
 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 GSK988T Turning CNC system made by GSK CNC Equipment Co., Ltd.

The user manual describes the programming, operation, installation and connection of this GSK988T Turning 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 cannot be explained one by one due to so many possibilities that may involve with, so the contents that are not specially stated in this manual shall be considered 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.

Book One Programming

**Technical Specification, Product Type, Command and
Program Format**

Book Two Operation

GSK988T CNC Operation Use

Book Three Installation and Connection

GSK988T CNC Installation, Connection and Setting

Appendix

CNC Ladder Function Allocation, Alarm List

contents

Volume I Programming

CHAPTER 1 PROGRAMMING FUNDAMENTALS.....	3
1.1 GSK988T introduction	3
1.1.1 Command system.....	4
1.1.2 Technical specifications	5
1.2 CNC system of machine tools and CNC machine tools	8
1.3 Programming fundamentals	9
1.3.1 Coordinates definition	9
1.3.2 Increment system	11
1.3.3 Max. travel	12
1.3.4 Machine coordinate system	12
1.3.5 Workpiece coordinate system	13
1.3.6 Local coordinate system	13
1.3.7 Reference point	13
1.3.8 Interpolation function	14
1.4 Structure of an NC program	15
1.4.1 Program name	16
1.4.2 Block format.....	16
1.4.3 Word	17
1.4.4 Block number.....	25
1.4.5 Main program and subprogram	26
1.5 Program run.....	26
1.5.1 Sequence of program run	26
1.5.2 Execution sequence of word	27
1.6 Coordinate Value and Dimension	28
1.6.1 Absolute programming and incremental programming	28
1.6.2 Diameter programming and radius programming	28
1.6.3 Decimal programming.....	29
1.6.4 Conversion between metric and inch	30
1.6.5 Linear axis and rotary axis	31
1.7 Grammar Rules	31
1.7.1 Lexical classification	31
1.7.2 Noun explanation	31
1.7.3 Example explanation	32
1.7.4 Expression.....	32
1.7.5 Morphology explanation	33
1.7.6 Related parameters.....	34
1.7.7 Grammar explanation.....	35
CHAPTER 2 MSTF COMMAND.....	40
2.1 M (miscellaneous function).....	40
2.1.1 End of program M02	40
2.1.2 End of program run M30	40

2.1.3	Program stop M00	40
2.1.4	Optional stop M01	41
2.1.5	Subprogram call M98	41
2.1.6	Subprogram call M98	42
2.1.7	Return from subprogram M99	42
2.1.8	M commands defined by standard PLC ladder	43
2.1.9	Spindle CW, CCW, stop M03, M04, M05	44
2.1.10	Cooling control M08, M09	44
2.1.11	Tailstock control M10, M11	44
2.1.12	Chuck control M12, M13	45
2.1.13	Lubricating control M32, M33	45
2.1.14	Spindle automatic gear shifting M41, M42, M43, M44	45
2.1.15	M command notes	45
2.2	Spindle Function	45
2.2.1	Spindle speed analog voltage control	46
2.2.2	Spindle override	46
2.3	Tool function	47
2.3.1	Tool select function	47
2.4	Feed function	50
2.4.1	Cutting feed mode	50
2.4.2	Manual feed mode	51
2.4.3	Automatic acceleration/deceleration	52

CHAPTER 3 G COMMANDS 55



3.1	Summary	55
3.1.1	Modal, non-modal and initial mode	56
3.1.2	Omitting word input	57
3.1.3	Related definitions	58
3.2	Rapid traverse movement G00	59
3.3	Linear interpolation G01	60
3.4	Arc interpolation G02, G03	62
3.5	Dwell G04	65
3.6	Polar Coordinate Interpolation G12.1, G13.1	66
3.7	Metric/Inch Switch G20, G21	69
3.7	Stored Travel Check G22, G23	69
3.8	Reference Point Function	70
3.8.1	Reference point return G28	70
3.8.2	2 nd , 3 rd , 4 th reference point return G30	71
3.9	Related Function of Coordinate System	72
3.9.1	Selecting machine coordinate system position G53	73
3.9.2	Workpiece coordinate system setting G50	75
3.9.3	Workpiece coordinate system selection command	76
3.9.4	Local coordinate system setting G52	78
3.9.5	Plane selection command G17~G19	79
3.10	Fixed cycle command	80
3.10.1	Axial cutting cycle G90	81
3.10.2	Radial cutting cycle G94	84

3.11	Multiple cycle commands.....	87
3.11.1	Axial Roughing Cycle G71	88
3.11.2	Radial Roughing Cycle G72	92
3.11.3	Closed Cutting Cycle G73.....	96
3.11.4	Finishing Cycle G70	101
3.11.5	Axial Grooving Multiple Cycle G74.....	102
3.11.6	Radial Grooving Multiple Cycle G75	105
3.11.7	Notes for multi cycle machining	109
3.12	Threading Cutting	109
3.12.1	Thread Cutting with Constant Lead G32.....	110
3.12.2	Thread cutting with variable lead G34	112
3.12.3	Thread cutting cycle G92.....	114
3.12.4	Multiple thread cutting cycle G76.....	118
3.13	Constant surface speed control G96, constant rotational speed control G97	123
3.14	Feedrate per minute G98, feedrate per rev G99	125
3.15	Macro command.....	126
3.15.1	Variable.....	126
3.15.2	System variable	128
3.15.3	Operation and jump command.....	132
3.15.4	Macro program statement and NC statement	137
3.15.5	Macro program call	138
CHAPTER 4	TOOL NOSE RADIUS COMPENSATION	141
4.1	Application.....	141
4.1.1	Overview.....	141
4.1.2	Imaginary tool nose direction	142
4.1.3	Compensation value setting.....	145
4.1.4	Command format	146
4.1.5	Compensation direction	147
4.1.6	Cautions	149
4.1.7	Application	150
4.2	Tool nose radius compensation offset path.....	151
4.2.1	Inner and outer side.....	151
4.2.2	Tool traversing when starting tool	152
4.2.3	Tool traversing in Offset mode	153
4.2.4	Tool traversing in Offset canceling mode	158
4.2.5	Tool interference check	159
4.2.6	Commands for canceling compensation vector temporarily	161
4.2.7	Particulars.....	165

Volume II Operation

VOLUME II OPERATION	171
Chapter I Overview.....	173
1.1 Operation panel	173
1.2 Introduction of the keypad	174
1.2.1 Editing the keypad	174
1.2.2 Function keys and soft keys	175
1.2.3 State indication.....	176
1.2.4 Machine panel	177
1.3 Overview of modes	181
Chapter II Power on, power off and safety protection	182
2.1 Power on	182
2.2 Power off.....	183
2.3 Overtravel protection	183
2.4 Overtravel protection in memory travel limit	183
2.5 Emergence operation.....	184
2.5.2 Emergency stop.....	185
2.5.3 Feed hold.....	185
2.5.4 Cutting off power supply	185
Chapter III Interfaces	186
3.1 Position interface.....	186
3.1.1 Absolute coordinate	188
3.1.2 Relative coordinate	189
3.1.3 Machine coordinate.....	190
3.1.4 Comprehensive coordinate.....	191
3.1.5 Setting the relative coordinate.....	192
3.1.6 Switching between the mode and the comprehensive information.....	193
3.2 Program interface.....	194
3.2.1 Executing the program	197
3.2.2 Opening the editing program	198
3.2.3 Creating the program	203
3.2.4 Program saved in other names	205
3.2.5 Deleting the program	206
3.2.6 Renaming the program.....	207
3.2.7 Output the program	208
3.2.8 Searching for the program.....	210
3.2.9 Sequence of the programs	211
3.2.10 MDI program	211

3.2.11 DIR/NXT	212
3.3 System interface	213
3.3.1 System parameter setting and rewriting interface	214
3.3.2 Screw pitch compensation setting and rewriting interface.....	221
3.3.3 System information and operation authority levels.....	223
3.3.4 System file management	227
3.3.5 Ladder diagram	229
3.4 Setting interface.....	230
3.4.1 Tool offset setting	230
3.4.2 CNC setting interface	233
3.4.3 Macro variable interface.....	237
3.5 Information interface.....	238
3.5.1 Checking alarm information interface.....	239
3.5.2 Checking the alarm record interface	241
3.5.3 Diagnosis interface.....	241
3.6 Ladder diagram interface	248
3.6.1 Monitor display of ladder diagram.....	249
3.6.2 Checking I/O state.....	251
3.6.3 Checking PLC data and setting	252
3.6.4 Stopping the program running.....	256
3.6.5 Display, creating, editing and managing PLC program directory.....	257
3.6.6 Editing the ladder diagram program.....	262
3.7 Graph interface.....	278
3.7.1 Setting graph parameter	279
3.7.2 Processing graph path.....	280
3.7.3 Simulation graph	281
3.8 Help interfaces.....	281
3.8.1 Operation help	282
3.8.2 Programming help	283
3.8.3 Alarm help	284
3.8.4 Parameter help.....	285
Chapter IV Manual operation.....	287
4.1 Manual reference point return	287
4.2 JOG (manual continuous) feeding	288
4.3 Increment feeding.....	289
4.4 MPG feeding	290
Chapter V Editing and managing the program.....	293
5.1 Creating and editing the program.....	293
5.1.1 Creating the program.....	293
5.1.2 Editing the program.....	295

5.2 Opening and rewriting the program	295
5.2.1 Opening the program	296
5.2.2 Editing and rewriting the program	297
5.2.3 Inserting and deleting the characters	297
5.2.4 Remarking the block	298
5.2.5 Skipping the blocks	298
5.2.6 Generating the block number	299
5.2.7 Shortcut keys	299
5.3 Deleting the program and the block	299
5.3.1 Deleting the program	299
5.3.2 Copying, pasting and deleting the block	300
5.4 Selecting and executing the block	301
5.4.1 Selecting the block	301
5.4.2 Executing the program	302
5.4.3 Executing from any block	303
Chapter VI Tool offset and setting tools	306
6.1 In-position tool setting	306
6.2 Trial tool cutting	307
6.3 Setting the tool offset and the wearing values	309
6.3.1 Direct input method	309
6.3.2 Measuring mode	310
6.3.3 +input mode	311
6.3.4 C input method	312
6.3.5 Clearing the offset value or the wearing value	313
Chapter VII Auto operation	314
7.1 Auto running	314
7.1.1 Selecting the running program	314
7.1.2 Program running	315
7.1.3 Running from any block  or 	316
7.1.4 Stop auto running	316
7.2 MDI running	318
7.2.1 Editing and running the program in MDI mode	318
7.2.2 Running from any block	319
7.2.3 Stop MDI running	319
7.3 DNC running	320
7.4 Auto running control	323
7.4.1 Machine and miscellaneous function lock	323
7.4.2 Dry running	324
7.4.3 Single block running	325

7.4.4 Feed rate override.....	325
7.4.5 Rapid movement override	326
Chapter VIII Setting and display graphs	327
8.1 Setting the graph parameter.....	327
8.2 Path graph display and operation	328
8.3 Simulation graph display and operation.....	330
Chapter IX Communication	332
9.1 Brief introduction of GSK988T communication software GSKComm.....	332
9.2 Creating, channeling and removing the project	333
9.2.1 Creating the project	333
9.2.2 Channeling the project	334
9.2.3 Removing the project	334
9.3 Creating, channeling, removing and editing the file	335
9.3.1 Creating the file.....	335
9.3.2 Editing the file.....	336
9.3.3 Adding the file.....	337
9.3.4 Removing the file.....	338
9.4 Downloading the file (PC→CNC)	338
9.4.1 Downloading the file.....	339
9.4.2 Downloading the single file	339
9.4.3 Uploading the file (CNC→PC)	340
9.5 DNC transmitting	341
9.6 Managing the part program	342
9.7 Preparation before communication	342
Chapter X Processing examples.....	344
10.1 Editing the program.....	345
10.2 Setting tool and running	346

Volume III Connection

Chapter I Installation layout	351
1.1 GSK988T connection	351
1.1.1 GSK988T back cover interface layout	351
1.1.2 Interface Introduction	352
1.1.3 General connection diagram	354
1.2 GSK988T installation	355
1.2.1 GSK988T outer dimension	355
1.2.2 Outer dimension of machine operational panel	356
1.2.3 Installation conditions of the cabinet	356
1.2.4 Method of shielding from interference	356
Chapter II Interface signal definition and connection	359
2.1 Connection with the drive unit	359
2.1.1 Definition of the drive interface	359
2.1.2 Command pulse signals and command direction signals	359
2.1.3 Drive alarm signal nALM	359
2.1.4 Axial enable signal nEN	360
2.1.5 Pulse forbidden signal nSET	360
2.1.6 Zero signal nPC	361
2.1.7 Connection with the drive unit	362
2.2 Connection with the spindle	364
2.2.1 The 5 th axis · spindle interface definition	364
2.2.2 Connection with the servo spindle drive unit	365
2.2.3 Connection with the spindle transducer interface	366
2.3 Connection with the spindle encoder	367
2.3.1 Interface definition of the spindle encoder	367
2.3.2 Signal introduction	367
2.3.3 Connection with the spindle encoder interface	367
2.4 Connection with MPG	368
2.4.1 Definition of MPG interface	368
2.4.2 Introduction of signals	368
2.4.3 Connectino with MPG	368
2.5 Connection with the 2 nd spindle	369
2.5.1: The 2 nd spindle (analog spindle) interface definition	369
2.5.2 Connection with the 2 nd spindle transducer interface	369
2.6 Connection with the external equipment	370
2.6.1 Definition of GSKLINK bus interface	370
2.6.2 Net interface definition	370
2.6.3 USB interface definition	371
2.6.4 RS-232 interface definition	371
2.7 Connection with the machine panel	372
Communication interface definition of the machine panel:	372
2.8 Connection with the power supply	372

2.9 GSK988T common I/O interface definition	373
2.9.1 Definition of input & output addresses	373
2.9.2 Input signal	374
2.9.3 Output signal	375
2.10 System function	377
2.10.1 Travel limit and emergency stop	377
2.10.2 Mechanical zero return	378
Chapter III Introduction of the parameters	382
3.1 Parameter for “setting”	383
3.2 Parameters about the interfaces of input and output	383
3.3 Parameters of axis control/settting unit	384
3.4 Parameter about the coordinate system	389
3.5 Parameter about the stroke detection	393
3.6 Parameter about the feedrate	397
3.7 Parameter about control of acceleration and deceleration	402
3.8 Parameter about servo and backlash compensation	405
3.9 Parameter about DI/DO	410
3.10 Parameter about display and editing	413
3.11 Parameter about programming	416
3.12 Parameter about the screw pitch error compensation	419
3.13 Parameter about the spindle control	422
3.14 Parameter about the tool compensation	428
3.15 Parameter about the fixed cycle	432
3.15.1 Parameter about the drilling fixed cycle	432
3.15.2 Parameter about the thread cutting cycle	432
3.15.3 Parameter about the combined fixed cycle	433
3.16 Parameter about the rigid tapping	435
3.17 Parameter about the polar coordinates interpolation	438
3.18 Parameter about the user macro program	440
3.19 Parameter about the jumping function	442
3.20 Parameter about the figure display	442
3.21 Parameter about display the running time and the quantity of parts	442
3.22 Parameter about MPG feed, MPG cutoff and MPG feed controlling the tool direction	443
3.23 Parameter about PLC axis control	447
3.24 Parameter about the basic function	450
3.25 Parameter about GSKLink communication function	451
Chapter IV MACHINE DEBUGGING	454
4.1 Emergency stop and limit	454
4.2 Setting drive unit	454
4.3 Gear ratio adjustment	456
4.4 Characteristics of adjusting acceleration and deceleration	459
4.5 Mechanical zero point adjustment	465
4.6 Spindle adjustment	470
4.6.1 Spindle encoder	470
4.6.2 Spindle break	470

4.6.3 Spindle speed switch control	470
4.6.4 Spindle speed analog voltage control	470
4.7 Backlash compensation	474
4.8 Thread error compensation	477
Chapter V PLC introduction	482
5.1 PLC specification.....	482
5.2 PLC address	483
5.2.1 X address (machine→PLC).....	483
5.2.2 Y address (PLC→machine).....	483
5.2.3 F address (CNC→PLC)	483
5.2.4 G address (PLC→CNC).....	483
5.2.5 The internal relay address (R)	483
5.2.6 Information display request address (A)	484
5.2.7 The relay address of maintenance type (K).....	484
5.2.8 The counter address (C)	484
5.2.9 Address of the counter preset value (DC)	485
5.2.10 The timer address (T)	485
5.2.11 Preset value address of the timer (DT)	485
5.2.12 Address of the data list (D)	486
5.2.13 The label address (L)	486
5.2.14 The Subprogram number (P)	486
5.3 PLC basic commands	486
5.3.1 LD, LDI, OUT and OUTN	487
5.3.2 AND and ANI	488
5.3.3 OR and ORI	488
5.3.4 ORB	489
5.3.5 ANB.....	489
5.3.6 MPS, MRD and MPP	490
5.4 PLC function command.....	492
5.4.1 Set.....	493
5.4.2 RST (reset).....	493
5.4.3 CMP (Data comparing in binary system)	493
5.4.4 TMRB (Timer)	494
5.4.5 CTRC (Counter in binary system)	495
5.4.6 MOVN (Sending data in binary system)	497
5.4.7 DECB (Decipher in binary system)	497
5.4.8 CODB (Codes conversion in binary system)	498
5.4.9 JMPB (Label number jumping)	499
5.4.10 LBL (Label number)	500
5.4.11 CALL (Calling subprograms)	500
5.4.12 ROTB (Rotation control in binary system)	501
5.4.13 PARI (Odd-even check)	502
5.4.14 ADDB (Data addition in binary system)	503

5.4.15	SUBB (Data subtraction in binary system)	504
5.4.16	DIFU (Setting rising edge check)	505
5.4.17	DIFD (Setting falling edge check)	505
5.4.18	MOVE (Logic multiplication)	506
5.4.19	ALT (Alternate output)	506
5.4.20	WAND (Bytes AND in binary system)	507
5.4.21	WOR (Bytes OR in binary system)	507
5.4.22	WXOR (Bytes AND-OR in binary system)	508
5.4.23	WINV (Bytes negate in binary system)	508
5.4.24	WSHL (Data shift left in binary system)	509
5.4.25	WSHR (Data shift right in binary system)	509
5.4.26	MULB (Data multiplication in binary system)	510
5.4.27	DIVB (Data division in binary system)	511
5.5	Control axis	512
5.5.1	Axial movement state	512
5.5.2	Servo ready signal	512
5.6	Running ready	513
5.6.1	Emergency stop	513
5.6.2	CNC ready signal	513
5.6.3	Alarm signal	513
5.6.4	Mode selection	513
5.6.5	State output	515
5.6.6	Overtravel detection	515
5.7	Manual operation	516
5.7.1	Manual feeding/increment feeding	516
5.7.2	MPG feeding	518
5.8	Mechanical zero-return	519
5.8.1	Mechanical zero-return	519
5.9	Auto running	521
5.9.1	Cycle start/feeding pause	521
5.9.2	Resetting	523
5.9.3	Machine lock	524
5.9.4	Dry run	525
5.9.5	Single block	526
5.9.6	Skipping any blocks	526
5.9.7	Manual absolute value	527
5.10	Feedrate control	527
5.10.1	Rapid movement signal	527
5.10.2	Rapid movement override	528
5.10.3	Feedrate override	528
5.10.4	Canceling override signal	529
5.11	MST function	530
5.11.1	The miscellaneous function (M function)	531
5.11.2	Spindle speed function (S function)	534
5.11.3	Tool function (T function)	534

5.11.4 MST function end	535
5.11.5 Miscellaneous function lock.....	536
5.12 Spindle speed function	537
5.12.1 Gear spindle.....	537
5.12.2 Analog spindle.....	537
5.12.3 Spindle jog function	541
5.12.4 Multi-spindle control	541
5.12.5 Switch between the spindle position/the speed.....	543
5.12.6 Spindle 8-point position function.....	544
5.13 Tool function	545
5.13.1 T command changing tools	545
5.14 Programming command	545
5.14.1 Switch between metric/inch system	545
5.15 Interpolation function	545
5.15.1 Thread cutting	545
5.16 Display/setting/editing	546
5.16.1 Display counting parts.....	546
5.17 PLC axis control function.....	546
5.17.1 Function overview	546
5.17.2 Basic steps.....	547
5.17.3 Control axis selecting signal.....	550
5.17.4 Axis control command signal.....	550
5.17.5 The axis control feedrate signal	554
5.17.6 Axis control data signal.....	557
5.17.7 Axis control command reading signal.....	559
5.17.8 Axis control command reading finish signal.....	559
5.17.9 Resetting signal.....	559
5.17.10 Axis control pause signal.....	560
5.17.11 Block stop signal	561
5.17.12 Block stop invalid signal	561
5.17.13 Miscellaneous function code signal.....	562
5.17.14 Miscellaneous function gating signal.....	563
5.17.15 Miscellaneous function 2 gating signal.....	563
5.17.16 Miscellaneous function 3 gating signal.....	563
5.17.17 Miscellaneous function finish signal	564
5.17.18 Buffer invalid signal	564
5.17.19 Control axis selecting state signal	565
5.17.20 In-position signal	566
5.17.21 Follow zero error check signal.....	566
5.17.22 Alarm signal.....	567
5.17.23 Axis movement signal	567
5.17.24 Miscellaneous function executing function	568
5.17.25 Negative overtravel signal.....	569
5.17.26 Positive overtravel signal.....	569
5.17.27 Feedrate override signal.....	569
5.17.28 Canceling override signal.....	570
5.17.29 Rapid traverse override signal.....	570

5.17.30 Rapid traverse override signal	570
5.17.31 Manual rapid traverse selecting signal	571
5.17.32 Override 0% signal	571
5.17.33 Distribution finish signal	571
5.17.34 Buffer area full of signals	572
5.17.35 Control signal	572

Appendix

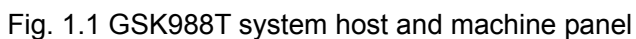
Appendix 1 Alarm List.....	575
A1.1 Program alarm (P/S alarm)	575
Appendix 2 Standard Ladder Function Allocation.....	585
B.1 Standard Machine Panel Key Allocation.....	585
B.2 Standard Ladder X, Y Address Definition.....	586
B.2.1 General machine IO interface.....	586
B.2.2 Servo spindle interface	590
B.2.3 Standard machine operation panel	592
B.3 Standard ladder function	598
B.3.1 Cycle start and feed hold	598
B.3.2 Feed/spindle hold	599
B.3.3 Program protection lock	600
B.3.4 Feedrate override adjustment	600
B.3.5 spindle override tune	600
B.3.6 Spindle CW/CCW control.....	601
B.3.7 Spindle jog.....	603
B.3.8 Spindle eight-point orientation function.....	603
B.3.9 Spindle position/speed switch function.....	604
B.3.10 Spindle speed switching control.....	605
B.3.11 Spindle automatic gear change control.....	606
B.3.12 Cooling control.....	607
B.3.13 Lubricating control	608
B.3.14 Chuck control	609
B.3.15 Tailstock control	611
B.3.16 Pressure low check	613
B.3.17 Each axis overtravel signal.....	613
B.3.18 Tool change control	614
B.3.19 Emergency stop control.....	620
B.4 Parameter explanation of ladder.....	620
B.4.1 K parameter	620
B.4.2 DT parameter.....	622
B.4.3 D parameter	623
B.5 PLC alarm (A address) explanation	624
B.6 G, F signal for Ladder.....	625
B.6.1 G signal.....	625
B.6.2 F signal	627

Volume I

Programming

1.1 GSK988T introduction

USB, ethernet, RS232, GSKLink.



1.1.1 Command system

1. Preparatory functions (G commands)

Table 1-1 G commands

Command	Function	Command	Function
G00	Rapid traverse(positioning)	G54	Select workpiece coordinate system 1
G01	Linear interpolation	G55	Select workpiece coordinate system 2
G02	CW arc interpolation	G56	Select workpiece coordinate system 3
G03	CCW arc interpolation	G57	Select workpiece coordinate system 4
G04	Dwell, exact stop	G58	Select workpiece coordinate system 5
G12.1	Polar coordinate interpolation mode	G59	Select workpiece coordinate system 6
G13.1	Polar coordinate interpolation mode cancel	G65	Non-modal macro program call
G17	XpYp plane select	G66	Marco program modal call
G18	ZpXp plane select	G67	Macro program modal call cancel
G19	YpZp plane select	G70	Finishing cycle
G20	Inch input	G71	Axial roughing cycle
G21	Mm input	G72	Radial roughing cycle
G22	Stored travel check ON	G73	Closed cutting cycle
G23	Stored travel check OFF	G74	Axial grooving cycle
G28	Automatic return to reference point	G75	Radial grooving cycle
G30	Return to 2 nd , 3 rd , 4 th reference point	G76	Multiple thread cutting cycle
G32	Invariable pitch thread cutting	G90	Axial cutting cycle
G34	Variable pitch thread cutting	G92	Thread cutting cycle
G40	Tool nose radius compensation cancel	G94	Radial cutting cycle
G41	Tool nose radius compensation left	G96	Constant surface speed control
G42	Tool nose radius compensation left	G97	Constant surface speed control cancel
G50	Coordinate system setting	G98	Feed per minute
G52	Local coordinate system setting	G99	Feed per rev
G53	Machine coordinate system setting		

2. PLC commands

Table 1-2 PLC commands

Basic command	Function	Basic command	Function
LD	Read normally-open contact	PARI	Parity check
LDI	Read normally-closed contact	ALT	Alternative output
OUT	Output coil	ROTB	Binary rotation control
OUTN	Drive output coil without discontented conditions	DECB	Binary decoding
AND	Normally-open contact in series	CODB	Binary code conversion

ANI	Normally-closed contact in series	JMPB	Program jump
OR	Parallel normally-open contact	LBL	Program jump label
ORI	Parallel normally-closed contact	CALL	Subprogram call
ORB	Parallel series circuit block	SP	Subprogram label
ANB	Parallel circuit block in series	SPE	End of subprogram
MPS	Logical result push	DIFU	Rising edge detection
MRD	Read logical result of stack top	DIFD	Falling edge detection
MPP	Pop logical result of stack top	MOVE	Logical AND
Function command	Function	ADDB	Binary addition
END1	End of a first-level program	SUBB	Binary subtraction
END2	End of a second-level program	MULB	Binary multiplication
SET	Calculates the logical OR and outputs the result	DIVB	Binary division
RST	Calculates the logical AND and outputs the result	WSHL	Binary data left move
CMP	Comparative set	WSHR	Binary data right move
CTRC	Counter	WOR	Binary data OR
TMRB	Timer	WXOR	Binary data OR, AND
MOVN	Data copy	WINV	Binary data reverse

1.1.2 Technical specifications

GSK988T technical specifications are as follows:

1. Feed axis

- Basic controllable axes: 2, controllable axes: 5(including Cs axis);
- Basic link controllable axes: 2, controllable axes: 3;
- Least command increment: 0.001mm or 0.0001mm, command range: $\pm 99999999 \times$ command increment;
- Position pulse+direction and AB-phase pulse output select;
- Position command gear ratio, position feedback gear ratio;
- Axis enabling output, axis servo alarm, servo ready (motor excitation) input;
- Matched with GSKLink bus servo unit to realize servo parameter read/write and servo unit monitor in real time.

2. Spindle function

- Spindle controllable axes: 2(one analog spindle, one Cs axis);
- Analog spindle interface: analog voltage output 0~10V, voltage resolution be less than 5mV;
- Cs spindle interface: analog voltage output, pulse+ direction and AB-phase pulse output select;
- Encoder interface: increment encoder A, B, Z difference signal input, encoder lines 100 p/r ~9999p/r;
- Spindle encoder transmission ratio: 1: 1, 1: 2, 1: 4, 1: 8;

- Speed command range: specified by S command or PLC signal, command range: 0~20000r/min, spindle override 50~120%, 8-level tuning in real-time;
- Constant surface speed control;
- Spindle orientation output function, Cs spindle providing contour control function;
- Spindle enabling output, zero-speed clamp, orientation output, spindle servo alarm, servo read(motor excitation), speed arrival, orientation completion input(define to I/O).

3. Interpolation function

- Interpolation mode: positioning, linear interpolation, arc interpolation, thread interpolation, polar coordinate interpolation;
- Thread interpolation: any one, two axes move with the spindle, single/multi metric/inch straight thread;

4. Feed function

- Max. speed: standard 60000mm/min (0.1 μ m:24000 mm/min);
- Rapid override: F0, 25%, 50%, 100%;
- Feedrate override: 0~150%, 16-grade tune in real time;
- Step/MPG step length: specified by parameter according to least input increment.

5. Acceleration/deceleration function

- Acceleration/deceleration characteristics: exponential and linear;
- Positioning: linear acceleration/deceleration;
- Feed: linear, exponential acceleration/deceleration;
- Thread: linear, exponential acceleration/deceleration;
- Initial speed, termination speed and acceleration/deceleration time are confirmed by parameters.

6. Compensation function

- Backlash: compensate all feed axes, cutting , and rapid positioning;
- Memory pitch error compensation: each feed axis with 1024 compensation points and each point $\pm 700 \times$ override;
- Tool nose radius compensation: C compensation;
- Tool length compensation;
- Tool wear compensation;
- Automatic tool offset function.

7. Tool function

- Tool quantities: defined by parameter;
- 99 tool length compensation, tool wear compensation, tool nose radius compensation data.
- 8, 16, 32 groups of tool life management;
- Toolsetting mode: fixed-point, trial-cut toolsetting;
- Tool compensation execution mode; modifying coordinates or moving tool.

8. Macro program function

- Support statement macro program programming;
- G65/66/67 support macro program call with variable;
- Local variable, common variable and system variable;

- Support GOTO, IF, WHILE and other macro program statements.

9. Program edit

- User program capacity: 25M;
- Program quantities: 10000(including subprograms, macro programs);
- Program format: ISO codes are compatible with FANUC, word spaces can have no blank space, absolute value, incremental value and decimal programming;
- Subprogram: 12-level subprogram nesting;
- Program edited at back stage;
- Edit, program/block/word search, modification, deletion, cancel/recover, block copy/deletion, program creation, copy and rename in full screen;
- System provides phrasing check function.

10. Working mode

- Edit: program edit, program input/output;
- MDI: parameter/tool compensation/variable edit, backup, input, output, many blocks MDI and execution;
- Auto: run, pause, run from any blocks, single block, skip, machine locked, auxiliary function locked; DNC run;
- Manual: rapid/feed;
- Step/MPG: step length can be set;
- Machine zero return.

11. PLC function

- PLC command: 13 basic commands and 30 functional commands;
- ded PLC, ladder edit on-line, monitor;
- Ladder serial port, USB, LAN upload and download;
- Max. steps of ladder: 5000;
- 1-level program refresh period: 8ms;
- Execution period of 2-level ladder with 2000 steps is less than 16ms.

12. Man-machine interface

- Display part: 8.4"640×480 color LCD, graphic operation interface;
- Two-dimensional tool path, solid display, 3-D tool path display;
- 10 screen soft function keys; 1-level display interface including: position, program, system, setting, information, graphic and help;
- Servo state monitor, parameter on-line configuration;
- Real-time clock;
- Parameters are classified according to their functions;
- On-line help.

13. Communication function and data interface

- RS232 interface: main system panel preposition, CNC software upgrade, PLC program, parameter, tool compensation, variable and files directional transmission;
- USB interface: one Host interface, main system panel preposition, CNC software upgrade, PLC program, parameter, tool compensation, variable and file bidirectional transmission;
- LAN interface: main system panel preposition, CNC software upgrade, PLC program,

parameter, tool compensation, variable and file bidirectional transmission, IP address configuration, remote monitor, remote control function technical storage;

14. Safety function

- Emergency stop;
- Hardware travel limit, many kinds of storage travel check;
- Data backup and recovery;
- Multi-level operation authority management.

1.2 CNC system of machine tools and CNC machine tools

CNC machine tool is an electro-mechanical integrated product, composed of Numerical Control Systems of Machine Tools, machines, electric control components, hydraulic components, pneumatic components, lubricating, coolant and other subsystems (components), and CNC systems of machine tools are control cores of CNC machine tools. CNC systems of machine tools are made up of computerized numerical control(CNC), servo (stepper) motor drive devices, servo (or stepper) motor etc.

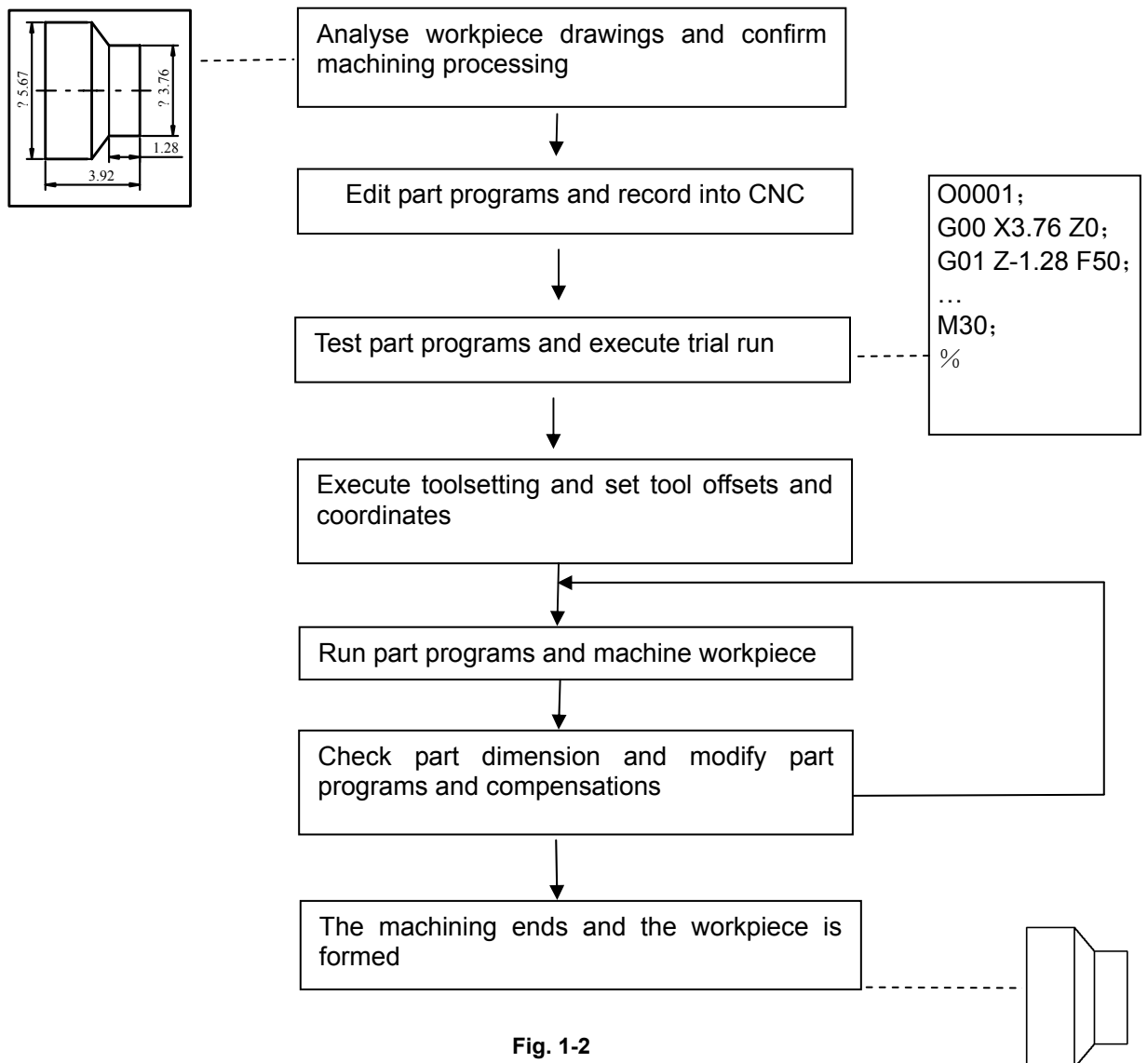
Operational principles of CNC machine tools: according to requirements of machining technology, edit user programs and input them to CNC, then CNC outputs motion control commands to the servo (stepper) motor drive devices, and last the servo (or stepper) motor completes the cutting feed of machine tool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

GSK988T Turning Machine CNC system has simultaneously motion control and logic control function to control two axes of CNC machine tool to move, and has ded PLC function. Edit PLC programs (ladder diagram) according to requirements of input and output control of machine tool and then download them to GSK988T Turning Machine CNC system, which realizes the required electric control requirements of machine tool, is convenient to electric design of machine tool and reduces cost of CNC machine tool.

Softwares used for controlling GSK988T Turning Machine CNC system are divided into system software (NC for short) and PLC software (PLC for short). NC system is used for controlling display, communication, edit, decoding, interpolation and acceleration/deceleration, and PLC system for controlling explanations, executions, inputs and outputs of ladder diagrams.

Standard PLC programs are loaded (except for the special order) when GSK980TDa Turning Machine CNC System is delivered, concerned PLC control functions in following functions and operations are described according to control logics of standard PLC programs, marking with "Standard PLC functions" in GSK980TDa Turning CNC System User Manual. Refer to Operation Manual of machine manufacturer about functions and operations of PLC control because the machine manufacturer may modify or edit PLC programs again.

Programming is a course of workpiece contours, machining technologies, technology parameters and tool parameters being edit into part programs according to special CNC programming G codes. CNC machining is a course of CNC controlling a machine tool to complete machining of workpiece according requirements of part programs. Technical flow of CNC machining is as following Fig. 1-1.



1.3 Programming fundamentals

1.3.1 Coordinates definition

Sketch map of CNC turning machine is as follows:

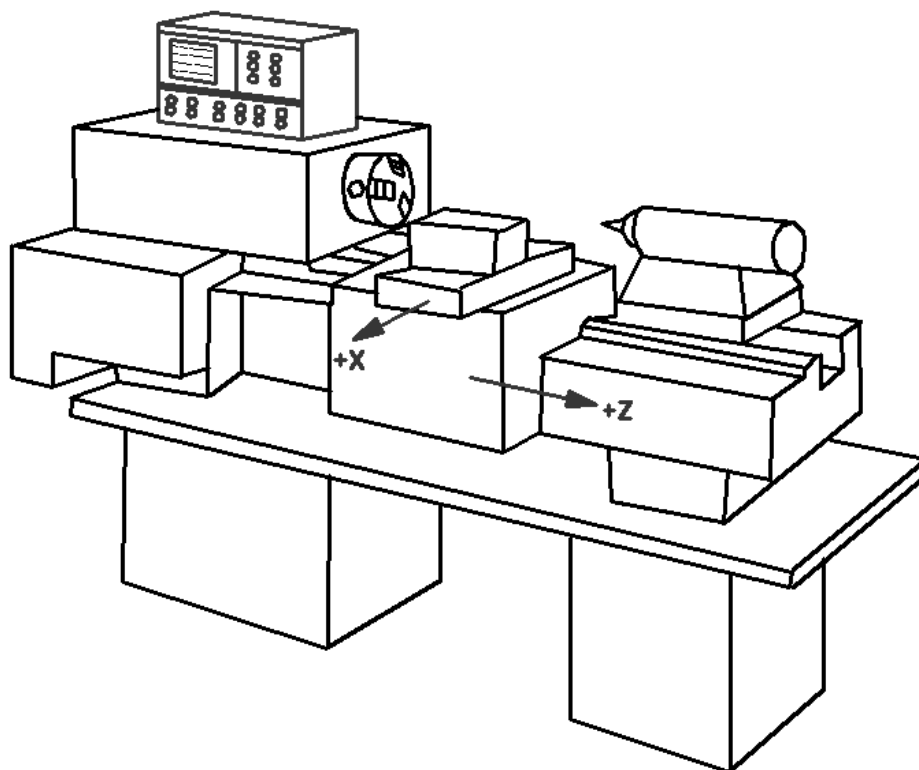


Fig. 1-3

Basic controllable axes of GSK988T Turning CNC System are two, controllable axes are five(including Cs axis); basic link controllable axes are three. The parameter 8130 and 1010 can separately set the total of the system controllable axis and CNC controllable axis. The relationship of each axis is as follows:

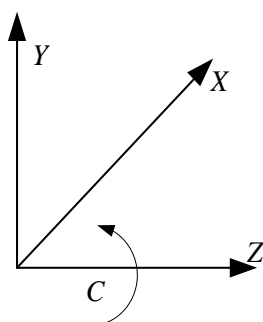


Fig. 1-4

GSK988T uses a rectangular coordinate system composed of X, Z axis. X axis is perpendicular with axes of spindle and Z axis is parallel with axes of spindle; negative directions of them approach to the workpiece and positive ones are away from it.

Parameter NO.1020 can set and modify program names of each axis and their responding relationship is as follows:

Table 1-3 (a)

Axis name	Setting value	Axis name	Setting value
X	88	Z	90
Y	89	A	65
B	66	C	67

There is a front tool post and a rear tool post of NC turning machine according to their relative position between the tool post and the spindle, Fig. 1-5 is a coordinate system of the front tool post and Fig. 1-6 is a rear toolpost one. It shows exactly the opposite of X axes, but the same of Z axes from figures. In the manual, it will introduce programming application with the front tool post coordinate system in the following figures and examples.

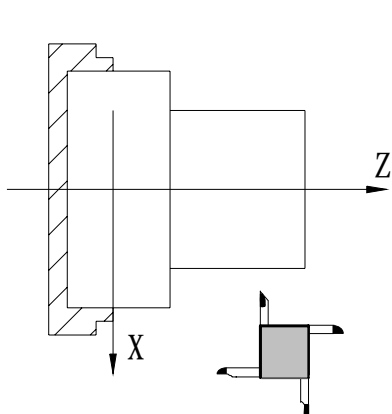


Fig.1-5 Front tool post coordinate system

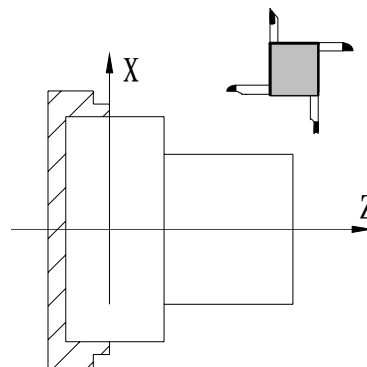


Fig.1-6 Rear tool post coordinate system

1.3.2 Increment system

Increment system includes least input increment (input) and least command increment (output). Least input increment is the least unit of programming movement distance. Least command increment is the least unit of tool movement on the machine tool. Their unit: mm, inch or degree. Increment systems are separately IS-B and IS-C. Bit 1 of NO. 1004 decides to select IS-B or IS-C. Bit 1 (ISC) setting of No.1001 is applied to all axes. For example: increment system of all axes is set to IS-C when the parameter selects IS-C.

Table 1-3 (b) increment system IS-B

		Least input increment	Least input increment
Metric machine	mm input	0.001mm (diameter) 0.001mm (radius) 0.001deg	0.0005mm 0.001mm 0.001deg
	Inch input	0.0001inch (diameter) 0.0001inch (radius) 0.001deg	0.0005inch 0.001inch 0.001deg
Inch machine	mm input	0.001mm (diameter) 0.001mm (radius) 0.001deg	0.00005mm 0.0001mm 0.001deg
	Inch input	0.0001inch (diameter) 0.0001inch (radius) 0.001deg	0.00005inch 0.0001inch 0.001deg

Table 1-3 (c) increment system IS-C

		Least input increment	Least input increment
Metric machine	mm input	0.0001mm (diameter) 0.0001mm (radius) 0.0001deg	0.00005mm 0.0001mm 0.0001deg
	Inch input	0.00001inch (diameter) 0.00001inch (radius) 0.0001deg	0.00005inch 0.0001inch 0.0001deg
Inch machine	mm input	0.0001mm (diameter) 0.0001mm (radius) 0.0001deg	0.000005mm 0.00001mm 0.0001deg
	Inch input	0.00001inch (diameter) 0.00001inch (radius) 0.0001deg	0.000005inch 0.00001inch 0.0001deg

1.3.3 Max. travel

Max. travel=least command increment * (±) 99999999

Table 1-3(d) max. travel IS-C

Increment system		Max. travel
IS-B	Metric machine system	±99999.999mm ±99999.999deg
	Inch machine system	±9999.9999inch ±9999.9999deg
IS-C	Metric machine system	±9999.9999mm ±9999.9999deg
	Inch machine system	±999.99999inch ±9999.9999deg

Note 1: The unit is diameter value in diameter programming, is radius value in radius programming in the above table.

Note 2: The input command cannot exceed max. travel command.

Note 3: The actual travel decides the machine tool.

1.3.4 Machine coordinate system

Machine tool coordinate system is a benchmark one used for CNC counting coordinates and a fixed one on the machine tool. **Machine tool zero** is a fixed point which position is specified by zero switch or zero return switch on the machine tool. Usually, the zero return switch is installed on max. stroke in axis positive direction. After the system is turned on, the reference point return is executed to set machine coordinate system. The machine coordinate system is not keeping until the system is turned off.

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

1.3.5 Workpiece coordinate system

The workpiece coordinate system is a rectangular coordinate system based on the part drawing, also called floating coordinate system. The workpiece coordinate system is set by the system in advance, can be changed by moving its coordinate origin point. The established workpiece is valid till it is replaced by a new one. The system has preset 6 workpiece coordinate systems (G54-G59).

1.3.6 Local coordinate system

When the system compiling programs in the workpiece coordinate system, sub-coordinate system of workpiece coordinate system can be set for easily programming, called local coordinate system as follows:

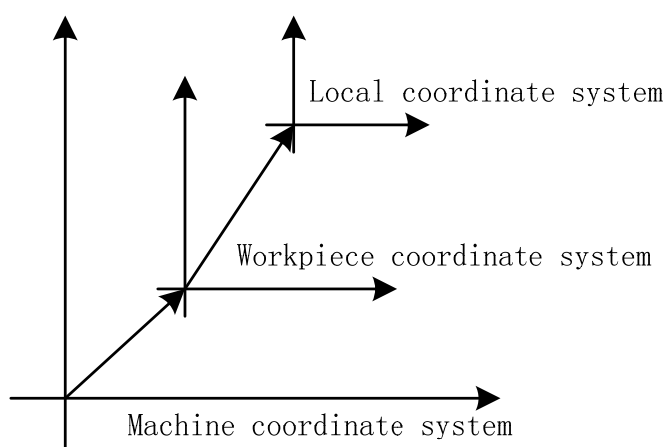


Fig. 1-7 local coordinate system

1.3.7 Reference point

Reference point is a fixed point on the machine tool. The tool can move to the position by executing the reference point return function. Generally, the reference point is used to tool change and setting coordinate system. GSK988T Turning CNC System can set 4 reference points by parameters as follows:

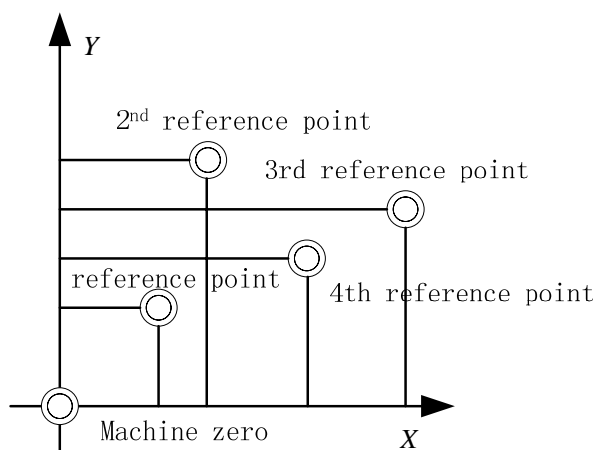


Fig. 1-8 reference point

1.3.8 Interpolation function

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

GSK988T has linear, arc and thread interpolation function.

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

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

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

Note 1: Xp, Yp, Zp are separately X or its parallel axis, Y or its parallel axis, Z or its parallel axis. The followings are the same as those.

Note 2: IP expresses the combination of X_Y_Z_(used in programming).

Example:

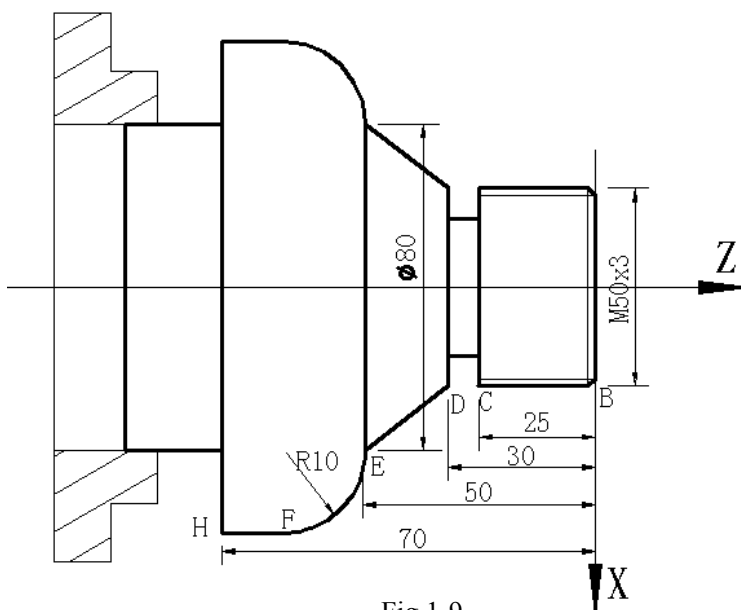


Fig.1-9

```

...
G32 W-27 F3;      ( B→C; thread interpolation )
G1 X50 Z-30 F100;
G1 X80 Z-50;      ( D→E; linear interpolation )
G3 X100 W-10 R10; ( E→F; arc interpolation )
...
M30;
    
```

1.4 Structure of an NC program

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

Program example:

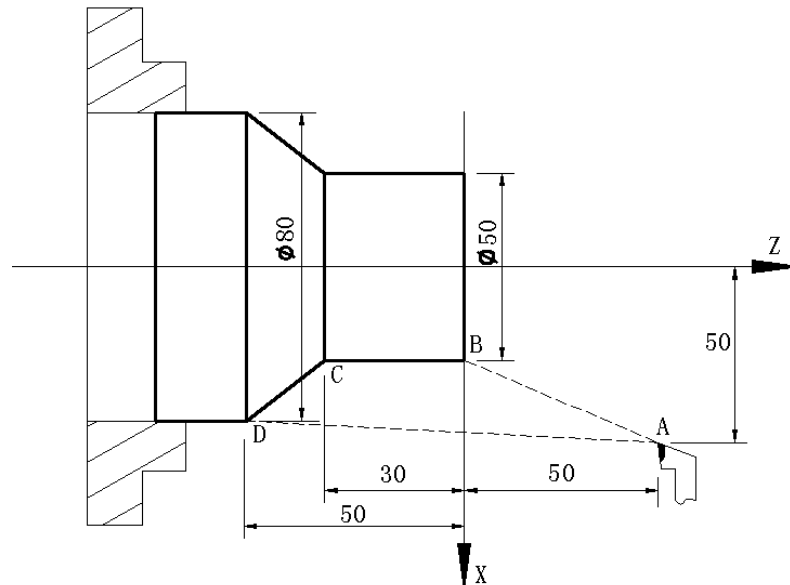


Fig. 1-10

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

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

A program consists of a sequence of blocks, beginning with "OXXXX"(program name)and ending with "%"; a block begins with block number (omitted) and ends with ";", or "*". See the general structure of program as Fig. 1-11:

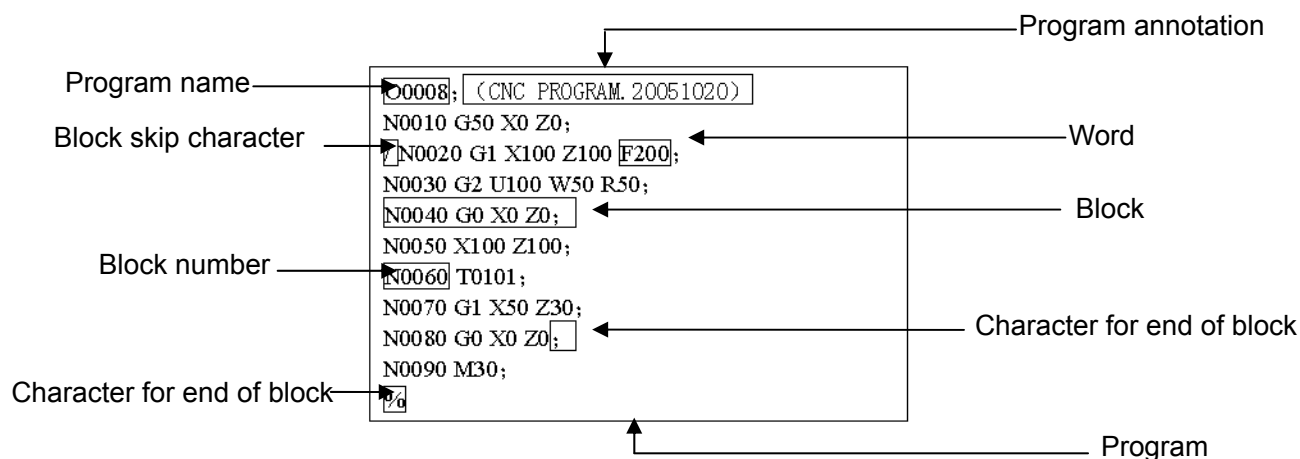
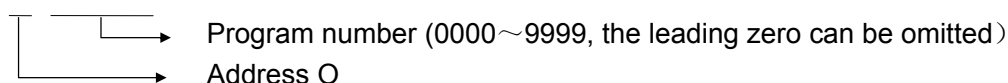


Fig. 1-11 Structure of a program

1.4.1 Program name

Format: O △△△△



△△△△ is number of a program name, its range is 4-digit integer 0000~9999, the system alarms when the negative program name is input. The system ignores NC commands when program are edited and other NC commands are edited in the first line.

1.4.2 Block format

1. Format: / N△△△△ countless words;

/: skip character. A block can have or not it, generally, it is placed in the initial position of a program; user can press "SKIP" on the operation panel to execute the operation when the skip function is valid, otherwise, the "SKIP" key on the operation panel is valid, i.e. the skip character in the block is invalid;

N△△△△△: block number. A block can have or not it; number △△△△△ following N is 5-digit positive integer 00001~99999, and the system alarms when the input number is decimal.

Countless words: one block can input countless words, and one block can have one or more words or have no words.

, : "EOB" is a end character when one block is completed, "," is displayed in LCD, there must be have one end character for one block;

2. Format requirements

- (1) In one block, there can be no blank space between block number and word, and can be countless blank space(the total characters of one block is within 255);
- (2) In one block, there can be not or be countless space between skip character and block number or words;
- (3) In one block, there can be not or be countless space between end character of block and its front word or blocks;

Each block can be up to 255 characters, including skip character, block number, command, space, end character of block ",";

(4) The system automatically ignores the content with small bracket “ (”, “) ”.

3. Parameters related block number:

(1) whether the system automatically creates block number or not:

User can set whether the system automatically creates block number or not in editing program by setting Bit 5(SEQ) of NO.0000;

(2) User can set the interval value in automatically creating block number by setting NO.3216.

Note: Sprit(/) explanations:

1. When the sprit (/) is used to skip character, it is generally placed the beginning of block, otherwise, and the messages from the sprit to EOB code are ignored. For example: U10.G00/04; when the skip function is started, the system executes U10. G00;(G00 U10.), when it stops, the system executes U10. G0004;(G04 U10.);

2. For cycle command buffer, when a block reads from memory to buffer memory, whether the skip function is valid or not has been executed. After a block reads into buffer memory, i.e. the system changes skip switch state, but does not influence the block which has read into the buffer memory;

3. Sprit (/) (closed in bracket[]) and sprit(/) right to value statement “=” in <Expression> are taken as division operation character instead of skip character.

1.4.3 Word

1. Format: address + number. There must not be space between address and number.

Presently, the system permissively input addresses: G, M, S, T, F, X, Y, Z, U, V, W, P, Q, I, J, K, R, L, A, B, C, H, N, O, and will add other;

Command number range following address is referred to the following table.

Table 1-4 word table

Address	Function	mm input	inch input	Related G codes
O	Program name	0~9999	0~9999	
N	Line label	1~99999	1~99999	
G	Preparatory function	See G code	See G code	
M	Miscellaneous function	0~9999	0~9999	
S	Spindle speed	(G96) 0~20000 m/min	(G96) 0~2000 feet/min	
		(G97) 0~20000 r/min	(G97) 0~20000 r/min	
T	Tool offset	0000~9999	0000~9999	G98
F	Feedrate per minute	(ISB system) 1 ~60000 mm/min	(ISB system) 0.01~2400 inch/min	
		(ISC system) 1 ~24000 mm/min	(ISC system) 0.01~960 inch/min	
	Feedrate per rev	(ISB system) 0.01~500mm/r	(ISB system) 0.01~9.99inch/r	
		(ISC system) 0.01~500mm/r	(ISC system) 0.01~9.99 inch/r	
	Pitch	0.01~500 mm	0.01~9.99inch	Relative commands for thread

				machining
X	X absolute coordinate value(linear axis), delay time (*1)	(ISB system) -99999.999~99999.999mm	(ISB system) -9999.9999~9999.9999inch	Relative command of axis, G04
		(ISC system) -9999.9999~9999.9999mm	-999.99999~999.99999inch	
Y	Y absolute coordinate value(linear axis) (*1)	(ISB system) -99999.999~99999.999 mm	(ISB system) -9999.9999~9999.9999 inch	Relative command of axis
		(ISC system) -9999.9999~9999.9999 mm	(ISC system) -999.99999~999.99999 inch	
Z	Z absolute coordinate value (linear axis) (*1)	(ISB system) -99999.999~99999.999 mm	(ISB system) -9999.9999~9999.9999 inch	Relative command of axis
		(ISC system) -9999.9999~9999.9999 mm	(ISC system) -999.99999~999.99999 inch	
A	A absolute coordinate value(linear axis) (*1)	(ISB system) -99999.999~99999.999 mm	(ISB system) -9999.9999~9999.9999 inch	Relative command of axis
		(ISC system) -9999.9999~9999.9999 mm	(ISC system) -999.99999~999.99999 inch	
B	B absolute coordinate value(linear axis) (*1)	(ISB system) -99999.999~99999.999 mm	(ISB system) -9999.9999~9999.9999 inch	Relative command of axis
		(ISC system) -9999.9999~9999.9999 mm	(ISC system) -999.99999~999.99999 inch	
C	C absolute coordinate value (rotary axis) (*1)	(ISB system) -99999.999~99999.999 deg	(ISB system) -99999.999~99999.999 deg	Relative command of axis
		(ISC system) -9999.9999~9999.9999 deg	(ISC system) -9999.9999~9999.9999 deg	
U	X relative coordinate value, finishing allowance in G71, G72, G73, X tool retraction distance and specified delay time(*1) in G73, (*1)	(ISB system) -99999.999~99999.999 mm	(ISB system) -9999.9999~9999.9999 inch	Relative command of axis,G71,G72,G73,G04
		(ISC system) -9999.9999~9999.9999 mm	(ISC system) -999.99999~999.99999 inch	
	Cut depth in G71(modify parameter manual) (*2)	(ISB system) 0.001~99999.999 mm	(ISB system) 0.0001~9999.9999 inch	G71
		(ISC system) 0.0001~9999.9999 mm	(ISC system) 0.00001~999.99999 inch	
V	Y relative coordinate value(linear axis) (*1)	(ISB system) -99999.999~99999.999 mm	(ISB system) -9999.9999~9999.9999 inch	Relative command of axis
		(ISC system) -9999.9999~9999.9999 mm	(ISC system) -999.99999~999.99999 inch	
W	Z relative coordinate value, Z finishing allowance in G71, G72, G73, Z tool retraction distance (*1) in G73 (*1)	(ISB system) -99999.999~99999.999 mm	(ISB system) -9999.9999~9999.9999 inch	Relative command of axis, G71, G72, G73,
		(ISC system) -9999.9999~9999.9999 mm	(ISC system) -999.99999~999.99999 inch	
	Cut depth (*2) in G72 (*2)	(ISB system) 0.001~99999.999 mm	(ISB system) 0.0001~9999.9999 inch	G72
		(ISC system)		

		0.0001~9999.9999 mm	0.00001~999.99999 inch	
R	Arc radius (*1)	(ISB system) -99999.999~99999.999 mm	(ISB system) -9999.9999~9999.9999 inch	G02,G03
		(ISC system) -9999.9999~9999.9999 mm	(ISC system) -999.99999~999.99999 inch	
	Taper and thread taper (*1) in G90, G92, G94, G76 (*1)	(ISB system) -99999.999~99999.999 mm	(ISB system) -9999.9999~9999.9999 inch	G90,G92,G94,G76
		(ISC system) -9999.9999~9999.9999 mm	(ISC system) -999.99999~999.99999 inch	
	Tool retraction (*2) in G71,G72 (*2)	(ISB system) 0~99999.999 mm	(ISB system) 0~9999.9999 inch	G71,G72
		(ISC system) 0~9999.9999 mm	(ISC system) 0~999.99999 inch	
	Roughing times in G73	1~999 (times)	1~999 (times)	G73
	Thread increment in variable pitch cutting	0.01~500.000 mm -0.01~-500.000 mm	0.01~9.99inch -0.01~-9.99inch	G34
	Tool retract movement after cutting in G74, G75 and tool retraction after cutting to end point (*2)	(ISB system) 0~99999.999 mm	(ISB system) 0~9999.9999 inch	G74,G75
		(ISC system) 0~9999.9999 mm	(ISC system) 0~999.99999 inch	
P	Finishing amount (*2) in G76	(ISB system) 0.001~99999.999 mm	(ISB system) 0.0001~9999.9999 inch	G76
		(ISC system) 0.0001~9999.9999 mm	(ISC system) 0.00001~999.99999 inch	
	Dwell time	0~99999999ms	0~99999999 ms	G04
	G30 returning to No.n reference point	2,3,4	2,3,4	G30 (default to 2)
	Commands for macro program number, subprogram and subprogram call times	1~9999	1~9999	G65,G66,M98 (default times is 1)
	Line number assignment in G70, G71, G72,G73	0~99999	0~99999	G70,G71,G72,G73
	X cycle movement (*3) in G74, G75	0 ~ 99999999 × least command unit	0~99999999×least command unit	G74,G75
	Thread cutting parameter in G76	Including 3 parameters: Thread finishing times: 1~99 Thread run-out length: 00~99 (*0.1 pitch) Angle between two teeth : 0°~99°	Including 3 parameters: Thread finishing times: 1~99 Thread run-out length: 00~99 (*0.1 pitch) Angle between two teeth : 0°~99°	G76
Q	Thread tooth height (*3) in G76	1 ~ 99999999 × least command unit	1~99999999×least command unit	G76
	Line number assignment in G70, G71, G72, G73	0~99999	0~99999	G70,G71,G72,G73
	Tool infeed amount(*3) in Z brokenly infeed in	0 ~ 99999999 × least command unit	0~99999999×least command unit	G74,G75

	G74,G75				
	Min. cutting amount (*3) in G76 thread roughing		0 ~ 99999999 × least command unit	0~99999999×least command unit	G76
	1 st thread cutting depth (*3) in G76 thread roughing		1 ~ 99999999 × least command unit	1~99999999×least command unit	G76
	Initial angle (*3)of 1 st circle in thread cutting (*3)		0 ~ 99999999 × least command unit (default to 0)	0~99999999×least command unit (default to 0)	G32,G34,G92
L	Macro program call times assignment		1~9999 (default to 1)	1~9999 (default to 1)	G65,G66
	Head quality of multi-thread		1~99 (default to 1)	1~99 (default to 1)	G92
I	Relative starting point of arc center is in X vector (*1)		(ISB system) -99999.999~99999.999mm	(ISB system) -9999.9999~9999.9999inch	G02,G03
			(ISC system) -9999.9999~9999.9999 mm	(ISC system) -999.99999~999.99999 inch	
J	Relative starting point of arc center is in Y vector (*1)		(ISB system) -99999.999~99999.999 mm	(ISB system) -9999.9999~9999.9999 inch	G02,G03
			(ISC system) -9999.9999~9999.9999 mm	(ISC system) -999.99999~999.99999 inch	
	Movement in short axis when thread run-out is executed (*1)		(ISB system) -99999.999~99999.999 mm	(ISB system) -9999.9999~9999.9999 inch	G32,G34,G92
			(ISC system) -9999.9999~9999.9999 mm	(ISC system) -999.99999~999.99999 inch	
K	Relative starting point of arc center is in Z vector (*1)		(ISB system) -99999.999~99999.999 mm	(ISB system) -9999.9999~9999.9999 inch	G02,G03
			(ISC system) -9999.9999~9999.9999 mm	(ISC system) -999.99999~999.99999 inch	
	Length in long axis when thread run-out is executed (*2)		(ISB system) 0~99999.999 mm	(ISB system) 0~9999.9999 inch	G32,G34,G92
			(ISC system) 0~9999.9999 mm	(ISC system) 0~999.99999 inch	
IF		Conditional judgement			
THEN	TH	Execution after IF conditional judgment is completed successfully			
GOTO	GO	Non-conditional skip			
WHILE	WH	Cycle judgment			
DO		Start to execute cycle			
END	EN	Return to WHILE			
EQ	==	Equal to	Judgement logic is used to brackets following IF, WHILE		
NE	<>	Not equal to			
GT	>	Greater than			
GE	>=	Greater than or equal to			
LT	<	Less than			
LE	<=	Less than or equal to			
SIN	SI	Sine	Functional function is used to count expression value		
ASIN	AS	Anti-sine			

COS	CO	Cosine
ACOS	AC	Anti-cosine
TAN	TA	Tangent
ATAN	AT	Anti-tangent
SQRT	SQ	Square root
ABS	AB	Absolute value
ROUN	RO	Rounding-off
FIX	FI	Down integer
FUP	FU	Up integer
LN		Nature logarithm
EXP	EX	Exponential function
OR		OR
XOR	XO	OR AND
AND	AN	AND
BIN	BI	Converse from BCD to BIN
BCD	BC	Converse from BIN to BCD
123456789	With to compose the value of word, the leading 0 can be omitted	
0	Word is 0 and is different with empty value	
+	Number count and number expression	
-		
*		
/	Skip command, selectively skip the commands following the character	
.	Floating point number with number	
=	Variable assignment	
[Prior operation of expression and conditional judgement prompt	
]		
#	Variable	
;	End of program in the block, following annotation	
(Annotation start in the block. Example: (X20.)W-10.; not execute X20.	
)	Annotation end in the block	
%	End of program	

Note 1: the 2-digit following the decimal point of F value is value, and the more following the two-digit is ignored.

Note 2: the expression can follow the word, the value counted by the expression is taken as the value of the word, and the expression should have[], and there must not be the space between the word and the expression. For example X[#1-#110] Z[#1+SIN[#120]].

*1): When the address values in the above table, X, Y, Z, C, A, B, C, U, V, W, H, I, J, K, R are taken as word address, their value ranges are controlled by the following parameters:

(1) No.0000#2 INI

INI input unit

0: metric

1: inch

(2) No.1006#0 ROTx

ROTx set linear axis or rotary axis

0: linear axis

1: rotary axis

(3) No.0004#1 ISC

ISC set least input unit and least command increment

ISC	Least setting unit	For short
0	0.001mm, 0.001deg or 0.0001inch	IS-B
1	0.0001mm, 0.0001deg or 0.00001inch	IS-C

Table 1-4-1 least command unit and value range

Address	Parameter setting			Least command unit	Range
X,Y,Z,C,A,B,C,U,V,W,H	ROTx=0 Rotary axis	Rotary axis is not related to INI	ISC=0 ISB	0.001deg	-99999.999 ~ 99999.999 deg
			ISC=1 ISC	0.0001deg	-9999.9999 ~ 9999.9999 deg
X,Y,Z,C,A,B,C,U,V,W,H,I,J,K,R	ROTx=1 Linear axis	INI=0 Metric	ISC=0 ISB	0.001mm	-99999.999 ~ 99999.999 mm
			ISC=1 ISC	0.0001mm	-9999.9999 ~ 9999.9999 mm
		INI=1 Inch	ISC=0 ISB	0.0001inch	-9999.9999 ~ 9999.9999 inch
			ISC=1 ISC	0.00001inch	-999.99999 ~ 999.99999 inch

When these word addresses follow data, data precision is least command unit, and excessive data is ignored. When a word address follows variable number or has [] expression, the word value has decimal data, and its precision is the least command unit, but its excessive data rounds.

(4) No.3401#0 DPI

DPI can use decimal address. When the decimal is omitted, its setting is as follows:

0: least setting unit

1: unit: mm, inch, sec

When parameter DPI is set to 1, word range is referred to Table 1-4-1;

When DPI is set to 0, and word omits its decimal, its value range is -99999999~99999999, data unit is the least command unit in Table 1-4-1.

*2) : Command value calculation method specified by U, W, R, K is the same that of *1) , they

meet the value range described in *1) and limit value range according to preparatory function.

*3): Position specified value commanded by P, Q is 0~99999999, data unit is the least command unit in Table 1-4-1. value range is limit by specific preparatory function.

2. Word value and state will change when the system runs, the following table separately explains each word omit and state when the system is ON, resets.

Table 1-5 word state

Character	Function	Initial value in power-on	Default value	Keep in the next block?	Value after pressing reset key	Related explanation
O	Program name	Value reserved by last power-on	Current value	Yes	Yes	None
G	Preparatory function	Initial mode in each group	Modal value	No	parameter (CLR) NO.3402#6	None
M	Miscellaneous function M00, M01, M02, M30, M98, M99		Current value	No (function reserved)	Current value	Specified by PLC, set by parameter
S	Analog spindle speed	0	Current value	Yes	Current value, output is invalid	
T	Tool offset	Value reserved by last power-on	Current value	Yes	Current value	
F	Feedrate per minute	Parameter value	Current value	Yes	parameter (CLR) NO.3402#6	
	Feedrate per rev	Empty	Current value	Yes	Current value	
	Pitch	Empty	Current value	Yes	Current value	
X	Delay time	Empty	0	No	0	
	X absolute coordinate value	0	Current value	Yes	Current value	
Y	Y absolute coordinate value	0	Current value	Yes	Current value	
Z	Z absolute coordinate value	0	Current value	Yes	Current value	
C	C absolute coordinate value	0	Current value	Yes	Current value	
U	Delay time	Empty	0	No	Empty	
	X relative coordinate value	0	0	No	Current value	
	X allowance in finishing	Empty	0	No	Empty	
	Cutting depth in G71	Parameter value	Parameter value	Yes	Parameter value	
V	Y relative coordinate value	0	0	No	Current value	

W	Z relative coordinate value	0	0	No	Current value	
	Z allowance in finishing	空	0	No	Empty	
	Cutting depth in G72	Parameter value	Parameter value	Yes	Parameter value	
H	C increment value	0	0	No	Current value	G00
		0	0	No	Current value	Polar coordinate interpolation
R	Arc radius	0	0	No	Current value	
	Taper G90, G92, G94 and thread taper	0	0	Yes	Current value	
	Tool retraction in G71, G72	Parameter value	Parameter value	Yes	Parameter value	
	Roughing times in G73	Parameter value	Parameter value	Yes	Parameter value	
	Clearance in G74,G75	Parameter value	Parameter value	Yes	Parameter value	
	Clearance to end point in G74,G75	0	0	No	Empty	
	Finishing cutting amount in G76	Parameter value	Parameter value	Yes	Parameter value	
P	Dwell time	Empty	0	No	Empty	
	G30 returning to No. n reference point	Empty	2	No	Empty	
	Macro program number, subprogram, subprogram call times	Empty	Alarm	No	Empty	
	Line assignment in G70, G71, G72, G73	Empty	Alarm	No	Empty	
	X cycle movement in G74,G75	Empty	0	No	Empty	
	Thread cutting in G76	Parameter value	Parameter value	Yes	Parameter value	
	Thread tooth height in G76	0	Alarm	No	Empty	
Q	Line assignment in G70, G71, G72, G73	Empty	Alarm	No	Empty	
	Z broken tool infeed amount in G74, G75	Empty	0	No	Empty	
	Least cutting amount in G76 roughing	Parameter value	Parameter value	Yes	Parameter value	

	1 st thread cutting depth in G76 thread roughing	Empty	Alarm	No	Empty	
	1 st circle start angle in thread cutting	Empty	0	No	0	
	Check offset in spindle fluctuation check	Empty	0	No (the parameter cannot be modified	0	
L	Macro program call times assignment	1	1	No	Empty	
I	X vector of circle center corresponding to starting point	0	0	No	Current value	
	X calculation direction in cancelling radius compensation	Empty	Empty	No	Empty	
J	Y vector of circle center corresponding to starting point	0	0	No	Current value	
	Y calculation direction in cancelling radius compensation	Empty	Empty	No	Empty	
K	Z vector of circle center corresponding to starting point	0	0	No	Current value	
	Pitch increment in variable pitch thread cutting	Empty	0	Yes	Current value	
	X travel lower limit value	Empty	Alarm	No	Current value	
	Z calculation direction in cancelling radius compensation	Empty	Empty	No	Empty	

1.4.4 Block number

Format: N △△△△△

△△△△△ is 5-digit integer 00001~99999, and its leading zero can be omitted.

(1) Can or not input a block number in one block(must input block number in target block in which program skips), when many block number are input in one block, only the last block number is valid;

(2) Block number can be placed any position of block but it is suggested that it should be placed at the initial position in order to search and read;

(3) There can be many same block number in one program, but the block number of target block of program skip has only one; otherwise, the program skips to the nearest block to the block;

(4) block number can be placed at will.(it is suggested that it should be placed by the rising or falling monotonously;

1.4.5 Main program and subprogram

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

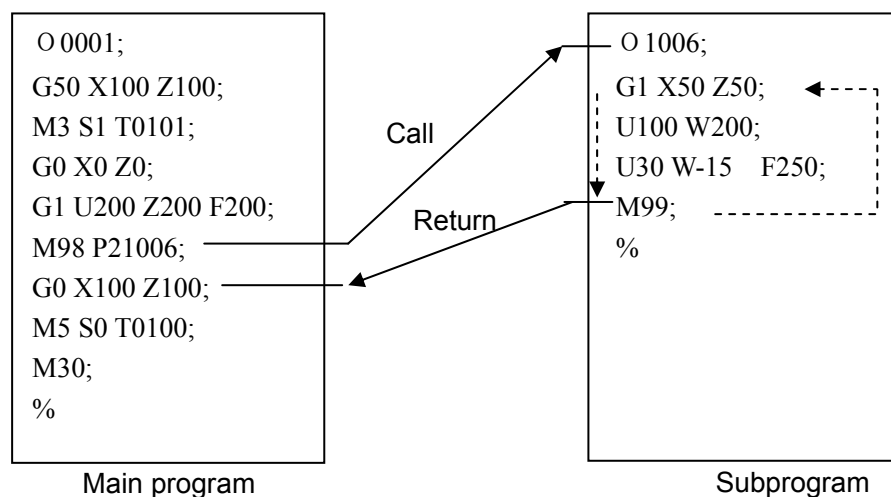


Fig.1-12

1.5 Program run

1.5.1 Sequence of program run

Running the current open program must be in Auto mode. GSK988T cannot open two or more programs at the same, and runs only program any time. When the first block is open, the cursor is located in the heading of the first block and can be moved in Edit mode. In the run stop state in Auto mode, the program starts to run by the cycle start signal (CYCLE START key is pressed or external cycle start signal) from a block pointed by current cursor, usually blocks are executed one by one according to their programming sequence, the program stops running till executing M02 or M30. The cursor moves along with program running and is located at the heading of the current block. Sequence and state of program running are changed in the followings:


- The program stops run after pressing RESET or EMERGENCY STOP button;
- The program stops running when the system or PLC alarms;
- The program runs and single block stops (the program run stops after the current block runs completely) in Edit, MDI mode, and then a block pointed by the current cursor starts running






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

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



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

- The program pauses after pressing  or the external cycle start signal is switched off, and it continuously runs from current position after pressing  or the external cycle start signal is switched on;
- When Single Block is ON, the program pauses after every block is executed completely, and then it continuously runs from the next block after  is pressed or the external cycle start signal is switched on;
- Block with “/” in the front of it is not executed when the block skipping switch is ON;
- The system skips to the target block to run after executing G65;
- Please see Section Three G Commands about execution sequence of G70~73;
- Call corresponding subprograms or macro program to run when executing M98 or M9000~M9999; the system returns to main program to call the next block when executing M99(if M99 specifies a target block number, the system returns to it to run) after the subprograms or macro programs run completely;
- The system returns to the first block to run and the current program is executed repetitively when M99 is executed in a main program.

1.5.2 Execution sequence of word

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

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

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

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

Execution sequence of G, M, S, T in the same block defined by GSK988T standard PLC program is as follows:

M3, M4, M8, M10, M12, M32, M41, M42, M43, M44, S□□, T□□□□ and G codes are executed simultaneously;

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

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

1.6 Coordinate Value and Dimension

1.6.1 Absolute programming and incremental programming

There are 2 methods to command tool movement: absolute value command and incremental value command. In absolute programming, the system uses end point coordinates; in incremental programming, it uses movement programming. The system selects the absolute program or incremental programming according to the address word as follows;

Table 1-6

	Absolute command	Incremental command
X movement command	X	U
Y movement command	Y	V
Z movement command	Z	W
C movement command	C	H

The system can select the incremental programming or absolute programming, or the compound programming including incremental and absolute programming;

X100.0 W100.0;

When the absolute value and incremental value of one axis are in the same block, the latter is value.

When X, Y, Z, C are in the same block, the latter is valid. And when U, V, W, H are in the same block, the latter is valid. When No.3403 BIT6(AD2) is set to1, the system alarms. U,W in some commands are specified by others. For example, these information are described in G code in G73.

When the user takes A, B as a position word, the system only uses absolute programming.

1.6.2 Diameter programming and radius programming

Because the workpiece section is the circle in CNC turning controlled program, X dimension can use two kind of method; diameter programming command and radius programming command.

1. The user can select the radius programming or diameter programming, which is set by state parameter (No. 1006 Bit 3(DIAX)).

2. Parameters related to diameter/radius programming:

State parameter No.1006 BIT3 (DIAX):

0—radius programming;

1—diameter programming;

State parameter No.5004 Bit1(ORC):

0—offset value is expressed with diameter;

1—offset value is expressed with radius;

Pay more attention to the conditions in the following table when X uses diameter programming:

Table 1-7 related addresses and data to the diameter or radius programming

	Word	Explanation	Diameter programming	Radius programming
Related addresses to diameter/radius programming	X	X coordinate, polar coordinate	Diameter value	Radius value
		G50 sets X coordinate	Diameter value	Radius value
	U	X increment	Diameter value	Radius value
		G71 infeed amount	Radius value	
		X finishing allowance in G71, G72, G73	Parameter definition	
		tool retraction amount in G73	Radius value	
	R	Clearance in G71, G72	Radius value	
		Clearance after cutting in G75	Diameter value	Radius value
		Clearance to end point in G74	Diameter value	Radius value
		Taper in G90, G92, G94, G76, radius in G02, G03, thread finishing amount in G76	Radius value	
	I	X amount of circle center	Radius value	
	F	G32,G34,G92,Pitch long axis is X in G76	Radius value	
		X federate display	Radius/rev, radius /min	
others	X or U value of position window	Display	Diameter value	Radius value

Note: Other related address word, data are expressed with radius value besides the above.

1.6.3 Decimal programming

Value can be input by decimal programming. Distance, time and speed can be input by decimal programming. The following addresses can use decimal point: X, Y, Z, A, B, C, U, V, W, H, I, J, K, R and F, and other addresses cannot use decimal programming.

There are two types of decimal point usage which is decided by No. 3401 Bit0(DPI).

When NO.3401 Bit 0(DPI) is set to 1, value without decimal point is with mm, inch.

When NO.3401 Bit0(DPI) is set to 0, input value is specified by least input increment.

Table 1-8

Parameter setting			Least command unit least command unit
ROTx=0 Rotary axis	Rotary axis is not related to parameter INI	ISC=0 ISB	0.001deg
		ISC=1 ISC	0.0001deg
ROTx=1 Linear axis	INI=0 Metric	ISC=0 ISB	0.001mm
		ISC=1 ISC	0.0001mm
	INI=1 Inch	ISC=0 ISB	0.0001inch
		ISC=1 ISC	0.00001inch

Example:

Table 1-9

Program command	The corresponding actual value when DPI is 1	The corresponding actual value when DPI is 0
X1000 without decimal command value	1000mm Unit: mm	1 mm Unit: least input increment(set to 0.001)
X1000.0 with decimal command value	1000mm unit: mm	1000mm Unit: mm

The decimal which is less than the least input increment unit is discarded in course of program being executed.

Example: X2.34567. When the least unit of input increment is 0.001mm, X2.34567 becomes X2.345, when the least unit is 0.0001inch, it becomes X2.3456.

The system alarms when the specified is more than 8-digit value.

1.6.4 Conversion between metric and inch

Metric input or inch input is set by NO.0000 Bit2(INI). G commands corresponding to metric/inch system is as follows:

G20: inch input ;

G21: mm input.

Input data unit becomes the inch or metric input unit when NO.0000 Bit2 (INI) setting is changed. But, the angle unit is not changed.

- F feedrate;
- position command;
- zero offset of workpiece;
- tool compensation value;
- graduation unit of MPG;
- movement distance in incremental feed.

NO.1001 Bit0(INM) can set MM or INCH input of least command increment in linear axis.

0: mm input(metric machine)

1: inch input (inch machine)

1.6.5 Linear axis and rotary axis

NO.1006 Bit0(ROTx) can set each axis to linear axis or rotary axis. NO. 1006 Bit 1 (ROSx) can be used to select the rotary type of each axis.

Absolute coordinate value is displayed circularly with the movement per rev set by NO.1260 when the cycle function is executed, which can prevent the rotary axis from overflowing. The cycle function is valid when NO.1008 Bit 0(ROAx) is set to 1.

For absolute value command, the coordinate values is the corresponding angle cycle value of per rev set by NO. 1260 after the machine moves. When NO.1008 Bit 1(RABx) is set to 0, the machine rotates according to the shortest distance(to the target point). For incremental command, the machine moves according to the angle defined by the command.

1.7 Grammar Rules

Programming needs some rules, and has some regulars which can help user compile prescriptive program and ensure the program is correct. There are rule tables for search, and their concrete details and related contents are referred to the related Chapters.

The grammatical rules including morphology, grammar, preparatory code function and so on.

1. 7.1 Lexical classification

Block lexical is divided into NC and macro lexical, which cannot exist in one block simultaneously.

Macro lexical of the system includes three kinds:

Assignment sign"=" lexical;

Key word lexical of IF, WHILE, GOTO, DO, END and other logical skip

G65, G66, G67 lexical;

Besides the above three kinds of lexical, others are NC lexical and concrete contents are referred to Chapter 3.15.4.

1. 7.2 Noun explanation

Character: user can input any number, letter, bracket, punctuation mark by MDI keyboard, such as: 1, A or X, # which consist of word or key word as follows:

Morphology: term is a word or a key word, such as X100, W[#100+1], GT, WHILE and so on.

Terms identified by the system are referred to 1.4.3 Word Table 1-4, and Morphology is a rule for terms, i.e.words compiled by user can correctly form these terms.

Semantic: it is the meaning of one statement to express. NC statement in the system is the semantic defined by related preparatory function, and statement in the macro program is the semantic defined by key word.

Grammar: it is the rules of words and expressions, and statement is to express user's semantic, and only to express one meaning without conflict rules. It expresses the correct or

mistaken statement.

1.7.3 Example explanation

(1) NC statement

G01 X20. W-10. F200.

After the user orderly inputs character by MDI keyboard: G01X20.W-10.F200

Character(word): G, 0, 1, X, 2, 0, ., W, -, 1, 0, ., F, 2, 0, 0, .;

Term(word): G01, X20, W-10., F200.;

Statement(block): G01 X20. W-10. F200.;

Semantic(functional definition): the tool moves at 200mm/min from the last point(the above) to X20, and moves negatively to Z10. the concrete movement unit and speed are defined by the parameter.

(2) Macro program

IF [#1 GT #110] THEN #2=0

After the user orderly inputs character by MDI keyboard: IF[#1GT#110]THEN#2=0:

Character(word): I, F, [, #, 1, G, T, #, 1, 1, 0,], T, H, E, N, #, 2, =, 0;

Term(key word): IF, [, #1, GT, #110,], THEN, #2, =, 0;

Statement(block): IF [#1 GT #110] THEN #2=0;

Semantic(functional definition): when # 1 variable is more than #110 variable,# variable is 0, otherwise, it does not change.

1.7.4 Expression

Expression is the basis of consisting of term, and the expression explanation of the system is referred to **Chapter 3.15.3**.

The expression supported by the system is data, variable sign, +-*/, functional function supported by the system, mathematic expression composed by []; data and operator in the [] can be separated by any blank space.

(1) The 5-level can be nested by the bracket[] .

(2) Data is composed by most 0-9, negative sign and decimal point, the negative sign is placed before the data, a data cannot have two decimal points, and the zero before or after the decimal point can be omitted, such as 123,-123,123,234 and so on. There must not be the blank space in the data.

(3) The '+' which is after the unit doublet operational character and which is before the data cannot be used. For example: 12++12 is illegal, but 12+[12] is correct,

(4) Directly use the negative in the digital in the operation, for example: -9/-3, -5*-5, -19—5, #110/-#2 are illegal;

(5) '/' in the expression should be in 1-layer bracket, and otherwise, is considered ad the skip character.

(6) When functional functions described in Chapter 1.4.3 Word Table1-4 exceed two characters, the first two characters is considered as its abbreviation.

(7) [#j] and [#k] in Chapter 3.15.3 are any expression, the bracket can be omitted when the data in the functional function is the data, its expression needs the bracket when the variable can omit the bracket, such as SIN45, EXP#102 ROUND[#1+SIN[#2-#3]], ABS[SIN[SIN#1/COS#2]+COS[COS#1*SIN#2]].

(8) Operation priority meets the operation rules described in mathematic expression, and its

concrete rules are referred.

Table 1-10 operator priority

Priority	Operator and function
5	"", "[", "]"
4	"#"
3	"SIN", "SI", "ASIN", "AS", "COS", "CO", "ACOS", "AC", "TAN", "TA", "ATAN", "AT", "SQRT", "SQ", "ABS", "AB", "ROUND", "RO", "FIX", "FI", "FUP", "FU", "LN", "EXP", "EX", "BIN", "BI", "BCD", "BC",
2	"AND", "AN", "*", "/",
1	"OR", "XOR", "XO", "+", "-",

1.7.5 Morphology explanation

The statement is composed of terms, and so we must explain the term when we describe the grammar.

The term in the system is divided into word, key word, and annotation symbol, and the word is the element to form NC statement, and the key word the element to form macro statement, and expression.

1) Word

Word: words identified by the system are referred to 1.4.3 Word Table 1-4.

Word format: word address+expression.

The middle between word and expression has no blank space, and the expression needs []. The bracket can be omitted when the expression is composed of data(data is composed of 0-9 digital, negative sign and decimal point), such as X10, W-10, R[SIN#7]. The system alarms when the expression following the word is illegal. The word addresses are divided into four groups:

(1) axis position word address (IP):

X,Y,Z,A,B,C,U,V,W,H

Axis position word is the position value, all axis position words are described to IP, and they can be followed by any legal expression. When the system has no the axis, it alarms. The axis relative information is referred to Chapter 1.6.1.

(2) Position command auxiliary word address:

I,J,K,R,F

The position command auxiliary word address and the position word address have the same use, and they can be followed by any legal expression and can specified at will. The system selects the corresponding word according to the corresponding preparatory function and the unused word is ignored.

(3) Preparatory function, auxiliary function, auxiliary word:

G, M, S, T, P, Q, L

Any legal expressions can follows these word addresses. They must meet the requirements when expressions consist of data. G codes are referred to Table 3-1 and others cannot have decimal point and negative sign, otherwise, the system alarms.

(4) File name and line number

N, O

Only data can follow the two words without decimal point and negative sign, otherwise, the

system alarms.

D and E in the system are illegal words, and the system alarms when they are used in NC statement.

2) Key word

The key words of the system are divided into transfer, logic judgement and operator. Key word with more than one character cannot have blank space. For example, ASIN cannot be written to A SIN, otherwise, the system alarms.

(1) Transfer command

IF, WHILE, GOTO, THEN, DO, END, the command exceeding two characters can use the first two characters: WH, GO, TH, EN

(2) Logic judgement command

GT, GE, EQ, NE, LE, LT can use the external input replacing character: >, >=, ==, <>, <=, <

Those only appear in the logic judgement brackets following IF, WHILE.

(3) Operator

Operators in Table 1-10, assignment operator="".

Operators mainly consist of expressions to complete the calculation.

3) Other symbol

(1) Annotation symbol

The annotation symbol have two types: Line end annotation symbol";", and annotation symbol in line"("and")".

The content following the annotation symbol is cleared when ";" is used;

When "("and")" is used, the code in "("and")" is cleared; when MDI keyboard cannot input, the external inputting program can realize the function.

(2) Skip symbol

"/" used in the system is taken as the skip symbol. The symbol can be placed any position of a block, the code between the symbol and the line end can optionally skip, and skips the whole line when it is placed at the line home.

Pay more attention to use them because the symbol and division sign are conflict. "/" in other positions are taken as the division sign when "/" outside of the first bracket is the skip symbol. "/" right to the assignment statement"=" is also taken as division sign.

(3) End character for file

The last line executed by the system is completed when there is "%" in the line.

1.7.6 Related parameters

After the user compiles one NC block, the system confirms its value according to the parameter. For example, G1 W100 values are different when they are set by the different system parameters. Parameters related to linear axis position are described here.

(1) DPI

DPI use decimal address. When the decimal point is omitted, the setting is as follows:

0: least setting unit

1: mm, inch, sec

Refer to **Chapter 1.6.3** about the parameter, and to the following ISC description about

the least setting unit.

(2) INI

INI input unit

0: metric

1: inch

Refer to **Chapter 1.6.4** about the parameter.

(2) ISC

Table 1-11 ISC sets least input unit and least command increment.

ISC	Least setting unit, least movement unit	Abbreviation
0	0.001mm, 0.001deg or 0.0001inch	IS-B
1	0.0001mm, 0.0001deg or 0.00001inch	IS-C

Refer to **Chapter 1.3.2** about the parameter.

For example: G1 W-100 results are different because the three parameter settings are different as follows:

Table 1-12 position command valuing method

Parameter combination			W100 value
DPI=0	INI=0	ISC=0	0.1mm
		ISC=1	0.01mm
	INI=1	ISC=0	0.01inch
		ISC=1	0.001inch
DPI=1	INI=0	ISC=0	100mm
		ISC=1	100mm
	INI=1	ISC=0	100inch
		ISC=1	100inch

The most input digit is the integer between -99999999 and +99999999, and the decimal point between -.99999999 and .99999999. The data which exceeds the least setting unit is omitted.

For I & Q position amount cannot be specified by the decimal point, and when their data meet DPI=0, they are taken as the least setting unit.

1.7.7 Grammar explanation

The block input by the user can be divided into NC and macro statement. The statement rules are described in the following.

(1) NC statement

NC statement can specify the run of all parts of machine, including line number, preparatory function, position command, cutting feedrate, miscellaneous function and so on.

/ O_ N_ G[] IP (annotation in the line) F[] S[] T[] M[] ; annotation
Word rules are referred to **Chapter 1.7.5**.

/ : skip character;

O_ : file name, to be omitted generally;

N_ : line number. Example: N30;;

[] : expression, program rules are referred to Chapter 1.7.4;

G[] : preparatory function talbe. Exmpale: G01;
 IP : axis position command list is the composition of position command, and the concrete is referred to 1.7.5 . it is expressed to X[] W[] and so on;
 (annotation in line) : annotation in statement;
 F[] : current specified preparatory function providing cutting feedrate;
 S[] : spindle speed;
 T[] : tool function;
 M[] : miscellaneous function table;
 ; : annotation explanation.

Note:

- 1) The blank spaces can be omitted or any blank spaces can be added between words. total quantities of character in one line cannot exceed 255.
- 2) Each word position in one line can be changed, and the system only has its one general expression described in the above. Example: the meanings of G04X4. and X4.G04 are to dwell 4s, but G words in the same group are not described here when they are compiled in one block.
- 3) Each word in the above described general expression can be omitted, and the system executes the block according to the previous block mode when all are omitted.
- 4)The system ignores it when there is a word which cannot be used by the preparatory function, and alarms when it the axis word. Example: G01W-100P2000, the system ignores P2000 word.
- 5) Words in the system are described in the following, and it is suggested that the programming should use the following standard format. Confirm preparatory function G, and then axis word, and select others according to the requirements.

$$O[] \quad N[] \quad G[] \quad \frac{X[] \quad U[]}{Y[] \quad V[]} \quad \frac{I[]}{J[]} \quad \frac{P[]}{Q[]} \quad F[] \quad S[] \quad T[] \quad M[]$$

$$\frac{Z[] \quad W[]}{C[] \quad H[]} \quad \frac{K[]}{R[]} \quad L[]$$

$$\frac{A[]}{B[]}$$

The preparatory function determines the meaning described by the block in NC command, and G formats are described in the following. See **Chapter 3 G commands** in detail.

IP : combination of axis word
 Xp : basic X and its parallel axis
 Yp : basic Y and its parallel axis
 Zp : basic Z and its parallel axis
 [] : expression

Table 1-13 G format reference

Group	G format	Explanation
00	G04 X[] G04 P[] G04 U[] G04	
	G28 IP	

	G30 P[] IP	
	G50 IP S[]	
	G52 IP	
	G53 IP (*2)	
	G65 P[] L[] [independent variable list] (*1)	Only one line is specified alone
	G70 P[] Q[]	
	G71 U[] R[] G71 P[] Q[] U[] W[]	
	G72 W[] R[] G72 P[] Q[] U[] W[]	
	G73 U[] W[] R[] G73 P[] Q[] U[] W[]	
	G74 R[] G74 Xp[] Zp[] P[] Q[] R[]	
	G75 R[] G75 Xp[] Zp[] P[] Q[] R[]	
	G76 P[] Q[] R[] G76 Xp[] Zp[] R[] P[] Q[] F[]	
01	G00 IP	
	G01 IP F[]	
	$G17 \left\{ \frac{G02}{G03} \right\} Xp[] Yp[] \left\{ \frac{R[]}{I[] J[]} \right\} F[]$	
	$G18 \left\{ \frac{G02}{G03} \right\} Xp[] Zp[] \left\{ \frac{R[]}{I[] K[]} \right\} F[]$	
	$G19 \left\{ \frac{G02}{G03} \right\} Yp[] Zp[] \left\{ \frac{R[]}{J[] K[]} \right\} F[]$	
	G32 IP F[] J[] K[] Q[]	
	G34 IP F[] J[] K[] Q[] R[]	
	G90 Xp[] Zp[] R[] F[]	
	G92 Xp[] Zp[] R[] J[] K[] F[] L[]	
	G94 Xp[] Zp[] R[] F[]	
02	G96 S[]	
	G97 S[]	
05	G98 F[]	
	G99 F[]	
06	G20	Only one line is specified alone
	G21	Only one line is specified alone

07	G40	
	G41	
	G42	
09	G22	Only one line is specified alone
	G23	Only one line is specified alone
12	G66 P[] L[] [independent variable list] (*1)	Only one line is specified alone
	G67	Only one line is specified alone
14	G54	
	G55	
	G56	
	G57	
	G58	
	G59	
16	G17	
	G18	
	G19	
21	G12.1 G112	Only one line is specified alone
	G13.1 G113	Only one line is specified alone

*1): Format of independent variable list: independent variable address+expression, independent address+expression and so on. Concrete format is referred to Chapter 3.15.5.

*2): G53 address is specified by the absolute value, and the relative value can be ignored.

(2) Macro statement

Macro command call format is referred to Table 1-13 G command format reference. There are another two formats as follows:

1) Assignment statement

#[] = [] ; [] is an expression

2) Transfer statement

Five formats of skip statement execution:

Conditional skip statement: IF[expression logic judgement expression] GOTO expression;

Conditional assignment statement: IF[expression logic judgement expression] THEN assignment statement;

Conditional skip statement: GOTO expression

Limit cycle statement: WHILE[expression logic judgement expression] DO [1, 2, 3]; ; ; END[1, 2,

3]

Unlimit cycle statement: DO [1, 2, 3]; ; ; END[1, 2, 3]

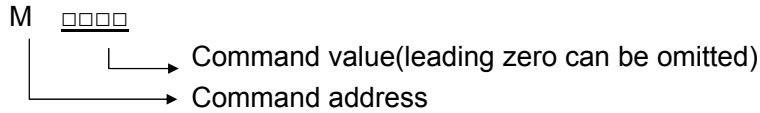
Note: One line only expresses one meaning of the above five formats, otherwise, the system alarms.

he line number must be in front of the skip statement.

CHAPTER 2 MSTF COMMAND

2.1 M (miscellaneous function)

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



There is one valid M code in one block. There are most specified 3 M codes in one block(set by NO.3404 Bit 7 (M3B)). The corresponding relationship between M codes and their functions are determined by the machine manufacturer. CNC sends M code signal and one strobe signal to PLC in executing M codes.

Except for M98, M198, M99, all M codes are executed in PLC.

The following M codes have special meanings.

2.1.1 End of program M02

Command format: M02 or M2

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

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

2.1.2 End of program run M30

Command format: M30

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

Besides the above-mentioned function executed by NC, M30 function is also defined by PLC ladder diagram as follows: the system closes M03, M04 or M08 signal output and outputs M05 signal after M30 is executed.

2.1.3 Program stop M00

Command format: M00 or M0

Command function: the system stops the automatic run after M00 block is executed, which is same that of the single block pausing to save the previous modal message, i.e. which is equal to the program pause function. Press the CYCLE START key on the operation panel to execute the follow block and the CNC continuously automatically runs.

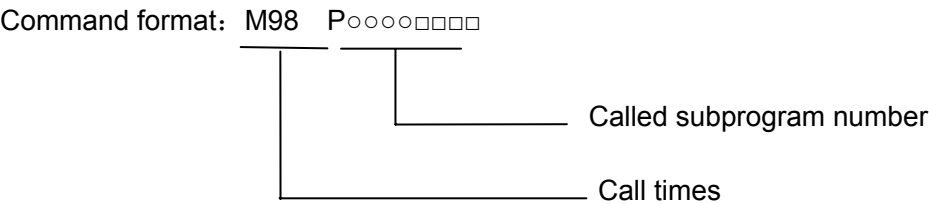
When M00 are other G command are in the same block, the system executes the command in

the block, then M00, and last stops running.

2.1.4 Optional stop M01

Command format: M01 or M1
Command function: after the block containing M01 is executed, the system stops the automatic run and the single block stopping signal lights. M01 is valid when the OPTIONAL STOP on the machine operation panel is pressed.

2.1.5 Subprogram call M98



Command function: In Auto mode, after other commands in the current block are executed in M98, CNC calls subprograms specified by P.

When the subprogram is called one time, ○○○○ can be omitted in inputting the number“○○○○□□□□” behind P, at the same time, the leading zero of the called subprogram number can be omitted and the system does not alarms. Example: M98 P12; it expresses to call the subprogram O0012 one time; the leading zero cannot be omitted when the subprogram call times are more than one.

The called subprogram name in M98 must be the program in the system and be less than 9999, and the subprogram name must be input.

The specified call times in M98 is 1~9999.

The called subprogram format in M98 is the following. The last end of the subprogram must be M99 instead of M30, its program compiling format is the same that of the main program compiling format.

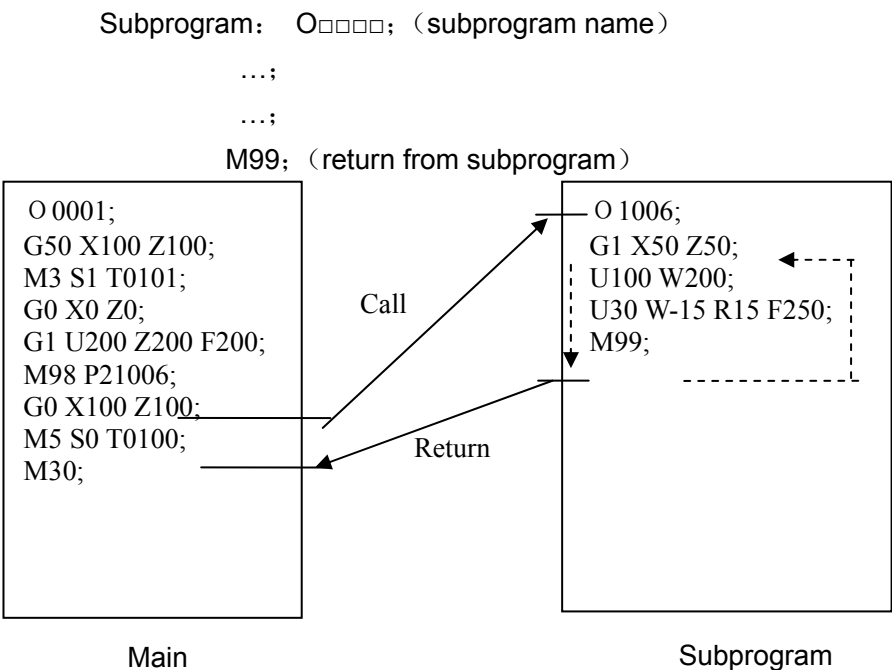


Fig.2-1 subprogram call

The called subprogram can call other subprograms. The subprogram called by the main program is called as the one-embedded subprogram, and the one called by the one-embedded subprogram is called as the two-embedded subprogram and so forth. One main program can call 12-embedded subprogram(including macro program call). The following is the four-embedded subprogram.

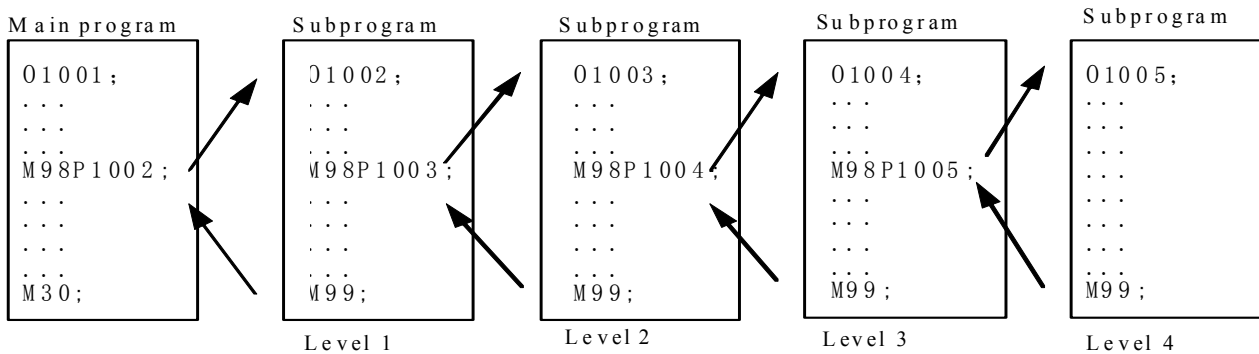


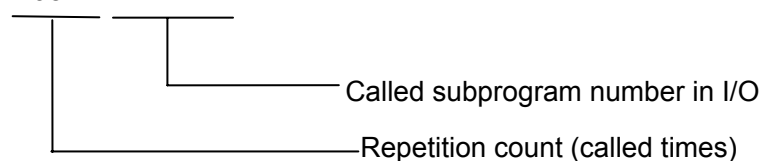
Fig. 2-2 Subprogram nesting

Note:

- (1) The system alarms when it has not searched the subprogram specified by P;
- (2) The system alarms when M98P__ is input in MDI, and the subprogram call cannot be executed;
- (3) The system alarms when P98P__ call itself;
- (4) The system alarms when M98 is commanded and the subprogram is called without P command.

2.1.6 Subprogram call M98

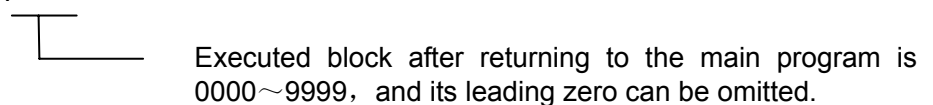
Command format: M198 P○○○○□□□



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

2.1.7 Return from subprogram M99

Command format: M99 P○○○○○



Command function: After other commands of current block in the subprogram are executed, the system returns to the main program and continues to execute next block specified by P, and calls a block following M98 of current subprogram when P is not input. The current program is executed

repeatedly when M99 is defined to end of program (namely, the current program is executed without calling other programs).

Example: Execution path of calling subprogram (with P in M99) as Fig. 2-3. Execution path of calling subprogram (without P in M99) as Fig. 2-4.

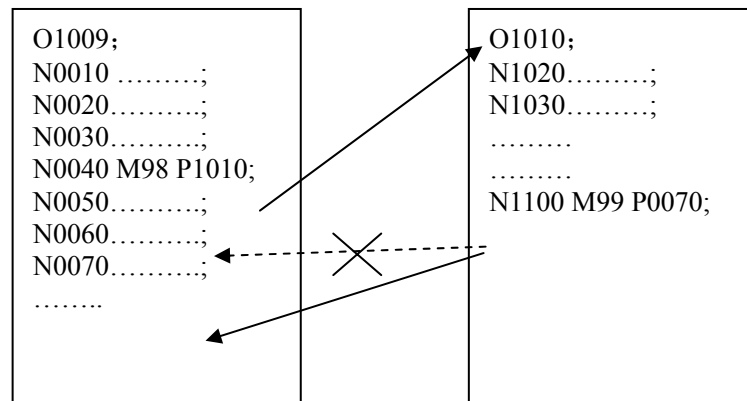


Fig.2-3

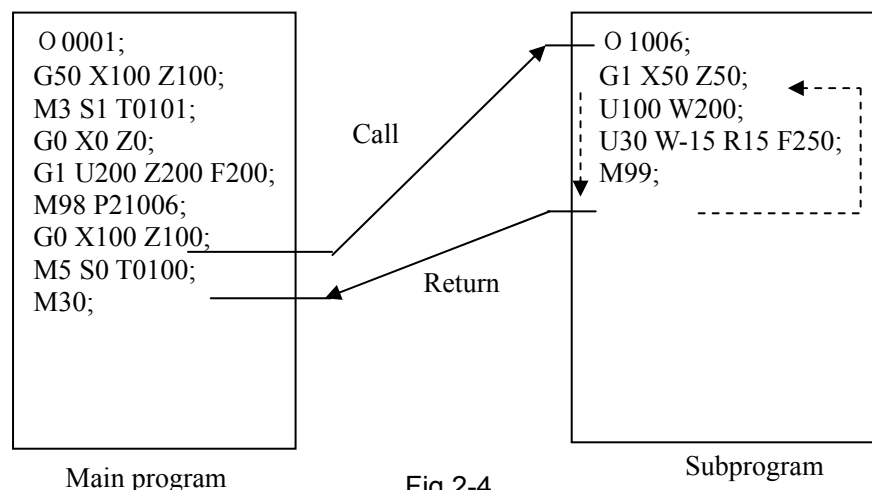


Fig.2-4

Notes:

- (1) M99 does not need to be specified in the alone block. Example: G00 X100 Z100 M99;
- (2) The system alarms when M99 has commanded the block number which does not exist;
- (3) In Auto mode, the program returns to the block which is placed in the front when the specified block number behind M99 is repetitive in the program;
- (4) In Auto mode, the system ignores the line and returns to the beginning of the file to perform the execution when the main program ends in M99 and specifies the line number following P.

2.1.8 M commands defined by standard PLC ladder

Besides the above commands(M02, M30, M98, M99), other M commands are defined by PLC. The following M commands defined by standard PLC are used to controlling GSK988T Turning CNC System. M function, meaning, control sequence and logic are referred to the manufacturer manual.

Table 2-1 M commands defined by standard PLC ladder

Command	Function	Remark
M00	Program pause	
M03	Spindle clockwise (CW)	Functions interlocked and states reserved
M04	Spindle counterclockwise (CCW)	
*M05	Spindle stop	
M08	Cooling ON	Functions interlocked and states reserved
*M09	Cooling OFF	
M10	Tailstock going forward	Functions interlocked and states reserved
M11	Tailstock going backward	
M12	Chuck clamping	Functions interlocked and states reserved
M13	Chuck releasing	
M32	Lubricating ON	Functions interlocked and states reserved

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

2.1.9 Spindle CW, CCW, stop M03, M04, M05

Command format: M03 or M3;

M04 or M4;

M05 or M5;

Function: M03: spindle CW ;

M04: spindle CCW;

M05: spindle stop.

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

2.1.10 Cooling control M08, M09

Command format: M08 or M8;

M09 or M9;

Command function: M08: Cooling ON;

M09: Cooling OFF.

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

2.1.11 Tailstock control M10, M11

Command format: M10;

M11;

Command function: M10: tailstock going forward;

M11: tailstock going backward.

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

2.1.12 Chuck control M12, M13

Command format: M12;
M13;

Command function: M12: chuck clamping;
M13: chuck releasing.

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

2.1.13 Lubricating control M32, M33

Command format: M32;
M33;

Command function: M32: lubricating ON;
M33: lubricating OFF.

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

2.1.14 Spindle automatic gear shifting M41, M42, M43, M44

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

Command function: the spindle automatically gears to № n gear when M4n is executed.

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

2.1.15 M command notes

(1) M00, M01, M02, M30, M98, M99, M198 are separately specified in one block, and other M command are ignored when it is in one block with others and the system executes only the above M commands; when the above 7 commands are commanded in the same one block, the first commanded M command is valid.

(2) CNC permits there are at most specified 3 M commands in one block (when No. 3404 Bit7 M3B is set to 1), and other M command cannot be specified simultaneously because of the limit of machine operations. For example: the spindle gear change command M41, M42, M43, M44.

(3) When the motion command and M auxiliary function command are specified in the same one block, there are two execution methods:

- a) the motion command M auxiliary function command are executed.
- b) the auxiliary function command is executed after the motion command is done.

The concrete method is referred to the machine manufacture's manual.

(4) No. 3010 sets the delay time of the strobe signal MF, SF, TF.

(5) No. 3011 sets the width of M, S, T function end signal (FIN) .

2.2 Spindle Function

S command is used to controlling spindle speed. In GSK988T spindle speed control, NC outputs 0~10V analog voltage signal to spindle servo device or inverter to realize the gradeless spindle speed.

2.2.1 Spindle speed analog voltage control

Command format: S □□□□□

Command function: the spindle speed is defined, and the system outputs 0~10V analog voltage to control spindle servo or converter to realize the stepless timing. S command value is not reserved, and it is 0 after the system is switched on.

Command explanation: spindle speed analog voltage control command

□□□□□ means the set spindle speed, its value range is referred to Table 1-4, and the leading zero can be omitted. When the value exceeds the range, the most spindle speed limit is specified in the program, and S value is specified to the most spindle speed; when it is not specified, the upper and lower limit of S value is specified. The system alarms when the decimal is input to the specified of the S value. The system can set the digit number by No.3031.

The first spindle of the CNC can execute 4-gear spindle speed, and the second spindle has 2-gear spindle speed. In executing S command, the system counts the analog voltage value corresponding to the specified speed according to setting value(corresponding to №037~№040) of max. spindle speed (analog voltage is 10V)of current gear, and then outputs to spindle servo or converter to ensure that the spindle actual speed and the requirement are the same.

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

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

2.2.2 Spindle override

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

The system supplies 8 steps for spindle override (50%~120% increment of 10%). The actual steps and tune of spindle override are defined by PLC ladder and introductions from machine manufacturer should be referred when using it. Refer to the following functions of GSK988T standard PLC ladder.

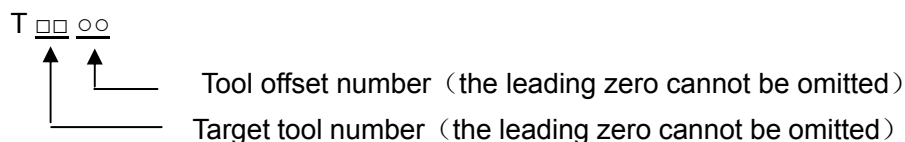
The spindle actual speed specified by GSK988T standard PLC ladder can be tuned real time by the spindle override tune key at 8 steps in 50%~120% and it is not reserved when the spindle override is switched off. Refer to the operations of spindle override in BOOK II OPERATION.

2.3 Tool function

T functions of GSK988T: automatic tool change and executing tool offset. Control logic of automatic tool change is executed by PLC and tool offset is executed by NC.

2.3.1 Tool select function

Command format:



Command function: The automatic tool post rotates to the target tool number and the tool offset of tool offset number commanded is executed. The tool offset number can be the same as the tool number, and also cannot be the same as it, namely, one tool can corresponds to many tool offset numbers. After executing tool offset and then T□□00, the system reversely offset the current tool offset and the system its operation mode from the executed tool length compensation into the non-compensation, which course is called the canceling tool offset, called canceling tool compensation. When the system is switched on, the tool offset number and the tool offset number displayed by T command is the state before the system is switched off, the tool offset number is in the cancelling state(i.e. 00 state).No. 3032 sets T code digit, and No.5002 Bit 0(LD1) sets the digit of tool offset number

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

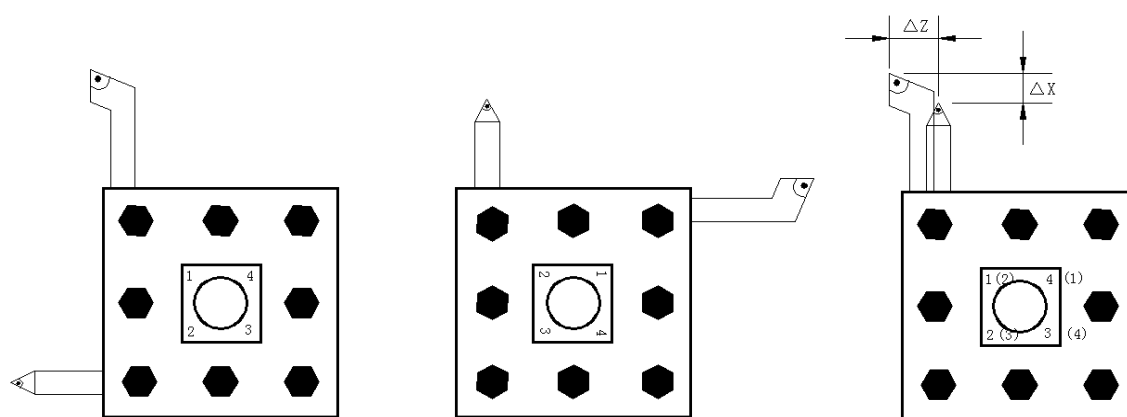


Fig.2-5 Tool offset

The tool offset is used for the programming. The offset corresponding to the tool offset number in T command is added or subtracted on the end point of each block. X tool offset in diameter or radius is set by No.5004 Bit1(ORC). For X tool offset in diameter or radius, the external diameter is changed along with diameter or radius when the tool length compensation is changed.

Example: When the state parameter No.5004 Bit1 is set to 0 and X tool length compensation value is 10mm, No.5004 Bit1 is set to 1 and X tool length compensation value is 10mm the diameter of workpiece external diameter is 20mm.

Fig. 2-6 is to create, execute and cancel the tool offset in movement mode.

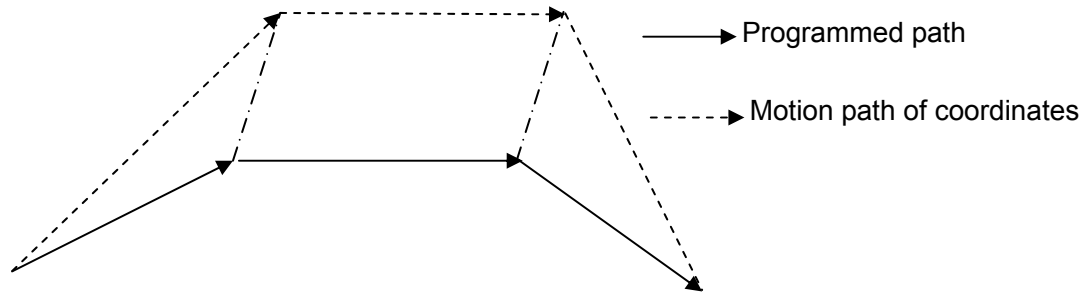


Fig. 2-6 Creation, execution and cancellation of tool length

G01 X100 Z100 T0101; (Block 1, start to execute the tool offset)
 G01 W150; (Block 2, tool offset)
 G01 X50 Z300 T0100; (Block 3, canceling tool offset)

There are two methods to execute the tool offset(they are set by No.5002 Bit4(LGT)):

- (1) **The tool length compensation is executed by the tool traversing;**
- (2) **The tool length compensation is executed by modifying the coordinates;**

Example:

Table 2-2

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

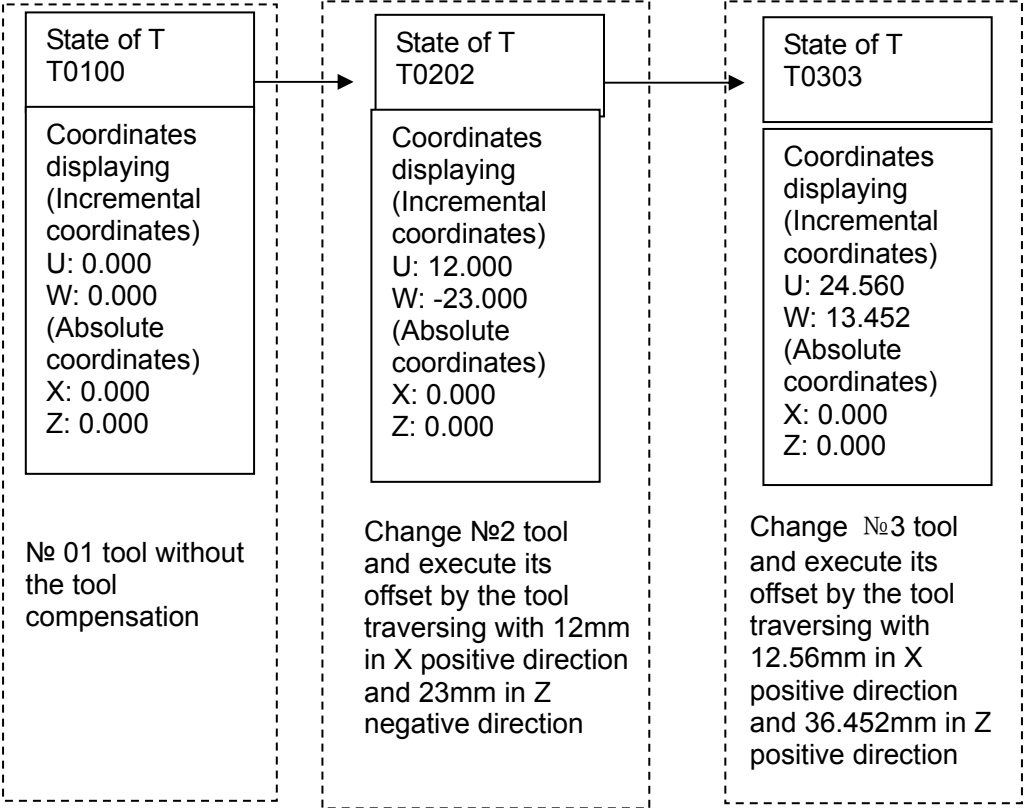


Fig. 2-7 Tool traversing mode to execute the tool offset

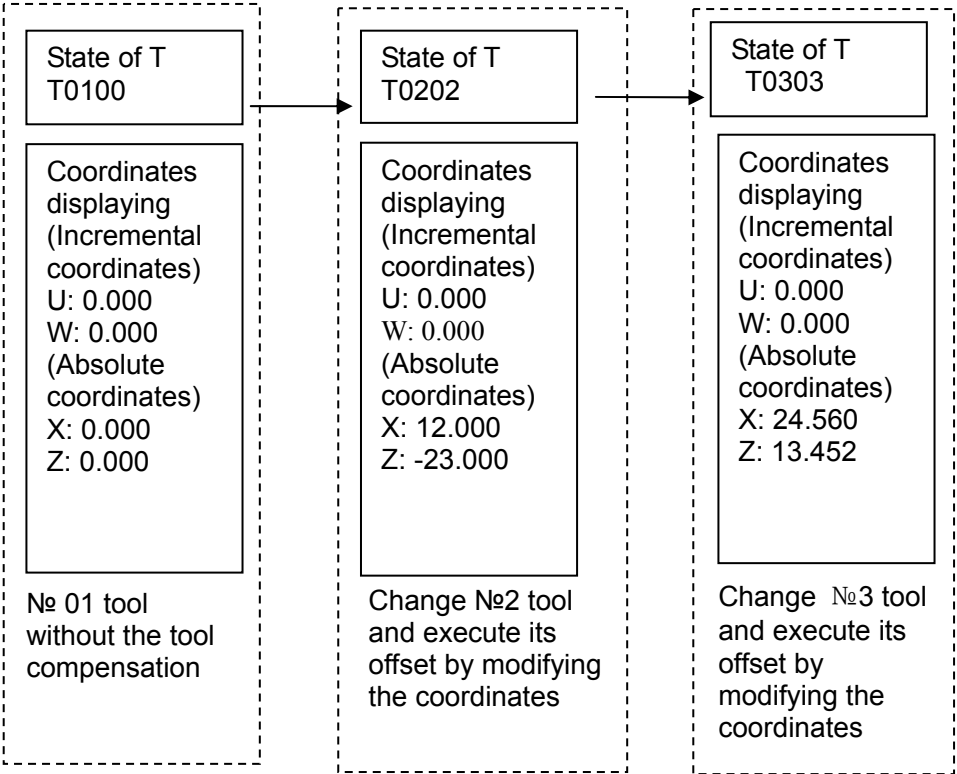


Fig. 2-8 Modifying the coordinates mode to execute the tool offset

When T command and the motion command are in the same block, they are executed simultaneously, in executing tool change, the system executes by adding the current tool offset to coordinates of motion command and whether the cutting feedrate or the rapid traverse speed is defined by the motion command.

Notes:

- (1) In tool traversing compensation mode, when the system executes the tool offset, NO.5002 Bit6 sets the valid method of the tool offset:
LWM=0: it is valid in the block of T code
LWM=1: it is valid in the axis movement block
- (2) NO.5001 Bit6(EVO) sets the valid method of modifying tool wear tool when the system executes the program:
EVO =0: it is valid in the next specified T code
EVO =1: it is valid in the next buffer block
Note: After the tool wear value is modified, the system executes the wear value in the tool traversing method to avoid the too big wear value.
- (3) It is suggested that the program should be complied according to the normative method, that is, the tool change is executed in the safe position and then the absolute value command is used to position to the starting point of the machining.
- (4) In the coordinate offset compensation mode, when the system executes the tool offset and executes T function command instead of movement command, it uses G50 to set the coordinate system, the displayed absolute coordinate value is that the coordinate value set by G50 adds or subtracts the unexecuted tool compensation value.
- (5) T command can use the leading zero. When T00□□ is commanded or only tool offset number is commanded in the program and the current tool number is not changed, the system only correspondingly modifies the current tool offset value.

2.4 Feed function

There are two kinds of feed function to control tool feedrate:

1. Cutting feed mode
2. Jog feed mode

2.4.1 Cutting feed mode

1. Cutting feed(G98/G99, F command)

Command format: G98 F__;

Command function: cutting feedrate is specified as mm/min, G98 is the modal G code. G98 cannot be input if the current command is G98 modal.

Command format: G99 F__;

Command function: cutting feedrate is specified as mm/min, G99 is a modal G code. G99 cannot be input if the current command is G98 modal. When G99F is executed, the arithmetic product of F command value(mm/rev) and current spindle speed(r/min) is taken as feedrate to command actual cutting feedrate which is changed along with spindle speed. The cutting feedrate per rev specified by G99 F_ is contributed to the equable cutting line on

the surface of workpiece. In G99, the machine tool must be employed with the spindle encoder to machine the workpiece on the machine tool.

G98, G99 are the modal G code in the same group and only one is valid. The modal defaulted by the system is set by No.3402 Bit4(FPM) when the system is switched on.

Reduction formula of feed between per rev and per min:

$$F_m = F_r \times S$$

F_m : feed per min;

F_r : feed per rev;

S : spindle speed.

F value is reserved after F is commanded.

The system supplies 16 steps for spindle override (0%~150%, increment of 10%).PLC ladder defines the actual feedrate override steps, reserving in power-on and the tune ways, which are referred to machine manufacturer's manual.

The cutting feedrate can be tuned real time by the feedrate override key on the operator panel or the external override switch, and the actual cutting feedrate is tuned in 0~150%.

2. Thread cutting

Thread cutting: The system specifies a pitch to execute thread cutting along with spindle rotating.

The tool moves a pitch when the spindle rotates one rev. Feedrate is relevant to the specified pitch, actual spindle speed. The system must be employed with spindle encoder which transmits the spindle actual speed to CNC in thread cutting. The thread cutting is not relevant to feedrate override and rapid override.

$$F = f \times S$$

F : Thread cutting feedrate;

f : Specified pitch;

S : Spindle actual speed.

2.4.2 Manual feed mode

1. Manual feed

Manual feed: the tool traverses at the current manual feedrate in X, Y, Z positive/negative direction.

The system supplies 16 steps for spindle override (0%~150%, increment of 10%).PLC ladder defines the actual feedrate override steps, reserving in power-on and the tune ways, which are referred to machine manufacturer's manual.

2. MPG/Step feed

MPG feed: X or Z positive/negative moves at the current increment in "MPG" mode but does not move simultaneously in the same direction.

Step feed: X or Z positive/negative moves at the current increment in "Step" mode but does not move simultaneously in the same direction.

JHD(NO.7100 Bit0) sets whether the MPG(handwheel) can be used. When it is set to 1, the MPG and single step are valid together.

Besides, HPF(NO.7100 Bit4) specifies as follows:

It is set to 0: the feedrate is clamped at rapid traverse rate and generated pulses exceeding the

rapid traverse rate are ignored. At the moment, the distance the tool is moved may not match the graduation on the manual pulse generator(MPG).

It is set to 1: the federate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are not ignored but accumulated in the CNC. At the moment, no longer rotating the MPG does not immediately stop the tool. The tool is moved by the pulses accumulated in the CNC before it stops.

HPF(NO.7117) specifies as follows:

It is set to 0: The federate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are ignored. The distance the tool is moved may not match the graduations on the MPG.

Other than 0: The feedrate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are not ignored but accumulated in the CNC until the limit specified in No.7117 is reached. No longer rotating the MPG does not immediately stop the tool. The tool is moved by the pulses accumulated in the CNC before it stops.

The NC provides 4-grade (0.001, 0.01, 0. 1, 1) MPG/ step increment, actual MPG/step increment grades, increment selection method. The selection method of currently valid axis is defined by PLC and is referred to the machine manufacturer's manual.

3. Manual rapid feed

1. Rapid feed in the CNC

- (1) For the rapid traverse in automatic run, the tool traverses at the rapid traverse rate set by No.1420 and the corresponding override.
- (2) When the manually rapid run is executed, the tool moves at the rapid traverse rate set by No. 1424 of each axis and the corresponding override.

2. Rapid feedrate override

The switch on the machine operation panel can regulate the rapid traverse rate by the override which is F0, 25%, 50%, 100%.

F0 is the fixed speed set by No. 1421 of each axis.

3. Notes

- (1) The rapid traverse rate in automatic run is set in the parameter and is not related to the commanded feedrate F in the block;
- (2) The rapid traverse rate is more than G00 rapid traverse, and also includes all rapid traverses, such as the fixed cycle positioning, automatic reference return;
- (3) When the manual rapid traverse rate of each axis is set to 0 by No. 1424, the system uses the value set by No.1420.

2.4.3 Automatic acceleration/deceleration

When the axis begins to move and before it stops, GSK988T can automatically accelerate/ decelerate contributed to the smooth speed to reduce impinge of run starting and stopping. GSK980TDa uses accelerations/decelerations as follows:

Rapid traverse: linear-post acceleration/deceleration

Cutting feed: No. 1610 Bit 0(CTLx) sets the acceleration/deceleration of cutting feed of each axis

Thread cutting: exponential/linear post-acceleration/deceleration

Manual feed: No. 1610 Bit4(JGLx) sets the acceleration/deceleration of cutting feed of each axis

MPG feed: linear-post exponential post-acceleration/deceleration

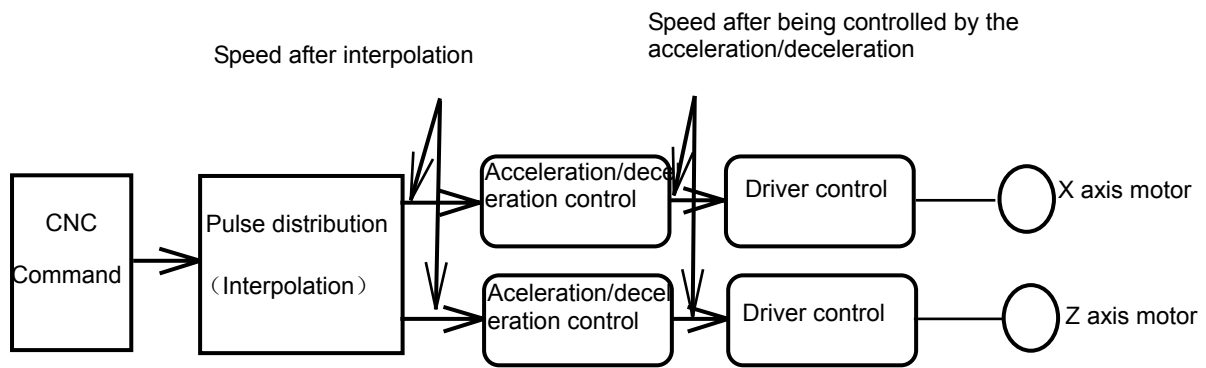


Fig. 2-9

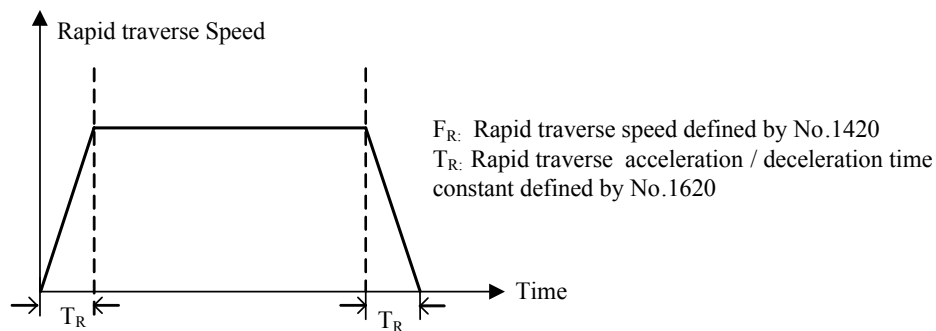


Fig.2-10 Rapid traverse speed

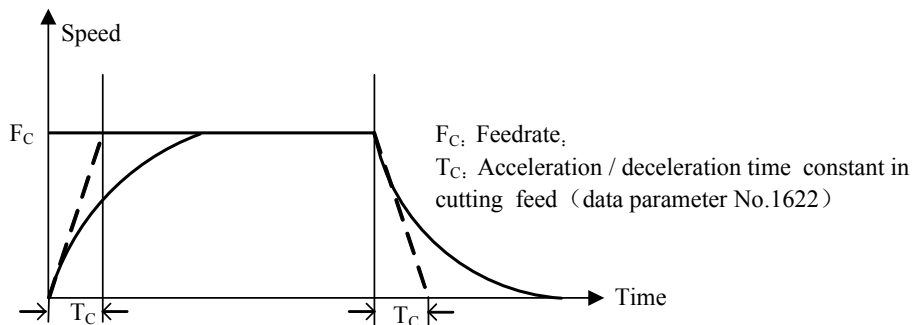


Fig. 2-11 Cutting feed, manual feed speed

When GSK988T executes the cutting feed, it uses exponential post-acceleration/deceleration, and there is a transitive arc caused by acceleration/ deceleration at the intersection of two paths, which is not positioned exactly on the intersection of the two paths, so there is a contour error between actual path and programming path, which is formed at a path intersection of neighboring two blocks in cutting feed. To avoid the contour error, the system executes G04 in two blocks or set №007 Bit3 to 1. At this moment, the previous block runs and positions exactly to its end point with zero mm/min and then the system starts to execute the next block, which increases program's running time and reduces machining efficiency.

Example (do not execute the exact positioning):

G01 U-100; (X negatively moves)

W-200; (Z negatively moves)

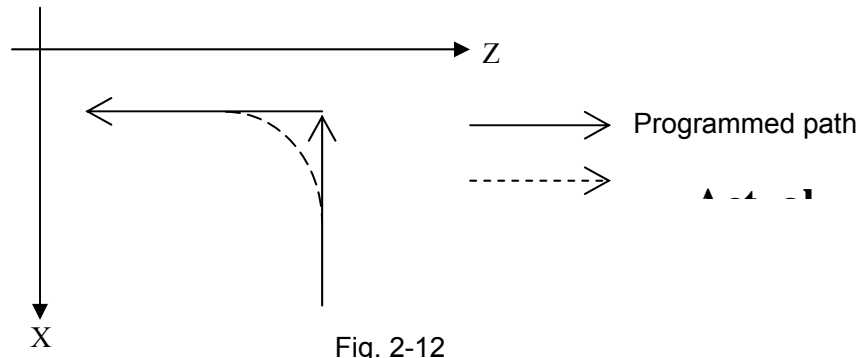
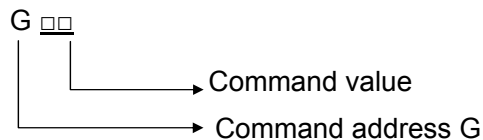


Fig. 2-12

CHAPTER 3 G COMMANDS

3.1 Summary

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



G words are divided into different groups. One block only has one G command in the same group, the last one is valid when two or more G commands in the same group are input and No.3403 Bit6(AD2) is set to 0, and the system alarms when it is set to 1. The words in the different groups without the same parameter (word) can be in the same block and their functions are valid without sequence at the same time. The system alarms when G words do not belong to Table 3-1 or they are optional functions without being supplied.

Table 3-1 G command list

G command	Group	Function
G00	01	Positioning(rapid traverse)
*G01		Linear interpolation
G02		Circular interpolation(CW)
G03		Circular interpolation(CCW)
G04	00	dwell
G12.1 (G112)	21	Polar coordinate interpolation mode
G13.1 (G113)		Polar coordinate interpolation mode cancel
G17	16	XpYp plane selection
*G18		ZpXp plane selection
G19		YpZp plane selection
G20	06	Inch input
*G21		mm input
*G22	09	Stored travel check ON
G23		Stored travel check OFF
G28	00	Return to reference point
G30		Return to 2 nd , 3 rd , 4 th reference point
G32	01	Constant pitch thread cutting
G34		Variable pitch thread cutting
*G40	07	Tool radius compensation cancel
G41		Cutter compensation left
G42		Cutter compensation right
G50	00	Workpiece setting or max. spindle speed setting
G52		Local coordinate system setting

G53		Machine coordinate system setting
*G54	14	Select workpiece coordinate system 1
G55		Select workpiece coordinate system 2
G56		Select workpiece coordinate system 3
G57		Select workpiece coordinate system 4
G58		Select workpiece coordinate system 5
G59		Select workpiece coordinate system 6
G65	00	Non-modal macro program call
G66	12	Macro program mode call
*G67		Cancel macro program mode call
G70	00	Finishing cycle
G71		Axial roughing cycle
G72		Radial roughing cycle
G73		Closed cutting cycle
G74		Axial grooving cycle
G75		Radial cutting multi-cycle
G76		Multi thread cutting cycle
G90	01	Axial cutting cycle
G92		Thread cutting cycle
G94		Radial cutting cycle
G96	02	Constant surface speed control
*G97		Constant speed control
*G98	05	Feed per minute
G99		Feed per rev

Note:

1. After the system is turned on, the modal G command in every group is in the state with *; G commands in Group 1, Group 5, Group 9 separately set the power-on state in No. 3402 Bit0(G01), Bit4(FPM) Bit7(G23), G commands in Group 06 sets the power-on state in No.0000 Bit2(INI);
2. When the system reset and No.3402 Bit6 (CLR) is set to 0, G modal keeps, when it is set to 1, G modal is switched into the initial mode in power-on, but G22 and G23 in Group 09 keep.
3. G command in Group 0 and G command in Group 1 are specified in the same block, G command in Group 00 is valid, G command in Group 1 only changes its modal.
4. The word required by the G command in one block can be compiled. The system ignores it when the word which cannot be used by G command is compiled; the system alarms when the ignored word format is not correct; there is no axis word in No.1020 including corresponding relative value address, the system alarms. The system alarms when the address D and E which are not defined by the system are compiled.
5. Commands in Group 06, 09, 21 cannot in the same block with those in other groups. The command in Group 12, G65 is executed in alone block, otherwise, the system alarms.

3.1.1 Modal, non-modal and initial mode

After G commands are executed, their defined functions and states are valid until they are changed by others in the same group, which commands are called modal G commands. After the modal G words are executed, and before their defined functions and states are changed, the G

command cannot be input again when they are executed by the following block.

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

After the system is switched on, the valid modal G commands not to be executed functions or states are called initial mode G command. Take it as the initial mode G command to be executed when it is not be input after the system is switched on. The initial commands of GSK988T are referred to the commands with “*” in Table 3-1.

3.1.2 Omitting word input

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

Table 3-2

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

Note:

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

metric thread.

2. They can be omitted not to input when the address characters X(U) , Z(W) are the coordinates of end point of block and the system defaults the current absolute coordinates in X or Z direction to the coordinate value of end point of block.

Example:

```
O0001;
G0 X100 Z100;    (rapid traverse to X100 Z100; the modal G0 is valid)
X20 Z30;         (rapid traverse to X20 Z30; the modal G0 is not input)
G1 X50 Z50 F300; (linear interpolation to X50 Z50, feedrate 300mm/min; the modal G1 is
                  valid)
X100;            (linear interpolation to X100 Z50, feedrate 300mm/min; Z coordinate is
                  not input and is the current coordinates Z50; F300 is kept, G1 is modal
                  and is not input)
G0 X0 Z0;        (rapid traverse to X0 Z0 and the modal G0 is valid)
M30;
```

Example 2:

```
O0002;
G0 X50 Z5;       (rapid traverse to X50 Z5)
G04 X4;          (dwell 4 seconds)
G04 X5;          (dwell 5 seconds again, G04 is non-modal and is needed to input
                  again)
M30;
```

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

```
O0003;
G98 F500 G01 X100 Z100;    (Feedrate per minute 500mm/min in G98)
G92 X50 W-20 F2 ;         (F value is a pitch and must be input in thread cutting)
G99 G01 U10 F0.01         (Feedrate per rev in G99 must be input again)
G00 X80 Z50 M30;
```

3.1.3 Related definitions

Definitions of word are as follows except for the especial explanations:

Starting point: position before the current block runs;

End point: position after the current block ends;

X: X absolute coordinates of end point;

Xp: absolute coordinate of X end point or one which is parallel to X;

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

Y: Y absolute coordinate of end point;

Yp: absolute coordinate of Y end point or one which is parallel to Y;

V: different value of Y absolute coordinate;

Z: Z absolute coordinates of end point;

Zp: absolute coordinate of Z end point or one which is parallel to Z;

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

C: C absolute coordinate of end point;

H: different value of C absolute coordinate between end point and starting point;

A: A absolute coordinate of end point;

B: B absolute coordinate of end point;

F: cutting feedrate.

IP: it is the combination of axes to execute the data provided by G command, the later specified address is valid when the absolute address and relative address of one axis are defined and are in the same block to be edit. The range of each axis in corresponding parameter is as follows:

Table 3-3

Parameter setting			Least command unit	Range
Rotary axis		ISB	0.001deg	-99999.999~99999.999 deg
		ISC	0.0001deg	-9999.9999~9999.9999 deg
Linear axis	Metric	ISB	0.001mm	-99999.999~99999.999 mm
		ISC	0.0001mm	-9999.9999~9999.9999 mm
	Inch	ISB	0.0001inch	-9999.9999~9999.9999 inch
		ISC	0.00001inch	-999.99999~999.99999 inch

3.2 Rapid traverse movement G00

Command format: G00 IP__; it can be omitted to G0

Command function: the tool rapidly traverses to the position specified by the absolute or incremental command in the workpiece coordinate system, and the system sets the modal value of G command in 01 group to G00; the motion path is as Fig. 3-1, 3-2.

Command specification: G00 is initial mode;

IP: IP is the combination of X_Y_Z_. It is the coordinates of end point in the absolute command and the tool movement distance in the incremental command. The latter specified command is value when the absolute and the incremental command of one axis are in the same block. Ranges of each command value are as follows:

Incremental system	Linear axis in metric input (mm)	Linear axis in inch input(inch)	Rotary axis input
ISB system	-99999.999 ~ 99999.999mm	-9999.9999 ~ 9999.9999 inch	-99999.999 ~ 99999.999deg
ISC system	-9999.9999 ~ 9999.9999 mm	-999.99999 ~ 999.99999 inch	-9999.9999 ~ 9999.9999deg

When the axis address word is not input, the system defaults the axis does not move; when the system specifies G00 instead of any axes, they do not move, and the system sets the modal in Group 01 to be G00.

Command path:

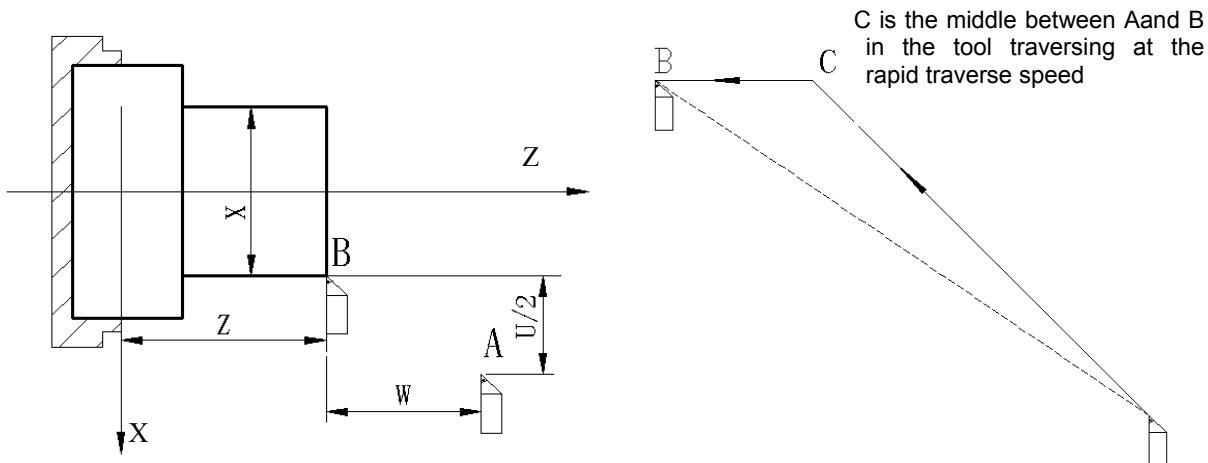
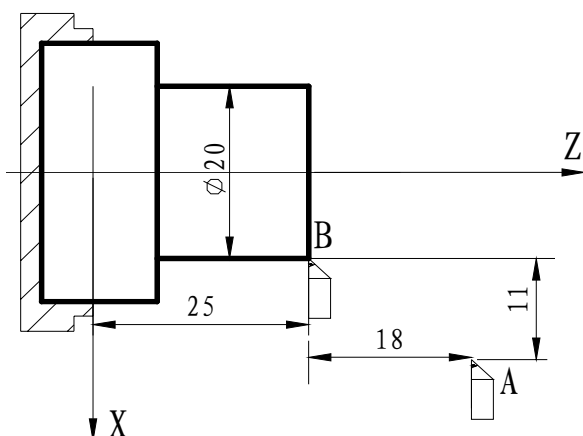


Fig.3-1 linear interpolation positioning

Note:

- 1) The rapid traverse speed in G00 is set by No. 1420, is not related to the feedrate F in the block.
- 2) No.3402 Bit 0(G01) determines the initial mode of 01group to be G00 or G01;

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



Program:

```
G0 X20.0 Z25.0; ( absolute programming )
G0 U-22.0 W-18.0; ( incremental programming )
G0 X20 .0 W-18.0; ( compound programming )
G0 U-22 .0 Z25.0; ( compound programming )
```

Fig. 3-2

3.3 Linear interpolation G01

Command format: G01 IP__ F__; it can be omitted to G1

Command function: The movement path is a straight line from starting point to end point, and the system sets G command in Group 1 sets to G01. The path is as Fig.3-3, Fig. 3-4.

Command explanation: G01 is modal;

IP_: it is coordinates of end point in the absolute value command, and it is the distance of the tool movement in the incremental value command.

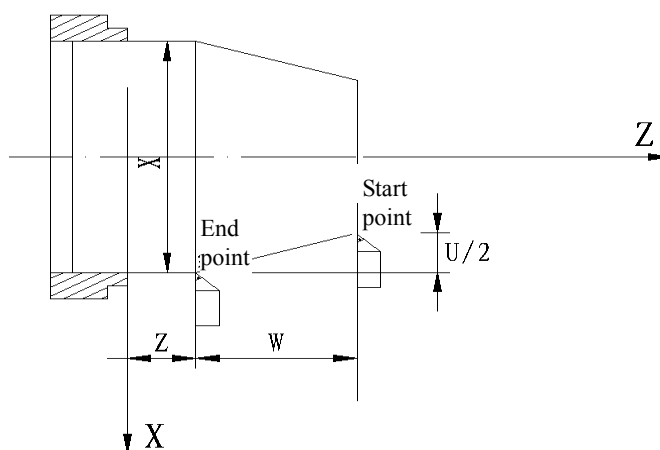
The system defaults the axis does not move when the input axis word in the command is omitted; when only G01 is specified and any axes are not specified, they do not move and the modal in Group is set to G01.

F_: tool federate, the actual cutting federate is the product between the feedrate override and F value.

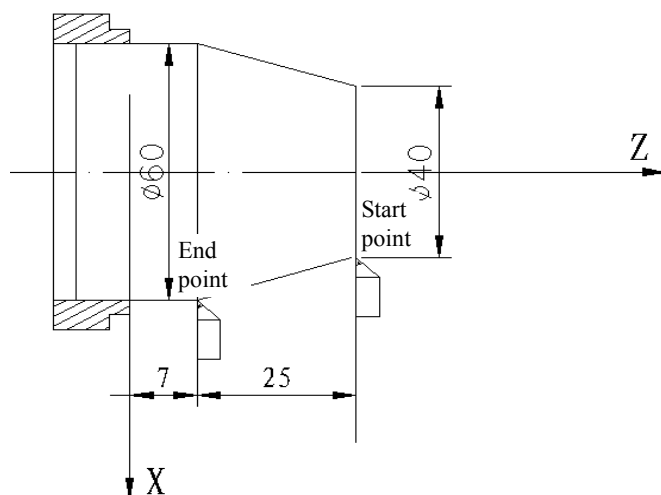
The federate specified by F is always valid till the new is specified, so it is not necessary to specify F value for each. F value range is as follows:

Feed mode		metric (mm) input	Inch (inch)input
G98	ISB system	1 ~ 60000 mm/min	0.01 ~ 2400 inch/min
	ISC system	1 ~ 24000 mm/min	0.01 ~ 960 inch/min
G99	ISB system	0.01 ~ 500mm/r	0.01 ~ 9.99inch/r
	ISC system	0.01 ~ 500mm/r	0.01 ~ 9.99 inch/r

Command path:



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



```

0 Z7 F500; (Absolute programming)
0 W-25;    (Incremental programming)
0 W-25;    (Compound programming)
0 Z7;      (Compound programming)

```

Fig. 3-4

3.4 Arc interpolation G02, G03

Command format:

$$G17 \left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} X_p - Y_p - \left\{ \begin{matrix} R - \\ I - J - \end{matrix} \right\} F -$$

$$G18 \left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} X_p - Z_p - \left\{ \begin{matrix} R - \\ I - K - \end{matrix} \right\} F -$$

$$G19 \left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} Y_p - Z_p - \left\{ \begin{matrix} R - \\ J - K - \end{matrix} \right\} F -$$

Command function:

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

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

Command path:

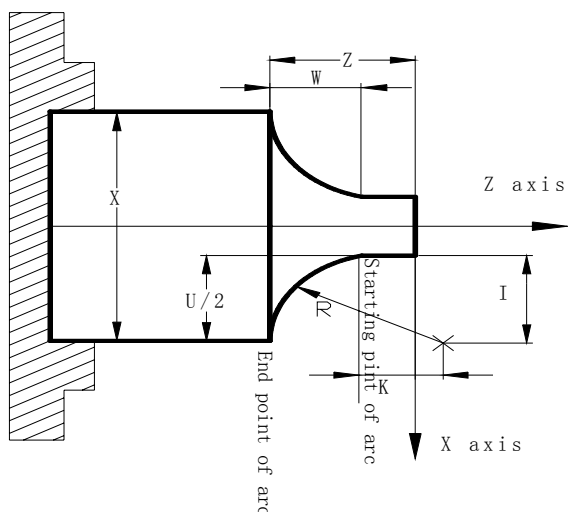


Fig. 3-5 G02 path

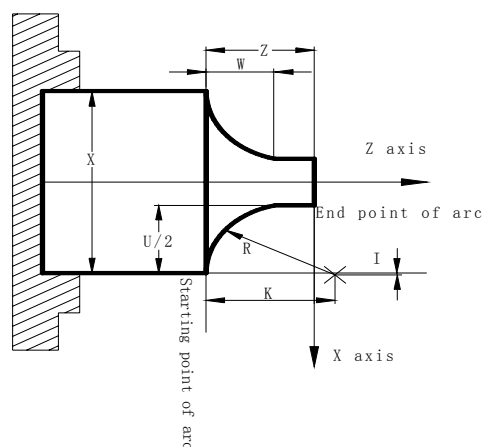


Fig. 3-6 G03 path

Command specification:

G17 specifies XpYp plane arc

G18 specifies ZpYp plane arc

G19 specifies YpZp plane arc

G02, G03 are modal

Xp_: command value of X or its parallel axis. It can be specified by the incremental command (U_) when it is the basic axis, its range is the same that of G00;

Yp_: command value of Y or its parallel axis. It can be specified by the incremental command (V_) when it is the basic axis, its range is the same that of G00;

Zp_: command value of Z or its parallel axis. It can be specified by the incremental command (W_) when it is the basic axis, its range is the same that of G00;

I_: Xp distance from starting point to arc center, with sign symbols, radius value and

its value range is as the following table;

J_: Yp distance from starting point to arc center, with sign symbols, radius value and its value range is as the following table;

K_: Zp distance from starting point to arc center, with sign symbols, radius value and its value range is as the following table;

R_: arc radius, and its value range is as the following table;

F_: feedrate along the arc, and its value range is the same that of the description in G01.

Address	Incremental system	Metric (mm) input	Inch(inch) input
I, J,	ISB system	-99999.999~99999.999mm	-9999.9999~9999.9999 inch
K, R	ISC system	-9999.9999~9999.9999 mm	-999.99999~999.99999 inch

Arc center is specified with I, J,K, which separately corresponds to vector weight of Xp, Yp, Zp from arc starting point to cycle center, and is incremental value; as Fig. 3-7.

I=circle center coordinates Xp(radius value)—arc starting point Xp (radius value) coordinate;

J=circle center coordinates Yp(radius value)—arc starting point Yp (radius value) coordinate;

K=circle center coordinates Zp(radius value)—arc starting point Zp (radius value) coordinate;

I, J, K have sign symbols according their directions, they are positive when their directions are the same those of Xp, Yp, Zp, otherwise, they are negative.

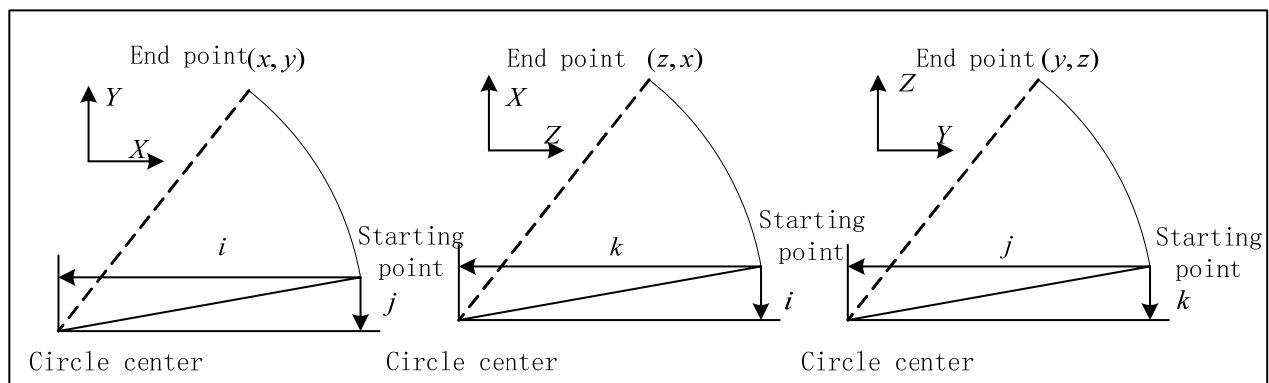


Fig. 3-7

Arc direction: the clockwise(G02) and the counterclockwise(G03) is from Zp coordinate(Yp or Xp) positive direction to negative direction to observe XpYp plane(ZpXp plane or YpZp) in the rectangular coordinate system. Taking example of G18 plane ZX, G02, G03 directions are as Fig.3-8:

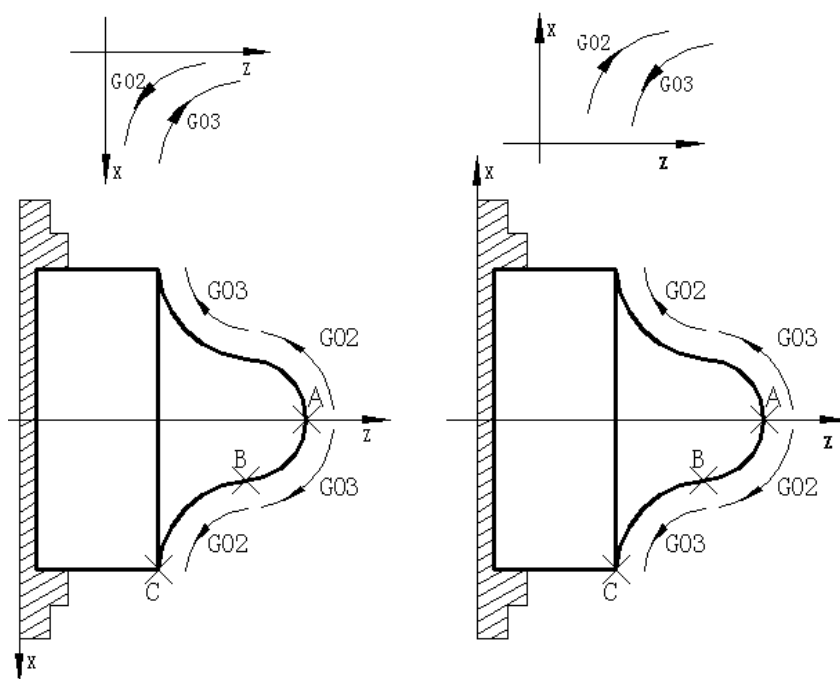


Fig. 3-8

Notes:

- (1) The system can omit one or all of X_p , Y_p , Z_p . Omitting one expresses the coordinates of the starting point are consistent with those of the end point; omitting all expresses the end point and the starting point are in the same position.
- (2) When $I = 0$, $J = 0$, $K = 0$, they can be omitted; when I , J , K , R are 0, the system executes the linear movement to the end point or alarms according to the setting value of No. 3403 Bit 5(CIR).
- (3) When I and J , J and K , I and K with R are input simultaneously, R is valid and I , J , K are invalid.
- (4) When the starting point and the end point are the same one, the path executed by G02/G03 with the circle center I, K is a full circle (360°); the circle specified R is 0° .
- (5) It is more than or less than 180° , and it is more than 180° arc when R is negative; it is less than or equal to 180° when R is positive.
- (6) In I , J , K programming, the system alarms when the radius difference between the starting point and end point of arc exceeds the setting value(except for 0) of No. 3410. when the difference does not exceed the setting value or the setting value is 0, the tool firstly executes the arc interpolation along the radius value between the arc and the center, and traverse linearly to the end point; in using R programming, R should be equal to or more than the half between the starting point and the end point; when the end point is not in the arc defined by R , the user can set whether the system alarms according to No. 3403 Bit4 (RER). It is suggested that the user should use R programming.
- (7) The system alarms when the other axes exceeding the current plane are commanded in G02/G03.

Example: command for arc cutting from $\Phi 45.25$ to $\Phi 63.06$ as Fig. 3-9.

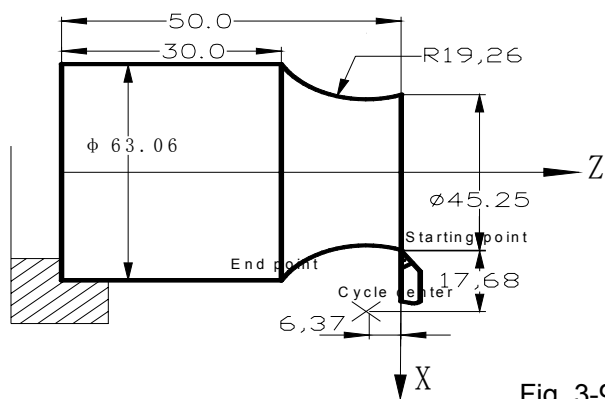


Fig. 3-9

Program:

G02 X63.06 Z-20.0 R19.26 F300 ; or
 G02 U17.81 W-20.0 R19.26 F300 ; or
 G02 X63.06 Z-20.0 I17.68 K-6.37 ; or
 G02 U17.81 W-20.0 I17.68 K-6.37 F300.

G02/G03 compound programming example:

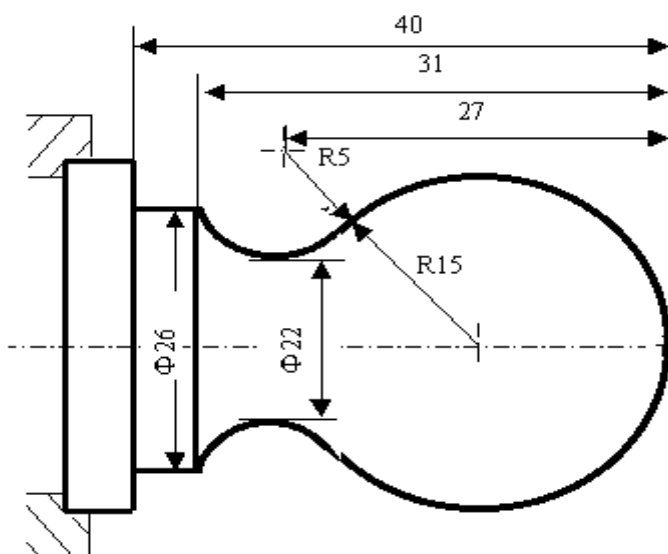


Fig. 3-10 Arc programming example

Program: O0001

N001 G0 X40 Z5; (rapidly positioning)
 N002 M03 S200; (spindle ON)
 N003 G01 X0 Z0 F900; (approach workpiece)
 N005 G03 U24 W-24 R15; (cut R15 arc)
 N006 G02 X26 Z-31 R5; (cut R5 arc)
 N007 G01 Z-40; (cutting $\phi 26$)
 N008 X40 Z5; (return to starting point)
 N009 M30; (end of program)

3.5 Dwell G04

Command format: G04 P__ ; or
 G04 X__ ; or
 G04 U__ ; or
 G04;

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

Command specification: G04 is non-modal.

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

X, U value can specify the decimal.

P value cannot have the decimal, otherwise, the system alarms.

Time of P__, X__ or U__ is as follows:

Address	P		U	X
Unit	DWT=1	0.001s		second
	DWT=0	ISB	0.001s	
		ISC	0.0001s	

Note: DWT is the setting value of No. 1015 Bit 7(DWT).

Value range of P__, X__ or U__ is as follows:

Address	Incremental system	Metric input	Inch input
X, U	ISB system	-99999.999~99999.999	-9999.9999~9999.9999
	ISC system	-9999.9999~9999.9999	-999.99999~999.99999
P	ISB, ISC	0~999999999	0~999999999

Note:

- (1) The system exactly stop a block when P, X, U are not input or P, X, U specify negative values.
- (2) X, U can command the negative value. The absolute value is taken as dwell time in G04, but the address P cannot command the negative value.
- (3) P time unit is set by No. 1015 Bit 7(DWT).
- (4) P, X, U are in the same block, P is valid; X, U are in the same block, the later specified command is valid.
- (5) The dwell can be executed after the current delay time is completed in executing the feed hold in G04.
- (6) When G04 and subprogram M98 /M99 P__are in the same block, the number following P is the time value of G04 dwell, and is also the message of M98/M99, i.e. subprogram skip message error.
- (7) G04 and the interpolation command in Group 1(such as G00, G01) are in the same block, G04 is valid, G0, G01 only change the modal value of G commands in Group 1.
- (8) When No.3403 Bit 6(AD2) is 0, G04 and G commands in Group 00 are in the same block, and the following specified command is valid.

3.6 Polar Coordinate Interpolation G12.1, G13.1

Command format: G12.1; enter the polar coordinate interpolation mode, written to G112;

-----;

-----;

G13.1; cancel the polar coordinate interpolation mode, written to

G113;

Command function: the contour is controlled by the programming command in the rectangle coordinate system being switched into one linear motion (tool motion) and one turn motion (workpiece turn motion). The function is used to end face cutting.

