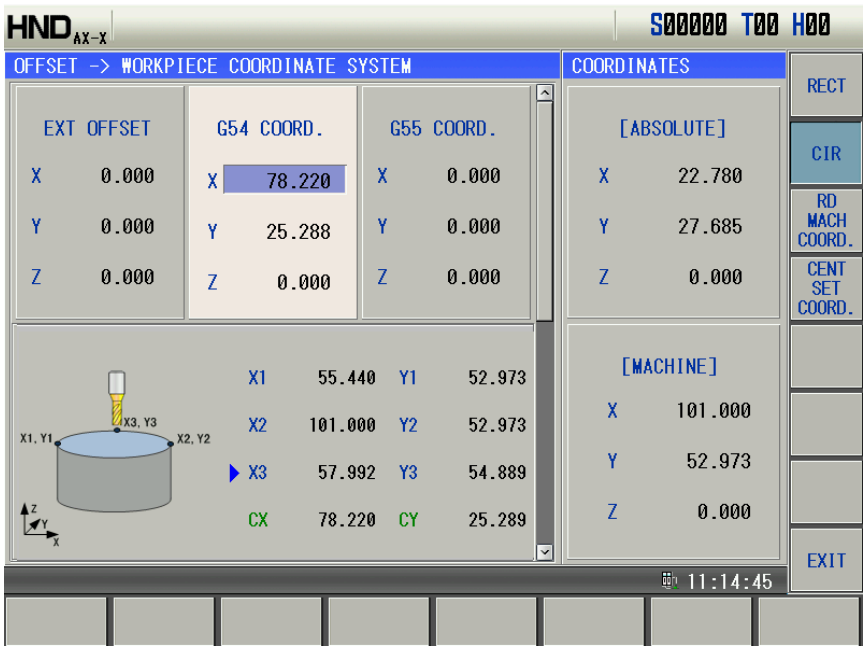
Move tool to the first measure point, press RD MACH COORD. to read its machine coordinate (X1，Y1) as shown in below screen-shot.



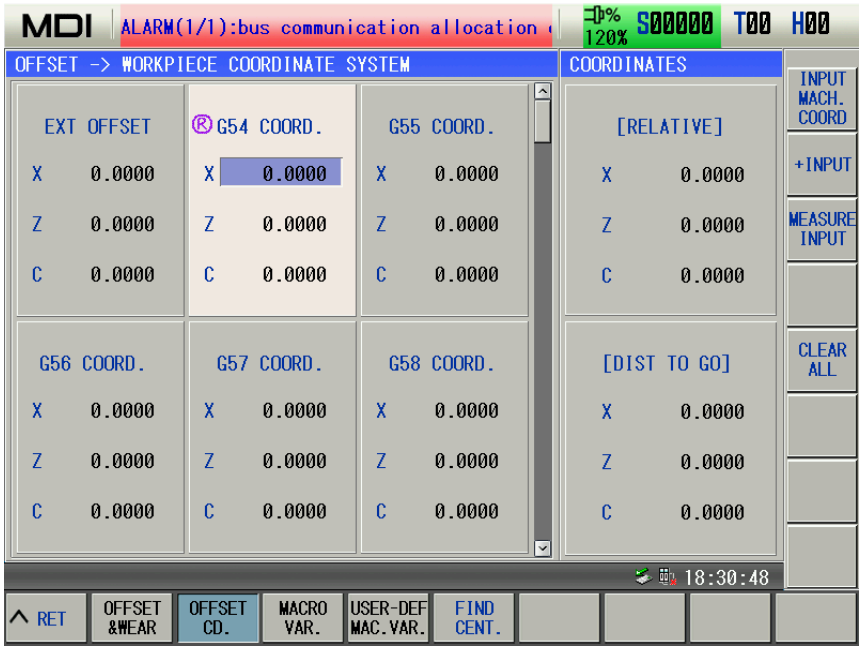
In above same way,read the machine coordinate of point no.2 and point no.3（X2,Y2）、(X3、Y3)



After reading of three measure point, press softkey CENT SET COORD. to write machine coordinate of center point (CX，CY)to the selected coordinate system.



Then press softkey EXIT to quite current interface and return to coordinate interface.



Use  to select measure point,use  to select the coordinate system to write center point coordinate.

Notes:

1、Only axis X and Y can be used to find the center point of rectangle part and arc part.

2、For arc part, when selected three points cannot establish a circle, alarm “three point measured cannot establish a circle,please re-entry”will be generated, meanwhile center point writing will be forbidden.

3、If reading of selected three measure points data is not completed, alarm “Arc three points data input uncompleted,cannot start finding center”will appear.

6.2 Z-axis Tool Alignment Operation

6.2.1 Basic Methods

There are two basic methods for Z-axis tool alignment: one is to align with a reference tool; 2： No benchmark knife for knife alignment.

1、 There are benchmark knife to knife operation methods:

1) Return the Z-axis to the zero point of the machine tool.
2) Using the first tool as the reference tool, move the reference tool to the surface of the G54 workpiece, press tool compensation ->workpiece coordinate system, move the cursor to the Z-axis coordinate of the G54 coordinate system, and enter the Z-axis machine coordinate by pressing "Enter Machine Coordinate";
3) Follow step 2 to set the values of the Z-axis in other coordinate systems.
4) Move the reference knife to the surface of G54 workpiece, press Position ->Relative Coordinates ->Z-axis Reset, and reset the Z-axis relative coordinate.
5) Set the geometric value of length compensation for 01 to 0 by pressing 'Knife Compensation ->Knife Deviation'.
6) Replace the 2nd knife, then move it to the surface of the G54 workpiece, and enter the Z-axis

relative coordinate value in the geometric value of the 02 length compensation at this time.

7) Follow step 6 to set the length compensation for other knives.
8) When writing the program, after changing the tool, use G43 to perform the corresponding length compensation, such as:
   T1 M6;
   G43 H01 Z50;

2、 Operation method for tool alignment without reference:

1) Return the Z-axis to the zero point of the machine tool.
2) Reset the Z-axis coordinate in the workpiece coordinate system to zero. (EXT zero offset and Z in G54 coordinate system need to be set to 0)
3) Input the difference between the surface of the workpiece in other coordinate systems and the surface of the G54 workpiece into the corresponding coordinate system Z-axis. If the surface of the G55 workpiece is 10mm higher than that of the G54 workpiece, then the Z-axis of the G55 coordinate system is set to 10. (Skip this step when using only G54 coordinate system)
4) Using the 1st tool, manually move the Z-axis to the surface of the G54 workpiece, enter the Z-axis machine coordinates in the 1st length compensation value geometry, and then reset the wear value of the 1st length compensation value to zero, completing the 1st tool alignment.
5) Follow the method in step 4 to complete the alignment of other knives.
6) When writing the program, after changing the tool, use G43 to perform the corresponding length compensation, such as:
   T1 M6;
   G43 H01 Z50;

Attention to issues:

1. No reference tool alignment, Z0 position in G49 mode is the zero point position of the machine tool, not the surface position of the workpiece.
2. Without a reference tool for tool alignment, the value of coordinate system Z is equivalent to the offset value of the tool alignment position. If G55 (Z)=10, then the position of Z0 in the G55 coordinate system is offset by 10 from the tool position.

6.2.2 Measurement Input Tool Alignment

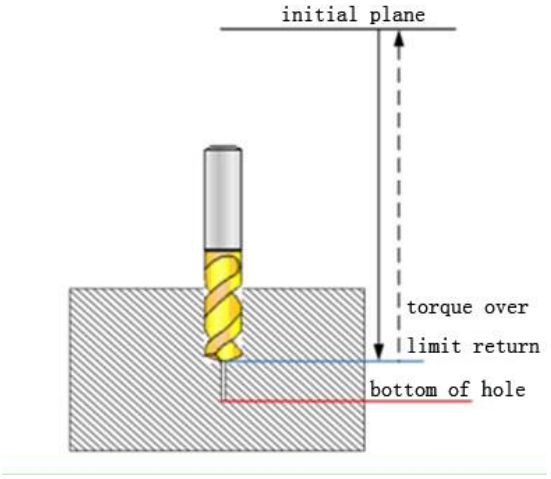
In addition to using basic methods for tool alignment, measurement input can also be used for tool alignment. The measurement input is to input the programming coordinates of the current position, and the system automatically calculates the tool length and the set values of the workpiece coordinate system. If the measured input coordinates are 0, then the absolute coordinates will be 0 after performing corresponding tool length compensation and workpiece coordinate system at the current position.

1) Length compensation page "Measurement input"
   Length compensation setting value=Z-axis machine coordinate - current coordinate system Z-axis setting value - input value.
2) Workpiece Coordinate System Page "Measurement Input"
   Workpiece coordinate system setting value=machine coordinates of the selected axis - input value - zero offset of the selected axis - length compensation value corresponding to the axis in the current plane and tool number state.

Chapter 7 Drilling And Tapping Current Detection

7.1 Functional Description

When drilling or tapping, if the current of the feed shaft or spindle exceeds a certain range, which will cause tool damage, the system will end drilling or tapping in advance and return to the initial plane, prompting the user to perform tool or process inspection.



7.2 Specify Current Detection Through Programming

Parameter Settings:

No.586								JOCK

JOCK =0 When the specified E-value detection current exceeds the limit for drilling and tapping, Z-axis detection is effective

=1 When the specified E-value detection current exceeds the limit for drilling and tapping, the spindle detection is effective

.....

FIXTIME	5128	Delay time after drilling detects current exceeding the limit（ms）

TAPTIME	5138	Delay time after current exceeding limit detected by tapping（ms）

Instruction Description (Taking G81 as an Example):

G81 X_ Y_ R_ Z_ E_

Instructions:

1) The E value specifies a percentage value, which is the percentage of the rated current of the motor. If 60 is specified, the detection value is 60% of the rated current, ranging from 0 to 150%. If it is greater than 150, it will still be calculated as 150.
2) When the E value is specified as 0, the detection function is invalid.
3) When specifying the E value, the parameter No.586 can be used to set whether the detection axis is valid.
4) The E value is the modal instruction.
5) If it is a drilling cycle and the continuous time of current exceeding the limit is less than the detection delay time No.5128, then the over limit return will not be executed, and the system will

continue to drill. If the continuous over limit time is greater than the detection time No.5128, then the over limit return will be executed immediately.

6) If it is a tapping cycle and the continuous time of current exceeding the limit is less than the detection delay time No.5138, the over limit return will not be executed, and the system will continue to perform tapping. If the continuous over limit time is greater than the detection time No.5138, the over limit return will be executed immediately.

7.3 Set the Current Limit Value Through Parameters

7.3.1 Drilling Set Current Exceeds the Limit

FIXZCK	5126	The percentage of Z-axis drilling detection current exceeding the limit and rated current (0-150%), 0: Z-axis detection is invalid
FIXSPCK	5127	The percentage of spindle drilling detection current exceeding the limit and rated current (0-150%), 0: spindle detection invalid

FIXTIME	5128	Delay time after drilling detects current exceeding the limit（ms）

Instructions:

1) When the E-value detection is specified for drilling, the function of detecting current exceedance through parameters is invalid.
2) When the E value is not specified, the function of detecting current exceedance through parameter settings is effective. If only one axis is detected, please set the detection value of the other axis to 0.
3) The set parameter is the percentage value. If 60 is specified, the detection value is 60% of the rated current of the motor.
4) When the drilling detects current exceeding the limit, if the continuous exceeding time is less than the detection delay time No.5128, the over limit return will not be executed, and the system will continue to drill. If the continuous exceeding time is greater than the detection time No.5128, the over limit return will be executed immediately.

7.3.2 Tapping Set Current Exceeding Limit Value

TAPZCK	5136	The percentage of Z-axis tapping detection current exceeding the limit and rated current (0-150%), 0: Z-axis detection is invalid
TAPSPCK	5137	The percentage of spindle tapping detection current exceeding the limit and rated current (0-150%), 0: spindle detection is invalid

TAPTIME	5138	Delay time after current exceeding limit detected by tapping（ms）

Instructions:

1) When tapping with specified E-value detection, the function of detecting current exceeding limit through parameters is invalid
2) When the E value is not specified, the function of detecting current exceedance through parameter settings is effective. If only one axis is detected, please set the detection value of the other axis to 0
3) The set parameter is the percentage value. If 60 is specified, the detection value is 60% of the rated current of the motor
4) If it is a tapping cycle and the continuous time of current exceeding the limit is less than the detection delay time No.5138, the over limit return will not be executed, and the system will continue to perform tapping. If the continuous over limit time is greater than the detection time No.5138, the over limit return will be executed immediately.

Chapter 8 DXF Drill And Tap

8.1 Transfer and Read DXF File

Steps to transfer DXF to CNC:

1）Copy DXF file to USB flash

2）Plug USB flash to CNC to read the flash.

3）Press 

then 

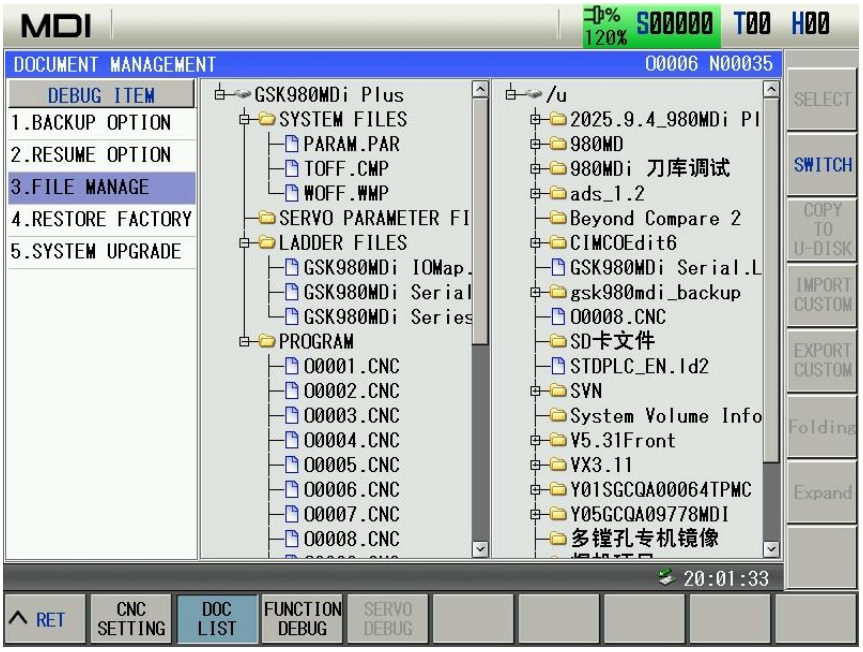
to enter File Management interface;

4）Use 

to select the DXF file in USB flash,in 

mode press 

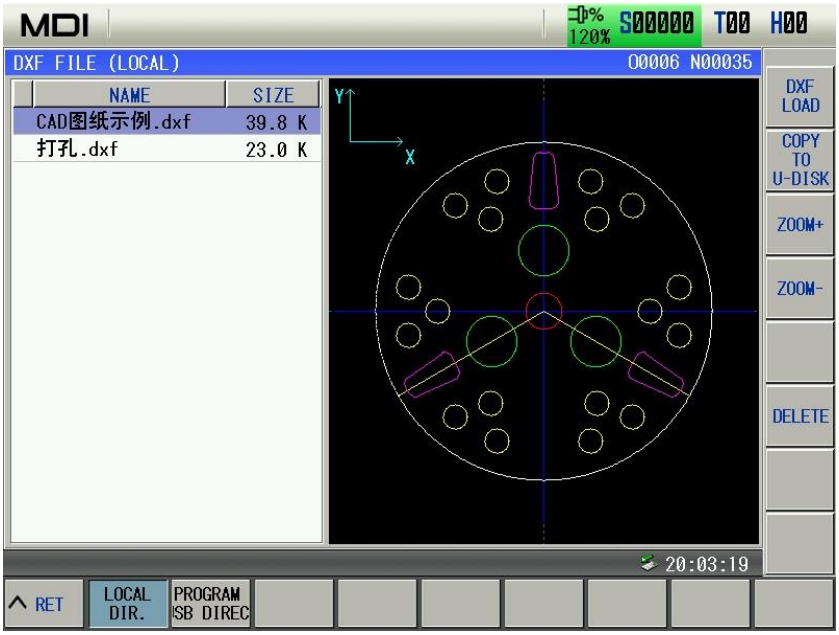
to transfer the selected DXF file to CNC.



After transfering DXF file,press 

then softkey 

to read the transferred DXF file.



8.2 File Drill Parameter Setting

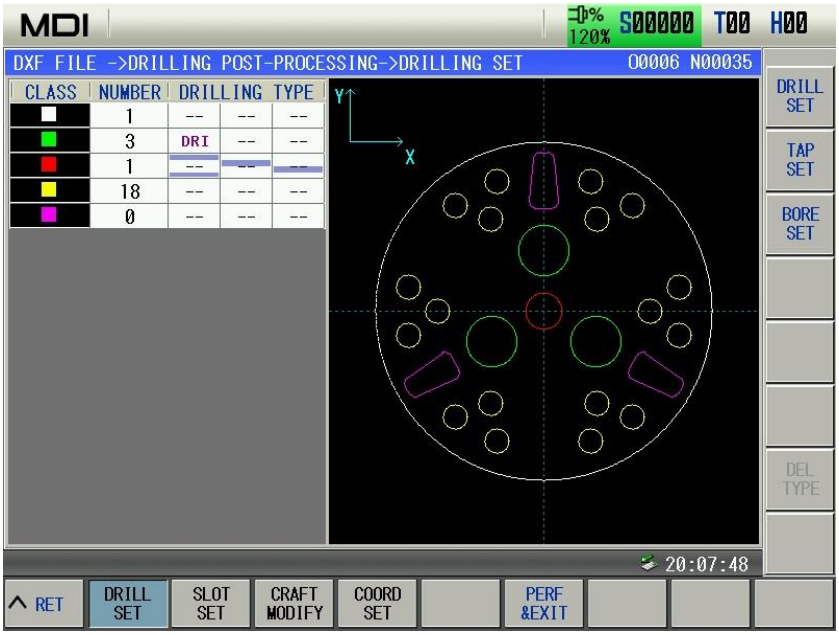
DXF drill parameter setting can work with only full circle data in the file. During editing DXF file, please note 1.CAD position must match the machining position after tool set. 2.Editor should classify the holes, holes with same color can operate three times.

- **Wor Coordinate Setting**

  In DXF file interface, press DRILL PROCESS to enter coordinate system setting interface, user can program for single work station or multi work station per requirement.

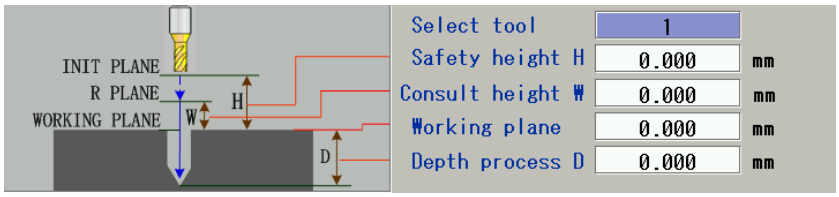
- **Hole Parameter Setting**

  Press DRILL SET to enter drill setting. There are three type of hole process: drill setting, tap setting, and boring setting.Holes with different color will be assorted, select hole color in the left chart, the related holes will be flick on the right. For concentric holes with the same color, max three process can be setted, but only one time tap or boring can be setted.



■ Basic info setting

Basic info setting tab is applied for drill, tap, and boring ,including tool selecting, safty height, reference height, part surface, machine dept,etc.



Tool Select: To select the drill tool. Input range is 1～32 for ATC, if the tool selection for all holes is 0, program will not execute tool change.

Safty Height: Means the start position of Z axis coordinate in fixed cycle. Its the relative value to part's surface.

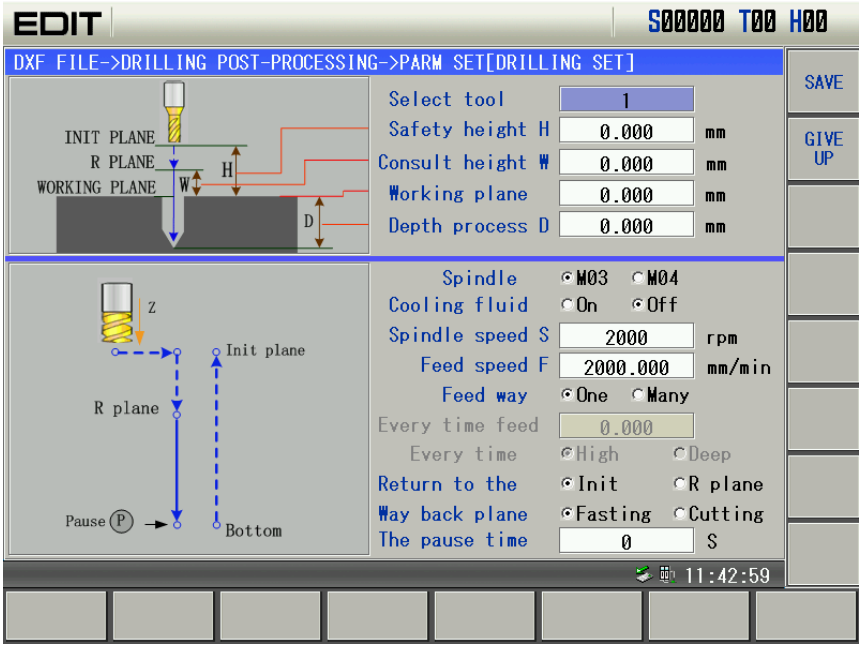
Reference Height: Height of R in fix cycle, cannot be bigger than safty height.

Part Surface: Absolute value of tool setting in Z axis and part surface. Can be input by softkey.

Machineing Depth:Relative value, machining depth from the part surface.

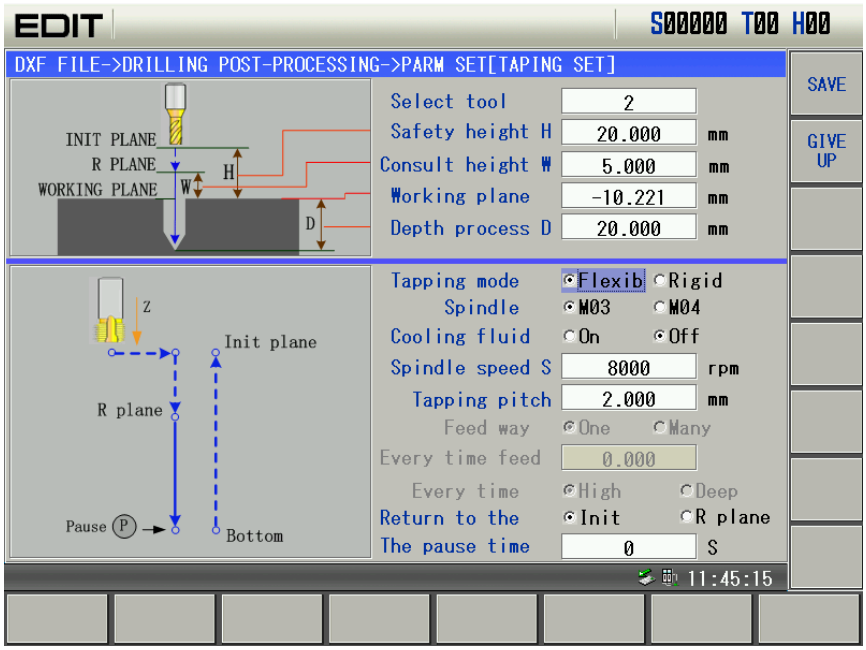
8.2.1 Drill Setting

Press DRILL SET to enter drill setting interface. CNC will automatically generate fixed cycle G code per basic info setting, such as rotate direction, feed mode, hole bottom hold time etc.



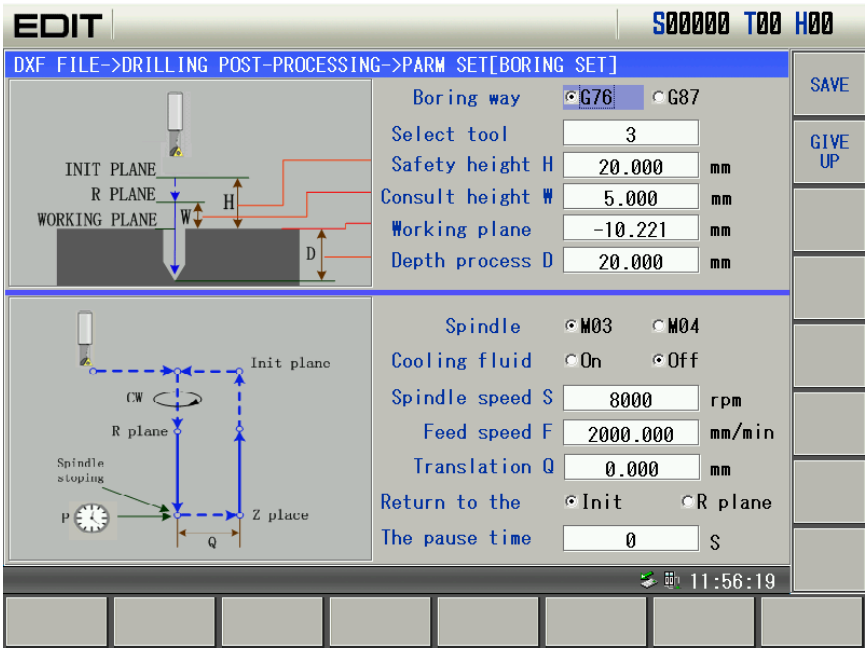
■ Tap Setting

Press TAP SET to enter tap setting interface. CNC will automatically generate fixed cycle G code per the info set by user, say tap mode, spindle direction, feed mode etc.



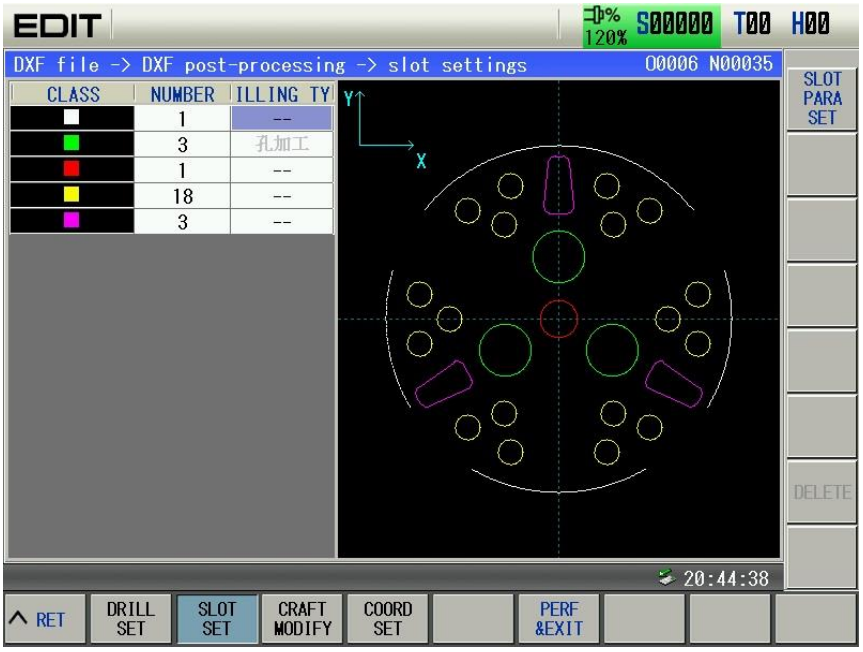
■ Boring Setting

Press BORE SET to enter boring setting interface. CNC will automatically generate fixed cycle G code per basic info set by user, say boring mode etc.



8.2.2 Slot Settings

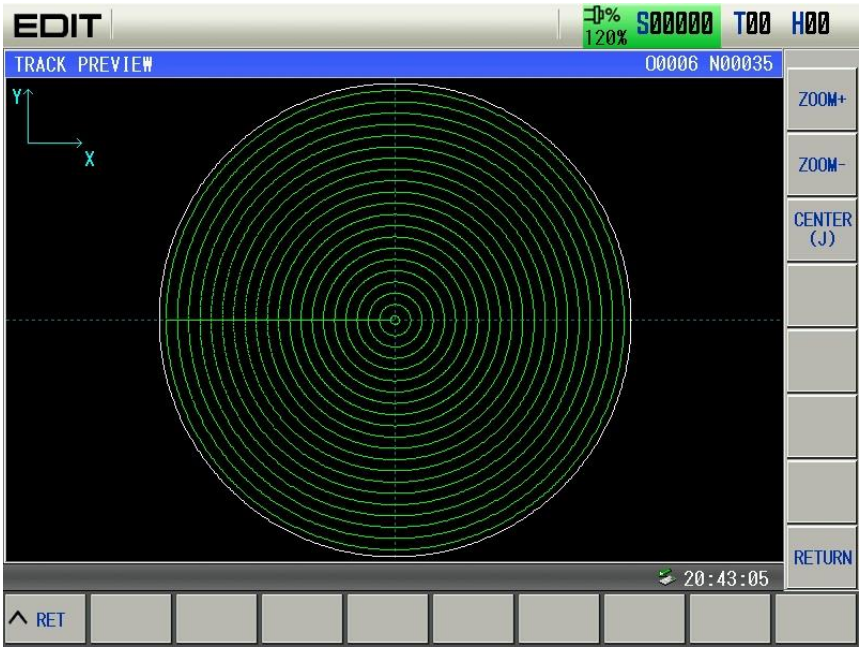
After entering the DXF post-processing page, select the software button to enter the milling groove settings. The system classifies the trajectory according to color, as shown in the following figure:



Slot parameter settings:

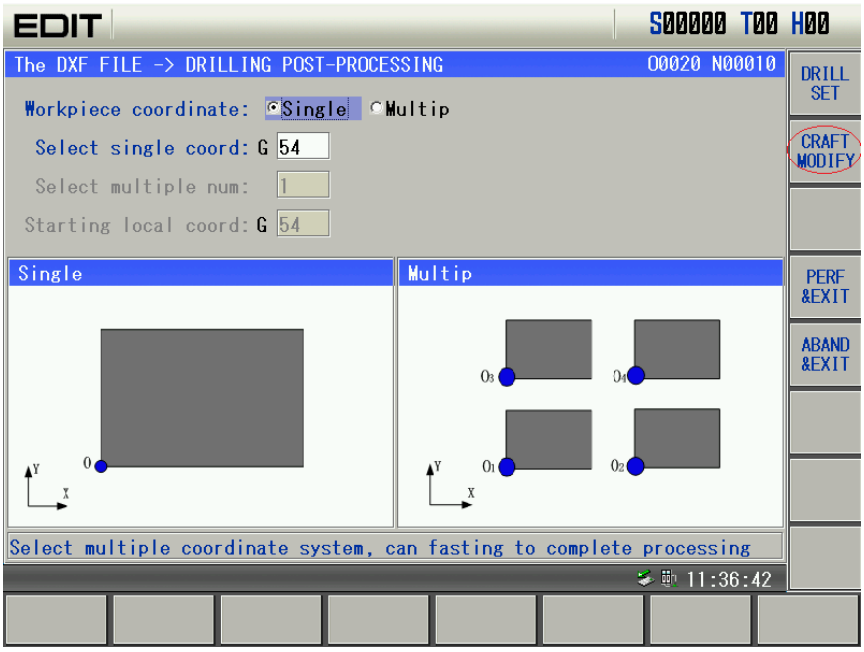
Press the "Slot Parameter Settings" menu to enter the slot parameter settings, and the interface is shown in the following figure

After completing the slot parameter settings, you can view the milling slot trajectory by pressing the "Trajectory Preview" menu, as shown in the following figure:



8.2.3 DXF File Process Modify

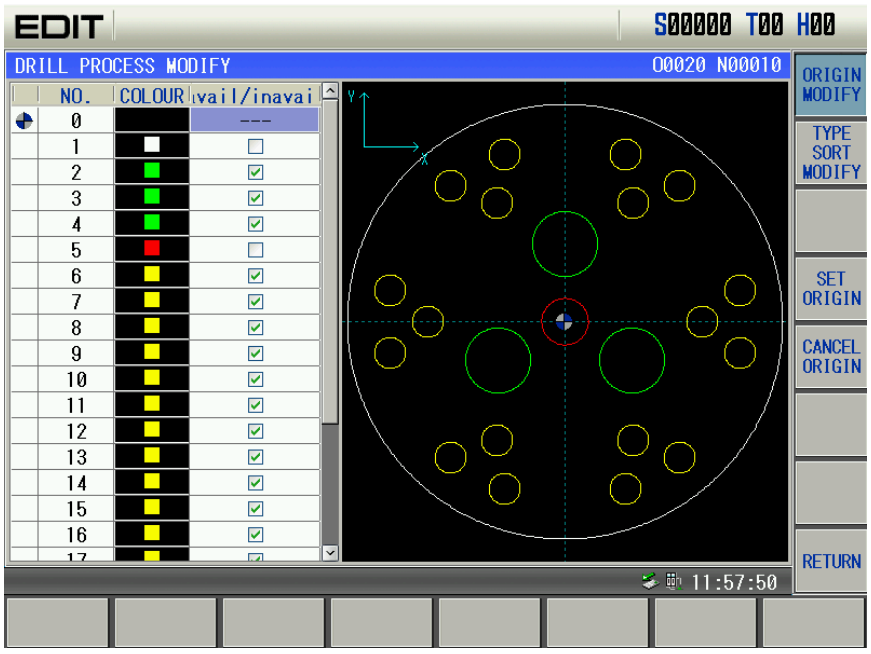
After input post process data of hole drill, the drill process can be modified, mainly to adjust drill initial point, and drill sequence, for easier tool setting and drill sequence optimize



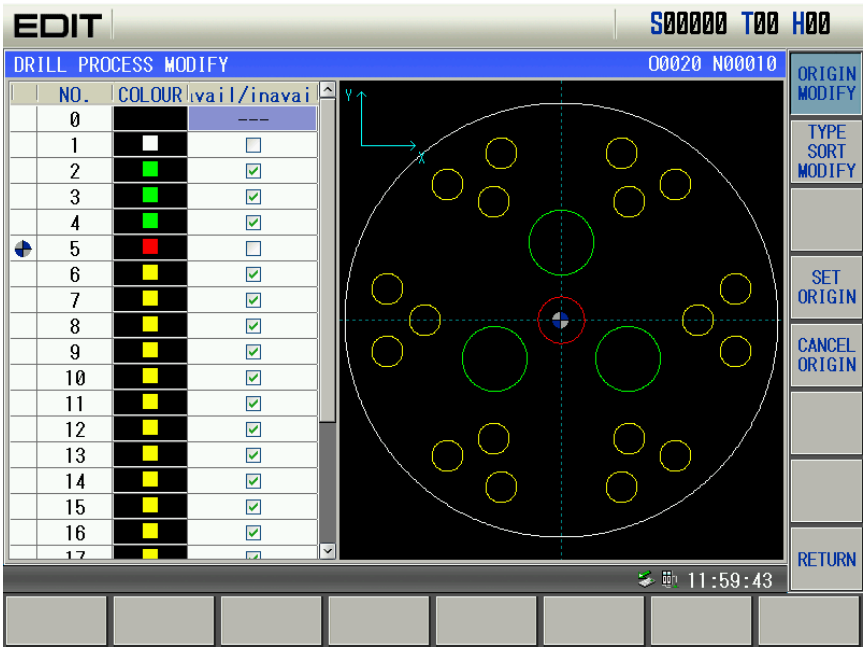
➢ **Initial Point Modify**

User can select one circle center in the file as initial point (point (0,0)in program), all other points can be offset.User can specify the program initial point for easier tool setting.

1：Process without initial point setting

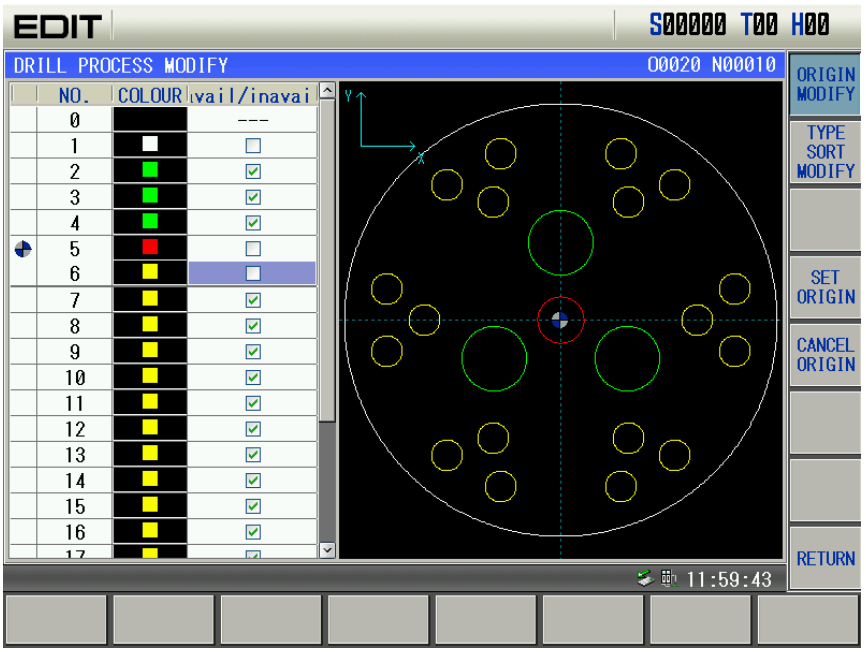


2：Set red circle (no.5) as the initial point



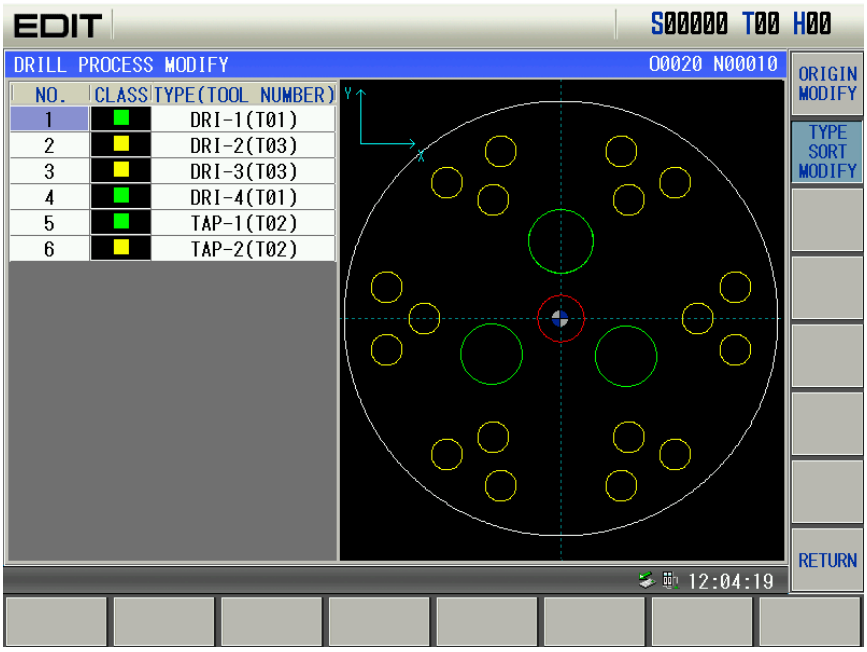
- **Hole set valid/invalid**

If some holes with the same color do not require drill are found after hole setting, these holes can be handled through Hose set valid/invalid function. Method as below: press Input to select, √ means hole set valid, blank means hole set invalid.



- **Type sequence order**

After setting drill parameter,press softkey CRAFT MODIFY ,then TYPE SORT MODIFY to enter interface.



Default Sorting Rule: CNC will default order sequence of the hole process per tool and concentric hole sequence after hole parameter setting.

CNC will read the first layer of drill data and judge if its with the same tool, holes with the same tool will be sorted in one type and be processed together.Then the second layer of drill date. If there is tap or boring in the setting, no matter in which layer, tapping and boring will be executed as the last process.

Example:

Drill Set as following:

1st Layer	2nd Layer	3rd Layer
Drill 1.1 （T03）	Drill1.2 （T06）	Tap1.3 （T08）
Drill 2.1 (T05)	Tap 2.2 （T08）	Drill2.3 （T06）
Drill 3.1 （T03）	Drill3.2 （T05）	Tap3.3 （T07）
Drill 4.1 （T05）	Drill4.2 （T06）	Tap4.2 （T08）

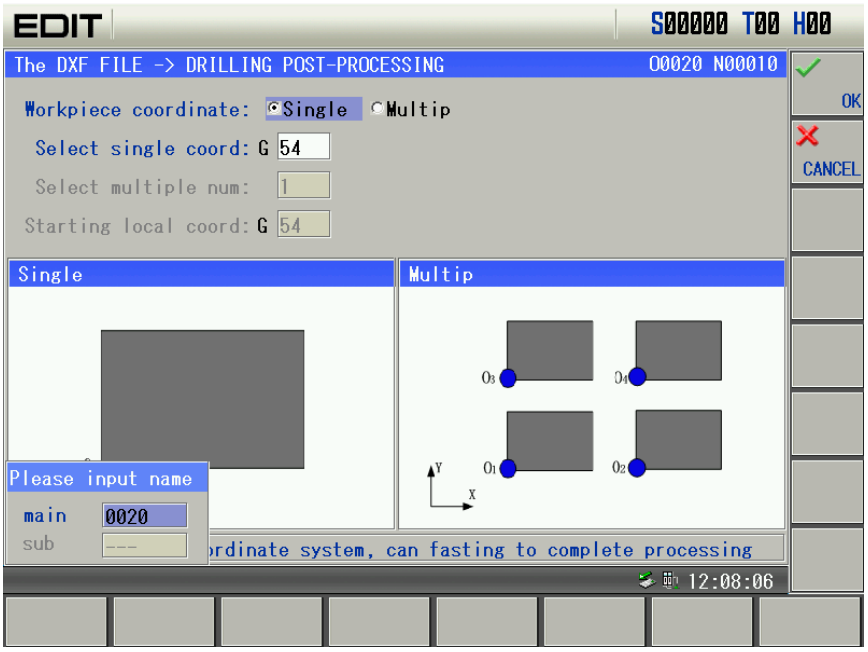
Default process sequence as below:

Drill 1.1 - Drill 3.1-Drill 2.1-Drill 3.1-Drill 3.2-Drill 1.2-drill 2.3-Drill 4.2-Tap 1.3-Tap 2.2-Tap 4.2-Tap 3.3

Note: Sequence of the holes with the same color is handled per principle of proximity. CNC will select the nearest hole per the last point. If the last point is not available, CNC will use the smallest X axis coordinate as the first point. If X axis coordinate is the same, point with the smalles Y axis will be selected as the first point.

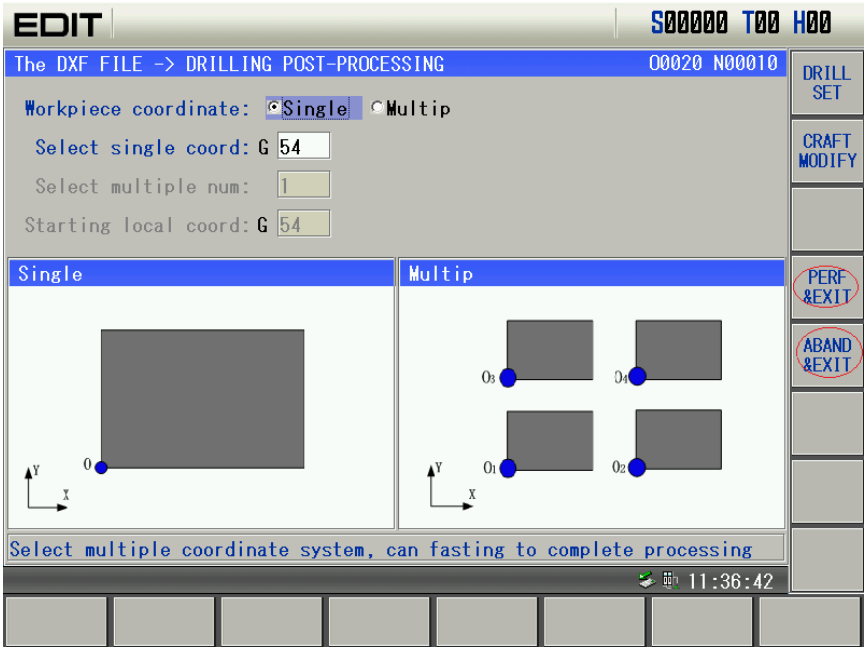
8.2.4 Coordinate System Setting

On the post-processing page of DXF, press "Coordinate System Settings" to enter the coordinate system settings. You can set the single station or multi station coordinate system for processing, as shown in the following figure:

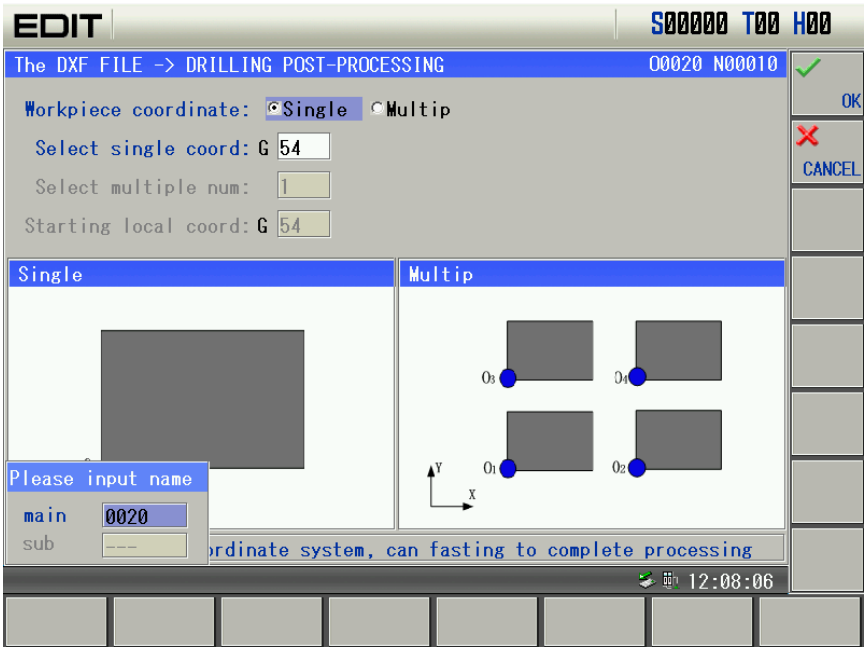


8.3 Program Generating and Parameter Control

After input all info, return to DXF post-process interface.



Press PERF &EXIT,in the dialog box input file name, program will be generated per the DXF post-process info:



If select ABAND &EXIT ,all hole setting will be invalided, program will not be generated.

Notes:

1） Hole setting interface can be entered only under Edit, MPG,and Manual mode

2） For single work station coordinate, a main program file will be generated under the related file name. For multi work station, besides the main program file, a called-sub-program file will be generated also.

3） If input data is non-compliant, when press SAVE Alarm will appear.If select GIVE UP , the current data will be recovered, and esc hole setting.

4） During program generating, if hole info require tool change but 0 appears in program, alarm will appear.

5） In hole setting interface, press DEL TYPE can cancel current hole setting info.

6） Hole setting info will be saved in DXF file, next time when using the same DXF to generate drill program, only press PERF &EXIT , the program will be generated.

7） When generating program, if circles with the same color are concentric, drill process will be executed once only. Repeat process of this type of hole will be avoid.

8） In one hole setting, one one tap or boring can be set.And tap or boring will be defaulted as sequence after drill.

Parameter Control:

300	***	***	***	***	***	***	***	DXFTOOL

.....

Example 1

When auto tool change activated, below program will be generated

G91 G30 Z0.0 P2

(Tool No 02, Drill No,10)

T02 M6 (T for tool change)

G98 G90 X10 Y10 M03 S1000

……
M30

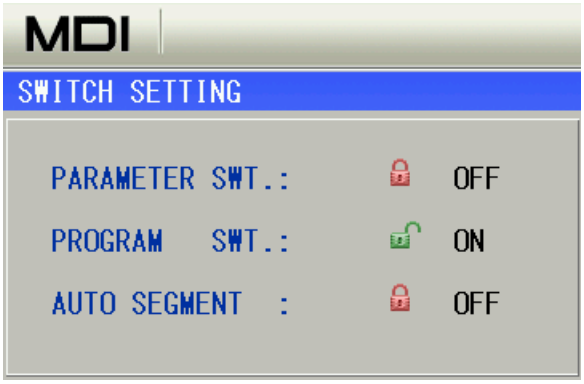
Example 2
When manual tool change activated,below program will be generated
G91 G30 Z0.0 P2
(Tool No.02,Drill No.10)
M05
M00 (Pause here for manual tool change)
……
M30

Chapter 9 Data Setting,Backup And Restore

9.1 Data Setting

9.1.1 Switch Setting

In SWITCH SETTING page, the ON-OFF state of PARM SWT (parameter switch), PROG SWT (program switch), AUTO SEG (auto sequence No.) can be displayed and set, the figure is as follows:



1. Press  key to enter the Setting interface, then press  to enter SWITCH SETTING page.
2. Press  or  key to move the cursor to the item to be set.
3. Press  and  to shift the ON-OFF state, press , "*" being OFF sets the switch to OFF, "*" being ON sets the switch to ON. Only the PARM SWT is set to ON, could the parameter be altered; so are PROG SWT and AUTO SEG.

Note : When parameter switch is shifted from "off" to "on", CNC alarm occurs. Alarm will occur again when parameter switch is shifted from "on" to "off" for the first time. Press , keys together to eliminate the alarm.

9.2 The Password Setting and Alteration

To prevent the part programs, CNC parameters from malignant alteration, this GSK980MD provides an authority setting function that is graded for 5 levels. By descending sequence, they are the 2nd level for machine builder, the 3rd level for equipment management, the 4th level for machine operation, the 5th level for basic operation and the 6th level for operation authority. CNC's current operation level is displayed by "CURRENT LEVEL:_" in the authority level page.

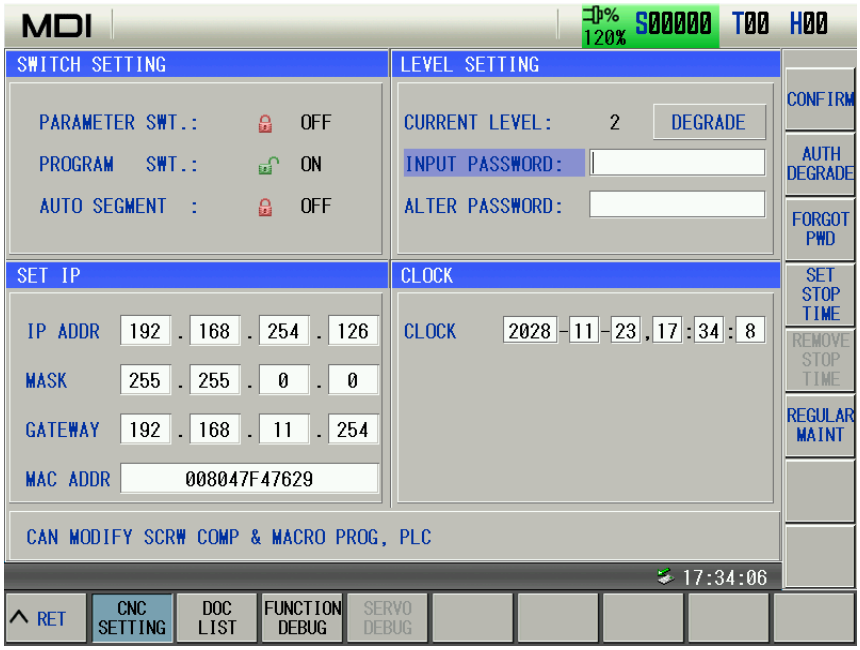
The 2nd level: Modification of the CNC bit parameter, data parameter, pitch data, tool offset data, part program edit (including macro programs), PLC ladder transmission etc. are allowed

The 3rd level: the CNC bit parameter, data parameter, tool offset data, part program edit operations are allowed;

The 4th level: tool offset data (for tool setting), macro variables, part program edit operations are allowed; but the CNC bit parameter, data parameter, pitch data operations are unallowed.

The 5th level: current programs and tool offset data modification are allowed.

The 6th level: Only the machine panel operation is allowed, and the operations of part program edit and selection, the alteration operations of CNC bit parameter, data parameter, pitch data, tool offset data are unallowed.



9.2.1 Entry of Operation Level

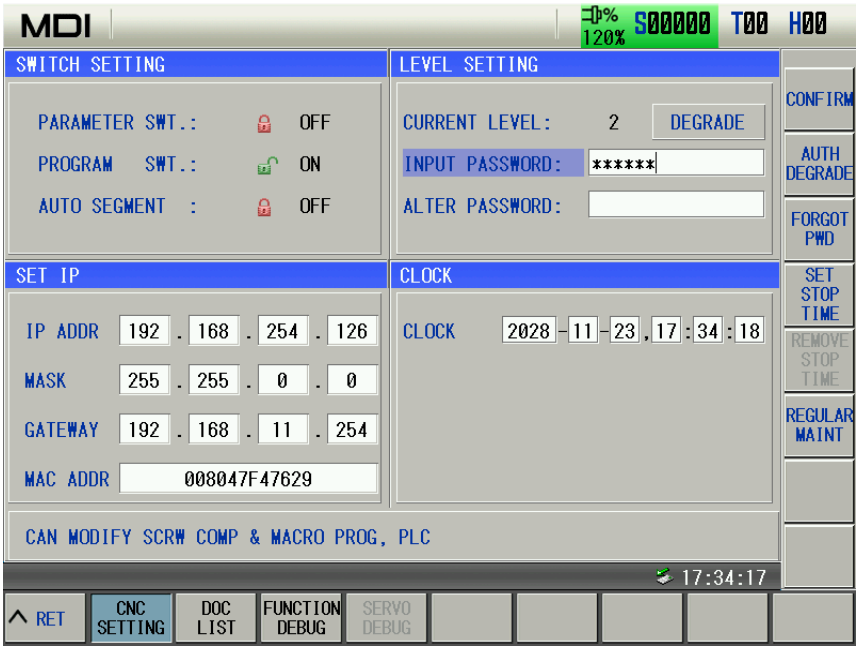
1. After entering the PASSWORD SETTING page, move the cursor to the "INPUT PASSWORD:" line;
2. Key in the password (an "*"sign added each time inputting a character)
3. Press  key to finish the inputting, and it will enter the corresponding password level.

The user can press  to degrade the higher operation level to the lower, but the user must input the password again before degradation.

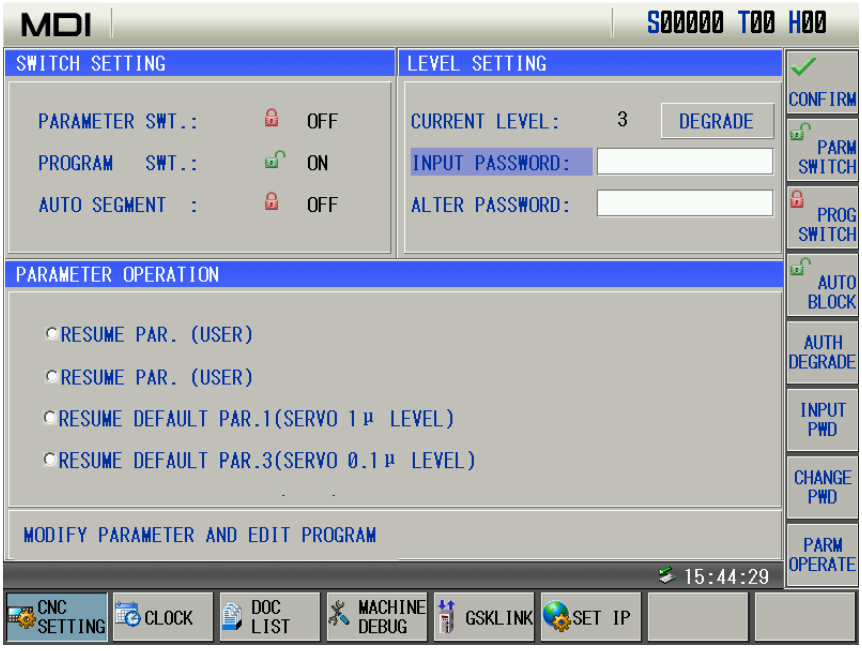
Note: The length of this GSK980MDi Plus system password corresponds to the operation level, which can't be added or decreased by user at will.

Operation level	Password length	Initial password
3rd level	5 digits	12345
4th level	4 digits	1234
5th level	3 digits	123
6th level	None	None

Example: The current CNC level is the 4th level, as the following page shows. The 3rd level password of CNC is 12345, please alter the current level to the 3rd level.



Move the cursor to the “INPUT PASSWORD:” line, key in 12345, then press the  key, the CNC prompts “Modify parameter and edit program”, “Password passed”, and the current level is the 3rd level. The page is as follows:



Note: When current operation authority is lower than or equal to the 3rd level (3rd, 4th, 5th or 6th level), the password level is not changed if repower the CNC system. If previous level is higher than the 3rd level (0, 1st, or 2nd level), it defaults the 3rd level.

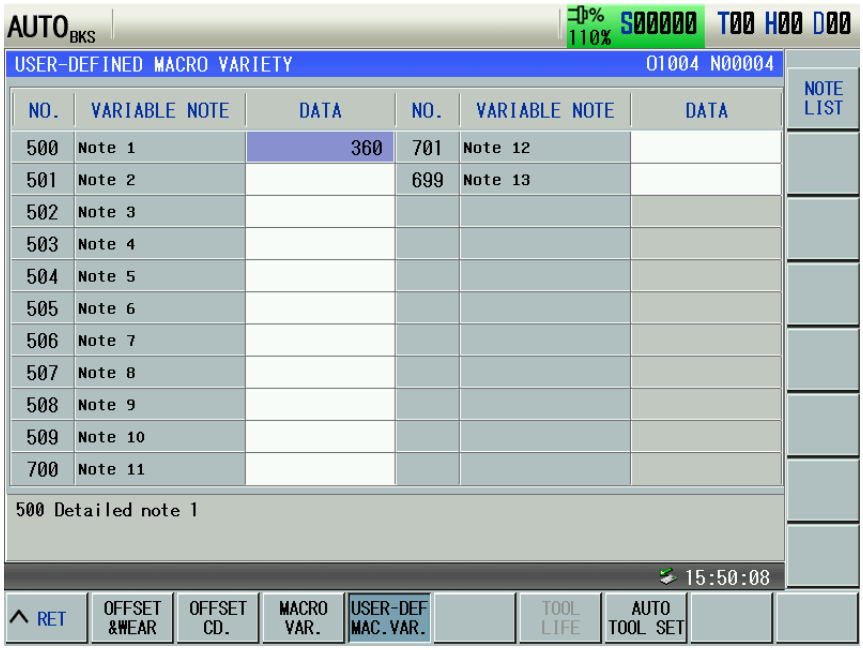
9.3 Macro Variable Note

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.

9.3.1 Macro Variable Note Page

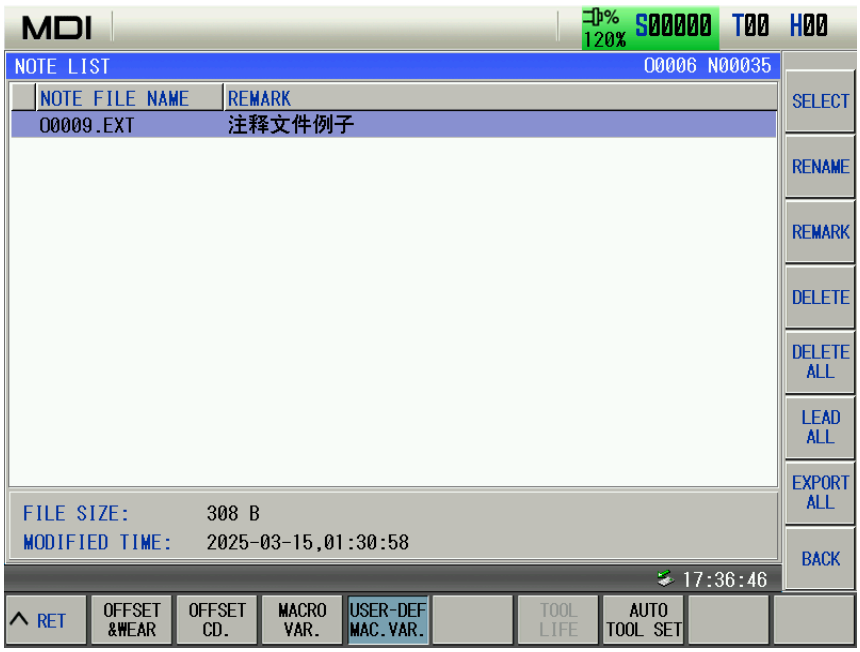
Press the  soft button to enter its corresponding customization page ,after entering the cutter compensation page by controlling the  button, The page is shown below:



The overloaded file data can be set on the “Custom macro variable” page, only 4 and above operation authorities can be performed.

9.3.2 Note List

Press the  button to enter the macro variable note list after entering the macro variable note page.



Function soft button explanations:



soft button: Select the note file to enable the file;



soft button: The file can be renamed for the current file, the note file at current loading can not be modified;



soft button: Modify or add the remark information, the note file at current loading can not be modified;



soft button: Delete the file at the cursor's position; (2 or above authorities are desired)



soft button: Clear the overall annotated files and contents;



soft button: The "EXT files" under the MAROEXT list of the U-disk are led to the system note list;



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 

can be performed in any authority, the 

or 

should be operated with 2 or above levels authorities; other functions should be performed with 3 or more authorities.

Note 2: The 

can be performed in any method, other functions should be operated in the Edit method.

9.3.3 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

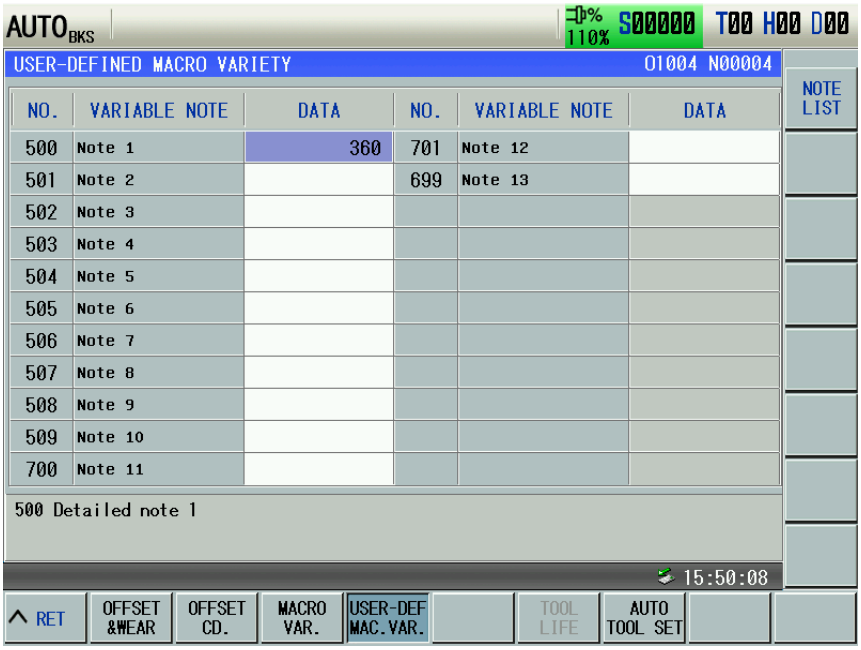
501, Note 2\t Detailed note 2

……

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.

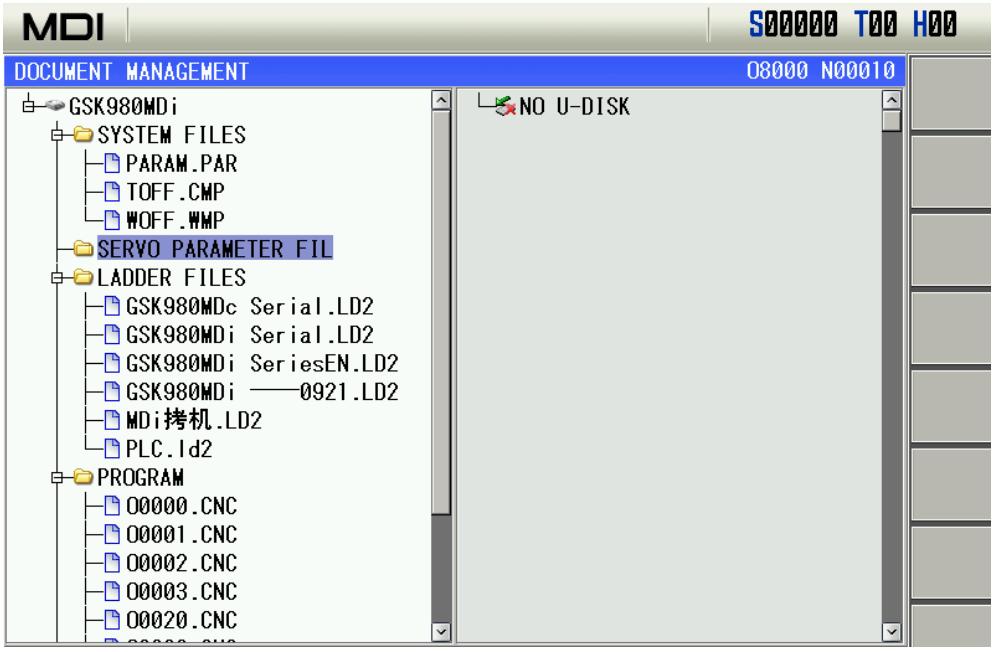
Leading-out operation

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 system root list.

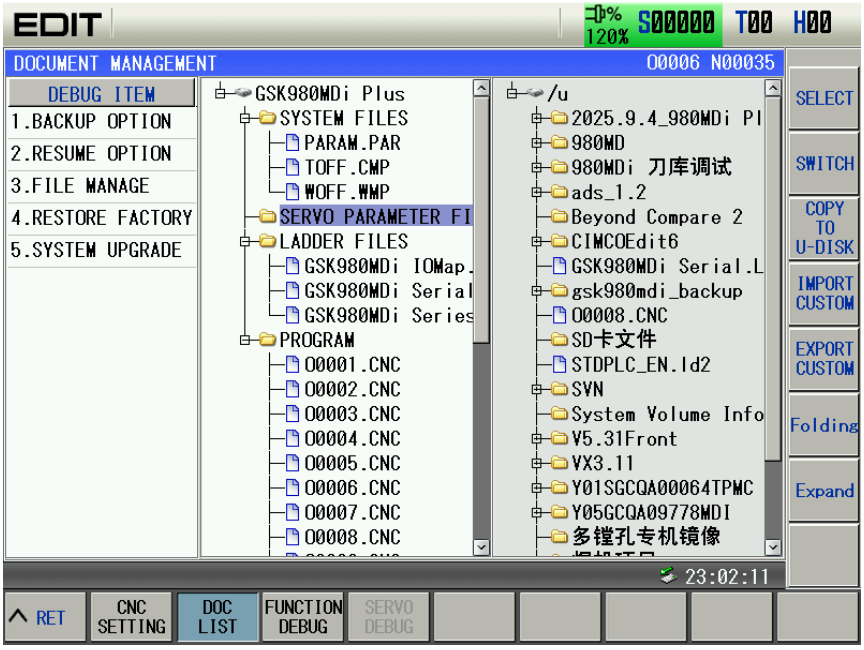
Chapter 10 Document Management

10.1 Document Management Page

Press  to select setting interface, press it repeatedly to enter [Document Management] page as follows:



The system automatically identifies the U disk after it is connected with the system. When it is not identified, the system prompts “Not inserted U”. When it is identified, the system displays file list of the U disk catalog as follows:



The left of the above page displays CNC catalog message and the right displays the U disk catalog message. When the system has not checked the U disk, the right does not display contents.

Notes:

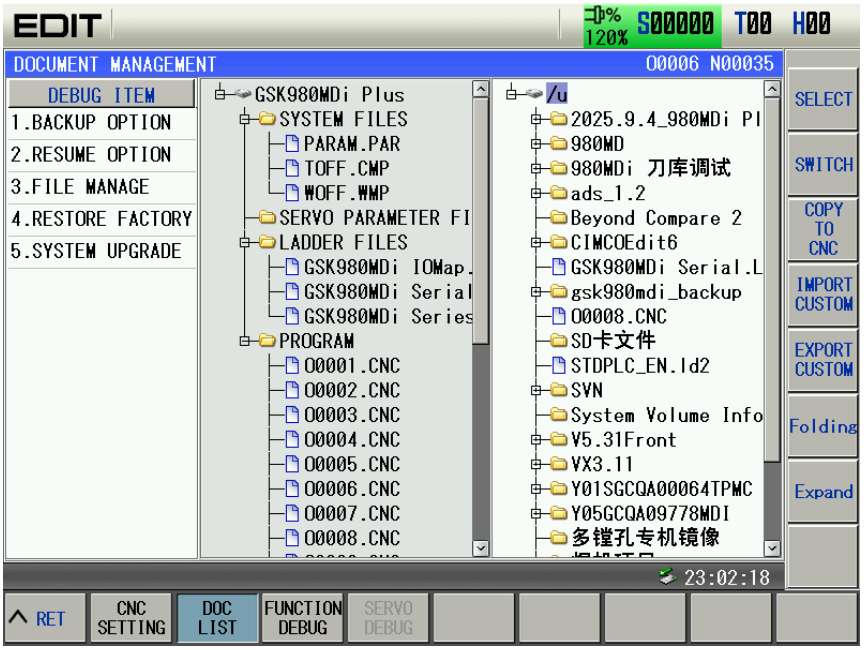
1. The current catalog page only displays catalog message of current open files.
2. The system does not support Chinese Traditional characters temporarily.

Note：File name composed of "O"+"4-bit digit "+".CNC " is considered to CNC format file, such as O1234.CNC.

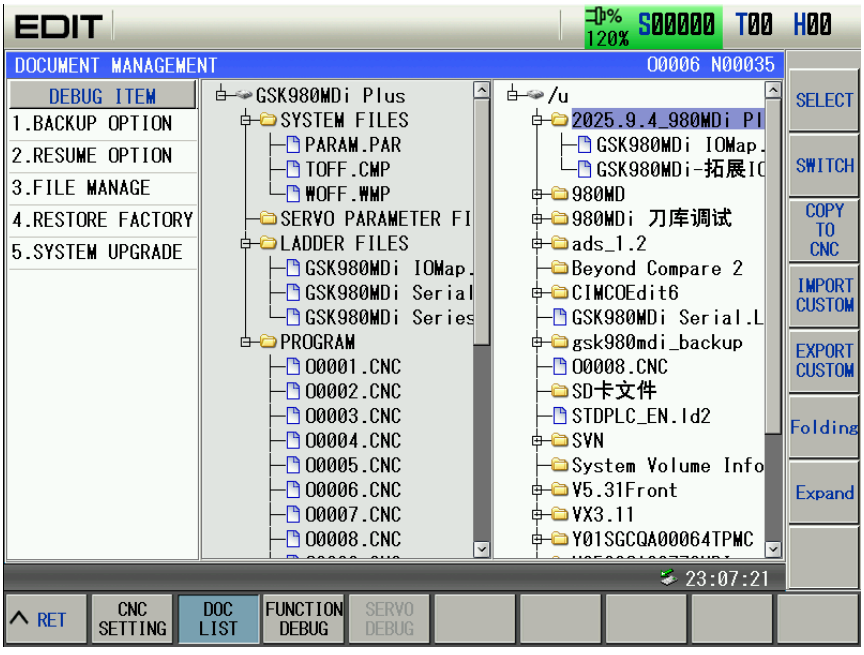
10.2 Often Used File Operation Function Introduction

10.2.1 File Extension and Return

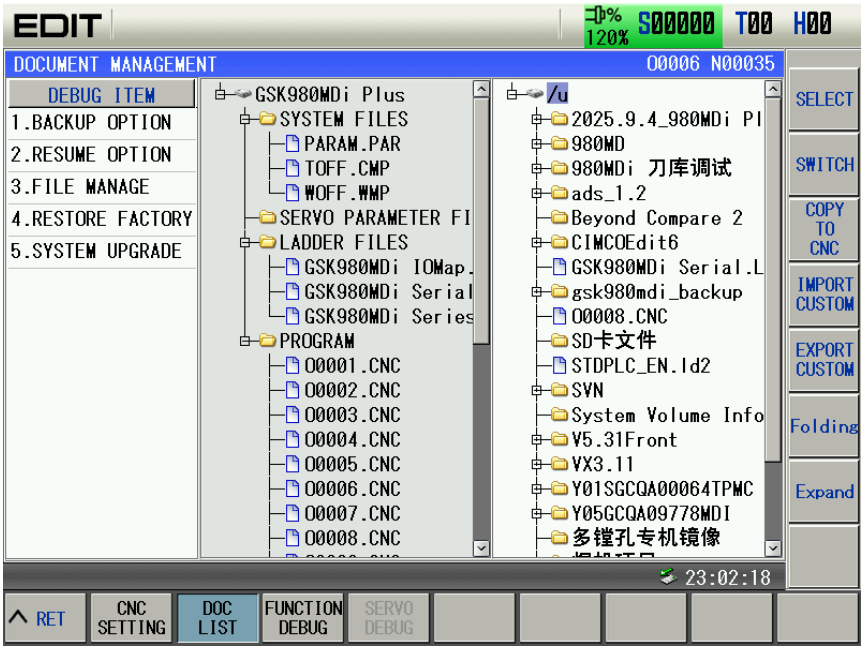
Move the cursor to the required extension file.



Press "EXPAND" to extend the file. The file is in the first line of the directory.



Press “FLODING” to return the previous catalog of the current file.



10.2.2 File Copy (Current Directory in C Disk ←→ Current Directory in U Disk)

In “MDI” mode, press 

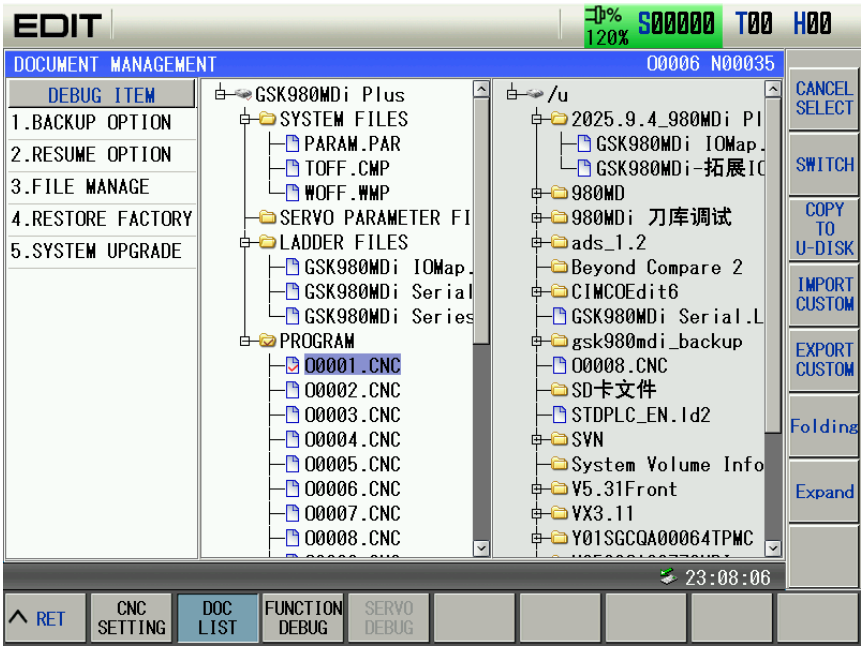
to select 

, and the cursor moves to the required file to copy, press 

or 

to select the file as follows:

① Selected O0001.CNC in the CNC：



② After the file is selected, 

is pressed to copy O0001.CNC to the U disk as follows:

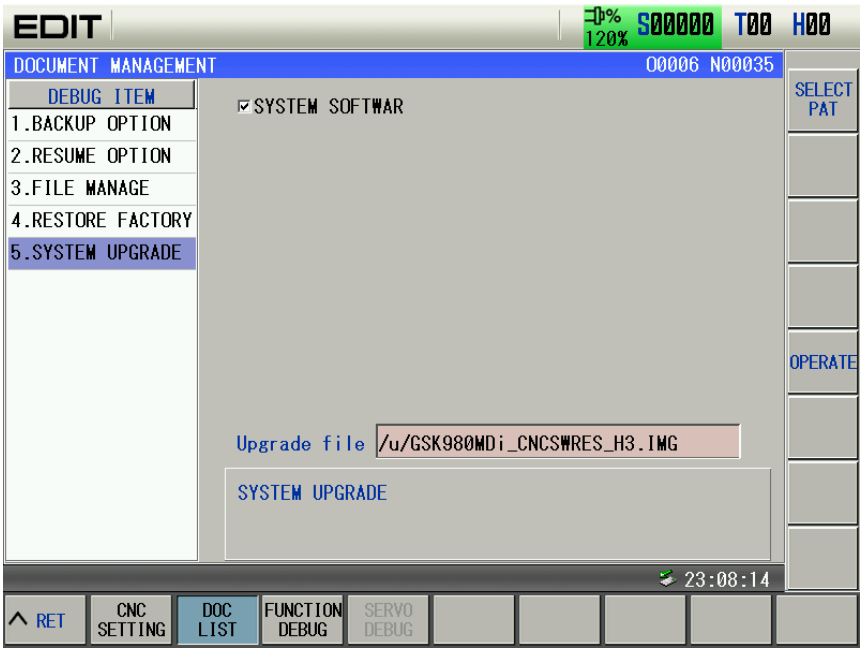
③ Copy methods from U disk to C disk are the same as the above.

10.2.3 System Update

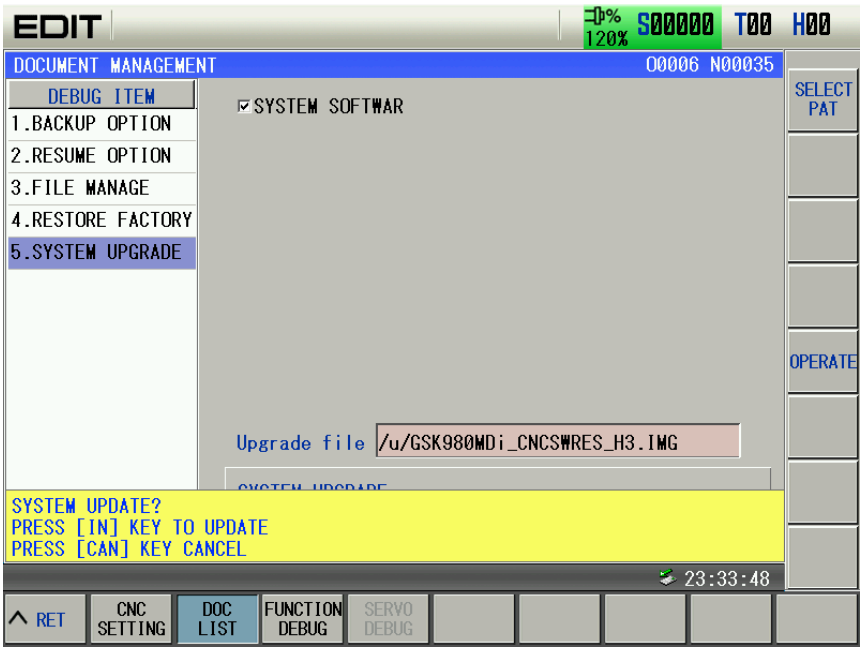
① In MDI mode, the system authority is set to 2-level, 

is pressed to select 

and the system pops up the following:

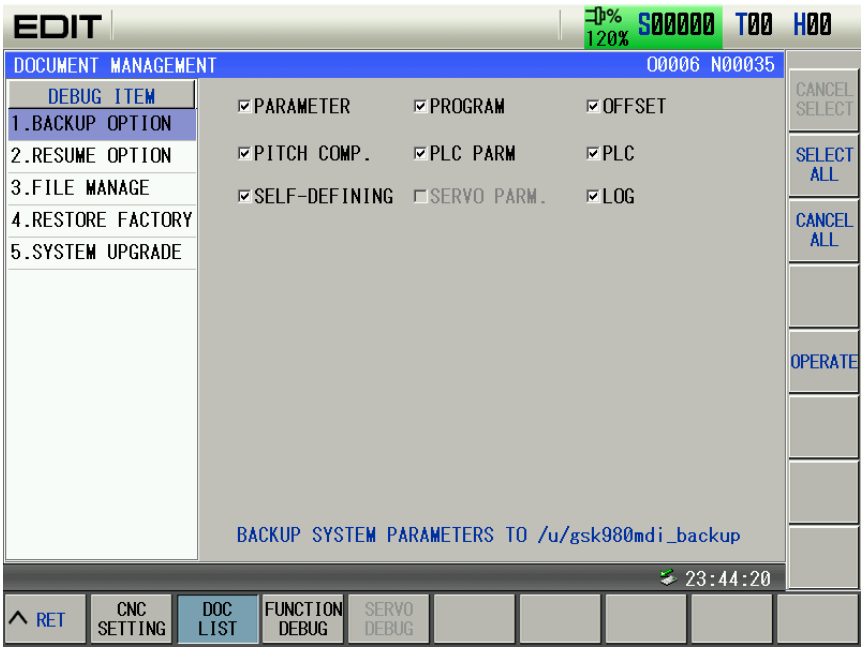


② In the above page, select “OPERATE” to enter the system update page. After the system update software is selected, the [IN] key is pressed to update the system:



Chapter 11 Resume And Backup

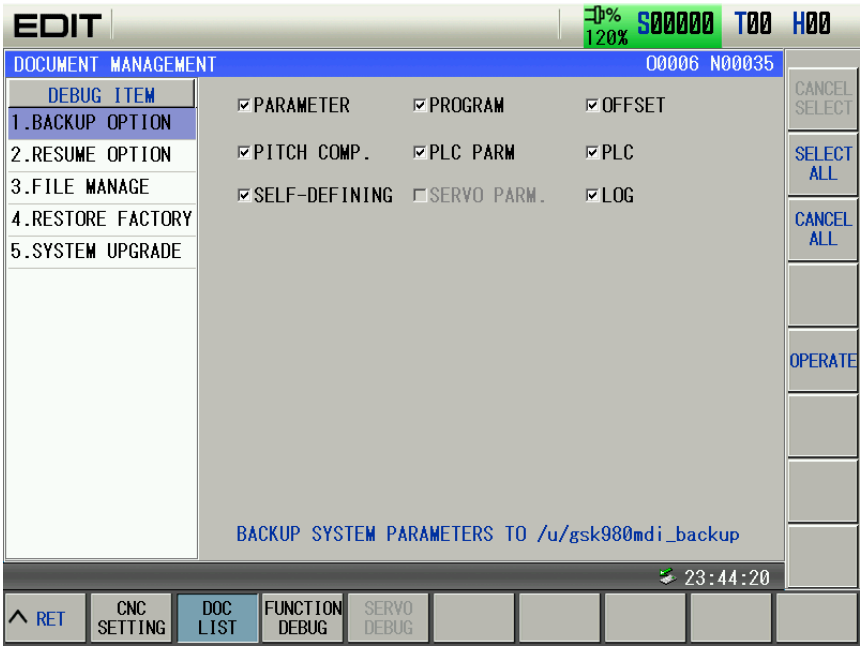
Resume and backup page of GSK980MDi Plus is in the file management of setting page. After the file management soft key is selected, backup/resume soft key is selected to enter the following page. In the page, MDI mode is selected to execute communication between CNC and U disk, and the system upgrading operation. Its transmission speed is much faster than traditional serial communication speed, greatly increases the efficiency of file transmission. More over, USB is easy to carry, to use and it supports hot plugging, plug and play at once.



11.1 Backup Operation Example

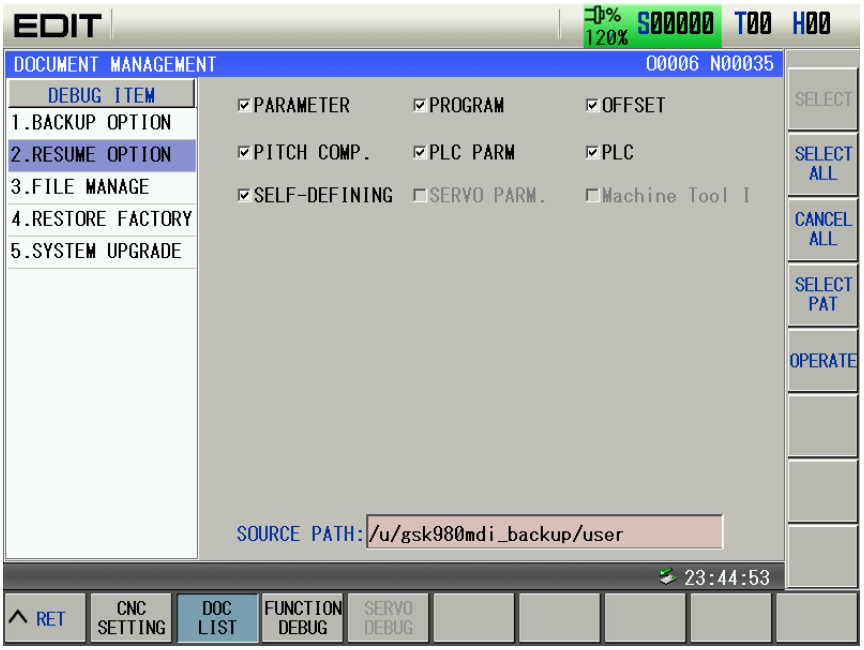
1. Backup log （take an example of log）;
2. Press 

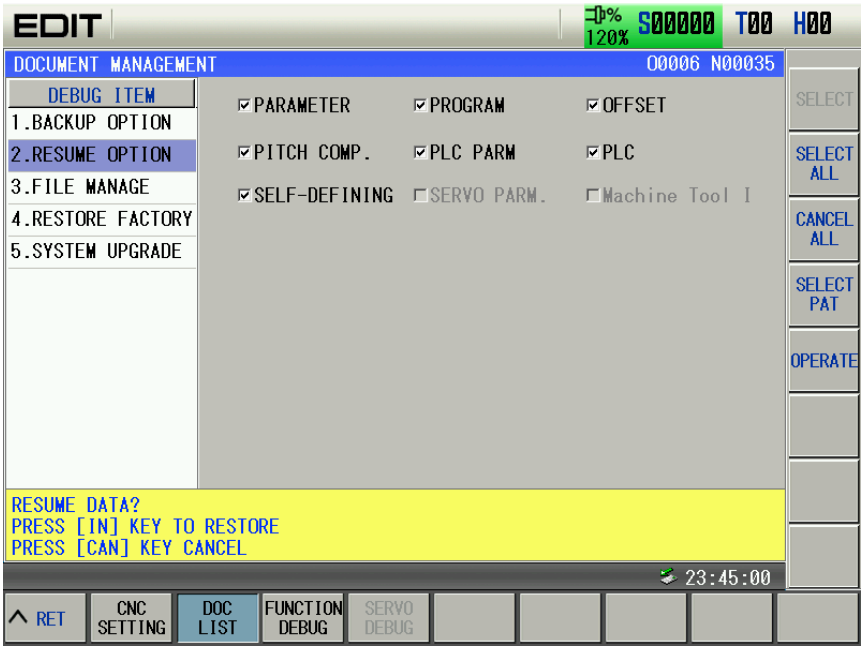
to enter the page, and the user can execute the following operations including canceling the selection, all selection and so on;
3. In MDI mode, press [IN] to select the log, and then press [OUT] to execute backup to successfully backup the log to the U disk. When the backup fails, the system prompts the fail as follows:



11.2 Resume Operation Example

1. In BACKUP page, press ↑ or ↓ to switch backup and resume;





2. After the system is switched to the resume page, a part program is selected by soft key;
3. In MDI mode, press soft keys or press [OUT] and the system prompts whether it executes resume; press [IN] to execute resume operation.

When the resume operation fails, the system prompts the failure.

Chapter 12 Waveform Analysis

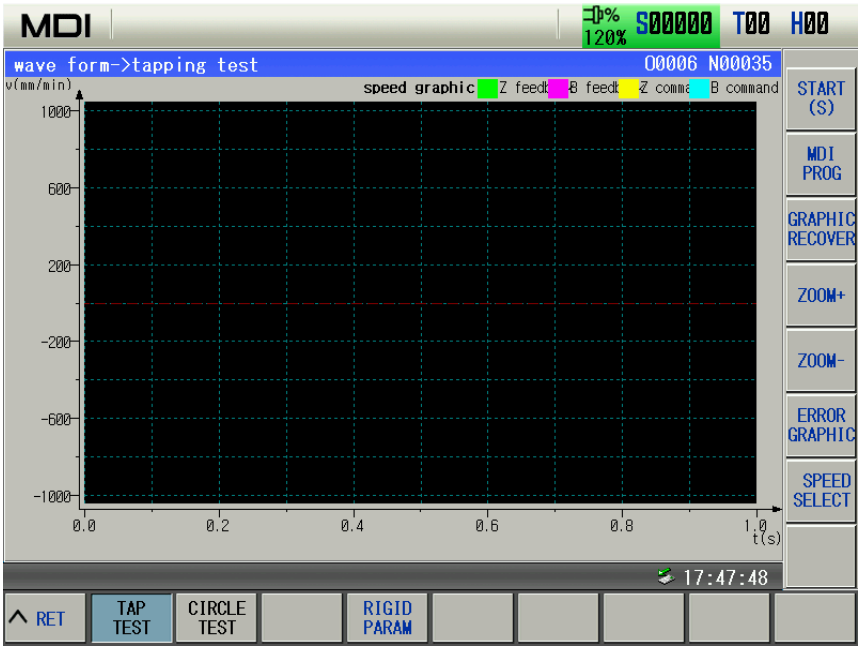
12.1 Rigid Tapping Test

Execute the rigid tapping test program in MDI mode and analyze the servo parameter settings through feedback data from the motor encoder.

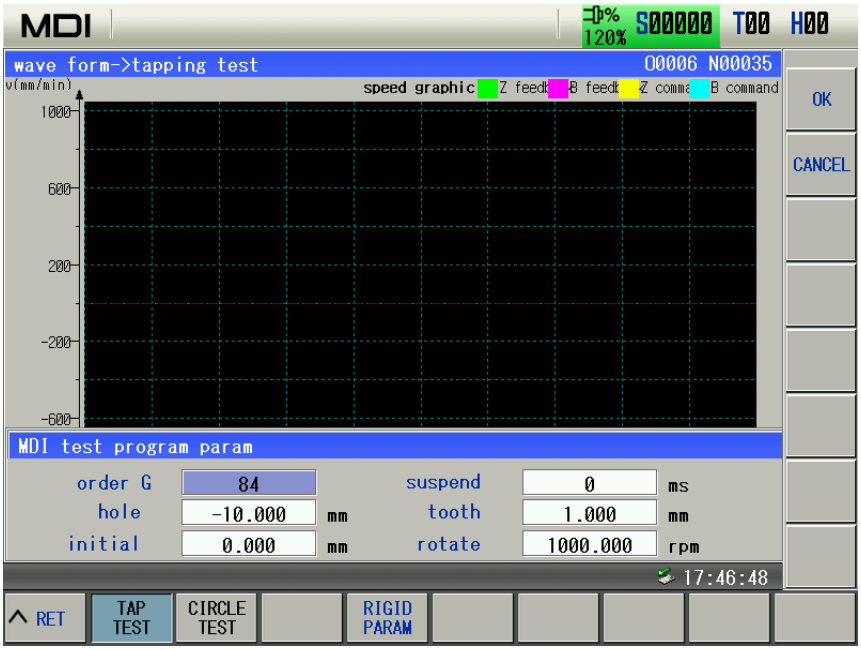
12.1.1 Generate Test Program

1） select 

, Place the system in MDI mode，enter to DIAG page，In the collection of graphic pages, switch the page to the TAP TEST page:



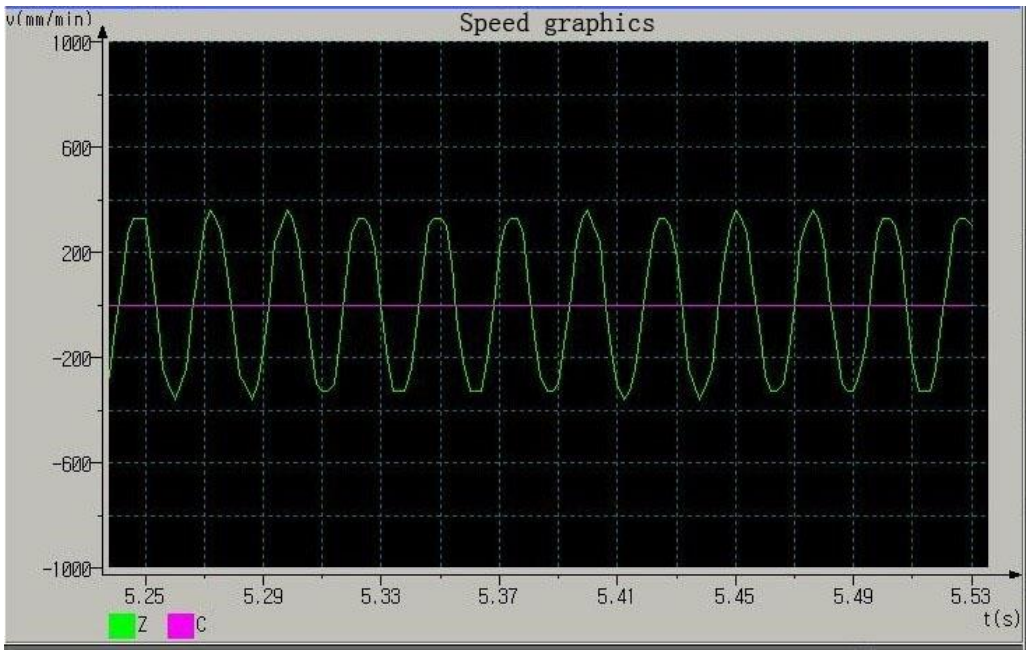
2）Select “MDI” to enter the test program parameter settings page, where you can set and modify the test program parameters as follows:



After setting the relevant program parameters, select OK to automatically generate the program on the MDI page, and select Cancel to not generate the program.

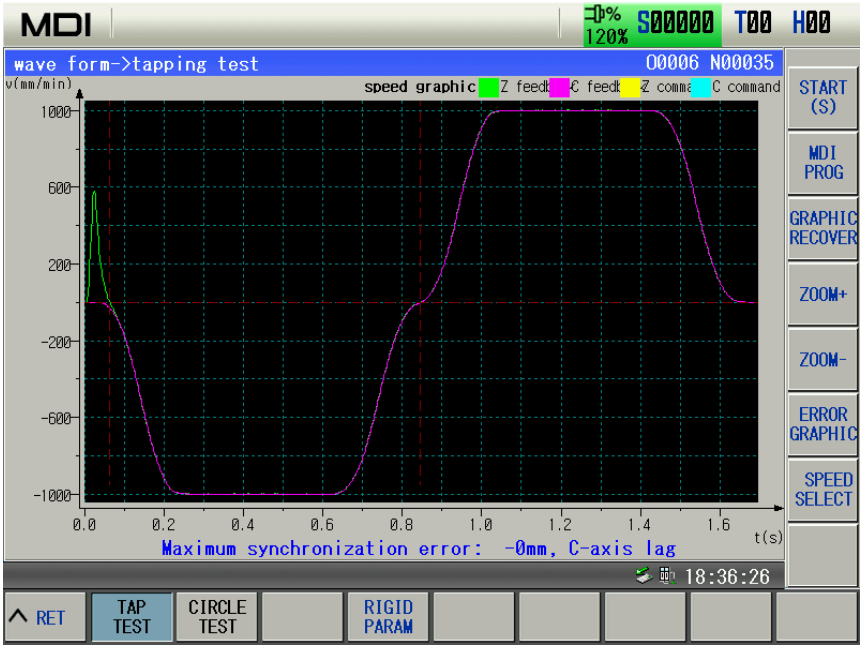
12.1.2 Sampling

After generating the MDI program, you can choose to enter the sampling mode after starting, select the loop start program to run, and start sampling:



After the program ends, the sampling automatically ends and generates a velocity waveform (selecting "stop" can also stop sampling).

12.1.3 Velocity Waveform



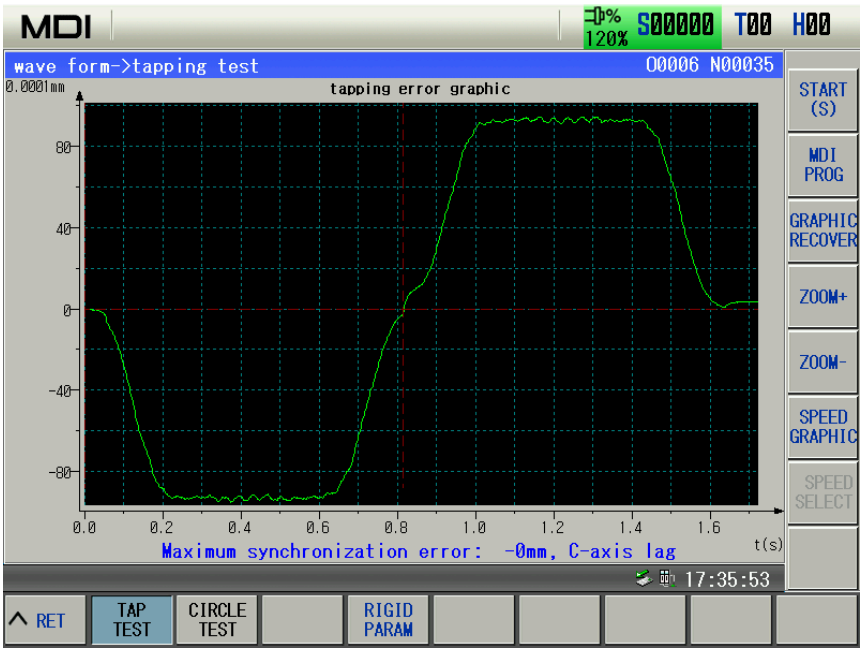
As shown in the figure, in the v-t image, the horizontal axis t is the time axis, measured in seconds, and the vertical axis v is the velocity axis, measured in millimeters per minute, representing the Z-axis waveform image and the C-axis waveform image.