Command explanation: G12.1: ensure it is valid in activating the polar coordinate interpolation mode; it

can be written to G112;

G13.1: cancel the polar coordinate interpolation mode; it can be written to G113;

G12.1: activate the polar coordinate interpolation mode and select a polar coordinate interpolation plane, and the polar coordinate interpolation is completed in the plane.

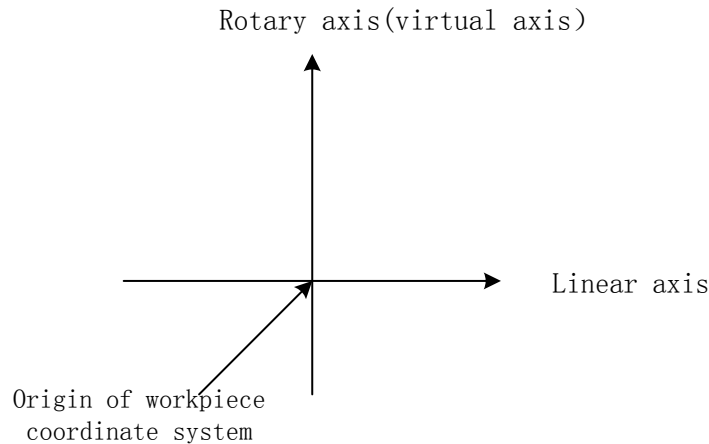


Fig. 3-11

Notes:

- (1) When the system is turned on or resets, the polar coordinate interpolation is cancelled(G13.1); G12.1 and G13.1 are modal;
- (2) The linear axis and turn axis for the polar coordinate interpolation must be set in advance in NO.5460, NO.5461; the axis undefined by the parameter does not execute the polar coordinate interpolation in spite of specifying the movement value in the polar coordinate interpolation mode;
- (3) The used plane (selected by G17, G18 or G19) before G12.1 is cancelled; after G13.1 cancels the polar coordinate interpolation, the plane recovers; when the system resets, the polar coordinate interpolation is cancelled and the system uses the plane selected by G17, G18 or G19;
- (4) In the polar coordinate interpolation mode, the program commands uses the rectangular coordinate command in the polar coordinate plane. The linear axis in the plane uses the diameter or radius programming and the turn axis uses the radius programming;
- (5) G codes in the polar coordinate interpolation mode can be used as follows:
 G01: linear interpolation;
 G02, G03: arc interpolation;
 G04: dwell;
 G40, G41, G42: tool nose radius compensation;
 G65, G66, G67: user macro program command;
 G98, G99: feed/rev, feed/minute;
 The system alarms when other G commands are executed in the polar coordinate interpolation mode.
- (6) F federate is the tangent speed with the polar coordinate interpolation plane(rectangular coordinate system) in the polar coordinate interpolation mode;
- (7) The arc interpolation commanding the arc radius address is determined by the linear axis of the interpolation plane in the polar coordinate interpolation plane as follows:
 Use I and J when the linear axis is X or its parallel and the turn axis uses J;
 Use J and K when the linear axis is Y or its parallel and the turn axis uses K;
 Use K and I when the linear axis is Z or its parallel and the turn axis uses I;

- (8) Must set a workpiece coordinate system before using G12.1, the center of the turn axis is the origin of the coordinate system. The coordinate system must not be changed in G12.1 mode.
- (9) cannot start or cancel the polar coordinate interpolation mode; command G12.1 or G13.1 in G40; otherwise, the system alarms;
- (10) When the tool traverses near to the workpiece center in the polar coordinate interpolation mode, C weight of federate changes, which exceeds max. C cutting speed to cause the system alarms.
- (11) The program command uses the rectangular coordinate command in the polar coordinate plane. The axis address of the turn axis is taken as the one of the 2nd axis(imaginary axis) in the plane. When the system executes G12.1, the tool position of the polar coordinate interpolation starts from the angle 0. So, the spindle must be positioned before the polar coordinate interpolation is executed.
- (12) The current position displays the actual coordinates in the polar coordinate interpolation. However, the remainder distance is displayed according to the coordinates in the polar coordinate interpolation plane(rectangular coordinate plane).
- (13) Must not switch the spindle gear in the polar coordinate interpolation. The system must be in the spindle speed control mode when the gear shifting is needed.

Example: Program for the polar coordinate interpolation based on X (linear axis) and C (turn axis)

X is used to the diameter programming and C is to the radius programming.

```

O0001;
N10 T0202
...
N100 G00 X150 C0 Z0;
N110 G12.1;
N120 G42 G01 X80 F200;
N130 C20;
N140 G03 X40 C40 R20;
N150 G01 X-40;
N160 G03 X-80 C20 R20;
N170 G01 C-20;
N180 G03 X-40 C-20 R20;
N190 G01 X40;
N200 G03 X80 C-20 R20;
N210 G01 C0;
N220 G40 X150.0;
N230 G13.1;
N240 Z100.0
...
N500 M30
    
```

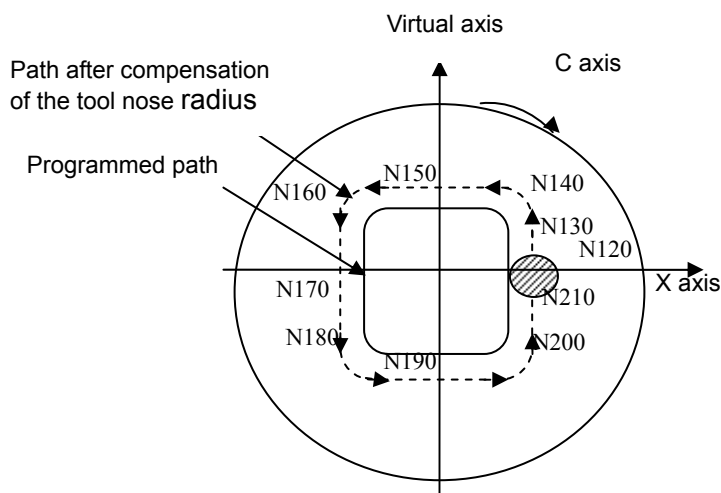


Fig. 3-12

3.7 Metric/Inch Switch G20, G21

Command format: G20; inch input
G21; metric input

Command function: realize the metric/inch switch of the system input mode.

Command explanation: G20/G21 is modal in Group 6, and can be set to the initial mode by No.0000 BIT2 (INI);
G20/G21

The units of the following value will change after they switch between the metric and the inch.

- F federate;
- position command;
- zero offset of workpiece;
- tool compensation value;
- scale unit of MPG;
- movement in incremental feed.

Notes:

- (1) The initial mode of G20/G21 is set by NO. 0000 BIT2 (INI) when the system is turned on.
- (2) When G20/G21 switches the current input mode, the system must set the beginning of the program and specify in an alone block, otherwise, the system alarms.
- (3) The tool compensation value must input the incremental unit and set it again. The tool compensation value can automatically change and cannot be set again when NO.5006 Bit0 is 1.
- (4) it modifies NO.0000 Bit2 (INI) when the system executes G20/G21. the displayed mode also changes when NO.0000 Bit 2 (INI) is changed.

3.7 Stored Travel Check G22, G23

Command format: G22; stored travel 2 check is turned on

-----;
-----;

G23; stored travel 2 check is turned off

Command function: create the forbidden area of stored travel limit check and limit the tool traverse range in one area.

Command explanation: G22,G23 are modal in Group 09, and they can be set to the initial mode by No.3402 Bit7 (G23);

G22: stored travel check is turned on;

G23: stored travel check is turned off;

Positive coordinates of the stored travel area is set by No.1322;

Negative coordinates of the stored travel area is set by No. 1323;

Limit area figure: taking examples of X, Y, Z limit area are as follows. X, Y, Z are positive coordinates, I, J, K are negative.

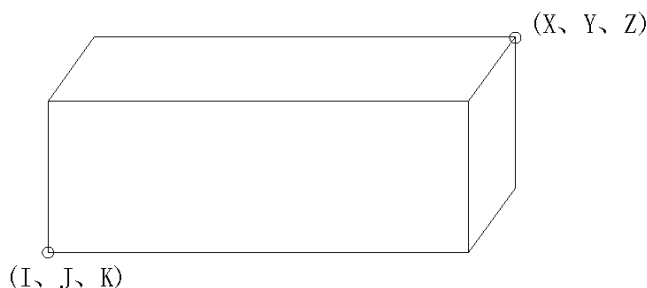


Fig. 3-13

Notes:

- (1) The initial mode of G22/G23 can be set by No. 3402 Bit 7(G23) when the system is turned on again.
- (2) G22 stored travel check is limited to the stored travel limit check 2, and the detailed is referred to **OPERATION**;
- (3) The data is set by the distance(min. command increment is taken as the unit) to the reference point when the parameter sets the top point of the forbidden area;
- (4) Whether the limit range is the inner side or outer side of the area is set by No. 1300 Bit0 (OUT) , and it is the inner side when it is set to 0;
- (5) The limit is valid after the system executes the reference point return; the system alarms when the reference point is in the limit area in G22;
- (6) The tool reversely traverses when the travel alarm appears;
- (7) G22/G23 is commanded in an alone block;
- (8) The system is switched from G23 to G22 in the forbidden area, there are as follows:
the system alarms in the next movement block when the forbidden area is in the inner side; alarms when the forbidden area is in the outer side;
- (9) When the set forbidden area is set by mistaken sequence, the system executes the area check of the two points as the top points;
- (10) When No.1310 Bit 0(OT2x) of the stored travel limit check 2 is set to 1(executing the stored travel limit 2 check), the system executes G22 and then the check; the system does not execute the check when it is G23.

3.8 Reference Point Function

3.8.1 Reference point return G28

Command format: G28 IP__ ;

Command function: move from the starting point at the rapid traverse speed to the middle position specified by IP_ and then return to the reference point; as Fig. 3-14, taking example of X-Z plane:

Command explanation: G28 is non-modal.

IP_: it is the middle point coordinates, is specified by the absolute value and incremental value. Omit one or all command address of each axis, omitting some axis means the axis does not return to the reference point, omitting all means the middle point is the tool starting point in the current workpiece coordinate system, and the tool does not return to the reference point and keeps stopping.

Command execution process: (as Fig. 3-14):

- (1) Rapidly position from the current position to the middle position of the command axis(A→B);
- (2) Rapidly position from the middle point to the reference point (B→R);

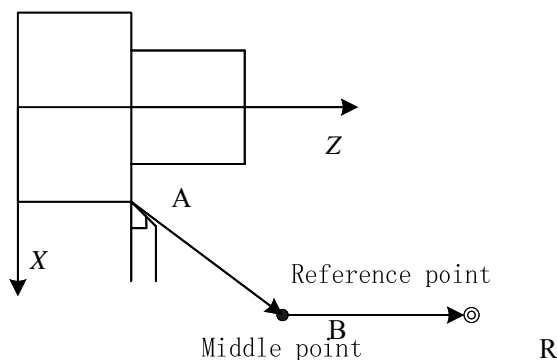


Fig. 3-14

Notes:

- (1) After the system is turned on, it does not execute the manual reference point return; when the system executes G28 reference point return, it judges it alarms or executes like the manual reference point return according to No. 1002 Bit 3(AZR) to use the deceleration block to execute the reference point return. But, when the reference point setting function without the block(No.1002 Bit1 (DLZ)) is set to 1 or NO.1005 Bit 1(DLZx) is set to 1, it is unrelated to AZR setting, the system alarms when the system executes G28 before the reference point is created.
- (2) Each axis separately moves at the rapid traverse speed from the starting point through the middle point to the reference point, i.e. G00 mode.
- (3) G28 or G30 in the tool radius compensation mode automatically cancels the tool radius compensation, and automatically recovers it in the next movement command.
- (4) The middle point will move to the new workpiece coordinate system when the workpiece coordinate system is changed.
- (5) Generally, G28 is specified in an alone line; when the system specifies simultaneously the same parameter address word of G00 or G01, IP_ is specified to G28 parameter, G00 or G01 only change the modal value of the corresponding G groups and does not execute the motion.

3.8.2 2nd, 3rd, 4th reference point return G30

Command format: G30 P2 IP__ ; return to the 2nd reference point

G30 P3 IP__ ; return to the 3rd reference point

G30 P4 IP__ ; return to the 4th reference point

Command function: move at the rapidly traverse speed to the middle point specified IP_ and then to the 2nd, 3rd and 4th reference point.

Command explanation: G30 is non-modal;

IP_: it is the middle point coordinates, is specified by the absolute value and incremental value. Omit one or all command address of each axis, omitting some axis

means the axis does not return to the reference point, omitting all means the middle point is the tool starting point in the current workpiece coordinate system, and the tool does not return to the reference point and keeps stopping.

Command execution process:

- (1) Rapidly position from the current position to the middle position of the command axis(A→B);
- (2) Rapidly position from the middle point to the reference point (B→R);

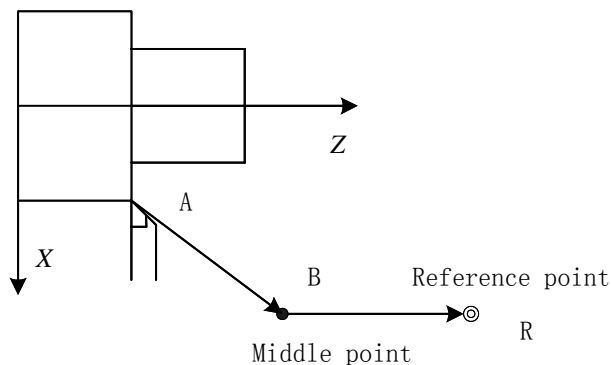


Fig.3-15

Notes:

- (1) Reference point position is set in NO.1241~NO.1243;
- (2) After the system is turned on, it executes the reference point return once before executing G30;
- (3) When P is omitted, the system executes it as P2 and returns to the 2nd reference point;
- (4) The middle point will move to the new workpiece coordinate system when the workpiece coordinate system is changed;
- (5) Each axis separately moves at the rapid traverse speed from the starting point through the middle point to the reference point, i.e. G00 mode.

3.9 Related Function of Coordinate System

The tool position is expressed with the coordinate value of the coordinate system, the coordinate value is specified by the programmed axis. GSK988T system has three kinds of coordinate system:

1. machine coordinate system
2. workpiece coordinate system
3. local coordinate system

Fig. 3-16 describes the relationship of the three coordinate systems:

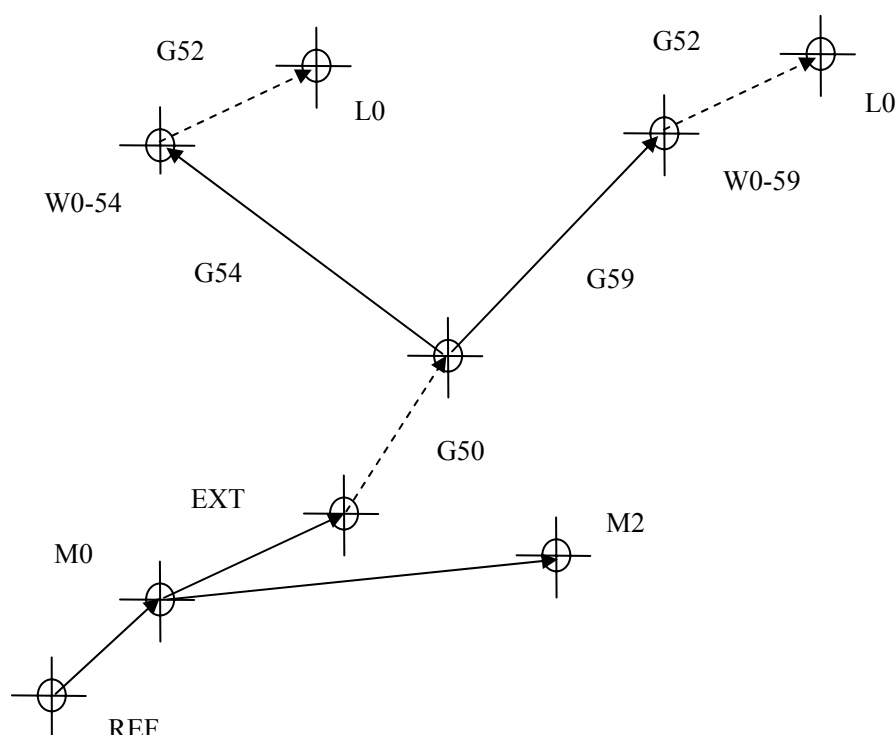


Fig. 3-16

REF: reference point ;

M0: origin of machine coordinate system is a fixed point on the machine, No. 1240 value confirms the relative position of the reference point and the machine origin;

M2: the 2nd reference point, No.1214 set the 2nd reference point position in the machine coordinate system;

EXT: the outer origin offset can be set by No. 1220 or in the coordinate setting window;

G50: the offset set by G50 is 0 when the system is turned on;

G54, G59: the offset of the workpiece coordinate system is set by No. 1221, No. 1226, and is also set in the coordinate window;

W0-54, W0-59: origin of the workpiece coordinate system;

G52: the offset of the local coordinate system is 0 when the system is switched on. All workpiece coordinate systems share, i.e. the local coordinate system offset set in one workpiece coordinate system can exist in other workpiece coordinate system.

L0: the origin of the local coordinate system.

Note: The system has created the above coordinate system after the 1st reference point return is executed. The coordinate system is created after the system is turned on with the absolute position encoder.

3.9.1 Selecting machine coordinate system position G53

A particular on the machine as the machining reference is called as the machine zero which is taken as the origin of the coordinate system is called as the machine coordinate system. After the system is turned on, executing the manual reference point return sets the machine coordinate system

which keeps till the system is turned off.

Command format: G53 IP__ ;

Command function: when the position of the machine coordinate system is commanded, the tool moves the position at the rapid traverse speed. Omitting one axis means the axis does not move; when the system only specifies G53 without specifying the positions of any axes, the system does not execute the motion.

Command explanation: G53 is non-modal;

IP_: the absolute coordinate value of each axis in the machine coordinate system must be specified by the absolute value.

As the following figure: the specified axis rapidly moves from A (20, 20) in the current workpiece coordinate system to B (-8, -10) in the machine coordinate system.

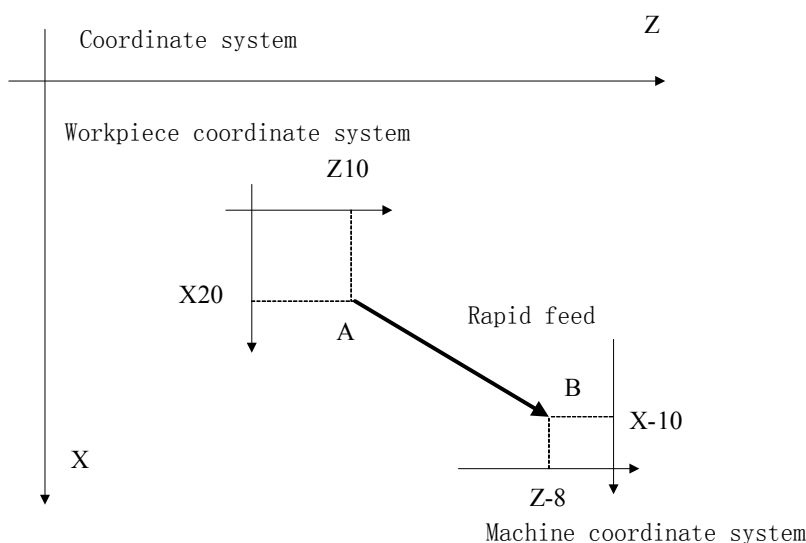


Fig.3-17

Notes:

- (1) G53 is non-modal, and is valid in other blocks;
- (2) G53 specifies the absolute position value in the machine coordinate system. The axis command is ignored when some axis uses the incremental value command;
- (3) When G53 is commanded, the system cancels the tool nose radius compensation;
- (4) After the system is turned on, the system performs the manual reference point return or G28 automatic reference point return, and automatically creates the origin position of the machine coordinate system according to the value set by No. 1240;
- (5) The machine coordinate system must be set before the system commands G53. So, the system must execute the manual reference point return or G28 automatic reference point return after it is turned on; the operation is not operated when the system uses the absolute position encoder;
- (6) The system executes G53 and G00, G01 in Group 01 in the same block, G00 or G01 only modifies G modal value in Group 01.

3.9.2 Workpiece coordinate system setting G50

The coordinate system used to machining the workpiece is called as the workpiece coordinate system.

The workpiece coordinate system can be set in advance. The set workpiece can change its origin position to set again the position of workpiece coordinate system in the machine coordinate system.

Command format: G50 IP__ ;

Command function: The absolute coordinate of the current position can be set by setting the absolute coordinate of current position to create the workpiece coordinate system (called as the floating coordinate system). After the workpiece coordinate system is created, the absolute coordinate programming inputs the coordinate value in the coordinate system till the new workpiece coordinate system in G50 is created.

Command explanation: G50 is non-modal G;

IP_: when the system uses the absolute command, it specifies the new absolute coordinate position of the current point in the coordinate system; when the system uses the incremental command, after its executes G50, the absolute coordinate value of the current point is equal to the sum between the absolute coordinate value before execution and the coordinate incremental value.

Notes:

- (1) After G50 changes the workpiece coordinate system, other workpiece coordinate system also performs the same offset;
- (2) In G50, the system can omit one or all command addresses of each axis, the current coordinate value is not input when the command value of each axis is not input. When the axis command address is omitted, the coordinate axis which is not input keeps its pervious coordinate value;
- (3) When G50 and G command (G00, G01) are in the same block, the system only modifies the modal value of Group 1, and the coordinate value in the block is specified by G50;
- (4) When the system does not set G50 offset value, it can set No. 1202 Bit(G50) to forbid G50;
- (5) In NC program, when LGT is set the coordinate offset mode to execute the tool offset, and
the system executes T function does not execute the absolute value command, the coordinate system is set by G50, the absolute coordinate value displayed by G50 is the one that the coordinate value set by G50 adding the tool compensation value which is not executed. The difference between the relative coordinates and the machine coordinates is (-80, 10) when the system executes N4, the difference value is caused because X100Z10 setting G50X20Z20 to create the workpiece coordinate system offset, i.e. the user does not think over the tool offset influence when G50 is set in NC program.

Program	Absolute coordinate	Relative coordinate	Machine coordinate
N1 T0100 G00 X100 Z10	X: 100 Z: 10	X: 100 Z: 10	X: 100 Z: 10
N2 T0101 (No.01 tool compensation value X12 Z23)	X: 88 Z: -13	X: 100 Z: 10	X: 100 Z: 10
N3 G50 X20 Z20	X: 8 Z: -3	X: 20 Z: 20	X: 100 Z: 10
N4 G00 X10 Z10	X: 10 Z: 10	X: 22 Z: 33	X: 102 Z: 23

3.9.3 Workpiece coordinate system selection command

Command format: G54 workpiece coordinate system 1;
G55 workpiece coordinate system 2;
G56 workpiece coordinate system 3;
G57 workpiece coordinate system 4;
G58 workpiece coordinate system 5;
G59 workpiece coordinate system 6;

Command function: one of G54~G59 is specified, one of workpiece coordinate system 1~6 can be selected. After the workpiece coordinate system is specified, the specified point in the block is in the specified workpiece till a new workpiece coordinate system is created as Fig. 3-18. The tool positions X60.0, Z20.0 in the workpiece coordinate system 3.

G56G00X 60.0Z 20.0

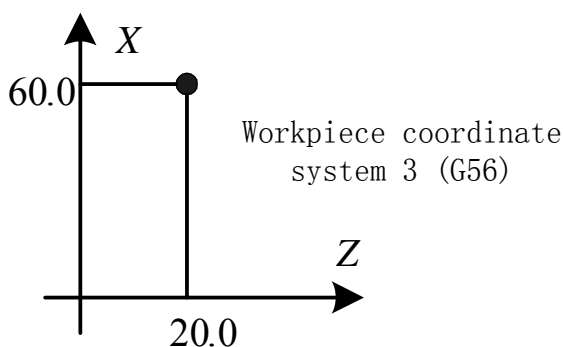


Fig. 3-18

Command explanation: G54~G59 are modal.

Notes:

- (1) The workpiece is created after the system is turned on and executes the reference point return. When the system is turned on, it automatically selects G54 as the current workpiece coordinate system;
- (2) G54-G59 describing the 6 workpiece coordinate systems can change their positions by the external workpiece zero offset value or workpiece zero offset value, and their relationship is as Fig. 3-19;

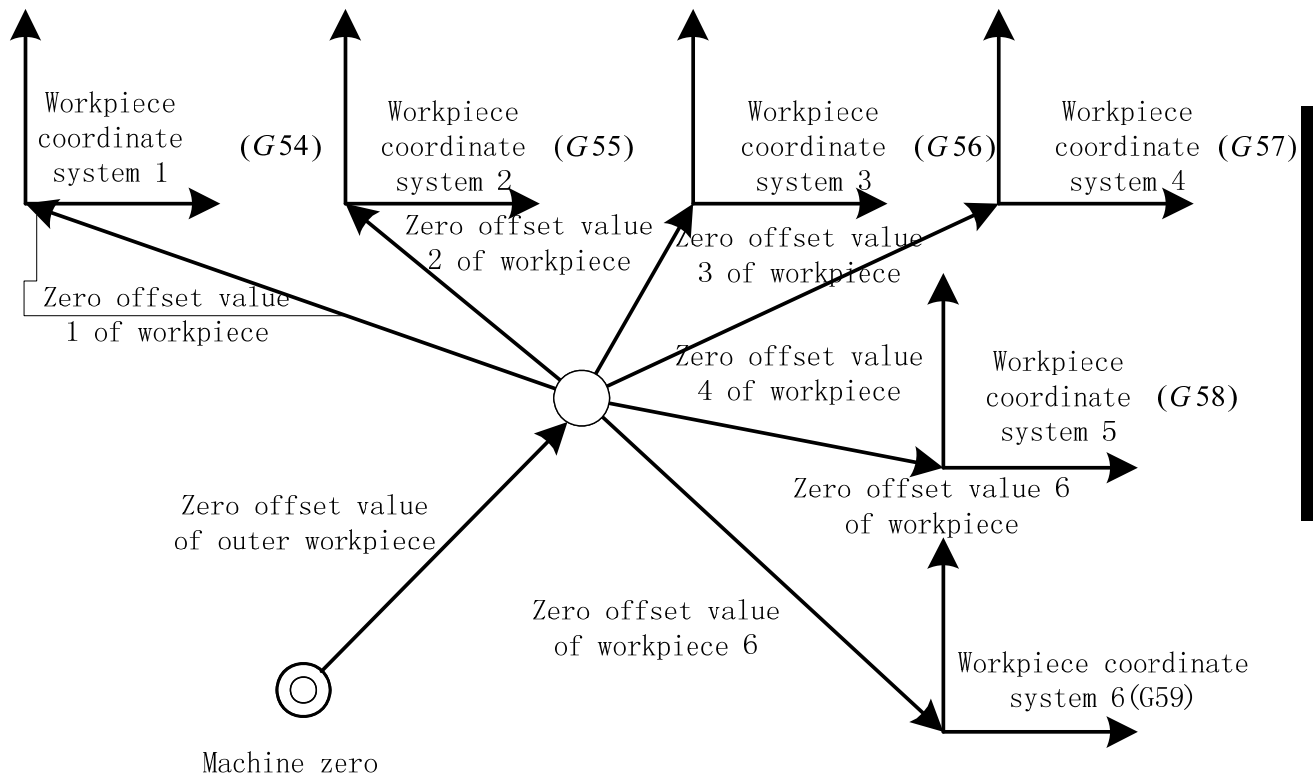


Fig. 3-19

(3) use the following method to change:

1) MDI input changes the workpiece coordinate system zero;

2) Use G50 to move the workpiece coordinate system;

Specifying G50 IP_ makes the workpiece coordinate system (G54~G59) to set a new workpiece coordinate system where the current tool position is consistent with the specified coordinates. When G50 specifies the relative value, the value adding the previous tool position coordinate value creates a new coordinate system, but the tool position does not change but the coordinate system executes the offset as Fig. 3-20:

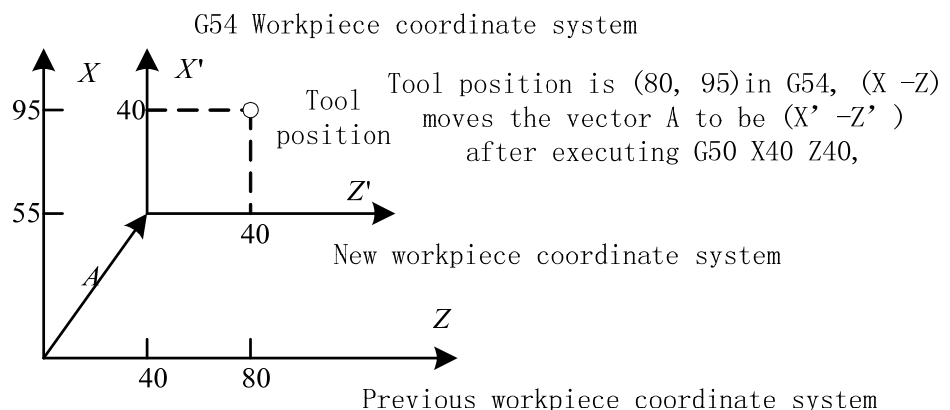


Fig. 3-20

(4) The coordinate offset value created by G50 adds to the one of all workpiece zero to make ensure that all workpiece coordinate systems offset the same value as Fig. 3-21:

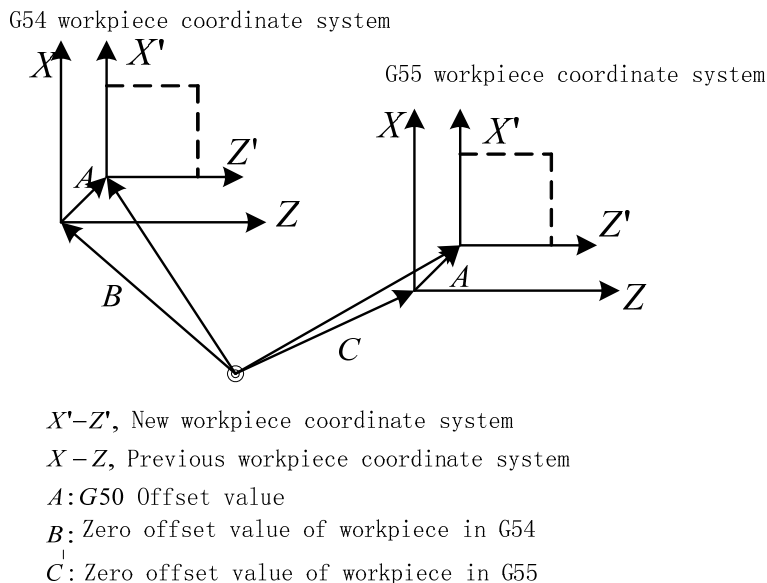


Fig. 3-21

- (5) The workpiece zero offset value of G54~G59 workpiece coordinate system can be set in the parameters and input in the coordinate setting window;
- (6) When the system is turned on, it defaults G54 as the current workpiece coordinate system; after the system executes the reference point return, it creates the coordinate system, uses G55~G59 to switch to other workpiece coordinate system; when the system resets, No.1201 Bit 7(WZR) determines whether the system returns to G54 workpiece coordinate system; when No. 3402 Bit 6(CLR) is set to 1, the modal returns to G54.

3.9.4 Local coordinate system setting G52

To be convenient to programming, the sub-coordinate system to set the workpiece coordinate system is called the local coordinate system.

Command format: G52 IP__; set the local coordinate system

.....

G52 IP0; cancel the local coordinate system (IP0 means the absolute value of each axis adds one zero)

Command function: commanding G52 in the program can set the local coordinate system in the workpiece coordinate system G54~G59. The origin of the local coordinate system can set in the position specified by IP_ in the workpiece coordinate system. The corresponding relationship is as Fig. 3-22.

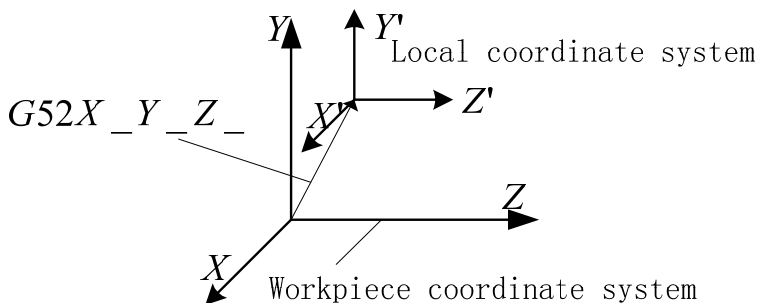


Fig. 3-22

Command explanation: G52 is non-modal;

IP_: when IP_ is absolute command, the system specifies the absolute coordinate value of origin of local coordinate system in the workpiece coordinate system; when IP_ is the incremental command, the system specifies the relative coordinate value of the origin of the local coordinate system related to the one of the workpiece coordinate system;

Once the local coordinate system is created, its coordinates are used to the axis motion command. Using G52 to command the zero of the new local coordinate system(workpiece coordinate system) can change the position of the local coordinate system;

Making the zero of the local coordinate system coincide with the one of the workpiece coordinate system can cancel the local coordinate system and returns to the workpiece coordinate system, i.e. command G52 X0 Z0.

Notes:

- (1) The local coordinate system setting does not change the workpiece coordinate system and the machine coordinate system.
- (2) Commanding G52 can temporarily cancel the offset in the tool nose radius compensation.
- (3) In local coordinate system, when G50 sets the workpiece coordinate system and the system has not specified the coordinate values to all axes in the local coordinate system, the axis which is not specified in G50 in the local coordinate system still keeps, the local coordinate system corresponding to G50 axis is cancelled; For example:

.....

G52 X50 Z50;

.....

G50 X100; at the moment, Z coordinate value is not change, the local coordinate system corresponding to X is cancelled

.....

- (4) When the system selects the workpiece coordinate system command (G54~G59) to change the workpiece coordinate system in the local coordinate system, the local coordinate system also moves to the new workpiece coordinate system.
- (5) Whether the local coordinate system in reset is cancelled is determined by No.1202 Bit 3(RLC), the local coordinate system is cancelled in reset when the parameter is set to 1.
- (6) Whether the local coordinate system in manual reference point return is cancelled is determined by No.1201Bit 2 (ZCL), the local coordinate system is cancelled in manual reference point return when the parameter is set to 1.

3.9.5 Plane selection command G17~G19

Command format: G17 selects XpYp plane;

G18 selects ZpXp plane;

G19 selects YpZp plane;

Command function: the plane selection command is used to the arc interpolation and the tool nose radius compensation selection plane. Once the system has selected the plane, it can execute the arc interpolation and tool nose radius compensation on the plane.

Command explanation: G17, G18, G19 are modal G commands.

Xp: X or its parallel axis

Yp: Y or its parallel axis

Zp: Z or its parallel axis

Xp, Yp, Zp are determined by the axis addresses of G17, G18, G19 in the block; when the axis addresses are omitted, the system defaults the omitted are the addresses of the basic axis; the plane keeps when the system does not command G17, G18, G19 blocks.

Notes:

- (1) When the system is turned on, its initialization is defaulted to G18 state, i.e. ZX plane;
- (2) When the system repetitively specifies G17~G19 in the same block, and No.3403 Bit 6(AD2) is 0, the last G17~G19 word is valid, the system alarms when the parameter is set to 1;
- (3) The multi-compound cycle command (G70~G76) and the fixed cycle command (G90, G92, G94) are used to ZX basic axis plane; when their functions are specified in other planes, the system alarms;
- (4) The motion command is not related to the plane selection, besides the arc interpolation and tool nose radius compensation command, when the system commands the axis beyond the planes, it does not alarm and the axis can move; when the system selects the axis motion beyond the plane in the arc interpolation command, the system alarms. For example:

```
.....;
G17;
G01 X100 Y50 Z20 F100; the system does not alarm, Z moves
.....;
G02 X20 Z50 R100; the system alarms
.....;
```

Example: the plane selection: when X and A are parallel axis:

```
G17 X_ Y_ ; select XY plane
G17 A_ Y_ ; select AY plane
G18 X_ Z_ ; select ZX plane
G17;      select XY plane
G17 A_    select AY plane
G18 Y_    select ZX plane, Y motion is not relative the plane
```

3.10 Fixed cycle command

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

G90: axial cutting cycle; G92: thread cutting cycle; G94: radial cutting cycle;
G92 thread cutting fixed cycle command is described in **Thread Function**.

3.10.1 Axial cutting cycle G90

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

G90 X (U) __ Z (W) __ R__ F__; (taper cutting)

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

Command specifications:

G90 is modal;

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

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

X: X absolute coordinates of cutting end point

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

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

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

R: different value (radius value) of X absolute coordinates between end point and start point of cutting.

When the sign symbols of R is not the same that of U, $R \leq |U/2|$; when $R=0$ or the input is default, the cylinder cutting is executed as Fig.3-23, otherwise, the cone cutting is executed as Fig. 3-24; unit: mm.

Address	Incremental system	metric (mm) input	Inch (inch) input
R	ISB system	-99999.999 ~ 99999.999mm	-9999.9999 ~ 9999.9999 inch
	ISC system	-9999.9999 ~ 9999.9999 mm	-999.99999 ~ 999.99999 inch

Cycle process:

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

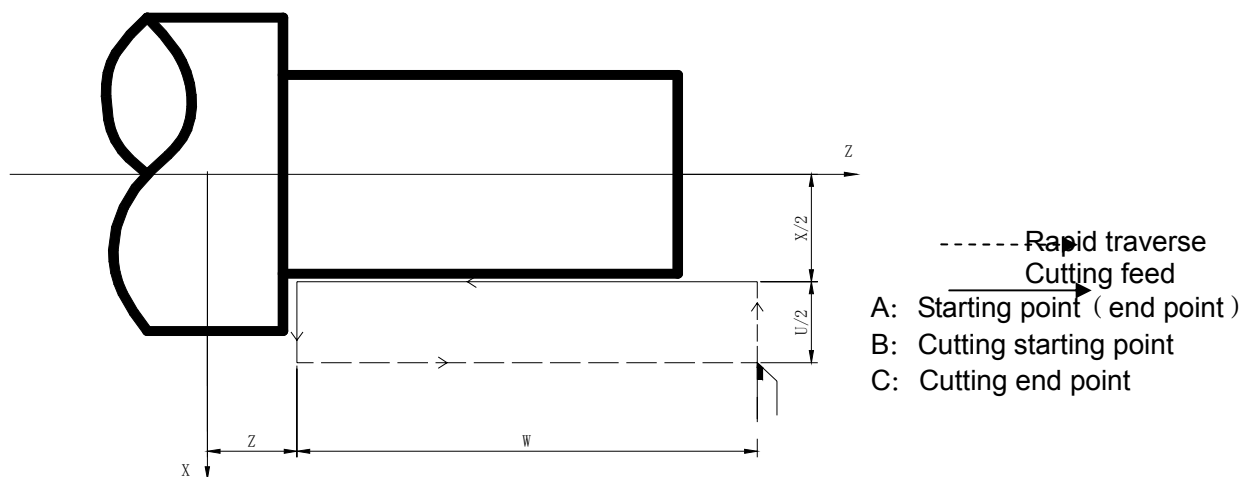
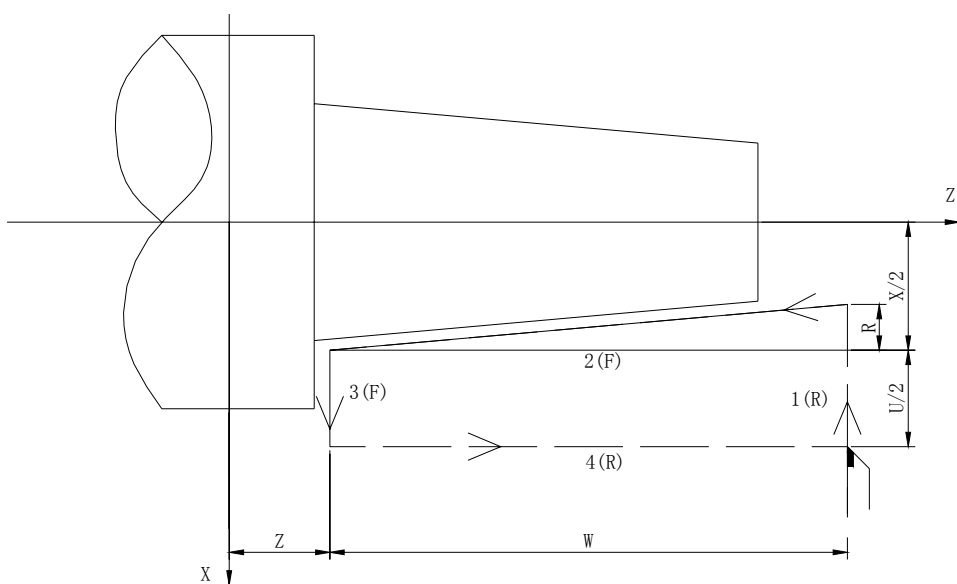


Fig.3-23



3-24

Cutting path: Relative position between cutting end point and starting point with U, W, R, and tool path of U, W, R with different sign symbols are as Fig. 3-25:

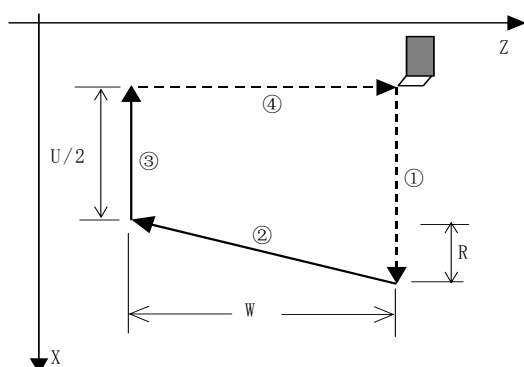
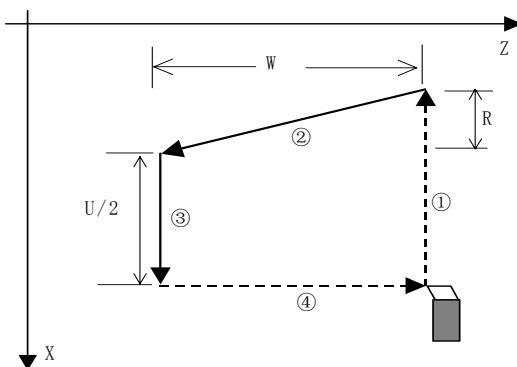
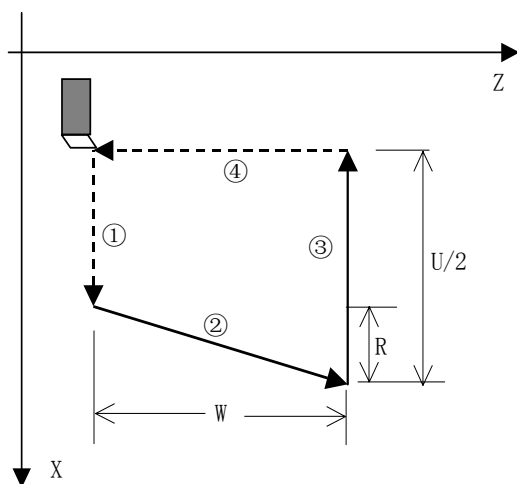
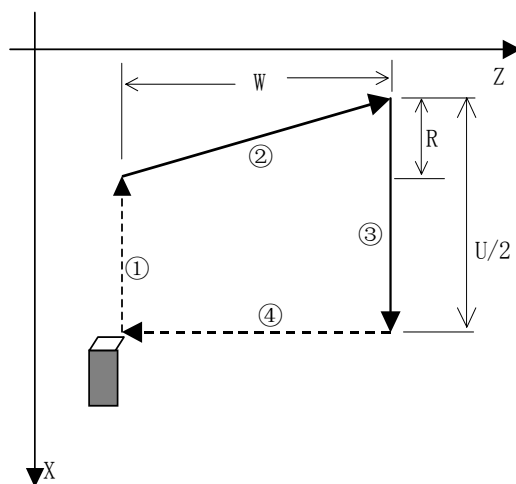
1) $U>0, W<0, R>0$ 2) $U<0, W<0, R<0$ 3) $U>0, W>0, R<0, |R| \leq |U/2|$ 4) $U<0, W>0, R>0, |R| \leq |U/2|$ 

Fig.3-25

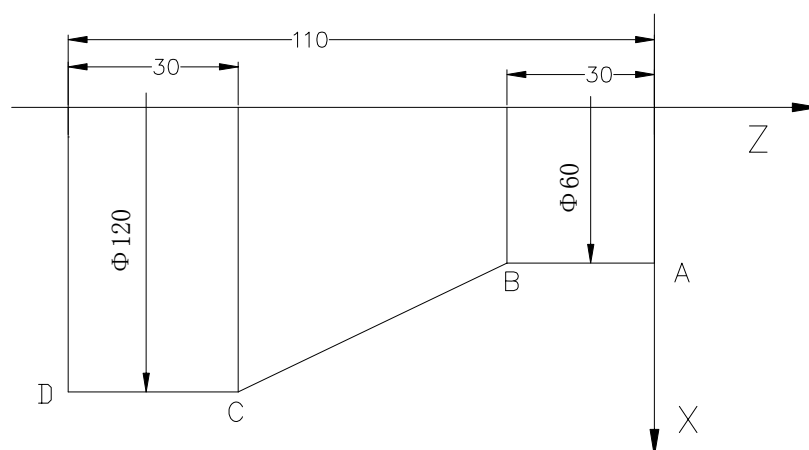
Example: Fig. 3-26, rod $\Phi 125 \times 110$ 

Fig.3-26

Program:

```

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

3.10.2 Radial cutting cycle G94

Command format: G94 X(U) __ Z(W) __ F__; (face cutting)
 G94 X(U) __ Z(W) __ R__ F__; (taper face cutting)

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

Command specifications:

G94 is modal;

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

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

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

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

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

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

R: different value(R value) of X absolute coordinates from end point to starting point of cutting.

When the sign symbols of R are not the same as those of U, R, $|R| \leq |W|$.

Radial linear cutting is as Fig. 3-27, radial taper cutting is as Fig. 3-28. Unit: mm

Address	Incremental system	Metric (mm) input	Inch (inch) input
R	ISB system	-99999.999 ~ 99999.999mm	-9999.9999~9999.9999 inch
	ISC system	-9999.9999 ~ 9999.9999 mm	-999.99999~999.99999 inch

Cycl

e process:

- ① Z rapidly traverses from starting point to cutting starting point;

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

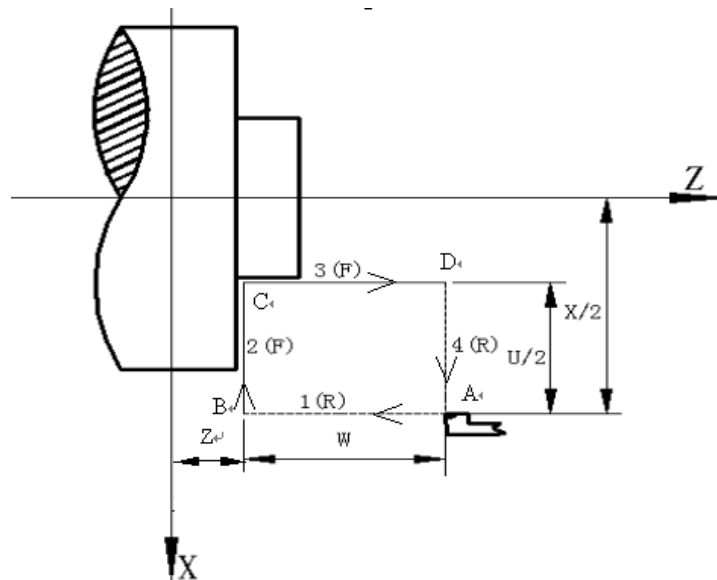


Fig.3-27

-----> Rapid traverse
 -----> Cutting feed
 A: Starting point(end point)
 B: Cutting starting point
 C: Cutting end point

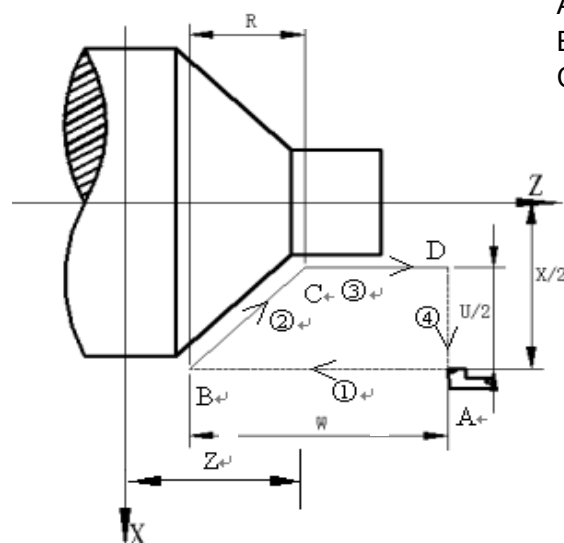


Fig.3-28

Cutting path: Relative position between cutting end point and starting point with U, W is as Fig.3-29:

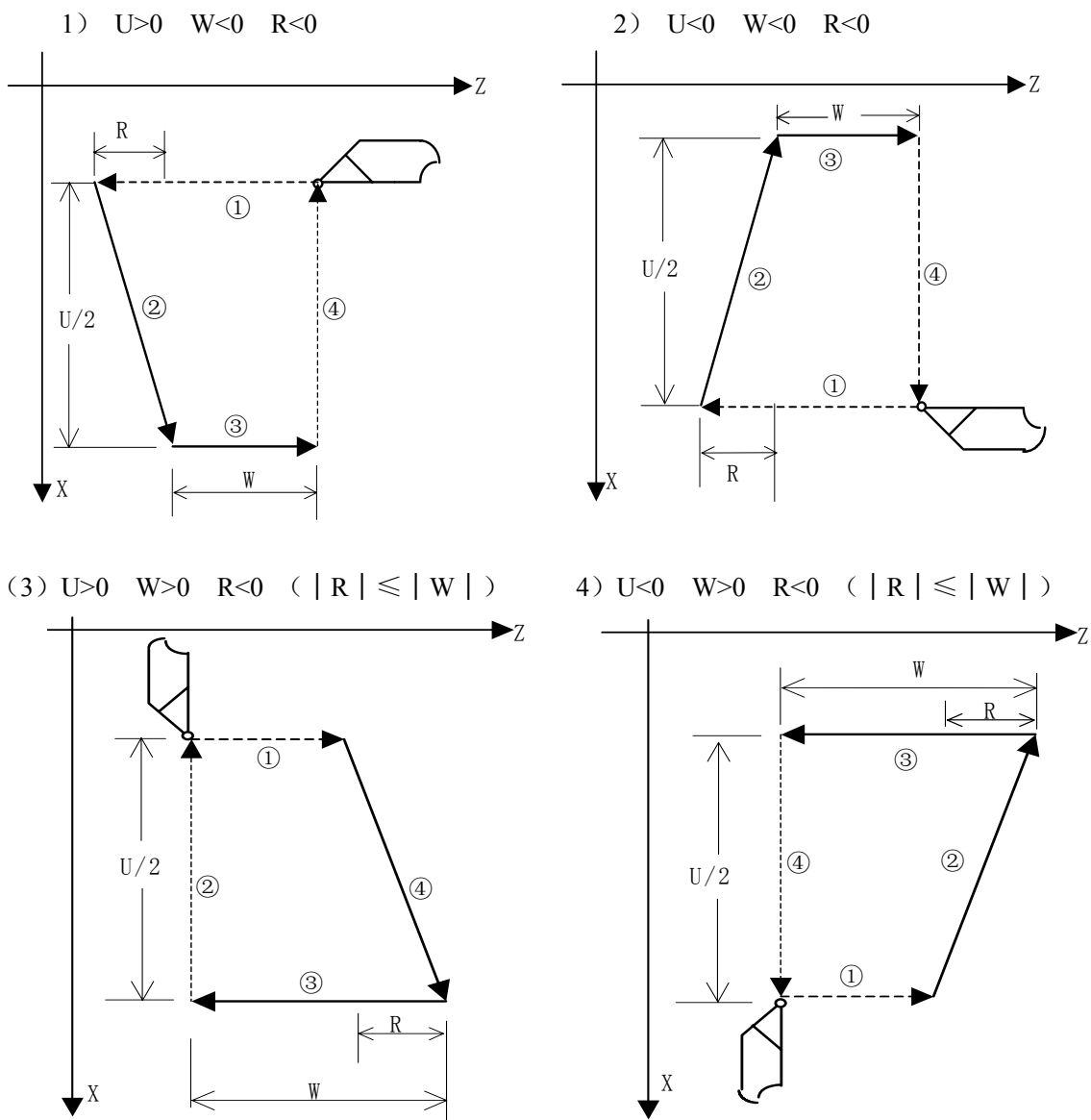


Fig. 3-29

Example: Fig. 3-30, rob Φ 125 \times 112

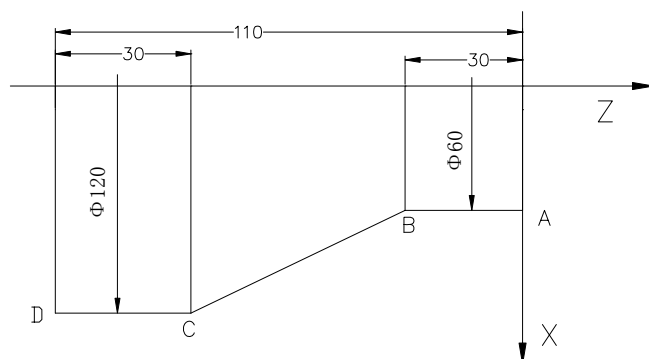


Fig.3-30

```

Program:
G00 X130 Z5 M3 S1;
G94 X0 Z0 F200          }      End face cutting
X120 Z-110 F300;         }      ( cut outer  $\Phi$ 120 )
G00 X120 Z0
G94 X108 Z-30 R-10      }
X96 R-20                }
X84 R-30                }      ( C→B→A, cut  $\Phi$ 60 )
X72 R-40                }
X60 R-50;               }
M30;

```

Notes

- (1) These fixed cycle commands are used to ZX plane. The system alarms when other axis motion in the block of the fixed cycle command is commanded;
- (2) After X(U) , Z(W) , R are executed in the canned cycle command, their command values are value if X(U) , Z(W) ,R are not redefined by executing a new canned cycle commands. The command values of X(U) ,Z(W) ,R are cleared if non-modal G command(00 Group) except for G04 or G00, G01, G02, G03, G32 is executed;
- (3) In MDI mode, the previous canned cycle can be executed by pressing the cycle start key after the canned cycle is completed;
- (4) One cycle cannot be executed repetitively in G90~G94 when the next block of G90~G94 is M, S, T command; the previous cycle is executed repetitively in G90~G94 when the next block is ended(EOB;).

Example ...

```

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

```

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

3.11 Multiple cycle commands

GSK988T multiple cycle commands include axial roughing cycle G71, radial roughing cycle G72, closed cutting cycle G73, finishing cycle G70, axial grooving multiple cycle G74, axial grooving multiple cycle G75 and multiple thread cutting cycle G76. When the system executes these commands, it automatically counts the cutting times and the cutting path according to the programmed path, travels of tool infeed and tool retraction, executes multiple machining cycle(tool infeed →cutting→retract tool→tool infeed) , automatically completes the roughing, finishing workpiece and the starting point and the end point of command are the same one.

76 multiple thread cutting cycle command is described in **Thread Function**.

3.11.1 Axial Roughing Cycle G71

Command format: G71 U (Δd) R (e) F__ S__ T__; (1)

G71 P (ns) Q (nf) U (Δu) W (Δw); (2)

N (ns) ;
 ;
 F;
 S;

 .
 N (nf) ;

(3)

Command function: G71 is divided into three parts:

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

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

Relevant definitions:

Finishing path: as Fig. 3-31, Part 3 of G71($ns \sim nf$ block) defines the finishing path, and the starting point of finishing path (starting point of ns block) is the same these of starting point and end point of G71, called A point; the first block of finishing path (ns block) is used to X rapid traversing or tool infeed, and the end point of finishing path is called to B point; the end point of finishing path (end point of nf block) is called to C point. The finishing path is A→B→C.

Roughing path: The finishing path is the one after offsetting the finishing allowance (Δu , Δw) and is the path contour formed by executing G71. A, B, C point of finishing path after offset corresponds separately to A', B', C' point of roughing path, and the final continuous cutting path of G71 is B'→C' point.

Δd : it is each travel (radius value) of X tool infeed in roughing without sign symbols, and the direction of tool infeed is defined by move direction of ns block. The command value Δd is reserved after executing U (Δd) and the value of NO.5132 is rewritten. The value of system parameter NO.5132 is regarded as the travel of tool infeed when U (Δd) is not input.

e : it is travel (radius value) of X tool retraction in roughing (radius value) without sign symbols, and the direction of tool retraction is opposite to that of tool infeed, the command value e is reserved and the value of system parameter NO.5133 is rewritten after R (e) is executed. The value of system parameter NO.5133 is regarded as the travel of tool retraction when R (e) is not input.

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

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

Δu : X finishing allowance range is as the following table (diameter) with sign symbols. X coordinate offset of roughing path compared to finishing path, i.e. the different value of X absolute coordinates between A' and A. The system defaults $\Delta u=0$ when U (Δu) is not input, i.e. there is no X finishing allowance for roughing cycle.

Δw : Z finishing allowance range is as the following table (diameter) with sign symbols. X coordinate offset of roughing path compared to finishing path, i.e. the different value of X absolute coordinates between A' and A. The system defaults $\Delta w=0$ when U (Δw) is not input, i.e. there is no Z finishing allowance for roughing cycle..

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

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

Address	Incremental system	metric (mm) input	inch(inch) input
U (Δd)	ISB system	0.001~99999.999 mm	0.0001~9999.9999 inch
	ISC system	0.0001~9999.9999 mm	0.00001~999.99999 inch
R (e)	ISB system	0~99999.999 mm	0~9999.9999 inch
	ISC system	0~9999.9999 mm	0~999.99999 inch
U (Δu)	ISB system	-99999.999~99999.999 mm	-9999.9999~9999.9999 inch
	ISC system	-9999.9999~9999.9999 mm	-999.99999~999.99999 inch
W (Δw)	ISB system	-99999.999~99999.999 mm	-9999.9999~9999.9999 inch
	ISC system	-9999.9999~9999.9999 mm	-999.99999~999.99999 inch

Execution process: as Fig. 3-31.

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

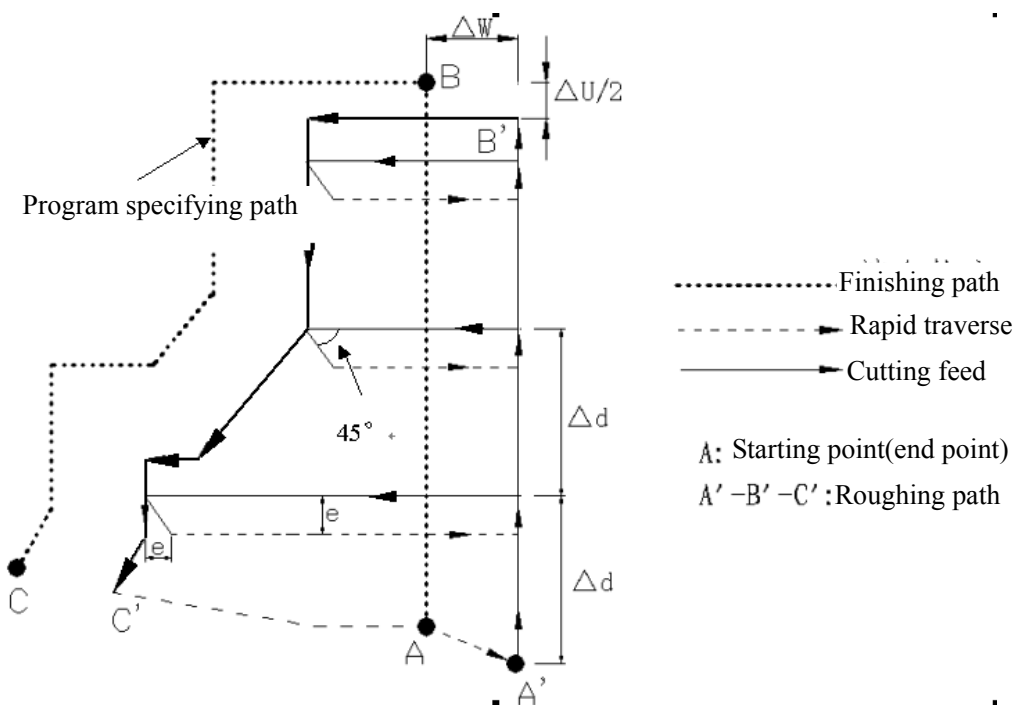


Fig. 3-31 G71 cycle path

Command specifications:

- (1) ns~nf blocks in programming must be followed G71 blocks. If they are in the front of G71 blocks, the system automatically searches and executes ns~nf blocks, and then executes the next program following nf block after they are executed, which causes the system executes ns~nf blocks repetitively;
- (2) ns~nf blocks are used to count the roughing path and the blocks are not executed when G71 is executed. F, S, T commands of ns~nf blocks are invalid when G71 is executed, at the moment, F, S, T commands of G71 blocks are valid. F, S, T of ns~nf blocks are valid when executing ns~nf to command G70 finishing cycle;
- (3) There are G00,G01 without the word Z(W) in ns block, otherwise the system alarms ;
- (4) X and Z dimensions must be changed monotonously (always increasing or reducing) for the finishing path;
- (5) In ns~nf blocks, there are only G commands: G01, G02, G03, G04, G96, G97, G98, G99, G40, G41,G42 and the system cannot call subprograms(M98/M99);
- (6) G96, G97, G98, G99, G40, G41, G42 are invalid in G71 and valid in G70, G96, G97, G98,
- (7) When G71 is executed, the system can stop the automatic run and manual traverse, but return to the position before manual traversing when G71 is executed again, otherwise, the following path will be wrong;
- (8) When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path;
- (9) Δd , Δu are specified by the same U and different with or without being specified P,Q commands;
- (10) G71 cannot be executed in MDI, otherwise, the system alarms;

Coordinate offset direction with finishing allowance:

$\Delta u, \Delta w$ define the coordinates offset and its direction of finishing, and their sign symbols are as

follows Fig. 3-32: B→C for finishing path, B'→C' for roughing path and A is the tool starting point

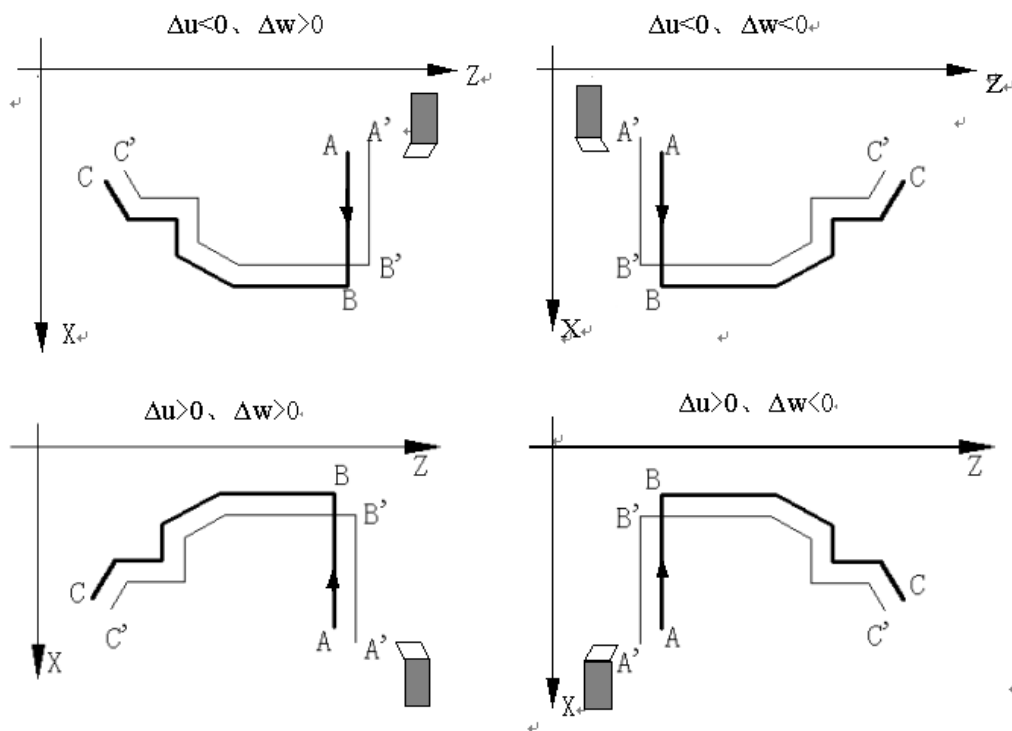


Fig.3-32

Example: Fig. 3-33

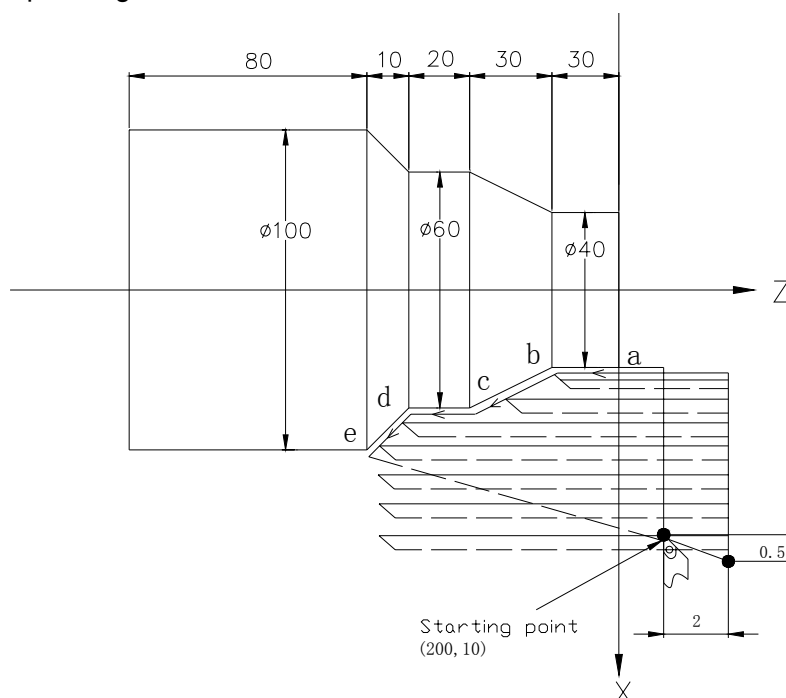


Fig. 3-33

Program: O0004;

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

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

G71 P80 Q120 U0.5 W0.2; (roughing a---e, X machining allowance 0.5mm , Z

0.2mm)

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

3.11.2 Radial Roughing Cycle G72

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

G72 P (ns) Q (nf) U (Δu) W (Δw) ; (2)

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

Command function: G72 is divided into three parts:

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

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

Relevant definitions:

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

Roughing path: The finishing path is the one after offsetting the finishing allowance (Δu, Δw)

and is the path contour formed by executing G72. A, B, C point of finishing path after offset corresponds separately to A', B', C' point of roughing path, and the final continuous cutting path of G72 is B'→C' point.

Δd : it is each travel of Z tool infeed in roughing without sign symbols, and the direction of tool infeed is defined by move direction of ns block. Δd is reserved after the system executes W (Δd) and NO.5132 value is modified. The value of system parameter NO.051 is regarded as the travel of tool infeed when W (Δd) is not input.

e: it is each travel of Z tool infeed in roughing without sign symbols, and the direction of tool retraction is opposite to that of tool infeed; after R(e) is executed, e value e is reserved and the system modifies No.5133 value. The value of system parameter NO.5133 is regarded as the travel of tool retraction when R (e) is not input.

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

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

Δu : X finishing allowance in roughing, (X coordinate offset of roughing path compared to finishing path, i.e. the different value of X absolute coordinate between A' and A, diameter value with sign symbols).

Δw : Z finishing allowance in roughing, its value: -9999.999~9999.999 (Z coordinate offset of roughing path compared to finishing path, i.e. the different value of X absolute coordinates between A' and A, with sign symbols).

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

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

Address	Incremental system	Metric (mm) input	Inch (inch) input
W (Δd)	ISB system	0.001~99999.999 mm	0.0001~9999.9999 inch
	ISC system	0.0001~9999.9999 mm	0.00001~999.99999 inch
R (e)	ISB system	0~99999.999 mm	0~9999.9999 inch
	ISC system	0~9999.9999 mm	0~999.99999 inch
U (Δu)	ISB system	-99999.999~99999.999 mm	-9999.9999~9999.9999 inch
	ISC system	-9999.9999~9999.9999 mm	-999.99999~999.99999 inch
W (Δw)	ISB system	-99999.999~99999.999 mm	-9999.9999~9999.9999 inch
	ISC system	-9999.9999~9999.9999 mm	-999.99999~999.99999 inch

Execution process :

- ① X rapidly traverses to A' from A point, X travel is Δu , and Z travel is Δw ;
- ② X moves from m A' is Δd (tool infeed), ns block is for tool infeed at rapid traverse speed with G0, is for tool infeed at G72 feedrate F in G1, and its direction of tool infeed is that of A→B point;
- ③ X executes the cutting feeds to the roughing path, and its direction is the same that of X coordinate B→C point;
- ④ X, Z execute the tool retraction e (45° straight line) at feedrate, the directions of tool retraction is opposite to that of tool infeed ;

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

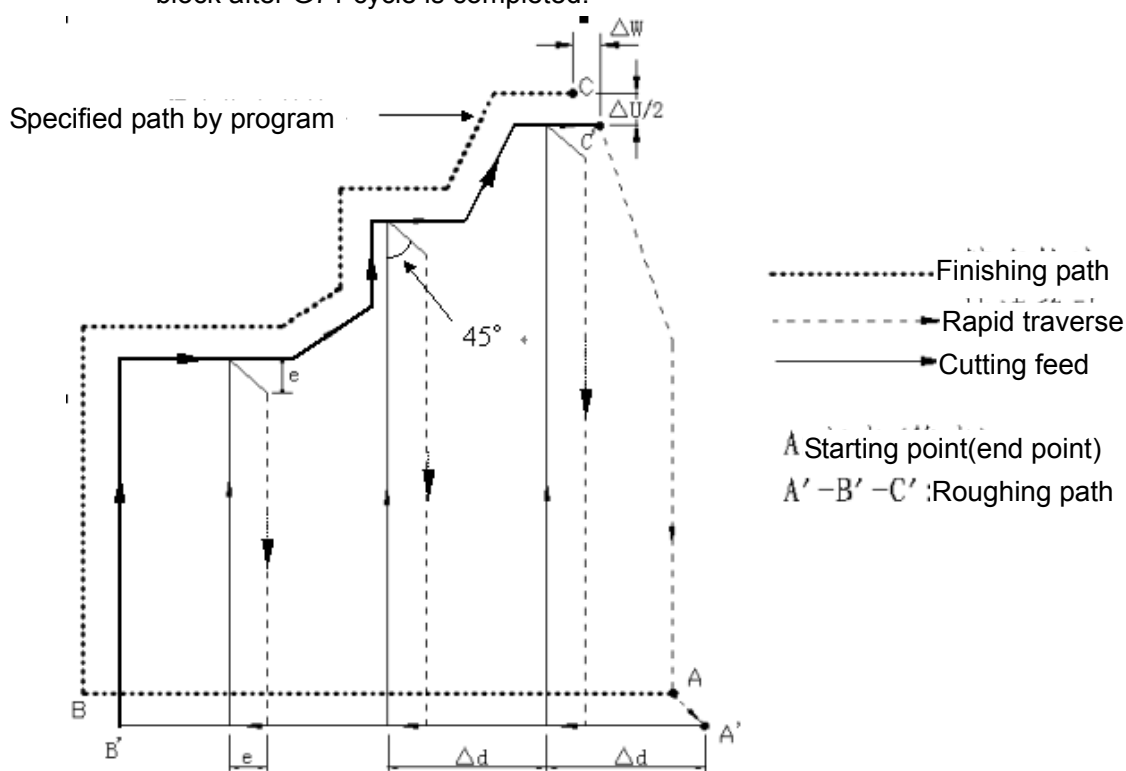


Fig. 3-34

Command specifications:

- (1) ns~nf blocks in programming must be followed G72 blocks. If they are in the front of G72 blocks, the system automatically searches and executes ns~nf blocks, and then executes the next program following nf block after they are executed, which causes the system executes ns~nf blocks repetitively.
- (2) ns~nf blocks are used for counting the roughing path and the blocks are not executed when G72 is executed. F, S, T commands of ns~nf blocks are invalid when G72 is executed, at the moment, F, S, T commands of G72 blocks are valid. F, S, T of ns~nf blocks are valid when executing ns~nf to command G70 finishing cycle;
- (3) There are G00, G01 without the word X(U) in ns block, otherwise the system alarms.
- (4) X, Z dimensions in finishing path (ns~nf blocks) must be changed monotonously (always increasing or reducing) for the finishing path;

- (5) In ns~nf blocks, there are only G commands: G01, G02, G03, G04, G96, G97, G98, G99, G40, G41, G42 and the system cannot call subprograms(M98/M99);
- (6) G96, G97, G98, G99, G40, G41, G42 are invalid in G72 and valid in G70;
- (7) When G72 is executed, the system can stop the automatic run and manual traverse, but return to the position before manual traversing when G72 is executed again, otherwise, the following path will be wrong;
- (8) When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path;
- (9) Δd , Δu are specified by the same U and different with or without being specified P,Q commands;
- (10) G72 cannot be executed in MDI, otherwise, the system alarms.

Coordinate offset direction with finishing allowance:

$\Delta u, \Delta w$ define the coordinates offset and its direction of finishing, and their sign symbols are as follows Fig. 3-35: B→C for finishing path, B'→C' for roughing path and A is the starting point.

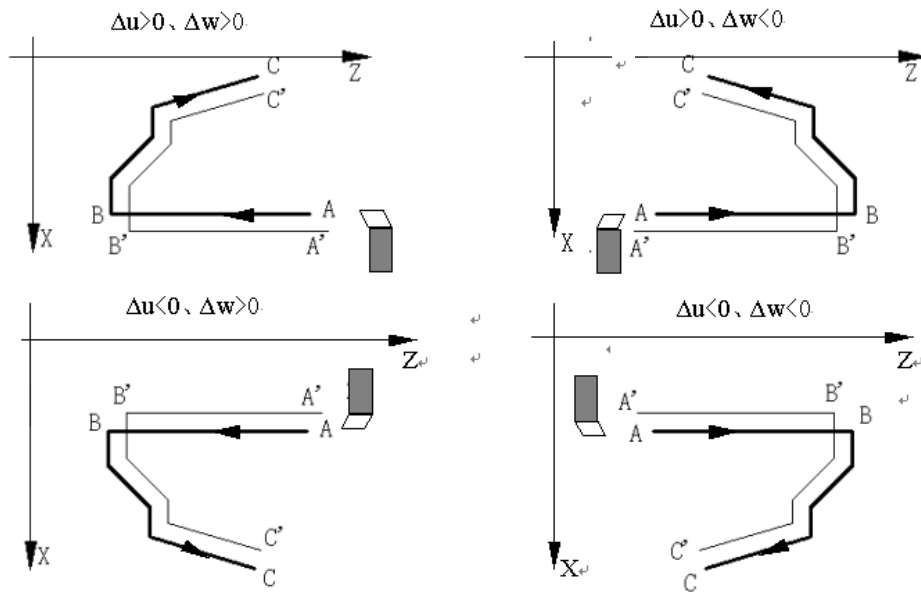


Fig.3-35

Example: Fig. 3-36

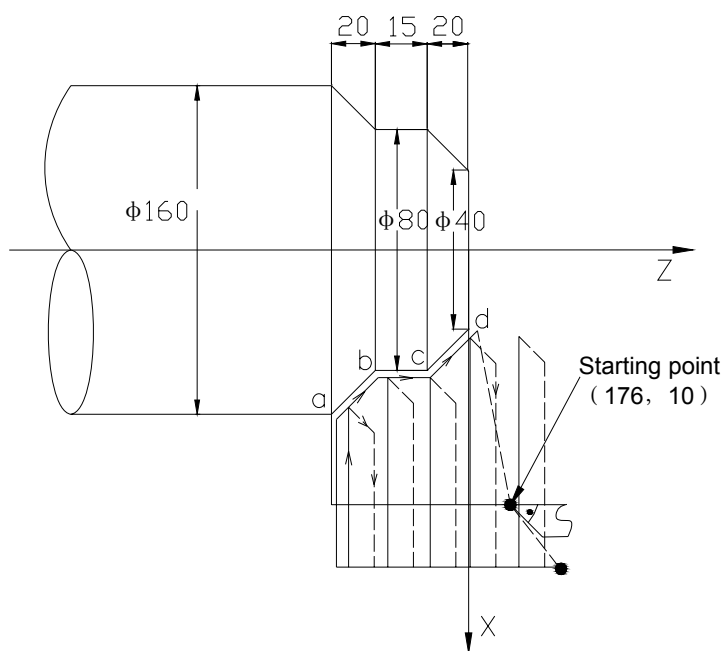


Fig.3-36

Program:

```

O0005;
G00 X176 Z10 M03 S500      ( Change No.2 tool and execute its compensation, spindle
                             rotation with 500 r/min )

G72 W2.0 R0.5 F300;        ( Tool infeed 2mm, tool retraction 2mm )
G72 P10 Q20 U0.2 W0.1;    ( Roughing a--d, X roughing allowance 0.2mm and Z 0.1mm )
N10 G00 Z-55 S800 ;       ( Rapid traverse )
G01 X160 F120;            ( Infeed to a point )
X80 W20;                 ( Machining a—b )
W15;                    ( Machining b—c )
N20 X40 W20 ;            ( Machining c—d )
G70 P050 Q090 M30;        ( Finishing a—d )
    
```

} Blocks for finishing path

3.11.3 Closed Cutting Cycle G73

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

G73 P (ns) Q (nf) U (Δu) W (Δw) ; (2)

$N_{\text{--}}(ns)$;
 ;
 F;
 S;
 ;
 .
 $N_{\text{--}}(nf)$;

(3)

Command functions: G73 is divided into three parts:

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

According to the finishing allowance, the travel of tool retraction and the cutting times, the system automatically counts the travel of roughing offset, the travel of each tool infeed and the path of roughing, the path of each cutting is the offset travel of finishing path, the cutting path approaches gradually the finishing one, and last cutting path is the finishing one according to the finishing allowance. The starting point and end point of G73 are the same one, and G73 is applied to roughing for the formed rod. G73 is non-modal and its path is as Fig.3-37.

Relevant definitions:

Finishing path: the above-mentioned Part 3 of G73($n_s \sim n_f$ block) defines the finishing path, and the starting point of finishing path (start point of n_s block) is the same these of starting point and end point of G73, called A point; the end point of the first block of finishing path (n_s block) is called B point; the end point of finishing path (end point of n_f block) is called C point. The finishing path is $A \rightarrow B \rightarrow C$.

Roughing path: It is one group of offset path of finishing one, and the roughing path times are the same that of cutting. After the coordinates offset, A, B, C of finishing path separately corresponds to A_n, B_n, C_n of roughing path (n is the cutting times, the first cutting path is A_1, B_1, C_1 and the last one is A_d, B_d, C_d). The coordinates offset value of the first cutting compared to finishing path is $(\Delta i \times 2 + \Delta u, \Delta w + \Delta k)$ (diameter programming), the coordinates offset value of the last cutting compared to finishing path is $(\Delta u, \Delta w)$, the coordinates offset value of each cutting compared to the previous one is $(\Delta i \times 2 / d - 1, \Delta k / d - 1)$.

Δi : Travel of X tool retraction in roughing is the following table (radius value with sign symbols), Δi is equal to X coordinate offset value (radius value) of A_1 point compared to A_d point. The X total cutting travel (radius value) is equal to $|\Delta i|$ in roughing, and X cutting direction is opposite to the sign symbol of Δi : $\Delta i > 0$, cut in X negative direction in roughing. It is reserved after Δi command value is executed and the system rewrites No.5135 value. NO.5135 value is regarded as the travel of X tool retraction of roughing when U (Δi) is not input.

Δk : Travel of Z tool retraction in roughing is the following table (radius value with sign symbols), Δk is equal to X coordinate offset value (radius value) of A_1 point compared to A_d point. The Z total cutting travel (radius value) is equal to $|\Delta k|$ in roughing, and Z cutting direction is opposite to the sign symbol of Δk : $\Delta k > 0$, cut in Z negative direction in roughing. It is reserved after Δk command value is executed and the system rewrites No.5136 value. NO.5136 value is regarded as the travel of X tool retraction of roughing when W (Δk) is not input.

d : It is the cutting times and its range is referred to the following table. R5 means the closed cutting cycle is completed by 5 times cutting. R (d) is reserved after it is executed and the system rewrites NO.5137. The value of system parameter NO.5137 is regarded as the cutting times when R (d is not input).

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

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

Δu : it is X finishing allowance as the following table (diameter value with sign symbols) and is the X coordinate offset of roughing contour compared to finishing path, i.e. the different value of X absolute coordinates of A_1 compared to A. $\Delta u > 0$, it is the offset of the last X positive roughing path

compared to finishing path. The system defaults $\Delta u=0$ when U (Δu) is not input, i.e. there is no X finishing allowance for roughing cycle.

Δw : it is Z finishing allowance as the following table -99.999~99.999 (unit: mm) and is the Z coordinate offset of roughing contour compared to finishing path, i.e. the different value of Z absolute coordinate of A_1 compared to A. $\Delta w>0$, it is the offset of the last roughing path compared to finishing path in Z positive direction. The system defaults $\Delta w=0$ when W (Δw) is not input, i.e. there is no Z finishing allowance for roughing cycle.

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

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

Address	Incremental system	Metric (mm) input	Inch (inch) input
U (Δi)	ISB system	-99999.999~99999.999 mm	-9999.9999~9999.9999 inch
	ISC system	-9999.9999~9999.9999 mm	-999.99999~999.99999 inch
W (Δk)	ISB system	-99999.999~99999.999 mm	-9999.9999~9999.9999 inch
	ISC system	-9999.9999~9999.9999 mm	-999.99999~999.99999 inch
R (d)	ISB, ISC	1~999 (times)	1~999 (times)
U (Δu)	ISB system	-99999.999~99999.999 mm	-9999.9999~9999.9999 inch
	ISC system	-9999.9999~9999.9999 mm	-999.99999~999.99999 inch
W (Δw)	ISB system	-99999.999~99999.999 mm	-9999.9999~9999.9999 inch
	ISC system	-9999.9999~9999.9999 mm	-999.99999~999.99999 inch

Execution process:(Fig. 3-37).

① $A \rightarrow A_1$: Rapid traverse;

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

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

$B_1 \rightarrow C_1$: Cutting feed.

③ $C_1 \rightarrow A_2$: Rapid traverse;

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

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

$B_2 \rightarrow C_2$: Cutting feed.

⑤ $C_2 \rightarrow A_3$: rapid traverse;

.....

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

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

$B_n \rightarrow C_n$: Cutting feed.

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

.....

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

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

in G1;

$B_d \rightarrow C_d$: Cutting feed.

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

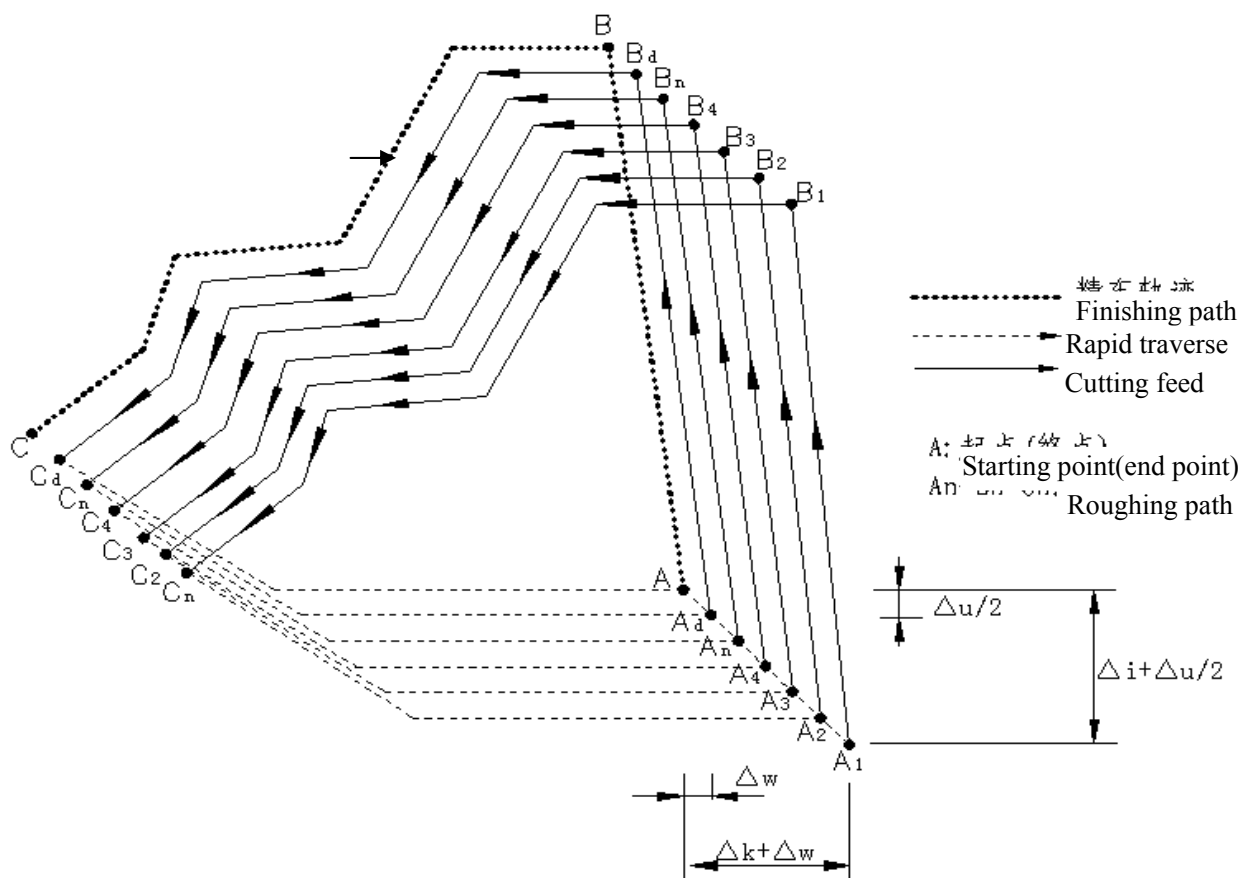


Fig. 3-37 G73 run path

Command specifications:

(1) $ns \sim nf$ blocks in programming must be followed G73 blocks. If they are in the front of G73 blocks, the system automatically searches and executes $ns \sim nf$ blocks, and then executes the next program following nf block after they are executed, which causes the system executes $ns \sim nf$ blocks repetitively.

(2) $ns \sim nf$ blocks are used for counting the roughing path and the blocks are not executed when G73 is executed. F, S, T commands of $ns \sim nf$ blocks are invalid when G71 is executed, at the moment, F, S, T commands of G73 blocks are valid. F, S, T of $ns \sim nf$ blocks are valid when executing $ns \sim nf$ to command G70 finishing cycle.

(3) There are only G00, G01 in ns block.

(4) In $ns \sim nf$ blocks, there are only G commands: G00, G01, G02, G03, G04, G96, G97, G98, G99, G40, G41, G42 and the system cannot call subprograms (M98/M99)

(5) G96, G97, G98, G99, G40, G41, G42 are invalid in G73 and valid in G70.

(6) When G73 is executed, the system can stop the automatic run and manual traverse, but return to the position before manual traversing when G73 is executed again, otherwise, the following path will be wrong.

(7) When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path.

(8) Δi , Δu are specified by the same U and Δk , Δw are specified by the same U, and they are

different with or without being specified P,Q commands.

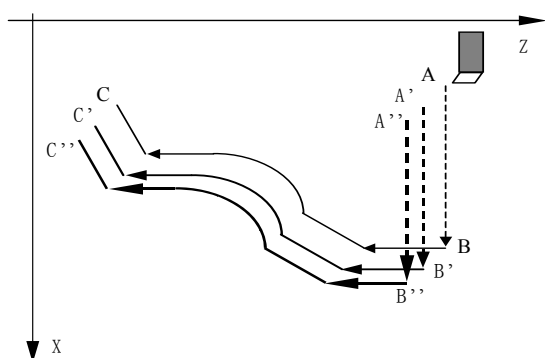
(9) G73 cannot be executed in MDI, otherwise, the system alarms.

Z must be the monotonous in the cycle body specified by P and Q. Z tool retraction and finishing allowance are set to 0 when the system executes X non-monotonous workpiece. When No. 5102 Bit0 (MRI) is set to 1, the system does not alarm.

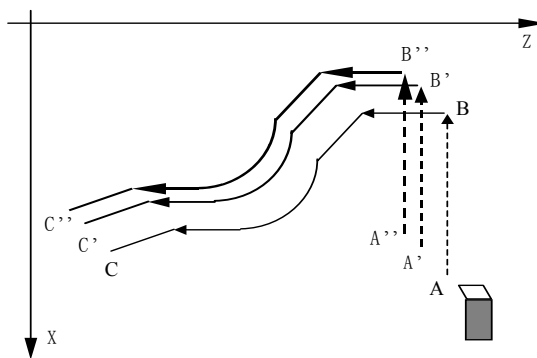
Coordinate offset direction with finishing allowance:

$\Delta i, \Delta k$ define the coordinates offset and its direction of roughing, $\Delta u, \Delta w$ define the coordinates offset and cut-in direction in finishing; $\Delta i, \Delta k, \Delta u, \Delta w$ can consist of many groups. Generally, the sign symbols of Δi and Δu are consistent, the sign symbols of Δk and Δw are consistent, there are four kinds of combination as Fig. 3-38, A for start-up tool point, B→C for workpiece contour, B'→C' for roughing contour and B''→C'' for finishing path.

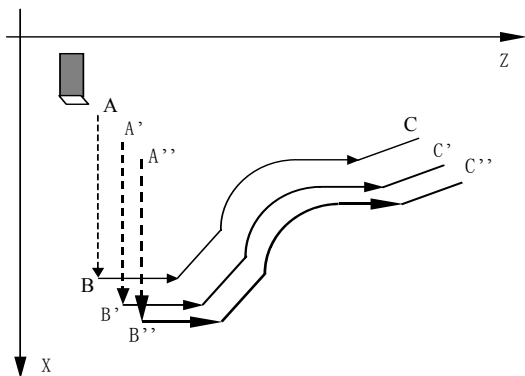
1) $\Delta i < 0, \Delta k > 0, \Delta u < 0, \Delta w > 0;$



2) $\Delta i > 0, \Delta k > 0, \Delta u > 0, \Delta w > 0;$



3) $\Delta i < 0, \Delta k < 0, \Delta u < 0, \Delta w < 0;$



4) $\Delta i > 0, \Delta k < 0, \Delta u > 0, \Delta w < 0;$

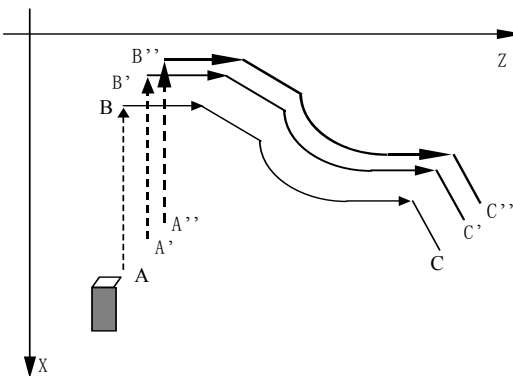


Fig.3-38

Example: Fig. 3-39

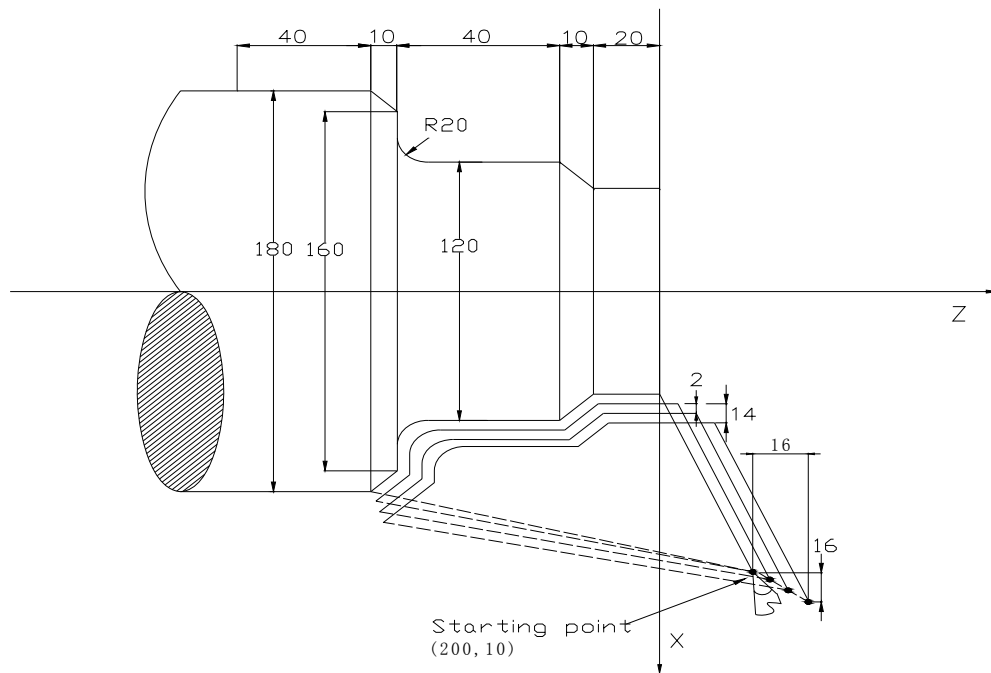


Fig. 3-39

Program: O0006;

G99 G00 X200 Z10 M03 S500; (Specify feedrate per rev and position starting point and start spindle)

G73 U1.0 W1.0 R3 ; (X tool retraction with 2mm, Z 1mm)

G73 P14 Q19 U0.5 W0.3 F0.3 ; (X roughing with 0.5 allowance and Z 0.3mm)

N14 G00 X80 W-40 ;

G01 W-20 F0.15 S600 ;

X120 W-10 ;

W-20 ;

G02 X160 W-20 R20 ;

N19 G01 X180 W-10 ;

G70 P14 Q19 M30;

} Blocks for finishing
(Finishing)

3.11.4 Finishing Cycle G70

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

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

ns: Block number of the first block of finishing path

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

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

```

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

. . .

```

} Blocks for finishing path

Command specifications:

- (1) G70 is compiled following ns~nf blocks. If they are in the front of G71 blocks, the system automatically searches and executes ns~nf blocks, and then executes the next program following nf block after they are executed, which causes the system executes ns~nf blocks repetitively.
- (2) F, S, T in ns~nf blocks are valid when executing ns~nf to command G70 finishing cycle.
- (3) G96, G97, G98, G99, G40, G41, G42 are valid in G70;
- (4) When G70 is executed, the system can stop the automatic run and manual traverse, but return to the position before manual traversing when G70 is executed again, otherwise, the following path will be wrong.
- (5) When the system is executing the single block, the program pauses after the system has executed end point of current path.
- (6) G70 cannot be executed in MDI mode, otherwise, the system alarms.
- (7) There are no the same block number in ns~nf when compound cycle commands are executed repetitively in one program.

3.11.5 Axial Grooving Multiple Cycle G74

Command format: G74 R (e);

G74 X (U) __ Z (W) __ P (Δi) Q (Δk) R (Δd) F__;

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

Relevant definitions:

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

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

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

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

Cutting end point: it is defined by X (U) ___ Z (W) ___, and is the end point B_f of last axial tool infeed.

R(e): it is the travel of tool retraction after each axial(Z) tool infeed without sign symbols as the following table. The command value is reserved after executing R (e) and the value of NO.5139 is rewritten. The value of NO.5139 is regarded as the travel of tool retraction when R (e) is not input.

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

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

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

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

P (Δi): travel of radial(X) cutting for each axial cutting cycle without sign symbols, and the value range is referred to the following table.

Q (Δi): travel of Z discontinuous tool infeed without sign symbols in axial(Z) cutting, and the value range is referred to the following table.

R(Δd): travel (radius value)of radial (X) tool retraction after cutting to end point of axial cutting. The value range is referred to the following table. The radial (X) tool retraction is 0 when R (Δd) is omitted and the system defaults the axial cutting end point. The radial (X) tool retraction is 0 when P (Δi) is omitted.

Address	Incremental system	metrci (mm) input	Inch (inch) input
P (Δi)	ISB system	0~999999999 (unit:0.001mm)	0~999999999 (unit:0.0001inch)
Q (Δk)	ISC system	0~999999999 (unit:0.0001mm)	0~999999999 (unit:0.00001inch)
R (e)	ISB system	0~99999.999mm	0~9999.9999 inch
R (Δd)	ISC system	0~9999.9999 mm	0~999.99999 inch

Command execution process: as Fig. 3-40.

- ① The system executes the axial (Z) cutting feed Δk from the starting point A_n of axial cutting cycle; when Z coordinate of cutting end point is less than that of starting point,

- the system executes Z negative feed, otherwise, positive feed;
- ② The system executes the axial(Z) rapid tool retraction e and its direction is opposite to the feed direction of ①;
 - ③ The system executes Z cutting feed ($\Delta k+e$) again, the end point of cutting feed is still in it between starting point A_n of axial cutting cycle and end point B_n of axial tool infeed; the system executes Z cutting feed ($\Delta k+e$) again and then executes ②; after it executes Z cutting feed ($\Delta k+e$) again, the end point of cutting feed is on B_n or is not between A_n and B_n , the system executes Z cutting feed to B_n and then executes ④;
 - ④ Radial(X) rapid tool retraction Δd (radius value) to C_n ; when X coordinate of B_f (cutting end point) is less than that of A (starting point), the system executes X positive tool retraction, otherwise, X negative tool retraction;
 - ⑤ Axial(Z axial) rapid retract tool to D_n , No. n axial cutting cycle is completed. If the current axial cutting cycle is not the last one, execute ⑥; if it is the previous one before the last axial cutting cycle, execute ⑦;
 - ⑥ Radial(X axial)rapid tool infeed, and its direction is opposite to that of ④tool retraction. When the end point of tool infeed is still on it between A and A_f (starting point of last axial cutting cycle) after the system executes X tool infeed ($\Delta d+\Delta i$) (radius value), i.e. $D_n \rightarrow A_{n+1}$ and then the system executes ① (start the next axial cutting cycle); after the system executes the tool infeed ($\Delta d+\Delta i$), the end point reaches A_f or is not between D_n and A_f , X rapidly traverse to A_f and executes ① to start the first axial cutting cycle;
 - ⑦ X rapidly traverse to return to A, and G74 is completed.

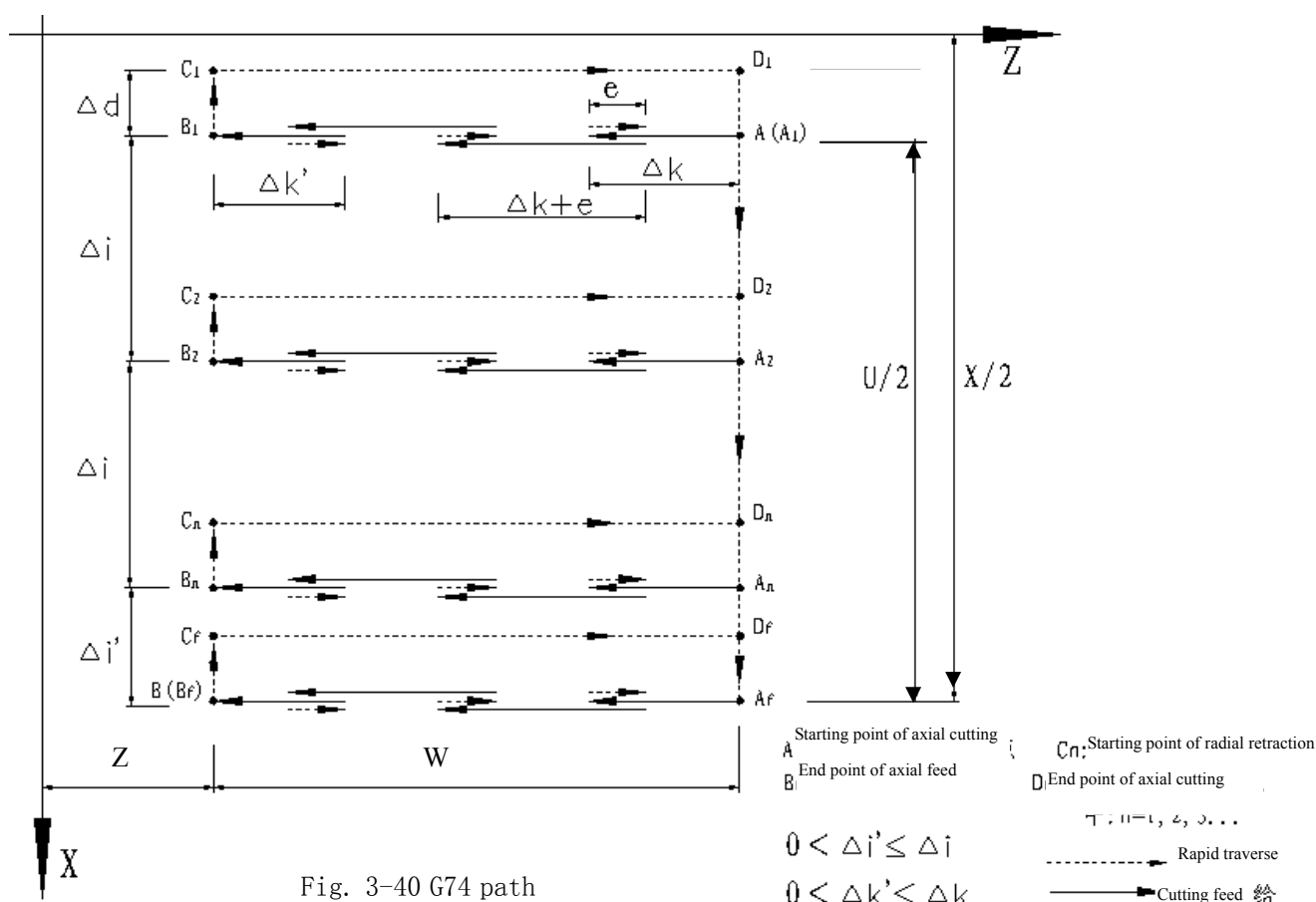
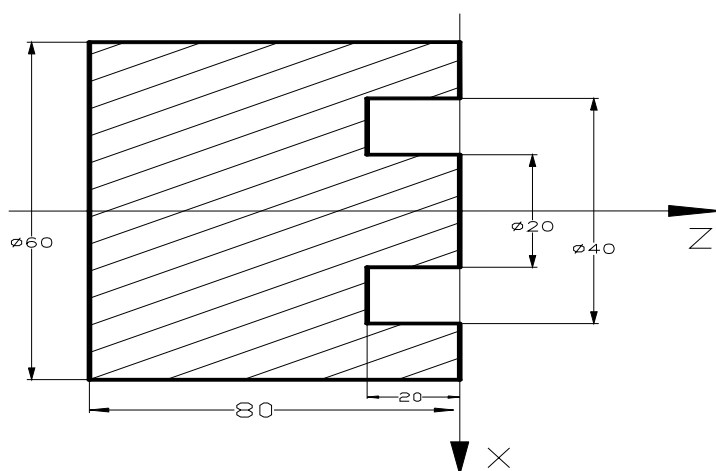


Fig. 3-40 G74 path

Command specifications:

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

Example: Fig.3-41



Program: O0007;

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

G74 R0.5 ; (Machining cycle)

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

M30; (End of program)

3.11.6 Radial Grooving Multiple Cycle G75

Command format: G75 R (e);

G75 X (U) __ Z (W) __ P (Δi) Q (Δk) R (Δd) F __;

Command function: Axial (Z) tool infeed cycle compounds radial discontinuous cutting cycle: Tool infeeds from starting point in radial direction, retracts, infeeds again, and again and again, and last tool retracts in axial direction, and retracts to position in radial direction, which is called one radial cutting cycle; tool infeeds in axial direction and execute the next radial cutting cycle; cut to end point of cutting, and then return to starting point (starting point and end point are the same one in G75), which is called one radial grooving

compound cycle. Directions of axial tool infeed and radial tool infeed are defined by relative position between end point X (U) Z (W) and starting point of cutting. G75 is used to machine the radial loop groove or column surface by radial discontinuously cutting, breaking stock and stock removal.

Relevant definitions:

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

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

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

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

Cutting end point: it is defined by X (U) ___ Z (W) ___, and is defined with B_f of the last radial tool infeed.

R (e): it is the travel of tool retraction after each radial(X) tool infeed without sign symbols and its value range is referred to the following table. The command value is reserved and the value of system parameter NO.5139 is rewritten after R (e) is executed. The value of NO.5139 is regarded as the travel of tool retraction when R (e) is not input.

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

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

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

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

P (Δi): it is the travel (diameter value) of radial(X) discontinuous tool infeed for each axial cutting cycle without sign symbols and its value range is referred to the following table.

Q (Δk): it is the travel of Z discontinuous tool infeed without sign symbols of the axial(Z) cutting, and the value range is referred to the following table.

R (Δd): it is the travel of axial (Z) tool retraction after cutting to end point of radial cutting with sign symbols and its value range is referred to the following table.

The system defaults the axial(Z) tool retraction is 0 when R (Δd) and Q (Δk) are omitted.

The system defaults to be the negative tool retraction when Z(W) is omitted.

Address	Incremental system	Metric (mm) input	Inch (inch) input
P (<u>Δi</u>)	ISB system	0~99999999 (unit:0.001mm)	0~99999999 (unit:0.0001inch)
Q (<u>Δk</u>)	ISC system	0~99999999 (unit:0.0001mm)	0~99999999 (unit:0.00001inch)
R (<u>e</u>)	ISB system	0~99999.999mm	0~9999.9999 inch
R (<u>Δd</u>)	ISC system	0~9999.9999 mm	0~999.99999 inch

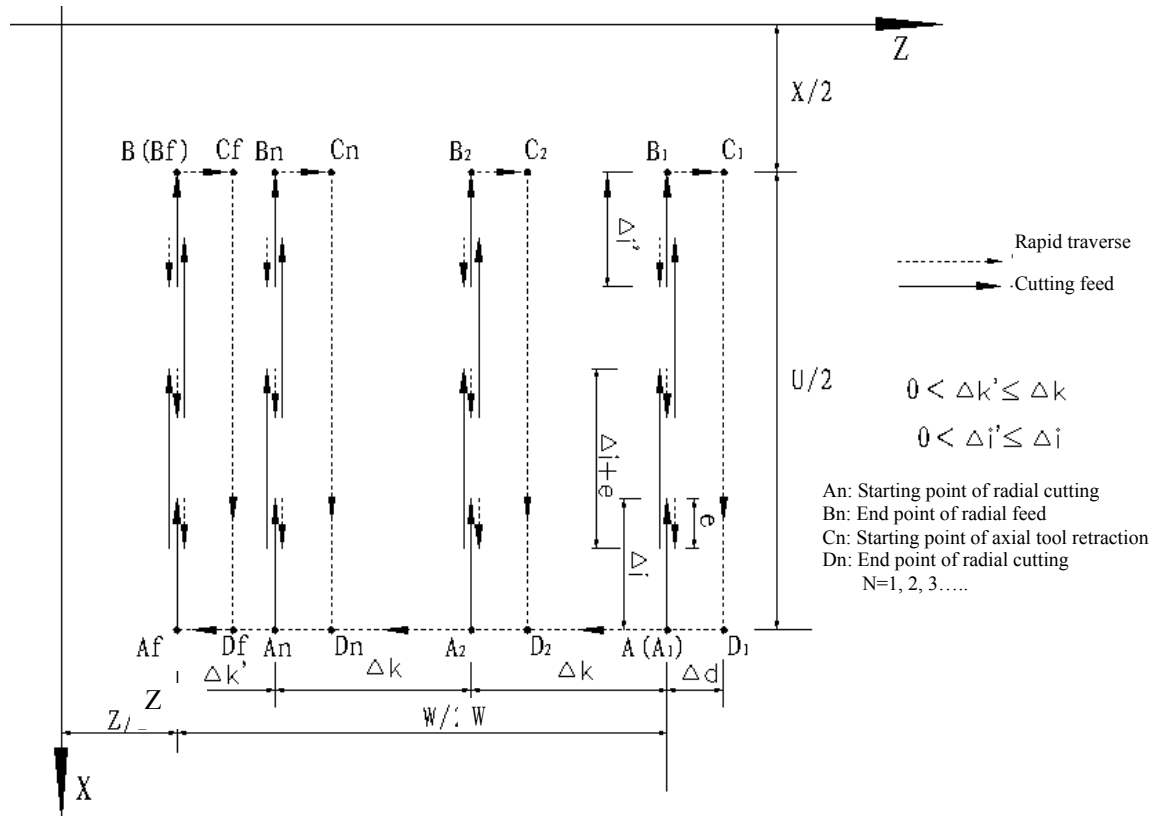


Fig. 3-42 G75 path

Execution process: as Fig. 3-42

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

of tool infeed is still on it between A and A_f (starting point of last radial cutting cycle) after Z tool infeed ($\Delta d + \Delta k$) (radius value), i.e. $D_n \rightarrow A_{n+1}$ and then execute ① (start the next radial cutting cycle); if the end point of tool infeed is not on it between D_n and A_f after Z tool infeed ($\Delta d + \Delta k$), rapidly traverse to A_f and execute ① to start the first radial cutting cycle;

- (1) The cycle movement is executed by X (W) and P (Δi) blocks of G75, G75 is not executed when there is no X(U) in G75 block. When only "G75 R (\underline{e}); " block is executed and only No.5139 value is modified, the cycle operation cannot be executed;
- (2) Δd and \underline{e} are specified by the same address R and whether there are X (U) and P (Δi) words or not in blocks can distinguish them;
- (3) The tool can stop in Auto mode and traverse in Manual mode when G75 is executed, but the tool must return to the position before executing in Manual mode when G75 is executed again, otherwise the following path will be wrong;
- (4) When the system is executing the single block, the program pauses after the system has executed end point of current path;
- (5) R (Δd) must be omitted in grooving, and so there is no travel of tool retraction when the tool cuts to radial cutting end point.

Example: as Fig.3-43

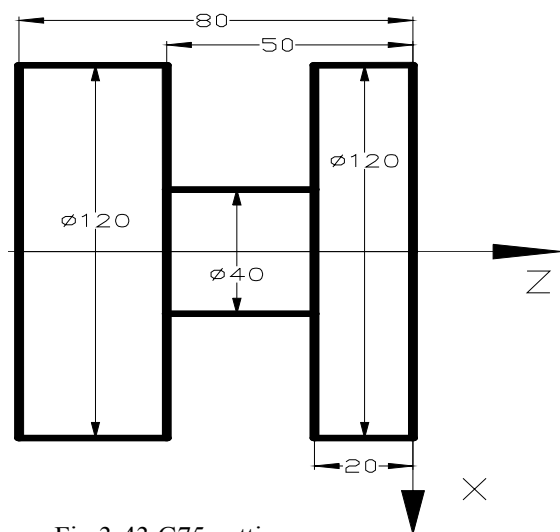


Fig.3-43 G75 cutting

Program:

O0008;

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

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

G75 R0.5 F150; (Machining cycle)

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

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

M30; (End of program)

3.11.7 Notes for multi cycle machining

1. When the multi cycle blocks are executed, they should be the specified address P, Q, X, Z, U, W, R of each block.
2. The block specified by P in G71, G72, G73 should be G00G01. When there is no command, the system alarms.
3. In MDI and DNC mode, G70, G71, G72 or G73 can not be specified, otherwise, the system alarms. But in MDI and DNC mode, G74, G75 or G76 can be specified.
4. The blocks in the serial numbers specified by P and Q in G71, G72 or G73 cannot specify the following command:
 - (1) non-modal G command except for G04 in group 00;
 - (2) all G commands except for G00, G01, G02, G03 in group 01;
 - (3) G20 and G21;
 - (4) M98 and M99;
5. The skip function should not be executed in the blocks of their serial number specified by P and Q. when the skip function is used in the blocks of their serial numbers specified by P and Q.
6. The tool nose radius compensation is invalid.
7. No.5104 Bit2 (FCK) sets whether G71, G72, G73 executes the outer check. When it is set to 1, the check is executed. The system alarms when the positioning point is in the cutting range.
8. No.5102 Bit1 (MRC) set whether the system alarm when the finishing cycle in G71, G72 is in non-monotonous, and it alarms when Bit1 is set to 1.

3.12 Threading Cutting

GSK988T CNC system can machine many kinds of thread cutting, including metric/inch single, multi threads, thread with variable lead and tapping cycle. Length and angle of thread run-out can be changed, multiple cycle thread is machined by single sided to protect tool and improve smooth finish of its surface. Thread cutting includes: continuous thread cutting G32, thread cutting with variable lead G34, Z thread cutting G33, Thread cutting cycle G92, Multiple thread cutting cycle G76

The machine used to thread cutting must be installed with spindle encoder, the transmission ratio between spindle and encoder is set by the parameter. X or Z traverses to start machine after the system receives spindle signal per rev in thread cutting, and so one thread is machined by multiple roughing, finishing without changing spindle speed.

GSK988T CNC system can machine many kinds of thread cutting, such as thread cutting without tool retraction groove. There is a big error in the thread pitch because there are the acceleration and the deceleration at the starting and ending of X and Z thread cutting, and so there is length of thread lead-in and distance of tool retraction at the actual starting and ending of thread cutting.

X, Z traverse speeds are defined by spindle speed instead of cutting feedrate override in thread cutting when the pitch is defined. The spindle override control is valid in thread cutting. When the spindle speed is changed, there is error in pitch caused by X and Z acceleration/deceleration, and so the spindle speed cannot be changed and the spindle cannot be stopped in thread cutting, which will cause tool and workpiece to be damaged.

3.12.1 Thread Cutting with Constant Lead G32

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

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

Command specifications: G32 is modal;

Pitch is defined to moving distance when the spindle rotates one -turn(radius value);

IP_: end point coordinate value. It can be specified by the absolute command value or incremental command value. The system specifies the different IP_ value to execute the straight thread cutting, end face thread cutting and taper thread cutting.

F: Metric pitch is moving distance of long axis when the spindle rotates one-turn and its value range is referred to the following table. After F is executed, it is valid until F with specified pitch is executed again. The pitch F value precision is the last two-digit of the decimal.

J: Travel in the short axis in thread run-out with positive/negative sign symbols and the value range is referred to the following table; the value is specified by the radius value.

K: Length in the long axis in thread run-out. The value range is referred to the following table. It has no direction.

Q: Initial angle between spindle rotation one-turn and starting point of thread cutting. The value range without the decimal is referred to the following table. Q is non-modal parameter, must be defined every time, it is 0^0 .when it is not specified, the system specifies Q different value can cut multi-thread.

Q rules:

1. Its initial angle is 0^0 if Q is not specified;
2. For continuous thread cutting, Q specified by its following thread cutting block except for the first block is invalid, namely Q is omitted even if it is specified;
3. In ISB mode, Q unit is 0.001^0 . ISC mode, Q unit is 0.0001^0 . Example, in ISB mode, Q180000 is input in program if it offsets 180^0 with spindle one rev; if Q180 or Q180.0, it is 0.18^0 . When the system specifies the value more than 360000, it counts based on 360000(180).
4. It is suggested that the system should use G97 instead of the constant surface cutting speed control in thread cutting.

Table 3-20

Address	Incremental system	Metric (mm) input	Inch (inch) input
F	ISB, ISC	0.01~500 mm	0.01~9.99inch
J	ISB	-99999.999~99999.999mm	-9999.9999~9999.9999 inch
	ISC	-9999.9999~9999.9999 mm	-999.99999~999.99999 inch
K	ISB	0~99999.999mm	0~9999.9999 inch
	ISC	0~9999.9999mm	0~999.99999 inch

Q	ISB	0~999999999(unit:0.001 degree)	0~99999999 (unit: 0.001 degree)
	ISC	0~999999999(unit:0.0001 degree)	0~99999999 (unit:0.0001 degree)

Difference between long axis and short axis is as Fig.3-44.

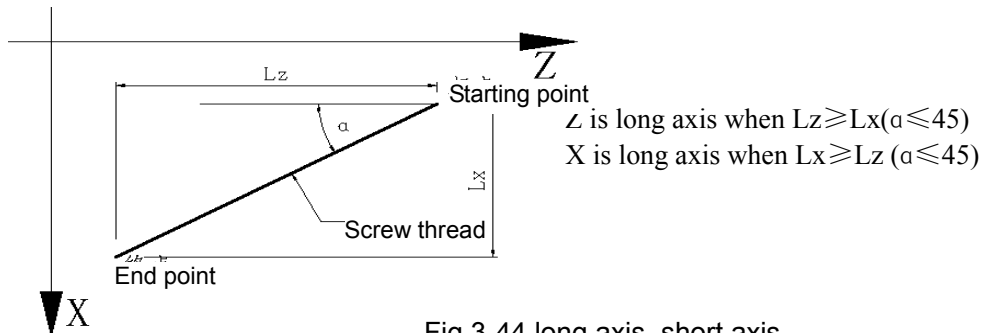


Fig.3-44 long axis, short axis

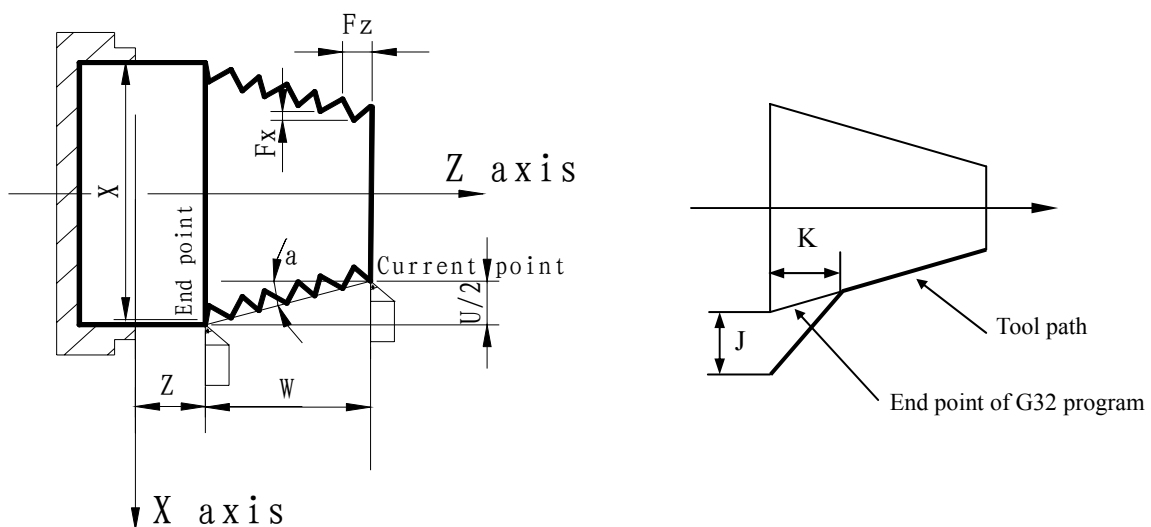


Fig.3-45 G32 path

Notes:

- (1) J, K are modal. The thread run-out is previous J, K value when they are omitted in the next block in continuous thread cutting. Their mode are cancelled when no thread cutting are executed;
- (2) There is no thread run-out when J, or J, K are omitted; $K=J$ is the thread run-out value when K is omitted;
- (3) There is no thread run-out when $J=0$ or $J=0, K=0$;
- (4) The thread run-out value $J=K$ when $J \neq 0, K=0$;
- (5) There is no thread run-out when $J=0$ or $K \neq 0$;
- (6) If the current block is for thread and the next block is the same, the system does not test the spindle encoder signal per rev at starting the next block to execute the direct thread cutting, which function is called as continuous thread machining.
- (7) After the feed hold is executed, the system displays "Pause" and the thread cutting continuously executes not to stop until the current block is executed completely; if the continuous thread cutting is executed, the program run pauses after thread cutting blocks are executed completely.
- (8) In Single block, the program stops run after the current block is executed. The program stops run after all blocks for thread cutting are executed.

- (9) The thread cutting decelerates to stop when the system resets, emergently stop or its driver alarms
- (10) The system alarms when the thread run-out length is more than the thread machined length of the long axis.
- (11) In G32, the basic axis command cannot be in the same block with its parallel axis command, otherwise, the system alarms.

● alarms.

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

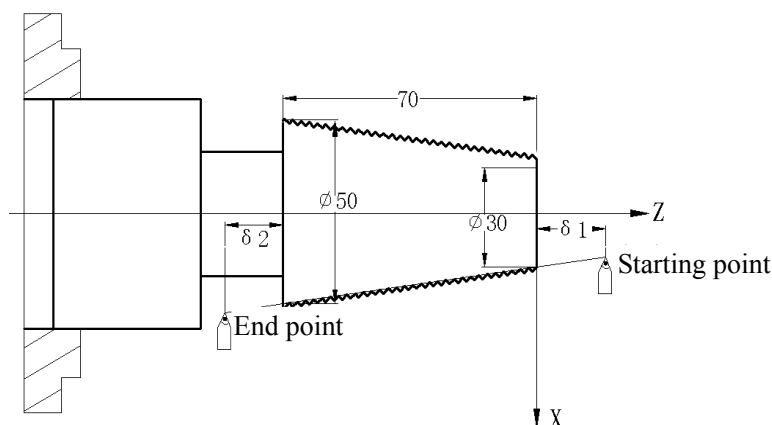


Fig. 3-46

Program:

```

O0009;
G00 X28 Z3;          (First cut-in 1mm)
G32 X51 W-75 F2.0;   (First taper cutting)
G00 X55;             (Tool retraction)
W75;                 (Z returns to the starting point)
X27;                 (Second tool infeed 0.5mm)
G32 X50 W-75 F2.0;   (Second taper thread cutting)
G00 X55;             (Tool retraction)
W75 ;                (Z returns to the starting point)
M30;
    
```

3.12.2 Thread cutting with variable lead G34

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

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

G34 can machine the metric, inch pitch. Machine metric or inch straight, taper, end face thread with variable pitch.

Command specifications: **G34 is modal;**

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

F: Metric thread of first pitch from starting point: 0.001~500 mm;

R: Increment or decrement of pitch per rev, $R=F1-F2$, with direction; $F1>F2$, pitch decreases when R is negative; $F1<F2$, pitch increases when R is positive (as Fig. 3-47);

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

$\pm 0.01 \sim \pm 9.99$ tooth/inch (inch thread).

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

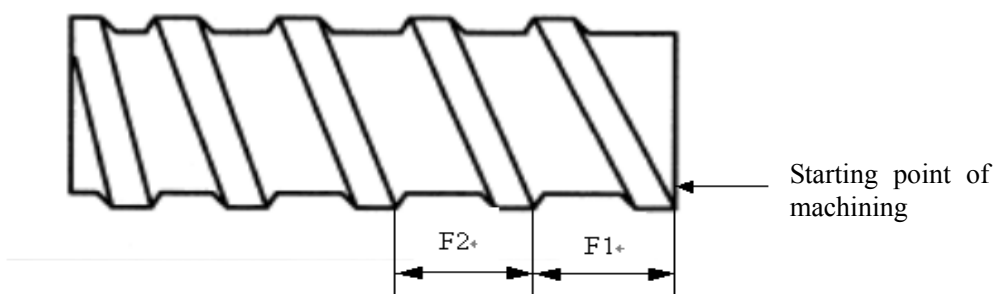


Fig. 3-47 Variable pitch thread

Caution:

- It is the same as that of G32.

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

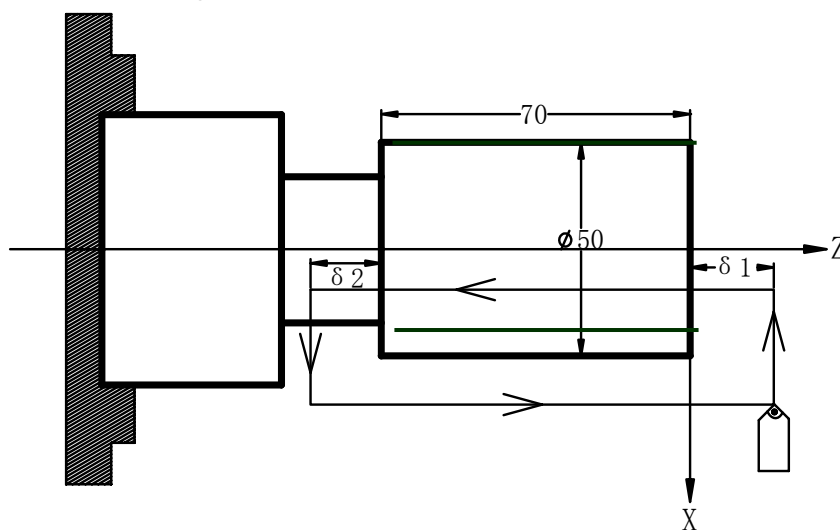


Fig. 3-48 Variable pitch thread machining

Value: $\delta 1 = 4\text{mm}$, $\delta 2 = 4\text{mm}$, total cutting depth 1mm, total cutting cycle 2 times; 1st tool infeed 0.7mm.

Program: O0010;

G00 X60 Z4 M03 S500;

G00 U-10;

G00 U-0.7;

G34 W-78 F4 J5 K2 R0.2;

G00 U10;

Tool infeed $\Phi 50$

Tool infeed

Variable pitch thread cutting

Tool retraction

Z returns to initial point

Z4;	Tool infeed again $\Phi 50$
G00 X50;	Tool infeed
G00 U-1.0;	Variable pitch thread cutting
G34 W-78 F4 J5 K2 R0.2;	Tool retraction
G00 U10;	Z returns to initial point
Z4;	
M30;	

3.12.3 Thread cutting cycle G92

Command format: G92 X (U) _ Z (W) _ F_ J_ K_ L_; (straight thread cutting cycle)
 G92 X (U) _ Z (W) _ R_ F_ J_ K_ L_; (taper thread cutting cycle)

Command function: Tool infeeds in radial(X) direction and cuts in axial(Z or X, Z) direction from starting point of cutting to realize straight thread, taper thread cutting cycle with constant thread pitch. Thread run-out in G92: at the fixed distance from end point of thread cutting, Z executes thread interpolation and X retracts with exponential or linear acceleration, and X retracts at rapidly traverse speed after Z reaches to end point of cutting as Fig. 3-49.

Command specifications:

G92 is modal;

Starting point of cutting: starting position of thread interpolation;

End point of cutting: end position of thread interpolation;

X: X absolute coordinate of end point of cutting, unit:mm;

U: different value of X absolute coordinate from end point to starting point of cutting, unit:mm;

Z: Z absolute coordinate of end point of cutting, unit:mm;

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

R: different value(R value) of X absolute coordinate from end point to starting point of cutting.

When the sign of R is not the same that of U, $R \leq |U/2|$, unit:mm.

Incremental system	Metric (mm) input	Inch (inch) input
ISB system	-99999.999~99999.999mm	-9999.9999~9999.9999 inch
ISC system	-9999.9999~9999.9999 mm	-999.99999~999.99999 inch

F: metric thread pitch is the same that of G32. After F value is executed, it is reserved and can be omitted;

J: Travel in the short axis in thread run-out is same that of G32 and cannot be less than 0 without direction (automatically define its direction according to starting position of program), and it is modal parameter. If the short axis is X, its value is specified by radius;

K: Travel in the long axis in thread run-out is same that of G32 without direction (automatically define its direction according to starting position of program), and it is modal parameter. If the long axis is X, its value is specified by radius;

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

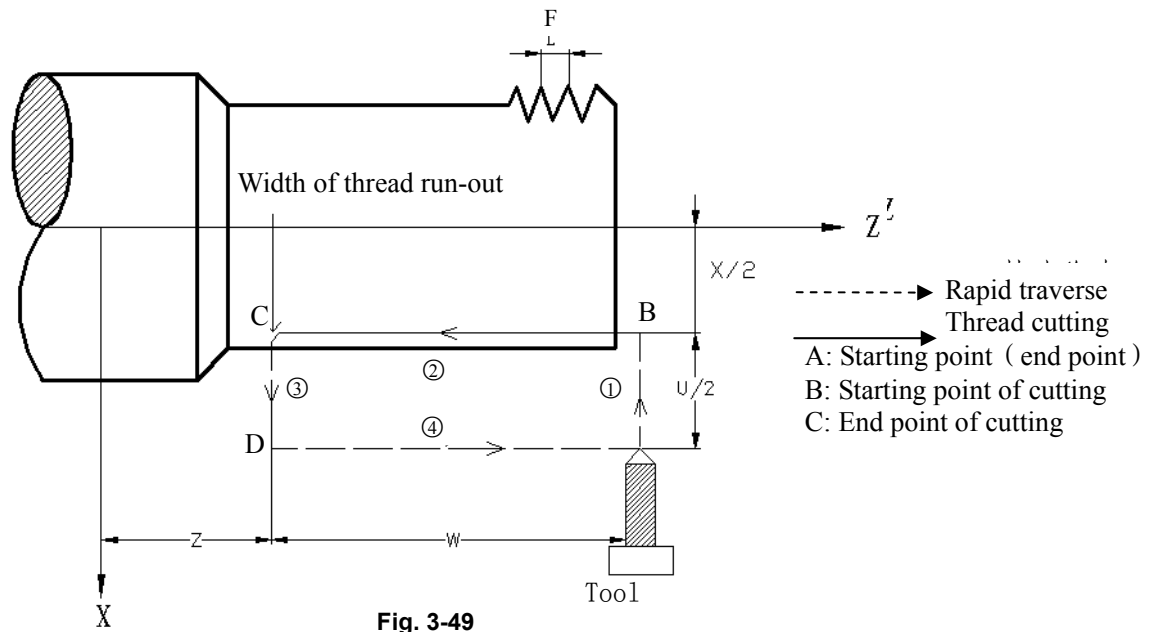


Fig. 3-49

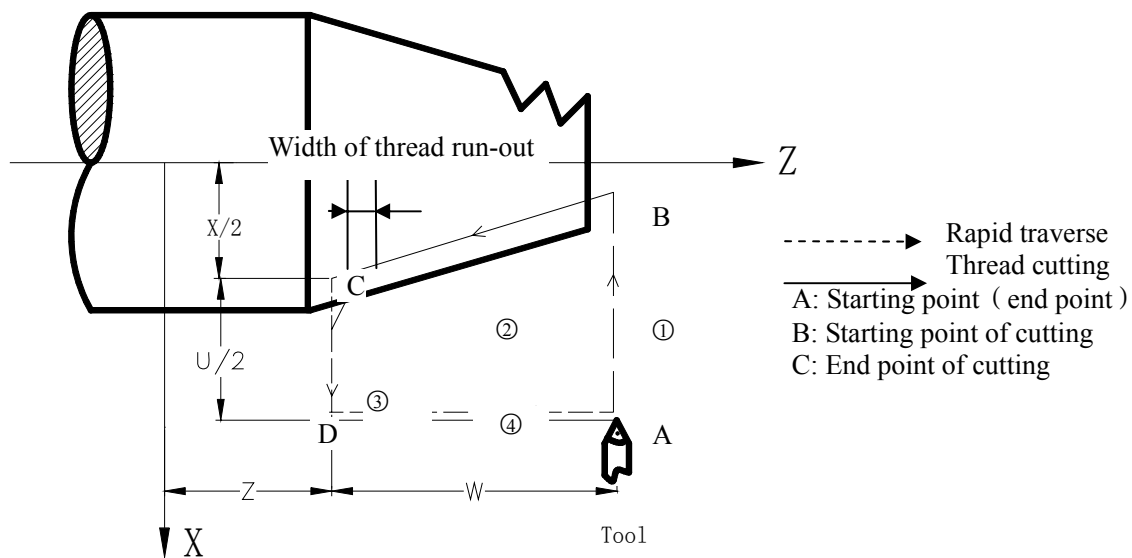


Fig. 3-50

The system can machine one thread with many tool infeed in G92, but cannot do continuous two thread and end face thread. Definition of thread pitch in G92 is the same that of G32, and a pitch is defined that it is a moving distance of long axis(X in radius) when the spindle rotates one revolution.

Pitch of taper thread is defined that it is a moving distance of long axis(X in radius). When absolute value of Z coordinate difference between B point and C point is more than that of X (in radius), Z is long axis; and reversely, X is the long axis.

Cycle process: straight thread as Fig.3-49 and taper thread as Fig.3-50.

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

Notes:

- (1) Length of thread run-out is specified by No. 5130 when J, K are omitted, and the thread run-out value=setting value x 0.1 x L, and L is the thread pitch;
- (2) Length of thread run-out is K in the long direction and is specified by No.5130 when J is omitted;
- (3) Length of thread run-out is J=K when K is omitted;
- (4) There is no thread run-out when J=0 or J=0, K=0;
- (5) Length of thread run-out is J=K when J≠0,K=0;
- (6) There is no thread run-out when J=0,K≠0;
- (7) After executing the feed hold in thread cutting, the system does not stop cutting until the thread cutting is completed with **Pause** on screen;
- (8) After executing single block in thread cutting, the program run stops after the system returns to starting point(one thread cutting cycle is completed);
- (9) Thread cutting decelerates to stop when the system resets, emergently stops or its driver alarms;
- (10) The system alarms when the thread run-out length of the long axis is more than the thread machining length of the long axis;
- (11) The system alarms when the thread run-out length of the short axis is more than the thread machining length of the short axis.

Command path: relative position between thread cutting end point and starting point with U, W, R and tool path and thread run-out direction with different U, W, R signs as Fig. 3-51:

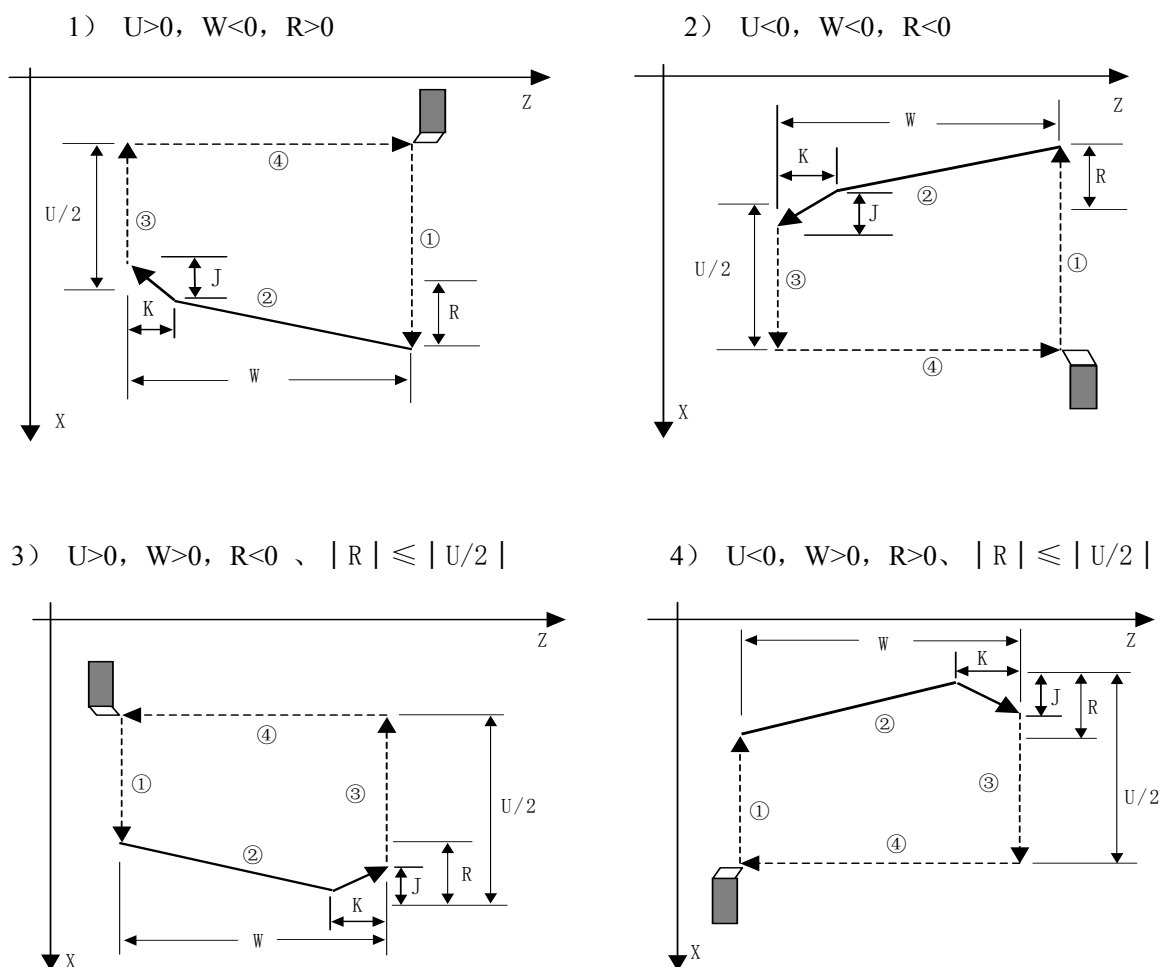


Fig. 3-51

Example: Fig. 3-52

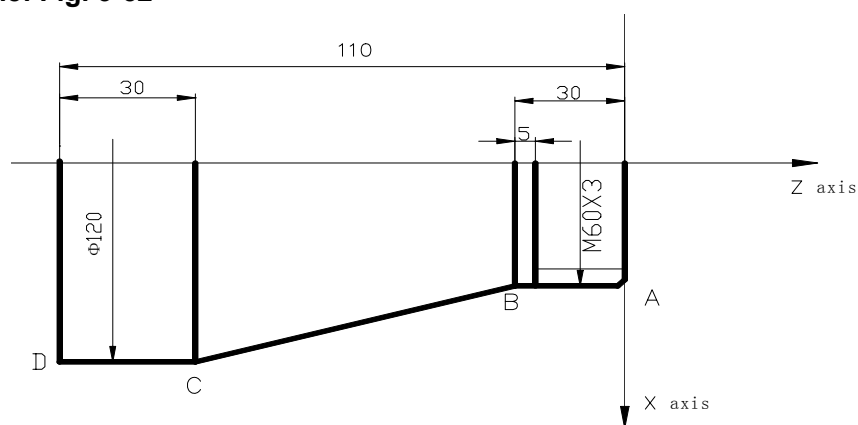


Fig. 3-52

Program:

O0012;

M3 S300 G0 X150 Z50 T0101; (Thread tool)

G0 X65 Z5; (Rapid traverse)

G92 X58.7 Z-28 F3 J3 K1; (Machine thread with 4 times cutting, the first tool infeed 1.3mm)

X57.7 ; (The second tool infeed 1mm)

X57; (The third tool infeed 0.7mm)
X56.9; (The fourth tool infeed 0.1mm)
M30;

3.12.4 Multiple thread cutting cycle G76

Command format: G76 P (m) (r) (a) Q (△dmin) R (d);

G76 X (U) __ Z (W) __ R (i) P (k) Q (△d) F__;

Command function: Machining thread with specified depth of thread (total cutting depth) is completed by multiple roughing and finishing, if the defined angle of thread is not 0°, thread run-in path of roughing is from its top to bottom, and angle of neighboring thread teeth is the defined angle of thread. G76 can be used for machining the straight and taper thread with thread run-out path, which is contributed to thread cutting with single tool edge to reduce the wear of tool and to improve the precision of machining thread. But G76 cannot be used for machining the face thread. machining path is as Fig. 3-53(a):

Relevant definitions:

Starting point(end point): position before block runs and behind blocks run, defined by A point;

End point of thread: end point of thread cutting defined by X(U) __ Z(W) __. The tool will not reach

the point in cutting if there is the thread run-out path;

Starting point of thread: its absolute coordinates is the same that of A point and the different value of

X absolute coordinates between C and D is i(thread taper with radius value). The tool cannot reach C

point in cutting when the defined angle of thread is not 0°;

Reference point of thread cutting depth: its absolute coordinates is the same that of A point and the different value of X absolute coordinate between B and C is k(thread taper with radius value). The cutting depth of thread at B point is 0 which is the reference point used for counting each thread cutting depth by the system;

Thread cutting depth: it is the cutting depth for each thread cutting cycle. It is the different value (radius value, without signs) of X absolute coordinate between B and intersection of reversal extension line for each thread cutting path and straight line BC. The cutting depth for each roughing is $\sqrt{n} \times \Delta d$, n is the current roughing cycle times, Δd is the thread cutting depth of first roughing;

Travel of thread cutting: different value between the current thread current depth and the previous one: $(\sqrt{n} - \sqrt{n-1}) \times \Delta d$;

End point of tool retraction: it is the end position of radial (X) tool retraction after the thread cutting in each thread roughing, finishing cycle is completed, is defined by E point;

$$\operatorname{tg} \frac{a}{2} = \frac{|Z \text{ replacement } t|}{|X \text{ replacement } t|}$$

a: thread angle;

Thread cut-in point: actual start thread cutting point in each thread roughing cycle and finishing cycle. It is defined by (n is the cutting cycle times), B_n is the first thread roughing cut-in point, B₁ is the last thread roughing cut-in point, B_e is the thread finishing cut-in point. B_n is the X, Z

displacement formula for B point:

X: X absolute coordinate of thread end point

U: difference value of X absolute coordinate between thread end point and starting point;

Z: Z absolute coordinate of thread end point;

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

P(m): Times of thread finishing: 00~99 (unit: times) with 2-digit digital. It is valid after m command value is executed, and the value of system parameter No.5142 is rewritten to m. The value of system parameter No.5142 is regarded as finishing times when m is not input. The thread is finished according to the programmed thread path, the first finishing cutting travel is d and the following one is 0,

P(r): Width of thread run-out 00~99(unit: $0.1 \times L$, L is the thread pitch) with 2-digit digital. It is valid after r command value is executed and the value of system parameter No.5130 is rewritten to r. The value of system parameter No.5130 is the width of thread run-out when r is not input. The thread run-out function can be applied to thread machining without tool retraction groove and the width of thread run-out defined by system parameter No.5130 is valid for G92, G76;

P(a): Angle at taper of neighboring two tooth is 0~99, unit: degree($^{\circ}$), with 2-digit digital. It is valid after a command value is executed and the value of system parameter No.5143 is rewritten to a. The value of system parameter No.5143 is regarded as angle of thread tooth. The actual angle of thread is defined by tool ones and so a should be the same as the tool angle;

$\Delta Q(\Delta d_{min})$: Minimum cutting travel of thread roughing (radius value without sign symbols). When $(\sqrt{n} - \sqrt{n-1}) \times \Delta d < \Delta d_{min}$, Δd_{min} is regarded as the cutting travel of current roughing, i.e. depth of current thread cutting is $(\sqrt{n-1} \times \Delta d + \Delta d_{min})$. Δd_{min} is applied because the cutting travel of roughing is undersize and the times of roughing is excessive, which is caused the cutting travel of thread roughing gradually decreases. After $Q(\Delta d_{min})$ is executed, the command value Δd_{min} is value and the value of system parameter No.5140 is rewritten to minimum cutting travel; when $Q(\Delta d_{min})$ is not input, the system takes No.5140 value as the least cutting value.

R(d): It is the cutting travel of thread finishing, and is the different value(radius value without sign symbols) of X absolute coordinates between cut-in point Be of thread finishing and Bf of thread roughing. After R(d) is executed, the command value d is value and the value of system parameter No.5141 is rewritten to $d \times 1000$ (unit: 0.001 mm). The value of system parameter No.5141 is regarded as the cutting travel of thread finishing when R(d) is not input.

R(i): It is thread taper and is the different value of X absolute coordinate between thread starting point and end point (unit: mm, radius value). The system defaults $i=0$ (straight thread) when i is not input;

P(k): It is the depth of thread tooth and is also the total cutting depth of thread(radius value without sign symbols), and the system alarms when P(k) is not input;

Q(Δd): It is the first depth of thread cutting (radius value without sign symbols).The system alarms when Δd is not input;

F: metric thread pitch is as follows:

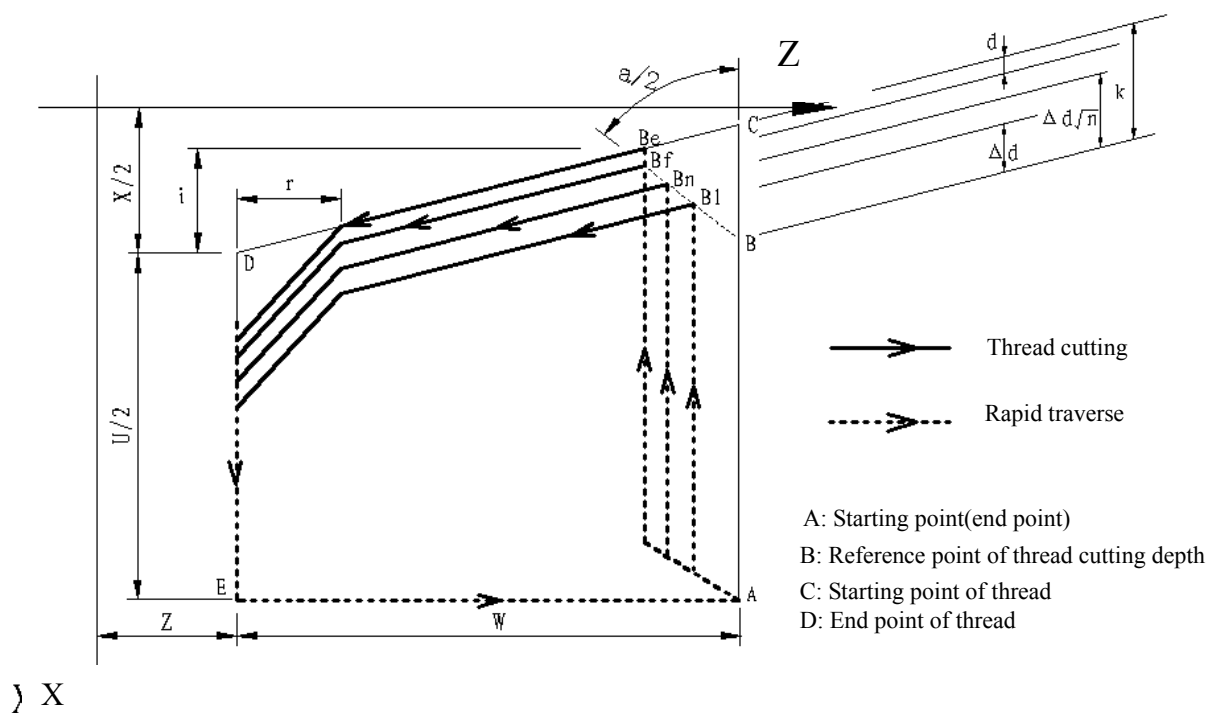


Fig. 3-53

Cut-in method as follows: Fig.3-54

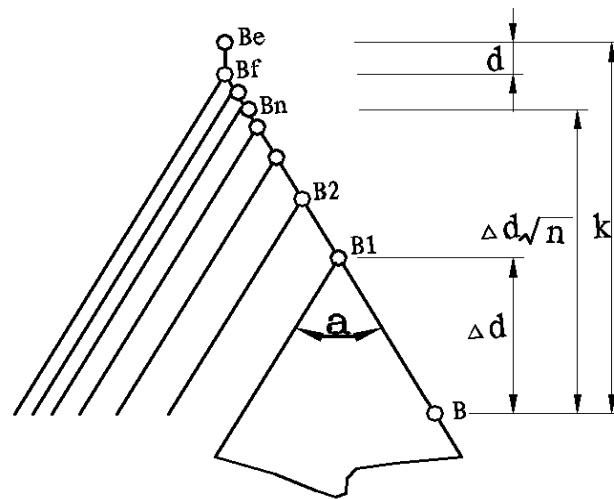


Fig. 3-54

Pitch is defined to moving distance (radius value in X direction) of long axis when the spindle rotates one rev. Z is long when absolute value of coordinate difference between C point and D point in Z direction is more than that of X direction (radius value, be equal to absolute value of i); and vice versa.

Execution process:

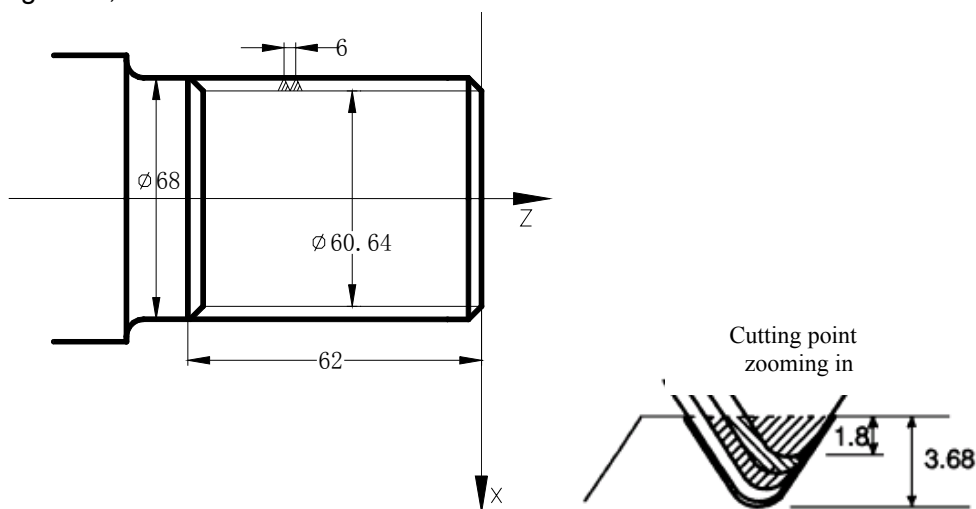
- ① The tool rapidly traverses to B_1 , and the thread cutting depth is Δd . The tool only traverses in X direction when $a=0$; the tool traverses in X and Z direction and its direction is the same that of $A \rightarrow D$ when $a \neq 0$;
- ② The tool cuts threads paralleling with $C \rightarrow D$ to the intersection of $D \rightarrow E$ ($r \neq 0$: thread run-out);
- ③ The tool rapidly traverses to E point in X direction;
- ④ The tool rapidly traverses to A point in Z direction and the single roughing cycle is completed;
- ⑤ The tool rapidly traverses again to tool infeed to B_n (is the roughing times), the cutting depth is the bigger value of ($\sqrt{n} \times \Delta d$), ($\sqrt{n-1} \times \Delta d + \Delta d_{min}$), and execute ② if the cutting depth is less than $(k-d)$; if the cutting depth is more than or equal to $(k-d)$, the tool infeeds $(k-d)$ to B_f , and then execute ⑥ to complete the last thread roughing;
- ⑥ The tool cuts threads paralleling with $C \rightarrow D$ to the intersection of $D \rightarrow E$ ($r \neq 0$: thread run-out);
- ⑦ X axis rapidly traverses to E point;
- ⑧ Z axis traverses to A point and the thread roughing cycle is completed to execute the finishing;
- ⑨ After the tool rapidly traverses to B (the cutting depth is k and the cutting travel is d), execute the thread finishing, at last the tool returns to A point and so the thread finishing cycle is completed;
- ⑩ If the finishing cycle time is less than m, execute ⑨ to perform the finishing cycle, the thread cutting depth is k and the cutting travel is 0; if the finishing cycle times is equal to m, G76 compound thread machining cycle is completed.

Notes:

- (1) In thread cutting, execute the feed hold, the system displays **Pause** after the thread cutting is executed completely, and then the program run pauses;

- (2) Execute single block in thread cutting and the program run stops after returning to starting point(one thread cutting cycle is completed);
- (3) The thread cutting decelerates to stop when the system resets and emergently stop or the driver alarms;
- (4) Omit all or some of G76 P(m)(r)(a) Q(Δd_{min}) R(d). The omitted address runs according to setting value of parameters;
- (5) m, r, a used for one command address P are input one time. Program runs according to setting value of №57, 19, 58 when m, r, a are all omitted; Setting value is a when address P is input with 1 or 2 digits; setting values are r, a when address P is input with 3 or 4 digits;
- (6) The direction of A→C→D→E is defined by signs of U,W , and the direction of C→D is defined by the sign of R(i) . There are four kinds of sign composition of U, W corresponding to four kinds of machining path;
- (7) When the set thread cutting depth is more than the thread total cutting depth, the system only executes roughing once, and its cutting depth is equal to the total cutting depth of the roughing;
- (8) In roughing, the system alarms when the least cutting amount or the finishing allowance is more than the thread tooth height;
- (9) The system alarms when the thread run-out length is more than its machining length.

Example: Fig. 3-55, thread M68×6.



Program:

```
O0013;
G50 X100 Z50 M3 S300;

G00 X80 Z10;
G76 P020560 Q150 R0.1;

G76 X60.64 Z-62 P3680 Q1800 F6;
G00 X100 Z50 ;
M30;
```

Fig. 3-55

(Set workpiece coordinate system, start spindle and specify spindle speed)

(Rapid traverse to starting point of machining)

(Finishing 2 times, chamfering width 0.5mm, tool angle 60°, min. cutting depth 0.15, finishing allowance 0.1)

(Tooth height 3.68, the first cutting depth 1.8)

(Return to starting point of program)

(End of program)

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

Command format: G96 Sxxxx;

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

Command explanation: G96 is modal G command. If the current modal is G96, G96 can not be input; it is the cutting surface speed in Sxxxxx constant surface control.

Command format: G97 Sxxxx;

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

Command explanation: G96 is modal G command. If the current modal is G97, G97 cannot be input;

It is the spindle speed in Sxxxxx constant speed control(r/min).

Relative command: G50

Command format: G50 Sxxxx;

Command function: define max. spindle speed limit (r/min) in the constant surface speed control(r/min).

Command explanation: after the system is turned on, and the max. spindle speed is not specified, the system does not limit the spindle speed state. Max. spindle speed limit is valid for G96, and is invalid for G97;

S value set by G50 is modal and is value before the new max. speed is set;

Note: when G50 S0 is executed, the spindle speed is limited in 0 r/min (the spindle does not rotate) in the constant surface control;

The system does not set the current workpiece coordinate system when G50 sets the constant surface speed control.

Address	Incremental system	Metric (mm) input	Inch (inch)input
S (G96)	ISB, ISC	0~20000 m/min	0~2000 feet/min
S (G97)	ISB, ISC	0~20000 r/min	0~20000 r/min

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

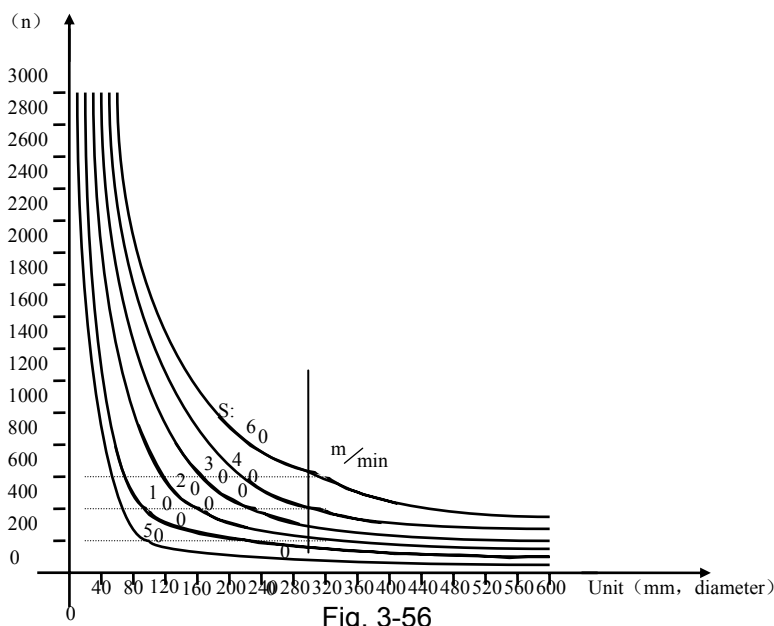
When the spindle speed controlled by the analog voltage is valid, the constant surface control is valid. The spindle speed is changed along with the absolute value of X absolute coordinate of programming path in the constant speed control. If the absolute value of X absolute coordinate adds, the spindle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

Surface speed=spindle speed× |X| × π ÷1000 (m/min)

Spindle speed: r/min

|X|: absolute value of X absolute coordinate value (diameter value), mm

$\pi \approx 3.14$



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

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

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

Notes:

- (1) G96, G97 are modal in the same group, and one of them is valid in the same time. G97 is initial word and is valid after the system is turned on.
- (2) In G96, S value commanded is reserved in G97. there is no new S is commanded and the S value in the last G96 state is recovered to the current valid surface speed after the system returns to G96 state, the system outputs the least surface speed in G96 when there is no saved value.
- (3) From G96 to G97, if none of S command (r/min) is commanded in the program block in G97, the last spindle speed in G96 is taken as S command in G97, namely, the spindle speed is not changed at this time;
- (4) The constant surface speed control function is still valid when the machine is locked(X, Z do not move when the system executes X, Z motion commands);
- (5) In G96, when the spindle speed counted by the cutting surface speed is more than max. speed of current spindle gear, at this time, the spindle speed is limited to max. one of current spindle gear;
- (6) In thread cutting, To gain the precise thread machining, it should not be adopted with the constant surface speed control but the constant rotational speed (G97) in the

course of thread cutting;

(7) No.3031 sets the numerical digit permitted by S.

Example:

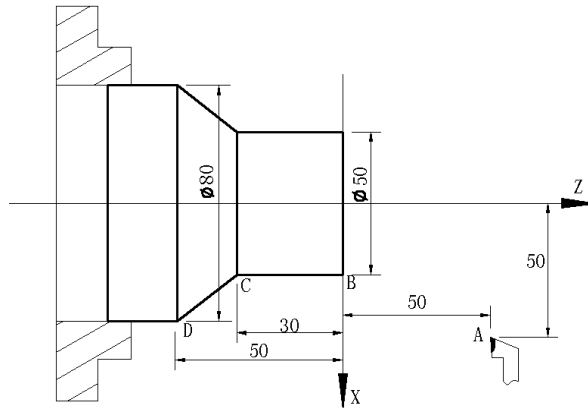


Fig. 3-57

Program:

```

M3 G96 S300;    (Spindle rotates clockwise, the constant surface speed control is valid
                  and the surface speed is 300m/min)
G0 X100 Z100;   (Rapid traverse to A point with spindle speed 955 r/min)
G0 X50 Z0;      (Rapid traverse to B point with spindle speed 1910 r/min)
G1 W-30 F200;   (Cut from B to C with spindle speed 1910 r/min)
X80 W-20 F150;  (Cut from C to D with spindle speed 1910 r/min and surface speed
                  1194 r/min)
G0 X100 Z100;   (Rapid retract to A point with spindle speed 955 r/min)
M30;            (End of program, spindle stop and coolant OFF)
  
```

3.14 Feedrate per minute G98, feedrate per rev G99

Command format: G98 Fxxx; (F0001~NO027, the leading zero can be omitted, feed rate per minute is specified, mm/min)

Command function: cutting feed rate is specified as mm/min, G98 is the modal G command. G98 cannot be input if the current command is G98 modal.

Command format: G99 Fxxx; (F0.0001~F500, the leading zero can be omitted)

Command function: cutting feed rate is specified as mm/min, G99 is the modal G command. G99 input may be omitted if current state is G99. The actual cutting feedrate is gotten by multiplying the F command value (mm/r) to the current spindle speed(r/min). If the spindle speed varies, the actual feedrate changes too. If the spindle cutting feed amount per rev is specified by G99 FXXXX, the even cutting texture on the surface of workpiece will be gotten. In G99 state, a spindle encoder should be fixed on the machine tool to machine the workpiece.

G98, G99 are the modal G commands in the same group and only one is valid. G98 is the initial state G command and the system defaults the modal can be set by No.3402 Bit4 (FPM) when the system turns on. F value ranges in G98, G99 are as follows:

Table 3-21

Address	Incremental system	Metric (mm) input	Inch (inch)input
F (G98)	ISB system	1 ~ 60000 mm/min	0.01~2400 inch/min
	ISC system	1 ~ 24000 mm/min	0.01~960 inch/min
F (G99)	ISB system	0.01~500mm/r	0.01~9.99inch/r
	ISC system	0.01~500mm/r	0.01~9.99 inch/r

Reduction formula of feed between per rev and per min:

$$F_m = F_r \times S$$

F_m : feed per min (mm/min) ;

F_r : feed per rev(mm/r) ;

S : spindle speed (r/min) .

F value is reserved after the system executes F command.

Notes:

- (1) In G99 modal, there is the uneven cutting feed rate when the spindle speed is lower than 1 r/min; there is the follow error in the actual cutting feed rate when there is the swing in the spindle speed. To gain the high machining quality, it is recommended that the selected spindle speed should be not lower than min. speed of spindle servo or converter.
- (2) No.1422 set the upper of the cutting feedrate. When the actual cutting feedrate (the value is multiplied by the override) exceeds the specified upper limit, it is clamped to the upper limit value.
- (3) No. 1403 Bit0(MIF)can set the cutting speed unit per minute and the detailed is referred to the parameter explanation.

3.15 Macro command

GSK988T provides the macro command which is similar to the high language, and can realize the variable assignment, and subtract operation, logic decision and conditional jump by user macro command, contributed to compiling part program for special workpiece, reduce the fussy counting and simplify the user program.

3.15.1 Variable

(1) variable use

The variable can specify the address value in the program. The variable value is assigned by the program command or is set directly by the keyboard. One program can use many variables which can be distinguished by their variable number.

● Variable expression

Use “#”+variable number to express;

Format: # i (i=200, 202, 203,);

Example: #205, #209, #225.

Besides, the expression can be used to specify the variable number. At the moment, the expression must be in the brackets.

Example: #[#20+#30/4]

● Variable reference

1, use variable to permute the number following address

Format: < address > + “# i” or < address > + “-# i” means to take the variable value or the negative value of value of the variable as the address value

Example: F#203...#203=15: it is the same those of F15 functions;

Z-#210...#210=250: it is the same those of Z-250 functions;

G#230...#230=3: it is the same those of G3 functions.

When the variable value is used in program, the decimal point can be omitted. Example: #1=123: the actual value of #1 is 123.000.

When the variable value followed the axis command address has the decimal point, the data less than the least setting unit executes the rounding. For example: #1=1.23456; the axis least setting unit is 0.001, the tool to execute G00 X#1 positions to 1.235 position.

2, use variable to permute variable number.

Format: “#”+[variable number]

Example: 5 uses #30 to execute the permutation in #5, is written to #[#30].

3, refer the undefined variable.

When the variable is not defined, it becomes the “empty” variable. When the variable #0 is empty, it is only read instead of being written.

When the system refers to the undefined variable, it ignores the variable and the word.

Example: when the variable #10 value is 0, the variable #!1 value is empty and the system executes G00 X#10 Y#11, the execution result is G00 X0, Y#11 to be ignored.

Beside using the empty to assign, the variable value is 0 in other conditions.

When #2=<空>时, #1=#2, #1=<empty>;

#1=#2*3, #1=0;

#1=#2+#2, #1=0;

<empty> in conditional expression is different with 0.

When #2=<empty>, #2 EQ #0, #2 NE 0, the condition is tenable.

When #2=0, #2 EQ #0, #2 NE 0, the condition is not tenable.

(2) Variable Type

The variable is divided into the different variable types according to the variable number, their use and property are different as follows:

NO.	Variable type	Function
#0	Null variable	The variable is null and is not assigned.
#1--#33	Local variable	The local variable is used to store data in the macro program, such as result. When the system is turned off, the local variable is initialized to be null. When the macro program is called, the argument assigns to the local.
#100--#199 #500--#999	Share variable	The share variable has the same meaning in the different macro program. When the system is turned off, the variable #100~#199 is initialized to be null, #500~#999 is saved and is not lost.
#1000--	System variable	The system variable is used to read all types of data when CNC runs.

(3) Variable range

The input range of the local variable and common variable is -99999999~99999999 which integer part and decimal part are up to 8-digit number. The system alarms when the assignment exceeds the valid range. The system alarms when the assignment value exceeds its range. The middle result in the macro variable count can be more than the valid input digital.

Note:

- (1) The variable cannot be referred to address O and N. The system cannot use O#200, N#220 to execute the programming;
- (2) When the variable exceeds the max. command value defined by the address, it cannot be used; for example: #230 = 120: M#230 exceeds the max. command value.
- (3) The system cannot identify -0 and + 0. # 4 = - 0: X # 4 is taken as X 0.
- (4) When the variable is used to the address data, the other except for the valid digit is rounded.
- (5) The number followed by the address can use <Formular> to replace. The system takes "Word address [<Formular>]" or word address-<Formular>" as a program, and take <Formular> value or its negative value as the command value of the address.
- (6) The decimal point which defines the variable in a program can be omitted. For example, #1=123 is defined, the actual value of #1 is 123.000;
- (7) The negative sign of variable value which changes the reference should be placed in the front of #, such as G00X-#1;
- (8) The variable #1--#33, #100--#199 are cleared out after they reset, which are set by NO.6001Bit7 (CLV) and Bit6 (CCV), and which cannot be executed in MDI mode.
- (9) When the variable value overflows, the command address referring to the variable is ignored.
- (10) NO.6000 Bit5 (SBM) sets whether the single block stop is valid in user macro program.
- (11) The number in expression (including brackets) can be omitted. For example, X[10] actual value is X10.000.

3.15.2 System variable

The system variable is used to read and write NC internal data. For example, some system variable only read the tool offset value and current position data. The system variable is the base of the automatic control and general machining program development.

(1) Interface signal

The interface signal can program the exchange message between the machine controller and user macro programs, i.e. it completes the exchange with PLC by G, F signals and the interfaces with IO are defined by PLC.

The input signal can be only read, and the output signal can be read and written.

System variable of interface signal		
Variable number	Function	Corresponding G, F signals
#1000--#1015	Read the signal with 16 bits according to its bit from PLC to user macro program.	corresponding to G54.0—G54.7, G55.0—G55.7 signal states
#1032	Read the signal with 6 bits one time.	Corresponding to G54,G55 signal states
#1100--#1115	Write the signal with 16 bits according to its bit to PLC.	Corresponding to F54.0—F54.7, F55.0—F55.7 signal states
#1132	Write the signal with 16 bits to PLC one time.	Corresponding to F54, F55 signal states
#1133	Write the signal with 32 bits to PLC one time. Specify from -99999999 to +99999999	Corresponding to F56, F57, F58, F59 signal states

(2) Tool compensation value

The system variable can read/write the tool compensation value. The system variable of the tool compensation storage area is 1501—2999. The variable numbers divided exactly in the above range are illegal. The variable number of 2201—2299, 2901—2999 alarm. The concrete range are referred to the following table.

Set the axis number to be $n(1-5)$, the compensation number to be $m(1-99)$, the offset variable number of the axis to be $1600+(n-1)*100+m$, the wear variable number to be $2300+(n-1)*100+m$.

Compensation number	1 st axis		2 nd axis		3 rd axis		4 th axis	
	Offset	Wear	Offset	Wear	Offset	Wear	Offset	Wear
1	1601	2301	1701	2401	1801	2501	1901	2601
...
99	1699	2399	1799	2499	1899	2599	1999	2699

Compensation number	5 th axis		Radius compensation value R		Tool nose T
	Offset	Wear	Wear	wear	
1	2001	2701	2101	2801	1501
...
99	2099	2799	2199	2899	1599

(3) Marco program alarm

There is the alarm and the alarm message specified by the user in program. The variable is only written instead of being read.

Variable	Function
#3000	<p>When the system executes the assignment statement of #3000=XXX, it stops the run and alarms.</p> <p>The alarm message only displays 26 characters(13 Chinese characters), and the system only displays the first 26 characters when there are more than it.</p> <p>The value of the alarm number being #3000 adds 3000, the alarm range is 3000 to 3200.</p> <p>When #3000 value is less than 0, the alarm number is 3000, when #3000 value is more than 200, the alarm number is 3200.</p>

Example:

#3000=6; the tool has not found

When the system executes the block, it stops and alarms and the alarm number is 3006. The alarm message is "TOOL NOT FOUND", The system maybe alarm in advance because of the buffer exists.

The alarm message can use the small brackets. For example, #3000=6(TOOL NOT FOUND). When the small brackets and the semicolon are in the block, the latter specified message is valid, such as #3000=6(TOOL NOT FOUND); TOOL NOT FOUND, the displayed message is "TOOL NOT FOUND".

(4) Stop message

The program execution is interrupted and the system displays one message. i.e. the single stops after the system executes the block, and the system displays only one prompt. The variable is only be written instead of being read.

Variable	Function
#3006	<p>When the system executes the assignment statement of #3006=1, it stops the run and displays only one prompt message.</p> <p>The alarm message only displays 26 characters(13 Chinese characters), and the system only displays the first 26 characters when there are more than it.</p> <p>The value of the alarm number being #3006 adds 3200, the prompt number range is 3201 to 3500.</p> <p>When #3006 value is less than 1, the alarm number is 3201, when #3006 value is more than 300, the alarm number is 3500.</p>

For example:

#3006=3; wait for run

When the system executes the block, it stops and displays one prompt and the prompt number is 3206. The prompt message is "WAITING FOR RUN". The format of the prompt message is the same that of description in the macro program alarm.

(5) Machine workpiece quantity

The required workpiece quantity and machined workpiece quantity are read and written

required workpiece quantity and machined workpiece quantity	
Variable	Function
#3901	Machined workpiece quantity(completed quantity)
#3902	Required workpiece quantity(target quantity)

When #3901 value is changed, the workpiece quantity displayed in POSITION window also changes.

When #3902 value is changed, No.6713 value also changes.

(6) Modal message

The previous modal message which is being processed can be read.

Variable number	Function	
#4001	G00, G01, G02, G03, G32, G34, G90, G92, G94	No. 1 group
#4002	G96, G97	No. 2 group
#4003		No. 3 group
#4004		No.4group
#4005	G98, G99	No.5group
#4006	G20, G21	No.6group
#4007	G40, G41, G42	No.7group
#4008	G25, G26	No.8group
#4009	G22, G23	No.9group
#4010	G80, G84, G88	No.10group
#4011		No.11group
#4012	G66, G67	No.12group
#4013		No.13group
#4014	G54, G55, G56, G57, G58, G59	No.14group
#4015		No.15group
#4016	G17, G18, G19	No.16group
...		...
#4022		No.22group
#4109	F command	
#4113	M command	
#4119	S command	
#4120	T command	

Example:

When the system executes #1=#4016, #1 value is 17, 18 or 19.

The system alarms when the reading/writing modal value is G command which cannot be used by the system.

(7) Current position

The position message is only read instead of being written.

Variable number	Position signal	Coordinate system	Tool compensation value
#5001--#5005	End point of block(absolute coordinate)	Workpiece coordinate system	Not including
#5021--#5025	Current position(machine coordinate)	Machine coordinate system	including
#5041--#5045	Current position(machine coordinate)	Workpiece coordinate system	including
#5061--#5065	Skip signal position	Workpiece coordinate system	including
#5081--#5085	Tool length compensation value		

The read is the position value after the last block execution.

The units digit from 1 to 5 of variable number corresponds the No. n axis.

(8) Compensation value of workpiece coordinate system

The workpiece zero offset value can be read and written.

Variable number	Function
#5201--#5205	External zero offset value
#5221--#5225	G54 workpiece zero offset value
#5241--#5245	G55 workpiece zero offset value
#5261--#5265	G56 workpiece zero offset value
#5281--#5285	G57 workpiece zero offset value
#5301--#5305	G58 workpiece zero offset value
#5321--#5325	G59 workpiece zero offset value

The units digit from 1 to 5 of variable number corresponds the No. n axis.

(9) Note

The system variable is the state value of the system, and is buffered in advance when multi cycles are executed, so, the attained system variable is the value before the multi cycle command instead of the current value to avoid using the system variable in the cycle body of the multi cycles.

3.15.3 Operation and jump command

(1) Operation command

Variables can execute all kinds of operations, and their operation command format is as following.

#i=<Expression>

The right <expression> of an operation command is a compose of constant, a variable, function and operator.

GSK988T defines the following operations and logic commands:

Function	Format	Use
assignment	#i=#j;	Assignment statement assigns #j value to #i; #i is empty when #j is empty;

addition	$\#i = \#j + \#k;$	Addition. When $\#j$ value is empty, it is taken as 0.0 value, and the following functions are the same that of it;
Subtraction	$\#i = \#j - \#k;$	Execute subtraction operation;
Multiplication	$\#i = \#j * \#k;$	Execute division operation;
Division	$\#i = \#j / \#k;$	Execute addition;
Sine	$\#i = \text{SIN}[\#j];$	Execute sine operation; Angle unit is degree;
Arc sine	$\#i = \text{ASIN}[\#j];$	Execute arc sine operation; $\#j$ value is from -1 to 1
cosine	$\#i = \text{COS}[\#j];$	Execute cosine operation ; Angle unit is degree;
Arc cosine	$\#i = \text{ACOS}[\#j];$	Execute arc cosine; $\#j$ value is from -1 to 1 Function range: $0^\circ \sim 180^\circ$
Tangent	$\#i = \text{TAN}[\#j];$	Execute tangent operation ; Angle unit is degree; $\#j$ value cannot be 0, 90, 270
Arc tangent	$\#i = \text{ATAN}[\#j]/[\#k];$	Specify the lengths of two sides, execute the arc tangent, $\#j$ is opposite with “/” to partition;
Square root	$\#i = \text{SQRT}[\#j];$	Execute square root operation; $\#j$ cannot be less than zero
Absolute value	$\#i = \text{ABS}[\#j];$	Execute absolute value operation;
Rounding	$\#i = \text{ROUND}[\#j];$	Execute rounding operation; In macro program, execute the rounding of one-digit of No., in NC statement, execute the rounding of the next digit of the least increment
FUP	$\#i = \text{FUP} [\#j];$	Floating UP integer In puls quantity, $\#i$ is more than or equal to $\#j$, in the negative, $\#i$ is less than or equal to $\#j$
FIX	$\#i = \text{FIX} [\#j];$	Floating FIX integer In puls quantity, $\#i$ is less than or equal to $\#j$, in the negative, $\#i$ is more than or equal to $\#j$
Natural logarithm	$\#i = \text{LN}[\#j];$	Execute natural logarithm The system alarms when $\#j$ is zero or less than

		zero
Exponential function	#i=EXP[#j];	Execute #j exponent #j value cannot be more than 80;
OR	#i=#j OR #k;	Execute the binary logic operation of input data #j, #k cannot be less than zero When there are the decimal points in #j, #k, the decimal parts are rounded
XOR	#i=#j XOR #k;	
AND	#i=#j AND #k;	
BCD to BIN	#i=BIN[#j];	Converse the decimal data into the binary The system alarms for the data which cannot the converse
BIN to BCD	#i=BCD[#j];	Converse the binary into the decimal

Command explanation:

(1) operation sequence:

Prior	Operator and function
5	"" [" , "]"
4	"#"
3	"SIN", "SI", "ASIN", "AS", "COS", "CO", "ACOS", "AC", "TAN", "TA", "ATAN", "AT", "SQRT", "SQ", "ABS", "AB", "ROUND", "RO", "FIX", "FI", "FUP", "FU", "LN", "EXP", "EX", "BIN", "BI", "BCD", "BC",
2	"AND", "AN", ".*", "/",
1	"OR", "XOR", "XO", "+", "-",

- (2) EXP function input value cannot be more than 80, otherwise, the system alarm;
- (3) "/" character in <expression>(in the right of assignment "=" or in the bracket []) is taken as the division operator instead of optional block skip code;
- (4) The bracket "[]" can use 5-level, including the used bracket in the function, and the system alarms when it exceeds 5-level;
- (5) The angle units of the triangle function SIN, COS, ASIN, ACOS, TAN and ATAN are degrees, for example: 90°30' is 90.5 degree;
- (6) #i=ASIN[#j] value range:
When NO.6004 No. 0-digit NAT is set to 0: 90°~270°
When NO.6004 No. 0-digit NAT is set to 1: -90°~90°
When #j exceeds between -1 and 1, the system alarms and #j can be a constant.
- (7) #i=ACOS[#j] range: 0°~180°.
When #j exceeds between -1 and 1, the system alarms and #j can be a constant.
- (8) In #i= ATAN[#j]/[#k], ATAN #j and #k are the weight length of two right-angle sides as follows:
When NO.6004 No. 0-digit NAT is set to 0: 0°~360°
Example: when #1=ATAN[-1]/[-1] is specified, #1=225°.

When NO.6004 No. 0-digit NAT is set to 1: $-180^{\circ} \sim 180^{\circ}$

Example: when #1=ATAN[-1]/[-1] is specified, #1=-135°

#j, #K can be the constant.

In division or TAN[90], the division is specified to 0, P/S alarms;

- (9) The function ROUND is used to NC command or macro statement, which rounds the data with the decimal point. It is used to NC statement, which rounds according to the least setting unit; when it is used to the macro statement, which rounds No. 1-digit decimal point;

In executing #2=ROUND[#3], when #3=1.2345, the variable #2 value is 1.

In ISB increment metric input, #2=1.2345, #3=2.5456:

G00 X#2; the tool moves to 1.235mm

G00 X#3; the tool moves to 2.546mm

- (10) For FUP, FIX, when the absolute value of the integer after execution is more than that of the original, it is FUP; when it is less than that, it is FIX.

When #2=1.2, #3=-1.2

In executing #4=FUP[#2], 2.0 is assigned to #4

In executing #4=FIX[#2], 1.0 is assigned to #4

In executing #4=FUP[#3], -2.0 is assigned to #4

In executing #4=FIX[#3], -1.0 is assigned to #4

- (11) Logic operation OR, XOR, AND firstly are converted the decimal into the binary, and

are

executed in the binary by one-digit to one digit.

Range: 0~99999999, when it has the decimal point, it is ignored.

Example:

#101=10 (the binary is: 00001010)

#102=12 (the binary is: 00001100)

#103=#101 OR #102 (or the operation result is : 00001110)

The window display result of macro variable is #101=10.000000 #102=12.000000 #103=14.000000

- (12) The function BIN converses the decimal into the binary displayed in 8421 format BCD.

The system cannot display and alarms when some digit in BCD code after conversion exceeds 9.

The function BCD converses the BCD code displayed in 8421 format into the decimal.

Example 1:

#101=55 (The binary: 00110111)

#102=BIN[#101]

Macro variable window display #102=37.000000

Example 2:

#101=37 (BCD 37 corresponds to the binary : 00110111)

#102=BCD[#101]

Macro variable window display #102=55.000000

(2) Transfer and repetition commands

The transfer and the repetition commands can change the control flow, and there are three

kind of transfer and repetition operation: the unconditional transfer GOTO, the conditional transfer IF...GOTO, IF...THEN and WHILE DO repetition.

Command format:

GOTO n;

Command function:

Skip to the line number n without condition;

Command format:

IF <Logical expression> THEN <expression>;

Command function:

When the logical expression is valid, the system executes one following THEN, otherwise, it executes the next block.

Command format:

IF < Logical expression > GOTO n;

Command function:

When the logical expression is valid, the system skips the block with the line number n to execute, otherwise, it executes the next block;

Command format:

WHILE < Logical expression > DO n;

.....;

END n

Command function:

When the logical expression is valid, the system executes the block between DO and END, otherwise, its execute the block following END. The numerical value n following DO and END is used to specify the execute range label of the specified program, n value is 1, 2, 3. The system alarms when n is not 1, 2, 3.

IF, WHILE logical operation character rules are as follows:

Operator	substitute	character definition
EQ	==	(=)
NE	<>	(≠)
GT	>	(>)
GE	>=	(≥)
LT	<	(<)
LE	<=	(≤)

Notes:

(1) When the system transfers to the block with the serial number n and specifies the another exceeding the serial number range between 1 and 99999, P/S alarms, and the expression can specifies the serial number;

(2) The conditional expression must include the operator which is inserted in the middle of two variables or the variable and the constant and is closed by the bracket[]. The expression can be replaced by the variable;

(3) The number following DO and the one following END specify the execution range label of the specified program, and the label value is 1, 2, 3. The system alarms when n is not 1, 2, 3;

(4) The label (1-3) in the repetition DO—END can be used many times, but P/S alarms when there is the cross repetition(superposition in DO range);

(5) When the system specifies DO instead of WHILE statement, it creates the limitless repetition between DO and END;

(6) In using EQ, NE logical operation expression, <Empty> and zero have the different result. <Empty> is taken as the zero in +, -, * conditional expression;

(7) The macro program statement cannot be used with NC statement together, and the macro program statement definition is as follows:

Block including arithmetic or logical operation(=);

Block including the control statement(such as TOTO, DO, END);

Block including macro program call command(such as G65, G66, G67 or other G codes, M code call macro program) ;

Any blocks except for macro program statements are NC statements;

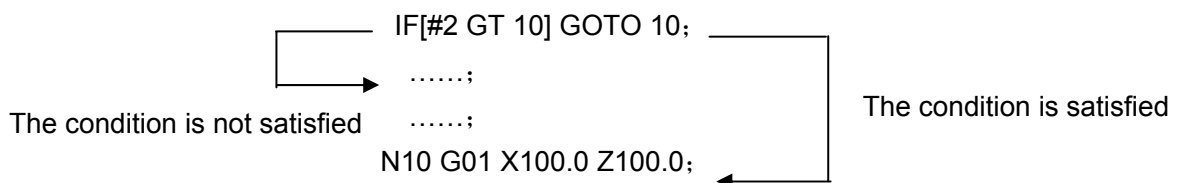
(8) The system can use the substitution character which is easily understood to replace the operator. '>', '<' can be edit in PC instead of on MDI keyboard and are uploaded into the system;

(9) When macro statement needs a line number, the line number must be compiled in the front the statement;

(10) In MDI mode, the system cannot execute the skip statement, otherwise, it alarms.

Example:

(1) GOTO example

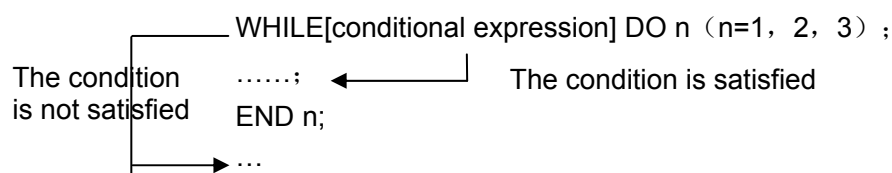


(2) IF <Logical expression> THEN <expression> example

IF[#2 EQ #3] THEN #4=0;

When #2 value is same that of #3, #4 value is 0.

(3) WHILE <Logical expression> DO n;...; ENDn example



3.15.4 Macro program statement and NC statement

The following blocks are macro program statements:

Including arithmetic or logical operation (=);

Including control statement(such as GOTO, DO, END);

Including macro program call command (G65, G66, G67).

Any NC blocks except for macro program statement are NC statements.

In Single Block mode, when No.6000 Bit5 (SBM) is set to 0, the system directly skips the macro program statement and the machine does not stop, but it is set to 1, the system stops run and enters the stop state.

One block cannot have the macro program statement and NC statement simultaneously.

3.15.5 Macro program call

(1) Non-modal call of macro program G65

Command format: G65 P __ L __ 〈argument list〉 ;

Command function: The system calls macro program L times specified by P and transfers the argument to the called macro program.

Command explanations: P: specify the macro program to be called;

L: times of calling the macro program, and its default is 1 and its range is 1—9999;

Argument list: data transferred to macro programs.

Argument specification:

Two types of argument specification are available. Argument specification I uses letters other than G, L, O, N and P once each. Argument specification II uses A, B and C once each and also uses I, J, and K up to ten times. The types of argument specification is determined automatically according to the letters used.

Argument specification I

Address	Variable No.	Address	Variable No.	Address	Variable No.
A	#1	I	#4	T	#20
B	#2	J	#5	U	#21
C	#3	K	#6	V	#22
D	#7	M	#13	W	#23
E	#8	Q	#17	X	#24
F	#9	R	#18	Y	#25
H	#11	S	#19	Z	#26

Addresses G, L, N, O and P cannot be used in arguments;

Addresses that need not be specified can be omitted and local variables corresponding to an omitted address are set to null;

Addresses do not need to be specified alphabetically. They conform to word address format.

Example: B_A_D_...J_K_ Correct

B_A_D_...K_J_ Incorrect

Argument specification II uses A, B and C once each and uses I, J, and K up to ten times. Argument specification II is used to pass values such as three-dimensional coordinates as arguments.

Argument specification II

Address	Argument No.	Address	Argument No.	Address	Argument No.
A	#1	K3	#12	J7	#23
B	#2	I4	#13	K7	#24
C	#3	J4	#14	I8	#25
I1	#4	K4	#15	J8	#26
J1	#5	I5	#16	K8	#27
K1	#6	J5	#17	I9	#28
I2	#7	K5	#18	J9	#29
J2	#8	I6	#19	K9	#30
K2	#9	J6	#20	I10	#31
I3	#10	K6	#21	J10	#32
J3	#11	I7	#22	K10	#33

Note:

- (1) G65 must be specified before any argument;
- (2) After G65, specify at address P and L. when P or L is repeated and No.3403 Bit6 (AD2) is set 0, the specification later takes precedence, otherwise, the system alarms;
- (3) Subscripts of I, J, K in the argument specification II for indicating the order of argument specification are not written in the actual program;
- (4) The CNC internally identifies argument specification I and argument specification II. If a mixture of argument specification I and argument specification II is specified, the type of argument specification specified later takes precedence.
- (5) Calls can be nested to a depth of four levels including simple calls G65 and modal calls G66. This does not include subprogram call M98.
- (6) Whether the units used for argument without a decimal point correspond to the least input increment of each address is related to the parameter DPI (No.3401#0);
- (7) G65, G66 cannot be in the same block with NC code, otherwise, the system alarms;
- (8) In macro program nesting call, the local variables from level 0 to 4 are provided for nesting. When the level of the main program is 0, each time a macro is call, the local variable level is incremented by one. The values of the local variables at the previous level are saved in the CNC. When M99 is executed in a macro program, control returns to the calling program. At that time, the values of the local variables saved when the macro was called are restored.
- (9) The line number of the command line of the macro statement must be home, otherwise, the system alarms.

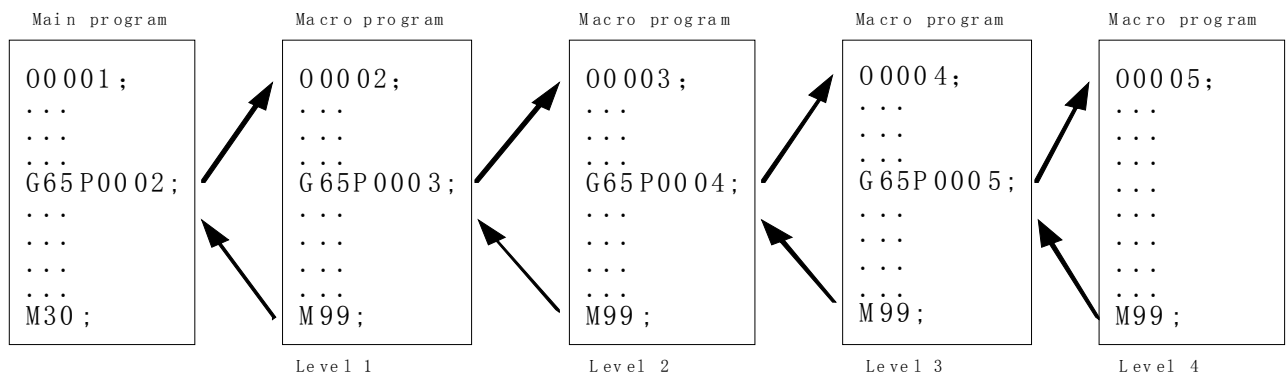
Macro program nesting example

Fig.3-58 Nesting macro program

(2) Modal call of macro program G66, G67

Command format: G66 P __ L __ 〈argument list〉 ;

.....;

G67;

Command function: set the modal message of the specified macro program L times for calling P, send the argument to the called macro program.

Command explanation:

G66: modal macro program call needs one line to be specified;

G67: call macro program call mode;

P: specify many called macro programs;

L: times for calling the macro program. It is default to 1, its range is 1—9999;

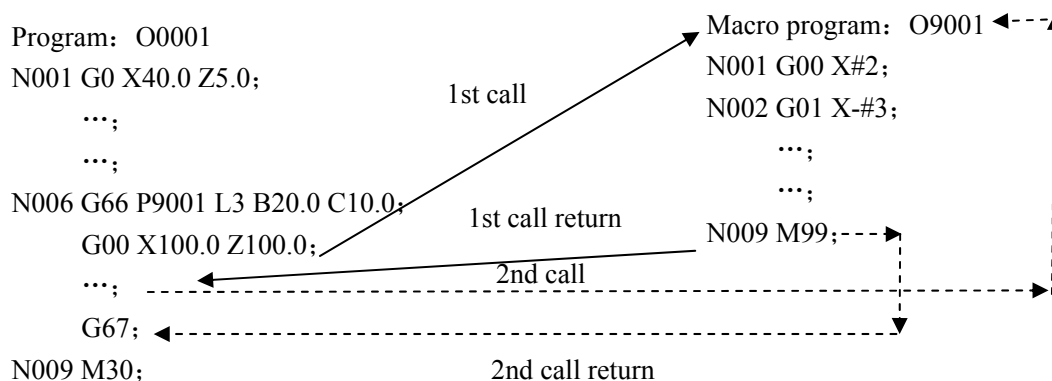
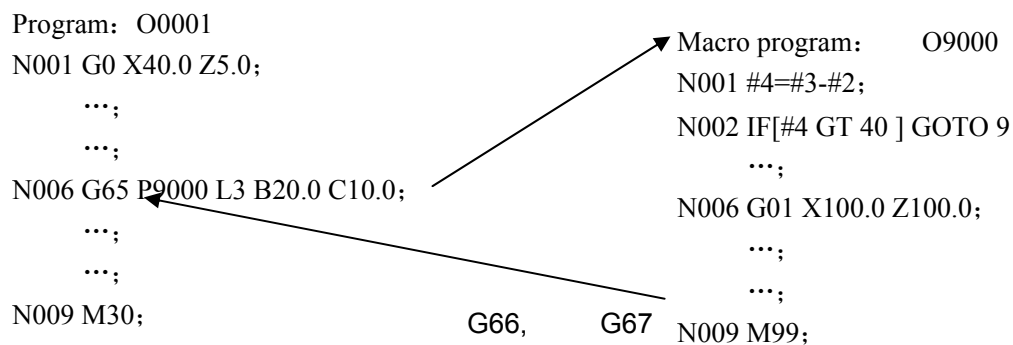
Argument list: data sending to macro program is referred to the explanations of G65.

Note:

- (1) Cannot call many macro programs in G66 block, but can call G66 again;
- (2) G66 is specified before P_, L_ and argument, and the use methods of P, L, the argument are the same those of G65;
- (3) Can't call macro program in the block without movement commands but with the auxiliary function;
- (4) The local variable (argument) is specified only in G66 block, and the system does not set it again when each modal call is executed;
- (5) Cannot specify the macro call command in MDI mode;
- (6) When the reset is executed by setting the parameter, whether the common variables of the local variables from #1 to #33 and from #100 to #149 are cleared to the empty value.
- (7) The system clears the call state of all user macro programs and subprograms and DO state, and returns to the main program;
- (8) In executing the macro program statement, when the feed pause is valid, the machine stops after the macro statement is executed, and the machine also stops when the system resets or alarms.

Application example:

(1) G65 example



CHAPTER 4 TOOL NOSE RADIUS COMPENSATION

4.1 Application

4.1.1 Overview

Part program is compiled generally for one point of tool according to a workpiece contour. The point is generally regarded as the tool nose A point in an imaginary state (there is no imaginary tool nose point in fact and the tool nose radius can be omitted when using the imaginary tool nose point to program) or as the center point of tool nose arc (as Fig. 4-1). Its nose of turning tool is not the imaginary point but one arc owing to the processing and other requirement in the practical machining. There is an error between the actual cutting point and the desired cutting point, which will cause the over- or under-cutting affecting the part precision. So a tool nose radius compensation is needed in machining to improve the part precision.

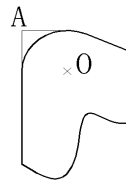


Fig. 4-1 Tool

B tool compensation is defined that a workpiece contour path is offset one tool nose radius, which cause there is excessive cutting at an intersection of two programs because of executing motion path of next after completing the previous block.

To avoid the above-mentioned ones, the system uses C tool compensation method (namely, tool nose radius compensation). The system will read the next block instead of executing it immediately after reading a block in C tool compensation method, and count corresponding motion path according to intersection of blocks. Contour can be compensated precisely because reading two blocks are pretreated as Fig.4-2.

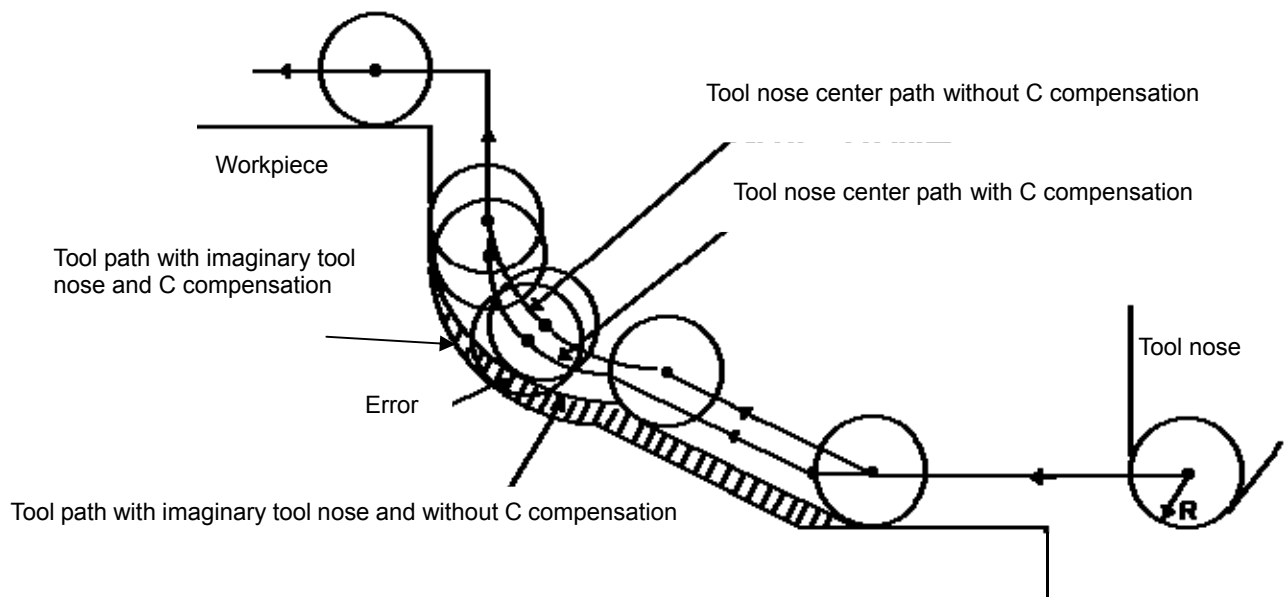
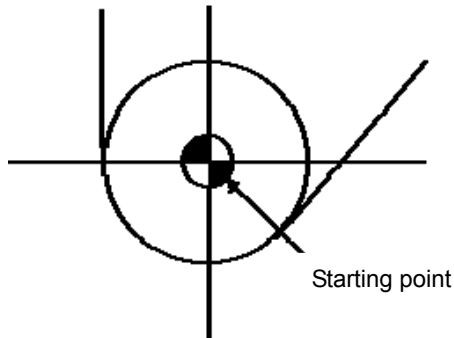


Fig. 4-2

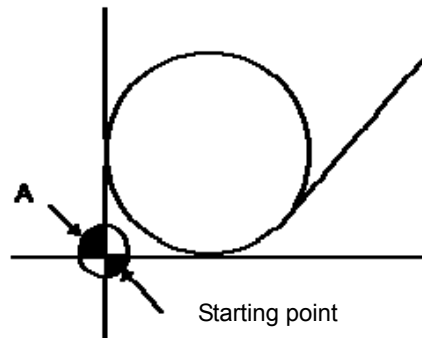
4.1.2 Imaginary tool nose direction

Suppose that it is generally difficult to set the tool nose radius center on the initial position as Fig. 4-3; suppose that it is easily set the tool nose on it as Fig. 4-4; The tool nose radius can be omitted in programming. Fig. 4-5 and Fig.4-6 correspond separately to the tool paths of tool nose center programming and imaginary tool nose programming when tool nose radius is executed or not.



Programming with tool nose center

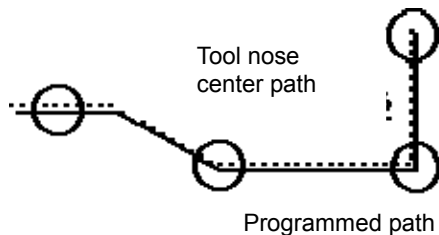
Fig. 4-3



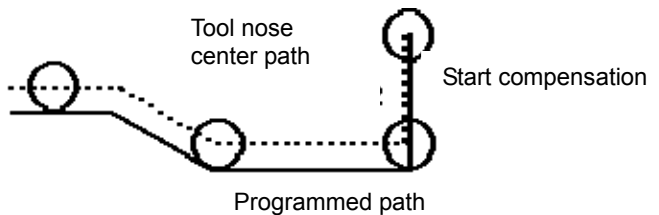
Programming with imaginary tool nose

Fig. 4-4

Tool nose path is the same as programming path without using tool nose radius compensation



Finishing when using tool nose radius compensation



Tool nose path is the same as programming path without using tool nose radius compensation

Finishing when using tool nose radius compensation

Fig. 4-5 Tool path in tool nose center programming

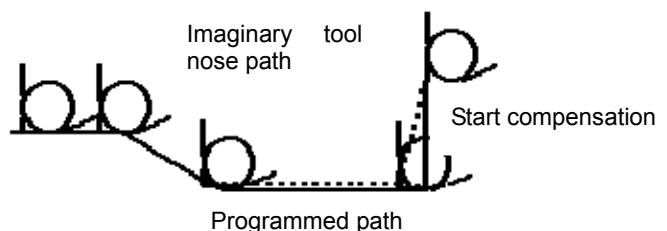
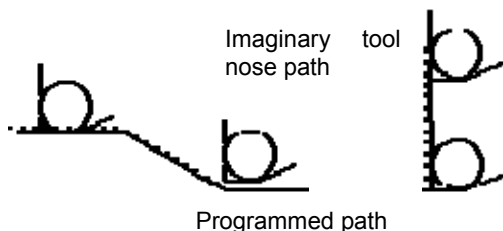
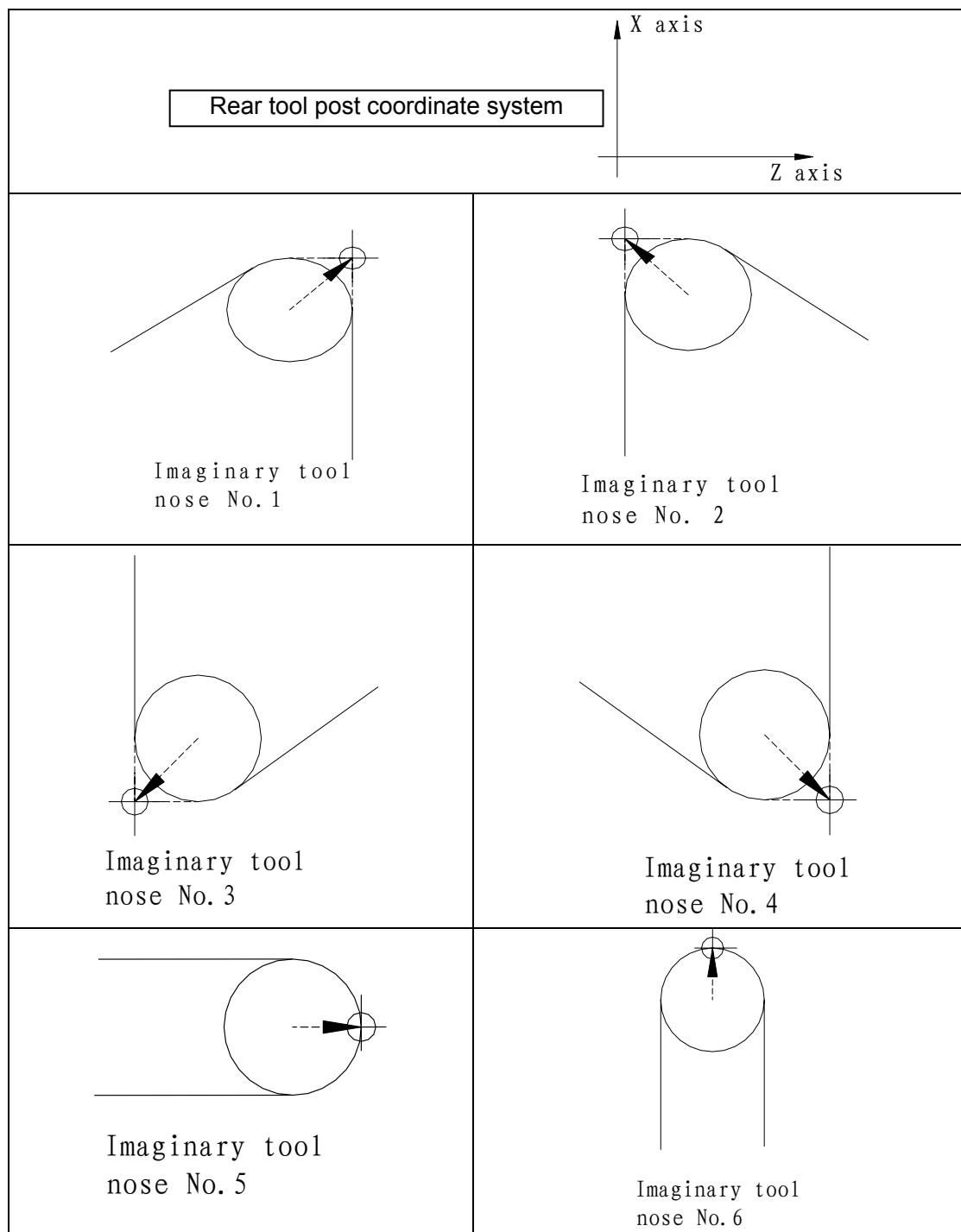


Fig. 4-6 Tool path in imaginary tool nose programming

The tool is supposed to one point in programming but the actual cutting blade is not one ideal point owing to machining technology. Because the cutting blade is not one point but one circular, machining error is caused which can be deleted by tool nose circular radius compensation. In actual machining, suppose that there are different position relationship between tool nose point and tool nose circular center point, and so it must create correct its direction of imaginary tool nose.

From tool nose center to imaginary tool nose, set imaginary tool nose numbers according to tool

direction in cutting. Suppose there are 10 (T0~T9) kinds of tool nose setting and 9 directions for position relationship. The tool nose directions are different in different coordinate system (rear tool post coordinate system and front tool post coordinate system) even if they are the same tool nose direction numbers as the following figures. In figures, it represents relationships between tool nose and starting point, and end point of arrowhead is the imaginary tool nose; T1~T8 in rear tool post coordinate system is as Fig. 4-7; T1~T8 in front tool post coordinate system is as Fig. 4-8. The tool nose center and starting point for T0 and T9 as Fig. 4-9.



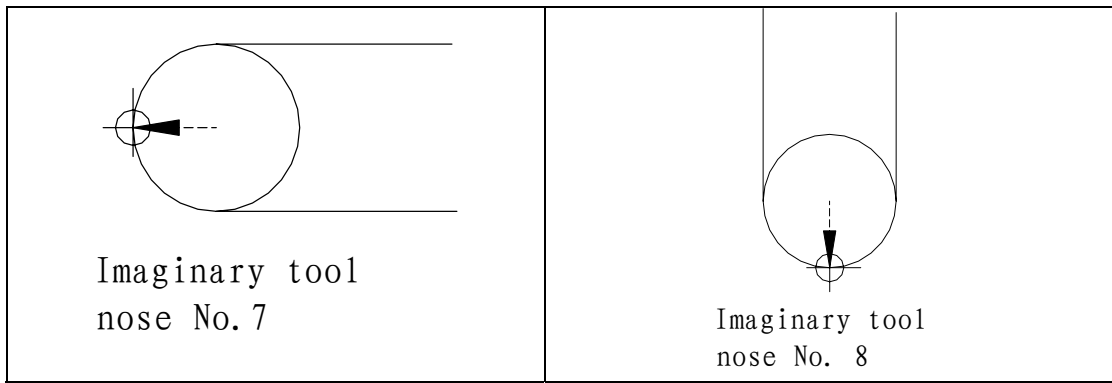
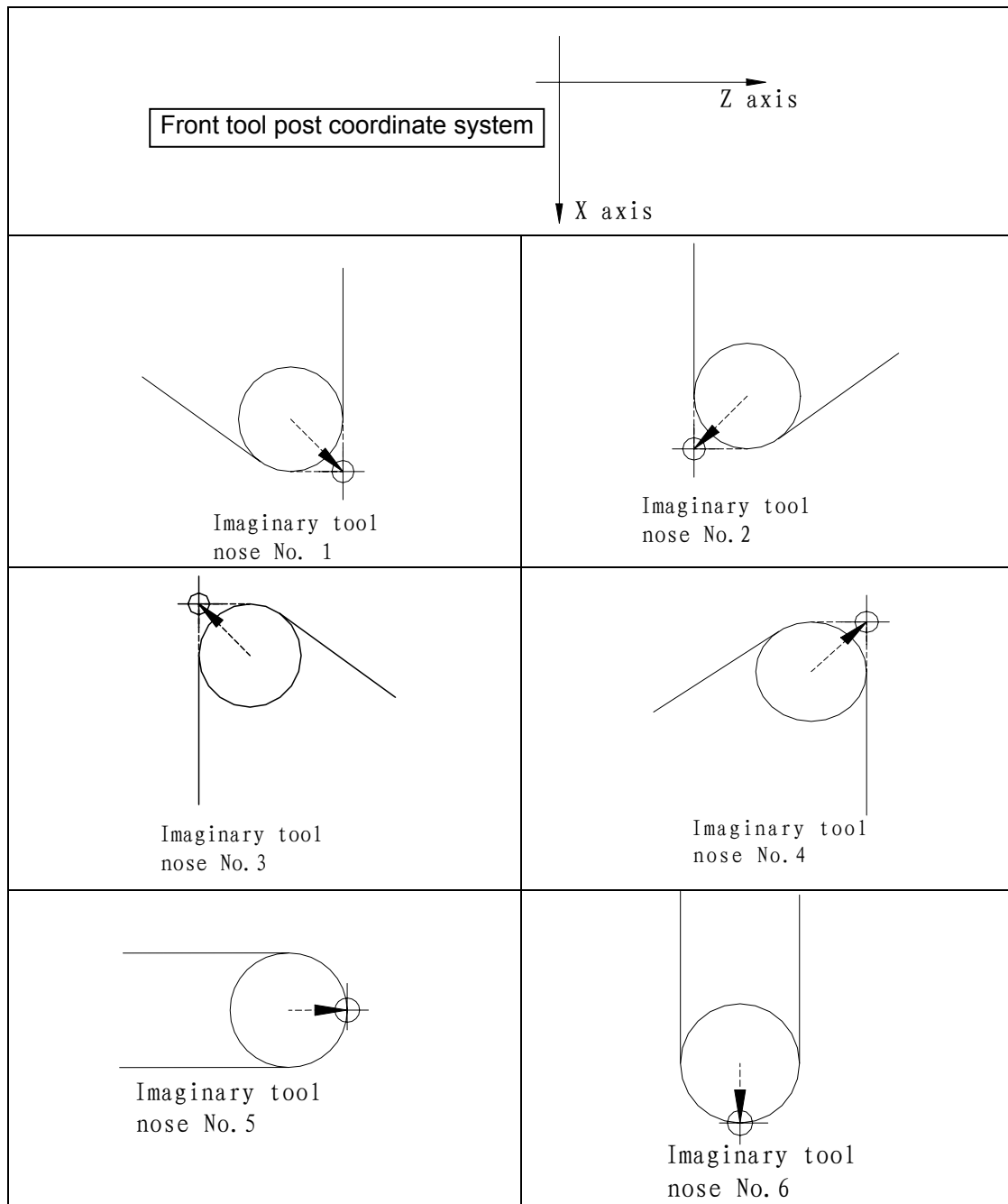


Fig. 4-7 Imaginary tool nose number in rear tool post coordinate system



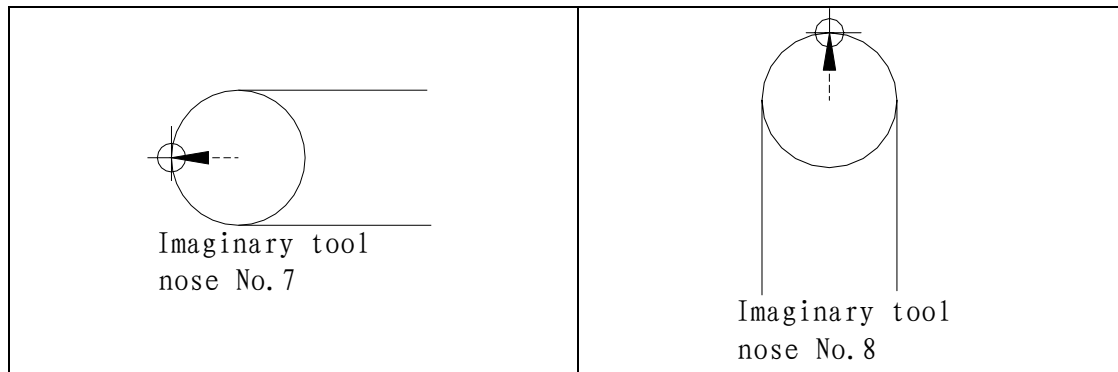


Fig. 4-8 Imaginary tool nose number in front tool post coordinate system

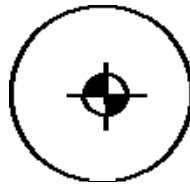


Fig. 4-9 Tool nose center on starting point

Note: the general imaginary tool nose direction 1~8 are used to G18 plane, the imaginary tool nose 0 or 9 is used to G17 and G19 planes. The imaginary tool 0 or 9 used to G18 is valid, but the imaginary tool nose direction 1~8 are used to G17 and G19 planes, the system uses the nose 0 to execute the compensation.

4.1.3 Compensation value setting

Preset imaginary tool nose number and tool nose radius value for each tool before executing tool nose radius compensation. Set the tool nose radius compensation value in “**TOOL OFFSET&WEAR**” window (as Fig. 4-1), R is tool nose radius compensation value, T is imaginary tool nose number, and the radius compensation value is the sum of offset radius and wear radius.

Table 4-1 Display window of system tool nose radius compensation value

Tool offset No.		X	Z	...	R	T
001	Offset	0.000	0.000	...	0.380	3
	Wear	0.000	0.000	...	0.000	
002	Offset	10.000	10.000	...	0.250	3
	Wear	0.020	0.040	...	0.000	
003	Offset	14.000	15.000	...	1.200	3
	Wear	1.020	0.123	...	0.000	
...	Offset
	Wear	
099	Offset	10.000	12.000	...	0.300	0
	Wear	0.050	0.058	...	0.000	

In toolsetting, the tool nose is also imaginary tool nose point of T_n (n=0~9) when taking T_n(n=0~9) as imaginary tool nose. For the same tool, offset value from standard point to tool nose radius center (imaginary tool nose is T₃) is different with that of ones from standard point to imaginary

tool nose(imaginary tool nose is T3) when T0 and T3 tool nose points are selected to toolsetting in rear tool post coordinate system, taking tool post center as standard point. It is easier to measure distances from the standard point to the tool nose radius center than from the standard point to the imaginary tool nose, and so set the tool offset value by measuring distance from the standard point to the imaginary tool nose(tool nose direction of T3).

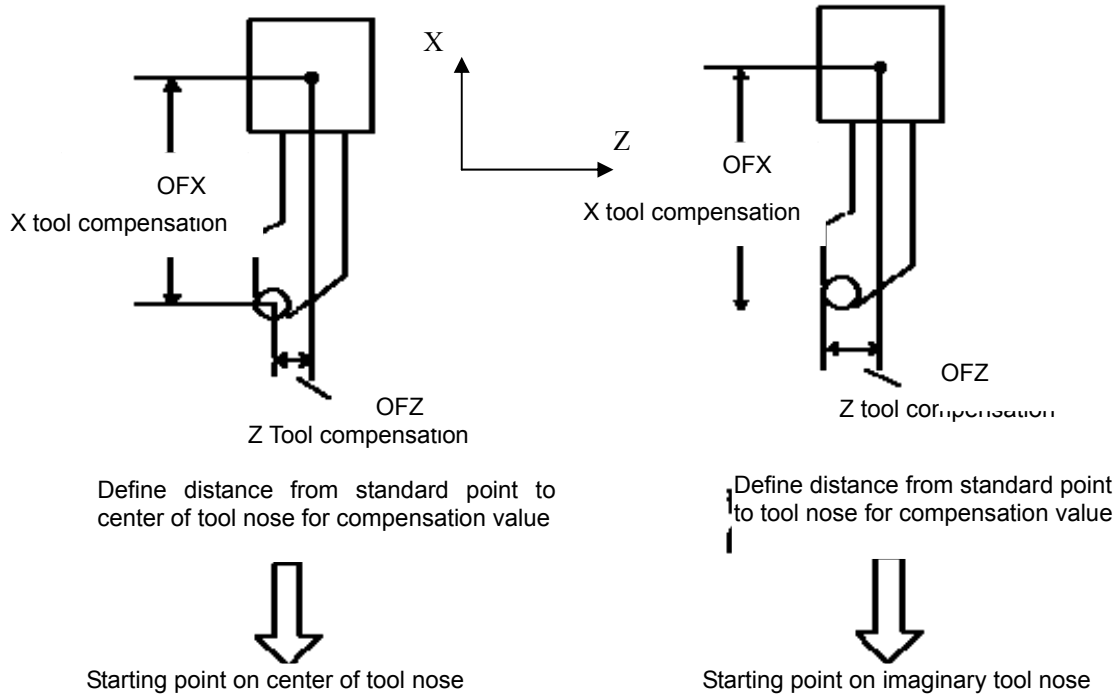


Fig. 4-10 Tool offset value of tool post center as benchmark

4.1.4 Command format

$$\left\{ \begin{matrix} G40 \\ G41 \\ G42 \end{matrix} \right\} \left\{ \begin{matrix} G00 \\ G01 \end{matrix} \right\} X_ Z_ T_ ;$$

In machining workpiece, the tool offset cannot easily compensate the precise workpiece because of the tool nose circle degree but the tool nose radius compensation function can automatically compensate the error.

G40 Xp__ Yp__ Zp__ I__ J__ K__

Command function:

Taking the previous and the current position increment as the programmed path can cancel the tool compensation mode, and its direction is the compensation direction of the previous. When the system specifies (I, J), (I, K) or (J, K), the vector defined by it can replace the current position increment to execute the count.

N1 G42 mode tool nose center moves to A point
 N2 G40 Xp__ Zp__ I__ K__ tool nose center moves to B point

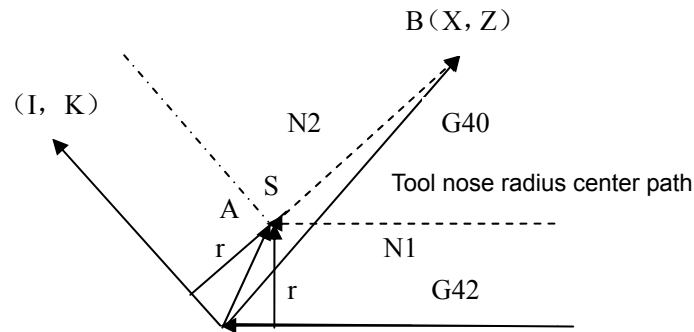


Fig. 4-11 G40 execution process

Command explanation:

Commands	Function specifications	Remark
G40	Cancel the tool nose radius compensation	See Fig.4-11 and 4-12
G41	Tool nose radius left compensation is specified by G41 in rear tool post coordinate system and tool nose radius right compensation is specified by G41 in front tool post coordinate system	
G42	Tool nose radius right compensation is specified by G42 in rear tool post coordinate system and tool nose radius left compensation is specified by G42 in front tool post coordinate system	
Xp	X and its parallel axis	
Yp	Y and its parallel axis	
Zp	Z and its parallel axis	
I	X and the cancel vector (radius value) of its parallel axis	
J	Y and the cancel vector (radius value) of its parallel axis	
K	Z and the cancel vector (radius value) of its parallel axis	

4.1.5 Compensation direction

Specify its direction according to relative position between tool nose and workpiece when executing tool nose radius compensation as Fig. 4-12 and Fig.4-13.

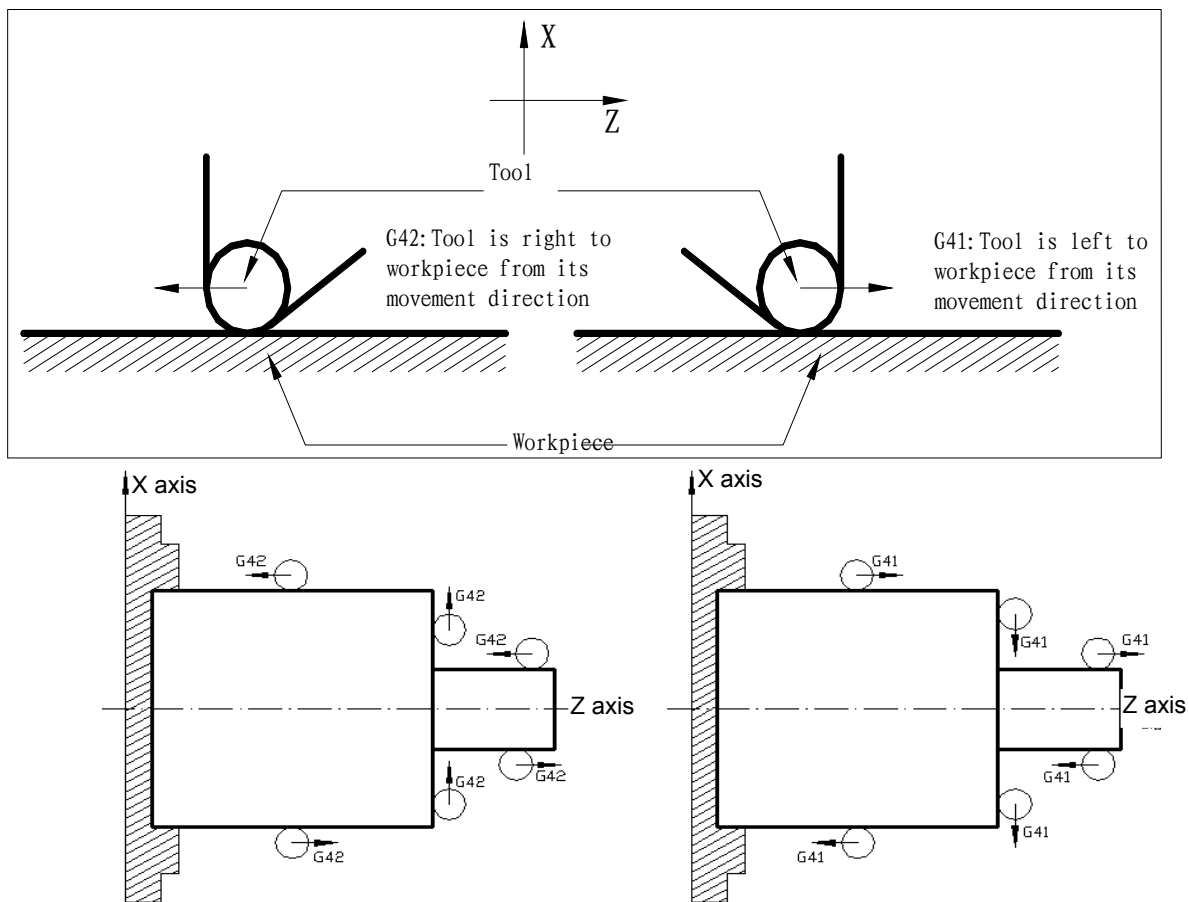


Fig. 4-12 Compensation direction of rear coordinate system

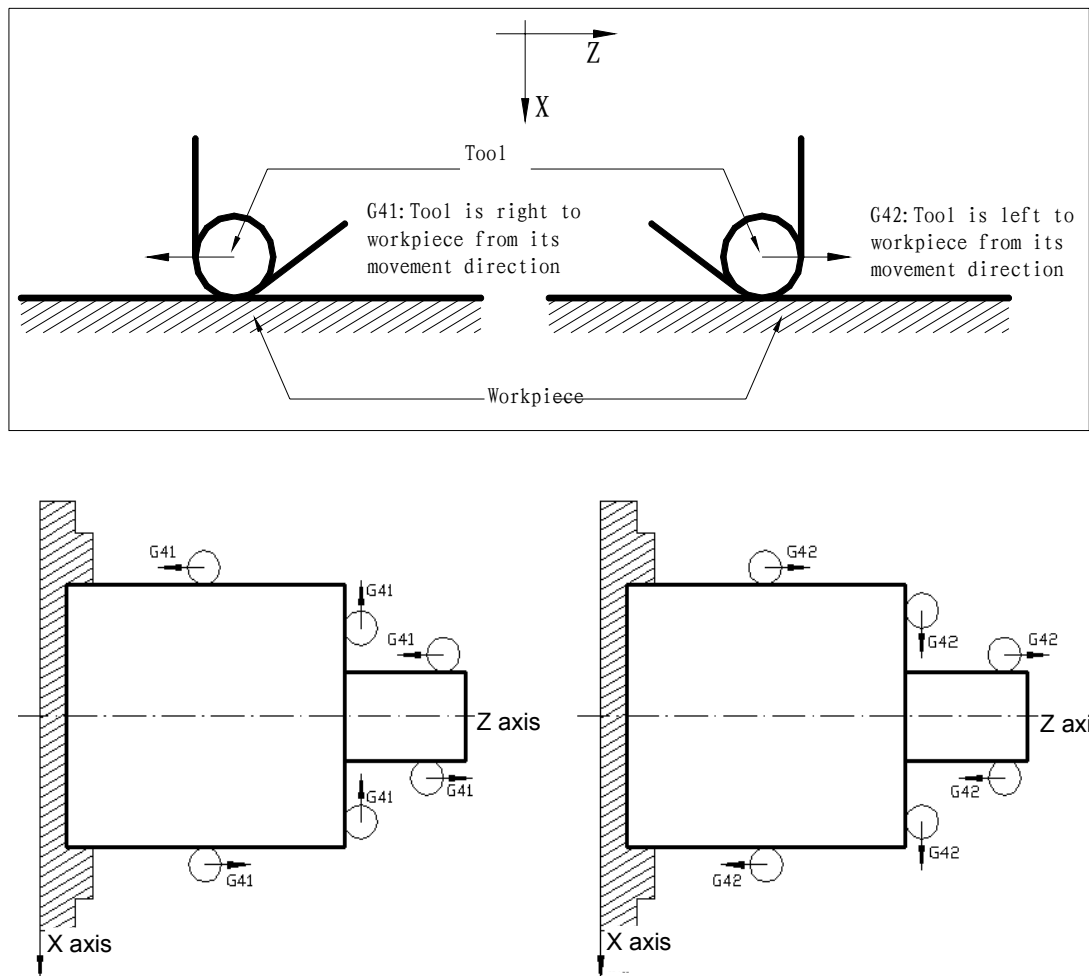


Fig. 4-13 Compensation direction of front coordinate system

4.1.6 Cautions

- In initial state, when the system is in the tool nose radius compensation cancel mode, and the offset compensation number is not 0 in G41 or G42, the system starts creating the tool nose radius compensation offset mode; when the offset compensation number is 0, G modal is the G40 state.
- In creating or cancelling tool compensation, the workpiece machining must not be executed, otherwise, it causes the overcut or undercut. The system takes the created first movement and the last movement command before being cancelled as the cutting command in normally machining workpiece.
- The tool does not create the offset and starts compensation in the next movement command when there is no movement command in creating the tool compensation. When there is no movement command in cancelling tool compensation, the tool does not create the offset and the system cancels the compensation vector in the next movement command.
- The next block to create the tool compensation block has the tool compensation cancel modal command, the system does not execute the tool compensation creation process, but at the moment, the modal command will change normally.
- The tool nose radius compensation creation and cancel only use G00 or G01 instead of G02

or G03. When they are specified, No.252 alarms.

- In tool nose radius compensation, the tool nose center moves to the end point of the last block and is vertical with the programmed path of the last when the system executes 3 or more than 3 blocks without movement command. At the moment, the overcut or undercut creates and the system should not machine the workpiece in the next block in programming. When 3 or more than 3 blocks without movement command following the movement command to create the tool nose radius compensation, the system does not create immediately the tool nose radius compensation but does it after the non-movement command.

- The system does not execute the tool nose radius compensation in G50, G52, G32, G34, G92, G71, G72, G73, G74, G75, G76 and temporarily cancels the compensation mode. Before the system temporarily cancels the compensation execution and when the system modal is G02 or G03, No.262 alarms.

- In G40, for the inner or outer machining, the system moves to the intersection of two paths, and executes the tool nose radius compensation cancel here, and then moves to the target point after the cancel. When there is no intersection and the tool reaches the normal line position of the end point of the last block, the system cancels the tool nose radius compensation and then moves the target point after the cancel. At the moment, the overcut creates, the workpiece must not be machined.

- In tool nose radius compensation mode, the system must not be switched to other planes, otherwise, No.253 alarms.

- In tool nose radius compensation mode, the system cancels the tool compensation mode in RESET, M30 or M02 mode.

- In MDI mode, the system cannot execute the tool nose radius compensation creation and its cancel. When the system specifies the tool nose radius compensation command, it executes the command according to No.5008 Bit4(MCR). When the parameter is set to 1, the system alarms.

4.1.7 Application

Machine a workpiece in the front tool post coordinate system as Fig. 4-14. Tool number: T0101, tool nose radius $R=2$, imaginary tool nose number $T=3$.

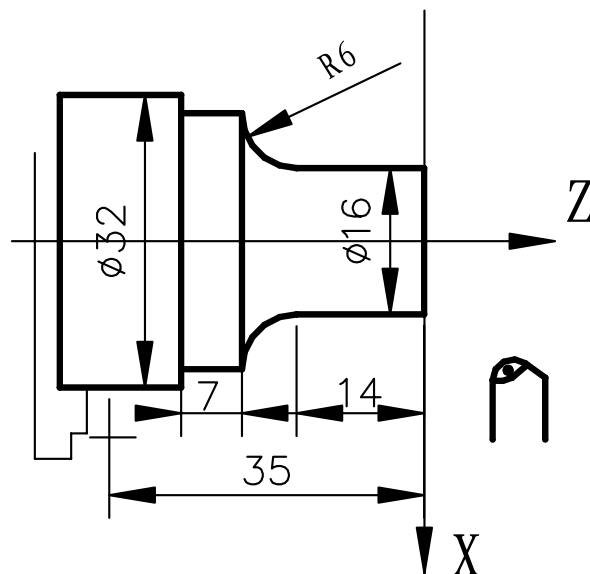


Fig. 4-14

Set the correct imaginary tool nose direction for executing the toolsetting in offset cancel

mode, Set the tool nose radius R and imaginary tool nose direction in “**TOOL OFFSET & WEAR**” window as following:

Table 3-7

No.	X	Z	Y	...	R	T
001				...	2.000	3
002
...
007

Program:

G00 X100 Z50 M3 T0101 S600; (Position, start spindle, tool change and execute tool compensation)

G42 G00 X0 Z3; (Set tool nose radius compensation)

G01 Z0 F300; (Start cutting)

X16;

Z-14 F200;

G02 X28 W-6 R6;

G01 W-7;

X32;

Z-35;

G40 G00 X90 Z40; (Cancel tool nose radius compensation)

G00 X100 Z50 T0100;

M30;

4.2 Tool nose radius compensation offset path

4.2.1 Inner and outer side

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

Outside is $0^\circ \sim 180^\circ$.

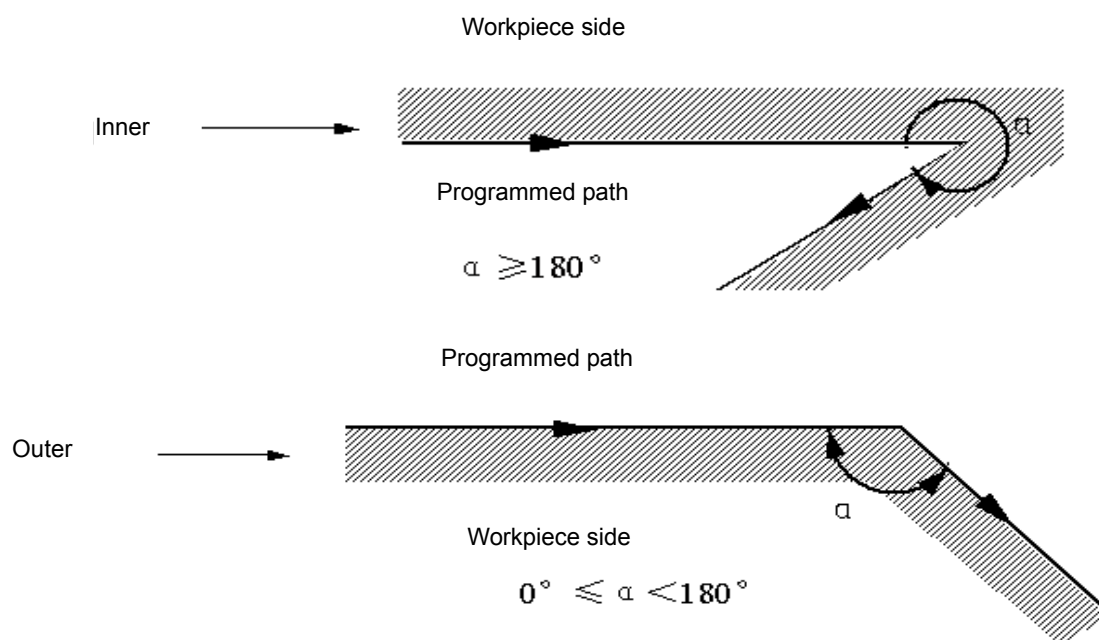


Fig. 4-15

4.2.2 Tool traversing when starting tool

3 steps to execute tool nose radius compensation: tool compensation creation, tool compensation execution and tool compensation canceling.

Tool traverse is called tool compensation creation (starting tool) from offset canceling to G41 or G42 execution.

Note: Meanings of S, L, C in the following figures are as follows:

S—Stop point of single block; **L**—linear; **C**—circular, **R** — — tool radius compensation; α —angle between two blocks.

(a) Tool traversing inside along corner($\alpha \geq 180^\circ$)

1) Linear —linear

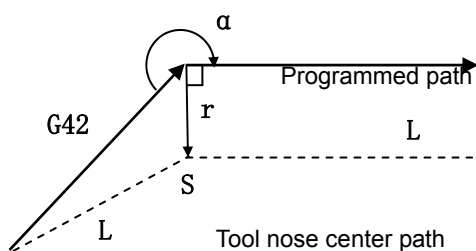


Fig.4-16 Linear —linear(starting tool inside)

2) Linear —circular

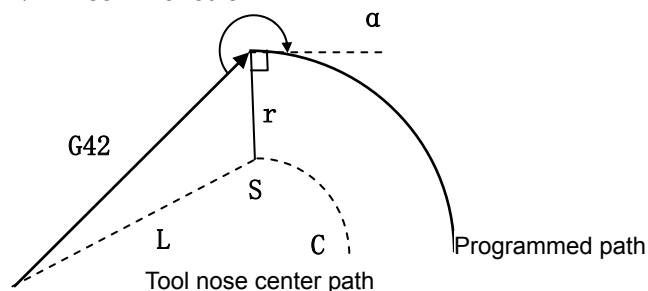


Fig. 4-17 Linear —circular (starting tool inside)

(b) Tool traversing inside along corner($180^\circ > \alpha \geq 90^\circ$)

1) Linear —linear

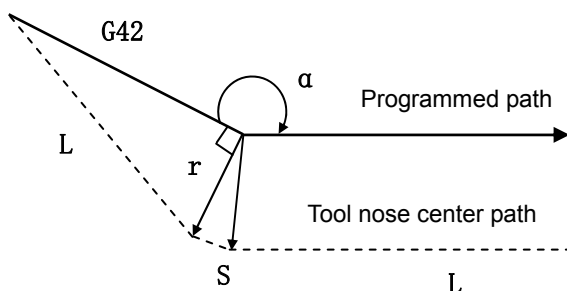


Fig.4-18 Linear —linear(starting tool outside)

2) Linear—circular

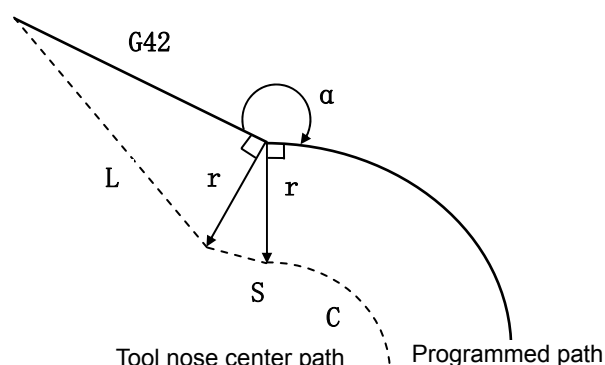


Fig.4-19 Linear—circular(starting tool outside)

(c) Tool traversing inside along corner ($\alpha < 90^\circ$)

1) Linear —linear

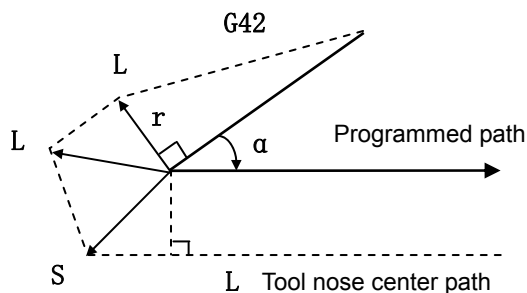


Fig.4-20 Linear —linear (starting tool outside)

2) Linear—circular

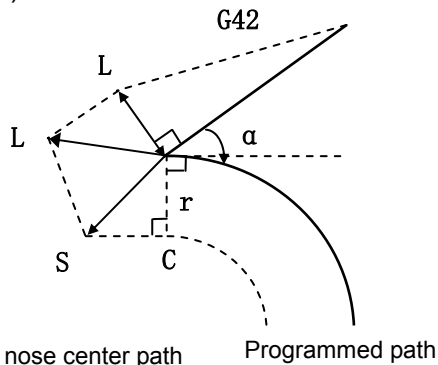
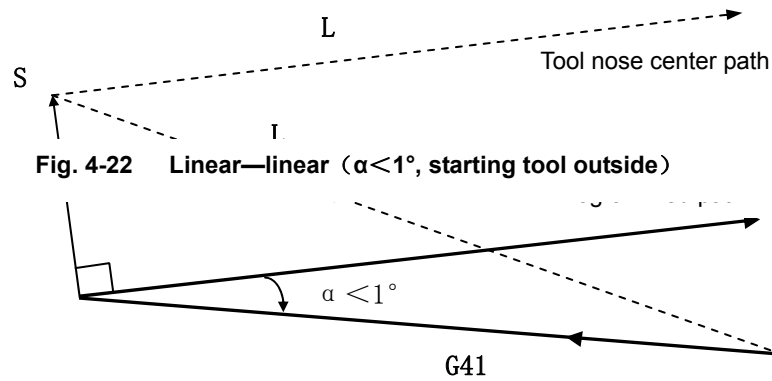


Fig. 4-21 Linear—circular (starting tool outside)

(d) Tool traversing inside along corner ($\alpha \leq 1^\circ$), linear \rightarrow linear



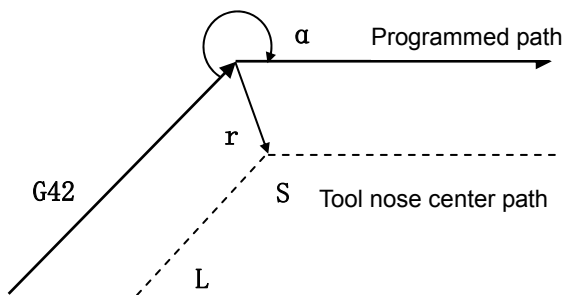
4.2.3 Tool traversing in Offset mode

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

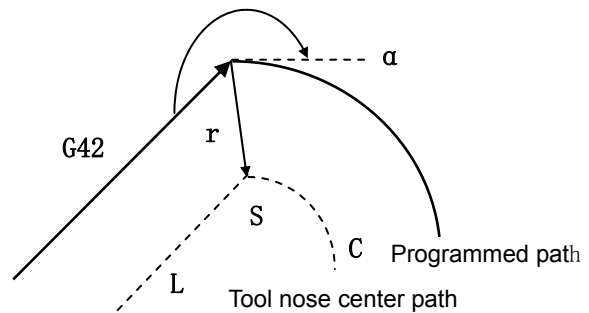
- **Offset path without changing compensation direction in compensation mode**

(a) Tool traversing inside along corner ($\alpha \geq 180^\circ$)

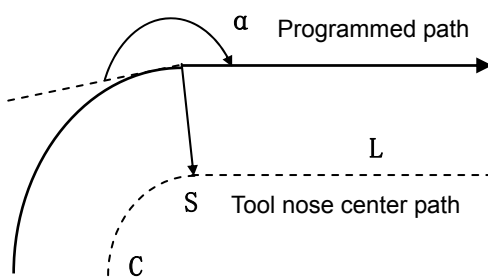
1) Linear—linear



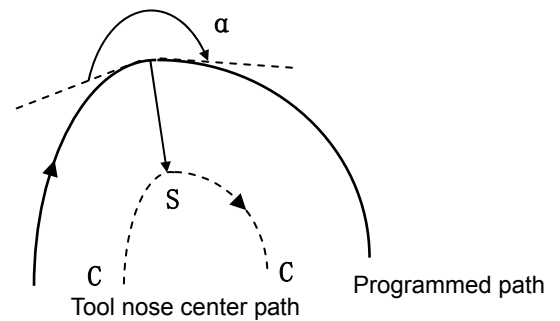
2) Linear—circular



3) Circular—linear



4) Circular—circular



Machining inside ($\alpha < 1^\circ$) and zoom in the compensation vector

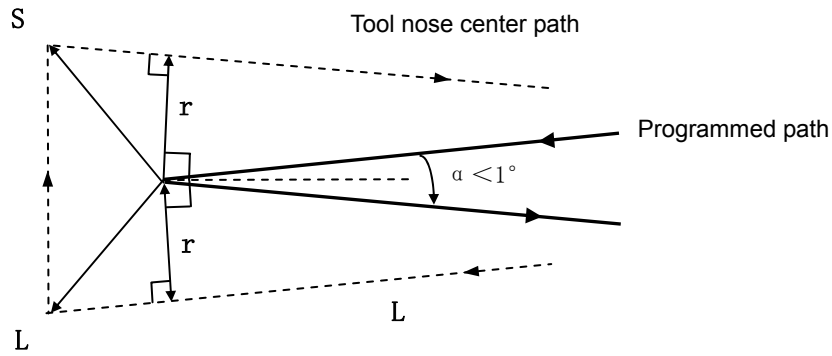


Fig. 4-27 Linear—linear ($\alpha < 1^\circ$, moving inside)

(b) Tool traversing outside along corner ($180^\circ > \alpha \geq 90^\circ$)

1) Linear—linear

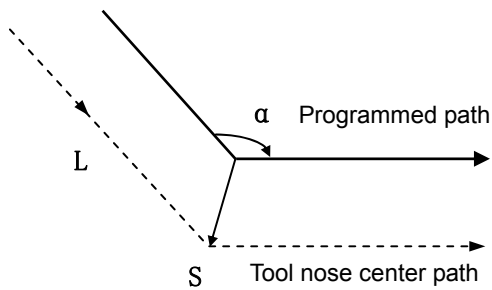


Fig. 4-28 Linear—linear (moving outside)

2) Linear—circular

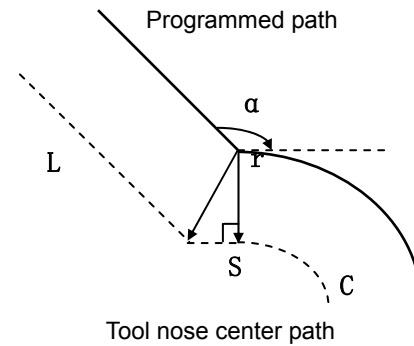


Fig. 29 Linear—circular (moving outside)

3) Circular—linear

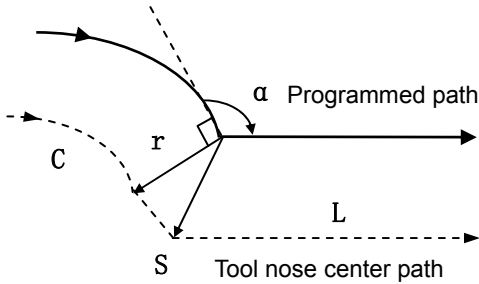


Fig. 4-30 Circular—linear (obtuse angle, moving outside)

4) Circular—circular

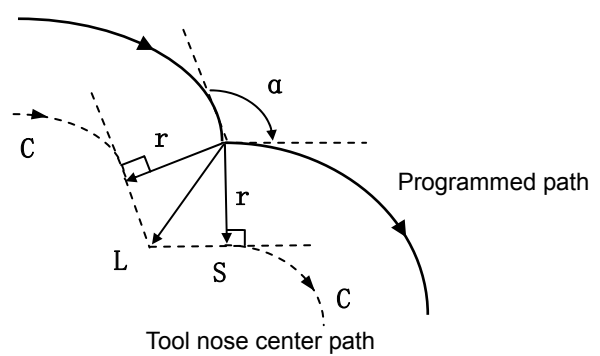


Fig. 4-31 Circular—circular (obtuse angle, moving outside)

(c) Tool traversing outside along corner ($\alpha < 90^\circ$)

1) Linear—linear

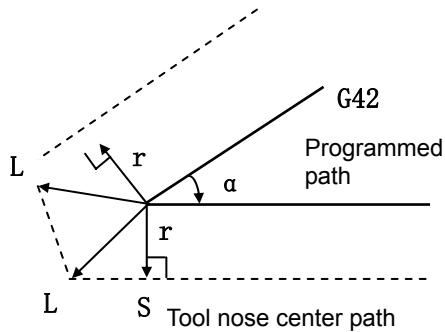


Fig. 4-32 Linear—Linear (moving outside)

2) Linear—circular

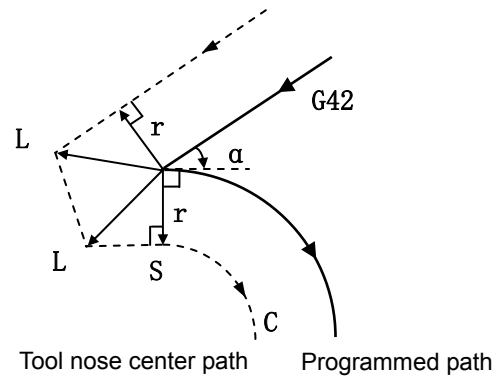


Fig. 4-33 Linear—circular (moving outside)

3) Circular—linear

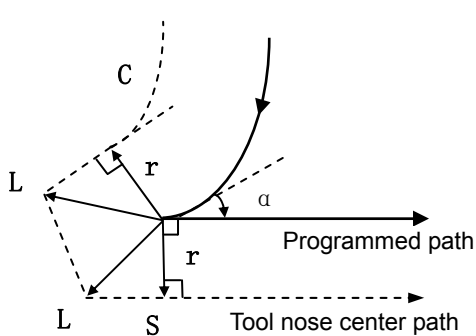


Fig. 4-34 Circular—linear (moving outside)

4) Circular—circular

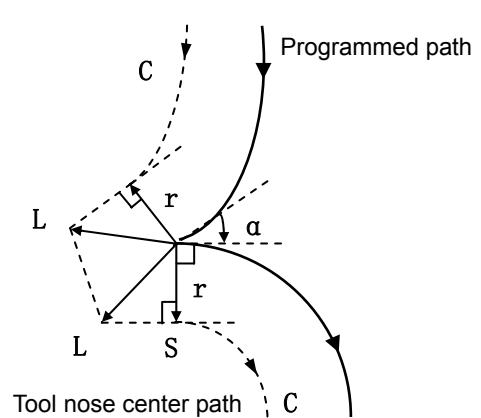


Fig. 4-35 Circular—circular (moving outside)

(d) Special cutting

1) Without intersection

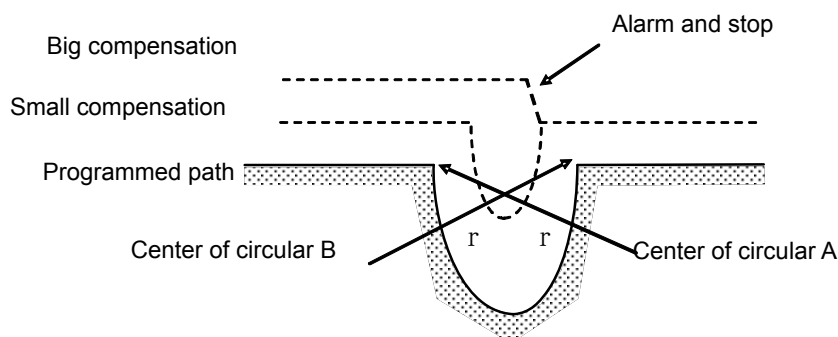


Fig. 4-36 Paths without intersection after offset

There is no intersection of compensation paths when the tool radius is small; no one when the radius is big and the tool stops at the end point of previous block and the system alarms.

2) Center point and starting point of circular being the same one

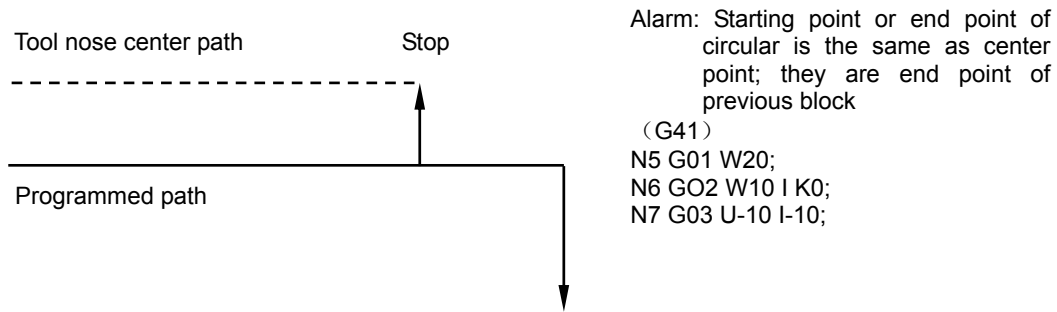


Fig. 4-37 Center point and starting point of circular being the same one

● Offset path of compensation direction in compensation mode

The compensation direction of tool nose radius is specified by G41 and G42 and the sign symbol is as follows:

Table 4-3

Sign symbol of compensation value	+	-
G Command		
G41	Left compensation	Right compensation
G42	Right compensation	Left compensation

The compensation direction can be changed in compensation mode in special cutting, it cannot be changed at starting block and its following one. There is no inside and outside cutting when the system changes the compensation direction. The following compensation value is supposed to be positive.

1) Linear—linear

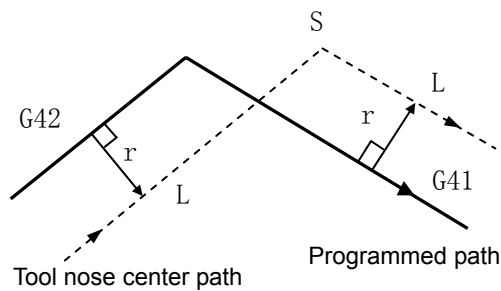


Fig. 4-38 Linear—linear
(changing compensation direction)

2) Linear—circular

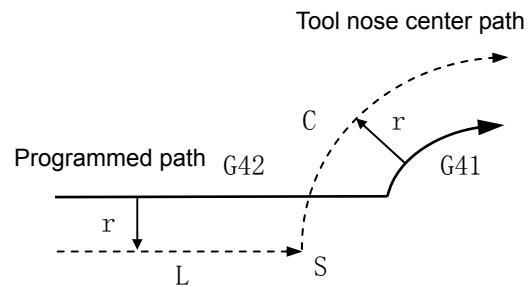
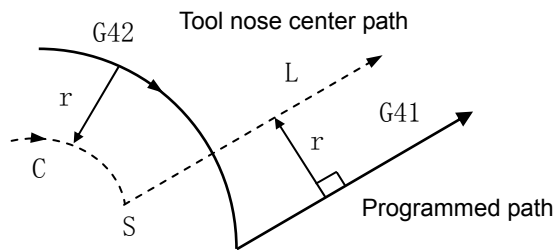


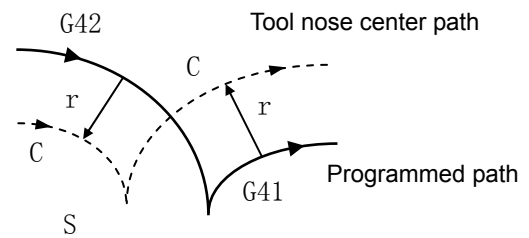
Fig. 4-39 Linear—circular
(changing compensation direction)

3) Circular—linear



**Fig. 4-40 Circular—linear
(changing compensation direction)**

4) Circular—circular



**Fig. 4-41 Circular—circular
(changing compensation direction)**

5) No intersection when compensation is executed normally

When the system executes G41 and G42 to change the offset direction between block A and B, a vector perpendicular to block B is created from its starting point.

i) Linear—Linear

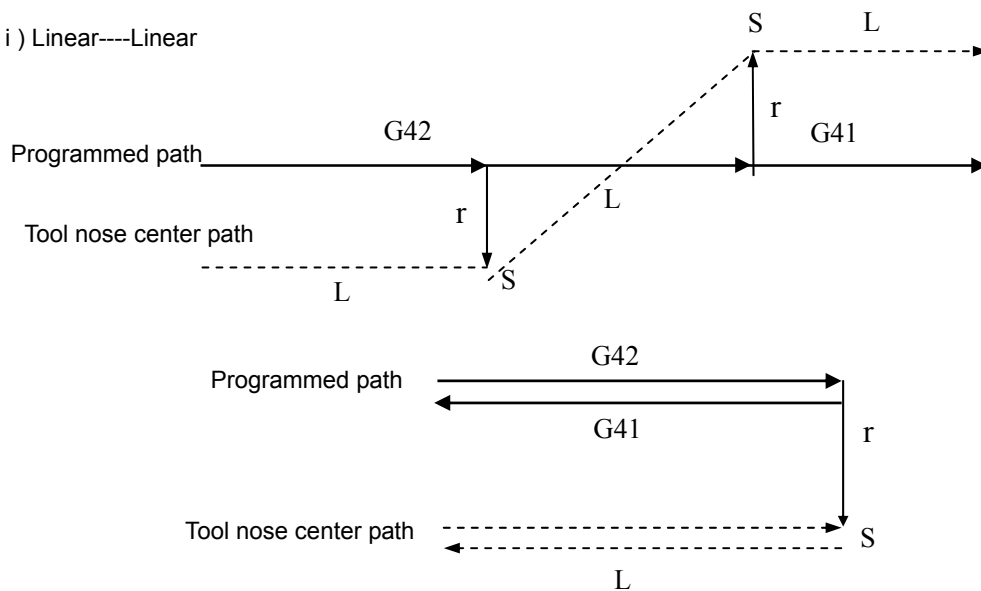


Fig. 4-42 Linear—linear, no intersection (changing compensation direction)

ii) Linear ---circular

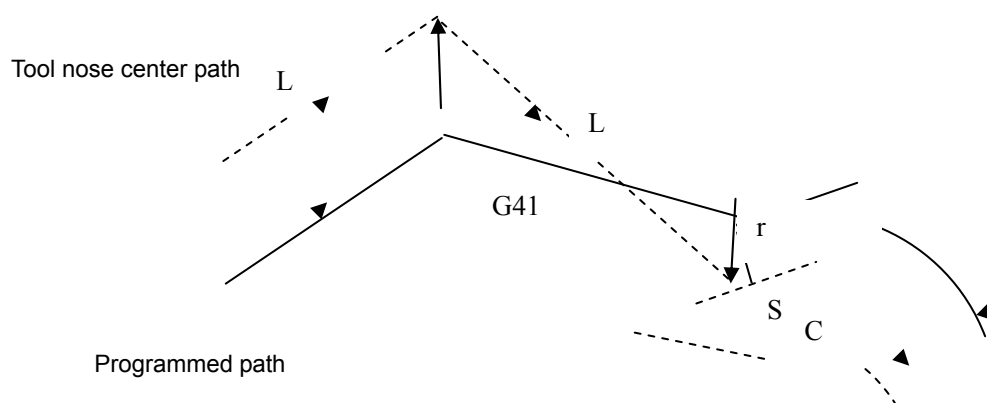


Fig. 4-43 Linear—circular without intersection (changing compensation direction)

iii) Circular-----circular

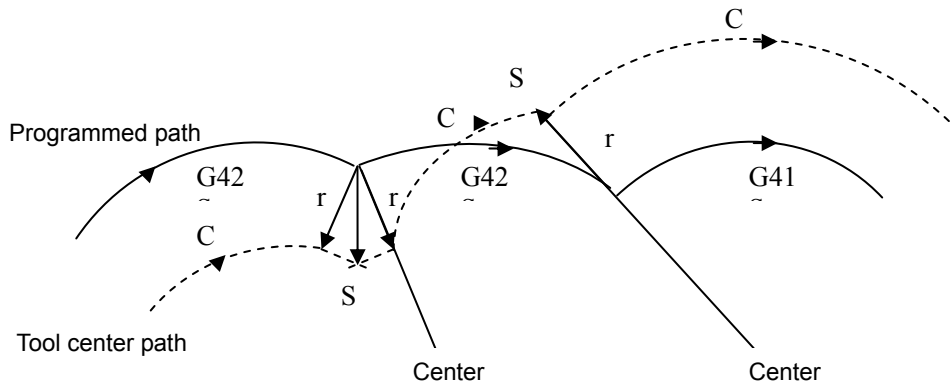


Fig. 4-44 Circular—circular without intersection (changing compensation direction)

4.2.4 Tool traversing in Offset canceling mode

In compensation mode, when the system executes G04, it enters the compensation canceling mode, which is defined to compensation canceling of block. The system cannot execute the circular command (G02 or G03) in canceling tool compensation mode, otherwise the system alarms and stops run.

(a) Tool traversing inside along corner ($\alpha \geq 180^\circ$)

1) Linear → Linear

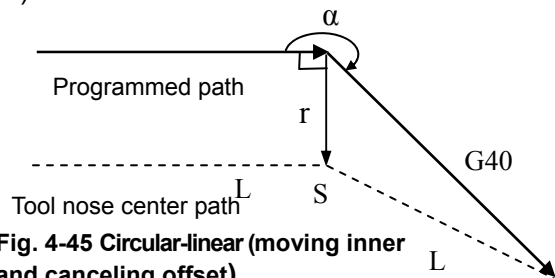


Fig. 4-45 Circular-linear (moving inner and canceling offset)

2) Circular → Linear

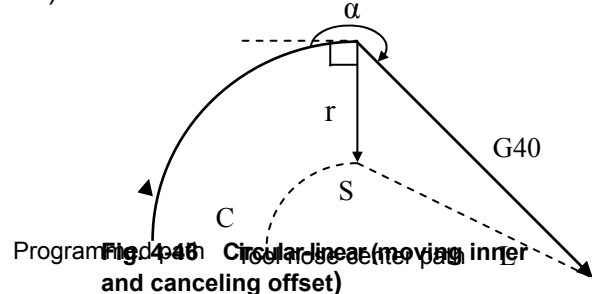


Fig. 4-46 Circular-linear (moving inner and canceling offset)

(b) Tool traversing outside along corner ($180^\circ > \alpha \geq 90^\circ$)

1) Linear → linear

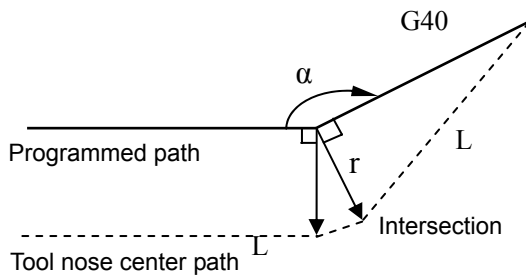


Fig. 4-47 Circular—linear (moving outside and canceling offset)

2) Circular → linear

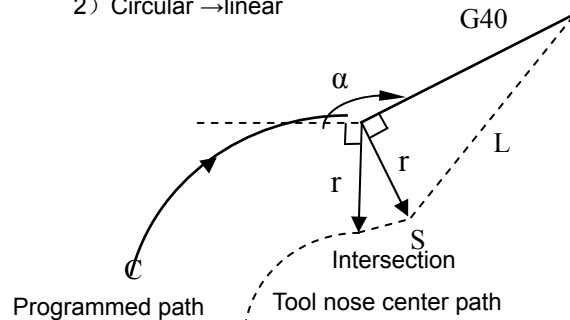
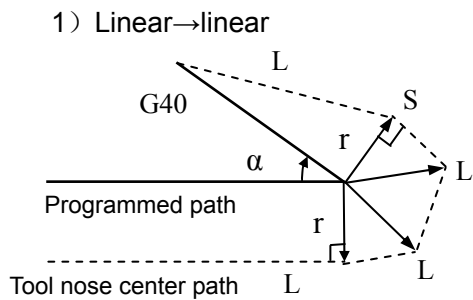
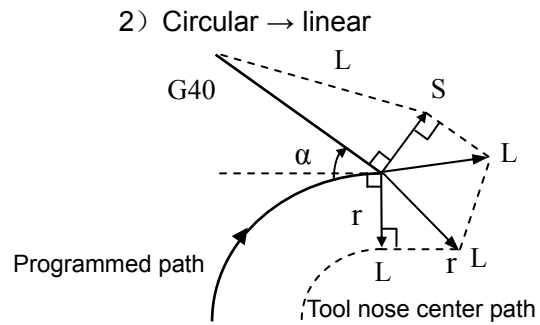
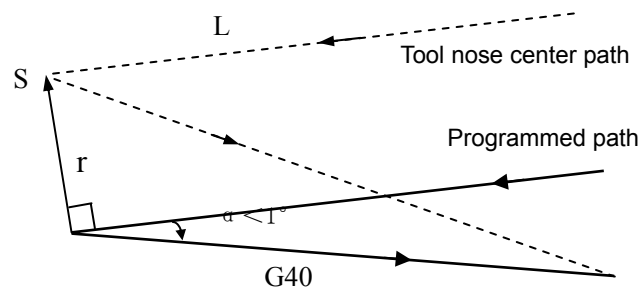


Fig. 4-48 Circular—linear (moving outside and canceling offset)

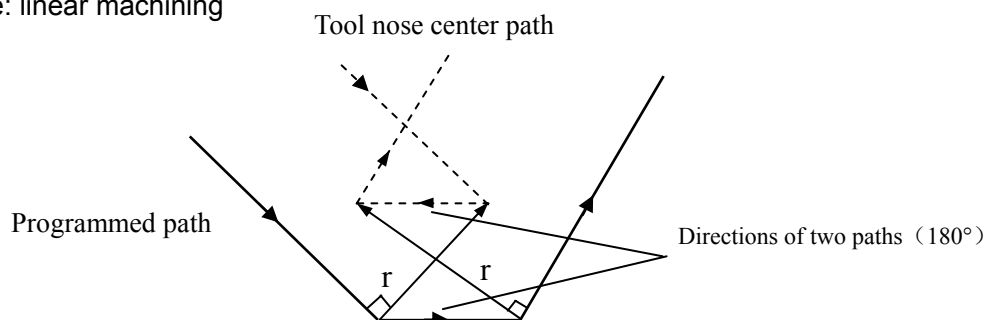
(c) Tool traversing outside along corner ($\alpha < 90^\circ$)**Fig. 4-49 Linear—linear (cutting outside and canceling offset)****Fig. 4-50 Linear—linear (cutting outside and canceling offset)****(d) Tool traversing outside along corner ($\alpha < 1^\circ$); linear \rightarrow linear****Fig. 4-51 Linear—linear ($\alpha < 1^\circ$ cutting outside and canceling offset)****4.2.5 Tool interference check**

“Interference” is defined that the tool cuts workpiece excessively and it can find out excessive cutting in advance, the interference check is executed even if the excessive cutting is not created, but the system cannot find out all tool interferences.

(1) Fundamental conditions

- 1) The tool path direction is different that of program path (angle is $90^\circ \sim 270^\circ$).
- 2) In machining arc, there is great difference the two angles ($\alpha > 180^\circ$), the one is between the starting point and the end point of the tool center path, and the other is between the starting point and the end point of the programmed path, or the system cuts the inner of the arc ($\alpha > 180^\circ$), and the tool cannot pass the entrance, No.256 alarms.

Example: linear machining

**Fig. 4-52 Machining interference, No.257 alarm appears**

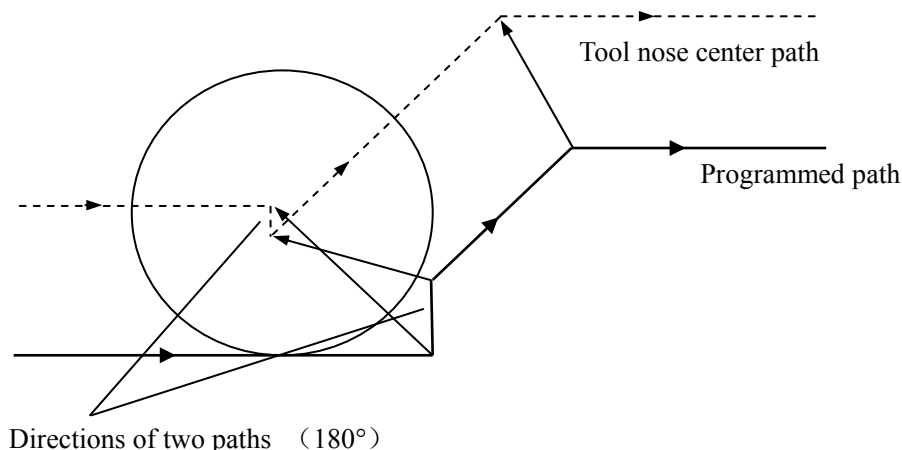


Fig. 4-53 Machining interference (2)

(2) Executing it without actual interference

1) Concave groove less than compensation value

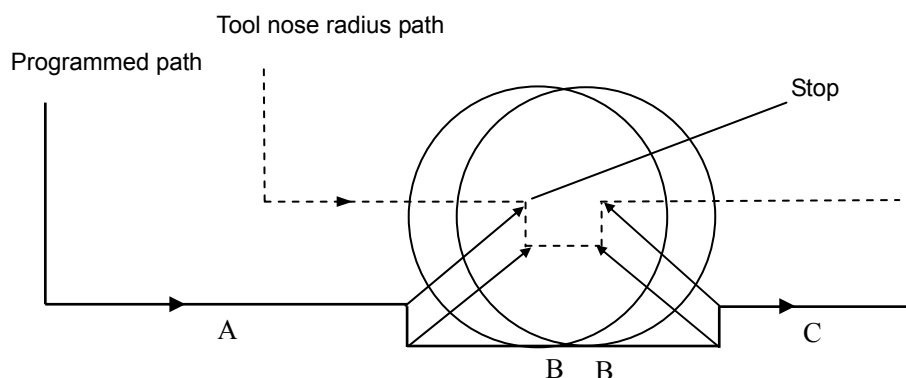


Fig. 4-54 Executing interference (1)

Directions of block B and tool nose radius compensation path are opposite without interference, the tools stops and the system alarms.

2) Concave channel less than compensation value

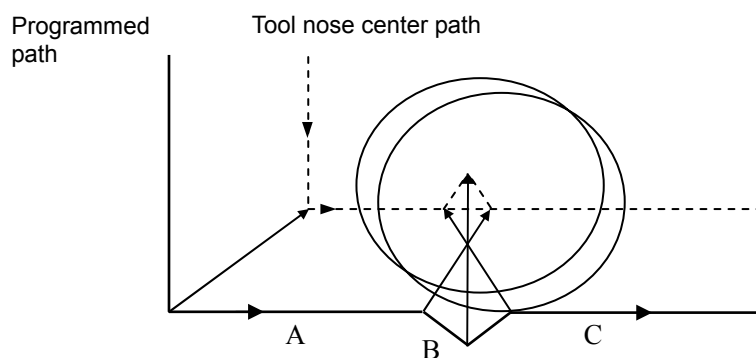


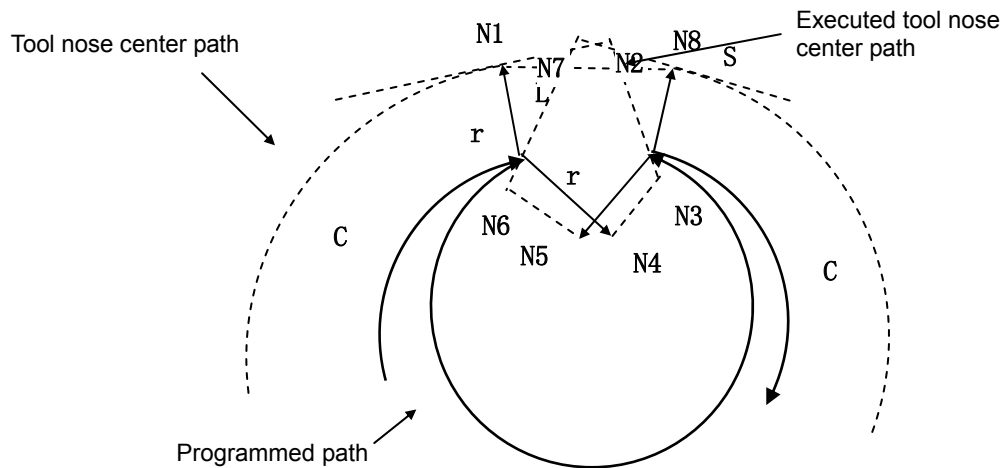
Fig. 4-55 Executing interference (2)

Directions of block B and tool nose radius compensation path are opposite without interference, the tools stops and No.257 alarms.

(3) Automatic interference vector clear

The system has the automatic interference vector clear function. For example, when the neighbor three blocks N10, N20, N30 execute the tool radius compensation, the section between N10 and N20 creates the vector V1, V2, V3 and V4, and the section between N20 and N30 creates V5, V6, V7, V8. The system executes the interference check to the last vectors in the above two group of

vector, i.e. V4 and V5. V4 and V5 are ignored when there is the interference; the system checks V3 and V6, and they are ignored when there is the interference; the system does V2 and V7, and they are ignored when there is the interference. When the system executes the interference check to the last vectors V1 and V8, and there is the interference, they cannot be ignored, the tool stops movement and the system alarms. Based on the above process, the system executes the interference check, and has checked the vector which is not interfered, the followings are not check, and the tool runs according to the path of the first group vector which does not create the interference. When the last group of vector creates the vector, they cannot be ignored, the tool stops movement and No.257 alarms.

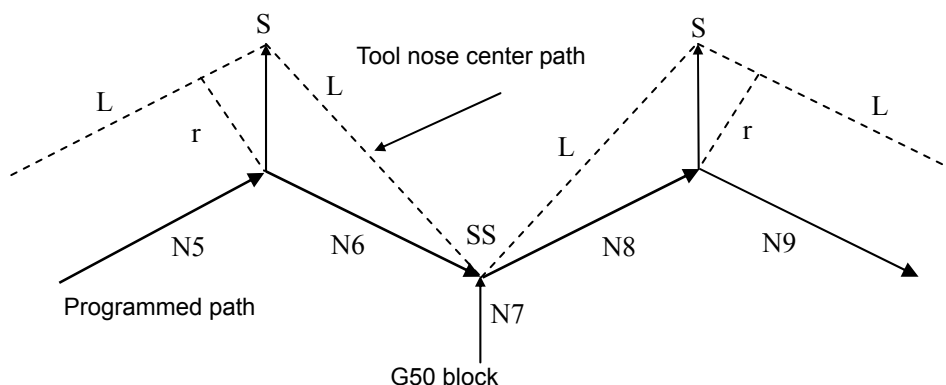
**Note:****Fig. 4-56 interference vector clear**

- (1) NO.5008 Bit 0 (CNI) can set whether the interference check is executed in tool nose radius compensation mode.
- (2) NO.5008 Bit 1 (CNC) can set whether the system alarms when the difference 90° - 270° between the movement direction and offset direction.
- (3) NO.5008 Bit 3 (CNV) can set whether the system executes the interference check and the vector clear.

4.2.6 Commands for canceling compensation vector temporarily

In compensation mode, when the system specifies G28, G30, G50, G52, G32, G34, the fixed cycle, multi cycle, drilling cycle command, the compensation vector is cancelled temporarily and is automatically resumed after executing the commands. At the moment, the compensation is cancelled temporarily and the tool directly moves from intersection to a point for canceling compensation vector. The tool directly moves again to the intersection after the compensation mode is resumed.

- **Setting coordinate system in G50, G52**

**Fig. 4-57 Temporary compensation vector in G50, G52**

Note: SS indicates a point at which the tool stops twice in Single mode.

- **Reference point automatic return G28, G30**

In compensation mode, the compensation is cancelled in a middle point and is automatically resumed after executing the reference point return in G28, G30.

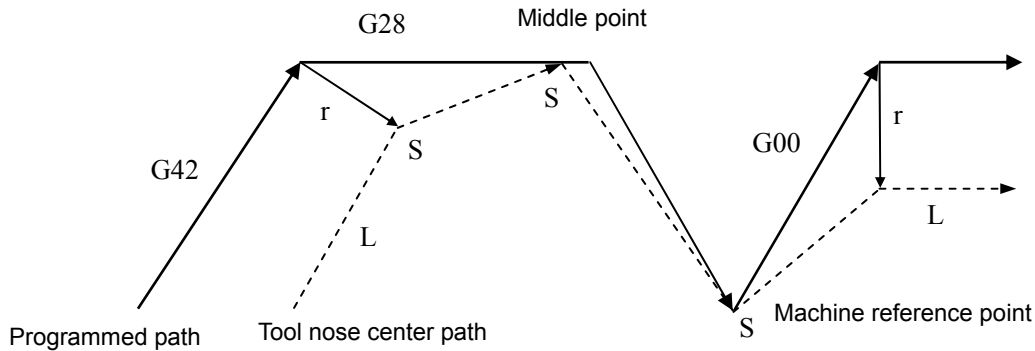
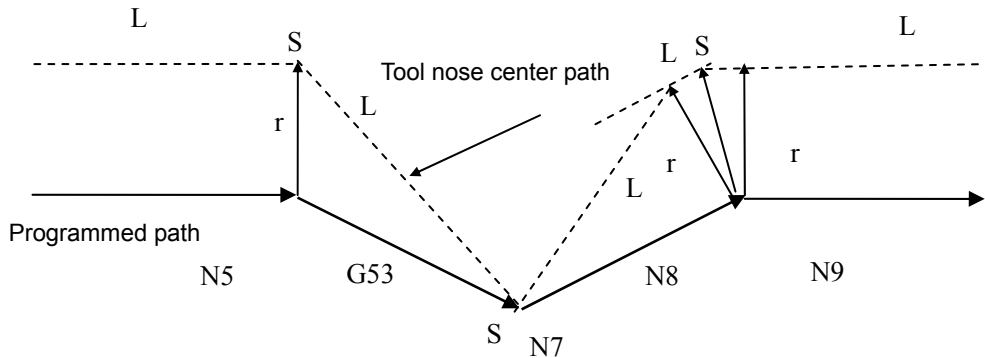


Fig. 4-58 Cancel compensation vector temporarily in G28

- **G53 automatic return to reference point**

In compensation mode, when G53 is commanded, the system creates the offset vector which is vertical with the tool motion direction before the end point of the last block. When the tool moves to G53 position, the compensation vector is cancelled. The compensation vector is automatically recovered when the system executes the next movement command.



Fi.g 4-59 G53 temporarily cancelling compensation vector

- **G71~G76 compound cycle; G92 fixed cycle, G84, G88 drilling cycle**

When executing G71~G76 , G92 fixed cycle, G84, G88 drilling cycle, the system does not execute the tool nose radius compensation and cancel it temporarily, and executes it in the next blocks of G00, G01, G70, CNC automatically recovers the compensation mode.

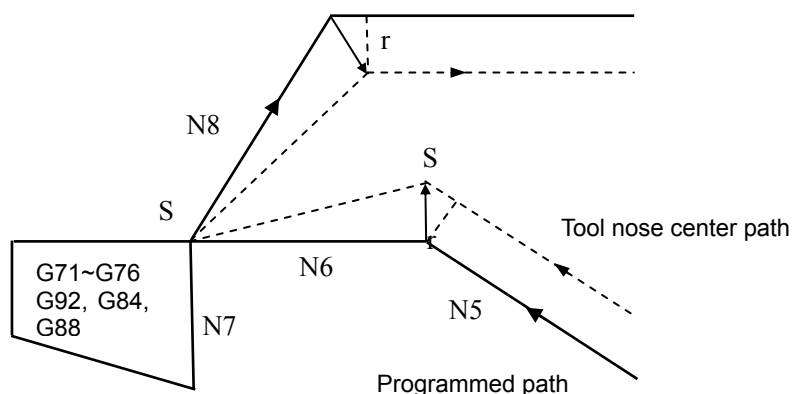


Fig. 4-60 Cancel compensation vector temporarily in cycle pause

● G32, G34 thread cutting

The system does not execute the tool nose radius compensation and temporarily cancels the tool nose radius compensation in G32, G34, and it automatically recovers the compensation mode in G00, G01.

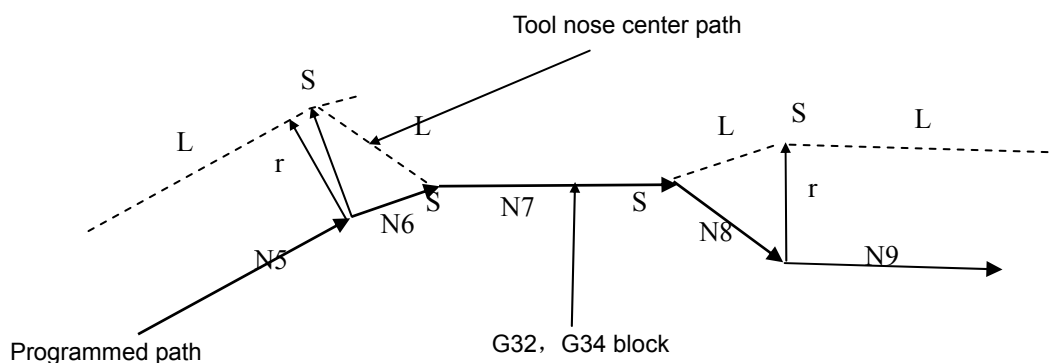


Fig.4-61 cancelling compensation vector in G32, G34 pause

● G90, G94

Compensation method of tool nose radius compensation in G90 or G94:

- Each cycle path and tool nose center path are parallel to program path.
- Offset directions are the same in G41 and G42 as the following figure, and the system determines the tool compensation direction according to the UW direction of starting point and end point, and executes the tool compensation according to the direction in the cycle process.
- In having creating C tool compensation state, the system firstly cancels C tool compensation state in G90, G94, and executes the infeed tool to the intersection point of the tool nose center based on the tool nose center parallel programmed path, and at last to the positioning point. The system creates C tool compensation again in the next G00, G01.
- After the system cancels the tool radius compensation, the imaginary tool nose point moves to the positioning point, and when the tool is in the cycle inner, the tool diameter exceeds the length of the rapid traverse of the first block, the overcut creates and No.255 alarms.

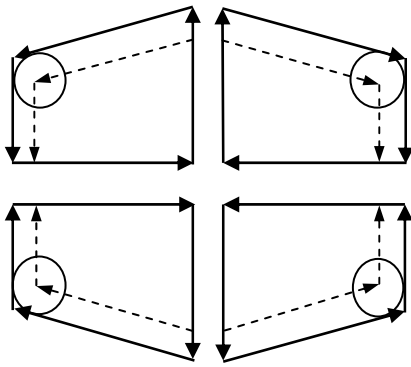


Fig. 4-62 Offset direction of tool nose radius compensation in G90

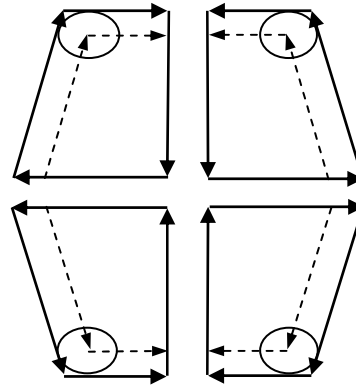


Fig. 4-63 Offset direction of tool nose radius compensation in G94

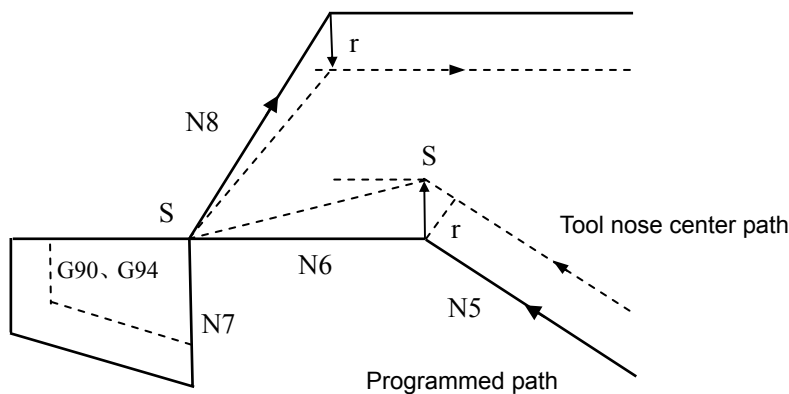


Fig. 4-64 G90, G94 radius compensation mode

● G70 command

When G71~G73 is executed, the system temporarily cancels C tool compensation. When G70 is specified again, the system automatically recovers the compensation mode. Because the system executes G71~G73, it does not execute the radius compensation, there must be the finishing allowance in programming to avoid the overcut in roughing.

In G70, the compensation mode is not cancelled after the cycle end, the system continuously executes the compensation in the fixed point, which causes the undercut of the finishing cycle in the last block, so, the last should exceeds one tool radius value of the workpiece in programming.

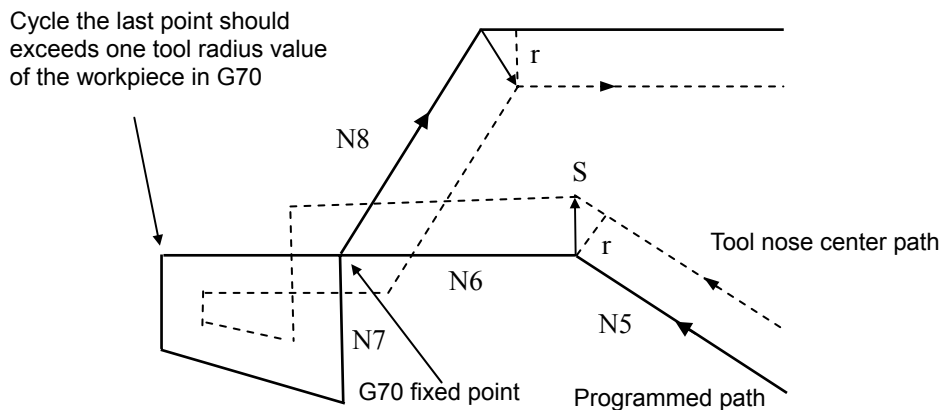


Fig. 4-65 G70 radius compensation mode

4.2.7 Particulars

- **Inside chamfer machining less than tool nose radius**

At the moment, the tool inside offset causes an excessive cutting. The tool stops and the system alarms (P/S41) when starting the previous block or chamfer moving. But the tool stops the end point of previous block when **Single** is ON.

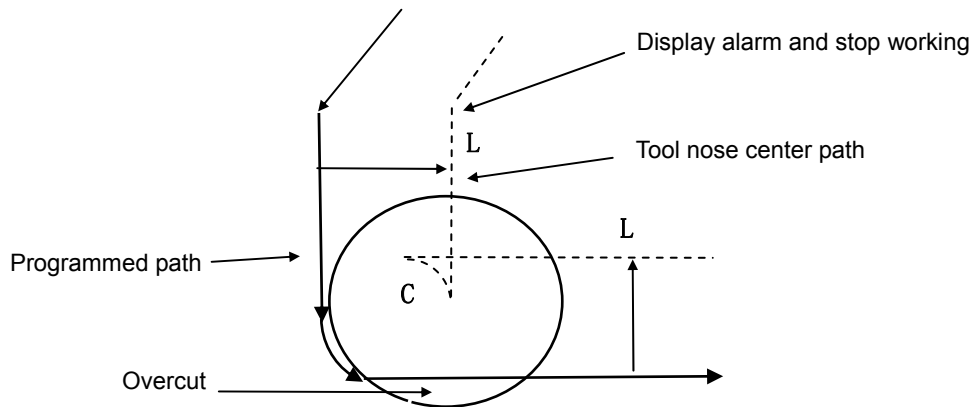


Fig.4-66 inner corner machining less than tool nose radius

- **Machining concave less than tool nose diameter**

There is an excessive cutting when the tool nose center path is opposite to program path caused by tool nose radius compensation. At the moment, the tool stops and the system alarms No.257 when starting the previous block or chamfer moving.

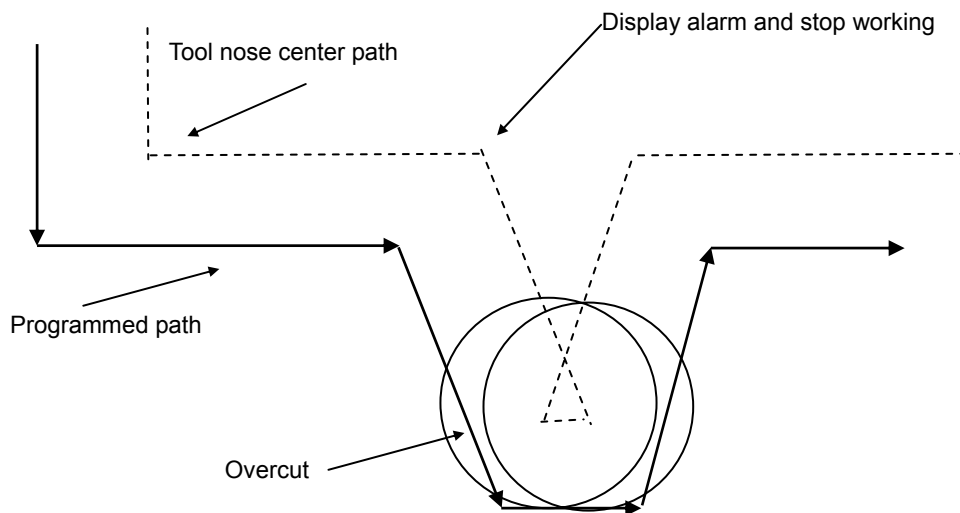


Fig. 4-67 machining a grooving less than tool nose radius

- **Machining a inner sidestep less than 90°**

When the system machines a inner sidestep less than or equal to 90° and the machining path length is less than the tool nose radius, there will be the too much undercut and No. 260 alarms. At the moment, No.5008 Bit6 (CNS) sets whether the system alarms in the condition.

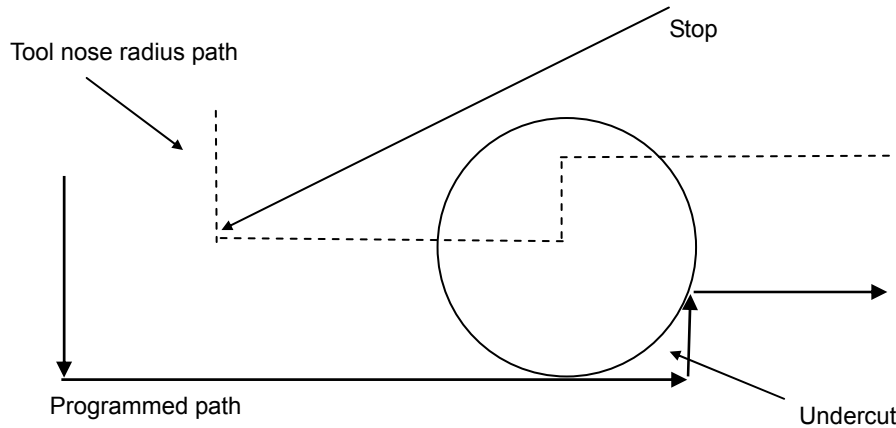


Fig.4-68 machining a inner sidestep less than 90°

● Corner motion

When two or more than movement vector in the end point of one block create, the tool moves to another vector from the vector linear, which is called the corner motion. When the single block is valid, the tool stops in the last vector.

When two vectors coincide, the system does not execute the corner motion and the second vector will be ignored. When the two-axis increments of the movement vector in the compensation plane are less than the setting values of No. 5010(CLV), the second vector is ignored, but it is not ignored when the interpolation block is the arc.

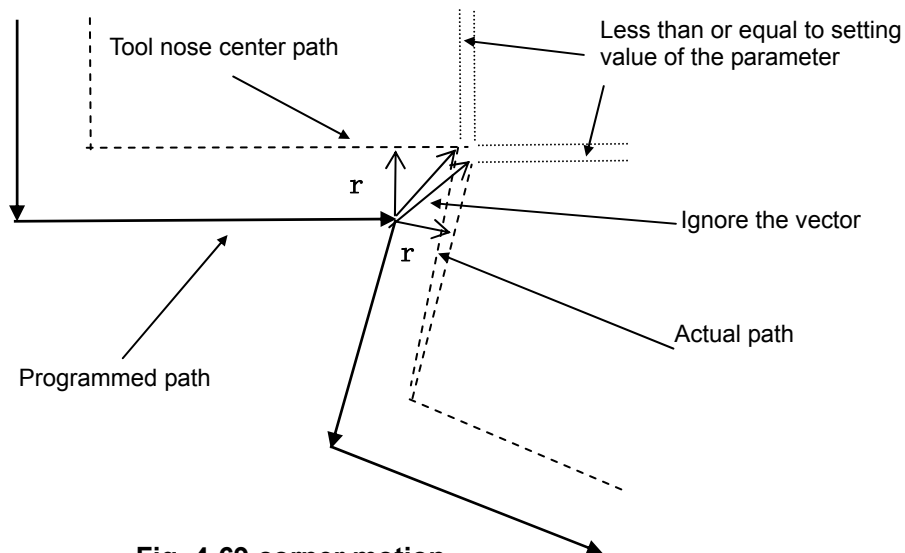


Fig. 4-69 corner motion

● Changing compensation value

- (a) The system executes the tool change in the compensation cancel mode, the compensation value is changed. When the compensation value is changed in the compensation mode, No.5001 Bit4(EVR) can set whether the compensation value change is valid from the nest T command or the next buffer block.

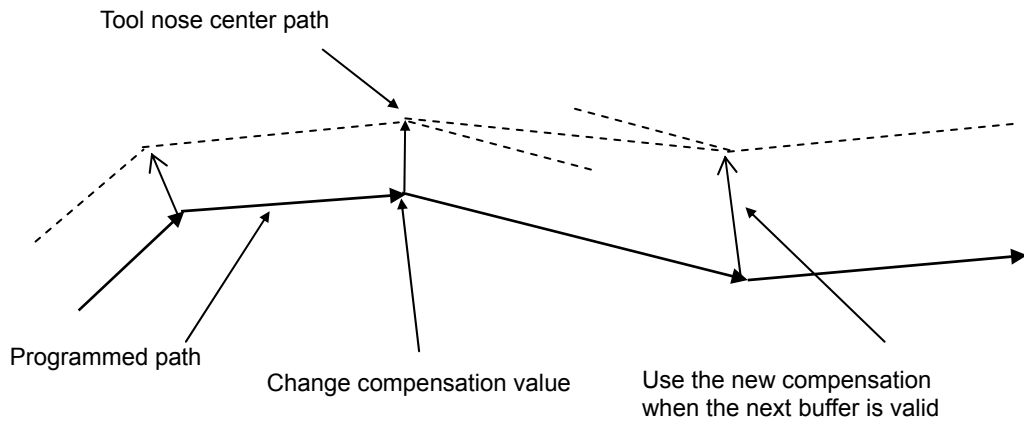


Fig.4-70 changing compensation value

(b) Positive/negative compensation value and tool nose center path

When the compensation value is negative(-), G41 and G42 exchange in programming. When the tool center moves along the workpiece outer, it moves along the inner, and vice versa.

Note: The compensation value is equal to the offset value adding the wear value. When the compensation sign is changed, the tool nose offset direction changes but the imaginary tool nose direction does not change. So, do not change the compensation sign optionally.

- **End point of programmed arc is not in the arc**

In the radius compensation process, when the system uses IJK to specify the circle center and the end point of the arc is not in the arc, the system positions again the circle center position specified by IJK, and confirms the circle center position according to the radius counted by IJK to execute the radius compensation. When the counted radius is too small not to reach the end point of the arc, No. 254 alarm creates.

Note: at the moment, there is a difference between the counted arc and the specified in programming, and the function is sued to regulate the error of the radius out-of-tolerance in some range in programming.

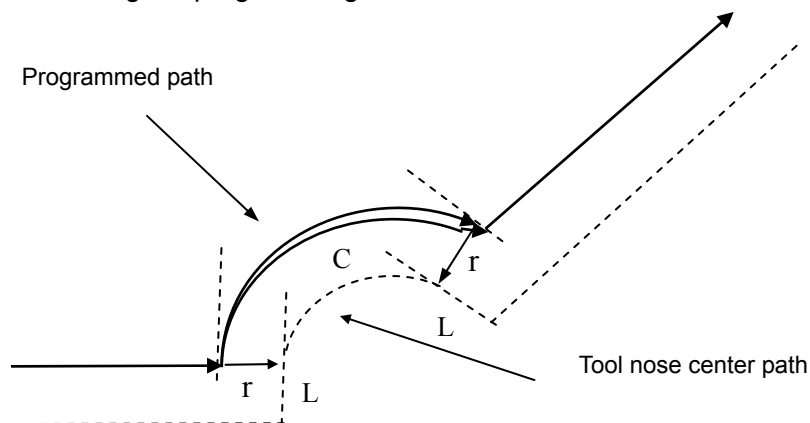


Fig. 4-71 End point of programmed arc is not in the arc

- **Continuous 3 or more than 3 blocks non-movement command**

In tool radius compensation process, when there are 3 or 3 blocks without movement command,

the tool nose center reaches the end point of the last block and is vertical with programmed path position of the last, at the moment, which causes the overcut, and the programmer should pay more attention it.

General non-movement command :

1.M03S300 only have M, S, T, F, O, N codes

2.#100=3 non-NC statement (when 6000#5 SBM is set to 1)

3.G04 X10 pause

4.G00 only have G code and do not specify the position command

5.G01 U0 the infeed distance is 0

6.G01X100 only specify the absolute value which is same with that of the last block

7.G01Y10 only specify the axis in non tool compensation plane

8.M98M99 statement for calling subprogram and subprogram return(the block has no

axis

increment command)

9.G66G67 statement for calling macro program and cancelling macro program call modal

10.; empty block

In non-movement block, when there is a command to cancel the radius compensation, the system does not cancel the vector and execute the command in the vertical vector. It cancels the radius compensation vector when the system cancels the radius compensation in G28, G30, G53, it executes the command in the vertical vector in G50, G52, G32, G34, fixed cycle, multi cycle, drilling cycle and other commands.

When there are 3 or more than 3 blocks without movement command following the block used to create the tool radius compensation, the system does not immediately create the tool radius compensation but does it in the block following the non-movement command.

The system executes the above vertical before the last movement command when there is a optional symbol "/" in tool radius compensation. So, please do not use the optional block function in the tool radius compensation to avoid the overcut.

When No.6000 Bit5 (SBM) is set to 1, the macro statement can stop in single block and is taken as the non-movement block in the tool nose radius compensation at the moment, which causes the abnormal path. It is suggested that No.6000 Bit5 (SBM) is set to 0 when the system uses the macro statement in the tool nose radius compensation mode in the course of normal machining.

G42
N6X100W10
N7X90W20
N8S200
N9M03
N10F100
N11X65W15
N12W20

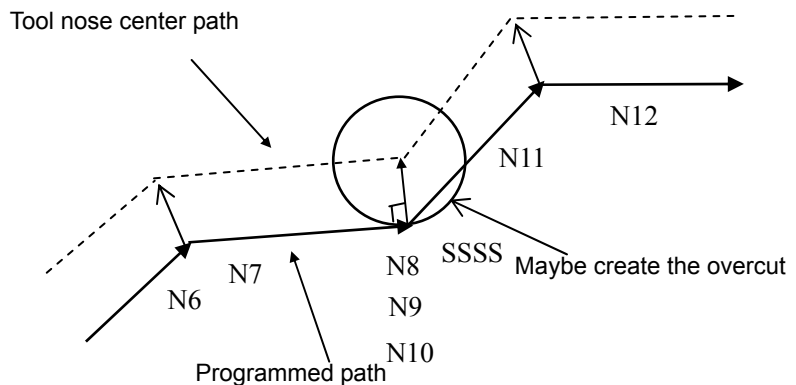


Fig. 4-72 continuous 3 or more than 3 blocks of non-movement command

- **Subprogram call and macro statement call in G code**

In tool nose radius compensation mode, when the system specifies the code for calling the subprogram, it can execute the normal compensation, the compensation method of calling program is transferred to the subprogram which is to execute the corresponding compensation.

The command for calling subprogram and subprogram return has no movement command, it is taken as the non-movement block. When the system specifies G code in the radius compensation mode in the subprogram, G code is valid, at the same time, the system cancels the radius compensation mode when the subprogram does not end, the compensation mode is transferred to the called program which will continuously executes the corresponding compensation.

- **Cutting inner of the whole circle**

In the tool nose radius compensation, when the system machines the inner of the whole circle and the compensation direction is not changed, the overcut or undercut creates, at the moment, it determines whether it alarms based on No.5008 Bit5 (CNF). When Bit5 is set to 0, No.259 alarms.

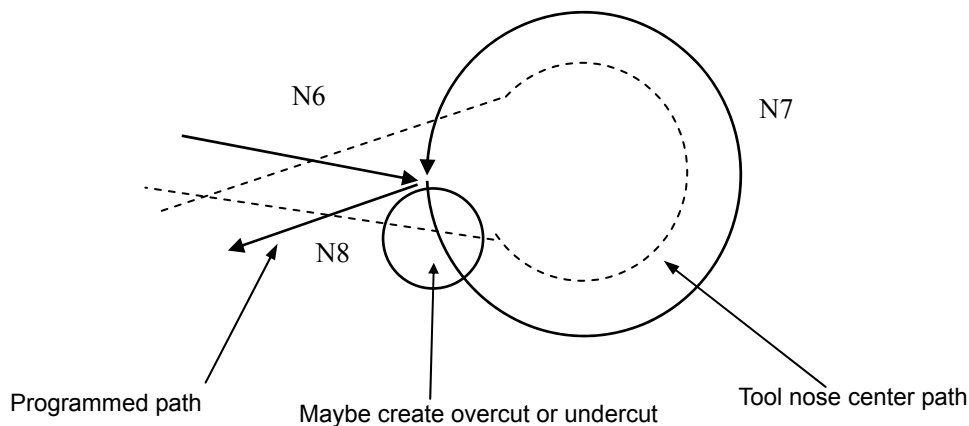


Fig. 4-73 overcut in machining inner of the whole circle

- **Inserting MDI operation in tool compensation**

In MDI mode, the system does not execute the tool nose radius compensation. When the system specifies G41 or G42, the system determines No.5008 Bit4 (MCR). When Bit is set to 1, No.258 alarms. The system does not alarm and ignores the specified G41 and G42 when it is set to 0,

When the system runs in AUTO mode in absolute command programming and the single block run stops to insert MDI mode, and then starts AUTO mode, at the moment, transfers the vector of starting point of the next block, and forms other vectors based on the next two blocks, the offset can be executed from PC, and the tool path is as follows:

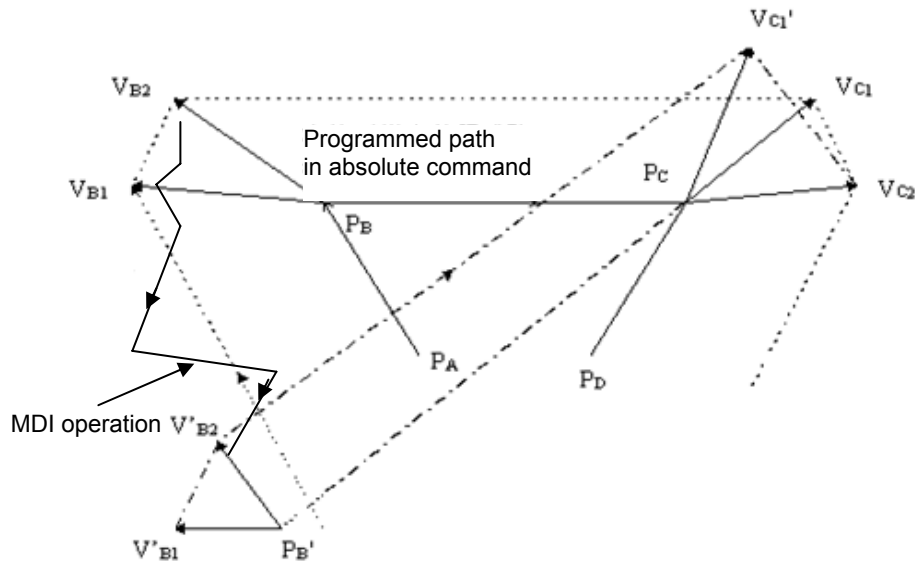


Fig.4-74 insert tool offset of block in MDI mode

When PA, PB, PC is programmed with absolute command, the single block run stops and the tool is moved in MDI after the block from PA to PB is executed. The vector VB1, VB2 are transferred to V'B1 and V'B2, V'C1, V'C2 of PB'→PC and PC→PD are calculated again.

But, the system can correctly execute the compensation following PC because the vector VB2 has not calculated again.

VOLUME II

OPERATION

Chapter I Overview

GSK988T system panel adopts 8.4"LCD and its appearance is shown as below:





1.1 Operation panel

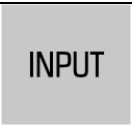


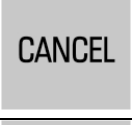

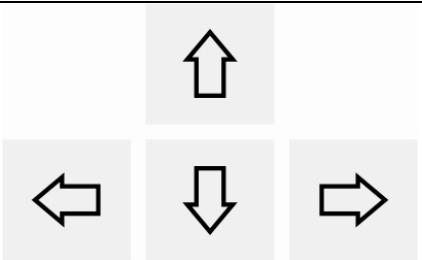
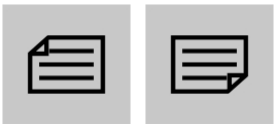
GSK988T operation panel is classified as below:



1.2 Introduction of the keypad

1.2.1 Editing the keypad

Keys	Name	Introduction of the function																								
	Reset key	CNC reset, feeding and output stop, etc																								
<table border="1" data-bbox="181 1561 598 1821"><tr><td>P O</td><td>Q N</td><td>R G</td><td>A 7</td><td>B 8</td><td>D 9</td></tr><tr><td>C X</td><td>Y Z</td><td>L F</td><td>[4</td><td>] 5</td><td>SP 6</td></tr><tr><td>I M</td><td>K S</td><td>J T</td><td>' 1</td><td># 2</td><td>= 3</td></tr><tr><td>H U</td><td>V W</td><td>E EOB</td><td>+ -</td><td>* 0</td><td>/ .</td></tr></table>	P O	Q N	R G	A 7	B 8	D 9	C X	Y Z	L F	[4] 5	SP 6	I M	K S	J T	' 1	# 2	= 3	H U	V W	E EOB	+ -	* 0	/ .	Keys of address, numerical and symbol	Input the address, number and symbol, press shift key and take the above address or symbol; Otherwise, take the address below.
P O	Q N	R G	A 7	B 8	D 9																					
C X	Y Z	L F	[4] 5	SP 6																					
I M	K S	J T	' 1	# 2	= 3																					
H U	V W	E EOB	+ -	* 0	/ .																					
	Shift key	Switch among keys of double addresses, double symbols, address symbol and numerical address, firstly press shift key and its indicator is on, and then press address key,																								

Keys	Name	Introduction of the function
		input the address above; or select one block with the cursor keys
	Input key	Input the data of parameter and compensation value, etc, and switch the line during editing the program.
	Change key	Switch between the information and the display, with function of Tab key, and forming the shortcut keys with the other keys during editing the program.
	Backspace key	Delete the program and the character, etc ahead
	Cancel key	Cancel the operation
	Delete key	Cancel the program and the character,etc backward
	Cursor keys	Control the cursors to move up,down, left and right
	Interface key	Switch the interfaces in one interface

1.2.2 Function keys and soft keys

1. Function keys

GSK988T system MDI keypad includes 8 function keys of the position and the program, etc; they respectively relative to 8 interfaces, which is shown as below:

POSITION	PROGRAM	SYSTEM	SETTING
MESSAGE	GRAPH		HELP

POSITION

Press it to switch into the position interface.

PROGRAM

Press it to switch into the program interface.

SYSTEM

Press it to switch into the system interface.

SETTING

Press it to switch into the setting interface.

MESSAGE

Press it to switch into the message interface.

GRAPH

Press it to switch into the graph interface.

Self-defined interface

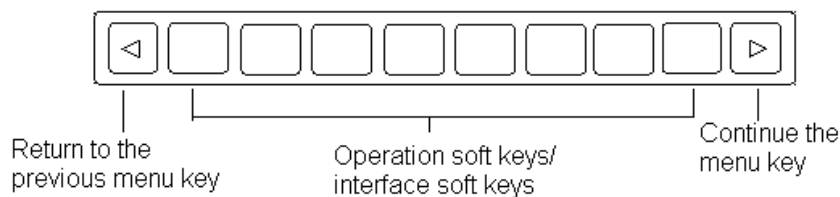
HELP

Press it to switch into the help interface.

2. Soft keys

Press function keys to switch the interfaces, and press the relative soft key to display the content of some interface, or input on the current interface.

The 10 soft keys of GSK988T are at the bottom of the screen, which is shown as the following sketch map.




Function of soft keys:

- ① Switch the secondary interface on the current interface;
- ② Operate on the current secondary interface, such as editing and rewriting the data or displaying the content, etc.

1.2.3 State indication


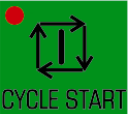



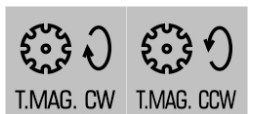
	The indicator of each axis reference point return		Running indicator
	Alarm indicator		Self-defined indicator








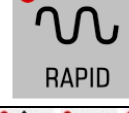
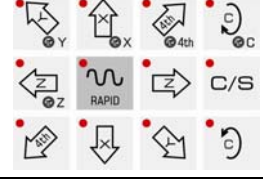


GEAR/TOOL NO. 	Gear/tool number indicator		
--	----------------------------------	--	--

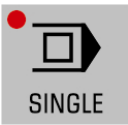










1.2.4 Machine panel









The function of keys on GSK988T machine panel is defined by PLC program (ladder diagram), and about the detailed function of each key, refer to the manual of the machine manufacturer. The machine panel is taken as the reference.

Function of the machine panel each key defined by GSK988T standard PLC program, refer to the following list:

Keys	Names	Function	Mode during valid function
	Feed hold key	The program and MDI command running pause	Auto mode, MDI mode and DNC mode
	Cycle start key	The program and MDI command running start	Auto mode, MDI mode and DNC mode
	Feed rate override knob	Adjusting the feed rate	Auto mode, MDI mode, edit mode, reference point return mode, MPG mode, single step mode, manual mode and DNC mode
	Rapid override keys	Adjusting the rapid movement speed	Auto mode, MDI mode, reference point return mode, manual mode and DNC mode
	Spindle override keys	Adjusting the spindle speed only when the spindle speed analog value control mode is valid	Auto mode, MDI mode, edit mode, reference point return mode, MPG mode, single step mode, manual mode and DNC mode
	Manual tool-change keys	Manual tool-change	Reference point return mode, MPG mode, single step mode and manual mode

Keys	Names	Function	Mode during valid function
 JOG	Jog key	Spindle jog on/off	MPG mode, single step mode and manual mode
 LUBRICATING	Lubricating key	Machine lubricating on/off	
 COOLING	Cooling key	Cooling on/off	
 CHUCK	Chuck key	Chuck clamp/release	Auto mode, MDI mode, edit mode, reference point return mode, MPG mode, single step mode, manual mode and DNC mode
 S.OVERRIDE  S.OVERRIDE  S.OVERRIDE	Spindle keys	Spindle CCW turn Spindle stop Spindle CW turn	MPG mode, single step mode and manual mode
 RAPID	Rapid speed switch	Switch between rapid speed/feed rate	Auto mode, MDI mode, manual mode and DNC mode
	Manual feeding keys	each axis moving positive/negative in manual or single step mode	Reference point return mode, single step mode and manual mode
	MPG control axes option keys	Each axis option in MPG mode	MPG mode
	Option keys of MPG/single step increment and rapid override	MPG movement value of each grid: 0.001/0.01/0.1/1 mm Single step movement value of each step: 0.001/0.01/0.1/1 mm Rapid override: Fo, F25%, 50% and F100%	Auto mode, MDI mode, reference point return mode, MPG mode, single step mode, manual mode and DNC mode

Keys	Names	Function	Mode during valid function
	Single block switch	Switch between the single block running/continuous running, when the single block is valid, its indicator is on.	Auto mode, MDI mode and DNC mode
	Block skip switch	Whether skip and switch the block with"/" at the beginning; When the block skip switch is on, its indicator is on.	Auto mode, MDI mode and DNC mode
	Machine lock switch	When the machine is locked, its indicator is on and each axis output is invalid	Auto mode, MDI mode, edit mode, reference point return mode, MPG mode, single step mode, manual mode and DNC mode
	Miscellaneous function lock switch	When miscellaneous function is locked, its indicator is on and the function of M, S and T output is invalid	Auto mode, MDI mode and DNC mode
	Dry run switch	When dry run is valid, its indicator is on and the machine program/MDI command block begins dry running	Auto mode, MDI mode and DNC mode
	Optional stop key	When optional stop is valid, its indicator is on; when there is M01 in the block, move to the block and the running stops	Auto mode, MDI mode and DNC mode
	Edit key	Access edit mode	Edit mode
	Auto key	Access the auto mode	Auto mode
	MDI key	Access MDI mode	MDI mode
	Reference point return key	Access reference point return mode	Reference point return mode
	Single step/MPG key	Access single step or MPG mode (One mode is selected by parameter)	Single step mode/MPG mode/manual mode

Keys	Names	Function	Mode during valid function
	Manual key	Access the manual mode	Manual mode
	DNC key	Access DNC mode	DNC mode
	Feed/spindle hold knob	Feed/axis hold function	Auto mode, MDI mode, edit mode, reference point return mode, MPG mode, single step mode, manual mode, DNC mode
	Emergency stop key	In emergency, the system and the machine stop running, all output is closed.	Auto mode, MDI mode, edit mode, reference point return mode, MPG mode, single step mode and DNC mode
	Power on/off keys	System power of/off switch	Auto mode, MDI mode, edit mode, reference point return mode, MPG mode, single step mode, manual mode and DNC mode
	Overtravel release key	Cancel machine limit	MPG mode and manual mode
	Program protection switch	The protection program can't be changed at random.	Auto mode, MDI mode, edit mode, reference point return mode, MPG mode, single step mode, manual mode and DNC mode
	MPG key	Control the machine movement	MPG mode

1.3 Overview of modes

There are modes of edit, auto, MDI, reference point return, manual/single step, manual and DNC running, etc.

- **Edit mode**

In edit mode, the machine program can be created, deleted and rewritten, etc.

- **Auto mode**

In auto mode, the program auto runs.

- **MDI mode**

In MDI mode, the parameter can be input and the block can be input and executed.

- **Reference point return mode**

In reference point return mode, each axis can return to the reference point, respectively.

- **MPG/single step mode**

In MPG / single step mode, CNC moves on the selected increment.

- **Manual mode**

In manual mode, operate the manual feeding, the manual rapid, the feeding override adjustment, the rapid override adjustment, the spindle on/off, the cooling on/off, the lubricating on/off, the spindle jog and the manual tool-change, etc.

- **DNC running mode**

During DNC running (RMT), the processing (DNC running) can be executed; meanwhile, the external programs can be read through reading the external interface.

Chapter II Power on, power off and safety protection

2.1 Power on

Before GSK988T powers on, they should be confirmed:

1. The machine is normal;
2. The power supply and the voltage comply with the requirements;
3. The connection is right and fixed.

After GSK988T powers on, the interface is shown as below:



Then, GSK988T self-detects and initializes. After completing the self-detection and the initialization, the interface of the present position (absolute coordinate) displays.

自动		复位		
绝对坐标			工艺数据	
X	0.000	mm	T	0100
Z	0.000	mm	F	0 mm/min
				1000 mm/min
			S	0 rev/min
				0
程序名 [00001]			综合信息	
1	00001(实例加工);		进给倍率	100% 手轮倍率 X1
2	G0 X150 Z50;	定位至安全位置换刀;	快速倍率	100% 加工件数 0
3	M12;	夹紧卡盘;	主轴倍率	100% 运行时间 00:00:00
4	M3 S800;	开主轴, 转速800;	手动倍率	0% 切削时间 00:00:00
5	M8;	开冷却液;		
6	T0101;	换第一把刀;		
7	G0 X136 Z2;	靠近工件;		
8	G71 U0.5 R0.5 F200;	切深1mm, 退刀1mm		
			15:50:38	
绝对坐标	相对坐标	机床坐标	综合坐标	模态
				相对坐标设置

2.2 Power off

Before power off, they should be confirmed:

1. Each axis of CNC stops;
2. The miscellaneous function switches off (such as the spindle and the water pump, etc)
3. Firstly cut off CNC power supply, and then cut off machine power supply.

Remark: About the operation of cutting off the machine power supply, refer to the manual of the machine manufacturer.

Chapter II Power on, power off and safety protection

2.3 Overtravel protection

To avoid the damage of the machine due to the overtravel of each axis, the machine must take the measure of overtravel protection.

Install the limit switches on the maximum stroke in each axis positive and negative directions on the machine. When it overtravels, the limit switch is on, the system decelerates till stopping and it alarms overtravel.

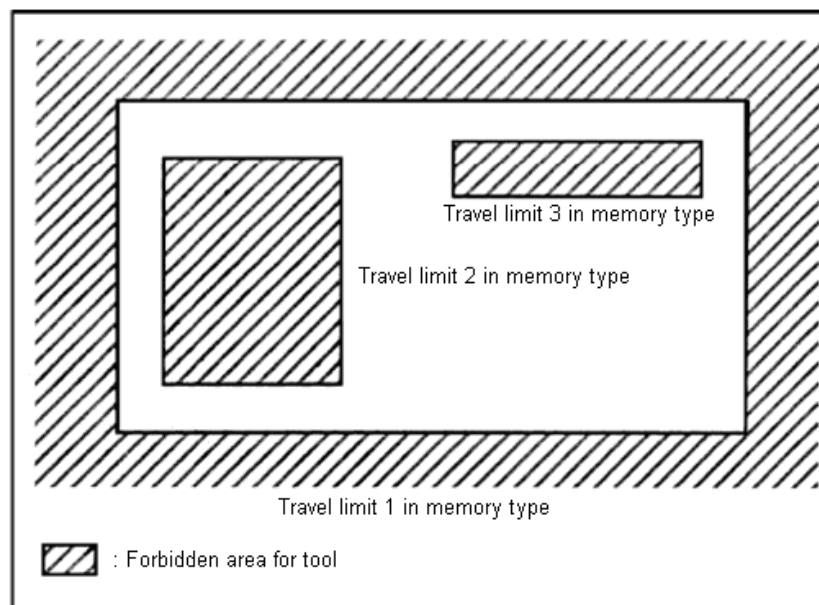
During auto running, when the machine moves along one axis and touches the limit switch, the tool decelerates and stops as long as it traverses along all axes and the system alarms overtravel.

During the manual operation, only the axis which the tool touches its limit switch decelerates and stops, while the tool still traverses along other axes.

The method of canceling the alarm of “overtravel”: In the manual mode, the working table moves in the opposite direction (For example: Overtravel is in the position direction, it moves negatively; negative, positively.) and leaves off the limit switch. Reset, the alarm is cleared.

2.4 Overtravel protection in memory travel limit

The tool can't enter the area stipulated by the travel limit check 1, 2 and 3 in memory type.



When the tool exceeds the travel limit in memory type, it alarms and the tool decelerates and stops. When the tool enters the forbidden area and alarms, the tool can traverse in the opposite

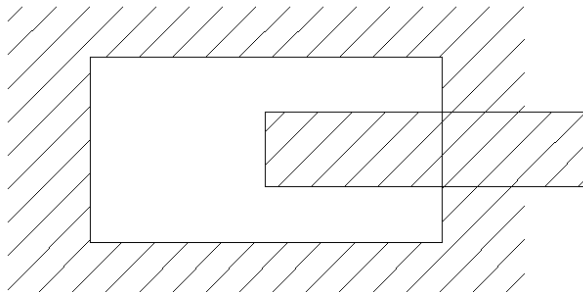
direction.

Travel limit check 1 in memory type: The board is set by parameter (#1320 and #1321 or #1326 and #1327); the outside of the range is set as the forbidden area. The machine manufacturer normally sets the area as the maximum stroke.

Travel limit check 2 (G22 G23) in memory type: It is set by parameter (#1322 and #1323) or commands. During programming, G22 forbids the tool enters the forbidden area; G23 allows the tool enters the forbidden area. In the program, G22 and G23 should be specified independently, which are independent blocks; about the details, refer to the introduction of G commands.

Travel limit check 3 in memory type: The internal board of the area set by parameters #1324 and #1325 as the forbidden area.

Overlap of the forbidden area: Each forbidden area can be overlapped (refer to the following figure), but the outside of the machine travel isn't limited.



The valid time of the forbidden area: After connecting the power supply and manual reference point return or auto reference point return through G28, each limit becomes valid. After connecting the power supply, if the reference point is in the limited area, it alarms immediately.

Display the alarm time: It alarms immediately before or after the tool enters the forbidden area, which is set by the 7th bit of #1300 of parameter BFA.

Overtravel alarm release: When the tool can't traverse in the forbidden area, switch into the manual mode and the tool traverses out of the forbidden area in the opposite direction (for example, overtravel is in the positive direction, it traverses negatively; negative, positively), press the resetting key, the alarm is cleared. If the setting is wrong, after rewriting and setting, the tool returns to the reference point.

Points for attention: During setting the forbidden area, if two points are set as same, the area is as below:


1. When the forbidden area is travel check 1 in memory type, all the areas are taken as the forbidden one.
2. When the forbidden area is travel check 2 or 3 in memory type, all the areas are taken as the moveable area.

2.5 Emergence operation

During the processing, due to the user programming, operation and the product default, etc, some unexpected situations may occur, then, GSK988T should stop working immediately. In this

chapter, it mainly introduces the measures taken in emergency. About the machine in emergency, refer to the relative introduction of the machine manufacturer.

2.5.1 Resetting

When GSK988T output and the coordinate axis moves abnormally, press  and GSK988T resets:

1. All axes movement stops;
2. Function of M and S output invalid;
3. Auto running completes, the mode function holds.

2.5.2 Emergency stop


During the machine running, in the dangerous or the emergency situation, press the emergency stop button and the external emergency stop signal is valid, and then CNC works in the emergency situation and the machine stops moving at once, all output is off, such as the revolving of the spindle and the cooling fluid. After releasing the emergency stop button, the alarm is released, CNC resets.

Remark 1: Before releasing the emergency stop alarm, confirm the trouble is shot;

Remark 2: Before power on and off, press the emergency stop button to reduce the electric shock of the equipment;

Remark 3: After releasing the emergency stop alarm, return to the reference point again to ensure the precision of the coordinate position.

2.5.3 Feed hold

During the machine running, press  to stop the running, temporarily. Pay attention to that during the thread cutting or the cycle command running, even press the button, the running can't stop immediately.

2.5.4 Cutting off power supply

During the machine running in the dangerous situation or emergency, the machine power supply can be cut immediately to avoid the accident. But, pay attention to that the coordinate displayed by CNC can't comply with its actual position after cutting off power supply, so it requires returning to the reference point, again.

Chapter III Interfaces

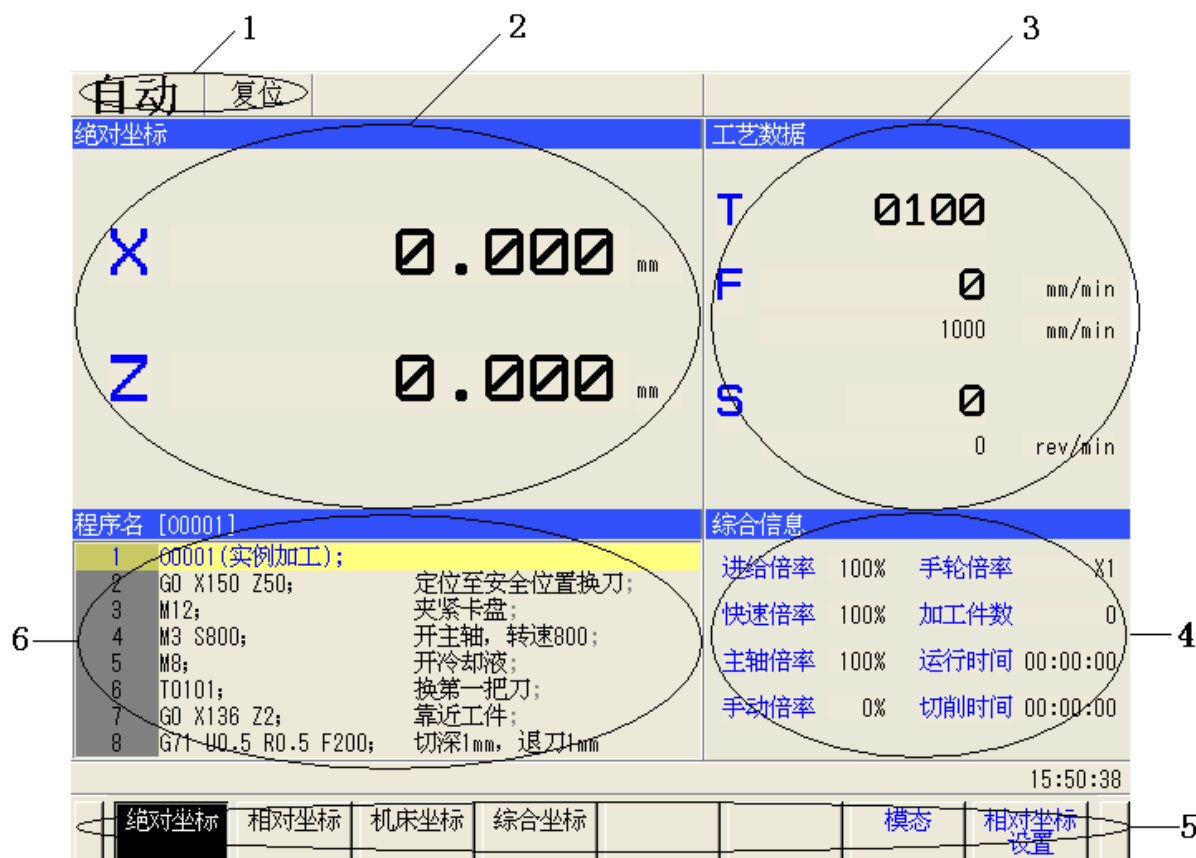
Based on the interfaces, this chapter introduces the relation among the switching interfaces, input and soft keys and the detailed operation method.

There are eight function keys including position, program and setting, etc on MDI panel in GSK988T system, each function key is relative to one main interface, and each main interface also includes many interfaces and the soft keys.

FUNCTION KEYS	REMARK
POSITION	Access the position interface. It mainly includes interfaces of relative, absolute, machine and comprehensive coordinates, etc.
PROGRAM	Access CNC program interface. It mainly includes interfaces of local directory, MDI program and DIR/NXT; if the system is with flash disc, it can also display its directory interface.
SYSTEM	Access the system interface. It mainly includes the interfaces of parameter, screw pitch compensation, system information, file management and ladder diagram, etc.
SETTING	Access the setting interface. It mainly includes the interfaces of setting tool offset, CNC and macro variable, etc.
MESSAGE	Access the information interface. It mainly includes interfaces of alarm information, records and diagnosis, etc.
GRAPH	Access graph interface. It mainly includes interfaces of graph setting, path display and simulation graph, etc.
	Self-defined interface
HELP	Access help interface. It mainly includes interfaces of operation, programming, alarm and parameter help.

3.1 Position interface

The initial interface is the position one after the system powers on; during auto resetting, the position interface without uploading the program is shown as below:



It mainly includes six zones:

Zone 1: Status information zone. Display the system operation mode and running status; display the modes of editing, auto and MDI and the information of auto, stop and resetting, etc.

Zone 2: Coordinate information zone is displayed in big font; display each coordinate name and the current coordinate position of each axis in big font.

Zone 3: Actual and command speed, speed values zone. Display the information of the current tool number, the actual feed rate and the actual speed of each spindle, etc.

Zone 4: Comprehensive and mode information zone; display the information of feeding, rapid, manual and spindle override, the system running and the cutting time and quantity of the processing work pieces, etc. Moreover, it can display the mode command information through soft keys.

Zone 5: Each soft key mark information zone, display the information corresponding to the soft and the operation keys in each interface.

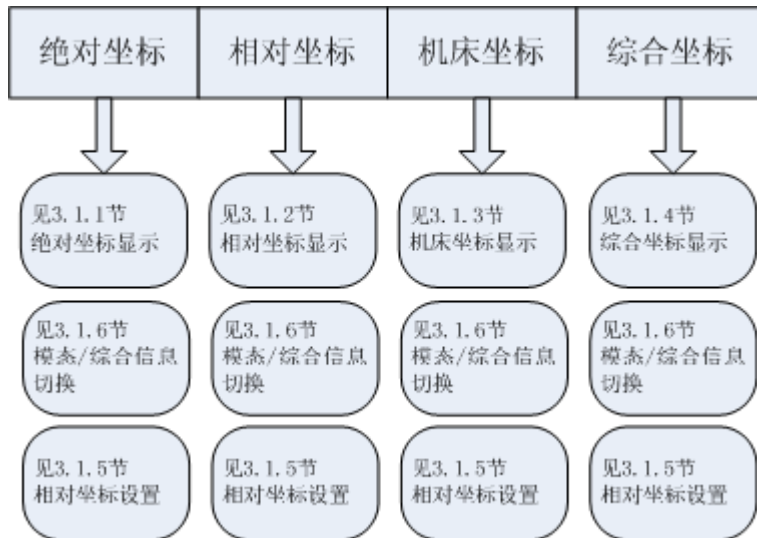
Zone 6: Block information zone, in auto mode, display the information of the uploaded, MDI and DNC programs.

Remark:

1. The quantity of axes, displayed in the interface, is set by parameter #1010 and #8130. Parameter #8130 sets the total quantity of the controlled axes and #1010 sets the quantity of axes controlled by CNC.

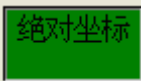
2. The name of each axis can be set by parameter #1020.

Press **POSITION** to access position interface and it includes interfaces of absolute, relative and machine coordinates, etc and check the content of each interface through pressing the corresponding soft keys. The structure of the soft key layers is shown as the following figure:



3.1.1 Absolute coordinate

绝对坐标

On the position interface, press  to switch into the absolute coordinate interface. During auto mode and resetting, the interface is shown as below. On the top left corner, display the coordinate value of X and Z axes as the absolute position of the current work piece coordinate system which the tool is.

自动 运行 G01直线切削中					
绝对坐标			工艺数据		
X	135.250	mm	T	0101	
Z	-6.247	mm	F	200	mm/min
				200	mm/min
			S	585	rev/min
				800	rev/min
程序名 [00001]			综合信息		
5	M8;	开冷却液;	进给倍率	100%	手轮倍率 X1
6	T0101;	换第一把刀;	快速倍率	100%	加工件数 0
7	G0 X136 Z2;	靠近工件;	主轴倍率	80%	运行时间 00:00:07
8	G71 U0.5 R0.5 F200;	切深1mm, 退刀1mm;	手动倍率	0%	切削时间 00:00:07
9	G71 P0060 Q0150 U0.25 W0.5; X轴预留0.5mm, Z轴0.5mm余量;				
10	N60 G0 X16;	靠近到工件端面;			
11	G1 Z-23;	车Φ16外圆;			
			17:09:35		
绝对坐标 相对坐标 机床坐标 综合坐标			模态		

T: Current tool number and tool offset number

Actual speed F: During actual processing, the actual processing speed after feeding override;

Programming speed: speed is set by F code in program;

Spindle actual speed S: The spindle speed feed back by the spindle encoder can display the actual speed of the spindle only after installing the spindle encoder;

Programming spindle speed S: The spindle speed is specified by S code in program;

Feeding override: It is selected by the feeding override switches;

Rapid override: It is selected by the rapid override switches;

Spindle override: It is selected by the spindle override switches;

Manual override: It is selected by the manual override switches;

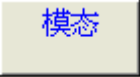
MPG override: Current MPG override;

Quantity of processing work pieces: The quantity of the processing work pieces pluses one after the program executes M02 or M30 or M codes set by parameter # 6710.

Cutting time: Executing time of auto running in one time without the time of stop and feeding pause, timing begins from 0 after auto running starts each time, the units in turn are hour, minute and second;

Running time: All execution time of system in auto mode without time of stop and feeding pause is the accumulative cutting time;

G function codes: The mode values of G codes in each group;

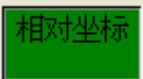
Switch between the mode and comprehensive information through pressing  and

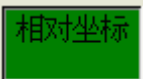
.

模态

综合信息

3.1.2 Relative coordinate

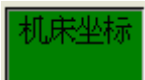
 相对坐标

In position interface, press  to switch into the relative coordinate interface. Then, on the left top corner, display the relative coordinate value. U and W coordinate value is the relative coordinate value of the current position. U and W coordinates can be cleared during stop and resetting state. The interface is shown as below:

自动		运行	G01直线切削中		
相对坐标				工艺数据	
U 145.247 mm				T 0101	
W -65.660 mm				F 200 mm/min	
				200 mm/min	
				S 585	
				800 rev/min	
程序名 [00001]				综合信息	
5	M8;	开冷却液;		进给倍率	100% 手轮倍率 X1
6	T0101;	换第一把刀;		快速倍率	100% 加工件数 0
7	G0 X136 Z2;	靠近工件;		主轴倍率	80% 运行时间 00:00:31
8	G71 U0.5 R0.5 F200;	切深1mm, 退刀1mm;		手动倍率	0% 切削时间 00:00:31
9	G71 P0080 Q0150 U0.25 W0.5;	X轴预留0.5mm, Z轴0.5mm余量;			
10	N60 G0 X16 ;	靠近到工件端面;			
11	G1 Z-23;	车Φ16外圆;			
				17:09:59	
绝对坐标		相对坐标	机床坐标	综合坐标	模态

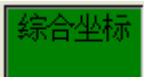
3.1.3 Machine coordinate

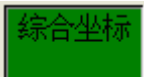
机床坐标

On the position interface, press  to switch into the machine coordinate interface. The machine coordinate system is set through the reference point. The interface is shown as below:

自动		暂停		
机床坐标			工艺数据	
<div>X145.247mm</div> <div>Z-0.220mm</div>			<div>T0101</div> <div>F0mm/min</div> <div>200mm/min</div> <div>S585</div> <div>800rev/min</div>	
程序名 [00001]			综合信息	
5	M8;	开冷却液;	进给倍率	100% 手轮倍率 X1
6	T0101;	换第一把刀;	快速倍率	100% 加工件数 0
7	G0 X136 Z;	靠近工件;	主轴倍率	80% 运行时间 00:00:11
8	G71 U0.5 R0.5 F200;	切深1mm, 退刀1mm;	手动倍率	0% 切削时间 00:00:11
9	G71 P0060 Q0150 U0.25 W0.5; X轴预留0.5mm, Z轴0.5mm余量;			
10	N60 G0 X16 ;	靠近到工件端面;		
11	G1 Z-23;	车Φ16外圆;		
			17:11:41	
绝对坐标	相对坐标	机床坐标	综合坐标	模态

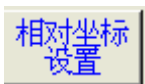
3.1.4 Comprehensive coordinate




In position interface, press  to switch into the comprehensive coordinate interface. Then, the comprehensive coordinate value is displayed on the top corner of the interface including the absolute, relative and machine coordinates and the surplus movement value. The interface is shown as below:







自动		运行		G00快速定位中			
绝对坐标		相对坐标		机床坐标		余移动量	
X17.833 mm		U17.833 mm		X17.833 mm		X131.547 mm	
Z8.916 mm		W8.916 mm		Z8.916 mm		Z40.773 mm	
程序名 [00001]				综合信息			
100001(实例加工);				进给倍率100%		手轮倍率X1	
2G0 X150 Z50;		定位至安全位置换刀;		快速倍率100%		加工件数0	
3M12;		夹紧卡盘;		主轴倍率100%		运行时间00:00:00	
4M3 S800;		开主轴, 转速800;		手动倍率0%		切削时间00:00:00	
5M8;		开冷却液;					
6T0101;		换第一把刀;					
7G0 X136 Z2;		靠近工件;					
8G71 U0.5 R0.5 F200;		切深1mm, 退刀1mm					
15:58:05							
绝对坐标		相对坐标		机床坐标		综合坐标	
						模态	

3.1.5 Setting the relative coordinate

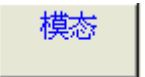

In position interface, press  to set the relative coordinate and the interface is shown as below. Then, the relative coordinate value of each coordinate axis can be set. The steps are as following:

(1) During resetting, press  to change the relative coordinate axis into the input state, the relative coordinate value U is shown as below:



- (2) Press  or  to select the coordinate axis to be set, which makes the axis to be input;
- (3) Input the relative coordinate axis to be set, press  to complete setting.
- (4) Firstly press , and then press  or  to select the other coordinate axes to set the relative coordinate value.

3.1.6 Switching between the mode and the comprehensive information

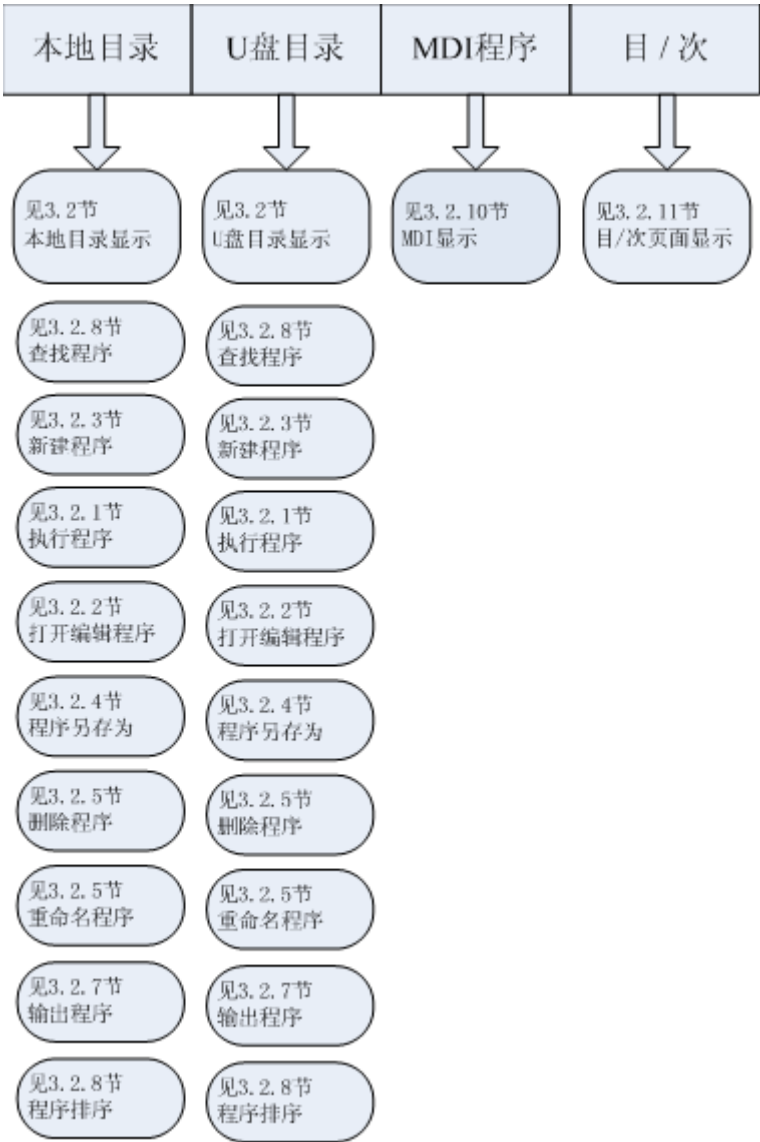
In position interface, press  and  to switch between the mode and the comprehensive information, the mode interface is shown as below:

自动		暂停			
机床坐标				工艺数据	
X		145.247 mm		T	0101
Z		-0.220 mm		F	0 mm/min 200 mm/min
				S	585 800 rev/min
程序名 [00001]				模态	
5	M8;	开冷却液;		G00 G97 G98 G21 G40 G25	
6	T0101;	换第一把刀;		G22 G80 G87 G54 G18 G113	
7	G0 X136 Z2;	靠近工件;			
8	G71 U0.5 R0.5 F200;	切深1mm, 退刀1mm;			
9	G71 P0060 Q0150 U0.25 W0.5;	X轴预留0.5mm, Z轴0.5mm余量;			
10	N60 G0 X16 ;	靠近到工件端面;			
11	G1 Z-23;	车Φ16外圆;			
16:03:54					
绝对坐标		相对坐标		机床坐标	
		综合坐标		综合信息	

3.2 Program interface

Press **PROGRAM** to access the program interface and it mainly includes the local directory, MDI program and DIR/NXT; when there is flash disc, its directory can also be displayed. Press

本地目录 to display the local CNC program directory and the programs in the directory can be loaded, opened, copied, pasted, created and saved in the other name, deleted, renamed and searched, etc. The structure of the soft key layers is shown as below:



The interface is shown as below:

自动	复位		
程序 -> 本地目录			
程序数: 7		占用空间(字节): 10,561	剩余空间(字节): 16,349,184
程序名	注 释	长度(字节)	修改时间
00001	实例加工	1,635	2009-03-25,16:11:27
00005	00005	14	2009-03-25,16:14:29
01111	说明书的所有程序	5,311	2009-03-25,15:18:31
08000	测试主程序	347	2009-03-25,15:18:33
08001	G00/G01测试程序	714	2009-03-25,15:18:36
08002	G02/G03测试程序	1,355	2009-03-25,15:18:37
08090	G90/G94测试程序	1,185	2009-03-25,15:18:38
17:44:08			
本地目录	MDI程序	目 / 次	查找 新建 执行 打开 >






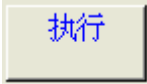
On the top status information zone, it displays the system current running mode and status; the total number of the programs in the current system, the total used space and the free space in all the programs.

In the list, it displays the program list of the current system and the capacity of each program and the latest rewritten date. The program on green background is the one selected by the current cursor, such as program O1111 shown as above. While the program in red font with is the executable program which is just uploaded to the position interface, such as program O0001 shown as above.



When the system port is with flash disc, display the soft key “flash disc directory” on the interface meanwhile, which is shown as below. Press “flash disc directory”, the window displays CNC program directory in “NCPROG”file. The file operation of the flash disc is same as that of the local directory.



3.2.1 Executing the program

- (1) Press  to access the program interface;
- (2) On the program interface, press , ,  or  to select the program to be executed;
- (3) During resetting, the operation authority is above level [4], press  and the selected program is uploaded to the block zone on the position interface and becomes the current executable program, the display interface jumps to the position interface, such as program # 00001 shown as below:

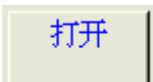
自动		复位			
绝对坐标			工艺数据		
X	0.000 mm		T	0100	
Z	0.000 mm		F	0 mm/min 200 mm/min	
			S	0 rev/min	
程序名 [00001]			综合信息		
4	M3 S800;	开主轴, 转速800;	进给倍率	100%	手轮倍率 X1
5	M8;	开冷却液;	快速倍率	100%	加工件数 0
6	T0101;	换第一把刀;	主轴倍率	80%	运行时间 00:00:00
7	G0 X136 Z2;	靠近工件;	手动倍率	0%	切削时间 00:00:00
8	G71 U0.5 R0.5 F200;	切深1mm, 退刀1mm;			
9	G71 P0060 Q0150 U0.25 W0.5;	X轴预留0.5mm, Z轴0.5mm余量;			
10	N60 G0 X16 ;	靠近到工件端面;			
9:33:46					
绝对坐标		相对坐标	机床坐标	综合坐标	
				模态	相对坐标设置

Then, press  to switch into auto mode; press  to run the loaded program.

Remark: Only during resetting, the operation authority above level [4] can operate the program loading.

3.2.2 Opening the editing program

In the program interface, press , ,  or  to select the programs, and

when the operation authority is above level [3], press  to open the selected program, which is shown as the following program #0001:



Then, when the program switch is on, the current program can be edited and rewritten.

① Press **执行** to upload the opened program to the block zone of the position interface and becomes the executable program and the current interface jumps to the position interface.

② Press **保存** to save the program which is edited and rewritten currently.

③ Press **撤消** to cancel the previous operation.

④ Press **恢复** to repeat the previous operation which is canceled.

⑤ Press **定位** and the interface is shown as below:



请输入行号

确定

Input the line number , press and the cursor can position the program line of which the line number is input.

⑥ Press . Firstly press or to move toward the block to be copied,

and press to copy the block which the cursor is.

⑦ Press . Firstly press or to move toward the line where the block is

pasted, and then press or to paste the copied block in the next line of current one.

⑧ Press . Press or to move toward the block to be deleted, and then

Press to delete the block where the cursor is.

Press , the interface is shown as below:



⑨ Press **语法检查**

to check whether there is any grammer mistake in the current program.

⑩ Press **搜索**

to search for the character string in the current program, the interface is shown as below:



请输入查找字符串

The characters to be searched can be input . For example, input

首行开始

G71, and press to search for G71 from the first line of the program, the searched characters displays

in blue-based color, and display the position of the characters which is shown as the following

下一个


前一个

graph. Press or to search for previous or next G71 from the current position.

取消

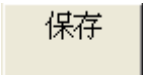
Press to cancel the search.



⑨ Press  to return to the previous menu.

Remark:

- (1) Only when the operation authority is above level (3), the programs can be opened, edited and rewritten.
- (2) When the file is rewritten, the program after the title displays “*”, which means the file has

been rewritten, after auto saving or press  , “*” will disappear.

3.2.3 Creating the program

In the program interface, press  to access the creating interface, which is shown as below:

自动		复位	
程序 -> 本地目录			
程序数: 6		占用空间(字节): 10,547	剩余空间(字节): 16,347,648
程序名	注 释	长度(字节)	修改时间
00001	实例加工	1,635	2009-03-25,16:11:27
01111	说明书的所有程序	5,311	2009-03-25,15:18:31
08000	测试主程序	347	2009-03-25,15:18:33
08001	G00/G01测试程序	714	2009-03-25,15:18:36
08002	G02/G03测试程序	1,355	2009-03-25,15:18:37
08090	G90/G94测试程序	1,185	2009-03-25,15:18:38
<div> <div>请输入新程序名</div> <div>0</div> </div>			
16:13:49			
确定	取消		

请输入新程序名

0

取消



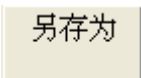
The new program name can be input , then, press to cancel the creating. Press to access the program editing interface, for example, input 0005, the interface jumps to the editing interface of program #O0005, which is shown as below:



Then, edit the new program through editing keypad, and the program can be loaded and saved, etc. The detailed operation is same as that of 2.2.2 about editing.

Remark: Only the operation authority is above level (3), the program can be created.

3.2.4 Program saved in other names

In program interface, press  or  to select the program, only when the operation authority is above level [3], press  to save the selected program in the other name. The interface is shown as below:



确定

Input a new program name and press

确定

example, input 2222, press and the program #00005 can be saved in the name of #02222, the cursor jumps to the new program name.

Remark: Only the operation authority is above level (3), the program can be saved in other name.

3.2.5 Deleting the program

In program interface, press



or

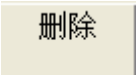
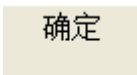
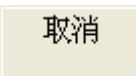


to select the program; when the operation

删除



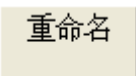
authority is above level (3), press to delete the selected program. The interface is shown as below:



For example: select program #0005, press  to remind whether delete the program, which is related with the value set by parameter CPD (NO.3202#5). And then, press  to delete the selected program, press  to cancel deleting and return to the previous menu.

- Remark:
- 1) The running program can't be deleted.
 - 2) Only when the operation authority is above level (3), the program can be deleted.

3.2.6 Renaming the program

In program interface, press  or  to select the program, when the operation authority is above level (3), press  to rename the selected program. The interface is shown as below:

自动

复位

程序 -> 本地目录

程序数: 7

占用空间(字节): 10,561

剩余空间(字节): 16,349,184

程序名	注 释	长度(字节)	修改时间
00001	实例加工	1,635	2009-03-25,16:11:27
00005	00005	14	2009-03-25,16:14:29
01111	说明书的所有程序	5,311	2009-03-25,15:18:31
08000	测试主程序	347	2009-03-25,15:18:33
08001	G00/G01测试程序	714	2009-03-25,15:18:36
08002	G02/G03测试程序	1,355	2009-03-25,15:18:37
08090	G90/G94测试程序	1,185	2009-03-25,15:18:38

请输入程序名

将程序 01111 重命名为

0 |

17:14:20

确定

取消

Input a new program name and press **确定** to rename the selected program as the new

name and return. Press **取消** to cancel renaming and return to the previous menu.

Remark:


- 1) The file which is uploaded or is running can't be renamed.
- 2) Only when the operation authority is above level (3), the program can be renamed.

3.2.7 Output the program

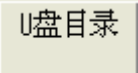
When the system is with flash disc, it is shown as the following graph:



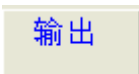


输出

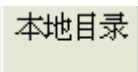
Press  to copy the program in flash disc directory to the local directory, vice versa. The detailed steps of the program in flash disc being copied to the system are as below:



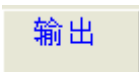
U盘目录

(1) Press  to access the directory of the flash disc;

(2) Press  or  to select the program to be copied; press  to copy the selected program to the local directory;

本地目录

(3) Press  to access the system program directory;

(4) Press  or  to select the program to be copied; press  to copy the selected program to the flash disc;

(5) When the copied program exists, remind: The program exists, whether replace? Program #00001 is shown as below:

自动

复位

程序 -> 本地目录

程序数: 7

占用空间(字节): 10,561

剩余空间(字节): 16,348,160

程序名	注 释	长度(字节)	修改时间
00001	实例加工	1,635	2009-03-25,16:11:27
00005	00005	14	2009-03-25,16:14:29
01111	说明书的所有程序	5,311	2009-03-25,15:18:31
08000	测试主程序	347	2009-03-25,15:18:33
08001	G00/G01测试程序	714	2009-03-25,15:18:36
08002	G02/G03测试程序	1,355	2009-03-25,15:18:37
08090	G90/G94测试程序	1,185	2009-03-25,15:18:38

粘贴

文件00001.CNC已经存在,是否覆盖?

[是]

直接覆盖

[否]

另存为

[取消]

取消粘贴

是

否

取消

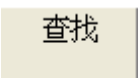
17:26:41

Then, press “yes” to replace the existed program; press “no” and the program can be saved in the other name, press “cancel” to cancel the operation.

Copying the system file to the flash disc is similar with copying the files in flash disc to the system.

Remark: Only when the operation authority is above level [3], the copying and pasting can be operated.

3.2.8 Searching for the program

In program interface, press  to access the search interface, which is shown as below:

自动

复位

程序 -> 本地目录

程序数: 7 占用空间(字节): 10,561 剩余空间(字节): 16,349,184

程序名	注 释	长度(字节)	修改时间
 00001	实例加工	1,635	2009-03-25,16:11:27
00005	00005	14	2009-03-25,16:14:29
01111	说明书的所有程序	5,311	2009-03-25,15:18:31
08000	测试主程序	347	2009-03-25,15:18:33
08001	G00/G01测试程序	714	2009-03-25,15:18:36
08002	G02/G03测试程序	1,355	2009-03-25,15:18:37
08090	G90/G94测试程序	1,185	2009-03-25,15:18:38

请输入查找的程序名

0

17:16:40

确定

取消

Input the program name

0

 to be searched, press

确定

 to search the input program and position the searched program name.

3.2.9 Sequence of the programs

In program interface, press

排列名称

,

排列大小

 or

排列时间

 and display the programs in the required sequence.

Repeatedly press

排列名称

,

排列大小

 or

排列时间

 and the sequence of programs is switched between plain sequence and reverse one.

3.2.10 MDI program

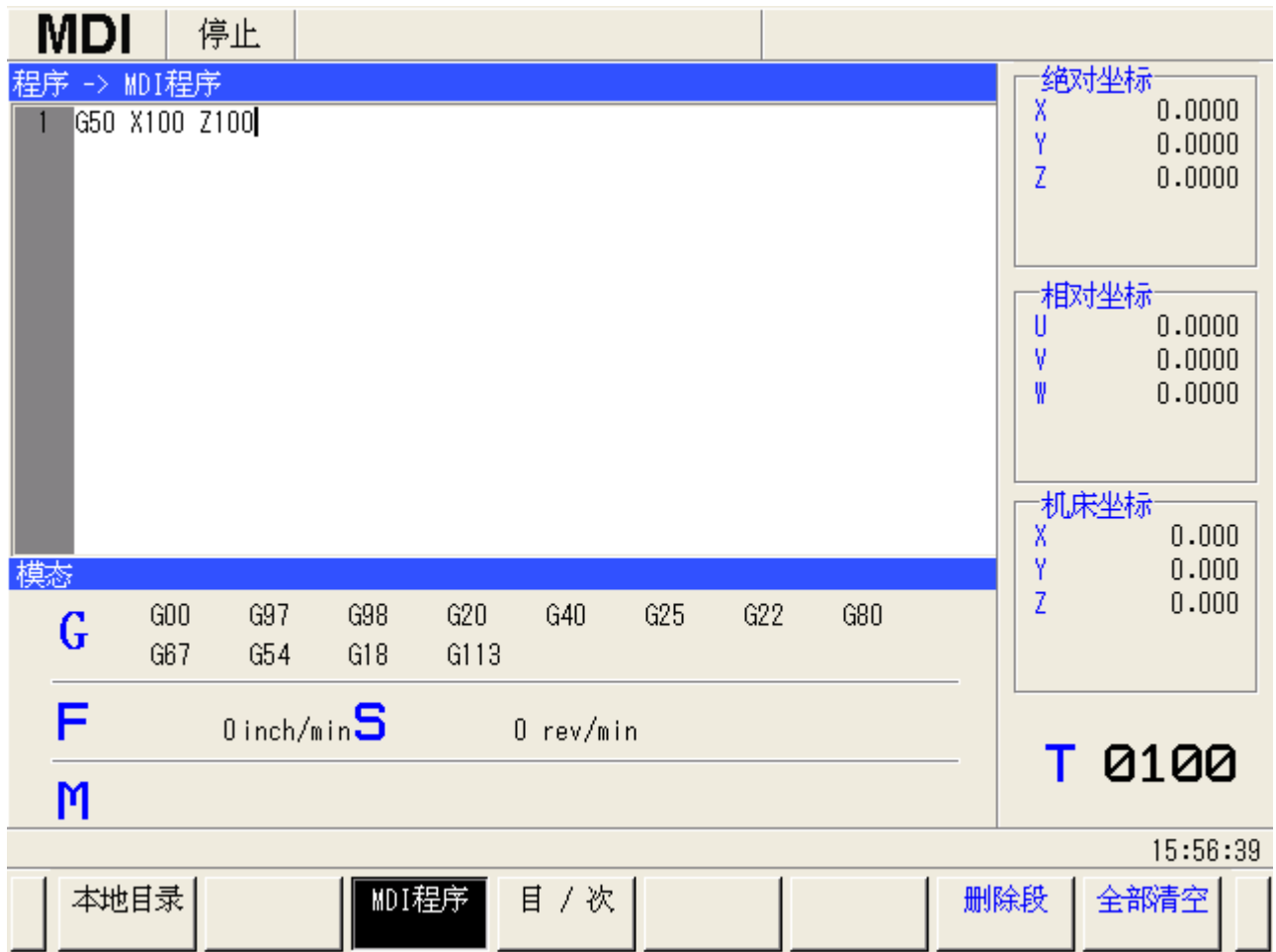
Press





PROGRAM

 to access the program interface, press


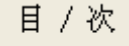
MDI程序

 to display MDI program input box, G, F and S modes and executed M codes. In MDI mode, input NC program not more than 10 lines in MDI program input box, which is shown as the following graph:



In MDI mode, it also displays  and . Press  to delete NC code which the cursor is and press  to clear all NC codes in MDI program input box.

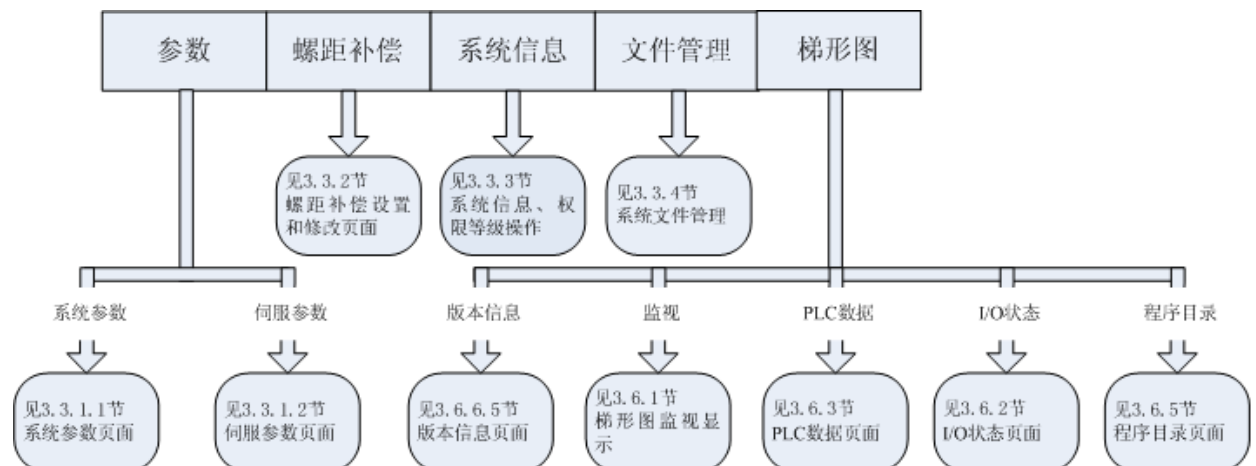
3.2.11 DIR/NXT

Press  to access program interface, press  to display the current block and NC command in next block, which is shown as the following figure:




3.3 System interface

Press **SYSTEM** to access the system interface. It mainly includes interfaces of parameter, screw pitch compensation, system information, file management and ladder diagram, etc. Check the content in each interface through the corresponding soft keys, and the structure of the soft key is shown as below:



3.3.1 System parameter setting and rewriting interface

参数

On the system interface, press  to access parameter setting interface, which is shown as the following figure:

自动

复位

系统 -> 参数 -> 系统参数

0000	SEQ				INI			
0	0	0	0	0	0	0	0	
0123	BPS 115200							
0138	OWN							
0	0	0	0	0	0	0	0	
1001					INM			
0	0	0	0	0	0	0	0	
1002					AZR		DLZ	
0	0	0	0	0	0	0	0	
1004	RPR				ISC			
0	0	0	0	0	0	0	0	
1005					HJZ _x		DLZ _x ZRN _x	
X	0	0	0	0	0	0	0	
Z	0	0	0	0	0	0	1	

0000 *--SEQ--INI--*

17:33:19

参数

螺距补偿

系统信息

文件管理

梯形图

系统参数

伺服参数

3.3.1.1 System parameter interface





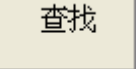
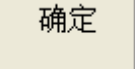

参数

系统参数

On the system interface, press  and  to access the system parameter setting interface.







The interface displays the detailed information of the user parameter, set and rewrite the system parameter in the interface, back up the parameter set currently, and initialize the parameter default by the system or the parameter of user backup.

In MDI mode, when the parameter switch is on and the operation authority is above level [3], the parameter can be set. Press , ,  or  to select the parameter to be rewritten; or press  to input the parameter sequence number to be selected; press  and the cursor positions in the parameter, like parameter #0000, which is shown as above; press , the parameter can be rewritten, parameter #0000 is shown as the following figure:

MDI		停止					
系统 -> 参数 -> 系统参数							
0000		SEQ				INI	
	00000000						
0123	BPS	115200					
0138	OWN	0	0	0	0	0	0
1001		0	0	0	0	0	INM 0
1002		0	0	0	AZR 0	DLZ 0	0
1004	RPR	0	0	0	0	ISC 0	0
1005					HJZ _x 0	DLZ _x 0	ZRN _x 1
X	0	0	0	0	0	0	1
Z	0	0	0	0	0	0	1
0000 *—*—SEQ—*—*—INI—*—*							
11:24:21							
^	备份	恢复					查找

Press the numerical key to rewrite the value of 8 bits in binary system, and press **INPUT** to confirm the setting is completed; if the value is less than 8 bits, zeroing in upper bit;





Moreover, set bit parameter based on the bits;

(1) In parameter setting interface, press , ,  or  to select the parameter to be set.





(2) Press  or  to select the parameter bit to be rewritten.

(3) Repeatedly press **INPUT** the parameter bit is switched between 0 and 1, and the value of the parameter bit is rewritten.

(4) Move the cursor to complete the setting.

Press , ,  or  to select the other parameters to be set.

The setting of parameter in decimal system is similar with that of parameter in bit type:

(1) Press , , ,  to select the parameter to be rewritten; Or press

查找

to input the parameter sequence number to be selected. When press **确定**, the

cursor positions to the parameter, parameter #0123 is shown as the following figure:

MDI		停止							
系统 -> 参数 -> 系统参数									
0000	0	0	SEQ	0	0	0	INI	0	0
0123	BPS								
115200									
0138	0	0	OWN	0	0	0	0	0	0
1001	0	0	0	0	0	0	0	0	INM
1002	0	0	0	0	AZR	0	DLZ	0	0
1004	0	0	RPR	0	0	0	0	ISC	0
1005	X	0	0	0	0	0	0	DLZx	ZRNx
	Z	0	0	0	0	0	0	0	1
0123 串口波特率(BPS)									
设置串口通讯的波特率(4800,9600,19200,38400,57600,115200)									
11:30:00									
^	备份	恢复						查找	

(2) Press and the selected parameter can be rewritten and parameter #0123 is shown as below:

MDI		停止							
系统 -> 参数 -> 系统参数									
0000	0	0	SEQ	0	0	0	INI	0	0
0123	BPS								
	115200								
0138	0	0	OWN	0	0	0	0	0	0
1001	0	0	0	0	0	0	0	0	INM
1002	0	0	0	0	AZR	0	DLZ	0	0
1004	0	0	RPR	0	0	0	0	ISC	0
1005	X	0	0	0	0	HJZx	0	DLZx	ZRNx
	Z	0	0	0	0	0	0	0	1
0123 串口波特率(BPS) 设置串口通讯的波特率(4800,9600,19200,38400,57600,115200)									
11:25:35									
^	备份	恢复						查找	

(3) Press the numerical key to input the numerical value to be set, and then press **INPUT** to confirm the setting is completed.

(4) Press , ,  or  to select the other parameter to be set.

Except set and rewrite the relative parameters, it can also back up the user parameter and restore the user backup parameter or the system default parameter.

Before the user rewrites the parameter, press **备份** to back up the parameter; when the parameter is wrongly rewritten or isn't required rewriting, press **恢复** to restore the parameter of user backup before rewriting or the parameter default by the system.

Press **备份** on the parameter interface, which is shown as below:

MDI		停止							
系统 -> 参数 -> 系统参数									
0000	0	0	SEQ	0	0	0	INI	0	0
0123	BPS								
115200									
0138	0	OWN	0	0	0	0	0	0	0
1001	0	0	0	0	0	0	0	0	INM
1002	0	0	0	0	AZR	0	DLZ	0	0
1004	0	RPR	0	0	0	0	ISC	0	0
1005	X	0	0	0	0	0	HJZx	DLZx	ZRNx
	Z	0	0	0	0	0	0	0	1
备份参数									
确定要备份参数吗?			4800,9600,19200,38400,57600,115200)						
11:26:10									
确定		取消							

Press to back up the parameter set by the user currently.

Press , the interface is shown as below:

MDI		停止							
系统 -> 参数 -> 系统参数									
0000	0	0	SEQ	0	0	0	INI	0	0
0123	BPS								
115200									
0138	0	OWN	0	0	0	0	0	0	0
1001	0	0	0	0	0	0	0	0	INM
1002	0	0	0	0	AZR	0	DLZ	0	0
1004	0	RPR	0	0	0	0	ISC	0	0
1005	X	0	0	0	HJZx	0	DLZx	0	ZRNx
恢复参数									
[用户参数] 恢复用户备份参数									
[默认参数] 恢复系统默认参数									
[取消] 取消操作			0,19200,38400,57600,115200)						
11:26:30									
用户参数		默认参数		取消					

Press **用户参数** to restore the parameter of user backup; press **默认参数** to restore the parameter default by the system.

On the parameter setting interface, press **螺距补偿**, **诊断**, **系统信息** and **文件管理** to respectively switch among the interfaces of the screw pitch compensation setting, the diagnosis, the system information check and the file management.

Remark:


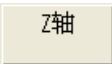

- 1) After rewrite the system parameter, some parameter can become valid immediately, some parameter becomes valid after the system powers on again, refer to 988T parameter introduction.
- 2) Only in MDI mode, when the parameter switch is on and the operation authority is above level (3), the parameter can be set and rewritten.



3.3.1.2 Servo parameter interface

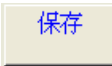
On the system interface, press **参数** and **伺服参数** to access the servo parameter setting interface.

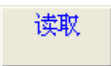


The servo parameter interface mainly includes checking the servo parameter and rewriting and saving the servo parameter in CNC side.

Switching the axes: Press ,  and  to switch the servo parameters among X, Z and S axes.

Rewriting the parameter: Press  to input the parameter value; press  again to complete the rewriting.

Saving the parameter: After rewriting the servo parameter, press  to save the rewritten parameter value after servo power off.

Reading the parameter: Press  to manually upload the servo parameter connected with the current axis.







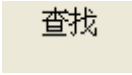
- Remark: 1) Before using, the servo system should be connected correctly and the configuration of the servo subunit number should be right.
- 2) Only in MDI mode, when the parameter switch is on and the operation authority is above level (3), the parameter can be set and rewritten.


3.3.2 Screw pitch compensation setting and rewriting interface

On the system interface, press  to access the screw pitch compensation interface, which is shown as below:


MDI		停止					
系统 -> 螺距补偿							
螺补号	补偿值	螺补号	补偿值	螺补号	补偿值	螺补号	补偿值
0000	2	0001	0	0002	0	0003	0
0004	0	0005	0	0006	0	0007	0
0008	0	0009	0	0010	0	0011	0
0012	0	0013	0	0014	0	0015	0
0016	0	0017	0	0018	0	0019	0
0020	0	0021	0	0022	0	0023	0
0024	0	0025	0	0026	0	0027	0
0028	0	0029	0	0030	0	0031	0
0032	0	0033	0	0034	0	0035	0
0036	0	0037	0	0038	0	0039	0
0040	0	0041	0	0042	0	0043	0
0044	0	0045	0	0046	0	0047	0
0048	0	0049	0	0050	0	0051	0
0052	0	0053	0	0054	0	0055	0
0056	0	0057	0	0058	0	0059	0
17:38:08							
参数		螺距补偿	系统信息	文件管理	梯形图		查找

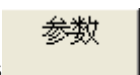
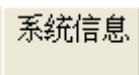
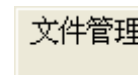
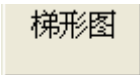
On the interface, the user can check and set the screw pitch compensation value corresponding to each screw pitch number.

On the screw pitch compensation interface, press  or  and , ,  or  to select the compensation value of screw pitch compensation number to be set, or press  to search for the screw pitch compensation number and the cursor positions to the compensation value of screw pitch compensation to be rewritten.

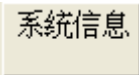
When the operation authority is above level (2), press  and the compensation value of the screw pitch compensation number can be rewritten, the compensation value #0000 is shown as below:

MDI	停止						
系统 -> 螺距补偿							
螺补号	补偿值	螺补号	补偿值	螺补号	补偿值	螺补号	补偿值
0000	4	0001	0	0002	0	0003	0
0004	0	0005	0	0006	0	0007	0
0008	0	0009	0	0010	0	0011	0
0012	0	0013	0	0014	0	0015	0
0016	0	0017	0	0018	0	0019	0
0020	0	0021	0	0022	0	0023	0
0024	0	0025	0	0026	0	0027	0
0028	0	0029	0	0030	0	0031	0
0032	0	0033	0	0034	0	0035	0
0036	0	0037	0	0038	0	0039	0
0040	0	0041	0	0042	0	0043	0
0044	0	0045	0	0046	0	0047	0
0048	0	0049	0	0050	0	0051	0
0052	0	0053	0	0054	0	0055	0
0056	0	0057	0	0058	0	0059	0
17:38:48							
参数	螺距补偿	系统信息	文件管理	梯形图		查找	

Input the compensation value through the numerical key, and then press  to complete rewriting.

On parameter setting interface, press , ,  and  to switch among the interfaces of the system parameter setting, the system information check, the file management and the ladder diagram.

3.3.3 System information and operation authority levels

On the system interface, press  to access the system information interface, which is shown as below:

自动	停止						
系统 -> 系统信息							
系统信息							
产品型号	GSK988T						
软件版本	V1.02						
硬件版本	V3.02.003						
BOOT版本	V1.7						
系统编号	454578878						
PLC型号	PLC-N1						
操作权限							
当前的操作权限等级 [2]							
机床厂家级。							
可修改参数,编辑零件程序,修改刀具补偿,编辑梯形图程序,螺距误差补偿数据录入。							
							15:46:37
参数	螺距补偿	系统信息	文件管理	梯形图	权限降级	修改密码	输入密码

On the system information display interface, it mainly displays the product type, software, hardware and BOOT versions, system serial number, PLC modal and the operation authority. On the interface, the operation authority level password can be rewritten and the operation authority level can be set, etc.

To realize the multi-level operation authority management of the development, maintainance, machine design and equipment management, etc, GSK988T CNC system sets operation authority of 5 levels, 1 is the superlative, 5 is the lowest;

☐ Level 1: Development with system software maintainance authority;

☐ Level 2: Machine manufacturer with the authority of PLC program editing, screw pitch error compensation data input and switch off the machine in limited time;

☐ Level 3: (User) equipement management with the authority of rewriting the parameter, editing the part program and the tool compensation data;

☐ Level 4: Machine operation level with the authority of editing the tool compensation data and selecting the part program (namely: operate the tool-setting, select the part program of auto running), but the parameter can't be rewritten and the part programs can't be edited;

☐ Level 5: operation limit level, without operation password (the operation password is canceled), the parameter can't be rewritten, the tool compensation data can't be edited, and the part program neither be selected nor edited (namely, the tool-setting is invalid, only run the current part program), manual, MPG, zero-return, MDI running and auto running can be operated, the part files of the system can back up rather than download.

Remark: Upload means uploading the files of CNC to PC and download means downloading the files to CNC.

The list of operation function relative with the operation authority levels:

Operation function	Operation authority level				
	Level 1 (Development)	Level 2 (Machine manufacturer)	Level 3 (Equipment management)	Level 4 (Machine operation)	Level 5 (limited operation)
System software upgrade	OK	NO	NO	NO	NO
Set the limited time of the system auto off	OK	OK	NO	NO	NO
PLC program editing, downloading and uploading	OK	OK	NO	NO	NO
Input the screw pitch error compensation data and download the screw pitch compensation file	OK	OK	NO	NO	NO
Upload and download the part program	OK	OK	OK	NO	NO
The parameter switch on (Allowable rewriting the parameter)	OK	OK	OK	NO	NO
The program switch on (Allowable editing the program)	OK	OK	OK	NO	NO
Set tool lift and download its files	OK	OK	OK	OK	NO
Input the macro variable	OK	OK	OK	OK	NO
Input the tool compensation data (allowable tool-setting) and download the tool compensation and the tool offset files	OK	OK	OK	OK	NO
Upload the screw pitch compensation files	OK	OK	OK	OK	OK
Upload the tool life file	OK	OK	OK	OK	OK
Upload the tool compensation and the tool offset files	OK	OK	OK	OK	OK

If execute the operation limited by the authority level, the corresponding authority must be obtained. Press **SYSTEM** on GSK988T panel to access the system interface, and then press

系统信息

权限降级

修改密码

输入密码

to access the password interface, finally press , or to access the corresponding setting, and input the password corresponding to the operation level, the relative authority is obtained. On the password setting interface, the password of the level or lower than the level can be rewritten, and the current password level can be degraded.

The operation authority of level 1 isn't saved after power off, and access level 2 after power on, again. The operation authority of levels 2~5 are saved, restore the operation authority level after power on again.

When execute the operation which doesn't reach the authority level, it reminds the current operation authority isn't enough in lower left; in auto mode, when the operation level isn't enough, the machine stops moving and alarms.

(1) Access the authority level

权限降级

Press to degrade the operation authority level, display the current operation authority level in the authority box.

输入密码

When is pressed to input the password corresponding to the level to access the level operation authority.

Remark: The corresponding relation between the initial password relative to each authority level is shown as below:

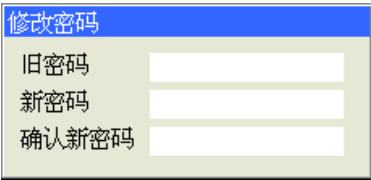

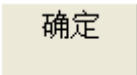
Operation levels	Initial password
Level 1	***
Level 2	222222
Level 3	333333
Level 4	444444
Level 5	Without password

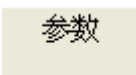
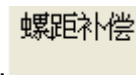
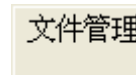
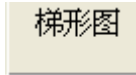
(2) Rewriting the password

修改密码

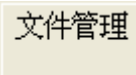
Firstly, access the operation authority level to rewrite the password; press the system authority register password, the interface is shown as below:



Input the old and new passwords in ; and press  to switch between the new and old passwords. Finally, press  to complete the rewriting password.

In parameter setting interface, press , ,  and  to switch among interfaces of the parameter setting, the screw pitch compensation setting, the file management and the ladder diagram.

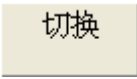
3.3.4 System file management

On the system interface, press  to access the file management interface. The interface is shown as below:







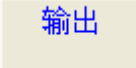

The window is divided into left and right columns. The left column displays the system files and the part program file directories; when the system is with the flash disc, the right column displays the file directory in the flash disc, which is shown as the following figure. Then, input or output the system files, the files in the system can be output to the flash disc, or the file in the flash disc can be input into the system.


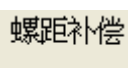
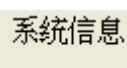
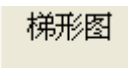


(1) Press  and the cursor can switch between the system file directory in left column and the file directory of flash disc in right column.

(2) When the cursor is on the file, press  or  to open or close the file.

(3) Press  or  and move to the document to be operated, press  to select the document, the selected document is ticked, such as the part programs 00098, 00003 and 00777 in the system file directory, which is shown as above. When the cursor is on the file, then, press  to select all documents in the file.

(4) Then , after select the files in the system, press  to output all the selected files to the flash disc; After selecting the files in the flash disc, press  to input all the selected files in the flash disc to the system file directory.

In the parameter setting interface, press , ,  and  to switch among interfaces of the system parameter setting, the screw pitch compensation setting, the system information check and the ladder diagram.

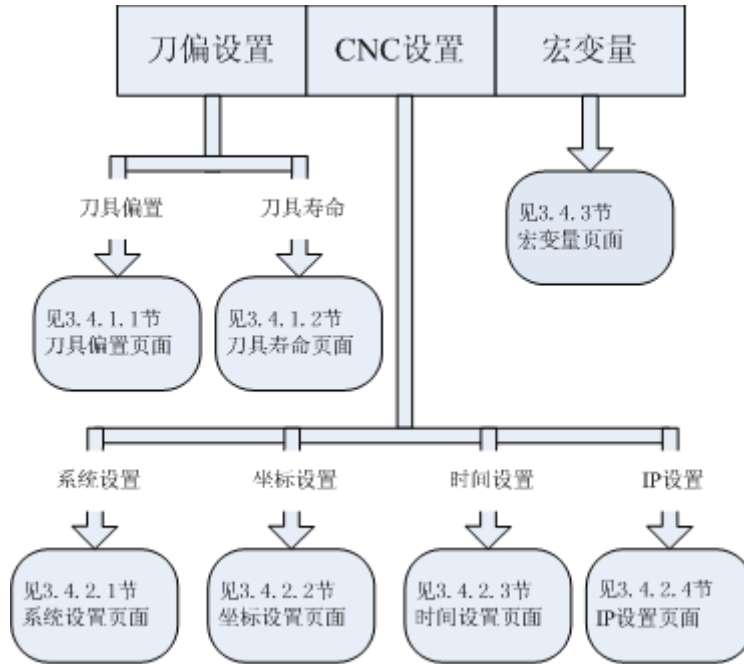
3.3.5 Ladder diagram

Because there are too many interfaces about the ladder diagram, it is introduced independently, about the ladder diagram interfaces refer to chapter 3.6.

3.4 Setting interface

SETTING

Press **SETTING** to access the setting interface. It mainly includes interfaces of the tool offset and CNC setting and macro variable, etc. The content can be checked through the corresponding soft keys. The structure of the soft key layers is shown as below:



3.4.1 Tool offset setting

刀偏设置

Press **刀偏设置** to access the tool compensation interface. It mainly includes two interfaces of the tool offset and the tool life management, and the content in each interface can be checked through pressing the corresponding soft keys, which is shown as below:



3.4.1.1 Tool offset

Press **刀具偏置** to access the tool offset setting interface, which is shown as below:



On this interface, check and set each axis tool offset value and wearing value corresponding to each tool offset number, about the detailed method, refer to chapter 6.6.

In the right column of the tool offset setting interface, it also displays the information of the current absolute coordinate and the relative coordinate values and the tool number of the current running program, etc.

Remark:

1. The quantity of axes in interface is set by parameters #1010 and #8130, and it doesn't display the rotation axis, the tool offset value is only valid to the straight axis rather than the rotation axis.
2. The straight axis or the rotation axis is set by composition of 0 and 1 bit of parameter #1006.
3. The name of each axis is set by parameter #1020.
4. The tool offset value of each axis is specified by the diameter or the radius value, which is set by the 1st bit of parameter #5004; and the movement value of each axis is specified by the diameter or the radius, which is set by the 3rd bit of parameter #1006.
5. Only the operation authority is above level (4), the tool offset and the wearing value can be set.

刀具寿命

In the tool offset interface, press 刀具寿命 to switch into the tool life interface.

3.4.1.2 Tool life

刀具寿命

In the tool compensation interface, press 刀具寿命 to access the tool life management interface and it displays the life set by the tool group of each tool group number and the actual life of the current tool, which is shown as below:

MDI

复位

设置 -> 刀偏设置 -> 刀具寿命

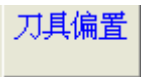
刀组号	刀具组设定寿命				当前刀具实际寿命			
1	10				0			
	0	1	2	3	4	5	6	7
	8	9	10	11	12	13	14	15
	16	17	18	19	20	21	22	23
	24	25	26	27	28	29	30	31
2	20				320			
	32	33	34	35	36	37	38	39
	40	41	42	43	44	45	46	47
	48	49	50	51	52	53	54	55
	56	57	58	59	60	61	62	63
3	30				640			
	64	65	66	67	68	69	70	71
	72	73	74	75	76	77	78	79

14:15:40

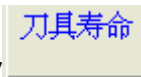
^

查找

On the interface, check the life set by each tool group and the actual life of the current tool through page keys and the cursors, but they can't be rewritten. The tool life can be rewritten and set by G commands in program.



On the tool life interface, press



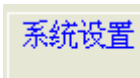
Remark: Whether display or not display, which is set by parameter #8132.0, namely, whether display the tool life interface.

3.4.2 CNC setting interface



On the setting interface, press to access CNC system setting interface and it mainly includes the system and the coordinate setting, system time and IP.

3.4.2.1 System setting interface



On CNC setting interface, press to access the system setting interface, which sets the program and the parameter switches, auto sequence number and input units, etc, which is shown as below:



On the interface, it mainly sets on or off of the program and the parameter switches, etc.



On the interface, press or to switch among the program switch, the parameter switch, auto sequence number and the input units, etc; In MDI, when the operation authority is above



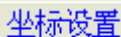
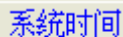

level (3), press , or to select on/off and metric /inch system.

In the right column, it also displays the current absolute position coordinate and the relative coordinate position value and the tool number of current running program.

坐标设置

系统时间

系统IP

Press  ,  and  to switch into the interfaces of coordinate setting, system time and IP setting.


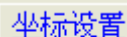
Remark:

- 1) Only when the program switch or the parameter switch is on, can the program and the parameter be edited, rewritten or set.
- 2) Only when the operation authority is above level (3), can CNC system be set.
- 3) Only when the program protection switch is on, which is installed on the machine panel, can the program and the parameter switches on/off be set.

3.4.2.2 Coordinate setting interface

SETTING

坐标设置

Press  to access the setting interface; on CNC setting interface, press  to access the coordinate setting interface, which is shown as below:



	X	Z	Y	B	C
原点偏移	0.0002	0.0003	0.0003	0.0003	0.0003
G54	0.0003	0.0003	0.0003	0.0003	0.0003
G55	0.0003	0.0003	0.0003	0.0003	0.0003
G56	0.0003	0.0003	0.0003	0.0003	0.0003
G57	0.0003	0.0003	0.0003	0.0003	0.0003
G58	0.0003	0.0003	0.0003	0.0003	0.0003
G59	0.0003	0.0003	0.0003	0.0003	0.0003

绝对坐标

X 0.8553

Z 0.0000

Y 0.0000

B 0.0000

C 0.0000

相对坐标

U 0.8553

W 0.0000

V 0.0000

B 0.0000

H 0.0000

机床坐标

X 26.2146

Z 14.4182

Y 35.3901

B 74.0559

C 360.0000

T 0000

18:25:17

In coordinate setting interface, it displays the origin offset value of each axis and the offset value of each coordinate axis in each coordinate system. Set the origin offset value relative to each axis and the offset value of each coordinate axis in each coordinate system.

In coordinate setting interface, press



or



to select the coordinate system to be set,

and press



or

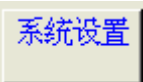
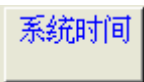



to select the coordinate axis to set the offset; in MDI mode, when the operation authority is above level (4), the coordinate offset data are rewritten through the numerical and the backspace keys.


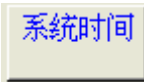
In the right column, it also displays the current absolute position coordinate, the relative coordinate position value and the tool number of current running program.

Remark:

- 1) Only in MDI mode, when the operation authority is above level (4), the coordinate offset can be set or rewritten.
- 2) The quantity of axes is set by parameters #1010 and #8130.
- 3) The name of each axis is set by parameter #1020.
- 4) The origin offset value of each coordinate in each coordinate system can be set by the parameter and the corresponding relation is shown as below:
Parameter #1220: The external work piece origin offset value of each axis.
Parameter #1221: Each axis origin offset value of work piece coordinate system 1 (G54).
Parameter #1222: Each axis origin offset value of work piece coordinate system 2 (G55).
Parameter #1223: Each axis origin offset value of work piece coordinate system 3 (G56).
Parameter #1224: Each axis origin offset value of work piece coordinate system 4 (G57).
Parameter #1225: Each axis origin offset value of work piece coordinate system 5 (G58).
Parameter #1226: Each axis origin offset value of work piece coordinate system 6 (G59).







Press ,  and  to switch between interfaces of setting system time or IP in CNC setting interface.







3.4.2.3 Setting system time interface




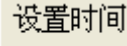
Press  to access setting interface; in CNC setting interface, press  to access setting system time interface, which is shown as below:









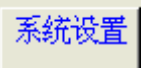
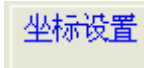
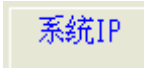
On the time setting interface, press  to switch among the date, month, year and time boxes in cycle.

Setting month: press  to switch into the month box, and it changes into green, press , ,  and  to change the month, press  to switch into the other box and the month setting completes.


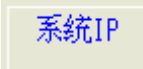
Setting year: press  to switch into the year box and it changes into green, press , ,  and  to change the year, press  to switch into the other box and the year setting completes.

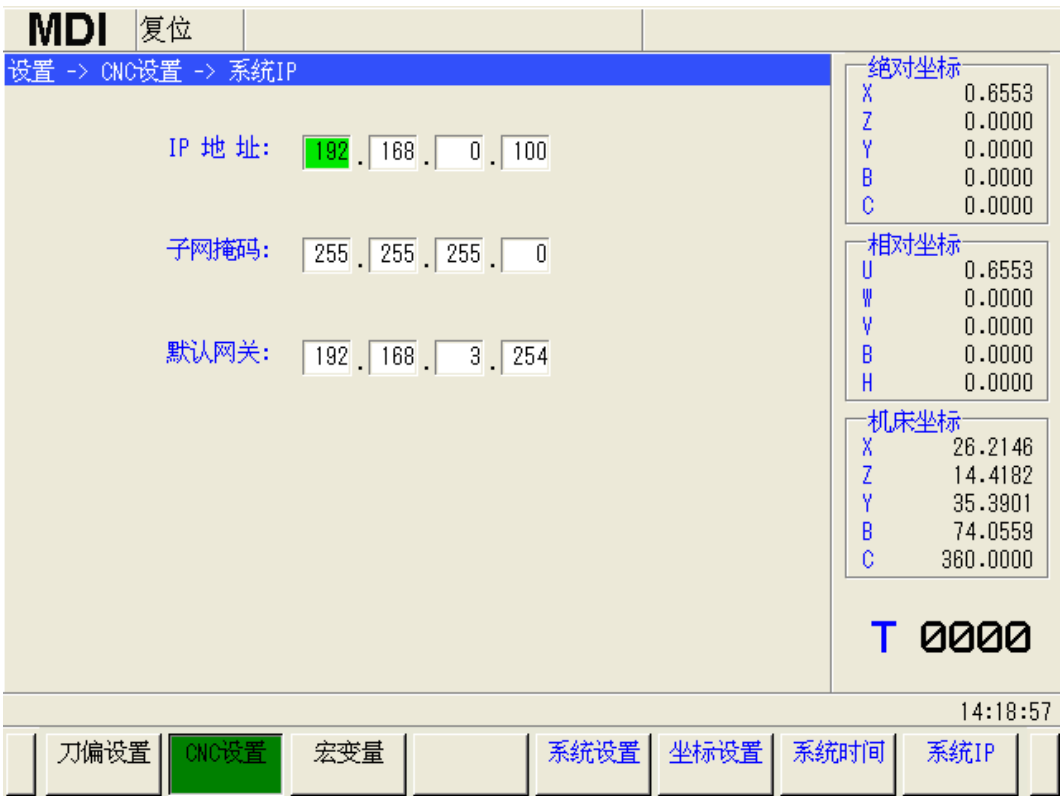
Setting time: press  to switch into the time box and it changes into green, press  and  to select the hour, the minute or the second and the selected boxes turn into green, and press address numerical keys to change the hour, the minute or the second, press  to complete the time setting. And then, the cursor moves to the date box, the time box changes into gray.





Setting date: press  to switch into the date box, the date box changes into green, press , ,  and  to change the date, press  to complete the date setting.

On setting time interface, press ,  and  to switch into the other interface of CNC setting interface.


3.4.2.4 Setting system IP interface

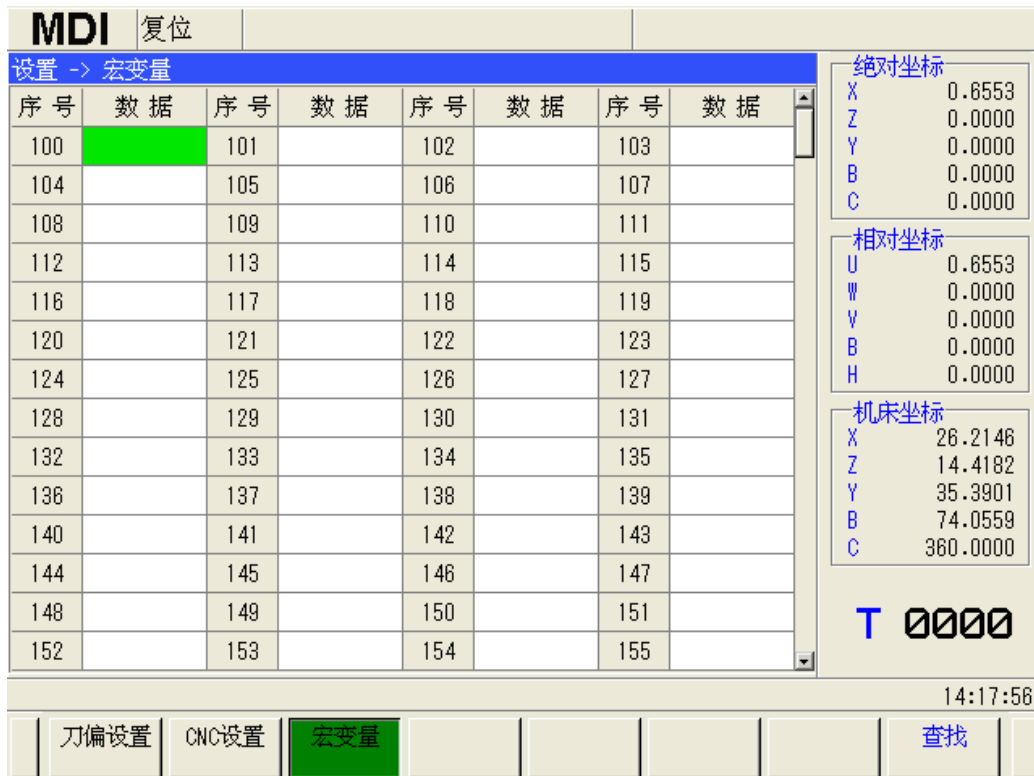
Press  to access the setting interface; on CNC setting interface, press  to access system IP setting interface, which is shown as below:









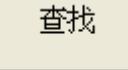
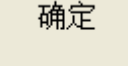
- 1) Press  and  to switch among IP address, subnet mast or default gateway column.
- 2) Press  and  to switch between each address box, input the address to be set.


3.4.3 Macro variable interface


On the setting interface, press  to access the macro variable interface, which is shown as below:



On the macro variable interface, check and set the value relative to each macro variable.


On macro variable interface, press ,  and , , ,  to select the macro variable to be rewritten, the selected macro variable changes into the green-based color, or press  to input the macro variable serial number to be selected, and then press  and the cursor positions in the data of the macro variable.

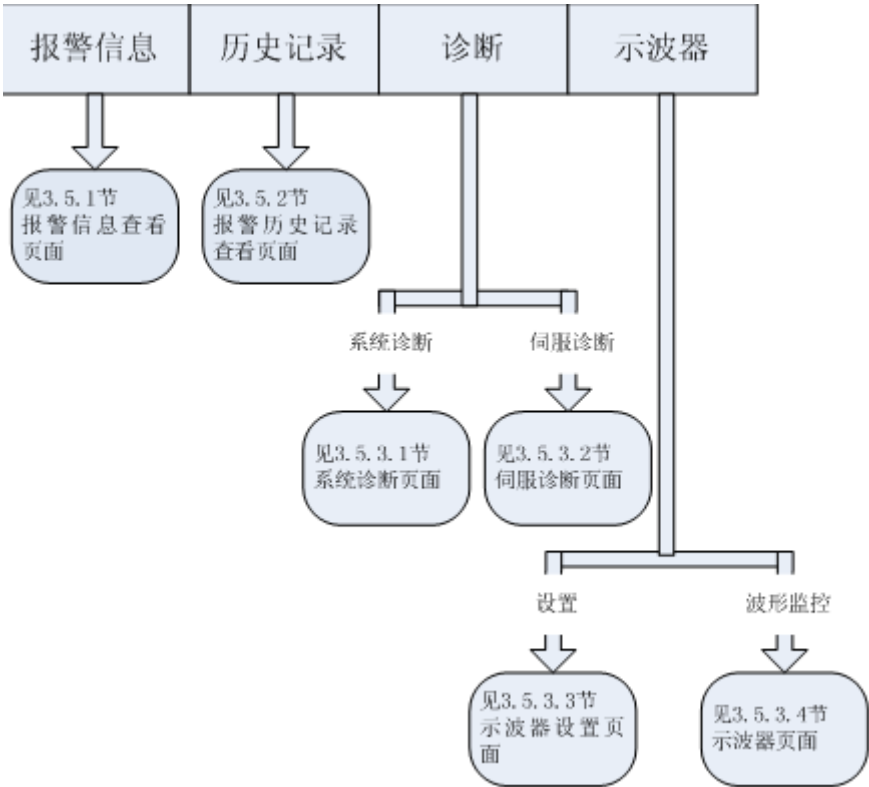
In MDI mode, the operation authority is above level (4), rewrite the macro variable data through numerical and backspace keys; or press  and the macro variable data can be rewritten, such as the macro variable data # 100, and rewrite the macro variable data through numerical and backspace keys.

And press , again to complete the rewriting.

On the macro variable interface, press  and  to switch into the other interface of setting interface.

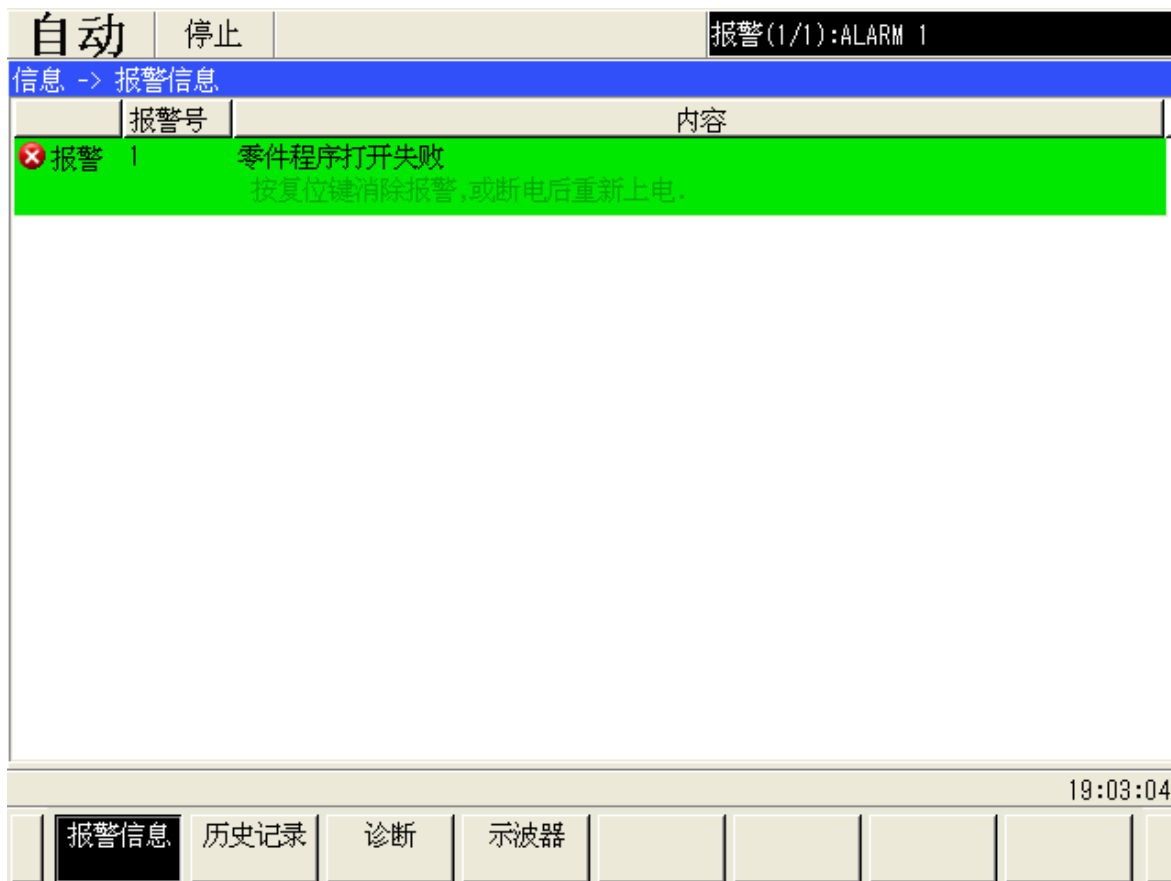
3.5 Information interface





Press  to access the alarm interface, there are three interfaces of alarm information, records and diagnosis, and check the content in each interface through pressing the corresponding soft keys. The structure of the software layers is shown as below:



3.5.1 Checking alarm information interface

On the information interface, press **报警信息** to access the alarm information interface, display the quantity CNC and PLC alarms and detailed information. The interface is shown as below:



On alarm information interface, the alarm information of CNC and PLC is listed in one window, and differed through the alarm number. Press  and  to scroll the list line by line, or press  and  to scroll the list page by page.

When PLC alarms or reminds, display information of address A in black; When CNC alarms or reminds, the reasons and trouble shooting is shown as black below the information line.

Cancel alarm: Press  to cancel all alarms.

Remark:

- (1) When PLC alarms or reminds, the information of address A displays in green below the information line;
- (2) When CNC alarms or reminds, the reason and the trouble shooting display in green below the information line.
- (3) Alarms of #0——1000 are CNC, alarms of #1000——2000 are PLC, after #2000, it is reminding information.
- (4) After the parameter is rewritten, which becomes valid after power on, the alarm can be cleared after power on again.



3.5.3.1 System diagnosis interface

Firstly access CNC diagnosis interface, press **系统诊断** to access the system diagnosis interface. In CNC system diagnosis interface, there is information of keypad and state diagnosis and

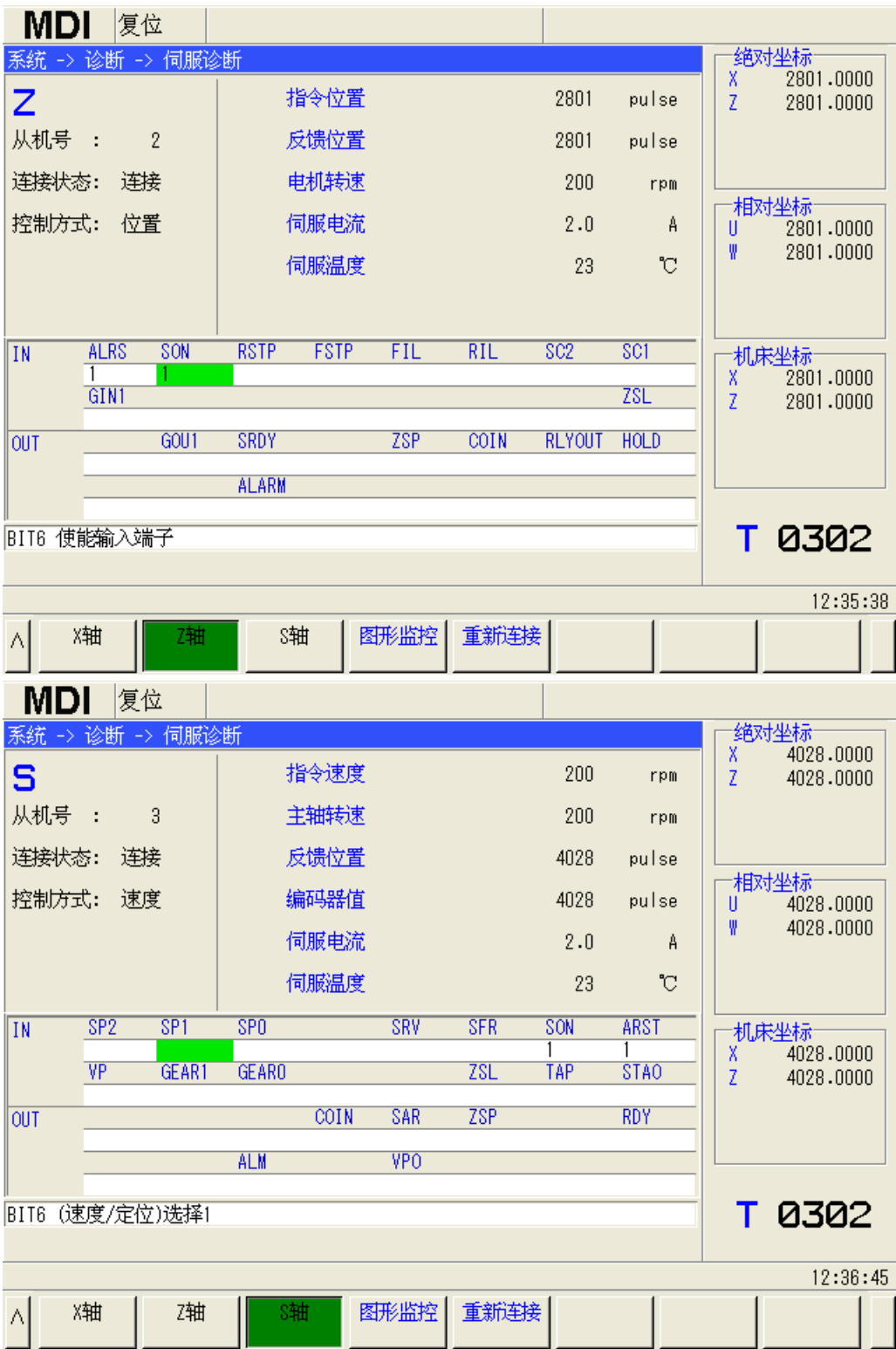
miscellaneous function parameter, etc. Press , , , , and to check the content. To prevent the corresponding function is operated during checking some keys, such as the direction and the interface keys, therefore, lock the current screen through

pressing **锁定屏幕**.

On CNC diagnosis interface, there are two lines to display the detailed content of the diagnosis numbers at the bottom, and the first line displays the diagnosis number; the second displays the meaning of some bit of the diagnosis number which the cursor is.

3.5.3.2 Servo diagnosis interface

On the system interface, press **诊断** to access the diagnosis interface and press **伺服诊断** to access the servo diagnosis interface.



There provides the following functions in 988T servo diagnosis mode:

Through the data of the servo communication feedback, real-time monitor the system control axis, then, the operator can learn the servo and the motor working state, etc, including:

- (1) The follow error analysis of the axis, the data are composed of two parts: The command data received by the servo and the data feed back by the encoder.
- (2) The axial state diagnosis information: The present operating current of the servo, the motor real-time speed, the internal temperature of the servo, the servo IO point state.

(3) The servo alarm information.

The introduction of each data in the servo diagnosis interface:

Z

: The name of the current selected axis

Subunit number: Number of the subunit connecting with the axis

Connecting state: Check whether the servo communication link layers are connected.

Control mode: The diagnosis data relative to the servo control mode, it may display as “position” and “speed”.

Command position: The quantity of the position pulses which the diagnosis data servo receives from the system.

Feedback position: The quantity of the position pulses (not include the servo gear ratio) feed back by the diagnosis data servo.

Command speed: The speed command value which the diagnosis data servo receives from the system.

Motor speed: The actual speed of the diagnosis data motor.

Spindle speed: The actual speed of the diagnosis data spindle.

Encoder value: The current value of the diagnosis data spindle encoder.

Servo current: Diagnosis the present operating current value of the diagnosis data servo.

Servo temperature: The measured temperature value in the diagnosis data servo.

IN: Value of the servo input point.

OUT: Value of the servo output point.

BIT7 alarm clear input terminal: The detailed explanation of the marked servo input and output points.

重新连接

: When the servo communication is wrong (alarm 2003: GSKLink communication is wrong.), connect all the servo subunits, again. If it fails, power on the servo and the system, again.

Remark: (1) About DAP03 spindle servo system, the subunit number of the spindle servo is the servo parameter PA119; the subunit number of DAT2050C feeding servo is servo parameter PA58, after rewriting, it should be saved.

(2) In control mode, display of the data is related with the setting of servo parameter PA4; about the details, refer to the relative servo manual.

(3) The data of spindle speed only displays when the axis is spindle or in C axis speed control mode.

(4) The data of the encoder value only displays when the axis is spindle or in C axis speed control mode.

(5) Before start the servo communication function, the relative parameter is collocated correctly; Otherwise, the servo community function isn't set, such as the parameters NO.9000, NO.9010, NO.9011 and NO.9012, about the details, refer to the parameter manual.

3.5.3.3 Oscillograph setting interface

Before use the oscillograph, set the units of the servo data and the waveform scaling corresponding to two monitor wave forms, the monitor type of the oscillograph and the sampling time, which is corresponding to the sampling type, in burst mode.



The introduction of the oscillograph items:

Selective axis: Select the axis to currently set the wave form correspondingly.

Operation: Press “input” and press UP/DOWN to select, and then press “input” again to confirm.

Monitor data: Select the servo data to currently set the wave form correspondingly

Operation: Press “input”, press UP/DOWN to select, finally press “input” again to confirm

At present, the data can be selected include:

The command position, the feedback position, the command speed, the feedback speed, the servo temperature and the servo current

Monitor mode: The oscillograph supports two monitor modes: memory mode and burst mode, select the suitable one for the user.

The difference between two monitor modes:

Property type	Sampling starting mode	Sampling finishing mode	Whether save the waveform data
Burst mode	Press start/stop	Sampling time up, auto stop	NO
Memory mode	Press start/stop	Servo alarm, auto stop	Save

Operation: Press “input”, and then press UP/DOWN to select, finally press “input” again to confirm.

Setting the unit (pulse/grid): The unit of the wave form on the vertical axis (the grid is taken as the unit.)

Operation: Directly input the numbers, and then press “input”. The input value out of the range is invalid.

Setting the sampling cycle (millisecond): At present, the data sampling cycle is fixed, which can't be rewritten.

Operation: Directly input the numbers, and then press "input". The input value out of the range is invalid.

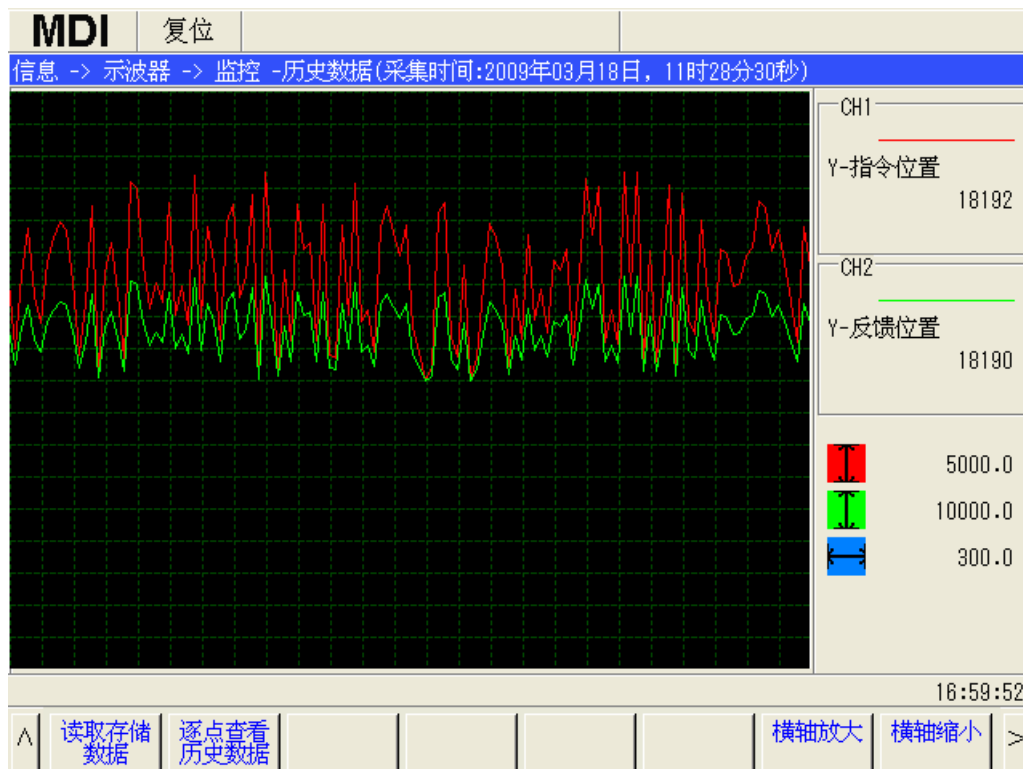
Setting the sampling time (millisecond): The monitor sampling time in burst mode is limited by the maximum sampling time.

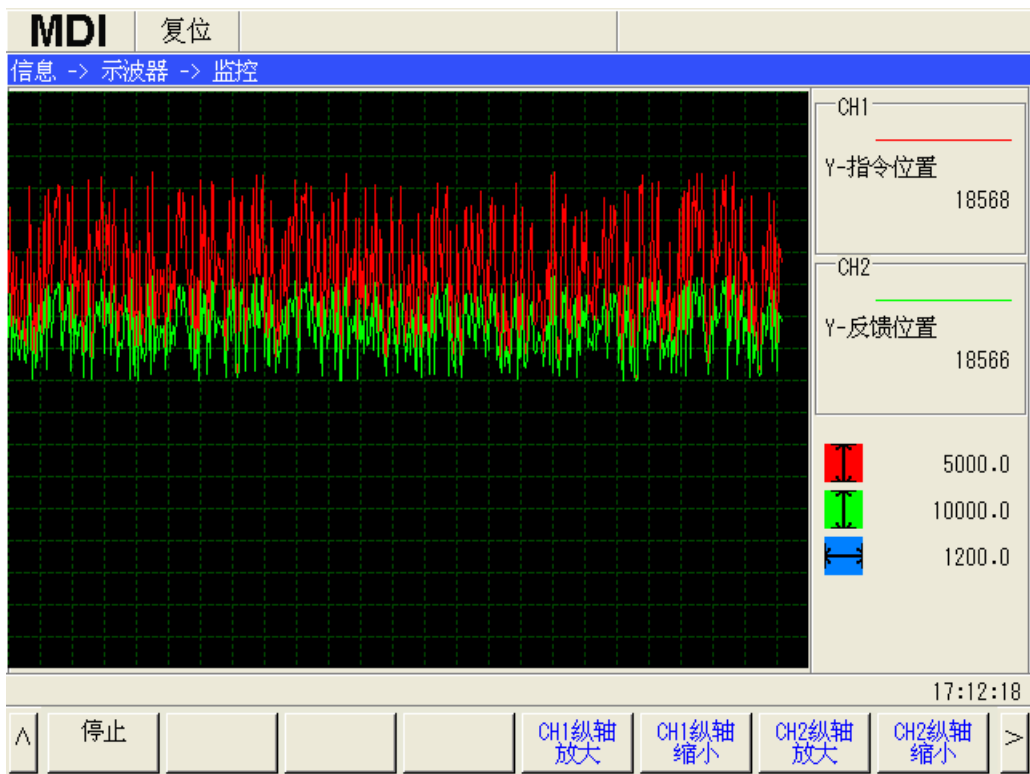
Operation: Directly input the numbers, and then press "input". The input value out of the range is invalid.

3.5.3.4 Oscillograph interface

The oscillograph interface is in the form of the oscillograph and displays the state data of GSKLink axis.

The user can select the oscillograph in memory mode or burst mode, respectively set the monitor data type relative to CH1 and CH2, real-time adjust the monitor wave form and check the lastest wave form of 1min when the monitor stops, or check the data which was saved in the oscillograph in memory mode last time.





Graph introduction:

—: CH1 (Channel I) wave form

—: CH2 (Channel II) wave form

—: CH1 Data unit

—: CH2 Data unit

—: Time axis unit

历史数据 (采集时间: 2009年01月15日, 10时20分03秒): Check the historical waveform data in the oscillograph in memory mode and the sampling time of waveform data.

Remark:

- (1) When the sampling cycle is 40m, the logest time limit of the historical data recorded by the system is about 1 min. If it exceeds the time limit, it will auto cover the previous memory area.
- (2) The unit of the monitor property data is same as that of the monitor data corresponding to the servo diagnosis interface.

Introduction of the soft key function

停止: Button of start/stop of the control data sample, it displays as “stop” during sampling, and it displays as “start” when the sampling stops.

CH1纵轴放大 CH1纵轴缩小 CH2纵轴放大 CH2纵轴缩小: Respectively scale the wave forms of CH1 and CH2 in the vertical axis.

横轴放大 横轴缩小: Scale the setting units of the time axis.

读取存储
数据

: Read in the historical data during sampling in the memory mode.

逐点查看
历史数据

: When stop sampling, select the relative data of the last point or the last grid, which displays in the right owned window.

3.6 Ladder diagram interface

PROGRAM

梯形图

Firstly press **PROGRAM**, and press **梯形图** to access the current PLC interface, and real-time check PLC executing situation. The ladder diagram interfaces mainly include the interfaces of the version information, monitor, I/O state, PLC data and the program directory, etc, and check the content in each interface through pressing the corresponding soft keys. The interface is shown as the following figure 1:



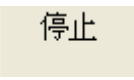
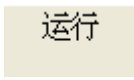
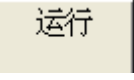
Fig.1

On the top of the interface, it displays the system current running mode and the state, the version information of the ladder diagram, the current running ladder diagram program and its running state, etc.


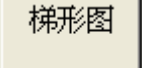

版本信息

停止

Press **版本信息** to access the version information interface, and then, press **停止** to stop the running state of the current ladder diagram program, and PLC state line displays "stop" and

its background changes into the red-based color, then,  turns into , and then, press  and the ladder diagram program can run, again, PLC state line displays “running” and its background becomes green-based color.

3.6.1 Monitor display of ladder diagram

Firstly press  and press  to access the ladder diagram interfaces, press  to access the monitor interface of the currently running ladder diagram program and the interface is shown as the following figure 2:

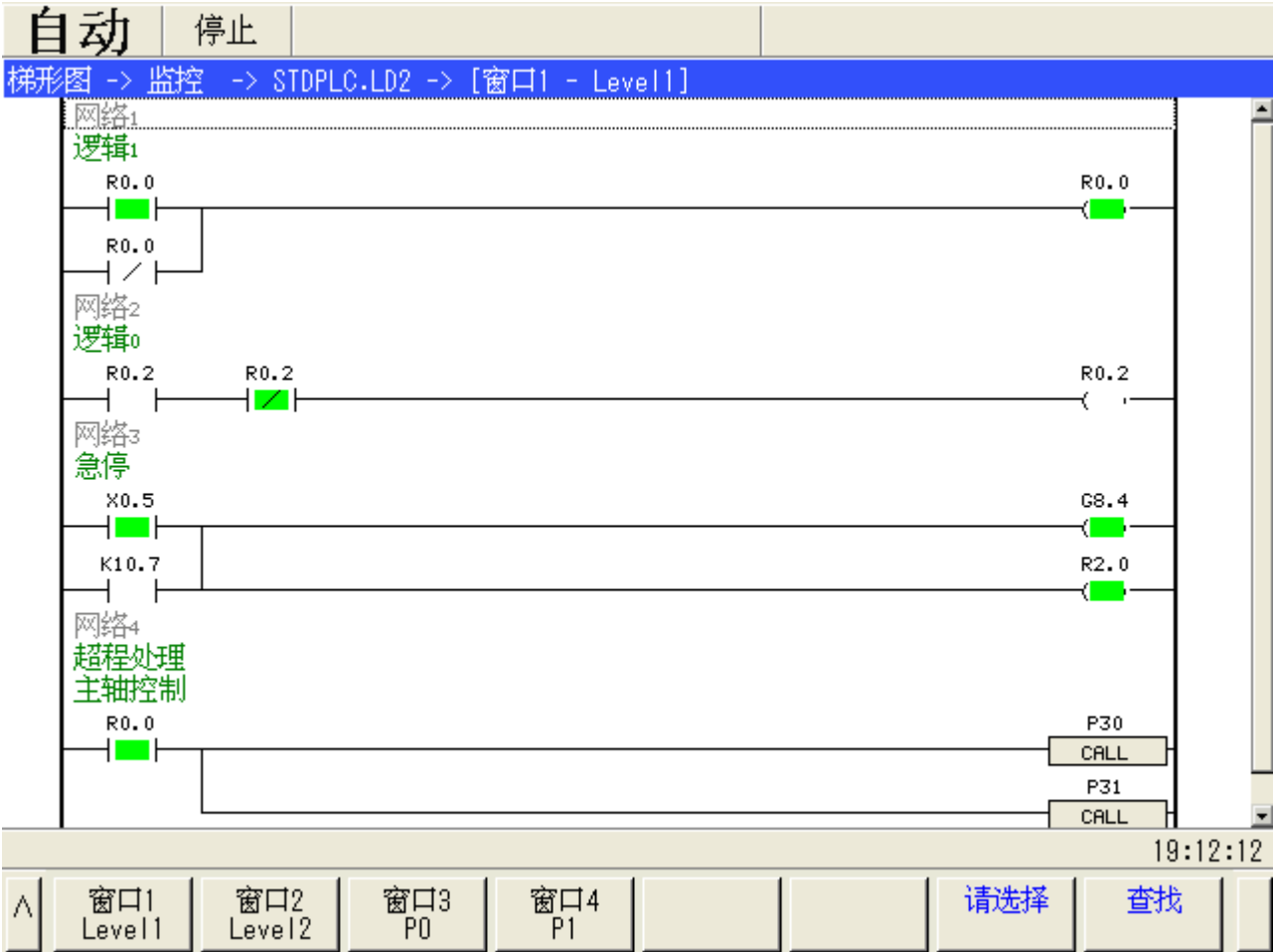




Fig.2

On the monitor interface, check the current contacts, connecting/disconnecting of the circuit and the current values of the timer and the counter. When the circuit and contacts are connected, it's the green-based color; disconnect, the based color is same as window one. For example,  means X0.5 contact is connected;  means Y25.2 circuit is disconnected.

1. Checking the window program

On the monitor interface, the programs on the four windows can be monitored at the same time.

Separately press **窗口1 Level1**, **窗口2 Level2**, **窗口3 P0** and **窗口4 P1**, respectively check the ladder diagram of the block corresponding to each window, and then, the ladder diagram of the block corresponding to the selected window displays on the screen.

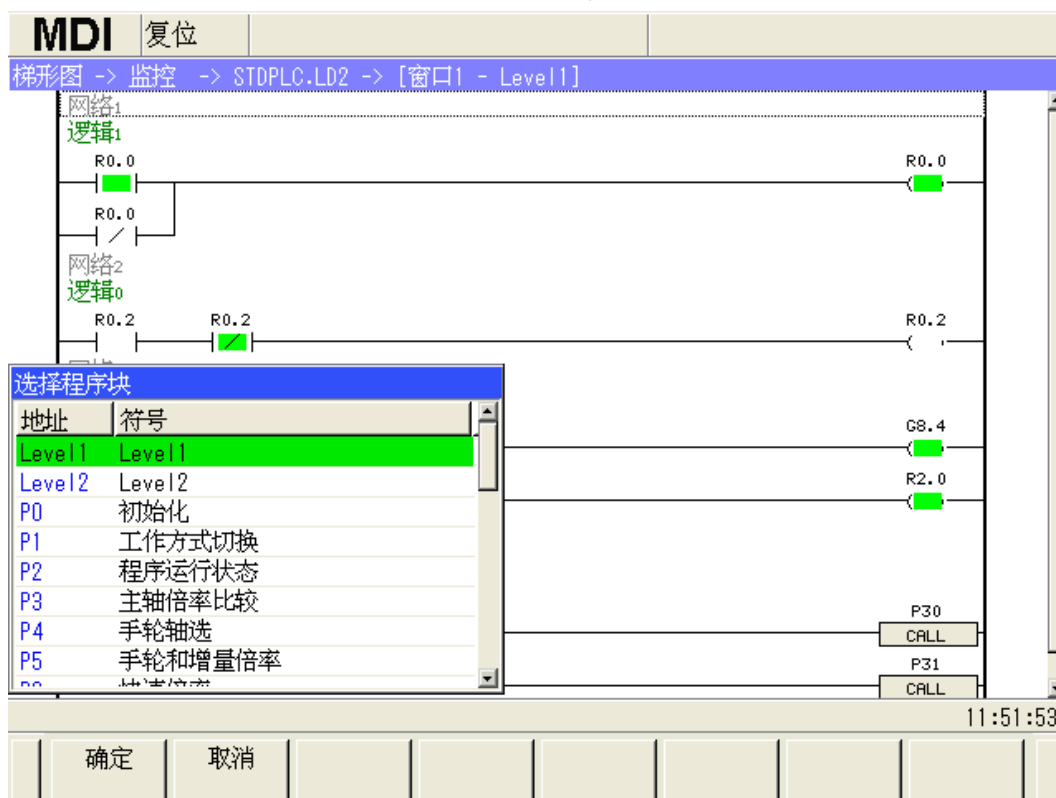
2. Selecting the window block

(1) Select the window to select the block, that is to say, respectively press

窗口1 Level1, **窗口2 Level2**, **窗口3 P0** or **窗口4 P1** to select the window.

请选择

(2) Press **请选择** to select the window program, then the interface is shown as below:



(3) Press **确定**, **取消**, **↑** or **↓** to select the ladder diagram of the block corresponding to the window.

(4) Press **确定** to confirm and return to the previous menu, press **取消** to cancel the operation and return to the previous menu.

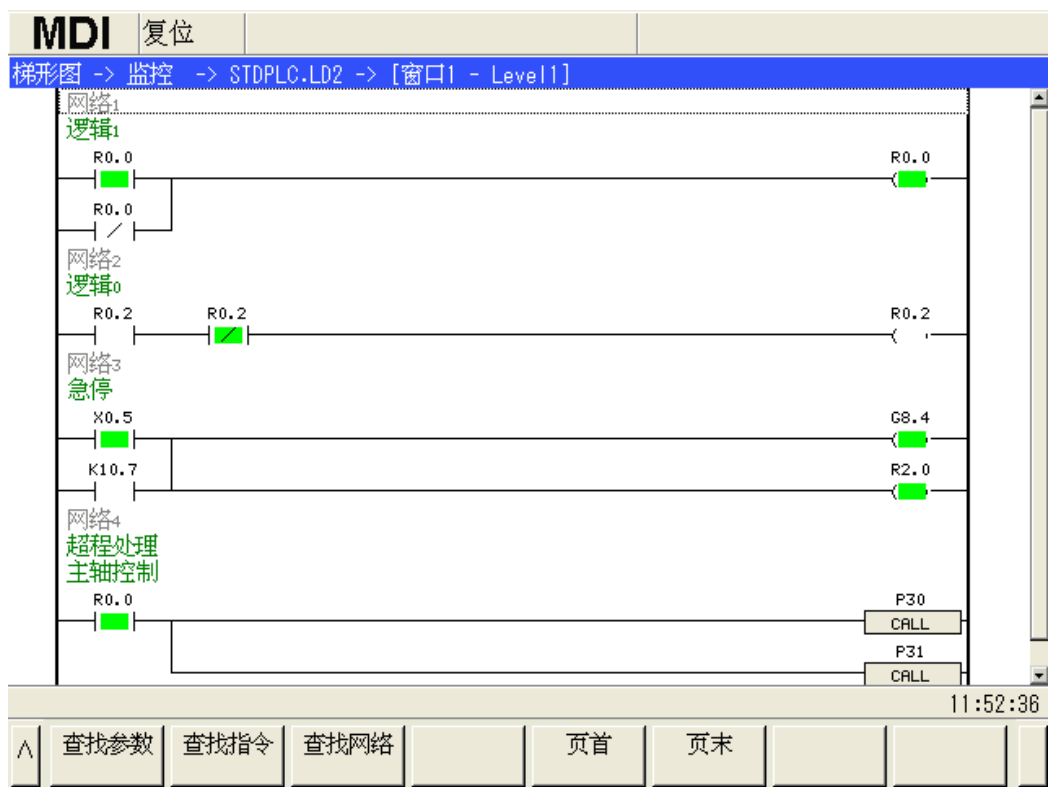
3. Searching for the parameter, the command and the internet

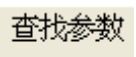
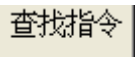
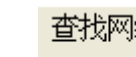
(1) Select the block window to search the command, the parameter and the internet, etc, that is


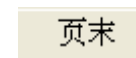
to say, respectively press **窗口1 Level1**, **窗口2 Level2**, **窗口3 P0** and **窗口4 P1** to select the window, and the corresponding block ladder diagram program can display on the window, and search for the command, the parameter and the internet, etc on the window.


查找

(2) Press **查找** to access the searching interface, which is shown as below:




(3) Respectively press ,  and , search for the corresponding parameter, the command and the internet in the corresponding block of the window and the cursor positions in the corresponding place.


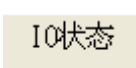
(4) Press  or , for checking, the cursor positions in the initial or the last line of the block corresponding to the window.

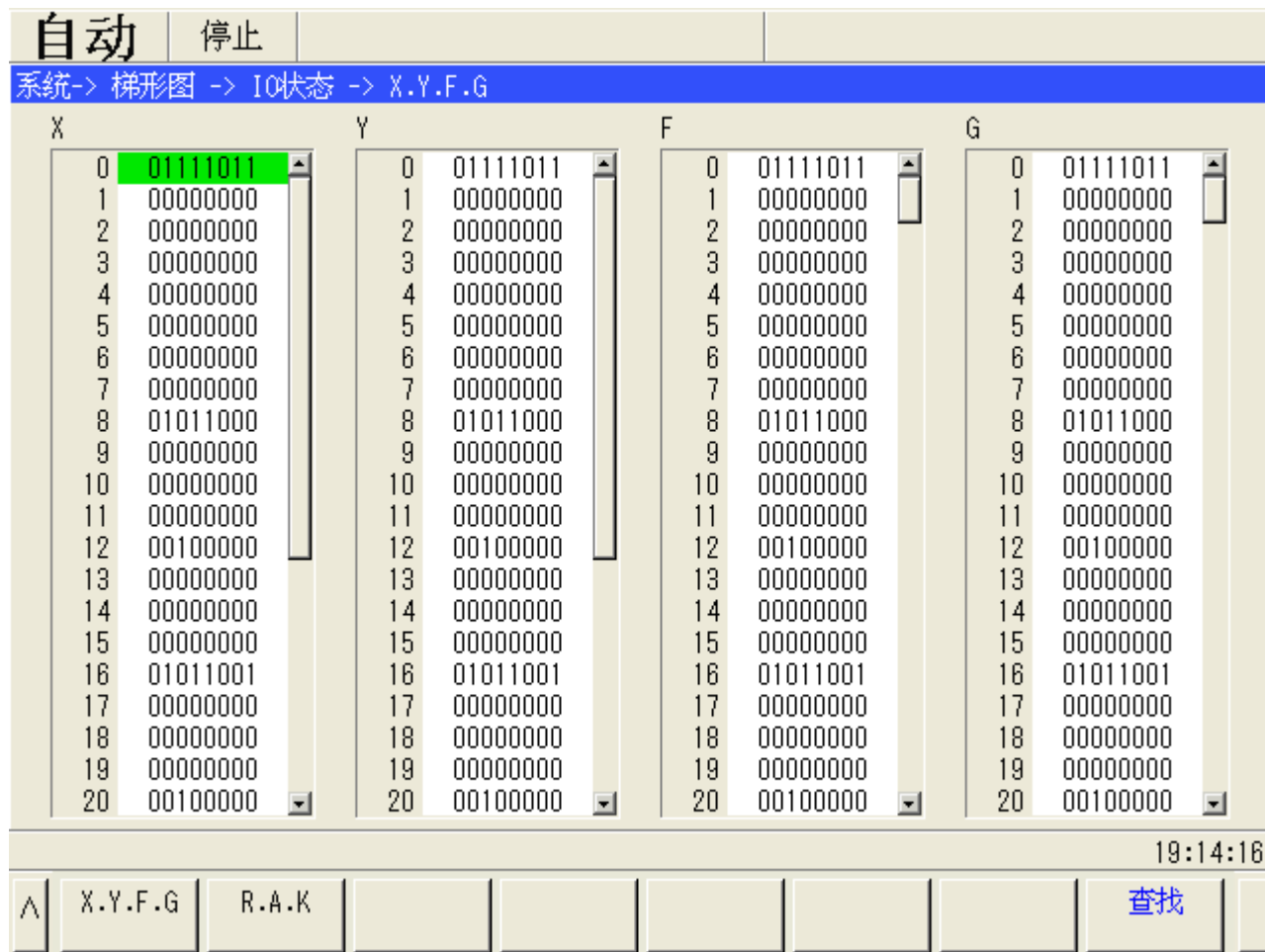
(5) Press , the screen displays it returns to the previous menu and its position is shown in figure 2,.

4. Return

In figure 2, press , the screen displays it returns to the previous menu, and its position is shown in figure 1.

3.6.2 Checking I/O state

On the ladder diagram interface, press  and then press  to access I/O state interface, and the interface is shown as below:



On the interface, press **X.Y.F.G**, the state information of each parameter X, Y, F and G

displays in window. Then, press or to switch among parameters X, Y, F and G; press

or to select and check among parameters X, Y, F and G.

Press **R.A.K** to check the state information of parameters R, A and K. Then, press

or to switch among parameters R, A and K; press or to select and check in parameters R, A and K.

Press **查找**, the cursor positions in the parameter to be searched. It searches in the entire interface, so it can be found only input the correct parameter name and number; it can't find if only the parameter number is input.

Press and the screen displays it returns to the previous menu and its position is shown in figure 1.

3.6.3 Checking PLC data and setting

PLC数据

On the ladder diagram interface, press **PLC数据** to access PLC data state interface including

the setting parameters K, D, DT and DC. The interface is shown as below:

	7	6	5	4	3	2	1	0
K0000	0	0	0	0	0	0	0	0
K0001	0	0	0	0	0	0	0	0
K0002	0	0	0	0	0	0	0	0
K0003	0	0	0	0	0	0	0	0
K0004	0	0	0	0	0	0	0	0
K0005	0	0	0	0	0	0	0	0
K0006	0	0	0	0	0	0	0	0
K0007	0	0	0	0	0	0	0	0
K0008	0	0	0	0	0	0	0	0
K0009	0	0	0	0	0	0	0	0
K0010	0	0	0	0	0	0	0	0
K0011	0	0	0	0	0	0	0	0
K0012	0	0	0	0	0	0	0	0
K0013	0	0	0	0	0	0	0	0
K0014	0	0	0	0	0	0	0	0

BIT7
K0000 工作方式记忆

19:15:25

^ K设置 D设置 DT设置 DC设置 查找地址





1. Setting parameter K

(1) On PLC data interface, press **K设置** to access setting parameter K interface.

(2) Press , , , ,  or  to select the parameter state bit

to be rewritten; Or press **查找地址** to input variable K to be selected; when press **确定** and the cursor positions in the parameter. At the bottom of the screen, it displays the meaning of the state bit.

(3) Repeatedly press **INPUT**, and the state bit can be switched between 0 and 1, the state of selected parameter K bit can be rewritten.

(4) Press , ,  or  to complete the rewriting.

Press **查找地址** to input K parameter address to be searched, and the cursor positions K parameter address to be searched.

2. Setting D parameter

(1) On PLC data interface, press **D设置** to access setting parameter D interface, which is





shown as below:

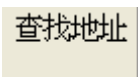
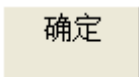
MDI 复位			
梯形图 -> PLC数据 -> D设置			
	数值	最小值	最大值
D0000	0	1	16
D0001	0		
D0002	0		
D0003	0		
D0004	0		
D0005	0		
D0006	0		
D0007	0		
D0008	0		
D0009	0		
D0010	0		
D0011	0		
D0012	0		
D0013	0		
D0014	0		


D0000 刀位数

11:54:28

K设置
D设置
DT设置
DC设置

(2) Press , ,  or  to select parameter D to be rewritten; Or press

 to input parameter D to be selected; when  is pressed, the cursor positions in the parameter. It displays the meaning of the parameter at the bottom of the screen:

(3) Press  and the selected parameter D can be rewritten, which is shown as numerical value D0000 in the following figure.

MDI

复位

梯形图 -> PLC数据 -> D设置

	数值	最小值	最大值
D0000	1	1	16
D0001	0		
D0002	0		
D0003	0		
D0004	0		
D0005	0		
D0006	0		
D0007	0		
D0008	0		
D0009	0		
D0010	0		
D0011	0		
D0012	0		
D0013	0		
D0014	0		

D0000 刀位数

11:58:02

K设置

D设置

DT设置

DC设置

>

(4) Input the rewritten numerical value, and press

INPUT

 again to complete the rewriting.

3. Setting DT parameter

(1) On PLC data interface, press

DT设置

 to access DT parameter setting interface, which is shown as below:

MDI

复位

梯形图 -> PLC数据 -> DT设置

	数值	最小值	最大值
DT0000	0	0	60000
DT0001	0	0	60000
DT0002	0	0	60000
DT0003	0	0	5000
DT0004	0	0	10000
DT0005	0	0	5000
DT0006	0	0	5000
DT0007	0	0	2000
DT0008	0	0	5000
DT0009	0	0	4000
DT0010	0	50	2000
DT0011	0	0	60000
DT0012	0		
DT0013	0		
DT0014	0		

DT0000 主轴换档时间1 (0-60000ms)

11:58:29

K设置

D设置

DT设置


DC设置

>

The setting method of DT parameter is same as that of parameter D.

4. Setting DC parameter

DC设置


On PLC data interface, press  to access DC parameter setting interface, which is shown as below:

MDI	复位		
梯形图 -> PLC数据 -> DC设置			
	数值	最小值	最大值
DC0000	0		
DC0001	0		
DC0002	0		
DC0003	0		
DC0004	0		
DC0005	0		
DC0006	0		
DC0007	0		
DC0008	0		
DC0009	0		
DC0010	0		
DC0011	0		
DC0012	0		
DC0013	0		
DC0014	0		
DC0000			
11:58:38			
K设置	D设置	DT设置	DC设置

The setting method of parameter DC is same as that of parameter D.


5. Return



On PLC data interface, press  to return to the previous menu, namely, the display interface returns to the interface shown in figure 1.

3.6.4 Stopping the program running

版本信息

On the ladder diagram interface, press  to access the version information interface;




停止

when there is running PLC program, press  to stop the running of the current program;


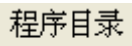
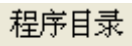
停止

运行

运行

then,  changes into . Then, press  again to run the currently selected program.

3.6.5 Display, creating, editing and managing PLC program directory

On the ladder diagram interface, when the operation authority is above level (2); press  and  then press , access the ladder diagram program management interface and the program can be created, deleted and edited, etc on the interface. The interface is shown as figure 3:

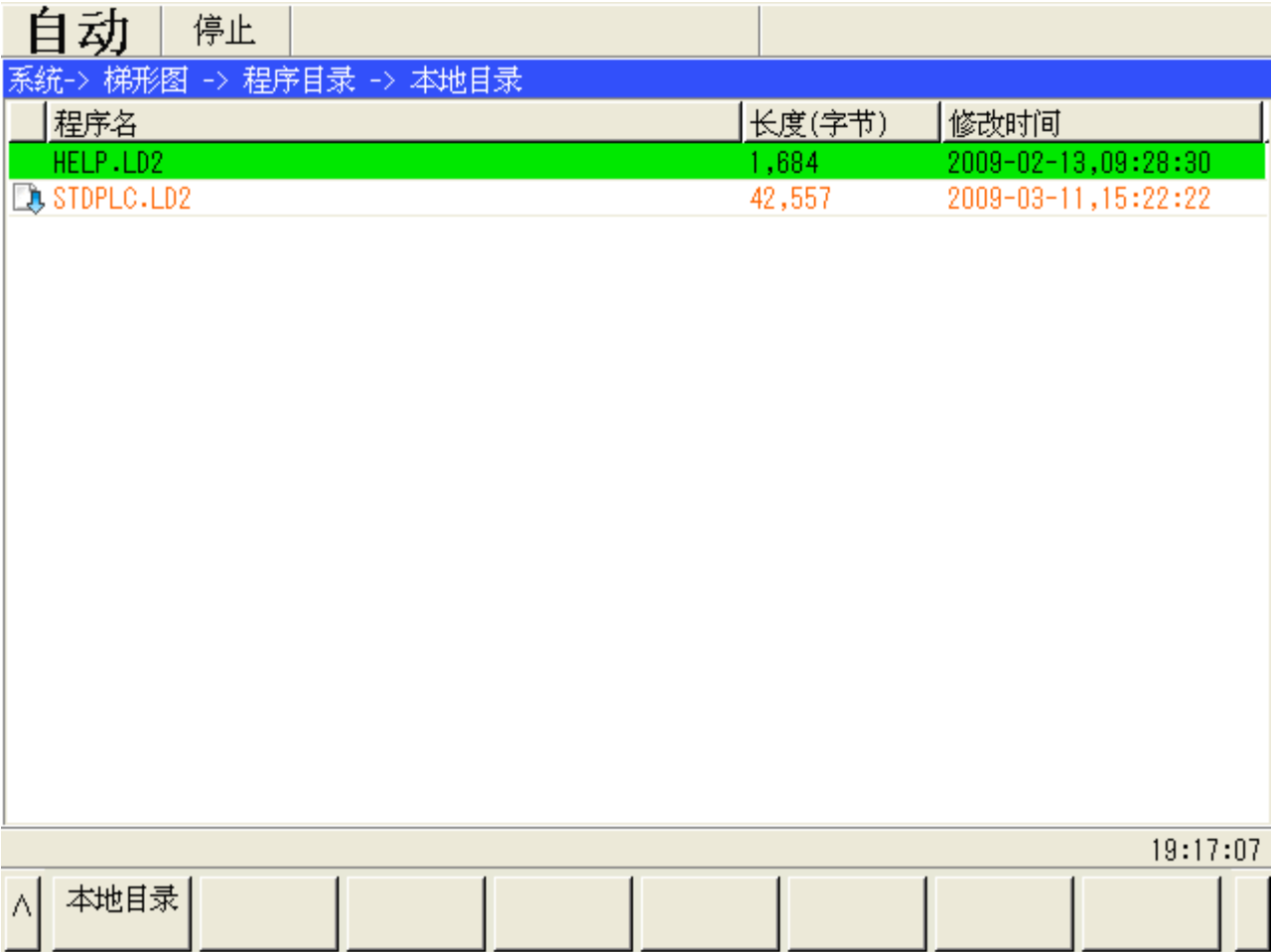
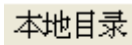
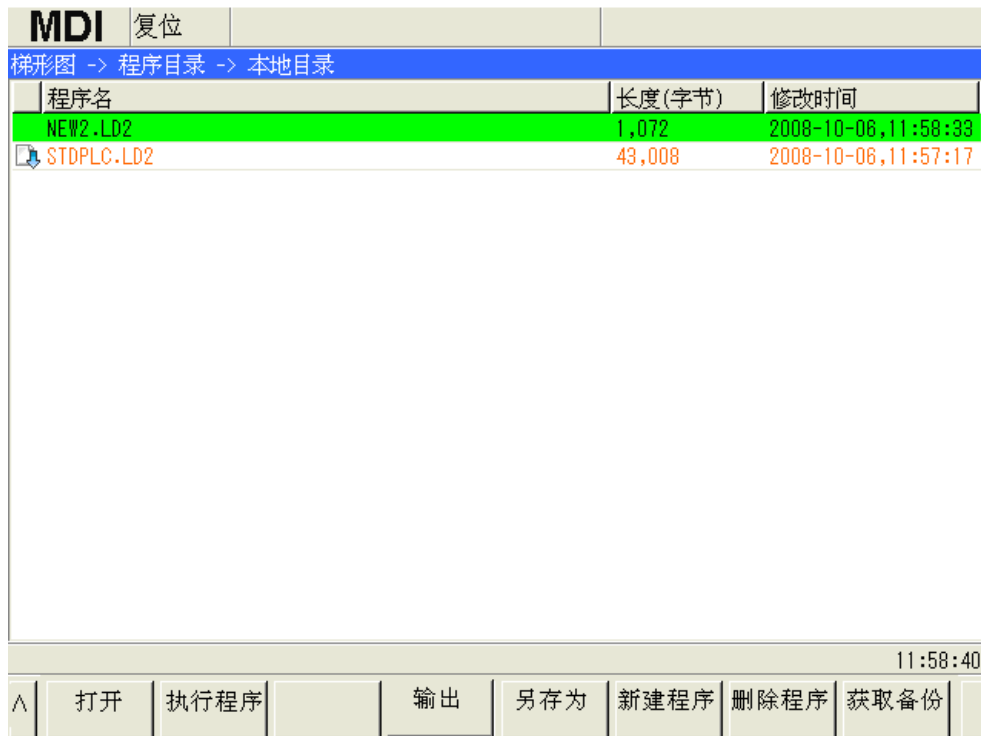


Fig. 3



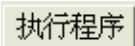

Press  and display all the ladder diagram directory names, the size of each program and the latest rewriting time in the current system.





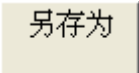
When there is flash disc in system USB port, there adds one “flash disc directory” soft key on the interface, which is shown as below. Press “flash disc directory” to display the ladder diagram program directory in the flash disc and the operation of flash disc directory is same as that of the local directory.

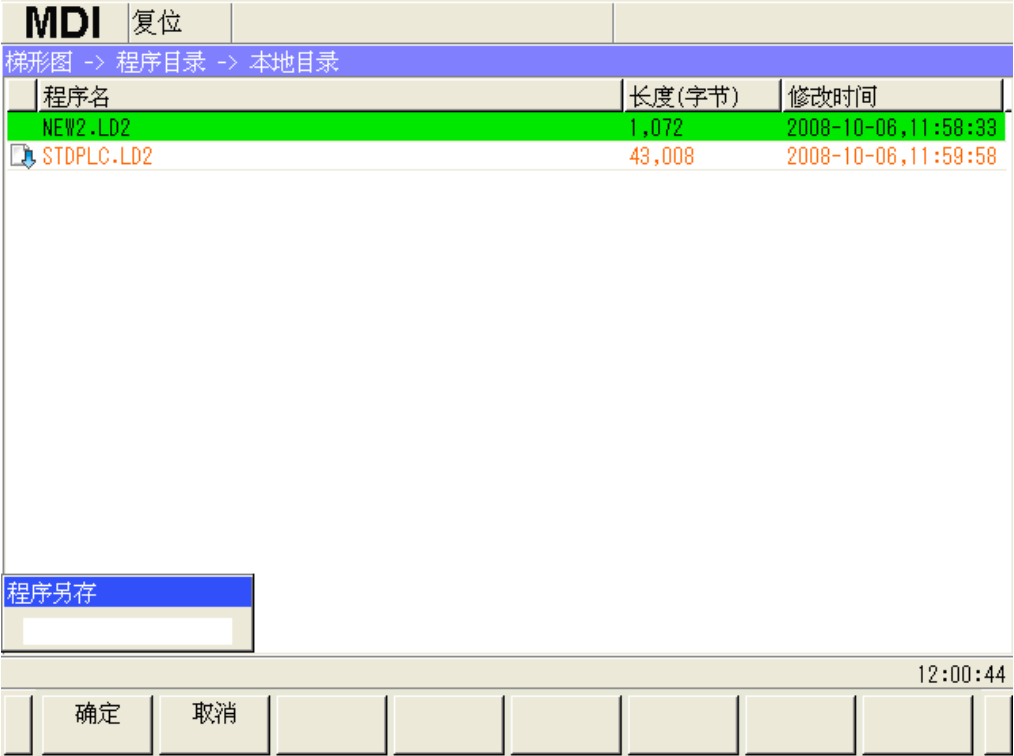


3.6.5.1 The program running

In figure 3, press  or  to select the program, press  to run the selected program and rewrite the currently running ladder diagram program. There is  mark before the running ladder diagram program column, such as STDPLC.LD2 in figure 3.

3.6.5.2 Program saved in the other name

In figure 3, press  or  to select the program; press  and the interface is shown as below. Input the new program name, and the selected program can be saved in the new program name.



3.6.5.3 Interchanging the program between the system and the flash disc

When there is flash disc in USB port, press  and the interface is shown as below:



输出

Press **输出**, the programs of the flash disc directory can be copied to the local directory, vice versa. The detailed steps are as below:

U盘目录

(1) Press **U盘目录** to access the flash disc file directory;



(2) Press **↑** or **↓** to select the ladder diagram program to be copied, press

输出

输出 to copy the selected program to the local directory.

The operation of copying files in local directory to the flash disc directory is similar as that of copying files in the flash disc to the local directory.

3.6.5.4 Backup the program

本地目录

On the ladder diagram directory interface, press **本地目录**, the extension interface is shown as following figure 4:

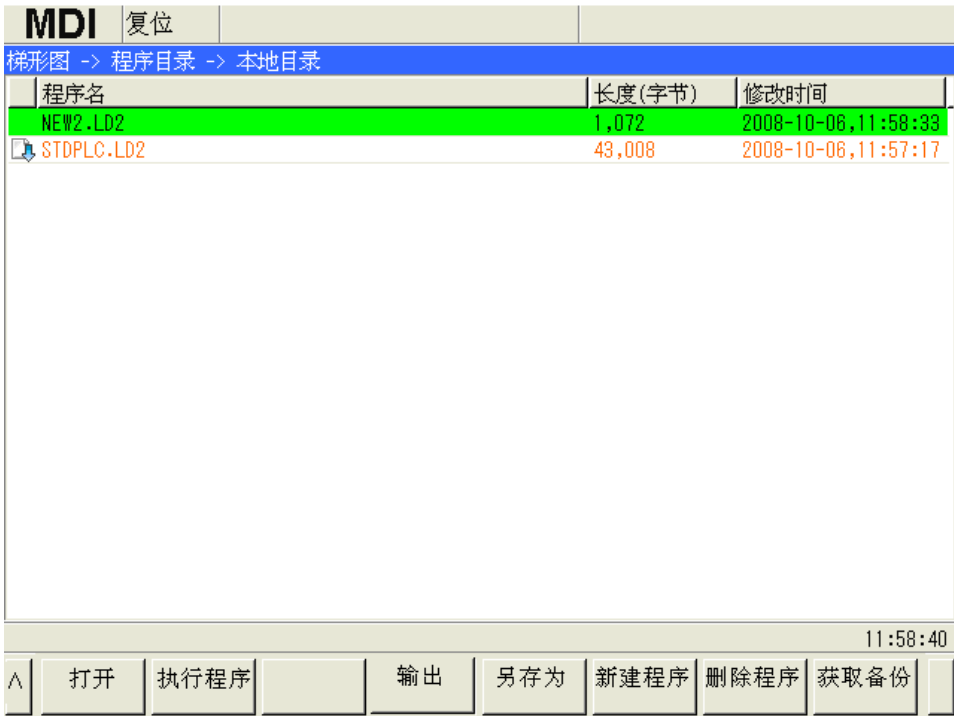


Fig. 4

Press **获取备份** in figure 4 and the interface is shown as below. Input a new program name, and then PLC program, default by the system, can be backed up in a new program name.



3.6.5.5 Creating and deleting the program




1) Creating the program

On the ladder diagram interface, press **本地目录** to access the ladder diagram program

请输入新程序名

确定

2) Deleting the program

In figure 4, press  or  to select the program name to be deleted, press  to delete the program.

删除




确定要删除文件“TD3.LD2”吗?

and there is 确定要删除文件 TD3.LD2 吗?, it reminds whether delete the program, press

确定

to delete the selected program.

On the program directory interface of the ladder diagram interfaces, which is shown as figure 4,

press  or  to select the program to be edited, press  to access the editing interface, then, the ladder diagram, code list, the information list and the initialization data list are respectively edited, which is shown as figure 5:

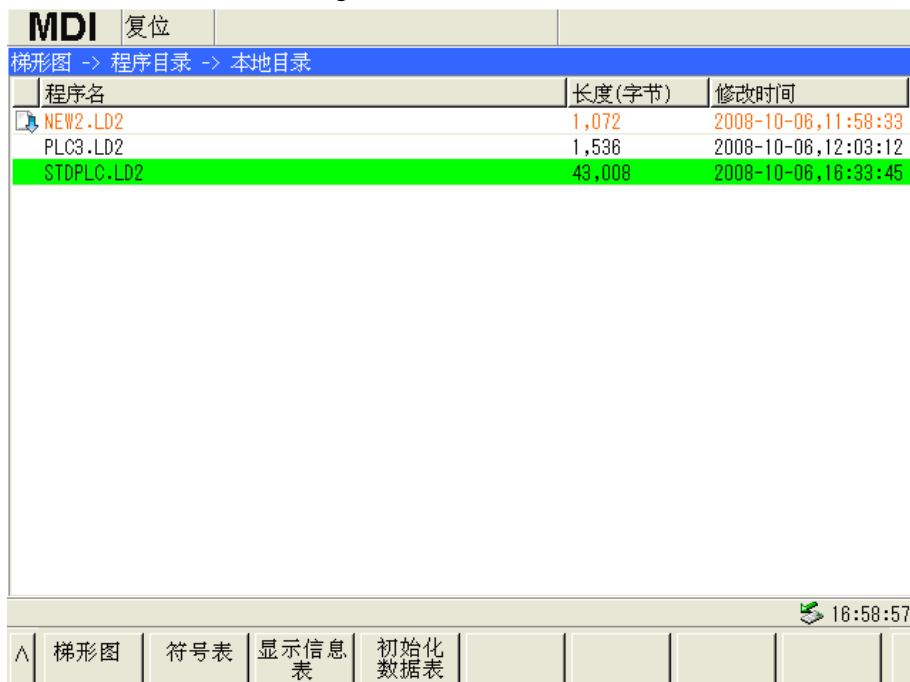




Fig. 5

打开

梯形图

On the editing interface shown in figure 5, firstly press , and press  to access interfaces of the ladder diagram and the editing window display, which is shown as figure 6:

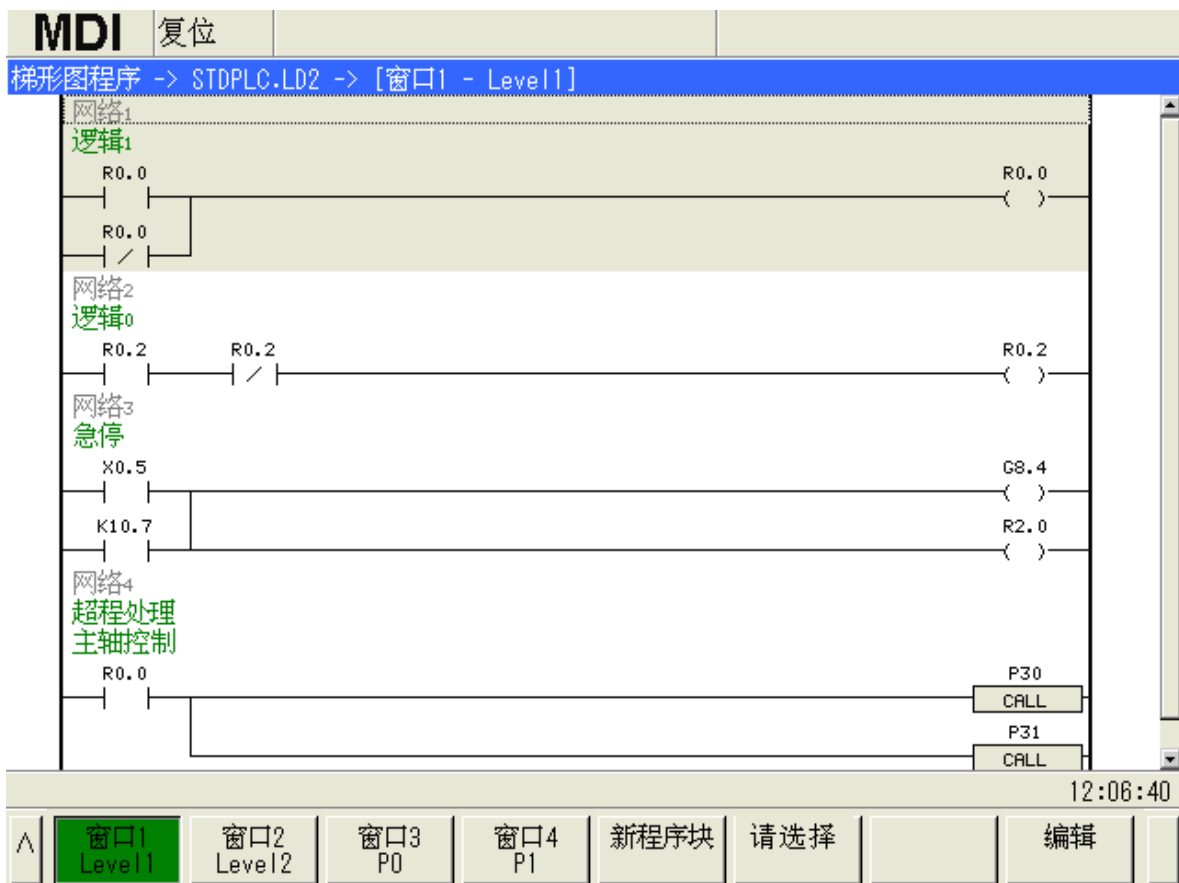


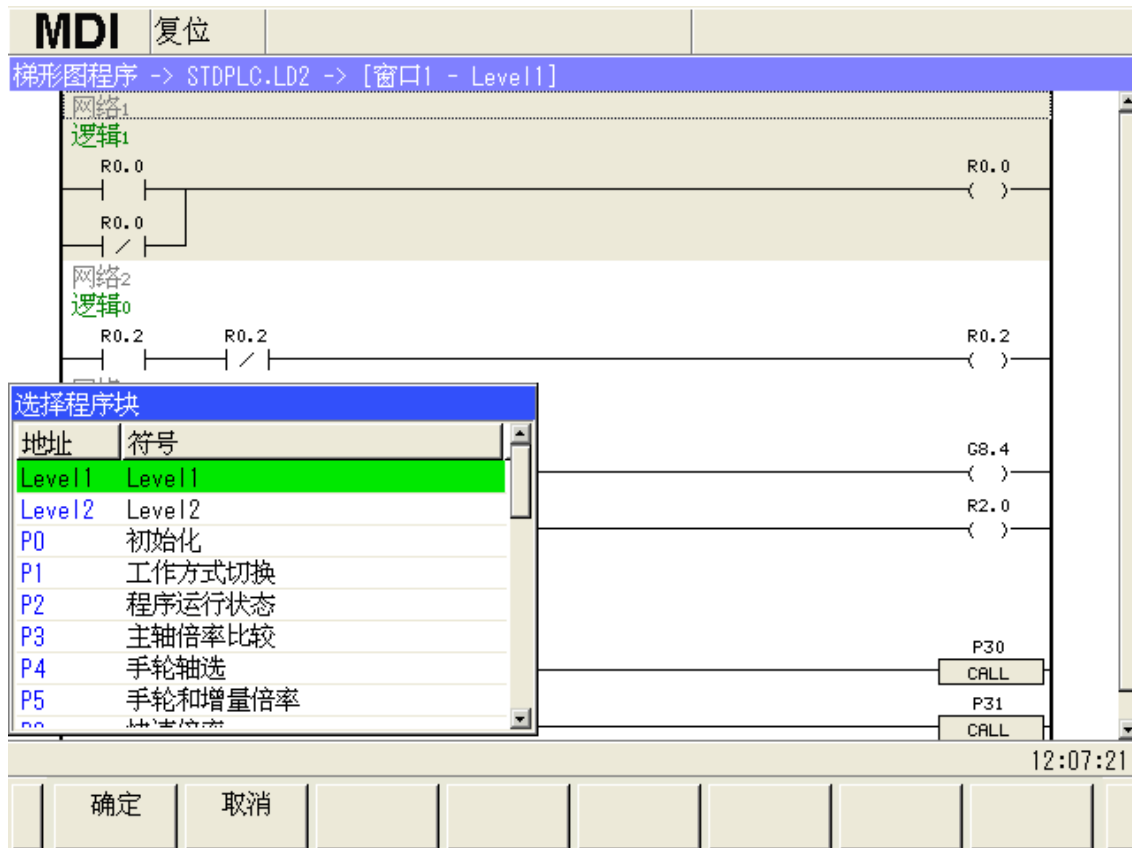
Fig. 6

In the figure, the position, which the cursor is, displays as the dotted box, and the internet area background color, which the cursor is, is a little deeper than the window one.

Respectively press **窗口1 Level1**, **窗口2 Level2**, **窗口3 P0** and **窗口4 P1**, it displays the corresponding block on the screen. On the top of the interface and below the display boxes of the running state and the mode, it displays the ladder diagram block name in the current window.

(1) Selecting the block for the window


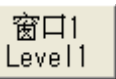
Firstly press **窗口1 Level1**, **窗口2 Level2**, **窗口3 P0** or **窗口4 P1** to select the window to rewrite the block, and press **请选择** to select the relative block. The interface is shown as below:




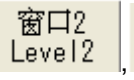
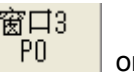
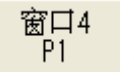
Press , ,  or  to select the block relative to the window, press

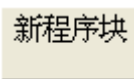
 确定

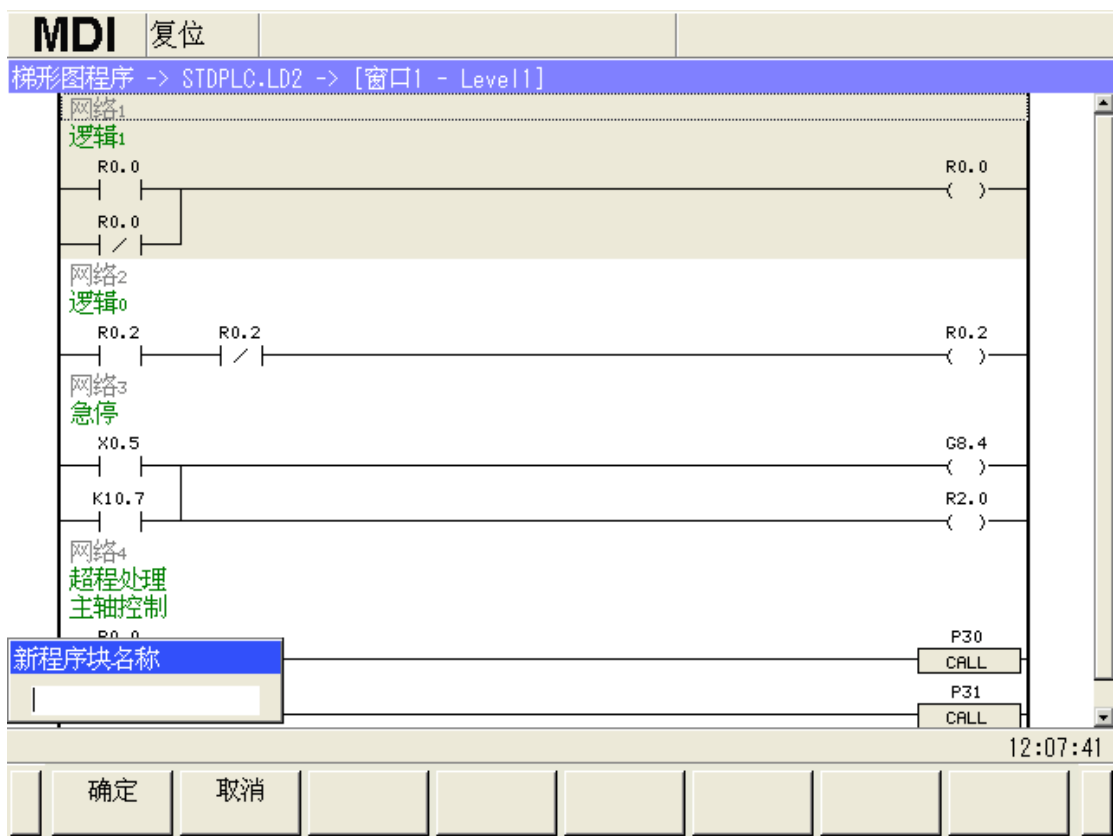
to select and return. Then, the corresponding block address displays in the window. For

example,  means window 1 corresponds to Level 1 block, namely, when press , the content of Level 1 block displays in the window.

(2) Creating the block

Firstly, press , ,  or  to select the window to create

the block, press  新程序块 and it is shown as below:



Input the new block name and press **确定** to create a blank block in the window. And then, press **编辑** to edit the new block.

(3) Editing the program

Firstly, press **窗口1 Level1**, **窗口2 Level2**, **窗口3 P0** or **窗口4 P1** to select the window block program to be edited or rewritten, press **编辑** to access the editing program interface, which is shown as the following figure 7:

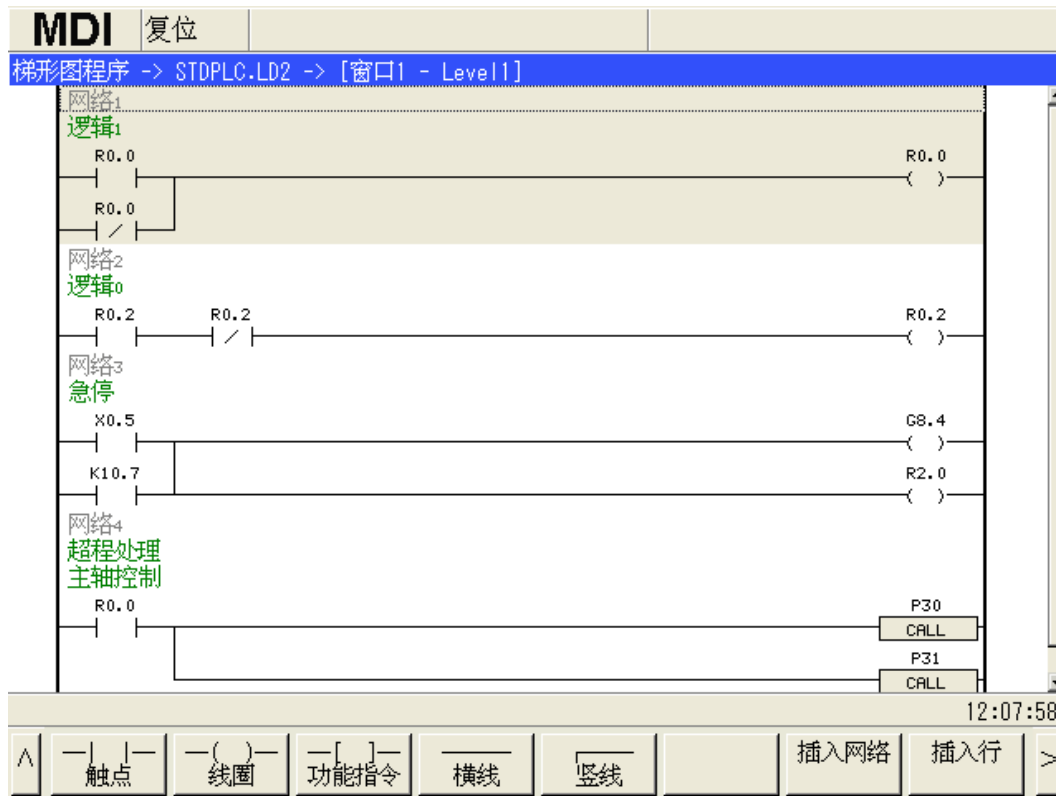


Fig. 7

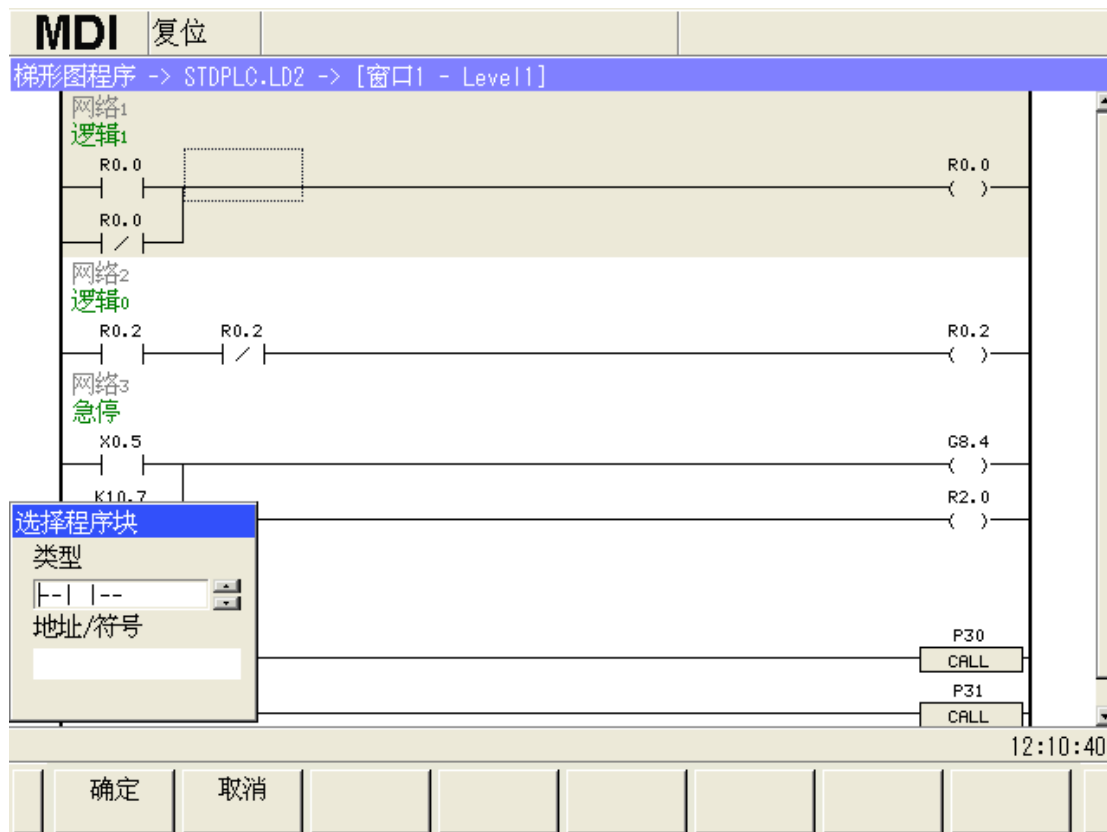
Press , , or and the cursor moves to the line to be rewritten.

Press , one internet is inserted after one internet which the cursor is; press and a new line is inserted after one line which the cursor is.

Press or and the cursor moves toward the address grid to insert the units, and click the soft keys of the contact and the circuit to insert the units.

For example:

① Press , the interface is shown as below:



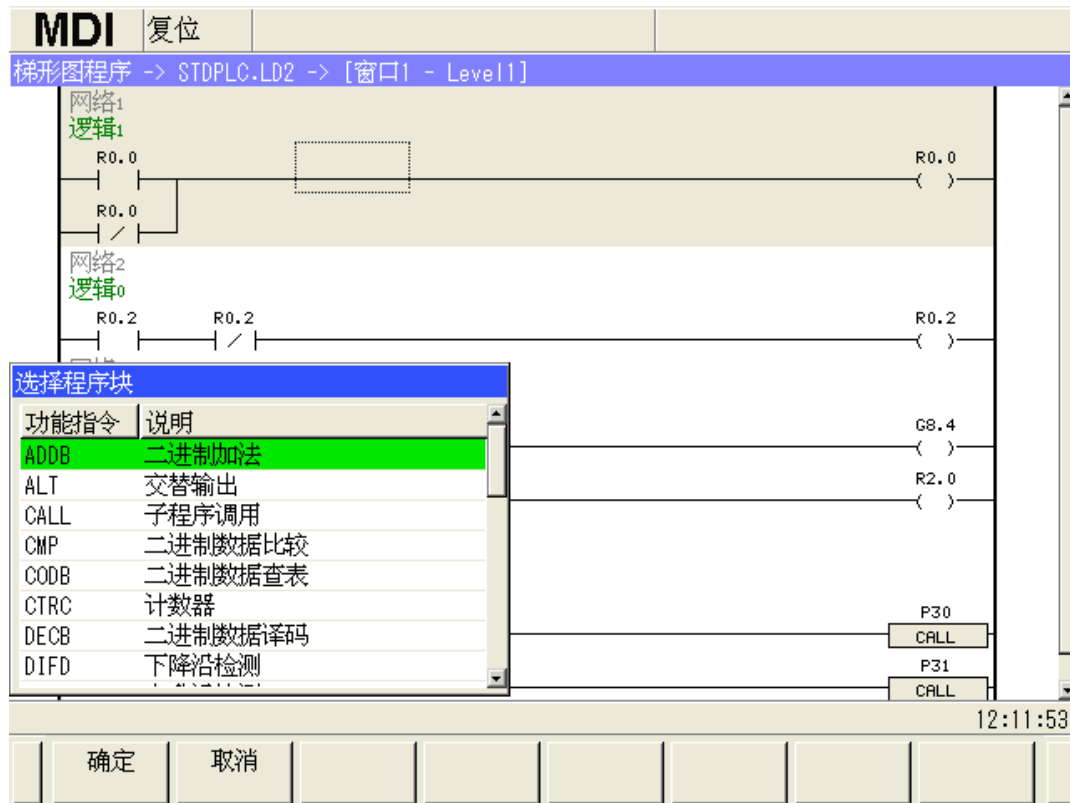
Then, the cursor is in the type selecting box, press , or , to switch the



types in the selecting box and select the normally open contact or the closed. Press and the cursor switches between the editing boxes of “type” and “address/code” , input address/code, press

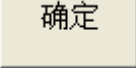
or press “input” to confirm the input is completed.

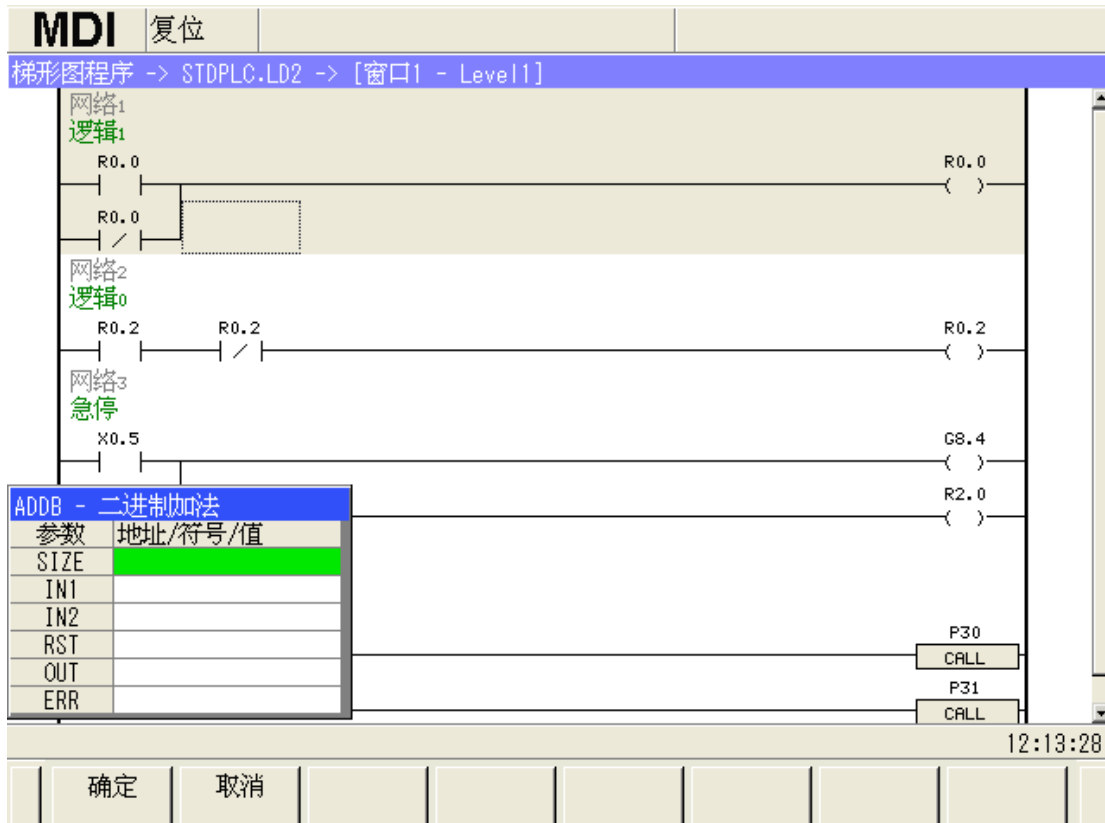
① Press , the operation is same as that of ;








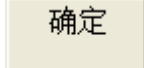
② Press , the interface is shown as below:




Press  or  to select the function commands to be inserted, such as ADDB

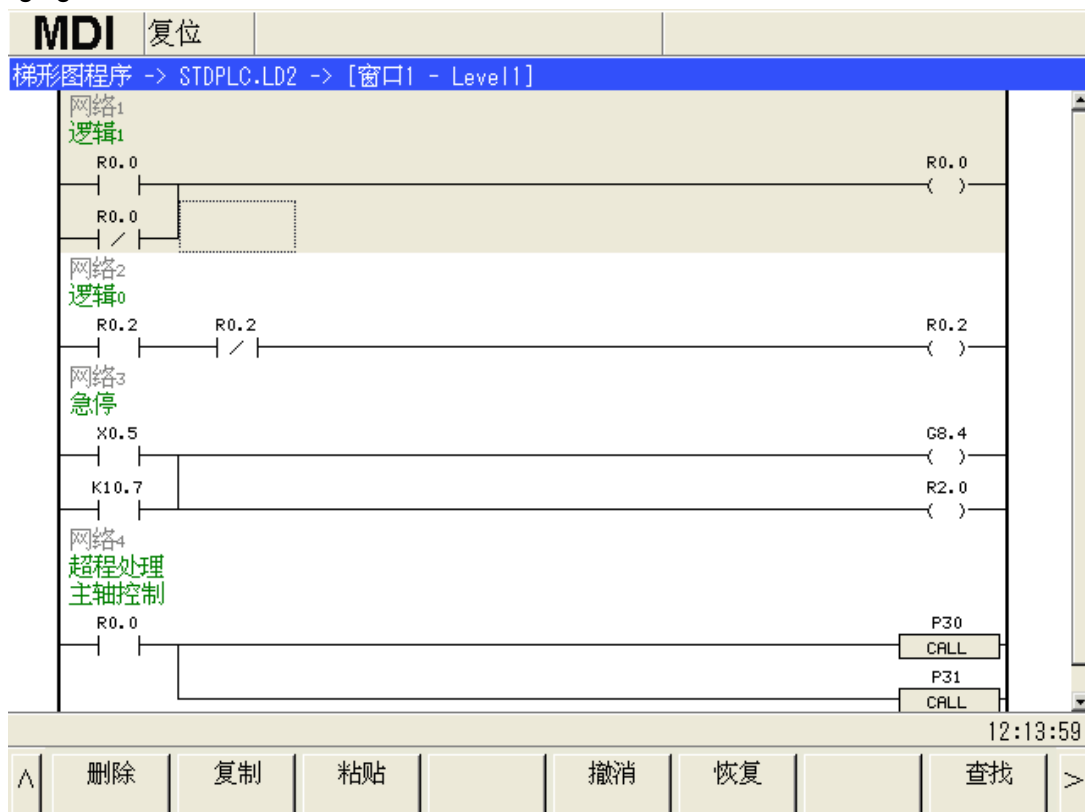
(addition in binary system) shown in the above figure, press  or "input", the interface is shown as below:



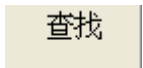
Then, press  or  to switch and select the editing box, input the address or the data, press  to confirm the rewriting; or press , the editing box, which the cursor is, can be input, and input the address or the data, press  again to confirm the rewriting. Press  or  to select the other editing box. After input, press  to confirm the editing is completed.

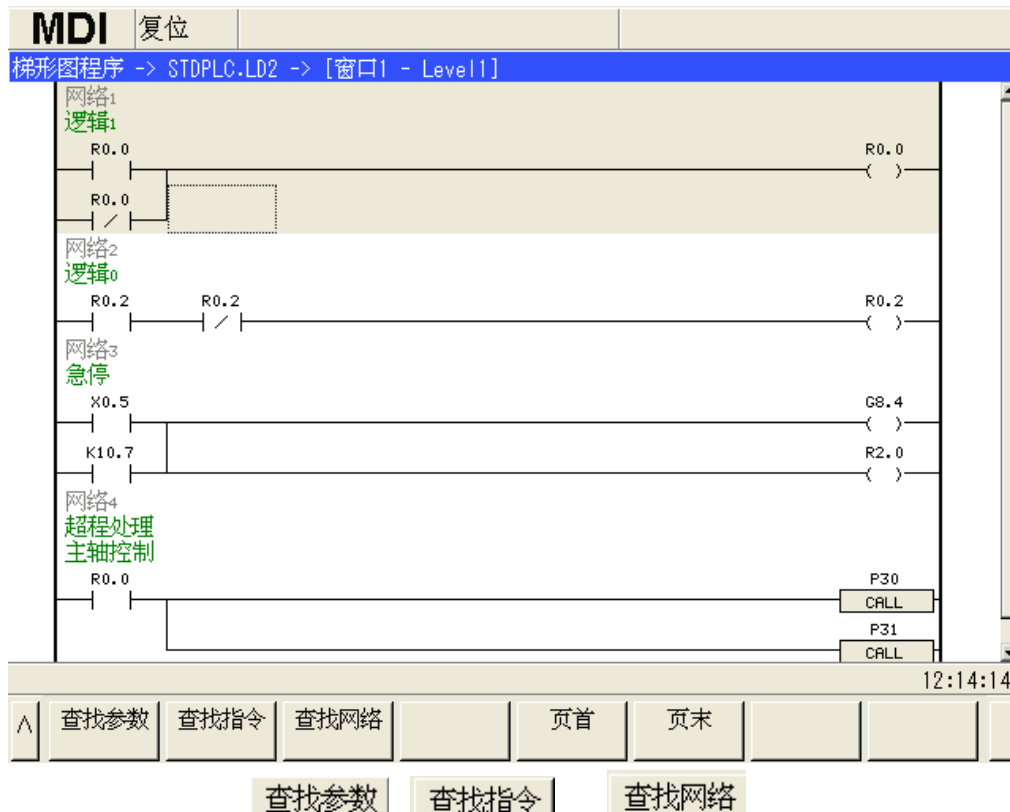
Input and editing the other function commands are stated as above.

On the interface of figure 7, press  to display the extension soft keys, which is shown as the following figure:



Then, the selected unit, line or network can be deleted, copied and pasted, etc, the previous operation can be canceled or the operation which is canceled can be done, again.

Press  to switch into the searching interface, which is shown as below:



Then, respectively press **查找参数**, **查找指令** or **查找网络** and the cursor positions in the parameter, the command or the network position to be searched; press **页首** or **页末** and the cursor can position at the starting or at the end of the block.

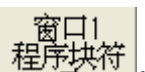
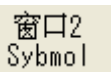
Press **^** to return to the previous menu.

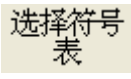
3.6.6.2 Checking and setting the symbol list

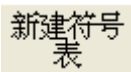
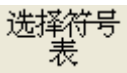
On the editing interface shown in figure 5, press **符号表** to access the symbol list interface, which is shown as following figure 7:

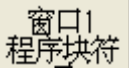


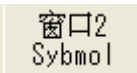
Fig. 7

(1) Respectively, press  or  to select the window, and display the relative symbol list information on the screen. Then, at the top of the window and below the display boxes of the running state and the mode, the window name and the corresponding symbol list name display in the current window.

Press  to select the symbol list corresponding to each window.

Press  to create a symbol list, and display in the current window (Remark: if there is a blank window, the new symbol list will display in the window in priority.). If it requires the original symbol list displays in the window, press  to select the original one.

In the above figure, press  to display the block symbol list, the address list corresponding to the program codes display in the block symbol list.

(2) Press  to display the sybmol list, which is shown as below:

Press      or 

On this interface, press 

On this interface, press  to display the extension soft keys, which is shown as below:

MDI

复位

窗口2(Sybmol)

	符号	地址	注释
1		DT0	主轴换挡时间1 (0-60000ms)
2		DT1	主轴换挡时间2 (0-60000ms)
3		DT2	压力低报警检测时间(0-60000ms)
4		DT3	换刀时移动一刀位的时间上限(ms)
5		DT4	换刀时移动最多刀位的时间上限(ms)
6		DT5	M代码执行持续时间(ms)
7		DT6	S代码执行持续时间(ms)
8		DT7	刀架从正转停止到反转输出的延迟时间(ms)
9		DT8	未收到刀架锁紧*TCP信号的报警时间(ms)
10		DT9	刀架反转锁紧时间(0-4000ms)
11		DT10	M05与主轴制动输出的延迟时间(ms)
12		DT11	主轴制动输出时间(0-60000ms)
13		DT12	主轴点动时间(0-60000ms)
14		DT13	润滑开启时间(0-60000ms)(0:润滑不限时)
15		DT16	自动润滑间隔时间
16		DT17	自动润滑输出时间
17		DT18	卡盘夹/松脉冲输出宽度

13:46:44

窗口1
程序块符

窗口2
Sybmol

窗口3
K

窗口4
符号表D







新建符号
表

选择符号
表

>

Then, press **查找** to input DT address to be searched, and the cursor positions in the **插入一行** address. Press **插入一行** to insert one blank line below the line which the cursor is, press **删除一行** to delete the line which the cursor is.

(3) Rewriting and editing the symbol list

Select the window symbol list to be rewritten, press  ,  ,  ,  ,  or  to select the symbol, address or explanation box to be rewritten, press **INPUT** and the selected box can be input with the symbol, the address or the explanation, press **INPUT** again to complete rewriting.

Remark: The symbol list of block can't be rewritten.

Press **窗口3** or **窗口4** to switch into the relative interface, and select, create or edit the data list.







3.6.6.3 Checking and rewriting the information list

On the editing interface shown in figure 5, press **显示信息表** to access the information list interface, which is shown as following figure 8:

MDI		复位	
显示信息表			
	信息号	显示内容	
A0000.0	1000	换刀时间过长	
A0000.1	1001	换刀完成时,刀架未到位	
A0000.2	1002	换刀未完成	
A0000.3	1003	尾座功能无效,不能执行M10/M11指令	
A0000.4	1004	主轴旋转时,不得退出尾座	
A0000.5	1005	主轴启动使能关闭,不能启动主轴	
A0000.6	1006	防护门未关	
A0000.7	1007	卡盘压力低	
A0001.0	1008	主轴旋转时,不得松开卡盘	
A0001.1	1009	卡盘夹紧未到位,禁止启动主轴	
A0001.2	1010	主轴旋转时,未检测到卡盘夹紧信号	
A0001.3	1011	卡盘松开,禁止启动主轴	
A0001.4	1012	卡盘功能无效,不能执行M12/M13指令	
A0001.5	1013	未检测到刀架锁紧信号	
A0001.6	1014	未定义功能的M代码	
A0001.7	1015	非模拟主轴,无法执行主轴点动功能	
A0002.0	1016	M03,M04代码指定错误	
13:47:42			
^	查找地址	查找信息	

Fig. 8


In the information list, it displays the information address A, its corresponding information number

and its content. Press , , , ,  or  to select and check each address, information number and its corresponding information.

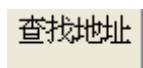
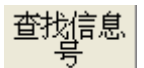
Rewriting the information number and the content:

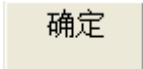
Press , , , ,  or  to select the information number or the

content to be rewritten; press  and the selected information number or the content can be

rewritten, and input the information number or the content to be rewritten, press  again to complete the rewriting.

Searching for the address and the information number:

Press  or  to input the address or the information number to be searched,

and press  to search and the cursor positions in the searched address or the information number.

3.6.6.4 Checking and setting the initialization data list

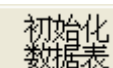
On the editing interface shown in figure 5, press  to access the initialization data list interface, which is shown as the following figure 9:



Fig. 9

(1) Setting parameter K

Press **窗口1 K设置** to select window 1 and it displays the detailed information of parameter K. At the bottom of the window, it displays the detailed meaning of the bit corresponding to parameter K which the cursor is.

Press **菜单**, **列表**, **上**, **下**, **左** or **右** to select some bit corresponding to parameter K to be set or rewritten.

Repeatedly press **INPUT** and the selected bit can be switched between 0 and 1.









(2) Initializing the data

Press **窗口2 InitData** to access the interface of InitData list, which is shown as below:


MDI 复位				
窗口2(InitData)				
	地址	数值	最小值	最大值
1	DT0	1000	0	60000
2	DT1	1000	0	60000
3	DT2	3000	0	60000
4	DT3	1000	0	5000
5	DT4	5000	0	10000
6	DT5	500	0	5000
7	DT6	500	0	5000
8	DT7	50	0	2000
9	DT8	1500	0	5000
10	DT9	1500	0	4000
11	DT11	50	0	60000
12	DT16	3000	0	3600000
13	DT17	1000	0	60000
14	DT19	1000	200	10000
15	DT21	2000	200	5000
DT0 主轴换挡时间1 (0-60000ms)				
13:49:39				
<div> <div>窗口1 K设置</div> <div>窗口2 InitData</div> <div>窗口3 数据表0</div> <div>窗口4</div> <div>删除数据表</div> <div>新建数据表</div> <div>选择数据表</div> </div>				

In the window 2, it displays the detailed information of DT parameter. At the bottom of the interface, it displays the meaning of DT parameter.

Rewriting and editing the data list:

Press  or  to select the interface, press , ,  or  to select the address value or the numerical value, the maximum value and the minimum value of some address to be rewritten, the selected value changes into the blue-based color; press  and the value can be rewritten, and press the numerical and the backspace keys to rewrite the data, and press  again to confirm the rewriting.

The data list can be deleted and created in the window, and the new data list can be selected or the data list can be selected, again.

On the interface, press  to display the extension soft key, which is shown as below:



Then, press **查找地址** and the cursor can position in any address. Press **插入一行** to insert one blank line below the line which the cursor is, press **删除一行** to delete the line which the cursor is.

(3) Press **窗口3** or **窗口4** to switch into the interface of the data list in window 3 or 4, and select, create, edit or rewrite the data list in window 3 or 4.

3.6.6.5 Version information

On the editing interface shown in figure 5, press **版本信息** to access the version information interface, which is shown as below:

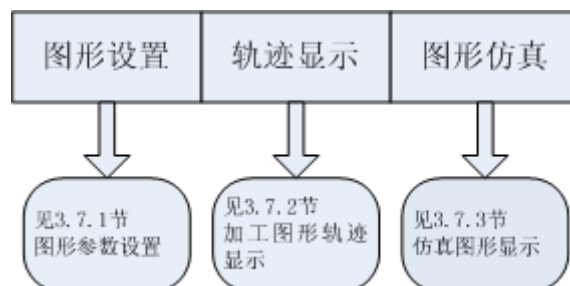
自动		复位	
系统-> 梯形图 -> 版本信息			
梯形图程序	STDPLC.LD2	PLC状态	运行
梯形图设计	广州数控	PLC系统版本	PLC-N1
梯形图版本	TD2-080523		
程序校验码	43ABF526		
程序创建日期	2008-11-27,09:14:31	当前扫描周期(ms)	8
最后修改日期	2009-03-11,15:22:22	最大扫描周期(ms)	8
		最小扫描周期(ms)	8
备注			
标准梯形图			
19:20:27			
^	停止		

On the top of the interface, it displays the system running mode and its state; the version information of the ladder diagram, the program name of currently running ladder diagram and its running state information, PLC executing current cycle, and the maximum/minimum cycle since it has been using.

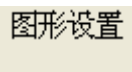
3.7 Graph interface

GRAPH

Press **GRAPH** to access the graph interfaces, and it mainly includes the interfaces of the graph setting, the path display and the simulation graph, etc, and check the content of each interface through pressing the corresponding soft keys. The structure of the software layers is shown as below:



3.7.1 Setting graph parameter




On the graph interface, press  to access the setting graph interface and it is shown as below:

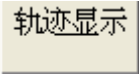
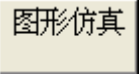


On the interface, the path and the simulation parameters can be set.

Firstly, set the horizontal and vertical axes of the graph, and set the offset of the coordinate axis and the magnification of the graph; if the simulation graph is required, set the simulation horizontal and vertical axes, the length and the diameter of the processing work piece and the magnification of the simulation graph.

In the right column, it displays the current absolute position coordinate and the relative coordinate position value and the tool number used in the currently running program at the same time.

Press  or  to switch between items; in MDI mode, press the numerical and  to confirm the setting is completed. About the details, refer to chapter 8.1.

On the setting graph interface, press  or  to switch into the path and the graph simulation display interfaces.

3.7.2 Processing graph path

轨迹显示

On the graph interface, press **轨迹显示** to access the path interface and it is shown as the following graph 1:

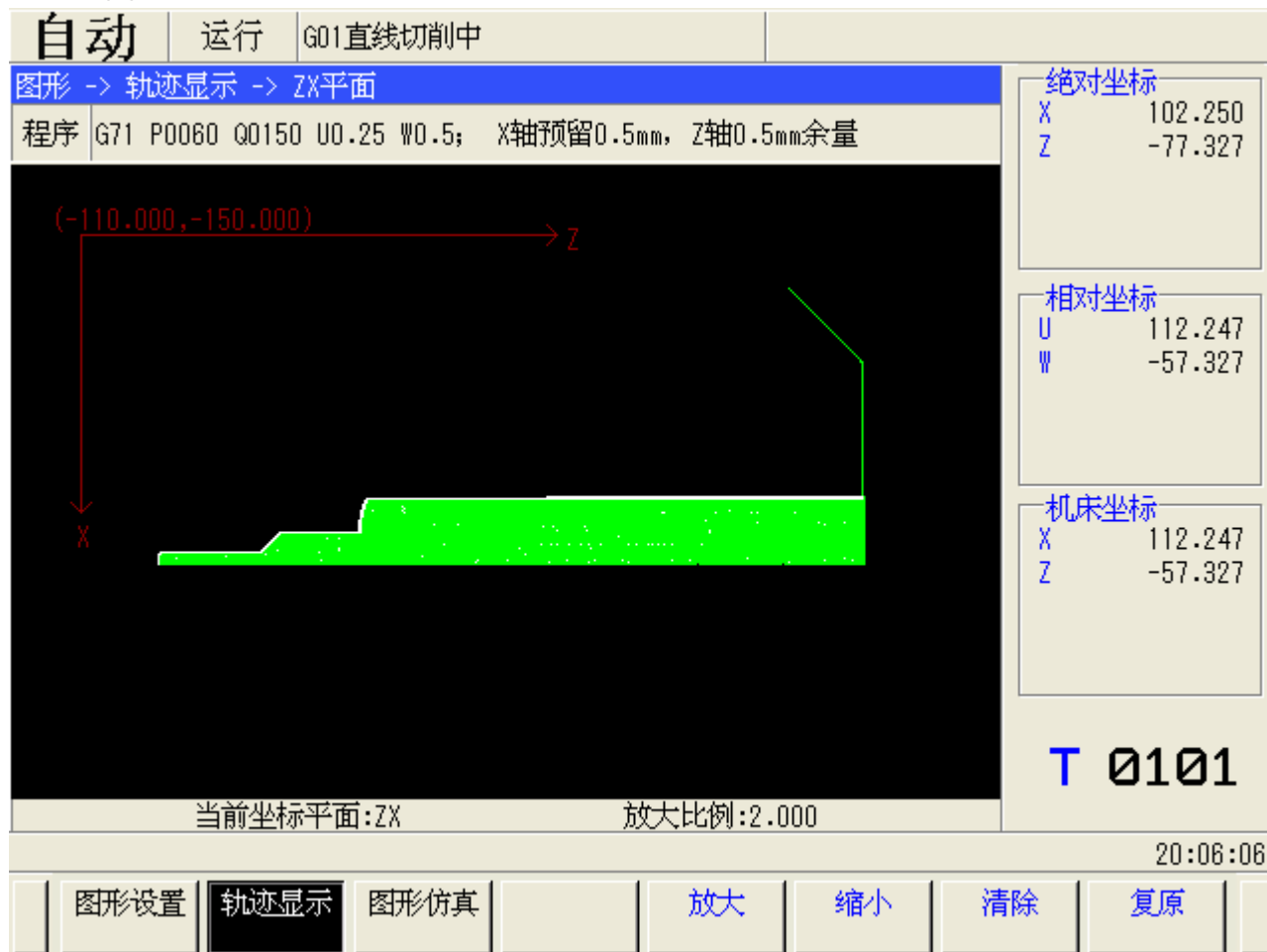


Fig. 1

In the figure, at the bottom of the path screen, it displays the coordinate plane of the present path and the scaling of the path graph.

In the right column, meanwhile, it displays the current absolute position coordinate and the relative coordinate position value and the tool number used in the currently running program.

Then, the graph can be zoomed in and out and the path can be cleared, and

press **↑**, **↓**, **←** or **→** to move the graph up, down, right or left.

Remark: The name of each axis is set by parameter #1020, and the names are set in the different letters.

图形设置

图形仿真

On the path interface, press **图形设置** or **图形仿真** to switch into the setting graph interface or the graph simulation one.

3.7.3 Simulation graph

图形仿真

On the graph interface, press to access the simulation graph interface and it is shown as the following figure 2:

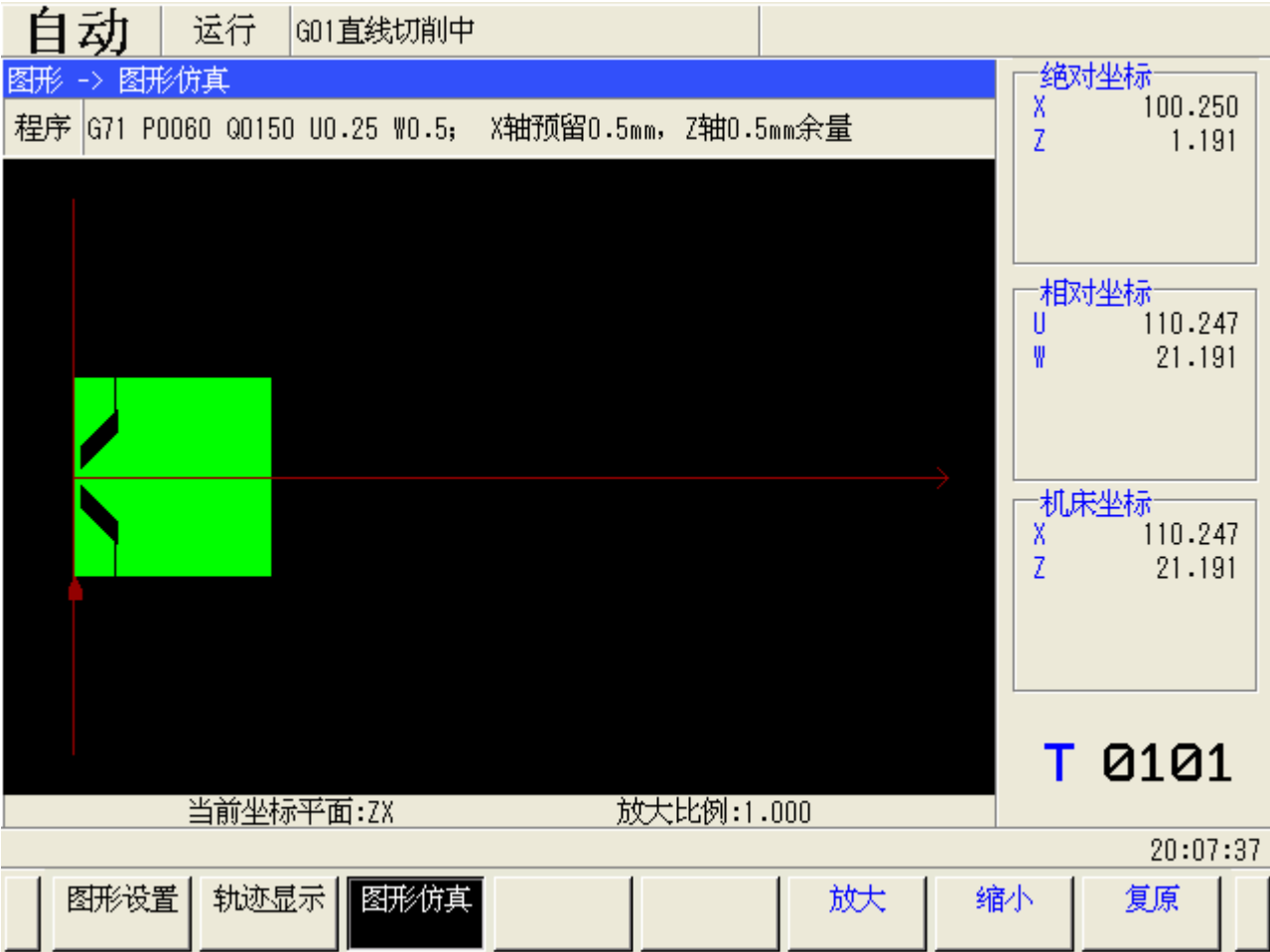






Fig. 2

In the figure, at the bottom of the simulation graph screen, it displays the coordinate plane of the present simulation graph and the scaling of the simulation graph.

On the simulation graph interface, only the graph simulation information of XZ plane is displayed. Then, the graph can be zoomed in and out and the path can be cleared, and press , ,  or  to move the graph up, down, right or left.

Remark: The name of each axis is set by parameter #1020, and the name can be set in different letters.

图形设置 轨迹显示

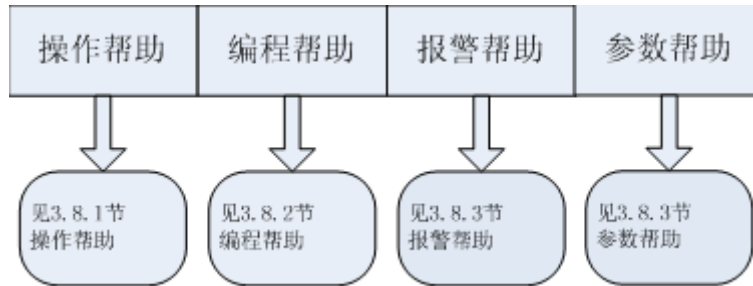
On the graph simulation interface, press to switch into the setting graph interface and the path one.

3.8 Help interfaces

HELP

Press to access help interface and it mainly includes the help of operation, programming, alarm and parameter interfaces, and check the content on the interfaces through pressing the

corresponding soft keys:



Each interface is separated as two parts, the left column and the right relative content. The shortcut keys can be operated as below:

Content: Interface up: turn to the last interface in the content;

Interface down: turn to the next interface in the content;

Directory: Upward direction key: Check the last directory;

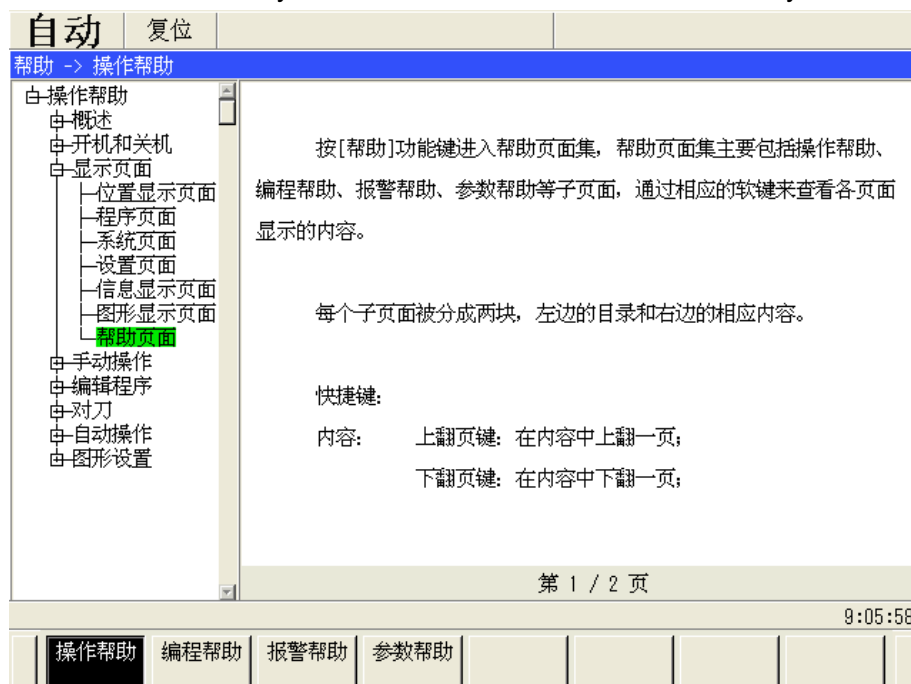
Downward direction key: Check the next directory;

Right direction key: Return to the previous directory;

Left direction key: Open the next directory;

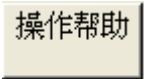
Alter + interface up key: turn to the last interface in the directory;

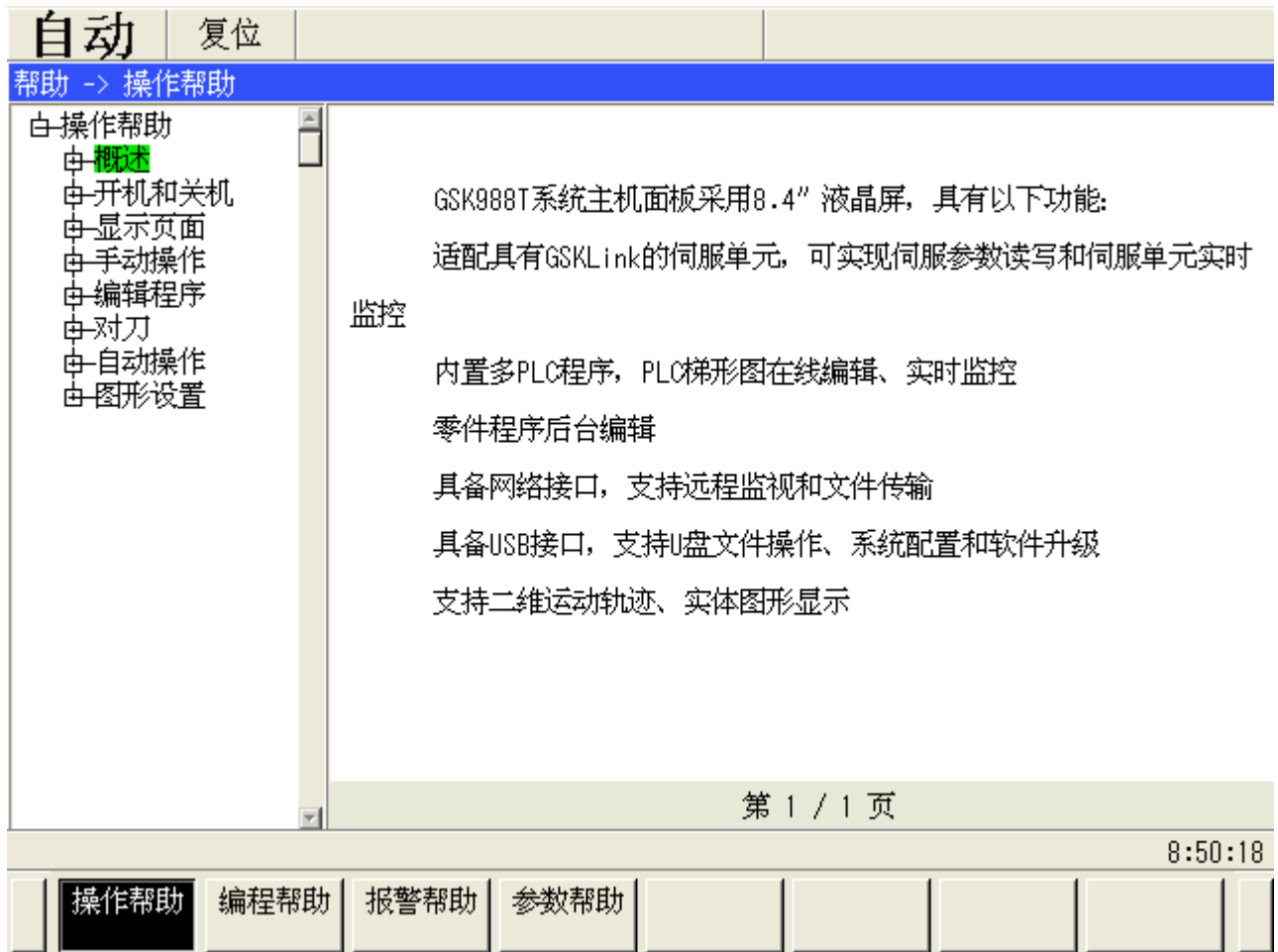
Alter + interface down key: turn to the next interface in the directory;



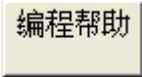
3.8.1 Operation help

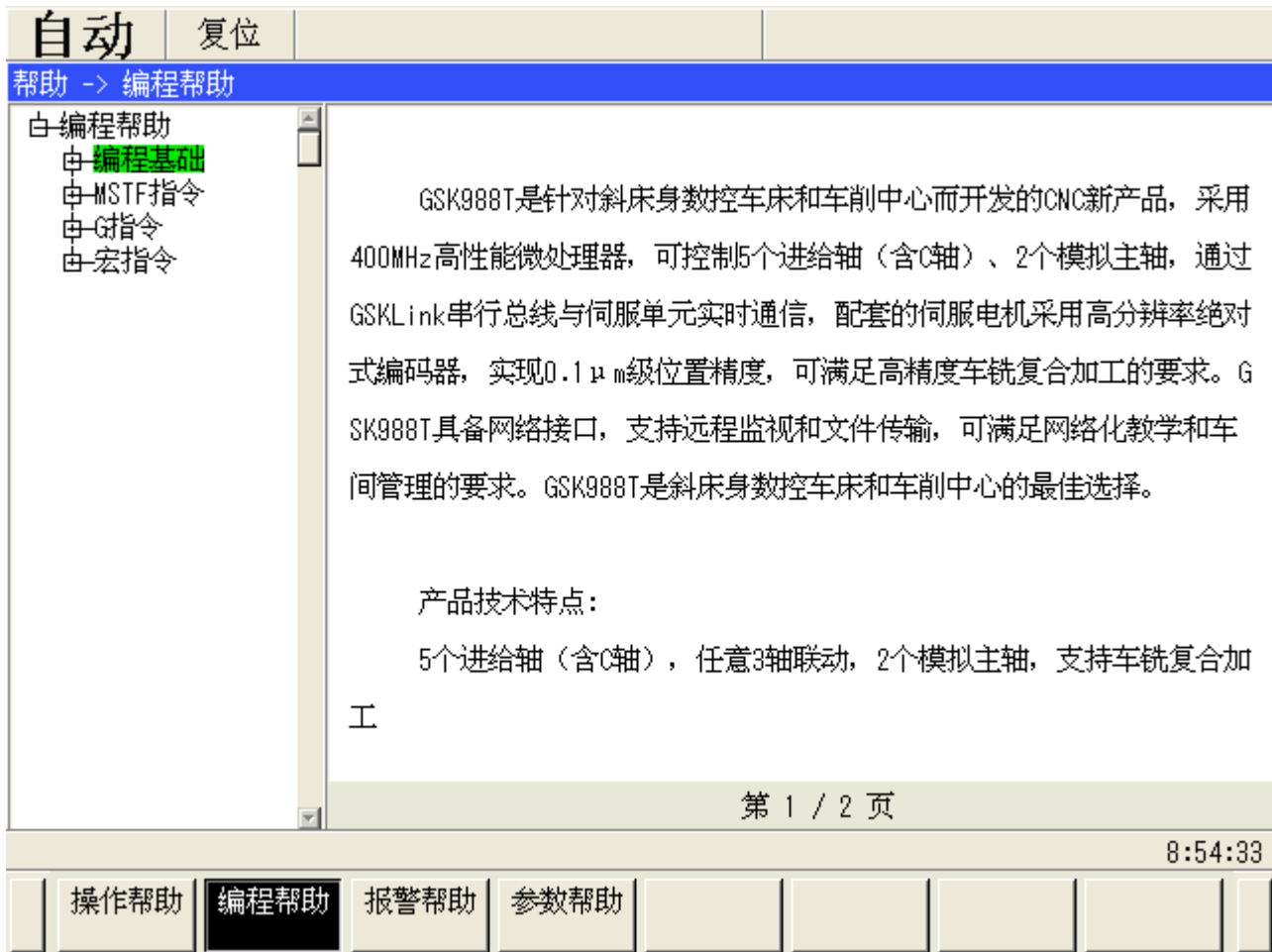
操作帮助

On the help interface, press  to access the operation help interface and it is shown as below:



3.8.2 Programming help

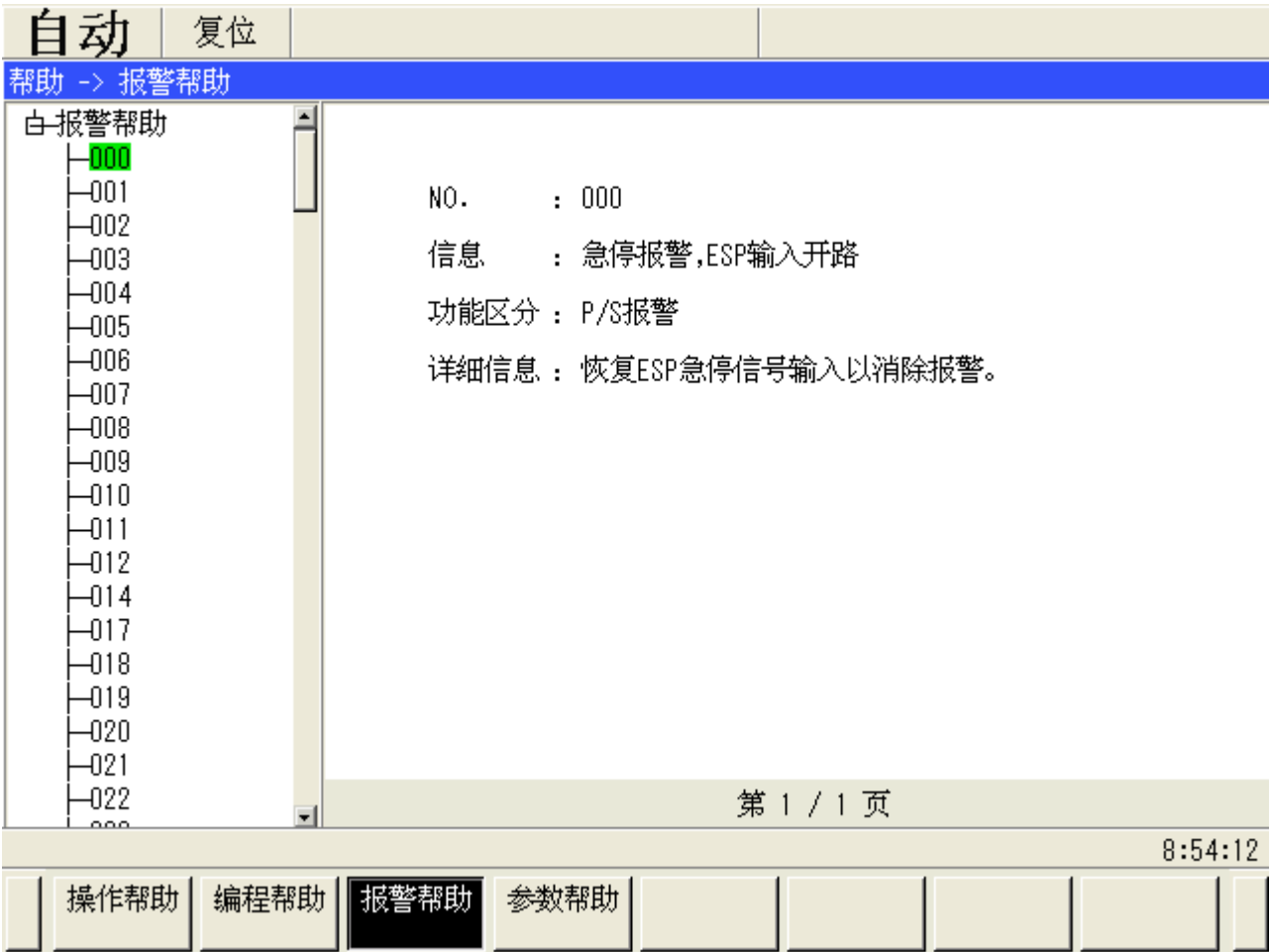
On the help interface, press  to access the programming help interface and it is shown as below:



3.8.3 Alarm help

报警帮助

On the help interface, press **报警帮助** to access the alarm help interface and it is shown as below:



3.8.4 Parameter help

On the help interface, press  to access the parameter help interface and it is shown as below:



Chapter IV Manual operation

4.1 Manual reference point return

There is one specified point in CNC machine, which can set the position of the machine working table. The specified point is taken as the reference point, in the position, the tool is changed and the coordinate system is set. After connecting the power supply, the tool traverses to the reference point. Manual reference point return is to use the switch and the button on the panel to traverse the tool to the reference point.

The process of reference point return of non-absolute encoder is as below:

The tool traverses in the direction which is set by parameter ZMI (the 5th bit of #1006), and each axis switch for reference point return is on the machine panel. The tool traverses to the decelerating point at the rapid traverse speed, and then traverse to the reference point at FL speed. Each axis rapid movement speed, rapid movement override F0 speed and FL speed of reference point return are respectively set by parameters #1420, #1421 and #1425.

During rapid movement, the rapid movement overrides of four gears are valid.

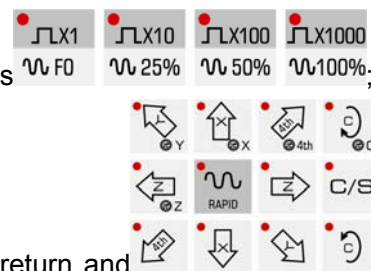
LED indicator is on after the tool returns to the reference point and completes.

Non-absolute encoder reference point return steps:



1, Press REF. RETURN and it is one of the mode selecting switches;

2. To decelerate, press one of the rapid traverse override switches



3. Press the feeding axis corresponding to the reference point return and execute the reference point return. The tool traverses to the deceleration point at the rapid traverse speed and then traverse to the reference point at FL speed set by parameter. After the tool returns to

the reference point, the reference point return finish indicator (LED) is on.

4. Execute the same operation with the other axes.

The steps of the absolute encoder reference point return:

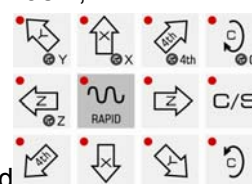


1. In MANUAL or MPG mode, move the machine to the reference point position to be set.

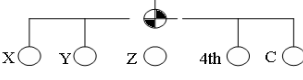
2. Set parameter APZx (NO.1815#4) as 0, the system alarms, and connect the power supply again after power off.



3. Press REF. RETURN, set manual reference point return select signal ZRN as 1;



4. Press the feeding axis corresponding to the reference point and , set the corresponding axis and direction selection signal Jx as 1, execute reference point return.

5. The system saves the current encoder position, auto set parameter APZx as 1, set the reference point return finish signal ZPx and the reference point creating signal ZRFx as 1, and the reference point return finish indicator  is on, the reference point return completes.

Remark:

1. Manual reference point return can only return to the 1st reference point; after the manual reference point return finishes, the coordinate system is auto set.
2. Once the reference point return finishes, “reference point return finish” indicator is on, the machine doesn’t move anymore until the reference point return switch is cut off.
3. When “reference point off” or “during emergency stop”, the reference point return finish indicator is off.
4. The direction of each axis reference point return is set by the 5th bit of parameter #1006.
5. Setting the 2nd bit of parameter 1404: After set the reference point, manually return to reference point, and moves to the reference point at the rapid feed rate or manual rapid feed rate.
6. After the system reference point of the absolute encoder is set, auto set the coordinate system after power on again, and it doesn’t require reference point return. But the non-absolute encoder system requires executing the reference point return after power on again.

The above is just one example; refer to the manual provided by the machine manufacture during the actual operation.

4.2 JOG (manual continuous) feeding


In JOG mode, press the feeding axes and direction selection switches on the machine panel, the machine moves along the selected axis.

Each axis manually continuous feed rate is set by parameter (#1423), and each axis manual continuous feed rate can be adjusted through manual continuous feed rate override dial.




Feed rate dial

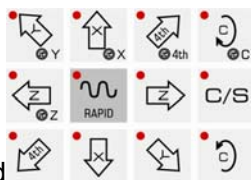



Press  and the machine moves at the rapid movement speed set by #1424 parameter, no matter where is JOG feed rate override dial, and the function is called as the manual rapid movement. During the manual operation, many axes can move at the same time.

JOG feeding steps:








1. Press  and it is one of mode selection switches,



2. Press feeding axis and , the machine moves along the corresponding axis in the corresponding direction. When the switch is pressed, the machine moves at the feed rate set by parameter (#1423); once the switch is released, the machine stops feeding;

3. Manual continuous feed rate can be adjusted through the manual continuous feed rate override dial;

4. If the feeding axis, the direction selection switch and  are pressed meanwhile, the machine moves at the rapid movement speed, and     can be selected and valid during the rapid movement;

The above is just one example; refer to the manual provided by the machine manufacturer during actual operation.

Remark:

1. Acceleration/deceleration;

Manual rapid movement speed, the time constant and the mode of acceleration/deceleration can be set by parameter 1610 and 1624.

2. Changing the mode:

During JOG feeding, when the mode is switched into the other mode, JOG feeding becomes invalid. To make JOG feeding valid, firstly access JOG feeding mode, and then press feeding axis and mode selection switch.

3. Rapid movement before the reference point return:

If the reference point doesn't return after connecting the power supply, even press "rapid movement" button, it can't run; while remain manual continuous feeding movement. The function can be set by parameter RPD (0 bit of #1401).

4. In manual mode, whether JOG override is valid, which is set by the 2nd bit of parameter #1402; when it is invalid, the override is fixed as 100%.

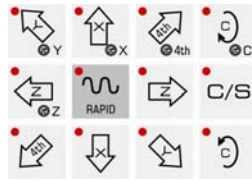
4.3 Increment feeding

In JOG mode, during MPG or manual feeding, whether the increment feeding is valid, this is set by parameter JHD (0 bit of #7100). The corresponding relation is shown as below:

	JHD=0		JHD=1	
	JOG mode	MPG mode	JOG mode	MPG mode
JOG feeding	O	×	O	×
MPG feeding	×	O	O	O
Increment feeding	×	×	×	O


O: Valid





×: Invalid



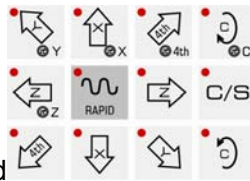
In increment mode, press feeding axis and on the machine panel, the machine moves one step in the selected axial direction. The minimum distance which machine moves is the minimum input increment, and each step can be 1 time, 10 times or 100 times of the minimum input increment.


Increment feeding steps:

1. Press  to select MPG mode;

2. Press     to select the movement distance of each step; moreover,

the distance of each step selected by   is rewritten by parameter 7113 and 7114;




3. Press the feeding axis and  , the machine moves along the selected axial direction. Press the switch for one time, it moves for one step. Its feed rate is same as the manual continuous feed rate.

4. Press rapid movement switch when the feeding axis and the direction selection switches are on, the machine moves at rapid movement speed.

The rapid movement override is valid during the rapid movement.

Remark: The minimum input unit (input) and the minimum command increment (output) are set by the 1st bit of parameter #1004. The minimum input increment is the minimum unit of the programming movement distance, the minimum command increment is the minimum unit of the tool traverse on the machine, and the two increments are represented by millimeter or inch.

4.4 MPG feeding






Press  to access MPG mode, the appearance of MPG is shown as below:



MPG outside drawing













In MPG mode, the machine moves continuously through rotating MPG on the operational panel.



And press      to select the movement axis. When MPG rotates one graduation, the minimum distance of the tool traverse is the minimum input increment. When MPG rotates one graduation, the tool traverse distance can be magnified 10 times or one of two

overrides is set by parameters #7113 and #7114.

MPG feeding steps:

1. Press  to access MPG mode;
2. Press      to select the axis which is moved by one machine;
3. Press     to select the override of the machine movement. When MPG rotates for one graduation, the minimum distance traversed by the machine is the product of the minimum input increment multiplying the current override. The override set by   can be rewritten by parameters #7113 and #7114;

4. Rotation MPG machine moves along the selection axis, MPG rotates for 360° and the machine movement distance is that of 100 graduations.

MPG feeding direction is set by MPG rotation direction. Normally, MPG CW feeds positively, CCW negatively.

The above is just one example; refer to the manual provided by the machine manufacturer during actual operation.

Remark:

1. In JOG mode (JHD), MPG is valid;

In JOG mode, whether MPG can be used, which is set by parameter JHD (the 0 bit of #7100), when parameter JHD (the 0 bit of #7100) is set as 1, MPG feeding and increment feeding are both valid.

The corresponding relation is shown as the following list:

	JHD=0		JHD=1	
	JOG mode	MPG mode	JOG mode	MPG mode
JOG feeding	O	×	O	×
MPG feeding	×	O	O	O
Increment feeding	×	×	×	O

O: Valid

×: Invalid

2. The commands of **MPG** exceed the rapid traverse speed (**HPT**);

The parameter HPT (the 4th bit of #7100) is stipulated as below:

Setting as 0: When the feed rate is limited by the rapid traverse speed, the impulse value exceeding the rapid traverse speed is invalid. (The machine movement distance doesn't comply with MPG graduation).

Setting as 1: The feed rate is limited by the rapid movement speed, and the impulse value exceeding the rapid movement speed is valid, but it is accumulated in CNC. (Although MPG isn't rotated, the machine can't stop. After MPG stops, the machine still moves due to the effect of CNC pulse.) The allowable value of the memory capacity is set by parameter #7117, then, the part exceeding the memory capacity is ignored.

3. Axial movement direction and MPG rotation direction:

Parameter HNGx (the 0 bit of #7102) switches into MPG direction which the tool traverses along the axis and it corresponds to MPG rotation direction.

4. Quantity of MPG

The maximum 2 manual pulse generators can be connected, which is set by parameter #7110. The two generators can operate one selected axis meanwhile.

Chapter V Editing and managing the program

On the program interface, the program can be created, selected, rewritten, copied and deleted, also channelled in and out.

To prevent the programs are rewritten and deleted by accident, the program switches are set in GSK988T. Before rewriting the program, the program switches must be on. About the setting of program switches, refer to chapter 3.4.2.1.

5.1 Creating and editing the program

Only when the operation authority is above level (3), the program can be created and edited.

5.1.1 Creating the program

(1) Firstly press **PROGRAM** and then press **本地目录** to access the program interfaces, which is shown as below:

自动	复位		
程序 -> 本地目录			
程序数: 7	占用空间(字节): 10,561	剩余空间(字节): 16,349,184	
程序名	注 释	长度(字节)	修改时间
00001 实例加工		1,835	2009-03-25,16:11:27
00005 00005		14	2009-03-25,16:14:29
01111 说明书的所有程序		5,311	2009-03-25,15:18:31
08000 测试主程序		347	2009-03-25,15:18:33
08001 G00/G01测试程序		714	2009-03-25,15:18:36
08002 G02/G03测试程序		1,355	2009-03-25,15:18:37
08090 G90/G94测试程序		1,185	2009-03-25,15:18:38
17:44:08			
本地目录	MDI程序	目 / 次	查找 新建 执行 打开 >

(2) On the program interface, press **新建** to access the creating interface, which is shown as below:

自动	复位		
程序 -> 本地目录			
程序数: 6		占用空间(字节): 10,547	剩余空间(字节): 16,347,648
程序名	注 释	长度(字节)	修改时间
00001	实例加工	1,635	2009-03-25,16:11:27
01111	说明书的所有程序	5,311	2009-03-25,15:18:31
08000	测试主程序	347	2009-03-25,15:18:33
08001	G00/G01测试程序	714	2009-03-25,15:18:36
08002	G02/G03测试程序	1,355	2009-03-25,15:18:37
08090	G90/G94测试程序	1,185	2009-03-25,15:18:38
<div> <div>请输入新程序名</div> <div>0</div> </div>			
16:13:49			
确定	取消		

请输入新程序名

0

(3) Input the new program name in , for example, input 0005, press

确定

to access #O0005 program editing interface, which is shown as the following figure 1:

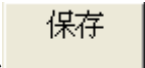


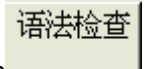
Fig. 1

Then, the new program is edited through the editing keypad and check whether there are grammar mistakes and then the new one is saved.

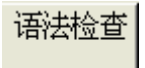
5.1.2 Editing the program

- (1) In 5.1.1, create one program or open one existing program;
- (2) Based on the program grammar rules, the program codes are edited in the program editing box.

- (3) Press  to save the edited programs.

- (4) After editing the programs, press  to check whether there are any grammar mistakes in the edited program.

Remark: When parameter 3404.6 is 0, there are the ending codes of M02, M30 and M99, etc;

otherwise, press , it reminds the mistake during checking the program; it alarms when the program runs.





5.2 Opening and rewriting the program

When the operation level is above level (3), the program can be opened, edited and rewritten.

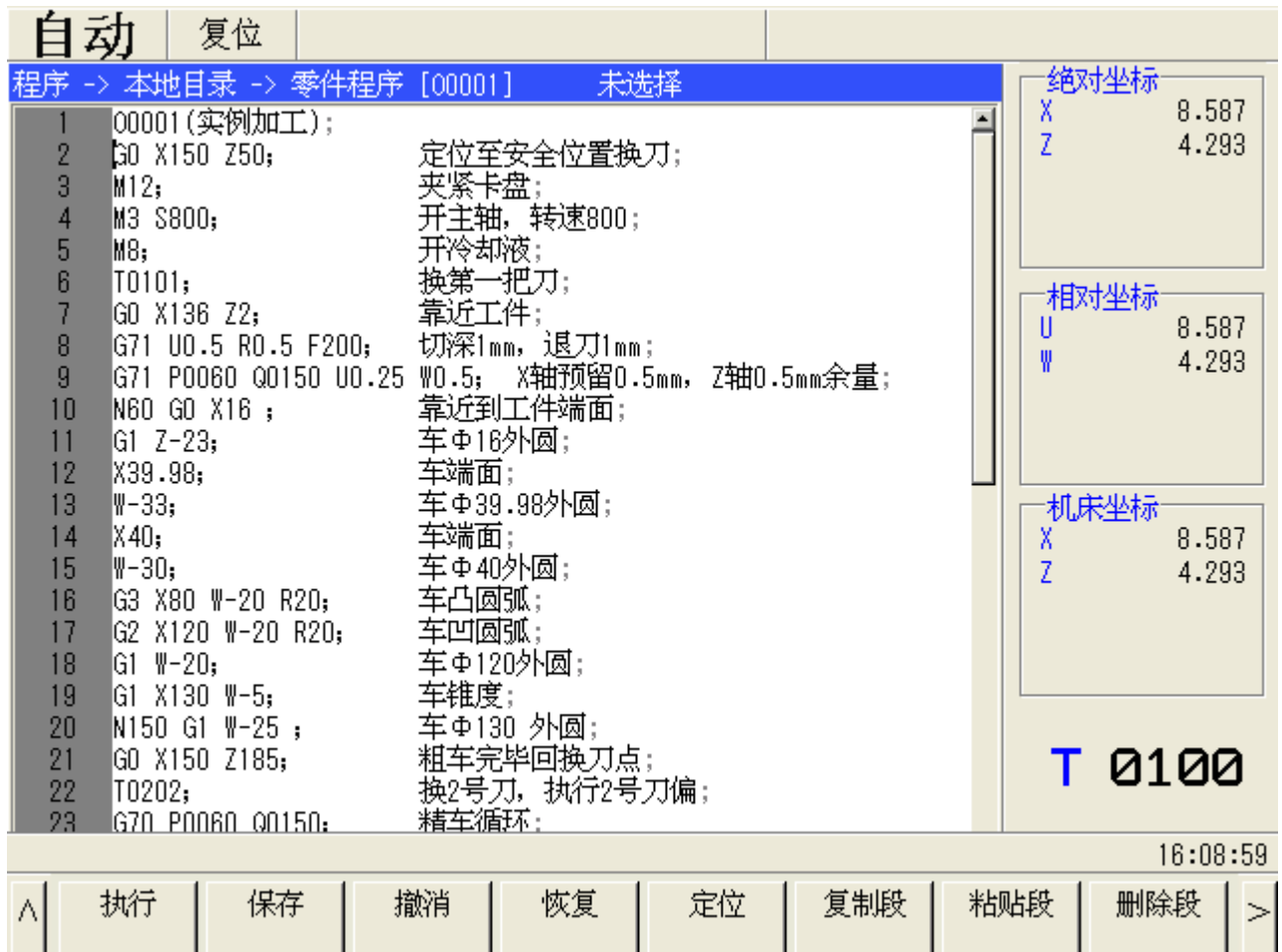
5.2.1 Opening the program

(1) Press **PROGRAM** and then press **本地目录** to access the program interfaces, which is shown as below:

自动	复位		
程序 -> 本地目录			
程序数: 7		占用空间(字节): 10,561	剩余空间(字节): 16,349,184
程序名	注 释	长度(字节)	修改时间
00001	实例加工	1,635	2009-03-25,16:11:27
00005	00005	14	2009-03-25,16:14:29
01111	说明书的所有程序	5,311	2009-03-25,15:18:31
 08000	测试主程序	347	2009-03-25,15:18:33
08001	G00/G01测试程序	714	2009-03-25,15:18:36
08002	G02/G03测试程序	1,355	2009-03-25,15:18:37
08090	G90/G94测试程序	1,185	2009-03-25,15:18:38
9:16:15			
本地目录	MDI程序	目 / 次	查找 新建 执行 打开 >

(2) In the program interfaces, press , ,  or  to select the program to be opened; or press **查找** to search, and input the program name to be opened, and then press **确定** to search, and the cursor positions in the program name, the background of the selected program name changes into green-based color, such as O0001 in the above figure;





(3) Press **打开** to open the codes of the selected program in the screen, which is shown as below:





Then, the current program can be edited and rewritten, but when the program is being executed, it must be edited in the editing mode.

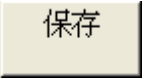
5.2.2 Editing and rewriting the program

- (1) Open the program based on the steps of 5.2.1 chapter;

(2) Press , ,  or , the cursor moves toward the program line to be rewritten;

Press  or  and the cursor moves to the character to be rewritten;





(3) Press the address and the numerical keys, which are on the editing keypad, to input the program codes to be rewritten;



(4) Press  to save the program which is edited.

5.2.3 Inserting and deleting the characters

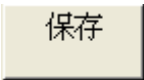
1. Inserting the characters

- (1) Open the programs based on the steps of 5.2.1 chapter;

(2) Press , ,  or , the cursor moves toward the program line to insert





the characters; Press  or  and the cursor moves toward the place to insert the characters;



(3) Input the characters to be rewritten;


(4) Press  to save the rewritten program.


2. Deleting the characters

(1) Open the programs based on the steps of 5.2.1 chapter;

(2) Press , ,  or  and the cursor moves toward the program line to

delete the characters; press  or  and the cursor moves toward the place to select the characters;

(3) Press  to delete one character before the cursor is.

(4) Press  to delete one character behind the cursor is.

5.2.4 Remarking the block

When the block should be remaked, press “EOB” and add up with “; ” behind the block, and the content behind“; ” is the remarked.

For example:


O0001;

G50 X0 Z0; Setting the coordinate zero point;


G00 X100 Z100; Rapidly move to the positions of X100 and Z100;

M30;

In the above program, add up with the remark in the 2nd and the 3rd block, the content after the

first semicolon is remarked, the second semicolon is the block ending code. Press  to enter a new line after editing one block, and the system auto adds the semicolon.

5.2.5 Skipping the blocks

Add “/” before the block, press  to start the skipping block mode, the indicator is on, press



to run the program and the block is skipped without running, which is shown as the 4th line of the following program:

O0001;

G50 X0 Z0; Setting the coordinate zero point;

G01 X100 Z100; Rapidly move to the positions of X100 and Z100;

/G0 X0 Z0;

M30;



Press to run the program and the 4th line is skipped.

5.2.6 Generating the block number

In the program, the block number can be edited or not edited; the program is executed based on the editing sequence of the block (except calling).

In the setting interfaces, CNC setting interface, when “auto generating number” switch is off, CNC can’t auto generate the block number, the block number can be edited manually during programming.

In the setting interface, CNC setting interface, when “auto generating number” switch is on, CNC auto generates the block number; during editing, press to enter a new line and auto generate the number of the next block, the increment value of the block number is set by CNC data parameter #3216.

5.2.7 Shortcut keys

During editing the program, the system provides the shortcut keys to adjust the cursor position, which are shown as below:

At the same time, press		and		, the cursor moves toward the ahead of the file;
At the same time, press		and		, the cursor moves toward the end of the file;
At the same time, press		and		, the cursor moves toward the ahead of the line;
At the same time, press		and		, the cursor moves toward the end of the line;





5.3 Deleting the program and the block



When the operation authority is above level (3), the program and the block can be deleted.

5.3.1 Deleting the program

(1) Press to access the program interfaces, which is shown as below:

自动	复位		
程序 -> 本地目录			
程序数: 7		占用空间(字节): 10,561	剩余空间(字节): 16,349,184
程序名	注 释	长度(字节)	修改时间
00001	实例加工	1,635	2009-03-25,16:11:27
00005	00005	14	2009-03-25,16:14:29
01111	说明书的所有程序	5,311	2009-03-25,15:18:31
08000	测试主程序	347	2009-03-25,15:18:33
08001	G00/G01测试程序	714	2009-03-25,15:18:36
08002	G02/G03测试程序	1,355	2009-03-25,15:18:37
08090	G90/G94测试程序	1,185	2009-03-25,15:18:38
17:44:08			
本地目录	MDI程序	目 / 次	查找 新建 执行 打开 >

(2) On the program interfaces, press , ,  or  to select the program to be deleted, the background of the selected program changes into the green-based color, which is shown as the above figure O1111;





(3) Firstly press  and then press  to delete the selected programs.

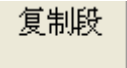
5.3.2 Copying, pasting and deleting the block

1. Copying the block

1. 1 Copying the single block





(1) Open the selected program based on the steps of 5.2.1 chapter;






(2) Press , ,  or  and the cursor moves to the block to be copied;

(3) Press  to copy the block which the cursor is;







2. Copying any block

(1) Open the selected program based on the steps of 5.2.1 chapter;

(2) Press , ,  or  and the cursor moves before the block to be copied;





(3) Press  + , ,  or  and the cursor moves behind the command to be copied, then, the block in the middle is selected and displays in the opposite color;

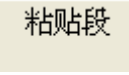


(4) Meanwhile, press  and  to copy the selected block.

Remark: During editing the program, the system provides some shortcut keys. For example, press  and  meanwhile to copy the selected block, press  and  to cut the selected block, and press  and  to paste the copied or cut block.

II. Pasting the block





(1) Open the program based on the steps of 5.2.1 chapter.

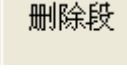
(2) Press , ,  or  and the cursor moves toward the place to paste the block;

(3) Press  to paste the copied block where the cursor is; Or press  +  to paste the copied block where the cursor is.

III. Deleting the block


(1) Open the selected program based on the steps of 5.2.1 chapter.

(2) Press , ,  or  and the cursor moves toward the block to be deleted;

(3) Press  to delete the block where the cursor is;





5.4 Selecting and executing the block

5.4.1 Selecting the block

(1) Press  to access the edit mode;

(2) Press function keys to access the program interfaces, which is shown as the following figure:

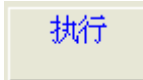
自动	复位		
程序 -> 本地目录			
程序数: 7		占用空间(字节): 10,561	剩余空间(字节): 16,349,184
程序名	注 释	长度(字节)	修改时间
00001	实例加工	1,635	2009-03-25,16:11:27
00005	00005	14	2009-03-25,16:14:29
01111	说明书的所有程序	5,311	2009-03-25,15:18:31
 08000	测试主程序	347	2009-03-25,15:18:33
08001	G00/G01测试程序	714	2009-03-25,15:18:36
08002	G02/G03测试程序	1,355	2009-03-25,15:18:37
08090	G90/G94测试程序	1,185	2009-03-25,15:18:38
9:16:15			
本地目录	MDI程序	目 / 次	查找 新建 执行 打开 >

(3) On the program interfaces, press , ,  or  to select the program to be opened, and the background of the selected program changes into the green-based color, which is shown as # O0001 program in the above figure.


5.4.2 Executing the program

(1) Select the program to be executed based on the steps of 5.4.1 chapter;

执行

(2) Press  and the program can be executed, and then the interface jumps into the position interface, and the program to be loaded displays on the block column in the position interface, which is shown as # O0001 program in the following figure:

自动		复位			
绝对坐标				工艺数据	
X		0.000 mm		T 0100	
Z		0.000 mm		F 0 mm/min 1000 mm/min	
				S 0 rev/min	
程序名 [00001]				综合信息	
1	00001 (实例加工);			进给倍率	100% 手轮倍率 X1
2	G0 X150 Z50; 定位至安全位置换刀;			快速倍率	100% 加工件数 0
3	M12; 夹紧卡盘;			主轴倍率	100% 运行时间 00:00:00
4	M3 S800; 开主轴, 转速800;			手动倍率	0% 切削时间 00:00:00
5	M8; 开冷却液;				
6	T0101; 换第一把刀;				
7	G0 X136 Z2; 靠近工件;				
8	G71 U0.5 R0.5 F200; 切深1mm, 退刀1mm				
15:50:38					
绝对坐标		相对坐标		机床坐标	
综合坐标		模态		相对坐标设置	

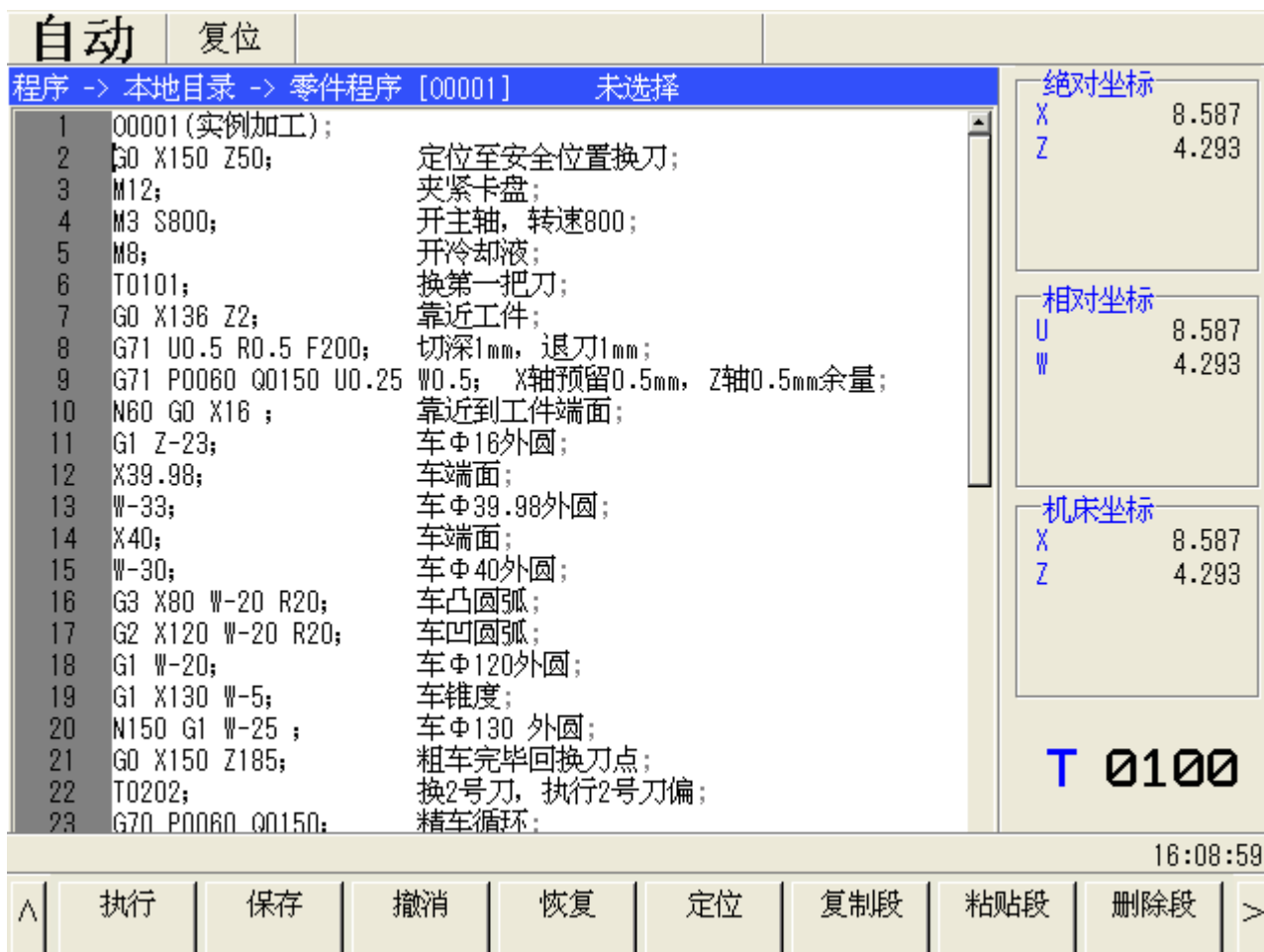
(3) Press  to access the auto mode;



(4) Press  to execute the loaded program.

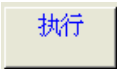
5.4.3 Executing from any block

(1) Select the program to be executed based on the steps of 5.4.1 chapter;

(2) Press  to open the selected program on the screen, which is shown as below:




(3) Press  or , the cursor moves toward the block to be executed. If execute from the 8th block in # O0001 program, the cursor moves toward the 8th block; if # O0001 program is being executed currently, adjust the running block and switch into the edit mode;

(4) Press  and the program can be executed, and then the block where the cursor is begins to display on the position interface, which is shown as below:

自动		复位	
绝对坐标		工艺数据	
X	0.000 mm	T	0100
Z	0.000 mm	F	0 mm/min 200 mm/min
		S	0 rev/min
程序名 [00001]		综合信息	
4	M3 S800; 开主轴, 转速800;	进给倍率	100% 手轮倍率 X1
5	M8; 开冷却液;	快速倍率	100% 加工件数 0
6	T0101; 换第一把刀;	主轴倍率	80% 运行时间 00:00:00
7	G0 X136 Z2; 靠近工件;	手动倍率	0% 切削时间 00:00:00
8	G71 U0.5 R0.5 F200; 切深1mm, 退刀1mm;		
9	G71 P0060 Q0150 U0.25 W0.5; X轴预留0.5mm, Z轴0.5mm余量;		
10	N60 G0 X16; 靠近到工件端面;		
9:33:46			
绝对坐标	相对坐标	机床坐标	综合坐标
		模态	相对坐标设置

(5) If it isn't the auto mode, press  and switch into the auto mode;

(6) Press  to execute the loaded program from the selected block.

Chapter VI Tool offset and setting tools

To simplify the programming, the actual position of the tool isn't taken into consideration during programming and GSK988T provides the methods of in-position tool-setting and trial cutting, etc, and get the tool offset data through setting tools.

6.1 In-position tool setting

In-position tool-setting is to set the tool offset data through C input mode.

The steps are as below:

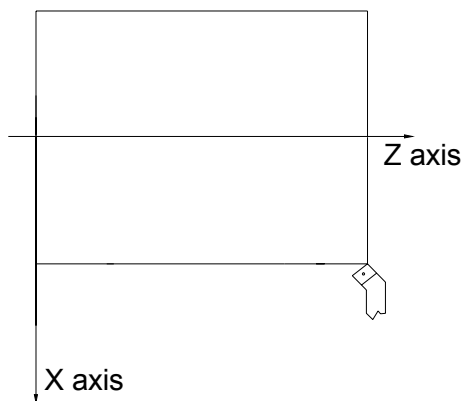


Fig. A

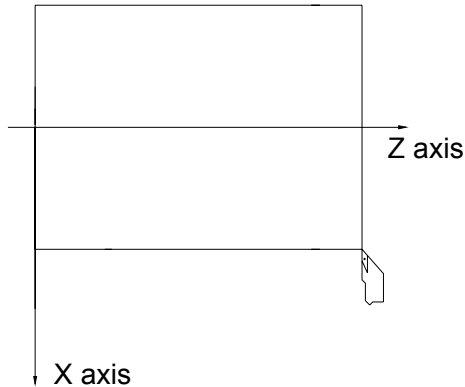




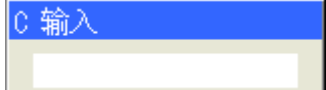


Fig. B

1. Firstly confirm the tool compensation value in X or Z direction is 0; if not, the tool compensation values of all tool numbers must be cleared;
2. The tool offset number is 00 (such as T0100 and T0300);
3. Select any tool (normally the first tool during processing is taken as the datum tool)
4. The nose of the datum tool positions in some point (tool-setting point), which is shown as figure A;
5. In MDI mode, G50 X__ Z__ command, on the program interface, sets the work piece coordinate system;
6. The value of relative coordinate (U, W) is cleared;
7. The tool traverses to the safe position, and the other tool is selected and traverses to the tool-setting point, which is shown as figure B;

8. On the setting interface, press **刀偏设置** to access the tool offset management interface, press **查找** to select the tool offset number, or press  or  to select the interface, and press  or  to select the tool offset or wearing data to be rewritten;

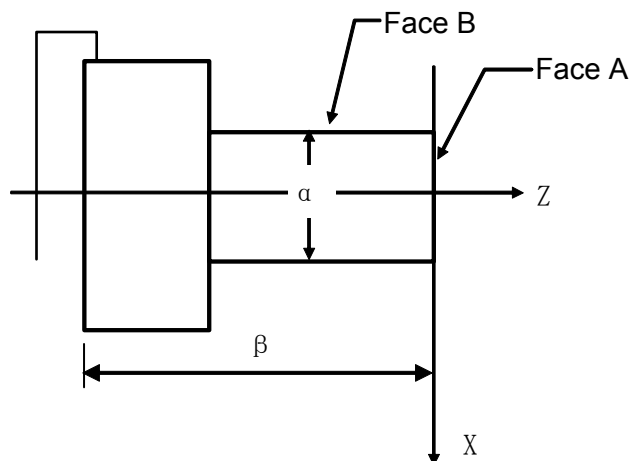
9. Press **C 输入** to access C input interface, input axial name , press **确定** and the tool offset value or the wearing value is set in the corresponding offset number;

10.Repeat the steps of 7~9, other tools can be set.

6.2 Trial tool cutting

After the coordinate system is set, the trial cutting tool is to set the tool offset value through measuring input method.

The steps are as below: (set the work piece coordinate system based on the work piece face):



1. Any tool is selected and the tool cuts along face A;
2. When Z axis remains still, the tool retracts along X axis and the spindle stops revolving;
3. Measure the distance " β " from zero point of the work piece coordinate system to surface A;

刀偏设置

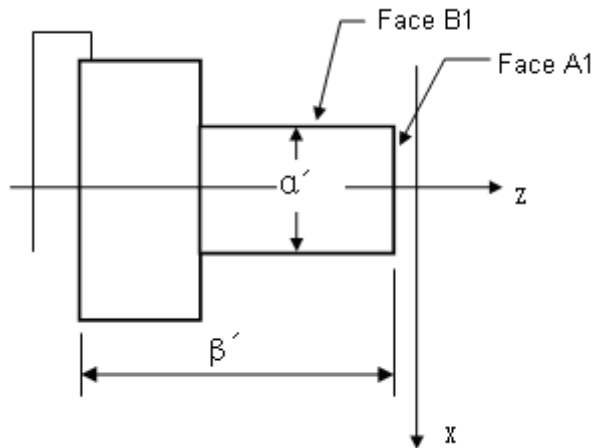
4. On the setting interface, press to access the tool offset management interface, press to select the tool offset number, or press or to select the interface, and press or to select the tool offset or the wearing data to be rewritten;

5. Press to access the measuring interface, input and measuring value β in , and then press , Z axis tool offset value or its wearing value is set in the corresponding offset number;

6. The tool cuts along surface B;
7. When X axis remains still, the tool retracts along Z axis and the spindle stops revolving;
8. Measure diameter " α ";

9. Press to access the measuring interface, input and the measuring value α in ; After pressing , X axis tool offset value or its wearing value is set in the corresponding offset number;

10. The tool traverses to the safe position for changing into the other one;



11. The tool cuts along face A1;
12. When Z axis remains still, the tool retracts along X axis and the spindle stops revolving;
13. Measure the distance " β " from face A1 to the origin of the work piece coordinate system;

刀偏设置

14. On the setting interface, press

查找

and press



or



to select the interface, and press



or



to select the tool offset or the wearing data to be rewritten;

测量输入

15. Press

$\begin{matrix} Y \\ Z \end{matrix}$

and measuring value

β' in

测量输入

确定

; After pressing, Z axis tool offset value or its wearing value is set in the corresponding offset number;

16. The tool cuts along face B1;

17. When X axis remains still, the tool retracts along Z axis and the spindle stops revolving;
18. Measure distance " α' ";

测量输入

19. Press

$\begin{matrix} C \\ X \end{matrix}$

and the measuring value

α' in

测量输入

确定

, and then press and X axis tool offset value or its wearing value is set in the corresponding offset number;

20. About the method of setting other tools, repeat the steps of 10~19.

Measuring method is to set the differential value between the tool reference position (such as the tool nose position) and the actual tool nose position during processing as the tool offset value. For example: when the coordinate value of face B is 50.0, the actual measured value is $\alpha=49.0$, then the tool offset value in X direction is 1.0.

6.3 Setting the tool offset and the wearing values

6.3.1 Direct input method

(1) On the setting interface, press **刀偏设置** to access the tool offset management interface, which is shown as below:

MDI 复位

设置 -> 刀偏设置 -> 刀具偏置

刀偏号		X	Z	Y	R	T
001	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
002	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
003	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
004	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
005	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
006	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
007	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	

绝对坐标

X 0.6553
Z 0.0000
Y 0.0000
B 0.0000
C 0.0000

相对坐标

U 0.6553
W 0.0000
V 0.0000
B 0.0000
H 0.0000

机床坐标

X 26.2146
Z 14.4182
Y 35.3901
B 74.0559
C 360.0000

T 0000

14:25:28







刀偏设置

CNC设置

宏变量

刀具偏置

刀具寿命

(2) On the interface, press  or  to select the interface, and press  or  to select the tool offset number to be rewritten, and press  or  to select the axial offset data, the wearing data or T value of the assumed tool nose direction to be rewritten, which is shown as X axis offset of #001 tool offset in the above figure; About the relative relation of the assumed tool nose, refer to the tool nose radius compensation in the 4th chapter in *programming introduction*.

(3) Directly rewrite the tool offset data, the wearing data or the relative assumed tool nose direction number T through the numerical keys or the backspace key; or press **INPUT** to make the selected tool offset value be input, such as X axis offset of #001 tool offset shown as the following figure, and then rewrite the tool offset data, the wearing data or the corresponding assumed tool nose direction number T through pressing the numerical keys or the backspace key;

MDI		复位				
设置 -> 刀偏设置 -> 刀具偏置						
刀偏号		X	Z	Y	R	T
001	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
002	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
003	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
004	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
005	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
006	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
007	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	

绝对坐标

X 0.6553
Z 0.0000
Y 0.0000
B 0.0000
C 0.0000

相对坐标

U 0.6553
W 0.0000
V 0.0000
B 0.0000
H 0.0000

机床坐标

X 26.2146
Z 14.4182
Y 35.3901
B 74.0559
C 360.0000

T 0000

14:25:51

刀偏设置 CNC设置 宏变量 刀具偏置 刀具寿命

(4) Press **INPUT** to complete the input or rewriting, or switch into the other interface to complete the rewriting.

(5) Move the cursor to set the other tool offset value, wearing value or T value of the assumed tool nose direction.

Remark: The maximum value of the tool wearing compensation value can be rewritten through parameter 5013.

6.3.2 Measuring mode

(1) On the setting tool offset interface, press **刀具偏置** to access the tool offset management interface;

(2) Press **测量输入** or **测量输入** to select the interface, and press **↑** or **↓** to select the tool offset number to be rewritten, or press **←** or **→** to select the axial tool offset data or the wearing value to be rewritten.

(3) Press **测量输入** to access the measuring interface to measure the tool offset value, which is shown as below:

MDI

复位

设置 -> 刀偏设置 -> 刀具偏置

刀偏号		X	Z	Y	R	T
001	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
002	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
003	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
004	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
005	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
006	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
测量输入		0.0000	0.0000	0.0000	0.0000	0
		0.0000	0.0000	0.0000	0.0000	0

绝对坐标

X0.6553
Z0.0000
Y0.0000
B0.0000
C0.0000

相对坐标

U0.6553
W0.0000
V0.0000
B0.0000
H0.0000

机床坐标

X26.2146
Z14.4182
Y35.3901
B74.0559
C360.0000

T 0000

14:29:20

确定取消

(4) Input “the coordinate axis number + axis value” to be measured in

press **确定** or **INPUT** for in-position measuring;

(5) Calculating the offset value:







If the cursor is in the tool offset box, the tool wearing value is cleared, the tool offset value = the relative coordinate value – the input coordinate value;

If the cursor is in the tool wearing box, the tool wearing value remains unchanged, the tool offset value = the relative coordinate value — the input coordinate value — the wearing value relative to the coordinate axis.

Remark: The lathe tool-setting isn’t with the tool compensation value.

6.3.3 +input mode

(1) On the setting tool offset interface, press **刀具偏置** to access the tool offset management interface;

(2) Press  or  to select the interface, and press  or  to select the tool offset number to be rewritten, and press  or  to select the axial tool offset data or the wearing value to be rewritten;