- Analysis results

  After the sampling is completed, the v-t waveform is automatically generated and the analysis results are provided.

12.1.4 Error Waveform

After the sampling is completed, selecting "Error Image" can generate an error image and perform error analysis on the sampled data:



As shown in the figure, in the e-t image, the horizontal axis t is the time axis, measured in seconds, and the vertical axis e is the error axis, representing the tracking error of the Z-axis relative to the C-axis, measured in millimeters.

12.2 Circularity Test

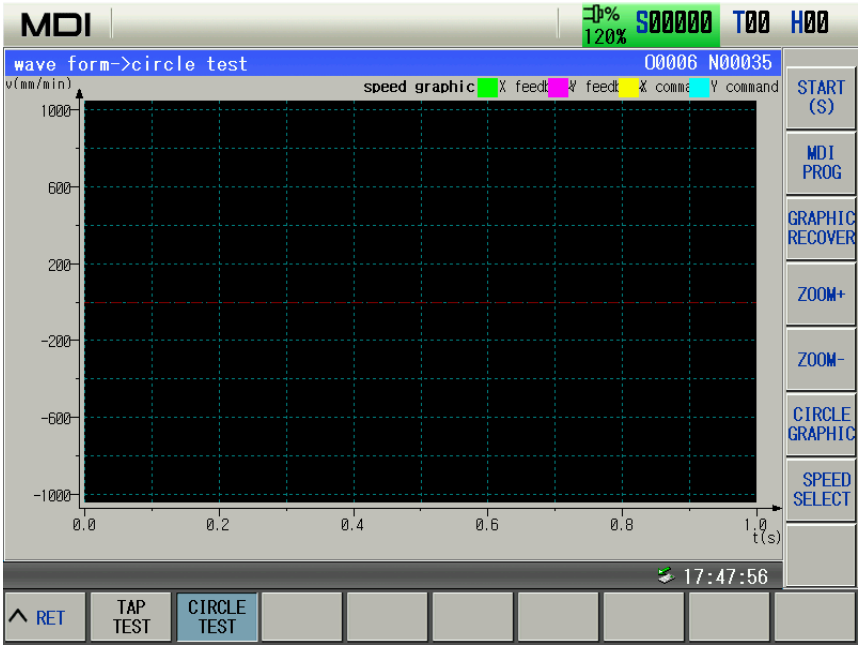
Execute roundness testing program in MDI mode, monitor motor encoder feedback data, and analyze the motion performance of circular axis servo parameters.

Attention: Before testing, temporarily set the reverse clearance between the two feed axes to 0, because the feedback signal detected by the system is read from the motor. Therefore, after setting the reverse clearance, there will be a significant jump in the detection trajectory when the motor trajectory exceeds the limit, which is not conducive to observation.

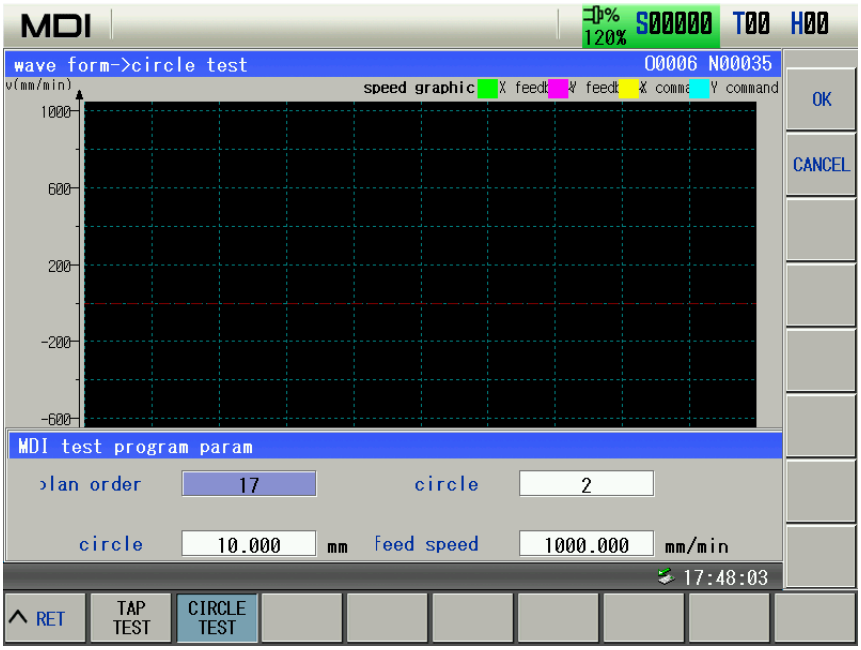
12.2.1 Generate Test Program

1） select 

, Place the system in MDI mode，enter to DIAG page，In the collection of graphic pages, switch the page to the CIRCLE TEST page:



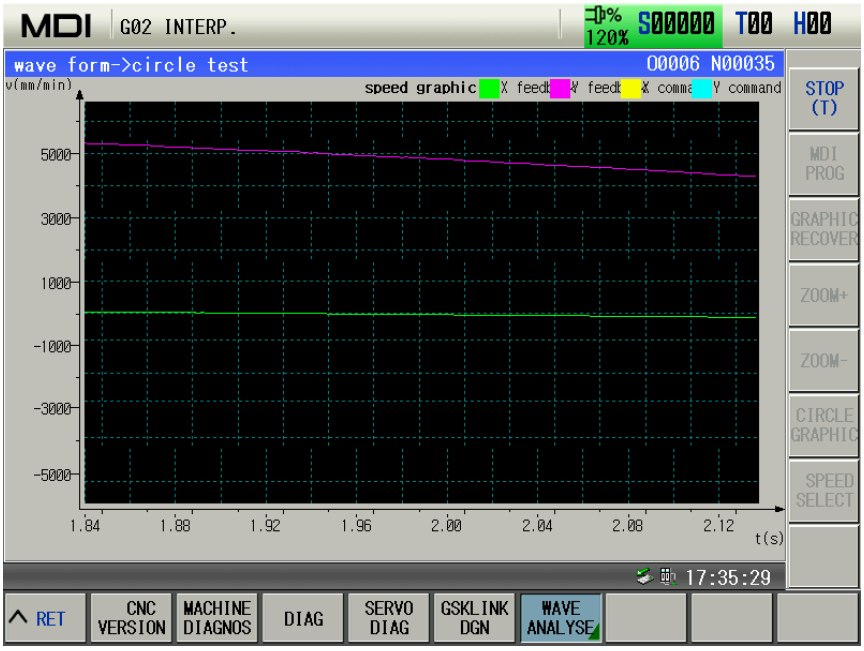
2） Select “MDI” to enter the MDI program page and set and modify the relevant program parameters for the round machining program, as follows:



After setting the relevant program parameters, select OK to automatically generate the program on the MDI page, and select Cancel to not generate the program.

12.2.2 Sampling

After generating the MDI program, you can choose to enter the sampling mode after starting, select the loop start program to run, and start sampling:



After the program ends, the sampling automatically ends and generates a velocity waveform (selecting "stop" can also stop sampling).

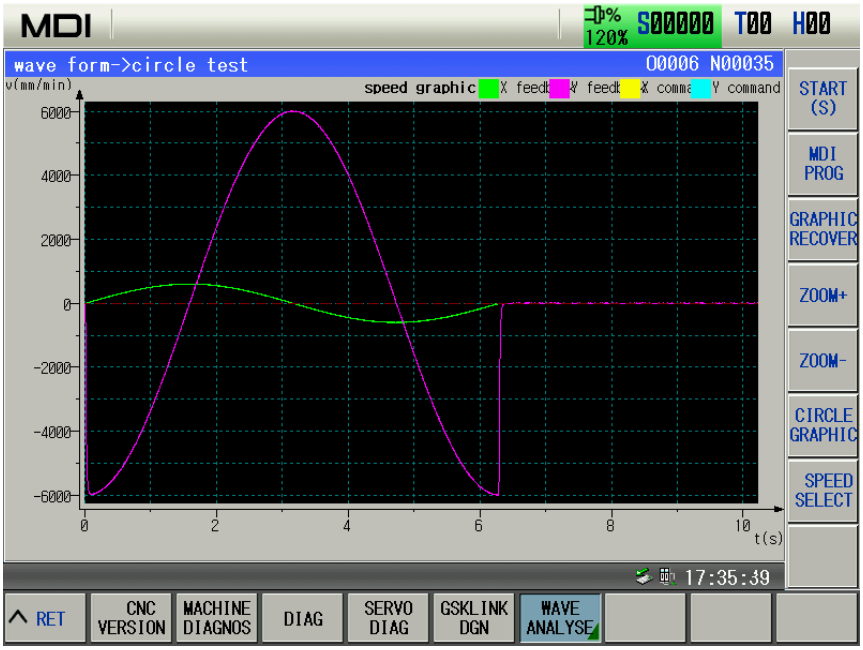
12.2.3 Velocity Waveform

As shown in the figure, in the v-t image, the horizontal axis t is the time axis, measured in seconds, and

the vertical axis v is the velocity axis, measured in millimeters per minute, representing the Z-axis waveform image and the Y-axis waveform image.

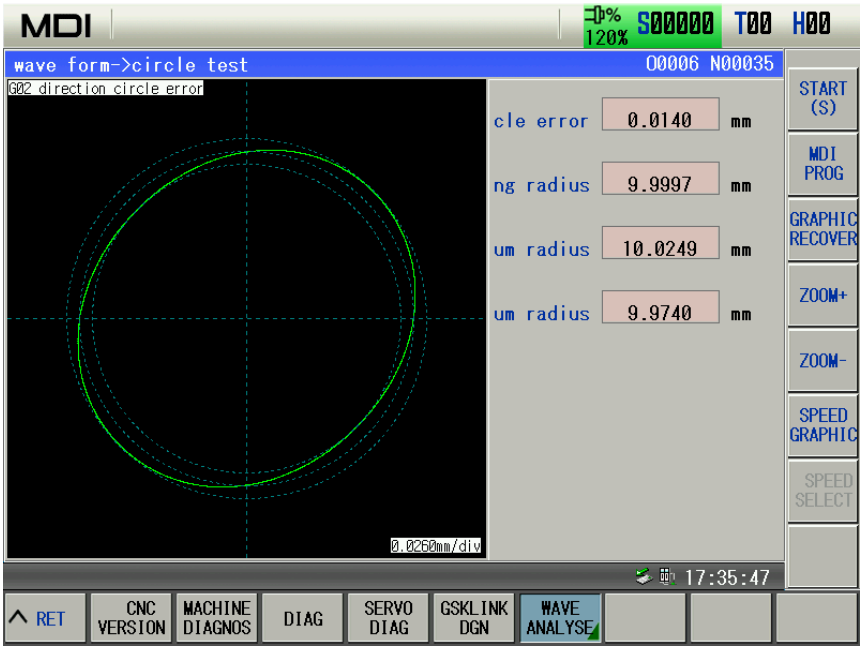
- **Speed waveform filtering**

  Select 'Speed Selection' to filter and compare the feedback speed waveform and command speed waveform.



12.2.4 Roundness Error

After the sampling is completed, selecting "Roundness Image" can generate a roundness image and perform roundness error analysis on the sampled data:



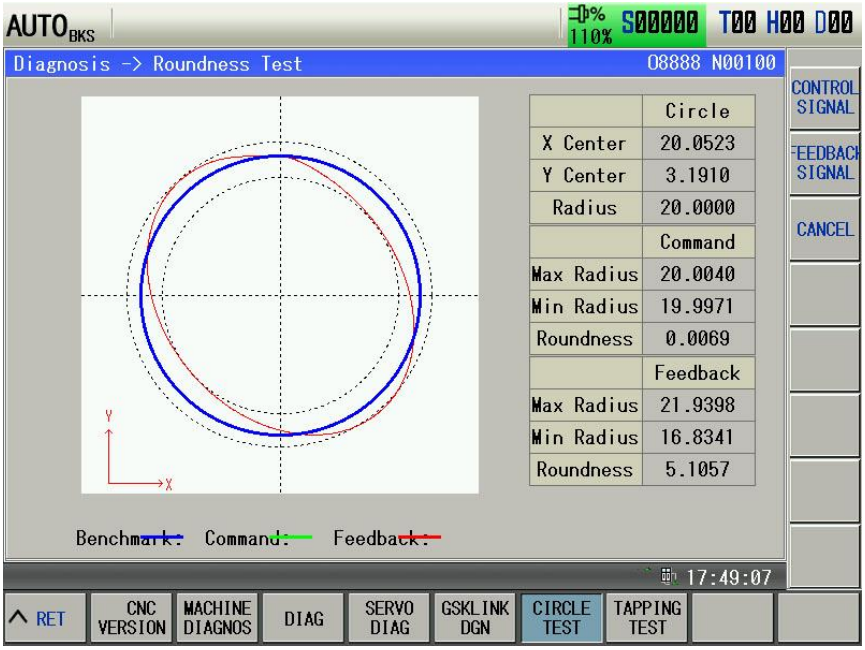
Related error parameters:

1) Roundness error: represents the degree of deviation of the sampled value from the theoretical value. The calculation formula is, where R is the programming radius.

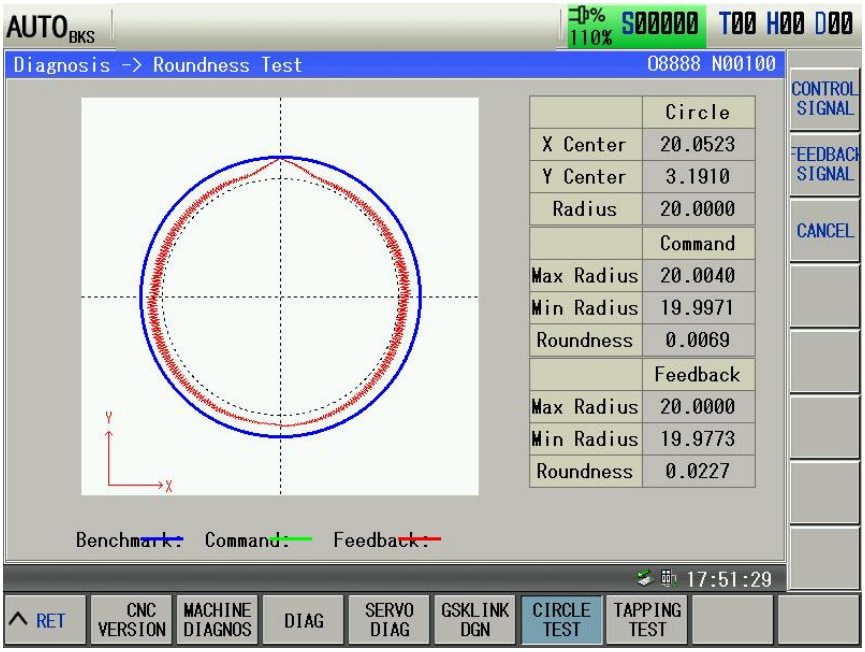
2) Fitting radius: used to represent the actual average radius of the sampled data. Based on the radius of the sampling points, calculate the average radius value of the fitting circle.
3) Maximum radius: Count the radii of all sampling points to obtain the maximum radius value.
4) Minimum radius: Count the radii of all sampling points to obtain the minimum radius value.

Analysis of roundness error image

1) G02 arc interpolation, when the servo position ring parameters are not coordinated, the roundness error contour appears elliptical:



2) G02 arc interpolation, when the servo speed loop parameters are not coordinated, the roundness error contour image shakes significantly, and even the contour may not be closed:



3) The feedback speed image also shakes greatly.

Chapter 13 Communication

13.1 Brief About GSK980MDi Plus Communication Software GSKComm

As CNC configuration managing software, GSKComm can realize files transfer and downloading between CNC and PC with high efficiency and reliability.

Below is the brief introduction about GSKComm communication and setting. For details please refer to GSKComm user manual.

13.1.1 PC System Required

Hardware: RS232 or LAN network interface, series communication cable (three core) or standard LAN cable.

Software: Microsoft Windows 98/2000/XP/2003.

13.1.2 Software Interface

Interface of GSKComm first run.

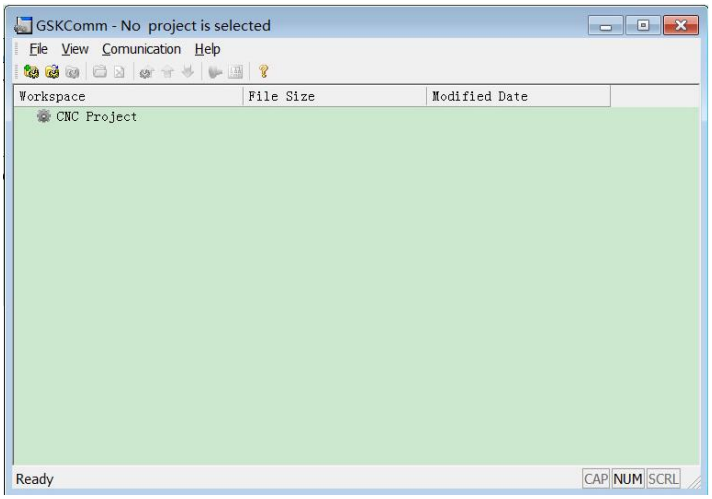
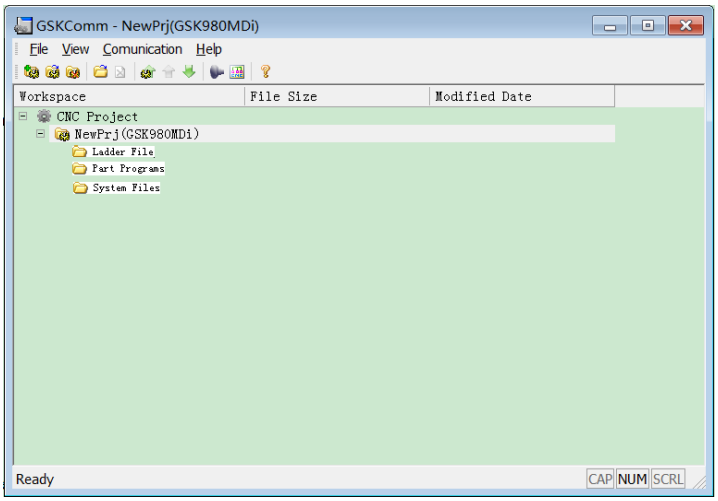


Fig. 13-1 file download（PC→CNC）interface

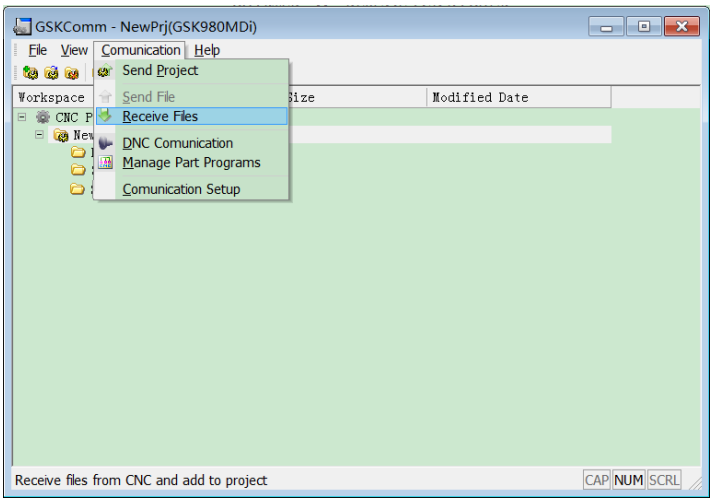
GSKComm software takes an administrator as a unit to complete management. When it run firstly, it does not upload project files, at the moment, the user should create a project program or upload project to execute the communication operation. The display is shown below after the project is created:



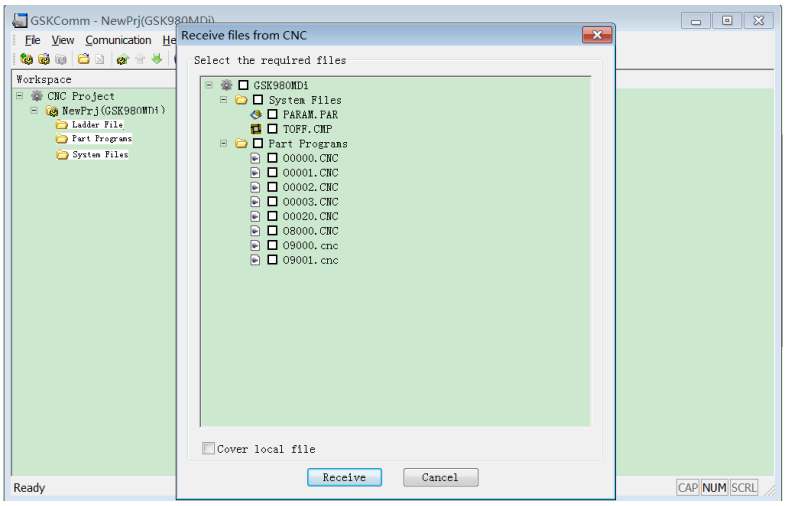
Then files can be saved from CNC to current project, or files in current project can be sent to CNC.

13.1.3 Transfer File (CNC-PC)

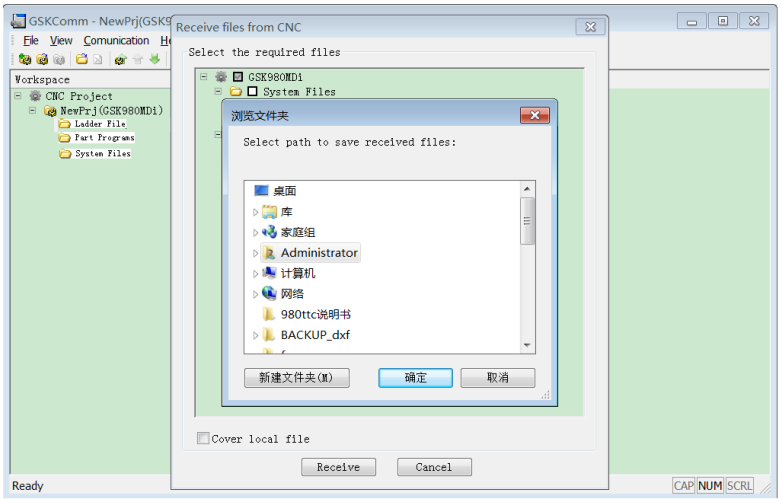
Select current project, click Communication menu “transfer file from CNC ”.



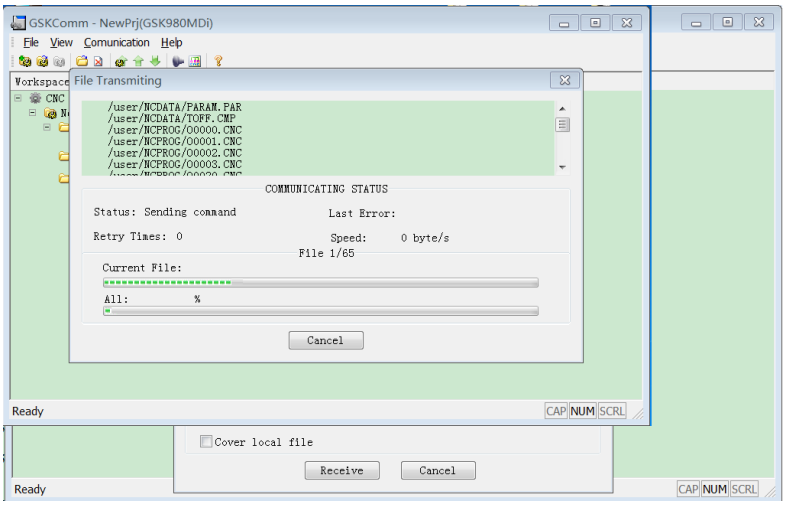
Select the file in the injected dialog box,it could be part program, tool set parameter (toff.cmp)Thread compensation (woff.wmp) and Parameter(param.pax).



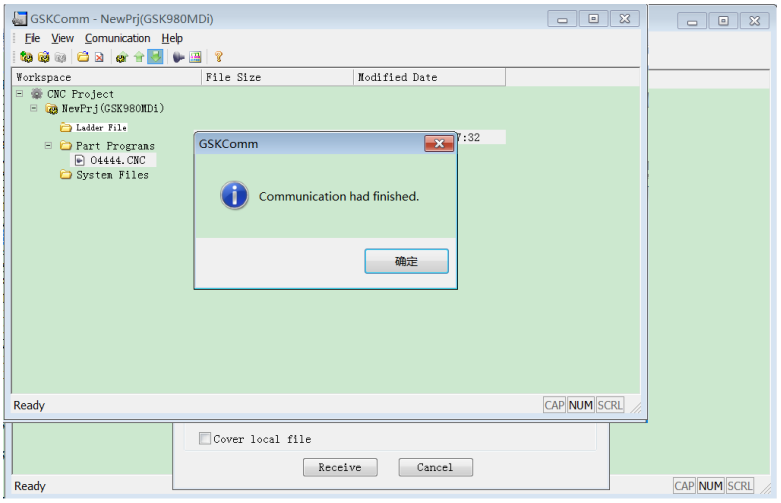
Select the file,press “start receiving”,select the saving path in the dialog box.

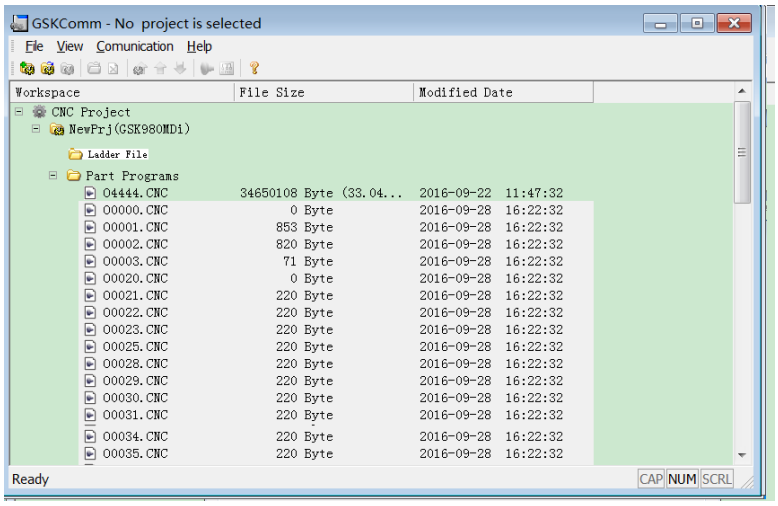


Select save path, press “OK” button,start transferring files from CNC to PC.



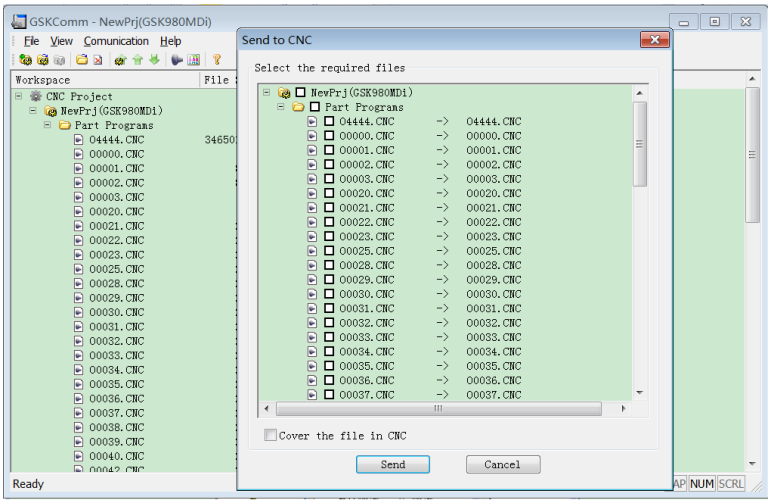
File transfer completed.





13.1.4 Send File PC-CNC

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).

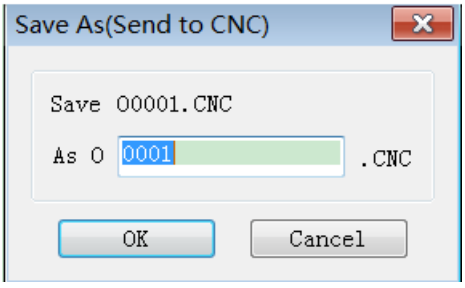


Fig. A

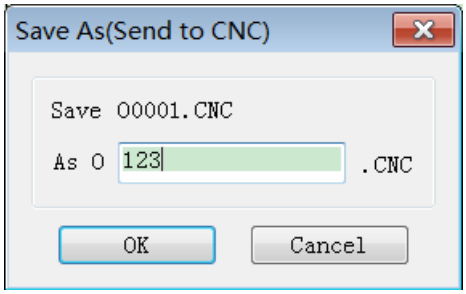
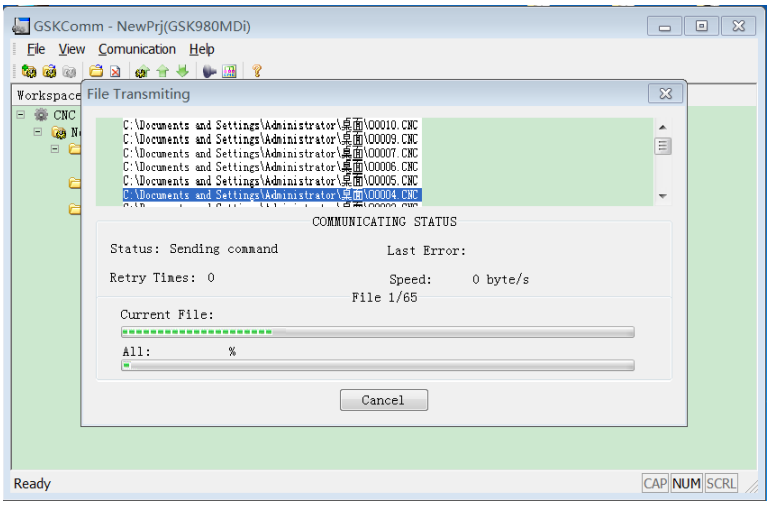


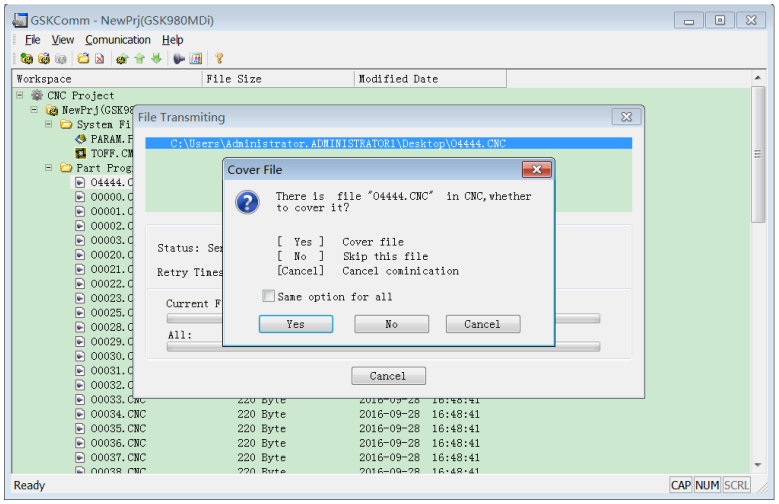
Fig. B

Select the required file to the CNC and confirm the saved file name, then press [start sending], and the file

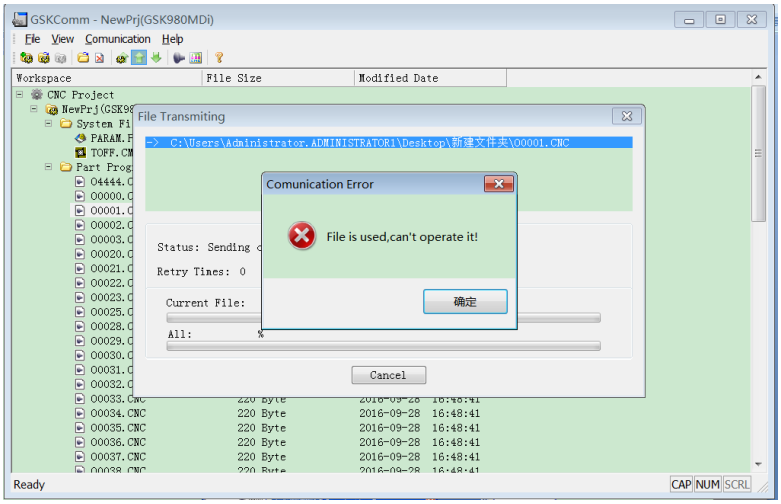
is started to send to the CNC as follows:



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:

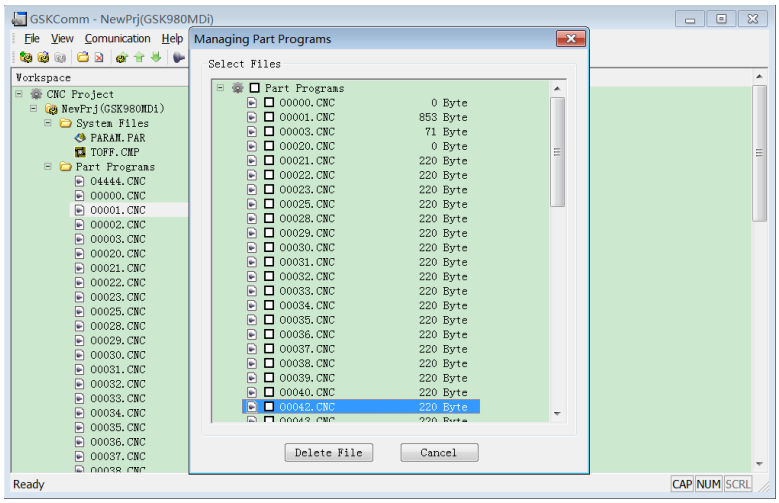


When the part program name is the same that of the transmitted file, the covering cannot be done, otherwise, the system pops up the wrong prompt as follows:



13.1.5 Part Program Managing

Select current project file, click Communication Menu “part program manage” can manage part programs. Below dialog box will be injected after click.



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 Third Party Communication Software Serial Port Transmission Program

13.2.1 Serial Port Settings

➢…. **CNC end settings**

port	COM1
Baud rate	Parameter setting for system 3020, unit bit/s
Data bit length	8 digits
parity check	No verification
stop bit	1 stop position

The baud rate in the above table is set in parameter 3020 of the system, while the rest are fixed settings.

➢…. **PC settings**

The PC software end must have the same settings as the CNC end in the table above.

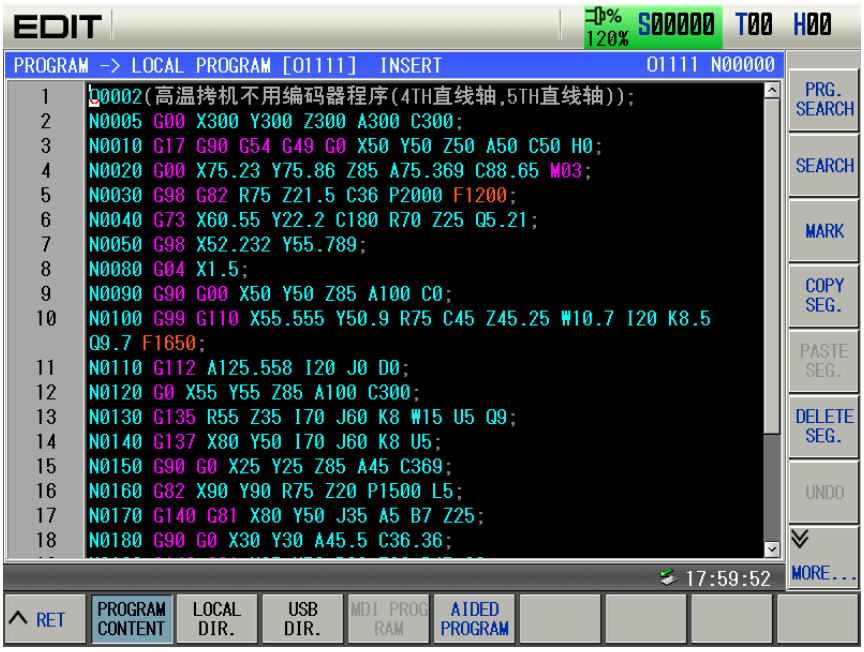
13.2.2 Transmission Program Operation

13.2.2.1 Input Program

1)....CNC Enter the waiting to receive program state

Enter the program page, select 

to enter the second page of the vertical submenu, as shown in the following figure:



Press the 

key to pop up the input/output program name input line, enter the program name.

Enter the program name, which is the name of the program saved in the received file. It must be a new program name and cannot overwrite an existing program.

Next, press the 'Enter' key to enter the receiving program state.

2)....PC software sends files

After clicking on 'send file' in the PC software, CNC starts receiving the program.

3)....If successfully received, open the receiving program. If the reception fails, an error message will be prompted in the status bar.

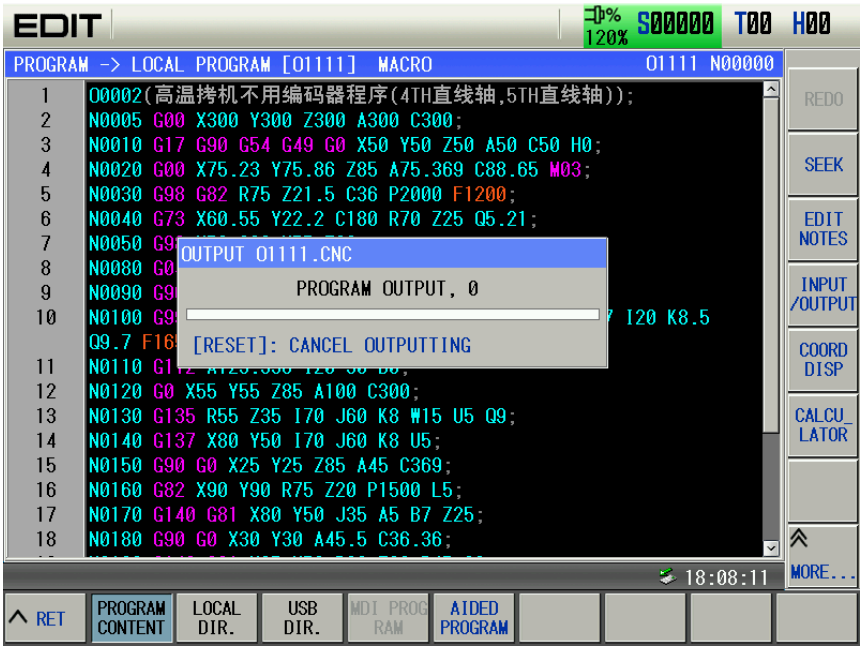
13.2.2.2 Output Programme

1）Click the PC software receive button, and the PC will be in the receive state.

2）Press the key to pop up the input/output program name input line, enter the program name.

Enter the name of the program to be sent. If no program name is entered, output the currently open program.

Next, press the "Output" button, and the system will output the specified program, as shown in the following figure:



3）After successfully sending the program, the status bar will prompt 'Program Oxxxx successfully output'.

13.2.3 Precautions

1）The system cannot switch between working modes and pages when transferring programs until the reception is completed or the reset button is pressed to cancel the reception.

2）When transferring programs, the system needs to be in editing mode, stopped state, and program switch on state.

3）When transferring programs, level 4 or higher permissions are required.

Ⅲ Installation

Chapter 1 Installation Layout

1.1 GSK980MDi Plus Connection

Interface and power box of GSK980MDi Plus, GSK980MDi Plus-V.

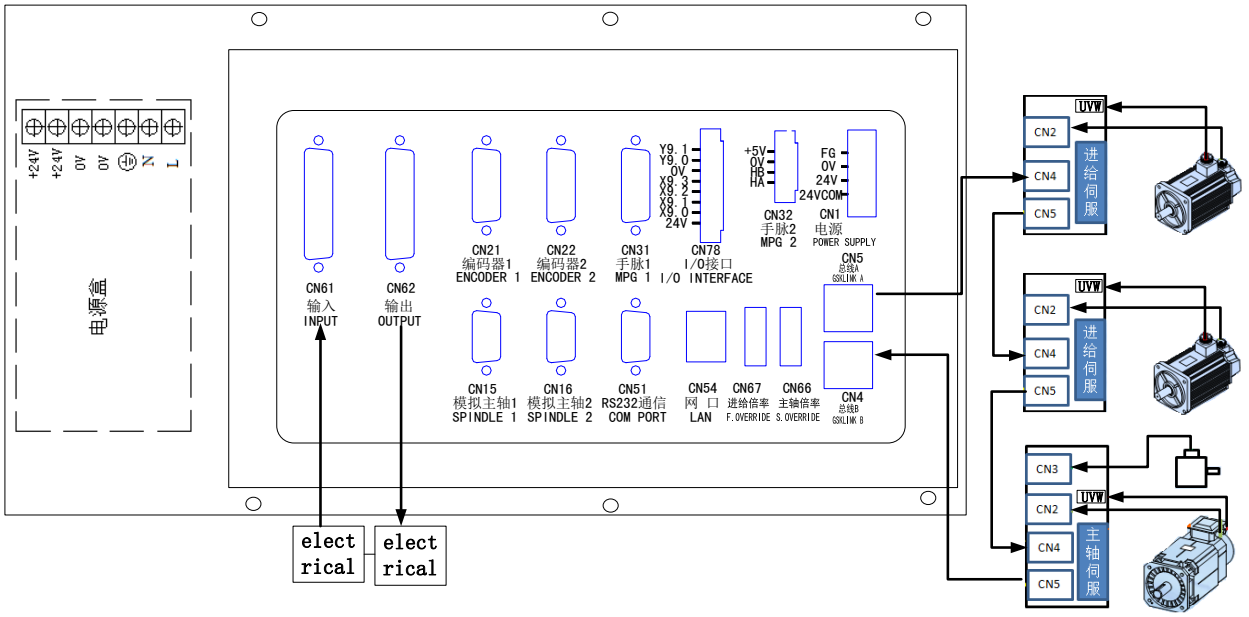


Fig 1-1 GSK980MDi Plus back cover interface layout

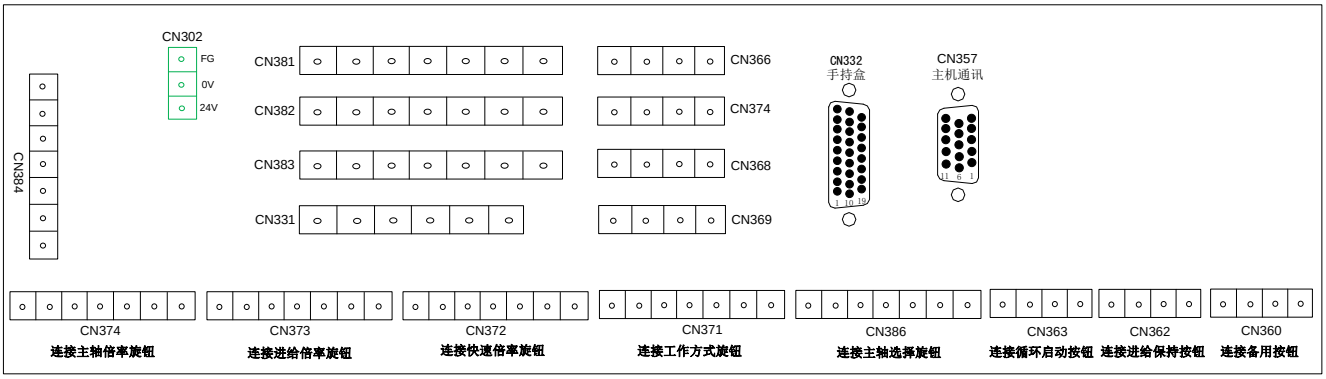


Fig 1-2 Machine tool operation panel back cover interface

1.2 GSK980MDi Plus Installation

1.2.1 GSK980MDi Plus External Dimensions

See Appendix Ⅰ, Ⅱ.

1.2.2 Measures Against Interference

In order to insure the CNC stable working, the anti-interference technology such as space electromagnetic

radiation shielding, impact current absorbing, power mixed wave filtering are employed in CNC design. And the following measures are necessary during CNC connection:

1. Make CNC far from the interference devices (inverter, AC contactor, static generator, high-voltage generator and powered sectional devices etc.);
2. To supply the CNC via an isolation transformer, the machine with the CNC system should be grounded, the CNC and drive unit should be connected with independent grounding wires at the grounding point;
3. To inhibit interference: connect parallel RC circuit at both ends of AC winding (Fig. 1-3), RC circuit should approach to inductive loading as close as possible; reversely connect parallel freewheeling diode at both ends of DC winding (Fig. 1-4); connect parallel surge absorber at the ends of AC motor winding (Fig. 1-5);

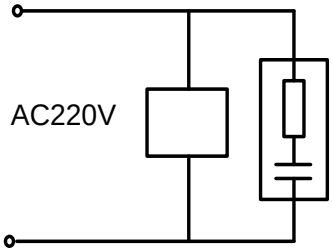


Fig.1-2

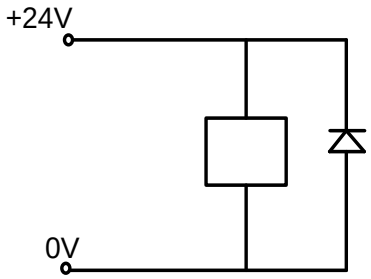


Fig. 1-3

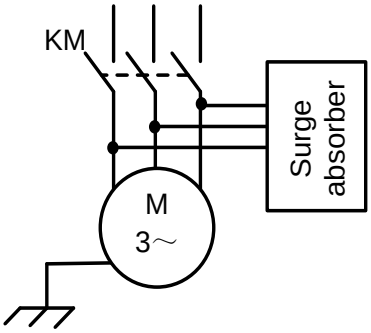


Fig. 1-4

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 Interface Signals

2.1 Connection to Drive Unit

Bus interfaces of GSK980MDi Plus are CN4 and CN5 (GSK-LinkB and GSK-LinkA), which is connected with the GSK-Link bus communication function, the feedrate servo drive unit, spindle drive unit and extended I/O communication. GSK-Link bus communication uses a loop control, and therefore, CN4 and CN5 interfaces should be used together.

GSK-Link bus connection is shown below:

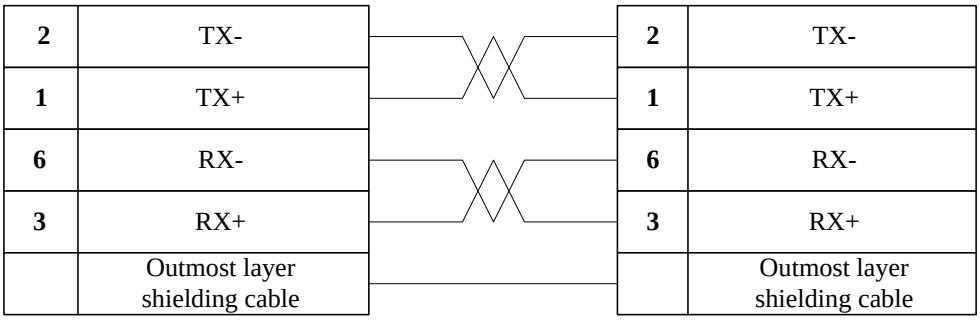
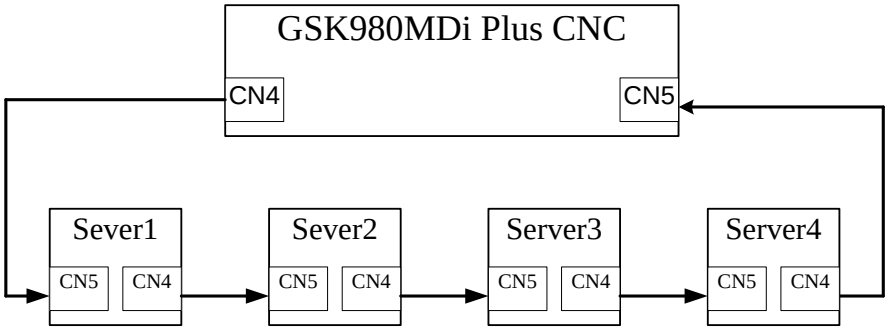


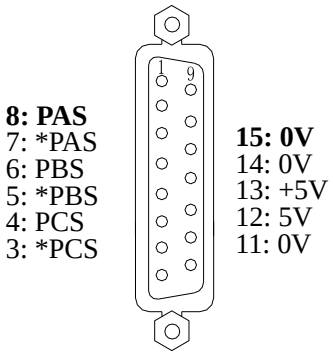
Fig. 2-1 GSK-Link communication connection

GSK980MDi Plus Connection example：



2.2 Being Connected with Spindle Encoder

2.2.1 Spindle Encoder Interface Definition



Name	Explanation
*PAS/PAS	Encoder A phase pulse
*PBS/PBS	Encoder B phase pulse
*PCS/PCS	Encoder C phase pulse

Fig.2-2 CN21,CN22 encoder interface (15-core D type male socket)

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 GSK980MDi Plus are set at will by parameter, the setting range is from 100 to 5000.

Its interior circuit is shown in Fig. 2-3: (n=A, B, C)

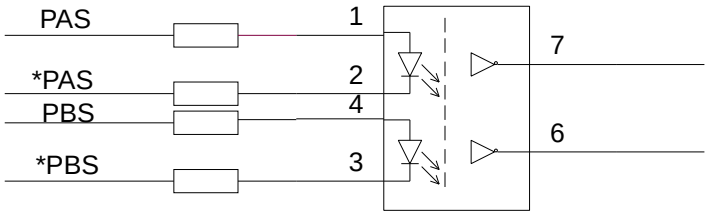


Fig. 2-3 Encoder signal circuit

2.2.3 Spindle Encoder Interface Connection

GSK980MDi Plus 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):

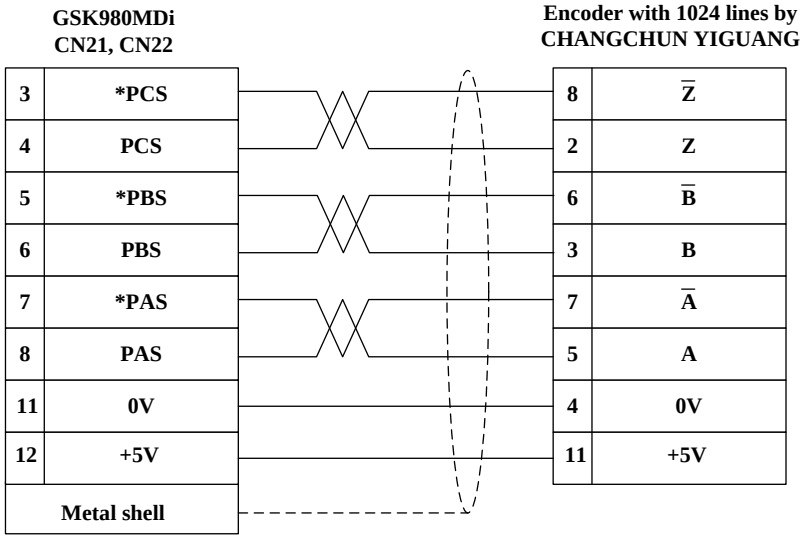


Fig. 2-4 GSK980MDi Plus is connected with the encoder

2.3 MPG Connection

2.3.1 MPG Interface Definition

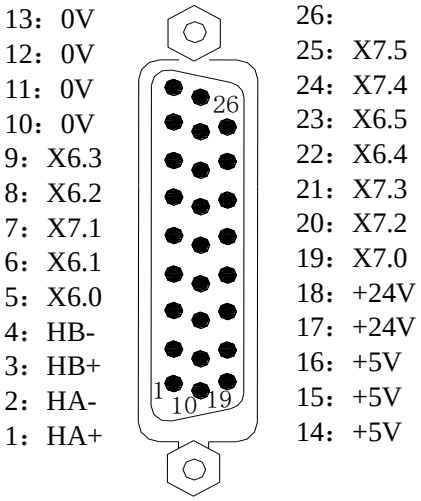


Fig.2-5 CN31 **MPG interface (26-core DB type male socket)**

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
X7.5	4TH_axis selection signal
X7.1	5TH_axis selection signal
X6.3	Increment0.001
X6.4	Increment0.01
X6.5	Increment0.1

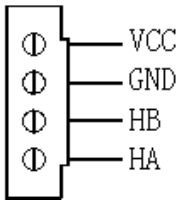


Fig. 2-6 CN32MPG interface (4-point wiring terminal)

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. 2-7:

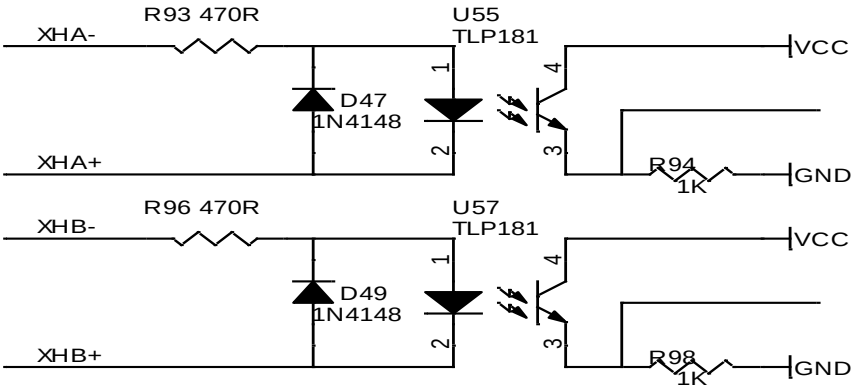


Fig. 2-7 MPG signal circuit

GSK980MDi Plus is connected with MPG shown in Fig. 2-8～Fig. 2-10.

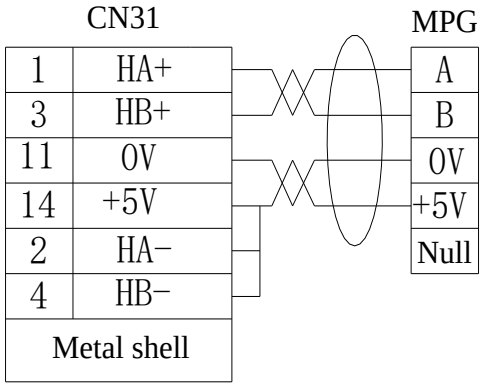


Fig. 2-8 CN31 connects with the single-port MPG input

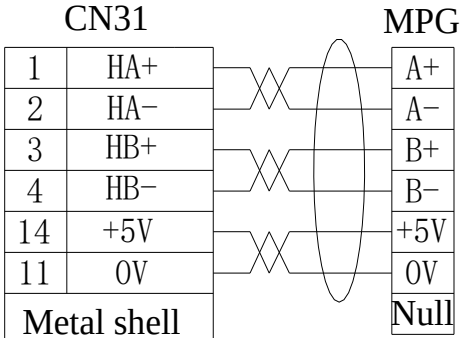


Fig. 2-9 CN31 connects with the difference MPG input

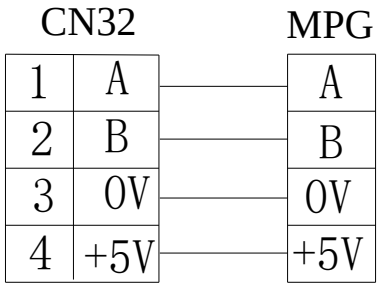


Fig. 2-10 CN32 connects with MPG

2.4 Analog Spindle Interface

2.4.1 Analog Spindle Interface Definition

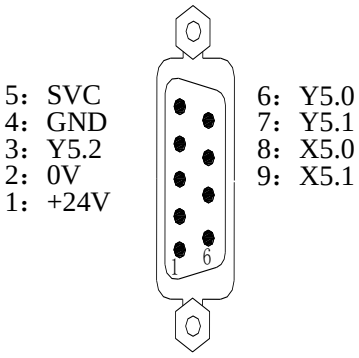


Fig.2-11 CN15 analogy spindle interface（male socket of 9-cord in type D）

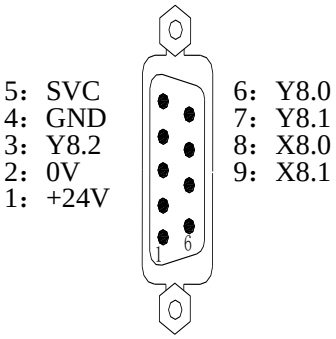


Fig.2-12 CN16 analogy spindle interface（male socket of 9-cord in type D）

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	

III Installation

2.4.2 Inverter Connection

The analog spindle interface SVC may output 0~10V voltage, its interior signal circuit is shown in Fig. 2-13:

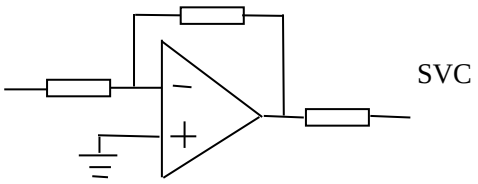


Fig. 2-13 SVC signal circuit

GSK980MDi Plus is connected with the inverter shown in Fig. 2-14:

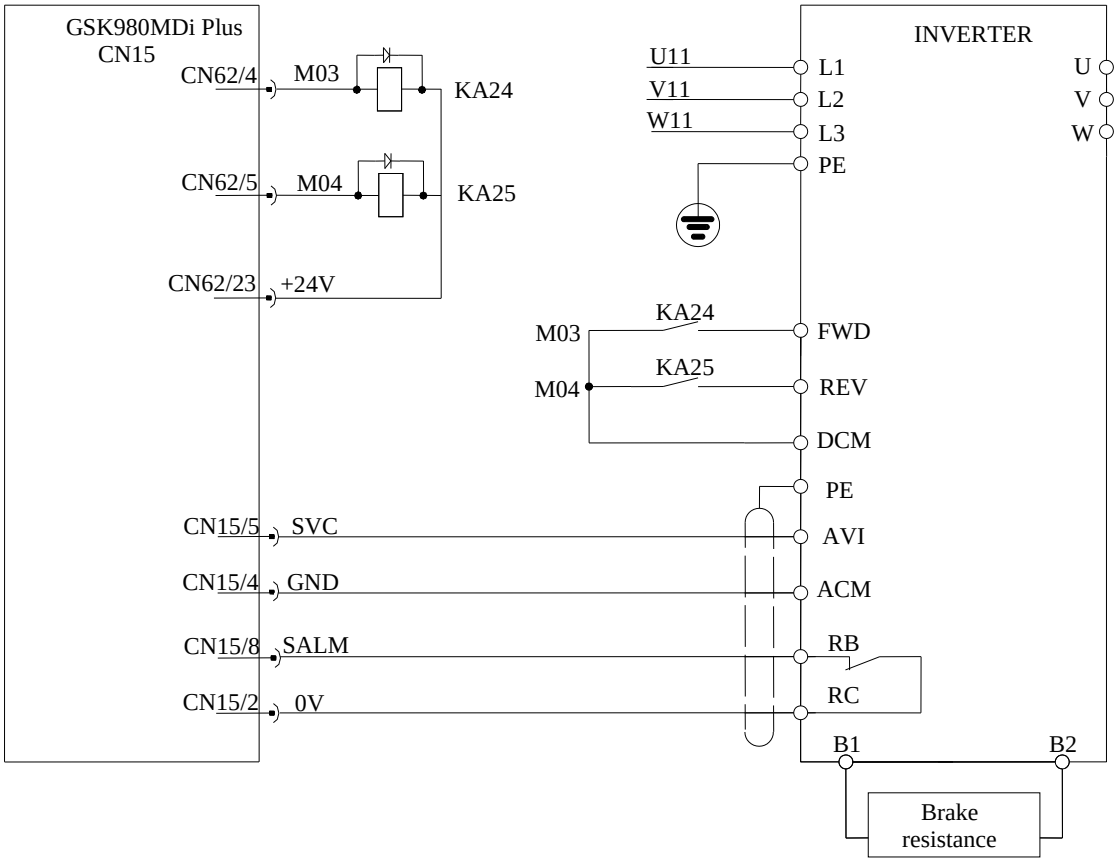
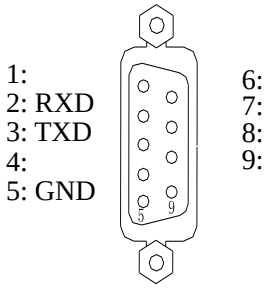


Fig. 2-14 GSK980MDi Plus is connected with the inverter

2.5 GSK980MDi Plus being Connected with PC

2.5.1 Communication Interface Definition



Signal	Explanation
RXD	Receiving data
TXD	Transmitting data
GND	Signal grounding

Fig. 2-15 CN51 communication interface (DB9-female)

2.5.2 Communication Interface Connection

GSK980MDi Plus can perform the communication by CN51 and PC(optional communication software). GSK980MDi Plus is connected with PC shown in Fig 2-16:

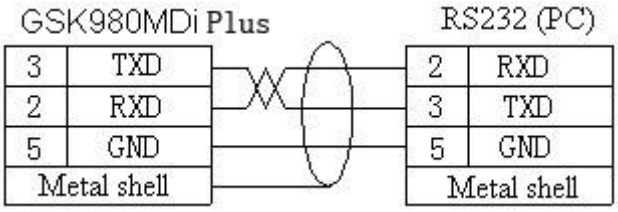


Fig. 2-16 GSK980MDi Plus is connected with PC

2.6 Network Interface

Network interfaces of GSK980MDi Plus 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 I/O Interface Definition

Note!

The I/O function significances of the unlabelled fixed addresses of this GSK980MDi Plus-H/GSK980MDi Plus-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.

CN61 is the interface of DB44 male socket (3-line), which definition is as follows:

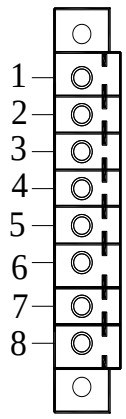
CN61 interface			
Pin-out No.	**Address**	**Symbol**	**Description**
21～24	0V		Power
17～20 25～28	Suspension	Suspension	
1	X0.0	TDEC	detect Cutter back to 0 in position
2	X0.1	SP	External feed hold signal
3	X0.2	TPCH	Cylinder pressure test
4	X0.3		
5	X0.4	TCOT	Tool post counting
6	X0.5	ESP	External emergency stop signal
7	X0.6	THCH	Toolcase horizontally in position detection/toolpot back in position detection
8	X0.7	TVCH	Toolcase vertically in position detection/toolpot forward in position detection

CN61 interface			
Pin-out No.	**Address**	**Symbol**	**Description**
9	X1.0	THSP	Stop detection of ATC arm of pot tool magazine
10	X1.1	THGT	Catching tool detection of the ATC arm of pot tool magazine
11	X1.2	THZP	Disc-tools tool changer arm origin detection
12	X1.3		
13	X1.4	ST	External cycle start signal
14	X1.5	JOGT	External spindle releasing/clamping tool input
15	X1.6	SAGT	Safe door signal
16	X1.7	TRCH	Spindle releasing tool detection
29	X2.0	TGCH	Spindle clamping tool detection
30	X2.1	THOV	motor of the ATC arm in pot magazine overload
31	X2.2	TTOV	Tool changer motor overloaded
32	X2.3		
33	X2.4	DEC4	Signal of 4th-axis deceleration
34	X2.5	DEC5	Signal of 5th-axis deceleration
35	X2.6	TZIN	External toolpot back to zero input signal
36	X2.7	LTXN	X positive limit(LT-)
37	X3.0	LTXP	X positive limit(LT+)
38	X3.1	LTYN	Y positive limit(LT-)
39	X3.2	LTYP	Y positive limit(LT+)
40	X3.3		
41	X3.4		
42	X3.5	SKIP	Skip signal
43	X3.6	LTZN	Z positive limit(LT-)
44	X3.7	LTZP	Z positive limit(LT+)

CN62 is the interface of DB44 female socket (3-line), which definition is as follows:

CN62 interface			
Pin-out No.	**Pin-out No.**	**Pin-out No.**	**Pin-out No.**
17～19, 26～28	0V	Power interface	Power 0V terminal
20～25	+24V	Power interface	Power ＋24V terminal
1	Y0.0	COOL	Cooling signal
2	Y0.1	LUBR	Lubricating signal
3	Y0.2		
4	Y0.3	SRV	Spindle CCW rotation
5	Y0.4	SFR	Spindle CW rotation
6	Y0.5		
7	Y0.6	TKOUT	chip removal
8	Y0.7	SPZD	Spindle brake signal
9	Y1.0	GEAR1	Spindle mechanical gear 1
10	Y1.1	GEAR2	Spindle mechanical gear 2

CN62 interface			
Pin-out No.	**Pin-out No.**	**Pin-out No.**	**Pin-out No.**
11	Y1.2	GEAR3	Spindle mechanical gear 3
12	Y1.3	GEAR4	Spindle mechanical gear 4
13	Y1.4	THOR	Toolcase vertical
14	Y1.5	TVER	toolcase horizontal
15	Y1.6	TCW	Toolpot motor CW
16	Y1.7	TCCW	Toolpot motor CCW
29	Y2.0	TROT	spindle releasing/clamping tool output
30	Y2.1	THOT	Disk-tools tool changer arm output motor
31	Y2.2	CLPY	Tri-colour indicator-yellow
32	Y2.3	CLPG	Tri-colour indicator -green
33	Y2.4	CLPR	Tri-colour indicator -red
34	Y2.5		
35	Y2.6	WKLT	Work light
36	Y2.7	ALTO	Turning output signal
37	Y3.0		
38	Y3.1	ITLS	Dividing table loosen signal
39	Y3.2	ITCP	Dividing table clamp signal
40	Y3.3		
41	Y3.4		
42	Y3.5	SPZD2	The 2nd spindle braking
43	Y3.6	ITEN	Dividing table enabled output
44	Y3.7		



CN78 **Pin.No**	**Address**	**Function**	
1	Y9.1	SPLAMP	feed hold signal lamp
2	Y9.0	STLAMP	cycle start signal lamp
3	0V	0V	
4	X9.3	EXESP	emergency stop signal of the machine panel
5	X9.2	reserved	
6	X9.1	SP	feed hold signal of the machine panel
7	X9.0	ST	cycle start signal of the machine panel
8	+24V	24V	

Fig.2-17 CN78

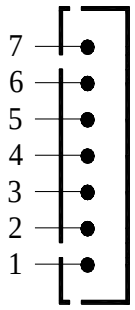


Fig.2-18 CN66

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

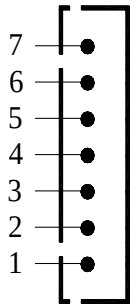


Fig.2-19 CN67

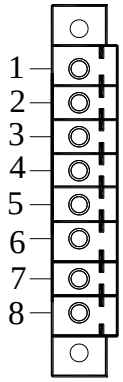
CN67 Pin.No	Address	Function	Explanation
1	X29.0	BAUD2_A	Feedrate wave-band switch signal 0
2	X29.2	BAUD2_B	Feedrate wave-band switch signal 2
3	Floating	Floating	Floating
4	+24V	24V	Power supply 24V
5	X29.3	BAUD2_E	Feedrate wave-band switch signal 3
6	X29.1	BAUD2_F	Feedrate wave-band switch signal 1
7	INH	BAUD2_INH	INH

Note 1: When the output signal is valid, it conducts internally with 0V.

Note 2: When the input signal is conducting with+24V, the input is valid.

2.8 System Specific Input/Output Interface

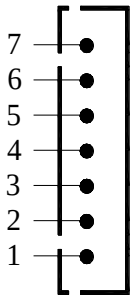
2.8.1 CN78 I/O Interface



pin number	address	function	Instructions
1	Y9.1	SPLAMP	External pause light
2	Y9.0	STLAMP	External cycle start light
3	0V	0V	Power supply 0V
4	X9.3	EXESP	External emergency stop signal
5	X9.2		Spindle feed three position knob spindle stop input
6	X9.1	SP	(0: Allow 1: Prohibit)
7	X9.0	ST	External pause
8	+24V	24V	External loop start

Fig.2-20 CN78 I/O Interface

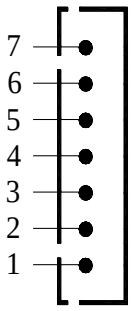
2.8.2 CN66 External Spindle Magnification Interface



Pin Number	Address	Function	Instructions
1	X29.4	BAUD1_A	Spindle magnification band switch signal 0
2	X29.6	BAUD1_B	Spindle magnification band switch signal 2
3	suspended	suspended	suspended
4	+24V	24V	Power supply 24V
5	X29.7	BAUD1_E	Spindle magnification band switch signal 3/Spindle feed three position knob feed stop input (0: allow 1: prohibit)
6	X29.5	BAUD1_F	Spindle magnification band switch signal 1
7	INH	BAUD1_INH	INH

Fig.2-21 CN66 external spindle magnification interface

2.8.3 CN67 External Input Magnification Interface



Pin Number	Address	Function	Instructions
1	X29.0	BAUD2_A	Feed rate band switch signal 0
2	X29.2	BAUD2_B	Feed rate band switch signal 2
3	suspended	suspended	suspended
4	+24V	24V	Power supply 24V
5	X29.3	BAUD2_E	Feed rate band switch signal 3
6	X29.1	BAUD2_F	Feed rate band switch signal 1
7	INH	BAUD2_INH	INH

Fig.2-22 CN67 external input magnification interface

2.9 GSK980MDi Plus Operation Panel Interface

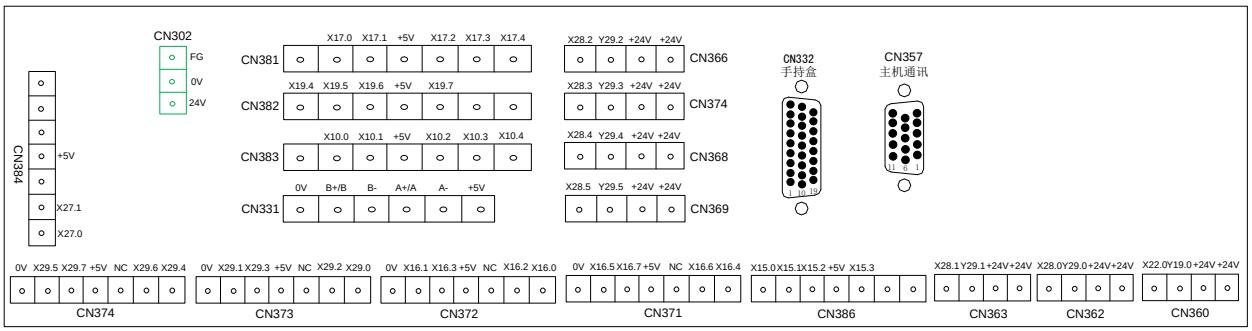
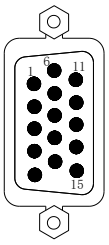


Fig.2-23 Operation Panel

2.9.1 Host Communication

1：TXD+
2：TXD-
3：GND
4：RXD+
5：RXD-



Signal	Instructions
TXD+	Sending data+
TXD-	Sending data -
GND	signal ground
RXD+	Receive data+
RXD-	Receiving data -

Fig.2-24 CN357 Host Communication Interface (15 Holes)

GSK980MDi Plus communicates with the CN357 interface of the control panel through the CN51 interface of the host, as shown in Fig.2-25:

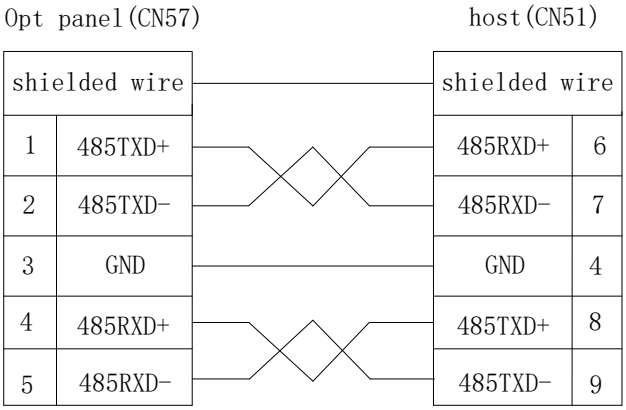


Fig.2-25 Connection between GSK980MDi Plus Host and Operating Panel

2.9.2 Hand Pulse Interface

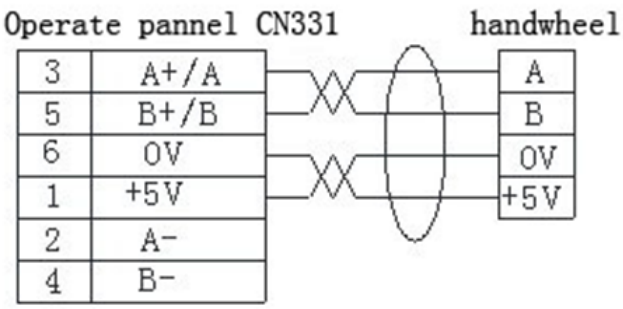


1：+5V
2：A-
3：A+/A
4：B-
5：B+/B
6：0V

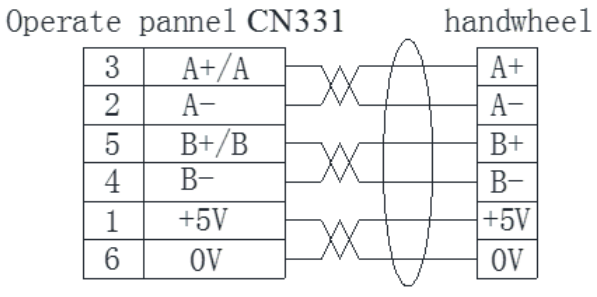
Signal	Instructions
A+/A、A-	Hand pulse A-phase signal
B+/B、B-	Hand pulse B-phase signal
+5V、0V	DC power supply

Fig.2-26 CN331 Hand Pulse Interface

The connection between the CN331 interface of the GSK980MDi Plus operation panel and the hand pulse is shown in Fig.2-27:



single-ended input



differential input

Fig.2-27

2.9.3 Expand I/O

Interface	Signal	Instructions
CN360	X22.0	Backup button input signal
	Y19.0	Backup button signal light
CN362	X28.0	Cycle start input signal
	Y29.0	Cycle start signal light
CN363	X28.1	Feed hold input signal
	Y29.1	Feed hold signal light
CN366	X28.2	
	Y29.2	
CN367	X28.3	
	Y29.3	
CN368	X28.4	
	Y29.4	
CN369	X28.5	
	Y29.5	
CN381	X17.0	
	X17.1	
	X17.2	
	X17.3	
	X17.4	
CN382	X19.4	
	X19.5	
	X19.6	
	X19.7	
CN383	X10.0	
	X10.1	
	X10.2	
	X10.3	
	X10.4	
CN384	X27.0	Spindle feed three position knob spindle stop input (0: allowed; 1: prohibited)
	X27.1	Spindle feed three position knob feed stop input (0: allowed; 1: prohibited)
CN386	X15.0	Spindle selection 0
	X15.1	Spindle selection 1
	X15.2	Spindle selection 2
	X15.3	Spindle selection 3
CN374	X29.4	Spindle magnification band switch 0
	X29.5	Spindle magnification band switch 1
	X29.6	Spindle magnification band switch 2
	X29.7	Spindle magnification band switch 3

Interface	Signal	Instructions
CN373	X29.0	Feed rate band switch 0
	X29.1	Feed rate band switch 1
	X29.2	Feed rate band switch 2
	X29.3	Feed rate band switch 3
CN372	X16.0	Fast magnification band switch 0
	X16.1	Fast magnification band switch 1
	X16.2	Fast magnification band switch 2
	X16.3	Fast magnification band switch 3
CN371	X16.4	Method selection multiplier band switch 0
	X16.5	Method selection multiplier band switch 1
	X16.6	Method selection multiplier band switch 2
	X16.7	Method selection multiplier band switch 3

Note 1: When the output function is valid, the output signal is internally connected to 0V.

Note 2: When the input signal is conducting with+24V, the input is valid.

Note 3: CN371~CN374, CN381~CN386 are 5V input signals and can only be connected to band switches. Incorrect connection may cause the system to burn out.

Chapter 3 Parameter

3.1 Parameter Description (Sorted in Order)

In this chapter the CNC bit and data parameters are introduced. Various functions can be set by these parameters.

The expression of bit parameter is shown as follows：

Parameter NO.	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0
0 0 0 1	***	***	***	***	HWL	***	ISC	***

HWL =1: MPG mode;

=0: Step mode.

ISC =1: Increment system IS-C(0.0001mm/0.00001inch)

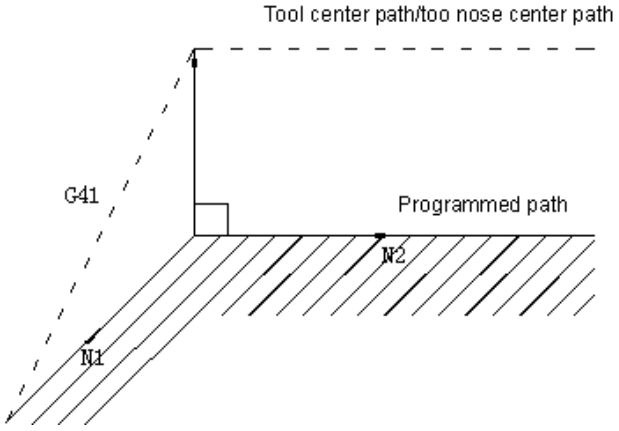
=0: Increment system IS-B(0.001mm/0.0001inch)

0 0 0 2	D/R	SUP	***	***	***	***	NRC	***

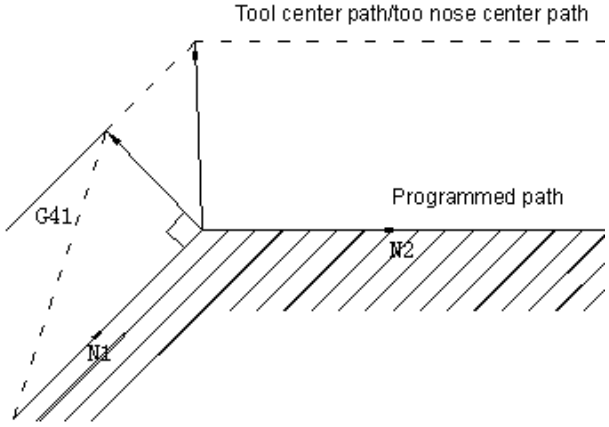
SUP =1: Tool movement of cutter compensation c in start up and cancel is B type;

=0: Tool movement of cutter compensation c in start up and cancel is A type.

A type mode：output vertical compensation vector at the next block after start-up or at the cancelled previous block as follows:



B type mode：output compensation vector and intersection point vector being vertical with start-up block/cancelled block as follows:



NRC =1: Tool nose radius compensation valid;
=0: Tool nose radius compensation invalid.
D/R =1: Tool offset D is diameter value;
=0: Tool offset D is radius value.

0 0 0 3	***	***	PCOMP	OFFSET	***	***	***	***

OFFSET =1: Tool offset by coordinate offset
=0: Tool offset by move
PCOMP =1: Screw-pitch error compensation valid;
=0: Screw-pitch error compensation invalid.

0 0 0 4	DEC5	DEC4	***	***	***	***	***	SCW

SCW =1：Inch output(inch system)valid after repower;
=0：Metric output(metric system)valid after repower

The functions of metric and inch system.

There are two kinds of input and output units for CNC numerical control system: metric unit, millimeter (mm) and English unit (inch).

Output increment unit is set by Bit0（SCW）of bit parameter №004 in GSK980MDc system. SCW=0 indicates that minimum command increment, parameter and screw–pitch values are in metric units; SCW=1 indicates that minimum command increment, parameter and screw–pitch values are in inches units. The setting of this parameter depends on machine tool.

G code: By selecting G20/G21 code, it is able to set whether minimum input increment values are in inch or in metric. Executing G21 indicates that minimum input increment values are in metric; and executing G20 indicates that values are in inch.

DECn =1：Deceleration signal high level for machine zero return;
=0：Deceleration signal low level for machine zero return.
by pulse and direction (need restart).

0 0 0 7	***	***	SMZ	***	***	***	***	***

SMZ =1：To execute next block till all moving blocks executed;
=0： For smooth transition between blocks.

0 0 0 8	***	***	***	DIR5	DIR4	DIRZ	DIRY	DIRX

DIRn =1：Direction signal (DIR)is high level as each axis moves positively;
=0：Direction signal (DIR)is low level as each axis moves negatively.

0 0 0 9	***	***	***	ALM5	ALM4	ALMZ	ALMY	ALMX

ALMn =1： axis alarm signal invalid;
=0： axis alarm signal valid.

0 0 1 0	CPF7	CPF6	CPF5	CPF4	CPF3	CPF2	CPF1	CPF0

CPF0～CPF7： Setting values of backlash compensation pulse frequency.

Set frequency =（2^7×CPF7+2^6×CPF6+2^5×CPF5+2^4×CPF4+2^3×CPF3+2^2×CPF2+2^1×CPF1+CPF0）Kpps

0 0 1 1	BDEC	BD8	***	***	***	ZNIK	***	***

BDEC =1：Backlash compensation type B, the compensation data are output by ascending type and the set frequency is invalid.

=0：Backlash compensation type A, the compensation data are output by the set frequency (by bit parameter No.010) or 1/8 of it.

BD8 =1：Backlash compensation is done by the 1/8 of the set frequency;

=0：Backlash compensation is done by the set frequency.

ZNIK =1：Direction keys locked during zero return, homing continues to end by pressing direction key once;

=0：Direction keys unlocked but should be held on during zero return.

0 0 1 2	G00CUT	JOGRATE	***	G01	***	***	SOT	ISOT

ISOT =1：Prior to machine zero return after power on, manual rapid traverse valid;

=0：Prior to machine zero return after power on, manual rapid traverse invalid.

SOT =1：Software limit is valid after zero return at power on;

=0：Software limit is valid once power on.

G01 =1：G01 status when power on;

=0：G00 status when power on.

JOGRATE =1：Manual feed speed, Using a fixed speed;

=0：Manual feed speed ，Using parameter (No.44)。

G00CUT =1：G00 Move in linear movemen;

=0：G00 Move in Non linear movemen.

0 0 1 3	CALH	***	***	MZR5	MZR4	MZRZ	MZRY	MZRX

MZRn =1：Machine zero return in negative each axis;

=0：Machine zero return in positive each axis.

CALH =1：Length offset not cancelled in reference point return;

=0：Length offset cancelled in reference point return.

0 0 1 5	CMRx：each axis multiplier coefficient

Setting range： 1～32767

0 0 1 6	CMDx：each axis frequency division coefficient

Setting range： 1～32767

0 0 2 2	Each axis max. rapid traverse speed

Setting range：IS-C：10～24000，thereinto 5th：10～6000（Unit：mm/min）

IS-B：10～60000（Unit：mm/min）

0 0 2 3	Acceleration&deceleration time constant of each axis rapid traverse (ms)

Setting range：10～4000（Unit：ms）

0 0 2 4	Rapid traverse speed when rapid override is F0

Setting range：6～4000（Unit：mm/min）

0 0 2 5	Feedrate of dry run

Setting range：10～60000 （Unit：mm/min）

0 0 2 6	Initial value of cutting feedrate when power on

Setting range：10～15000 (Unit：mm/min)

0 0 2 7	Axes top feedrate of cutting

Setting range：10～60000（Unit：mm/min）

0 0 2 9	Time constant of cutting feed

Setting range：10～4000（Unit：ms）

0 0 3 0	Acceleration start speed and deceleration end speed in cutting feed

Setting range：0～8000（Unit：mm/min）

0 0 3 1	High speed of each axis machine zero return

Setting range：10～60000 （Unit：mm/min）

0 0 3 2	Low speed of each axis machine zero return

Setting range：10～1000（Unit：mm/min）

0 0 3 4	Each axis backlash offset

Setting range：0～2000(Unit:0.001mm)

0 0 4 1	Start speed for handwheel moving（mm/min）

Setting range：0～8000（Unit：mm/min）

0 0 4 2	acceleration&deceleration time constant of manual feed

Setting range：10～4000（Unit：ms）

0 0 4 3	Max. rapid traverse speed of the handwheel of each axis

Setting range：10～60000

When No. 175.7 is set to 0, pulse exceeding No. 43 setting speed is ignored and so the speed cannot exceed No. 43 setting value;

When No. 175.7 is set to 1, pulse exceeding No. 43 setting speed is not ignored. When setting value of No. 43 is less than that of No. 41, Max. speed is not limited by No. 43.

0 0 4 4	Manual feedrate(feedrate override 100%)

Setting range：10～5000

0 0 4 5	Max. each axis coordinate value of software limit
0 0 4 6	Min. each axis coordinate value of software limit

Setting range：-99999999～+99999999 (Unit:0.001mm)

0 0 6 0	Acc.&dec.time constant of CS axis

Setting range：10～10000（Unit：ms）

0 0 7 1	G31 speed

Setting range：10～15000（Unit：mm/min）

0 0 7 6	division axis max. rapid traverse speed

Setting range：1～8000（Unit:mm/min）

0 1 7 2	***	MST	MSP	MOT	MESP	***	***	***

MST =1：External cycle start signal (ST) invalid，

=0：External cycle start signal (ST) valid.

MSP =1：External stop signal (SP) invalid，

=0：External stop signal (SP) valid with external stop switch connected, otherwise CNC shows “stop” .

MOT =1：Not detect software stroke limit;

=0：Detect software stroke limit.

MESP =1：Emergency stop invalid;

=0：Emergency stop valid.

0 1 7 3	***	TMANL	SMAL	***	***	***	ESST	ESCD

ESCD =1：S code off at emergency stop;

=0：S code not off at emergency stop.

ESST =1： break off at emergency stop

=0： slow down at emergency stop

SMAL =1：Spindle manual gear shift for S command;

=0：Spindle auto gear shift for S command.

TMANL =1：Manual tool change for T code;

=0：Auto tool change for T code.

0 1 7 4	SPFD	SAR	***	VAL5	VAL4	VALZ	VALY	VALX

SPFD =1：Cutting feed stops if spindle stops;

=0：Cutting feed not stop after spindle stop.

SAR =1：Detect spindle SAR signal prior to cutting;

=0：Not detect spindle SAR signal prior to cutting.

VAL5 =1：For 5^{th} axis move key, ←is positive，→is negative;

=0：For 5^{th} axis move key, →is positive，←is negative.

VAL4 =1：For 4^{th} axis move key, ↖is positive，↘is negative;

=0：For 4^{th} axis move key, ↘is positive，↖is negative.

VALZ =1：For Z axis move key,↑ is positive，↓is negative;

=0：For Z axis move key, ↓is positive，↑is negative.

VALY =1：For Y axis move key, ↗is positive，↙is negative;

=0：For Y axis move key, ↙is positive，↗is negative.

VALX =1：For X axis move key, →is positive，←is negative;

=0：For X axis move key, ←is positive，→is negative

0 1 7 5	HPF	***	***	HW5	HW4	HWZ	HWY	HWX

HPF =1：When speed of handwheel exceeds the max.speed set in Para 43#, the exceeded handwheel pulses are not neglected;

=0：When speed of handwheel exceeds the max.speed set in Para 43#, the exceeded handwheel pulses are neglected.

HWn=1：Coordinate is increscent when the MPG in ccw;

=0：Coordinate is increscent when the MPG in cw.

0 1 8 7	IS1x	IS0x	***	***	***	***	ROSx	ROTx

ROSx, ROTx：Set the type of 4th, 5th;

	Linear	Rotary A	Rotary B	invalid
ROTx	0	1	1	0
ROSx	0	0	1	1

IS1x, IS0x:Selecte increment system of 4th, 5th.

IS1x	IS0x	Increment System of 4th
0	0	Same to the X, Y, Z
0	1	IS-A
1	0	IS-B
1	1	IS-C

0 1 8 8	***	RRTx	***	***	***	RRLx	RABx	ROAx

RRTx =1：Zero mode D is used on 4th or 5th rotary axis;

=0：Zero mode A,B,C are used on 4th or 5th rotary axis.

RRLx =1：4th or 5th rel.coor.cycle func.is valid;

=0：4th or 5th rel.coor.cycle func.is invalid.

RABx =1：4th or 5th rotates according to symbol direction;

=0：4th or 5th rotates according to nearby rotation.

ROAx =1：4th or 5th abs.coor.cycle func.is valid;

=0：4th or 5th abs.coor.cycle func.is invalid.

Note 1: Parameter ROAx is valid for only rotary axis (ROTx=1).

Note 2: Only parameter ROAx =1, is RABx valid.

Note 3: Only parameter ROAx =1, is RRLx valid.

0 2 0 0	***	***	***	***	***	***	***	HWI

HWI =1：dividing ax can be moved in JOG\MGP\STEP mode;

=0：dividing ax can't be moved in JOG\MGP\STEP mode.

0 3 0 0	***	***	***	***	SUBSPOFF	G30HAVE	***	DXFTOOL

DXFTOOL =1：DXF file dirll trading tool mode is AUTO;

=0：DXF file dirll trading tool mode is MANUAL.

G30HAVE =1：DXF file dirll last program no have G30;

=0：DXF file dirll last program have G30.

SUBSPOFF =1：DXF file sub program over close spindle;

=0：DXF file sub program over no close spindle.

0 5 2 0	NAT	***	***	***	***	***	***	RDRN

NAT =1 : Define the range of user macro program asin(-90~+0), atan(-180~180);

=0：Define the range of user macro program asin(90~270), atan(0~360).

RDRN =1：G00 rapid traverse，speed = federate ×dry run speed;

=0：G00 speed = rapid override × rapid tranverse speed .

0 5 2 8	***	***	***	***	***	***	***	RSTP

RSTP =1: Polar coordinate is canceled after cnc is reseted;
=0: Polar coordinate isn't canceled after cnc is reseted

0 5 4 0	SCRD	RSCD	***	DELE	***	PPD	CIME	***

SCRD =1：Coordinate system holding on at power down;
=0：Coordinate system not holding on at power down, G54 coordinate system is set after power on.
RSCD =1：G54 coordinate system when reset 4;
=0：Coordinate system not changed when reset.
DELE =1：G54.1,G54~G59 remove offset of G92；
=0：G54.1,G54~G59 remained offset of G92。
PPD =1：Relative coordinate set by G92, G52, G54.1, G54～G59;
=0：Relative coordinate not set by G92, G52, G54.1, G54～G59.
CIME =1：modify current coordinate system (G54.1,G54～G59),take effect at once;
=0：modify current coordinate system (G54.1,G54～G59),take effect at next coordinate system (G54.1,G54～G59).

0 5 4 2	***	***	***	***	***	RLC	MRC	ZCL

RLC =1：Local coor. system is canceled after cnc is reseted;
=0：Local coor. system isn't canceled after cnc is reseted.
MRC =1：Local coor. system is canceled after M02 or M30 is executed;
=0：Local coor. system isn't canceled after M02 or M30 is executed.
ZCL =1：Local coor. system is canceled after machine ref. position is setted;
=0：Local coor. system isn't canceled after machine ref. position is setted.

0 5 5 1	***	***	***	***	***	G31F	SKPI	G31P

G31F =1：Feedrate of G31 come form para NO.71;
=0：Feedrate of G31 come form para F of program。
SKPI =1：High level valid for skip signal;
=0：Low level valid for skip signal.
G31P =1：G31 immediately stops when skip signal is valid;
=0：G31 slows down to stop when skip signal is valid.

0 5 5 3	***	***	***	***	***	***	TCHKI	***

TCHKI =1：Auto tool length compensation signal(for G37) high signal valid;
=0：Auto tool length compensation signal(for G37) low signal valid.

0 5 6 2	RIN	***	***	XSC	***	SCLZ	SCLY	SCLX

RIN =1: Rotation angle of coordinate rotation is defined by G90,G91;
=0: Rotation angle of coordinate rotation is defined by G90.
XSC =1: Scale rate of X,Y,Z is defined by I,J,K or para NO.5421;
=0: Scale rate of X,Y,Z is defined by P or para NO.5421.
SCLZ =1: Z scale function is valid;
=0: Z scale function is invalid.
SCLY =1: Y scale function is valid;
=0: Y scale function is invalid.
SCLX =1: X scale function is valid;

=0: X scale function is invalid.

0 5 8 2	BKST	***	STWP	***	***	LPTK	RPTK	BRCH

BKST =1：when Milled to be plane or chessboard hole,the tool don't return to initial XY position;

=0：when Milled to be plane or chessboard hole,the tool return to initial XY position.

STWP =1：Orientation of circle or line interval drill，start from jumping-off point;

=0：Orientation of circle or line interval drill，strat from next point.

LPTK =1：Hole locating is done by cutting feed on line continuous drilling;

=0：Hole locating is done by rapid feed on line continuous drilling.

RPTH =1: Hole locating is cutting path in circle and rectangle continuous drilling;

=0：Hole locating is rapid path in circle and rectangle continuous drilling.

BRCH =1：Plane returning is selected by G98 and G99 in continous drilling;

=0：Plane returning is selected by G99 in continuous drilling.

0 5 8 4	***	***	***	***	***	***	RD2	RD1

RD2 =1：Retraction in Y axis direction of G76, G87;

=0：Retraction in X axis direction of G76, G87.

RD1 =1：Retraction in Y axis direction of G76, G87;

=0：Retraction in X axis direction of G76, G87.

0 5 8 8	RTORI	RHD5	RTPCP	***	***	RTCRG	***	***

RTORI =1：M29 is executed, Spindle need to return zero;

=0：M29 is executed, Spindle need not to return zero.

RHD5 =1：5^{th} axis coordinate information is hided;

=0：5^{th} axis coordinate information isn't hided.

RTPCP =1：Rigid tapping is the high-speed deep hole cycle(G73) ;

=0：Rigid tapping is the high-speed deep hole cycle (G83).

RTCRG=1：Do not wait for G61.0 to be 1 as executing next program block after rigid tapping cancelled;

=0：Do wait for G61.0 to be 1 as executing next program block after rigid tapping cancelled.

0 6 0 0	***	***	***	***	***	***	GTT	SGB

SGB =1: M-type gear shift mode B;

=0：M-type gear shift mode A.

GTT =1：Analog spindle gear shit is M;

=0：Analog spindle gear shit is T.

0 6 5 0	***	***	***	***	***	***	PRPD	PLA

PLA =1：PLC axis control is valid;

=0：PLC axis control is invalid.

PRPD =1：PLC axis rapidly uses input value;

=0：PLC axis rapidly uses input parameter.

0 7 0 0	***	***	***	***	***	***	***	CHPW

CHPW =1：Real-time detection for the stability of voltage isn't performed；

=0：Real-time detection for the stability of voltage is performed.

0 7 0 1	***	***	***	***	***	***	***	EXTIO

EXTIO =1: External remote I/O is valid;
=0: External remote I/O is invalid.

0 7 6 0	ALK	***	***	***	***	***	***	APC

APC =1: The encoder type of axis is absolute encoder;
=0: The encoder type of axis is incremental encoder。

ALK =1: The servo axis didn't connect servo drive;
=0: The servo axis connected servo drive。

0 7 8 0	***	***	***	***	***	***	***	GSKLink

GSKLink =1: GSK-Link is valid;
=0: GSK-Link is invalid.

0 8 0 0	***	***	***	***	***	L2	L1	L0

L2, L1, L0：Interface language selection:

Language	L2	L1	L0
Chinese	0	0	0
English	0	0	1
France	0	1	0
Spanish	0	1	1
Germen	1	0	0
Italian	1	0	1
Russian	1	1	0
Korean	1	1	1

0 8 0 1	DISP	***	***	***	A45_CAN	***	PROD	***

PROD =1：Rel.coord. don't with comp
=0：Rel.coord. with comp

A45_CAN =1：synchronizing axis displayed;
=0：synchronizing axis don't displayed。

DISP =1：Enter absolute page after power on;
=0：Enter relative page after power on.

0 8 1 0	***	***	***	MER	MCL	MKP	***	M30R

MER =1：After the last block is executed in MDI operation, the executed programs are not deleted;
=0：After the last block is executed in MDI operation, the executed programs are deleted.

MCL =1：The programs edited in MDI operation are deleted by the reset operation;
=0：The programs edited in MDI operation are not deleted by the reset operation.

MKP =1：In MDI operation, when M02 or M30 is executed, the edited MDI programs are not deleted;
=0：In MDI operation, when M02 or M30 is executed, the edited MDI programs are deleted.

M30R =1：Cursor return after executing M30 in Auto mode;
=0：Cursor don't return after executing M30 in Auto mode.

0 8 3 0	***	***	***	***	KEY1	***	***	***

KEY1 =1：Prog. switch ON after power on;

=0：Prog. switch OFF after power on.

0 8 3 1	***	***	***	***	***	CTPANE	***	MGEH

CTPANE =1：Custom working haven't cords;

=0：Custom working have cords.

MGEH =1：Aided programming function is shielded;

=0：Aided programming function is unshielded.

1 0 1 0	Amount of axes

Setting range：3～5.

1 0 2 0	Define the name of the each axis

Setting range:65～67 65-A，66-B，67-C.