(3) Press **+ 输入**, the selected tool offset value or the wearing value adds up one input value, which is shown as below:

MDI 复位

设置 -> 刀偏设置 -> 刀具偏置

刀编号		X	Z	Y	R	T
001	偏置	13.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
002	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
003	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
004	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
005	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
006	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
+ 输入		0.0000	0.0000	0.0000	0.0000	0
		0.0000	0.0000	0.0000	0.0000	

绝对坐标
X 0.8553
Z 0.0000
Y 0.0000
B 0.0000
C 0.0000

相对坐标
U 0.8553
W 0.0000
V 0.0000
B 0.0000
H 0.0000

机床坐标
X 26.2146
Z 14.4182
Y 35.3901
B 74.0559
C 360.0000

T 0000

14:30:11

确定 取消

(4) Input one numerical value in , the value can be negative. Press

确定

or

INPUT

to complete the input;

(5) Calculating the offset value: The offset value or the wearing value = the original offset value or the original wearing value + the input numerical value.

6.3.4 C input method

刀具偏置

(1) On the setting tool offset interface, press to access the tool offset management interface;

(2) Press or to select the interface, and press or to select the

tool offset number to be rewritten, and press or to select the axial tool offset data or the wearing value to be rewritten;

C 输入

(4) Press to access C input interface, which is shown as below:

MDI 复位

设置 -> 刀偏设置 -> 刀具偏置

刀偏号		X	Z	Y	R	T
001	偏置	13.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
002	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
003	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
004	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
005	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
006	偏置	0.0000	0.0000	0.0000	0.0000	0
	磨损	0.0000	0.0000	0.0000	0.0000	
C 输入		0.0000	0.0000	0.0000	0.0000	0
		0.0000	0.0000	0.0000	0.0000	0

绝对坐标
X 0.8553
Z 0.0000
Y 0.0000
B 0.0000
C 0.0000

相对坐标
U 0.8553
W 0.0000
V 0.0000
B 0.0000
H 0.0000

机床坐标
X 26.2146
Z 14.4182
Y 35.3901
B 74.0559
C 360.0000

T 0000

14:30:57

确定 取消

C 输入

(5) Input the coordinate axia name to be measured in , press **确定** for in-position measuring;







(6) Then, calculate the offset value;
Press C input button to input the axial number.

If the cursor is on the tool offset box, the tool wearing value remains unchanged, write in the tool offset value = the relative coordinate value – the tool wearing value;

If the cursor is on the tool wearing box, the tool offset value remains unchanged, write in the tool wearing value = the relative coordinate value – the tool offset value.

Remark: The lathe tool-setting isn't with the tool compensation value.

6.3.5 Clearing the offset value or the wearing value

On the tool offset management interface, press  or  to select the interface, and press  or  to select the tool offset number to be rewritten, and press  or  to select the tool offset data, the wearing value or the tool number to be cleared; press **数据清0**

to clear the selected tool offset value, the wearing value or the assumed tool nose direction number relative to the axis.



Chapter VII Auto operation

7.1 Auto running

The program should be saved in the memorizer in advance, when one program is selected and




is pressed on the machine operation panel, the program automatically runs and the cycle start

indicator is on. During cycle, press , auto running pauses. When  is pressed once more,

auto running starts again. When  is pressed on MDI panel, auto running ends and resets.





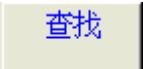
7.1.1 Selecting the running program

(1) In auto or edit mode, press  to access the program interfaces, the interface is shown as the following figure 1:

自动	复位		
程序 -> 本地目录			
程序数: 7		占用空间(字节): 10,561	剩余空间(字节): 16,349,184
程序名	注 释	长度(字节)	修改时间
00001	实例加工	1,635	2009-03-25,16:11:27
00005	00005	14	2009-03-25,16:14:29
01111	说明书的所有程序	5,311	2009-03-25,15:18:31
 08000	测试主程序	347	2009-03-25,15:18:33
08001	G00/G01测试程序	714	2009-03-25,15:18:36
08002	G02/G03测试程序	1,355	2009-03-25,15:18:37
08090	G80/G84测试程序	1,185	2009-03-25,15:18:38
9:16:15			
本地目录	MDI程序	目 / 次	查找 新建 执行 打开 >

Fig. 1

On the interface, it displays the total number of programs, the total occupied space and free space. In the list, it displays the program list, the size of each program and the latest rewritten date. Among them, the green-based program is the one the cursor is, which is shown as above # O0001 program; while the red-based program should be uploaded into the position interface currently, which is shown as above # O8000 program.

(2) In the program interfaces, press  ,  ,  or  , and the cursor moves to select the program name, or press  to search the program name to run. The selected program line displays on the green base, which is shown as above O0001 program.

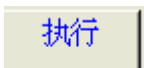


(3) During resetting, press  , the selected program is uploaded into the block area in the position interface and it can be executed, the current interface switches into the position interface, which is shown as the following figure 2:



Fig. 2

Then, O0001 program is selected to run.
Remark: Only during resetting, the files can be uploaded.

7.1.2 Program running

1. Press  to select the auto mode;
2. Press  to start the program, and the program auto runs and the cycle starting indicator is on; while running ends, the indicator is off. When the last block specifies M99, it can return to the beginning of the program to run the program in cycle after running ends.
3. To stop during running or cancel the memorizer running, there are following methods:

1) Stop the memorizer running



Press on the machine operation panel and its indicator is on, while the cycle start indicator is off. The machine responds as below:

- When the machine is moving, feed running decelerates till stopping.
- When pause (stopping) is being executed, the running stops.
- When M, S and T functions are executed, the running stops after completing M, S and T functions.



When feed hold indicator is on, press on the machine panel, the machine runs, again.

2) Stop the memorizer running

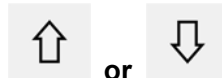


Press on MDI panel, auto running ends and resets.



Remark: The program runs from the line which the cursor is, before is pressed, check whether the cursor is on the block to run.

7.1.3 Running from any block



1. In the above figure 2, press to access the auto mode, press to access the program interface, press or to select the program content interface: press or , the cursor moves toward the block to run; or on the program interface shown as the above figure 1, press or to select the program to run, press to access the program editing interface, and then press or , the cursor moves toward the block to run, and then press , it returns to the position interface.

2. If the mode defaults (G, M, T and F commands) in the block which the cursor is, and the mode doesn't comply with that of the block, the next step can be operated only after the corresponding mode function is executed;




3. Press to access the auto mode, and press to start the program, the program begins executing from the selected block.

7.1.4 Stop auto running

The memorizer running can be stopped through the following methods: Command stopping or

press the relative keys, which are on the machine operation panel, to stop.


- Command stopping (M00, M01, M02 and M30)

After executing the block with M00 or M01 (the selecting stop button on the panel is on), running automatically stops, the mode function and the state all are saved. Press , the program continues to execute. When read in M02 or M30 (command at the end of the main program), the program running ends and resets.


The operations of different machines are not same; about the details, refer to the manual of the machine manufacturer.

- Pressing relatives keys to stop

1. During auto running, press  and the machine is shown as below:

- (1) Machine feeding decelerates till stopping;
- (2) The mode function and the state are saved;
- (3) Press , the program continues to execute.

2. Press 

- (1) All axes running decelerates till stopping;
- (2) M and S functions output invalid (After pressing , whether auto switch off signals of spindle CW/CCW, lubricating, cooling, etc is set by the parameter.)
- (3) After auto running ends, the mode function is hold.

3. Press emergence stop button

During machine running, in the dangerous or the emergency case, press the emergency stop button (the external emergency stop signal is valid), CNC accesses the emergency stop, then the machine running stops immediately, all output is off, such as the spindle revolving and the cooling fluid, etc. Press the emergency stop button, the emergency stop alarm clears, and CNC resets.

4. Switching the operation mode

During auto running, switch into the reference point return, MPG/single or manual, the current block “pause” at once; during auto running, switch into edit or MDI mode, the running stops after running the current block.

Remark 1: Confirm the trouble is shot before clearing the emergence stop alarm;

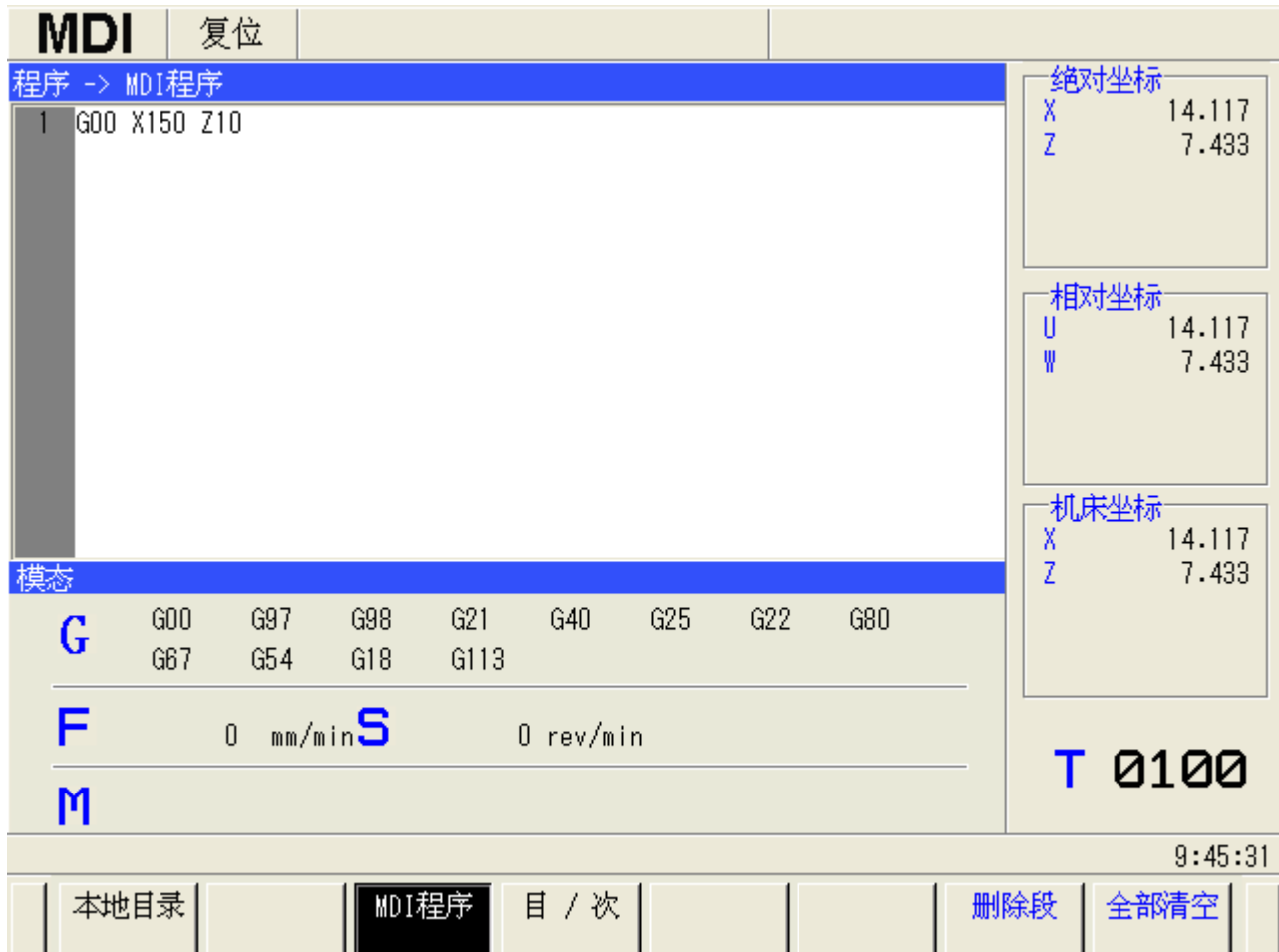
Remark 2: Before power on or shutdown, press emergency stop button to reduce the electric shock to the equipment;

Remark 3: After clearing the emergency stop alarm, return to the reference point, again to guarantee the correctness of the coordinate position;


7.2 MDI running



7.2.1 Editing and running the program in MDI mode

(1) the program interface, press  to access MDI mode, then the interface is shown as below:



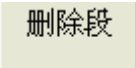
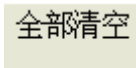
(2) The running block is input (maximum 10 lines) in the edit bar which is at the downside of the block (MDI); The editing method is similar with that of editing the common program. If the program is created in MDI mode, the characters can be rewritten and deleted. About editing the program, refer to chapter 5th.


(3) After the block is input, the cursor moves toward the beginning of the block, and executes. If the cursor is in somewhere of the program, the program is begun to execute. Press , MDI command characters are executed from the line which the cursor is. When the program end codes (M02 or M03) are executed, the program running ends rather than return to the beginning of the program. After running the program, the system accesses the stop mode.

(4) During running, press ,  or the emergency stop button to stop MDI command characters.

Remark:

1) Deleting the program

a. In MDI mode, press  to delete the block which the cursor is, press  to clear all the blocks in MDI edit bar.

b. When parameter MCL (NO.3203#7) is set as 1, press  and the program is auto cleared.

c. When parameter MER (NO.3203#6) is set as 1, in single block mode, after running the last block, the program is auto cleared.




2) Restart

When MDI running stops, after editing,  is pressed to run again, the running starts from the position where the cursor is.

3) The program which is created in MDI mode can't be saved.

4) In MDI mode, the subprogram and the macro program can't be called.

7.2.2 Running from any block


In the position interface, in MDI mode, press  or , the cursor moves toward the block to run, press  to start the program and the program begins executing from the block which the cursor is.

7.2.3 Stop MDI running

MDI running can be stopped through the following methods: Command stopping or press the relative keys on the machine panel to stop.

- **Command stopping (M00 or M01)**

After executing the block with M00 or M01 (the selecting stop button is on, which is on the machine operation panel), MDI running stops, the mode function and the state all are saved. Press

, the program continues to execute.


When read in M02 or M03, which commands at the end of the main program, the program running ends.

The operations of different machines are not same, about the details, refer to the manual of the machine manufacturer.

- **Press relative keys to stop**

1. During auto running, press  and the machine is shown as below:

- (1) Machine feeding decelerates till stopping;
- (2) During executing dwell command (G04 command), dwell when G04 is executed;
- (3) The mode function and the state are saved;

(4) After pressing , the program continues to execute.



2. Press

(1) All axes movement stops;



(2) M and S function output is invalid (After pressing , whether automatically switch off signals of spindle CW/CCW turning, lubricating or cooling, etc is set by parameter);

(3) Running ends and the mode function holds.

3. Press emergency stop button

During the machine running, in the dangerous or the emergency case, press emergency stop button (the external emergency stop signal is valid), CNC accesses the emergence stop, then the machine running stops immediately, all output (such as the spindle or the coolant revolving) are off. After releasing the emergency stop button, the emergency button alarm is cleared and CNC resets.

4. Switching modes

During MDI auto running, switch into the modes of the reference point return, MPG/single step and the manual, the current block “pause” immediately, and then access MDI mode, press “cycle start” to continue running; in MDI auto running, switch into the edit or the auto mode, and it stops after the current block running.

7.3 DNC running

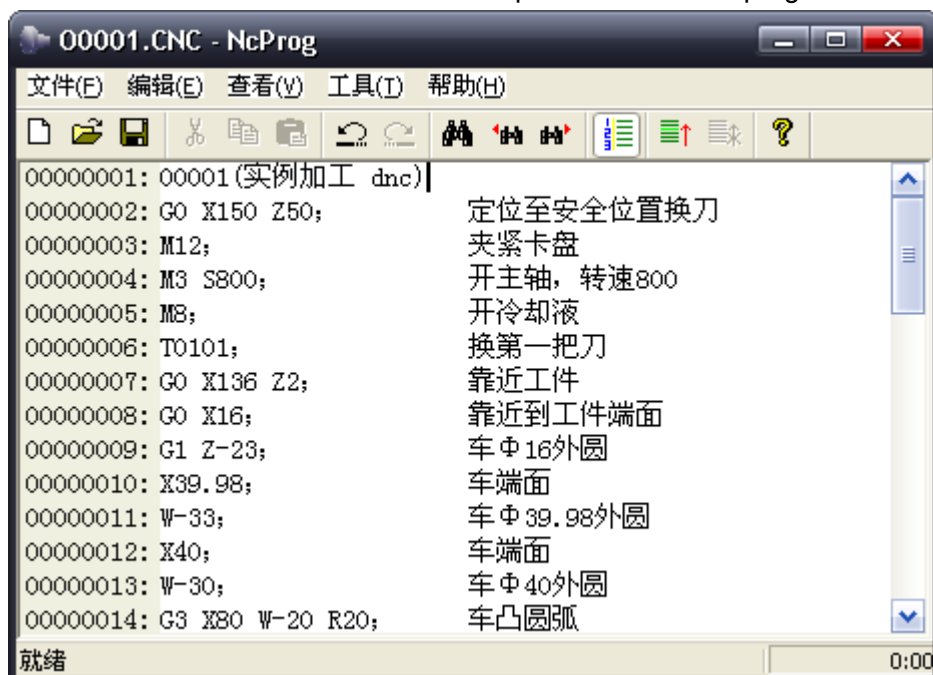
988T is equipped with DNC function, and DNC communication software is connected with CNC, then the program is running in high speed and large capacity.



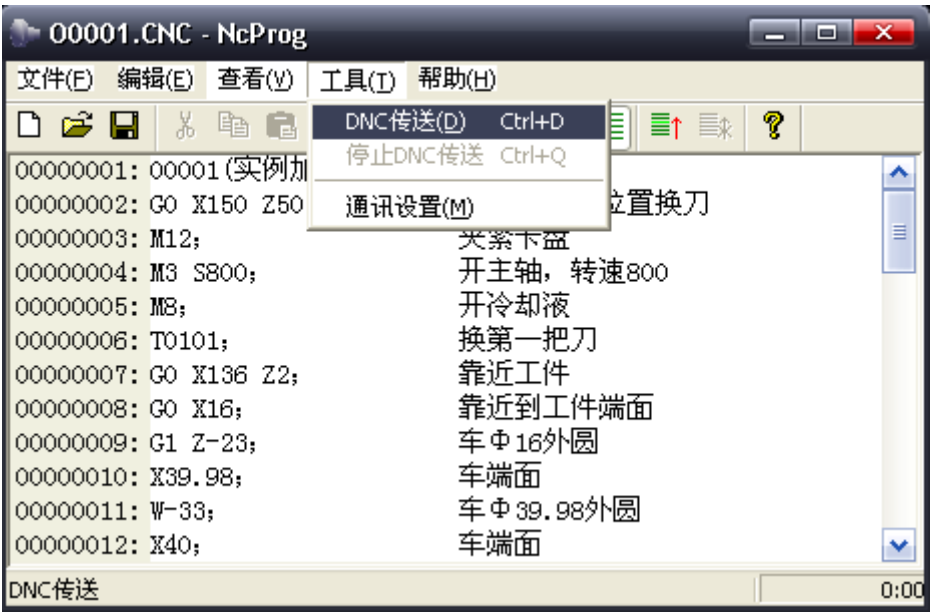
Press on the machine panel, access DNC mode, after PC is ready, press cycle start key and start the program for DNC processing.


About the detailed method, refer to the introduction of DNC communication software.

1. Communication software GSKComm selects and opens the machine program.




2. Connect CNC system



3. Press  to select DNC mode:




4. Press , the program automatically starts and the cycle start indicator is on. After automatic running ends, the cycle start indicator is off.

DNC		运行	G00快速定位中					
绝对坐标					工艺数据			
<div>X136.000mm</div> <div>Z16.171mm</div>					<div>T0101</div> <div>F401mm/min</div> <div>1000mm/min</div> <div>S585</div> <div>800rev/min</div>			
程序名 [DNC]					综合信息			
3	M12;	夹紧卡盘;			进给倍率	100%	手轮倍率	X1
4	M3 S800;	开主轴, 转速800;			快速倍率	100%	加工件数	0
5	M8;	开冷却液;			主轴倍率	80%	运行时间	00:00:02
6	T0101;	换第一把刀;			手动倍率	0%	切削时间	00:00:02
7	G0 X136 Z2;	靠近工件;						
8	N80 G0 X16 ;	靠近到工件端面;						
9	G1 Z-23;	车Φ16外圆;						
10	X39.98;	车端面						
10:15:01								
绝对坐标		相对坐标	机床坐标	综合坐标		模态		


5. Stop during running



Press , the feed hold indicator is on, while the cycle start indicator is off. The machine responds as below:

- When the machine is running, the feeding decelerates till stopping.
- When the pause (the tool stops running) is being executed, the running stops.
- When functions of M, S and T are executed, running stops after completing the functions of M, S and T.



When feed hold indication is on, press  on the machine operation panel, the machine runs again.

6. Running end



Press  on MDI panel or DNC program executes M30 command, reset after running ends.

DNC 复位			
绝对坐标		工艺数据	
X	150.000 mm	T	0100
Z	50.000 mm	F	0 mm/min 200 mm/min
		S	0 rev/min 100 rev/min
程序名 [DNC]		综合信息	
		进给倍率	100% 手轮倍率 X1
		快速倍率	100% 加工件数 1
		主轴倍率	80% 运行时间 00:03:48
		手动倍率	0% 切削时间 00:03:48
		10:26:10	
绝对坐标	相对坐标	机床坐标	综合坐标
			模态 相对坐标设置

Remark: In DNC program, the program calling and jumping commands can't be executed.

7.4 Auto running control


7.4.1 Machine and miscellaneous function lock

Use the machine lock and execute the machine program, but the machine remains still, only the tool position changing situation displays. All axes are locked, and the movement of all axes is stopped. Moreover, the locking miscellaneous function can lock the commands of M, S and T. Same as the machine lock, it's for checking the programs.

7.4.1.1 Machine lock

Execute the machine program, but the machine remains still, only the tool position changing situation displays, and then, the machine is locked to check the program. When the machine is locked, the movement of all axes is stopped.



Press  on the operation panel, the machine is still, but each axis position on the monitor is changing. About the machine lock, refer to the manual provided by the machine manufacturer.

Remark:

1) Position relation between the work piece coordinate system and the mechanical coordinate system may be different before or after automatically use the machine lock. Then, the coordinate sets


the commands or execute the manual reference point return to set the work piece coordinate system.

2) When the machine is locked and G28 or G30 command is sent, the command can be received rather than move to the reference point and the reference point return indicator is off.

7.4.1.2 Miscellaneous function lock

Locking the miscellaneous function can lock the commands of M, S and T. Same as the machine lock, it's for checking the program.




Press  on the machine operation panel, when M, S and T codes are invalid, they can't be executed. About the miscellaneous function lock, refer to the manual provided by the machine manufacturer.

Remark:

- 1) When the machine is locked, M, S and T commands can still be executed;
- 2) Even the miscellaneous function is locked, commands of M00, M01, M02, M30, M98 and M99 (subprogram calling function) can be executed.


7.4.2 Dry running



Press  on the operation panel, the machine moves at the speed set by the parameter without considering the feed rate specified in the program, which can check the machine movement which the work piece unloads from the working table.

Steps of dry running:



During automatically running, press  on the machine operation panel, the machine moves at the feed rate set by the parameter, and the rapid movement switch can change the feed rate. About the details of dry running refer to the manual provided by the machine manufacturer.

According to the rapid movement switch and the parameter, the dry running speed change is shown as below:

Rapid movement button	Program commands	
	Rapid movement	Feeding
ON	Rapid movement speed	Dry running speed *JVmax
OFF	Rapid movement speed	Dry running speed *JV

JVmax: The maximum graduation value of the feed rate override

JV: The graduation value of the feed rate override

Remark:


1. The maximum cutting feed rate is set by parameter #1422;
2. The rapid movement speed is set by parameter #1420;
3. The dry running speed is set by parameter #1410.

7.4.3 Single block running

When execute the program at the first time, select the single block running to prevent the malfunction due to the programming mistakes.

In auto mode, the method of opening the single block switch is as below:




Press  to start the single block mode and the single block indicator is on. In single block





mode, press  to execute one block, and then the machine stops; continue to execute the next



block, press  again, repeatedly, until the program running ends. In the single block mode, check the program through executing the blocks one by one.

Steps of the single block running:



1. Press  on the machine operation panel, press  to execute one block in the program. After executing the current block, the machine stops;

2. Press the cycle start button and execute the next block; after executing the block, the machine stops.



Remark:

- 1、Reference point return and single block running: if commands of G28 and G29 are sent, the single block function in the intermediate point is valid.
- 2、Subprogram block and single block running: with M98P_ or M99, or in G65 block, the single block stops.
- 3、About executing the fixed cycle and multiply cycle in the single block mode, refer to the relative content in the command manual.

7.4.4 Feed rate override

The feed rate of programming can be decreased or increased through selecting the percent (%) on the override dial, which is for checking the program. For example, the machine move at 50mm/min when the specified feed rate is 100mm/min in the program and the override is set as 50%.

The steps of changing the feed rate override: before automatical running or during running, the feed rate override dial can be set as the expected percent (%).



Feed rate override button

The override can be specified from 0 to 150%. For some machine, the range is stipulated in the manual.

Override of the thread cutting: During the thread cutting, the override is invalid but the feed rate

specified by the program is still valid.

7.4.5 Rapid movement override

For the rapid movement speed, there are four overrides (F0, 25%, 50% and 100%). The rapid movement speed of each axis is set by parameter #1420; F0 is set by parameter #1421.

The step of changing the rapid movement override: During the rapid movement, select one

override through pressing .

The following types of the rapid movement are valid and the rapid movement override can apply to them:

1. G00 rapid movement
2. Rapid movement during the fixed cycle
3. Manual rapid movement
4. Rapid movement during the manual reference point return
5. Rapid movement during G28 and G30

Chapter VIII Setting and display graphs

8.1 Setting the graph parameter

Before display the path, the relative information of the path display or the graphic simulation must be set.

Chapter VIII Setting and display graphs

The graph information mainly sets the offset value of each coordinate axis, the length and the diameter of the processing work piece, the magnification ratio of the graph path and that of the graph simulation. The detailed steps are as below:

(1) Press **GRAPH** to access the graph interface;

(2) On the graph interface, press **图形设置** to access the setting graph parameter interface and it is shown as below:

自动 复位

图形 -> 图形设置

轨迹设置		仿真设置	
横轴选择	Z	横轴选择	Z
纵轴选择	X	纵轴选择	X
放大比例	2	放大比例	1
Z轴偏移(mm)	-100.000	工件长度(mm)	100
X轴偏移(mm)	0.000	工件直径(mm)	100
		Z轴偏移(mm)	30.000
		X轴偏移(mm)	60.000

绝对坐标

X 7.093

Z 3.547

相对坐标

U 7.093

W 3.547

机床坐标

X 7.093

Z 3.547

T 0100

19:28:53

图形设置 轨迹显示 图形仿真

(3) Press **↑** or **↓** to select the item to be set, such as the cross axis shown as below:

(4) Press **INPUT** and the selected item can be input, such as the cross axis shown as below:

自动 复位

图形 -> 图形设置

轨迹设置		仿真设置	
横轴选择	Z	横轴选择	Z
纵轴选择	X	纵轴选择	X
放大比例	2	放大比例	1
Z轴偏移(mm)	-100.000	工件长度(mm)	100
X轴偏移(mm)	0.000	工件直径(mm)	100
		Z轴偏移(mm)	30.000
		X轴偏移(mm)	60.000

绝对坐标

X 0.000

Z 0.000

相对坐标

U 0.000

W 0.000

机床坐标

X 0.000

Z 0.000

T 0100

10:41:46

图形设置 轨迹显示 图形仿真

(5) Press **↑** or **↓** to select the item to be set and press **INPUT** to confirm the rewriting is completed.

(6) Repeat the above operation to set the other parameter.

Remark: The setting on the interface is only for the path display and the display on the graph simulation interface.

8.2 Path graph display and operation

Through the graph path display, real-time check the path which the tool traverses.

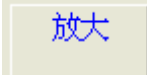
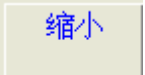
(1) Press **GRAPH** to access the graph interface;

(2) On the graph interface, press **轨迹显示** to access the path interface, display the program path which is being executed and it is shown as below:

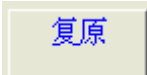






At the bottom of the path screen in the figure, it displays the coordinate plane of the current path and the magnification ratio of the path graph. On the top of the figure, it displays the running mode and the state of the current system. On the right of the screen, it displays the current absolute coordinate value, the relative coordinate value and the mode command.

The path graph can be operated as below:

(1) Press  or , the path graph can be zoomed out or in, and the previous ones can be cleared.

(2) Press  to clear the screen path.


(3) Press  and the path graph can be restored as the original normal position and the previous ones can be cleared.

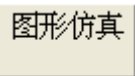
(4) Repectively press , ,  or  to move the path graph up, down, left or right.

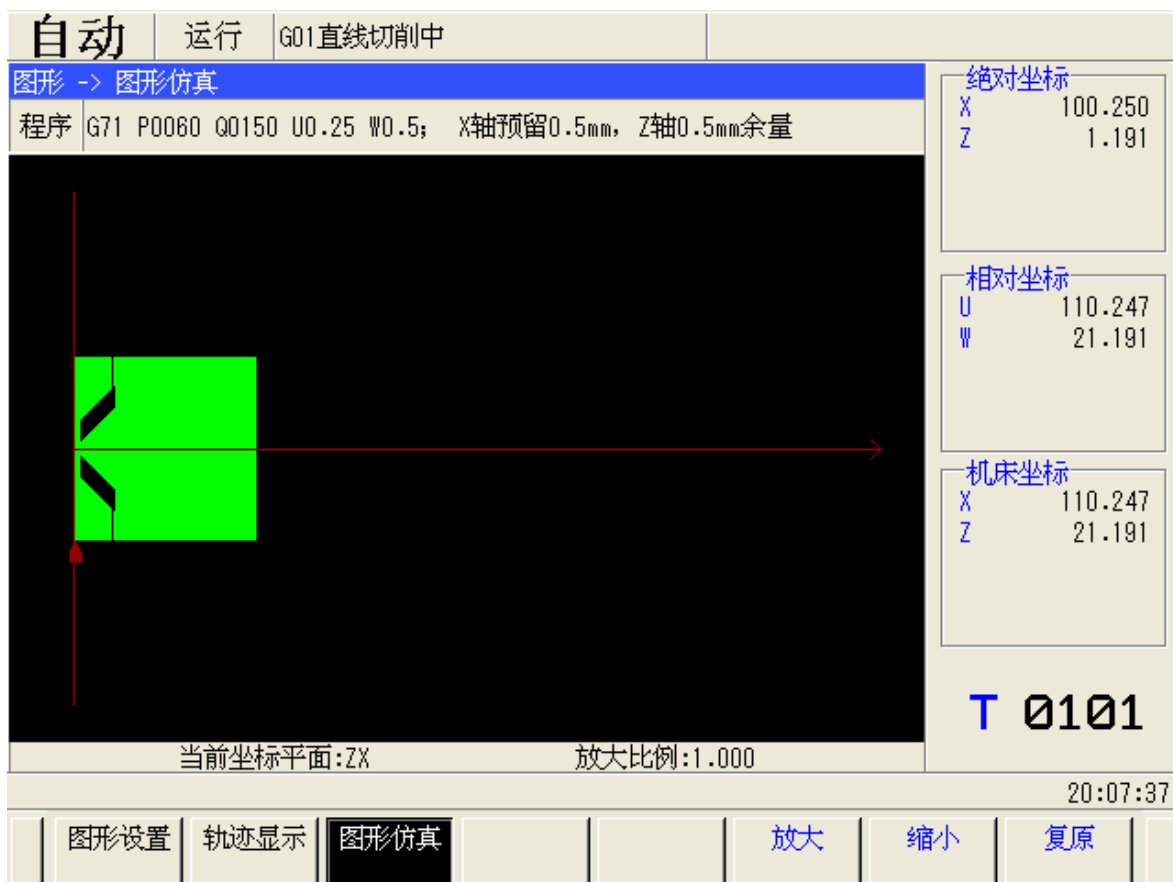
Remark: The name of each axis can be set by parameter #1020, and the name can be set as different letters, and then, at the bottom of the path interface, name of each coordinate plane and that of the path coordinate change correspondingly.

8.3 Simulation graph display and operation

Through the graph simulation, real-time check the complete cutting process of the part.

(1) Press  to access the graph interface;

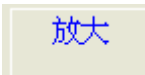
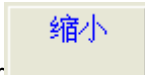
(2) On the graph interface, press  to access the simulation graph interface and it is shown as below:








On the top of the figure, it displays the running mode and the state of the current system; on the right of the screen, it displays the information of the current absolute coordinate value, the relative coordinate value and the current tool number, etc.

In the figure, it only displays the simulation graph information of XZ coordinate plane; at the bottom of the graph simulation screen, it displays the coordinate plane which the current simulation graph is, and the magnification ratio of the simulation graph.

During the graph simulation process, the simulation graph can be operated as below:

(1) Press  or , the simulation graph can be zoomed in or out, and the previous simulation graph information can be cleared;

(2) Press  and the simulation graph can be restored as the original size and position, and the previous simulation graph information can be cleared.

(3) Respectively press , ,  or , and the simulation graph can move up, down, left or right.

Remark: The name of each axis is set by parameter #1020, and each axis name can be set as the different letters, and then, at the bottom of the path interface, the coordinate plane and the path coordinate names can change correspondingly.

Chapter IX Communication

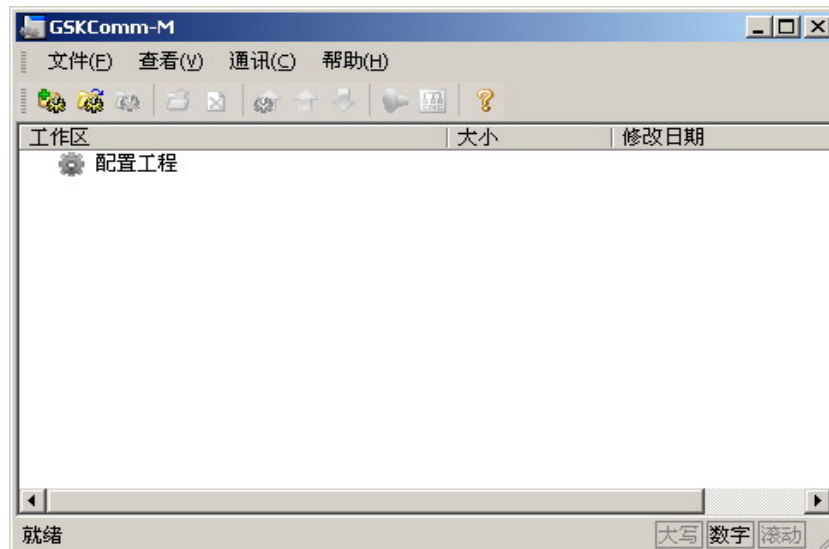
9.1 Brief introduction of GSK988T communication software GSKComm

GSKComm especially provides the user the communication management software. It can realize the uploading and downloading between PC and CNC; DNC communication; editing CNC parameter, managing the part programs; checking the tool compensation and the thread compensation and editing the ladder diagram, etc. And it is with the characteristics of the simple operation, high communication efficiency and reliability.

- Requirements of GSKComm system (PC)
Hardware: Common PC with RS232 serial port or network port, serial port communication cable (three-wire)
Operating system: Microsoft Windows 98/2000/XP/2003
- Requirements of GSKComm software
Firstly install PLC editing software GSKLadder.
- Difference between GSKComm-M and GSKComm-U
GSKCOMM-M communication software especially provides for the machine manufacturer.
GSKCOMM-U communication software especially provides for the user.

Function	GSKComm-U	GSKComm-M
Transmitting and editing the part program	OK	OK
DNC communication	OK	OK
Managing the part program	OK	OK
Transmitting the system files (the tool compensation, the thread compensation, the parameter, PLC program)	OK	OK
Checking the tool compensation and the thread compensation	NO	OK
Editing the parameter	NO	OK
Editing PLC program	NO	OK

- Software interface
The software interface of GSKComm is very simple, the interface after the software running is shown as below:



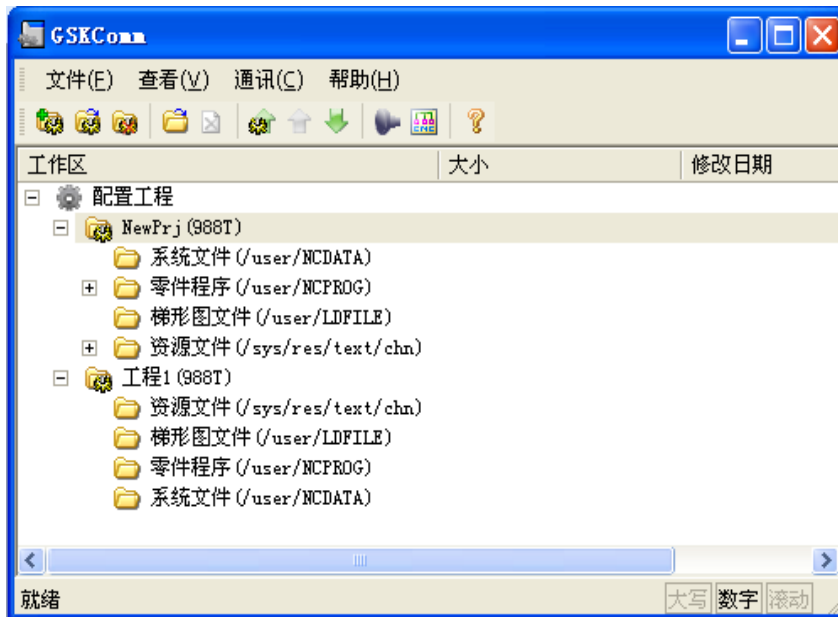
9.2 Creating, channeling and removing the project

9.2.1 Creating the project

Click  or select "file—> creating project" to create a project.



Because many projects can be created in GSKComm, each project can be differed and marked through "label".



If the file name exists, rewrite in “saved as” box. Click the right key of the box to select the path to save the project file.

9.2.2 Channeling the project

If the project file exists, it doesn't require creating a project.


Click  or select “file—>channeling the project” to channel one project.

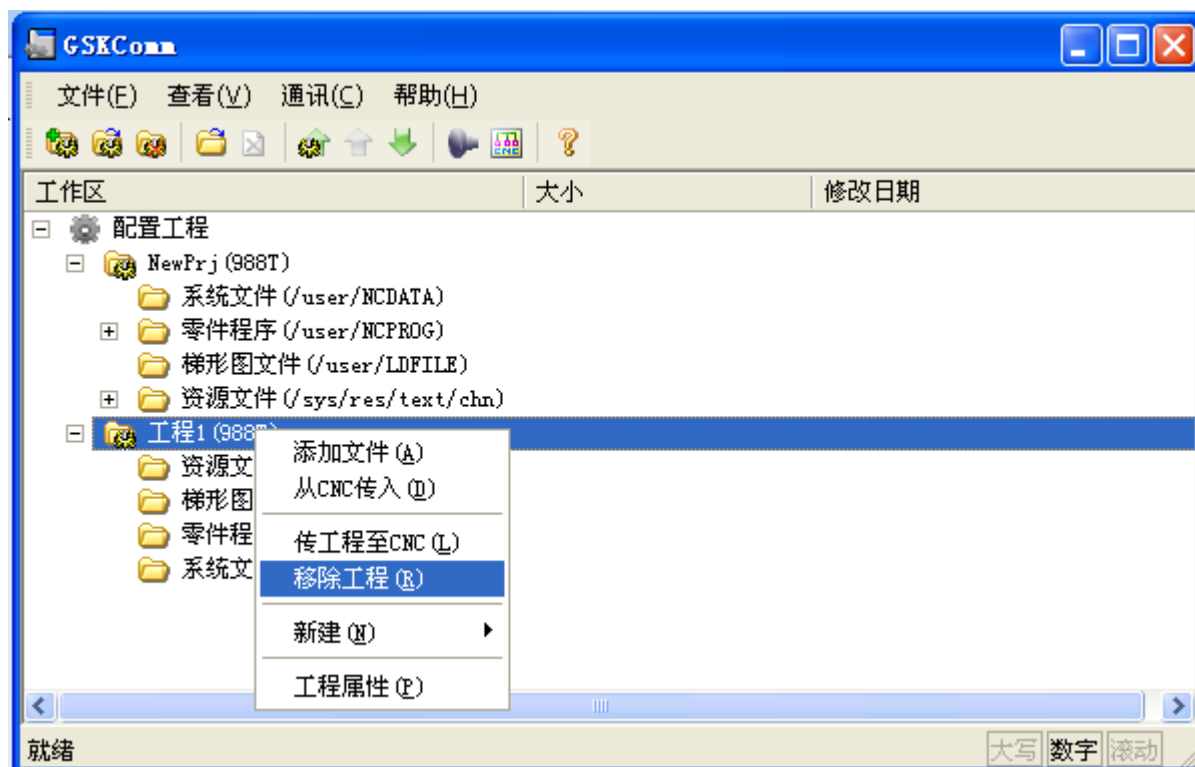


9.2.3 Removing the project

If it doesn't require editing some project in GSKComm, select the project and remove it from GSKComm.

Firstly, select the project to remove;

And then, click  or the right key to select “removing the project”, and then the project can be removed from GSKComm.



Remark: Removing the project doesn't mean deleting it. The removed project still exists, and it can be added into GSKComm through "channeling the project".

9.3 Creating, channeling, removing and editing the file

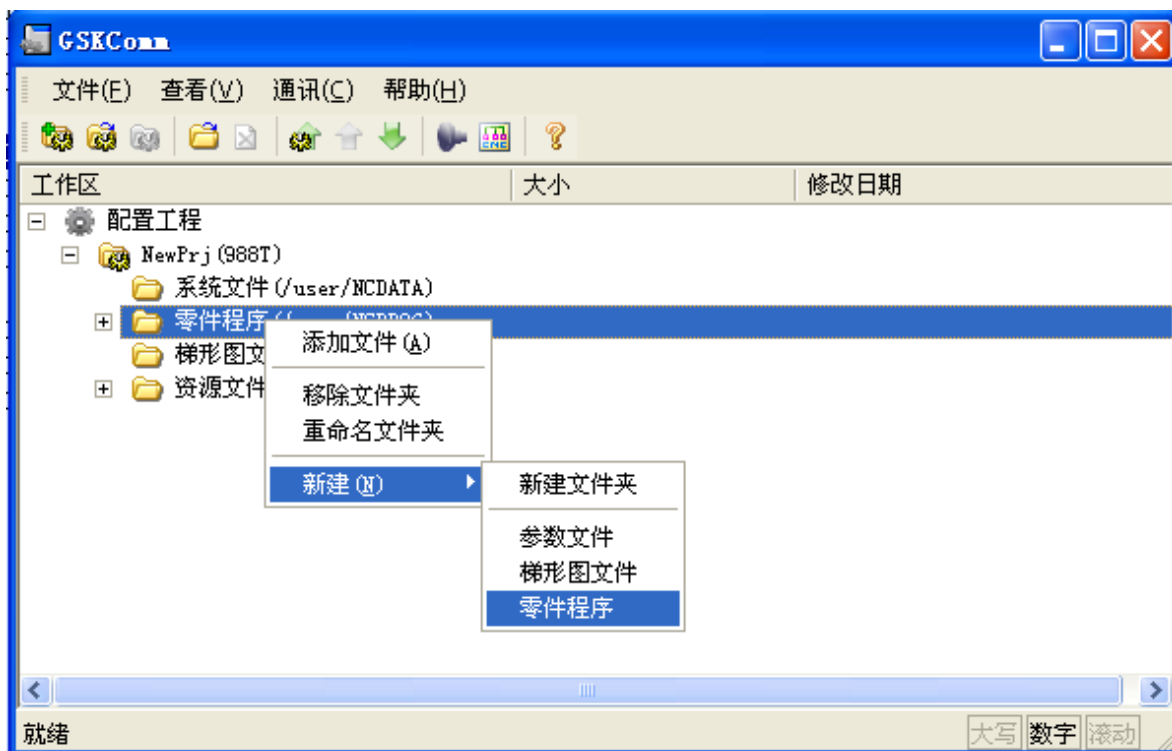
Create and edit the part program, the parameter file and PLC program in the project, and the existed file can be channeled or removed.

9.3.1 Creating the file

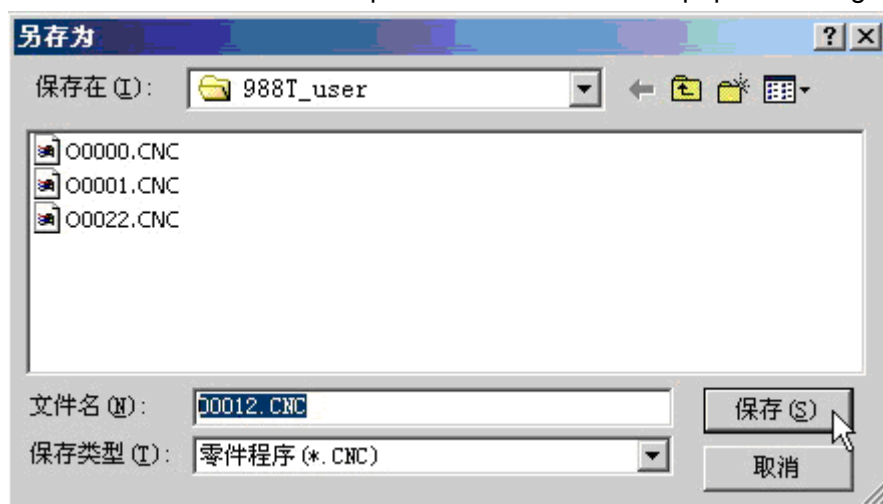
In the project, create the part program, the parameter file and PLC program. Creating the part program can be taken as an example to introduce the process of creating the file, which is shown as below:

Firstly, select the corresponding project.

And then, click the right key, select "creating—> the part program", a part program is created.

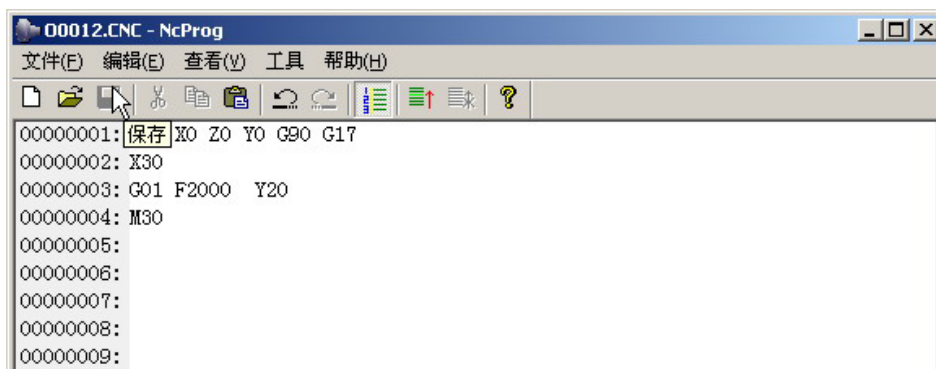


The file name can be rewritten and the path can be saved in the popped message box.

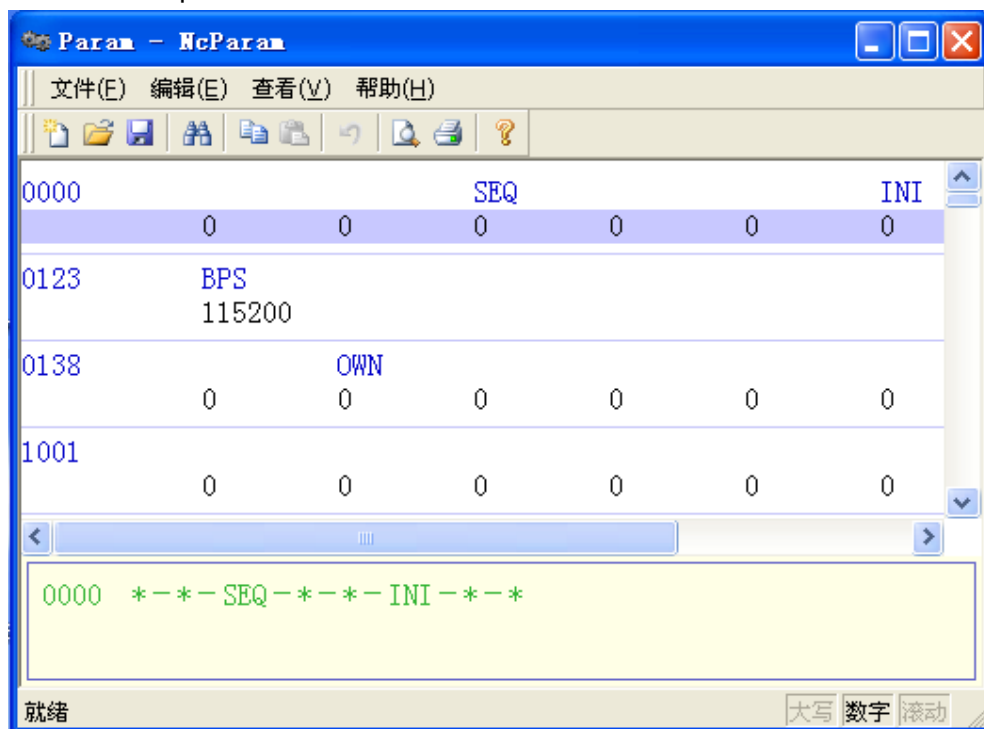


9.3.2 Editing the file

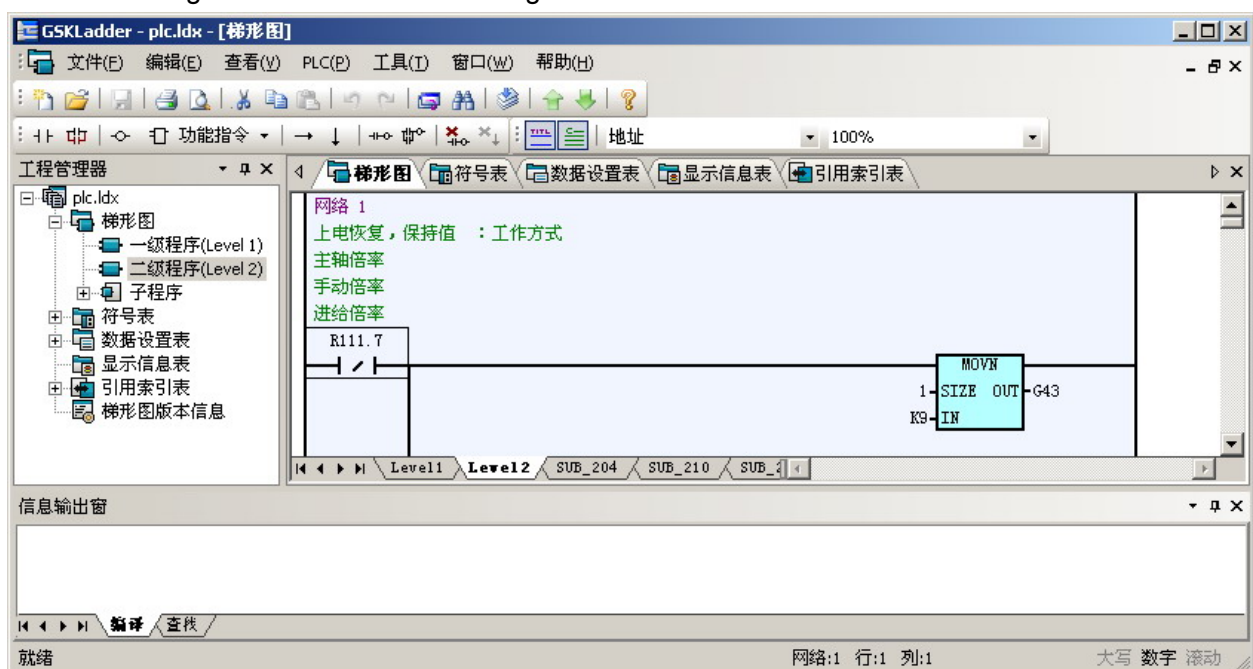
As long as double click the part program, the parameter file and PLC program to be edited, the editing interface corresponding files pops up. The editing interface of the part program is shown as below:



The editing interface of the parameter file:




The editing interface of the ladder diagram:

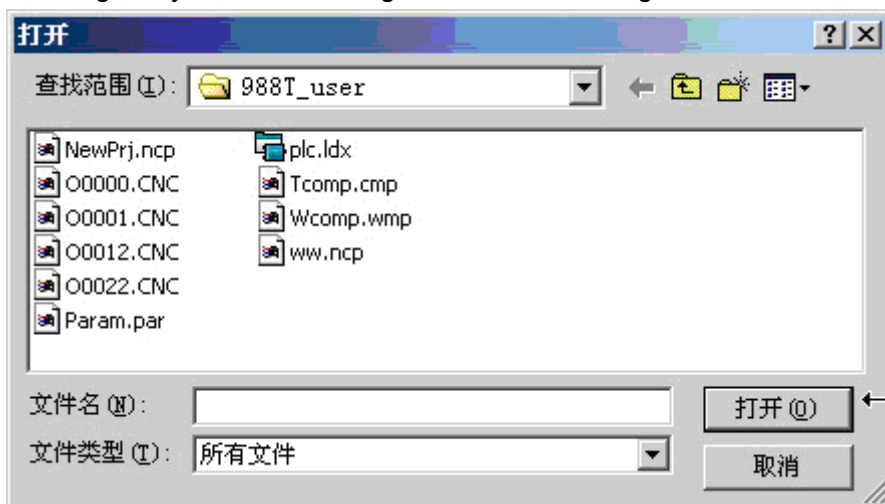


On the editing interface, the corresponding files can be edited and rewritten. After editing, click “save” to save the current file.

9.3.3 Adding the file


If the existed file is edited, the file should be added into the project previously.

Click  or click the right key to select “adding the file”, the message box to add the file can pop up.



In the message box, select the file to be added.

9.3.4 Removing the file

If some file is useless, it can be removed from the project. Firstly select the file to remove, and then click  or click the right key to select “removing the file”, and then it can be removed from the project.


Remark: Removing the file doesn't mean deleting it. The removed file still exists and it can be added into GSKComm through “Adding the file”.

9.4 Downloading the file (PC→CNC)

GSKComm can transmit all the files or one file in the project to CNC. Before downloading the files, CNC should be set as the following list; otherwise, the file can't be downloaded.

Data downloading from PC	CNC authority (minimum)	Remark
Part program	Level 3	The program switch is on
Macro variable	Level 4	The program switch is on
Tool offset	Level 4	
Parameter	Level 3	The parameter switch is on
Thread data	Level 5	The parameter switch is off
PLC files	Level 2	
The tool life files	Level 5	

9.4.1 Downloading the file

Firstly, select the project to transmit; and then, click  or click the right key, select “transmitting the project to CNC” and the message box of transmitting the file to CNC pops up.




In the message box, click the item which is in the left of the file name to select the file to be transmitted.

File name “->” means the file name which can be saved in CNC. Double click to rewrite the saved file name.



Click “start sending”, and all the selected files in the saved names can be sent to CNC.

9.4.2 Downloading the single file


Select the file to be downloaded, and then click  or the right key to select “transmitting the file to CNC”, the message box pops up:

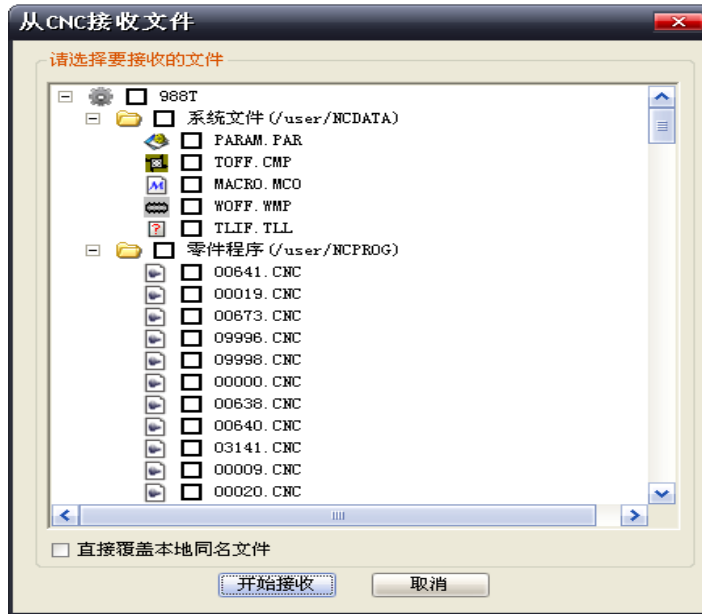


The saved file name in CNC can be rewritten. Click “confirm”, the file can be sent to CNC.

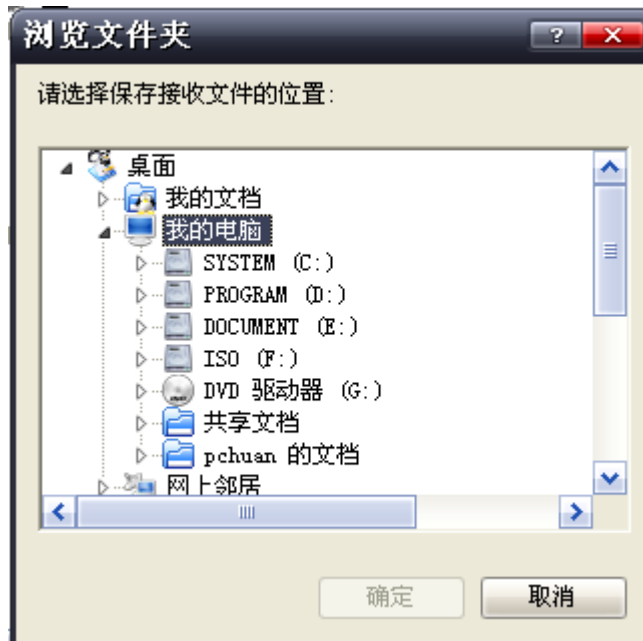
9.4.3 Uploading the file (CNC→PC)

Firstly, select a project.

And then, click  or select “communication—>receive the file from CNC”, the message box of “receive the file from CNC” pops up:




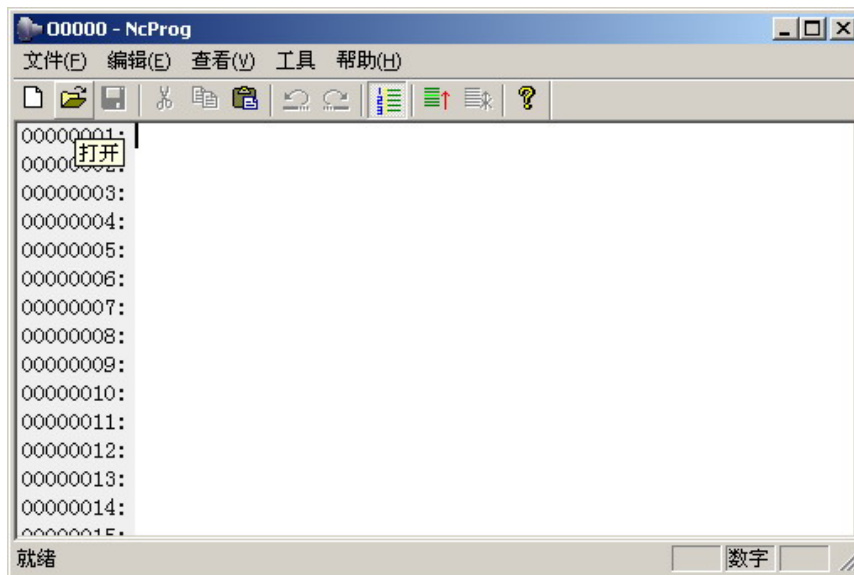
Select the file to be uploaded, and then, click “begin receiving” and the message box of “browse the files” pops up:



Select the folder where the uploaded file is saved.
Click “confirm” to upload the selected file from CNC.


9.5 DNC transmitting

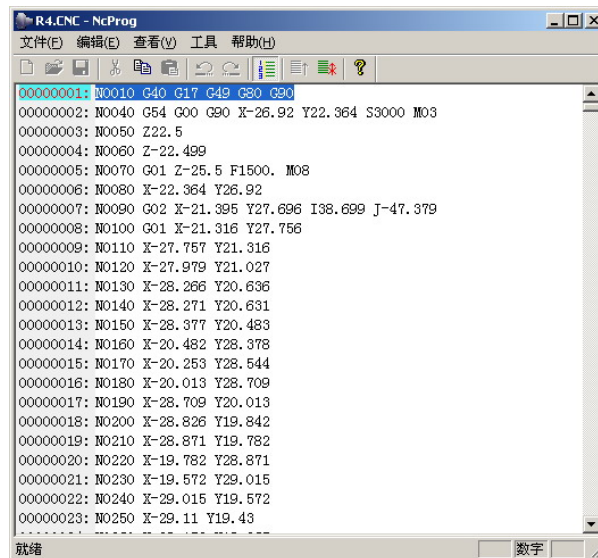
Click  or select “communication—>DNC communicaton”, DNC communication interface pops up:



Click “open” to select the part program to be processed.




And then, move the cursor to the initial line of the program, and then click  or select “the tool —> DNC transmitting”, and then it is ready for DNC transmitting;

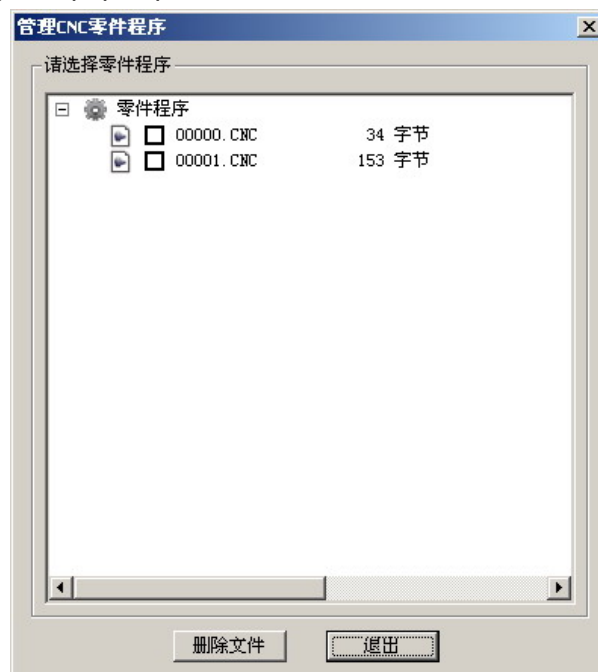


Finally, switch CNC into DNC mode, and then press “cycle start” button, DNC communication processing can be operated.

9.6 Managing the part program

Managing CNC part program is to check CNC current part program list and delete the part program.

Click  or select “communication —> managing the file program” and the interface of “managing CNC part program” pops up.



List all current CNC part programs on the interface. The user selects the corresponding program and clicks “delete the file”, and it can be deleted.

9.7 Preparation before communication

1. When PC and CNC power off, connect the communication cable;

Connection between PC and CNC: DB9 pin plug can insert RS-232 communication interface of

CNC, DB9 hole plug can insert the serial port of 9 pins in PC (COM0 or COM1);

1. Setting the communication baud rate

● Setting the baud rate in CNC

GSK988T CNC serial port communication baud rate is set by data parameter No.0123, if transmit the data between CNC and PC, the value should be set more than 4800. The standard setting before leaving the factory: 115200

● Setting PC baud rate:

After the communication software running, click the menu through the left key of the mouse, select “communication—>communication setting” and the interface is shown as below:



Selecting the port: Select the port (COM1, COM2, COM3 or COM4) for communication

Baud rate: Select the communication baud rate (4800, 9600, 19200, 38400, 57600 or 115200 (unit: bps))

Chapter X Processing examples

Process the work piece shown as below and the bar stock dimension is $\Phi 136 \times 180 \text{ mm}$.

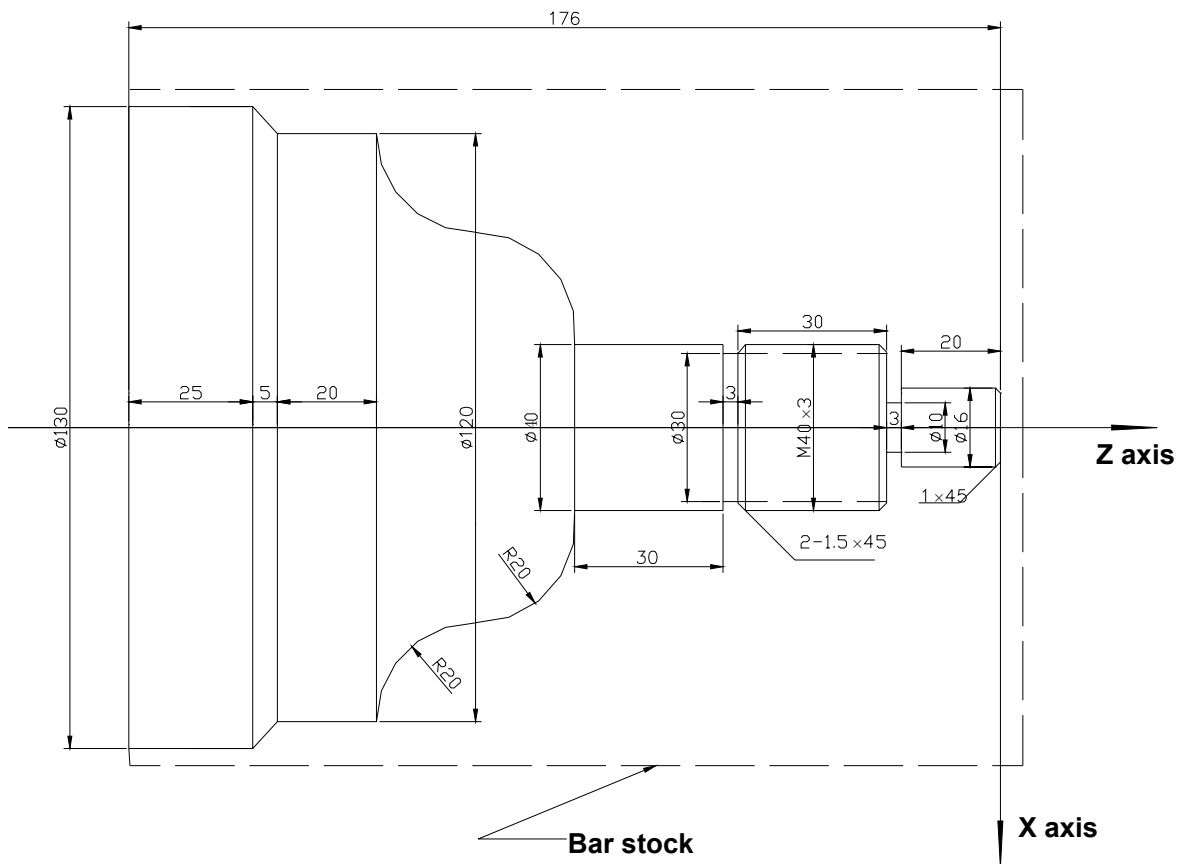


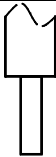



Fig. 12-1

Process in four different tools and the details are as below:

TOOL NO	TOOL TYPE	REMARK
#1 tool		Outer circular roughing tool
#2 tool		Outer circular turning tool
#3 tool		Grooving tool, its width is 3mm
#4 tool		Thread tool, the nose angle is 60°

10.1 Editing the program

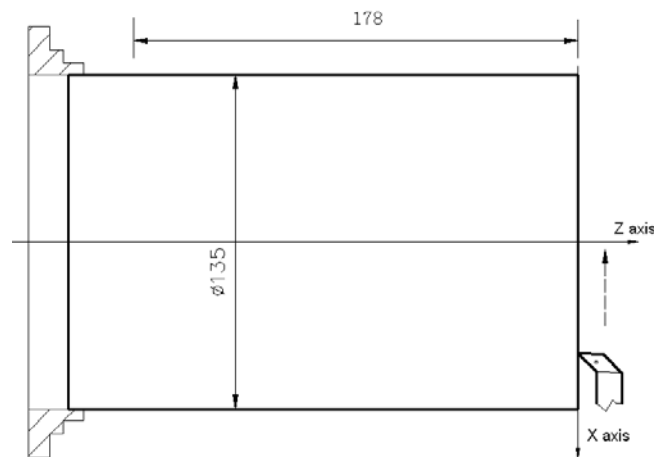
According to the mechanical processing and introduction of the commands in the manual, set the work piece coordinate system shown as fig 12-1; edit the programs shown as below:

O 0 0 0 1 ;		Part program name
N 0 0 0 0	G0 X150 Z50;	Position to the safe place to change the tool
N 0 0 0 5	M12;	Clamp the chuck
N 0 0 1 0	M3 S800;	The spindle is on, and its speed is 800
N 0 0 2 0	M8;	The cooling is on
N 0 0 3 0	T0101;	Change into the 1 st tool
N 0 0 4 0	G0 X136 Z2;	Close to the work piece
N 0 0 5 0	G71 U0.5 R0.5 F200;	Cutting depth is 1mm, the tool retracts for 1mm.
N 0 0 5 5	G71 P0060 Q0150 U0.25 W0.5;	X axis leaves for 0.5mm, 0.5mm surplus in Z axis
N 0 0 6 0	G0 X16;	Close to the work piece face
N 0 0 7 0	G1 Z-23;	Turning $\Phi 16$ outer circle
N 0 0 8 0	X39.98;	Turning face
N 0 0 9 0	W-33;	Turning $\Phi 39.98$ outer circle
N 0 1 0 0	X40;	Turning face
N 0 1 0 5	W-30;	Turning $\Phi 40$ outer circle
N 0 1 1 0	G3 X80 W-20 R20;	Turning convexo arc
N 0 1 2 0	G2 X120 W-20 R20;	Turning concave arc
N 0 1 3 0	G1 W-20;	Turning $\Phi 120$ outer circle
N 0 1 4 0	G1 X130 W-5;	Taper turning angle
N 0 1 5 0	G1 W-25;	Turning $\Phi 130$ outer circle
N 0 1 6 0	G0 X150 Z185;	Return to the tool-change point after roughing
N 0 1 7 0	T0202;	Change into #2 tool, execute #2 tool offset
N 0 1 8 0	G70 P0060 Q0150;	Finishing cycle
N 0 1 9 0	G0 X150 Z185;	Return to the tool-change point after roughing
N 0 2 0 0	T0303;	Change into #3 tool, execute #3 tool offset
N 0 2 1 0	G0 Z-56 X42;	Close to the work piece
N 0 2 2 0	G1 X30 F100;	Grooving $\Phi 30$
N 0 2 3 0	G1 X37 F300;	Return
N 0 2 4 0	G1 X40 W1.5;	Chamfering
N 0 2 5 0	G0 X42 W30;	Leave the grooving tool width
N 0 2 6 0	G1 X40 ;	

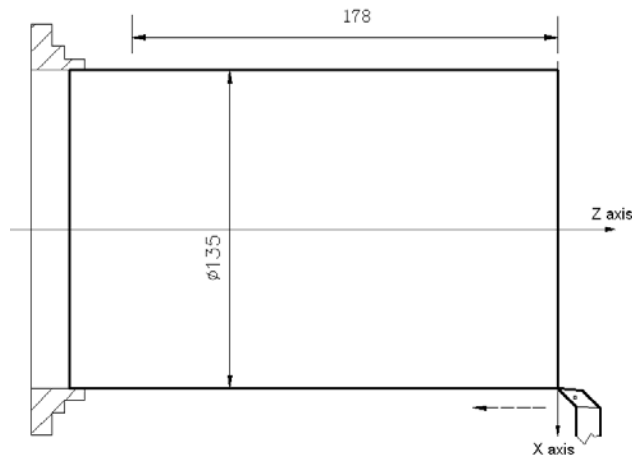
N 0 2 6 2	G1 X37 W1.5;	Chamfering
N 0 2 6 4	G1 X10;	Grooving $\Phi 10$
N 0 2 6 6	G0 X17 Z-1;	
N 0 2 6 8	G1 X16;	
N 0 2 7 0	G1 X14 Z0 F200;	Chamfering
N 0 2 8 0	G0 X150 Z50;	Return to the tool-change point
N 0 2 9 0	T0404 S100;	Change into #4 tool, set the spindle speed as 200 r/min.
N 0 3 0 0	G0 X42 Z-54;	Close to the work piece
N 0 3 1 0	G92 X39 W-34 F3;	Threading cycle
N 0 3 2 0	X38;	Feed 1mm and cut the 2 nd time
N 0 3 3 0	X36.4;	Feed 0.6mm and cut the 3 rd time
N 0 3 3 2	X36;	Feed 0.4mm and cut the 4 th time
N 0 3 4 0	G0 X150 Z50;	Return to the tool-change point
N 0 3 5 0	T0100;	Change into #1 tool
N 0 3 6 0	M5;	The spindle is off
N 0 3 7 0	M9;	The cooling is off
N 0 3 8 0	M13;	Release the chuck
N 0 3 9 0	M30;	Program end

10.2 Setting tool and running

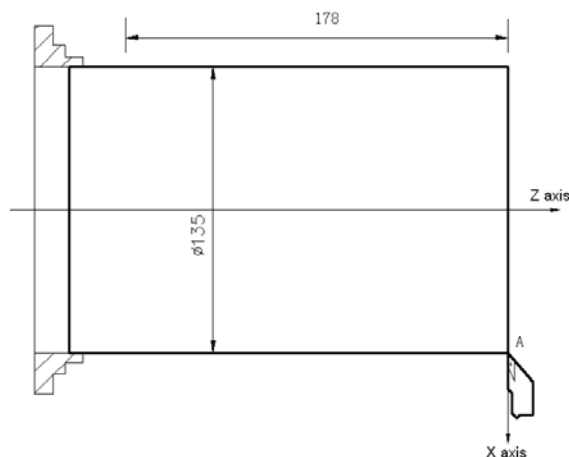
1. The tool traverses to the safe position; in MDI mode, execute T0100 U0 W0 and cancel the tool offset on the program interface;
2. The tool traverses and cuts along the work piece face;



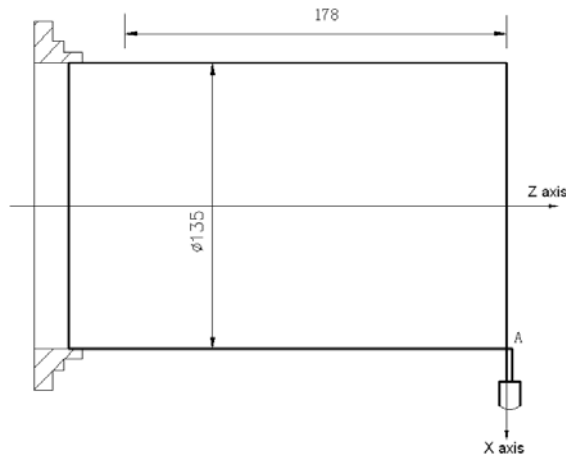
3. When Z axis remains still, release the tool along X axis, and stop the spindle revolving, in MDI mode, execute G50 Z0 and set Z axis coordinate on the program interface,;
4. Switch into the tool offset interface, input Z0 in #001 offset;
5. The tool traverses and cuts along the outer circle of the work piece;



6. When X axis remains still, release the tool along Z axis and stop the spindle revolving, measure the outer dimension of the work piece (for example, the measured value is 135mm);
7. In MDI mode, execute G50 X135 and set X axis coordinate on the program interface;
8. Switch into the tool offset interface, input X135 in #001 offset;
9. The tool traverses to the safe position, press tool-change key to execute the 2nd tool in manual mode;
10. Start the spindle and the tool traverses to the tool-setting position, such as point A shown as below:



11. Switch into the tool offset interface, the cursor moves to #002 offset, press 测量输入 to access the measuring interface, and input X135 in 测量输入, and then press 确定. Use the same method to input **Z0**;
12. The tool traverses to the safe position, in manual mode, press tool-change key to execute #3 tool;
13. Start the spindle, and the tool traverses to the tool-setting position, such as point A shown as below;

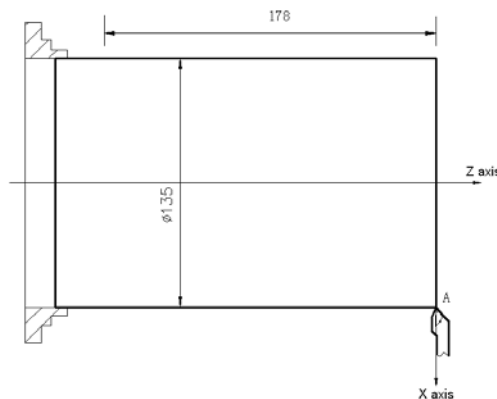


14. Switch into the tool offset interface, move the cursor to #003 offset, input X135 and Z0, the input

operation is same as above step 11;

15. The tool traverses to the safe position, press tool-change key to execute #4 tool in the manual mode;

16. The tool traverses to the tool-setting point, such as point A shown as below;



17. Switch into the tool offset interface, move the cursor to #004 offset, input X135 and Z0, the operation is same as step 11.

18. After tool-setting, the tool traverses to the safe position;

19. In auto mode, press  to automatically process;

20. Measure the work piece dimension, if there is offset between the actual part dimension and the work piece dimension, the tool wearing value can be rewritten until the part dimension is in the tolerance.

VOLUME III

CONNECTION

Chapter I Installation layout

1.1 GSK988T connection

1.1.1 GSK988T back cover interface layout

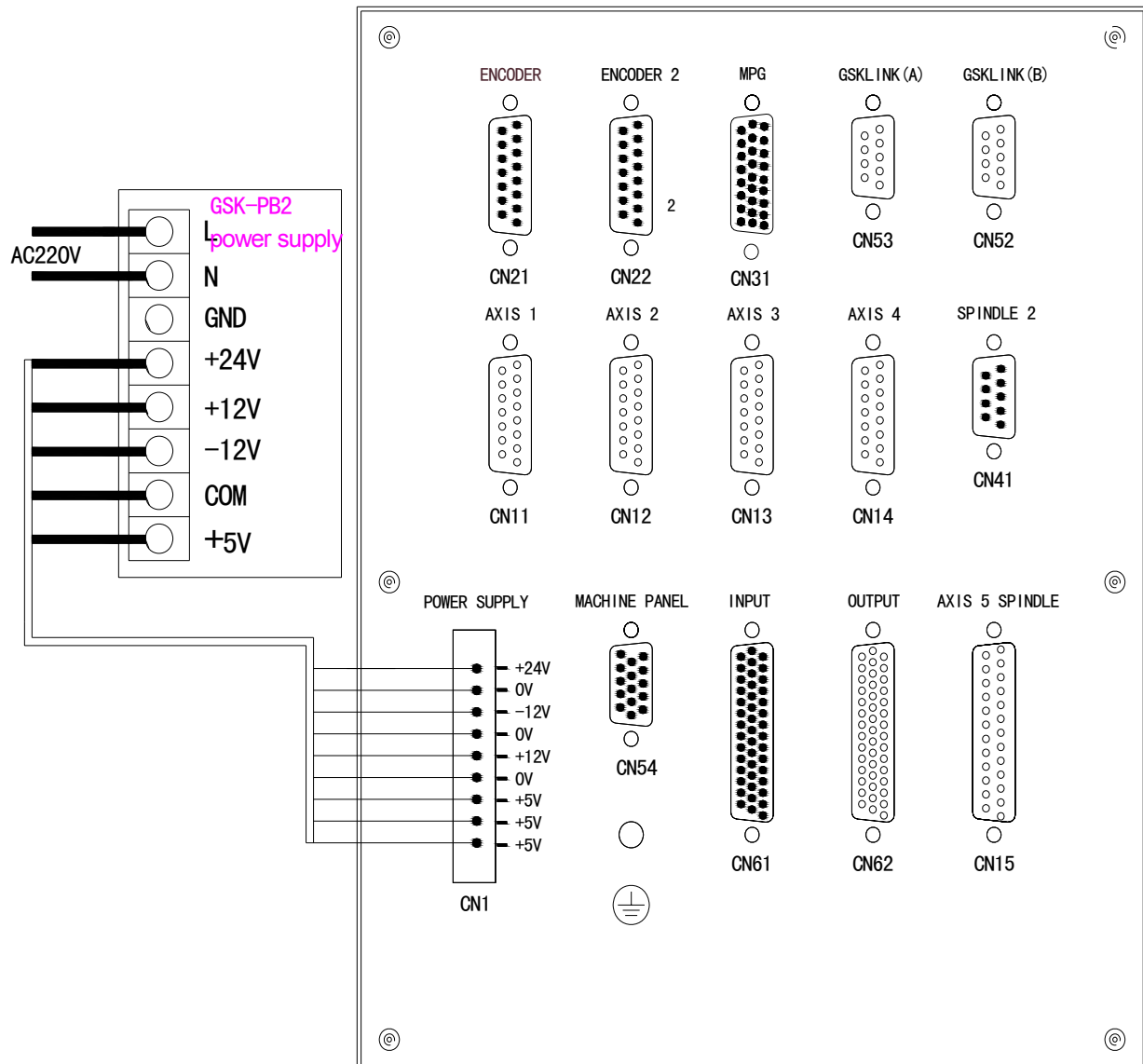


Fig. 1-1 GSK988T back cover interface layout

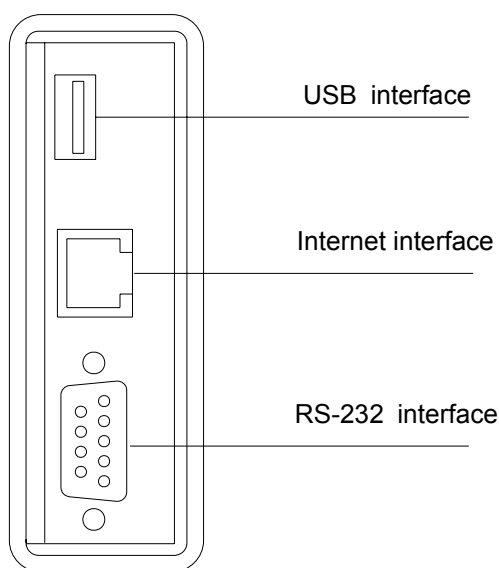


Fig. 1-2 GSK988T front cover interface layout

1.1.2 Interface Introduction

Back cover interface:

- Power supply box: GSK-PB2 power supply box, providing +5V, +24V, +12V, -12V and 0V power supply
- CN11: the first axis interface, 15 cords, pin socket of type D, connect with the first servo axis drive;
- CN12: the second axis interface, 15 cords, pin socket of type D, connect with the second servo axis drive;
- CN13: the third axis interface, 15 cords, pin socket of type D, connect with the third servo axis drive;
- CN14: the forth axis interface, 15 cords, pin socket of type D, connect with the forth servo axis drive;
- CN15: the fifth axis interface · spindle interface, 25 cords, pin socket of type D, connect with the spindle servo drive unit/spindle transducer/the fifth servo axis drive;
- CN41: the second spindle interface, 9 cords, pin socket of type D, connect with the second spindle transducer;
- CN21: encoder interface, 15 cords, pin socket of type D, connect with spindle encoder;
- CN22: encoder 2 interface, 15 cords, pin socket of type D, connect with spindle encoder 2 (expandable to the second MPG interface)
- CN31: MPG interface, 26 cords, pin socket of type D, connect with MPG
- CN53: GSKLINK serial bus A, 9 cords, pin socket of type D, communicate between CNC and the servo drive;
- CN52: GSKLINK serial bus B, 9 cords, pin socket of type D, communicate between CNC and remote IO unit;
- CN54: machine panel communication interface, 15 cords, pin socket of type D, connect with the machine panel;
- CN61: input interface, 44 cords, pin socket of type D, interface for CNC receiving the

machine signals;

- CN62: output interface, 44 cords, pin socket of type D, interface for CNC sending the signals to the machine;
- CN1: interface of power supply

Front cover interface:

- USB interface: read/ write flash memory data;
- RS-232 interface: connect with PC communication;
- Internet interface: connect with Ethernet

1.1.3 General connection diagram

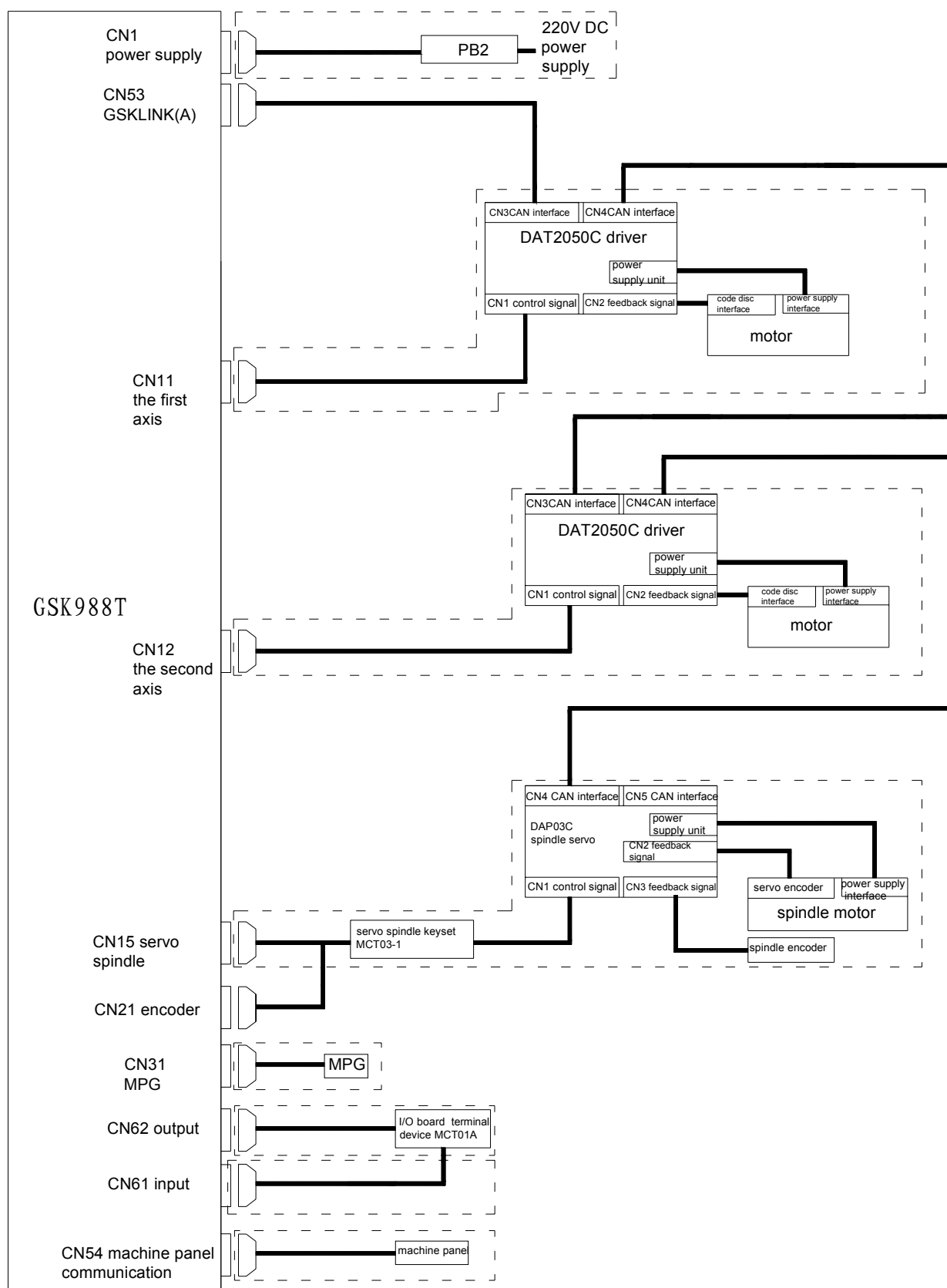


Fig. 1-3 GSK988T general connection diagram

1.2 GSK988T installation

1.2.1 GSK988T outer dimension

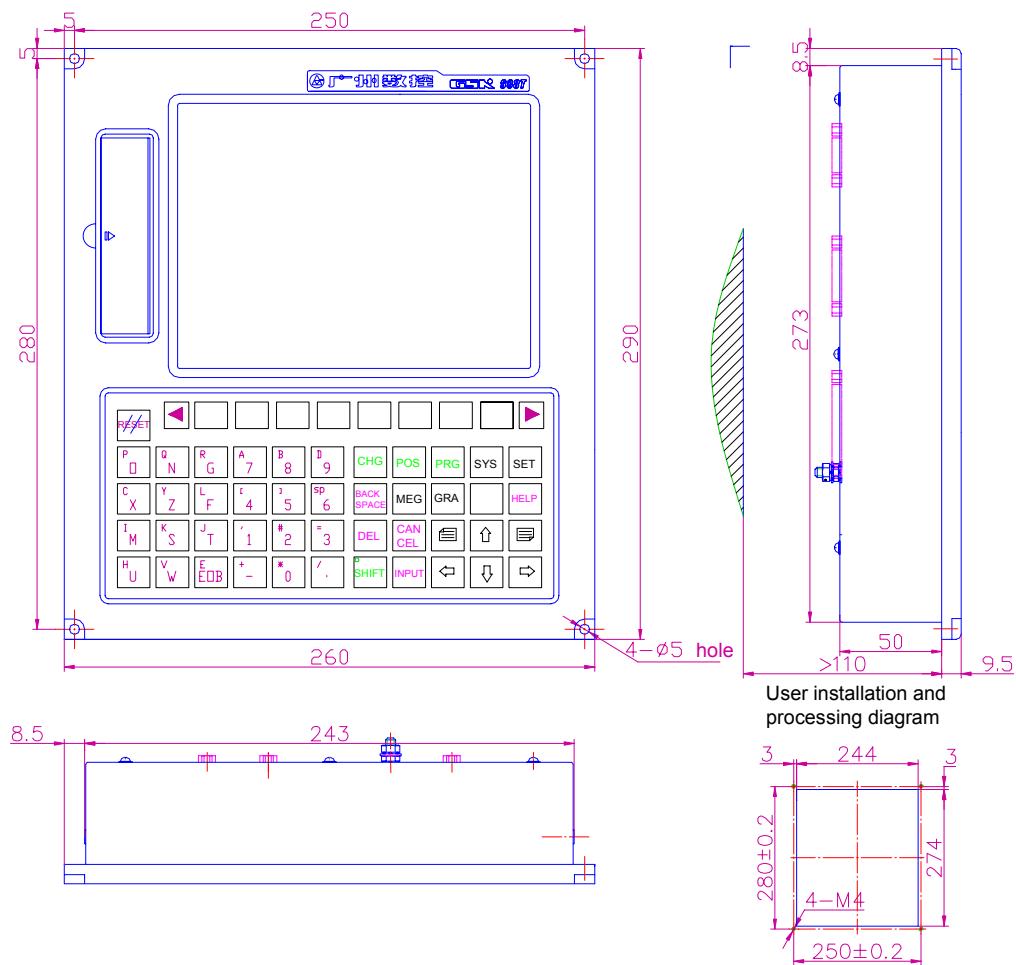


Fig. 1-4 GSK988T outer dimension

1.2.2 Outer dimension of machine operational panel

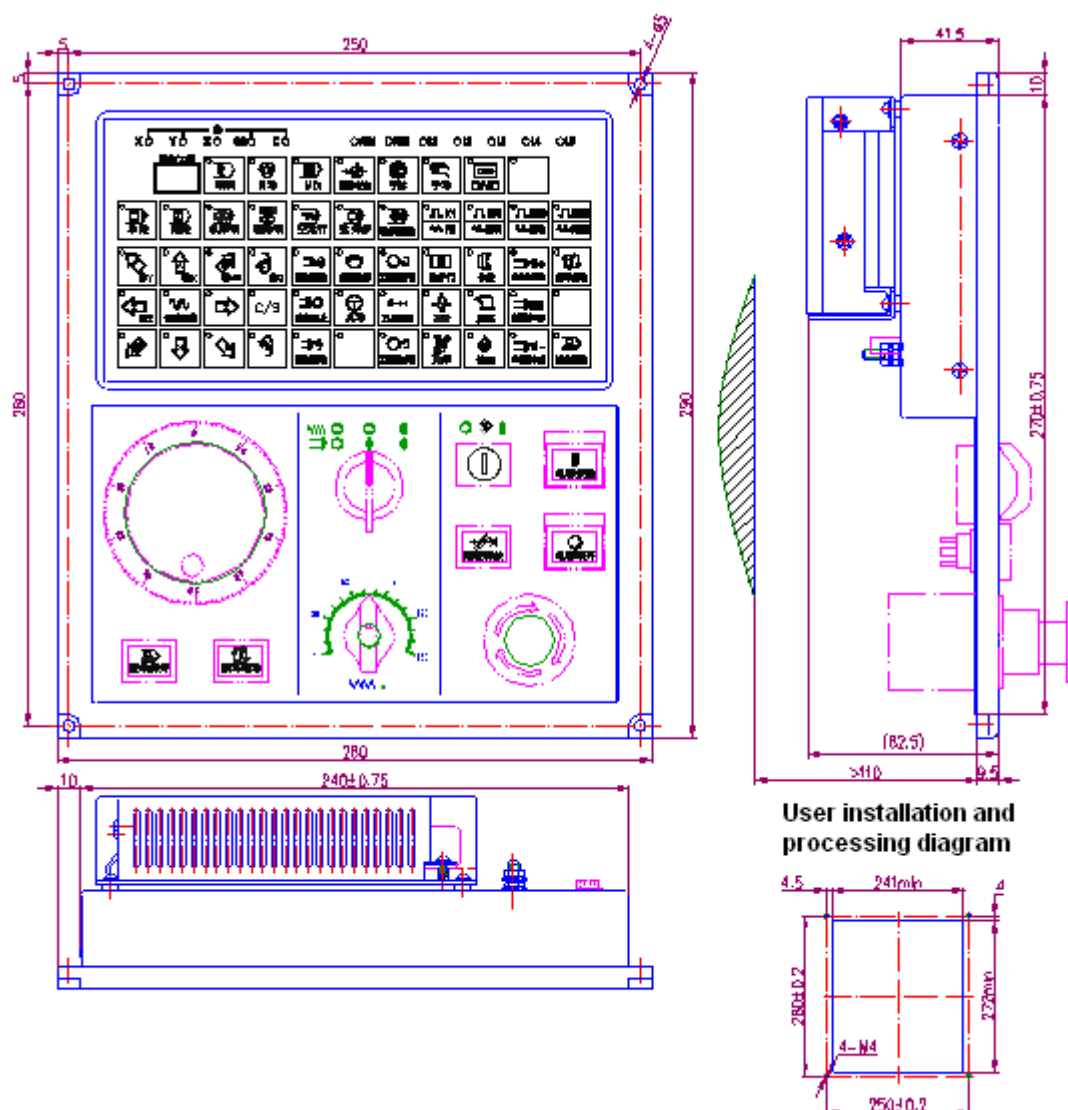


Fig. 1-5 Outer dimension of machine operational panel

1.2.3 Installation conditions of the cabinet

- The cabinet can prevent the dust, coolant and organic liquid;
- When the cabinet is designed, the distance between CNC back cover and the cabinet can not be less than 20cm; in consideration of the temperature rising in the cabinet, the temperature between inside and outside of the cabinet can not exceed 10℃;
- To guarantee the internal air circulation, the fan should be installed in the cabinet;
- The display panel should be installed in the position which can't be jetted by the coolant;
- Try to lower the external electrical interference to avoid the interfering with CNC transmission during designing the cabinet.

1.2.4 Method of shielding from interference

CNC has already used the anti-interference measures of shielding space electromagnetic

radiation, absorbing impulse current and filtering the clutter power supply, etc, to the extent, which can prevent CNC system from the external interference source. To guarantee CNC system work normally, the necessary steps should be used when connecting with CNC:

1. CNC should keep far away from the equipment of interference (such as transducer, AC contactor, static generator, HV generator and section device of main lead, etc)
2. Supply power to CNC through the isolation transformer, and CNC machine must be connected with the ground; CNC and the drive must connect with the independent ground wire from the ground point.
3. Barrage jamming: Connect RC loops in parallel at the ends of AC coil (shown in fig1-6), and try to be close to the perceptual load during installing RC loops; connect the fly-wheel diode in parallel reversely at the ends of DC coils (shown in fig 1-7); connect surge absorber in parallel at AC motor winding end (shown in fig 1-8).

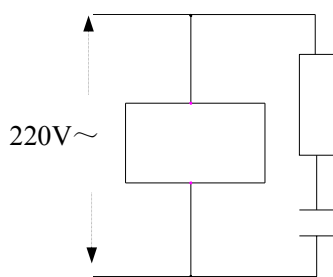


Fig. 1-6

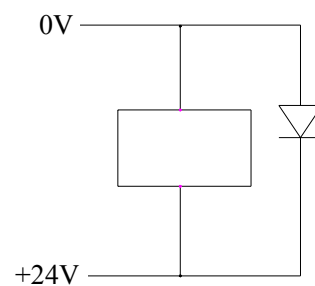


Fig. 1-7

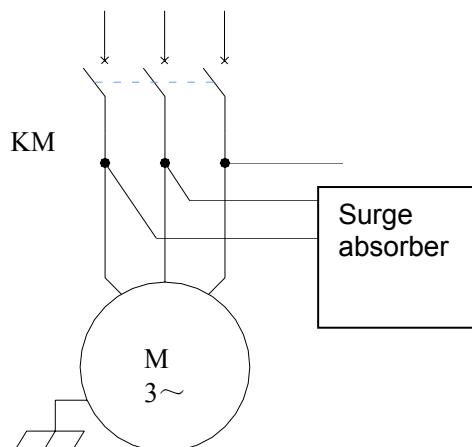


Fig. 1-8

4. CNC lead-out cable uses twisted shielded cable or shielded cable, the shielded layer of the cable should connect ground with the single end at CNC side, and the holding wire should be as shorter as possible.

5. To reduce the mutual interference between CNC signal cable and the heavy-current cable, the following principles should be observed during wiring:

Table 1-1

GROUP	TYPE OF CABLES	REQUIREMENTS OF WIRING
A	AC power-supply line	Seperately bind the cables of group A, that of group B, and group C, and keep at least 10cm from each other, or magnetic shield group A cable.
	AC coil	
	AC contactor	
B	DC coil (24VDC)	Seperately bind the cables of group A and that of group B or shield group B cable; the cable of group B should keep as further as possible from that of group C.
	DC relay (24VDC)	
	Cable between CNC and the heavy-current cabinet	
	Cable between CNC and the machine	
C	Cable between CNC and servo drive	Seperately bind the cables of group C and that of group A or shield group C cable; and keep at least 10cm between group C cable and that of group B and the cable uses twisted pair line.
	Positional feedback cable	
	Position encoder cable	
	MPG cable	
	Other shielded cable	

Chapter II Interface signal definition and connection

2.1 Connection with the drive unit

2.1.1 Definition of the drive interface

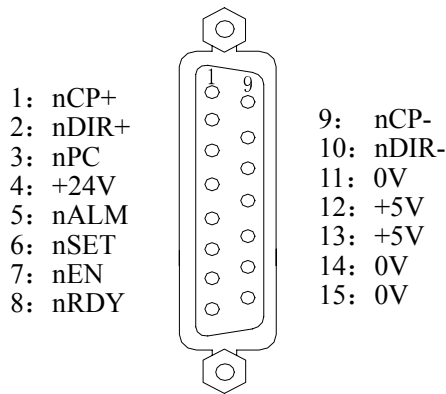


Table 1-1

DEFINITION OF SIGNAL	INTRODUCTION OF SIGNAL
nCP+、nCP-	Command pulse signal
nDIR+、nDIR-	Command direction signal
nPC	Zero signal
nALM	Drive alarm signal
nEN	Axial enable signal
nSET	Pulse forbidden signal
nRDY	Servo ready signal

Fig. 2-1 CN11, CN12, CN13 and CN14
interfaces (15 cords, pin socket of D
type)

Remark: CN1 is the 1st servo axis interface, CN2 is the 2nd one, CN3 the 3rd one, and CN4 the 4th one. Each control axis outputs the corresponding servo axis interface, which is set by parameter NO.1023.

2.1.2 Command pulse signals and command direction signals

nCP+ and nCP- are command pulse signals, nDIR+ and nDIR- are command direction signals, the two groups of signals all are difference (AM26LS31) output, the external is suggested to use AM26LS32 for receiving, refer to the following Fig.2-2 about the internal circuit:

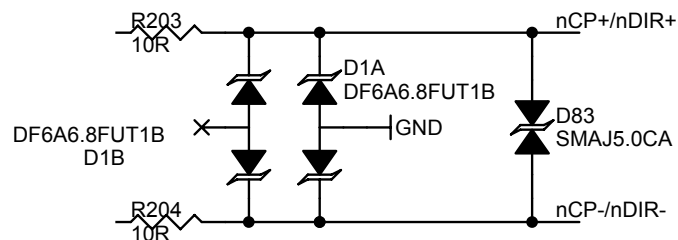


Fig. 2-2 Internal circuit of command pulse signals and command direction signals

2.1.3 Drive alarm signal nALM

The drive alarm level is low or high, which is set by 0 bit of parameter 1816; refer to Fig. 2-3 about the internal circuit:

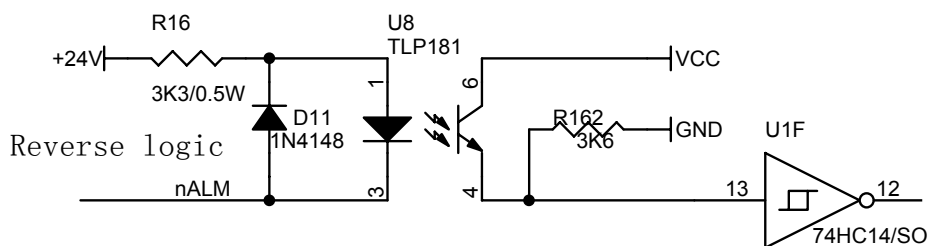


Fig. 2-3. Internal circuit of the drive alarm signal

Input circuit of this type requires that the drive should provide the signal through the methods in the following Fig. 2-4:

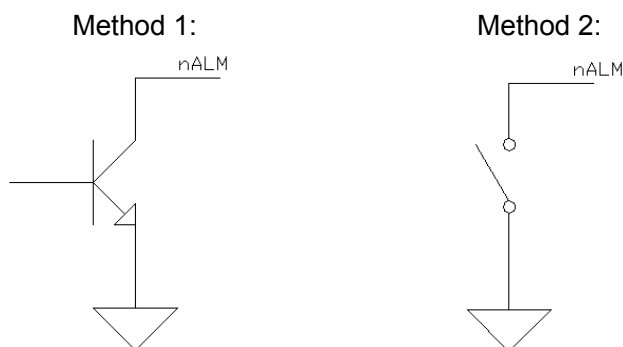


Fig. 2-4. Methods of the drive providing signals

2.1.4 Axial enable signal nEN

When CNC is running normally, nEN signal output is valid (nEN signal connects with 0V), and the drive or the emergency stop alarms, CNC switched off, nEN signal outputs (nEn signal cuts off 0V). About the internal interface circuit, refer to the following Fig. 2-5:

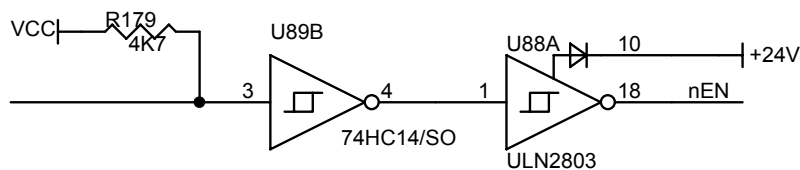


Fig. 2-5. Internal interface circuit of axial enable signal

2.1.5 Pulse forbidden signal nSET

nSET signal is for controlling the servo input forbid. To improve the anti-interference ability between CNC and the drive, the signal is low level when CNC outputs the pulse signal, if there isn't any pulse signals, it is high level; refer to the following Fig. 2-6 about the internal interface circuit:

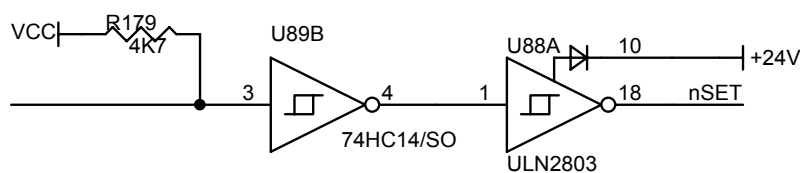


Fig. 2-6 Pulse forbidden signal circuit

2.1.6 Zero signal nPC

Take one-turn signal of motor encoder or proximity switch signal as zero signals. About the internal connection circuit, refer to the following Fig. 2-7:

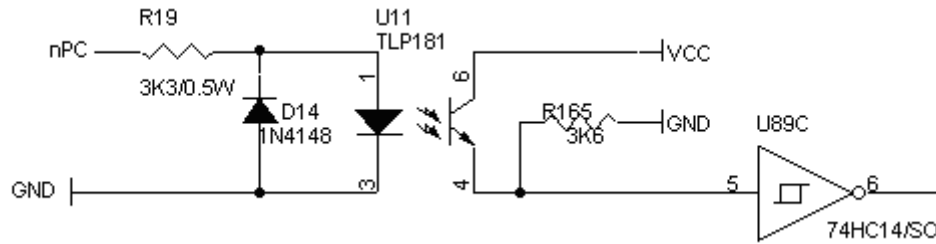


Fig. 2-7 zero signal circuit

a): The user should provide the wave of PC signals shown in Fig. 2-8:

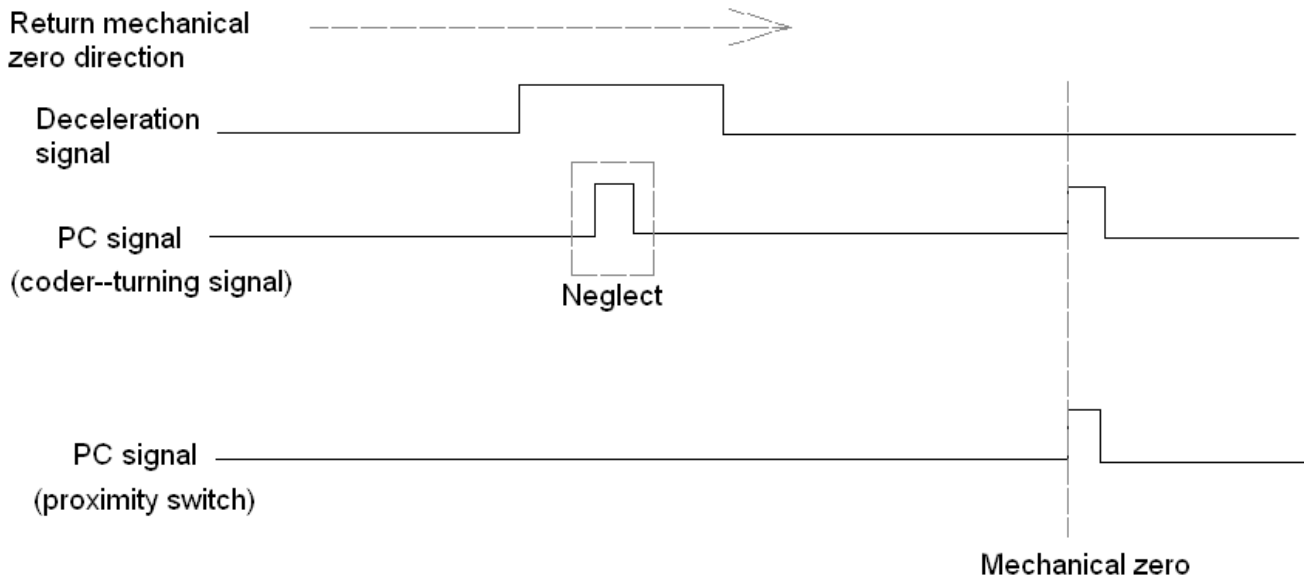


Fig. 2-8 PC signal oscillogram

Remark: During mechanical zero return, after releasing the deceleration switch, CNC determines the position of the reference point through detecting PC signal jumping, and the rising edge check and the falling edge check are both valid.

b): Refer to the following Fig. 2-9 about the connection method of taking one Hall unit in NPN type as the deceleration signal:

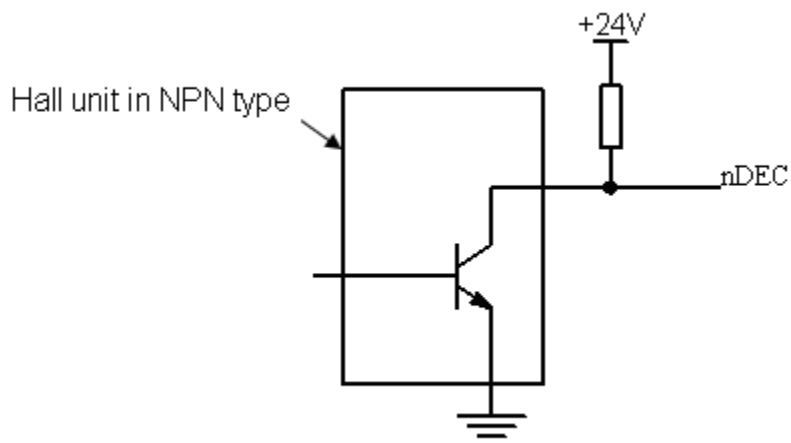


Fig. 2-9 Connection with Hall unit in NPN type

c): Refer to the following Fig. 2-10 about the connection method of taking one Hall unit in PNP type as one deceleration signal:

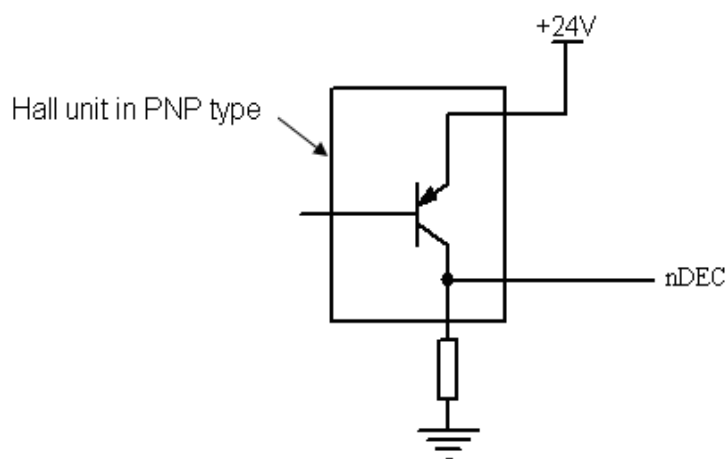


Fig. 2-10 Connection with Hall unit in PNP type

2.1.7 Connection with the drive unit

The connection between GSK988T system and GSK DA98B drive unit is shown in the following Fig. 2-11:

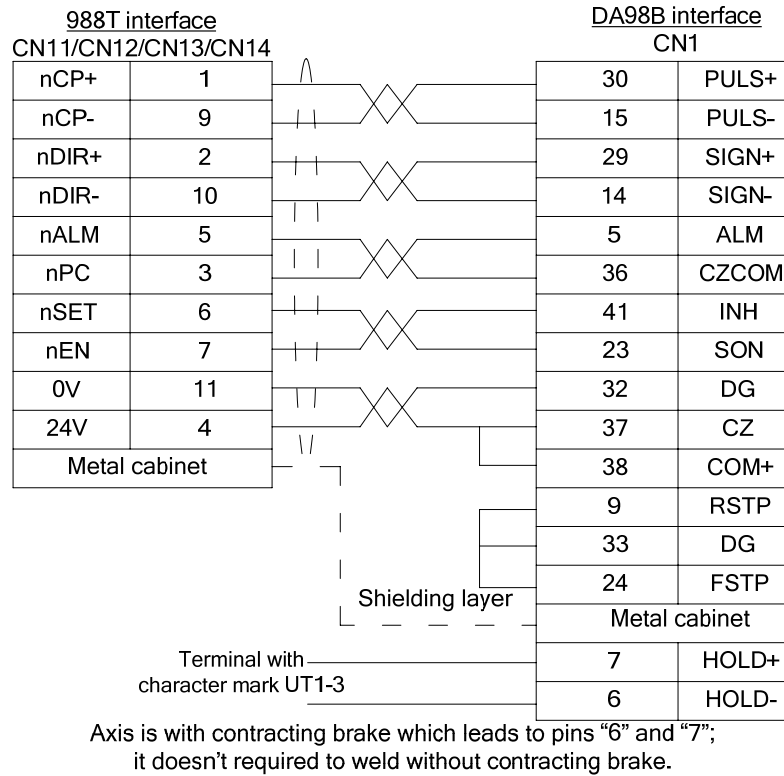


Fig. 2-11 Connection between GSK988T and DA98B

The connection between GSK988T and GSK DAT2050C drive unit is shown as below:

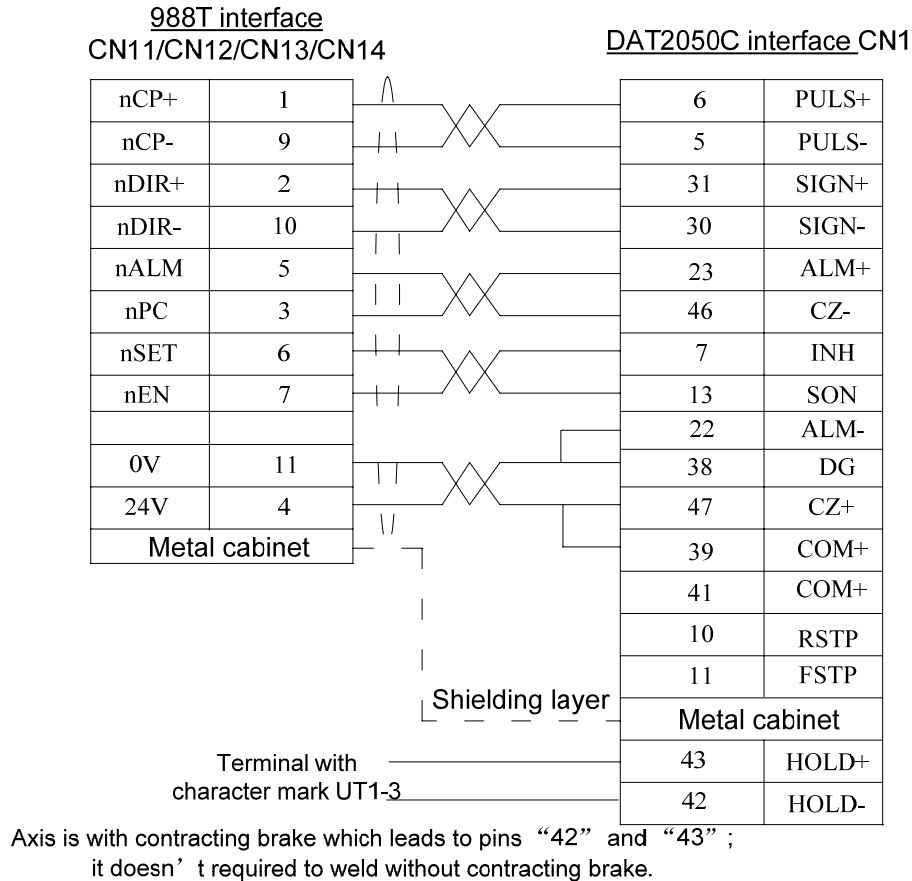


Fig. 2-12 Connection between GSK988T and DAT2050C drive

2.2 Connection with the spindle

The spindle interface of GSK988T is CN15 (the fifth axis · spindle interface), and the interface is also equipped with the function of pulse output and analog voltage output, and adopted with the servo spindle drive unit or the common spindle transducer, or taken as the independent 5th servo axial interface. Moreover, GST988T system is also equipped with the 2nd spindle interface CN41 (about the details, refer to the following chapters), and it can output 0~+10V analog voltage for extending the 2nd spindle or the unit head.

2.2.1 The 5th axis · spindle interface definition

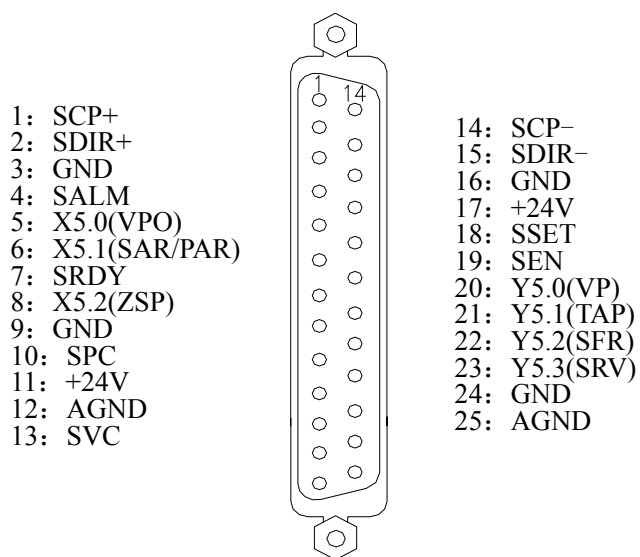


Fig 2-13 CN15 servo spindle interface (25 cords, pin socket of D type)

Table 1-2

DEFINITION OF SIGNAL	INTRODUCTION OF SIGNAL	DEFINED FUNCTION OF STANDARD PLC ADDRESS
SCP+, SCP-	Command pulse signal	/
SDIR+,SDIR-	Command direction signal	/
SALM	Drive unit alarm signal	/
SRDY	Servo ready signal	/
SSET	Pulse forbidden signal	/
SEN	Axial enable signal	/
SPC	Zero signal	/
SVC	0~+10V analog voltage output	/
AGND	Analog voltage output ground	/
X5.0 (VPO)	Address of PLC definition, switch input	Spindle speed/position status signal
X5.1 (SAR/PAR)	Address of PLC definition, switch input	Spindle position/speed reaching signal
X5.2 (ZSP)	Address of PLC definition, switch input	Spindle output at zero speed signal
Y5.0 (VP)	Address of PLC definition, switch output	Spindle speed/position switch signal
Y5.1 (TAP)	Address of PLC definition, switch output	The 2 nd gain selective signal of spindle speed loop for tapping
Y5.2 (SFR)	Address of PLC definition, switch output	Spindle CW signal
Y5.3 (SRV)	Address of PLC definition, switch output	Spindle CCW signal
+24V	+24V	/
GND	0V (Switch input & output signal ground)	/

2.2.2 Connection with the servo spindle drive unit

Connection between GSK988T and GSKDAP03C servo spindle drive unit is shown as the following Figure:



The 5th axis spindle interface (CN15) SVC port outputs 0~+10V voltage, the connection between GSK988T and the spindle transducer is shown as the following figure:



2.3 Connection with the spindle encoder

GSK988T is equipped with encoder input interfaces (CN21 and CN22) of two routes, only use CN21 interface as feedback input of spindle speed during default. When multi-spindle control function is started, select the encoder interface, which receives the feedback pulse for the system control, through the selective signal PC2SLC (G28.7) of spindle encoder in PLC. When the interface (CN22) of encoder 2 doesn't connect the encoder, the selective signal PC2SLC of the position encoder isn't set as 1, CN21 interface is taken as the feedback input of the spindle speed.

2.3.1 Interface definition of the spindle encoder

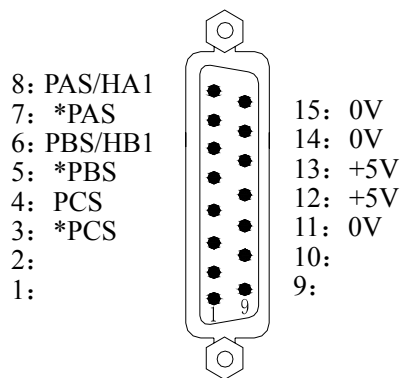


Fig 2-16 Encoder interface of CN21 and CN22 (15 cords, pin socket of D type)

Table 1-3

DEFINITION OF SIGNAL	REMARK
*PAS/PAS	Encoder phase A pulse
*PBS/PBS	Encoder phase B pulse
*PCS/PCS	Encoder phase C pulse
HA1 (Only CN22 is with the signal)	The 2 nd MPG phase A signal (When it's not used in the 2 nd spindle encoder, it can be used to extend the 2 nd MPG)
HB1 (Only CN22 is with the signal)	The 2 nd MPG phase B signal (When it's not used in the 2 nd spindle encoder, it can be used to extend the 2 nd MPG.)

2.3.2 Signal introduction

*PCS/PCS, *PBS/PBS and *PAS/PAS are respectively difference input signals of phase C, B and A; *PAS/PAS and *PBS/PBS is the cross square wave with difference of 90°, the maximum signal frequency <1MHz: The quantity of GSK988T encoder pulses is set by parameter NO.3873 (the quantity of the spindle encoder pulses) and NO.3803 (the quantity of the 2nd spindle encoder).

2.3.3 Connection with the spindle encoder interface

The connection between GSK988T and the spindle encoder with the twisted pair line is shown as the following figure, and Changchun Yiguang ZLF-12-102.4BM-C05D encoder is taken as one example:

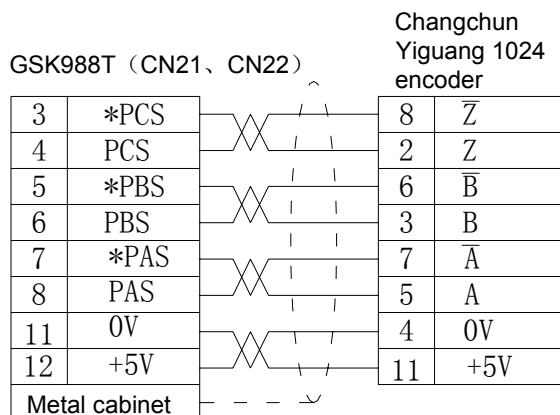


Fig. 2-17 Connection between GSK988T and the encoder

2.4 Connection with MPG

2.4.1 Definition of MPG interface

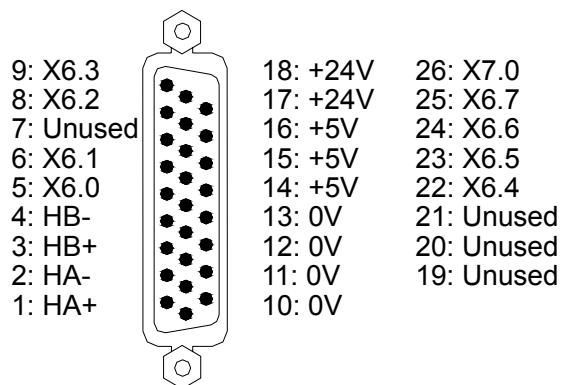


Fig. 2-18 CN31 MPG interface
(26 cords, pin socket of D type)

Table 1-4

DEFINITION OF SIGNAL	REMARK
HA+, HA-	MPG phase A signal input
HB+, HB-	MPG phase B signal input
X6.0~X7.0	Address of PLC definition, switch input

2.4.2 Introduction of signals

HA+, HA- and HB+, HB- are respectively difference input signals of MPG phase A and B. X6.0~X7.0 interfaces are input addresses defined by PLC interface, and it can also be used for axial selection of external MPG box and gear signal input.

2.4.3 Connectino with MPG

The typical connection between GSK988T and MPG is shown as the following figure:

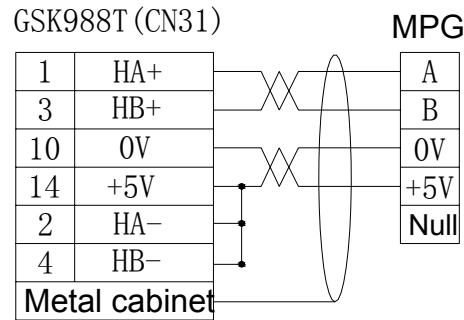


Fig. 2-19 Connection between GSK988T and MPG

2.5 Connection with the 2nd spindle

GSK988T supports the control function of multi-spindle with output interfaces of spindle analog voltage in two routes. One route is in the 5th axis-spindle (CN15) interface; The other one is in the interface of the 2nd spindle (CN41), and signals from PLC select the analog voltage output to control the two spindle interfaces. The 2nd spindle interface is used to extend the 2nd transducer spindle or the unit head.

2.5.1: The 2nd spindle (analog spindle) interface definition

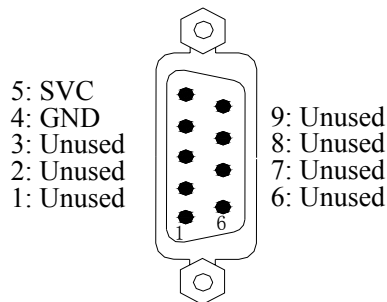


Table 1-5

DEFINITION OF SIGNAL	REMARK
SVC	0~+10V analog voltage output
GND	Analog voltage output ground

Fig. 2-20 CN41 analog spindle interface (9 cords, pin socket of D type)

2.5.2 Connection with the 2nd spindle transducer interface

The 2nd spindle interface SVC port output 0~+10V voltage, the connection between GSK988T and the 2nd spindle transducer is shown as the following Fig. 2-19:

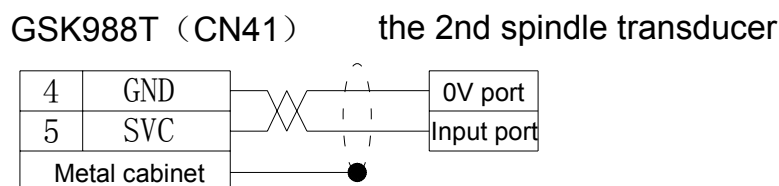


Fig. 2-21 Connection between GSK988T and the 2nd spindle transducer

2.6 Connection with the external equipment

2.6.1 Definition of GSKLINK bus interface

GSK988T is with GSKLink interfaces of two routes for connecting with the remote IO units and the servo drive unit with GSKLink communication function. Among them, CN53 (GSKLINK serial bus A) is for communication between CNC and the servo drive unit to realize servo parameter configuration at CNC port and real-time monitor of servo unit; CN52 (GSKLINK serial bus B) is for communication between CNC and remote IO unit.

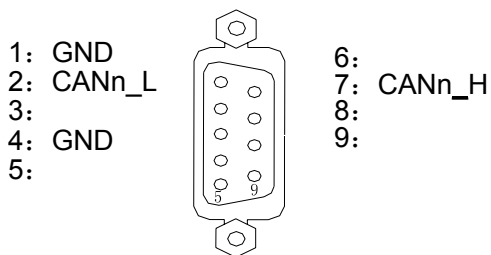


Fig 2-22 GSKLink bus interface CN53 and CN52 (9 cords, pin socket of D type)

Table 1-6

DEFINITION OF SIGNALS	REMARK
CANn_L	Low level of data difference signal
CANn_H	High level of data difference signal
GND	Signal ground

2.6.2 Net interface definition

There are three interfaces on the left side of GSK988T LCD display screen: USB (flash driver), internet and RS-232 interfaces, which are shown in the following figure. All the three interfaces can be used for processing the file, two-way transmission between the system para file and PLC file and upgrading the system software. Among them, the internet interface can also be used for remote monitor from PC to 988T system.

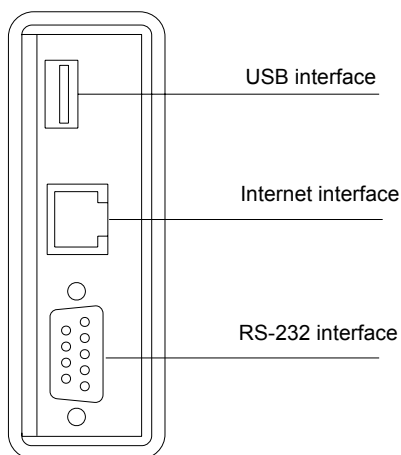


Fig. 2-23 GSK988T front panel interface

Table 1-7 Internet interface (standard interface):

PIN NUMBER	SIGNAL	PIN NUMBER	SIGNAL
1	TXDLAN+	11	LAN_LED
2	TXDLAN-	10、12	VDD33
3	RXDLAN+	13、14	Chassis ground
6	RXDLAN-		
9	LINK_LED		

2.6.3 USB interface definition

Table 1-8 Main USB interface (standard interface):

PIN NUMBER	SIGNAL
1	VCC(+5V)
2	USB_DN0
3	USB_DP0
4	GND
5, 6	Chassis ground

2.6.4 RS-232 interface definition

RS-232 communication interface:

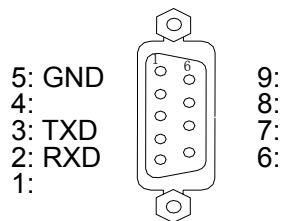


Table 1-9

PIN NUMBER	SIGNAL
2	RXD
3	TXD
5	GND

Fig 2-24 RS-232 interface
(9 cords, pin socket of D type)

2.7 Connection with the machine panel

Connect between GSK988T system and the machine panel through communication.

Communication interface definition of the machine panel:

Table 1-10

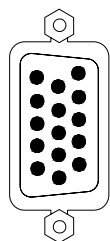


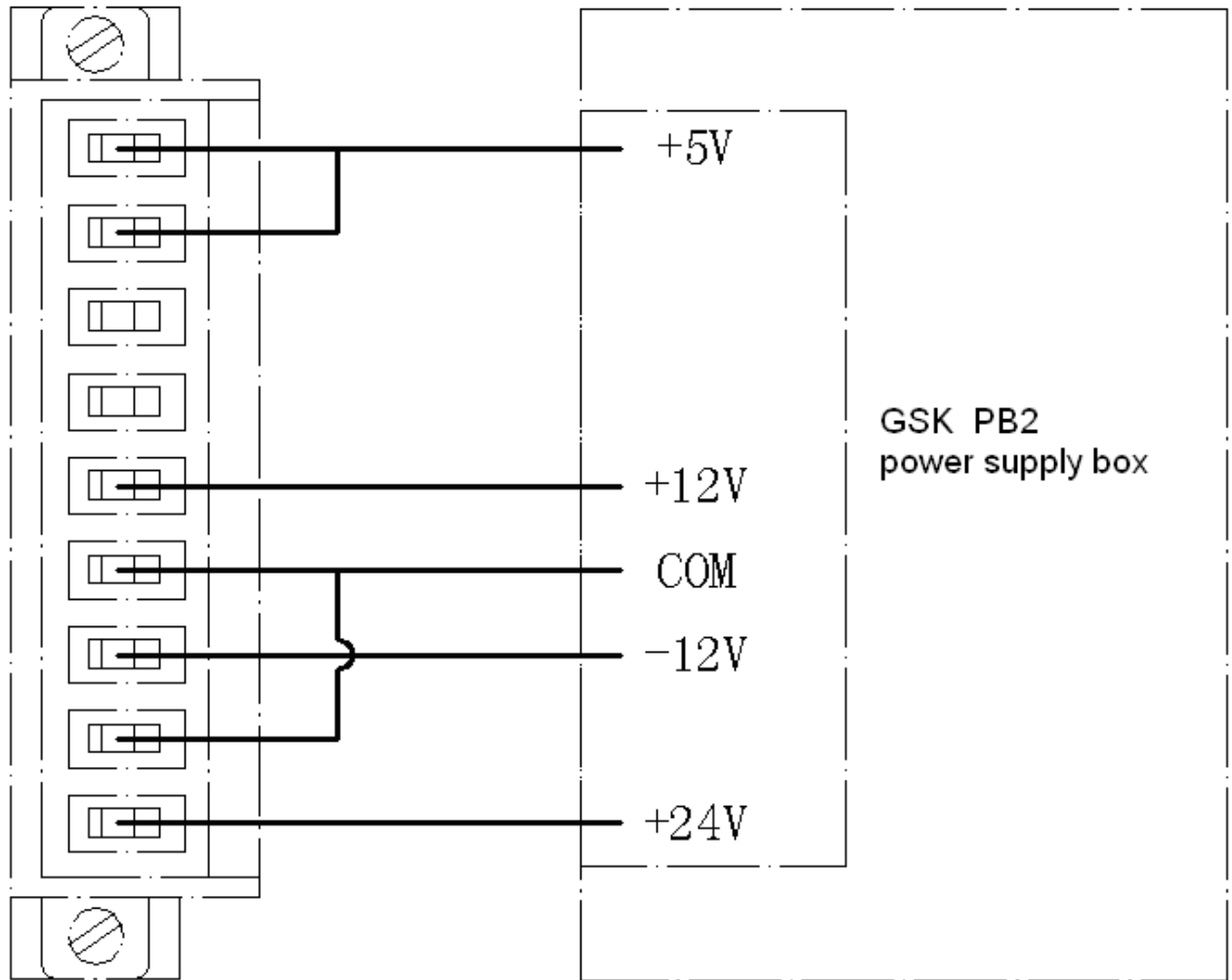
Fig2-25 Standard machine panel interface CN54 (15 cords, pin socket of D type)

PIN NUMBER	SIGNAL	IN/OUT	REMARK
1	RXDA	IN	Receive data difference signal
2	RXDB	IN	Receive data difference signal
4	TXDA	OUT	Send data difference signal
5	TXDB	OUT	Send data difference signal
7	RESET	OUT	Panel resetting signal

2.8 Connection with the power supply

GSK988T uses GSK-PB2 power supply boxes, total 4 groups of voltage: +5V (3A) , +12V (1A) , -12V (0.5A) and +24V (0.5A) , and common port COM (0V). When GSK988T is dispatched from the factory, GSK-PB2 power supply box and GSK988T power supply interface has been already connected, so the user just connects 220V AC power supply.

The connection between GSK-PB2 power supply box and GSK988T power supply interface is shown as the following Fig. 2-26:



CN1 power supply interface

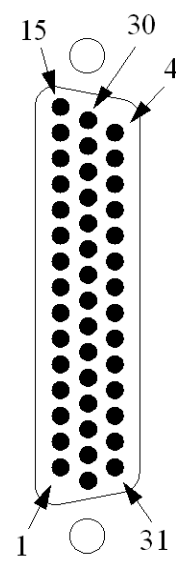
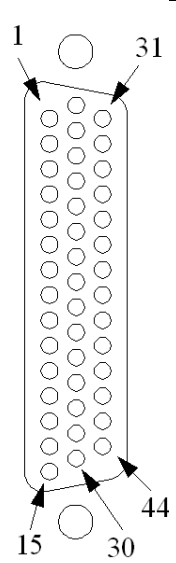
Fig 2-26 Connection between 988T system and GSK PB2 power supply box

2.9 GSK988T common I/O interface definition

2.9.1 Definition of input & output addresses

Table 1-11 Definition of input & output addresses

INTERFACE	CN61 PIN NUMBER	PLC ADDRESS	INTERFACE	CN62 PIN NUMBER	PLC ADDRESS
	1	X0.0		1	Y0.0
	2	X0.1		2	Y0.1
	3	X0.2		3	Y0.2
	4	X0.3		4	Y0.3
	5	X0.4		5	Y0.4
	6	X0.5		6	Y0.5
	7	X0.6		7	Y0.6
	8	X0.7		8	Y0.7
	9	X1.0		9	Y1.0

 <p>CN61 (pin) input</p>	10	X1.1	 <p>CN62 (hole) output</p>	10	Y1.1
	11	X1.2		11	Y1.2
	12	X1.3		12	Y1.3
	13	X1.4		13	Y1.4
	14	X1.5		14	Y1.5
	15	X1.6		15	Y1.6
	16	X1.7		16	Y1.7
	29	X2.0		29	Y2.0
	30	X2.1		30	Y2.1
	31	X2.2		31	Y2.2
	32	X2.3		32	Y2.3
	33	X2.4		33	Y2.4
	34	X2.5		34	Y2.5
	35	X2.6		35	Y2.6
	36	X2.7		36	Y2.7
	37	X3.0		37	Y3.0
	38	X3.1		38	Y3.1
	39	X3.2		39	Y3.2
	40	X3.3		40	Y3.3
	41	X3.4		41	Y3.4
	42	X3.5		42	Y3.5
	43	X3.6		43	Y3.6
	44	X3.7		44	Y3.7
	17	X4.0		17~19, 26~28	0V
	18	X4.1		20~25	+24V
	19	X4.2			
	20	X4.3			
	25	X4.4			
	26	X4.5			
	27	X4.6			
	28	X4.7			
	21~24	0V			

2.9.2 Input signal

Input signal is the one which the machine electric wire or the machine panel transmits to CNC, and after connecting the input signal and +24V, the input is valid; if they are cut off, the input is invalid. The input signal of contacts on the machine side should satisfy the following conditions:

Contact capacity: DC30V, above 16mA;

Leakage current between contacts during opening: Below 1mA;

Potential drop between contacts during closing: Below 2V (Current 8.5mA, including the cable potential drop).

There are two methods of external input for input signals: one is switch input with contacts, the connection is shown in Fig. 2-27:

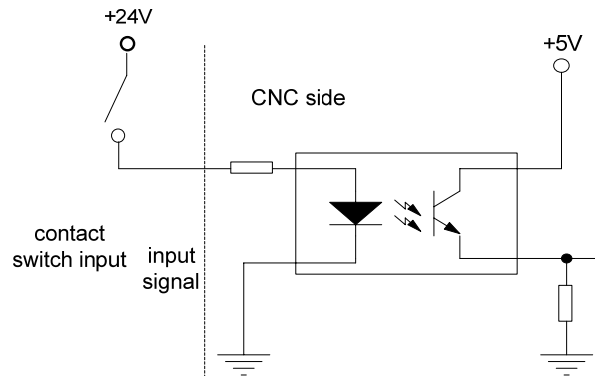


Fig. 2-27

The other is switch (transistor) input free of contacts; connection is shown in Fig. 2-28 and Fig. 2-29.

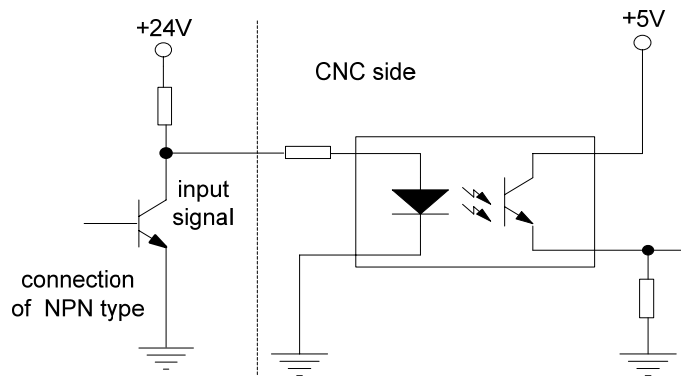


Fig. 2-28 Connection in NPN type

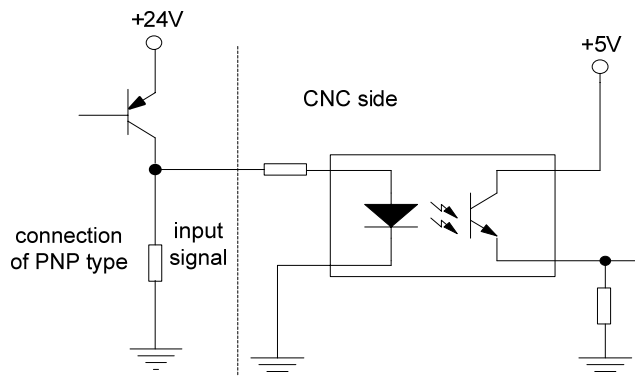


Fig. 2-29 Connection in PNP type

2.9.3 Output signal

Output signal is used for the drive machine electrical wire side or the relay and the indicator on the machine panel side. When the output signal connects with 0V, the output function is valid (Y output signal is 1); cut off 0V, the output function is invalid (Y output signal is 0). The circuit is shown in the following Fig. 2-30:

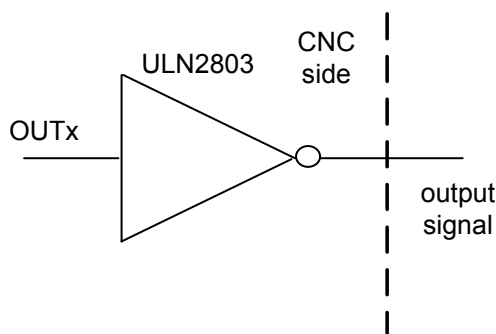


Fig. 2-30. Internal circuit structure diagram of the output signals

Therefore, the signal has two output statuses: OV output or high resistance. The typical application is as below:

- Drive light diode

Use ULN2803 to output drive light diode and need the serial connection with one resistance, limit the current from light diode (normally 10mA), which is shown in Fig. 2-31:

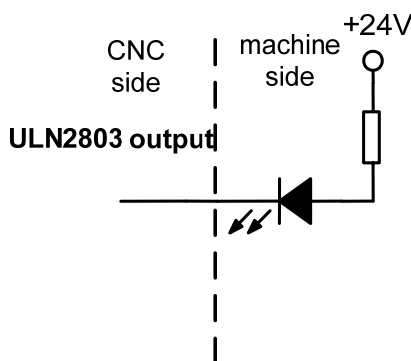


Fig. 2-31 Drive light diode

- Indicator in drive filament type

ULN2803 is used to output the indicator in drive filament type, and externally connect with one preheated resistance to reduce the current shock during break-over, and the value of the preheated resistance is based on that the indicator is off, which is shown in Fig. 2-32:

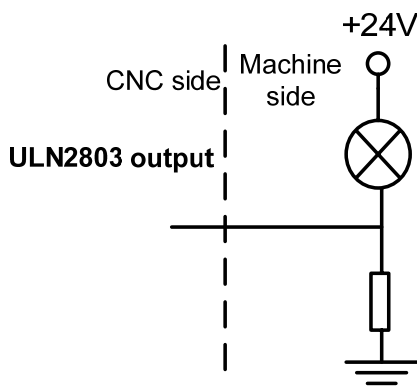


Fig. 2-32

- Drive nastic loading (such as the relay)

Output the drive nastic loading in ULN2803 type and it requires connecting the fly-wheel diode close to the circuit, which is to protect the output circuit and reduce the interference, which is shown in Fig. 2-33:

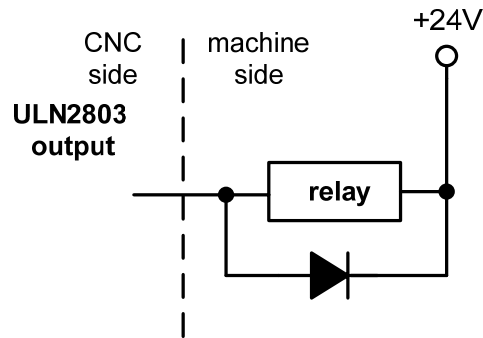


Fig. 2-33

2.10 System function

2.10.1 Travel limit and emergency stop

- Address definition

X0000			ESP					
Corresponding interface			CN61.6					

ESP: Emergency stop signal, emergency stop alarms when +24V is cut off.

- G signal

G	0	0	8				ESP				
---	---	---	---	--	--	--	-----	--	--	--	--

ESP =0: Emergency stop is valid.

=1: Emergency stop is invalid.

- Connecting the signals

Emergency stop signal (ESP) circuit is shown in the following Fig. 2-34:

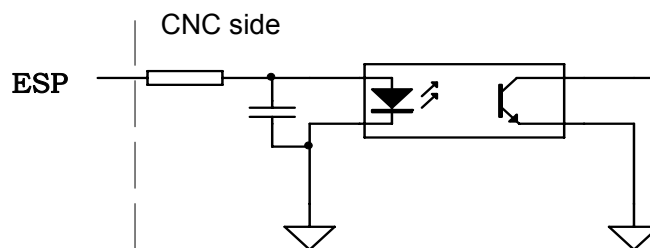


Fig. 2-34

- External connection with the machine

The connection method of the emergency stop and the limit switch is shown as the following Fig.2-35:

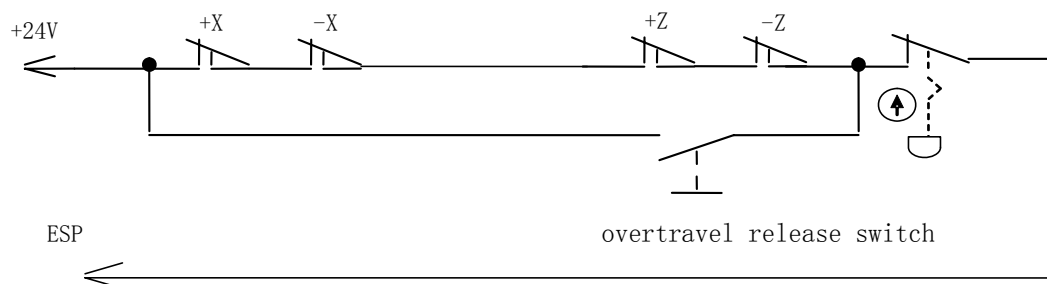


Fig.2-35

2.10.2 Mechanical zero return

GSK988T system supports the three methods of setting mechanical zero point (reference point): setting the reference point free of block, the reference point with block and the encoder reference point in absolute type.

When the machine is equipped with the absolute position encoder and its function is valid (set the system parameter NO.1815 BIT5 APCx as 1), after setting the mechanical zero point (the reference point of the encoder in absolute type) at the first time, since then, auto set the mechanical zero point (reference point) after the system powers on without manual reference point return. About setting the reference point of encoder in absolute type, refer to the other chapters of the manual.

Set the reference point free of block, GSK988T system uses one-turn signal (nPC) of the motor encoder to set the reference point.

Set the reference point with block, and GSK988T system uses external deceleration switch input signal (DECn) and one-turn signal (nPC) of the motor encoder to set the reference point.

This chapter mainly introduces GSK988T system and the machine without encoder in absolute type, and how to set the machine zero point (reference point) in the method of setting the reference point with block.

● Address definition

X0000					DECX			
Corresponding interface					CN61.4			

X0002					DECZ			
Corresponding interface					CN61.32			

DECX and DECZ: Zero return deceleration signal in X and Z axes, fixed address

● Control parameter

1	0	0	5							ZRNx
---	---	---	---	--	--	--	--	--	--	-------------

ZRNx: When the reference point is not set, during auto running (MEM, DEM or MDI), the movement commands except G28 are specified; set whether the system alarms.

0: Alarm

1: Not alarm

1	0	0	6			ZMlx					
---	---	---	---	--	--	-------------	--	--	--	--	--

ZMlx: Set each axis in the direction of reference point return

0: Positive direction

1: Negative direction

Remark: It is valid when power on again.

1	2	4	0	Machine coordinate value of each axis at the 1st reference point						
---	---	---	---	--	--	--	--	--	--	--

Parameter #1240~#1243 set the coordinate value from the 1st to the 4th reference points in the mechanical coordinate system.

SETTING UNIT	IS-B	IS-C	UNIT
Machine in metric system	0.001	0.0001	mm
Machine in inch system	0.0001	0.00001	inch
Rotation axis	0.001	0.0001	deg

1	3	0	0		LZR						
---	---	---	---	--	-----	--	--	--	--	--	--

LZR: After connecting the power supply, set whether detect the first stroke in memory type before manual reference point return.

- 0: Detect
- 1: Not detect

1	4	2	5		FL speed of each axis reference point return						
---	---	---	---	--	--	--	--	--	--	--	--

Set the speed (FL speed) of each axis after deceleration during reference point return.

SETTING UNIT	DATA UNIT	VALID RANGE	DEFAULT SETTING
Machine in metric system	1 mm/min	6~15000	0
Machine in inch system	0.1 inch/min	6~6000	0
Rotation axis	1 deg/min	6~15000	0

3	0	0	9			DECx					
---	---	---	---	--	--	------	--	--	--	--	--

DECx Deceleration signal during the reference point return

- 0: Decelerate when the signal is 0
- 1: Decelerate when the signal is 1

● Connecting the signals

The internal connection circuit of deceleration signals is shown as Fig.2-36:

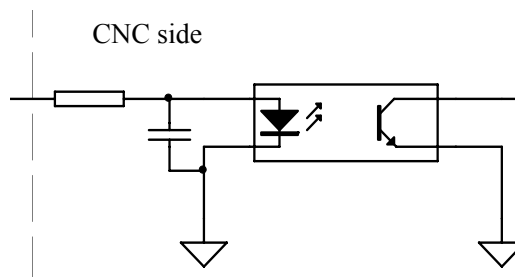


Fig 2-36

- The servo motor one-rotation signal is taken as the zero signal during mechanical zero return

① The sketch map is as below:

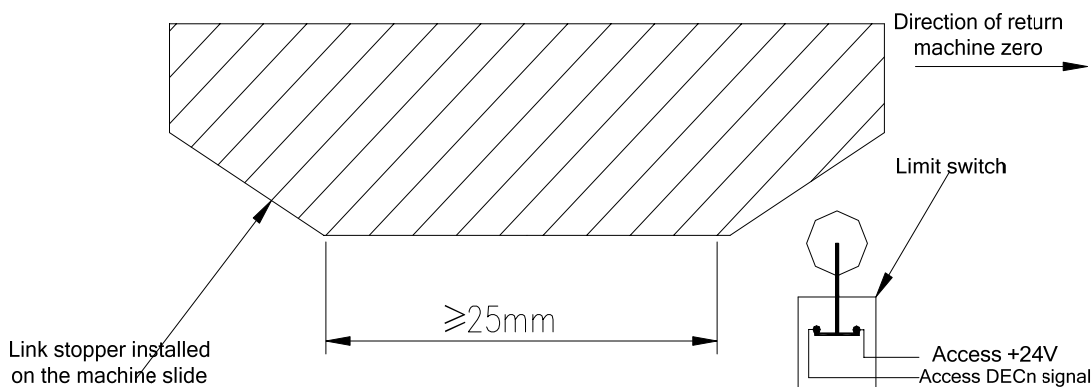


Fig.2-37

② Connection circuit of the deceleration signals

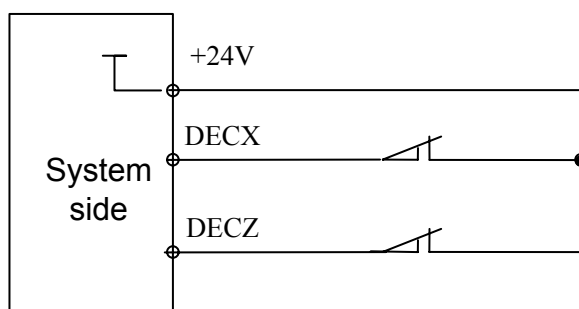


Fig.2-38

③ Process of mechanical zero return

A: When the mechanical zero return mode is selected, press manual positive or negative feed key (the mechanical zero return direction of the axis is set by ZMlx of parameter No.1006), and the corresponding axis moves in the mechanical zero direction at the rapid movement speed. It runs until the deceleration switch is pressed, and then the deceleration signal contact cuts off, the feedrate decelerates immediately and it runs at the fixed low speed.

B: After deceleration switch releases, the deceleration signal contacts are off again, CNC begins to detect one-turn signal nPC of the encoder. If the signal level jumps and changes, the movement stops; meanwhile the zero return finish indicator is on, which is on the panel and the operation of mechanical zero return completes.

④ Movement sequence of mechanical zero return

When BIT5 (ZMlx) of status parameter No.1006 is set 0, and BIT5 (DECx) of status parameter No.3009=0, the selective axis mechanical zero return is valid while the direction is positive and the deceleration signal is low level. Then, the movement sequence of the mechanical zero return is shown as the following figure:

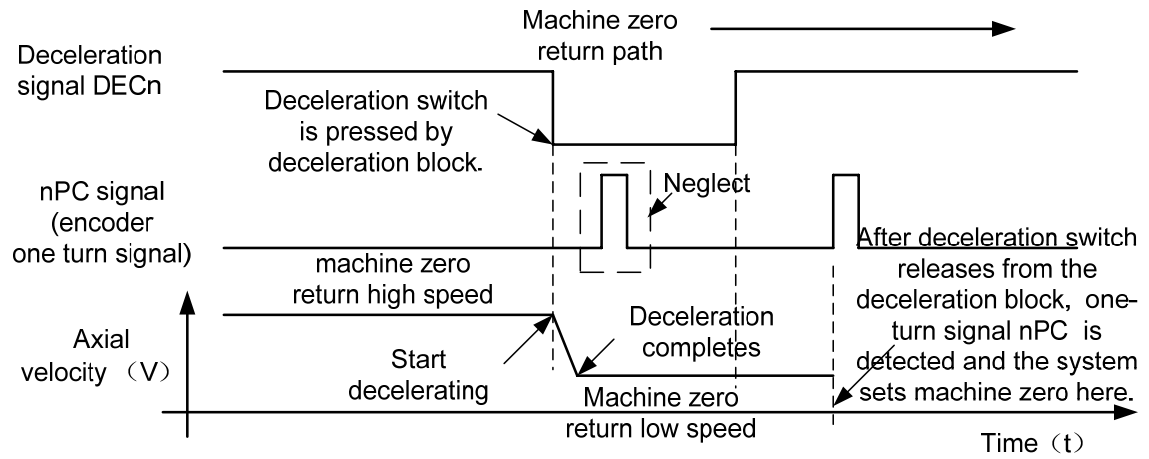


Fig.2-39

Chapter III Introduction of the parameters

This chapter mainly introduces CNC state and data parameters through setting different parameters to realize the different requirements of function. The data types of parameter mainly has the following four types:

Table 3-1

Formula of data	Range
(1) Bit type	8 digits, 0 or 1
(2) Bit axial type	
(3) Charater type	-99 999 999~+99 999 999
(4) Charater axial type	

About the parameters of the above character type (3) or character axial type (4), the span is set by the detailed parameters.

Each parameter should include the following information:

『Parameter type』 : (Bit type, digit axial type, character type or character axial type)

『Default setting』 : (8 digits in binary system, or integral value in 32 digits)

『Range』 : (Interval, enumerate or special judge)

『Authority to rewrite』 : (System, machine or user)

『Valid method』 : (Valid immediately or power on)

『Additional requirements』 : (Hiding and reserving, etc)

The examples of each parameter are as below:

Bit (axial) type:

Parameter number	#7	#6	#5	#4	#3	#2	#1	#0
0000								ABCx

『Type of parameter』 : Bit axial type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : System

『Valid method』 : Power on

『Additional requirements』 : None

ABCx About the introduction of the parameter digit (axial) type

0: OK

1: Forbid

(2) Charater (axial) type:

1000	Parameter name
------	----------------

『Parameter type』 : Character axial type

『Default setting』 : 600

『Range』 : 0~999

『Authority to rewrite』 : User

『Valid method』 : Power on

『Additional requirements』 : None

Explanation information of parameter in character (axial) type

3.1 Parameter for “setting”

	#7	#6	#5	#4	#3	#2	#1	#0
0000			SEQ			INI		

『Parameter type』 : Bit axial type

『Default』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : User

『Valid method』 : Valid immediately

『Additional requirements』 : None

INI Input unit

0: Metric system

1: Inch system

SEQ Whether auto insert the serial number

0: No

1: Yes

Programming in the edit and MDI modes, the serial number can be auto inserted.

The increment value is set in parameter NO.3216.

3.2 Parameters about the interfaces of input and output

0123	Baud rate of serial port (BPS)						
-------------	---------------------------------------	--	--	--	--	--	--

『Parameter type』 : Character type

『Default setting』 : 115200

『Range』 : 4800, 9600, 19200, 38400, 57600, 115200

『Authority to rewrite』 : User

『Valid method』 : Valid immediately

『Additional requirements』 : None

	#7	#6	#5	#4	#3	#2	#1	#0
0138		OWN						

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : User

『Valid method』 : Valid immediately

『Additional requirements』 : None

OWN When NC data or the programs are input or output, whether display the covered information.

0: Display

1: No display

3.3 Parameters of axis control/settting unit

	#7	#6	#5	#4	#3	#2	#1	#0
1001								INM

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

INM The minimum movement unit of linear axis is:

0: Metric system (The machine in the metric system)

1: Inch system (The machine in the inch system)

	#7	#6	#5	#4	#3	#2	#1	#0
1002					AZR		DLZ	

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

DLZ Whether the setting function of the reference point free of the link stopper is valid or not

0: invalid

1: valid (all axes)

Remark:

When parameter DLZ is 0, parameter DLZx (No.1005#1) sets each axis valid or invalid.

AZR G28 command without setting the reference point

0: Use the deceleration link stopper to return the reference point, which is same as the manual reference point return.

1: P/S alarms

Remark:

When the reference point free of the link stopper (parameter DLZ (No.1002#1) is set as "1" or parameter DLZx (No.1005 #1) is "1", and there is no relation with the setting of AZR. P/S alarms if G28 is executed before setting the reference point.

	#7	#6	#5	#4	#3	#2	#1	#0
1004		RPR					ISC	

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

ISC setting the minimum input unit and the minimum command increment

Table 3-2

ISC	Min setting or traverse unit	Abbreviation
0	0.001mm, 0.001deg or 0.0001inch	IS-B
1	0.0001mm, 0.0001deg or 0.00001inch	IS-C

RPR whether set the minimum input unit of the rotation axis as 10 times of the minimum command increment

0: Not set it as 10 times

1: Set as it 10 times

	#7	#6	#5	#4	#3	#2	#1	#0
1005					HJZx		DLZx	ZRNx

『Parameter type』 : Bit axial type

『Default setting』 : 0000 1000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

ZRNx Whether the system alarms if the other traverse commands are specified except G28 before setting the reference point in auto running (MEM, DNC or MDI).

0: Alarm

1: Not alarm

DLZx Whether setting the reference point free of the link stopper is valid.

0: Invalid

1: Valid

Remark:

Parameter DLZ (No.1002#1) is valid when it is "0". When DLZ (No.1002#1) is "1", there is no connection with the parameter, and setting the reference point free of the link stopper is valid for all axes.

HJZx After the reference point is set, manually return to the reference point.

0: Use the deceleration link stopper to return to the reference point

1: No connection with the deceleration link stopper, rapidly position in the reference point.

	#7	#6	#5	#4	#3	#2	#1	#0
1006			ZMlx		DIAx		ROSx	ROTx

『Parameter type』 : Bit axial type

『Default』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

ROTx, ROSx sets linear axis or rotation axis

Table 3-3

ROSx	ROTx	Content
0	0	Linear axis Switch between metric and inch system. All coordinate values are the linear axial type. The screw pitch error compensation in memory type is the linear axial type.
0	1	Rotation axis (type A) It can't switch between metric and inch system. The machine coordinate value displays in 0~360° cycle. The screw pitch error compensation in memory type is the rotation axial type. Auto return to the reference point at the direction of the reference point return (G28 and G30), the traverse amount can not exceed one turn.
1	0	Invalid setting
1	1	Rotation axis (type B) It can't switch between metric and inch system. The machine coordinate value, the relative coordinate value and the absolute coordinate value are in the linear axial type, which can't display in cycle of 0~360°. The screw pitch error compensation of memory type is the linear axial type. The cycle function and the graduation function of the rotation axia can not be used at the same time.

DIAx sets the traverse amount of each axis

0: specified by the radius

1: specified by the diameter

ZMlx sets the direction of each axis reference point return

0: positive direction

1: negative direction

	#7	#6	#5	#4	#3	#2	#1	#0
1008						RRLx	RABx	ROAx

『Parameter type』 : Bit axial type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

ROAx sets whether the cycle display function of the rotation axis valid.

0: Invalid

1: Valid

Remark:

ROAx is just valid for the rotation axis and parameter ROTx (No.1006#0) must be 1.

RABx sets the rotation direction of the axis during the absolute command.

0: Rotation direction close to the target

1: Direction specified by the command value coder

Remark:

RABx is valid only when parameter ROAx is 1.

RRLx Relative coordinate

0: Not cycle as the movement amount of each turn

1: Cycle as the movement amount of each turn

Remark:

1. RRLx is valid only when ROAx is 1.

2. The movement amount of each turn is set by parameter No.1260.

1010

Quantity of CNC control axes (CCA)

『Parameter type』 : Character type

『Default setting』 : 2

『Range』 : 0~total number

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

Set the total number of axes which is directly controlled by CNC, the other can be controlled by PLC.

	#7	#6	#5	#4	#3	#2	#1	#0
1015	DWT	WIC						

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

WIC: the offset measured value of the work piece origin is directly input

0: Only valid for the selected work piece coordinate system

1: Valid for all coordinate systems

DWT When the pause time is specified by P, the data units are

0: IS-B is 1ms, IS-C is 0.1ms.

1: 1 ms

1020

Programmimg name of each axis (CAN)

『Parameter type』 : Character axial type

『Default setting』:

Table 3-4

Control axes numbers	Default values
1	88
2	90
3	0
4	0
5	0

『Range』 : 88 (X), 89 (Y), 90 (Z), 65 (A), 66 (B), 67 (C)

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the axial name of each control axis.

Remark: the same axial name can not be set.

The address used by the 2nd miscellaneous function can not be taken as the axial name.

1022

The property of each axis in the basic coordinate system

『Parameter type』 : Character axial type

『Default setting』:

Table 3-5

Control axes numbers	Default values
1	1
2	3
3	0
4	0
5	0

『Range』 : 0~7

『Authority to rewrite』 : System

『Valid method』 : Power on

『Additional requirements』 : None

To ensuer the planes of the arc interpolation, the tool offset and the tool nose radius, etc.

G17: X—Y plane

G18: Z—X plane

G19: Y—Z plane

Set each control axis as one of three basic axes---X, Y and Z axes in the basic coordinate system, or the parallel axis which is paralleled with these axes. Only one axis of the basic three axes can be set: X, Y and Z; the parallel axes can be set as two more axes (which is paralleled with the basic axis).

Setting value	Meaning
0	They are neither basic three axes nor the parallel axes,
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Parallel axis of X axis
6	Parallel axis of Y axis
7	Parallel axis of Z axis

1023**Servo axis number of each axis (NSA)**

『Parameter type』 : Character axial type

『Default setting』 :

Table 3-6

Control axes numbers	Default values
1	1
2	2
3	3
4	4
5	5

『Range』 : 1~quantity of control axes

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

Set each control axis as the corresponding Nth servo axis. Generally, the setting value of the control axial number and that of the servo axial number are same. The so-called control axis number is to set parameter in the axial type or the serial number of the signal in the axial type. When the spindle is taken as the control axis, it is set as 5.

3.4 Parameter about the coordinate system

	#7	#6	#5	#4	#3	#2	#1	#0
1201	WZR					ZCL		

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

ZCL After manually return to reference point, the part coordinate system

0: Not cancel

1: Cancel

WZR Work piece coordinate system during resetting

0: Not return to G54

1: Return to G54

	#7	#6	#5	#4	#3	#2	#1	#0
1202					RLC	G50	EWS	EWD

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : System

『Valid method』 : Valid immediately

『Additional requirements』 : None

EWD: The movement direction of the coordinate system caused by the external work piece origin offset amount

0: It is same as the direction specified by the external work piece origin offset amount.

1: It is opposite to the direction specified by the external work piece origin offset amount.

EWS: The work piece coordinate system movement amount and the external work piece zero point offset amount

0: Saved in each memorizer

1: Saved in one memorizer (the work piece coordinate system movement amount are same as the external work piece zero point offset amount)

G50: When G50 is commanded and the coordinate system is set,

0: Not alarm, but execute G50

1: P/S alarms (No.010), not execute G50

RLC After resetting, the part coordinate system

0: Not cancel

1: Cancel

1220

The origin offset amount of each axis external work piece coordinate system (EWO)

『Parameter type』 : Character axial type

『Default』 : 0

『Range』 : -9999 9999~9999 9999

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

This is one parameter to set the origin location of the work piece coordinate system (G54~G59) . The parameter is the valid common offset amount for all work piece coordinate system.

Table 3-7

Setting unit	IS-B	IS-C	Unit
Linear axis (input in metric system)	0.001	0.0001	mm
Linear axis (input in inch system)	0.0001	0.00001	inch
Rotation axis	0.001	0.0001	deg

1221	Origin offset amount of each axis in G54 work piece coordinate system (WO1)
1222	Origin offset amount of each axis in G55 work piece coordinate system (WO2)
1223	Origin offset amount of each axis in G56 work piece coordinate system (WO3)
1224	Origin offset amount of each axis in G57 work piece coordinate system (WO4)
1225	Origin offset amount of each axis in G58 work piece coordinate system (WO5)
1226	Origin offset amount of each axis in G59 work piece coordinate system (WO6)

『Parameter type』 : Character axial type

『Default setting』 : 0

『Range』 : -99 999 999~+99 999 999

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

This is one parameter to set the origin location of the work piece coordinate system (G54~G59).

The parameter is the valid common offset amount for all the work piece coordinate system.

Table 3-7

SETTING UNIT	IS-B	IS-C	UNIT
Linear axis (input in metric system)	0.001	0.0001	mm
Linear axis (input in inch system)	0.0001	0.00001	inch
Rotation axis	0.001	0.0001	deg

1240	Each axis machine coordinate value of the 1 st reference point (RF1)
1241	Each axis machine coordinate value of the 2 nd reference point (RF2)
1242	Each axis machine coordinate value of the 3 rd reference point (RF3)
1243	Each axis machine coordinate value of the 4 th reference point (RF4)

『Parameter type』 : Character axial type

『Default setting』 : 0

『Range』 : -99 999 999~+99 999 999

『Authority to rewrite』 : Machine manufacture

『Valid method』 : 1240 valid after power on; 1241~1243 valid immediately.

『Additional requirements』 : None

Set the coordinate values from the 1st to the 4th reference points in the mechanical coordinate system

Table 3-8

SETTING UNITS	IS-B	IS-C	UNITS
Machine in metric system	0.001	0.0001	mm
Machine in inch system	0.0001	0.00001	inch
Rotation axis	0.001	0.0001	deg

1260	Each turn movement amount of each axis in rotation axis (PRA)
------	---

『Parameter type』 : Character axial type

『Default setting』 :

Table 3-9

SETTING UNIT	IS-B	IS-C
DATA UNITS	0.001 deg	0.0001 deg
DEFAULT SETTING	360 000	360 0000

『Range』 : 1 000~9 999 999

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

Set the movement amount of each turn in rotation axis.

3.5 Parameter about the stroke detection

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA	LZR	RL3			LMS		OUT

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : User

『Valid method』 : Valid immediately

『Additional requirements』 : None

OUT The restricted area of the stroke detection 2 in memory type is set by parameters (No.1322 or No1323) .

0: Internal area

1: External area

LMS Whether the switching signal EXLM of the stroke detection in memory type is valid

0: Invalid

1: Valid

Remark:

Stroke detection 1 in memory type possesses the parameter of the restricted area set by two groups, signals are switched through the stroke limit in memory type and the set restricted area is selected.

(1) Restricted area I: Parameter No.1320 or No.1321

(2) Restricted area II: Parameter No.1326 or No.1327

RL3: Whether it is valid that the stroke detection 3 releases signal RLS0T3

0: Invalid

1: Valid

LZR: After power on before manual reference point return whether detect the stroke 1 in the memory type

0: Detect

1: Not detect

Remark:

There isn't any connection with the setting when the absolute position encoder is being using, the power is on and the reference point is set. After power on, the stroke is directly detected in memory type.

BFA When the command of overrun memory is sent

0: Alarm after overrun

1: Alarm before overrun

Remark:

The tool stops before or after the maximum distance F/7500 (mm) far away from the boundary. (F: Feedrate during reaching the boundary (Unit: mm/min))

	#7	#6	#5	#4	#3	#2	#1	#0
1310							OT3x	OT2x

『Parameter type』 : Bit axial type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : System

『Valid method』 : Valid immediately

『Additional requirements』 : None

OT2X: Whether each axis detects the stroke 2 in memory type

0: Not detect

1: Detect

OT3X Whether detect the stroke 3 in memory type in each axis

0: Not detect

1: Detect

1320	Coordinate value in positive direction boundary of each axis stroke detection 1 in memory type (PC1)
------	--

1321	Coordinate value in negative direction boundary of each axis stroke detection 1 in memory type (NC1)
------	--

『Parameter type』 : Character axial type

『Default setting』 : NO.1320 is 99 999 999, NO.1321 is -99 999 999

『Range』 : -99 999 999~99 999 999

『Authority to rewrite』 : User

『Valid method』 : valid immediately

『Additional requirements』 : None

Respectively set the coordinate values of boundaries in positive and negative directions in the mechanical coordinate system in each axis stroke detection 1 in memory type. Set the outside of boundary as the restricted area to tools.

Table 3-9

SETTING UNITS	IS-B	IS-C	UNIT
Machine in metric system	0.001	0.0001	Mm
Machine in inch system	0.0001	0.00001	Inch
Rotation axis	0.001	0.0001	Deg

Remark:

1. The axes specified by diameter are set by diameter value.
2. When (parameter No.1320) < (parameter No.1321) and the limit is infinite, it can not detect the stroke 1 in memory type. (The stroke limit switching signal in memory type is invalid.) If the absolute command is specified, the coordinate value may overflow, the normal movement can not be executed.
3. If parameter LMS (No. 1300#2) is "1", and the stroke limit switching signal in memory type EXLM is also "1", the restricted area is invalid set by the parameter. Parameter No.1326 and No.1327 set the restricted area.

1322**Coordinate value of each axis stroke 2 in memory type in positive direction boundary (PC2)****1323****Coordinate value of each axis stroke 2 in memory type in negative direction boundary (NC2)**

『Parameter type』 : Character axial type

『Default setting』 : NO.1322 is 99 999 999, NO.1323 is—99 999 999

『Range』 : —99 999 999~99 999 999

『Authority to rewrite』 : User

『Invalid method』 : Valid immediately

『Additional requirements』 : None

Respectively set the coordinate values of boundaries in positive and negative directions in the mechanical coordinate system in each axis stroke detection 2 in memory type. The outside or inside of boundary is the restricted area, which is set by parameter OUT (No.1300#0) .

Table 3-10

SETTING UNIT	IS-B	IS-C	UNIT
Machine in metric system	0.001	0.0001	mm
Machine in inch system	0.0001	0.00001	inch
Rotation axis	0.001	0.0001	deg

Remark:

The axis specified by diameter must be set by the diameter value.

1324**Coordinate value in positive direction boundary of each axis stroke detection 3 in memory type (PC3)****1325****Coordinate value in negative direction boundary of each axis stroke 3 in memory type (NC3)**

『Parameter type』 : Character axial type

『Default setting』 : NO.1324 is99 999 999, NO.1325 is —99 999 999

『Range』：—99 999 999~99 999 999

『Authority to rewrite』：User

『Valid method』：Valid immediately

『Additional requirements』：None

Respectively set the coordinate values of boundaries in positive and negative directions in the mechanical coordinate system in each axis stroke detection 3 in memory type. Set inside of the boundary as the restricted area to tools.

Table 3-11

SETTING UNITS	IS-B	IS-C	UNITS
Machine in metric system	0.001	0.0001	mm
Machine in inch system	0.0001	0.00001	inch
Rotation axis	0.001	0.0001	deg

Remark:

The axis specified by the diameter must be set by the diameter value.

1326

Coordinate value II in positive direction boundary of each axis stroke detection 1 in memory type (PC12)

1327

Coordinate value II in negative direction boundary of each axis stroke detection 1 in memory type (NC12)

『Parameter type』：Character axial type

『Default setting』：NO.1326 is 99 999 999, NO.1327 is —99 999 999.

『Range』：—99 999 999~99 999 999

『Authority to rewrite』：User

『Valid method』：Valid immediately

『Additional requirements』：None

Respectively set the positive and negative boundary coordinate values of each axis stroke detection 1 in memory type in the machine coordinate system. Set outside of the boundary as the restricted area. When parameter LMS (No.1300#2) is “1”, and the stroke limit switching signal EXLM (G7.6) in memory type is “1”, the restricted area is valid, but it is invalid if it is set by No.1320 and 1321.

Table 3-12

SETTING UNIT	IS-B	IS-C	Unit
Machine in metric system	0.001	0.0001	mm
Machine in inch system	0.0001	0.00001	inch
Rotation axis	0.001	0.0001	deg

Remark:

1. The axes programmed by the diameter must be set by the diameter value.
2. The parameter is invalid when parameter LMS (No.1320#2) is "0", or the stroke limit switching signal EXLM (G7.6) in the memory type is "0". Then, the restricted area set by parameter No.1320 or No. 1321 is valid.

3.6 Parameter about the feedrate

	#7	#6	#5	#4	#3	#2	#1	#0
1401		RDR	TDR					RPD

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : System

『Valid method』 : Valid immediately

『Additional requirements』 : None

RPD: Manually rapid run from power on to the reference point return

0: Invalid (JOG speed)

1: Valid

TDR: Dry run during thread cutting or tapping

0: Valid

1: Invalid

RDR: Execute the dry run toward the rapid running command

0: Invalid

1: Valid

	#7	#6	#5	#4	#3	#2	#1	#0
1402						JOV		

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : System

『Valid method』 : Valid immediately

『Additional requirements』 : None

JOV: JOG override

0: Valid

1: Invalid (fixed as 100%)

	#7	#6	#5	#4	#3	#2	#1	#0
1403	RTV							MIF

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : System

『Valid method』 : Power on

『Additional requirements』 : None

MIF: The minimum unit of F command (the cutting feedrate) of feeding/min

0: 1mm/min (input in metric system) or 0.01inch/min (input in inch system)

1: 0.001mm/min (input in metric system) or 0.00001inch/min (input in inch system)

RTV: During thread cutting cycle, the override of the tool run-out is

0: Valid

1: Invalid

	#7	#6	#5	#4	#3	#2	#1	#0
1404						F8A	DLF	

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : System

『Valid method』 : Valid immediately

『Additional requirements』 : None

DLF : After setting the reference point, manually return to the reference point

0: Move to the reference point (No.1420) at the rapid feedrate

1: Move to the reference point (No.1424) at the manual rapid feedrate

F8A 每分进给时的 F 指令范围 F command range feed/min

0: 按照参数 MIF (No.1403#0) 的设置 Set according to parameter MIF (No.1403#0)

1:

Table 3-13

SETTING UNITS	UNIT	IS-B	IS-C
Input in metric system	mm/min	0.001~60000	0.001~24000
Input in inch system	inch/min	0.00001~2400	0.00001~960
Rotation axis	deg/min	1~60000	1~24000

1410

Dry run speed (DRR)

『Parameter type』 : Character type

『Setting data』 :

Table 3-14

SETTING UNITS	DATA UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1mm/min	6~15000	6~12000	1000
Machine in inch system	0.1inch/min	6~6000	6~4800	

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the speed during dry run.

1411

Feedrate in auto mode after power on (IFV)

『Parameter type』 : Character type

『Range』 :

Table 3-15

SETTING UNITS	DATA UNITS	VALID RANGE	DEFAULT SETTING
Machine in metric system	1 mm/min	6~32767	1000
Machine in inch system	0.1 inch/min	6~32767	

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

It doesn't require changing the cutting speed in the machine during the processing. And the cutting feedrate can be set by the parameter, then the cutting feedrate is not required to be set in the program. But the actual feedrate is limited by parameter NO.1422 which set the maximum cutting feedrate for all axes.

1420

Each axis rapid movement speed (RTT)

『Parameter type』 : Character axial type

『Range』 :

Table 3-16

SETTING UNITS	DATA UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1 mm/min	30~60000	6~24000	8000
Machine in inch system	0.1 inch/min	30~24000	6~9600	
Rotation axis	1 deg/min	30~60000	6~24000	

『Authority to rewrite』 : Machine

『Valid method』 : Immediately

『Additional requirements』 : None

Set the rapid movement speed of each axis when the rapid movement override is 100%.

1421

F0 speed of each axis rapid override (F0R)

『Parameter type』 : Character axial type

『Range』 :

Table 3-17

SETTING UNITS	DATA UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1 mm/min	30~15000	30~12000	400
Machine in inch system	0.1 inch/min	30~6000	30~4800	
Rotation axis	1 deg/min	30~15000	30~12000	

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the speed when the rapid movement override of each axis is 0.

1422

Maximum cutting feedrate of all axes (MFR)

『Parameter type』 : Character type

『Range』 :

Table 3-18

SETTING UNITS	DATA UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1mm/min	6~60000	6~24000	8000
Machine in inch system	0.1inch/min	6~24000	6~9600	

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the maximum cutting feedrate for all axes.

1423

JOG feedrate of each axis (JFR)

『Parameter type』 : Character axial type

『Range』 :

Table 3-19

SETTING UNITS	DATA UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1mm/min	6~32767		1000
Machine in inch system	0.1inch/min			
Rotation axis	1 deg/min			

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the feedrate of each axis during continually manual feeding (JOG feeding), the actual feedrate is limited by parameter NO.1422 (the maximum cutting feedrate of all axes) .

1424

Manual rapid speed of each axis (MRR)

『Parameter type』 : Character axial type

『Range』 :

Table 3-20

SETTING UNITS	DATA UNIT	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1 mm/min	30~60000	30~24000	8000
Machine in inch system	0.1 inch/min	30~24000	30~9600	
Rotation axis	1 deg/min	30~60000	30~24000	

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the speed of each axis manual rapid movement when rapid movement override is 100%.
Set the maximum speed of MPG feeding.

Remark:
If it is set as 0, use the setting value of parameter 1420.

1425

FL speed of each axis reference point return (FLR)

『Parameter type』 : Character axial type

『Range』 :

Table 3-21

SETTING UNITS	DATA UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1 mm/min	6~15000	6~12000	200
Machine in inch system	0.1 inch/min	6~6000	6~4800	
Rotation axis	1 deg/min	6~15000	6~12000	

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

After deceleration, set the speed (FL speed) of each axis during the reference point return.

3.7 Parameter about control of acceleration and deceleration

	#7	#6	#5	#4	#3	#2	#1	#0
1601				RTO				

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : System

『Valid method』 : Valid immediately

『Additional requirements』 : None

RTO: During rapid running, the block is

0: No overlapping

1: Overlapping

	#7	#6	#5	#4	#3	#2	#1	#0
1610				JGLx				CTLx

『Parameter type』 : Bit axial type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : System

『Valid method』 : Valid immediately

『Additional requirements』 : None

CTLx: Acceleration and deceleration of cutting and feeding (include feeding during dry run)

0: Acceleration and deceleration in index type

1: Acceleration and deceleration in linear type after interpolation

JGLx: The acceleration and deceleration during JOG feeding

0: Acceleration and deceleration in index type

1: Acceleration and deceleration in linear type after interpolation

1620	Time constant T of linear acceleration and deceleration of each axis rapid movement (TT1)
-------------	--

『Parameter type』 : Character axial type

『Default setting』 : 100

『Range』 : 0~4000 ms

『Authority to rewrite』 : System

『Valid method』 Valid immediately

『Additional requirements』 : None

Set the time constant of acceleration and deceleration during rapid movement.

1622	Time constant of acceleration and deceleration during cutting and feeding after each axis interpolation (ATC)
-------------	--

『Parameter type』 : Character axial type

『Default setting』 : 100

『Range』 : 0~4000 ms

『Authority to rewrite』 : System
 『Valid method』 Valid immediately
 『Additional requirements』 : None

Set the acceleration and deceleration of each axis cutting and feeding in index type, or the time constant of acceleration and deceleration in linear type after interpolation. And the detailed type is set by parameter CTLx (NO.1610#0). If CTLx sets the acceleration and deceleration in linear type after linear interpolation, the maximum time constant of acceleration and deceleration is limited in 512ms and even it exceeds 512ms, it is still dealt as 512ms.

Except the special usage of the parameter, all axes must be set as the same time constant. If the different time constants are set, the correct linear or circular can't be shaped.

1623

**FL speed of acceleration and deceleration
in index type of each axis cutting and feeding (FLC)**

『Parameter type』 : Character axial type
 『Range』 :

Table 3-22

SETTING UNITS	DATA UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1 mm/min	0, 6~15000	0, 6~12000	30
Machine in inch system	0.1 inch/min	0, 6~6000	0, 6~4800	30
Rotation axis	1 deg/min	0, 6~15000	0, 6~15000	30

『Authority to rewrite』 : System
 『Valid method』 : Valid immediately
 『Additional requirements』 : None
 Set the low limit speed (FL speed) of acceleration and deceleration in index type of each axis cutting and feeding.

1624

**Time constant of acceleration and deceleration of
each axis JOG feeding after interpolation (JET)**

『Parameter type』 : Character axial type
 『Default setting』 : 100
 『Range』 : 0~4000ms
 『Authority to rewrite』 : System
 『Valid method』 : Valid immediately
 『Additional requirements』 : None

Set the acceleration and deceleration in index type of each axis JOG feeding, and the time constant of acceleration and deceleration in linear type after interpolation.

The detailed type is set by parameter JGLx (NO.1610#4). If JGLx sets the acceleration and deceleration in linear type after interpolation, the maximum time constant of acceleration and deceleration is limited in 512ms and even it exceeds 512ms, it is dealt as 512ms.

1625

**FL speed of acceleration and deceleration
in index type during each axis JOG feeding (FLJ)**

『Parameter type』 : Character axial type

『Range』 :

Table 3-23

SETTING UNITS	DATA UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1 mm/min	0, 6~15000	0, 6~12000	30
Machine in inch system	0.1 inch/min	0, 6~6000	0, 6~4800	30
Rotation axis	1 deg/min	0, 6~15000	0, 6~15000	30

『Authority to rewrite』 : System

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the low limit speed (FL speed) of acceleration and deceleration in index type during each axis JOG feeding.

1626

**Time constant of acceleration and deceleration
during each axis thread cutting cycle (TET)**

『Parameter type』 : Character axial type

『Default setting』 : 100

『Range』 : 0~4000ms

『Authority to rewrite』 : System

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the time constant of acceleration and deceleration in linear and index types during each axis thread cutting cycle.

1627

**FL speed of acceleration and deceleration
in index type during each axis thread cutting cycle (FLT)**

『Parameter type』 : Character axial type

『Range』 :

SETTING UNITS	DATA UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1 mm/min	0, 6~15000	0, 6~12000	30
Machine in inch system	0.1 inch/min	0, 6~6000	0, 6~4800	30

『Authority to rewrite』 : System

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set low limit speed (FL speed) of acceleration and deceleration in index type during each axis thread cutting cycle.

3.8 Parameter about servo and backlash compensation

	#7	#6	#5	#4	#3	#2	#1	#0
1800	BDEC	BD8		RBK				

『Parameter type』 : Bit type

『Default setting』 : 1000 0000

『Range』 : 0 or 1

『Authrity to rewrite』 : System

『Valid method』 : Valid immediately

『Additional requirements』 : None

RBK: Cutting feeding and rapid movement are respectively compensated with backlash

0: No compensate

1: Compensate

BD8: Impulse output frequency of the backlash compensation

0: Compensate at the frequency set by parameter #1853

1: Compensate at 1/8 of frequency set by parameter #1853

BDEC: Backlash compensation mode

0: fixed pulse frequency output , which is set by parameter #1853 and #1800.6.

1: Pulse frequency output based on the acceleration and deceleration characteristics

	#7	#6	#5	#4	#3	#2	#1	#0
1811						POD		ABP

『Parameter type』 : Bit axial type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : System

『Valid method』 : Power on

『Additional requirements』 : None

ABP: Selecting pulse drive modes

0: Pulse +direction mode

1: AB phases pulse mode

POD: Selecting output directions of each axis pulse

0: Not negate

1: Negate

	#7	#6	#5	#4	#3	#2	#1	#0
1815			APCx	APZx				APRx

『Parameter type』 : Bit axis type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : System

『Valid method』 : Power on

『Additional requirements』 : None

APRx: The position direction of the absolute position detector during using the absolute position encoder

0: Not negate

1: Negate

APZx: The mechanical position and the absolute position detector position during using the absolute position detector

0: Not consistency

1: Consistency

Remark:

When use the absolute position detector, during the initial setting or after changing the absolute position encoder, the parameter must be set as 0, and connect power supply, again after power off and manually return to the reference point. Therefore, the mechanical position consists with that of the position encoder, and the parameter will be auto set as 1.

APCx Position encoder

0: Not use the absolute position detector

1: Use the absolute position detector (the absolute pulse encoder)

	#7	#6	#5	#4	#3	#2	#1	#0
1816		DM3x	DM2x	DM1x				ISAx

『Parameter type』 : Bit axial type

『Default』 : 0001 0001

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

ISAx Servo alarm signal

0: High level of alarm signal is valid

1: Low level of alarm signal is valid

DM1x-DM3x: Set the detection multiply ratio of each axis(DMR)

SETTING VALUE			DETECTION MULTIPLY RATIO (DMR)
DM3x	DM2x	DM1x	
0	0	0	1/2
0	0	1	1
0	1	0	3/2
0	1	1	2
1	0	0	5/2

1	0	1	3
1	1	0	7/2
1	1	1	4

1820**Command multiply ratio of each axis (CMR)**