1 0 2 2	Name of each axis

Setting range：5～7 parallel of X—5，parallel of Y—6，parallel of Z—7.

1 0 2 6	Set synchro axis of the 4th axis,0:invalid 1:X axis 2:Y axis 3:Z axis
1 0 2 7	Set synchro axis of the 5th axis,0:invalid 1:X axis 2:Y axis 3:Z axis

1 0 3 0	the number of axis for circular dividing table 0:invalid; 4:4th axis; 5:5th axis

1 2 4 0	Each axis machine coordinate of 1st reference point
1 2 4 1	Each axis machine coordinate of 2nd reference point
1 2 4 2	Each axis machine coordinate of 3rd reference point
1 2 4 3	Each axis machine coordinate of 4th reference point

Setting range：-99999999～99999999 (0.001mm)

1 2 4 6	Each axis axis machine zero offset

Setting range：-99999～99999 (Unit:0.001mm)

1 2 6 0	Movement per rotation of each axis

Setting range：-360000～360000 （Unit:0.001deg）

3 0 1 0	Delay time (ms) of strobe pulse signal SF during M gear shift

Setting range：0～1000 (Unit：ms）

3 0 1 7	Reset output time

Setting range：16～4080（Unit：ms）

3 0 2 0	Serial communication baudrate

Setting range：1200, 2400, 4800, 9600, 19200, 38400, 57600, 115200（Unit：bit/s）

3 0 2 1	Serial communication slave ID

3 0 3 0	Quantity of MPG (1~2)
3 0 3 1	MPG interface of MPG 1(0:CN31 1:CN32)
3 0 3 2	MPG interface of MPG 2(0:CN31 1:CN32)

3 2 1 6	Block No. increment for block No. auto insertion

Setting range：1～100

3 4 1 0	Arc radius error limit

Setting range：0～1000 (Unit:0.001mm), On arc code （G02,G03）, if error exceeds the difference running limit between initial point radius and end point radius, alarm will be issued.

3 4 1 2	Max Offset of absolute encoder at power on
3 6 2 0	Screw-pitch error compensation position number of each axis machine zero

Setting range： 0～1023

3 6 2 1	Min. position number of each axis for pitch error compensation

Setting range：0～1023

3 6 2 2	Max. position number of each axis for pitch error compensation

Setting range：0～1023

3 6 2 4	Interval of each axis screw-pitch error compensation

Setting range：1000～999999 (Unit:0.001mm)

3 7 1 0	Quantity of spindle（1~3）
3 7 1 2	Type of spindle(0:switching volume ; 1:analog voltage; 2:series spindle)
3 7 2 0	spindle encoder pulses/rev

Setting range：0～5000 （Unit：p/r）0: Not detect spindle encoder in G74, G84 tapping.

3 7 2 1	Transmission ratio of encoder and - spindle gear teeth
3 7 2 2	Transmission ratio of encoder and - encoder gear teeth

Setting range:1～255

3 7 2 4	encoder sign form (0:encoder invalid 1:CN21 2:CN22 3:series spindle)
3 7 2 6	Spindle svc init data (mV)
3 7 3 0	Voltage offset value when spindle max. speed analog voltage 10V output

Setting range：-2000～2000（Unit：mV）

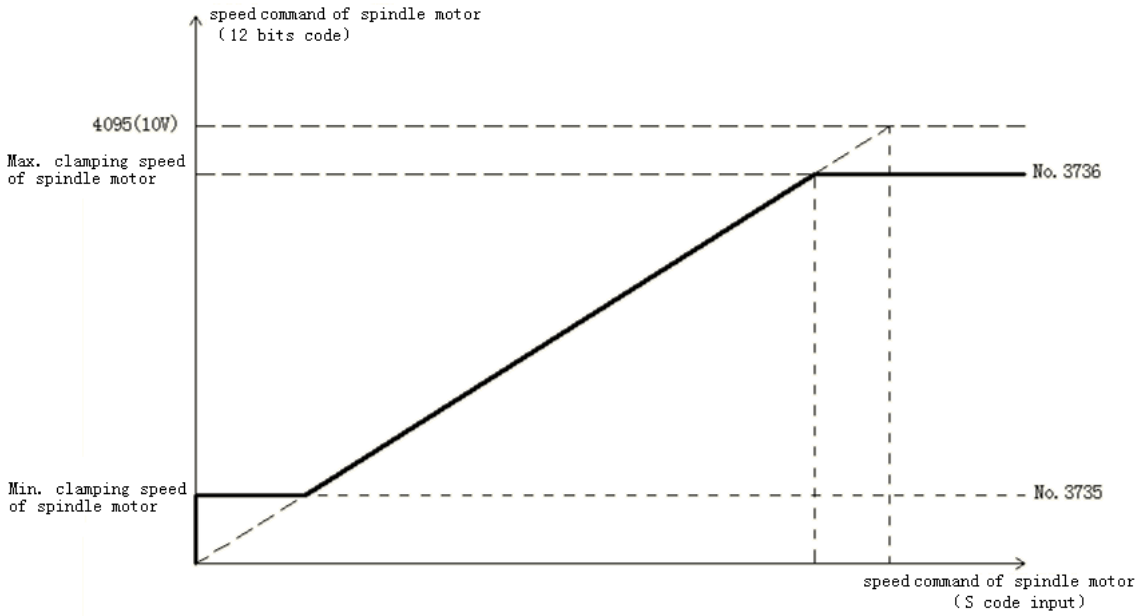
3 7 3 1	Voltage compensation for 0V analog voltage output

Setting range：-1000～1000（Unit:mV）

3 7 3 5	Min. clamping speed of spindle motor (12 bits code value) in gearing type M
3 7 3 6	Max. clamping speed of spindle motor (12 bits code value) in gearing type M

Setting range：0～4095

No. 3735 setting value:



3 7 4 0	Delay of spindle speed in-position signal detection

Setting range：0～4080 （Unit:ms）

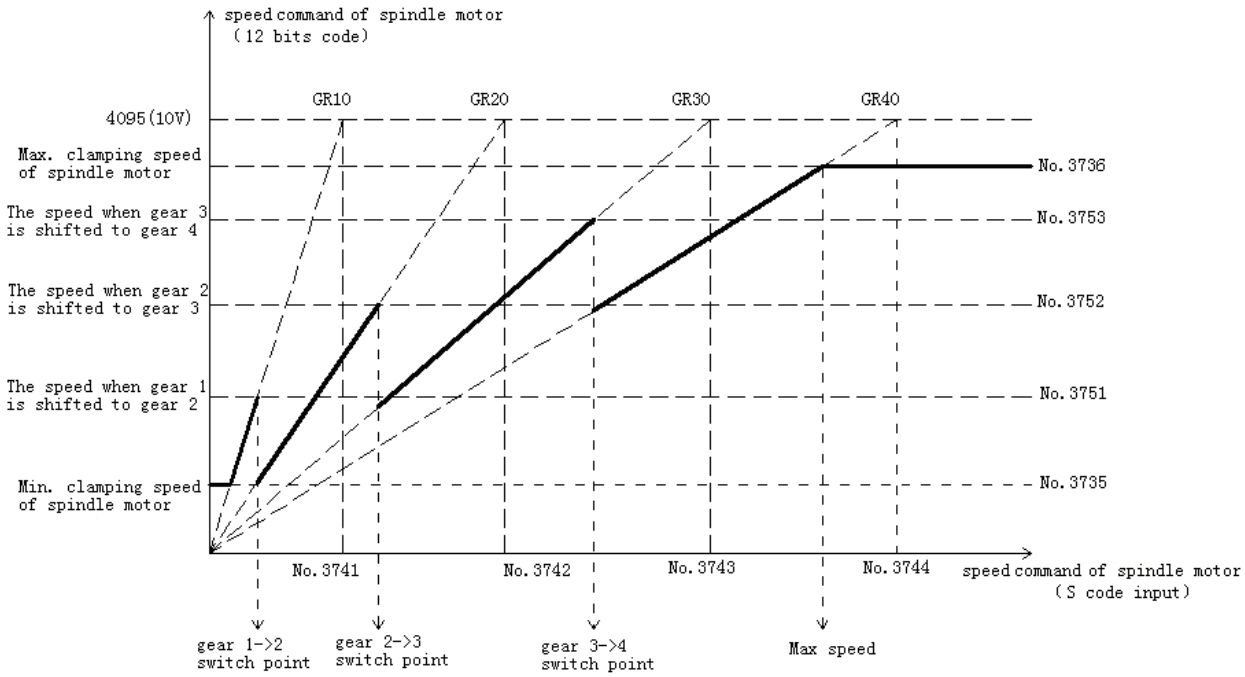
3 7 4 1	Max spindle speed of 1st gear when analog voltage output is 10V
3 7 4 2	Max spindle speed of 2nd gear when analog voltage output is 10V
3 7 4 3	Max spindle speed of 3rdt gear when analog voltage output is 10V
3 7 4 4	Max spindle speed of 4th gear when analog voltage output is 10V

Setting range：10～9999 （Unit:r/min）

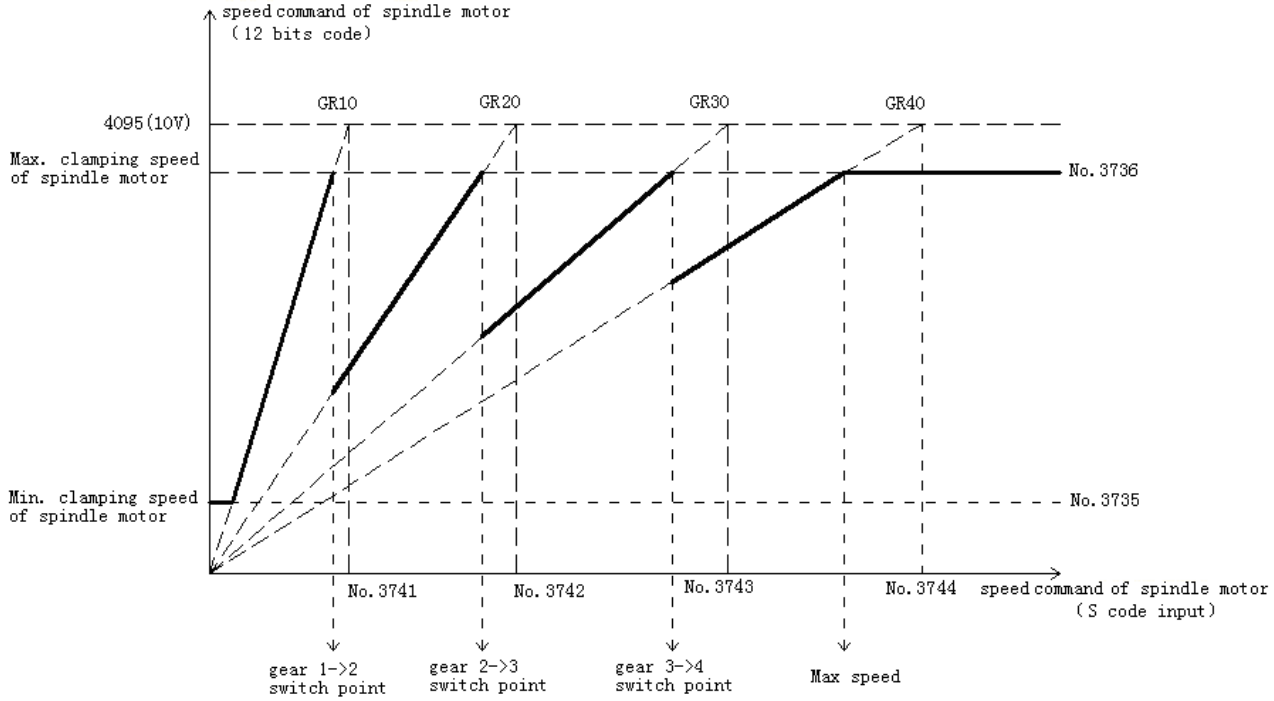
3 7 5 1	Spindle motor speed when gear 1 is shifted to gear 2 (12 bits code value) in gearing type M
3 7 5 2	Spindle motor speed when gear 2 is shifted to gear 3 (12 bits code value) in gearing type M
3 7 5 3	Spindle motor speed when gear 3 is shifted to gear 4 (12 bits code value) in gearing type M

Setting range：0～4095

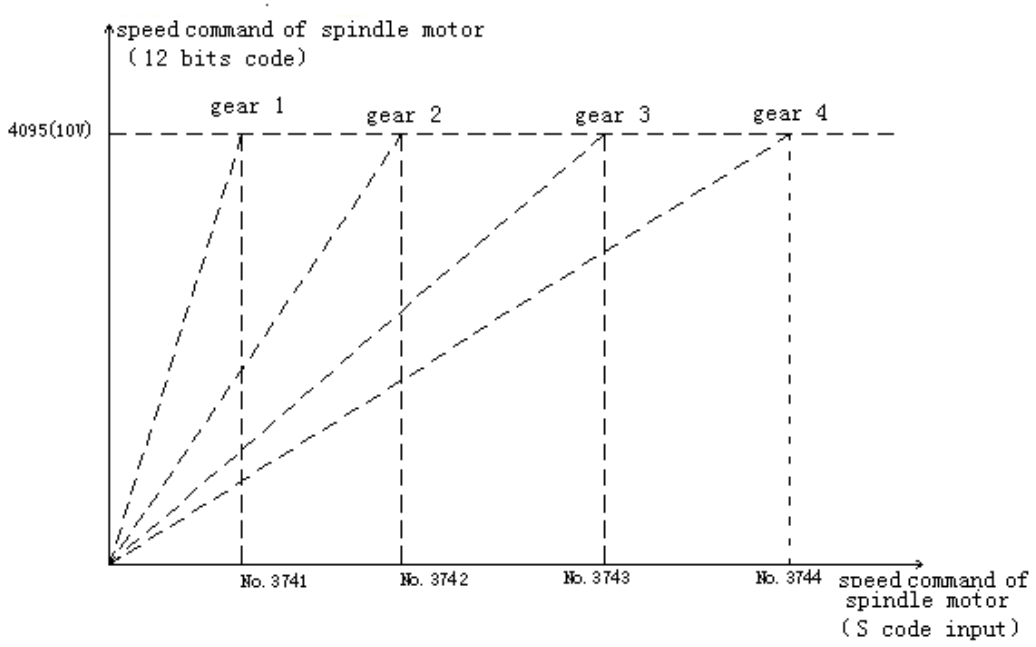
M gear shift type B mode:



M type gear shift A mode:



T type gear shift ：



3 7 6 0	CS control axis (0:CS control invalid; 1:X axis; 2:Y axis; 3:Z axis; 4:4th axis; 5:5th axis)
3 7 6 1	Spindle analog voltage output (0:invalid ; 1:CN15; 2:CN16; 11~14:extended I/O unit)
4 9 0 0	Max. spindle speed fluctuation allowed by system

Setting range：50～1000（Unit:r/min）

4 9 0 1	Feed stop alarm time to the spindle stopping(ms)

4 9 0 2	Time of detect SAR sig.(ms)

4 9 6 0	G76,G87 Spindle orientation M code selection

Setting range：13～100

5 0 1 2	Max. permissible angle error when the angle between two lines of radius compensation is 0 °or 180°.

Setting range：IS_B： 0～1000(Unit:0.001°)
IS_C: 0～10000(Unit:0.0001°)

5 0 2 5	initial tool No.

Setting of initial tool number must be less than that of max. tool number, otherwise, an alarm occurs.
Setting range：0～99

5 0 2 6	Max. tool No.

Setting of max. tool number must be more than that of initial tool number, otherwise, an alarm occurs.
Setting range：0～99

5 0 3 2	The number of PLC D data is corresponding to the number of start tool sheath

Setting range：300～999

5 0 3 3	The number of PLC D data is corresponding to the number of exchanging tool sheath

Setting range：300～999

5 0 4 0	The value of the measuring speed in G37

5 0 4 3	The γ value of the slowdown disdance from the absolute coordinante given by G37
5 0 4 4	The ε value of the measure disdance from the absolute coordinante given by G37

5 1 0 8	Fixed loop return wait time(ms)

Setting range：0～9999999

5 1 1 0	orientation of interval drilling cut moving speed

5 1 1 4	Retraction amount of G73 high deep hole drilling cycle

Setting range：0～32767000 (Unit:0.001mm),

5 1 1 5	Cutting initial point of G83 high deep hole drilling cycle

Setting range：0～32767000 (Unit:0.001mm),

5 1 2 2	G110,G111,G134,G135 Lead of helical tool infeed

Setting range：0～999999（unit 0.001mm）

If setting value is less than 10, helical feeding is invalid for rough milling command G110, G111, G134, G135, and it feeds by linear type.

If setting value is more than or equal to 10, it feeds by helical type for rough milling command G110, G111, G134, G135.

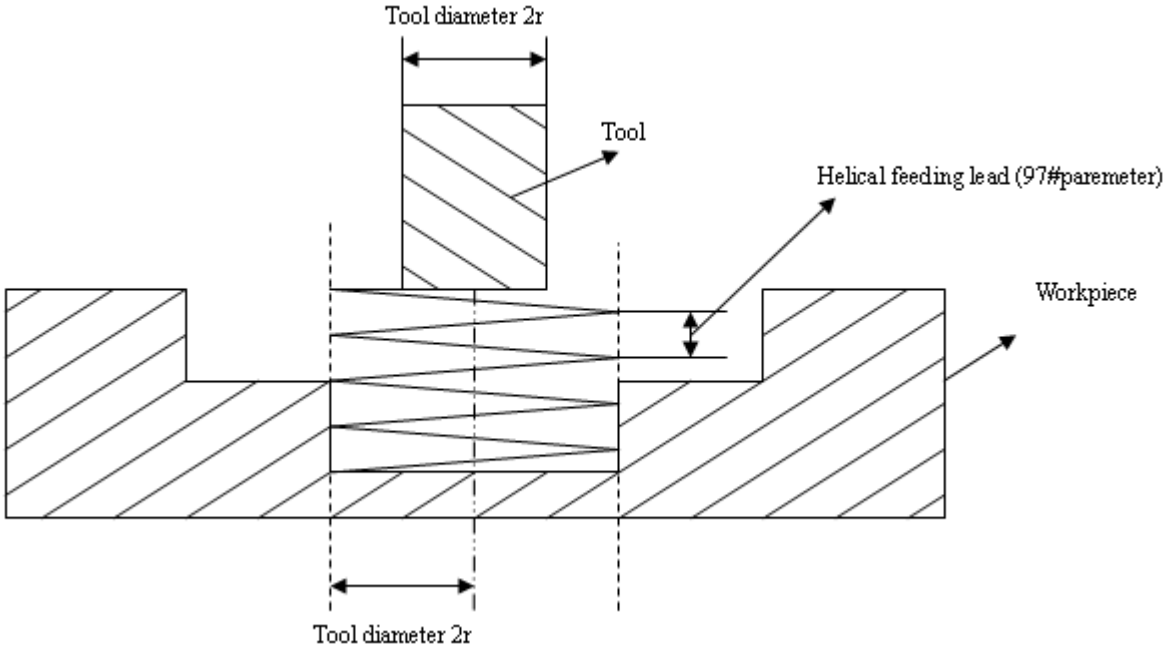
Rough milling command（G110,G111,134,G135）helical feed function:

Namely, for Z axis depth cutting of rough milling command G110, G111, 134, G135, the tool feeds not by linear type, but by helical type. So the workpiece with no groove may be rough milled directly.

Note 1: when the Z axis cutting depth is less than 10μm each time, the helical feeding is invalid.

Note 2: when the tool radius is less than 1mm, the helical feeding is also invalid.

The helical feeding path is shown in follows:



5 2 1 1	Override value in rigid tapping tool retract(0: override is set to 100%)

Setting range：0～200, 0: override is set to 100%

5 2 1 3	Tool retract amount in deep hole rigid tapping(high-speed, standard)

Setting range：0～32767000（Unit：0.001mm）

5 2 1 6	Spindle move amount per revolution in rigid taping

Setting range：1～9999999（Unit :0.001mm）

5 2 2 1	Multiplier coefficient of rigid taping spindle command(gear 1(rpm))
5 2 2 2	Multiplier coefficient of rigid taping spindle command(gear 2(rpm))
5 2 2 3	Multiplier coefficient of rigid taping spindle command(gear 3(rpm))
5 2 2 4	Multiplier coefficient of rigid taping spindle command(gear 41(rpm))

Setting range： 1～32767

5 2 3 1	Division coefficient of rigid taping spindle command(gear 1(rpm)
5 2 3 2	Division coefficient of rigid taping spindle command(gear 2(rpm)
5 2 3 3	Division coefficient of rigid taping spindle command(gear 3(rpm)
5 2 3 4	Division coefficient of rigid taping spindle command(gear 4(rpm)

Setting range： 1～32767

5 2 6 1	Linear acc.&dec. time constant in rigid tapping tool infeed

Setting range：10～10000（Unit：ms）

5 2 7 1	Linear acc.&dec. time constant in rigid tapping tool retract

Setting range：0～4000（Unit：ms）, 5261 setting value is used when it is set to 0.

5 3 2 1	Backlash comp of rigid taping spindle command(gear 1)
5 3 2 2	Backlash comp of rigid taping spindle command(gear 2)

5 3 2 3	Backlash comp of rigid taping spindle command(gear 3)
5 3 2 4	Backlash comp of rigid taping spindle command(gear 4)

Setting range：0～4095

5 4 1 0	Rotation angle of coordinate rotation without R

Setting range：-360000～360000 （Unit :0.001deg）

5 4 1 1	If NO.562 XSC is 0,scale rate is this par. without P

Setting range：1～99999999；

5 4 2 1	If NO.562 XSC is 1,scale rate is this par.

Setting range：-99999999～99999999，moreover it isn't zero；

6 0 4 4	Call starting codes of M code of the subprogram

Setting range: 3~8999

6 0 4 5	Call the starting program No. of a subprogram through M code

Setting range：0~9999

6 0 4 6	The quantity of M code of the called subprogram

Setting range：0~8000

When specify several subprograms call based on the M code at one time, set by data parameters №6044, №6045, №6046. This call is invalid when the data parameter is set to 0.

[Example] When data parameters №6044=2000, №6045=300, №6046=100 are set,

M2000 → O300

M2001 → O301

M2002 → O302

:

M2099 → O399

Specify the subprogram call of the above listed 100 groups.

Note

1. If the system satisfies the following conditions, the calls based on this setting are invalid.

① The value exceed the data scope is set in parameter

② When (№6045+(№6046-1)>9999

2. M execution explanations:

Set called M range	Execution result
M00, M01, M02, M29, M30, M98, M99	The system executes basic M function instead of M calling.
M9000～M9999	Call a macro program and the called program number is the command value of M code.
Other M codes	The system executes M calling subprograms instead of basic M function.

8 0 1 0	Setting the PLC control axis DI/DO channel 1~4 of each axis

Setting range：0～4, When set to 0, the channel is invalid.

Chapter 4 Machine Debugging

The trial run methods and steps at initial power on for this GSK980MDi Plus are described in this chapter. The corresponding operation can be performed after the debugging by the following steps.

4.1 ESP & Limit

4.1.1 ESP & Limit Series-Connection

The connection is as follows (3 axes):

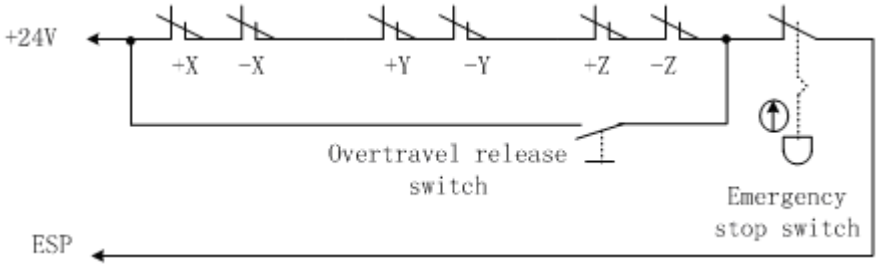


Fig. 4-1

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):

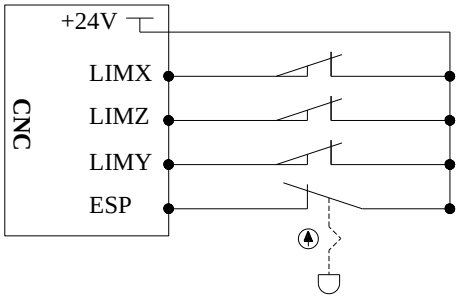


Fig. 4-2

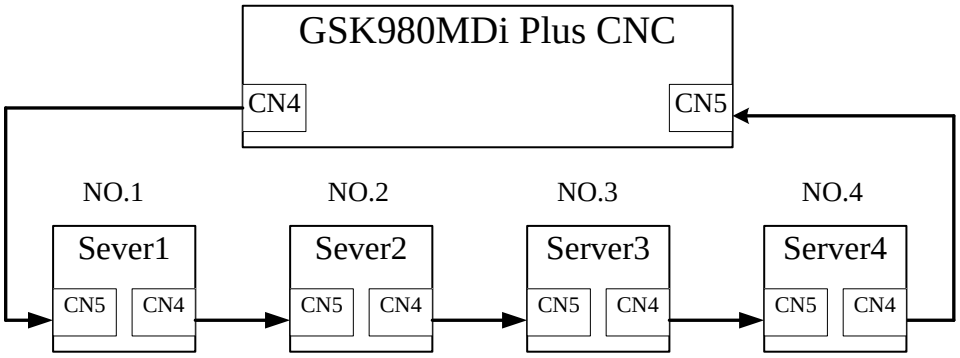
The Bit 3 of status parameters №172 equals to 0: ESP function is enabled;

PLC parameter K15: The overtravel function valid/invalid.

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

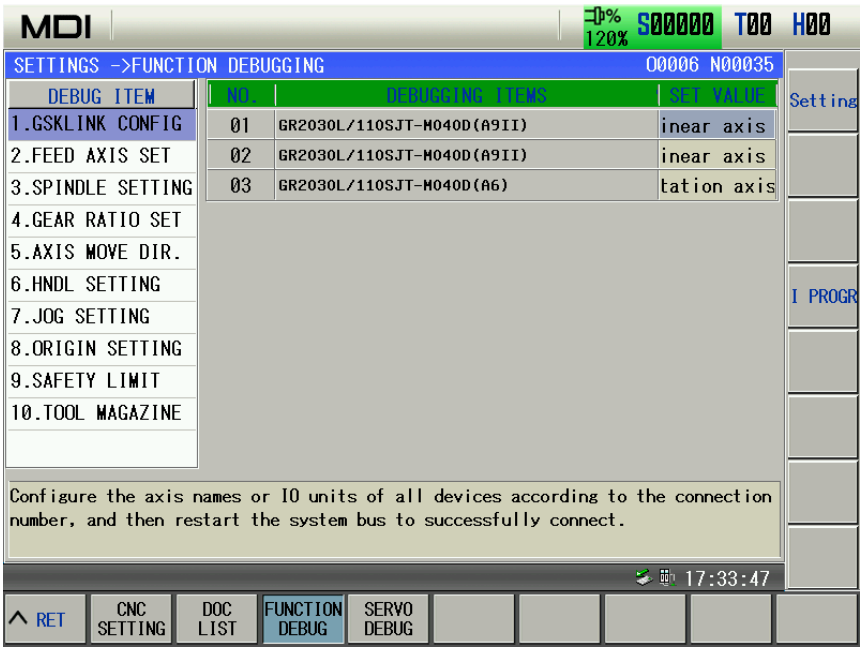
GSK980MDi Plus Connection example：



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->GSK-Link” based upon the 2-level authorities.

Under the 2-level or more authorities, enter to the setting page by 

, and then to the GSK-Link setting page by”Function Debuging”, the display is shown below:



Select the servo drive by  or  button, set the selected corresponding axis of the servo drive by 

: 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).

4.3 Backlash Offset

The backlash offset is input by diameter value with the unit 0.001mm, which is irrelevant to the programming by diameter or by radius. It can be measured by a dial indicator,a micrometer or a

laser detector. Because the backlash offset can improve the machining precision only by accurate compensation, it is not recommended to measure it in MPG or Step mode, but the following method is suggested:

- Program editting
  O0001;
  N10 G01 Z10 F800 G91 ;
  N20 Z15 ;
  N30 Z1 ;
  N40 Z-1 ;
  N50 M30 .
- Set the backlash error offset to 0 before measuring:
- Run the program by single blocks, search the measuring benchmark A after 2 positioning operations, record the current data, move 1mm in the same direction, then move 1mm reversely to point B, read the current data.

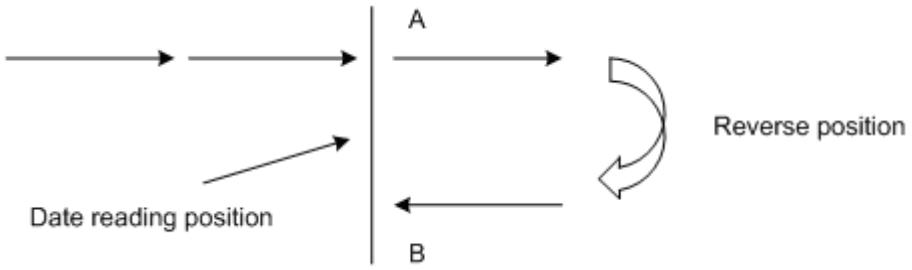


Fig. 4-3 Schematic map of backlash measuring methods

Backlash error offset value =| data of point A –data of point B |.Input the calculated data to the CNC data parameter №034. Calculation for other axes are the same as this.

Data A ： dial-indicator data at point A

Data B ： dial-indicator data at point B

Chapter 5 Diagnosis Message

5.1 CNC Axis Motion Status and Data Diagnostic Information

0	4	8	Number of pulses output by X-axis CNC
0	4	9	Number of pulses emitted by the X-axis bus
0	5	0	Number of pulses received by X-axis servo (servo command position)
0	5	1	Position of X-axis motor encoder
0	5	4	Y-axis CNC output pulse count
0	5	5	Number of pulses emitted by the Y-axis bus
0	5	6	Number of pulses received by Y-axis servo (servo command position)
0	5	7	Y-axis motor encoder position
0	6	0	Z-axis CNC output pulse count
0	6	1	Number of pulses emitted by Z-axis bus
0	6	2	Number of pulses received by Z-axis servo (servo command position)
0	6	3	Position of Z-axis motor encoder
0	6	6	4th axis CNC output pulse count
0	6	7	Number of pulses emitted by the 4th axis bus
0	6	8	4th axis servo receiving pulse number (servo command position)
0	6	9	4th axis motor encoder position
0	7	2	5th axis CNC output pulse count
0	7	3	Number of pulses emitted by the 5th axis bus
0	7	4	5th axis servo receiving pulse number (servo command position)
0	7	5	5th axis motor encoder position
0	8	6	Value of X-axis reverse clearance compensation
0	8	7	Value of Y-axis reverse clearance compensation
0	8	8	Value of Z-axis reverse clearance compensation
0	8	9	The value of reverse clearance compensation for the 4th axis
0	9	0	Value of 5th axis reverse clearance compensation
0	9	1	Value of 6th axis reverse clearance compensation
0	9	2	Value of X-axis pitch error compensation
0	9	3	Value of Y-axis pitch error compensation
0	9	4	Value of Z-axis pitch error compensation
0	9	5	Value of 4th axis pitch error compensation
0	9	6	Value of 5th axis pitch error compensation
0	9	7	Value of 6th axis pitch error compensation
2	0	0	Spindle 1 speed (r/min)
2	0	1	Spindle 2 speed (r/min)
2	0	2	Spindle 3 speed (r/min)
2	1	0	Output value of spindle 1 (analog voltage in mV when simulating spindle, speed in 0.01rpm when serial spindle)
2	1	1	Output value of spindle 2 (analog voltage in mV when simulating spindle, speed in 0.01rpm when serial spindle)
2	1	2	Output value of spindle 3 (analog voltage in mV when simulating spindle, speed in 0.01rpm when serial spindle)

2	2	0	Actual rotation direction of spindle encoder 1 (rear cover interface CN21)
2	2	1	Count value of spindle encoder 1 (rear cover interface CN21)
2	2	2	Actual rotation direction of spindle encoder 2 (rear cover interface CN22)
2	2	3	Count value of spindle encoder 2 (rear cover interface CN22)
5	0	0	Hand wheel 1 speed (r/min) (rear cover hand pulse interface CN31)
5	0	1	Hand wheel 2 speed (r/min) (rear cover hand pulse interface CN32)
5	0	2	Handwheel 3 speed (r/min) (operation panel interface CN331+CN332)
5	1	0	Handwheel interface 1 count Value (rear cover hand pulse interface CN31)
5	1	1	Handwheel interface 2 count Value (back cover hand pulse interface CN32)
5	1	2	Handwheel interface 3 count Value (operation panel interface CN331+CN332)
6	0	1	PLC execution time (ms)
6	0	2	Total system running time (h)

Chapter 6 Screw-Pitch Compensation

6.1 Screw-Pitch Compensation

0	0	0	3

***	***	PCOMP	***	***	***	D/R	***

PCOMP =1: Screw-pitch error compensation valid;

=0: Screw-pitch error compensation invalid.

3	6	2	1	Min. position number of each axis for pitch error compensation
3	6	2	2	Max. position number of each axis for pitch error compensation
3	6	2	0	Screw-pitch error compensation position number of each axis machine zero

3	6	2	4	Interval of each axis screw-pitch error compensation

Setting range：1000～999999 (Unit:0.001mm)

6.2 Compensation Value

The axes screw-pitch offset values are set in the page of screw-pitch parameter. Refer to the following table.Input range: -2550~2550.The offset value is input by diameter with the unit 0.001mm. （Take X, Y, Z axes as example）

Offset No.	X	Y	Z
000	…	…	…
001	5	-2	3
002	-3	4	-1
…	…	…	…
199	-2	3	1
…	…	…	…

6.3 Cautions for Offset Setting

① The setting and modification of screw-pitch offset can only be done at the authority of password level 2 and switch on parameter switch.

② Offset is not allowed if the offset interval entered is 0.

③ After the parameter of screw-pitch offset is set, only the machine zero is returned could the compensation be done.

6.4 Examples of Offset Parameters Setting

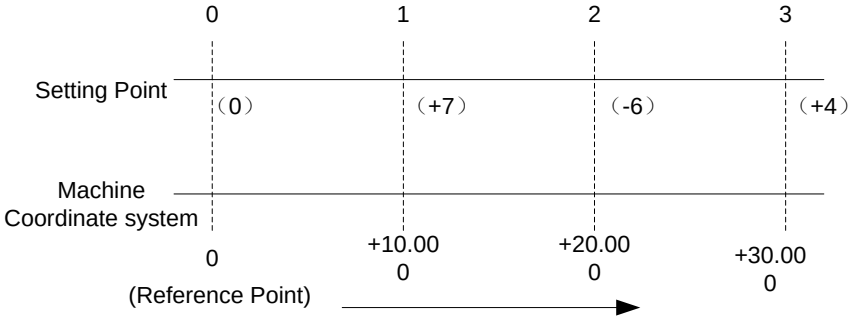
① parameter №3620（screw-pitch error origin point）=0, Data parameter №3624（screw-pitch offset interval）=0,

Parameter №3621（min. position number of each axis for pitch error compensation）=0，parameter№3622（max. position number of each axis for pitch error compensation）=199

When the screw-pitch error origin is set to 0: The offset value for the 1st section is set in screw-pitch

compensation parameter list №0001, the offset value for the 2nd section is set in screw-pitch compensation parameter list №0002, and the offset value for the Nth section is set in screw-pitch compensation parameter list №(0000+N).

The machine zero is regarded as the reference point of screw-pitch error origin point; it begins to compensate the position №0001 in the offset table from the machine zero. So the screw-pitch error compensation can only be performed in the positive moving of the machine zero coordinate system. So, at the moment, pitch error origin 0 is invalid when reverse motion is done even if it is valued.



The position No.0000 in the offset table corresponds to the reference point (i.e screw-pitch error origin 0), the offset point 1 corresponds to a point 10.000 positive moving from this reference point, and there is a compensation point from this point every 10.000 distance. The 127th compensation point is the offset value at position 1270.000. Therefore, at compensation point 1, set an compensation value moving from 0 to 10.000, at offset point 2, set an offset value moving from 10.000 to 20.000. At offset point N, set an offset value moving from (N-1) × (offset clearance) to N × (offset clearance).

Above is the example of following offset interval errors：

Offset clearance	Offset value
0～10.000	+7
10.000～20.000	-6
20.000～30.000	+4

Machine coordinate system	Offset parameter No.	Offset value	Drive unit current command pulses before offsetting	Drive unit current command pulses after offsetting
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 reference point to the point of +30.000, the screw-pitch compensation is: (+7)+(-6)+(+4)=(+5)

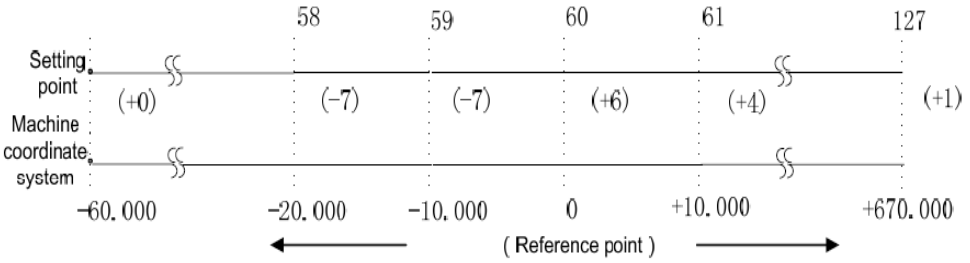
② Data parameter№3620（screw-pitch error origin）=60，№03624（compensation interval）=10.000 Parameter №3621（min. position number of each axis for pitch error compensation）=0，parameter№3622（max. position number of each axis for pitch error compensation）=199

When the screw-pitch error origin is set to 60: For the positive moving, the compensation value for the

1st section is set by the position №0061 in the compensation table. The compensation value for the 2nd section is set by the position №0062 in the compensation table. The compensation value for the Nth is set by position №0060+N in the compensation table.

For the negative moving, the 1st section error compensation is set by position №0060 in the compensation table, the 2nd section by position №059. The Nth section error compensation is set by position №0060-N in the compensation table.

By taking the machine zero as the reference point, the screw-pitch error origin moves from the positive coordinate system of machine zero to compensate the corresponding position No.0061 in the compensation table, and from the negative coordinate system to compensate the position No.0060. Therefore the screw-pitch compensation can be done when moving in the positive or the negative coordinate system of machine zero.



The position No.0060 in the screw-pitch error compensation parameters corresponds to the reference point (60), compensation point 61 to a point positive10.000 moving from origin. So there is a compensation point every 10.000 distance. The 127th offset point is the compensation at position +670.000. While the compensation point 59 corresponds to a point negative 10.000 moving from reference point. Also there is a compensation point every 10.000 .The offset point 0 is the compensation value at -600.000 position. Therefore, at compensation point N, set a compensation value when moving from (N-61) × (compensation interval) to (N-60) × (compensation interval).

Above is the example of following compensation interval errors

Offset interval	Offset value
0～10.000	+4
-10.000～0	+6
-20.000～-10.000	-7
-30.000～-20.000	-7

Machine coordinate system	Offset parameter No.	Offset value	Drive unit current command pulses before offsetting	Drive unit current command pulses after offsetting
-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	…		

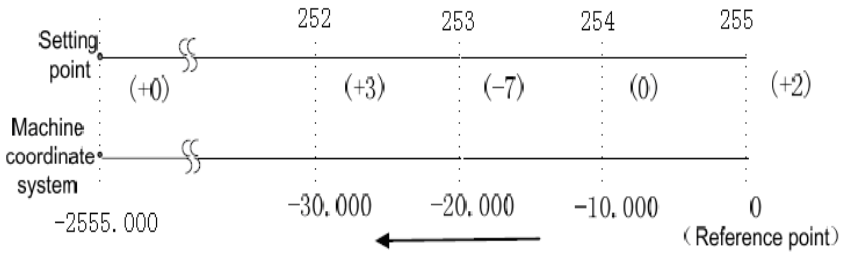
Actually the machine moves from -30.000 point to the point of +10.000, the screw-pitch compensation

is: (-7)+(-7)+(+6)+(+4)=(-4)

③ Data parameter №3620（screw-pitch error origin）=255，№3624（compensation interval）=10000 Parameter №3621（min. position number of each axis for pitch error compensation）=0，parameter№3622（max. position number of each axis for pitch error compensation）=255

When the screw-pitch error origin is set to 255: The compensation value for the 1st section is set by the position №0255 in the compensation table, the compensation value for the 2nd section is set by the position №0254 in the compensation table, and the compensation value for the Nth section is set by the position №0256-N in the compensation table.

The machine zero is regarded as the reference point of screw-pitch error origin. It begins to compensate the position №255 in the compensation table from the machine zero. So the screw-pitch error compensation can only be done in the negative moving of the machine zero coordinate system.



The compensation point 254 corresponds to a point moving 10.000 in negative direction from the reference point. There is a compensation point every -10.000 distance. Therefore, set an offset value moving from 0 to -10.000 at compensation point 255; set an offset value moving from -10.000 to -20.000 at offset point 254. At compensation point N, set an offset value moving from (N-256)×(compensation interval) to (N-255)×(compensation interval).

The above is the example of following compensation interval errors:

Compensation interval	Compensation value
0～-10.000	+2
-20.000～-10.000	0
-30.000～-20.000	-7
-40.000～-30.000	+3

Machine coordinate system	Compensation parameter No.	Compensation value	Drive unit current command pulses before offsetting	Drive unit current command pulses after offsetting
Reference point 0			0	0
-10.000	255	2	10000	10002
-20.000	254	0	20000	20002
-30.000	253	-7	30000	29995
-40.000	252	3	40000	39998

Actually the machine moves from the point -40.000 to the reference point, the screw-pitch compensation is: (+3)+(-7)+(0)+(+2)=(-2)

④ Pitch error compensation of rotary axis

When the additional axis is linear or rotary (B type), the pitch error compensation mode is the same as

the linear axis. The following example explains the pitch error compensation function when the additional axis is taken as the rotary axis (A type).

No. 1260 (movement amount of rotary per rev) = 360，№3624（pitch error compensation interval）= 45

№3620（pitch error compensation number corresponded to machine zero）= 60

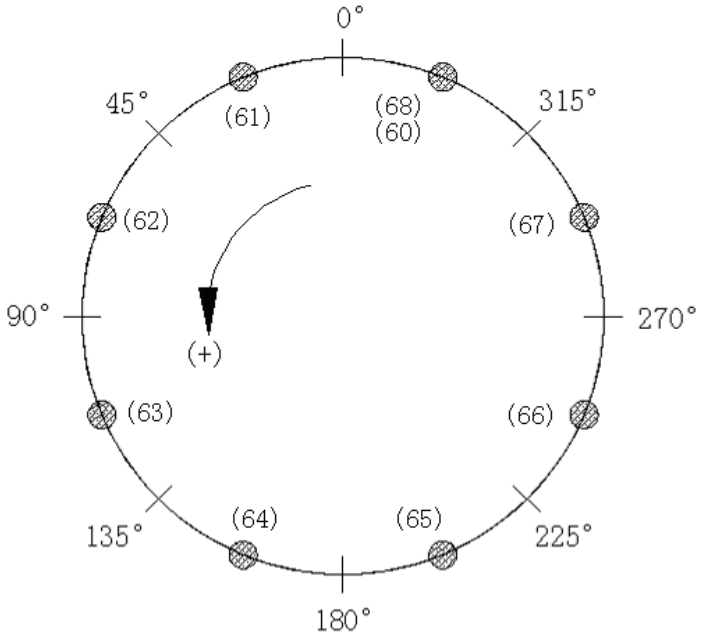
Set the minimum pitch error compensation position number No. 3621 of the rotary axis=50, the maximum pitch error compensation position number No. 3622=100

After the above parameters are set, the farthest pitch error compensation position number of the rotary axis in the negative direction is equal to compensation position number of reference point.

The farthest compensation position number in the positive direction is:

Compensation position number of reference point+（movement amount per rev/compensation position interval）= 60+360/45 = 68

Relations between machine coordinates and compensation position number:



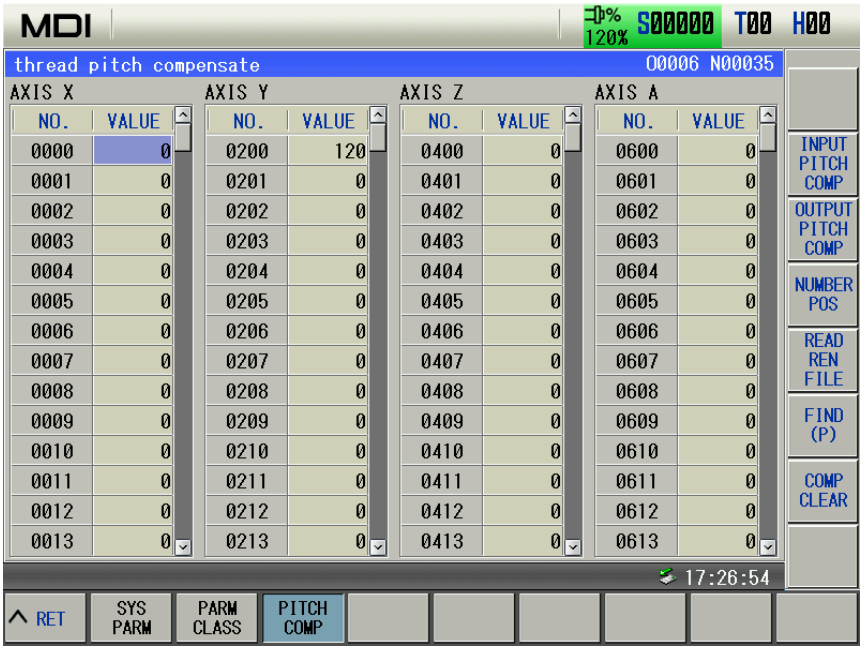
When compensation sum of the position 60～68 is not 0, a position error occurs. The sum is an accumulation of pitch error compensation value per rev. the same compensation values must be set at the position 60 and 68 as follows:

Compensation position number	60	61	62	63	64	65	66	67	68
Set compensation value	+1	-2	+1	+3	-1	-1	-3	+2	+1

6.5 Import REN Format Screw File

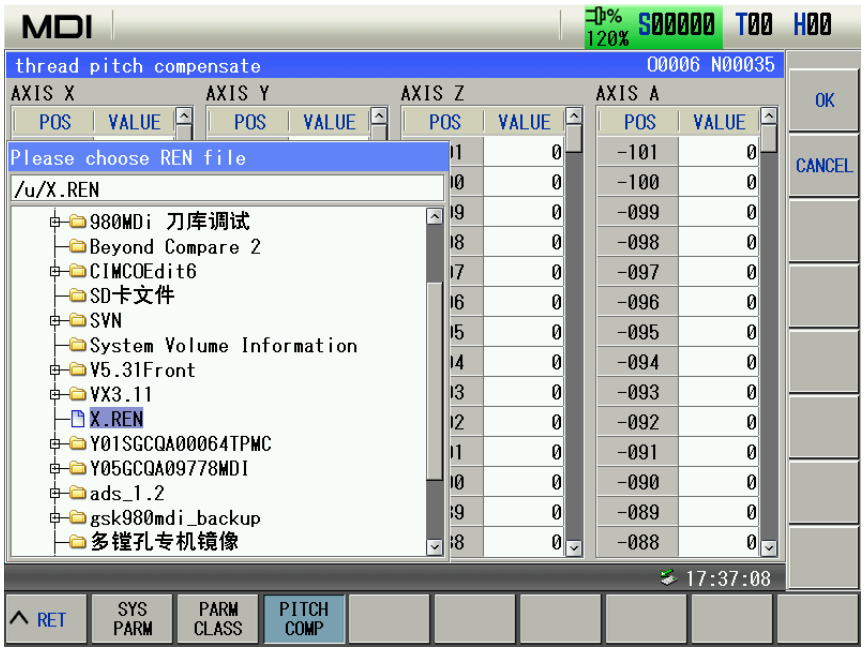
6.5.1 Import Page

Insert the USB drive into the system, turn on the parameter switch and enter the level 2 permission password. On the pitch compensation page, import the thread compensation file as shown in the following figure:

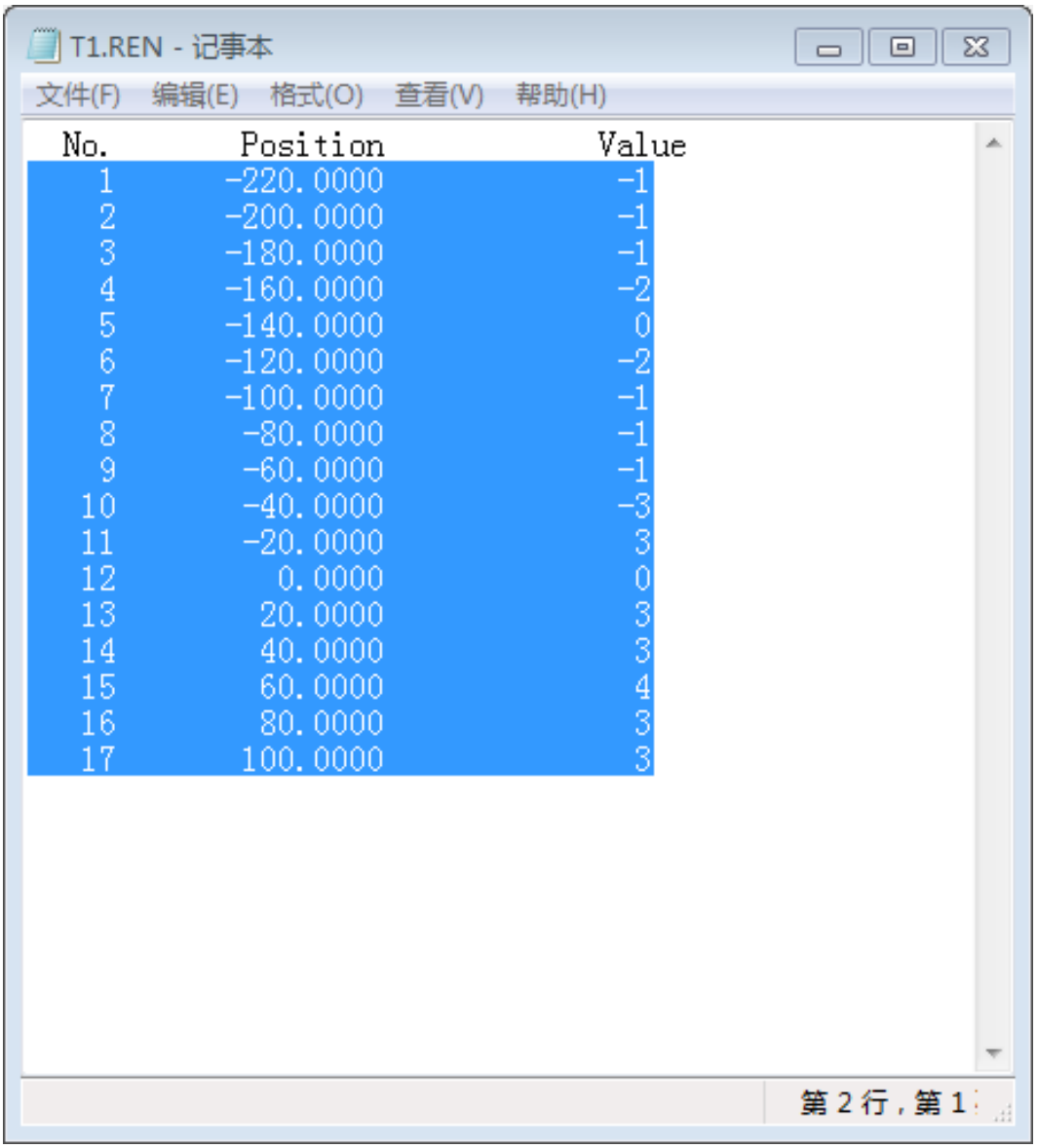


6.5.2 Import Operation

Press "Import REN" to pop up the file selection dialog box, select the REN file that needs to be imported, and then press "OK" to write the data in the REN screw file to the currently selected axis, as shown in the following figure:



6.5.3 REN Example of Screw Repair File



Instructions:

1) The cursor selection section in the file is mandatory data, while other data is negligible and will not affect the import of screws.
2) The rotation axis does not support importing screw files.
3) When there is reverse gap data in the file, if the data value obtained by superimposing the reverse gap data with the existing reverse gap compensation value is greater than 0, the superimposed value will be written into the reverse gap parameter. Otherwise, the reverse gap parameter will not be modified.

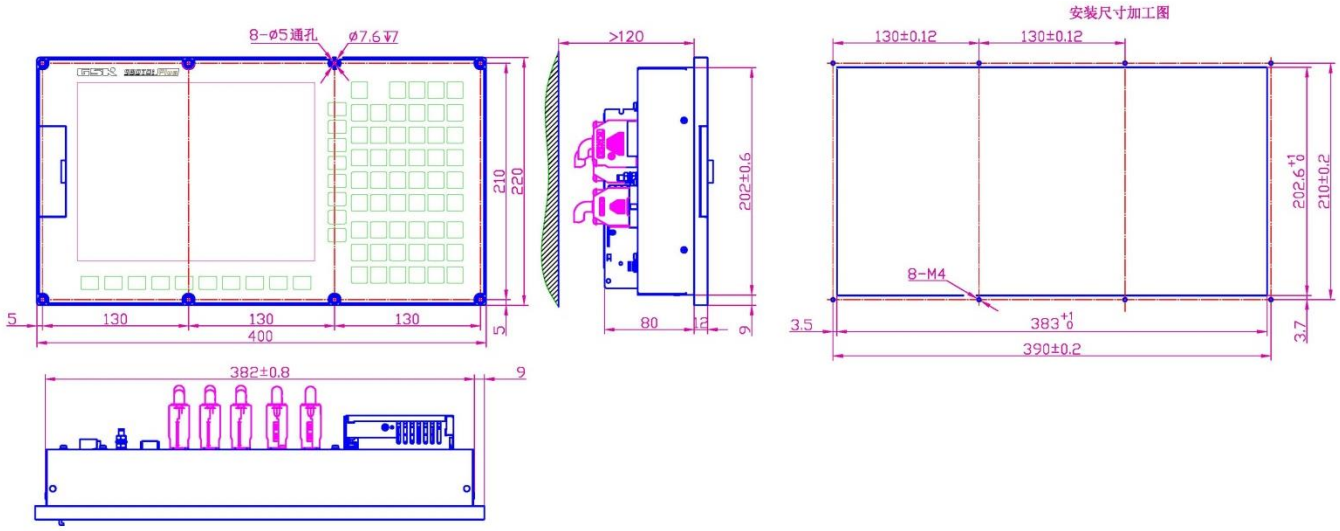
Appendix

Appendix

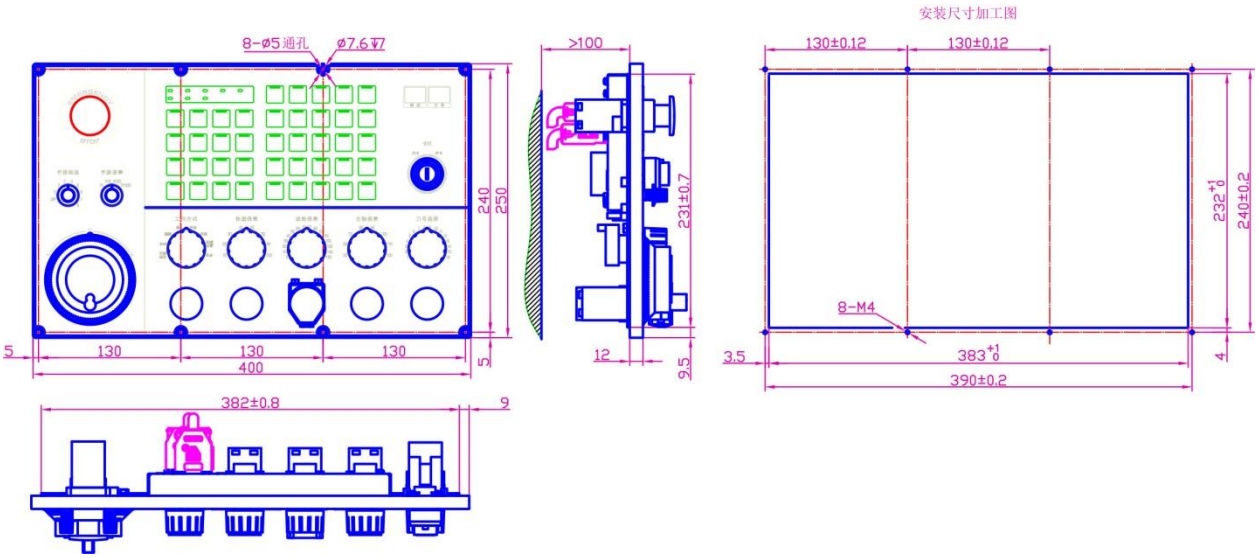
Appendix 1 GSK980MDi Plus Series External Dimensions

1.1 GSK980MDi Plus-H External Dimensions

- Host



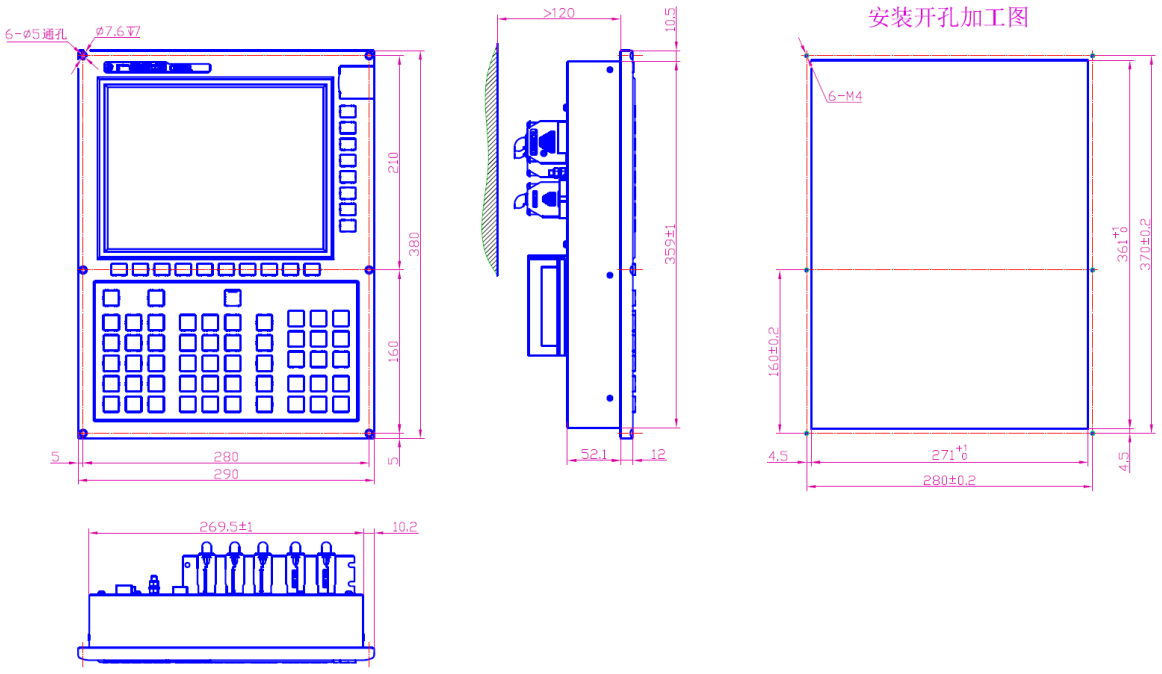
- Operator Panel



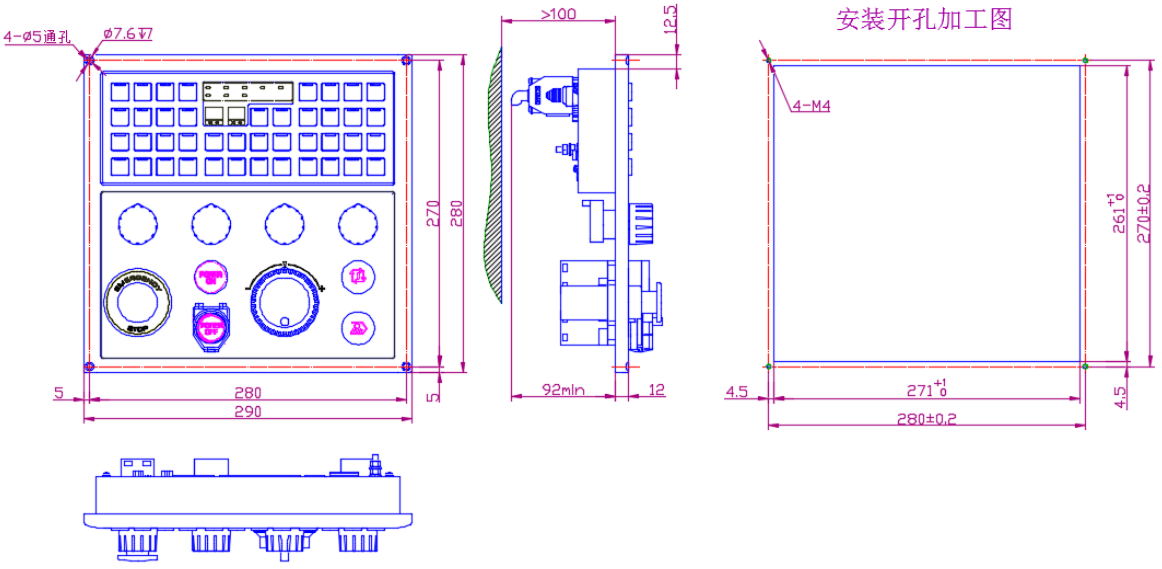
Appendix

1.2 GSK980MDi Plus-V External Dimensions

- Host



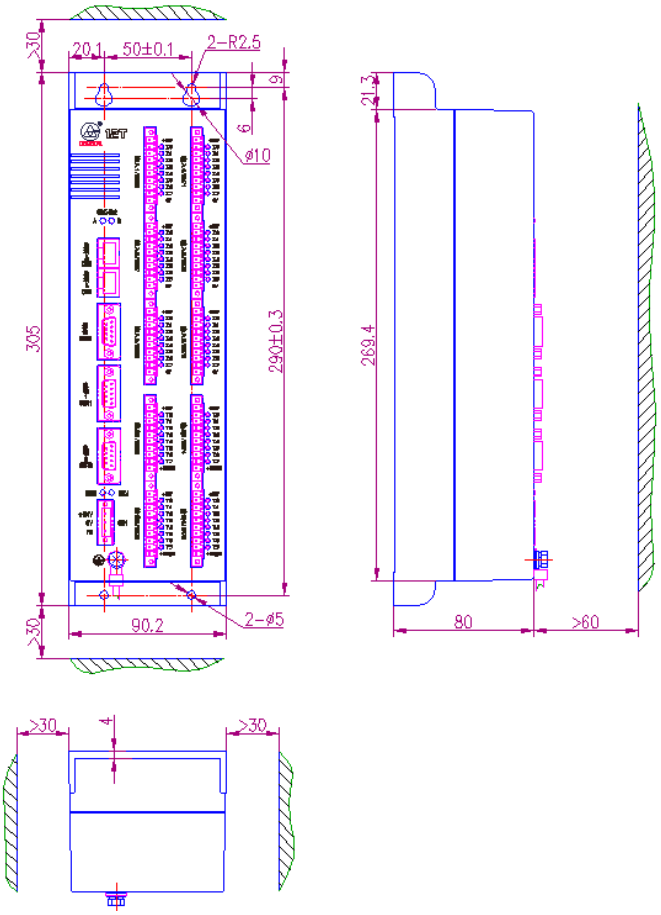
- Operator Panel



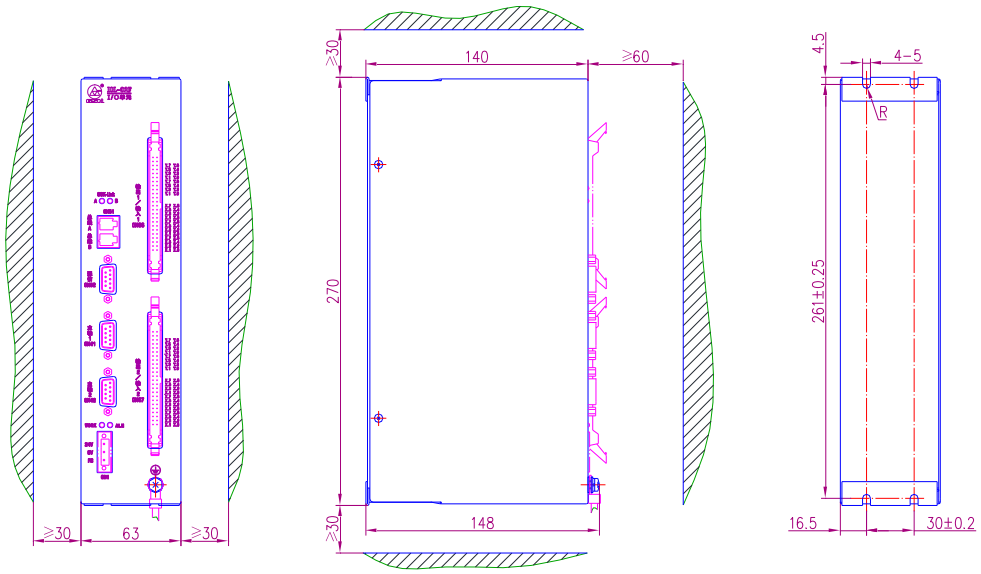
Appendix

Appendix 2 Diagram of I/O Deconcentrator

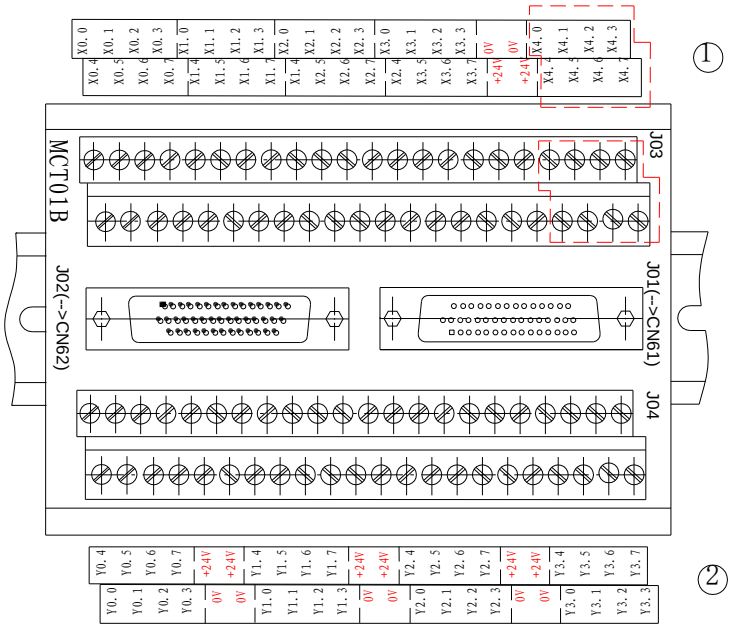
2.1 IOR-04T / IOR-44T



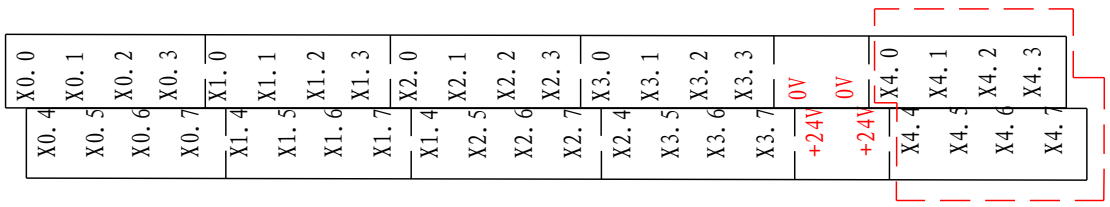
2.2 IOR-44F



2.3 MCT01B

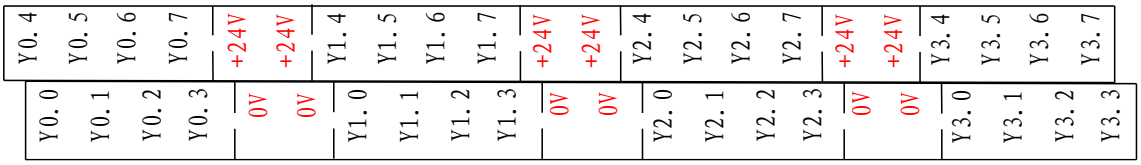


① The enlarged diagram is as follows: (The part with dotted line is invalid)

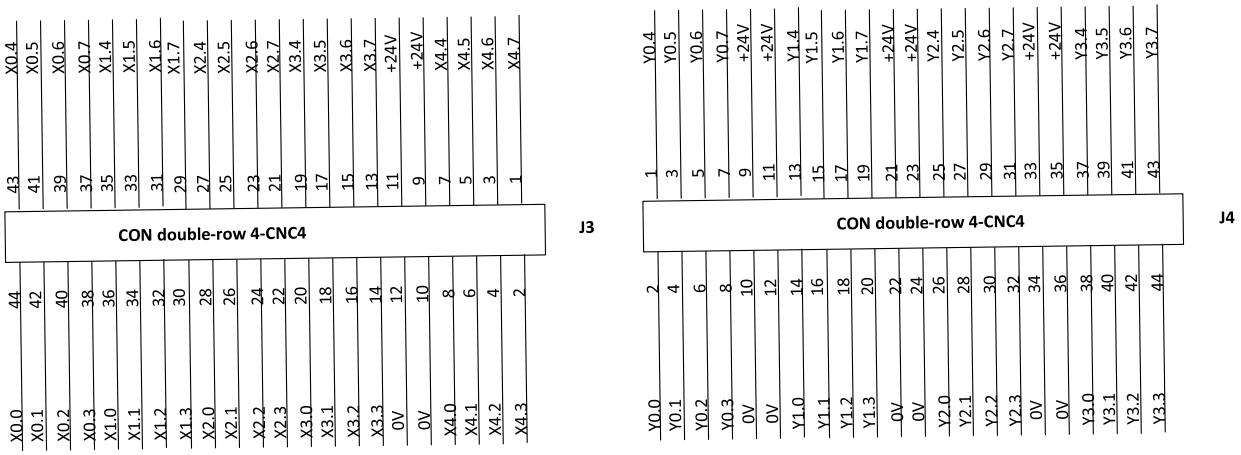


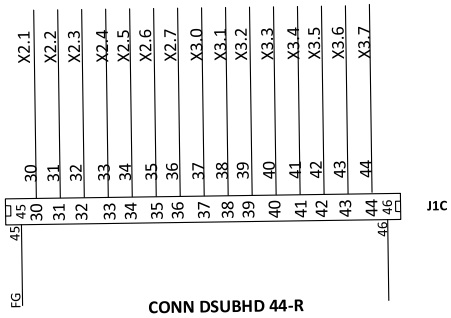
② The enlarged diagram is as follows:

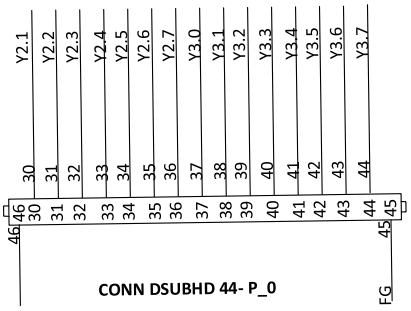
Appendix

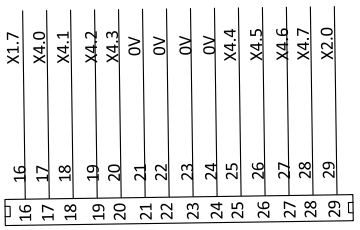


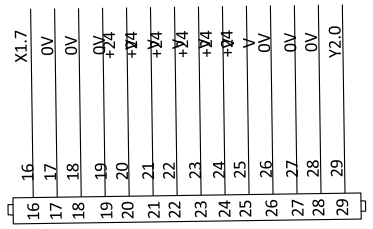
Circuit diagram is as follows:

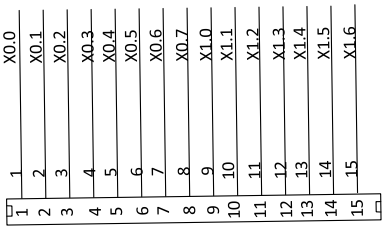


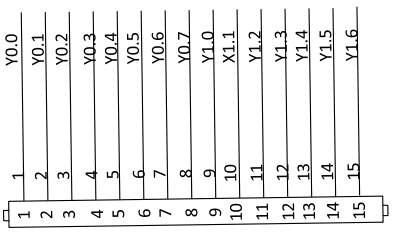






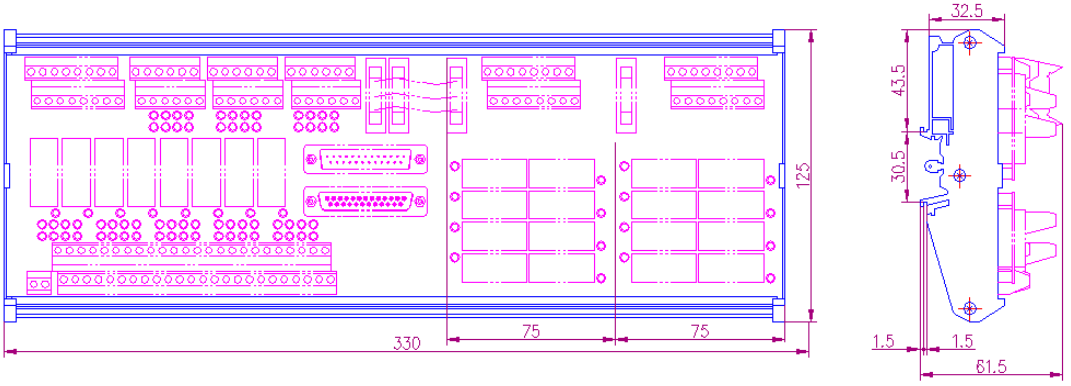






2.4 MCT06-R

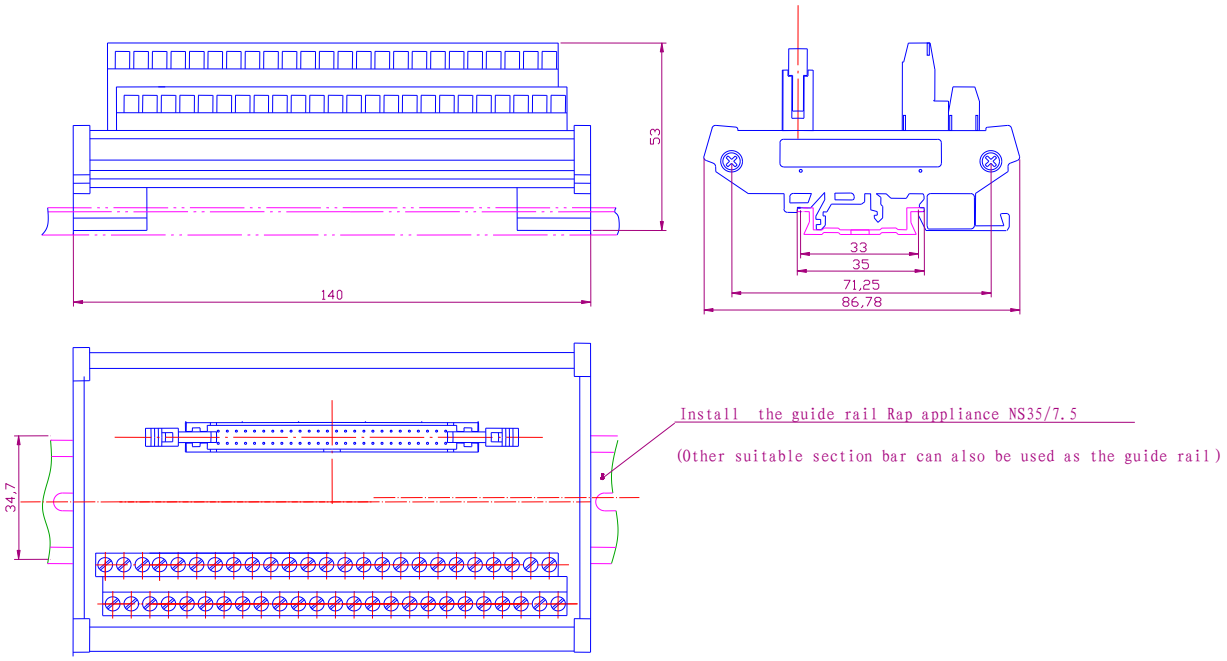
For installation, debugging and maintenance of the product, our company has already developed the brand-new I/O deconcentrator MCT06-R with the relay to reduce the faults, such as the product running abnormal and the component damage, etc due to the incorrect connection during installation and connection. 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.



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**）

2.5 MCT07

MCT07 Suit the IOL-10F expand I/O unit.



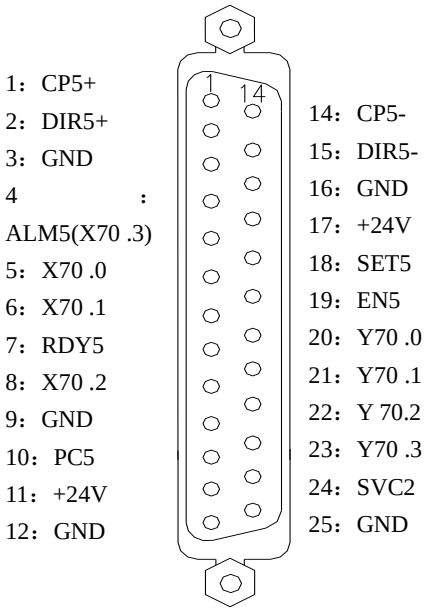
Appendix

2.6 IOR-20TP(Bus to Pulse I/O Unit)

By converting the IOR-20TP bus to a pulse I/O unit, the GSK980MDi Plus can adapt to spindle servo drives with pulse interfaces.

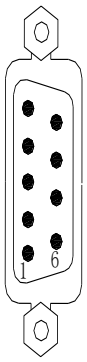
- **Interface address**

  The interface of the bus to pulse I/O unit is as follows (the spindle interface is consistent with that of GSK980MDc):



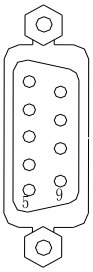
CN22 pulse spindle interface (25 core DB hole)

Signal	Explanation
CP5+、CP5-	5th axis pulse signal
DIR5+、DIR5-	5th axis direction signal
ALM5（X70.3）	5th axis/main axis alarm signal
RDY5	Axis 5 ready signal
PC5	Zero point signal of the 5th axis
SVC	Analog voltage output
SET5	5th axis pulse prohibition signal
EN5	5th axis enable signal



CN41 simulation spindle interface 1 (9-core DB pin)

Standard ladder diagram function definition		
Address	**symbol**	**function**
X70.0	VPO	Spindle speed/position status in place signal
X70.1	SAR	Speed arrival signal
X70.2	COIN	Spindle positioning completed
X70.3	SPAL	Spindle alarm signal
Y70.0	VP	Spindle speed/position switching signal
Y70.1	TAP	Rigid tapping signal
Y70.2	SRV	The spindle rotates counterclockwise
Y70.3	SFR	The spindle rotates clockwise



CN52 communication interface (9-core DB hole)

Appendix

Instructions:

1) The X and Y addresses in the CN22 pulse axis interface start from X70 and Y70, while other X and Y addresses on the IO unit start from X71 and Y71.
2) The CN41 simulated spindle interface and CN52 communication interface are reserved interfaces and will not be used temporarily.

- **Related parameters**

6	0	2

LTOP							

LTOP =1: The pulse conversion function of each spindle bus is effective;

=0: The pulse conversion function of each spindle bus is invalid.

3	7	1	0
3	7	1	2
3	7	6	0

Number of spindles (1-3)
Spindle type (0: gear spindle; 1: analog spindle; 2: serial spindle)
The feed axis associated with each Cs spindle (0: Cs axis is invalid; 1: X-axis; 2: Y-axis; 3: Z-axis; 4: 4th axis; 5: 5th axis)

0	0	9

***	***	***	ALM5	ALM4			

ALM4 =1:4th axis alarm signal is a low-level alarm;

=0:4th axis alarm signal is a high-level alarm.

ALM5 =1:5th axis alarm signal is a low-level alarm;

=0:5th axis alarm signal is a high-level alarm.

0	0	5

			ABP5	ABP4			

ABP4 =1:4th axis pulse output in two orthogonal directions (needs to be restarted);

0:4th axis pulse output in (pulse+direction) (needs to be restarted).

ABP5 =1:5th axis pulse output in two orthogonal directions (needs to be restarted);

0:5th axis pulse output in (pulse+direction) (needs to be restarted).

- **Attention to issues**

1) The spindle in the 980MDi Plus needs to be set as a serial spindle and the CS function is valid before the bus to pulse function can be used.
2) When using the bus to pulse IO unit, it is not necessary to set the remote IO (parameter 701.0) to be valid.
3) When using directional control, the OSTA directional signal of the driver needs to be connected to Y3.0 (pin 37) of CN62.
4) When the pulse spindle needs to return to zero, the encoder should be connected to the IO unit encoder interface (CN21), and the electronic gear ratio should be set to the system parameters. The return to zero method D should be used.

- **Reference wiring diagram (IOR-20TP with GS3000 economical spindle servo):**

CNC			GS series economical spindle servo drive		
Connect to CN15 (25PIN, 2 rows of pins)	22	SRV（Y5.2）	20	CCW	Connect to CN1 (44PIN 3-pin)
	23	SFR（Y5.3）	5	CW	
	20	VP(Y5.0)	38	PSTI	
	21	TAP(Y5.1)	6	GAIN	
	19	EN5	23	SON	
	3 或 9 或 16	0V	24	COM-	
			28	COIN-	
			29	ZOUT-	
			26	PSTO-	
			25	ALM-	
	11	+24V	39	COM+	
	4	ALM5	9	ALM+	
	5	VPO	10	PSTO+	
	6	X5.1	41	PSR	
	8	X5.2	12	COIN+	
	13	SVC	44	VCMD+	
	12	SVC-GND	14	VCMD-	
	1	CP5+	2	PULS+	
	14	CP5-	17	PULS-	
	2	DIR5+	1	SIGN+	
	15	DIR5-	16	SIGN-	
	10	PC5	13	ZOUT+	
Connected to CN21 (15PIN, 2 rows of holes)	8	PAO+	19	PAO+	
	7	PAO-	4	PAO-	
	6	PBO+	18	PBO+	
	5	PBO-	3	PBO-	
	4	PZO+	31	PZO+	
	3	PZO-	32	PZO-	
			8	OSTA	
Metal shell connected to shielding					

Appendix

Appendix 3 Explanations of Rigid tapping

3.1 Setting of Spindle Electronic Gear Ratio

Calculation formula of CNC:

$$\frac{CMR}{CMD} = \frac{\delta}{\alpha} \times \frac{Z_M}{Z_D}$$

$$\alpha = \delta \times \frac{Z_M}{Z_D} \times \frac{CMD}{CMR}$$

CMR: Multiplier coefficient of spindle command.

CMD: Frequency division coefficient of spindle command.

α : Pulse volume, motor rotation angle for a pulse.

δ: Min. input command unit of CNC (0.001°, 0.01° or 0.0001°), this value corresponds to the travel amount of a revolution of the spindle (related parameters 5216). When 5216 is set to 1000, δ=360/1000=0.360°

ZM: Gear teeth of lead screw.

ZD: Gear teeth of motor.

Calculation formula of spindle servo drive unit:

Parameter 12, 13 of drive unit correspond to the pulse frequency division numerator of position command respectively. The calculation formulas for pulse frequency division denominator of position command and gear ratio of drive unit are as follows:

$$P \times G = 4 \times N \times C$$

And:

$$G = \frac{4 \times N \times C}{P} = 4 \times N \times C \times \frac{\alpha}{360} = \frac{4 \times C}{360} \times \delta \times \frac{Z_M}{Z_D} \times \frac{CMD}{CMR}$$

P: Correspondence between required pulse amounts for motor rotates 360 degrees and CNC end: $P = 360 / \alpha$.

G: Electronic gear ratio of drive unit, G= position command pulse frequency division numerator/ position command pulse frequency division denominator.

N: Set motor rev number to 1.

C: Wire number of feedback encoder.

In order to enhance the machining accuracy, the gear ratio of spindle servo drive is usually set to 1:1, namely, G=1 in the above formula, and the evolving process is as follows:

$$\frac{4 \times C}{360} \times \delta \times \frac{Z_M}{Z_D} \times \frac{CMD}{CMR} = 1 \qquad \frac{CMR}{CMD} = \frac{4 \times C}{360} \times \delta \times \frac{Z_M}{Z_D}$$

In order to match with DAP03, C=1024, spindle connects to the motor, Z_M/Z_D=1, and it is suggested that the data parameter 5216 of CNC is set to 1000 (Now the incremental system of the 5th axis is 0.001°) (If the incremental system of the 5th axis is 0.0001°, the recommended value of this parameter is 10000), namely, $\delta = \frac{360}{1000}$ (°).

$$\frac{CMR}{CMD} = \frac{4 \times 1024}{360} \times \frac{360}{1000} \times \frac{1}{1} = \frac{512}{125}$$

Therefore, data parameter 5221 is set to 512, and 5231 is set to 125.

3.2 Related Parameter Setting

The parameters relating to rigid tapping is as follows:

Data parameter	Significance of the parameter	Adjustment explanation	Recommended range
5221	Multiplier coefficient of spindle command in rigid tapping(gear 1)	Refer to section 7.3 for detailed setting method	512
5222	Multiplier coefficient of spindle command in rigid tapping(gear 2)		1
5223	Multiplier coefficient of spindle command in rigid tapping(gear 3)		1
5224	Multiplier coefficient of spindle command in rigid tapping(gear 4)		1
5231	Frequency division coefficient of spindle command in rigid tapping(gear 1)		125
5232	Frequency division coefficient of spindle command in rigid tapping(gear 2)		1
5233	Frequency division coefficient of spindle command in rigid tapping(gear 3		1
5234	Frequency division coefficient of spindle command in rigid tapping(gear 4		1
5371	Pulse feed speed of M29 calculating rigid tap gear ratio	Refer to section 7.4 for detailed setting method	200
5261	Acceleration/deceleration time constant in rigid tapping	The faster tapping speed is, the bigger setting time is	80~110
5271	Acceleration/deceleration time constant in tool retraction of rigid tapping(when set to 0, time constant for feeding is used)	When the tapping speed is low (below 500), the parameter setting is not necessary. Set this parameter at high speed to avoid affecting process in tool retraction.	100~200
5211	Override value in tool retraction of rigid tapping	Add sub-parameter properly if the teeth are not required in tool retraction	0
5216	The travel amount of a revolution of the spindle	Set the value according to the wire number of feedback encoder of spindle servo drive	1000(1u),10000(0.1u)

Appendix 4 Analog Spindle Function

The spindle is classified into mechanical spindle、analog spindle and serial spindle, according to the way how CNC control the spindle. To mechanical spindle, CNC controls its speed by inputting the switch values which are changed from S codes, into the spindle; to analog spindle, CNC changes S codes into analog values which are inputted to the spindle to control the speed; to serial spindle, CNC transmit S codes by bus to control the spindle speed.CNC informs the PLC of the current spindle control mode by issuing SIMSPL and SERSPX signal.

Spindle Mode Signal SIMSPL、SERSP

- Signal Type: NC→PLC
- Signal Function: By setting SIMSPL and SERSP to 0 or 1 to decide the spindle control mode, NC then transfers to PLC, messaging PLC the current spindle control mode.
- Signal Activity:

control mode	**SIMSPL**	**SERSP**
mechanical spindle	0	0
analog spindle	1	0
serial spindle	1	1

- Signal Address

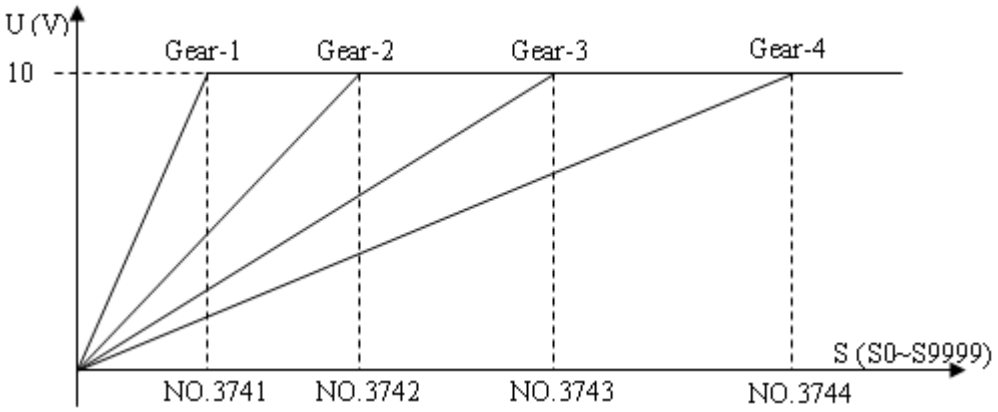
F200				SIMSPL				
F225						SERSP3	SERSP2	SERSP1

4.1 T Type Gear Shift

- S code/SF signal output
  The control function of analog spindle in CNC turns S instruction values into binary coding and then outputs them to PLC(F22~F25),SF not outputted.
- Gear shift method
  The maximum speed of the spindle corresponding to each gear-stage in the 1st spindle is decided by CNC data parameter. And the gear-stage select signal is 2-bit coding signal (GR1, GR2). The maximum speed of the spindle corresponding to each gear-stage in the 2nd spindle is decided by CNC data parameter too, but gear-stage decided by GR21, GR22. The relationship between signal and gear-stage is as follows:

	GR2(G28.2)	GR1(G28.1)	gear-stage	Maximum spindle speed parameter NO.
1st spindle	0	0	1	data parameterNO.3741 S1
	0	1	2	data parameterNO.3742 S1
	1	0	3	data parameterNO.3743 S1
	1	1	4	data parameterNO.3744 S1
	GR22(G29.1)	**GR21(G29.0)**	**gear-stage**	**Maximum spindle speed parameter NO.**
2nd spindle	0	0	1	data parameterNO.3741 S2
	0	1	2	data parameterNO.3742 S2
	1	0	3	data parameterNO.3743 S2
	1	1	4	data parameterNO.3744 S2

For the same S instruction but at different gear-stage, output analog voltages and S instructions have linear relationship as follows:



Assume data parameter S1:NO.3741＝1000;NO.3742＝2000;NO.3743=4000;NO.3744=5000. When spindle speed S＝800,

G28.1=0, G28.2=0, at gear-1:

$$U1 = \frac{800}{1000} \times 10 = 8V$$

G28.1=1, G28.2=0, at gear-2:

$$U2 = \frac{800}{2000} \times 10 = 4V$$

G28.1=0, G28.2=1, at gear-3:

$$U3 = \frac{800}{4000} \times 10 = 2V$$

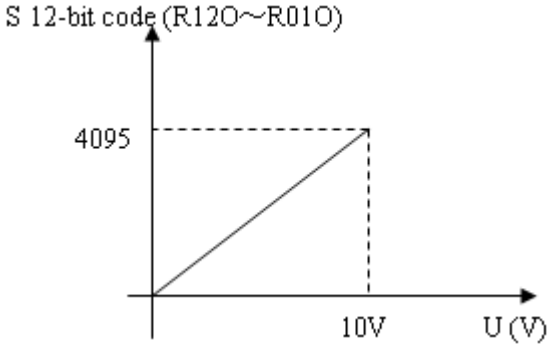
G28.1=1, G28.2=1, at gear-4:

$$U4 = \frac{800}{5000} \times 10 = 1.6V$$

Output voltage value:

$$U = \frac{\text{S instruction value}}{\text{Maximum speed at current gear}} \times 10$$

- S 12-bit code(R12O～R01O,F36～F37) (R12O2～R01O2,F236～F237) output
  Through the gear shift treatment, NC computes the spindle speed for every gear stage, that is, S 12-bit code signal R12O～R01O(0~4095),R12O2～R01O2(0~4095)outputs to spindle frequency converter and PLC. The relationship between 10V voltage and S 12-bit code is as follow:



  Identified output of S 12-bit code:
  That whether S 12-bit code data(R12O～R01O)(R12O2～R01O2), computed by NC or 12-bit code input signal(spindle motor speed instruction signalR12I～R01I,R12I2～R01I2), appointed by PLC, is used to identify the 1st /2nd spindle speed instructions decided by SIND/SIND2 signal(G33.7/G35.7). When spindle speed outputted by PLC is chosen, then spindle speed instructions are set by 12-bit code input signal.
- voltage offset:
  After spindle speed instruction S 12-bit code (0~4095) gotten, data parameterNO.3730 (voltage compensate value when analog voltage output is 10V) and data parameterNO.3730 (voltage compensate value when analog voltage output is 0V) can be used to finally compensate and clamp adjustment to spindle speed instructions. The formula is as follows; take the 1st spindle for example:

$$\text{Spindle speed output} = \frac{9999 + NO.021 - NO.036}{9999} \times S12\text{-bit code} + NO.036 \times \frac{4095}{9999}$$

- Stop outputting conditions:
  When S0 instruction is used, instructions output to spindle are all reset to 0. At this situation, none of M05, emergency or CNC reset can make spindle speed output instruction of NC reset to 0.
- analog spindle interface:
  CNC outputs adjusted spindle speed output value, in the form of analog voltage, via analog spindle interface circuit, to spindle frequency converter.

4.2 M Type Gear Shift

This gear shift pattern is valid only in single analog spindle.

To choose M type gear shift, according to data parameter set value and S instruction value, CNC makes a decision and outputs to PLC that which gear stage(GR40,GR30,GR2O,GR1O<F034.3,F034.2,F034.1,F034.0>) is needed for the appointed spindle speed.

After S code outputted , only when CNC issues gear shift instructions to PLC, do CNC output SF signal.

When there is auto running of any of input instructions S0～S9999, CNC outputs spindle motor rotation

instructions, which are corresponding to speed range of the 4 gears pre-set via data parameter, to spindle motor. At meantime it outputs current gear select signals as well as SF.

The meaning of gear select signal is as follows.

GR10(F34.3)	GR10(F34.2)	GR20(F34.1)	GR10(F34.0)	Selective gear-stage
0	0	0	1	gear-1
0	0	1	0	gear-2
0	1	0	0	gear-3
1	0	0	0	gear-4

To spindle motor analog spindle output speed instructions like this: as to analog voltage 0～10V, output to analog voltage signal SVC.

M type gear shift has two types to choose, set by bit parameter SGB.

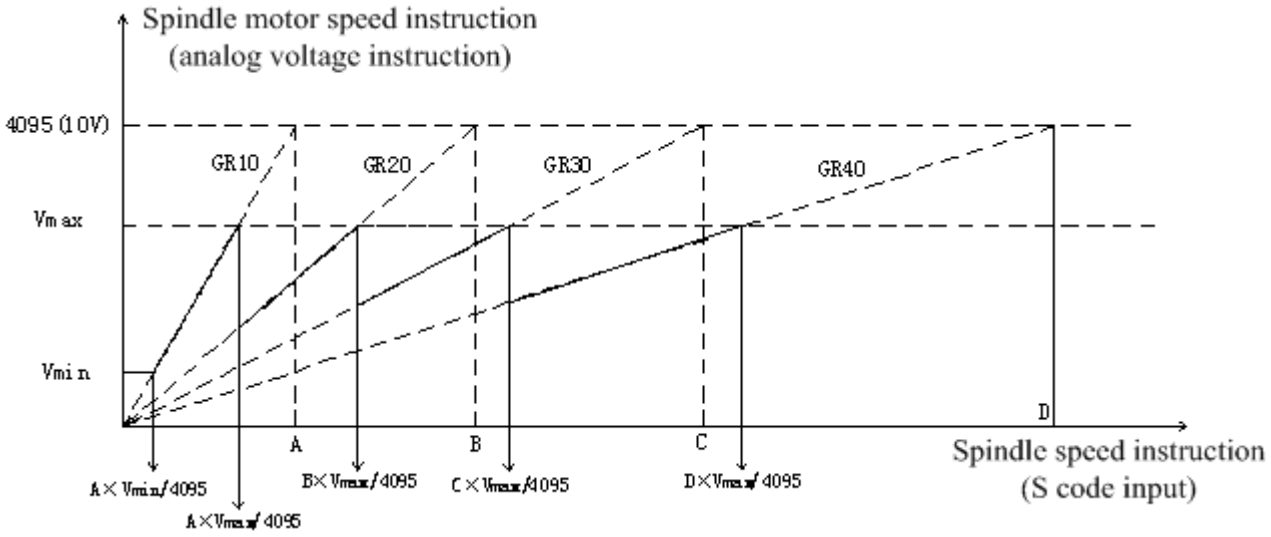
SGB: gear shift types:

0: M type gear shift Type A;

1: M type gear shift Type B

The following are examples of the two gear shift.

M type gear shift Type A:



As to the data in the image above, descriptions as follows:

Constant Vmax:spindle motor Maximum restraint speed (S12-bit code value), that is data parameter NO.3736.

Constant Vmin:spindle motor minimum restraint speed (S12-bit code value), that is data parameter NO.3735.

Spindle speed A: Maximum speed when instruction voltage is10V, that is, data parameter NO.3741..

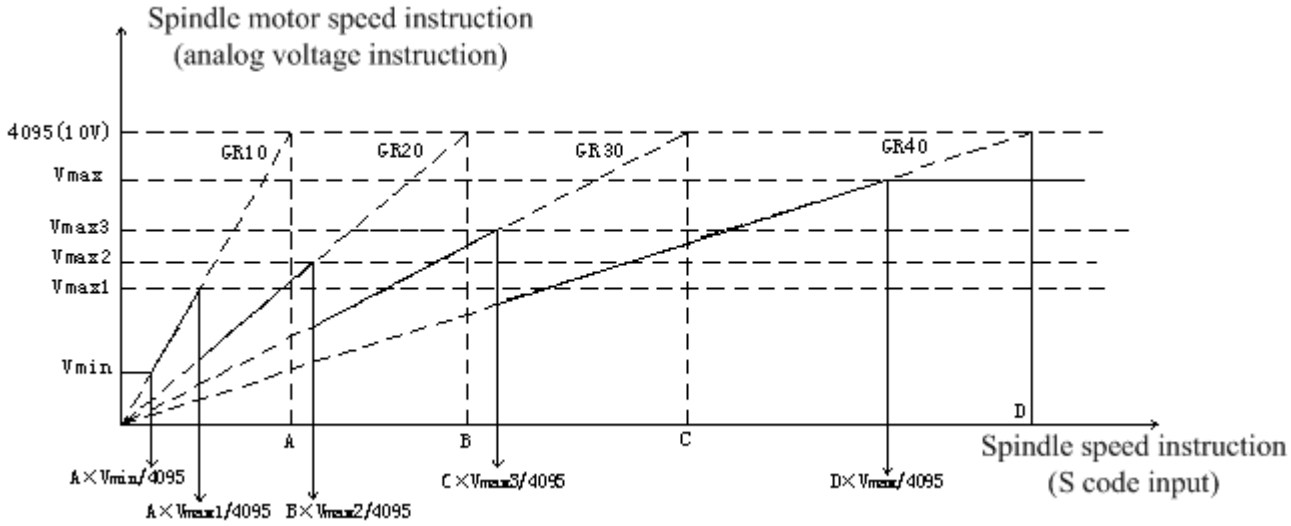
Spindle speed B: Maximum speed when instruction voltage is10V, that is, data parameter NO.3742

Spindle speed C: Maximum speed when instruction voltage is10V, that is, data parameter NO.3743.

Spindle speed D: Maximum speed when instruction voltage is10V, that is, data parameter NO.3744.

M type gear shift Type B:

This gear shift mode allows setting the switching speed of the gears respectively in data parameters.



Some parameters related to the figure above are as follows:

Constant Vmax:spindle motor Maximum restraint speed (S12-bit code value), that is, data parameter NO. 3736.

Constant Vmin:spindle motor minimum restraint speed (S12-bit code value), that is, data parameter NO. 3735.

Vmax.1: from gear-1 switch to the gear-2, the current spindle motor speed (S 12-bit code value), that is, data parameter NO.3751.

Vmax.2: from gear-2 switch to the gear-3, the current spindle motor speed (S 12-bit code value), that is, data parameter NO.3752.

Vmax.3: from gear-3 switch to the gear-4, the current spindle motor speed (S 12-bit code value), that is, data parameter NO.3753.

Spindle speed A: Maximum speed when instruction voltage is10V, that is, data parameter NO.3741.

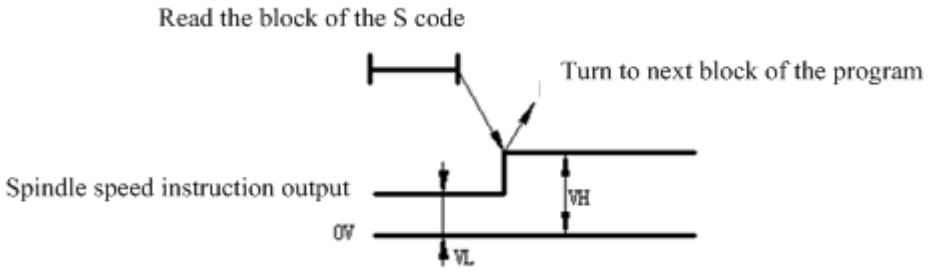
Spindle speed A: Maximum speed when instruction voltage is10V, that is, data parameter NO.3742.

Spindle speed A: Maximum speed when instruction voltage is10V, that is, data parameter NO.3743.

Spindle speed A: Maximum speed when instruction voltage is10V, that is, data parameter NO.3744.

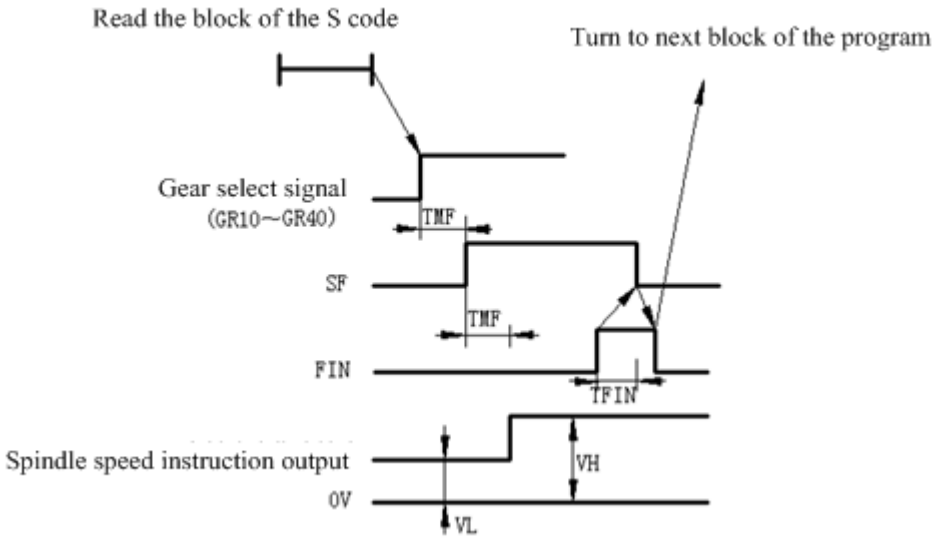
- When executing S code, sequence chart related as follows:

  Do not switch the gear select signal:



  In this case, do not output SF signal, and when new value spindle speed instruction outputted, it will automatically transfer to implement the next block.

  Do switch the gear select signal:



To switch gear select signal, firstly CNC outputs gear select signal, then after a time (TMF) set by data parameter No.3010, outputs SF signal. Repeatedly after another same TMF, CNC outputs new spindle speed instruction. PLC changes the gear at the moment receiving the SF signal, after finished, then sending accomplishment signal FIN. SF changes with FIN just the way S code output.

The rest signals are same with related signals in single analog spindle.

Appendix 5 Explanations for Standard Ladder Function

5.1 Address Definition

CN61	PLC address	Symbol	Description	Note
1	X0.0	TDEC	detect Cutter back to 0 in position	BT40 pot/turret
2	X0.1	SP	External feed hold signal	
3	X0.2	TPCH	Cylinder pressure test	BT40 pot /turret
4	X0.3			
5	X0.4	TCOT	Tool post counting	BT40 pot/turret
6	X0.5	ESP	External emergency stop signal	fixed address
7	X0.6	THCH	Toolcase horizontally in position detection/toolpot back in position detection	BT40 pot/turret
8	X0.7	TVCH	Toolcase vertically in position detection/toolpot forward in position detection	BT40pot/turret
9	X1.0	THSP	Stop detection of ATC arm of pot tool magazine	BT40pot/turret
10	X1.1	THGT	Catching tool detection of the ATC arm of pot tool magazine/the current place dry detection in turret tool magazine	BT40pot/turret
11	X1.2	THZP	Disc-tools tool changer arm origin detection	BT40 pot
12	X1.3			
13	X1.4	ST	External cycle start signal	
14	X1.5	JOGT	External spindle releasing/clamping tool input	BT40pot/turret
15	X1.6	SAGT	Safe door signal	
16	X1.7	TRCH	Spindle releasing tool detection	BT40 pot/turret
29	X2.0	TGCH	Spindle clamping tool detection	BT40 pot/turret
30	X2.1	THOV	motor of the ATC arm in pot magazine overload	BT40 pot
31	X2.2	TTOV	Tool changer motor overloaded	BT40pot/turret
32	X2.3			
33	X2.4	DEC4	Signal of the 4th axis deceleration	fixed address
34	X2.5	DEC5	Signal of the 5th axis deceleration	fixed address

CN61	PLC address	Symbol	Description	Note
35	X2.6			
36	X2.7	LTXN	X positive limit(LT-)	
37	X3.0	LTXP	X positive limit(LT+)	
38	X3.1	LTYN	Y positive limit(LT-)	
39	X3.2	LTYP	Y positive limit(LT+)	
40	X3.3	DTLO	indexing axis loosen in place	
41	X3.4	DTCL	indexing axis clamp in place	
42	X3.5	SKIP	Skip signal	fixed address
43	X3.6	LTZN	Z positive limit(LT-)	
44	X3.7	LTZP	Z positive limit(LT+)	
17~20	Suspension			
21~24	0V			
25~28	Suspension			

CN62	PLC address	Symbol	Description	Note
1	Y0.0	COOL	Cooling signal	
2	Y0.1	LUBR	Lubricating signal	
3	Y0.2	SRDY	Brake release	
4	Y0.3	SRV	Spindle CCW rotation	
5	Y0.4	SFR	Spindle CW rotation	
6	Y0.5			
7	Y0.6	TKOUT	chip removal	
8	Y0.7	SPZD	Spindle brake signal	
9	Y1.0	GEAR1	Spindle mechanical gear 1	
10	Y1.1	GEAR2	Spindle mechanical gear 2	
11	Y1.2	GEAR3	Spindle mechanical gear 3	
12	Y1.3	GEAR4	Spindle mechanical gear 4	
13	Y1.4	THOR	Toolcase vertical	BT40pot/turret
14	Y1.5	TVER	toolcase horizontal	BT40pot/turret
15	Y1.6	TCW	Toolpot motor CW	BT40pot/turret
16	Y1.7	TCCW	Toolpot motor CCW	BT40 pot/turret
29	Y2.0	TROT	spindle releasing/clamping tool output	BT40 pot/turret
30	Y2.1	THOT	Disk-tools tool changer arm output motor	BT40 pot
31	Y2.2	CLPY	Tri-colour indicator-yellow	
32	Y2.3	CLPG	Tri-colour indicator -green	
33	Y2.4	CLPR	Tri-colour indicator -red	

CN62	PLC address	Symbol	Description	Note
34	Y2.5			
35	Y2.6	WKLT	Work light	
36	Y2.7	ALTO	Turning output signal	
37	Y3.0			
38	Y3.1	ITLS	Dividing table loosen signal	
39	Y3.2	ITCP	Dividing table clamp signal	
40	Y3.3			
41	Y3.4			
42	Y3.5	SPZD2	The 2nd spindle braking	The 2nd spindle control
43	Y3.6	ITEN	Dividing table enabled output	
44	Y3.7			
17~19	0V			
20~25	24V			
26~28	0V			

CN31	PLC Address	Symbol	Description	Note
5	X6.0	EHDX	External hand wheel X selected	
6	X6.1	EHDY	External hand wheel Y selected	
8	X6.2	EHDZ	External hand wheel Z selected	
19	X7.0	EHD4	External hand wheel 4th selected	
7	X7.1	EHD5	External hand wheel 5th selected	
9	X6.3	EMP0	External X1 override	
22	X6.4	EMP1	External X10 override	
23	X6.5	EMP2	External X100 override	
1	HA+			
2	HA-			
3	HB+			
4	HB-			
7	Suspend			
10~13	0V			
14~16	5V			
17~18	24V			
19~21	Suspend			
24~26	Suspend			

CN15	PLC Address	Address symbol	Function of address definition	Remarks
8	X5.0	SPAL	The first spindle alarm input signal	
9	X5.1			
6	Y5.0	SRV	The first spindle CCW	
7	Y5.1	SFR	The first spindle CW	
3	Y5.2			
4	GND			
5	SVC			
2	0V			
1	+24V			

CN16	PLC Address	Address symbol	Function of address definition	Remarks
8	X8.0	SPAL2	The second spindle alarm input signal	
9	X8.1			
6	Y8.0	SRV2	The second spindle CCW	
7	Y8.1	SFR2	The second spindle CW	
3	Y8.2			
4	GND			
5	SVC			
2	0V			
1	+24V			

Other address definition, please see to the third piece- installation.

5.2 Function Configuration

5.2.1 Spindle Rotation Control

- **Related signals**

Type of signal	Symbol	Significance	Pin-out No	PLC State	CNC Diagnosis
Input signal		Machine panel spindle rotates CCW		X21.7	
		Machine panel spindle rotates CW		X21.3	
		Machine panel spindle stop key		X21.5	
	SPAL	Spindle alarm signal		X5.0	
Output signal	SRV	Spindle rotates CCW signal	CN62.4	Y0.3	
				Y5.1	
	SFR	Spindle rotates CW signal	CN62.5	Y0.4	
				Y5.0	
	SPZD	Spindle brake signal	CN62.8	Y0.7	
		Indicator for spindle rotates CCW		Y23.1	
		Indicator for spindle rotates CW		Y19.1	
		Indicator for spindle stop		Y18.0	

Type of signal	Symbol	Significance	Pin-out No	PLC State	CNC Diagnosis
Command output	M03	Command signal for spindle rotates CCW			
	M04	Command signal for spindle rotates CW			
	M05	Command signal for spindle stop			

- **Control parameter**

3712	Spindle type(0: mechanical spindle; 1: analog spindle; 2: serial spindle)

K0010		MTSS					RSJG	

RSJG =1: Spindle, coolant and lubrication are not closed by the CNC during reset;

=0: Spindle, coolant and lubrication are closed by the CNC during reset.

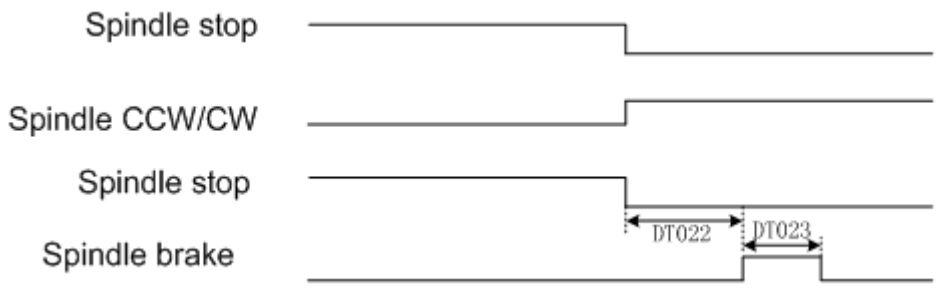
MTSS =1: Do not stop the spindle when running M30;

=0: Stop the spindle when running M30.

DT0021	M command execution time
DT0022	Delay time from spindle stop to brake output
DT0023	Spindle brake output time

- **Action sequence**

Operation sequence of spindle is as follows:



Note: DT022 is the time from spindle stop signal issuing to spindle brake signal issuing. DT023 is spindle brake holding time.

- **Control logic**

SSTP output is valid after the CNC power on. M03 or M04 is executed when SSTP output is valid. When SFR or SRV output is valid and held on, SSTP output is stopped at the same time. M05 is executed when SFR or SRV is valid. When SFR or SRV is stopped, SSTP output is valid and held. Spindle brake SPZD signal output delay time is set by PLC data DT022 (the delay time between spindle stop commands output to spindle brake SPZD signal output). The holding time of brake signal is set by PLC data DT023 (spindle brake output time).

If the current spindle is in the state of CCW/CW rotation, PLC alarm A0.3 (M03, M04 specify the error) occurs when M04 or M03 is executed.

Note 1: When the CNC external stop or spindle alarm, the spindle rotation signal is stopped and SSTP signal is output at the same time.

Note 2: In CNC reset, RSJG of K0010 of the PLC set whether cancel SFR, SRV output.

When RSJG＝0, CNC reset to stop SFR, SRV output.

When RSJG＝1, CNC reset SFR, SRV output state is unchanged.

Note 3: 436 alarm occurs (spindle alarm) after the APAL signal is detected by the CNC, and F35.0 is 1.

Note 4: In spindle analog quantity control, spindle enable signal is valid when output voltage is greater than 0.

Note 5: Serial spindle don't need I/O signal, CNC communicate with servo by G、F signal(spindle CW signal:G70.5, spindle CCW signal:G70.4)

5.2.2 The 2nd Spindle Rotation Control

- **Related signals**

Signal type	Signal symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal		Spindle CW key of the 2nd spindle machine panel		X21.7	
		Spindle CCW key of the 2nd spindle machine panel		X21.3	
		The 2nd spindle machine panel stop key		X21.5	
Output signal	SRV2	CCW signal of the 2nd spindle	CN62.41	Y3.4	
	SFR2	signal of the 2nd spindle	CN62.40	Y3.3	
	SPZD	Breaking signal of the 2nd spindle	CN62.42	Y3.5	
Instruction input	M73	CW instruction signal of the 2nd spindle			
	M74	CW instruction signal of the 2nd spindle			
	M75	Stop instruction signal of the 2nd spindle			

Note: for the 1st spindle relative signals, please see to spindle rotation control description.

- **Parameter control**

3710	Sum of CNC spindles
3712	Spindle type(0: mechanical spindle; 1: analog spindle; 2: serial spindle)

DT0021	Duration of the implement of M code
DT0022	The delay time for the spindle from stop to brake output
DT0023	Spindle braking output time

K0013		EMS						

EMS =0: Multi-spindle function is invalid

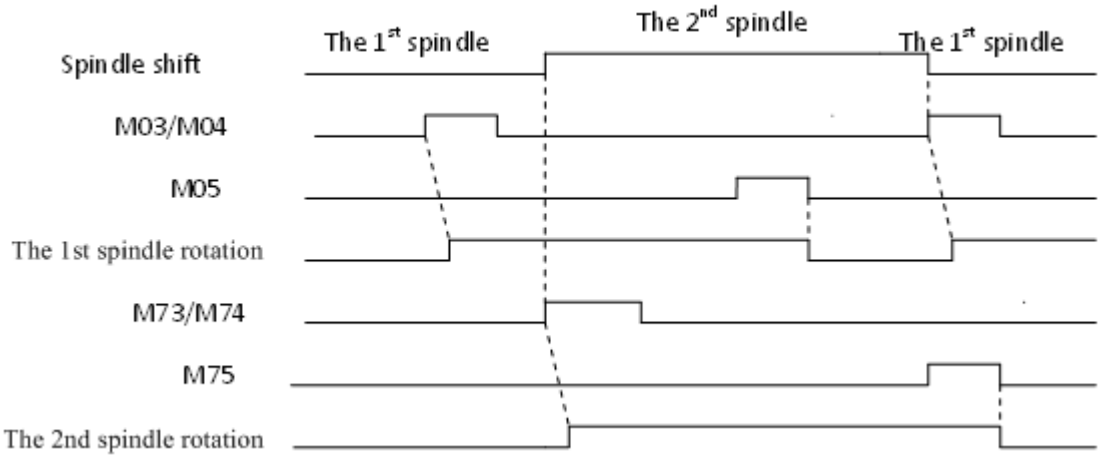
=1: Multi-spindle function is valid

- **Description for multi-spindles function**

  S code appoints the speed of the spindle, and if the corresponding signal for a spindle does not pass through, then, the spindle will continue rotating at the former speed. In this way, the CNC can realize multi-spindles spinning at different speed at the same time.

- **Description for multi-spindles**
  - Multi-spindles function is valid only in the analog voltage control mode.
  - M03/M04 S□□□□ shift to the 1st spindle and spindle spinning CW and CCW.
  - M73/M74 S□□□□ shift to the 2nd spindle and spindle spinning CW and CCW.
  - M05/M75 stop the 1st spindle / the 2nd spinning CW and CCW.
  - In operation panel, the CCW key light, spindle stop key light, and CW key light only reflect the 1st spindle rotational states, not the 2nd spindle.

- **Sequence chart**



- **Control logic**

  When CNC is power on, the current used spindle is the 1st spindle. By M03, M04, CNC controls the 1st spindle CW and CCW; By M73, M74, CNC shifts to control outputting of the 2nd spindle CW and CCW. The shift completed, the control logic to current spindle is the same as the spinning spindle.

  For example, currently, the 2nd spindle is spinning CW or CCW, and then if M74 or M73 is executed, there will be PLC alarm A0.7:M73, M74 specified error.

Note 1: Input S value must be the speed allowed for the certain spindle.

Note 2: When change the spindle, if the spindle to be changed is spinning, then you should input M instruction to make the new changed spindle retaining the same direction as before, or stop the spindle that will be changed, else here will be PLC alarm: Appoint wrong spindle rotation.

Note 3: When CNC emergence or spindle alarm appearing, CNC shuts multi-spindle rotation output signal, at the same time outputs stop signal.

Note 4: Serial spindle don't need I/O signal, CNC communicate with servo by G、F signal(spindle CW signal:G70.5, spindle CCW signal:G70.4).

5.2.3 The 3nd Spindle Rotation Control

- **Related signals**

Signal type	Signal symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal	SPAL	3nd Spindle alarm signal		X5.0	
Output signal	SRV1	CCW signal of the 3nd spindle		Y5.0	
	SFR2	CW signal of the 3nd spindle		Y5.1	
Instruction input	M83	CW instruction signal of the3nd spindle			
	M84	CW instruction signal of the 3nd spindle			
	M85	Stop instruction signal of the 3nd spindle			

- **Parameter control**

3710	sum of CNC spindles
3712	Spindle type(0: mechanical spindle; 1: analog spindle; 2: serial spindle)

K0013		EMS		SP3				

EMS =0: Multi-spindle function is invalid

=1: Multi-spindle function is valid

SP3 =0: 3nd spindle invalid

=1: 3nd spindle valid

- **Description for multi-spindles**
  - Multi-spindles function is valid only in the analog voltage control mode.
  - M03/M04 S□□□□ shift to the 1st spindle and spindle spinning CW and CCW.
  - M73/M74 S□□□□ shift to the 2nd spindle and spindle spinning CW and CCW.
  - M83/M84 S□□□□ shift to the 2nd spindle and spindle spinning CW and CCW.
  - M05/M75 stop the 1st spindle / the 2nd spinning CW and CCW.
  - In operation panel, the CCW key light, spindle stop key light, and CW key light only reflect the 1st spindle rotational states, not the 3nd spindle.
- **Control logic**

  The third spindle rotation control logic is consistent with the second spindle rotation control logic. When the third spindle is set to active, X5.0, Y5.0, and Y5.1 are the control signals for the third spindle. When the third spindle type is set to analog spindle, the analog spindle 1 (CN15) output port is used.

Note 1: input S value must be the speed allowed for the certain spindle.

Note 2: When change the spindle, if the spindle to be changed is spinning, then you should input M instruction to make the new changed spindle retaining the same direction as before, or stop the spindle that will be changed, else here will be PLC alarm: Appoint wrong spindle rotation.

Note 3: when CNC emergence or spindle alarm appearing, CNC shuts multi-spindle rotation output signal, at the same time outputs stop signal.

Note 4: Serial spindle don't need I/O signal, CNC communicate with servo by G、F signal(spindle CW signal:G78.5, spindle CCW signal:G78.4).

5.2.4 T- type Spindle Auto Gear Shift

- **Related signals**

Signal type	Signal symbol	Significance	Pin-out No.	PLC Status	CNC diagnose
Input signal	ASP1	Gear-1 spindle in position signal	CN61.7	X0.6	
	ASP2	Gear-2 spindle in position signal	CN61.8	X0.7	
	ASP3	Gear-3 spindle in position signal	CN61.9	X1.0	
	ASP4	Gear-4 spindle in position signal	CN61.10	X1.1	
Output signal	GEAR1	Spindle mechanical gear-stage signal1	CN62.9	Y1.0	
	GEAR2	Spindle mechanical gear-stage signal 2	CN62.10	Y1.1	
	GEAR3	Spindle mechanical gear-stage signal 3	CN62.11	Y1.2	
	GEAR4	Spindle mechanical gear-stage signal 4	CN62.12	Y1.3	
Instruction input	M41	Gear 1of the spindle auto gear shift			
	M42	Gear 2 of the spindle auto gear shift			
	M43	Gear 3 of the spindle auto gear shift			
	M44	Gear 4 of the spindle auto gear shift			

- **Function description**

  T type spindle auto gear shift function is used to control auto shifting mechanical gear-stage. operating S code, CNC computes analog voltage which output to spindle servo or frequency converter, according to the current M instruction controlling the corresponding parameters(M41～M44 respectively control data parameterNO.3741～NO.3744) to control the actual speed of spindle is the same as S code appointing.

- **Parameter control**

  K parameter:

K13	SPCD						SPCH	SPSE

SPCD =1: spindle auto gear shift function is valid;
=0: spindle auto gear shift function is invalid.
SPCH =1: The automatic shift detection signal for the spindle is invalid;
=0: The automatic shift detection signal for the spindle is valid .
SPSE =1: spindle auto gear shift output gear-stage signal maintains;
=0: spindle auto gear shift output gear-stage signal does not maintain.

Data parameter:

3741 S1	GRMAX.1(Max. speed for the 1st spindle gear-1)
3742 S1	GRMAX.2(Max. speed for the 1st spindle gear-2)
3743 S1	GRMAX.3(Max. speed for the 1st spindle gear-3)
3744 S1	GRMAX.4(Max. speed for the 1st spindle gear-4)

PLC parameter:

DT32	speed setting for spindle auto gear shift (0～4095)

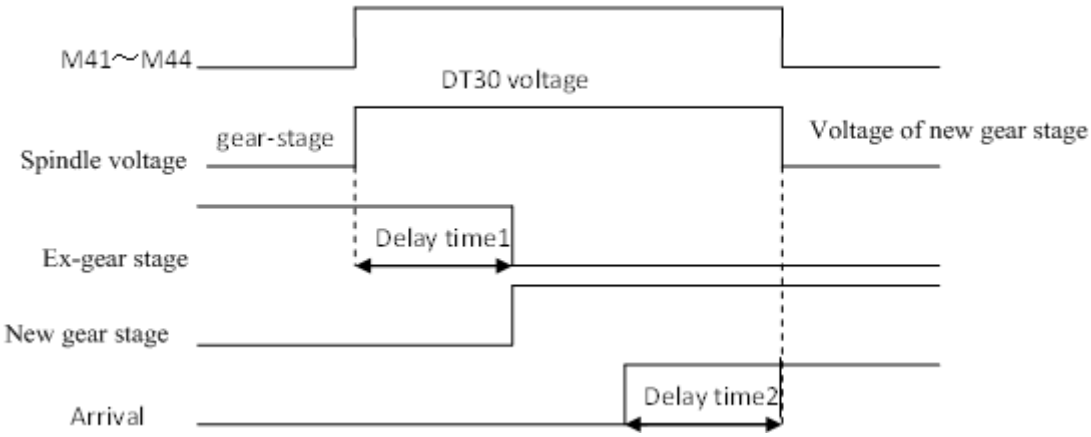
DT33	Shutting time for spindle to auto close the old gear-stage
DT34	Delay time after spindle auto gear shift in position
DT35	Check time of the auto gear shift
DT50	Shift swing interval time
DT51	Shift swing output time

DT33: The initialization for 5 seconds, users can set the parameter to adjust it. but don't set too short time to cause the spindle damage due to little time for spindle switching in to lower gear stage; too long times means Timeliness is relatively low and processing efficiency affected.

DT34: The initialization for 5 seconds, users can set the parameter to adjust it, but don't set too short time that spindle cannot reach the appointed speed during the M instruction implement; too long to affect the processing efficiency.

- **Action sequence**

The action sequence is as below according to control logic:



- **Control logic**
  - When K13.7 parameter set to 1, T type spindle auto gear shift function is valid;
  - To implement any of M41,M42,M43,M44, judge input gear-stage is consistent with the current one or not, else gear stage should shift;
  - Gear shift starts, CNC outputs analog voltage set by parameter DT30 to spindle servo or frequency converter;
  - After a time set by parameter DT31(gear shift time1), shut gear-stage output signal and output new gear shift signal;
  - Check gear shift completed or not, if not, wait for complete signal;
  - In a time set by DT33, if gear shift is not completed then perform alarm A1.0.

  If gear shift is completed, delaying for a time set by parameter DT32(gear shift time 2), then CNC outputs analog voltage set by parameterNO.3741～NO.3744, according to the current gear-stage. That gear shift process is completed.

Note 1: only the number of spindle revolutions is in the analog voltage control, is the spindle auto gear shift function valid;

Note 2: if spindle function is invalid, perform M41 ~ M44 alarm;

Note 3: It needs checking in position signal that spindle gear shift function takes actions. After having checked in position signal, the gear shift function can complete.

5.2.5 M-type Spindle Auto Gear Shift

- **Related signals**

Signal type	Signal symbol	Significance	Pin-out No.	PLC Status	CNC diagnose
Input signal	ASP1	Gear-1 spindle in position signal	CN61.7	X0.6	
	ASP2	Gear-2 spindle in position signal	CN61.8	X0.7	
	ASP3	Gear-3 spindle in position signal	CN61.9	X1.0	
	ASP4	Gear-4 spindle in position signal	CN61.10	X1.1	
Output signal	GEAR1	Spindle mechanical gear-stage signal1	CN62.9	Y1.0	
	GEAR2	Spindle mechanical gear-stage signal 2	CN62.10	Y1.1	
	GEAR3	Spindle mechanical gear-stage signal 3	CN62.11	Y1.2	
	GEAR4	Spindle mechanical gear-stage signal 4	CN62.12	Y1.3	
Instruction input	M41	Gear 1of the spindle auto gear shift			
	M42	Gear 2 of the spindle auto gear shift			
	M43	Gear 3 of the spindle auto gear shift			
	M44	Gear 4 of the spindle auto gear shift			

- **Function description**

  When selecting the M-type gear shift, based on the set values of the data parameters and the S command value, the CNC makes a judgment on whether to shift and outputs it to the PLC to specify the required gear number for the spindle speed.

- **Parameter control**

  K parameter:

K13	SPCD	EMS	SPMT				SPCH	SPSE

SPCD =1: spindle auto gear shift function is valid
=0: spindle auto gear shift function is invalid

EMS =1: Multi spindle function is effective
=0: Multi spindle function is invalid

SPMT =1: The spindle automatic shifting type is M-type
=0: The spindle automatic shifting type is T-shaped

SPCH =1: The automatic shift detection signal for the spindle is invalid
=0: The automatic shift detection signal for the spindle is valid

SPSE =1: spindle auto gear shift output gear-stage signal maintains
=0: spindle auto gear shift output gear-stage signal does not maintain

Data parameter:

3735	Minimum clamping speed of spindle motor during M-type gear shifting (12 digit code value)
3736	Maximum clamping speed of spindle motor during M-type gear shifting (12 digit code value)
3741 S1	GRMAX.1(Max. speed for the 1st spindle gear-1)
3742 S1	GRMAX.2(Max. speed for the 1st spindle gear-2)

3743 S1	GRMAX.3(Max. speed for the 1st spindle gear-3)
3744 S1	GRMAX.4(Max. speed for the 1st spindle gear-4)

PLC parameter:

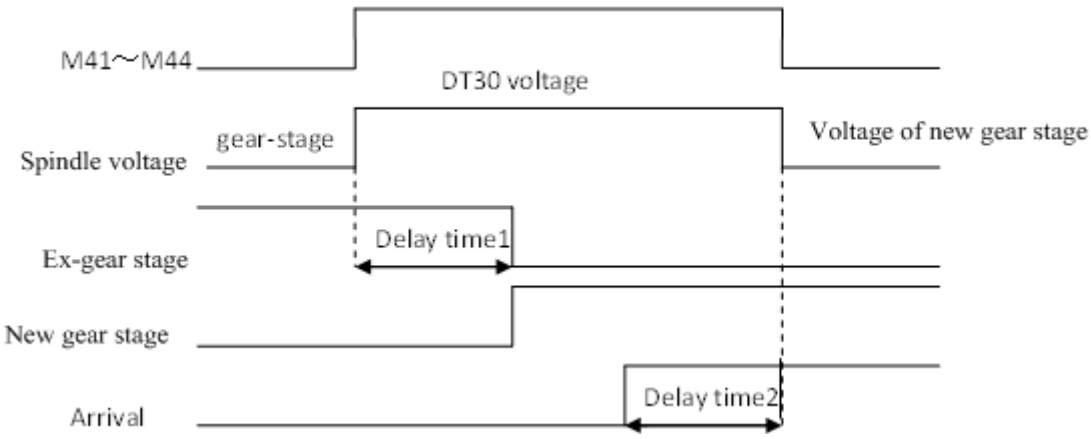
DT32	Speed setting for spindle auto gear shift (0～4095)
DT33	Shutting time for spindle to auto close the old gear-stage
DT34	Delay time after spindle auto gear shift in position
DT35	Check time of the auto gear shift
DT50	Shift swing interval time
DT51	Shift swing output time

DT33: The initialization for 5 seconds, users can set the parameter to adjust it. but don't set too short time to cause the spindle damage due to little time for spindle switching in to lower gear stage; too long times means Timeliness is relatively low and processing efficiency affected.

DT34: The initialization for 5 seconds, users can set the parameter to adjust it, but don't set too short time that spindle cannot reach the appointed speed during the M instruction implement; too long to affect the processing efficiency.

- **Action sequence**

The action sequence is as below according to control logic:



- **Control logic**
  - When K13.7 and K 13.5 parameter set to 1, M type spindle auto gear shift function is valid;
  - To implement any of M41,M42,M43,M44, judge input gear-stage is consistent with the current one or not, else gear stage should shift;
  - Gear shift starts, CNC outputs analog voltage set by parameterDT30 to spindle servo or frequency converter;
  - After a time set by parameter DT31(gear shift time1), shut gear-stage output signal and output new gear shift signal;
  - Check gear shift completed or not, if not, wait for complete signal;
  - In a time set by DT33, if gear shift is not completed then perform alarm A1.0.

If gear shift is completed, delaying for a time set by parameter DT32(gear shift time 2), then CNC outputs analog voltage set by parameterNO.3741～NO.3744, according to the current gear-stage. That gear shift process is completed.

Note 1: The automatic shifting function of the spindle must be effective only when the spindle speed is selected as the analog voltage control mode;

Note 2: M-type automatic shifting is only effective under a single simulated spindle;

Note 3: The main shaft shift function conversion requires the detection of the in place signal, and the shift function can only be completed after the detection of the in place signal.

5.2.6 Spindle Switching Volume Control

- **Related signal**

Signal Type	Symbol	Significance	Pin-out No.	PLC State	C NC Diagnosis
Output signal	GEAR1	Spindle mechanical gear signal 1	CN62.9	Y1.0	
	GEAR 2	Spindle mechanical gear signal 2	CN62.10	Y1.1	
	GEAR 3	Spindle mechanical gear signal 3	CN62.11	Y1.2	
	GEAR 4	Spindle mechanical gear signal 4	CN62.12	Y1.3	
Command Input	S01	Spindle gear signal 1 command signal			
	S02	Spindle gear signal 2 command signal			
	S03	Spindle gear signal 3 command signal			
	S04	Spindle gear signal 4 command signal			
	S00	Spindle gear signal cancel command signal			

- **Control parameter**

0	0	1
Corresponding F address		

			ACS				
			F200.4				

ACS =1: Analog voltage control of spindle speed

=0: Switching volume control of spindle speed

0	1	2
Corresponding F address		

			ESCD				
			F211.4				

ESCD =0: S code is active in emergency stop

=1: S code is inactive in emergency stop

DT0019	S code execution time
DT0024	Gear shifting delay time

- **Control logic**

GEAR1～GEAR4 output are inactive at power on. When any code of S01, S02, S03, and S04 is executed, the corresponding S signal output is active and held on, and the other 3 S signal outputs are cancelled. GEAR1～GEAR4 outputs are cancelled when executing S00 command, and only one in GEAR1～GEAR4 is active.

5.2.7 Cycle Start and Feed Hold

- **Related signal**

Signal Type	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal		External cycle start signal		X9.0	
		External feed hold signal		X9.1	
		Cycle start key signal		X23.0	
		Feed hold key signal		X22.7	
		OUT key cycle start signal of MDI mode		F197.1	
Output signal		Cycle start indicator of the panel		Y20.0	
		Feed hold indicator of the panel		Y21.0	
Command input	M00	Feed hold signal		F9.7	

- **Control parameter**

1	7	2
Corresponding F address		

	MST	MSP					
	F210.6	F210.5					

MST =1: External cycle start signal is inactive

=0: External cycle start signal is active

MSP =1: External feed hold signal is inactive

=0: External feed hold signal is active, the stop switch is needed, or “stop” is displayed by the CNC.

5.2.8 Coolant Control

- **Related signal**

Signal Type	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal		Coolant key signal		X21.4	
Output signal		Coolant start indicator		Y23.0	
	COOL	Coolant output signal	CN62.1	Y0.0	
Command input	M08	Coolant start command signal			
	M09	Coolant stop command signal			

- **Control parameter**

K0010						MTCC	RSJG	

RSJG =1: In reset, spindle, coolant and lubrication output signal are not stopped by the CNC;

=0: In reset, spindle, coolant and lubrication output signal are stopped by the CNC.

MTCC =1: Run M30,coolant retain;

=0: Run M30,coolant close.

- **Function description**

COOL output is invalid after the CNC power on. COOL output is valid when M08 command is executed and the coolant is on. COOL output is cancelled when M09 is executed and the coolant is off.

Note1: In CNC reset, RSJG of K10 of the PLC set whether close the coolant output.

Note 2: There is no corresponding output signal for M09. M08 output is cancelled if the M09 is executed.

Note 3: Coolant output is closed when the M30 is executed.

5.2.9 Lubrication Control

- **Related signals**

Signal Type	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal		Lubrication key signal on the machine panel		X21.6	
Output signal		Lubrication start indicator		Y20.7	
	LUBR	Lubrication output signal	CN62.2	Y0.1	
Command input	M32	Lubrication start command signal			
	M33	Lubrication stop command signal			

- **Control parameter**

DT0016	Interval time of automatic lubricating
DT0017	0:Non-automatic lubricating ;>0:Automatic lubricating time
DT0018	In automatic lubrication, 0:Turning lubrication >0: Regularly lubrication output time

- **Function description**

There are two types of lubrication function defined by the GSK980MDi Plus standard ladder diagram: non-automatic and automatic lubrication, which are set by PLC data.

DT0017 =0:Non-automatic lubrication

>0: Automatic lubrication, DT0016 lubricating time DT0017 and lubricating interval time DT0016 can be set

DT0018 =0:Non-automatic lubrication, turning lubrication

>1:Non-automatic lubrication, regularly lubrication

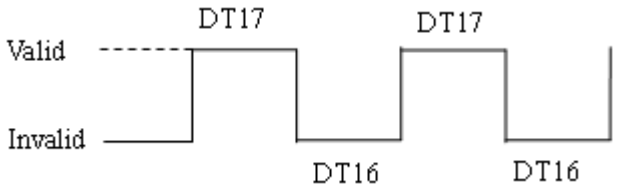
1. Non-automatic lubrication function

When PLC data DT0018=0, it is turning lubrication output. Lubrication is output by pressing lubrication key once and it is cancelled by pressing the key again. Lubrication output when M32 is executed. The lubrication is cancelled when M33 is executed.

When PLC data DT0018>1, it is turning lubrication output. By pressing lubrication key, lubrication is output, and it is cancelled after a setting time by the PLC data DT0018. By executing M32, lubrication is output, and it is cancelled after a setting time by the PLC data DT0018. If the DT0018 setting time is not yet up, M33 is executed to cancel the lubrication output.

2. Automatic lubrication

After the CNC system power on, it is lubricating for a time set by DT0017, then the lubrication output stops. After a time set by DT0016, the lubrication is output again, and it cycles by sequence. In automatic lubrication, M32, M33 commands as well as the lubrication key on the panel are all inactive.



5.2.10 Block Skip

If a block in a program needs not to be executed and deleted, the block skip function can be selected. When the block is headed with "/" sign, and the block skip switch is turned on (machine panel key or external input of the block is active), the block will be skipped without execution in auto running.

- **Related signal**

Signal type	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal		Block skip key on the machine panel		X18.7	
Output signal		Block skip indicator on the machine panel		Y18.6	

- **Function description**

1. When the block skip signal is valid, the block with "/" sign is skipped without being executed.
2. The block skip function is only valid in Auto, MDI and DNC mode.

5.2.11 Machine Lock

- **Related signal**

Signal type	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal		Machine lock on the machine panel		X19.0	
Output signal		Machine lock indicator on the machine panel		Y18.5	

- **Function description**

1. Machine lock is valid in any mode.
2. The state of the machine lock cannot be changed in program executing.

5.2.12 Auxiliary Lock

- **Related signal**

Signal type	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal		Auxiliary lock on the machine panel		X19.1	
Output signal		Auxiliary lock indicator on the machine panel		Y18.4	

- **Function description**

Auxiliary lock is valid in Auto, MDI or DNC mode.

5.2.13 Single Block

- **Related signal**

Signal type	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal		Single block key on the machine panel		X18.6	
Output signal		Single block indicator on the machine panel		Y18.7	

- **Function description**

Single block is valid in Auto, MDI or DNC mode.

5.2.14 Dry Run

- **Related signal**

Signal type	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal		Dry run key on the machine panel		X19.2	
Output signal		Dry run indicator on the machine panel		Y18.3	

- **Function description**

  1. Dry run is valid in Auto, MDI or DNC mode.

  2. The state of the dry run cannot be shifted in program execution.

5.2.15 Optional Stop

- **Related signal**

Signal type	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal		Optional key on the machine panel		X20.0	
Command input	M01	Optional command		F9.6	
Output signal		Optional indicator on the machine panel		Y21.7	

- **Function description**

  In Auto, MDI and DNC mode, press 

key to light up the indicator of optional stop key, and enter the optional stop state.

  The program is "stopped" when M01 is being executed. Press 

to continue the execution.

5.2.16 Stroke Limit and Emergency Stop

- **Related signal**

  1、plus-minus positive limit signal independent dispose:

Signal type	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal	ESP	External emergency stop signal	CN61.6	X0.5	
	LTXN	X positive limit(LT-)	CN61.36	X2.7	
	LTXP	X positive limit(LT+)	CN61.37	X3.0	
	LTYN	Y positive limit(LT-)	CN61.38	X3.1	
	LTYP	Y positive limit(LT+)	CN61.39	X3.2	
	LTZN	Z positive limit(LT-)	CN61.43	X3.6	
	LTZP	Z positive limit(LT+)	CN61.44	X3.7	
	LT4N	4^{th} positive limit(LT-)	CN61.40	X3.3	
	LT4P	4^{th} positive limit(LT+)	CN61.41	X3.4	
	LT5N	5^{th} positive limit(LT-)	CN61.30	X2.1	
	LT5P	5^{th} positive limit(LT+)	CN61.31	X2.2	

  2、plus-minus positive limit signal combine dispose:

Signal type	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal	LTXP	X positive limit	CN61.37	X3.0	
	LTYN	Y positive limit	CN61.38	X3.1	
	LTYP	Z positive limit	CN61.39	X3.2	
	LT4N	4th positive limit	CN61.40	X3.3	
	LT4P	5th positive limit	CN61.41	X3.4	

- **Control parameter**

1	7	2
Corresponding F address		

				MESP			
				F210.3			

MESP =0: External emergency stop signal is valid

=1: External emergency stop signal is invalid

K15

LTEN1	LTEN2	LT5	LT4				LTHL

LTHL =1: Alarm for X, Y, Z axis overtravel signal is disconnected with +24V;

=0: Alarm for X, Y, Z axis overtravel signal is connected with +24V;

LT4 =1: 4th positive limit valid.

=0: 4th positive limit invalid.

LT5 =1: 5th positive limit valid.

=0: 5th positive limit invalid.

LTEN1,LTEN2 =10: plus-minus positive limit signal independent dispose.

=01: plus-minus positive limit signal combine dispose.

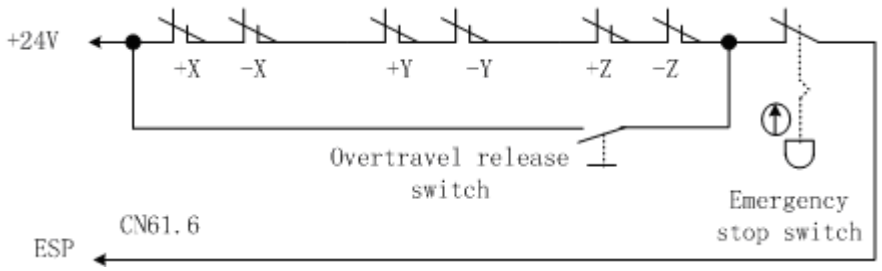
=00: positive limit invalid.

=11： positive limit invalid.

Note: when there is eternal overtravel alarm, change the work mode into manual or MPG mode, move the alarming axis in another direction until the overtravel signal overturned, finally press the Reset key to release the alarm.

- **External connection of the machine**

The connection of the emergency stop and travel switch is as follows (take 3-axis as example)



- **Control logic**

Emergency stop control: When the contact of the emergency stop switch is cut off, the ESP signal and +24V is disconnected, and the CNC issues emergency alarm. Meanwhile the CNC turns off the enable (ENB) signal to stop the pulse output. Except the functions are processed, the other

functions can also be defined by the ladder diagram when the emergency alarm is issued.

Positive limit control:

1、when appear positive limit alarm, change the work mode to JOG or MPG mode, then move the axis reverse until the positive limit signal turn. Press the reset key to clear the alarm.

2、Using the plus-minus positive limit signal combine dispose mode, every axis only have one positive limit signal. Judge the plus or minus positive limit through the axis move direction.

5.2.17 Tri-colour Indicator

- **Related signal**

Signal types	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Output signal	CLPY	Tri-colour indicator-yellow	CN62.31	Y2.2	
	CLPG	Tri-colour indicator -green	CN62.32	Y2.3	
	CLPR	Tri-colour indicator -red	CN62.33	Y2.4	

- **Function description**

Yellow indicator (normal state, non-operation, non-alarm), green indicator (in automatic running), red indicator (system alarming).

5.2.18 Reset and Cursor Return

- **Related signal**

Signal types	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal		Reset key on the MDI panel		X24.0	

- **Control parameter**

K0010								RESB

RESB =1: The functions of reset and the cursor return are valid

=0: The functions of reset and the cursor return are invalid

- **Function description**

When RESB of K10 is set to 1, press reset key (X26.0 in Auto mode, the system reset and the cursor returns to the beginning.

5.2.19 Rigid Tapping

- **Related signal**

Signal types	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input command	M29	Specified signal of rigid tapping			
G、F signal	F76.3	Position mode shift signal			
	F45.6	Servo rigid tapping state			
	G70.3	Servo rigid tapping parameter shift signal			
	G61.0	Rigid tapping shift successful			

- **Function description**

When M29 is being executed, F76.3 signal is output and the servo spindle is shifted from speed to the position.At the same time output G70.3 signal to invoking the rigid tapping parameter. After the shifting, servo spindle sends the F45.6 signal, and the signal is received by the PLC to set G61.0 to 1, then M29 is executed over. Execute process as follows:

M29—>F76.3=1—>G70.3=1—>F45.6=1—>G61.0=1—>G4.3

In the execution of M29, if the signal F45.6 is not detected at the time set by DT15, the alarm A0.2 (M29 is executed overtime, rigid tapping feedback signal is not detected) will be issued.

5.2.20 Spindle Exact Stop

- **Related signal**

Signal type	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal		Exact stop key on the panel		X25.7	
Output signal		Indicator signal of the spindle exact stop		Y21.3	
Input command	M19	Command of the spindle exact stop			
G、Fsignal	G70.2	Start signal of the spindle positioning			
	F45.2	Completion signal of the spindle positioning			

- **Function description**

The spindle positioning function is realizable in the speed mode of the servo spindle. The spindle rotary output is cancelled in the spindle exact stop. The spindle exact stop id cancelled in the spindle rotary output. After the spindle positioning signal G70.2 is output, the alarm A0.4 (Completion signal of the spindle positioning is detected overtime in spindle positioning) will be issued if the completion signal COIN is not detected at the time set by DT14.

5.2.21 External MPG Control

- **Related signal**

Signal types	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal	EHDX	External MPG X-axis selection	CN31.5	X6.0	
	EHDY	External MPG Y-axis selection	CN31.6	X6.1	
	EHDZ	External MPG Z-axis selection	CN31.8	X6.2	
	EHD4	External MPG 4th-axis selection	CN31.25	X7.0	
	EHD5	External MPG 5th-axis selection	CN31.7	X7.1	
	EMP0	External MPG/increment ×1	CN31.9	X6.3	
	EMP1	External MPG/increment ×10	CN31.22	X6.4	
	EMP2	External MPG/increment×100	CN31.23	X6.5	

- **Function description**

The standard ladder diagram supports external MPG of 5-axis, PSG-100-05E/L, ZSSY2080 external MPG can be matched. Please refer to the related data for the wiring of the MPG.

5.2.22 Cs Axis Switching

- **Related signals**

Signal type	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Control command	M14	CS axis switches from the speed to the position			
	M15	CS axis switches from the position to the speed			
G、Fsignal	G27.7	CS axis switches from the speed to the position			
	F44.1	Position mode shift finish			

- **Function description**

When the function of rotary axis CS is valid, the speed mode can be switched to the position mode by executing M14, and the position mode can be switched to the speed mode by executing M15. The spindle rotary output is cancelled by executing M14/M15 to switch.

When the speed mode of CS axis is switched to the position mode, alarm A0.5 will be issued if the state output signal F44.1 is not received at the time set by DT29.

5.2.23 Index Table Tightness Control

- **Related signals**

signal type	symbol	Signal significance	Pin	PLC static	CNC diagnosis
Input signal	DTLO	Release signal of the indexing table in place	CN61.40	X3.3	
	DTCL	Signal for clamping the dividing table in place	CN61.41	X3.4	
		Machine tool panel indexing table elastic button		X26.4	
	BUCLP	Release signal of dividing table		F61.0	
	BCLP	Dividing table clamping signal		F61.1	
Output signal		Release the output of the dividing table	CN62.38	Y3.1	
		Dividing table clamping output	CN62.39	Y3.2	
		4^{th} axis enable off signal		G126.3	
		5^{th} axis enable off signal		G126.4	
		Machine panel indexing table elastic button indicator light		Y26.4	
	BEUCL	Feedback signal indicating completion of release of the dividing table		G38.0	
	BECLP	Feedback signal indicating completion of clamping on the dividing table		G38.1	
Instruction input	M22	Release the dividing table			
	M23	Dividing table clamping			

- **Related parameters**

K0017	ITI	SVOFF					DAAN	ADHL

ADHL =1: The output signal of the dividing table is maintained;

=0: The output signal of the dividing table is not maintained;

DAAN =1: The control axis of the indexing table is the 5^{th} axis;

=0: The control axis of the dividing table is the 4^{th} axis;

SVOFF =1: The servo disconnect function is effective when clamping and releasing;

=0:When clamping and releasing, the servo disconnect function is invalid.

ITI =1: The indexing function of the indexing table is effective;

=0: The indexing function of the indexing table is invalid.

Note: The K17.1 parameter must be consistent with the system NO.1030 data parameter settings to ensure correct manual movement and zeroing operations of the dividing axis.

DT0012	The timeout for loosening/clamping the dividing table is set to 10000ms by default.

If the X3.3 release signal cannot be detected within the DT12 set time when the dividing table is released, the system will sound an alarm;

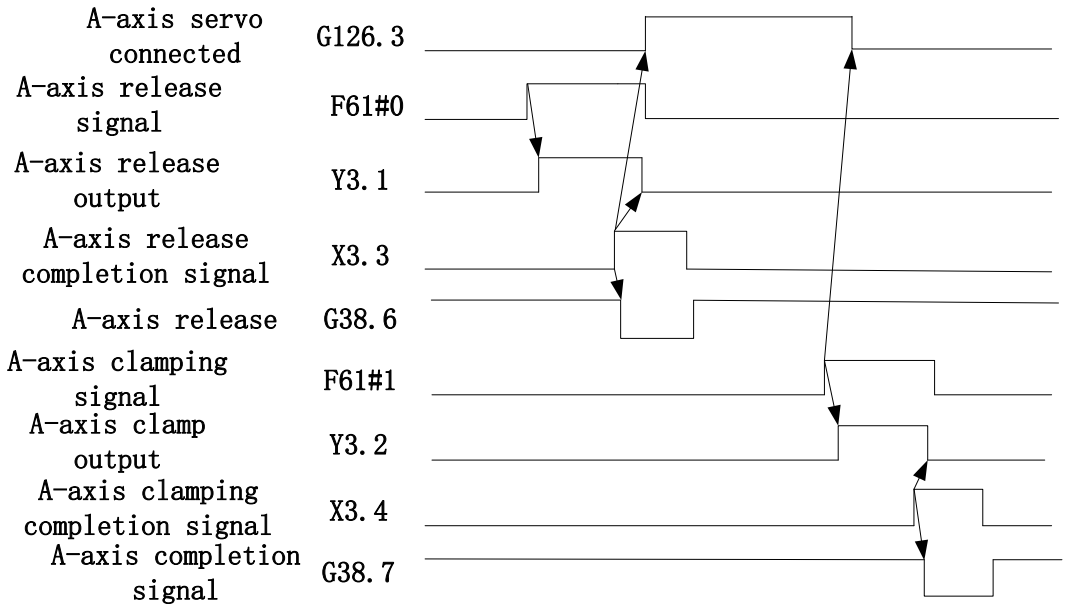
If the X3.4 clamping in place signal cannot be detected within the DT12 setting time when the dividing table performs clamping, the system will sound an alarm.

DT0013	0: It is necessary to detect the signal of the degree of tightness of the dividing table; >0: Failure to detect in place signal, execution time for the looseness of the dividing table

The default value is 0, and the clamping of the dividing table needs to be detected with a proper signal.

DT0038	Delay the output servo enable time when the indexing table is released
DT0039	Delay the time for closing the servo enable when clamping the dividing table

- **Action sequence diagram**



- **Control logic**

Release of the indexing table: When executing the indexing axis movement command or M22, output Y3.1. When receiving the signal X3.3 to release the indexing table, stop Y3.1 output and output the indexing table servo enable signal.

Clamping of the dividing table: When the dividing table rotates or M23 is executed, Y3.2 is output, and the servo enable signal of the dividing table is turned off at the same time. Stop Y3.2 output when receiving the clamping signal X3.4 from the dividing table.

When manually moving the indexing axis, the indexing table needs to be loosened in place before it can be moved, otherwise an alarm will be triggered.

5.2.24 Safe Gate Function

- **Related signals**

Signal type	Signal symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
input signal	SAGT	Safe gate open/close feedback signal	CN61.15	X1.6	

- **Related parameters**

K0014						SGSP	SPB	SDR

SDR =1: Safe gate function is valid;

=0: Safe gate function is invalid;

SPB =1: SAGT is connected with +24V and safe gate is closed;

=0: SAGT is disconnected with +24V and safe gate is closed;

SGSP =1: Open the safe gate in the running and do not shut spindle and coolant;

=0: Open the safe gate in the running and shut spindle and cooling.

- **Function description**

GSK980MDi Plus standard ladder diagram offers safe gate function. And if safe gate parameter K14.0 setting is active, open the safe gate, the CNC will remind that: safe gate is not closed yet.

In auto mode, press CYCLE START, if the safe gate is not closed then the CNC will promote alarm: safe gate is not closed yet, forbidden auto running.

If open the safe gate during the auto running mode, the CNC will cease in any minute, and promote: safe gate is not closed yet. At this time, shutting the spindle and cooling or not is decided by parameter K14.2. When shutting the spindle and cooling is set, then close the safe gate then restart the CNC, which will run as the former state; if shutting the spindle and cooling is not set, then close the safe gate then restart the CNC, open the spindle and cooling.

5.2.25 Holding Brake Control

- **Related signals**

Signal type	address symbol	signal meaning	CNC Pin PLC	PLC Address	CNC Diagnosis
Output signal	SRDY	Brake control signal output	CN62.3	Y0.2	

- **Related parameters**

K0018			SRID					

SRID =1: The system controls the brake function effectively;

=0: The system control brake function is invalid.

DT37	The delay time for releasing the brake after the system is powered on. Initialize to 4s

- **Function description**

  After the system is powered on, the delay time set by DT37 outputs the brake release signal Y0.2. When an emergency stop alarm or drive alarm occurs, turn off the Y0.2 output to brake the Z-axis motor until the alarm is eliminated before outputting Y0.2.

5.2.26 Spindle Releasing/Clamping Tool

- **Related signals**

Signal Type	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal	JOGT	Manual spindle releasing/clamping tool control	CN61.14	X1.5	
	TACT	Check spindle releasing tool in position	CN61.16	X1.7	
	TGGH	Check spindle clamping tool in position	CN61.29	X2.0	
Output signal	Y2.0	Spindle releasing/clamping tool output	CN62.29	Y2.0	
Control instruction	M54	Spindle releasing tool			
	M55	Spindle clamping tool			

- **Control parameter**

DT0030	Check delay time of spindle releasing/clamping tool

When the spindle is performing clamping tool, if the CNC cannot detect the X2.0 clamping tool in position signal in the time DT30 set, it will alarm;

When the spindle is performing releasing tool, if the CNC cannot detect the X1.7 releasing tool in position signal in the time DT30 set, it will alarm.

- **Function description**

  There is a strict interlock relationship between spindle releasing/clamping tool function and spindle rotation, to avoid damaging the spindle during the use, the interlock relationship is as below:

  Forbidden starting spindle when spindle releasing tool is using, or CNC will alarm;

  Forbidden starting spindle releasing/clamping tool, when spindle is rotating or CNC will alarm;

  Manual spindle releasing/clamping tool is only can be used by MPG, and hand wheel, or CNC will alarm. Manual releasing/clamping tool is active if long pressed. Press the releasing tool key not release means that the spindle releases the tool, and if release it then the spindle clamps the tool.

5.2.27 Pot Tool Magazine

- **Used range**

  This ladder diagram applies to BT40, BT50 and other similar logic tools.

- **Related signals**

Signal Type	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal	TDEC	back to 0 in position detection	CN61.1	X0.0	
	TPCH	Cylinder pressure test	CN61.3	X0.2	
	TCOT	Tool post counting	CN61.5	X0.4	
	THCH	Toolcase horizontally in position	CN61.7	X0.6	
	TVCH	Toolcase vertically in position	CN61.8	X0.7	
	THSP	ATC arm of pot tool magazine stopping detection	CN61.9	X1.0	
	THGT	Check ATC arm of pot tool magazine catching in position	CN61.10	X1.1	
	THZP	ATC arm of pot tool magazine original point detection	CN61.11	X1.2	
	JOGT	Spindle releasing/clamping tool control	CN61.14	X1.5	
	TRCH	Spindle releasing tool in position detect	CN61.16	X1.7	
	TGCH	Spindle clamping tool in position detect	CN61.29	X2.0	
	THOV	Motor controlling ATC arm of pot tool magazine overloaded	CN61.30	X2.1	
	TTOV	ATC arm motor overloaded	CN61.31	X2.2	
	JOGT	Spindle releasing/clamping tool control	CN61.14	X1.5	
Output signal	THOR	Toolcase vertical output	CN62.13	Y1.4	
	TVER	Toolcase horizontal output	CN62.14	Y1.5	
	TCW	Toolpot motor CW	CN62.15	Y1.6	
	TCCW	Toolpot motor CCW	CN62.16	Y1.7	
	TROT	Releasing tool output	CN62.29	Y2.0	
	THOT	ATC arm output motor	CN62.30	Y2.1	
		Tool change indicator		Y19.0	

- **Related parameter**

K0011	HALT1	HALT2					APCH	DBUG

DBUG =1: Open tool debug mode
=0: close tool debug mode

APCH =1:Don't need to test cylinder pressure is too low or not
=0: Test the cylinder pressure is too low or not

HALT1, HALT2 parameter set as follows:

	BT40tool magazine	Turret tool magazine	Tool magazine function invalid	Tool magazine function invalid
HALT1	1	0	0	1
HALT2	0	1	1	1

K0012	CTRZ	TKEY			TCHL			

TCHL =1: The signal of tool places count is low level
=0: The signal of tool places count is high level

TKEY =1: The tool magazine key in CNC panel shield

=0: The tool magazine key in CNC panel not shield

CTRZ =1: Return the tool zero by T.MAG.CW valid

=0: Return the tool zero by T.MAG.CW invalid

- **Data parameter:**

5025	Original tool No.
5026	Max. tool No.
5032	Original toolcase corresponding to D address No. in PLC
5033	Tool No. placing tool change position corresponding to sequence of D address
6044	Call the initiate M code of the subroutine
6045	Call the initiate program No. (0-9999) corresponding to M code
6046	Allow number of M code calling subroutine (0-8000;0:this function invalid)

- **M function instructions**

M54: Spindle tool releasing.
M55: Spindle tool clamping.
M65: Toolcase falling down..
M66: Toolcase rising up
M68: ATC arm catching tools (motor controlling tool changing starts for the first time).
M69: ATC arm changing tools (motor controlling tool changing starts for the second time).
M70: ATC arm back to situ place (motor controlling tool changing starts for the third time).
M19: Spindle orienting.

- **PMC Data Sheet**

1. D addresses (One-byte binary number):
   D408 is the target toolcase No., initial value 0.
   D409 is the current toolcase No., initial value 0.
   D410 tool No. in spindle, default 0
   D411-430tool No. 1-20, initial value is 1-20 respectively.
2. C address:
   C21: manual, toolpot zero counter.
   C22: auto tool changing counter.
   C23: auto tool changing counter.

- **Pre-tool -chosen function**

When running the program, input T instruction to change tools. Toolpot will search for the target toolcase, at meantime the T auxiliary function ends. The program continues to be executed. After calling M6the tool change macro program, if the toolpot don't stops, please wait till it stops before the toolcase operations. The working efficiency can be enhanced by shorting the tool change time when in the programming, the toolpot rotation accomplished before M6 calling the macro program, e.g.:

```
T10              //toolpot rotates, and the system executes the next block of the program
G01 X10 Y10      //program running
G00 X0 Y0
M6               //tool change, change the tool in spindle into Tool No.10
```

- **Debug instructions**
  - **Parameters debugging**
    - Set K11.7,K11.6 valid for pot tool magazine;
    - Set K11.5 valid for spindle releasing/clamping tool function;
    - Other parameters K11, K12 stay the same, and change them until needed when debugging. After debugging, recover the parameters to initial conditions.
    - Set the number of tool magazine:

5025	Initial tool No.
5026	Max. tool No.

Set the Max. tool No. the total toolcase numbers in the current machine tool magazine. To change the initial tool No., the ladder diagram needed to be altered, so do not mend the parameter unless really necessary.

- TOOL MAGAZINE parameter setting :

5032	Initial tool case number corresponding to sequence of D address in PLC
5033	Tool No. placing tool change position corresponding to sequence of D address

The parameters above are suitable for the current standard ladder diagram setting. If want to change the ladder diagram, you have to set the parameters above.

- Parameters setting for CNC calling subroutine:

6044	6
6045	9001
6046	1

The setting completed, tool change macro program P9001can be called by M6, no need by M98.

- **Toolpot zero position**

It is only in back to reference point mode, do the toolpot back to zero position can be implemented. So when execute this function, please make sure whether zero signal exists or not. CNC cannot implement this function without a zero signal.

1: Only when the toolcase is risen up, that do toolpot can back to zero position, otherwise not.

2:If the toolpot wants back to zero position, the CNC will detect X0.0 signal. If this signal turns one circle and the toolpot keeping spinning and cannot find 0 point, please check the zero signal in machine tools electric is correct or not.

3: After the toolpot back to zero point, the target and current toolcase data will be all set to1, indicating that the current toolcase No. 0 is 1.If the data corresponding to the toolpot is not 1, please check machine tools electric is correct or not.

4: There is no 

key in GSK980MDi Plus panel. If other keys needed to be set, the ladder diagram should be altered.

- **Manual toolpot rotation**

In GSK980MDi Plus system, in the mode of MPG, STEP or MANUAL, press the tool magazine rotation key, the toolpot will rotate CW a tool location. If the toolpot will not rotate:

1: Please make sure the tool magazine parameter K11.6 and K11.7 set in pot tool magazine valid;

2: Please make sure that the CNC is in the mode of MPG, STEP or MANUAL;

3: Press the key, check up in ladder->PLC, to see Y1.6 has an output or not. If Y1.6 has an output, please check the machine tools electric part that whether it has inputted to toolpot motor to start toolpot spin.

4: There is no 

function in 980MDi-V panel. If other keys needed to be set, the ladder diagram should be altered.

- **Manual spindle releasing/clamping tool**

Please make sure there is no tool in spindle when test the degree of tightness of the spindle to avoid tool damage.

In the mode of MPG, STEP or MANUAL, press the release/clamp tool key in the machine spindle. Press and do not release which means asking spindle to release the tool, air pressure outputted. Release the key means asking spindle to clamp the tool. If the releasing/clamping tool does not work:

1: Make sure K11.5 spindle releasing/clamping tool parameter set to1;

2: When spindle clamping tool, observe X1.5 is changing or not in Ladder->PLC. If it is not changed, check the machine tools electric.

3: If X1.5 changed, please observe Y2.0 has an output or not in Ladder->PLC. If it do has, check the machine tools electric, and the motor controlling spindle releasing/clamping tool is started or not.

4: If there is an overtime alarm when tool releasing/clamping, please observe X2.0 or X1.7 in Ladder->PLC is in position or not. If not, please check the machine tools electric.

- **Auto spindle releasing/clamping tool**

When test the degree of tightness of the spindle, please make sure that there is no tool in spindle to avoid tool damage.

In AUTO or MDI mode, and spindle stopped spinning, input M54 (spindle releasing tool), or M55 (spindle clamping tool) function, then press CYCLE START key. If the following situations happen:

1: Illeage M code alarming, please check K11.5 parameter is open or not;

2: If the releasing or clamping command has been input, but there is no output about the tightness. Please observe Y2.0 outputted or not in Ladder->PLC. If it is outputted, please check the machine tools electric, and whether the motor controlling spindle releasing/clamping tool is started or not.

3: If there is an overtime alarm when releasing/clamping tool, please observe X2.0 or X1.7 in Ladder->PLC is in position or not. If not, please check the machine tools electric.

- **Toolcase falling and rising**

Start toolcase falling or raising, please make sure that the toolpot ends spinning, or there may be a danger. The system has a control of this condition, so when toolcase spinning, the falling or rising action cannot achieve.

M65 toolcase falls, M66 toolcase raises. If it is not completed:

1: When implementing M65, observe Y1.4 in Ladder->PLC to see whether it is outputted. When implementing M65, observe Y1.4 in Ladder->PLC to see whether it is outputted. If it is outputted, but the toolcase does not move, then please check the machine tools electric ,

the connection to motor and the motor in good condition or not.

2: If either Y1.4 or Y1.5 has no output, please check the ATC arm is in situ or not. If ATC arm is in situ, toolcase can not fall because of the falling definitely causes the damage to the ATC arm.

3: If either Y1.4 or Y1.5 has no output, please check the ATC arm is in situ or not, and signal X1.2 is 0 or not. If the signal turns out to be 1 that means the ATC arm is in situ and toolcase actions forbidden.

4: The tool magazine forward and backward keys in 980MDi-V panel can realize the functions of M65 and M66. To backward and fall down the tool magazine, make sure that toolpot spinning stopped and the ATC is in situ, or the operation cannot be achieved. If the operations cannot be done even if the massages mentioned above is ensured, please check according to Step 1 and 2.

■ **Start the motor of ATC arm**

Before debugging and starting the motor of ATC arm, whether the spindle is in the safe position or spindle is in the tool change position should be made sure. If the spindle is not in the safe position, ATC arm will hit against it and be damaged. If there is a tool in the spindle, first implement spindle orientation function, and make sure that the orientation can make ATC arm accomplishes clamping the tool, or the tool will be damaged.

The system has a protection mechanism when the ATC arm is started:

1: To let spindle orientate and back to NO.2 reference position, to debug, please set parameter K11.2 to 0.

2: If the current toolcase is fallen, to ensure the tools will not drop, to debug, set K11.4 to 1.

3: The ATC arm changes the tool and is back to situ, and if the spindle releasing/clamping tool detection needed, to debug, please set K11.3 to 1.

Conditions permitting, the parameters cannot be changed, but if they changed during the debugging, please reset them later.

During the debugging, input M68 (tool change motor the 1st time catching tool) M69 (tool change motor the 2nd time catching tool) M70 (tool change motor the 3rd time catching tool).

1: Please check the ATC arm is available and debugging parameter is set correctly when the ATC arm responds nothing to the start signal.

2: set the parameter, then input M instruction, observe Y2.1 in Ladder->PLC to see whether it starts the rotation of the ATC arm or not. If Y2.1 has an output, please check the machine tools electric or the motor of the ATC arm.

3: Please check whether the in position signals, signalX1.0, X1.1, are changing in Ladder->PLC, when the ATC arm cannot be in position or the rotation of it cannot stop. Please make sure the electric of the two signals feedbacks normally, if both of signals do not change.

■ **Reference point setting and machine tool interlock**

The setting of reference point of the tool change is the key point to tool change. If the setting is not correct, the ATC arm will strike, causing machine damage by this. There is a machine tool function in GSK980MDi Plus system configure. That is, the Z axis (spindle) cannot move when the ATC arm is changing tools to avoid the damage of the machine tool. The debug setting method:

1: Set K11.0 to 0(interlock function invalid).

2: Move Z axis to a safe position to make sure when the ATC arm is spinning, it will not hit

the spindle.

3: Make the ATC arm changing tools (implement M68 or M69).

4: Move Z axis slowly to the ATC arm. The hand wheel is recommended.

5: Configure the equipments the tool change debugging needs. M19 is used to adjust the orientation of the changing tool and the tool change position of the system.

6: After the tool change position is set, input machine coordinate of the current tool change position into data parameter NO.1241(the Z axis machine coordinate of the 2nd reference point of all-axis).please compute and confirm the smallest unit of the current system to avoid mistakes.

■ **Overall debugging of pot tool magazine:**

Only ensure that the debugging can complete the functions of toolpot spinning, toolcase falling and down, spindle orientation accomplishment and the ATC arm action, can overall debugging be taking.

1) When T instruction inputted to change tools, the system will still go on to implement the program and at this moment the toolpot will search the target by preprocessing. At meantime T instruction and M tool change program instruction can in the same block. However in this way, this program runs less efficient because of waiting for the toolpot stops before tool changed. The program is as follows:

```
T10
G01 X10 Y10
G00.....
M6
G00.....
T11 M6
.....
```

2) The subroutine of the tool change is as follows. And please run subroutine in STEP mode for the first time, and make sure there is no tools in the spindle to avoid damage of the tool machine if debug errors happen.

```
O9001 (O9001)
N1 #501=#4003                  // save G9091
N2 IF [#1002EQ1] GOTO 12       //if the target tool of the tool change is the current one, then skip to the next
N3 #1101=1                     //issue macro program signal
N4 G91 G30 Z0                  //back to the reference point of the tool change
N5 IF [#1000EQ1] GOTO 7        //wait for toolpot stopping, spindle orientation accomplished and toolcase falling
N6 GOTO 5                      //toolpot not stopping yet
N7 M68                         //the motor of the ATC arm starts for the first time (the ATC arm catches the tool)
N8 M54                         //the spindle releases the tool
N9 M69                         // the motor of the ATC arm starts for the second time (the ATC arm changes the tool)
N10 M55                        // the spindle clamps the tool
N11 M70                        // the motor of the ATC arm starts for the third time (the ATC arm returns to the situ)
```

N12 G#501 //recover the value of G9091
M99

■ **Exception Handling**

Handle the chaos tools in toolpot:

1: When the current toolcase is in the right position for tool falling, then open the switch of the parameter, in the MDI mode, and input the right corresponding tool to the toolcase number in ATC SET.

2:When the current fallen toolcase number is not correspond to the toolpot, then press the CHANGE key in ATC SET ,do not release, at meantime moving the up and down keys to the proper place. In this way, the sequence number of the fallen toolcase No. is adjusted.

3: Please alter toolcase No. 0 when there is an error of the tool NO. in spindle .

4: When input the tool No. and toolcase No., the one-to-one relationship should be noticed. If there are reduplicate tools in the toolcase sequence, the system will issue alarms, and errors will happen when tools are changed.

■ **Related signals**

Alarm address	Alarm No.	Alarm message	Interlock logic
X0.2	A1.1	Cylinder low pressure	Cannot change tools correctly when in low pressure
	A1.2	too long rotation time for the toolpot	Avoid the damage of toolpot when spinning
X2.2	A1.3	Toolpot motor overload	Toolpot overloaded and stop operations on tool magazine
X0.6	A1.4	Toolcase must be horizontally in position when the toolpot is rotating	Avoid damage of the toolpot
X2.1	A1.5	the ATC arm motor overloaded	Motor overloaded and stop operations on tool magazine
X1.7	A2.0	overtime detection for releasing tool alarm	releasing tool is not in position and releasing tool time is too long
X2.0	A2.1	overtime detection for clamping tool alarm	clamping tool is not in position and clamping tool time is too long
X1.7	A2.2	Cannot make spindle spinning when releasing tool	Cannot spinning when the spindle is releasing tool
X1.5	A2.3	No spindle releasing/clamping tool when the spindle is spinning	No spindle releasing/clamping tool when the spindle is spinning
X0.7	A4.0	Toolcase of pot tool magazine falling overtime, please check accomplishment signal(X0.7)	Have not detected the fallen accomplishment signal; toolcase falling action costs too long time.
X0.6	A4.1	Toolcase of pot tool magazine raising overtime, please check accomplishment signal(X0.6)	Have not detected the raised accomplishment signal; toolcase raising action costs too long time.
X1.0 X1.1	A4.2	The tool change detection signal of the ATC arm of pot tool magazine is overtime, please check signal(X1.0,X1.1)	Have not detected the catching tools and stop signal; the ATC arm acts overtime.

X1.2	A4.3	back to situ detection signal of the ATC arm of pot tool magazine is overtime, please check accomplishment signal(X1.2)	Have not detected the ATC arm back to 0 signal; the ATC arm acts overtime.
X1.0 X1.1 X1.2	A4.4	Rotation check accomplishment signal of the auto ATC arm of pot tool magazine is overtime, please check signal(X1.0,X1.1,X1.2)	The falling accomplishment is not detected in MANUL; the key of toolcase falling has been pressed too long time.

- **Note:**

1、T displaying in the CNC interfaces stands for the tool No. of the current tool used by the spindle, not the toolpot No. of the current toolpot;

2、In AUTO, MDI, and DNC mode, the appointed T instruction stands for tool no. not the toolpot No.;

3、When toolpot chooses tool, T displaying in the CNC interfaces is not changed, but the toolpot is rotating and pre-choose the corresponding tool;

4、When machine is back to 0,press the tool change key in the panel, the toolpot will rotate to Toolcase No.1,and T displaying in the CNC interfaces is not changed;

5、When M69 is implemented, the tool No. of the current tool in the spindle will change with tool in current toolcase and then update the display of the current tool;

6、The setting value of data parameter 5026 must be consistent with the numbers of the toolcase in the toolpot;

7、Overall debugging, the first time for testing, the system should be in the mode of STEP and with no tools. During the test, please press RESET immediately, when find the ATC arm after back to the reference point, changing tools in not correct position where will crush the system. Then, reset the reference point of tool change to avoid machine damage.

5.2.28 Turret Tool Magazine

- **Related signals**

Signal type	Symbol	Description	Pin-out No	PLC State	CNC Diagnosis
Input signal	TDEC	Back to 0 in position detection	CN61.1	X0.0	turret
	TPCH	Cylinder pressure test	CN61.3	X0.2	turret
	TCOT	Tool post counting	CN61.5	X0.4	turret
	THCH	Toolpot backward in position	CN61.7	X0.6	turret
	TVCH	Toolpot forward in position	CN61.8	X0.7	turret
	JOGT	Spindle releasing/clamping tool control	CN61.14	X1.5	turret
	TRCH	Spindle releasing tool in position detection	CN61.16	X1.7	turret
	TGCH	Spindle clamping tool in position detection	CN61.29	X2.0	turret
	TTOV	Tool change motor overloaded	CN61.31	X2.2	turret
Output signal	THOR	Toolpot forward output	CN62.13	Y1.4	turret
	TVER	Toolpot backward output	CN62.14	Y1.5	turret
	TCW	Toolpot motor spinning CW	CN62.15	Y1.6	turret
	TCCW	Toolpot motor spinning CCW	CN62.16	Y1.7	turret
	TROT	Spindle releasing/clamping tool output	CN62.29	Y2.0	turret

- **Related parameter**

K0011	HALT1	HALT2					APCH	DBUG

DBUG =1: Open tool debug mode;

=0: Close tool debug mode.

APCH =1:Don't need to test cylinder pressure is too low or not;

=0: Test the cylinder pressure is too low or not.

HALT1, HALT2 parameters set as follows:

	BT40 tool magazine	**Turret tool magazine**	**tool magazine function invalid**	**tool magazine function invalid**
HALT1	1	0	0	1
HALT2	0	1	1	1

K0012	CTRZ	TKEY	ARSE		TCHL			

TCHL =1: The signal of tool places count is low level;

=0: The signal of tool places count is high level.

ARSE =1:Bamboo hat model tool magazine advance and retreat transmission signal to keep;

=0:Bamboo hat model tool magazine advance and retreat transmission signal to not keep.

TKEY =1: The tool magazine key in CNC panel shield;

=0: The tool magazine key in CNC panel not shield.

CTRZ =1: Return the tool zero by T.MAG.CW valid;

=0: Return the tool zero by T.MAG.CW invalid.

Data parameters:

5025	Initial tool No.
5026	Max. tool No.
6044	Call the initiate M code of the subroutine
6045	Call the initiate program No. (0～9999) corresponding to M code
6046	Allow number of M code calling subroutine (0～8000;0:this function invalid)

- M function instructions

M54: Make the spindle release tool.

M55: Make the spindle clamp tool.

M65: Make the toolpot turn right (close to the spindle).

M66: Make the toolpot turn left (away from the spindle).

M60: Start tool magazine rotating.

M61: Finish tool change, and update the system tool messages.

M19: The spindle orientates.

- PMC Data Sheet

1. D addresses (One-byte binary number):

D409 is the current toolcase No., initial value 0.

D408 is the target toolcase No., initial value 0.

2. C address:

C21: manual, toolpot zero counter

C22: auto tool changing counter.

C23: auto tool changing counter.

- Debug description
  - **Parameters debugging**

Set K11.7,K11.6 valid for pot tool magazine;

Set K11.5 valid for spindle releasing/clamping tool function;

Other parameters K11, K12 stay the same, and change them until needed when debugging. After debugging, recover the parameters to initial conditions.

Set the number of tool magazine:

5025	Initial tool No.
5026	Max. tool No.

Set the Max. tool No. the total toolcase number in the current machine tool magazine. To change the initial tool No., the ladder diagram needed to be altered, so do not mend the parameter unless really necessary.

parameters setting for CNC calling subroutine:

6044	6
6045	9000
6046	1

The setting completed, tool change macro program P9001can be called by M6, no need by M98.

  - **Toolpot back to 0 operations**

It is only in back to reference point mode, do the toolpot back to zero position can be implemented. So when execute this function, please make sure whether the zero signals exist or not. CNC cannot implement this function without a zero signal.

1: Only when the toolpot is back in position, that do toolpot can back to zero position, otherwise not.

2: When the toolpot wants back to zero position, the CNC will detect X0.0 signal. If this signal turns one circle and the toolpot keeping spinning and cannot find 0 point, please check the zero signal in machine tools electric is correct or not.

3: After the toolpot back to zero point, the target and current toolcases data will be all set to1, indicating that the current toolcase No. 0 is 1.If the data corresponding to the toolpot is not 1, please check machine tools electric is correct or not.

4: There is no 

key in GSK980MDi Plus panel. If other keys needed to be set, the ladder diagram should be altered.

  - **Manual spindle releasing/clamping tool**

Please make sure there is no tool in spindle when test the degree of tightness of the spindle to avoid tool damage.

In the mode of MPG, STEP or MANUAL, press the release/clamp tool key in the machine

spindle. Press and do not release which means asking spindle to release the tool, air pressure outputted. Release the key means asking spindle to clamp the tool. When the releasing/clamping tool does not work:

1: Make sure K11.5 spindle releasing/clamping tool parameter set to1.

2: When spindle clamping tool, observe X1.5 is changing or not in Ladder->PLC. If it is not changed, check the machine tools electric.

3: If X1.5 changed, please observe Y2.0 has an output or not in Ladder->PLC. If it do has, check the machine tools electric, and the motor controlling spindle releasing/clamping tool is started or not.

4: When there is an overtime alarm when releasing/clamping tool, please observe X2.0 or X1.7 in Ladder->PLC is in position or not. If not, please check the machine tools electric.

5: When there is a releasing/clamping tool key in system panel, please open there switch of K12.0 parameter. The release/clamp tool keys in system panel and in machine should not be used at the same time, for only one of them is valid, to avoid disoperation.

■ **Auto spindle releasing/clamping tool**

Please make sure there is no tool in spindle when test the degree of tightness of the spindle to avoid tool damage.

In AUTO, or MDI mode, when the spindle is still, input M54 (spindle releasing tool), or M55 (spindle clamping tool) function, press CYCLE START key. When the spindle does not work out:

1: When illegal M code alarm exists, please make sure K11.5 parameter is open.

2: When after being inputted, the releasing/clamping tool has no output, please observe Y2.0 has an output or not in Ladder->PLC. If it do has, please check the machine tools electric, and make sure the motor starts the spindle releasing/clamping tool or not.

3: When there is an overtime alarm when releasing/clamping tool, please observe X2.0 or X1.7 in Ladder->PLC is in position or not. If not, please check the machine tools electric.

■ **Reference point setting**

There are two reference points needed to be set in GSK980MDi Plus turret tool magazine, that is, the 2nd reference point and the 3rd reference point (data parameter No.1241an data parameter NO.1242).

The main aim of setting these two reference points is that G28 is not needed to impact the zero block every time. When the distance of the travel is long enough (the spindle moves to not higher than the block of the toolpot). That the 3rd reference point is set at a safe and not block hitting position when the tools are changed is suggested. When the distance of the travel is not long enough, then there is no need to set the 3rd reference point. In this situation the 3rd reference point is 0, the machine zero.

To set the 2nd reference point, first raise the spindle to safe place, push the manipulator to the right, and move the Z axis slowly to the place where the manipulator can reach by hand wheel or in MANUAL mode. During the process the debugging tools should be collocated, because the positions are needed to be relatively exact to avoid possible tool hitting in the next step tool changing.

■ **Manipulator boosting and retreating**

The spindle needs to accomplish the orientation and return to the 2nd reference point, as well as #1100=1 when the tool magazine is boosting or retreating. In debugging, no need to orient or return to the reference point, please set K11.2 to 0. After debugging, set it to 1, to improve

the safety of the tool magazine.

M65: The manipulator catches tools boosting to the right (in catching, make sure where to catch the tool and make the spindle orientate to avoid manipulator hitting the machine)

M66: The manipulator retreats to the left (in retreating, make sure the tool clamped to avoid that the tool falls from it)

If the function is not worked out:

1: Parameter K11.2=1, make sure the spindle orientation accomplished and has returned to the reference point. If both are done, please #1100=1, in mode of MDI or AUTO. If the function does not worked out, please make sure Y1.4 (manipulator boosting) or Y1.5 (manipulator retreating) is 1 in Ladder->PLC. If it is 1, please check the machine tools electric;

2: If there is a manipulator operation key, the above action can be achieved in non-auto mode.

- **Overall debugging**

Only every step above has been finished correctly that can the overall program debugging of tool change be taking in MDI or AUTO mode.

Program:

Main program:

```
O0001 (O0001)                    // Tool change main program
T01 M6                           // call subroutine by M instructions other than M98.
T05 M6
M30
```

Tool change subroutine:

```
O9000 (O9000)
N1 #501=#4003                    //save G9091
N2 IF [#1000EQ1] GOTO 17         //if the target tool of the tool change is the current one, then
                                 skip to the next
N3 G91 G30 Z0                    //back to the 2nd reference point
N4 M19                           //spindle orientation accomplished
N5 #1100=1                       // back to the reference point and spindle orientation
                                 accomplished
N6 M65                           //toolpot boosting to the right
N7 M54                           //tool releasing
N8 G30 Z0 P3                     // back to the 3rd reference point
N9 #1100=0                       //away from the 2nd reference point
N10 M60                          //star toolpot rotation
N11 G30 Z0  P2                   // back to the 2nd reference point
N12 M55                          //tool clamping
N13 #1100=1                      //notify to back to the 2nd reference point
N14 M66                          // toolpot boosting to the left
N15 M61                          //tool changes completed and update the tool message
N16 #1100=0                      //clear sign
N17 G#501                        // recover the value of G9091
M99
```

- **Exception Handling**

Handle the chaos tools in toolpot:

When the current toolcase is in the right position for tool falling, then open the switch of the parameter, in the MDI mode, and just input the current toolcase No. into D409 to adjust.

■ **Related alarm and retractions**

Alarm address	Alarm No.	Alarm message	Interlock logic
X0.2	A1.1	Cylinder low pressure	Cannot change tools correctly when in low pressure
	A1.2	Too long rotation time for the toolpot	Avoid the damage of toolpot when spinning
X2.2	A1.3	Toolpot motor overload	Toolpot overloaded and stop operations on tool magazine
X0.6	A1.6	When turret tool magazine starts toolpot rotating,Z axis is not away from the 2nd reference point	Avoid damage of the toolpot
X1.7	A2.0	Overtime detection for releasing tool alarm	Releasing tool not in position and overtime
X2.0	A2.1	Overtime detection for clamping tool alarm	Clamping tool not in position and overtime
X1.7	A2.2	Cannot make spindle spinning when releasing tool	The spindle cannot rotate when releasing tool
X2.0	A2.3	No spindle releasing/clamping tool when the spindle is spinning	The spindle cannot rotate when releasing/clamping tool
X0.7	A4.5	Toolcase of pot tool magazine falling overtime, please check accomplishment signal(X0.7)	Have not detected toolpot forward in position signal and overtime
X0.6	A4.6	Toolcase of pot tool magazine raising overtime, please check accomplishment signal(X0.6)	Have not detected toolpot backward in position signal and overtime

■ **Note:**

1、T displaying in the CNC interfaces stands for the tool No. of the current tool used by the spindle, not the toolpot No. of the current toolpot.

2、In AUTO, MDI, and DNC mode, the appointed T instruction stands for tool no. not the toolpot No..

3、When toolpot chooses tool, T displaying in the CNC interfaces is not changed, but the toolpot is rotating and pre-choose the corresponding tool.

4、In MANUAL, MPG, or STEP mode, tool magazine CW/CCW input valid, the toolpot rotates CW/CCW. The toolcase plus/minus 1, T displaying in the CNC interfaces is not changed.

5、When machine is back to 0,press the tool change key in the panel, the toolpot will rotate to Toolcase No.1,and T displaying in the CNC interfaces is not changed.

6、The setting value of data parameter 5026 must be consistent with the number of the toolcases in the toolpot.

7、The first time for testing, the system should be in the mode of STEP and with no tools. If the positions of the manipulator boosting and tool change are not correct, press RESET key immediately to avoid impacting and tool magazine damage.

5.2.29 Turret Knife Magazine

- **Related signals**

Signal type	Address symbol	Signal significance	CNC pin	PLC address	CNC diagnosis
Input signal	TDEC	Zero in place detection	CN61.1	X0.0	
	TPCH	Cylinder pressure detection	CN61.3	X0.2	
	TCOT	Knife position counting	CN61.5	X0.4	
	TTOV	Overloading of cutterhead motor	CN61.31	X2.2	
Output signal	TCW	The cutterhead motor rotates clockwise	CN62.15	Y1.6	
	TCCW	The cutterhead motor rotates counterclockwise	CN62.16	Y1.7	

- **Related parameters**

K0011	HALT1	HALT2					APCH	DEBG

DEBG =1: Enable tool library debugging mode;

=0: Turn off the tool library debugging mode.

APCH =1: Detecting low cylinder pressure;

=0: Do not detect low cylinder pressure HALT1、HALT2.

The parameter settings are as follows:

	BT40 type knife magazine	Douli style knife magazine	The knife magazine function is invalid	Turret knife magazine
HALT1	1	0	0	1
HALT2	0	1	0	1

K0012					TCHL			

TCHL =1: The low level of the knife position counting signal is valid;

=0: The high level of the tool position counting signal is valid;

System status parameters:

173		TMANL						

TMANL =0: Automatic tool change

=1: Manual tool change

546								EXLM

EXLM =0: The second set of soft limit is invalid;

=1: The second set of soft limit is effective.

Data parameters:

5026	Maximum knife position number

6044	The actual M code for calling subroutines
6045	The starting program number (0～9999) corresponding to the M code that calls the subroutine
6046	The allowed number of M codes for calling subroutines (0-8000; 0: indicates that the function is invalid)

Set the above three parameters as:

6044	6
6045	9002
6046	1

After setting up, call the tool change subroutine O9002 through M6.

1270	The second group of soft limit positive position machine coordinates

When changing the tool, the Z-axis soft limit will switch to the second set of positive limit.

- **M function instruction**

  M60: Start tool magazine rotation

  M1000: Initialize the tool number. D409 and D410 are set to 1.

- **PMC data table**

  1. D address (one byte binary number):

     D409 is the current tool set number, with an initial setting value of 0.

     D408 is the target tool set number, with an initial set value of 0

  - **Operating instructions**

    After setting K parameters K11.7=1, K11.6=1, and status parameter No173.6=0, operations can be performed on the turret magazine

  - **Mechanical zeroing of the cutterhead**

    In the zeroing mode, press the tool magazine zeroing button to rotate the cutter head. When both the zero signal (X0.0) and the tool position counting signal (X0.4) are detected, the cutter head stops rotating. The system has updated the knife number to knife number 1.

  - **Manual knife selection**

    In the handwheel, manual, and single step modes, press the clockwise/counterclockwise tool selection button on the system panel, and the tool disc will rotate clockwise/counterclockwise by one tool position to complete the manual tool change.

  - **Related alarms and restrictions**

Alarm address	alarm number	Alarm content	Interlocking logic
F45.2	A0.4	When the spindle is oriented, the detection of orientation feedback signal timeout (F45.2)	Check spindle orientation timeout, stop orientation
X0.2	A1.1	Low cylinder pressure	Low pressure prevents proper tool change
	A1.2	The rotation time of the cutterhead is too long	Avoid rotating and damaging the cutterhead
X2.2	A1.3	Overloading of cutterhead motor	Stop operating the tool magazine when the cutterhead is overloaded
	A6.2	The spindle is not oriented and cannot enter the tool changing area	Prevent damage to the cutterhead
	A6.3	The Z-axis is not at the second reference point, so the cutterhead cannot be rotated	Prevent damage to the cutterhead
	A6.4	The Z-axis is not at the second reference point, so the tool magazine cannot be reset to zero	Prevent damage to the cutterhead
	A6.5	Cannot start the program in the tool changing area to avoid tool collision	Prevent damage to the cutterhead

- **Precautions**

  1. When the cutterhead rotates, the Z-axis needs to return to the second reference point (G91 G30 Z0) first.
  2. When executing the T command alone, the system does not rotate the cutterhead. After executing M60, the cutterhead rotation control is performed.

- **Macro program related**
  - **Macro variables**

    1、#1000 (G54.0) T code equals the tool number on the spindle, tool change is complete

  - **Macro program**

```
Main program:
O0001 (O0001)//Main program for tool change
T * * M06//Call the tool changing program to change the tool
……
M30

Knife changing program:
O9002(O9002)
N1 # 501=# 4003          //Save G90/G91 information
N2 # 502=# 4002          //Save G17/G18/G19 information
N3 IF [# 1000EQ1] GOTO 10      //If the target tool for tool change is equal to the tool on the
                                 current spindle, then jump
N4 G17 G91 G28 Z0       //Return to the first reference point
N5 M19                   //Spindle orientation
N6 G30 Z0               //Return to the second reference point and have the tool and spindle
                          disengage mechanically
N7 M60                  //Start cutterhead rotation
```

```
N8 G28 Z0          //Return to the first reference point and have the spindle and tool
                   engaged mechanically
N9 M05             //Cancel targeting
N10 G # 501 G # 502//Restore the saved values of G90/G91 and G17/G18/G19
M99
```

- **Debugging steps**

1) Set parameters 5026, 6044~6046, and K11.
2） K11.0=1， Open debugging mode.
3) Set the orientation position of the spindle.
4) Move the Z-axis to the position where the tool disengages from the spindle, and set this position as the second reference point of the Z-axis (parameter 1241).
5) Move the Z-axis back to the zero position of the machine tool.
6) Press the single-stage button to execute T * * M6 for tool change.
7) After debugging is complete, set K11.0 to 0 and turn off debugging mode.

5.2.30 M10/M11 Turn Control

- **Related signals**

Signal type	Signal symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Output signal	ALTO	Turn output signal	CN62.36	Y2.7	
Command input	M10	Y2.7 output			
	M11	Y2.7 output close			

5.2.31 Spindle Ratio

- **Related parameter**

K0010	SPDR							

SPDR =0: Spindle ratio rang is 50%~120%

=1: Spindle ratio rang is 0%~150%

The encoding spindle ratio rang is 50%~120%:

Decimal system	Gn30.3～Gn30.0	Spindle ratio
7	0111	50％
6	0110	60％
2	0010	70％
3	0011	80％
1	0001	90％
0	0000	100％
4	0100	110％
5	0101	120％

The encoding spindle ratio rang is 0%~150%:

Decimal system	Gn30.3～Gn30.0	Spindle ratio
7	0111	50%
6	0110	60%
2	0010	70%
3	0011	80%
1	0001	90%
0	0000	100%
4	0100	110%
5	0101	120%
8	1000	130%
9	1001	140%
11	1011	150%
10	1010	0%
14	1110	10%
15	1111	20%
13	1101	30%
12	1100	40%

5.2.32 Work Light

- **Related signal**

Signal Type	Symbol	Significance	Pin-out No.	PLC State	CNC Diagnosis
Input signal		Work light key signal		X19.5	
Output signal		Work light key indicator		Y19.6	
	WKLT	Work light output signal	CN62.7	Y0.6	

5.2.33 Chip Removal

- **Related signal**

Signal Type	Signal Symbol	Signal Significance	Corresponding Pins	PLC State	CNC Diagnosis
Input signal		Chip removal button on machine panel		X19.6	
		Machine tool panel chip removal reverse button		X24.5	
Output signal		Machine tool panel chip removal button indicator light		Y19.5	
		Machine tool panel chip removal reverse indicator light		Y27.5	
	TKOUT	Chip discharge output signal	CN62.35	Y2.6	
	TKNOUT	Chip removal reversal output signal	CN62.34	Y2.5	

M38: Effective chip removal and forward rotation output.

M39: The chip removal forward rotation output is invalid.

5.2.34 Addition Panel Signal

- **Related signal**

Signal Type	Signal Symbol	Signal Significance	Corresponding Pins	PLC State	CNC Diagnosis
Input signal		Spindle magnification knob		X23.4~X23.7	
		Feed rate knob		X23.0~X23.3	
		Cycle Start	CN65	X28.0	
		Feed hold	CN65	X28.1	
		Emergency stop signal	CN78.4	X9.3	
		Three position switch feed stop	CN63	X27.1	
		Three position switch spindle stop	CN63	X27.0	
Output signal		Cycle start button light	CN65	Y29.0	
		Cycle start button light	CN65	Y29.1	

- **Related parameter**

K0019								
				SWI		APESP	APRI	

APRI =0：Addition panel ratio adjust switch invalid.
=1：Addition panel ratio adjust switch valid.

APESP =0：Emergency stop signal invalid.
=1：Emergency stop signal valid.

SWI =0: The three position switch on the additional panel is invalid;
=1: The three position switch on the additional panel is valid.

- **Function description**

1) When the system is connected to the additional panel, set K19.0=1 to use the spindle magnification and feed magnification band switches of the additional panel.
When the additional panel magnification switch is set to valid, the spindle magnification and feed magnification adjustment buttons on the main panel are invalid.

2) When the three position switch is turned to the feed stop position during program execution, the program pauses. When reset to the feed allowance position, the program continues to run.
When the three position switch is turned to the spindle stop position during program execution, the spindle stops. When the spindle is moved back to the allowed position, the spindle restarts.
The spindle orientation, gear shift swing, and program non running state are not controlled by the three position switch.

5.2.35 Handwheel Trial Cutting

- **Related signal**

Signal Type	Signal Symbol	Signal Significance	Corresponding Pins	PLC State	CNC Diagnosis
Input signal		K2 button		X19.7	
		Hand pulse test cutting button (GSK980MDi Plus-V)		X27.7	
		Hand pulse test cutting button (GSK980MDi Plus-H)		X26.2	

- **Function Description**

  In automatic mode, press the handwheel trial cutting button to switch to the handwheel trial cutting state, and then press the cycle start button to run the machining program by shaking the handwheel. The cutting speed of the handwheel is set through system data parameter 790.

5.2.36 Workpiece Blowing

- **Related signal**

Signal Type	Signal Symbol	Signal Significance	Corresponding Pins	PLC State	CNC Diagnosis
Output signal	BLOW	Output signal of workpiece blowing air		Y0.5	
Instruction input	M07	Workpiece blowing start command			
	M09	End instruction for workpiece blowing			