『Parameter type』 : Character axial type

『Range』 :

COMMAND MULTIPLY RATIO (CMR)	VALID RANGE OF VALUE SET BY NO.1820	DEFAULT SETTING
1/2~1/27	102~127	2
1 ~ 48	2~96	

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements 』 : None

Set the command multiply ratio (CMR) of each axis.

1. When the command multiply ratio (CMR) is 1/2~1/27, the setting value = 1 / CMR+100;
2. When the command multiply ratio (CMR) is 1~48, the setting value = 2×CMR.

Gear ratio output by each axis=CMR/ DMR

Detection unit=minimum movement unit/ CMR

The relations between the setting units and the minimum movement units:

			MINIMUM SETTING UNITS	MINIMUM MOVEMENT UNITS
IS—B	Machine in metric system	Input in metric system	0.001mm（specified by the diameter）	0.0005mm
			0.001mm（specified by the radius）	0.001mm
		Input in inch system	0.0001 inch（specified by the diameter）	0.0005mm
			0.0001 inch（specified by the radius）	0.001mm
	Machine in inch system	Input in metric system	0.001mm（specified by the diameter）	0.00005 inch
			0.001mm（specified by the radius）	0.0001 inch
		Input in inch system	0.0001 inch（specified by the diameter）	0.00005 inch
			0.0001 inch（specified by the radius）	0.0001 inch
	Rotation axis		0.001deg	0.001deg

			MINIMUM SETTING UNITS	MINIMUM SETTING UNITS
IS—C	Machine in metric system	Input in metric system	0.0001mm (specified by the diameter)	0.00005mm
			0.0001mm (specified by the radius)	0.0001mm
		Input in inch system	0.00001 inch (specified by the diameter)	0.00005mm
			0.00001 inch (specified by the radius)	0.0001mm
	Machine in inch system	Input in metric system	0.0001mm (specified by the diameter)	0.000005 inch
			0.0001mm (specified by the radius)	0.00001 inch
		Input in inch system	0.00001 inch (specified by the diameter)	0.000005 inch
			0.00001 inch (specified by the radius)	0.00001 inch
	Rotation axis		0.0001deg	0.0001deg

1851

Backlash compensation value of each axis (BCV)

『Parameter type』 : Character axial type
 『Default setting』 : 0
 『Range』 : -9999~+9999 (Detection unit)
 『Authority to rewrite』 : Machine
 『Valid method』 : Valid immediately
 『Additional requirements』 : None

Set the backlash compensation value of each axis.

After connecting power supply, it compensates the backlash at the first time when the machine moves in the direction opposite with that of the reference point return.

Decetion units are related with parameter No.1820 (command multiply ratio CMR) and the minimum movement units, about the relations between the setting units and the minimum movement units, refer to parameter No.1820 introduction.

1852

Backlash compensation value during each axis rapid movement (BCVR)

『Parameter type』 : Character axial type
 『Default setting』 : 0
 『Range』 : -9999~+9999 (Detection units)
 『Authority to rewrite』 : Machine
 『Valid method』 : Valid immediately
 『Additional requirements』 : None

Set the backlash compensation value during each axis rapid movement. It is valid when parameter NO.1800#4(RBK) is set as 1. It can change the backlash compensation value based on the cutting feedrate/rapid movement speed to process in higher precision.

Remark:

1. Manually continuous feeding (JOG) is taken as cutting feed.
2. After connecting power supply and before the reference point return completes at the first time, it doesn't compensate the backlash in cutting feed/rapid movement. No matter the compensation value is the cutting feed or the rapid movement, it should be compensated based on parameter NO.1851.
3. When parameter NO.1800#4(RBK) is set as 1, parameter NO.1851 is the backlash compensation value of cutting feed, parameter NO.1852 is the backlash compensation value of rapid movement. When parameter NO.1800#4(RBK) is set as 0, parameter NO.1851 is the backlash compensation value of cutting feed/rapid movement.

	#7	#6	#5	#4	#3	#2	#1	#0
1853				CPF5	CPF4	CPF3	CPF2	CPF1

『Parameter type』 : Bit type

『Default』 : 0000 0111

『Range』 : 0 or 1

『Authority to rewrite』 : System

『Valid method』 : Valid immediately

『Additional requirements』 : None

CPF1~CPF5:Setting value of the backlash compensation pulse frequency (in BCD code)

Setting frequency= (setting value +1) Kpps

CPF5	CPF4	CPF3	CPF2	CPF1	SETTING FREQUENCY (Kpps)
0	0	0	0	0	1
0	0	0	0	1	2
0	0	0	1	0	3
0	0	0	1	1	4
0	0	1	0	0	5
0	0	1	0	1	6
0	0	1	1	0	7
0	0	1	1	1	8
0	1	0	0	0	9
0	1	0	0	1	10
0	1	0	1	0	11
0	1	0	1	1	12
0	1	1	0	0	13
0	1	1	0	1	14
0	1	1	1	0	15
0	1	1	1	1	16
1	0	0	0	0	17

1	0	0	0	1	18
1	0	0	1	0	19
1	0	0	1	1	20
1	0	1	0	0	21
1	0	1	0	1	22
1	0	1	1	0	23
1	0	1	1	1	24
1	1	0	0	0	25
1	1	0	0	1	26
1	1	0	1	0	27
1	1	0	1	1	28
1	1	1	0	0	29
1	1	1	0	1	30
1	1	1	1	0	31
1	1	1	1	1	32

2071

Each axis backlash acceleration and deceleration valid time constant (BAT)

『Parameter type』 : Character axial type

『Default setting』 : 40

『Range』 : 0~100 ms

『Authority to rewrite』 : System

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set each axis backlash acceleration and deceleration valid time constant.

3.9 Parameter about DI/DO

#7 #6 #5 #4 #3 #2 #1 #0

3003

ESP

『Parameter type』 : Bit type

『Default setting』 : 1000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

ESP: External emergency stop alarm input signal (X0.5)

0: When the signal is 0 (low level), emergency stop alarms

1: When the signal is 1 (high level), emergency stop alarms

#7 #6 #5 #4 #3 #2 #1 #0

3004

OTH

『Parameter type』 : Bit type

『Default setting』 : 0010 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid Immediately

『Additional requirements』 : None

OTH: Overtravel limit signal

0: Check

1: Not check

	#7	#6	#5	#4	#3	#2	#1	#0
3006								GDC

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

GDC: Deceleration signal of the reference point return

0: Use X signal

1: Use G196 (X signal is invalid)

	#7	#6	#5	#4	#3	#2	#1	#0
3009			DECx					

『Parameter type』 : Bit axial type

『Default setting』 : 0010 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

DECx: Deceleration signal of the reference point return

0: When the signal is 0 (low level), decelerate.

1: When the signal is 1 (high level), decelerate.

3010	Dwell time of the gating signals MT, TF and SF(MFT)
-------------	--

『Parameter type』 : Bit type

『Default setting』 : 16

『Range』 : 16 ms~32767 ms

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the time from sending codes M, S, T and B, till MF, SF, TF and BF being sent.

3011	Minimum width (MAW)of finish signals (FIN) of M, T and S
-------------	---

『Parameter type』 : Character type

『Default setting』 : 16

『Range』 : 16~32767 ms

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the minimum width of the finish signals (FIN) of M, S, T and B function.

Remark:

Time is set by 8ms, if the setting value is not the multiple of 8, it should be carried into the multiple of 8.

3017

Output time of the resetting signal (RST)

『Parameter type』 : Character type

『Default setting』 : 32

『Range』 : 0~255

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the dwell time when the resetting signal RST is output.

RST signal output time =resetting time + the parameter value X 16ms.

3030

Allowable digits of M code (MCB)

『Parameter type』 : Character type

『Default setting』 : 2

『Range』 : 2~8

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the allowable digits of M code.

3031

Allowable digits of S code (SCB)

『Parameter type』 : Character type

『Default setting』 : 4

『Range』 : 1~5

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the allowable digits of S code.

Maximum 5 digits in S code is allowed.

3032

Allowable digits of T code (TCB)

『Parameter type』 : Character type

『Default setting』 : 4

『Range』 : 2~8

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the allowable digits of T code.

3.10 Parameter about display and editing

	#7	#6	#5	#4	#3	#2	#1	#0
3101				BGD				

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

BGD: Background editing selects the programs selected at the foreground

0: Editable

1: Uneditable

	#7	#6	#5	#4	#3	#2	#1	#0
3102					CHI			

『Parameter type』 : Bit type

『Default setting』 : 0000 1000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

CHI: Display language

0: English

1: Chinese

Set the selected language for display.

	#7	#6	#5	#4	#3	#2	#1	#0
3104	DAC	DAL	DRC	DRL				MCN

『Parameter type』 : Bit type

『Default setting』 : 1100 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

MCN: Display the machine position

0: Display based on the output units

(There isn't any connection with the metric system or the inch system, the metric machine displays as the metric units, the inch machine displays as the inch units.)

1: Display based on the input units

(When it is input in the metric system, display in the metric system; when it is input in the inch system, display in the inch system)

DRL: Display the relative position

0: Display the actual position including the tool offset (T serial)

1: Display the programming position without the tool offset (T serial)

Remark:

In T serial, the movement coordinate system compensates the tool appearance, (parameter LGT (NO.5002#4) is 0), display the programming position which ignores the tool compensation (the parameter is set as 1). However, the programming position without the tool appearance compensation value can not display.

DRC: Display the relative position

0: Display the actual position including the tool nose radius compensation (T serial)

1: Display the programming position without the tool nose radius compensation (T serial)

DAL: Display the absolute position

0: Display the actual position including the tool offset (T serial)

1: Display the programming position without the tool offset (T serial)

Remark:

In T serial, the movement coordinate system compensates the tool appearance (parameter LGT (NO.5002#4) is 0), and display the programming position which ignores the tool compensation (the parameter is set as 1). However, the programming position without the tool appearance compensation value can not display.

DAC: Display the absolutely position

0: Display the actual position including the tool nose radius compensation (T serial)

1: Display the programming position without the tool nose radius compensation (T serial)

	#7	#6	#5	#4	#3	#2	#1	#0
3107				SOR	REV	DNC		

『Parameter type』 : Bit type

『Default setting』 : 0001 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

DNC: Whether clear display of DNC running programs during resetting

0: Not clear

1: Clear

REV: Display the actual speed in feeding/rev mode

0: mm/min or inch/min

1: mm/rev or inch/rev

SOR: Display orders of program directory

0: Based on the time sequency

1: Based on the program numbers

	#7	#6	#5	#4	#3	#2	#1	#0
3110						AHC		

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

AHC: Whether the alarm record can be cleared by keys

0: Yes

1: No

	#7	#6	#5	#4	#3	#2	#1	#0
3114								IPC

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

IPC : On the current interface, press the function keys

0: Switch into the interface

1: Not switch into the interface

	#7	#6	#5	#4	#3	#2	#1	#0
3202			CPD					

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

CPD When NC program is deleted, confirm information and keys

0: Not display

1: Display

	#7	#6	#5	#4	#3	#2	#1	#0
3203	MCL	MER						

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

MER: When the single block runs in MDI mode, after the last block is executed in the program, whether the excuted programs are

0: Not deleted

1: Deleted

Remark:

Even MER is 0, when “%” (end code) is read in and executed, the program is also deleted (“%”is auto inserted at the end of the program).

MCL: Whether delete the programs edited in MDI mode through resetting

0: Not delete

1: Delete

	#7	#6	#5	#4	#3	#2	#1	#0
3209								MPD

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

MPD: When the subprogram is executed, whether display the main program number

0: Not display

1: Display

3216	the increment value (INC) during the serial number being auto inserted							
-------------	---	--	--	--	--	--	--	--

『Parameter type』 : Character type

『Default setting』 : 10

『Range』 : 0~9999

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

When the serial number (parameter SEQ(NO.0000#5) is 1) is auto inserted, it is the increment value of the serial number in each block.

3.11 Parameter about programming

	#7	#6	#5	#4	#3	#2	#1	#0
3401						NCK		DPI

『Parameter type』 : Bit type

『Default setting』 : 0000 0001

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid immediately』 : Valid immediately

『Additional requirements』 : None

DPI: The address is with the decimal point, but when the decimal point is omitted, the setting is as below:

- 0: Take them as the minimum setting units
- 1: Take them as the units of mm, inch and sec

NCK: During grammar checking, there are same N numbers

- 0: Alarm
- 1: Not alarm

	#7	#6	#5	#4	#3	#2	#1	#0
3402	G23	CLR		FPM				G01

『Parameter type』 : Bit type

『Default setting』 : 0001 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

G01: Mode during connecting the power supply

- 0: G00 mode (orientation)
- 1: G01 mode (linear interpolation)

FPM: System defaults after power on

- 0: Feeding/rev
- 1: Feeding/min

CLR: Press the resetting key on MDI panel, the external resetting signal and the emergency stops, G code mode and the feedrate are

- 0: Hold mode
- 1: Switched to the power on state

G23 When the power supply is connected, it is

- 0: G22 mode (Check the memory stroke)
- 1: G23 mode (Not check the memory stroke)

	#7	#6	#5	#4	#3	#2	#1	#0
3403		AD2	CIR	RER				

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

RER: During arc interpolation, when R goes over the minor finishing point and isn't in the arc, and the radius doesn't exceed error:

- 0: Calculate the new radius, the path is semicircle
- 1: P/S alarms

CIR: In arc interpolation commands (G02,G03), there are no distance (I, J, K) from the starting point of the command to the center, and the arc radius isn't commanded, either.

- 0: Linear interpolation moves to the finishing point
1: P/S alarms

AD2: In one block, two or two more same addresses are commanded

- 0: The following commands are valid.
1: The program is taken as wrong, P/S alarms.

Remark:

It alarms when the parameter is 1 and two or two more G codes of one group are commanded in one block.

	#7	#6	#5	#4	#3	#2	#1	#0
3404	M3B	EOR	M02	M30				

『Parameter type』: Bit type

『Default setting』: 0000 0000

『Range』: 0 or 1

『Authority to rewrite』: Machine

『Valid method』: Valid immediately

『Additional requirements』: None

M30: During auto running, process M30 command

- 0: M30 auto searches the program header when it is sent to the machine side. Therefore, the program is executed from the beginning when the resetting isn't executed or return to the resetting and the finish signal FIN returns.
1: M30 is sent to the machine side, but it doesn't return to the beginning of the program.

M02 During auto running, process M02 command

- 0: M02 auto searches the program header when it is sent to the machine side. The program is executed from the beginning when resetting isn't executed or resetting returns and finish signal FIN returns.
1: M02 is sent to the machine side, but it doesn't return to the beginning of the program.

EOR : During executing the program, read in “%” (program end)

- 0: P/S alarms (stop auto running, display alarm state)
1: Not alarm (auto running stops, the system resets)

M3B: The quantity of M codes which can be commanded in one block

- 0: One
1: Maximum three

3410

Circular radius allowable error(CRE)

『Parameter type』: Character type

『Default setting』: 0

『Range』: 0~9999 9999

SETTING UNITS	IS—B	IS—C	UNITS
input in mm	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch

『Authority to rewrite』: Machine

『Valid method』: Valid immediately

『Additional requirements』 : None

Set the allowable error value of arc interpolation (G02, G03) starting point radius and its finishing point radius. P/S alarms when arc interpolation radius error is more than the limit value.

Remark:

When the setting value is 0, it doesn't require checking the arc radius error.

3.12 Parameter about the screw pitch error compensation

3620

**Screw pitch error compensation number
in each axis reference point (NPR)**

『Parameter type』 : Character axial type

『Default setting』 : 0

『Range』 : 0~1023

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

3621

**Number of the furthest screw pitch error compensation
point of each axis in negative direction (NEN)**

『Parameter type』 : Character axial type

『Default setting』 : 0

『Range』 : 0~1023

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

The parameter sets the number of the furthest screw pitch error compensation point of each axis in negative direction.

3622

**Number of the furthest screw pitch error compensation
point of each axis in positive direction (NEP)**

『Parameter type』 : Character axial type

『Default setting』 : 0

『Range』 : 0~1023

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

The parameter sets the number of the furthest screw pitch error compensation point of each axis in positive direction.

The parameter setting value should be greater than that of parameter NO.3620.

3623

Each axis screw pitch error compensation override (PCM)

『Parameter type』 : Character axial type

『Default setting』 : 0

『Range』 : 0~100

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

Set the override of each axis screw pitch error compensation.

If the override is set as 1, the detection unit is same as that of compensation.

If the override is set as 0, the override is same as one when it is set as 1.

3624

Each axis screw pitch error compensation point interval (PCI)

『Parameter type』 : Character axial type

『Default setting』 : 0

『Range』 : 0~99 999 999

SETTING UNITS	IS—B	IS—C	UNITS
Input in metric system	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch
Revolving axis	0.001	0.0001	deg

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

The screw pitch compensation points are distributed in equal interval, and the interval value of each axis is set respectively. The minimum value of the interval is limited and set by the following formula: the minimum value = the maximum feedrate (rapid feedrate) / 7500.

Unit: Screw pitch compensation minimum interval: mm, inch and deg.

Maximum feedrate: mm/min, inch/min and deg/min.

For example: When the maximum feedrate is 15000mm/min, the minimum value of the screw pitch error compensation interval is 2mm.

But, according to the setting override, when the absolute value of the compensation point value exceeds 100, the interval of the compensation point is magnified by the override which is calculated by the following formula.

Override = Max compensation amount (absolute value)/128 (round up the digits after the decimal point)

Screw pitch compensation minimum interval = Value, which is obtained from the above maximum feedrate X override.

Remark:

The unit of the screw pitch compensation value is same as that of the detection.

The detection unit is relative with parameter No.1820 (command magnify ratio CMR) and the minimum movement unit, about the relation between the setting units and the minimum movement units, refer to the introduction of parameter No.1820.

	#7	#6	#5	#4	#3	#2	#1	#0
3628				NPF5	NPF4	NPF3	NPF2	NPF1

『Parameter type』 : Bit type

『Default setting』 : 0000 0111

『Range』 : 0 or 1

『Authority to rewrite』 : System

『Valid method』 : Valid immediately

『Additional requirements』 : None

NPF1~NPF5: Setting value of the screw pitch compensation pulse frequency (in BCD code).

Setting frequency=(setting value +1) Kpps

NPF5	NPF4	NPF3	NPF2	NPF1	Setting frequency (Kpps)
0	0	0	0	0	1
0	0	0	0	1	2
0	0	0	1	0	3
0	0	0	1	1	4
0	0	1	0	0	5
0	0	1	0	1	6
0	0	1	1	0	7
0	0	1	1	1	8
0	1	0	0	0	9
0	1	0	0	1	10
0	1	0	1	0	11
0	1	0	1	1	12
0	1	1	0	0	13
0	1	1	0	1	14
0	1	1	1	0	15
0	1	1	1	1	16
1	0	0	0	0	17
1	0	0	0	1	18
1	0	0	1	0	19
1	0	0	1	1	20
1	0	1	0	0	21
1	0	1	0	1	22
1	0	1	1	0	23
1	0	1	1	1	24
1	1	0	0	0	25
1	1	0	0	1	26
1	1	0	1	0	27
1	1	0	1	1	28
1	1	1	0	0	29
1	1	1	0	1	30
1	1	1	1	0	31
1	1	1	1	1	32

3.13 Parameter about the spindle control

	#7	#6	#5	#4	#3	#2	#1	#0
3705				EVS				

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

EVS: For S command, use spindle control function (spindle analog output or spindle serial output)

0: Not output S code and SF

1: Output S code and SF

	#7	#6	#5	#4	#3	#2	#1	#0
3706							PG2	PG1

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

PG2 and PG1: Gear ratio between the spindle and the position encoder.

Gear ratio=spindle speed/position encoder speed

Gear ratio	PG2	PG1
×1	0	0
×2	0	1
×4	1	0
×8	1	1

	#7	#6	#5	#4	#3	#2	#1	#0
3707							P22	P21

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

P22 and P21: Gear ratio between the spindle and the second position encoder.

Gear ratio= spindle speed/position encoder speed

Gear ratio	P22	P21
×1	0	0
×2	0	1
×4	1	0
×8	1	1

Remark:
The parameter is valid only when multi-spindle control.

	#7	#6	#5	#4	#3	#2	#1	#0
3708		TSO					SAT	SAR

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

SAR: Whether check the spindle speed reaching signal

0: Not check

1: Check

SAT: Whether check the spindle speed reaching signal when the thread cutting block is begun to be executed.

0: Check or not, which is set by parameter SAR (NO.3708#0)

1: Must check, which isn't connected with parameter SAR

Remark:
When the thread cutting block is continually executed, the spindle speed reaching signal isn't checked in the thread cutting block after the 2nd block.

TSO: Whether the spindle override is valid during thread processing or tapping cycle

0: Invalid (fixed as 100%)

1: Valid

Remark:
In rigid tapping, the override is fixed as 100%, and there isn't any connection with the setting of the parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3709						MSI		SAM

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

SAM: Times of sampling in spindle average speed

- 0: Four times (Generally it is set as 0)
- 1: One time

MSI: SIND signal is valid during multi-spindle control

- 0: It is only valid for the 1st spindle .(SIND signal of the 2nd spindle becomes invalid.)
- 1: No matter whether each spindle is selected or not, it is valid for all spindles. (Each spindle has its own SIND signal.)

3730

Increment adjustment data of the spindle speed analog output (AGS)

『Parameter type』 : Character type

『Default setting』 : 1000

『Data unit』 : 0.1%

『Range』 : 700~1250

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the increment adjustment data of the spindle speed analog output. (Adjusting method)

- (1) Set the standard setting value 1000,
- (2) Command the spindle speed when the spindle speed analog output maximum voltage is 10V.
- (3) Measure the output voltage.
- (4) Set the value in the following formula in parameter No.3730:

$$\text{Setting value} = \frac{10 (\text{V})}{\text{Measured voltage (V)}} \times 1000$$

- (5) After setting the parameter, command the spindle speed analog output as the spindle speed of the maximum voltage, again, and confirm the output voltage as 10V.

3731

Compensation value of the spindle speed analog output offset voltage (CSS)

『Parameter type』 : Character type

『Default setting』 : 0

『Range』 : -1024~+1024

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

The parameter sets the compensation value of the spindle speed analog output offset voltage.

1. Set the standard setting value as 0.
2. Command the analog output voltage as 0V, which is the theoretical spindle speed.
3. Measure the output voltage.
4. Set the value in the following formula in parameter No.3731.

$$\text{Setting value} = \frac{-8191 \times \text{offset voltage (V)}}{12.5}$$

5. After setting the parameter, command the analog output voltage as 0V, again, which is the theoretical spindle speed and confirm the voltage as 0V.

3740	Dwell time of the detection spindle speed reaching signal (SAD)
-------------	--

『Parameter type』 : Character type

『Default setting』 : 6000

『Range』 : 0~255ms

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the dwell time from executing S function to detecting the spindle speed reaching signal.

3741	Spindle maximum speed of gear 1 (MSG1)
-------------	---

3742	Spindle maximum speed of gear 2 (MSG2)
-------------	---

3743	Spindle maximum speed of gear 3 (MSG3)
-------------	---

3744	Spindle maximum speed of gear 4 (MSG4)
-------------	---

『Parameter type』 : Character type

『Default setting』 : 6000

『Range』 : 0~32767r/min

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

The parameter sets the spindle maximum speed of each gear.

3770	Axis as the calculation reference during the constant surface speed control (ACS)
-------------	--

『Parameter type』 : Character type

『Default setting』 : 0

『Range』 : 0, 1~quantity of the control axes

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

The parameter sets the axis as the calculation reference during the constant surface speed control.

Remark:

When it is set as 0, default X axis. Then, P value commanded in G96 block is not significant to the constant surface speed.

3771

**Constant surface speed control mode (G96)
spindle minimum speed (CFL)**

『Parameter type』 : Character type

『Default setting』 : 0

『Range』 : 0~32767r/min

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

The parameter sets the spindle minimum speed when the constant surface speed control. During the constant surface speed control (G96) , if the spindle speed is lower than the speed set by the parameter, it is limited in the parameter speed.

3772

Maximum spindle speed (MSS)

『Parameter type』 : Character type

『Default setting』 : 6000

『Range』 : 0~32767r/min

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

The parameter sets the maximum spindle speed. The actual spindle speed is limited by the maximum speed set by the parameter when the commanded spindle speed exceeds the maximum spindle speed, or the spindle speed after override exceeds the maximum spindle speed.

Remark:

1. When the constant surface speed controls, no matter whether G96 or G97 is commanded, the spindle speed is limited by the maximum spindle speed.
2. When the setting value is 0, it is not limited by the speed.
3. When PLC controls the spindle speed, the parameter is invalid and the spindle speed isn't limited by the maximum speed.
4. When multi-spindle control, the maximum speed of each spindle is set through the following parameters:
The maximum speed of the 1st spindle is set by parameter NO.3772.
The maximum speed of the 2nd spindle is set by parameter NO.3802.

3773

Quantity of the spindle encoder pulses (CNT)

『Parameter type』 : Character type

『Default setting』 : 1024

『Range』 : 100~9999

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

The parameter sets the quantity of the spindle encoder pulses.

3802

Maximum speed of the 2nd spindle (MSS2)

『Parameter type』 : Character type

『Default setting』 : 6000

『Range』 : 0~32767r/min

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

The parameter sets the maximum speed of the 2nd spindle. The actual spindle speed is limited by the maximum speed set by the parameter when the commanded spindle speed exceeds the maximum spindle speed, or the spindle speed after override exceeds the maximum spindle speed.

Remark:

1. When the multi-spindle controls, the parameter is valid.
2. When the constant surface speed controls, no matter whether G96 or G97 is commanded, the spindle speed is limited by the maximum speed.
3. When the setting value is 0, parameter NO.3772 is valid (the maximum speed of the 1st spindle). When parameter NO.3772 is 0, the spindle speed is not limited.
4. When PLC controls the spindle speed, the parameter is invalid and the spindle speed isn't limited by the maximum speed.

3803

Quantity of the 2nd spindle encoder pulses (CNT2)

『Parameter type』 : Character type

『Default setting』 : 1024

『Spna』 : 100~9999

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

The parameter sets the quantity of the 2nd spindle encoder pulses.

3811

Spindle maximum speed of the 2nd spindle gear 1 (M2G1)

3812

Spindle maximum speed of the 2nd spindle gear 2 (M2G2)

『Parameter type』 : Character type

『Default setting』 : 6000

『Range』 : 0~32767r/min

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

The parameter sets the maximum speed of each gear in the 2nd spindle.

Remark:
It is for multi-spindle control.

3.14 Parameter about the tool compensation

	#7	#6	#5	#4	#3	#2	#1	#0
5001		EVO		EVR				

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

EVR: In tool nose compensation mode C, when the tool compensation value is changed

0: It becomes valid from the next block which specifies T code.

1: It becomes valid from the next buffer block.

EVO: The rewritten value becomes valid when the compensation value of the tool position compensation mode is changed.

0: It is valid from the next block which specifies T code.

1: It is valid from the next buffer block.

	#7	#6	#5	#4	#3	#2	#1	#0
5002		LWM		LGT				LD1

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

LD1: The tool offset number

0: Specify through the last two digits of T code

1: Specify through the last one digit of T code

LGT : Tool appearance compensation

0: Compensate through the coordinate system offset (there isn't any connection with LWM, and compensate in the block of T code)

1: Compensate through the tool traverse

LWM: Tool offset (When LGT is 1, compensate the appearance and wearing)

0: Execute in T code block

1: Execute with axis movement meanwhile

Remark:
When LGT is 0, the offset is executed in T code block, and there isn't

any connection with the parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
5003		LVC				CCN		

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

CCN: In the tool nose radius compensation mode, when the auto reference point return (G28) is commanded,

0: The compensation vector of the tool nose radius is canceled when the tool nose traverses to the intermediate point.

1: The compensation vector of the tool nose radius isn't canceled when the tool nose traverses to the intermediate point. But it is canceled until it traverses to the reference point.

LVC: Tool offset value is

0: Not cleared during resetting

1: Cleared during resetting

	#7	#6	#5	#4	#3	#2	#1	#0
5004							ORC	

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

ORC: Tool offset value

0: Specified by the diameter value (axes programmed by the diameter value)

1: Specified by the radius value

	#7	#6	#5	#4	#3	#2	#1	#0
5006							TGC	OIM

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

OIM: Switch between the inch system and the metric system, whether the tool offset value is auto changed

- 0: Not changed
1: Changed

TGC: Command T code in G50, G04 or G10 block

- 0: Not alarm
1: P/S alarms

	#7	#6	#5	#4	#3	#2	#1	#0
5008		CNS	CNF	MCR	CNV		CNC	CNI

『Parameter type』: Bit type

『Default setting』: 0000 0000

『Range』: 0 or 1

『Authority to rewrite』: Machine

『Valid method』: Valid immediately

『Additional requirements』: None

CNI: The tool nose radius compensation is interference checked

- 0: Execute
1: Not execute

CNC: When the tool nose radius compensation is interference checked and the difference between the programming movement direction and the offset movement direction is 90~270°

- 0: P/S alarms
1: Not alarm

CNV: The tool nose radius compensation (T serial) is interface checked and the vector is cleared

- 0: Execute
1: Not execute

MCR: If G41/G42 tool nose radius compensation is commanded in MDI mode, whether alarm

- 0: Not alarm
1: P/S alarm

Remark:

In MDI mode, the tool nose radius isn't compensated even it is set by the parameter.

CNF: When the tool nose radius compensation is interference checked, whether alarm when the internal full circle is cut

- 0: P/S alarms
1: Not alarm

CNS: The tool nose radius compensation is interference checked, whether alarm when the step is less than the tool radius

- 0: P/S alarms
1: Not alarm

5010

During the tool nose compensation, the limit value of the vector is ignored when the tool traverses along the corner outside (CLV)

『Parameter type』: Character type

『Default setting』 : 0

『Range』 : 0~16383

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

The limit value of the minor traverse value is ignored when the tool nose radius compensation is set and the tool traverses along the corner outside.

SETTING UNITS	IS-B	IS-C	UNITS
Input in metric system	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch

5013

Maximum value of the tool wearing compensation value (MTW)

『Parameter type』 : Character type

『Default setting』 : 10

『Range』 :

SETTING UNITS	IS-B	IS-C	UNITS
Input in metric system	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch

SETTING RANGE	IS-B	IS-C
Input in metric system	0~9 999 999	0~99 999 999
Input in inch system		

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

The parameter sets the maximum value of the tool wearing compensation value.

When the set absolute value of the tool wearing compensation value exceeds the maximum value, it alarms:

Input from MDI.....alarm: too many digits. Exceed range (XXXX—XXXX) (input range is in the bracket).

Input through G10.....alarm: The offset value input by G10 is out of the specified range.

3.15 Parameter about the fixed cycle

3.15.1 Parameter about the drilling fixed cycle

	#7	#6	#5	#4	#3	#2	#1	#0
5102							MRC	

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

MRC: The non-monotonic target shape is defined in multi-cycle command (G71 or G72), or non-monotonic Z axis is in G73 cycle and the run-out value is in Z axis or the finishing surplus X axis is non-monotonic

0: Not alarm

1: Alarm

	#7	#6	#5	#4	#3	#2	#1	#0
5104						FCK		

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

FCK : In combined fixed cycles (G71, G72 and G73), the processing appearance is

0: Not checked

1: Checked

5110	M code locking C axis in the fixed cycle of drilling holes (CMD)
------	--

『Parameter type』 : Character type

『Default setting』 : 0

『Range』 : 0~99

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set M code , which can lock C axis, during the fixed cycle of drilling holes.

3.15.2 Parameter about the thread cutting cycle

5130	Beveling value of the thread cutting cycle (G76, G92) (THD)
------	---

『Parameter type』 : Character type

『Default setting』 : 0

『Range』 : 0~99× (0.1 screw pitch)

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

The parameter sets the beveling value of G76 and G92 thread cutting cycle.

3.15.3 Parameter about the combined fixed cycle

5132

Cutting value of the combined fixed cycle G71 and G72 (THC)

『Parameter type』 : Character type

『Default setting』 : 1000

『Range』 : 0~99 999 999

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the cutting value of G71 and G72 combined fixed cycle.

	IS-B	IS-C	UNITS
Input in metric system	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch

5133

Run-out value of G71 and G72 combined fixed cycle (MCE)

『Parameter type』 : Character type

『Default setting』 : 0

『Range』 : 0~99 999 999

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the run-out value of G71 and G72 combined fixed cycle.

	IS-B	IS-C	UNITS
Input in metric system	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch

5135

**Run-out value of G73 combined
fixed cycle along X axis direction (G73XE)**

5136

**Run-out value of G73 combined
fixed cycle along Z axis direction (G73ZE)**

『Parameter type』 : Character type

『Default setting』 : 0

『Range』 : -99 999 999~99 999 999

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the run-out value of G73 combined fixed cycle along with X and Z axes direction

SETTING UNITS	IS-B	IS-C	UNITS
Input in metric system	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch

5137

Partition times of G73 combined fixed cycle (G73DC)

『Parameter type』 : Character type

『Default setting』 : 1

『Range』 : 1~99 999 999

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the partition times of G73 combined fixed cycle.

5139

Reversal value of G74 and G75 combined fixed cycles (G74G75R)

『Parameter type』 : Character type

『Default setting』 : 0

『Range』 : 0~99 999 999

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the reversal value of G74 and G75 combined fixed cycle.

SETTING UNITS	IS-B	IS-C	UNITS
Input in metric system	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch

5140

Minimum cutting value of G76 combined fixed cycle (G76MID)

『Parameter type』 : Character type

『Default setting』 : 0

『Range』 : 0~99 999 999

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the minimum cutting value of G76 combined fixed cycle.

SETTING UNITS	IS-B	IS-C	UNITS
Input in metric system	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch

5141

Finishing surplus of G76 combined fixed cycle (G76FA)

『Parameter type』 : Character type

『Default setting』 : 500

『Range』 : 1~99 999 999

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the finishing surplus of G76 combined fixed cycle.

5142

Finishing cycle times of G76 combined fixed cycle (G76FC)

『Parameter type』 : Character type

『Default setting』 : 1

『Range』 : 1~99

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the finishing cycle times of G76 combined fixed cycle.

5143

Tool nose angle of G76 combined fixed cycle (G76TNA)

『Parameter type』 : Character type

『Default setting』 : 60

『Range』 : 0~99 (deg)

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the tool nose angle of G76 combined fixed cycle.

3.16 Parameter about the rigid tapping

	#7	#6	#5	#4	#3	#2	#1	#0
5200		FHD		DOV		CRG		G84

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

G84: Method of commanding the rigid tapping

0: M code commands the rigid tapping before command G84/G88 (refer to parameter NO.5210).

1: M code doesn't command the rigid tapping. G84/G88 is taken as G code of the rigid tapping, and the common tapping is not used.

CRG: After the command of canceling the rigid tapping method, rigid tapping:

0: After the rigid tapping signal RGAP changes to 0, the method is canceled.

1: Before the rigid tapping signal RGAP changes to 0, the method is canceled.

DOV: Override during the rigid tapping run-out

0: Invalid

1: Valid, override value is set by parameter 5211

FHD: Feed pause and single block running in rigid tapping

0: Forbid

1: Allow

	#7	#6	#5	#4	#3	#2	#1	#0
5201	TXZ	TDK				TDR		

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

TDR: The time constant of the rigid tapping cutting and that of the tapping run-out:

0: Same

1: Different

TDK: Specify K in tapping command

0: Take it as the cycle times

1: Ignore

TXZ: Non-tapping axis is taken as the orientation in tapping command

0: Allow to use

1: Alarm

5210	M code commanding the rigid tapping (RTMC)
-------------	---

『Parameter type』 : Character type

『Default setting』 : 29

『Range』 : 0~255

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

M code is set to specify the rigid tapping method. When it is set as 0, CNC takes it as M29.

5211**Speed override value in tapping run-out (RTOV)**

『Parameter type』 : Character type

『Default setting』 : 120

『Data unit』 : 1% or 10%

『Range』 : 0~200

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the override value during the rigid tapping run-out, and it is valid only when parameter DOV (NO.5200 BIT4) is set as 1.

5241**Spindle maximum speed in rigid tapping (RTMS)**

『Parameter type』 : Character type

『Default setting』 : 1000

『Range』 : 0~9999

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the spindle maximum speed in rigid tapping.

5261**Time constant of linear acceleration and deceleration in rigid tapping (RTLT)**

『Parameter type』 : Character type

『Default setting』 : 200

『Range』 : 0~4000ms

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

During the rigid tapping, the time constant of linear acceleration or deceleration of the spindle and the tapping axis is the time (parameter NO.5241) of the spindle maximum speed when the spindle reaches the rigid tapping. The actual time is the ratio between the specified spindle speed and the maximum speed multiplies by the parameter.

5271**Linear acceleration or deceleration time constant in rigid tapping run-out (RTET)**

『Parameter type』 : Character type

『Default setting』 : 200

『Range』 : 0~4000ms

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

During the rigid tapping run-out, the time constant of linear acceleration or deceleration of the spindle and the tapping axis. The parameter is valid only when parameter TDR (NO.5201 BIT2)

is set as 1.

3.17 Parameter about the polar coordinates interpolation

	#7	#6	#5	#4	#3	#2	#1	#0
5450							AFC	

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

AFC: Whether use the auto override and the auto speed in the polar coordinates interpolation mode.

0: Not use

1: Use

Remark:

In the polar coordinates interpolation mode, the more closely the tool is near to the work piece center, the bigger the speed vector of the revolving axis is. If the center part exceeds the maximum cutting speed (parameter NO.5462), the servo (NO.411) alarms. Auto feedrate override and auto feedrate limit function auto controls the feedrate, then, the speed vector of the revolving axis doesn't exceed the maximum cutting feedrate.

5460	Specify the polar coordinates interpolation axis (linear axis) (LAI)
------	--

5461	Specify the polar coordinates interpolation axis (revolving axis) (RAI)
------	---

『Parameter type』 : Character type

『Default setting』 : NO.5460 is 0; NO.5461 is 5

『Range』 : 1~quantity of the control axes

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the control axis numbers of the linear axis and the revolving axis for polar coordinates interpolation.

5462	Maximum cutting feedrate of the polar coordinates interpolation (MFI)
------	---

『Parameter type』 : Character type

『Default setting』 : 8000

『Range』 :

	IS-B	IS-C	UNITS
Machine in metric system	0, 6~24 000	0, 6~10 000	mm/min
Machine in inch system	0, 6~9 600	0, 6~4 800	inch/min
Rotation axis	0, 6~24 000	0, 6~10 000	deg/min

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the valid maximum feedrate of the polar coordinates interpolation. If the commanded speed is greater than the value, the speed is limited by the maximum one. When the parameter is set as 0, the speed in the polar coordinates interpolation is limited by the maximum cutting feedrate (parameter NO.1422) value.

5463

**Allowable auto override percentage
in polar coordinates interpolation (API)**

『Parameter type』 : Character type

『Default setting』 : 0

『Range』 : 0~100 (%)

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

When the polar coordinates interpolation is set, the percentages of the auto override are allowed to limit the cutting feedrate of the revolving axis.

The allowable speed of the revolving axis = Maximum cutting feedrate X override percentage

In polar coordinates interpolation, the more closely the tool is near to the work piece center, the bigger the speed vector of the revolving axis is. When it exceeds the allowable speed, the feedrate automatically multiplies by the override value calculated through the following formula:

Override = Allowable speed of the revolving axis/the speed vector of the revolving axis X 100%

If the revolving speed after timing the override still exceeds the allowable speed, the feedrate is limited in the allowable maximum cutting feedrate (auto speed limit function) .

Remark:

When the parameter value is set as 0, it is taken as 90%;

To limit the auto speed override and the auto speed, the parameter AFC (NO.5450#1) is set as 1.

3.18 Parameter about the user macro program

	#7	#6	#5	#4	#3	#2	#1	#0
6000			SBM					G67

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

G67: Macro program mode calling (G66) mode is not set, but mode calling command (G67) is canceled.

0: P/S alarms (NO.122)

1: Ignore G67

SBM: Whether use the single block to stop in the user macro program

0: Not use

1: Use

	#7	#6	#5	#4	#3	#2	#1	#0
6001	CLV	CCV						

『Parameter type』 : Bit type

『Default setting』 : 0100 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

CCV: After resetting, the user macro program common vector 100~199 is

0: Cleared as null

1: Not cleared

CLV: After resetting, the user macro program part vector 1~33 is

0: Cleared as null

1: Not cleared

	#7	#6	#5	#4	#3	#2	#1	#0
6004							MFZ	NAT

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

NAT: The function command ATAN of the user macro program

0: Result of ATAN is 0~360.0

Result of ASIN is 270.0~0~90.0

1: Result of ATAN is -180.0~0~180.0

Result of ASIN is -90~0~90

MFZ : The angles of STN, COS or TAN, which are operation commands of the user macro program, are 1.0×10^{-8} or less, or the operation result is not exact 0

0: Underflow process 1:Reduction to 0

3.19 Parameter about the jumping function

	#7	#6	#5	#4	#3	#2	#1	#0
6200	SKF						SK0	

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : System

『Valid method』 : Valid immediately

『Additional requirements』 : None

SK0: Set the valid state of the jumping signal

0: It is valid when the input signal is “1”

1: It is valid when the input signal is “0”

SKF: Dry run and override is valid or invalid for G31 jumping command

0: Invalid

1: Valid

3.20 Parameter about the figure display

Parameter about the figure display/the dynamic figure display

	#7	#6	#5	#4	#3	#2	#1	#0
6500					DPA			

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

DPA: In the figure display interface, the current position displays

0: Display the actual position including the tool compensation and offset

1: Display the programming position excluding the tool compensation and offset

3.21 Parameter about display the running time and the quantity of parts

	#7	#6	#5	#4	#3	#2	#1	#0
6700							PRT	PCM

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

PCM: M codes counting the total quantity of the processing parts and the quantity of the processing parts

0: M codes specified by M02 and M30 and parameter NO.6710

1: M codes only specified by parameter NO.6710

PRT: During setting, the signal PRTSF (F62.7) of the sufficient quantity of the processing parts is

0: Cut off

1: Not cut off

6710

**M codes counting the total quantity of the processing parts
and the quantity of the processing parts (MPC)**

『Parameter type』 : Character type

『Default setting』 : 0

『Range』 : 0~9999

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

The machine program executes M codes set by the parameter, total quantity of the processing parts and quantity of the processing parts plus 1, respectively.

Remark:

When the setting value is 0, it is invalid (M00 can't count the parts).
And it can't be set as 98 and 99, neither.

6713

Quantity of the required parts (RPM)

『Parameter type』 : Character type

『Default setting』 : 0

『Range』 : 0~9 999

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

When the quantity of the processing parts equals to that of the parts required being processed, the signal PRTSF (F62.7) of the enough quantity of the required parts outputs to PLC. However, if the quantity is 0, it is regarded as infinitely great, not output to PRTSF.

3.22 Parameter about MPG feed, MPG cutoff and MPG feed controlling the tool direction

	#7	#6	#5	#4	#3	#2	#1	#0
7100				HPF				JHD

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

JHD: MPG feeding in JOG mode, or increment feeding in MPG feed mode

0: Invalid

1: Valid

	JHD=0		JHD=1	
	JOG MODE	MPG MODE	JOG MODE	MPG MODE
JOG feeding	O	×	O	×
MPG feeding	×	O	O	O
Increment feeding	×	×	×	O

HPF: When MPG feedrate exceeds the manual rapid movement speed

0: The speed is limited in the manual rapid movement speed, the pulse exceeding the manual rapid movement part is ignored (The scale of MPG doesn't comply with the movement amount)

1: The speed is limited in the manual rapid movement speed, the exceeding part isn't ignored but saved in CNC. (Although MPG is stopped, the machine still moves the pulse value saved in CNC and then stops.)

	#7	#6	#5	#4	#3	#2	#1	#0
7102								HNGx

『Parameter type』 : Bit axial type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

HNGx: Revolving direction of each axis movement direction and that of MPG

0: Same

1: Opposite

7110	Quantity of MPG (NMP)
-------------	------------------------------

『Parameter type』 : Character type

『Default setting』 : 1

『Range』 : 1~2

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the quantity of MPG.

7113**MPG feeding override M (MFM)**

『Parameter type』 : Character type

『Default setting』 : 100

『Range』 : 1~127

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set the override when MPG feeding movement value selection signals MP1=0, MP2=1.

7114**MPG feeding override N (MFN)**

『Parameter type』 : Character type

『Default setting』 : 1000

『Range』 : 1~1000

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Set MPG feeding override when MPG feeding movement value selecting signals MP1=1, MP2=1.

MOVEMENT VALUE SELECTING SIGNAL		MOVEMENT VALUE (MPG FEEDING)
MP2	MP1	
0	0	Minimum setting unit * 1
0	1	Minimum setting unit * 10
1	0	Minimum setting unit * M
1	1	Minimum setting unit * N

7117**Allowable pulse cumulative value in MPG feeding (APM)**

『Parameter type』 : Character type

『Default setting』 : 10000

『Range』 : 0~99999999

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

When MPG feeding instance exceeds the rapid movement speed, the pulse exceeding the rapid movement is not canceled but saved. The parameter sets the allowable value of the memory capacity.

Remark:

When overrides, such as X100 or more than it, are selected, MPG rapidly turns round. MPG feeding is more than the rapid movement speed, the speed is limited by the rapid movement speed. The pulse exceeding the rapid movement speed is ignored, therefore, the scale value of MPG doesn't comply with the actual movement value. Then, If the allowable value is preset in the parameter, the pulse exceeding the rapid movement speed is not canceled, but saved in CNC temporarily (the part exceeding the allowable value is ignored). When MPG revolving speed becomes slower or the revolving stops, the saved pulse changes into the movement command and outputs. Pay attention to it if the allowable value is set too big, even MPG is stopped revolving, CNC won't stop until the remaining pulse is completed.

3.23 Parameter about PLC axis control

	#7	#6	#5	#4	#3	#2	#1	#0
8001			NCC		RDE	OVE		MLE

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

MLE: Whether the locking machine signal MLK of PLC control axis is valid

0: Valid

1: Invalid

OVE: Signals relative with the dry run and the override controlled by PLC axis

0: Same signals controlled by CNC

1: Signals especially used in PLC

RDE: In PLC axes control, whether the dry run is valid for the rapid feeding commands

0: Invalid

1: Valid

NCC: For PLC control axes (the control axes select the axes chosen by the signal), command the program to command the movement

0: According to the axis control command, PLC controls the axis, P/S (No.139) alarms; the axis is not controlled, CNC command is valid.

1: P/S (No.139) alarms.

	#7	#6	#5	#4	#3	#2	#1	#0
8002	FR2	FR1	PF2	PF1	F10		DWE	RPD

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

RPD: The rapid movement speed of PLC control axis

0: Feedrate set by parameter No.1420

1: In axis control command, feedrate set by feedrate data

DWE When use the increment system IS-C, the minimum time specified by the pause command during PLC axis control.

0: 1ms

1: 0.1ms

F10: In PLC axis control, the minimum increment units of the cutting feedrate (per minute)

F10	Input in metric system	Input in inch system
0	1mm/min	0.01inch/min
1	10mm/min	0.1inch/min

FR1, FR2: The feedrate units of per revolution feeding during PLC axis control

FR2	FR1	Input in metric system	Input in inch system
0	0	0.0001mm/rev	0.000001inch/rev
1	1		
0	1	0.001mm/rev	0.00001inch/rev
1	0	0.01mm/rev	0.0001inch/rev

	#7	#6	#5	#4	#3	#2	#1	#0
8003								PIM

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

PIM : If PLC control axis is linear axis, the control commands are

0: Affected by inch system/metric system

1: Not affected by inch system/metric system

	#7	#6	#5	#4	#3	#2	#1	#0
8004	NDI	NCI	DSL			JFM	NMT	CMV

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

CMV: According to the commands sent by CNC, PLC sends the axis control command after moving along the axis and before receiving the command signal of the miscellaneous function.

0: P/S No.130 alarms

1: The axis is processed as one PLC axis and is executed the set movement.

NMT: When PLC is processing one control command of some axis, and CNC sends another command to command the axis, PLC control axis is still

0: P/S No.133 alarms

1: Not alarm

JFM: Feedrate units of continuous feeding (06h) of PLC control axis

INCREMENT SYSTEM	JFM	INPUT IN METRIC SYSTEM	INPUT IN INCH SYSTEM	ROTATION AXIS
IS-B	0	1mm/min	0.01inch/min	1deg/min
	1	200mm/min	2.00inch/min	200deg/min
IS-C	0	0.1mm/min	0.001inch/min	0.1deg/min
	1	20mm/min	0.200inch/min	20deg/min

DSL: When selecting the axes controlled by PLC is forbidden, if the axes are tried to exchange,

0: Failed and P/S No.139 alarms

1: Axes, without commanding the channel, are executed exchanging

NCI: During decelerating the axes controlled by PLC, in-position check is

0: Executed

1: Not executed

NDI: When PLC control axis selects the diameter programming, under PLC axis control

0: The radius programming specifies the movement distance and the feedrate

1: The diameter programming specifies the movement distance and the feedrate

	#7	#6	#5	#4	#3	#2	#1	#0
8005							CDI	

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

CDI : When PLC control axis selects the diameter programming, under PLC axis control

0: Radius programming specifies the movement distance and the feedrate

1: The diameter programming specifies the movement distance and the radius programming specifies the feedrate

8010	Selecting each axis DI/DO group controlled by PLC (PAS)
-------------	--

『Parameter type』 : Character type

『Default setting』 : 0

『Range』 : 0~4

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

Each DI/DO group controlled by each PLC axis, which is shown as the following list:

NUMERICAL VALUE	REMARK
0	The axis is not controlled by PLC
1	DI/DO in group A is used
2	DI/DO in group B is used
3	DI/DO in group C is used
4	DI/DO in group D is used

8022	Maximum feedrate of feeding/per revolution controlled by PLC axis (PAMS)
-------------	---

『Parameter type』 : Character type

『Default setting』 : 6

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』: None

『Range』: Set the maximum feedrate of feeding/per revolution controlled by PLC axis.

INCREMENT SYSTEM	DATA UNITS	VALID DATA RANGE	
		IS-B	IS-C
Machine in metric system	1mm/min	6~15000	6~12000
Machine in inch system	0.1inch/min	6~6000	6~4800
Rotation axis	1deg/min	6~15000	6~12000

8028

For each PLC control axis, the linear acceleration or deceleration time constant specified by speed command during JOG feeding (PALT)

『Parameter type』: Character axial type

『Default setting』: 200

『Range』: 0~3000ms

『Authority to rewrite』: Machine

『Valid method』: Valid immediately

『Additional requirements』: None

Specify the linear acceleration or deceleration time constant during JOG feeding

Points for attention:

If it is specified as "0", the system doesn't control the acceleration and deceleration.

3.24 Parameter about the basic function

8130

Total quantity of the control axes (TCA)

『Parameter type』: Character axis

『Default setting』: 2

『Range』: 2~5

『Authority to rewrite』: Machine

『Valid method』: Power on

『Additional requirements』: None

Set the total quantity of the axes controlled by CNC system.

#7 #6 #5 #4 #3 #2 #1 #0

8131

HPG

『Parameter type』: Bit type

『Default setting』: 0000 0001

『Range』: 0 or 1

『Authority to rewrite』: Machine

『Valid method』: Power on

『Additional requirements』: None

HPG: Whether use MPG feeding

0: Not use

1: Use

	#7	#6	#5	#4	#3	#2	#1	#0
8132								TLF

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

TLF : Whether use the tool work life management function

0: Not use

1: Use

	#7	#6	#5	#4	#3	#2	#1	#0
8133					MSP	SCS	AXC	SSC

『Parameter type』 : Bit type

『Default setting』 : 0000 0001

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Power on

『Additional requirements』 : None

SSC : Whether use the function of the constant surface speed (G96) control

0: Not use

1: Use

AXC :Whether the use spindle orientation function

0: Not use

1: Use

SCS: Whether use CS outline control function

0: Not use

1: Use

MSP: Whether use the multi-spindle control function

0: Not use

1: Use

3.25 Parameter about GSKLink communication function

	#7	#6	#5	#4	#3	#2	#1	#0
9000								ACAN

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Equipment management level

『Valid method』 : Power on

『Additional requirements』 : None

ACAN: Whether the system servo communication function is valid

0: Invalid

1: Valid

9010

Baud rate of the system servo communication (ABPS)

『Parameter type』 : Character type

『Default setting』 : 500 (kbps)

『Range』 : 500, 600, 800 or 1000 (kbps)

『Authority to rewrite』 : Equipment management level

『Valid method』 : Power on

『Additional requirements』 : None

ABPS: The parameter sets the baud rate of the system servo communication

9011

Subunit number corresponding to each axis during servo communication (SIDx)

『Parameter type』 : Character axial type

『Default setting』 :

CONTROL AXES NUMBERS	DEFAULT VALUE
1	1
2	2
3	0
4	0
5	0

『Range』 : 0~5

『Authority to rewrite』 : Equipment management level

『Valid method』 : Power on

『Additional requirements』 : None

SIDx The parameter sets the subunit number corresponding to each axis during servo communication.

Remark:

“0” represents the axis doesn't connect with the servo subunit.
“1~5” represent the servo subunit number corresponding to each axis.

9012

扩展伺服主轴通讯对应的从机号 (SIDS) Subunit number corresponding to the extended servo spindle communication

『Parameter type』 : Character type

『Default setting』 : 0

『Range』 : 0~5

『Authority to rewrite』 : Equipment management level

『Valid method』 : Power on

『Additional requirements』 : None

SIDS: During the servo spindle communication, the parameter sets the corresponding subunit numbers out of the range of the controlled axes total quantity.

Remark:

“0” represents the axis doesn’t connect with the servo subunit. “1~5” represent the analog spindle subunit number corresponding to the axis.

Chapter IV MACHINE DEBUGGING

This chapter introduces the trial running method and its steps when GSK988T powers on at the first time, and debug according to the following operation steps and operate the machine correspondingly.

4.1 Emergency stop and limit

GSK988T is equipped with the software limit function; for safety, it is suggested to use the hardware limit measure, meanwhile. And install the limit switches in each axis in positive and negative directions, the connection is shown as the following figure:

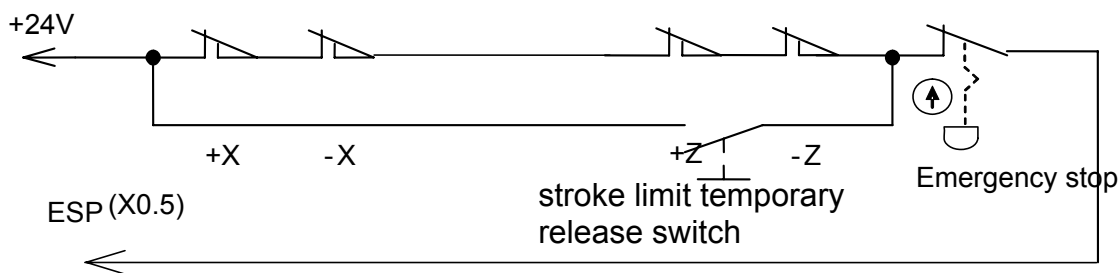


Fig 4-1

In manual or MPG mode, slowly move each axis to verify the validity of the overrun limit switch, the correctness in the alarm display and the validity of the overrun release button; when it overruns or the emergency stop button is pressed, CNC alarms "emergency stop". The alarm can be cleared through pressing the overrun release button and the axis moves in the reverse direction, or shield PLC emergency stop parameter which makes the switch invalid, and then press the resetting key to clear the emergency stop limit alarm, and the axis moves toward the working table in the reverse direction and is off from the limit switch.

Relative parameter:

3003	ESP							
------	-----	--	--	--	--	--	--	--

『Parameter type』 : Bit type

『Default setting』 :1000 0000

『Valid method』 : Valid immediately

ESP: External emergency stop alarm signal (X0.5)

0: Emergency stop alarms when the input signal is 0 (low level)

1: Emergency stop alarms when the input signal is 1 (high level)

4.2 Setting drive unit

After the servo drive unit correctly connects with the system, the detailed setting is as below:

1. Set the alarm level of the axis corresponding to the servo axis based on the alarm logic level setting ISAx (NO.1816#0) of the servo drive unit set by axis.

2. Select the current axis output pulse mode through setting parameter ABPx(NO.1811#0). 988T system supports two types of the pulse command output, one is pulse + direction pulse output; the other is pulse output of two phase (AB phases) right-angle intersection. The parameter should be set correctly.

3. According to the encoder type of the servo motor, whether the servo uses the encoder of the

absolute type is set by parameter APCx (NO.1815#5) .

4. Correctly set the gear ratio CMR/DMR, which is set by parameters NO.1816 and NO.1820 based on the gear ratio of the machine, which makes the machine movement distance in compliance with the command distance value (refer to the next chapter).

5. If the machine movement direction doesn't comply with that of the displacement command, set the parameter PODx (NO.1811#2) to negate the movement direction output by its corresponding servo axis command.

6. It can be adjusted through parameter APRx (NO.1815#0) if the direction detected by the absolute position encoder can not comply with the actual direction.

The relative parameter is as below:

1. ABP

1811						POD		ABP
------	--	--	--	--	--	-----	--	-----

『Parameter type』 : Bit axial type

『Default setting』 : 0000 0000

『Valid method』 : Power on

ABP: Selecting the pulse drive mode

0: Pulse + direction mode

1: AB phase pulse mode

POD: Selecting each axis pulse output direction

0: Not negate

1: Negate

2. APRx, APZx and APCx

1815			APCx	APZx				APRx
------	--	--	------	------	--	--	--	------

『Parameter type』 : Bit axial type

『Default setting』 : 0000 0000

『Valid method』 : Power on

APRx: When the absolute position encoder is used, the position direction of the absolute position detector

0: Not negate

1: Negate

APZx: When the absolute position detector is used, the mechanical position and that of the absolute position detector

0: Not comply

1: Comply

Remark:
When the absolute position detector is used, the parameter must be set as 0 during initial debugging or after replacing the absolute position detector. And then, cut power supply, power on, again, and manually return to the reference point. Then, the mechanical position complies with that of the position detector; the parameter is auto set as 1.

APCx: The position detector

0: Not use the absolute position detector

1: Use the absolute position detector (the absolute pulse encoder)

3. ISAx and DM1x~ DM3x

1816		DM3x	DM2x	DM1x				ISAx
------	--	------	------	------	--	--	--	------

『Parameter type』 : Bit axial type

『Default setting』 : 0001 0001

『Valid method』 : Power on

ISAx: The servo alarm signal

0: High level of the alarm signal is valid

1: Low level of the alarm signal is valid

DM1x-DM3x: Set each axis detection multiply ratio (DMR)

SETTING VALUES			DETECTION OVERRIDE RATIO (DMR)
DM3x	DM2x	DM1x	
0	0	0	1/2
0	0	1	1
0	1	0	3/2
0	1	1	2
1	0	0	5/2
1	0	1	3
1	1	0	7/2
1	1	1	4

4. CMR

1820	Each axis command multiply ratio (CMR)
------	--

『Parameter type』 : Character axial type

『Range』 :

COMMAND MULTIPLY RATIO (CMR)	VALID RANGE OF NO.1820 SETTING VALUE	DEFAULT SETTING
1/2~1/27	102~127	2
1 ~ 48	2~96	

『Valid method』 : Power on

Set the command multiply ratio (CMR) of each axis.

1. The setting value= $1/\text{CMR} + 100$ when the command multiply ratio (CMR) is 1/2~1/27.

2. The setting value= $2 \times \text{CMR}$ when the command multiply ratio (CMR) is 1~48.

4.3 Gear ratio adjustment

Understand the following concepts before adjusting.

Minimum input increment: It is the minimum input unit of the programming movement value. The minimum input increment is also called as the minimum input unit or the minimum setting unit.

Minimum command increment: The minimum command unit sent from CNC to machine is also the minimum increment of the tool traverse on the machine, which is also called as the minimum traverse unit.

Detection unit: It is the minimum unit to detect the machine position.

Command multiply ratio (CMR): the constant of the equivalence of CNC command pulse

matching with that of the detector pulse, which is also called as the command multiply coefficient.

Detection multiply ratio (DMR): the constant of the equivalence of CNC command pulse matching with that of the detector pulse, which is also called as the command frequency coefficient.

The following relations between the minimum input increment (minimum setting unit) and the minimum command increment (minimum traverse unit) are as below:

			MINIMUM SETTING UNITS	MINIMUM TRAVERSE UNITS
IS—B	Machine in metric system	Input in metric system	0.001mm（Specified by the diameter）	0.0005mm
			0.001mm（Specified by the radius）	0.001mm
		Input in inch system	0.0001 inch（Specified by the diameter）	0.0005mm
			0.0001 inch（Specified by the radius）	0.001mm
	Machine in inch system	Input in metric system	0.001mm（Specified by the diameter）	0.00005 inch
			0.001mm（Specified by the radius）	0.0001 inch
		Input in inch system	0.0001 inch（Specified by the diameter）	0.00005 inch
			0.0001 inch（Specified by the radius）	0.0001 inch
	Rotation axis		0.001deg	0.001deg

			MINIMUM SETTING UNITS	MINIMUM SETTING UNITS
IS—C	Machine in metric system	Input in metric system	0.0001mm (Specified by the diameter)	0.00005mm
			0.0001mm (Specified by the radius)	0.0001mm
		Input in inch system	0.00001 inch (Specified by the diameter)	0.00005mm
			0.00001 inch (Specified by the radius)	0.0001mm
	Machine in inch system	Input in metric system	0.0001mm (Specified by the diameter)	0.000005 inch
			0.0001mm (Specified by the radius)	0.00001 inch
		Input in inch system	0.00001 inch (Specified by the diameter)	0.000005 inch
			0.00001 inch (Specified	0.00001 inch

		by the radius)	
	Rotation axis	0.0001deg	0.0001deg

The relation between the detection unit and the minimum traverse unit is as below:

Detection unit=minimum traverse unit/command multiply ratio (CMR)

Remark: The units of the axis backlash compensation value and the axis screw pitch error compensation setting value are detection units, pay attention to it:

The calculation formula of the electrical gear ratio is shown as below:

Electrical gear ratio output by each axis =command multiply ratio (CMR) /detection multiply ratio (DMR)

The meaning of the electrical gear ratio is to make the movement distance of the machine axis carriage comply with the distance value of the command (the movement distance of the machine coordinate). If axis is programmed by the radius (about the setting radius/diameter programming, refer to NO.1006#3), the movement distance of the actual axis on the machine equals to that of the machine coordinate displayed on the system; if the axis is programmed by the diameter, the two times of the actual axis movement distance on the machine equals to the movement distance of the machine coordinate displayed on the system.

For example:

There is one tilted-bed lathe; the feed screw lead of X axis is 6mm, Z axis lead is 8mm, the connection between X axis motor, Z axis motor and the feed screw is straight (the gear ratio between the motor and the feed screw is 1:1). With GSK988T system, the system parameter NO.1004ISC, NO.0000INI and NO.1004INM are set as 0. X axis is programmed by the diameter, Z axis is programmed by the radius. The encoder of the motor is increment type, the quantity of the pulses is 5000 (4 times of frequency, resolution: 200000), and the motor revolves one cycle, which requires the system sends 20000 pulses to the system. The gear ratio of drive unit is set as 1:1, and it requires setting the correct gear ratio on GSK988T system.

Explanation:

From system parameter INI and INM setting values, it can be learned that the system is input and the machine is in metric system. Because ISC is set as 0, the current minimum setting unit of the system is 0.001mm.

X axis gear ratio calculation:

X axis is programmed by the diameter, and the corresponding minimum movement unit is 0.0005mm. If X axis carriage movement value is 6mm (When X axis moves at the distance of one lead, X axis motor rotates for one cycle), the corresponding machine coordinate variety value is 12mm, and the quantity of the pulses output by X axis drive interface is 12000 (12000 = 6mm / 0.0005mm) . However, when X axis motor rotates for one cycle, it requires the system to send 20000 pulses to drive. It requires the number of the output pulses of X axis drive interface multiplies one magnified coefficient, then, the magnified coefficient is X axis electrical gear ratio.

Electrical gear ratio output by X axis = 20000 / 12000
= 5 / 3
= command multiply ratio (CMR)/detection multiply ratio (DMR)

Z axis gear ratio calculation:

Z axis is programmed by the radius, and the corresponding minimum movement unit is 0.001mm. If Z axis carriage movement value is 8mm (When Z axis moves at the distance of one lead, Z axis motor rotates for one cycle), the corresponding machine coordinate variety value is 8mm, then the quantity of pulses output by Z axis drive interface is 8000 ($8000 = 8\text{mm} / 0.001\text{mm}$). While Z axis motor rotates for one cycle, it requires the system to send 20000 pulses to the drive unit, the same principle is as below:

<p>Electrical gear ratio output by Z axis = $20000 / 8000$ $= 5 / 2$ = Command multiply ratio (CMR) / detection multiply ratio (DMR)</p>
--

After calculating the gear ratio, correctly set NO.1820 and NO.1816 according to the parameter meaning, which is shown as the following list.

	COMMAND MULTIPLY RATIO (CMR)	NO.1820 SETTING VALUE	DETECTION MULTIPLY RATIO (DMR)	NO.1816 SETTING VALUE
X axis	5	10	3	DM3~DM1: 101
Z axis	5	10	2	DM3~DM1: 011

When the electrical gear ratio molecule is greater than the denominator, the maximum speed allowed by CNC reduces. For example: the command multiply ratio CMR=2, the maximum speed allowed by Z axis is reduced to half of the output maximum speed in theory.

When the electrical gear ratio molecule is not equal to the denominator, the orientation precision of CNC may reduce. For example, the pulse doesn't output when the detection multiply ratio DMR=2, the input increment is 0.001; while output one pulse when the input increment is 0.002.

Normally, to guarantee the orientation precision and the speed index of CNC, when the servo drive unit is equipped with the function of the electrical gear ratio, it is suggested that the electrical gear ratio of CNC is set as 1:1, and the calculated electrical gear ratio is set in the numerical servo.

When it is equipped with the stepped drive unit, try to choose the drive unit with the stepped subdivision function, properly select the mechanical gear ratio, and try to keep the electrical gear ratio of CNC as 1:1; meanwhile, avoid too big gap between the molecule and the denominator of CNC electrical gear ratio.

4.4 Characteristics of adjusting acceleration and deceleration

Adjust the relative CNC parameters based on the characteristics of the drive unit and the motor and the machine loading capacity:

1	4	2	0	Each axis rapid movement speed
---	---	---	---	--------------------------------

『Range』 :

SETTING UNITS	DATA UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	

Machine in metric system	1 mm/min	30~60000	6~24000	8000
Machine in inch system	0.1 inch/min	30~24000	6~9600	
Rotation axis	1 deg/min	30~60000	6~24000	

Set the rapid movement speed of each axis when the rapid movement override is 100%.

1	4	2	1
---	---	---	---

The lowest speed of each axis rapid movement override (F0)

『Range』 :

SETTING UNITS	DATA UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1 mm/min	30~15000	30~12000	400
Machine in inch system	0.1 inch/min	30~6000	30~4800	
Rotation axis	1 deg/min	30~15000	30~12000	

Set the speed when each axis rapid movement override is 0.

1	4	2	2
---	---	---	---

The maximum cutting feedrate of all axes

『Range』 :

SETTING UNITS	DATA UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1mm/min	6~60000	6~24000	8000
Machine in inch system	0.1inch/min	6~24000	6~9600	

Set the maximum cutting feedrate to limit all axes.

1	4	2	3
---	---	---	---

Each axis JOG feedrate

『Range』 :

SETTING UNITS	DATA UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1mm/min	6~32767		1000
Machine in inch system	0.1inch/min			

Rotation axis	1 deg/min		
---------------	-----------	--	--

Set the manual rapid movement speed of each axis when the rapid movement override is 100%.

1	4	2	4	Manual rapid movement speed of each axis
---	---	---	---	---

『Range』 :

SETTING UNITS	DATA UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1 mm/min	30~60000	30~24000	8000
Machine in inch system	0.1 inch/min	30~24000	30~9600	
Rotation axis	1 deg/min	30~60000	30~24000	

Set the feedrate when each axis manual continuous feed (JOG feed).

1	6	0	1				RTO				
---	---	---	---	--	--	--	------------	--	--	--	--

RTO: During rapidly running, the block is

- 0: Not overlapped
- 1: Overlapped

1	6	1	0				JCLx				CTLx
---	---	---	---	--	--	--	-------------	--	--	--	-------------

CTLX: Acceleration and deceleration during cutting feeding (including dry run feeding) is

- 0: Acceleration and deceleration in index type
- 1: Acceleration and deceleration in linear type after interpolation

JGLX: Acceleration and deceleration of JOG feeding is

- 0: Acceleration and deceleration in index type
- 1: Acceleration and deceleration in linear type or in bell type after interpolation

1	6	2	0	Acceleration and deceleration time constant T in linear type or T1 in bell type during each axis rapid feeding							
---	---	---	---	--	--	--	--	--	--	--	--

Set acceleration and deceleration time constant during rapid movement.

1. T1 is set in the parameter, T2 in parameter No.1621 when the function of acceleration and deceleration is used in bell type during rapid movement,
2. When the function of acceleration and deceleration is used free of bell type, acceleration and deceleration time constant of linear type is set in parameter.

Remark:

1. When parameter No.1621 (acceleration and deceleration time constant T2 in bell type during rapid movement) is set as 0, acceleration and deceleration in linear type is still used even with the function of acceleration and deceleration in bell type. Under such circumstances, the parameter represents acceleration and deceleration time constant in linear type during rapid movement.
2. Sometimes, the set time constant value makes the movement speed less than the rapid movement speed a little, and it reaches the rapid movement speed after acceleration for a while. To solve the

problem, the time constant is set as multiple of 8.

1	6	2	2
---	---	---	---

**Acceleration and deceleration time constant
of cutting feed after each axis interpolation**

『Range』：

0~4000 ms (Cutting and feeding of acceleration and deceleration in index type)

0~512 ms (After cutting and feeding interpolation, acceleration and deceleration in linear type)

Set each axis cutting and feeding of acceleration and deceleration in index type, or acceleration and deceleration time constant in linear type after interpolation. The detailed type is set by parameter CTLX or CTBX (No. 1610#0 or #1). Except the special usage of the parameter, all axes must be set as the same time constant. If the different time constant is set, it's impossible to get the correct linear or arc shape. The parameter is valid to thread cutting and without any connection with the acceleration and deceleration. In G76 thread cutting cycle, the parameter is valid except for acceleration and deceleration in index type.

1	6	2	3
---	---	---	---

**FL speed of acceleration and deceleration
in index type during each axis cutting and feeding**

『Range』：

SETTING UNITS	DATA UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1 mm/min	0, 6~15000	0, 6~12000	30
Machine in inch system	0.1 inch/min	0, 6~6000	0, 6~4800	30
Rotation axis	1 deg/min	0, 6~15000	0, 6~15000	30

Set the minimum speed (FL speed) of acceleration and deceleration in index type during each axis cutting and feeding.

Remark: Except the special usage of the parameter, each axis must be set as 0. If the value is set except 0, the correct linear or arc can not be shaped.

1	6	2	4
---	---	---	---

**acceleration and deceleration time constant of each axis JOG feeding
after interpolation**

『Range』：

0~4000 (Acceleration and deceleration in index type during JOG feeding)

0~512 (Acceleration and deceleration in linear type after JOG feeding interpolation)

Set acceleration and deceleration time constant in index type during each axis JOG feeding, the acceleration and deceleration time constant in linear type after interpolation.

The detailed type is set by parameter CTLx or JGLx (No.1610#0 or #4).

1	6	2	5
---	---	---	---

FL speed of acceleration and deceleration
in index type during each axis JOG feeding

『Range』 :

SETTING UNITS	DATA UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1 mm/min	0, 6~15000	0, 6~12000	30
Machine in inch system	0.1 inch/min	0, 6~6000	0, 6~4800	30
Rotation axis	1 deg/min	0, 6~15000	0, 6~15000	30

Set the minimum speed (FL speed) of acceleration and deceleration in index type during each axis JOG feeding.

1	6	2	6
---	---	---	---

Acceleration and deceleration time constant
in index type during each axis thread cutting cycle

『Range』 : 0~4000

Set the acceleration and deceleration time constant in index type during each axis G76 thread cutting cycle. Parameter No.1622 is valid if it is not acceleration and deceleration in index type.

1	6	2	7
---	---	---	---

FL speed of acceleration and deceleration
in index type during each axis thread cutting cycle

『Range』 :

SETTING UNIT	DATA UNIT	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Machine in metric system	1 mm/min	0, 6~15000	0, 6~12000	30
Machine in inch system	0.1 inch/min	0, 6~6000	0, 6~4800	30

Set the minimum speed (FL speed) of acceleration and deceleration in index type during each axis G76 thread cutting cycle.

The bigger the acceleration and deceleration time constant is, the slower the acceleration and deceleration is, the smaller the machine running impact, the lower the processing efficiency is; the smaller the acceleration and deceleration time constant is, the quicker the acceleration and deceleration is, the bigger the machine running impact, the higher the processing efficiency is.

When the acceleration and deceleration time constants are same, the higher the starting/finishing speed of acceleration and deceleration is, the bigger the machine running impact, the higher the processing efficiency is; the lower the starting/finishing speed of acceleration and deceleration is, the slower the acceleration and deceleration is, the smaller the machine running impact, the lower the processing efficiency is.

The principle of acceleration and deceleration adjustment is to properly reduce the acceleration and deceleration time constant, to increase the starting/finishing speed of acceleration and deceleration, and to improve processing efficiency in the preconditions that the drive unit doesn't alarm, the motor doesn't step out and the machine running doesn't impact seriously. If the acceleration and deceleration time constant is too small, and the starting/finishing speed is set too high, which easily cause the drive alarm, the motor out-of-step or the machine vibration.

With the step motor drive unit, it tends to result in the motor out-of-step if the rapid movement speed is too high, the acceleration and deceleration time constant is too small, and the starting/finishing speed of acceleration and deceleration is too high.

With AC servo drive unit, to improve the processing efficiency, the starting speed can be set very high and the acceleration and deceleration time constant can be set very small. If try to get the best acceleration and deceleration, the time constant of acceleration and deceleration can be set as 0 through adjusting AC servo acceleration and deceleration parameter.

When the system is equipped with the machine output in metric system (parameter NO.1004#1 ISC is set as 0), the electrical gear ratio is 1:1, and the parameter is suggested to set as the following list:

	With stepped motor	With servo motor
Each axis rapid movement speed (parameter NO.1420)	≤ 4000 mm/min	≤ 8000 mm/min
F0 speed of each axis rapid movement override (parameter NO.1421)	≤ 400 mm/min	≤ 800 mm/min
Time constant of acceleration and deceleration in linear type during each axis rapid feeding (parameter NO.1620)	≥ 300	≤ 200
Acceleration and deceleration time constant of cutting and feeding after each axis interpolation (parameter NO.1622)	≥ 150	≤ 200
FL speed of acceleration and deceleration in index type during each axis cutting and feeding (parameter NO.1623)	≤ 50 mm/min	≤ 100 mm/min
Each axis JOG feedrate (parameter NO.1423)	≤ 500 mm/min	≤ 1000 mm/min
Each axis manual rapid movement speed (parameter NO.1424)	≤ 1500 mm/min	≤ 5000 mm/min
Acceleration and deceleration time constant during each axis JOG feeding after interpolation (parameter NO.1624)	≥ 150	≤ 200
FL speed of acceleration and deceleration in index type during each axis JOG feeding (parameter NO.1625)	≤ 50 mm/min	≤ 100 mm/min
Time constant of acceleration and deceleration during each axis thread cutting cycle (parameter NO.1626)	≥ 150	≤ 200
FL speed of acceleration and deceleration in index type during each axis thread cutting cycle (parameter NO.1627)	≤ 50 mm/min	≤ 100 mm/min
FL speed during each axis reference point return (parameter NO.1425)	≤ 250 mm/min	≤ 500 mm/min

The above parameter setting value is suggested value; about the detailed setting, refer to the actual situation of the drive unit and the motor characteristics and the machine loading capacity.

4.5 Mechanical zero point adjustment

GSK988T system supports three methods of setting the mechanical zero-point (also called as the reference point): the reference point free of the block, the reference point with the block and the encoder reference point in absolute type.

METHOD OF SETTING THE REFERENCE POINT (ZERO-RETURN MODE)	SETTING PARAMETER	REMARK
Encoder reference point of absolute type	System parameter NO.1815 APCx is set as 1	When the absolute position detector is used (absolute pulse encoder), set the reference point through the method.
Reference point free of block	System parameter NO.1815 APCx is set as 0; NO.1002 DLZ or NO.1005 DLZx is set as 1 (DLZ or DLZx is set as 1).	Axis doesn't use the absolute pulse encoder, if DLZ (no matter whether the reference point free of block is valid, all axes are valid) or DLZx (parameter in bit axial type) is set as 1, set the referene point through method free of block.
Reference point with block	The system parameter NO.1815 APCx is set as 0; NO.1002 DLZ and NO.1005 DLZx are set as 0.	Axis doesn't use the absolute pulse encoder, and the reference point free of block function is invalid, set the reference point through method with block.

The relative parameter of zero-return mode is as below:

- **Control parameter**

1815			APCx	APZx				APRx
-------------	--	--	-------------	-------------	--	--	--	-------------

『Parameter type』 : Bit axial type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : System

『Valid method』 : Power on

『Additional requirements』 : None

APRx: When use the absolute position encoder, the position direction of the absolute position detector

0: Not negate

1: Negate

APZx: When use the absolute position detector, the mechanical position and that of the absolute position detector

0: Not comply

1: Comply

APCx: Position detector

0: Not use the absolute position detector

1: Use the absolute position detector (absolute pulse encoder)

1002					AZR		DLZ	
------	--	--	--	--	-----	--	-----	--

『Parameter type』 : Bit type

『Default setting』 : 0000 0000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

DLZ: Whether the reference point free of block is valid

0: Invalid

1: Valid (all axes valid)

Remark:

When parameter DLZ is 0, parameter DLZx (No.1005#1) sets each axis valid or invalid.

AZR: when the reference point is not set, G28 command

0: Same as manual reference point return, use the deceleration block to return to reference point

1: P/S alarms

1005					HJZx		DLZx	ZRNx
------	--	--	--	--	------	--	------	------

『Parameter type』 : Bit axial type

『Default setting』 : 0000 1000

『Range』 : 0 or 1

『Authority to rewrite』 : Machine

『Valid method』 : Valid immediately

『Additional requirements』 : None

ZRNx: When the reference point isn't set, in auto running (MEM, DNC or MDI), the movement commands except G28 are specified, whether the system alarms

0: Alarm

1: Not alarm

DLZx: Whether the reference point free of block is valid

0: Invalid

1: Valid

Remark:

When it is "0" parameter DLZ (No.1002#1) is valid. When DLZ (No.1002#1) is "1", there isn't any connection with the parameter, the function of the reference point free of block is valid for all axes.

HJZx: After setting the reference point, manually return to the reference point

0: Use the deceleration block to return to the reference point

1: No connection with the deceleration block, rapidly position to the reference point

1006			ZMIx					
------	--	--	------	--	--	--	--	--

『Parameter type』 : Bit axial type

『Default setting』 : 0000 0000

『Valid method』 : Power on

ZMlx: Set each axis reference point return direction

- 0: Positive direction
- 1: Negative direction

3009			DECx					
------	--	--	------	--	--	--	--	--

『Parameter type』 : Bit axial type

『Default setting』 : 0010 0000

『Valid method』 : Valid immediately

DECx: Deceleration signal during reference point return



- 0: Decelerate when the signal is 0 (low level)
- 1: Decelerate when the signal is 1 (high level)


● **Setting the reference point of encoder in absolute type**

When the machine is equipped with the absolute position encoder and its reference point return function is valid, it requires setting the absolute position encoder reference point while the system doesn't set the reference point or readjust the reference point. After setting the reference point, the system auto saves the reference point position after power off, therefore, it doesn't require setting the reference point position, again when it powers on in the next time.


The steps of setting the reference point of encoder in absolute type are as below:

- 1. In MDI mode, the system parameter APZx (NO.1815#4) is set as 0, cut off power supply, and power on, again, the system alarms: Set the reference point (it requires manual reference point return).

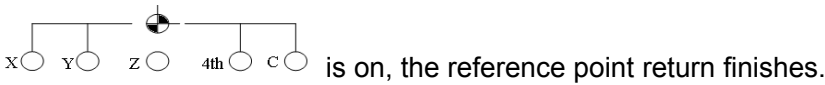
- 2. In mode of manual  or MPG  , the axis carriage is moved to the reference point to be set.

- 3. Press  on the panel, and switch the current mode into the reference point return mode;



- 4. Press the switches  relative to the reference point return, the system reads in the corresponding axis and direction selection signals and sets the reference point (the machine zero point) in the position of the current axis carriage.



- 5. The system saves the current encoder position, auto sets the parameter APZ x as 1, the reference point return finishing signal ZPx and the reference point setting signal ZRFx as 1, the reference point return finishing indicator (operational panel indicator)




is on, the reference point return finishes.

● **Setting the reference point free of block**

The steps of setting the reference point free of block are as below:

1. In mode of manual  or MPG , the axis carriage is moved close to the position of the reference point to be set.

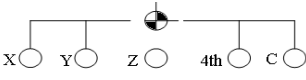
2. Press  on the machine panel, the current mode switches into the reference point return;



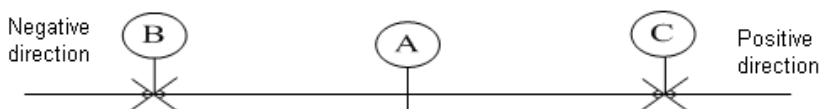
3. Press the switches relative to the reference point return, the system reads in the corresponding axis and direction selection signals and returns to the reference point.

4. The axis carriage is moved to the reference point in the direction set by parameter ZMlx (the 5th bit of parameter NO.1006) and at FL speed (zero-return low speed) set by parameter 1425.

5. When the system detects the first motor one-turn signal nPC, set the reference point return finishing signal ZPx and the reference point setting signal ZRFx as 1, the reference point

return finishing indicator (LED)  is on, the reference point return completes.

The relation between the reference point setting position and the machine initial position is as below:



A: Execute the position before reference point return free of block


B: Execute the reference point position after reference point return in negative direction, namely, the position of the first PC signal after point A moves negatively.


C: Execute the reference point position after reference point return in positive direction, namely, the position of the first PC signal after point A moves positively.

● Setting the reference point with block

The steps of setting the reference point free of block are as below:

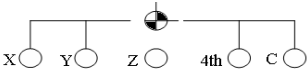
1. Only after confirming the overtravel limit switch valid, the mechanical zero-return can be operated.

2. Press  on the machine panel, switch the current mode into the reference point return;

3. To reduce zero-return speed, the rapid movement override switches  are firstly switched into the low-speed gear;



4. Press switches relative to the reference point return; the system reads in the corresponding axis and direction selection signals and returns to the reference point.
5. The carriage moves to the deceleration point at the rapid movement speed, the deceleration signal DECx is valid (the signal valid level set by parameter NO.3009 DECx), the movement speed reduces to 0, and then moves at zero-return speed set by parameter 1425.
6. When the tool is off from the deceleration switch position, the deceleration signal DECx is set as 1, the system begins to detect motor one-turn signal nPC.
7. After the system detects the first nPC signal of motor, set the reference point return finishing signal Zpx and the reference point setting signal ZRFx as 1, the reference point return finishing

indicator (LED)  is on, the reference point return completes.

Remark: If the system zero-return method is set as the reference point free of block/with block, set the reference point through executing the reference point setting when the system powers on every time.

Usually, the mechanical zero-point block is installed in the position of the maximum stroke, the zero-return block stroke is valid more than 25mm, to gurantee enough deceleration distance and ensure the required speed can be reduced, then, the zero-return can be completed correctly. The quicker the speed of the mechanical zero-return is, the longer the zero-return link stopper is; otherwise, the zero-return precision gets affected due to not enough deceleration distance, which is caused by CNC acceleration and deceleration, the machine inertia which makes the carriage break through the zero-return block. Moreover, for safety, before zero-return, ensure the carriage doesn't interfere with the other parts of the machine during zero-return process.

The connection method of adapted AC servo motor: respectively use the sketch map of one limit switch and one-turn signal of the servo motor.

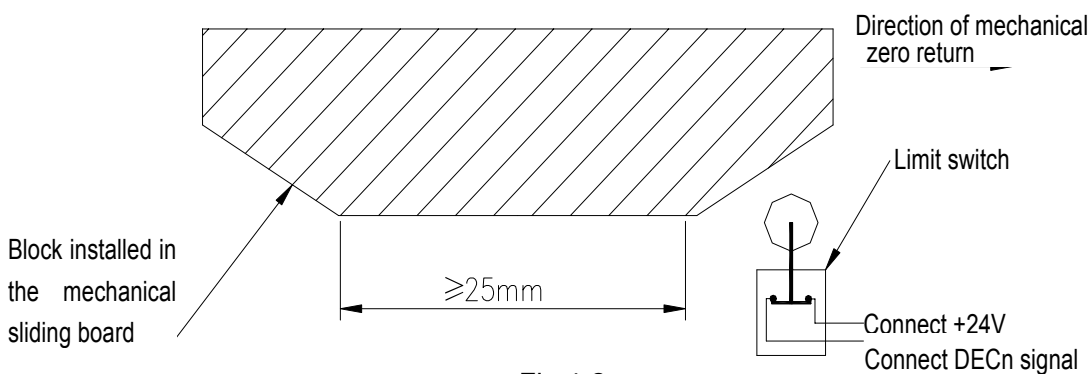


Fig 4-2

When use the method, after release the deceleration switch during mechanical zero-point return, one-turn signal of the encoder should avoid be in the critical point after release the limit switch, and ensure the motor rotates for half of a cycle to reach one-turn signal of the encoder to improve the zero-return precision. The block position can be adjusted a little to reduce zero-return error.

4.6 Spindle adjustment

4.6.1 Spindle encoder

3	7	0	6							PG2	PG1
---	---	---	---	--	--	--	--	--	--	-----	-----

PG2, PG1: The gear ratio of the spindle and the position encoder

Gear ratio=spindle speed/position encoder speed

Gear ratio	PG2	PG1
×1	0	0
×2	0	1
×4	1	0
×8	1	1

3	7	0	7							P22	P21
---	---	---	---	--	--	--	--	--	--	-----	-----

P22, P21 The gear ratio between the spindle and the 2nd position encoder.

Gear ratio=spindle speed/ position encoder speed

GEAR RATIO	PG2	PG1
×1	0	0
×2	0	1
×4	1	0
×8	1	1

Remark:

The parameter is only valid during multi-spindle control.

If the machine processes the thread, the encoder must be installed, and the quantity of the encoder pulses is 100~9999, which is set by parameter 3773 (the 1st spindle) and 3803 (the 2nd spindle).

4.6.2 Spindle break

After executing M05 command, to stop the spindle immediately and improve the processing efficiency, the proper spindle break time must be set, because the long time of breaking easily cause the motor burnout during using the motor energy consumption break.

4.6.3 Spindle speed switch control

The machine uses the multi-speed motor for control, and controls the spindle gears through I/O port of PLC.

4.6.4 Spindle speed analog voltage control

Realize spindle speed analog voltage control through setting CNC parameter, the analog voltage 0V~10V output by interface controls the transducer to realize the infinitely variable speeds; the relative parameter is adjusted as below:

NRF: The first movement command after the serial spindles switch into CS outline control axes (G00 and G01, etc)

1: usual positioning

EVS: When use spindle control function (spindle analog output or spindle serial output), S command

1: output S code and SF

[illegible]

Gear ratio=spindle speed/position encoder speed

Gear ratio=spindle speed/position encoder speed

Remark: The parameter is only valid when multi-spindle controls.

471

check in the thread cutting block after the 2nd block.

TSO: During thread processing or tapping cycle, the spindle override

0: Invalid (fixed in 100%)

1: Valid

Remark: In rigid tapping, the override is fixed as 100% and there isn't any connection with the parameter.

3	7	0	9
---	---	---	---

								SAM
--	--	--	--	--	--	--	--	-----

SAM: Sampling times during getting the spindle average speed

0: 4 times (usually set as 0)

1: 1 time

3	7	3	1
---	---	---	---

Compensation value of the spindle speed analog output offset voltage

〔Range〕 : -1024~+1024

The parameter sets the compensation value of the spindle speed analog output offset voltage.

1. Set standard setting value as 0.

2. Command the theoretical spindle speed when the analog output voltage is 0V.

3. Measure the output voltage.

4. After setting the parameter, command the theoretical spindle speed, again when the analog output voltage is 0V, and the voltage must be 0V.

3	7	4	1
3	7	4	2
3	7	4	3
3	7	4	4

Spindle maximum speed of gear 1
Spindle maximum speed of gear 2
Spindle maximum speed of gear 3
Spindle maximum speed of gear 4

〔Range〕 : 0~32767r/min

The parameter sets the spindle maximum speed of each gear.

3	7	7	0
---	---	---	---

The axis as calculation benchmark during constant surface speed control

〔Range〕 : 1~quantity of the control axes

The parameter sets the axis as the calculation benchmark during constant surface speed control.

Remark: When it is set as 0, default it as X axis. Then, commanded P value in G96 block doesn't have any meaning to the constant surface speed control.

3	7	7	1
---	---	---	---

The spindle minimum speed in constant surface speed control mode (G96)

〔Range〕 : 0~32767

The parameter sets the spindle minimum speed during the constant surface speed control. During the constant surface speed control (G96), it is limited in the parameter speed when the spindle speed is lower than the speed set by the parameter.

3	7	7	2
---	---	---	---

Spindle maximum speed

〔Range〕 : 0~32767r/min

The parameter sets the spindle maximum speed. The actual spindle speed is limited in the maximum speed set by the parameter when the commanded spindle speed or the spindle speed after override exceeds the spindle maximum speed.

- Remark: 1.When the constant surface speed controls, no matter whether command G96 or G97, the spindle speed is limited by the maximum spindle speed.
2. When the setting value is 0, it isn't limited by the speed.
3. When PMC controls the spindle speed, the parameter is invalid and it isn't limited by the maximum speed.
4. During multi-spindle control, the following parameters set each spindle maximum speed:

When parameter is №.3772, set the maximum speed of the 1st spindle.

When parameter is №.3802, set the maximum speed of the 2nd spindle.

When parameter is №.3822, set the maximum speed of the 3rd spindle.

3	8	0	2	The maximum speed of the 2 nd spindle
『Range』 : 0~32767r/min				

The parameter sets the maximum speed of the 2nd spindle. The actual spindle speed is limited in the maximum speed set by the parameter when the commanded spindle speed or the spindle speed after override exceeds the spindle maximum speed.

- Remark: 1.When the multi-spindle control, the parameter is valid.
2. When the constant surface speed controls, no matter whether command G96 or G97, the spindle speed is limited by the maximum speed.
3. When the setting value is 0, parameter №.3772 is valid (the maximum speed of the 1st spindle). When parameter №.3772 is 0, the spindle speed isn't limited.
4. When PMC controls the spindle speed, the parameter is invalid and not limited by the maximum speed.

3	8	1	1	The spindle maximum speed of the 2 nd spindle gear 1
3	8	1	2	The spindle maximum speed of the 2 nd spindle gear 2
『Range』 : 0~32767r/min				

The parameter sets the maximum speed of each gear in the 2nd spindle.

Remark: It's for multi-spindle control.

S command of the analog spindle control: S command is input by the machine program and specifies the analog spindle speed controlled by CNC. About the constant surface speed cutting (G96 mode), CNC switches the specified surface linear speed into the spindle speed.

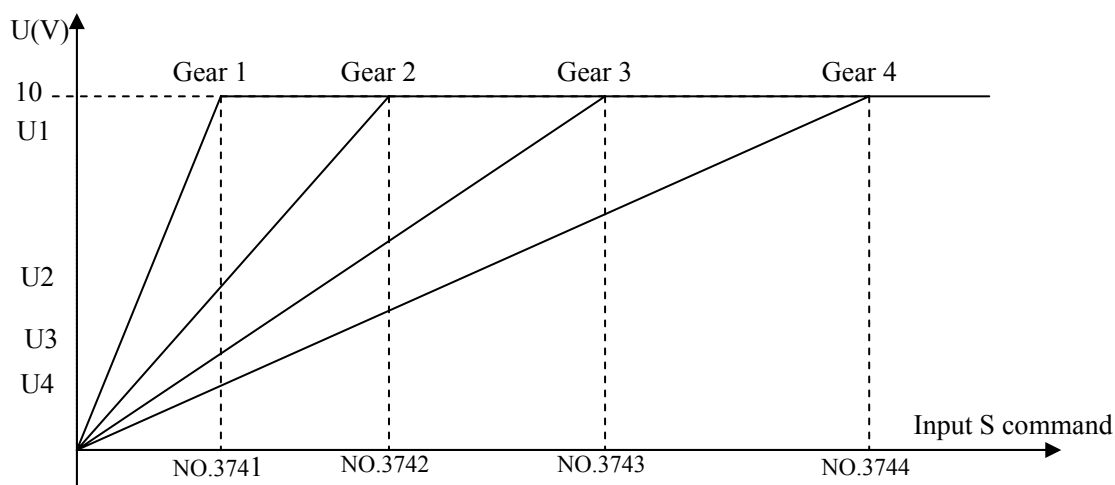
S code/SF signal output: The analog spindle control function in CNC can output S command value into PLC in codes of the binary system rather than SF.

Switching gears: Although S command specifies the spindle speed, the actual controlled object is the spindle motor. Therefore, there is the certain corresponding relation between the spindle motor speed and the gear in CNC. The machine sets the gear, and CNC outputs the spindle speed relative to the gear.

Method of switching gears: To switch the gears, the maximum spindle speed corresponding to each gear must be set by the data parameter №.3741~№.3744 (the 2nd spindle is set by №.3811~№.3814.). The gear selecting signal is coding signal in 2 bits (GR1 and GR2) , the relation between the signal and the gear is as below:

GR1	GR2	GEAR	SPINDLE MAXIMUM SPEED PARAMETER NUMBER
0	0	1	Data parameter №.3741
1	0	2	Data parameter №.3742
0	1	3	Data parameter №.3743
1	1	4	Data parameter №.3744

The data parameter №.3741~№.3744 respectively set the spindle maximum speed in the 1st ~ 4th gear, the analog voltage 10V relative to the maximum speed of the spindle motor. Then, one S command in different gears, the analog voltage output is displayed as the following linear relation:



Data parameter №.3741~№.3744: The maximum speed is limited in the spindle speed 1~4 gears; Default the spindle in the 1st gear when CNC powers on.

The basic parameter adjusted by the transducer:

Selecting CW-CCW command mode: set by terminal VF;

Selecting frequency setting mode: set by terminal FR.

When the speed specified by the program doesn't comply with that detected by the encoder, it can be adjusted by data parameter №.3730 and №.3731 to get the specified speed in compliance with the actual speed.

Speed adjusting method: Select the spindle in the 1st gear, input the commands of the 1st gear maximum speed (S + value set by No.3741) in MDI interface and make the spindle revolve, the calculation increment= $1000 \times \text{No.3741 setting value} \times \text{the actual speed}$, and input the calculation result into parameter №3730;

In MDI interface, input SO, the actual speed is input into parameter №3731. The calibrated voltage offset compensation value set by parameter №3731 is adjusted before delivery, and normally it doesn't require adjusting, again.

4.7 Backlash compensation

The backlash compensation value is related to the programming system (diameter programming or radius programming). (diameter/radius programming directly affects the minimum movement unit.) If the axis is the diameter programming, the value set by the parameter is the diameter value; if the axis is the radius programming, the value is the radius value, all the units are detection units (the minimum movement units/CMR), refer to the introduction of the detection units, the minimum movement units and the command override ratio in chapter 4.3.

Detection units = the minimum movement unit/command override ratio (CMR)

It can use the micrometer gauges, or laser apparatus to measure and the backlash should be compensated correctly, then, the precision of the processing can be improved. Therefore, MPG or single step isn't recommended to measure the thread backlash, it's suggested to measure the backlash through the following methods:

- Editing programs:

```
O0001;
N10 G01 W10 F800 ;
N20 W15 ;
N30 W1 ;
N40 W-1 ;
N50 M30 。
```

- The backlash error compensation value is set as 0 before measuring;
- When the single block runs, find the measure datum A after positioning for two times, record the current data, and moves 1mm in the same direction, and then moves 1mm to point B in opposite direction, read the current data.

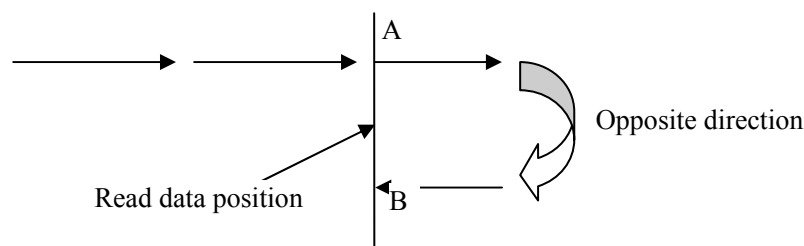


Fig 4-4 Sketch map of backlash measuring method

- Backlash error compensation value = |record data of point A - record data of point B|; and convert the calculated data into the detection units and then input to CNC data parameter №1851.

Data A: In position A, the data of micrometer can be read;

Data B: In position B, the data of micrometer can be read;

Detection unit = Minimum movement unit/CMR;

For example: The system is set as IS-B through parameter (parameter NO.1004#1 ISC is set as 0), the machine in metric system (parameter NO.1001#0 INM is set as 0), if parameter NO.1820 (command override ratio of setting each axis) sets the value as 2, the command override ratio CMR=1 based on the parameter meaning; the axis is programmed by the radius.

Therefore: Detection unit = the minimum movement unit/CMR = 0.001mm / 1 = 0.001 mm;

If the backlash error compensation value measured by the multimeter gauge is 0.030mm, the parameter NO.1851 is set as 30.

The above is the backlash compensation of cutting feed; if the backlash compensation value can be changed through the cutting feedrate/rapid movement speed, the processing can be operated in higher precision. The backlash compensation value during rapid movement is set by parameter №1852, for example, the measured backlash during cutting feed is A, the measured backlash during rapid feeding is B, the compensation values are shown as the following list based on the changing feedrates and the movement directions:

CHANGING FEEDRATES AND MOVEMENT DIRECTIONS	FROM CUTTING FEED TO CUTTING FEED	FROM CUTTING FEED TO RAPID MOVEMENT	FROM RAPID MOVEMENT TO RAPID MOVEMENT	FROM RAPID MOVEMENT TO CUTTING FEED
Same direction	0	$\pm (-d)$	0	$\pm d$
Opposite direction	$\pm A$	$\pm (B+d)$	$\pm B$	$\pm (B+d)$

When $d = (A-B)/2$, the compensation direction is the movement direction no matter it is position or negative.

The relative parameter is as below:

1	8	0	0	BDEC	BD8		RBK				
---	---	---	---	------	-----	--	-----	--	--	--	--

RBK: Respectively compensate the backlash during cutting feed and rapid movement

0: Not compensate

1: Compensate

BD8: Pulse output frequency of the backlash compensation

0: Compensate in the frequency set by parameter # 1853

1: Compensate in 1/8 of frequency set by parameter # 1853

BDEC: Backlash compensation mode

0: Output in the fixed pulse frequency (set by parameter # 1853 and #1800.6)

1: Output pulse frequency based on the acceleration and deceleration

1	8	5	1	Backlash compensation value of each axis							
---	---	---	---	--	--	--	--	--	--	--	--

『Range』: -9999~+9999 (Detection units)

Set the backlash compensation value of each axis.

After connecting the power supply, the machine moves in the opposite direction of the reference point return and compensate the backlash at the first time.

1	8	5	2	Backlash compensation value during each axis rapid movement							
---	---	---	---	---	--	--	--	--	--	--	--

『Range』: -9999~+9999 (Detection units)

Set the backlash compensation value during each axis rapid movement. It is valid when parameter NO.1800#4(RBK) is set as 1. To improve the higher precision of processing, the backlash compensation value can be changed based on the cutting feedrate/rapid movement speed.

1	8	5	3				CPF5	CPF4	CPF3	CPF2	CPF1
---	---	---	---	--	--	--	------	------	------	------	------

『Range』: 0 or 1

CPF1~CPF5: Setting value of backlash compensation pulse frequency (in BCD codes)

Setting frequency = (setting value +1) Kpps

2	0	7	1	Acceleration or deceleration valid time constant of each axis backlash							
---	---	---	---	--	--	--	--	--	--	--	--

『Range』: 0~100 ms

Set the acceleration or deceleration valid time constant of each axis backlash.

Remark: 1. Manual continuous feeding (JOG) is taken as the cutting feed.

2. After connecting power supply, before return to the reference point at the first time, the backlash

of cutting feed/rapid movement isn't compensated. The normal compensation value is compensated through parameter NO.1851, no matter it is during cutting feed or rapid movement.

3. When the backlash is respectively compensated during cutting feed/rapid movement, it is valid only when parameter NO.1800#4(RBK) is 1. When it is 0, the backlash is compensated normally.

Remark : Check the backlash once when the machine is used in three months.

4.8 Thread error compensation

If the screw pitch error compensation value is defined, the thread error compensation of each axis can be compensated based on the detection units of each axis.

Set the screw pitch error compensation data for each compensation position, and its compensation position is set based on the space between each axis. The compensation origin is the reference position for the tool return.

When compensate the screw pitch error, the following parameters must be set:

Parameter 3620: Each axis is in the position number of the screw pitch error compensation in the reference point.

Parameter 3621: The minimum position number of each axis screw pitch error compensation

Parameter 3622: The maximum position number of each axis screw pitch error compensation

Parameter 3623: The scalling rate of each axis screw pitch error compensation

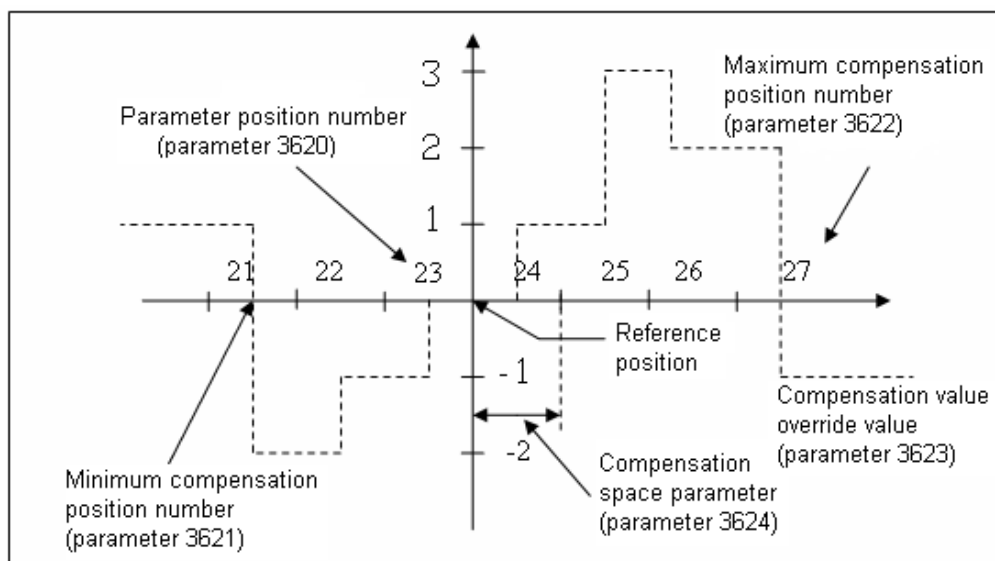
Parameter 3624: The position space between each axis screw pitch error compensation

The space value, which is set by parameter 3624, between the screw pitch error compensation points isn't relative with the radius or the diameter programming, all the setting values are the radius values, and the units are as below:

The setting values all are radius values, no matter the interval value, which is set by parameter, of thread error compensation point (parameter 3624 setting value) and its axes are programmed by the diameter or the radius, the setting units are as below:

	IS-B system	IS-C system
Machine in metric system	0.001mm	0.0001mm
Machine in inch system	0.001inch	0.0001inch

About the setting value of the screw pitch error compensation value in each detailed point (parameter→the setting value in the screw pitch compensation interface), the units are the detection units (the minimum movement unit/CMR). The axis is relative with the radius or the diameter programming, if it is the diameter programming, the compensation setting value is the diameter value; if it is the radius programming, the setting value is the radius value (diameter/radius programming directly affects the system minimum movement units.)



COMPENSATION POSITION NUMBER	21	22	23	24	25	26	27
SET COMPENSATION VALUE	-3	+1	+1	+1	+2	-1	-3

Define the compensation position: To specify the compensation position for each axis, specify the positive or negative direction for compensation based on the reference point. If the machine stroke exceeds the specified range in positive or negative direction, the screw pitch error compensation doesn't work outside of the range.

Compensation position number: In screw pitch error compensation setting interface, total 1024 compensation positions 0~1023 can be used. The parameter can be used as any distribution position number of each axis.

In each axis, it requires setting the compensation position number (parameter 3620) of the reference point; the compensation minimum position number (parameter 3621) and the compensation maximum position number (parameter 3622).

For example:

1. Linear axis

Machine stroke: $-400\text{mm} \sim +800\text{mm}$

Space between the screw pitch error compensation positions: 50mm

Compensation position number of the reference point: 70

After the above is defined, the furthest compensation position number in negative direction is as below:

The compensation position number of the reference point $-$ (machine stroke in negative direction/space between compensation positions) $= 70 - 400/50 + 1 = 63$

The furthest compensation position number in positive direction is as below:

Compensation position number of the reference point $+$ (machine stroke in positive direction/space between compensation positions) $= 70 + 800/50 = 86$

The corresponding relation between the machine and the compensation point position number is shown as below:

PARAMETER	SETTING VALUE
3620: Parameter point compensation number	70
3621: Minimum compensation position number	63
3622: Maximum compensation position number	86
3623: Compensation magnification	1
3624: Space between the screw pitch error compensation positions	50000

2. Rotation axis

Movement value/ revolution: 360°

Space between the screw pitch error compensation positions: 45°

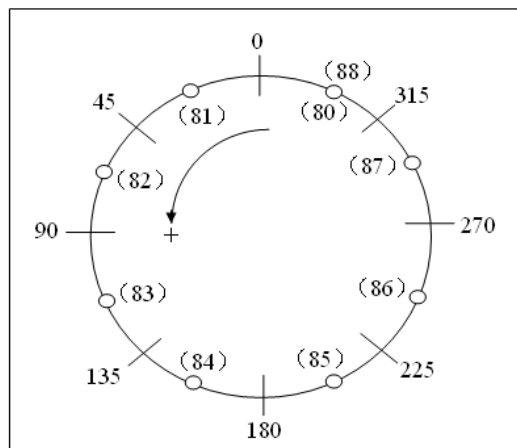
Compensation position number of the reference point: 80

After defining the above parameter, the furthest compensation position number in negative direction of the rotation axis is the compensation position number of the reference point.

The furthest compensation position number in positive direction is as below:

The compensation position number of the reference point + (movement value of each revolution/ space between compensation positions) = $80 + 360/45 = 88$

The corresponding relation between the machine coordinate and the compensation position number is as below:



The parameter is set as below:

PARAMETER	SETTING VALUE
3620: Compensation number of the reference point	80
3621: Minimum compensation position number	80
3622: Maximum compensation position number	88
3623: Compensation magnification	1
3624: Space between screw pitch error compensation positions	45000

For the rotation axis, there may result in the position offset if the sum of the compensation value of positions 81~88 isn't 0. The sum is the accumulation of screw pitch error compensation value of each revolution. Moreover, in the compensation positions of 80 and 88, the same compensation value must be set.

For example:

NUMBER OF COMPENSATION POSITION	80	81	82	83	84	85	86	87	88
SET COMPENSATION VALUE	+1	-2	+1	+3	-1	-1	-3	+2	+1

3620

Screw pitch error compensation number of each axis reference point (NPR)

『Range』 : 0~1023

Set the screw pitch error compensation number of the reference point in each axis.

3621

**Furthest screw pitch error compensation point number
of each axis in negative direction (NEN)**

『Range』 : 0~1023

The parameter sets the furthest screw pitch error compensation point number of each axis in negative direction.

3622

**Furthest screw pitch error compensation point number
of each axis in positive direction (NEP)**

『Range』 : 0~1023

The parameter sets the furthest screw pitch error compensation point number of each axis in positive direction.

The value set by the parameter should be greater than that of parameter NO.3620.

3623

Each axis screw pitch error compensation override (PCM)

『Range』 : 0~100

Set the override of each axis screw pitch error compensation.

If the override is set as 1, the detection unit should be same as the compensation unit.

If the override is set as 0, the override is same as that when it is set as 1.

3624

Space between each axis screw pitch error compensation points (PCI)

『Range』 : 0~99 999 999

SETTING UNITS	IS—B	IS—C	UNITS
Input in metric system	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch
Rotation axis	0.001	0.0001	deg

The screw pitch compensation points are distributed as the same space. The minimum value of the space is limited by the following formula: the minimum value = the maximum feedrate (rapid feedrate)/7500.

Units: Minimum space between the screw pitch compensation: mm, inch and deg.

Maximum feedrate: mm/min, inch/min and deg/min.

(For example) When the maximum feedrate is 15000mm/min, the minimum value of the screw pitch error compensation space is 2mm.

But, based on the set override, when the absolute value of the compensation value of its point exceeds the 100, the space between each compensation points is magnified through the override which is calculated through the following formula:

Override = Maximum compensation value (absolute value)/128 (round up after the decimal point)

Screw pitch compensation minimum space = the value calculated from the above maximum feedrate * the override

Remark:
The unit of the screw pitch compensation value and that of the detection unit are same.
The detection unit is relative with the parameter No.1820 (command override ratio CMR) and the minimum movement unit; refer to the introduction of parameter No.1820 about the relation between the setting unit and the minimum movement unit.

	#7	#6	#5	#4	#3	#2	#1	#0
3628				NPF5	NPF4	NPF3	NPF2	NPF1

『Range』 : 0 or 1

The setting value of the screw pitch compensation pulse frequency (in BCD codes) .

Setting frequency= (setting value +1) Kpps

Chapter V PLC introduction

5.1 PLC specification

PLC system version		NP1
Programming language		Ladder diagram
Programming software		GSKCC-Ladder
Program level		2
Program execution cycle of the 1 st level		8ms
Basic command average processing time		<2μs
Program maximum steps		5000 steps
Programming command		Basic command + function command
Programming address	Internal relay address (R)	R0000~R0999
	Information display request address (A)	A0000~A0024
	Timer address (T)	T0000~T0099
	Counter address (C)	C0000~C0099
	Data list address (D)	D0000~D0999
	Relay address of maintenance type (K)	K0000~K0039
	Counter preset value address (DC)	DC0000~DC0099
	Timer preset value address (DT)	DT0000~DT0099
	Subprogram address (P)	P0000~P9999
	Label address (L)	L0000~L9999
	Machine→PLC address (X)	X0000~X0029
	PLC→machine address (Y)	Y0000~Y0029
	CNC→PLC address (F)	F0000~F0255
	PLC→CNC address (G)	G0000~G0255

5.2 PLC address

The diagnosis is for detecting machine→PLC, PLC→machine, CNC→PLC, PLC→CNC and the signal state of alarm information address A, which can't be rewritten.

5.2.1 X address (machine→PLC)

Fixed address definition:

Address	Function
X0.3	X axis deceleration signal
X0.5	Emergency stop input signal
X1.3	Z axis deceleration signal
X2.3	Y axis deceleration signal
X2.4	Deceleration signal of the 4 th axis
X2.5	Servo spindle deceleration signal
X3.5	G31 jumping signal

Refer to the appendix 2 about the standard PLC ladder diagram address X definition.

5.2.2 Y address (PLC→machine)

Refer to appendix 2 about the standard PLC ladder diagram address Y definition.

5.2.3 F address (CNC→PLC)

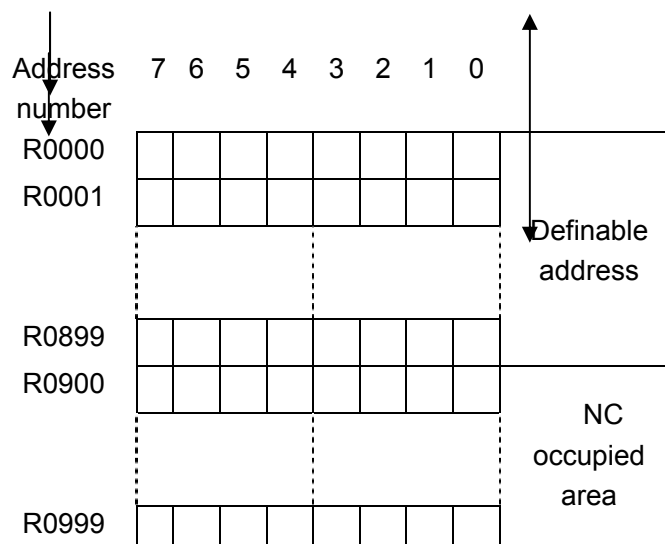
Refer to appendix 2 about the standard PLC ladder diagram address F definition.

5.2.4 G address (PLC→CNC)

Refer to the appendix 2 about the standard PLC ladder diagram address G definition.

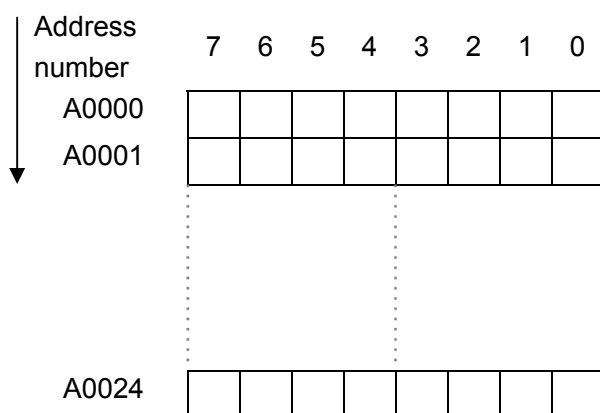
5.2.5 The internal relay address (R)

Address range: R0000.0~R0999.7, span: 0, 1. The address area is cleared zero when CNC powers on.



5.2.6 Information display request address (A)

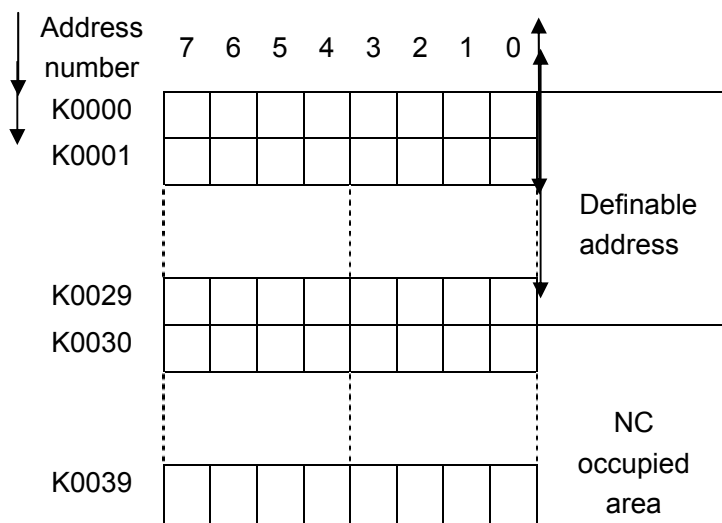
Address range: A0000.0~A0024.7. When CNC powers on, the address area is cleared zero for PLC alarm.



Refer to the appendix 2 about the standard PLC ladder diagram address A definition.

5.2.7 The relay address of maintenance type (K)

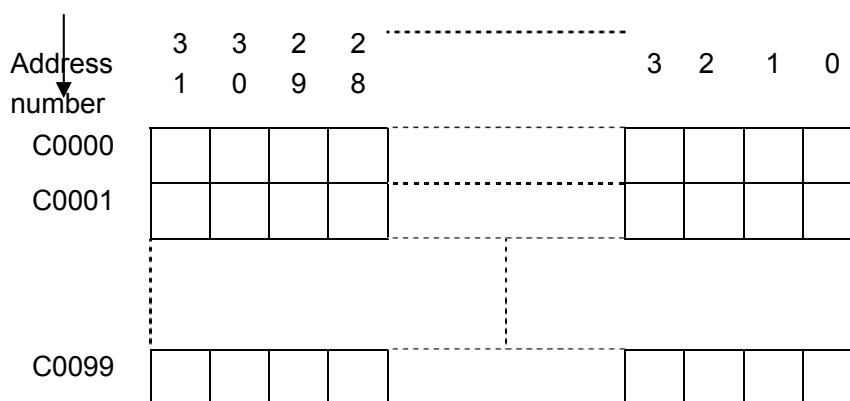
The address area is for relay of maintenance type and setting PLC parameter, the data are saved when power is off, the address range: K0000.0~K0039.7, span:0, 1.



Refer to the appendix 2 about the standard PLC ladder diagram address K definition.

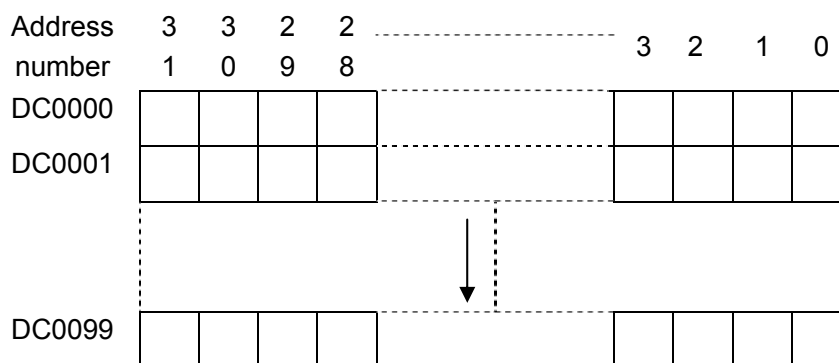
5.2.8 The counter address (C)

The address area is for saving the current calculation value of counter, the data is saved in the area when power is off, address range: C0000~C0099. Span: 0~21,4748,3647.



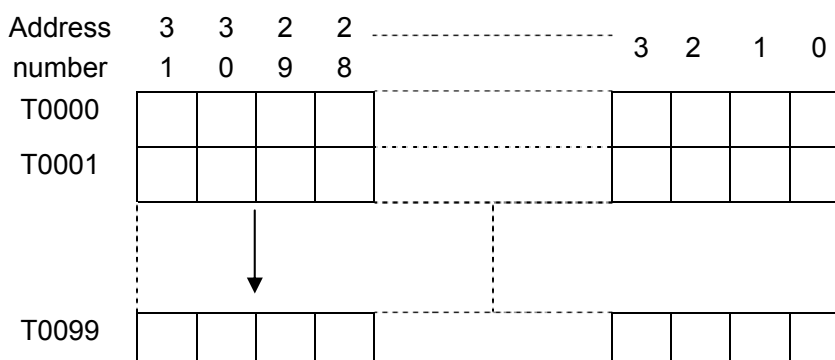
5.2.9 Address of the counter preset value (DC)

The address area is for saving preset value of the counter, and the data is saved when power is off. Address range: DC0000~DC0099, span: 0~21,4748,3647.



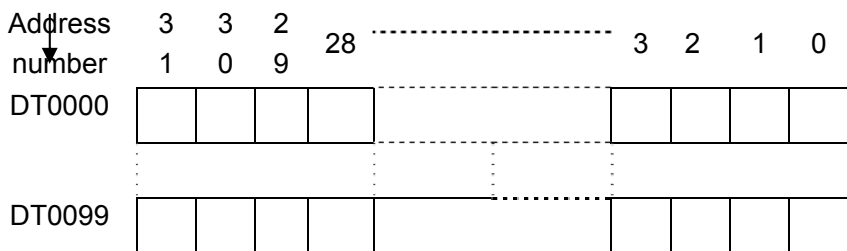
5.2.10 The timer address (T)

The address area is for saving the current numerical value of timer, and the data are saved when power is off. Address range: T0000~T0099, span: 0~21,4748,3647.



5.2.11 Preset value address of the timer (DT)

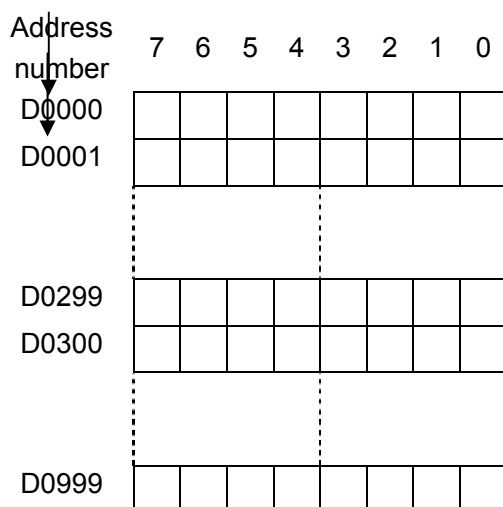
The address area is for saving preset value of the timer, and data are saved when power is off. Address range: DT0000~DT0099, span: 0~21,4748,3647.



Refer to the appendix 2 about the standard PLC ladder diagram DT address definition.

5.2.12 Address of the data list (D)

D0000~D0999 address data are saved when power is off. Span: 0~255



Refer to the appendix 2 about the standard PLC ladder diagram address D definition.

5.2.13 The label address (L)

It is for specifying the label number of the jumping target in JMPB command and the label number of LBL command.

Span: L0~L9999

5.2.14 The Subprogram number (P)

It is for specifying the called target subprogram number in CALL command and the subprogram number in SP command.

Span: P0000~P9999

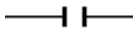
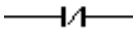
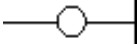

5.3 PLC basic commands

The basic commands are ones which has been used for maximum times during designing the sequence program, and the single-digit calculation is executed. There are the following basic commands in CNC:

Command name	Function	Operable parts
LD	Read normally open contact	X, Y, F, G, R, K, A
LDI	Read normally closed contact	X, Y, F, G, R, K, A
OUT	Drive output circuit	Y, G, R, K, A
OUTN	Drive outputs circuit when the conditions aren't satisfied.	Y, G, R, K, A
AND	Normally open contacts in series connection	X, Y, F, G, R, K, A
ANI	Normally closed contacts in series connection	X, Y, F, G, R, K, A
OR	Normally open contacts in parallel connection	X, Y, F, G, R, K, A
ORI	Normally closed contacts in parallel connection	X, Y, F, G, R, K, A
ORB	Series circuit in parallel connection	Void
ANB	Parallel circuit blocks in series connection	Void
MPS	Logic results stack	Void
MRD	Read stacking top logic results	Void
MPP	Display logic results of stacking top	Void

5.3.1 LD, LDI, OUT and OUTN

- Mnemonic symbols and function

Mnemonic symbols	Function	Symbols of ladder diagram
LD	Read in normally open contact status	
LDI	Read in normally closed contact status	
OUT	Drive output circuit	
OUTN	Output none	

- Command introduction

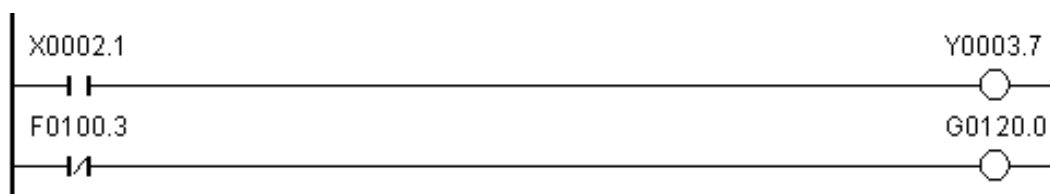
A: LD and LDI commands are for connecting the contacts with the bus-bar. The other methods are combined with ANB command, and it can also be used at the starting point of the branches.

B: OUT commands are those of drive output relay and internal relay circuit, which can't be used for input relay.

C: The parallel OUT commands can be used continuously for many times.

D: OUTN command negates the drive condition and outputs, the other usage method is same as that of OUT.

- Programming example



Program introduction:

Take X0002.1, output Y0003.7 if it is 1;

Take F0100.3, output G0120.0 if it is 0.

5.3.2 AND and ANI

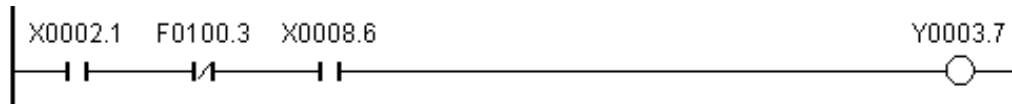
- Mnemonic symbols and function

Mnemonic symbols	Function	Symbols of the ladder diagram
AND	Normally open contacts in series connection	
ANI	Normally closed contacts in series connection	

- Command introduction

AND and ANI commands can series connect one contact. The quantity of series contacts is not limited, and the commands can be used for many times.

- Programming example



Program introduction:

Take X0002.1,

Take F0100.3 and X0002.1 in serial connection,

Take X0008.6 serial connects with F0100.3 and X0002.1,

If X0002.1 and X0008.6 are 1, F0100.3 are 0, output Y0003.7.

5.3.3 OR and ORI

- Mnemonic symbols and function

MNEMONIC SYMBOLS	FUNCTION	CODES OF LADDER DIAGRAM
OR	Normally open contacts in parallel connection	
ORI	Normally closed contacts in parallel connection	

- Command introduction

A: OR and ORI commands can parallel connect one contact. If there are two more contacts in

series connection, and the serial loop blocks parallel connect with the other loops, use the ORB command which is introduced in the following chapter.

B: OR and ORI start from the step of the command, and parallel connect with the above mentioned LD and LDI commands.

• Programming example



Program introduction:

Take X0002.1,

Take F0100.3 and X0002.1 in parallel connection,

If X0002.1 is 1 or F0100.3 is 0, output Y0003.7.

5.3.4 ORB

• Mnemonic symbol and function

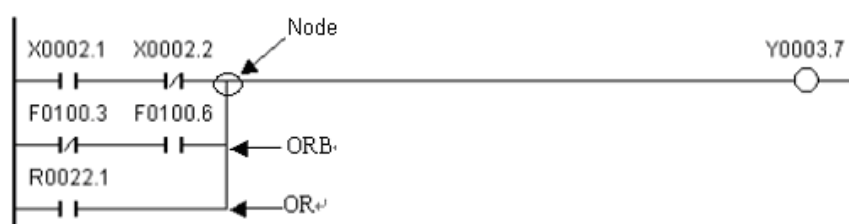
MNEMONIC SYMBOL	FUNCTION	CODES OF LADDER DIAGRAM
ORB	Parallel connection of serial circuits	

• Command introduction

A: The loop, which the two more contacts serial connect, is called as loop block of serial connection. When parallel connect serial loop blocks, the branches begin with commands of LD and LDI, and end with ORB command.

B: ORB command is the independent command free of the address.

• Programming example



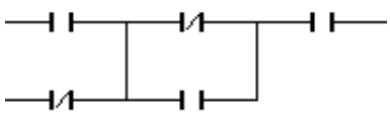
Program introduction:

When there are three circuit branches 0002, 0003 and 0004 from the left bus-bar to the node, the circuit branches 0002 and 0003 all are serial circuit blocks, which is shown in the above figure; when there are serial circuit blocks in parallel connection from the bus-bar to the node or from one node to the other node, ORB command is used at the end of branches except for the first branch. However, circuit branch 0004 isn't serial circuit block, just use OR command.

ORB and ANB are commands without operation units, which means OR or AND relation between circuit blocks.

5.3.5 ANB

• Mnemonic symbol and function

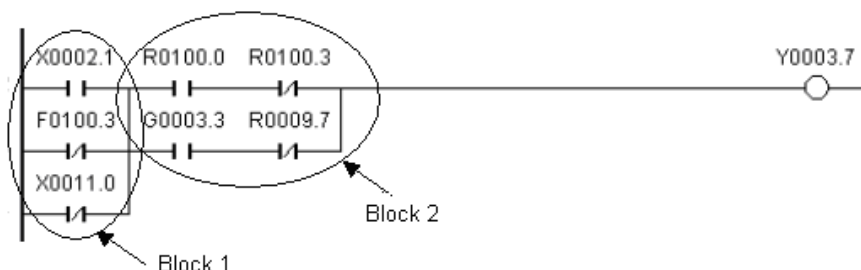
MNEMONIC SYMBOL	FUNCTION	CODES OF LADDER DIAGRAM
ANB	Parallel circuit in serial connection	

- Command introduction

A: ANB command is used when branch loops (loop blocks in parallel connection) serial connect with the previous loop. The starting point of branch uses LD and LDI commands, ANB command serial connects with the previous loops after completing the loop blocks in parallel connection.

B: ANB command is independent free of the address

- Programming example



5.3.6 MPS, MRD and MPP

GSK988T version is PLC system of NPI which supports multiple-level output.

MPS/MRD/MPP is called as the stack commands for multiple output circuit and it's convenient for programming. NPI supports 32 layers.

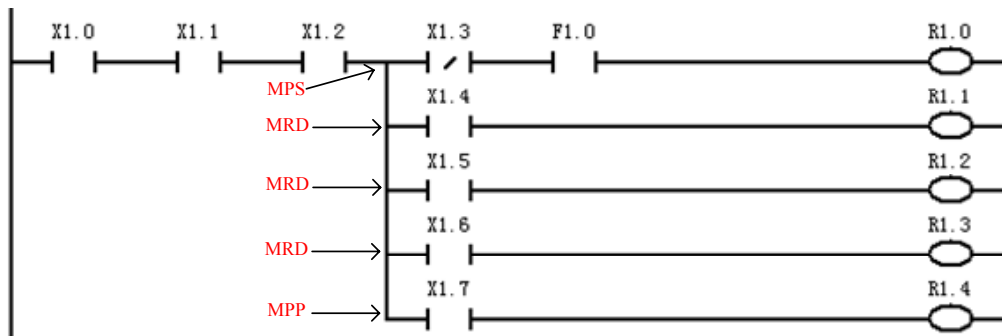
(1) MPS (push command) sends the calculation result to the first section of the stack memorizer and moves the previous data to the next section of block in order.

(2) MRD (reading stack command) reads the data, which is the last pushed, in the first section of the stack memorizer and continues to save the data in the first section of the stack memorizer; the data in the stack don't move.

(3) MPP (pop command) reads the data, which is the last pushed, in the first section of the stack memorizer and the data disappear from the stack, and the following data move upward in order.

The information after pushing can be limitlessly used; if the information is used at the last time, MPP command will pop up.

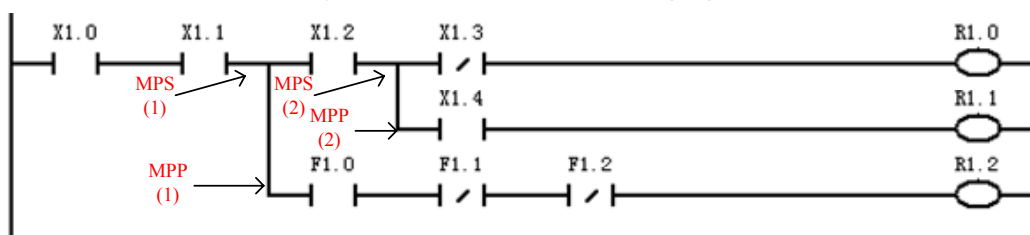
The stack of the first layer is shown as the following fig:



Command list:

```
LD      X0001.0
AND     X0001.1
AND     X0001.2
MPS
ANI     X0001.3
AND     F0001.0
OUT     R0001.0
MRD
AND     X0001.4
OUT     R0001.1
MRD
AND     X0001.5
OUT     R0001.2
MRD
AND     X0001.6
OUT     R0001.3
MPP
AND     X0001.7
OUT     R0001.4
```

The stack of the second layer is shown as the following fig:



Command list:

```
LD      X0001.0
AND     X0001.1
MPS
AND     X0001.2
MPS
ANI     X0001.3
OUT     R0001.0
MPP
AND     X0001.4
OUT     R0001.1
```

MPP
 AND F0001.0
 ANI F0001.1
 ANI F0001.2
 OUT R0001.2

The usage introduction of the stack command:

- 1) The stack command is free of target unit;
- 2) MPS and MPP must be used in pair.

5.4 PLC function command

When the basic commands are difficult to complete some function, the function commands can realize. PLC is with the following function commands:

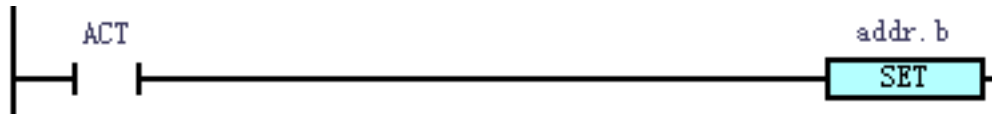
SR.NO	COMMAND NAME	FUNCTION
1	SET	Output after logic OR the logic calculation result and address value
2	RST	Nagate the logic calculation result, and then output after logic AND the result and the address value
3	CMP	Compare setting
4	CTRC	Counter
5	TMRB	Timer
6	MOVN	Data copy
7	PARI	Odd-even check
8	ALT	Alternate output
9	ROTB	Rotation control in binary system
10	DECB	Codes in binary system
11	CODB	Codes switch in binary system
12	JMPB	Program jumping
13	LBL	Program jumping label number
14	CALL	Subprogram calling
15	DIFU	Rising edge check
16	DIFD	Falling edge check
17	MOVE	Logic multiplication
18	ADDB	Addition in binary system
19	SUBB	Subtraction in binary system
20	MULB	Multiplication in binary system
21	DIVB	Division in binary system
22	WSHL	Data moves left in binary system
23	WSHR	Data moves right in binary system
24	WAND	Characters AND in binary system
25	WOR	Data OR in binary system
26	WXOR	Data AND-OR in binary system
27	WINV	Data negate in binary system

5.4.1 Set

- Command function

Set the specified address as 1.

- Format of ladder diagram



- Control conditions

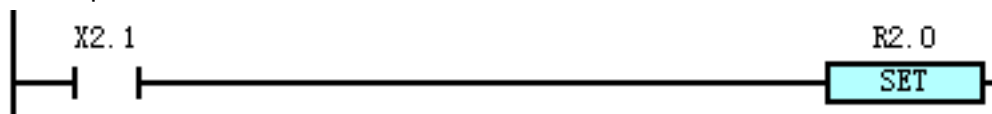
ACT =0: addr.b remains unchanged.

=1: addr.b is set as 1.

- Relative parameter

addr.b: Setting unit address bit, it can be contacts and output circuit, addr= Y、G、R、K、A.

- Program example:



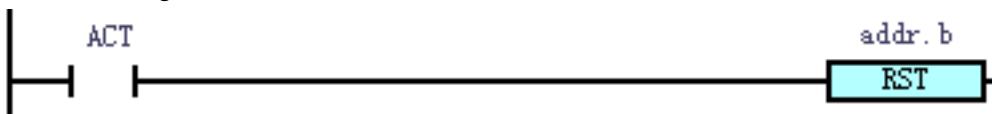
Remark: When X0002.1 is 1, R0002.0 is set as 1: when X0002.1 is 0, R0002.0 remains unchanged.

5.4.2 RST (reset)

- Command function

Set the specified address as 0.

- Format of ladder diagram



- Control conditions

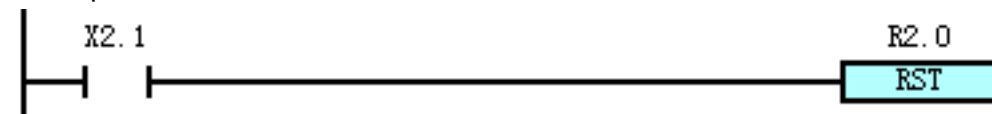
ACT =0:addr.b remains unchanged.

=1:addr.b is set as 0.

- Relative parameter

addr.b: Resetting unit address bit, it can be contanct or output circuit, addr= Y, G, R, K, A.

- Program example:



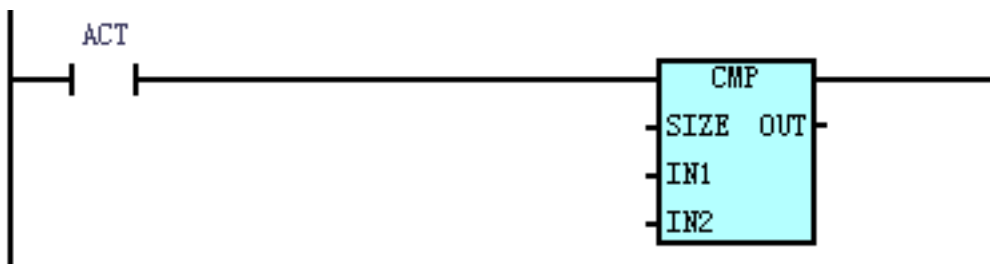
Remark: When X0002.1 is 0, R0020.0 remains unchanged; 1, R0020.0 is set as 0.

5.4.3 CMP (Data comparing in binary system)

- Command function

Compare the data for which is greater in binary system, output the comparing result.

- Format of ladder diagram



• Control conditions

It is supposed that the address of OUT is reprinted by addr.b, then

ACT=0: addr.b remains the original value

= 1: Compare IN1 and IN2, which one is greater, the output result is as below:

	addr.(b+2)	addr.(b+1)	addr.(b+0)
IN1> IN2	0	0	1
IN1= IN2	0	1	0
IN1< IN2	1	0	0

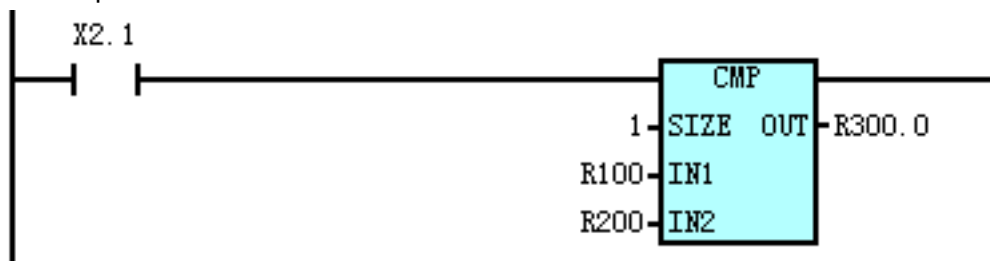
• Relative parameter

Size: Specify the size of data, when 1, 2 or 4 is set, the data size respectively is 1 byte, 2 bytes or 4 bytes.

IN1, IN2: The content of comparing original data 1 and 2 can be the constant or the address number (It can't be address bit, for example, addr.b is illegal.) The address numbers are R, X, Y, F, G, K, A, D, T, C, DC and DT, etc.

OUT: It can be comparing output result. It can be R, Y, G, K or A, etc.

• Program example:



Remark: When X0002.1 is 0, it can't be compared, and R0300.0 remains unchanged;

When X0002.1 is 1, compare, the result is as below:

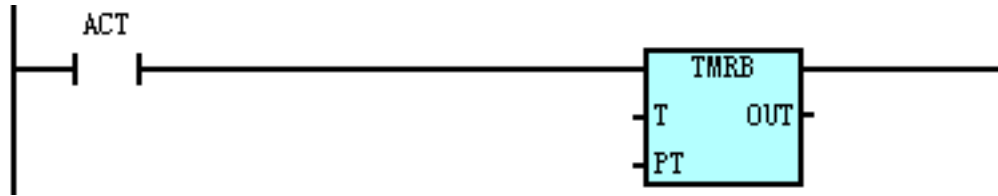
	R0300.2	R0300.1	R0300.0
R0100>R0200	0	0	1
R0100=R0200	0	1	0
R0100<R0200	1	0	0

5.4.4 TMRB (Timer)

• Command function

Delay break-over timer.

- Format of ladder diagram

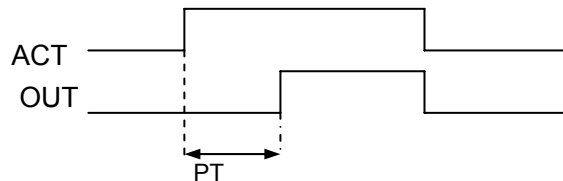


- Control conditions

ACT =0: T and OUT reset.

=1: T starts timing from 0, when reach the preset time, OUT=1.

The logic relation is as below:



- Relative parameter:

T : Number of timer, the digit range is T0000~T0099.

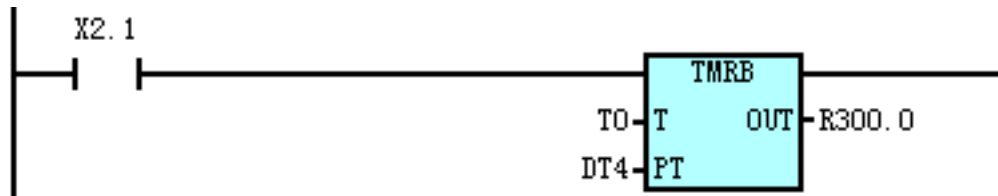
PT : Time constant or the data register beginning with DT, the setting range of DT: 0~21,474,3647(ms).

OUT: Timer output address, it can be R, Y, G, K or A, etc.

Remark:

Timer T is executed for one time in 8ms and 8ms is the unit for timing.

- Program example:



Remark: When X0002.1 is 0, T0002 and R0300.0 all are 0;

When X0002.1 is 1, T0002 begins timing; after reach the time set by DT0004, R0300.0 is set as 1.

5.4.5 CTRC (Counter in binary system)

- Command function

The data of counter uses binary system, and the following function is used based on the detailed situations.

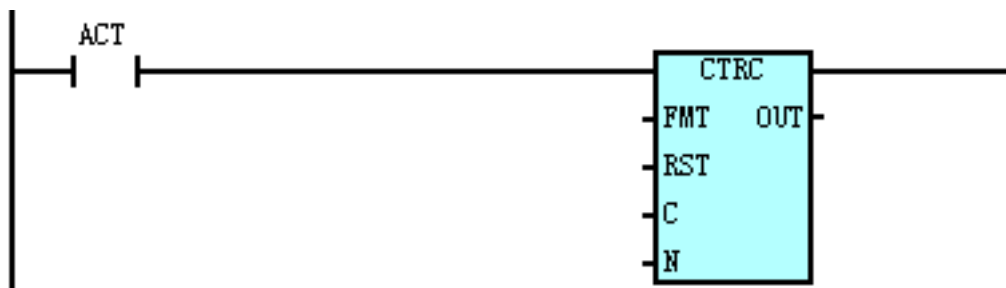
A: The counter in preset type: Preset the count value, if the value reaches the preset one, output the corresponding signal.

B: Counter in loop: When the counter reaches the preset value, input the counting signal and reset the initialized value, and count again.

C: Counter of addition and subtraction: Reversible counter can add and subtract.

D: Selection of initial value: The initial value can be 0 or 1.

- Format of ladder diagram



● Control conditions

ACT is rising edge check:

Adding the value: From the initial value, C begins adding the value, and the rising edge check for one time, C adds the counting value for one time; when reach the preset value (N), OUT=1. When C is less than N, OUT=0; if the rising edge check again, C initializes the value and begins calculating, at the same time, OUT=0.

Subtracting the value: From the set value (N), C begins subtracting the value, and the rising edge check for one time, C subtracts the counting value for one time; after reach the preset value, OUT=1. And when C is greater than N, OUT=0; if the rising edge check again, C initializes the value and begins calculating, OUT=0.

When ACT=0, C and OUT remains the original value.

● Relative parameter:

FMT: Data format:

0	0	CN0	U/D
---	---	-----	-----

Specify adding/subtracting values

0: Adding values, the counter starts from CN0

1: Subtracting values, the counter starts from the preset value

Specify initial value of counter

0: Count from 0

1: Count from 1

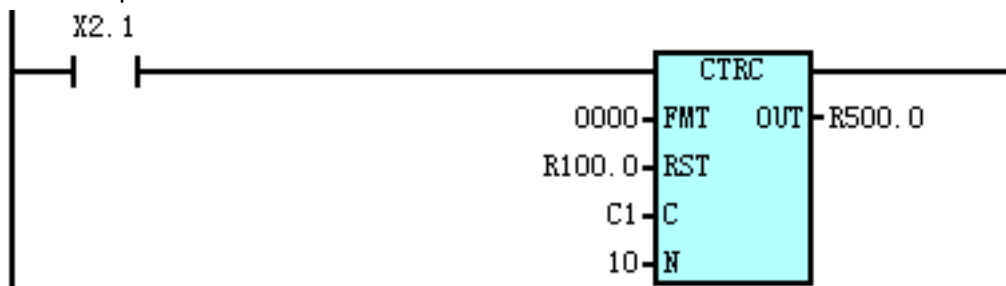
RST: It is 1, no matter whatever state ACT is, C=CN0, OUT =0. RST can be: X, Y, G, F, R, K or A, etc.

C : Specify number of the counter, represented by Cxxx, xxx is the digits (0~99).

N: Preset value of the counter can be constant, or the data register beginning with DC. If it is the constant, its range is 0~21,4748,3647.

OUT: The output position is 1 when reach the count value. OUT can be R, Y, G, K or A, etc.

● Program example



Remark: When R0100.0 is 1, C0001=0, R0500.0=0;

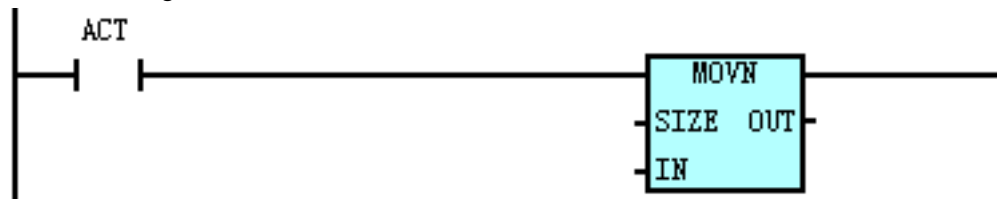
When R0100.0 is 0, X0002.1 rising edge check for one time, C0001 adds the value for one time; till 10, R0500.0 is set as 1. X0002.1 rising edge check again, C0001 initializes as 0 and begins counting, R0500.0 is reset as 0.

5.4.6 MOVN (Sending data in binary system)

- Command function

Send the data of source address or specified data in binary system to the target address.

- Format of ladder diagram



- Control conditions:

ACT =0: OUT remains the original value.

=1: Copy value of IN or constant to OUT.

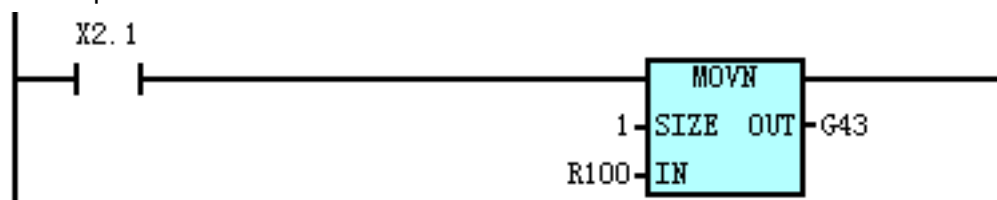
- Relative parameter

SIZE: Size of copied data (1, 2 or 4 bytes)

IN : The address starting character of source data address or constant, the address number is R, X, Y, F, G, K, A, D, T, C, DC and DT, etc.

OUT: The target address starting byte, the address number is R, Y, G, K, A, D, T, C, DC and DT, etc.

- Program example:



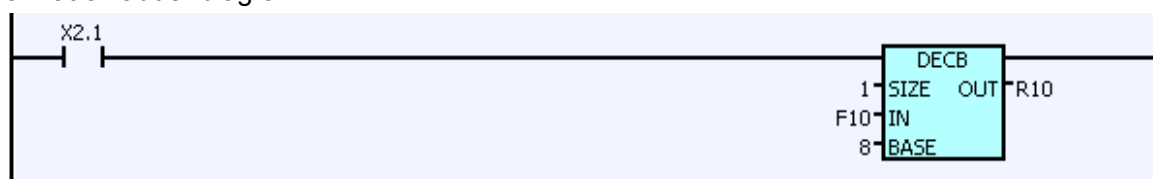
Remark: When X0002.1 is 1, send value of R0100 to G0043.

5.4.7 DECB (Decipher in binary system)

- Command function

DECB can decode the code data in binary system, and one of the specified eight continuous data is same as the code data, the corresponding output data is 1; if it's different, the output data is 0. The command is for data decipher of M or T function.

- Format of ladder diagram



- Control conditions

ACT =0: Eight data digits of OUT all are reset.

=1: The content value of decipher address (IN) is compared with the eight continuous data

beginning with BASE. If the content value of IN equals with anyone of eight data, and then the corresponding digit of the output address (OUT) is set as 1 based on the equaled data in which digit of the eight data.

- Relative parameter

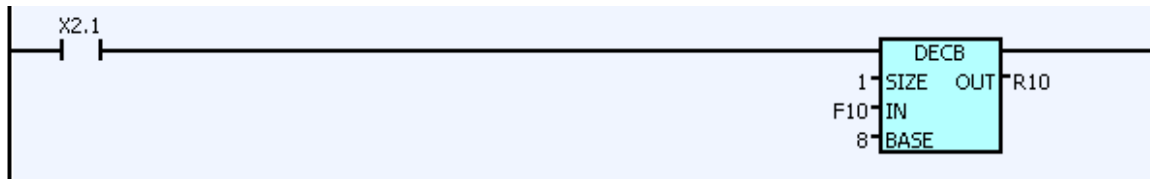
SIZE: Specify the size of IN1 address (1, 2 or 4 bytes).

IN: The starting address of decipher, the address number is R, X, Y, F, G, K, A, D, T, C, DC and DT, etc.

BASE: Compare the basic value of constant.

OUT: Output the comparing result, the address number is R, Y, G, K and A, etc.

- Program example



When X0002.1=1;

F0010=8, R0010.0=1;

F0010=9, R0010.1=1;

.....

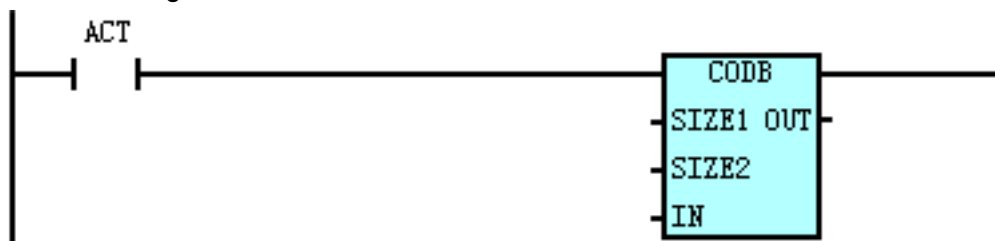
F0010=15, R0010.7=1

5.4.8 CODB (Codes conversion in binary system)

- Command function

The command is for converting the data in binary system.

- Format of ladder diagram

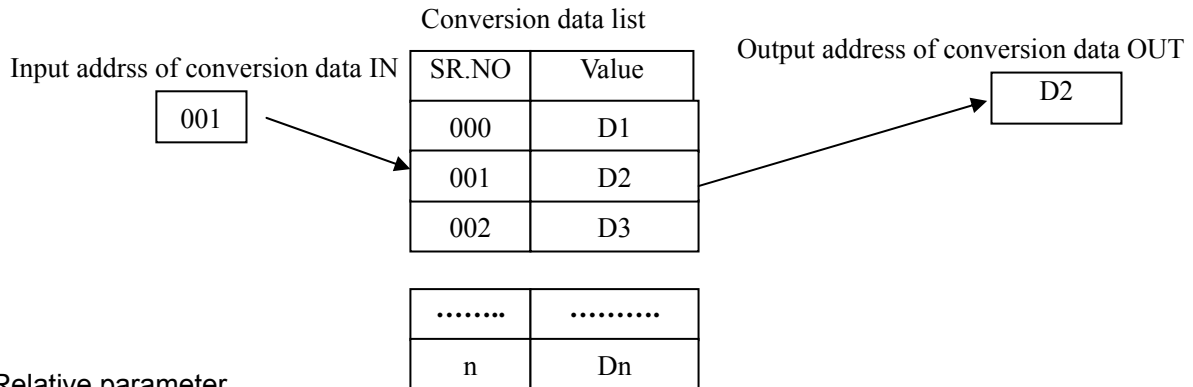


- Control conditions

ACT =0: The values in OUT remain unchanged.

=1: The values of “converting input data address (IN)”

are taken as the list number of conversion list, take the corresponding conversion data which corresponds the list number from the conversion list, and output to the output address (OUT) of the conversion data.



- Relative parameter

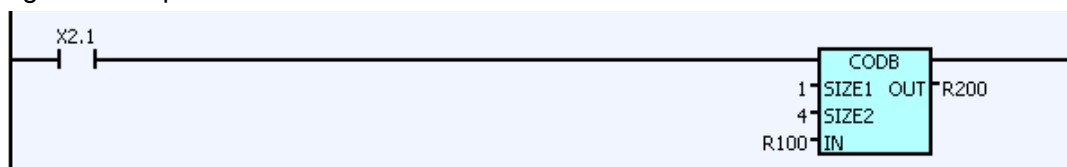
SIZE1: The data size in binary system and output address size of conversion data in its list, they are: 1-1 byte, 2-2 byte, 4-4 byte.

SIZE: Size of conversion list, the size and conversion data correspond with each other.

IN: Input address of the conversion data, the address only requires data of one byte. The address can be R, X, Y, G, F, A, K or D, etc.

OUT: The output address of conversion data, the address can be R, X, Y, G, F, K, A, D, DT and DC, etc.

Program example



For example

When X0002.1=1

X0002.1=1, R0100=0: R0200=1

X0002.1=1, R0100=1: R0200=2

X0002.1=1, R0100=2: R0200=3

X0002.1=1, R0100=3: R0200=4

Data conversion list

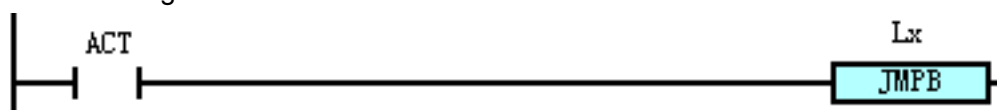
SR.NO	Value
000	1
001	2
002	3
003	4

5.4.9 JMPB (Label number jumping)

- Command function

The program can be transferred immediately to the program position set by the label number, which has the following characteristics: Many jumping commands can use same label number; it's not allowed to jump out of the subprogram, but it can jump forward or backward.

- Format of ladder diagram



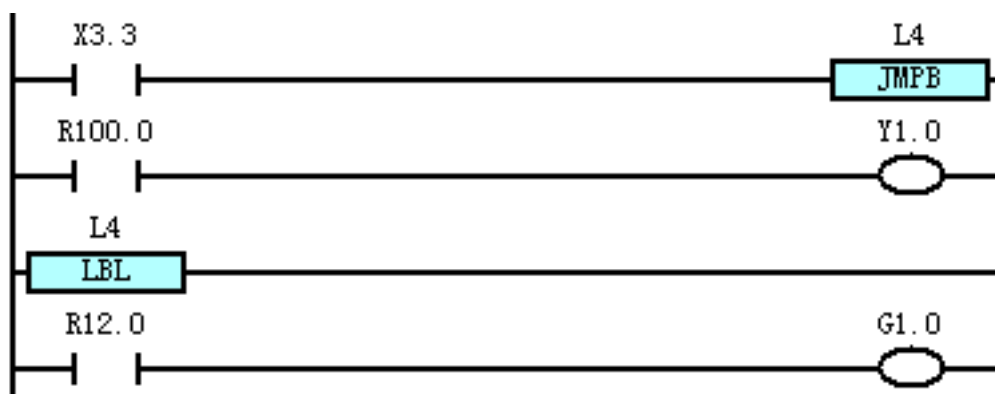
- Control conditions

ACT =0: Not jump, execute the next command behind JMPB.

= 1: Jump to the specified label number; execute the next command behind label number.

- Relative parameter

Lx: Specify the target label number of jumping, the label number must begin with L address, and specify one value from L1 to L9999.



- Program

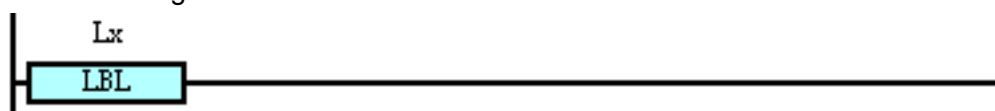
Remark: When X0003.3 is 1, the program skips line 0003, and executes from line 0004 in sequence; if X0003.3 is 0, execute from line 0003 in order.

5.4.10 LBL (Label number)

- Command function

Specify one label number in ladder diagram, which is the target position to jump, specified by JPB, one Lx label number can be specified by LBL once, otherwise, invalid.

- Format of ladder diagram



- Command parameter

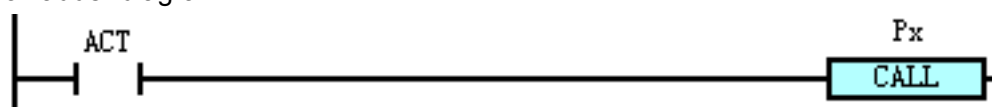
Lx: Specify target label number to jump, the label number must begin with specified L address, and specify one value from L1 to L9999.

5.4.11 CALL (Calling subprograms)

- Command function

Specified subprograms are called and there are the following characteristics: Many calling commands can call same subprogram; the calling commands can be nested.

- Format of ladder diagram



- Control conditions

ACT =0: Execute the next command after CALL.

=1: The subprog of specified subprogram number is called.

- Relative parameter

Px: Specify the label number of the subprogram, the subprogram label number must begin with

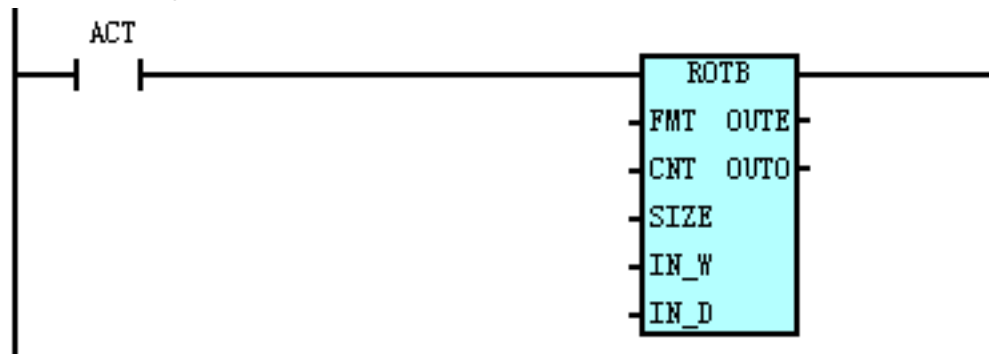
specified P address, and specify one value from P1 to P9999.

5.4.12 ROTB (Rotation control in binary system)

- Command function

It is for rotation control, such as the tool post and the rotary table, etc. The command has the following functions: select the rotation direction of the short path; calculate the number of steps from the current position to the target position, or calculate the number of steps from the previous position of the current to the previous one of the target; calculate the position number of the previous position of the target.

- Format of ladder diagram



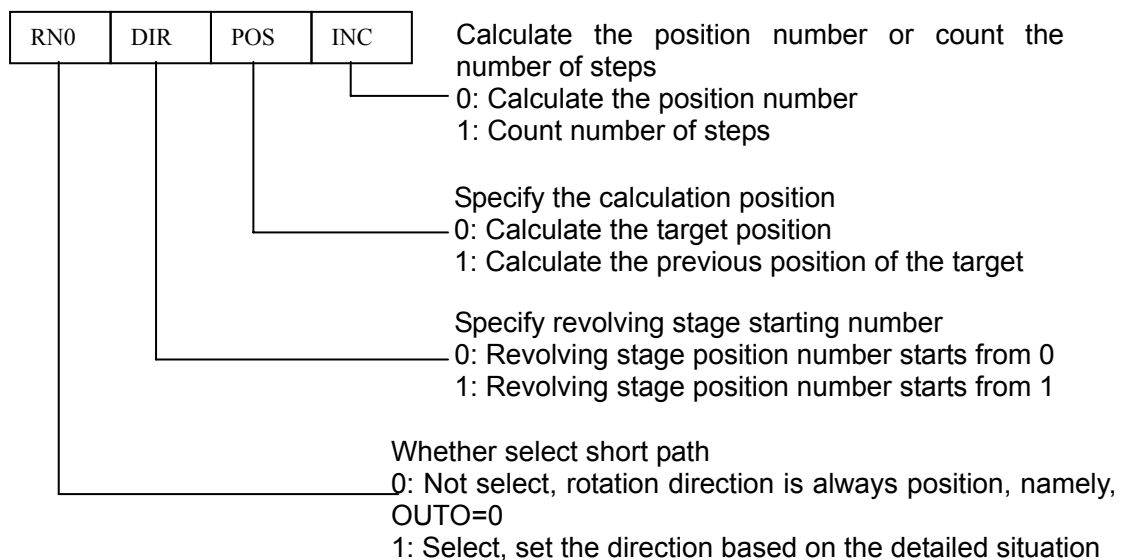
- Control conditions

ACT =0: The command isn't executed, and OUTE and OUTO remain original value.

=1: The command is executed, and output the result between OUTE and OUTO.

- Relative parameter

FMT: Data format:



CNT: Position number of revolving stage graduation

SIZE: Specify address sizes of IN_W , IN_D and OUTE. (1, 2 or 4 bytes)

IN_W: The current position address, the current position number is saved. The address numbers are R, X, Y, F, G, K, A, D, DC and DT, etc.

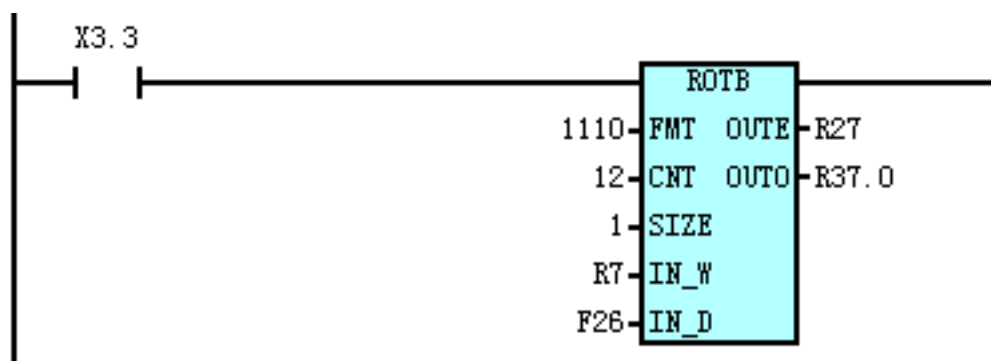
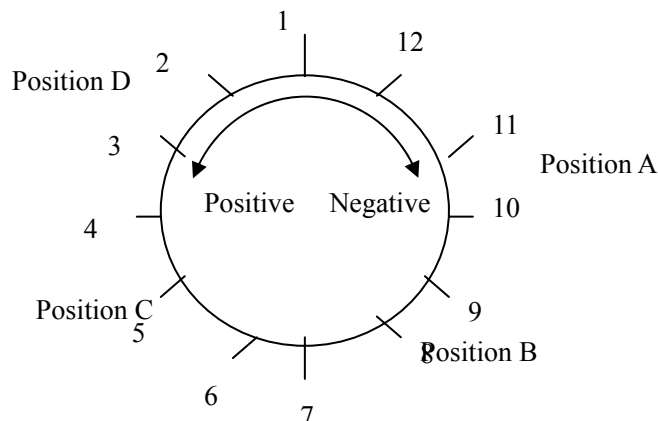
IN_D: Target position address, target position number is saved. The address numbers are R, X, Y, F, G, K, A, D, DC and DT, etc.

OUTE: Calculation result output address. The address number are R, Y, G, K, A, D, DC and DT, etc.

OUTO: Output in rotation direction, the direction which can increase the position number of revolving stage is positive (FOR); otherwise, negative (REV). OUTO=0, it rotates in positive direction; OUTO=1, negative. The address numbers are R, Y, G, K or A, etc.

• Program example

For example: There is one revolving stage tool post is as below, the current position is in #1 tool position:



When rotate the short path, calculate the position number of the previous position of the target. When the current position number is R0007=1, the position number of revolving state graduation CNT=12; And when X0003.3=1:

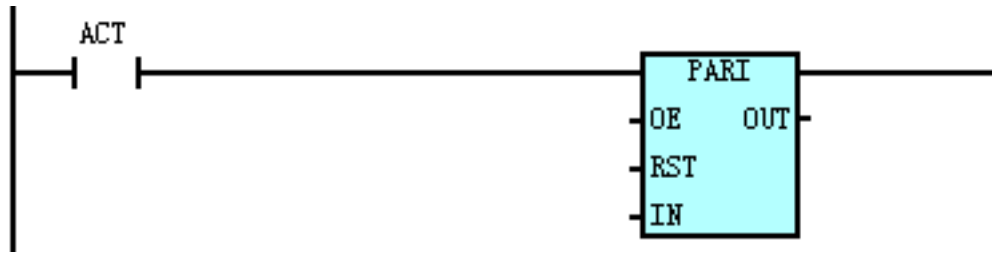
F0026=10 target position is A, R0027=11, R0037.0=1 the target position is A,
F0026=8 target position is B, R0027=9, R0037.0=1 the target position is B
F0026=5 target position is C, R0027=4, R0037.0=0 the target position is C,
F0026=3 target position is D, R0027=2, R0037.0=0 the target position is D

5.4.13 PARI (Odd-even check)

• Command function

Odd-even check the input data, the input data is one byte (8 digits).

• Format of ladder diagram



- Control conditions

ACT=1: Odd-even check the input data, if the input data can't comply with it specified by OE, OUT is 1; otherwise, OUT is 0.

ACT=0: The command isn't executed, OUT doesn't remain the original value.

- Relative parameter

OE =0: Number of "1" in input data is even.

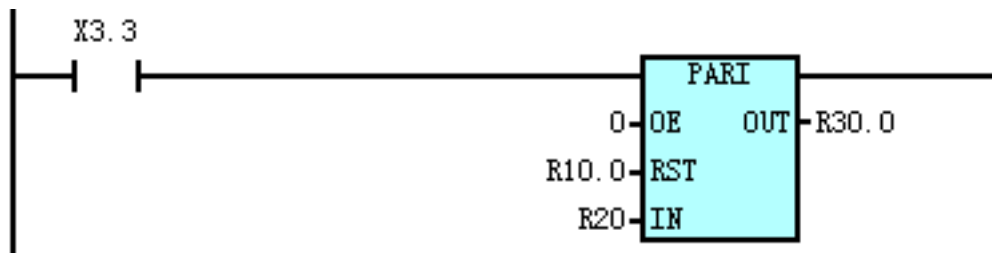
=1: Number of "1" in input data is odd.

RST: If it is 1, OUT is reset as 0, the address can be X, Y, G, R, F, A or K, etc.

IN: Input data address, the address can be X, Y, G, R, F, A, K or D, etc.

OUT: Check result output address, the address can be Y, G, R, A or K, etc.

- Program example



Remark: When X0003.3 is 1, PARI command is executed; OE=0000, odd check. When R0010.0 is 1, R0030.0 is reset as 0, without checking. When R0010.0 is 0, check; when numerical value of R0020 includes the even number of 1, R0030.0 is 0; when numerical value of R0020 includes the odd number of 1, R0030.0 is 1.

5.4.14 ADDB (Data addition in binary system)

- Command function

Data addition in binary system

- Format of ladder diagram



- Control conditions

ACT=1: Execute $OUT=IN1+IN2$. If the calculation is wrong, ERR is 1; otherwise, ERR is 0.

ACT=0: The command isn't executed, and OUT and ERR remain unchanged.

- Relative parameter

SIZE: 1-1 byte, 2-2 byte, 4-4 byte.

IN1: The summand can be constant or address. The address number can be R, X, Y, F, G, A, K, D, T, C, DC or DT, etc.

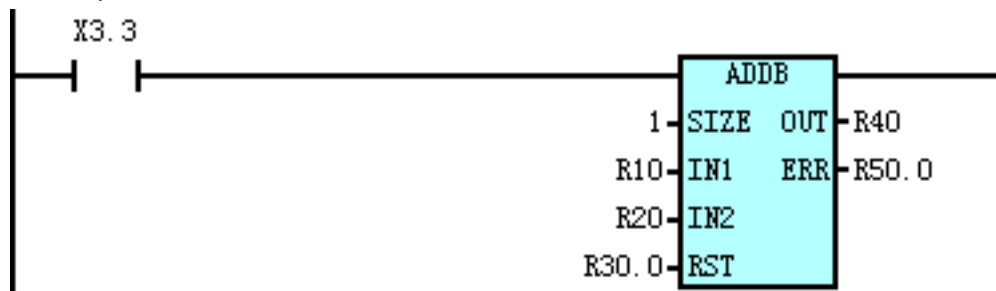
IN2: The addend can be constant or address. The address number can be R, X, Y, F, G, A, K, D, T, C, DC or DT, etc.

RST: When it is 1, ERR is reset as zero, OUT remains unchanged. The address number can be R, X, Y, F, G, A or K, etc.

OUT: Running result output data address. The address can be Y, G, R, A, K, DC, DT, D, C or T, etc.

ERR: The calculation result wrongly outputs the address, the address can be Y, G, R, A or K, etc.

• Program example



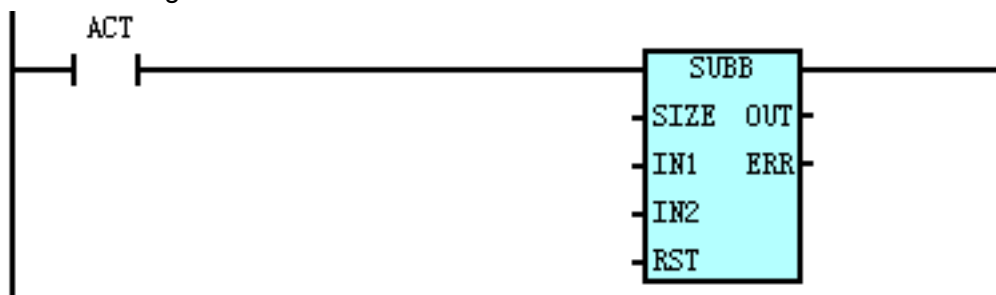
Remark: X0003.3=1, ADDB command is executed. R0040=R0010+R0020, if the calculation is wrong, R0050.0 is 1, otherwise, it is 0. When R0030.0 is 1, R0040 remains unchanged, R0050.0 is reset as 0.

5.4.15 SUBB (Data subtraction in binary system)

• Command function

Data subtraction in binary system

• Format of ladder diagram



• Control conditions

When ACT=1: OUT= IN1-IN2 is executed. If the calculation is wrong, ERR is 1; otherwise, it is 0.

ACT=0: The command isn't executed, and OUT and ERR remain unchanged.

• Relative parameter

SIZE: 1, 2 and 4 is respectively relative to 1 byte, 2 bytes or 4 bytes.

IN1: The minuend can be constant or address. The address number can be R, X, Y, F, G, A, K, D, T, C, DC or DT, etc.

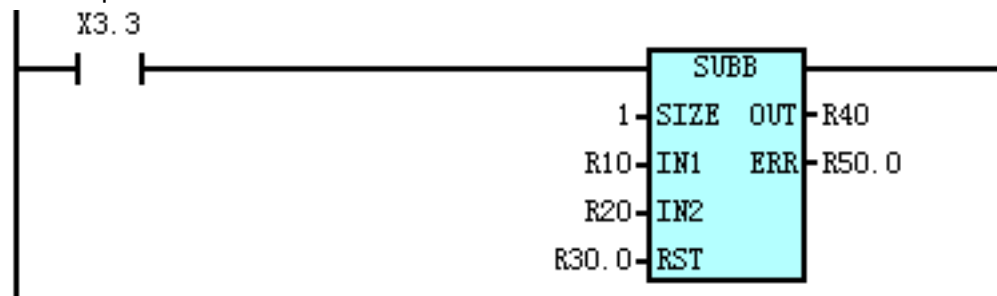
IN2: The subtrahend can be constant or address. The address number can be R, X, Y, F, G, A, K, D, T, C, DC or DT, etc.

RST: When it is 1, ERR is reset. The address number are be R, X, Y, F, G, A or K, etc.

OUT: Running result output data address, the address can be Y, G, R, A, K, DC, DT, D, C or T, etc.

ERR: Calculation result wrongly outputs address, the address can be Y, G, R, A or K, etc.

- Program example



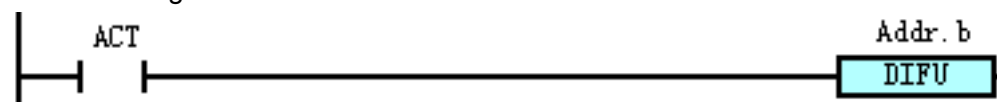
Remark: X0003.3=1, SUBB command is executed, R0040=R0010-R0020; if the calculation is wrong, R0050.0 is 1; otherwise, R0050.0 is 0; when R0030.0 is 1, R0040 remains unchanged, R0050.0 is reset as 0.

5.4.16 DIFU (Setting rising edge check)

- Command function

In the scan cycle of the input signal of the rising edge check, the output signal is set as 1.

- Format of ladder diagram



- Control conditions

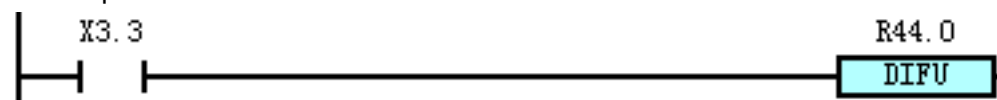
Input signal ACT: At the rising edge check ($0 \rightarrow 1$) of ACT, the output signal is set as 1.

Output signal Addr.b: When the function is executed, Addr.b remains 1 in one scan cycle of ladder diagram.

- Relative parameter

Addr.b: The calculation result output address, the address can be Y, G, R, A or K, etc.

- Program example



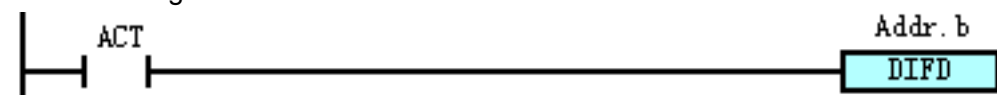
Remark: When X0003.3 is in the rising edge check, R0040.0 outputs 1.

5.4.17 DIFD (Setting falling edge check)

- Command function

The output signal is set as 1 when input the signal in the scan cycle of falling edge check.

- Format of ladder diagram



- Control conditions

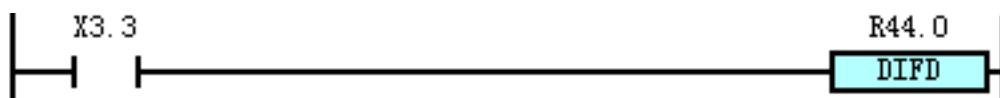
Input signal ACT: At the falling edge check ($1 \rightarrow 0$) of ACT, the output signal is set as 1.

Output signal Addr.b: When the function is executed, Addr.b remains 1 in one scan cycle of ladder diagram.

- Relative parameter

Addr.b: Calculation result output address, the address can be Y, G, R, A or K, etc.

- Program example



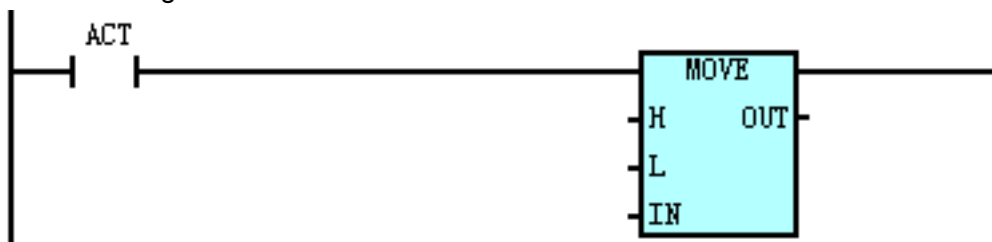
Remark: When X0003.3 is in the falling edge check, R0044.0 outputs 1.

5.4.18 MOVE (Logic multiplication)

- Command function

Logic AND the logic multiplier and input data, and output the result to the specified address.

- Format of ladder diagram



- Control conditions

When ACT=1: Logic AND the logic multiplier (H, L) and input data (IN), and output the result to the specified address (OUT). The digits which aren't required are excluded from one signal of 8 digits in the specified address.

ACT=0: OUT remains the original value.

- Relative parameter

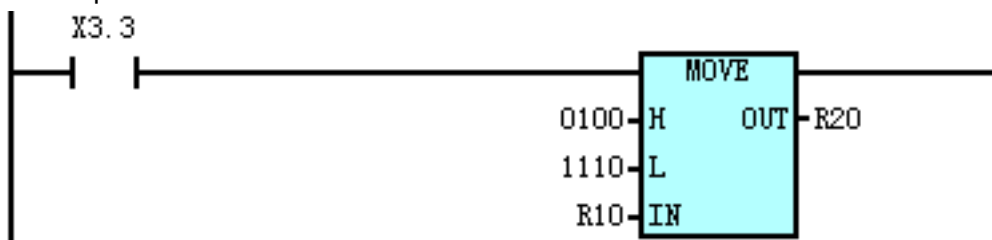
H: Logic multiplier of high four digits

L: Logic multiplier of low four digits

IN: Input data address, the address number can be R, A, K, X, Y, F, G or D, etc.

OUT: Output the data address, the address number can be R, A, K, Y, G or D, etc.

- Program example



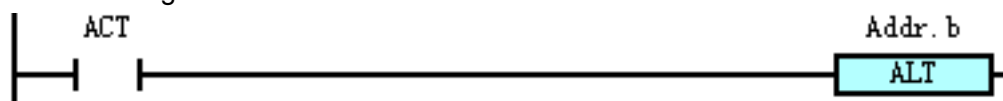
Remark: When X0003.3 is 1, R0010 AND 01001110, and the result is saved in R0020.

5.4.19 ALT (Alternate output)

- Command function

Alternate output the commands; when input signal changes in the rising edge check each time, the output signal CCW outputs.

- Format of ladder diagram



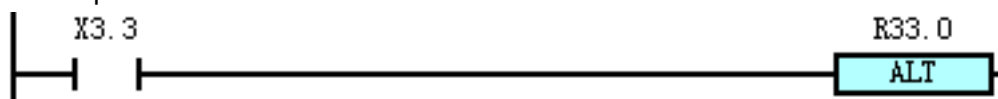
- Control conditions

When input signal ACT changes 0—>1 each time, output signal Addr.b CCW outputs.

- Relative parameter

Addr.b: The output signal, the address can be Y, G, R, A or K, etc.

- Program example



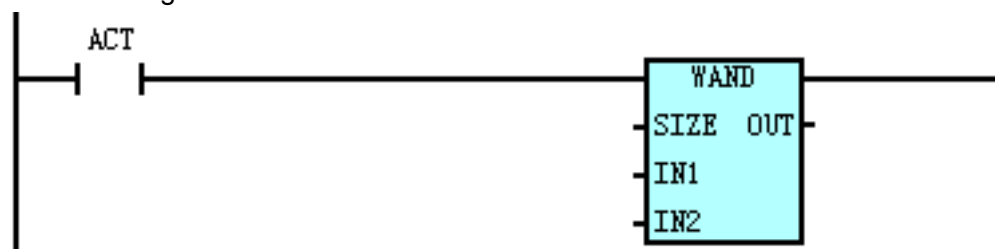
Remark: X0003.3 comes to the rising edge check each time, R0033.0 rotates for one time.

5.4.20 WAND (Bytes AND in binary system)

- Signal function

WAND AND the bytes of two input data (1, 2 and 4 bytes) based on the bits, output the calculation result to the output address.

- Format of ladder diagram



- Command format

WAND	SIZE	IN1	IN2	OUT
------	------	-----	-----	-----

- Control conditions

ACT=0: Value of OUT remains unchanged.

ACT=1: AND content values of IN1 and IN2, and output the result to OUT address.

- Relative parameter

SIZE: Specify the size of IN1 and IN2 (1, 2 or 4 bytes).

IN1, IN2: The starting address of input data or constant, the address number can be R, X, Y, F, G, K, A, D, T, C, DC or DT.

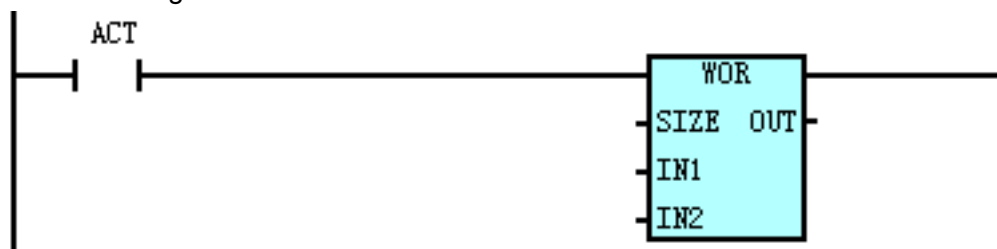
OUT: Result output address, the operation address can be R, Y, G, K, A, D, T, C, DC or DT.

5.4.21 WOR (Bytes OR in binary system)

- Command function

WOR OR the bytes of two input data (1, 2 or 4 bytes) based on the bits, output the calculation result to the output address.

- Format of ladder diagram



- Command format

WOR	SIZE	IN1	IN2	OUT
-----	------	-----	-----	-----

- Control conditions

ACT=0, value of OUT remains unchanged.

ACT=1, OR the content values of IN1 and IN2, output the result to OUT address.

- Relative parameter

SIZE: Specify the size of IN1 and IN2 (1, 2 or 4 bytes).

IN1, IN2: The starting address of input data or constant, the address can be R, X, Y, F, G, K, A, D, T, C, DC or DT.

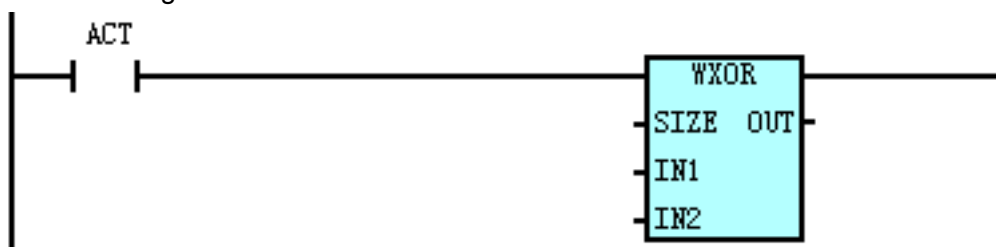
OUT : Result output address, the operation address can be R, Y, G, K, A, D, T, C, DC or DT.

5.4.22 WXOR (Bytes AND-OR in binary system)

- Command function

WXOR AND-OR the bytes of two input data (1, 2 or 4 bytes) based on the bits, output the calculation result to the output address.

- Format of ladder diagram



- Command format

WXOR	SIZE	IN1	IN2	OUT
------	------	-----	-----	-----

- Control conditions

ACT=0, value of OUT remains unchanged.

ACT=1, operate AND-OR in content values of IN1 and IN2, and output the result to OUT address.

- Relative parameter

SIZE: Specify the size of IN1 and IN2 addresses (1, 2 or 4 bytes)

IN1, IN2: Data input address starting bytes or constant. The address can be R, X, Y, F, G, K, A, D, T, C, DC or DT.

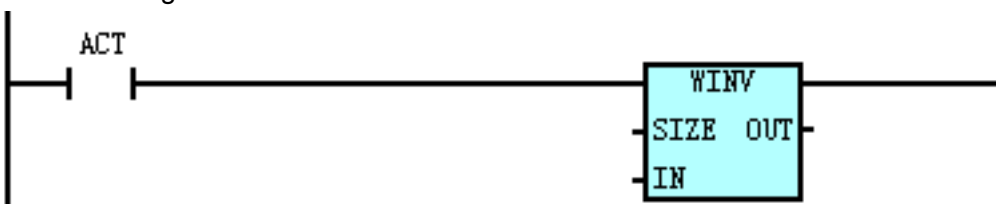
OUT: Result output address, the address can be R, Y, G, K, A, D, T, C, DC or DT.

5.4.23 WINV (Bytes negate in binary system)

- Function

Negate the data of input address or constant and save in the output address.

- Format of ladder diagram



- Command format

WINV	SIZE	IN	OUT
------	------	----	-----

- Control conditions

ACT=0, OUT remains the original value.

ACT=1, negate the value of IN and OUT address is saved.

- Relative parameter

SIZE: The size of the data (1, 2 or 4 bytes)

IN: The data input address starting byte or constant, the input address can be R, X, Y, F, G, K, A, D, T, C, DC or DT.

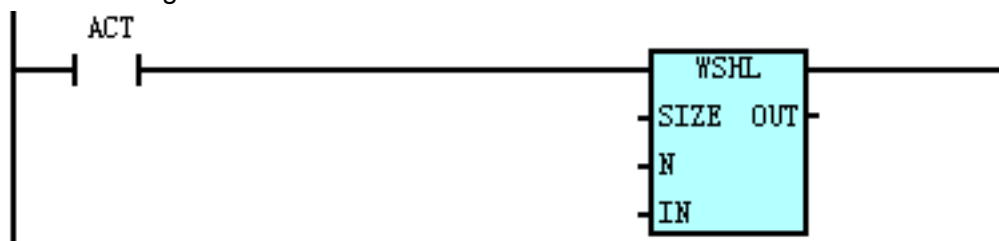
OUT: Output address starting bytes, the address can be R, Y, G, K, A, D, T, C, DC or DT.

5.4.24 WSHL (Data shift left in binary system)

- Command function

WSHL shifts the two input data (1, 2 or 4 bytes) left based on the specified bits, and output the result to the output address.

- Format of ladder diagram



- Format of command

WSHL	SIZE	N	IN	OUT
------	------	---	----	-----

- Control conditions

ACT=0, value of OUT remains unchanged.

ACT=1, shift the value of IN left for N bits and outputs the result to OUT address.

- Relative parameter

SIZE: Specify the data size of IN (1, 2 or 4 bytes)

N: The carry number address or the constant. The address can be R, X, Y, F, G, K, A, D, T, C, DC or DT;

IN: The data input address starting bytes or constant. The address can be R, X, Y, F, G, K, A, D, T, C, DC or DT.

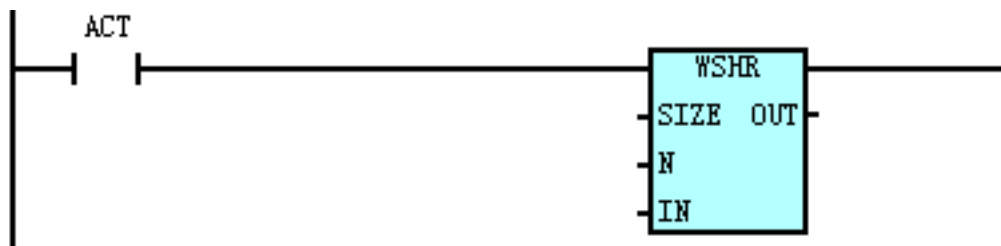
OUT: Result output address, the address can be R, Y, G, K, A, D, T, C, DC or DT.

5.4.25 WSHR (Data shift right in binary system)

- Command function

WSHR shifts the two input data (1, 2 or 4 bytes) right based on the specified bits, and output the result to the output address.

- Format of ladder diagram



• Format of command

WSHR	SIZE	N	IN	OUT
------	------	---	----	-----

• Control conditions

ACT=0, value of OUT remains unchanged.

ACT=1, shift value of IN right in N bits and output the result to OUT address.

• Relative parameter

SIZE: Specify data size of IN (1, 2 or 4 bytes)

N: The carry number address or the constant, the address can be R, X, Y, F, G, K, A, D, T, C, DC or DT;

IN: The data input address starting bytes or the constant. The address can be R, X, Y, F, G, K, A, D, T, C, DC or DT.

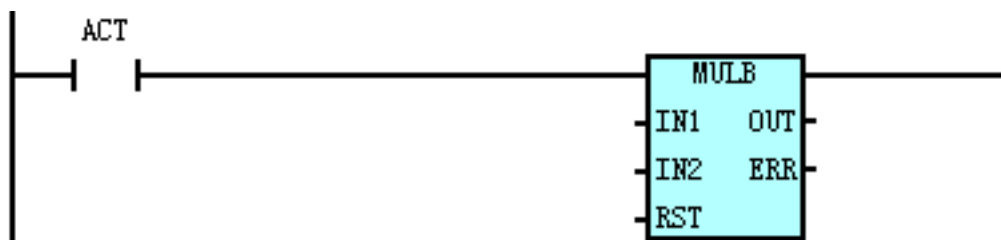
OUT: The result output address, the address can be R, Y, G, K, A, D, T, C, DC or DT.

5.4.26 MULB (Data multiplication in binary system)

• Command function

MUL multiplies the two input data (integer in 16 digits), and the product in 32 digits is obtained and save in the output address (32 digits).

• Format of ladder diagram



• Format of command

MUL	IN1	IN2	RST	OUT	ERR
-----	-----	-----	-----	-----	-----

• Control conditions

RST = 0: ERR and OUT remain unchanged.

RST = 1: ERR and OUT are reset.

ACT=0: Value of OUT remains unchanged.

ACT=1, value of IN1 and that of IN2 multiply, output the result to OUT address.

• Relative parameter

IN1, IN2: The multiplier input address starting bytes or the constant, the address can be R, X, Y,

F, G, K, A, D, T, C, DC or DT; If use the single byte address R, X, Y, F, G, K, A or D (8 digits), the command takes two continuous bytes as the multiplier; If use the double bytes address of T, C, DC or DT (32 digits), the command takes its low 16 digits as the multiplier.

OUT: The result output address, the address can be R, Y, G, K, A, D, T, C, DC or DT.

RST: Command resetting signal input address (bit address).

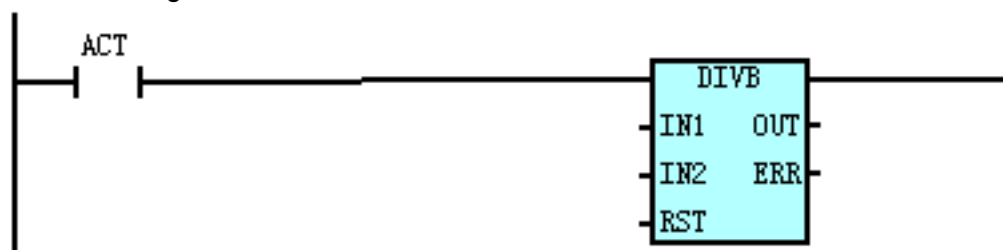
ERR: Calculation wrongly output address.

5.4.27 DIVB (Data division in binary system)

- Command function

DIV makes one input data (integer in 16 digits) is divided by the other, obtain the result in 32 digits, including the remainder in 16 digits (upper digit) and the quotient in 16 digits (lower digit) and save in output address (32 digits).

- Format of ladder diagram



- Command format

DIV	IN1	IN2	RST	OUT	ERR
-----	-----	-----	-----	-----	-----

- Control conditions

RST =0: ERR and OUT remains unchanged.

RST =1: ERR and OUT are reset.

ACT=0: Value of OUT remains unchanged.

ACT=1: IN1 is divided by IN2; output the result to OUT address.

- Relative parameter

IN1, IN2: The data input address starting bytes or the constant, the address can be R, X, Y, F, G, K, A, D, T, C, DC or DT. If use the single byte address of R, X, Y, F, G, K, A or D (8 digits), the command takes two continuous bytes as divisor. If use T, C, DC and DT, which are address of double bytes (32 digits), the command takes lower 16 digits as the divisor;

OUT: The result output address, the address can be R, Y, G, K, A, D, T, C, DC or DT.

RST: Command resetting signal input address (bit address).

ERR: The calculation wrong input address.

5.5 Control axis

5.5.1 Axial movement state

NC can send the current axial movement state to PLC, PLC moves based on the axial movement state.

5.5.1.1 Axial movement signal

MV1~MV5 (F102.0~F102.4)

- Signal type: NC->PLC
- Signal function: MV1, MV2, MV3, MV4 and MV5 are respectively relative to the movement signals of the 1st, 2nd, 3rd, 4th and 5th axes. When axis moves, the corresponding axial movement signal is set as 1; when the axis is still, the corresponding axial movement signal is 0. After PLC receives the axial movement signal sent by NC, it moves based on the state of signal.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F102				MV5	MV4	MV3	MV2	MV1

5.5.1.2 Axial movement direction signal

MVD1~MVD5 (F106.0~F106.4)

- Signal type: NC->PLC
- Signal function: MVD1, MVD2, MVD3, MVD4 and MVD5 are respectively relative to the movement direction signals of the 1st, 2nd, 3rd, 4th and 5th axes. When some axis moves in negative direction, the movement direction signal of corresponding axis is 1; when some axis moves in positive, it is 0; when the axis stops, the movement direction signal of corresponding axis is set as 1 or 0 according to the movement state before stop, PLC operates the next movement after PLC receives the axial movement direction signal.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F106				MVD5	MVD4	MVD3	MVD2	MVD1

5.5.2 Servo ready signal

SA (F0.6)

- Signal type: NC->PLC
- Signal function: When NC receives the alarm signal of the servo system, it alarms and sets SA signal as 0, and it informs PLC that it doesn't ready and the axis can't be moved; when the alarm cancels, SA is set as 1 and the axis can be moved.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F0		SA						

5.6 Running ready

5.6.1 Emergency stop

ESP (G8.4) : Emergency stop signal

✎ **Signal type:** PLC->NC, it is valid when it is 0.

- Signal function: When G8.4 is level 0, it will alarm emergency stop when NC detects the signal.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G8				ESP				

5.6.2 CNC ready signal

MA (F1.7):

- Signal type: NC->PLC
- Signal function: CNC ready signal means CNC has been ready.
- Output condition: After CNC powers on and gets ready, the signal is set as 1, which is set as 1 in several seconds after power on. If CNC alarms or executes emergency stop, the signal switches into 0.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F1	MA							

5.6.3 Alarm signal

AL (F1.0):

- Signal type: NC->PLC
- Signal function: When CNC alarms, the alarm is displayed on the screen, and AL is set as 1, after PLC receives the signal, control based on the state of alarm signal; three alarms display: servo alarm, P/S alarm and overtravel alarm. When CNC resets, clear alarm, AL is 0.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F1								AL

5.6.4 Mode selection

Mode selection signals include MD1, MD2, MD4, DNC1 and ZRN; the seven modes: edit, auto, MDI, manual, MPG/single step, mechanical zero return and program zero return. CNC detects signal through mode and informs PLC system about the current mode.

5.6.4.1 Selecting the signal in different modes

MD1、MD2、MD4(G43.0~G43.2) DNC1(G43.5) ZRN (G43.7):

- Signal type: PLC->NC
- Signal function: the codes of mode are listed as below:

SR.NO	Code signal Mode	ZRN	DNC1	MD4	MD2	MD1
1	EDIT	0	0	0	1	1
2	AUTO	0	0	0	0	1
3	MDI	0	0	0	0	0
4	MPG/SINGLE STEP	0	0	1	0	0
5	JOG	0	0	1	0	1
9	MECHANICAL ZERO RETURN	1	0	1	0	1

After PLC receives the input signal of mode, assign the values to the coder signal, and then send to NC, NC confirms the mode of CNC based on the code signals.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G43	ZRN		DNCI			MD4	MD2	MD1

5.6.4.2 Detecting the signal in different modes

MINC(F3.0)、MH(F3.1)、MJ(F3.2)、MMDI(F3.3)、MRMT (F3.4)、MMEM(F3.5)、MEDT(F3.6)、MREF(F4.5)

- Signal type:NC->PLC
- Signal function: When CNC is in some mode, the corresponding F signal is set as 1, and then send F signal to PLC, PLC operates the next movement according to the mode.

Detecting the signal in single step	MINC
Detecting the signal in MPG	MH
Detecting the signal in JOG	MJ
Detecting the signal in MDI	MMDI
Detecting the signal in DNC	MRMT
Detecting the signal in AUTO	MMEM
Detecting the signal in EDIT	MEDT
Detecting the signal in mechanical zero return	MREF

- Signal function:

	#7	#6	#5	#4	#3	#2	#1	#0
F3		MEDT	MMEM	MRMT	MMDI	MJ	MH	MINC
F4			MREF					

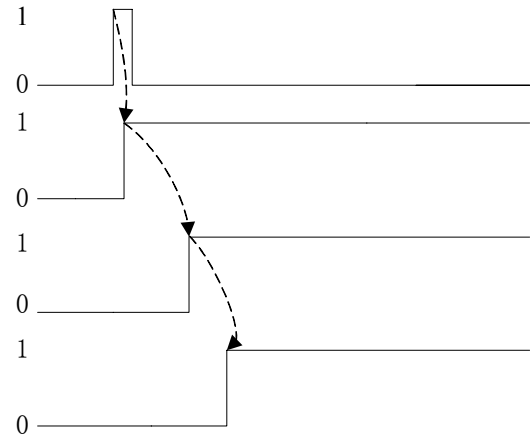
5.6.4.3 The signal time sequence of mode

Key input mode (X 2 0 . 0 , X 2 0 . 1 ,
X 2 0 . 2 , X 2 0 . 3 , X 2 0 . 4 , X 2 0 . 5 , X 2 1 . 3)

Selecting signal
mode G 4 3

Detecting signal mode (F 3 . 0 , F 3 . 1 ,
F 3 . 2 , F 3 . 3 , F 3 . 5 , F 3 . 6 , F 4 . 5 , F 4 . 6)

Indicator lamp of mode (Y 5 . 3 ,
Y 5 . 2 , Y 5 . 5 , Y 5 . 6 , Y 5 . 7 , Y 5 . 4 , Y 6 . 2)



5.6.5 State output

5.6.5.1 Rapid feeding signal

RPDO (F2.1):

- Signal type: NC->PLC
- Signal function: When CNC is in manual rapid feeding mode, the axial movement is executed, and RPDO is set as 1.
- Points for attention: When axis feeds rapidly, RPD0 is set as 1, after feeding stops, state of PRDO remains unchanged and it is still 1. If non-rapid feeding mode is selected, RPDO signal is reset as 0 after the axis moves.
- Singal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F2							PRDO	

5.6.6 Overtravel detection

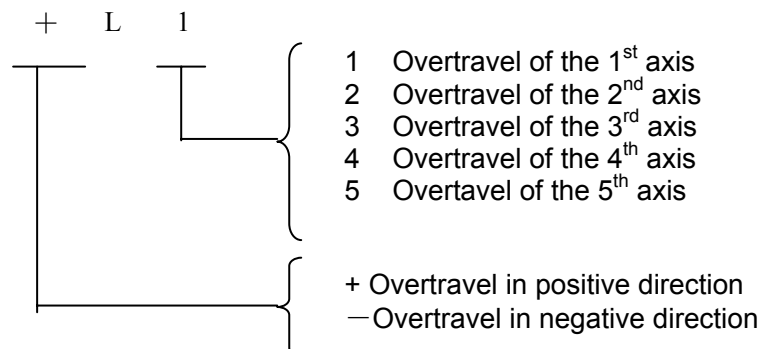
5.6.6.1 Overtravel signal

+L1 ~ +L5 (G114#0~G114#4)

-L1 ~ -L5 (G116#0~G116#4)

•Signal type: PLC->NC

- Signal function: It means control axis has already reached the travel limit; each direction of each control axis all has the signal. The codes of “+” and “-” signal name mean direction, the bit corresponds to the control axis.



[Movement] When the above signal is “0”, the control unit movement is as below:

- * When it auto operates, even only one axial overtravel signal changes into 0, all axes decelerate and stop, it alarms and running cuts off.
- * During manual operation, the axis movement signal is 0 and it decelerates and stops, and then moves in the opposite direction.
- * Once the axial overtravel signal changes into 0, its movement direction is saved. Even the signal changes into 1, before the alarm is cleared, the axis can't move along the direction.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G114				+L5	+L4	+L3	+L2	+L1
G116				-L5	-L4	-L3	-L2	-L1

5.6.6.1 Memory travel check 1

Memory travel check selective signal

EXLM (G7.6)

- Signal type: PLC->NC
- Signal function: When the signal is 1, use parameter #1326 and #1327 to check travel 1; when the signal is 0, use parameter #1320 and #1321 to check travel 1.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G7		EXLM						

5.6.6.2 Memory travel check 2, 3

Travel check 3 releasing signal

RLSOT3 (G7.4)

- Signal type: PLC->NC
- Signal function: Selected whether execute memory travel check 3. When the signal is 1, memory travel 3 isn't checked; when the signal is 0, parameter #1300.5 and #1310.1 are 1, check memory travel 3.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G7				RLSOT3				

5.7 Manual operation

5.7.1 Manual feeding/increment feeding

Manual feeding: In manual mode, set the feeding axis and direction selecting signal as 1 on the machine operation panel, and then the machine continuously moves along the selected axis in the selected direction.

Increment feeding: In single step mode, the feeding axis and direction selecting signal are set as 1 on the machine operation panel. And then the machine moves by one step along the selected axis in the selected direction, the machine movement minimum distance is the minimum input increment;

each step has three override values: 0.001, 0.010 and 0.100.

The only difference between the manual feeding and the increment feeding is to select the feeding distance. During manual feeding, when feeding axis of +J1, -J1, +J2, -J2, +J3 or -J3 and the direction selecting signal are 1, the machine can continuous feed. In the increment feeding, the machine is the single step feeding.

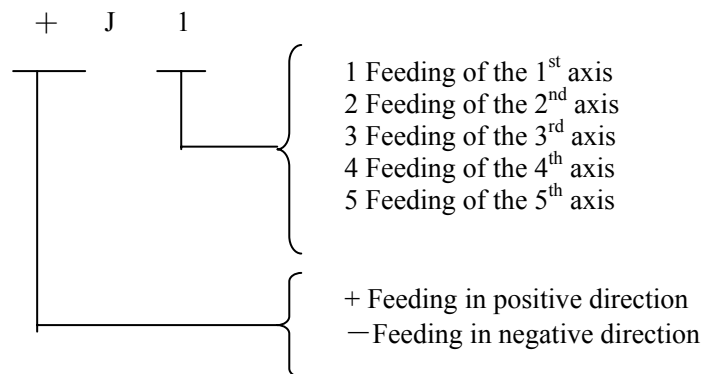
Through manual rapid feeding selecting switch, the machine can move on the rapid feedrate. The single step distance of increment feeding can be selected through MP1 and MP2.

5.7.1.1 Feeding axis and direction selecting signal

+J1~+J5 (G100.0~G100.4)

-J1~-J5 (G102.0~G102.4)

- Signal type: PLC →NC
- Signal function: During manual feeding or increment feeding, select the required feeding axis and direction, axial movement is executed, and then the corresponding axes and direction selecting signal is set as 1, PLC controls the next movement after PLC receives the signal. “+” and “-” mean the feeding direction, and the bits correspond to the control axes.



- Points for attention: A: During manual feeding, CNC makes the corresponding axes continuous move; during increment feeding, CNC makes the corresponding axes feed based on the step distance defined by MP1 or MP2.
B: When the axis moves, the corresponding axis and direction selecting signal is set as 1; when the movement stops, the signal is reset.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G100				+J5	+J4	+J3	+J2	+J1
G102				-J5	-J4	-J3	-J2	-J1

5.7.1.2 Manual feeding override signal

JV00~JV15 (G10、G11) :

- Signal type: PLC →NC
- Signal function: Select manual movement speed, the corresponding relation between signal and manual feedrate is shown as the following list. When PLC receives the external override input signal, assigns the values to G10 and G11, and then sends to NC, finally, CNC displays the corresponding movement velocity.

G11	G10	Feeding override (%)
0000 0000	0000 1111	0
0000 0000	0000 1110	10
0000 0000	0000 1101	20
0000 0000	0000 1100	30
0000 0000	0000 1011	40
0000 0000	0000 1010	50
0000 0000	0000 1001	60
0000 0000	0000 1000	70
0000 0000	0000 0111	80
0000 0000	0000 0110	90
0000 0000	0000 0101	100
0000 0000	0000 0100	110
0000 0000	0000 0011	120
0000 0000	0000 0010	130
0000 0000	0000 0001	140
0000 0000	0000 0000	150

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G10	JV7	JV6	JV5	JV4	JV3	JV2	JV1	JV0
G11	JV15	JV14	JV13	JV12	JV11	JV10	JV9	JV8

5.7.1.3 Manual rapid feeding selecting signal

RT (G19.7)

- Signal type: PLC → NC
- Signal function: Select manual rapid feedrate. After PLC receives the manual rapid feeding input signal, RT is set as 1 and then sends to NC.
During manual rapid feeding, RT switches from 1 to 0, or from 0 to 1, the feedrate decelerates into 0 and then accelerates into the set value. During acceleration and deceleration, the state of the feeding axis and the direction selecting signal remain unchanged.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G19	RT							

5.7.2 MPG feeding

In MPG feeding mode, rotation MPG makes the machine move a little, and select the movement axis through MPG feeding axis selecting signal.

5.7.2.1 MPG feeding axis selecting signal

HS1A~HS1D (G18.0~G18.3), HS2A~HS2D (G18.4~G18.7)

- Signal type: PLC → NC
- Signal function: MPG feeding axis selecting signal, after PLC receives MPG feeding axis input

signal and assigns the values to HSnA~HSnD, NC selects the corresponding axes to feed based on the signal states of HSnA~HSnD. The corresponding relations of these signals and MPG feeding axes are shown as the following list:

HSnD	HSnC	HSnB	HSnA	Feeding axis
0	0	0	0	Without feeding axis
0	0	0	1	Axis 1 feeding
0	0	1	0	Axis 2 feeding
0	0	1	1	Axis 3 feeding
0	1	0	0	Axis 4 feeding
0	1	0	1	Axis 5 feeding

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G18	HS2D	HS2C	HS2B	HS2A	HS1D	HS1C	HS1B	HS1A

5.7.2.2 MPG override signal

MP1(G19.4)、MP2(G19.5):

- Signal type: PLC →NC
- Signal function: When MP1 or MP2 selects MPG feeding, the movement distance of MPG each pulse or that of increment feeding each step can be selected. After PLC receives MPG/increment feeding input signal, assigns the values to MP1 or MP2, and then sends to NC, and selects the corresponding feeding amount. The corresponding relations between MP1 or MP2 and the movement amount are as below:

MP2	MP1	Movement amount (mm)
0	0	0.001
0	1	0.010
1	0	0.100
1	1	1.000

Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G19			MP2	MP1				

5.8 Mechanical zero-return

5.8.1 Mechanical zero-return

In mechanical zero-return mode, set the feeding axis and direction selecting signal as 1, the machine moves along the set direction and returns to the machanical zero. After mechanical zero-return, CNC sets the work piece coordinate system based on the value set by data parameter 1240~1243.

5.8.1.1 Mechanical zero-return finish signal

ZP1~ZP5(F94.0~F94.4)

ZP21~ZP25(F96.0~F96.4)

ZP31~ZP35(F98.0~F98.4)

ZP41~ZP45(F100.0~F100.4)

- Signal type: NC->PLC
- Signal function: After machanical zero-return of some axis, NC sets the corresponding F signal as 1, and then sends to PLC, and then PLC logic controls according to the signal state. ZPn1, ZPn2, ZPn3, ZPn4 and ZPn5 are respecitively relative to the mechanical zero-return finish signal of the 1st axis, the 2nd axis, the 3rd axis, the 4th axis and the 5th axis.
- Points for attention: When mechanical zero-return (including G28 command zero-return) completes, and the current position is in the setting area, the mechanical zero-return finish signal changes into 1.

When the machine moves from the mechanical zero point, the emergency stop or the drive alarms and the mechanical zero-return finish signal changes into 0.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F94				ZP5	ZP4	ZP3	ZP2	ZP1
F96				ZP25	ZP24	ZP23	ZP22	ZP21
F98				ZP35	ZP34	ZP33	ZP32	ZP31
F100				ZP45	ZP44	ZP43	ZP42	ZP41

5.8.1.2 Mechanical zero point setting signal

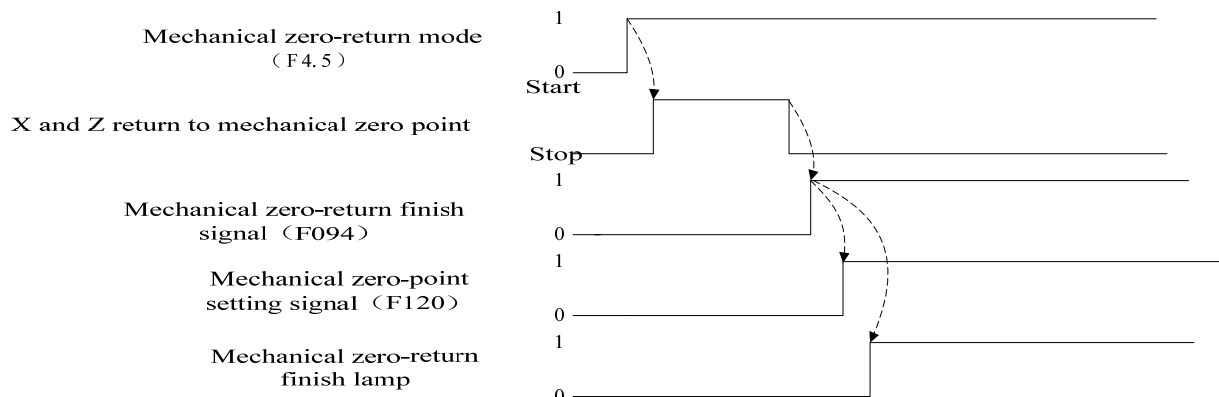
ZRF1~ZRF5(F120.0~F120.4)

- Signal type: NC->PLC
- Signal function: After CNC mechanical zero-return and the mechanical zero point is set, and the corresponding mechanical zero point sets the signal as 1, and then sends to PLC. PLC sets the state of signal and logic controls based on the mechanical zero. ZRF1, ZRF2, ZRF3, ZRF4 and ZRF5 set the signal based on the mechanical zero of the corresponging 1st axis, 2nd axis, 3rd axis, 4th axis and 5th axis.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F120				ZRF5	ZRF4	ZRF3	ZRF2	ZRF1

5.8.1.3 Mechanical zero-return signal time sequence



5.9 Auto running

5.9.1 Cycle start/feeding pause

- Cycle start (start auto running):

In auto or MDI mode, auto running start signal ST is valid, the programs starts running.

1. Signal ST is ignored during the following situations:

- A: Modes except auto or MDI mode;
- B: Feeding pause signal (SP) is 0
- C: Emergency stop (ESP) is 0
- D: External resetting signal (ERS) is 1
- E: Press “resetting” on the panel
- F: CNC is in the state of alarm
- G: Auto running has already started
- H: Program restart signal (SRN) is 1
- I: CNC is searching for one sequence number.

2. During auto running, CNC feeding pauses in the following states:

- A: Feeding pause signal (SP) is 0
- B: Switch into mode of jog, MPG, single step, mechanical zero-return or program zero-return
- C: Single block commands end during single block running
- D: Running has already ended in MDI mode
- E: Alarm in CNC
- F: After switch into edit mode, the commands of single block have already ended.

3. During auto running, CNC resets in the following states, the running stops:

- A: Emergency stop signal (ESP) is set as 0
- B: External resetting signal (ERS) is set as 1
- C: Press “resetting” on the panel

- Feeding pause (auto running cut-off):

During auto running, the pause signal SP is 0, CNC accesses into pause state and stops running. At the same time, the cycle start indicator signal STL is set as 0. If the feeding pause signal SPL is set as 1; even SP signal is reset as 1, the auto running won't restart again. When SP signal is set as 1, ST signal is valid and auto running restarts.

When execute the block with M, S and T function commands, SP signal is set as 0, STL signal changes into 0 immediately; when the signal SPL is 1, CNC accesses into pause state. When PLC sends FIN signal, CNC continues to execute the cut-off block. After the block ends, SPL signal is set as 0, (STL signal pause is 0), and CNC accesses the auto running stop state.

5.9.1.1 Cycle start signal

ST (G7.2):

- Signal type: PLC →NC, the falling edge check is valid.
- Signal function: In auto or MDI mode, after PLC receives the starting pulse input signal, set G7.2 as 1, and then 0. PLC sends to NC, auto running starts.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G7						ST		

5.9.1.2 Feeding pause signal

SP (G8.5)

- Signal type: PLC →NC, it is valid when it is 0.
- Signal function: After PLC receives the pause signal, set G8.5 as 0 and then send to NC, stop auto running. When SP input signal is 0, auto running can't be started.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G8			SP					

5.9.1.3 Cycle start signal

STL (F0.5)

- Signal type: NC→PLC
- Signal function: When CNC auto runs, CNC sets STL as 1, and then sends to PLC. PLC logic controls based on STL state.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F0			STL					

5.9.1.4 Feeding pause signal

SPL (F0.4)

- Signal type: NC→PLC
- Signal function: When CNC is in the pause state, set SPL as 1 and send to PLC. PLC logic controls based on SPL state.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F0				SPL				

5.9.1.5 Auto running signal

OP (F0.7)

- Signal type: NC→PLC
- Signal function: When CNC auto runs, set OP as 1, and then send to PLC. PLC logic controls according to the state of OP.

	Cycle start indicator signal STL	Feed pause indicator signal SPL	Auto running signal OP
Cycle start state	1	0	1
Feeding pause state	0	1	1
Auto running stop state	0	0	0
Resetting state	0	0	0

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F0	OP							

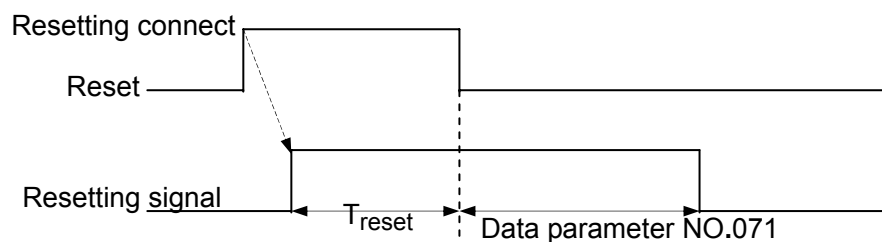
5.9.2 Resetting

In the following situations, CNC accesses the resetting state:

- A: Emergency stop signal (ESP) is set as 0.
- B: External resetting signal (ERS) is set as 1.
- C: Press “resetting” on the panel

When the above conditions are released, after the time set by data parameter NO.071, the resetting signal RST changes into 0.

$RST\ time = T_{reset} (\text{resetting time}) + \text{setting value of data parameter NO.071}$



During auto running, when CNC is reset, auto running stops and the machine decelerates and stops along the movement direction of control axis. When CNC is reset during executing M, S and T, signals of MF, SF and TF in 16ms is set as 0.

5.9.2.1 External resetting signal

ERS (G8.7)

- Signal type: PLC →NC
- Signal function: After PLC receives the external resetting input signal, reset G8.7, and then send to NC, reset CNC. When CNC is reset, RST signal changes into 1.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G8	ERS							

5.9.2.2 Resetting signal

RST (F1.1)

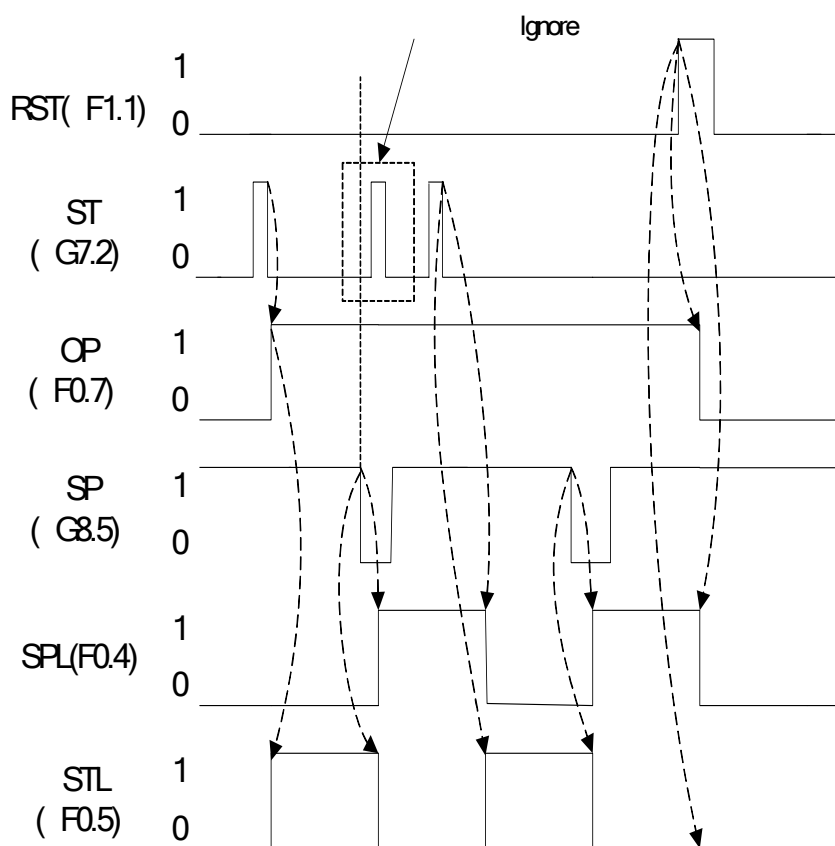
- Signal type: NC→PLC
- Signal function: When CNC is in the resetting state, set RST as 1, and then send to PLC, finally PLC logic controls based on the state of RST.

- Points for attention: In the following situations, RST is set as 1:
 - A: External emergency stop input signal (ESP) is set as 0.
 - B: External resetting signal (ERS) is set as 1.
 - C: Press “resetting” on the panel.
 After the above three states are released, the time set by data parameter NO.071 has already completed, RST is set as 0.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F1							RST	

5.9.2.3 Start running time sequence



5.9.3 Machine lock

Before processing, check the programs, the machine lock function is on, the machine locking all axes signal MLK or machine locking each axis signal MLK1~MLK4 are set as 1. During manual or auto running, stop output pulse to servo motor, while CNC still distributes commands and coordinates of absolute and relative also refresh. Check whether the edited programs are correct through monitor coordinate position change.

5.9.3.1 Machine locking all axes signal

MIK (G44.1)

- Signal type: PLC → NC
- Signal function: After PLC receives machine locking all axes signal, set MLK as 1, and then

send to NC, finally CNC sets all axes as machine lock.

- Points for attention: when MIK is 1 and during manual or auto running, CNC doesn't output the pulse to the axial servo motor and the machine table doesn't move.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G44							MIK	

5.9.3.2 Machine locking all axes check signal

MMLK (F4.1)

- Signal type: NC->PLC
- Signal function: When all axes are locked, CNC sets the locking all axes check signal as 1, and then send to PLC.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F4							MMLK	

5.9.4 Dry run

Dry run is valid in auto mode, the machine moves at the constant feedrate rather than the feedrate defined in the program. The function can check the movement of machine even the machine doesn't install the work piece, the running speed depends on the manual feeding override signal (JV0~JV15) .

Manual rapid feed selecting signal (RT)	Dry run speed
1	Manual rapid feedrate
0	Manual feedrate

5.9.4.1 Dry run signal

DRN (G46.7) :

- Signal type: PLC ->NC
- Signal function: After PLC receives the dry run input signal, set DRN as 1, and then send to NC, finally, CNC accesses dry run state.
- Points for attention: A: When DRN is 1, the machine moves at the feedrate set by dry run; when it is 0, the machine moves normally.
B: When DRN changes from 0 to 1 or from 1 to 0 during machine movement, the machine running speed decelerates into 0, and then accelerates into the specified feedrate.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G46	DRN							

5.9.4.2 Dry run check signal

MDRN (F2.7):

- Signal type: NC->PLC
- Signal function: When CNC is in the dry run state, set MDRN as 1, and then send to PLC.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F2	MDRN							

5.9.5 Single block

Single block running is only valid to auto running, when single block signal (SBK) is set as 1 during auto running. After execute the current block, CNC accesses the feeding pause state. When the single block signal (SBK) is set as 0, the program runs again.

5.9.5.1 Single block signal

SBK (G46.1):

- Signal type: PLC →NC
- Signal function: After PLC receives the running input signal of the single block, SBK is set as 1, and then sends to NC, finally, CNC accesses the single block running state.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G46							SBK	

5.9.5.2 Single block check signal

MSBK (F4.3) :

- Signal type: NC→PLC
- Signal function: When CNC is single block running state, set MSBK as 1 and then send to PLC.
- Points for attention: A: During thread cutting, SBK signal changes into 1 during thread cutting. After executing the thread cutting command, running stops after the first non-thread cutting block.
B: Fixed cycle running: During fixed cycle running, set SBK signal as 1. Running stops when it is close to drilling or run-out during positioning each time rather than stop at the end of block.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F4					MSBK			

5.9.6 Skipping any blocks

In auto running, when specify one slash at the starting of block, and skipping any block signal BDT is set as 1, the block is ignored.

5.9.6.1 Skipping any block signal

BDT (G44.0) :

- Signal type: PLC →NC
- Signal function: When PLC receives skipping any block input signal, set BDT as 1, and then send to NC, finally CNC accesses the state of skipping any blocks; in the program, block with "/" isn't executed.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G44								BDT

5.9.6.2 Skipping any block check signal

MBDT (F4.0) :

- Signal type: NC->PLC
- Signal function: When NC is in skipping any block state, set MBDT as 1, and then send to PLC.
PLC logic controls based on MBDT state.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F4								MBDT

5.9.7 Manual absolute value

During manual running (JOG feeding and manual feeding) and machine moves, select whether the movement amount adds in the current position of the work piece coordinate system, and output one chek signal to set the manual absolute value is on or off in CNC.

5.9.7.1 Manual absolute value signal

ABSM (G6.2)

- Signal type: PLC ->NC
- Signal function: Set manual absolute value signal on or off. When the signal is set as 1, the manual absolute value signal is invalid; 0, valid.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G6						ABSM		

5.9.7.2 Manual absolute value check signal

MABSM (F4.2)

- Signal type: NC ->PLC
- Signal function: Inform PLC about the state of manual absolute value signal. When ABSM (G6.2) signal is 0, the signal is 1, the manual absolute value function is valid; ABSM (G6.2) signal is 1, the signal is 0, the manual absolute value function is invalid.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F4						MABSM		

5.10 Feedrate control

5.10.1 Rapid movement signal

Rapid movement speed of each axis can be controlled by data parameter NO.1420 rather than programming, but it can adjust the speed of rapid movement through rapid movement override.

RPD0 (F2.1):

- Signal type: NC->PLC
- Signal function: When CNC executes the movement command at the rapid movement speed, sets RPD0 as 1, and then sends to PLC.
- Points for attention: A: After RPD0 is 1, which means select the rapid movement, some axis begins to move; if it is 0, which means non-rapid movement speed, some axis begins to move.
B: During auto running, the rapid movement includes all the rapid movement, such as the fixed cycle positioning and the mechanical zero-return, etc, not only GOO movement commands. Manual rapid movement also includes the rapid movement of mechanical zero-return.
C: Once rapid movement is selected, the signal remains bit 1 until the other feedrate is selected even during the cease time.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F2							RPD0	

5.10.2 Rapid movement override

Rapid movement override can be four levels of F0, 25%, 50% and 100%, and the speed defined by FO is set by parameter NO.1421.

In auto or manual mode (including mechanical zero-return, program zero-return), the actual movement speed is the product of the value, which is set by data parameter NO.1420, multiplying by the override value.

5.10.2.1 Rapid movement override signal

ROV1、ROV2 (G14.0、G14.1)

- Signal type: PLC ->NC
- Signal function: When PLC receives the rapid movement override input signal, assigns values ROV1 and ROV2, and then sends to NC, and sets the speed of rapid movement. The override values corresponding to ROV1 and ROV2 are as below:

ROV1	ROV2	Override value
0	0	100%
0	1	50%
1	0	25%
1	1	FO

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G14							ROV2	ROV1

5.10.3 Feedrate override

The feedrate can be adjusted through the feedrate override signal, and can also check the program. For example, when the specified feedrate in the program is 100mm/min, if the feedrate override is 50%, the machine moves at the speed of 50mm/min.

5.10.3.1 Feedrate override signal

FV0~FV7 (G12):

- Signal type: PLC →NC
- Signal function: After PLC receives the feedrate override input signal, it assigns values to FV0~FV7, and then sends to NC, finally, set the feedrate. The override values relative to the codes of FV0~FV7 in binary system are as below:

FV7~FV0 (G012.7~G012.0)	Cutting feed override
0000 1111	0%
0000 1110	10%
0000 1101	20%
0000 1100	30%
0000 1011	40%
0000 1010	50%
0000 1001	60%
0000 1000	70%
0000 0111	80%
0000 0110	90%
0000 0101	100%
0000 0100	110%
0000 0011	120%
0000 0010	130%
0000 0001	140%
0000 0000	150%

- Points for attention: during auto running, the actual feedrate is the product of the speed specified by cutting feed multiplying by the override value of the feedrate override signal.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G12	FV7	FV6	FV5	FV4	FV3	FV2	FV1	FV0

5.10.4 Canceling override signal

OVC (G6.4):

- Signal type: PLC →NC
- Signal function: After PLC receives the canceling override input signal, set OVC as 1, and then send to NC, finally fix the feedrate override as 100%.
- Points for attention: When OVC is 1, the operation of CNC is as below:
 - A: No matter how about the feedrate override signal is, the feedrate override is fixed as 100%.
 - B: The rapid movement override and the spindle speed override are not affected.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G6				OVC				

5.11 MST function

When the maximum 8 digits following M, S and T are specified, the corresponding code signal and gating signal are sent to PLC, and then PLC logic controls according to the state of the signals. The relative signals are as below:

Function	Program address	NC→PLC			(PLC→NC) Finish signal
		Code signal	Gating signal	Distributing finish signal	
Miscellaneous function	M	M00~M31	MF	DEN	FIN
Spindle speed function	S	S00~S31	SF		
Tool function	T	T00~T31	TF		

The processing is as below: (Change M code into S and T codes, namely, it is the spindle speed function, the tool function process)

A: It is supposed that M XXX is specified in the program; if CNC isn't set, it alarms.

B: After the code signals M00~M31 are specified, the gating signal MF is set as 1, the code signals are represented as the program command value XXX in binary system. If movement pauses, when the spindle speed or other functions are commanded with miscellaneous function at the same time, the code signals of miscellaneous function are sent, the other functions are begun to execute.

C: When the gating signal is 1, PLC reads code signal and executes the corresponding operation.

D: In one block, movement pauses or other functions end, execute the other operation till the distributing finish signal DEN changes into 1.

E: When operation ends, PLC sets the finish single FIN as 1. The finish signal is used in miscellaneous function, spindle speed and tool function. If these functions run at the same, the finish signal FIN is set as 1 after all functions end.

F: The finish signal FIN is 1 and must hold for some time, CNC sets the gating signal as 0 and informs that the finish signal has been received.

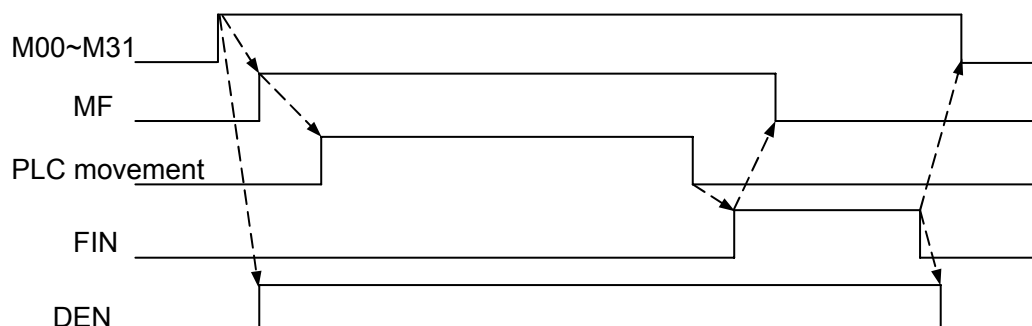
G: When the gating signal is 0, PLC sets FIN signal as 0.

H: When FIN signal is 0, CNC sets all code signals as 0, and end all miscellaneous functions in order.

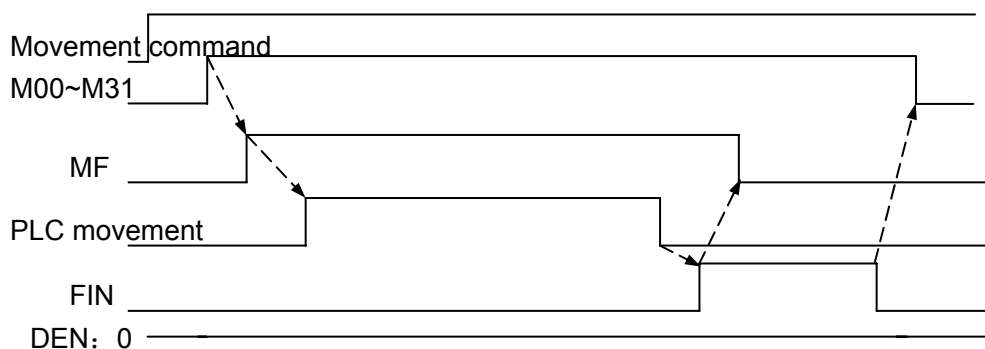
I: When the commands of one block are executed, CNC executes the next block.

The control time sequence is as below:

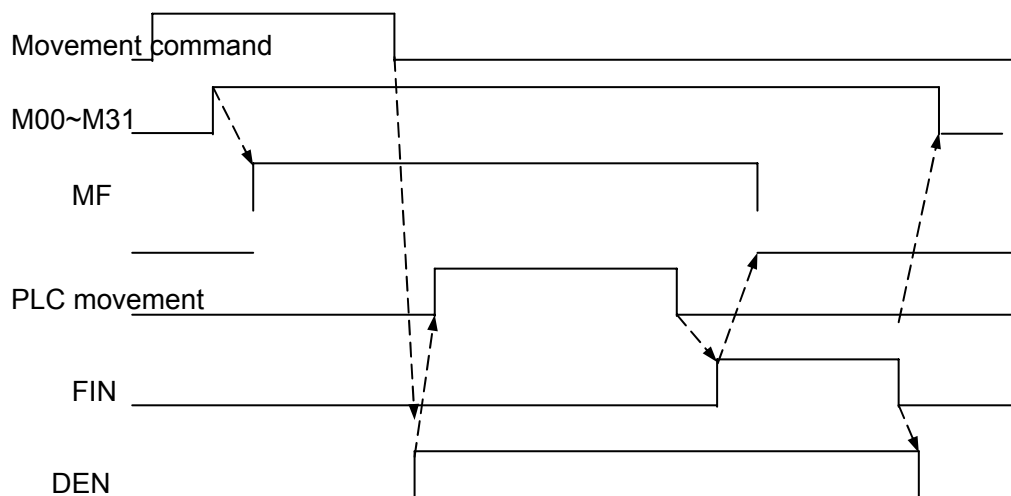
When there is one miscellaneous function in the block:



When the movement command and the miscellaneous function are in one block, execute the miscellaneous function even the movement command is in operation:



When the movement command and the miscellaneous function are in one block, execute the miscellaneous function after the movement command ends:



5.11.1 The miscellaneous function (M function)

5.11.1.1 The miscellaneous function code signal and the gating signal

The miscellaneous function code signal: M00~M31 (F10~F13)

The miscellaneous function gating signal: MF (F7.0)

- Signal type: NC->PLC
- Signal function: After the program executes M codes, set the corresponding F code signal and MF as 1, and then send to PLC, finally, logic control. About output conditions and execution process, refer to the above introduction of the execution process. The code corresponding relation of M commands and signal codes is as below:

F13~F10	M commands
F13, F12, F11, 00000000	M00
F13, F12, F11, 00000001	M01
F13, F12, F11, 00000010	M02
F13, F12, F11, 00000011	M03
F13, F12, F11, 00000100	M04
F13, F12, F11, 00000101	M05
F13, F12, F11, 00000110	M06
F13, F12, F11, 00000111	M07
F13, F12, F11, 00001000	M08

--	--

• Points for attention: Even the following miscellaneous function commands are commanded in CNC, they can't be output,

A: M98, M99, M198

B: Calling M codes of subprogram

C: Calling M codes of the user macro program

2: The following miscellaneous functions except code signals and gating signals can be output, the decipher signal can also be output: M00, M01, M02 and M30.

3: M00~M31 assign M codes in binary system, such as M5 relative to 00000000, 00000000, 00000000 and 00000101, which is shown on the above list.

• Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F10	M07	M06	M05	M04	M03	M02	M01	M00
F11	M15	M14	M13	M12	M11	M10	M09	M08
F12	M23	M22	M21	M20	M19	M18	M17	M16
F13	M31	M30	M29	M28	M27	M26	M25	M24
F7								MF

5.11.1.2 M decipher signal

DM00 (F9.7)、DM01 (F9.6)、DM02 (F9.5)、DM30 (F9.4):

- Signal type: NC->PLC, when it is 1, it's valid.
- Signal function: When CNC executes M00, M01, M02 and M30 commands, the corresponding decipher signals DM00, DM01, DM02 and DM30 are set as 1.

Program command	Output signal
M00	DM00
M01	DM01
M02	DM02
M30	DM30

• Points for attention: in the following conditions, M decipher signal is 1:

The corresponding miscellaneous function is specified and the other movement and pause commands end in one block (M decipher signal doesn't output if NC has already received FIN signal before movement and pause commands end.)

In the following conditions, M decipher signal is 0: when FIN signal is 1 or reset.

• Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F9	DM00	DM01	DM02	DM30				

5.11.1.3 Many M commands in one block

Maximum three M codes in one block can be commanded in CNC and output to the machine at the same time. Comparing the traditional method of one M code in one block, it takes the shorter processing cycle time.

The 2nd and 3rd M function code signals: M200~M215 (F14~F15), M300~M315 (F16~F17)

The 2nd and 3rd M function gating signals: MF2 (F8.4) and MF3 (F8.5)

- Signal type: NC->PLC
- Signal function: After the program executes the 2nd and 3rd M codes, set the corresponding F code signal, MF2 and MF3 as 1, and then send to PLC, finally, logic controls. The code corresponding relation between M codes and the code signals is as below:

F15~F14	2M command
F15, 00000000	M00
F15, 00000001	M01
F15, 00000010	M02
F15, 00000011	M03
F15, 00000100	M04
F15, 00000101	M05
F15, 00000110	M06
F15, 00000111	M07
F15, 00001000	M08
.	.

F17~F16	3M command
F17, 00000000	M00
F17, 00000001	M01
F17, 00000010	M02
F17, 00000011	M03
F17, 00000100	M04
F17, 00000101	M05
F17, 00000110	M06
F17, 00000111	M07
F17, 00001000	M08
.	.

- Points for attention: 1. Because of mechanical operation limit, some M codes can't be specified at the same time. Refer to the manuals of each machine manufacturer about specifying the mechanical operation limit of many M codes in one block meanwhile.
- 2. When M00, M01, M02 or M30 and other M codes are commanded together, other M codes are ignored and not executed; when M00, M01, M02 or M30 are commanded together, M code, which is commanded firstly, is valid, other M codes are ignored and not executed.

For example: One M command in one block

M03;

M10;

M12;

G01 X100 Z100;

.....

Many M commands in one block

M03 M10 M12;
G01 X100 Z100;
.....

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F14	M207	M206	M205	M204	M203	M202	M201	M200
F15	M215	M214	M213	M212	M211	M210	M209	M208
F16	M307	M306	M305	M304	M303	M302	M301	M300
F17	M315	M314	M313	M312	M311	M310	M309	M308
F8			MF3	MF2				

5.11.2 Spindle speed function (S function)

Spindle speed code signals S00~S31 (F22~F25) , spindle speed gating signal SF (F7.2)

- Signal type: NC->PLC
- Signal function: When execute S commands, NC sets the corresponding S code signals and SF as 1, and then sends to PLC, finally logic controls. About the relative output conditions and execution process, refer to the previous relative introduction. The corresponding code relation between S commands and code signals in binary system is shown as the following list:

F25~F22	S commands
F25, F24, F23, 00000000	S00
F25, F24, F23, 00000001	S01
F25, F24, F23, 00000010	S02
F25, F24, F23, 00000011	S03
F25, F24, F23, 00000100	S04
.	.
.	.
.	.
.	.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F22	S07	S06	S05	S04	S03	S02	S01	S00
F23	S15	S14	S13	S12	S11	S10	S09	S08
F24	S23	S22	S21	S20	S19	S18	S17	S16
F25	S31	S30	S29	S28	S27	S26	S25	S24
F7						SF		

5.11.3 Tool function (T function)

5.11.3.1 Tool function

Tool function code signals

T00~T31 (F26~F29) , Tool function gating signal TF (F7.3)

- Signal type: NC->PLC
- Signle function: When NC commands T command, set the corresponding T code signal and TF as 1, and then send to PLC for logic control. About the output conditions and execution process, refer to the previous relative introduction, the code corresponding relation between T command and T code signal in binary system is shown as the following list:

F29~F26	T command
F29, F28, F27, 00000000	T00
F29, F28, F27, 00000001	T01
F29, F28, F27, 00000010	T02
F29, F28, F27, 00000011	T03
F29, F28, F27, 00000100	T04

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F26	T07	T06	T05	T04	T03	T02	T01	T00
F27	T15	T14	T13	T12	T11	T10	T09	T08
F28	T23	T22	T21	T20	T19	T18	T17	T16
F29	T31	T30	T29	T28	T27	T26	T25	T24
F7					TF			

5.11.4 MST function end

5.11.4.1 Finish signal

FIN (G4.3)

- Signal type: PLC->NC
- Signal function: After the functions of miscellaneous, spindle speed and tool are executed, PLC sets FIN as 1 and then sends to NC.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G4				FIN				

MFIN (G5.0)

- Signal type: PLC->NC
- Signal function: After the miscellaneous function is executed, PLC sets MFIN as 1 and then sends to NC.
- Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G5							MFIN	

SFIN (G5.2)

- Signal type: PLC->NC
- Signal function: After the spindle speed function is executed, PLC sets SFIN as 1 and then sends to NC.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G5						SFIN		

TFIN (G5.3)

- Signal type: PLC→NC
- Signal function: After the tool function is executed, PLC sets TFIN as 1, and then sends to NC.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G5					TFIN			

5.11.4.2 Finish signals of M2 and M3

MFIN2 (G4.4), MFIN3 (G4.5)

- Signal type: PLC →NC
- Signal function: When M2 and M3 functions are ended, PLC sets the finish signal as 1 and then sends to NC.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G4			MFIN3	MFIN2				

5.11.4.3 Distributing finish signal

DEN (F1.3):

- Signal type: NC->PLC
- Signal function: When the miscellaneous, the spindle speed, and the tool functions and other commands (such as the movement and pause commands) share one block, after the other commands are executed, NC sets DEN as 1, and waits for FIN signal sent by PLC. After the block is executed, DEN turns into 0.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F1					DEN			

5.11.5 Miscellaneous function lock

5.11.5.1 Miscellaneous function lock signal

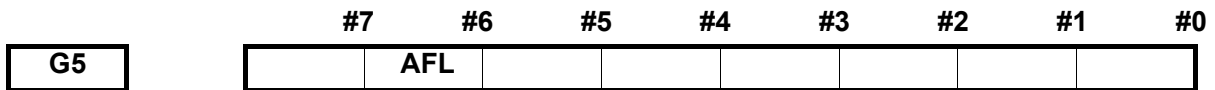
AFL (G5.6):

- Signal type: PLC →NC
- Signal function: When PLC receives the miscellaneous function locking input signal, AFL is set as 1, and then sent to NC, and it's not allowed to execute M, S and T functions.
- Points for attention: When AFL signal is 1, CNC processes as below:
 1. About the running in AUTO and MDI mode, CNC doesn't execute M, S and T functions, that is to say, the code signal and the gating signal are not output.
 2. After output the code signal, AFL is set as 1, and execute in the normal method till end (Till receive FIN signal and the gating signal is set as 0).
 3. When AFL is 1, commands of M00, M01, M02 and M30 can be executed, the corresponding signals of code, gating and decipher should be output based on

the normal mode.

4. When AFL is 1, the miscellaneous functions M98 and M99 are still executed on the normal mode while the executed result isn't output.
5. When AFL is 1, the spindle analog amount can be output.

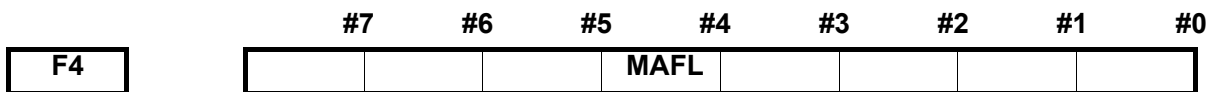
- Signal address:



5.11.5.2 Miscellaneous function locking check signal

MAFL (F4.4):

- Signal type: NC->PLC
- Signal function: When CNC is in the miscellaneous function locking state, MAFL is set as 1, and then sent to PLC.
- Signal address:



5.12 Spindle speed function

CNC classifies the spindle based on the spindle control mode: the gear spindle and the analog spindle. PLC parameter K10.4 sets the spindle type as gear or analog.

5.12.1 Gear spindle

Gear spindle is the spindle actual speed controlled by the mechanical gear of the machine, CNC changes S codes into the switch output, controls the machine mechanical gear and then controls the spindle speed.

5.12.2 Analog spindle

5.12.2.1 Analog spindle speed control

S commands controlled by the analog spindle: S commands are input by the machine program and specify the analog spindle speed controlled by CNC. About constant surface speed cutting (G96 mode), CNC switches the specified surface linear speed into the spindle speed.

S code/SF signal output: When parameter #3705.4 is 1, the analog spindle control function in CNC outputs values of S commands and SF gating commands to PLC; When parameter #3705.4 is 0, the analog spindle control function in CNC doesn't output values of S commands and SF gating commands to PLC.

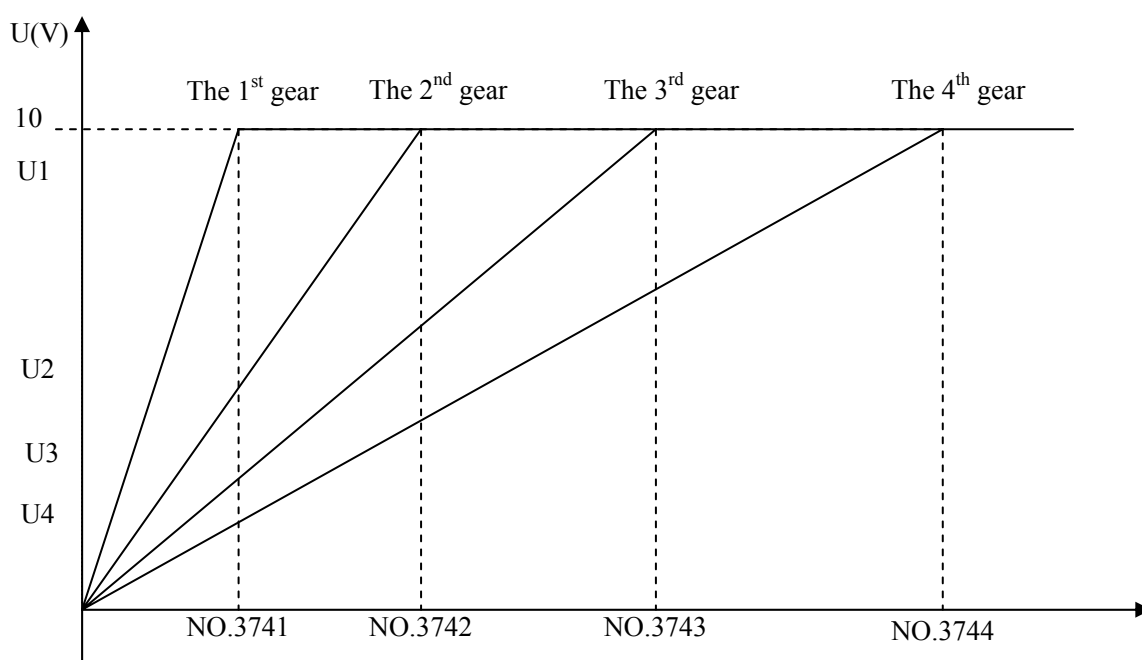
Switching gears: Although S commands specify the spindle speed, the actual controlled object is the

spindle motor. Therefore, CNC must have certain relative relation between the spindle motor speed and its gear. The machine sets the gear; CNC outputs the spindle speed corresponding to its gear.

Method of switching gears: To switch the gears, the maximum spindle speed corresponding to each gear is set by the data parameters NO.3741~NO.3744. Gears selecting signals are two coding signal (GR1 and GR2), the relation between the signal and the gear is as below:

GR1	GR2	Gear	The parameter number of the 1 st spindle maximum speed
0	0	1	Data parameter NO.3741
1	0	2	Data parameter NO.3742
0	1	3	Data parameter NO.3743
1	1	4	Data parameter NO.3744

The data parameters NO.3741~NO.3744 respectively set the spindle maximum speed of the 1st ~the 4th gears, the analog voltage 10V is relative to the spindle motor maximum speed, namely, one S command is in different levels, the analog voltage output has the following linear relations:



It is assumed that the data parameters are NO.3741=1000; NO.3742=2000; NO.3743=4000; NO.3744=5000. When the spindle speed is: S=800,

G28.1=0, G28.2=0 are in the 1st gear,

$$U1 = \frac{800}{1000} \times 10 = 8V$$

G28.1=1, G28.2=0 are in the 2nd gear,

$$U2 = \frac{800}{2000} \times 10 = 4V$$

G28.1=0, G28.2=1 are in the 3rd gear,

$$U3 = \frac{800}{4000} \times 10 = 2V$$

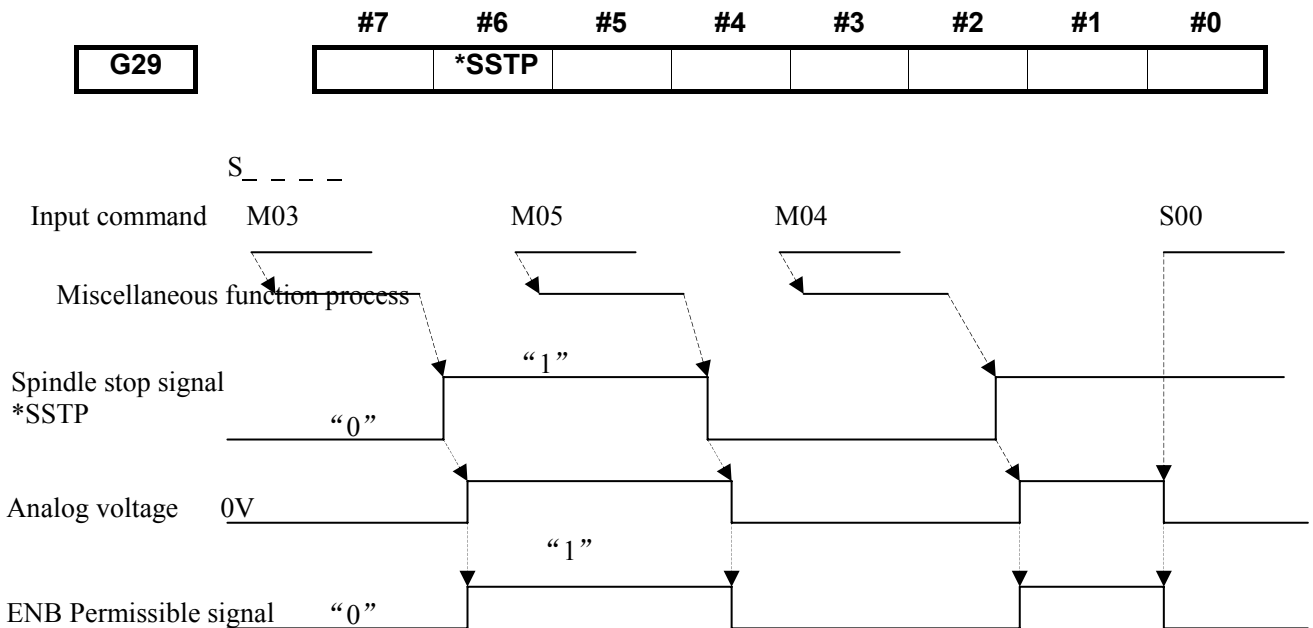
G28.1=1, G28.2=1 are in the 4th gear,

$$U4 = \frac{800}{5000} \times 10 = 1.6V$$

Calculation of the output voltage value: $U = \frac{\text{S command value}}{\text{the maximum speed of current gear}} \times 10$

Spindle stop signal* SSTP (G29.6) :

- Signal type: PLC →NC
- Signal function: Stop the output of the spindle speed command. The signal sets S command in NC as 0. About the time sequence, it is shown as the following figure:
- Points for attention: When the spindle stop signal *SSTP is 0, the output voltage changes into 0V. When the signal is 1, the analog voltage outputs the command value. When the signal isn't used, set the signal as 1, and CNC executes the spindle speed control.
- Signal address:



Spindle speed override signal: SOV00~SOV07 (G30)

- Signal type: PLC →NC
- Signal function: After PLC receives the spindle speed override input signal, assign the corresponding values to SOV00~SOV07, and then send to NC, finally set the different spindle speed override. The corresponding relation between SOV00~SOV07 codes and override values is as below:

SOV7~SOV0 (G30.7~G30.0)	Spindle override
0000 0101	50%
0000 0110	60%
0000 0111	70%
0000 1000	80%
0000 1001	90%
0000 1010	100%
0000 1011	110%
0000 1100	120%

- Points for attention: During the tapping cycle and the thread cutting, the spindle override function is invalid.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G30	SOV07	SOV06	SOV05	SOV04	SOV03	SOV02	SOV01	SOV00

Spindle permissible signal ENB (F1.4)

- Signal type: NC->PLC
- Signal function: It means whether there is the spindle command.
- Points for attention: When the non-zero command is output to spindle, ENB is 1; if the command is 0, ENB signal changes into logic 0. In the analog spindle, even the command is 0 and output to the spindle, namely, the analog voltage is 0V, it will result in the spindle motor rotates at low speed because the transducer has the floating voltage. In this case, ENB signal can stop the motor.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F1				ENB				

Gear selecting signal GR1,GR2 (G28.1, G28.2)

- Signal type: PLC->CNC
- Signal function: The signal informs CNC the selected gear currently, about the detailed movement, refer to the previous introduction.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G28						GR2	GR1	

Constant surface cutting speed signal CSS (F2.2)

- Signal type: PLC->CNC
- Signal function: When the signal is 1, it means it is executing the constant surface cutting speed control mode (G96); 0, it means it hasn't been executed.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F2						CSS		

Spindle speed reaching signal SAR (G29.4)

- Signal type: PLC->CNC
- Signal function: The signal informs CNC spindle has already reached the speed specified by the spindle.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G29				SAR				

Spindle motor speed selecting command signal SIND (G33.7)

- Signal type: PLC->CNC
- Signal function: The signal is to select the speed command of the spindle motor.

SIND 1: Control the spindle motor based on the speed command sent by PLC.

0: Control the spindle motor based on the speed command sent by CNC, namely, the spindle speed specified by S command.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G33	SIND							

Spindle S12 digits code signal R010~R120 (F36#0~F37#3)

- Signal type: CNC->PLC
- Signal function: The signal converts the spindle speed command calculated by CNC into the code information of 0~0Xfff.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F36	R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O
F37					R12O	R11O	R10O	R09O

PLC inputs the spindle motor speed command signal R01I~R12I (G32#0~G33#3)

- Signal type: PLC->CNC
- Signal function: The signal is to input the spindle motor speed command sent by PLC.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G32	R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I
G33					R12I	R11I	R10I	R09I

5.12.3 Spindle jog function

When the spindle jog function is valid, only the analog spindle is controlled; and the function is executed in the modes of manual, MPG/single, mechanical zero-return and program zero-return, etc.

5.12.3.1 Spindle jog signal

SPHD (G200.0)

- Signal type: PLC →NC
- Signal function: When the spindle jog function is valid, after PLC receives the spindle jog input signal, set G200.0 as 1, and then send to NC, the specified spindle jog is valid. When the spindle normally revolves, G200.0 is 0.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G200								SPHD

5.12.4 Multi-spindle control

Spindle selecting signal

SWS1 (G27.0)

SWS2 (G27.1)

- Signal type: PLC->NC
- Signal function: The multi-spindle controls whether S command of NC outputs to the spindle.
SWS1 1: Output the speed command to the 1st spindle
0: Not output the speed command to the 1st spindle

SWS2 1: Output the speed command to the 2nd spindle

0: Not output speed command to the 2nd spindle

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G27							SWS2	SWS1

Each spindle stop signal

SSTP1 (G27.3)

SSTP2 (G27.4)

- Signal type: PLC->NC
- Signal function: It is only valid to many spindles; each axis can be stopped by the signal.

SSTP1 1: Not output 0 rev/min to the 1st spindle

0: Output 0 rev/min to the 1st spindle

SSTP2 1: Not output 0 rev/min to the 2nd spindle

0: Output 0 rev/min to the 2nd spindle

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G27				SSTP2	SSTP1			

Gear selecting signal

GR21 (G29.0)

- Signal type: PLC->NC
- Signal function: It is to select the gear (2 gears) of the 2nd spindle during installing the multi-spindle.

The 2nd spindle is set by the data parameter NO.3811~NO.3812, the relation between the signal and the gear is as below:

GR21	Gear	Maximum speed parameter number of the 2 nd spindle
0	1	Data parameter NO.3811
1	2	Data parameter NO.3812

GR21 1: Select the 2nd gear for the 2nd spindle.

0: Select the 1st gear for the 2nd spindle.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G29								GR21

Selecting signal of the 2nd position encoder

PC2SLC (G28.7)

- Signal type: PLC->NC
- Signal function: The selecting signal of the position encoder

PC2SLC 1: The feedback pulse, which is obtained by the 2nd spindle encoder, is to control.

0: The feedback pulse, which is obtained by the 1st spindle encoder, is to control.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G28	PC2SLC							

The enable signal of the 2nd spindle

ENB2 (F38.2)

- Signal type: NC->PLC
- Signal function: Inform PLC whether output the command signal to the 2nd spindle during multi-spindle control. The signal is the condition to stop the analog spindle.
ENB2 1: Allow the 2nd spindle speed control
0: Not allow the 2nd spindle speed control

- Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F38						ENB2		

The 2nd spindle motor speed selecting command signal SIND2 (G35.7)

- Signal type: PLC->CNC
- Signal function: The signal is to select the speed command of the 2nd spindle motor.
SIND2 1: Control the 2nd spindle motor based on the speed command sent by PLC.
0: Control the 2nd spindle motor based on the speed command sent by CNC, namely, it is the spindle speed specified by S command.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G35	SIND2							

PLC inputs the 2nd spindle motor speed command signal R01I2~R12I2 (G34#0~G35#3)

- Signal type: PLC->CNC
- Signal function: The signal is to input the 2nd spindle motor speed command sent by PLC.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G34	R08I2	R07I2	R06I2	R05I2	R04I2	R03I2	R02I2	R01I2
G35					R12I2	R11I2	R10I2	R09I2

5.12.5 Switch between the spindle position/the speed

Spindle outline control switch signal

CON (G27.7)

- Signal type: PLC->NC
- Signal function: The signal specifies Cs outline control function. The servo spindle is switched between the spindle speed control mode and Cs outline control mode. When the signal is "1", the spindle is switched from the speed control mode to Cs outline control mode. The signal changes into "0", switch from Cs to the speed.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G27	CON							

Spindle outline control switch finish signal

FSCSL (F44.1)

- Signal type: NC->PLC
- Signal function: When the signal is “0”, it means the controlled axes are in the spindle speed control mode; “1”, it means the controlled axes are in Cs outline control mode.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F4							FSCSL	

5.12.6 Spindle 8-point position function

Spindle orientation signal SORI (Y3.4)

Spindle position signal SEC0~SEC2 (Y3.5~Y3.7)

Spindle position finish input signal COIN (X2.7)

M51~M58 are spindle position commands:

Remark: M51~M58 can't share the same block with the movement commands.

M51~M58 commands output correspondingly.

Programming commands	Output signals		
	SEC2 (Y3.7)	SEC1 (Y3.6)	SEC0 (Y3.5)
M51	0	0	0
M52	0	0	1
M53	0	1	0
M54	0	1	1
M55	1	0	0
M56	1	0	1
M57	1	1	0
M58	1	1	1

● Logic control

1. After executing the position function commands M51~M58, PLC→Drive sends the position selecting signals SEC0, SEC1 and SEC2, specify the position;
2. Delay 40 millisecond, PLC→Drive sends spindle position signal SORI;
3. Drive begins positioning;
4. After Drive positions, Drive →PLC outputs the spindle position finish signal COIN;
5. After PLC sends the position selecting signal, the position finish signal hasn't been received in 6000ms; the system alarms “Spindle position takes too much time”.
6. Before position, the spindle is in the rotation or stop states; after positioning, the spindle is in the stop state.

5.13 Tool function

When T code or HDT signal is specified, NC compares the required tool numbers with the current tool numbers NOWT00~NOWT31. If the tool number can't comply with each other, change tools; therefore, it causes the code and the gating signals of the required tool number, and the machine selects tools based on the signals.

CNC can change tools through T commands in auto and MDI modes, or change the tools in order through HDT signal in jog mode.

5.13.1 T command changing tools

The user can change tools by T commands in auto and MDI modes. After NC explains T commands, sends the tool post number specified by T commands and the gating signal, and then waits the PLC changing tools end.

Current tool number signal

NOWT00~NOWT31 (G201~G204)

- Signal type: PLC → NC
- Signal function: After PLC detects the current tool post signal, sets the corresponding values of NOWT00~NOWT31 (G201~G204), and then sends to NC about the current tool number. These signals are represented by the tool number in binary system.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G201	NOWT07	NOWT06	NOWT05	NOWT04	NOWT03	NOWT02	NOWT01	NOWT00

5.14 Programming command

5.14.1 Switch between metric/inch system

Input signal in inch system

INCH (F2.0)

- Signal type: NC → PLC
- Signal function: When the signal is 1, it means input in inch system (G20); 0, input in metric system (G21).
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F2								INCH

5.15 Interpolation function

5.15.1 Thread cutting

Thread cutting signal

THRD (F2.3)

- Signal type: NC → PLC

- Signal function: The signal means the thread is being cut.

The signal is 1 under the following circumstances:

1. Thread cutting mode;
2. Thread is being cut.

The signal is 1 in the following situations: It isn't in thread cutting mode; the thread isn't cut, either.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F2				THRD				

5.16 Display/setting/editing

5.16.1 Display counting parts

Target parts counting reaching signal

PRTSF (F62.7)

- Signal type: NC->PLC
- Signal function: When the quantity of the processing parts reaches the required part number set by parameter #6713, PRTSF signal is output to PLC; When #6713 is set as 0, PRTSF signal isn't output.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F62	PRTSF							

5.17 PLC axis control function

5.17.1 Function overview

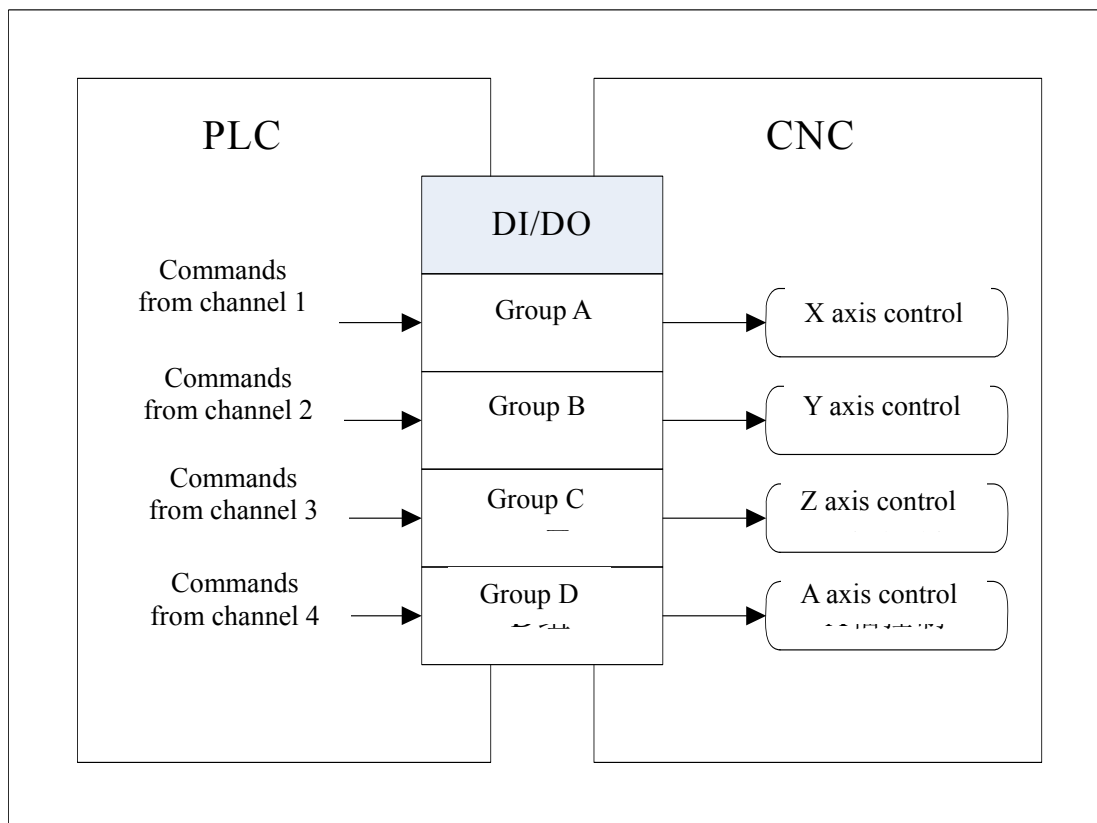
PLC can independently control the axes which are directly controlled by CNC. In other words, PLC can input the commands to traverse the tools along the axes which are not controlled by CNC, such as, command the traverse distance and the feedrate. PLC can control the coordinate axis, the tool post, the exchange work table, the graduation work table and the other external devices.

The quantity of the maximum controllable axes of GSK988T system is 5, parameter №1010 and №8010 set the quantity of CNC and PLC control axes. The specified axial control signal (EAX) sets one axis is controlled by CNC or PLC. PLC can directly control the following operations:

- (1) The distance of the rapid movement command;
- (2) The cutting feeding (feeding/min), the distance of the movement command;
- (3) The cutting feeding (feeding/rev), the distance of the movement command;
- (4) Pause;
- (5) Continuous feeding;
- (6) Manual reference point return;
- (7) The 1st reference point return;
- (8) The 2nd reference point return;
- (9) The 3rd reference point return;
- (10) The 4th reference point return;

- (11) Feedrate control;
- (12) The miscellaneous function, the 2nd miscellaneous function, and the 3rd miscellaneous function;
- (13) Machine coordinate system selection.

PLC provides 4 controlled channels, use signals of input and output to control the operations. Send commands through 4 channels, PLC can control 5 independent axes at the same time. Parameter №8010 sets the channels to control the corresponding axes. Commands can be sent to two or more axes through one channel, then, PLC can use one channel to control many axes. The control sketch is shown as below:



In the following introduction, the signals of input/output in four channels are respectively group A (channel 1), group B (channel 2), group C (channel 3) and group D (channel 4). The names of input/output signals controlled by PLC always include the small caps “g”, which respectively correspond to the signals of four groups.

5.17.2 Basic steps

Basic stops controlled by PLC are as below:

- (1) In parameter №8010, specify DI/DO signal group (A, B, C or D) controls the corresponding axis during PLC control.

When control two or more axes in one group, it checks whether the parameter setting is same with each controlled axis and the setting is relative with the feedrate (the rapid movement speed, the time constant of acceleration and deceleration, diameter/radius and linear axis/rotation axis, etc).

To make PLC direct control axes valid, set the selecting signal of the controlled axes (from EAX1 to EAX5) as 1.

(2) Setting the operation type

The axial control command signal (from EC0g to EC6g) command operation type. The axial control feedrate signal (from EIF0g to EIF15g) commands the feedrate. The axial control data signals (from EID0g to EID31g) command the movement distance and other data.

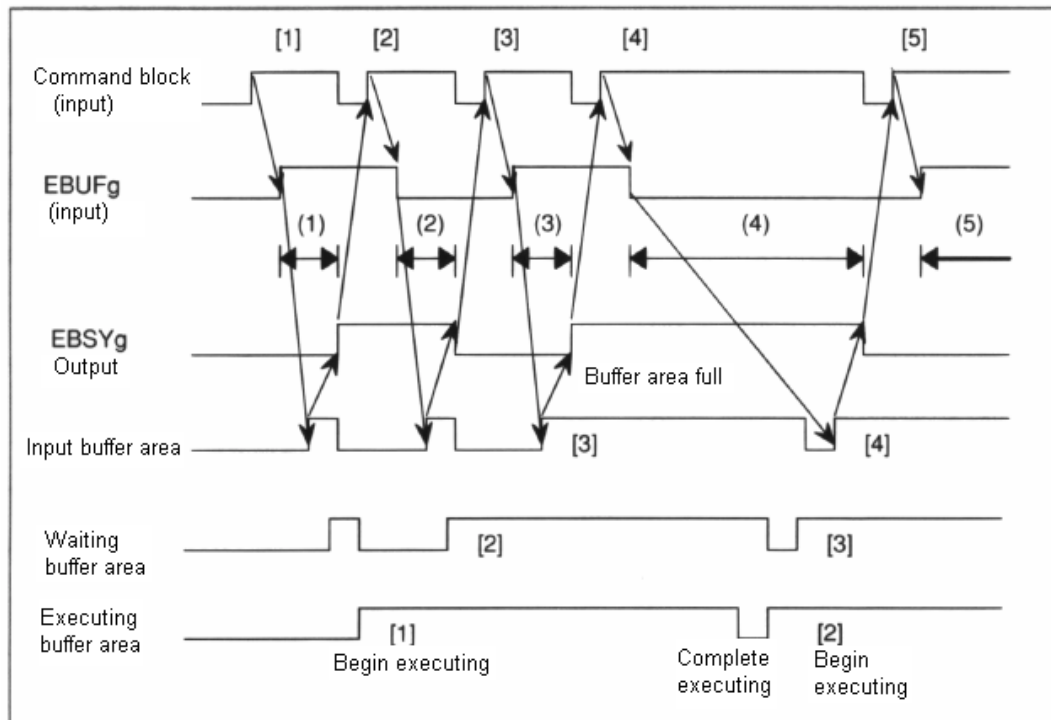
These signals and signal of forced service EMSBKg are together to set one complete operation. Namely, execute one block during CNC auto control. These signals are collectively called as the axial control block data signal.

The list is that PLC axis can control the signals in one block:

GENERAL TERMS	SIGNAL NAME	CODES	DATA TYPE
The block data signals controlled by axes	The block forced service signals	EMSBKg	Bit
	The axial control command signals	EC0g~EC6g	Byte
	The axial control feedrate signal	EIF0g~EIF15g	Byte
	The axial control data signal	EID0g~EID31g	Double bytes

- (3) After specify one complete operation (one block) data, CCW axial control command reading logic state of signal EBUFg is from “0” to “1”, or contrary. Therefore, the axial control command reading finish signal EBSYg must be same with the logic state of EBUFg.

CNC can save PLC axial control function in its buffer area; therefore, it can execute multiple PLC control in order. If the buffer area is void, CNC can receive the new block commands sent by PLC during executing the other block. There are three buffer areas in CNC, respectively, they are buffer areas of input, waiting and executing, and the time sequence diagram of the command is as below:



The state of CNC buffer area can be set through AND-OR of the axial control reading signal EBUF which is input from PLC, and that of the axial control command reading finish signal EBSYg which is output from CNC.

EBUFg	EBSYg	(XOR)	STATE OF CNC BUFFER AREA
0	0	0	The previous block has read CNC buffer area and PLC can send the next block.
1	1		
0	1	1	The previous block has't been read, is being read or waits CNC buffer area changes into usable state, so, the next block isn't output, logic state of EBUFg isn't turned, either. The state of CCW EBUFg makes the sent block invalid.
1	0		

(4) Repeat the steps (3) and (4) till all blocks are sent.

When the last block has been sent, the control axis selecting signals from EAX1 to EAX5 are set as "0". However, before set these signals as "0", check the input in CNC memory and wait till all blocks in buffer area are executed. When one block is being executed, set the signal as "0", or there are still blocks in the buffer area, which will result in P/S alarm. The alarm stops executing the block and makes the blocks which are saved in input and waiting buffer area become invalid.

To gurantee there isn't any blocks being executed, or without any blocks in the input and waiting areas, check control axis selecting state signal *EAXSL should be set as "0".

About those axes controlled by PLC, such as, the control tool post, the exchange work table and those axes of ATC, gurantee the signals from EAX1 to EAX5 are set as "1". After sending the commands from PLC to CNC, it doesn't require setting these signals as "0". When all the blocks have been executed (without the surplus blocks to be executed), CNC auto stops executing.

(5) When the control axis selecting signals from EAX1~EAX5 are from "1" to "0", control returns to CNC.

5.17.3 Control axis selecting signal

EAX1~EAX5

(G136.0~G136.4)

- Signal type: PLC->NC
- Signal function: When the signal is set as “1”, the corresponding axes are controlled by PLC.

When the signal is set as “0”, PLC control becomes invalid. Only when the control axis selecting state signal *EAXSL is set as “0”, the setting of the control axis selecting signal can be changed. When *EAXSL is set as “1”, the axial selecting signal can be changed, P/S (№139) alarms. The alarm signal EIALg is set as “1”.

When the fifth bit (NCC) of parameter №8001 is set as “0”, meanwhile, the control axis selecting signal is set as “1”, and the signal *EAXSL is set as “0”, execute the commands sent from CNC. When the parameter is set as “1”, execute the above operation, P/S alarms (№139). Remark: During manual continuous feeding mode, when the tool traverses along the axis, the command is invalid.

When CNC is executing one command, if the control axis selecting signal is set as “1”, P/S (№139) alarms. During manual continuous feeding mode, set the signal as “1”, cut off executing the command. When the control axis selecting signal is set as “1”, P/S (№139) alarms; Meanwhile, when *EAXSL is set as “0”, the state of alarm signal EIALg can't be changed into “1”. In such case, even CNC alarms, the axes are still controlled by PLC.

- Points for attention: After setting the control axis selecting signals from EAX1 to EAX5 as “1”, it takes minimum 8ms before PLC sends commands to CNC.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G136				EAX5	EAX4	EAX3	EAX2	EAX1

5.17.4 Axis control command signal

EC0g~EC6g

(G141.0~6)

(G151.0~6)

(G161.0~6)

(G171.0~6)

- Signal type: PLC->NC
- Signal function: Command the following operations through each channel.

AXIS CONTROL COMMANDS (Code in hexadecimal system)	OPERATION
00h	Rapid movement (linear acceleration or deceleration)
	Execute the same operation with G00 of CNC.
01h	Cutting feed/min (index acceleration or deceleration after interpolation)
	Execute the same operation with G98G01of CNC
02h	Cutting feed/rev (index acceleration or deceleration after interpolation)

	Execute the same operation with G99G01 of CNC
04h	Pause
	Execute the same operation with G04 of CNC
05h	Reference point return
	Tool traverses at the rapid traverse mode in the reference point return direction set by the fifth bit (ZMIx) of parameter №1006, and then, execute the same operation with the manual reference point return usually executed by CNC.
06h	Continuous feeding (index acceleration or deceleration)
	Tool traverses at JOG mode in the specified direction; execute the same operation with JOG of CNC.
07h	The 1 st reference point return
	Execute the same operation with positioning the tool in the reference point and passing the intermediate point, which is specified by G28 of CNC
08h	The 2 nd reference point return
	Execute the same operation with positioning in the reference point passing the intermediate point, which is specified by G30P2 of CNC
09h	The 3 rd reference point return
	Execute the same operation with positioning in the reference point passing the intermediate point, which is specified by G30P3 of CNC
0Ah	The 4 th reference point return
	Execute the same operation with positioning in the reference point passing the intermediate point, which is specified by G30P4 of CNC
10h	Speed command (Linear acceleration or deceleration)
	Execute the continuous feeding at the specified speed
12h	The miscellaneous function
	Execute the same function with the miscellaneous function (M function) used by CNC.
14h	The 2 nd miscellaneous function
	Execute the same function with the miscellaneous function (M function) used by CNC
15h	The 3 rd miscellaneous function
	Execute the same function with the miscellaneous function (M function) used by CNC
20h	Machine coordinate system selection
	Execute the same operation with that of G53 function (M function) used by CNC

Rapid movement speed

When the rapid movement command (from EC0g to EC6g: 00h) is used, the feedrate can be same as the one used by CNC (№1420) or specified by PLC axis feedrate signals from EIF0g to EIF15g. And it can be set by 0 bit (RPD) of parameter №8002.

Reference position return free of the block

Reference point return command (from EC0g to EC6g: 05h) realizes the following operations: The 1st bit (DLZ) of parameter №1002 sets all controlled axes reference point return without the block; the 1st bit (DLZx) of parameter №1005 sets one axes reference point return free of the block. After one of the above is set and each axis doesn't return to reference point after power on, and then, reference point return commands (from EC0g to EC6g: 05h) are sent, the tool traverses in the reference point return direction set by the 5th bit (ZMlx) of parameter №1006, and return to reference position without deceleration signal (The tool is set in the nearest grid dot from the current position) .

After setting the reference position, the reference position return command is sent (from EC0g to EC6g: 05h) and the reference position return can be executed at the high speed without considering the reference position return direction set by the 5th bit of parameter №1006.

The 1st reference position return without the block

When use the 1st reference position return command (from EC0g to EC6g: 07h), if the 1st bit (DLZ) of parameter №1002 commands all axes reference point return free of the block or the 1st bit (DLZx) of parameter №1005 commands one axis reference position return free of the block as valid; moreover, the tool has never returned to the reference position since power on, send the 1st reference position return command (from EC0g to EC6g: 07h), P/S (№090) alarms.

Return from the 1st to the 4th reference position

When use return commands from the 1st to the 4th reference position (from EC0g to EC6g: from 07h to 0Ah), the feedrate can be specified by the zero bit (RPD) of the parameter №8002, and the method is same as that of the rapid traverse command (from EC0g to EC6g: 00h). Remark: When return to the 1st reference position, after power on, if the tool has never returned to the reference position manually, then the system defaults the feedrate set by parameter №1424.

Speed command

When use speed command (from EC0g to EC6g: 10h), the controlled axes specified by the 0 bit (ROTx) of parameter №1006 is revolving axes.

When the continuous feeding command (from EC0g to EC6g: 06h) is executing the position control, the speed command (from EC0g to EC6g:10h) controls the speed of the servo motor, and then, the speed can change dynamically during continuous feeding. The command can be used for the tool revolved by the servo motor.

When use parameter №8028, set the time constant of linear acceleration or deceleration in each axis.

Remark: When the speed commands are executing JOG feeding, the coordinate value doesn't change, which results in the tool position missing. Therefore, after the continuous feeding end, before executing the movement command, the tool always returns to the reference position.

Machine coordinate system selection

The machine coordinate system selecting command (from EC0g to EC6g: 20h) executes the absolute position, the tool traverses to the position specified by the machine coordinate system at the rapid speed. The command is to traverse the tool to the position set by the machine, such as: the tool exchange position.

About the revolving axis, it can command the short path for revolving. When use this command, the tool offset and nose radius compensation should be canceled.

Before using this command, the machine coordinate system must be set. After power on, use manual or G28, the tool returns to the reference position. When use absolute position detector, the tool isn't required returning to the reference position because the tool position is saved in the memoriser.

The corresponding relation between the axis control commands and their data is shown in the following list:

COMMAND BLOCK		
OPERATION	AXIS CONTROL CODE SIGNAL FROM EC0g TO EC6g	COMMAND DATA
Rapid movement	00h	The total movement distance from EID0g to EID31g, the rapid movement speed from EIF0g to EIF15g, when the 0 bit of parameter №8002 is set as “1”, the rapid movement speed is valid.
Feeding/min	01h	The total movement distance from EID0g to EID31g, the feedrate from EIF0g to EIF15g
Feeding/rev	02h	The total movement distance from EID0g to EID31g, the feeding amount/rev from EIF0g to EIF15g
Pause	04h	Pause time from EID0g to EID31g
Reference position return	05h	Null
JOG feeding	06h	Feeding direction EID31g JOG feedrate from EIF0g to EIF15g
The 1 st reference position return	07h	The rapid movement speed from EIF0g to EIF15g, when the 0 bit (RPD) of parameter №8002 is set as “1”, the rapid traverse speed is valid.
The 2 nd reference position return	08h	
The 3 rd reference position return	09h	
The 4 th reference position return	0Ah	
The speed command	10h	Continuous feedrate from EIF0g to EIF15g
The miscellaneous function	12h	The miscellaneous function codes from EID0g to EID15g
The 2 nd miscellaneous function	14h	
The 3 rd miscellaneous function	15h	
The machine coordinate system setting	20h	The machine coordinate system setting (the absolute value) from EID0g to EID31g The rapid movement speed from EIF0g to EIF15g, when the 0 bit (RPD) of parameter №8002 is set as “1”, the rapid movement speed is valid.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G141		EC6A	EC5A	EC4A	EC3A	EC2A	EC1A	EC0A
G151		EC6B	EC5B	EC4B	EC3B	EC2B	EC1B	EC0B
G161		EC6C	EC5C	EC4C	EC3C	EC2C	EC1C	EC0C
G171		EC6D	EC5D	EC4D	EC3D	EC2D	EC1D	EC0D

5.17.5 The axis control feedrate signal

EIF0g~EIF15g

(G142,G143)

(G152,G153)

(G162,G163)

(G172,G173)

- Signal type: PLC->NC
- Signal function: Command the feedrate value of PLC axis.
Rapid movement (from EC0g to EC6g: 00h)
The 1st reference position return (from EC0g to EC6g:07h)
The 2nd reference position return (from EC0g to EC6g: 08h)
The 3rd reference position return (from EC0g to EC6g: 09h)
The 4th reference position return (from EC0g to EC6g: 0Ah)

About these commands, when the 0 bit (RPD) of parameter №8002 is set as “1”, the signal commands the rapid movement speed in binary system. However, about the return of the 1st reference position, after connecting power supply, if the manual reference position return has never been executed, use the rapid movement speed set by parameter №1424.

Data units are shown in the following list:

		DATA UNIT		UNIT
		IS-B	IS-C	
Straight axis	The machine in metric system	1		mm/min
	The machine in inch system	0.1		inch/min
Rotation axis		1		deg/min

The valid data range is shown as the following list:

		DATA UNIT		UNIT
		IS-B	IS-C	
Straight axis	The machine in metric system	30~15000	30~12000	mm/min
	The machine in inch system	30~6000	30~4800	inch/min
Rotation axis		30~15000	30~12000	deg/min

Cutting feed/min (from EC0g to EC6g: 01h)

About this command, the signal commands the feedrate of one axis in binary system. The data of command can be set by the 3rd bit (F10) of parameter №8002 * 10.

Data units are shown as the following list:

The 3rd bit (F10) of parameter №8002 is set as "0":

		DATA UNIT		UNIT
		IS-B	IS-C	
Straight axis	The machine in metric system	1	0.1	mm/min
	The machine in inch system	0.01	0.001	inch/min
Rotation axis		1	0.1	deg/min

The 3rd bit (F10) of parameter №8002 is set as "1":

		DATA UNIT		UNIT
		IS-B	IS-C	
Straight axis	The machine in metric system	10	1	mm/min
	The machine in inch system	0.1	0.01	inch/min
Rotation axis		10	1	deg/min

The valid data range is listed as below:

		DATA UNIT		UNIT
		IS-B	IS-C	
Straight axis	The machine in metric system	1~100000	0.1~12000.0	mm/min
	The machine in inch system	0.01~4000.00	0.001~480.000	inch/min
Rotation axis		1~100000	0.1~12000.0	deg/min

Cut feeding/rev (from EC0g to EC6g: 02h)

About the command, the signal is to specify the tool traverse amount relative to the spindle each revolution.

The data increment units are set by the 6th bit (FR1) and the 7th bit (FR2) of parameter №8002, which is shown as the following list:

PARA.		INPUT IN METRIC SYSTEM (mm/rev)	INPUT IN INCH SYSTEM (inch/rev)	ROTATION AXIS (deg/rev)
FR2	FR1			
1	1	0.0001	0.000001	0.0001
0	0			
0	1	0.001	0.00001	0.001
1	0	0.01	0.0001	0.01

The valid data range is shown as the following list:

		DATA UNIT		UNIT
		IS-B	IS-C	
Straight axis	The machine in metric system	0.0001~500.0000		mm/rev
	The machine in inch system	0.000001~9.999999		inch/rev
Rotation axis		0.0001~500.0000		deg/rev

Continuous feeding (from EC0g to EC6g: 06h)

Set the feedrate as setting cut feeding/min (from EC0g to EC6g:01h). During the continuous feeding, the feedrate can be changed. Signals EIF0g~EIF15g can command the feedrate. During the continuous feeding, after CCW axial control command reading signal EBUFg, the tool traverses at the new feedrate. 9

Because JOG feeding command isn't buffered, it doesn't require checking axis control command reading finish signal EBUFg. The specified feedrate can be set through the third bit (F10) of parameter NO8002 *10 and the second bit (JFM) of parameter NO8004 *200.

The speed command (from EC0g to EC6g: 10h)

About the command, the signal specifies the speed of the servo motor in binary system.

The positive commands rotate in positive direction; the negative commands rotate in negative.

When the new servo motor speed is commanded, the logic of the axial control command reading signal EBUFg negates, which makes the servo motor accelerate or decelerate to the new speed value.

- Points for attention: When it is set as "0", CNC continuous to execute buffer rather than to traverse the tools. In this case, input resetting signal ECLRg to release the buffer. The cutting speed limit is invalid.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G142	EIF7A	EIF6A	EIF5A	EIF4A	EIF3A	EIF2A	EIF1A	EIF0A
G143	EIF15A	EIF14A	EIF13A	EIF12A	EIF11A	EIF10A	EIF9A	EIF8A

	#7	#6	#5	#4	#3	#2	#1	#0
G152	EIF7B	EIF6B	EIF5B	EIF4B	EIF3B	EIF2B	EIF1B	EIF0B
G153	EIF15B	EIF14B	EIF13B	EIF12B	EIF11B	EIF10B	EIF9B	EIF8B

	#7	#6	#5	#4	#3	#2	#1	#0
G162	EIF7C	EIF6C	EIF5C	EIF4C	EIF3C	EIF2C	EIF1C	EIF0C
G163	EIF15C	EIF14C	EIF13C	EIF12C	EIF11C	EIF10C	EIF9C	EIF8C

	#7	#6	#5	#4	#3	#2	#1	#0
G172	EIF7D	EIF6D	EIF5D	EIF4D	EIF3D	EIF2D	EIF1D	EIF0D
G173	EIF15D	EIF14D	EIF13D	EIF12D	EIF11D	EIF10D	EIF9D	EIF8D

5.17.6 Axis control data signal

EID0g~EID31g

(G144,G145,G146,G147)

(G154,G155,G156,G157)

(G164,G165,G166,G167)

(G174,G175,G176,G177)

- Signal type: PLC->NC
- Signal function: Command data controlled by PLC.

The data units are shown as the following list:

		DATA UNIT		UNIT
		IS-B	IS-C	
Straight axis	Input in metric system	0.001	0.0001	mm
	Input in inch system	0.0001	0.00001	inch
Rotation axis		0.001	0.00001	deg

Rapid movement (from EC0g to EC6g: 00h)

Cut feeding/min (from EC0g to EC6g: 01h)

Cut feeding/min (from EC0g to EC6g: 02h)

About these commands, use signals EID0g~EID31g based on the axis input increment and specify the increment movement distance in binary system.

The valid data range is listed as below:

		DATA UNIT		UNIT
		IS-B	IS-C	
Straight axis	Input in metric system	±99999.999	±9999.9999	mm
	Input in inch system	±9999.9999	±999.99999	inch
Rotation axis		±99999.999	±9999.9999	deg

The 3rd bit (DIAx) of parameter №1006 sets the diameter programming, and the 1st bit (CDI) of parameter №8005 sets radius or diameter should be used in commands.

Pause (from EC0g to EC6g: 04h)

About this command, the signal commands the pause time in binary system.

DATA RANGE	UNIT
1~99999999	ms

Continuous feeding (from EC0g EC6g: 06h)

About this command, signal EID31g commands the continuous feeding direction, which is shown as below:

- 0: Positive direction
- 1: Negative direction

Signals EID0g~EID30g are not defined.

The miscellaneous function (from EC0g to EC6g: 12h)

The 2nd miscellaneous function (from EC0g to EC6g: 14h)

The 3rd miscellaneous function (from EC0g to EC6g: 15h)

About the command, the signal sends the miscellaneous function codes from PLC to CNC in binary system. The miscellaneous function codes can use one or two bytes of signals EID0g~EID15, which is set by the 6th bit (AUX) of parameter №8001.

The machines coordinate system selection (from EC0g to EC6g: 20h)

About the command, according to the increment system used by axis, the signal commands the absolute coordinate system in binary system.

- Points for attention: When use IS-C increment system, the minimum input increment of pause time is 0.1ms, which is set by the 1st bit (DWE) of parameter №8002.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G144	EID7A	EID6A	EID5A	EID4A	EID3A	EID2A	EID1A	EID0A
G145	EID15A	EID14A	EID13A	EID12A	EID11A	EID10A	EID9A	EID8A
G146	EID23A	EID22A	EID21A	EID20A	EID19A	EID18A	EID17A	EID16A
G147	EID31A	EID30A	EID29A	EID28A	EID27A	EID26A	EID25A	EID24A

	#7	#6	#5	#4	#3	#2	#1	#0
G154	EID7B	EID6B	EID5B	EID4B	EID3B	EID2B	EID1B	EID0B
G155	EID15B	EID14B	EID13B	EID12B	EID11B	EID10B	EID9B	EID8B
G156	EID23B	EID22B	EID21B	EID20B	EID19B	EID18B	EID17B	EID16B
G157	EID31B	EID30B	EID29B	EID28B	EID27B	EID26B	EID25B	EID24B

	#7	#6	#5	#4	#3	#2	#1	#0
G164	EID7C	EID6C	EID5C	EID4C	EID3C	EID2C	EID1C	EID0C
G165	EID15C	EID14C	EID13C	EID12C	EID11C	EID10C	EID9C	EID8C
G166	EID23C	EID22C	EID21C	EID20C	EID19C	EID18C	EID17C	EID16C
G167	EID31C	EID30C	EID29C	EID28C	EID27C	EID26C	EID25C	EID24C

	#7	#6	#5	#4	#3	#2	#1	#0
G174	EID7D	EID6D	EID5D	EID4D	EID3D	EID2D	EID1D	EID0D
G175	EID15D	EID14D	EID13D	EID12D	EID11D	EID10D	EID9D	EID8D
G176	EID23D	EID22D	EID21D	EID20D	EID19D	EID18D	EID17D	EID16D
G177	EID31D	EID30D	EID29D	EID28D	EID27D	EID26D	EID25D	EID24D

5.17.7 Axis control command reading signal

EBUFg

(G140.7)

(G150.7)

(G160.7)

(G170.7)

- Signal type: PLC->NC
- Signal function: Command CNC reads the command data block controlled by PLC axis. The signal changes from “0” to “1” or from “1” to “0”. About the detailed running situation, refer to “basic steps”.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G140	EBUFA							
G150	EBUFB							
G160	EBUFC							
G170	EBUFD							

5.17.8 Axis control command reading finish signal

EBSYg

(F140.7)

(F150.7)

(F160.7)

(F170.7)

- Signal type: NC->PLC
- Singal function: Inform the system that CNC has already read one command data block controlled by PLC axis and saved in the input buffer area. About the details of output conditions and steps, refer to “basic steps”.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F140	EBSYA							
F150	EBSYB							
F160	EBSYC							
F170	EBSYD							

5.17.9 Resetting signal

ECLRg

(G140.6)

(G150.6)

(G160.6)

(G170.6)

- Signal type: PLC->NC

- Signal function: Reset the corresponding PLC control axes.

When the signal is “1”, execute the following operations:

- (1) When the tools traverse along the axis: decelerate and stop the tools.
- (2) When the tool pauses: operation is cut off.
- (3) When the miscellaneous function is being executed: stop operation.

Meanwhile, all buffered commands are cleared. When the signal is set as “1”, ignore all control commands.

Continuous feeding commands (from EC0g to EC06g: 06h) can set the resetting signal ECLRg as “1” to end executing. When these commands end, the servo motor decelerates and stops, the axial movement signal EGENg is set as “0”, and the control axis selecting state signal *EAXSL changes into “0”. Before the control axis selecting state signal *EAXSL changes into “0”, the resetting signal can’t be “0”.

Speed command (from EC0g to EC6g: 10h) can also be ended through setting the resetting signal as “1”. Before the command is ended, the servo motor decelerates and stops, and the axis moving signal EGENg is set as “0”. Before sending the next command, the axis moving signal EGENg must be “0”. Before the axis moving signal EGENg is set as “0”, the resetting signal ECLRg can’t be “0”.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G140		ECLRA						
G150		ECLRB						
G160		ECLRC						
G170		ECLRD						

5.17.10 Axis control pause signal

ESTPg

(G140.5)

(G150.5)

(G160.5)

(G170.5)

- Signal type: PLC->NC
- Signal function: When the signal is set as “1”, the operation is executed as below:
 - (1) When the tool traverses along the axis, decelerate and stop the tool.
 - (2) When the tool pauses: operation is cut off.
 - (3) When the miscellaneous function is being executing: When the miscellaneous function finish signal EFING is input, stop operation.

Set the signal as “0”, and the operation, which has been stopped, restarts.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G140			ESTPA					
G150			ESTPB					
G160			ESTPC					
G170			ESTPD					

5.17.11 Block stop signal

ESBKg

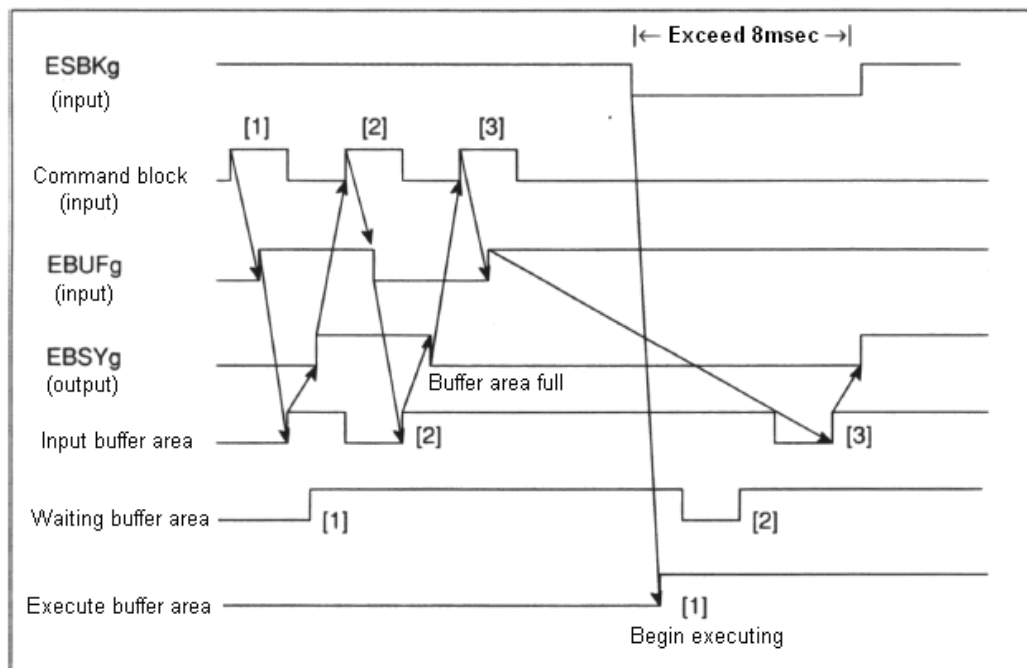
(G142.3)

(G154.3)

(G166.3)

(G178.3)

- Signal type: PLC->NC
- Signal function: When PLC sends the commands, the block stop signal ESBKg is set as "1", the block, which is being executed, ends, and the axis control stops. When the signal is set as "0", execute the commands in the buffer memory. The time sequence of command operation is shown as below:



- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G140					ESBKA			
G150					ESBKB			
G160					ESBKC			
G170					ESBKD			

5.17.12 Block stop invalid signal

EMSBKg

(G141.7)

(G151.7)

(G161.7)

(G171.7)

- Signal type: PLC->NC

• Signal function: When block stop invalid signal EMSBKg is set as “1” in the block, the block stop signal ESBKg is invalid.

• Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G141	EMSBKA							
G151	EMSBKB							
G161	EMSBKC							
G171	EMSBKD							

5.17.13 Miscellaneous function code signal

EM11g~EM48g

(F142, F143)

(F152, F153)

(F162, F163)

(F172, F173)

• Signal type: NC->PLC

• Signal function: When the miscellaneous function command (from EC0g to EC6g:12h), the 2nd miscellaneous function command (from EC0gt to EC6g:14h) , the 3rd miscellaneous function command (from EC0g to EC6g:15h) are sent from PLC, the miscellaneous function sends the command in one byte (signals EID0g~EID7g) or two bytes (EID0g~ EID15g) , which is set by the 6th bit (AUX) of parameter №8001.

• Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F142	EM28A	EM24A	EM22A	EM21A	EM18A	EM14A	EM12A	EM11A
F143	EM48A	EM44A	EM42A	EM41A	EM38A	EM34A	EM32A	EM31A

	#7	#6	#5	#4	#3	#2	#1	#0
F152	EM28B	EM24B	EM22B	EM21B	EM18B	EM14B	EM12B	EM11B
F153	EM48B	EM44B	EM42B	EM41B	EM38B	EM34B	EM32B	EM31B

	#7	#6	#5	#4	#3	#2	#1	#0
F162	EM28C	EM24C	EM22C	EM21C	EM18C	EM14C	EM12C	EM11C
F163	EM48C	EM44C	EM42C	EM41C	EM38C	EM34C	EM32C	EM31C

	#7	#6	#5	#4	#3	#2	#1	#0
F172	EM28D	EM24D	EM22D	EM21D	EM18D	EM14D	EM12D	EM11D
F173	EM48D	EM44D	EM42D	EM41D	EM38D	EM34D	EM32D	EM31D

5.17.14 Miscellaneous function gating signal

EMFg

(F141.0)

(F151.0)

(F161.0)

(F171.0)

- Signal type: NC->PLC
- Signal function: When the miscellaneous function commands (from EC0g to EC6g:12h) and the code commands of the miscellaneous function are sent, the signal is set as “1”.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F141								EMFA
F151								EMFB
F161								EMFC
F171								EMFD

5.17.15 Miscellaneous function 2 gating signal

EMF2g

(F141.2)

(F151.2)

(F161.2)

(F171.2)

- Signal type: NC->PLC
- Signal function: The signal is set as “1” when the 2nd miscellaneous function (from EC0g to EC6g:14h) commands and the code commands of the miscellaneous function are sent.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F141						EMF2A		
F151						EMF2B		
F161						EMF2C		
F171						EMF2D		

5.17.16 Miscellaneous function 3 gating signal

EMF2g

(F141.3)

(F151.3)

(F161.3)

(F171.3)

- Signal type: NC->PLC
- Signal function: The signal is set as “1” when the 3rd miscellaneous function (from EC0g to EC6g:15h) commands and the code commands of the miscellaneous function are sent.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F141					EMF3A			
F151					EMF3B			
F161					EMF3C			
F171					EMF3D			

5.17.17 Miscellaneous function finish signal

EFINg

(G140.0)

(G150.0)

(G160.0)

(G170.0)

- Signal type: PLC->NC
- Signal function: CNC sends the miscellaneous function codes to the miscellaneous function code signals EM11g ~ EM28g and EM31g ~ EM48g, and waits for the miscellaneous function finish signal EFINg. When the miscellaneous function finish signal EFINg returns, CNC begins to execute the next block.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G140								EFINA
G150								EFINB
G160								EFINC
G170								EFIND

5.17.18 Buffer invalid signal

EMBUFg

(G140.2)

(G150.2)

(G160.2)

(G170.2)

- Signal type: PLC->NC
- Signal function: When the signal is “1” and is being executed or is waiting, or the input buffer area includes one block, the commands of PLC aren’t read. If the signal is “1”, when any buffer area includes one block and the block is executed, read the commands in order only when all buffer areas are void.

To distinguish the buffer invalid state, only when all buffer areas are void and the commands are read, CNC outputs the axis control command reading finish signal EBSYg.

For the following commands, no matter what the state of buffer invalid signal EMBUFg is, the buffer is invalid:

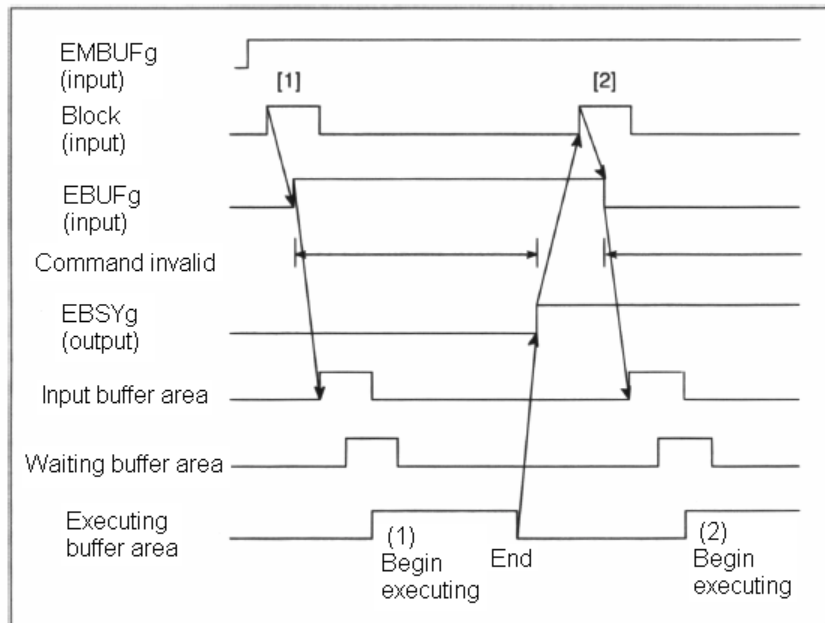
- (1) Reference position return (from EC0g to EC6g: 05h)
- (2) The 1st reference position return (from EC0g to EC6g: 07h)
- (3) The 2nd reference position return (from EC0g to EC6g: 08h)
- (4) The 3rd reference position return (from EC0g to EC6g: 09h)

- (5) The 4th reference position return (from EC0g to EC6g: 0Ah)
- (6) The machine coordinate system selection (from EC0g to EC6g: 20h)

The following commands are ended by the resetting signal ECLRg. Execute it when the buffer memory is invalid, namely, the following block is canceled rather than executed.

- (1) Continuous feeding (from EC0g to EC6g: 06h)
- (2) Continuous command (from EC0g to EC6g: 10h)

The time sequence diagram of command operation is shown as below:



- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G140						EMBUFA		
G150						EMBUFB		
G160						EMBUFC		
G170						EMBUFD		

5.17.19 Control axis selecting state signal

*EAXSL

(F129.7)

- Signal type: NC->PLC
- Signal function: When the signal is set as "0", the control axis selecting signals EAX1~EAX5 can be changed.

In the following situations, the signal is "1":

- (1) The tool traverses along PLC control axis;
- (2) When one block is the reading buffer area;

When the signal is "1", the control axis selecting signals EAX1~EAX5 can't be changed. If these signals are changed, P/S №139 alarms.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F129	*EAXSL							

5.17.20 In-position signal

EINPg

(F140.0)

(F150.0)

(F160.0)

(F170.0)

- Signal type: NC->PLC
- Signal function: When the corresponding PLC control axis is in-position state, the signal is set as "1".

When the tool decelerates, execute the in-position check, and execute the commands in the next block till the tool accesses the in-position area. However, whether skip the in-position check, which is set by the 6th bit (NCI) of parameter №8004 to reduce cycle time.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F140								EINPA
F150								EINPB
F160								EINPC
F170								EINPD

5.17.21 Follow zero error check signal

ECKZg

(F140.1)

(F150.1)

(F160.1)

(F170.1)

- Signal type: NC->PLC
- Signal function: The signal is set as "1" when PLC control axis is executing follow zero error check or in-position check.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F140							ECKZA	
F150							ECKZB	
F160							ECKZC	
F170							ECKZD	

5.17.22 Alarm signal

EIALg

(F140.2)

(F150.2)

(F160.2)

(F170.2)

- Signal type: NC->PLC
- Signal function: The signal is “1” when there is the servo alarm, the overtravel alarm or P/S alarms №130 and №139 in PLC control axis. After alarm releases, when the resetting signal ECLRg is “1”, the signal is “0”, which is introduced as below.

Servo alarm

Clear the reason of alarm, and then reset CNC.

Overtravel alarm

The tool traverses in the area of the memory travel limit and then reset CNC.

During the overtravel alarm, the following commands the tool traverses to the area of the memory travel limit:

- (1) Rapid traverse (from EC0g to EC6g: 00h)
- (2) Cut feeding/min (from EC0g to EC6g: 01h)
- (3) Cut feeding/rev (from EC0g to EC6g: 02h)
- (4) Continuous feeding (from EC0g to EC6g: 06h)

P/S alarm

Reset CNC.

In the above situations, the resetting signal ECLRg can't reset CNC, use the resetting button set on the panel.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F140						EIALA		
F150						EIALB		
F160						EIALC		
F170						EIALD		

5.17.23 Axis movement signal

EGENg

(F140.4)

(F150.4)

(F160.4)

(F170.4)

- Signal type: NC->PLC
- Signal function: The signal is “1” when the tool traverses along PLC control axis based on the commands, such as, the rapid traverse (from EC0g to EC6g: 00h) and the cut feeding (from ECOg to EC6g: 01h).

- Points for attention: When the axes distribution ends, the signal is set as "0". During decelerating, the signal is set as "0".
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F140				EGENA				
F150				EGENB				
F160				EGENC				
F170				EGEND				

5.17.24 Miscellaneous function executing function

EDENg

(F140.3)

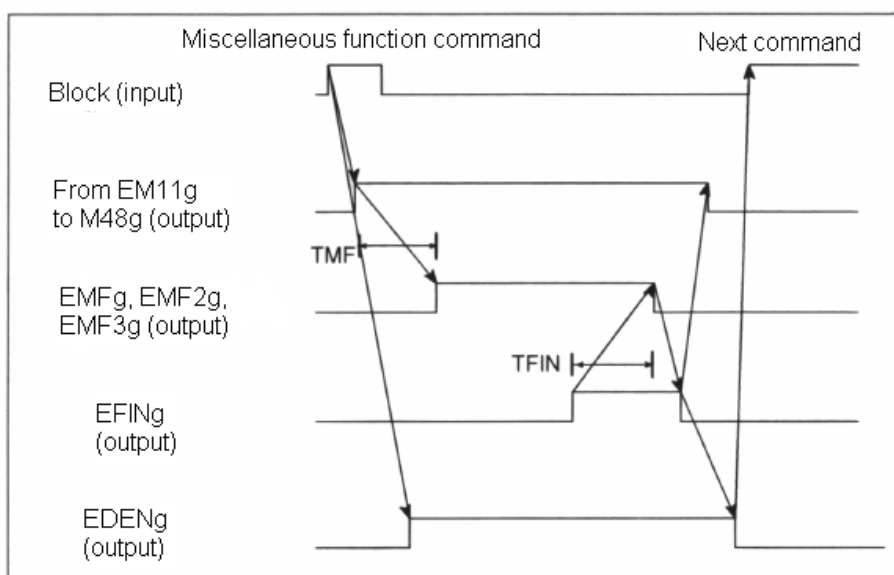
(F150.3)

(F160.3)

(F170.3)

- Signal type: NC->PLC
- Signal function: The signal is "1" when PLC commands the miscellaneous function (from EC0g to EC6g:12h) , the miscellaneous function co00000des from EID0g to EID15g are sent to the miscellaneous function code signals EM11g~EM48g, till the miscellaneous function finish signal EFING returns.

The time sequence diagram of command operation is shown as following:



- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F140				EDENA				
F150				EDENB				
F160				EDENC				
F170				EDEND				

5.17.25 Negative overtravel signal

EOTNg

(F140.6)

(F150.6)

(F160.6)

(F170.6)

- Signal type: NC->PLC
- Signal function: When exceed negative travel limit, the signal EOTNg is "1". Meanwhile, the alarm signal EIALg is "1". These signals are "0" when the overtravel alarm is released and the resetting signal ECLRg is "1".
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F140		EOTNA						
F150		EOTNB						
F160		EOTNC						
F170		EOTND						

5.14.26 Positive overtravel signal

EOTPg

(F140.6)

(F150.6)

(F160.6)

(F170.6)

- Signal type: NC->PLC
- Signal function: The signal EOTPg is "1" when exceed the positive travel limit. Meanwhile, the alarm signal EIALg is "1". These signals are "0" when the overtravel alarm is released and the resetting signal ECLRg is "1".
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F140			EOTPA					
F150			EOTPB					
F160			EOTPC					
F170			EOTPD					

5.17.27 Feedrate override signal

*FV0E~*FV7E

(G138)

- Signal type: PLC->NC
- Signal function: Like CNC feedrate override signals *FV0~*FV7, these signals select the override of the cutting feedrate. Use the 2nd bit (OVE) of parameter №8001, the setting override of PLC control axis isn't connected with CNC. The override calculation method is same as that of CNC. When all the signals are "0", the override is taken as 0%; when all the signals are "1", it's also 0%.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G138	*FV7E	*FV6E	*FV5E	*FV4E	*FV3E	*FV2E	*FV1E	*FV0E

5.17.28 Canceling override signal

OVCE

(G137.5)

- Signal type: PLC->NC
- Signal function: When the 2nd bit (OVE) of parameter №8001 is “1”, the feeding override of PLC isn’t connected with CNC. When the signal is set as “1”, the cutting feed override is fixed as 100%. The signal doesn’t affect the rapid traverse override.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G137			OVCE					

5.17.29 Rapid traverse override signal

ROV1E, ROV2E

(G137.0, G137.1)

- Signal type: PLC->NC
- Signal function: The signal is for the override of the rapid traverse speed, the 2nd bit (OVE) of parameter №8001 is set, the rapid override of PLC isn’t connected with CNC.

Rapid traverse override signal		Override value
ROV2E	ROV1E	
0	0	100%
0	1	50%
1	0	25%
1	1	F0

F0 is the low speed set by parameter №1421.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G137							ROV2E	ROV1E

5.17.30 Rapid traverse override signal

DRNE

(G137.7)

- Signal type: PLC->NC
- Signal function: Set the 2nd bit (OVE) of parameter №8001, the signal executes dry running and it’s not connected with CNC. When the signal DRNE of dry running is set as “1”, the specified rapid traverse speed and the cutting feedrate all are ignored. The tool traverses at the speed which the dry running speed (set by parameter №1410) multiplies by the specified override. The 3rd bit (RDE) of

parameter №8001 sets the dry running is valid or invalid to the rapid traverse.

Manual rapid traverse selecting signal	Commands from PLC	
	Rapid traverse	Cutting feed
1	Rapid traverse speed	Maximum cutting feedrate
0	Dry running feedrate ×FV or rapid traverse speed	Dry running feedrate ×FV

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G137	DRNE							

5.17.31 Manual rapid traverse selecting signal

RTE

(G137.6)

- Signal type: PLC->NC
- Signal function: During dry running, when the manual rapid traverse selecting signal RTE is set as “1”, the tool rapidly traverses at the rapid traverse speed, and cut and feed at the maximum cutting and feeding speed. When the signal is set as “0”, the tools traverse at the dry running speed. When the dry running signal DRNE is set as “0”, the specified rapid traverse speed or the cutting speed is restored.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
G137		RTE						

5.17.32 Override 0% signal

EOV0

(F129.5)

- Signal type: NC->PLC
- Signal function: When the feeding override is 0%, the signal is “1”.
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F129			EOV0					

5.17.33 Distribution finish signal

EADEN1~EADEN5

(F112.0~F112.4)

- Signal type: NC->PLC
- Signal function: When PLC commands the tools traverse, these signals are set as “0”. Except during executing the traverse commands, the axis control pause signal ESPg makes the axis traverse pause, and the tools remains still in the other cases and these signals are set as “1”.

- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F112				EADEN5	EADEN4	EADEN3	EADEN2	EADEN1

5.17.34 Buffer area full of signals

EABUFg

(F141.1)

(F151.1)

(F161.1)

(F171.1)

- Signal type: NC->PLC
- Signal function: When input buffer area includes one block, the signal is "1".
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F141							EABUFA	
F151							EABUFB	
F161							EABUFC	
F171							EABUFD	

5.17.35 Control signal

EACNT1~EACNT5

(F192.0~F192.4)

- Signal type: NC->PLC
- Signal function: When the control axis selecting state signal *EAXSL is set as "1", correspondingly, the signal EACNTn of the controlled axis is set as "1".
- Signal address:

	#7	#6	#5	#4	#3	#2	#1	#0
F192				EACNT5	EACNT4	EACNT3	EACNT2	EACNT1

Appendix

Appendix 1 Alarm List

A1.1 Program alarm (P/S alarm)

No.	Message	Content
000	Emergency stop, ESP open circuit	Recover ESP signal input to clear alarm.
001	Program open failure	Press reset key to clear alarm or power on after power off.
002	One block cannot exceed 256 characters	Modify the program because of too many characters in it.
003	Data exceeds the permitted range	Data input exceeds the permitted range or the specified data exceeds 8 numbers. Modify the data.
004	Addresss not found	Only input numerical value or sign instead of address at the head of the block. Modify the program.
005	No data following addresses	Data does not follow the address or the expression followed the address is mistaken without brackets. Modify the program.
006	Incorrect negative sign	Sign"-" is used by mistake(use it in mistaken position or input them too many). Modify the program.
007	Incorrectly used decimal point	The decimal point "." is used by mistake(use it in mistaken position or input them too many). Modify the program.
008	Illegal address input	The unavailable addresses are input in the valid message area. Modify the program.
009	Incorrect G code	An unusable G code or G code corresponding to a not provided function is specified. Modify the program.
010	Address duplication error	The same addressed appears more than once in a block. Alternatively, a block contains two or more G codes belonging to the same group. Please refer to parameter 3403#6 AD2. Modify the program.

011	Command cannot run in DNC occurs	Command that cannot run in DNC is found. Modify the program.
012	Too many M codes	Specifying multiple M codes in the same block is not allowed. Please refer to parameter 3404#7 M3B. Modify the program.
014	Divided by zero	Division by zero is specified (including $\tan 90^\circ$). Modify the program.
017	Parameter writing failure	Please check whether the parameter file is in normal state. Pay attention that the user area may be corrupted.
018	Part program operation failure	Press "RESET" key to cancel the alarm.
019	End of record	The end of record (%) is specified, or the program end is not specified. Please refer to parameter 3404#6 EOR. Modify the program.
020	DNC time out	DNC transmission failure; Please check it.
021	The setting value of feedrate is not within the range.	During cutting feed, the feedrate is not specified or incorrectly specified. The values for modal G98 and G99 are different. Please check modal G98 and G99, and modify the program.
022	The setting value of spindle speed is out of the range.	Spindle rotational speed or surface speed is set incorrectly. Please refer to parameter 3031 SCB and modify the program.
023	M command value is out of the range.	A wrong M code is specified. Please refer to parameter 3030 MCB and modify the program.
024	G code usage error	The G code needs to be used independently, and cannot coexist with other G codes in the same block. Modify the program.
025	Illegal tool number	An inexistent tool number is specified. Please refer to parameter 3032 TCB and modify the program.
028	T code is not allowed in this block.	G50, G10 and G04 cannot be specified in the same block with T codes. Please refer to parameter 5006#1 TGC and modify the program.
031	Too many axes commanded	The number of commanded axes exceeds that of simultaneously controlled axes. Modify the program.
032	Axis that cannot perform	Axis not included in the selected plane is specified

	interpolation command is specified.	or the basic axis and the parallel axis are specified simultaneously. Modify the program.
033	Illegal plane axis commanded	In circular interpolation, axis not included in the selected plane is specified. Modify the program.
034	No arc radius command	In circular interpolation, neither R nor I, J, K is specified. Refer to parameter 3403#5 CIR and modify the program.
035	Illegal radius command	In circular interpolation, address R value is wrong. Please refer to parameter 3403#4 RER and modify the program.
036	Exceeds the radius difference range	In circular interpolation, the difference between the distance from the start point to the center point and distance from the end point to the center point exceeds the value set by parameter. Please refer to parameter and modify the program.
037	Thread run-out length J, K value commanded incorrectly in thread cutting.	The run-out length exceeds the permitted range. K value is less than zero in G32, G34 commands; J value or K value is less than zero in G92 command. Modify the program.
038	Illegal lead command	Lead command value F is out of the range; or in variable lead thread cutting, the lead variation exceeds the range. Modify the program.
039	In thread cutting command, the thread run-out length of long axis is excessive.	The thread run-out length of long axis exceeds the thread cutting length. Modify the program.
040	In thread cutting command, the thread run-out length of short axis is excessive.	In G92 command, the thread run-out length of the short axis exceeds the distance from the starting point to the end point.
041	Illegal plane selection	In plane selection command, more than one parallel axes are specified in the same direction. Modify the program.
042	Metric/inch conversion command error	The metric/inch conversion command is not specified alone in a line, or is not specified in the first line. Metric/inch conversion is performed when a subprogram is called. Modify the program.
047	The specified axis does not	The specified axis does not return to the reference

	return to the reference point.	point by cycle start. Please perform reference point return.
048	Wrong reference point commanded	In G30 block, a value other than 2~4 is specified by P. Modify the program.
058	G31 cannot be used in feed-per-rotation mode	In feed-per-rotation mode, skip cutting command is specified. Modify the program.
059	G31 cannot be used in tool nose radius compensation mode	Skip cutting command is specified in tool nose radius compensation mode. Modify the program.
065	The offset value are accumulated excessively	In G50 offset accumulation, the accumulated offset exceeds the permitted range. Modify the program.
074	Illegal command G12.1/G13.1	When the polar coordinate interpolation is started or cancelled, the condition is wrong. 1). G12.1/G13.1 is specified in the mode that is not specified by G40. 2). Error is found in plane selection. The parameter specifying is erroneous. Modify the program or the parameter.
075	An unusable G code is specified in polar coordinate interpolation.	An unusable G code is specified in polar coordinate interpolation. Modify the program.
081	Undefined address P	In the programs commanded by M98, G65, G66, the address P (program number) is not defined. Modify the program.
082	Subprogram nesting error	The nested subprogram exceeds 12 levels.
083	Program number not found	In the blocks that contain M98, M99, G65, G66, the program number specified by address P is not found. Modify the program.
084	Subprogram call error	A higher-level program or the subprogram itself is called by M98, G66 or G66. Modify the program.
085	Program call statement cannot be run in MDI and DNC modes.	Macro program call and subprogram call in MDI and DNC modes are not supported. Modify the program.
090	Axis command error in spindle constant surface speed control mode	In modal G96, the basic calculation axis commanded by parameter does not exist. Modify the program.
121	Canned cycle command is specified in non-ZX plane.	The canned cycle command is not specified in the basic ZX coordinate system. Modify the program.
122	Axis not included in the basic ZX	Axis not included in the ZX coordinate system is

	coordinate system is specified in canned cycle.	specified in canned cycle. Modify the program.
123	The R value (radius value) is greater than the U value (absolute value) in G90, G92 commands.	In G90, G92 commands, when the plus or minus signs for the R and U are different, the absolute value of R value (radius value) is greater the U value (absolute value). Modify the program.
124	In G94 command, the R absolute value is greater than W absolute value	In G94 command, when the plus or minus signs for the R and W are different, the absolute value of R is greater that of the W value. Modify the program.
126	Illegal plane selection in multiple-cycle command	Cycle command is not specified in ZX plane. Modify the program.
127	Axis not included in the ZX plane is specified in G70~G76.	Axis not included in the ZX plane is specified in G70~G76 commands or the G70~G76 loop. Modify the program.
128	Incorrect G code in G70~G73	An unusable G code is commanded between the two blocks which is specified by addresses P and Q in G70~G73. Modify the program.
129	G70~G73 commands cannot be run in MDI mode	G70~G73 commands including addresses P, Q are specified in MDI mode.
130	Macro statement execution is not allowed in G70~G73 loop	Macro statement execution is not allowed in G70~G73 loop. Modify the program.
131	Subprogram is called in G70~G73 loop	Subprogram cannot be called in G70~G73 loop. Modify the program.
132	Subprogram is called in G70~G73 command lines	Subprogram cannot be called in G70~G73 command lines. Modify the program.
133	In G70~G73 commands, the addresses P or Q is out of the range	In G70~G73 commands, the addresses P or Q is undefined or out of the range. Modify the program.
134	The sequence number is not found in G70~G73 commands	The sequence number specified by address P or Q is not found in G70~G73 commands. Modify the program.
135	P and Q commands error in G70~G73 commands	In G70~G73 commands, the command values of P and Q are the same. Modify the program.
136	Two continuous blocks is not found in G71~G73 commands	Two continuous blocks is not found in G71~G73 commands, which will cause error. Modify the program.

137	In G71~G73 commands, the number of Ns~Nf blocks exceeds 100.	In G71~G73 commands, the Ns-Nf blocks are excessive. Modify the program.
138	In G71~G73 commands, the Ns-Nf blocks are non-monotonic	In multiple cycle command (G71 or G72), a non-monotonic object structure is defined; or in G73 cycle, the Z axis is non-monotonic; when the Z axis is set with retract amount or finishing allowance, the X axis is non-monotonic. Please refer to parameter 5102#1 MRC and modify the program.
139	The positioning point commanded by G71~G73 is within the cutting range	When the positioning point commanded by G71~G73 is within the cutting range, tool collision may occur. Please refer to parameter 5104#2 FCK and modify the program.
141	In G73 cycle, the tool retract direction of X axis is inconsistent with the finishing allowance direction.	In G73 cycle, the tool retract direction of X axis is opposite to the finishing allowance direction. Modify the program.
142	In G73 cycle, the tool retract direction of Z axis is inconsistent with the finishing allowance direction.	In G73 cycle, the tool retract direction of Z axis is opposite to the finishing allowance direction. Modify the program.
144	G00 or G01 is not commanded in starting block of the G71~G72 loop.	G00 or G01 needs to be commanded in starting block of the G71~G72 loop. Modify the program.
145	None of G00-G03 is commanded in starting block of the G73 loop	G00, G01, G02 or G03 is not commanded in the starting block of the G73 loop. Modify the program.
146	Only X axis increment is needed in the starting block of G71 loop	X axis is not commanded in the starting block of the G71 loop, or the X axis increment is zero, or Z axis is commanded. Modify the program.
147	Only Z axis increment is needed in the starting block of G72 loop	Z axis is not commanded in the starting block of the G71 loop, or the Z axis increment is zero, or X axis is commanded. Modify the program.
148	The single feeding amount in G71 or G72 command is less than zero	The single feeding amount in G71 or G72 command is less than zero. Modify the program.
149	The single tool retracting amount	The single retracting amount R(e) in G71 or G72

	R(e) in G71 or G72 command is less than zero	command is less than zero. Modify the program.
151	The number of repetition R(d) in G73 command is out of the permitted range	The number of repetition R(d) in G73 command is less than 1 or greater than 000 after rounding. Modify the program.
156	R(e) is less than zero in G74 or G75 command	Single tool retracting amount R(e) is less than zero in G74 or G75 command. Modify the program.
157	R(Δ d) is less than zero in G74 or G75 command	The tool retracting amount R(Δ d) is less than zero in G74 or G75 command when the cutting feed reaches the end point. Modify the program.
161	The repetition number of G76 is less than 1 or greater than 99	The repetition number of G76 is less than 1 or greater than 99. Modify the program.
165	The G76 finishing allowance R(d) exceeds the permitted range	The G76 finishing allowance R(d) is less than a minimum increment. Modify the program.
166	During taper thread cutting commanded by G76, the R value and U value are unmatched	During taper thread cutting commanded by G76, the start point of machining is between the thread start point and thread end point. Modify the program.
167	Thread height P value is not specified in G76 command	Thread height P value is not specified in G76 command. Modify the program.
168	The G76 thread height is less than the finishing allowance or the minimum cutting amount	The G76 thread height is less than the finishing allowance or the minimum cutting amount. Modify the program.
169	The Q value in G76 command is not within the range	The first cutting depth is not defined in G76 command: the Q value is not within the range or not input. Modify the program.
180	Illegal S command in rigid tapping	The S code in rigid tapping is undefined or out of the range. Modify the program.
181	Illegal K command in rigid tapping	The specified repetition number K value is out of the range in rigid tapping. Modify the program.
182	Illegal F command in rigid tapping	The cutting feedrate value is F is out of the range in rigid tapping. Please check the modal G98 and G99 and modify the program.
183	Program error in rigid tapping	The M code and S value is not in the same block in rigid tapping. Modify the program.
184	Illegal axis operation in rigid tapping	A move axis is specified between the M code and G84 command in rigid tapping. Modify the

		program.
185	The spindle cannot perform rigid tapping	In rigid tapping, the spindle is not selected. Modify the parameter.
186	Plane alteration during rigid tapping	During rigid tapping, a non-G18 plane is switched or the rigid tapping is enabled in non-G18 plane. Modify the program.
187	Data error in rigid tapping	The specified distance is too short or too long in rigid tapping. Modify the program.
188	Data repetition in rigid tapping	The same M code or S code is repeated between M code and G84 in rigid tapping. Modify the program.
189	M code repetition in rigid tapping	In rigid tapping, the M code cannot be in the same block with the M code which locks C axis in drilling canned cycle. Modify the program.
190	Servo spindle command occurs in rigid tapping	The increment of the servo spindle occurs in positioning command in rigid tapping. Modify the program.
201	Incorrect command used in macro program	An unusable function is specified in custom macro program. Modify the program.
202	Format error in macro program	There is a format error in <Formula>. Modify the program.
203	Illegal variable number is used in macro program.	A value not defined as variable number is designated in the custom macro. Modify the program.
204	Macro program call repetition	M98, G65 or G66 is called in G66 modal state in the same program. Modify the program.
205	Bracket nesting error	The number of bracket nesting level exceeds 5. Modify the program.
206	Illegal operation data	The argument of SQRT is a negative value; The arguments of BCD and BIN are negative values, or the BIN argument value cannot converted to correct BCD code. Modify the program.
207	Excessive macro program modal call	Macro call or macro program modal call nesting exceeds 4 levels. Modify the program.
208	Branch of macro program cannot used in DNC and MDI operation	Branch of macro program is used in DNC and MDI operation. Modify the program.

209	End statement absent	DO-END is not 1: 1; the END block contains other illegal command or the branch cannot be made to a location within the loop. Modify the program.
210	Limited authority	Argument assignment cannot be executed in MDI or DNC mode due to limited authority. Modify the program.
211	Illegal repetition number	Condition $1 \leq n \leq 3$ is not fulfilled (n in Don). Modify the program.
212	NC statement and macro call statement coexist in the same block	NC statement and macro call statement are used mixedly. Modify the program.
213	Illegal macro sequence number	The defined sequence number in branch command is not within 1~99999, or they cannot be searched. Modify the program.
214	Illegal argument address	An unallowable address is specified in <argument>. Modify the program.
216	Illegal argument value	The argument value is erroneous or illegal. Modify the program.
217	Data error in logical operation command	The data in logical operation command OR, XOR, AND are negative values. Modify the program.
218	G67 modal call cancel is commanded	When G66 macro modal call is not specified, G67 modal call cancel is commanded. Check if it is necessary to write G66 command. Please refer to parameter 6000#0G67 and modify the program.
251	The intersection point cannot be determined in tool nose radius compensation mode	The intersection point cannot be determined in tool nose radius compensation mode. Modify the program.
252	Tool nose radius compensation mode cannot be set or canceled in circular interpolation	Tool nose radius compensation mode is set or canceled in circular interpolation. Modify the program.
253	Compensation plane switching is not allowed in tool nose radius compensation	Compensation plane is changed in tool nose radius compensation mode. Modify the program.
254	Interference is generated in circular block in tool nose radius compensation mode	In tool nose radius compensation mode, the start point or end point of an arc is the same with the center point, or the end point is not on the arc, which

		may cause overcut. Modify the program.
255	In tool nose radius compensation mode, interference occurs in G90 or G94 block	Overcut may occur when tool nose radius compensation is commanded in G90 or G94 block. Modify the program.
256	Overcut occurs during interference check in tool nose radius compensation mode	The overcut may occur in tool nose radius compensation mode. Modify the program.
257	The cutter path direction is different with the programmed path direction in tool nose radius compensation mode.	The tool path direction is different with the programmed path direction in tool nose radius compensation mode (90°~270° difference). Overcut may occur. Modify the program.
258	G41 or G42 execution is not allowed in MDI mode	G41 or G42 (tool nose radius compensation) is specified in MDI mode. Please refer to parameter 5008#4 MCR and modify the program.
259	Overcut is produced within the cutting full circle	Overcut is produced within the cutting full circle in tool nose radius compensation mode. Please refer to parameter 5008#5 CNF and modify the program.
260	Overcut may be produced when a step less than the tool radius is machined	In tool nose radius compensation mode, overcut may be produced when a step less than the tool radius is machined. Please refer to parameter 5008#6 CNS and modify the program.
261	The circular radius is less than the tool radius when a inner circle is machined	In tool nose radius compensation mode, overcut may occur if the circular radius is less than the tool radius when an inner circle is machined. Modify the program.
262	Circular command occurs when tool nose radius compensation is temporarily cancelled or set	In tool nose radius compensation mode, when G command for which the compensation mode needs to be temporarily cancelled is specified, circular command is specified to set or cancel the compensation mode. Modify the program.
263	Error is found in tool nose radius compensation mode	Programming error or operation error is found in tool nose radius compensation mode. Modify the program.

Appendix 2 Standard Ladder Function Allocation

B.1 Standard Machine Panel Key Allocation

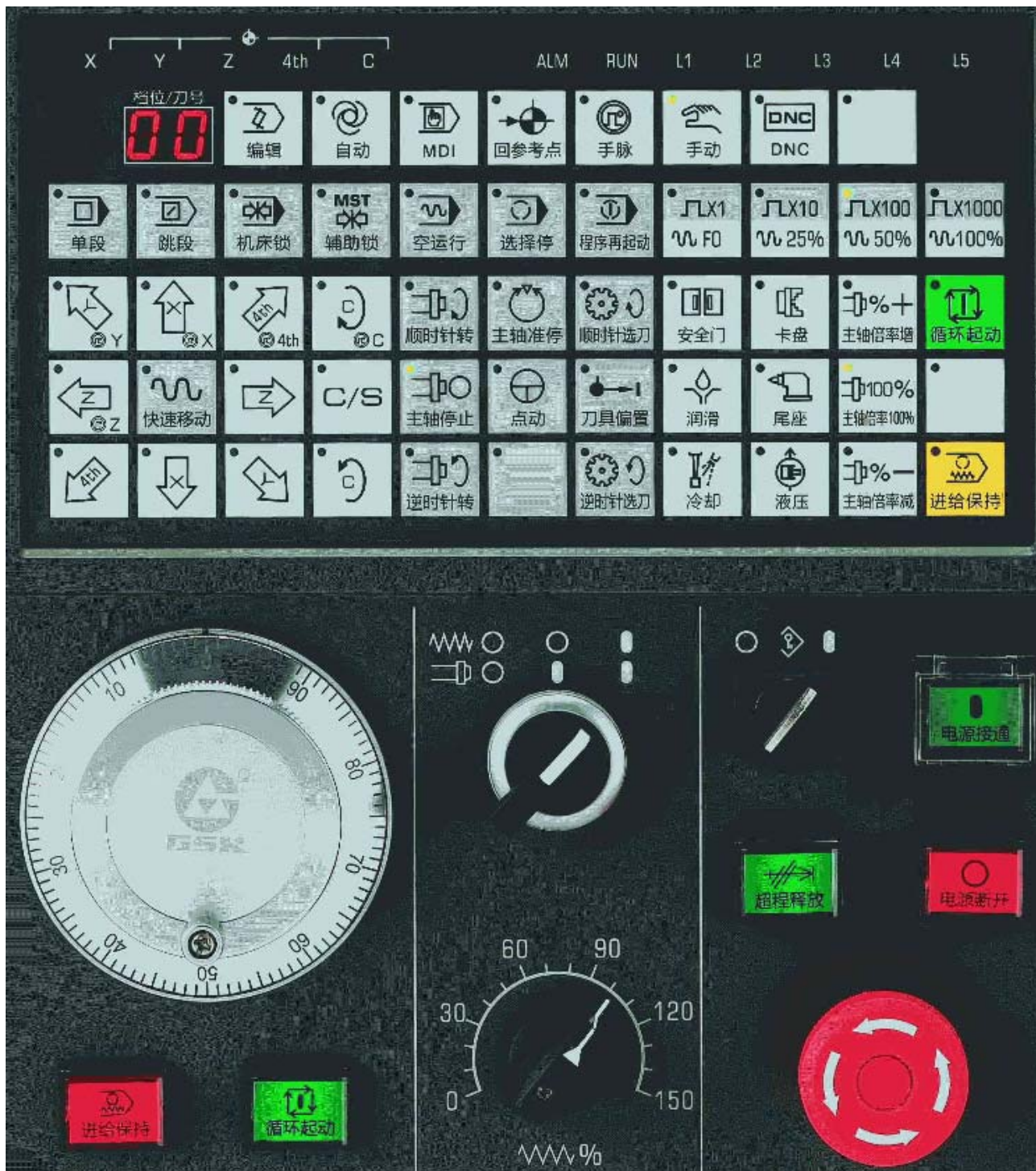


Fig. B-1 Standard machine operation panel layout

B.2 Standard Ladder X, Y Address Definition

B.2.1 General machine IO interface

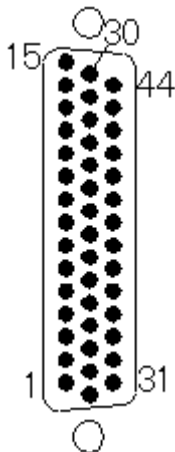


Fig. B-2 CN61(male) input

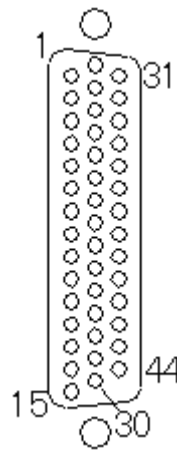


Fig. B-3 CN62(female) output

DB pin	PLC address	PLC Address definition	PLC address definition function	Remark
CN61.1	X0.0	SAGT	Safety door check signal	
CN61.2	X0.1		standby	
CN61.3	X0.2	DIQP	Chuck input signal	
CN61.4	X0.3	DECX	X deceleration signal	Fixed address
CN61.5	X0.4	DITW	tailstock control signal	
CN61.6	X0.5	ESP	Emergency stop input signal	fixed address
CN61.7	X0.6	PRES	Pressure check signal	
CN61.8	X0.7	T05	Tool signal 5/tool post worktable pregraduation signal(Yantai AK31)/ Sensor E (Liuxin Tool post)	
CN61.9	X1.0	T06	Tool signal 6/tool post worktable strobe signal(Yantai AK31) /Sensor F (Liuxin Tool Post)	
CN61.10	X1.1	T07	Tool signal 7/tool post worktable overheat signal(Yantai AK31)	
CN61.11	X1.2	T08	Tool signal 8	

DB pin	PLC address	PLC Address definition	PLC address definition function	Remark
CN61.12	X1.3	DECY	Y deceleration signal	fixed address
CN61.13	X1.4		standby	
CN61.14	X1.5	M41I	Shift to the 1 st gear in-position	
CN61.15	X1.6	M42I	Shift to the 2 nd gear in-position	
CN61.16	X1.7	T01	Tool signal 1/T1(Yantai AK31)/ Sensor A (Liuxin Tool Post)	
CN61.29	X2.0	T02	Tool signal 2/T2(Yantai AK31)/ Sensor B (Liuxin Tool Post)	
CN61.30	X2.1	T03	Tool signal 3/T3(Yantai AK31)/ Sensor C (Liuxin Tool Post)	
CN61.31	X2.2	T04	Tool signal 4/T4(Yantai AK31)/ Sensor D (Liuxin Tool Post)	
CN61.32	X2.3	DECZ	Z deceleration signal	fixed address
CN61.33	X2.4	DEC4	4th deceleration signal	fixed address
CN61.34	X2.5	DEC5	Servo spindle deceleration signal	fixed address
CN61.35	X2.6	TCP	Tool post lock signal	
CN61.36	X2.7	COIN	Spindle positioning completion signal	
CN61.37	X3.0	LMIX+	X positive(+) overtravel signal	
CN61.38	X3.1	LMIY+	Y positive (+)overtravel signal	
CN61.39	X3.2	LMIZ+	Z positive(+) overtravel signal	
CN61.40	X3.3	WQPJ	Chuck in-position signal (outer clamping/inner releasing in-position)	
CN61.41	X3.4	NQPJ	Chuck in-position signal (outer releasing/inner clamping in-position)	
CN61.42	X3.5	SKIP	G31 skip signal	fixed address
CN61.43	X3.6		standby	
CN61.44	X3.7		standby	
CN61.17	X4.0	LMIX-	X negative (-)overtravel signal	
CN61.18	X4.1	LMIY-	Y negative (-)overtravel signal	

DB pin	PLC address	PLC Address definition	PLC address definition function	Remark
CN61.19	X4.2	LMIZ-	Z negative (-)overtravel signal	
CN61.20	X4.3	LMI4+	4 th positive(+) overtravel signal	
CN61.25	X4.4	LMI4-	4 th negative(-) overtravel signal	
CN61.26	X4.5	LMI5+	Positive(+) overtravel signal of servo spindle	
CN61.27	X4.6	LMI5-	Negative(-) overtravel signal of servo spindle	
CN61.28	X4.7		standby	
CN61.21~CN61.24	0V			

CN62.1	Y0.0	M08	Cooling output signal	
CN62.2	Y0.1	M32	Lubricating output signal	
CN62.3	Y0.2		Standby	
CN62.4	Y0.3	M03	Spindle signal (CW)	
CN62.5	Y0.4	M04	Spindle signal (CCW)	
CN62.6	Y0.5	M05	Spindle stop signal	
CN62.7	Y0.6		standby	
CN62.8	Y0.7	SPZD	Spindle brake output signal	
CN62.9	Y1.0	M41	Spindle 1 st output signal	
CN62.10	Y1.1	M42	Spindle 2 nd output signal	
CN62.11	Y1.2	M43	Spindle 3 rd output signal	
CN62.12	Y1.3	M44	Spindle 4 th output signal	
CN62.13	Y1.4	M12(DOQP J)	Outer chuck clamping output signal/inner chuck releasing output	
CN62.14	Y1.5	M13(DOQP S)	Outer chuck releasing output signal/inner chuck clamping output	
CN62.15	Y1.6	TL+	Tool post (CW)output signal	
CN62.16	Y1.7	TL-	Tool post (CCW)output signal	

DB pin	PLC address	PLC Address definition	PLC address definition function	Remark
CN62.29	Y2.0		Tool post worktable motor brake signal (Yantai AK31)/ tool post releasing output (Liuxin tool post)	
CN62.30	Y2.1		Tool post worktable pregraduation electromagnet signal (Yantai AK31)/tool pot clamping output (Liuxin Tool Post)	
CN62.31	Y2.2		standby	
CN62.32	Y2.3		standby	
CN62.33	Y2.4		standby	
CN62.34	Y2.5	M10	Tailstock forward output signal	
CN62.35	Y2.6	M11	Tailstock backward output signal	
CN62.36	Y2.7		standby	
CN62.37	Y3.0		standby	
CN62.38	Y3.1		standby	
CN62.39	Y3.2		standby	
CN62.40	Y3.3		standby	
CN62.41	Y3.4	SORI	Spindle orientation signal	
CN62.42	Y3.5	SEC0	Spindle orientation selection signal 1	
CN62.43	Y3.6	SEC1	Spindle orientation selection signal 2	
CN62.44	Y3.7	SEC2	Spindle orientation selection signal 3	
CN62.17~CN62.19,CN62.26~ CN62.28			0V	
CN62.20~CN62.25			+24V	

B.2.2 Servo spindle interface

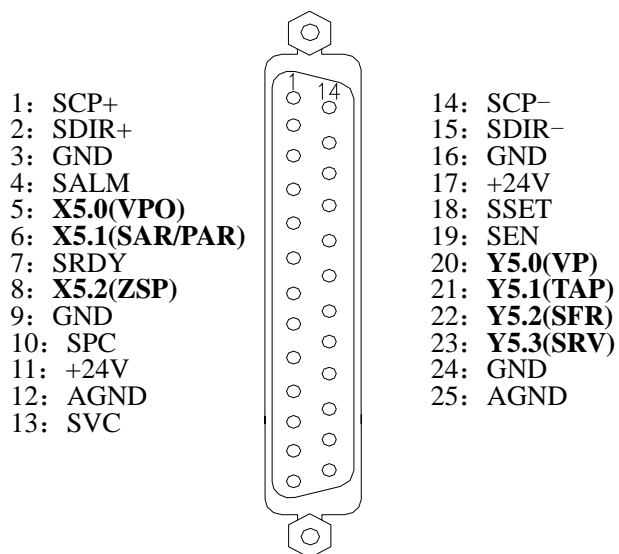


Fig. B-13 CN15 servo spindle interface
(25-core D female socket)

DB pin	Signal definition	Signal explanation	PLC address defined function
CN15.1,CN15.14	SCP+, SCP-	Command pulse signal	/
CN15.2,CN15.15	SDIR+,SDIR-	Command direction signal	/
CN15.4	SALM	Drive unit alarm signal	/
CN15.7	SRDY	Servo ready signal	/
CN15.18	SSET	Pulse forbidden signal	/
CN15.19	SEN	Axis enabling signal	/
CN15.10	SPC	Zero signal	/
CN15.13	SVC	Spindle analog voltage output	/
CN15.12,CN15.25	AGND	Spindle analog voltage output ground	/
CN15.5	X5.0 (VPO)	Address defined by PLC, switching input	Spindle speed/position state signal
CN15.6	X5.1 (SAR/PAR)	Address defined by PLC, switching input	Spindle position/speed arrival signal
CN15.8	X5.2 (ZSP)	Address defined by PLC, switching input	Spindle zero speed output signal
CN15.20	Y5.0 (VP)	Address defined by PLC, switching output	Spindle speed/position switch signal
CN15.21	Y5.1 (TAP)	Address defined by PLC, switching output	Spindle speed loop 2 nd gain selection signal, used to tapping
CN15.22	Y5.2 (SFR)	Address defined by PLC, switching output	Spindle signal(CW)
CN15.23	Y5.3 (SRV)	Address defined by PLC, switching output	Spindle signal(CCW)
CN15.11,CN15.17	+24V	+24V	/
CN15.3,CN15.9, CN15.16,CN15.24	GND	0V (switching signal ground)	/

B.2.3 Standard machine operation panel

1) X address

Address defined by PLC	Corresponding machine panel key	Remark
X18.0	Skip block	
X18.1	Auxiliary function lock	
X18.2	Spindle override increasing	
X18.3	Single block	
X18.4	Machine lock	
X18.5	Dry run	
X18.6	Spindle override decreasing	
X18.7	Spindle override 100%	
X19.0	C negative movement(C-)/MPG C	
X19.1	C/S switch	
X19.2	Cycle start	
X19.3	tailstock	
X19.4	4 th negative movement(4th-)/MPG 4th	
X19.5	Z negative movement (Z-)/ MPG Z	
X19.6	Y negative movement (Y-)/ MPG Y	
X19.7	X negative movement (X-)/ MPG X	
X20.0	Safety door	
X20.1	Tool post CW	
X20.2	Tool offset	
X20.3	Tool post CCW	
X20.4	Cooling	
X20.5	Spindle stop	
X20.6	Manual rapid traverse	
X20.7	Optional stop	
X21.0	Program restart	
X21.1	Spindle CW	
X21.2	Spindle	
X21.3	Spindle CCW	

X21.4	4 th positive movement(4th+)	
X21.5	C positive movement (C+)	
X21.6	Spindle orientation	
X21.7	Feed hold	
X22.0	MPG	
X22.1	DNC right spacebar	
X22.2	Manual mode	
X22.3	MDI mode	
X22.4	DNC mode	
X22.5	Auto mode	
X22.6	Reference point return mode	
X22.7	Edit mode	
X23.0	Rapid override 100%/MPG×1000	
X23.1	Z positive movement (Z+)	
X23.2	Rapid override 50%/ MPG×100	
X23.3	Rapid override 25%/ MPG×10	
X23.4	Y positive movement (Y+)	
X23.5	Rapid override F0/MPG×1	
X23.6	X positive movement (X+)	
X23.7	Hydraulic	
X24.0	Spacebar under cycle start key	
X24.1	Chuck	
X24.2	Lubricating	
X24.3	Spacebar in the right of spindle CW key	
X24.4	Undefined	System reservation
X24.5	Undefined	System reservation
X24.6	Undefined	System reservation
X24.7	Undefined	System reservation
X25.0	Outlet to terminal block	reservation to user
X25.1	Outlet to terminal block	reservation to user
X25.2	Outlet to terminal block	reservation to user
X25.3	Outlet to terminal block	reservation to user
X25.4	Outlet to terminal block	reservation to user
X25.5	Outlet to terminal block	reservation to user

X25.6	Outlet to terminal block	reservation to user
X25.7	Outlet to terminal block	reservation to user
X26.0	Outlet to terminal block	reservation to user
X26.1	Outlet to terminal block	reservation to user
X26.2	Outlet to terminal block	reservation to user
X26.3	Outlet to terminal block	reservation to user
X26.4	Outlet to terminal block	reservation to user
X26.5	Outlet to terminal block	reservation to user
X26.6	Outlet to terminal block	reservation to user
X26.7	Outlet to terminal block	reservation to user
X27.0	Outlet to terminal block	reservation to user
X27.1	Outlet to terminal block	reservation to user
X27.2	Outlet to terminal block	reservation to user
X27.3	Outlet to terminal block	reservation to user
X27.4	Outlet to terminal block	reservation to user
X27.5	Outlet to terminal block	reservation to user
X27.6	Outlet to terminal block	reservation to user
X27.7	Outlet to terminal block	reservation to user
X28.0	Outlet to terminal block	Connecting with panel wave band switch(spindle overrideOV1)
X28.1	Outlet to terminal block	Connecting with panel wave band switch(spindle overrideOV2)
X28.2	Outlet to terminal block	Connecting with panel wave band switch(spindle overrideOV3)
X28.3	Outlet to terminal block	Connecting with panel wave band switch(spindle overrideOV4)
X28.4	Outlet to terminal block	Connecting with panel wave band switch(feedrate overrideOV1)
X28.5	Outlet to terminal block	Connecting with panel wave band switch(feedrate overrideOV2)
X28.6	Outlet to terminal block	Connecting with panel wave band switch(feedrate overrideOV3)
X28.7	Outlet to terminal block	Connecting with panel wave band switch(feedrate overrideOV4)
X29.0	Outlet to terminal block	Connecting with panel button(cycle start)

X29.1	Outlet to terminal block	Connecting with panel button(feed hold)
X29.2	Outlet to terminal block	Connecting with panel key switch button(program protection lock)
X29.3	Outlet to terminal block	Connecting with panel knob normally-open terminal(permitting spindle rotation)
X29.4	Outlet to terminal block	Connecting with panel knob normally-closed terminal (permitting feed)
X29.5	Outlet to terminal block	reservation to user
X29.6	Outlet to terminal block	reservation to user
X29.7	Outlet to terminal block	reservation to user

Note: PLC X18~X24 are the fixed key input addresses on the machine operation panel, and their function definitions are fixed; X25~X29 addresses are outlet to the terminal block in the back of the panel and their functions are defined by the ladder.

2) Y address

Address defined by PLC	Corresponding machine panel input	Remark
Y18.0	Skip key indicator	
Y18.1	Auxiliary lock key indicator	
Y18.2	L5 indicator	
Y18.3	Single key indicator	
Y18.4	Machine lock key indicator	
Y18.5	Dry run key indicator	
Y18.6	C/S switch key indicator	
Y18.7	CS negative(-) key indicator	
Y19.0	C positive movement (C+) key indicator	
Y19.1	4 th positive movement (4th+) key indicator	
Y19.2	Cycle start key indicator	
Y19.3	Feed hold key indicator	
Y19.4	Program restart key indicator	
Y19.5	Optional stop key indicator	
Y19.6	Spindle override (-)key indicator	
Y19.7	Spindle override 100% key indicator	
Y20.0	Spindle override (+) key indicator	

Y20.1	Hydraulic key indicator	
Y20.2	Tailstock key indicator	
Y20.3	Lubricating key indicator	
Y20.4	Safety door key indicator	
Y20.5	Tool post CW key indicator	
Y20.6	Tool offset key indicator	
Y20.7	Tool post CCW key indicator	
Y21.0	Numerical indicator tube(right) output (1)	
Y21.1	Numerical indicator tube(right) output (2)	
Y21.2	Numerical indicator tube(right) output (4)	
Y21.3	Numerical indicator tube(right) output (8)	
Y21.4	Numerical indicator tube(left) output (1)	
Y21.5	Numerical indicator tube(left) output (2)	
Y21.6	Numerical indicator tube(left) output (4)	
Y21.7	Numerical indicator tube(left) output (8)	
Y22.0	MPG mode key indicator	
Y22.1	Spacebar key indicator in the right of DNC	
Y22.2	Manual mode key indicator	
Y22.3	MDI mode key indicator	
Y22.4	DNC mode key indicator	
Y22.5	Auto mode key indicator	
Y22.6	Reference point return mode key indicator	
Y22.7	Edit mode key indicator	
Y23.0	Rapid override 100% key indicator	
Y23.1	Z positive movement (Z+) key indicator	
Y23.2	Rapid override 50% indicator	
Y23.3	Rapid override 25% key indicator	
Y23.4	Y positive movement (Y+)	
Y23.5	Rapid override F0 key indicator	
Y23.6	X positive movement (X+)key indicator	
Y23.7	System alarm indicator ALM	
Y24.0	Cooling key indicator	
Y24.1	Chuck key indicator	
Y24.2	Spacebar indicator in the right of spindle CW key	

Y24.3	Spindle orientation key indicator	
Y24.4	Spindle stop key indicator	
Y24.5	Spindle CCW key indicator	
Y24.6	Spindle JOG key indicator	
Y24.7	Spindle CW key indicator	
Y25.0	4 th negative movement(4th-)key indicator	
Y25.1	Z negative movement (Z-)key indicator	
Y25.2	Y negative movement (Y-)key indicator	
Y25.3	Z machine zero indicator	
Y25.4	Y machine zero indicator	
Y25.5	X machine zero indicator	
Y25.6	X negative movement (X-)key indicator	
Y25.7	Rapid key indicator	
Y26.0	Spacebar indicator under the cycle start key	
Y26.1	L4 indicator	
Y26.2	L3 indicator	
Y26.3	L2 indicator	
Y26.4	L1 indicator	
Y26.5	System run indicator RUN	
Y26.6	C machine zero light	
Y26.7	4th machine zero light	
Y27.0	Outlet to terminal block	reservation to user
Y27.1	Outlet to terminal block	reservation to user
Y27.2	Outlet to terminal block	reservation to user
Y27.3	Outlet to terminal block	reservation to user
Y27.4	Outlet to terminal block	reservation to user
Y27.5	Outlet to terminal block	reservation to user
Y27.6	Outlet to terminal block	reservation to user
Y27.7	Outlet to terminal block	reservation to user
Y28.0	Outlet to terminal block	reservation to user
Y28.1	Outlet to terminal block	reservation to user
Y28.2	Outlet to terminal block	reservation to user
Y28.3	Outlet to terminal block	reservation to user
Y28.4	Outlet to terminal block	reservation to user
Y28.5	Outlet to terminal block	reservation to user

Y28.6	Outlet to terminal block	reservation to user
Y28.7	Outlet to terminal block	reservation to user
Y29.0	Outlet to terminal block	Pressing panel button light(cycle start)
Y29.1	Outlet to terminal block	Pressing panel button light(feed hold)
Y29.2	Outlet to terminal block	reservation to user
Y29.3	Outlet to terminal block	reservation to user
Y29.4	Outlet to terminal block	reservation to user
Y29.5	Outlet to terminal block	reservation to user
Y29.6	Outlet to terminal block	reservation to user
Y29.7	Outlet to terminal block	reservation to user

Note: PLC Y18~Y26 are the fixed key output addresses on the machine operation panel, and their function definitions are fixed; Y27~Y29 addresses are outlet to the terminal block in the back of the panel and their functions are defined by the ladder.

B.3 Standard ladder function

B.3.1 Cycle start and feed hold

There are one group of key and one group of external big button on the standard machine operation panel, used to the cycle start and the feed hold, and their addresses are different.

● Address definition

X0019							BIT2	
X0021	BIT7							
Y0019						BIT3	BIT2	

X19.2: panel cycle start key input

X21.7: panel feed hold key input

Y19.2: panel cycle start key indicator output

Y19.3: panel cycle start key indicator output

X0029							BIT1	BIT0
-------	--	--	--	--	--	--	------	------

Y0029							BIT1	BIT0
-------	--	--	--	--	--	--	------	------

X29.0: external cycle start button input

X29.1: external feed hold button input

Y29.0: external feed hold button output

Y29.1: external cycle start button output

● Control logic

In the course of auto run, one of the feed hold key or the external feed hold button is pressed, the

auto run stops. In the stop state of auto mode, one of them is pressed, the system enters the auto run state.

B.3.2 Feed/spindle hold

● Address definition

X0029				BIT4	BIT3			
-------	--	--	--	------	------	--	--	--

X29.3: permissible input of feed (connect with panel feed/spindle knob)

X29.4: permissible input of spindle rotation (connect with panel feed/spindle knob)

● Control parameter

K0010					KNEN			
-------	--	--	--	--	------	--	--	--

KNEN =1: machine panel feed knob hold function is valid;

=0: machine panel feed knob hold function is invalid;

● Control logic

Feed/spindle hold knob controls the enabling of spindle rotation and the cycle start run;

When the spindle is rotating and the knob rotates to the spindle hold position, the spindle output closes;

When the spindle does not rotate and the knob rotates to the spindle hold position, the spindle cannot start;

When the knob rotates to the feed hold position in auto run, the feed stops and the system prompts "Pause";

In Auto mode, when the knob rotates to the feed hold position and the "Cycle start" button is pressed, the program cannot run.

B.3.3 Program protection lock

- **Address definition**

X0029						BIT2		
-------	--	--	--	--	--	------	--	--

BIT2: program protection signal input

- **Control logic**

When X29.2 is valid, the program switch and the parameter switch can be open;

When X29.2 is invalid, the program switch and the parameter switch cannot be open.

B.3.4 Feedrate override adjustment

- **Address definition**

X0028	BIT7	BIT6	BIT5	BIT4				
-------	------	------	------	------	--	--	--	--

X28.4: feedrate override OV0 signal

X28.5: feedrate override OV1 signal

X28.6: feedrate override OV2 signal

X28.7: feedrate override OV3 signal

- **Control logic**

Use the digital wave switch to tune. The encoding is the binary complement.

B.3.5 spindle override tune

- **Address definition**

X0018	BIT7	BIT6				BIT2		
-------	------	------	--	--	--	------	--	--

X18.2: spindle override+

X18.6: spindle override—

X18.7: spindle override100%

Y0019	BIT7	BIT6						
-------	------	------	--	--	--	--	--	--

Y0020								BIT0
-------	--	--	--	--	--	--	--	------

Y19.6: spindle override—key indicator

Y19.7: spindle override100% key indicator

Y20.0: spindle override+key indicator

- **Relative parameter**

DT0023	spindle override indicator flash period (100-1000ms)
--------	--

- **Control logic**

When the spindle override is more than 100%:

a) When it is less than 120%, the spindle override+key indicator flashes, the flash interval time is set by DT23;

b) When it is equal to 120%, the spindle override+key indicator lights;

When the spindle override is equal to 100%:

The spindle override 100% key indicator lights;

When the spindle override is less than 100%:

a) When it is more than 50%, the spindle override-key indicator flashes, and the flash interval time is set by DT23;

b) When it is equal to 50%, the spindle override-key indicator lights;

Note: when the CNC executes the thread cutting, it forbids the spindle override adjustment.

B.3.6 Spindle CW/CCW control

- **Address definition**

Y0000	SPZD		M5	M4	M3			
-------	------	--	----	----	----	--	--	--

Y0.3: spindle CW (M3)

Y0.4: spindle CCW (M4)

Y0.5: spindle stop (M5)

Y0.7: spindle brake output signal (SPZD)

X0020			BIT5					
-------	--	--	------	--	--	--	--	--

X20.5: spindle stop key

X0021					BIT3		BIT1	
-------	--	--	--	--	------	--	------	--

X21.1: spindle CCW key

X21.3: spindle CW key

Y0024	BIT7		BIT5	BIT4				
-------	------	--	------	------	--	--	--	--

Y24.4: spindle stop indicator

Y24.5: spindle CCW indicator

Y24.7: spindle CW indicator

● Control parameter

K0010							BIT1	
-------	--	--	--	--	--	--	------	--

K10.1 =1: when the system resets, CNC closes M03, M04, M08, M32 output signals.

=0: when the system resets, CNC closes M03, M04, M08, M32 output signals

DT0005	MTIME
--------	-------

M code executive durable time (ms), value range: 100~5000ms

DT0010	SPDDL T
--------	---------

Delay time of M05 and the spindle brake output (ms), value range: 0~10000ms

DT0011	SPZD TIME
--------	-----------

Spindle brake output time. value range: 50~60000ms

● Operation time sequence

Spindle operation time sequence as follows:

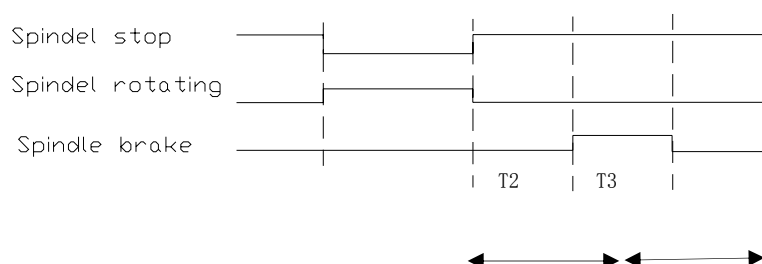


Fig. B-15 spindle CW, CCW time sequence

Note: T2 is the delay time from sending the spindle stop signal to sending the spindle brake signal.

● Control logic

After the CNC is turned on, M05 output is valid.

When ,05 output is valid, M03 or M04 is executed, and its output is valid and remains unchanged, at the time, M05 output is closed;

When M03 or M04 output is valid, M05 is executed, M03 or M04 output is closed, M05 output is valid and remains unchanged;

The spindle brake SPZD signal output delay is set by DT0010, and the brake signal hold time is set by DT0011.

When M03 (or M04) output is valid, the system alarms because M04(or M03) is executed.

Note: When the CNC is in emergency stop, M03 or M04 signal output is closed, at the time, M05 signal is output.

B.3.7 Spindle jog

- **Address definition**




X0024		BIT6						
-------	--	------	--	--	--	--	--	--

BIT6: spindle jog mode signal

- **Control parameter**

DT12	Spindle jog time (ms)
------	-----------------------

- **Control logic**

In Increment, MPG, Manual mode, press  and the system enters the spindle jog mode. Press  and the spindle jogs clockwise; press  and the spindle jogs counterclockwise. The spindle jog time is set by DT12.

B.3.8 Spindle eight-point orientation function

- **Address definition**

Y0003	SEC2	SEC1	SEC0	SORI				
-------	------	------	------	------	--	--	--	--

SORI: spindle orientation signal

SEC0~SEC2: spindle orientation selection signal

X0002	COIN							
-------	------	--	--	--	--	--	--	--

COIN: spindle position completion signal

- **Control logic**

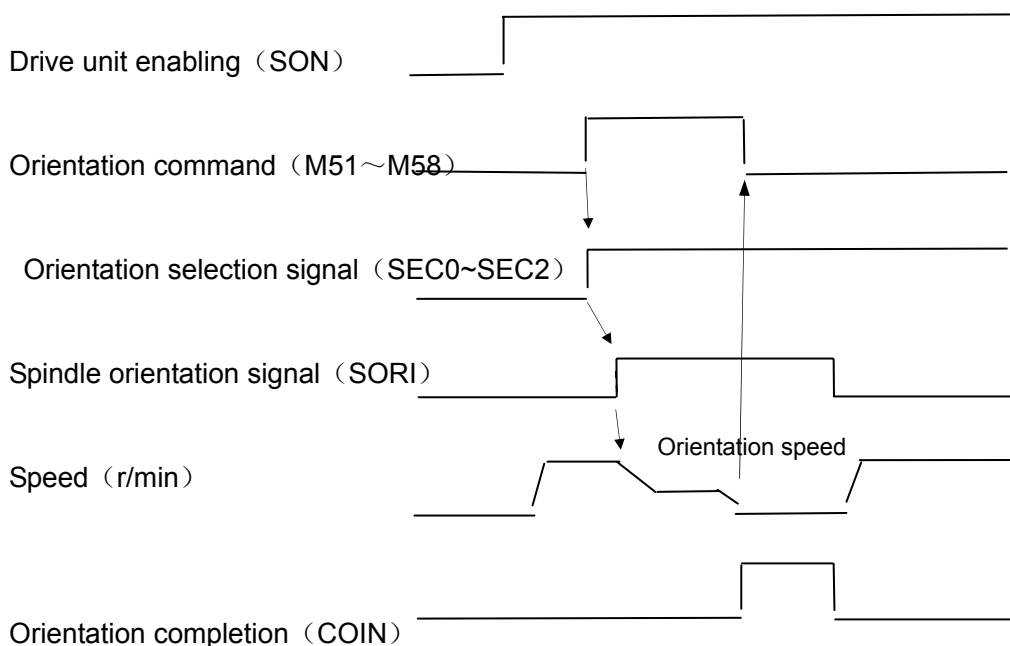
M51~M58 are the spindle orientation commands.

Note: M51~M58 cannot be with movement commands in a same block.

- 1, After M51~M58 are executed, PLC→Drive sends the orientation selection signals SEC0, SEC1, SEC2 to confirm the orientation position;
- 2, After the system delays 40ms, PLC→Drive sends the spindle orientation signal SORI;
- 3, The drive unit starts the orientation;
- 4, After the drive unit orientation is completed, Drive →PLC outputs the spindle orientation completion signal COIN;
- 5, When the system has not received the position completion signal within 6000ms after the PLC has sent the position selection signal, it alarms “the spindle orientation time tool long”.
- 6, The spindle rotates or stops before orientation. The spindle is in the state of stop after the

orientation is completed.

● Control sequence



Corresponding output of M51~M58

Programming command	Output signal		
	SEC2(Y3.7)	SEC1(Y3.6)	SEC0(Y3.5)
M51	0	0	0
M52	0	0	1
M53	0	1	0
M54	0	1	1
M55	1	0	0
M56	1	0	1
M57	1	1	0
M58	1	1	1

B.3.9 Spindle position/speed switch function

● Address definition

Y0005								VP
-------	--	--	--	--	--	--	--	----

VP: speed/position switch signal

X0005								VPO
-------	--	--	--	--	--	--	--	-----

VPO: speed/position state signal

X0019							C/S	
-------	--	--	--	--	--	--	-----	--

C/S: C/S switch

● Control parameter

K0010			VPC					
-------	--	--	-----	--	--	--	--	--

VPC =1: the spindle position/speed switch function is valid

=0: the spindle position/speed switch function is invalid

● Control logic

1. In Manual mode, press C/S key on the panel, PLC→Drive sends the switch signal VP of the spindle speed/position control mode;

2. The drive unit starts the switch control mode, and then Drive→PLC sends the speed/position switch state signal VPO after the switch is completed;

3. After PLC receives the spindle speed/position switch state signal VPO, the spindle contour control switch signal CON (G27.7) is switched, then, the spindle control mode switch is completed.

B.3.10 Spindle speed switching control

● Address definition

Y0001					S04/M4	S03/M4	S02/M4	S01/M4
					4	3	2	1

S01~S04: spindle speed switching control signal;

M41~M44: spindle automatic gear change signal.

● Control parameter

K0010				BIT4				
-------	--	--	--	------	--	--	--	--

K10.4 =1: gear spindle: the spindle speed is controlled by the gear

=0: analog spindle: the spindle speed is controlled by the analog voltage

● Control logic

When K10.4 is set to 1, the spindle speed switching control is valid. When the CNC is turned on, S1~S4 output are invalid.

One of S01,S02,S03,S04 is executed, the corresponding S signal output is valid and remains unchanged, at the time, the output of other three signals are cancelled, i.e. only one of them is valid in the same time.

When S00 is executed, S1~S4 output is cancelled.

B.3.11 Spindle automatic gear change control

● Address definition

Y0001					S04/M4	S03/M4	S02/M4	S01/M4
					4	3	2	1

X0001		M42I	M41I					
-------	--	------	------	--	--	--	--	--

M41~M44: spindle automatic gear output signal

M41I~M42I: spindle automatic gear change to No. 1, 2 gear in-position signal

● Control parameter

K0010				BIT4				
-------	--	--	--	------	--	--	--	--

K10.4 =1: gear spindle: spindle speed is controlled by the gear

=0: analog spindle: the spindle speed is controlled the analog voltage

K0013	AGER	AGIN	AGIM	ASTR				
-------	------	------	------	------	--	--	--	--

AGER =1: the spindle automatic gear change function is valid

=0: the spindle automatic gear change function is invalid

AGIN =1: check the gear in-position signal when the spindle is in the automatic gear change

=0: do not check the gear in-position signal when the spindle is in the automatic gear change

AGIM =1: it is valid when the gear change in-position signal is not connected with +24V.

=0: it is valid when the gear change in-position signal is connected with +24V.

ASTR =1: the spindle gear power-down memorizes

=0: the spindle gear power-down does not memorize

3	7	4	1
3	7	4	2
3	7	4	3
3	7	4	4

MSG1
MSG2
MSG3
MSG4

MSG1,MSG2,MSG3,MSG4: when the spindle analog voltage output is 10V, they separately

correspond max. spindle speed of No. 1, 2, 3, 4 gear. When the spindle automatic gear change is valid, they separately correspond to the spindle speed of M41,M42,M43,M44. when the spindle gear power-down does not memorize, and the system is turned on, the No. 1 gear speed is the default value.

DT0000	SFT1TME
--------	---------

Automatic gear change signal outputs the delay time 1

DT0001	SFT2TME
--------	---------

Automatic gear change signal outputs the delay time 2

● Function description

When the automatic gear change function is used to control the automatic switch spindle mechanical gear. When the system executes S____, it counts the analog voltage to output to the spindle servo or inverter based on the parameters (M41~M44 separately corresponds NO.3741~NO.3744) corresponding to the gear controlled by the current M4n, which makes the actual spindle speed be consistent with S command speed.

B.3.12 Cooling control

● Address definition

Y0000								M08
-------	--	--	--	--	--	--	--	-----

Y0.0: cooling signal output (M08)

X0020				BIT4				
-------	--	--	--	------	--	--	--	--

X20.4: cooling key input

Y0024								BIT0
-------	--	--	--	--	--	--	--	------


Y24.0:cooling key indicator

● Functional description

After the CNC is turned on, M09 is valid, i.e. M08 output is invalid.

M08 is executed, M08 output is valid and the cooling is ON;

M09 is executed, M08 output is cancelled and the cooling is OFF.

When  key on the machine panel is pressed, M08 output state is changed in two-way alternation.

Note 1: when the system is in the emergency stop, M08 output is cancelled;

Note 2: when the system is in reset, K10 Bit1 sets whether M08 output is cancelled:

Bit1=0: when the system is in reset, M08 output is cancelled;

=1: when the system is in reset, M08 output state remains unchanged.

Note 3: M09 has no corresponding output signal, M09 is executed and M08 output is cancelled.

B.3.13 Lubricating control

● Address definition

Y0000							M32	
-------	--	--	--	--	--	--	-----	--

Y0.1: lubricating output signal (M32)

X0024						BIT2		
-------	--	--	--	--	--	------	--	--

X24.2: lubricating key

Y0020					BIT3			
-------	--	--	--	--	------	--	--	--

Y20.3: lubricating key indicator

DT0013	Manual lubricating output time
--------	--------------------------------

Lubricating starting time (0-60000ms); when it is set to 0, the lubricating output remains.

DT0016	Automatic lubricating interval time
--------	-------------------------------------

Automatic lubricating interval time (0-60000ms)


DT0017	Automatic lubricating output time
--------	-----------------------------------

Automatic lubricating output time (0-60000ms)

● Function description

The lubricating function of GSK988T PLC includes non-automatic lubricating and automatic lubricating. When DT16 = 0 or DT17 = 0, the automatic lubricating function is invalid.



a) Non-automatic lubricating

DT13>0: the lubricating is output at regular time,  on the panel is valid or when M32 is executed, the lubricating Y0.1 output is valid, at the time, the light signal Y20.3 output is valid, the lubricating Y0.1 and Y20.3 output is cancelled after the time set by DT13; when the time has not reached the one set by DT13 and M33 is executed, Y0.1 and Y20.3 output is cancelled.

DT13=0: the lubricating outputs in two-way alternation,  on the panel is valid or when

M32 is executed, Y20.3 output is valid; when  is valid again or M33 is excluded, Y0.1 output is closed, Y20.3 is closed.

b) Automatic lubricating

DT16 > 0, DT17>0: after the system is turned on, it counts the time set by DT16 and then the lubricating is output within the time set by DT17, and then is stopped, the lubricating is executed circularly. In automatic lubricating, when the system is in the lubricating interval time,  key on the panel and M32, M33 are valid;  key on the panel and M32, M33 are invalid when the system is in lubricating output time.

B.3.14 Chuck control

● Address definition

Y0001			DOQPS	DOQPJ				
-------	--	--	-------	-------	--	--	--	--

Y1.4: outer chuck clamping output signal/inner chuck releasing output

Y1.5: outer chuck releasing output signal/inner chuck clamping output

X0000						DIQP		
-------	--	--	--	--	--	------	--	--

X0.2: chuck control input signal (DIQP)

X0003				NQPS	WQPJ			
-------	--	--	--	------	------	--	--	--

X3.3: outer chuck clamping in-position signal/inner chuck releasing in-position (WQPJ)

X3.4: outer chuck releasing in-position signal/inner chuck clamping in-position (NQPJ)

X0024							BIT1	
-------	--	--	--	--	--	--	------	--

X24.1: chuck key

Y0024							BIT1	
-------	--	--	--	--	--	--	------	--

Y24.1: chuck key indicator

● Control parameter

K0013							SLSP	SLQP
-------	--	--	--	--	--	--	------	------

K13.0= 1: chuck control function is valid;

0: chuck control function is invalid.

K13.1= 1: when the chuck function is valid, the system checks whether the chuck is clamped;

0: when the chuck function is valid, the system does not check whether the chuck is clamped; when the chuck does not clamp, the spindle cannot be started.

K0014						PB2		PB1
-------	--	--	--	--	--	-----	--	-----

K14.0= 1: check the chuck in-position signal;

0: do not check the chuck in-position signal.


K14.2= 0: in outer chuck mode, WQPJ is the outer chuck clamping signal, NQPJ is the outer chuck releasing signal;

1: in inner chuck mode, NQPJ is the outer chuck clamping signal, WQPJ is the outer chuck releasing signal.

● Control logic

Signal in outer chuck mode	Signal in inner chuck mode
Chuck clamping	Chuck clamping
WQPJ (X3.3): chuck clamping in-position signal	NQPJ (X3.4): chuck clamping in-position signal
DOQPJ (Y1.4): chuck clamping output signal	DOQPS (Y1.5): chuck clamping output signal
Chuck releasing	Chuck releasing
NQPJ (X3.4): chuck releasing in-position signal	WQPJ (X3.3): chuck releasing in-position signal
DOQPS (Y1.5): chuck releasing output signal	DOQPJ (Y1.4): chuck releasing output signal

When the system is turned on, DOQPJ (outer chuck clamping/inner chuck releasing) and DOQPS(outer chuck releasing/inner chuck clamping) signals remain the previous states of the last power-off, i.e. DOQPJ and DOQPS signals have the power-down memory function.

When the chuck control input (DIQP) is valid or the chuck key  on the panel is pressed, the chuck clamping/releasing signal outputs alternatively, i.e. the output state changes once when the chuck control input signal each time is valid.

When the spindle rotates, DIQP input is invalid and the chuck key is disabled; when executing M13 is disabled, the system alarms and its output state remains unchanged.

When the system is in reset or emergency stop state, , DOQPJ,DOQPS output state remains unchanged.

● Sequence chart :

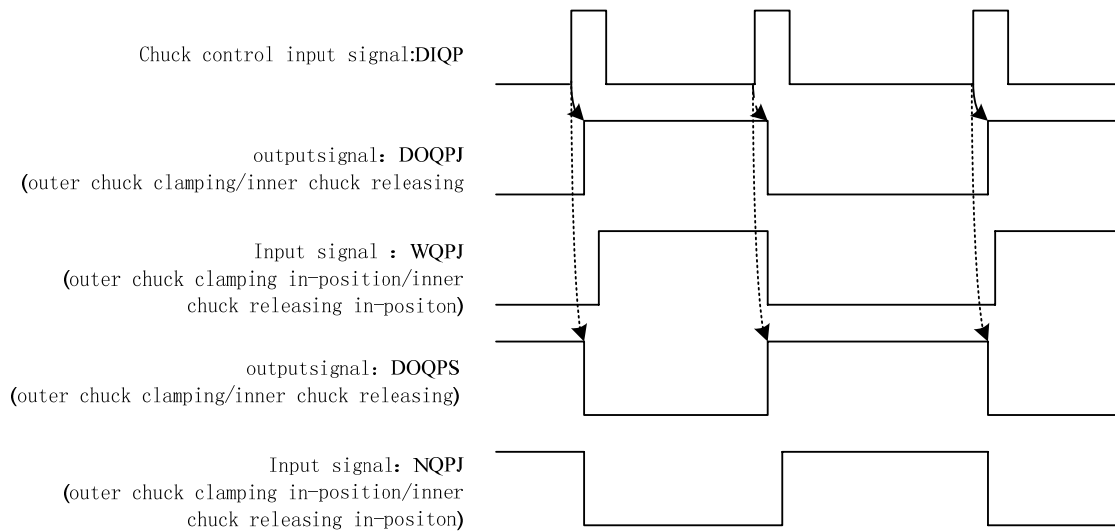


Fig. B-16 chuck control sequence

The system checks whether the chuck clamps (K13.1) and the logic requirements of the chuck in-position signal (K14.0).

K13.1=1, K14.0=0:

When the chuck clamping signal outputs, the spindle can be started, otherwise, the system alarms.

K13.1=1, K14.0=1:

When the chuck clamps and the in-position signal is valid, the spindle can be started, otherwise, the system alarms.

K13.1=0, K14.0=0:

No matter what the chuck clamps or not , the spindle can be started.

K13.1=0, K14.0=1:

When the chuck clamping in-position signal is valid, the spindle can be started, otherwise, the system alarms.

B.3.15 Tailstock control

● Address definition

Y0002		M11	M10					
-------	--	-----	-----	--	--	--	--	--

Y2.5: tailstock forward output signal (DOTWJ);

Y2.6: tailstock backward output signal (DOTWS)

X0000				DITW				
-------	--	--	--	------	--	--	--	--

X0.4: tailstock control input signal

X0019					BIT3			
-------	--	--	--	--	------	--	--	--

X19.3: tailstock key on the panel

Y0020					BIT2			
-------	--	--	--	--	------	--	--	--

Y20.2: tailstock key indicator on the panel

● Control parameter

K0013					SLTW			
-------	--	--	--	--	------	--	--	--

K13.2 =1: the tailstock control function is valid.

=0: the tailstock control function is invalid.

● Operation sequence

Tailstock control sequence is shown in Fig. 7:

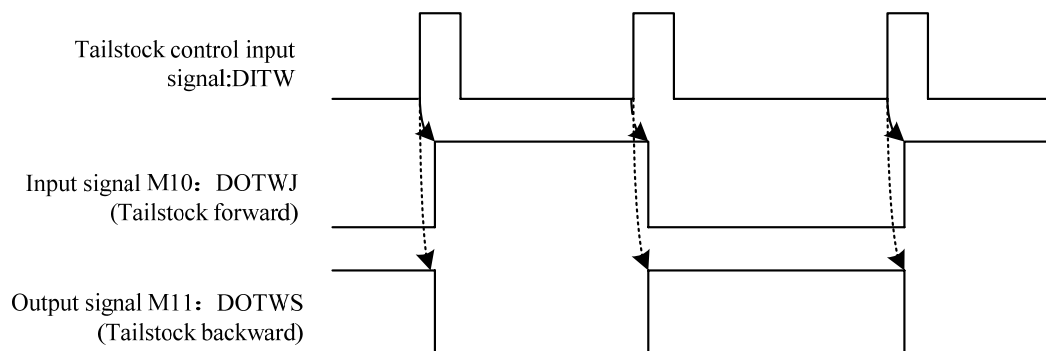



Fig. B-17 Time sequence of tailstock control

When the system is turned on, DOTWJ and DOTWS signals keep the previous states; i.e. DOTWJ and DOTWS signals has the power-down memory function.

When the tailstock control input (DITW) is valid or pressing  is valid on the panel's tailstock, the tailstock forward/backward signal outputs alternately, i.e. when the tailstock control input signal each time is valid, the output state changes once.

After M10 is executed, DOTWJ signal outputs and the tailstock goes forward; after M11 is executed, DOTWS signal outputs and the tailstock retreats.

When the spindle rotates, the tailstock control DITW input and the tailstock key in the panel are invalid; the system alarms when executing M11 is disabled, its output state remains unchanged.

When the system is in reset or emergency stop state, DOTWJ/DOTWS output state remains unchanged.

B.3.16 Pressure low check

- **Address definition**

X0000		PRES						
-------	--	------	--	--	--	--	--	--

X0.6: pressure low alarm check signal (PRES)

- **Control parameter**

K0014			BIT5	BIT4				
-------	--	--	------	------	--	--	--	--

K14.4 =0: alarm for the HIGH. When PRES is connected with +24V, the system alarms for the pressure low.

=1: alarm for LOW. When PRES is not connected with +24V, the system alarms for the pressure low.

K14.5 =0: pressure low check function is invalid.

=1: pressure low check function is valid.

DT0002	Checking time on alarm for pressure low
--------	---

Wait time before alarm for pressure low (0-60000ms)

- **Functional description**

After the system selects the pressure low alarm check function, it finds the pressure low alarm signal PRES is valid, it alarms after it waits in the time set by DT0002, at the time, the feed axis pauses, the spindle stops, and the automatic cycle cannot be started, and then the reset key or power-off is executed to cancel the alarm.

B.3.17 Each axis overtravel signal

- **Address definition**

X0003						LMIZ+	LMIY+	LMIX+
-------	--	--	--	--	--	-------	-------	-------

X0004		LMI5-	LMI5+	LMI4-	LMI4+	LMIZ-	LMIY-	LMIX-
-------	--	-------	-------	-------	-------	-------	-------	-------

X3.0: X positive overtravel signal

X4.0: X negative overtravel signal

X3.1: Y positive overtravel signal

X4.1: Y negative overtravel signal

X3.2: Z positive overtravel signal

X4.2: Z negative overtravel signal

X4.3: the 4th axis positive overtravel signal

X4.4: the 4th axis negative overtravel signal

X4.5: the 5th axis positive overtravel signal

X4.6: the 5th axis negative overtravel signal

● Control parameter

K0010						BIT2		
-------	--	--	--	--	--	------	--	--

K10.2 =1: overtravel signal LOW of each axis is valid;

=0: overtravel signal HIGH of each axis is valid.

B.3.18 Tool change control

● Control parameter

K0011	BIT7	BIT6	BIT5	BIT4	BIT3	BIT2	BIT1	BIT0
-------	------	------	------	------	------	------	------	------

K11.0 1: tool post locking signal is LOW;

0: tool post locking signal is HIGH.

K11.1 1: tool signal of the tool post is LOW;

0: tool signal of the tool post is HIGH.

K11.3 1: signal for checking the tool after the tool change completion;

0: signal for not checking the tool after the tool change completion.

K11.4 1: signal for checking tool post locking;

0: signal for not checking tool post locking.

DT0007	Delay time from tool post stopping CW rotation to the rotation CCW output (0-1000ms)
--------	--

DT0008	Alarm time for not having received the tool post locking TCP signal (0-1000ms)
--------	--

DT0009	Locking time of tool post rotation CCW (0-1000ms)
--------	---

The standard ladder diagram supports three kinds of tool post control logic; setting the corresponding control bit of K parameter selects optional tool post. Composition of Bit 7 and Bit8, and Bit 2 select the tool change mode.

K0011	BIT7	BIT6				BIT2		
-------	------	------	--	--	--	------	--	--

K11.6=0, K11.7=0: standard tool change mode (K11.2 selects the tool change mode A or B)

K11.2=1: tool change mode A (optional to Jingcheng Tool Post).

K11.2=0: tool change mode B (optional to Changzhou Tool Post)

K11.6=1, K11.7=0: optional to Yantai tool post AK31 series(8,10,12-tool).

K11.6=0, K11.7=1: optional to Liuxin 8-tool hydraulic tool post in Taiwan

● Address definition

BIT7	BIT6	BIT2	Tool post type	Required address corresponding to the tool post
0	0	1	Tool change mode A	X1.7 (T1) ,X2.0 (T2) ,X2.1 (T3) ,X2.2 (T4) ,X0.7 (T5),X1.0(T6),X1.1(T7),X1.2(T8),X2.6(TCP),Y1.6 (TL+) ,Y1.7 (TL-)
0	0	0	Tool change mode B	
0	1	/	Yantai tool post AK31 series (8,10,12-tool)	X1.7 (T1) ,X2.0 (T2) ,X2.1 (T3) ,X2.2 (T4) ,X2.6 (TCP) ,X1.0 (tool post worktable pregraduation proximity switch) ,X0.7 (tool post worktable strobe signal),X1.1(tool post overheat check),Y1.6(TL+),Y1.7 (TL-) ,Y2.0 (TZD tool post worktable brake) ,Y2.1 (tool post worktable pregraduation electromagnet)
1	0	/	Liuxin hydraulic tool post LS120 (8-tool)	X1.7 (T1) ,X2.0 (T2) ,X2.1 (T3) ,X2.2 (T4) ,X0.7 (sensor for tool pot stopping rotation and locking) ,X1.0 (sensor for tool pot releasing\locking output) ,Y2.0 (tool releasing output) ,Y2.1 (tool pot locking output),Y1.6(tool pot rotation CW output),Y1.7 (tool pot rotation CCW output)

● Control logic

a) K11.7=0, K11.6=0, K11.2=1: tool change mode A

- ① In Manual, MDI or Auto mode, when the tool change is executed, CNC outputs the tool post rotation(CW) signal (TL+), and checks the tool signal, and then, after it has checks it to close the rotation signal (TL+) , it checks whether the tool signal changes, if change, it outputs the tool post rotation (CCW) signal (TL-), and then it delays the time set by DT009, it closes the signal (TL-).
- ② When K0011 Bit4 is set to 1 (checking the locking signal), the system checks the tool post locking signal, it alarms when it has not received TCP signal in the time set by DT008.
- ③ When K0011 Bit3 is set to 1 (checking the tool signal after tool change completion), the system confirms whether the current tool input signal is consistent with the current tool number after the tool post rotation(CCW) ends, if not, it alarms.
- ④ Tool change ends.

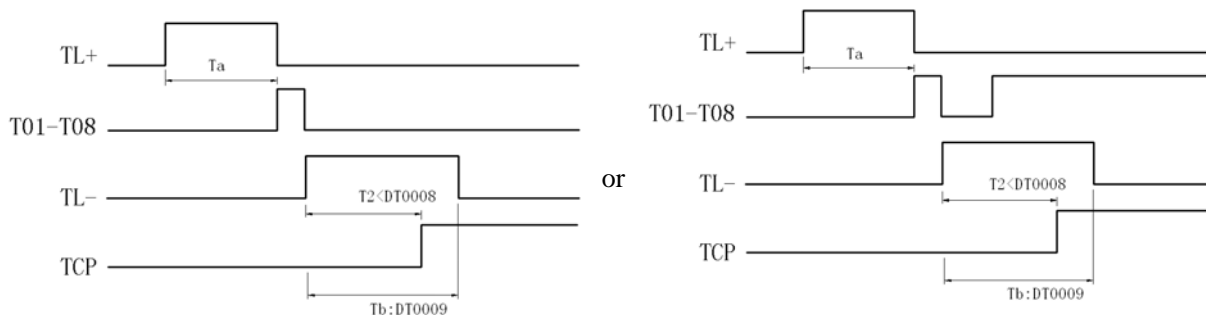


Fig. B-18 Time sequence of tool change mode

b) K11.7=0, K11.6=0, K11.2=0: tool change mode B

- ① When the tool change is executed, CNC outputs the tool post rotation(CW) signal (TL+), and checks the tool signal, and then, after it has checks it to close TL +, it checks whether the tool signal changes, if change, it outputs the tool post rotation (CCW) signal (TL-), and then it delays the time set by DT007, it closes the signal (TL-).
- ② When K0011 Bit4 is set to 1 (checking the locking signal), the system checks the tool post locking signal, it alarms when it has not received TCP signal in the time set by DT008.
- ③ When K0011 Bit3 is set to 1 (checking the tool signal after tool change completion), the system confirms whether the current tool input signal is consistent with the current tool number after the tool post rotation(CCW) ends, if not, it alarms.
- ④ Tool change ends.

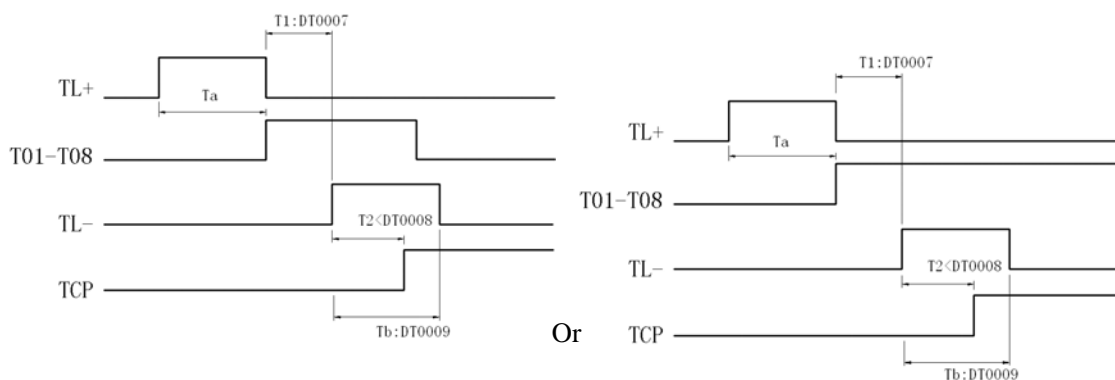


Fig. B-19 Time sequence of tool change mode B

c) K11.7=0, K11.6=1: Yantai tool post AK31 series

1) Tool change process

- ① Ensure the tool post workpiece brake signal TZD is OFF.

- ② The system executes the shortest path judgement according to the target tool number and the current tool number, selects the output rotation direction based on the neighboring selecting tool, and outputs the forward direction signal (TL+) or reverse direction signal (TL-), and then the tool workpiece rotates to select the tool.
- ③ During the rotation, the system encodes according to the tool encoding signal T1~T4 input and identifies the current tool number. When the tool post rotates to the previous tool of the target tool, and starts to check the strobe signal change of the tool post worktable. The strobe signal of the previous tool before the target tool falls edge, the system outputs the pregraduation electromagnetic signal of the tool post worktable.
- ④ When the pregraduation proximity input signal of the tool post worktable is HIGH, the system closes the tool post rotation output signal (TL+ or TL-) and then the motor stops.
- ⑤ The system delays 50ms, and outputs the direction signal(TL+ or TL-) opposite to the previous output direction, and the tool post worktable rotates inversely.
- ⑥ When the proximity input signal of the tool post worktable locking is HIGH, the system closes the rotation signal (TL—,TL+), and the motor stops. Then, the system outputs the brake signal TZD of the worktable and the motor brake device is ON.
- ⑦ The system delays 200ms, and closes the output signal of the electromagnet, and then the electromagnet is OFF.
- ⑧ The system confirms again the current tool number to confirm whether the current tool encoding signal is consistent with the target tool number.
- ⑨ The system confirms again whether the locking proximity switch signal is HIGH.
- ⑩ When the above steps are correct, the system closes the tool post worktable brake signal TZD, and the tool change is completed.
- During the tool change, when the system has checked the motor overheating , it alarms to close the output of all signals.

2) Tool change flow chart

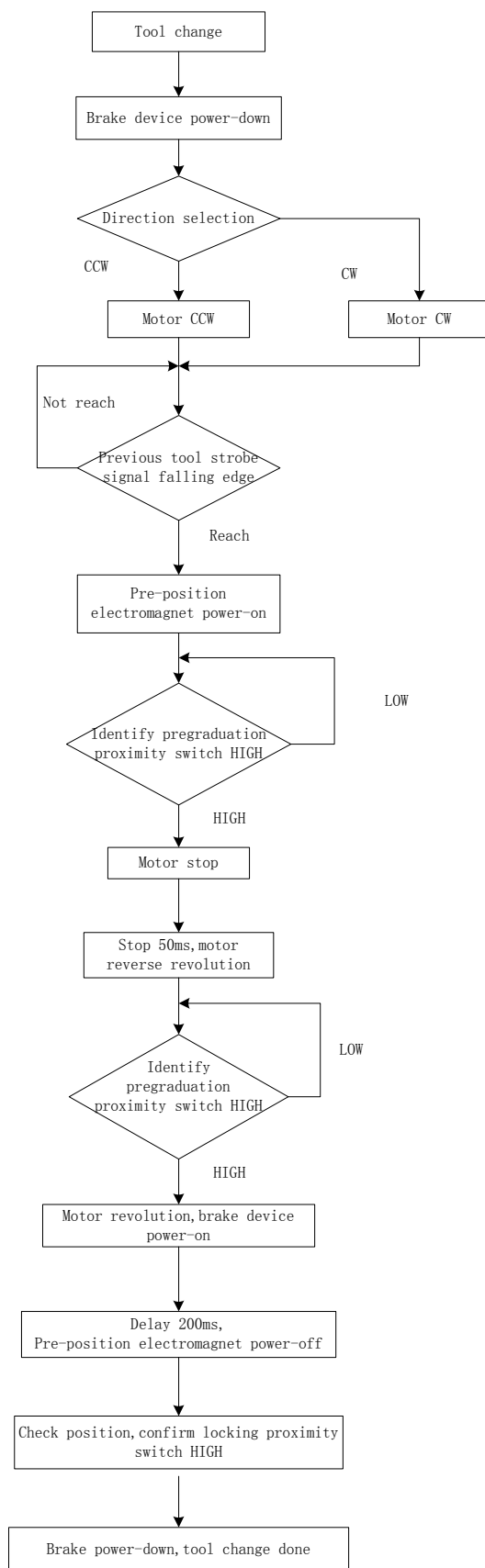


Fig. B-20 tool change flow chart of AK31 too post

c) K11.7=1, K11.6=0: Liuxin 8-tool hydraulic tool post**1) input and output allocation**

Sensor A: tool check sensor ———→T1 (X1.7): tool signal

Sensor B: tool check sensor ———→T2 (X2.0): tool signal

Sensor C: tool check sensor ———→T3 (X2.1): tool signal

Sensor D: tool check sensor ———→T4 (X2.2): tool signal

Sensor E: tool pot stopping and being locked ———→SSE (X0.7): signal for tool pot stopping and being locked

Sensor F: too pot releasing/locking ———→SSF(X1.0): signal for tool pot releasing/locking

Sol A: electromagnetic valve for tool pot releasing \ locking ———→ Y3.0: tool pot releasing output Y3.1: tool pot locking output

Sol B: electromagnetic valve for tool pot rotation(CW/CCW)

———→TL+ (Y1.6): tool pot output (CW)

———→TL- (Y1.7): tool pot output(CCW)

2) corresponding table between position and signal

	1	2	3	4	5	6	7	8
A			●		●	●	●	
B	●				●		●	●
C				●	●	●		●
D		●				●	●	●
E	●	●	●	●	●	●	●	●

3) signal explanation

Sensor A,B,C,D: are only used to the tool check instead of be starting signal of any operations

Sensor E: it is inductive once after the system executes one tool change, and it is the starting signal of the tool pot stopping rotation and being clocked. When the tool pot rotates to the required in-position, Sensor E is inductive, i.e., the electromagnetic valve to control the tool pot rotation is turned off, which makes the tool pot stops rotation to start the electromagnetic valve to lock the tool pot.

Sensor F: releasing/clamping confirmation signal. When Sensor F is not inductive, i.e., the tool pot has been released, the tool pot is started to rotated; when the tool pot is inductive, i.e., the tool pot is locked, i.e., the tool change operation is completed.

Sol A: control tool pot releasing/clamping

Sol B: control tool pot rotation(CW/CCW)

4) Tool change process

Example: the system switches No. 1 tool into No. 4 tool

Step 1: Sol A is turned on (tool post is released)

Step 2: Sensor F is not inductive, Sol B is turned on and the hydraulic motor rotates

Step 3: The system checks the tool signal (note: Sensor E is inductive in No. 1, 2, 3 tool, its locking operation does not execute when it does not reach No. 4 tool). When there is No. 3 tool signal, Sensor E ready operation should be set and the tool pot should rotate to No. 4 tool and Sensor E is inductive, i.e. Sol B is turned off, the tool pot stops rotation and Sol A is controlled to make the tool pot is locked.

B.3.19 Emergency stop control

● Address definition

X0000			ESP					
-------	--	--	-----	--	--	--	--	--

ESP: emergency stop input signal

● Control parameter

K0010	ESP							
-------	-----	--	--	--	--	--	--	--

ESP =1: alarm for external emergency stop input signal(X0.5) HIGH

=0: alarm for external emergency stop input signal(X0.5) LOW

3	0	0	9	ESP						
---	---	---	---	-----	--	--	--	--	--	--

ESP =1: emergency stop alarm when the external emergency stop signal(X0.5) is 1

=0: emergency stop alarm when the external emergency stop signal(X0.5) is 0

Note: K10.7 must be consistent with that of No.3003#7.

B.4 Parameter explanation of ladder

B.4.1 K parameter

Address	Parameter meaning	Initial value
K0	Working mode memory	0
K1	spindle override memory	0
K2.7	Chuck state memory (1: clamping, 0: releasing)	0

K3.4, K3.5	MPG and increment override	0
K3.6, K3.7	Spindle gear encoding	0
K4	Tool memory	0
K5.0~K5.2	MPG axis selection encoding	0
K8.0	X manual movement key direction (1: reverse, 0: not)	0
K8.1	Y manual movement key direction (1: reverse, 0: not)	0
K8.2	Z manual movement key direction (1: reverse, 0: not)	0
K8.3	The 4 th axis manual movement key direction (1: reverse, 0: not)	0
K8.4	C axis manual movement key direction (1: reverse, 0: not))	0
K9.0	Shield program protection lock (1: shield, 0: not)	0
K10.0	Feedrate override reverses (1:reverse, 0:not)	0
K10.1	Close the spindle, cooling, lubricating in reset (1: keep, 0: close)	0
K10.2	Overtravel signal of each axis LOW/HIGH (1:LOW, 0: HIGH)	0
K10.3	Feed knob valid/invalid on machine panel (1:valid, 0:invalid)	1
K10.4	Spindle type(1:gear, 0:analog)	0
K10.5	Spindle position/speed switch function (0: invalid 1: valid)	0
K10.7	External emergency stop input signal (X0.5)(1: HIGH alarm 0: LOW alarm)	1
K11.0	Tool post locked signal level selection (1: LOW, 0: HIGH)	0
K11.1	Tool signal level selection (1:LOW, 0:HIGH)	0
K11.2	Tool change mode (1: A mode, 0: B mode)	1
K11.3	Check tool signal after completing the tool change (1: check, 0: not)	0
K11.4	Check tool post locking signal (1: check, 0: not)	1
K11.5	Spindle function and tailstock (1:interlock, 0: not)	0
K11.6	Tool post selection(PB8 PB7: 00 standard tool post/10 Yantai tool post/01 Liuxin tool post)	0
K11.7	Tool post selection(PB8 PB7: 00 standard tool post/10 Yantai tool post/01 Liuxin tool post)	0
K12.0	1/0: manually reverse tool change is valid/invalid	0
K12.2	Zero return operation direction key is self-locking (1:self-locking, 0: not)	0
K13.0	Chuck control function (1:valid, 0:invalid)	0
K13.1	Check the chuck clamping when the chuck function is enabled (1:check, 0: not)	0

K13.2	Tailstock control function (1:valid, 0:invalid)	0
K13.3	Allowable input of spindle rotation (1:invlaid, 0:invalid)	1
K13.4	Whether analog spindle gear power-down is memorized (1:memory, 0:not)	1
K13.5	Gear in-position is valid (1:LOW, 0:HIGH)	0
K13.6	Spindle gear in-position signal (1:check, 0: not)	0
K13.7	Automatic gear change function of the analog spindle (1: valid, 0: invalid)	0
K14.0	Chuck clamping/releasing in-position signal (1: check, 0: not)	0
K14.2	Chuck inner/outer mode (1:outer, 0:inner)	0
K14.4	Alarm level of pressure low (1:LOW, 0:HIGH)	0
K14.5	Pressure low check function (1:valid, 0:invalid)	0
K14.6	Input signal level of protection door closed (1:LOW, 0:HIGH)	0
K14.7	Protection door check function (1:valid, 0:invalid)	0
K15.0	Operation mode MD1 in starting up	0
K15.1	Operation mode MD2 in starting up	0
K15.2	Operation mode MD4 in starting up	0
K15.4	Operation mode in starting up (1:MD1, MD2, MD4; 0: the previous mode in the last starting up)	0

B.4.2 DT parameter

DT address	Initial value of ladder	Input min. value	Input max. value	Parameter meaning
DT0	1000	0	60000	Spindle gear change time 1(ms)
DT1	1000	0	60000	Spindle gear change time 2(ms)
DT2	3000	0	60000	Pressure low alarm check time(ms)
DT3	5000	100	5000	The upper time to move one tool in tool change (ms)
DT4	15000	1000	60000	The upper time to move the most tool in tool change (ms)
DT5	500	100	5000	M execution durable time(ms)
DT6	500	100	5000	S execution durable time (ms)
DT7	1000	0	4000	Delay time from tool post stopping rotation(CW)to rotation(CCW) output (ms)

DT8	500	0	4000	Alarm time for having not received the tool post locked signal *TCP (ms)
DT9	1000	0	4000	Locked time of tool post rotation(CW) (ms)
DT10	0	0	10000	Delay time between M05 and spindle brake output (ms)
DT11	50	0	60000	Output time of spindle brake (ms)
DT12	3000	0	60000	Spindle jog time(ms)
DT13	0	0	60000	Lubricating start time (0-60000ms)(0:lubricating)
DT16	0	0	60000	Automatic lubricating interval time (ms)
DT17	0	0	60000	Automatic lubricating output time (ms)
DT18	0	0	60000	Chuck clamping/releasing pulse output width (ms)
DT19	1000	200	60000	Chuck function execution time for not checking in-position signal (ms)
DT21	1000	100	60000	Spindle stop, chuck operation enabling delay (ms)
DT22	1000	100	1000	Alarm light flash period (100 - 1000) (ms)
DT23	1000	100	1000	spindle override flash period (100 - 1000) (ms)
DT24	100	0	2000	Feedrate override switch delay time (ms)
DT32	10000	0	60000	Tool change alarm time of Liuxin 8-tool hydraulic tool post (ms)

B.4.3 D parameter

D address	Initial value of ladder	Input min. value	Input max. value	Parameter explanation
D0	4	1	16	Total tool count of the tool post
D1	1	0	5	Internal controlled axis number corresponding to manual axis movement key of X (the press key is invalid when it is set to 0)
D2	0	0	5	Internal controlled axis number corresponding to manual axis movement key of Y (the press key is invalid when it is set to 0)
D3	2	0	5	Internal controlled axis number corresponding to manual axis movement key of Z (the press key is invalid when it is set to 0)
D4	0	0	5	Internal controlled axis number corresponding to manual axis movement key of the 4 th axis

				(the press key is invalid when it is set to 0)
D5	0	0	5	Internal controlled axis number corresponding to manual axis movement key of C (the press key is invalid when it is set to 0)

B.5 PLC alarm (A address) explanation

Address	Alarm No.	Alarm message
A0000.0	1000	Tool change time is too long
A0000.1	1001	tool post not in-position after rotation (CW) ends
A0000.2	1002	tool change not finished
A0000.3	1003	Can't execute M10 and M11 under invalid tailstock function
A0000.4	1004	Can't retreating tailstock when spindle rotating
A0000.5	1005	Can't start cycle when cycle start enabling is closed
A0000.6	1006	Protection door not closed
A0000.7	1007	Chuck low pressure alarm
A0001.0	1008	Can't unclamp chuck when spindle rotating
A0001.1	1009	Chuck clamped is not in-position when spindle rotating
A0001.2	1010	Have not checked chuck clamping/releasing in-position signal
A0001.3	1011	Can't start spindle when chuck is released
A0001.4	1012	Can't execute M12 or M13 under invalid chuck function
A0001.5	1013	Have not checked tool post locked signal
A0001.6	1014	Undefined function M code
A0001.7	1015	It is the non-analog spindle, and the spindle JOG function cannot be executed
A0002.0	1016	M03, M04 designation is mistaken
A0002.1	1017	T command tool does not exist
A0002.2	1018	The total tool count (D0) setting value is 0
A0002.3	1019	It is non-analog spindle and M41/42/43/44 cannot be executed
A0002.4	1020	The automatic gear change function is disabled, and K13.7 should be checked
A0002.5	1021	The cycle start at the feed hold knob position cannot be permitted
A0002.6	1022	The feed hold button is open-circuit and the system cannot execute the cycle start
A0002.7	1023	Have checked the button short-circuit on the machine panel

A0003.0	1024	Commanded tool exceeds the total tool spacing count(D0) setting
A0003.1	1025	Commanded M code is invalid
A0003.2	1026	Spindle position time is too long
A0003.3	1027	Have received the chuck clamping/releasing in-position signal
A0004.0	1032	Have not received pre-graduation proximity switch signal alarm
A0004.1	1033	Have not received locked proximity switch signal alarm
A0004.2	1034	The tool is not consistent with the target tool spacing when the tool change is completed
A0004.3	1035	No locking proximity switch signal when the tool change is completed
A0004.4	1036	The tool post motor is overheat
A0004.5	1037	Total tool count (D0) setting of the tool post is mistaken (it is only 8, 10, 12)
A0005.0	1040	Have not found the target tool
A0005.1	1041	Have not received the tool post stopping rotation and locked signal
A0005.2	1042	No locked signal when the tool change is completed
A0005.3	1043	The current tool is not consistent with the target cutting spacing after the tool change is completed
A0005.4	1044	Tool count (D0) setting of the tool post is mistaken(it is only 8)

B.6 G, F signal for Ladder

B.6.1 G signal

Address	Function	Symbol	Chapter index
G4.3	Miscellaneous function end signal	FIN	5.11.4.1
G4.4	2M function end signal	MFIN2	5.11.4.2
G4.5	3M function end signal	MFIN3	5.11.4.2
G5.0	Miscellaneous function end signal	MFIN	5.11.4.1
G5.2	Spindle function end signal	SFIN	5.11.4.1
G5.3	Tool function end signal	TFIN	5.11.4.1
G5.6	Miscellaneous function locked signal	AFL	5.11.5.1
G6.2	Manual absolute signal	ABSM	5.9.7.1
G6.4	Override cancelled signal	OVC	5.10.4

Address	Function	Symbol	Chapter index
G7.2	Cycle start signal	ST	5.9.1.1
G7.4	Travel check 3 released signal	RLSOT3	5.6.6.2
G7.6	Stored travel limit selection signal	EXLM	5.6.6.1
G8.4	Emergency stop signal	ESP	5.6.1
G8.5	Feed pause signal	SP	5.9.1.3
G8.7	External reset signal	ERS	5.9.2.1
G10,G11	Manual traverse speed override signal	JV0~JV15	5.7.1.1
G12	Feedrate override signal	FV0~FV7	5.10.3.1
G14.0, G14.1	Rapid feedrate override signal	ROV1,ROV2	5.10.2.1
G18.0~G18.3	MPG 1 feed axis selection signal	HS1A~HS1D	5.7.2.1
G18.4~G18.7	MPG 2 feed axis selection signal	HS2A~HS2D	5.7.2.1
G19.4, G19.5	MPG/Step override signal	MP1,MP2	5.7.2.2
G19.7	Manual rapid feed selection signal	RT	5.7.1.3
G27.0	The 1 st spindle selection signal	SSW1	5.12.4
G27.1	The 2 nd spindle selection signal	SSW2	5.12.4
G27.3	The 1 st spindle stop signal	SSTP1	5.12.4
G27.4	The 2 nd spindle stop signal	SSTP2	5.12.4
G27.7	Spindle contour control switch signal	CON	5.12.5
G28.1, G28.2	Gear selection signal	GR1, GR2	5.12.2.1
G28.7	The 2 nd position encoder selection	PC2SLC	5.12.4
G29.0	The 2 nd spindle gear selection	GR21	5.12.4
G29.4	Spindle speed arrival signal	SAR	5.12.2.1
G29.6	spindle stop signal	SSTP	5.12.2.1
G30	spindle override signal	SOV0~SOV7	5.12.2.1
G43.0 ~ G43.2,G43.5,G43.7	Mode selection signal	MD1,MD2,MD4,D NC1,ZRN	5.6.4.1
G44.0	Skip optional block signal	BDT1	5.9.6.1
G44.1	All axes machine locked signal	MIK	5.9.3.1
G46.1	Single block signal	SBK	5.9.5.1
G46.7	Dry run signal	DRN	5.9.4.1
G100.0~G100.4		+J1~+J5	5.7.1.1

Feed axis and direction selection

Address	Function	Symbol	Chapter index
G102.0~G102.4	signal	-J1~-J5	5.7.1.1
G114.0~G114.4	Overtravel signal	+L1~+L5	5.6.6.1
G116.0~G116.4		-L1~-L5	5.6.6.1
G200.0	Spindle Jog function signal	SPHD	5.12.3.1
G201	Current tool signal	NT00~NT07	5.13.1

B.6.2 F signal

Address	Function	Symbol	Chapter index
F0.4	Feed pause signal	SPL	5.9.1.4
F0.5	Cycle start signal	STL	5.9.1.3
F0.6	Servo ready signal	SA	5.5.2
F0.7	Automatic operation signal	OP	5.9.1.5
F1.0	Alarm signal	AL	5.6.3
F1.1	Reset signal	RST	5.9.2.2
F1.3	Distribution end signal	DEN	5.11.4.3
F1.4	Spindle enabling signal	ENB	5.12.2.1
F1.7	CNC ready signal	MA	5.6.2
F2.0	Inch input signal	INCH	5.14.1
F2.1	Rapid feed signal	RPDO	5.6.5.1
F2.2	Constant surface cutting signal	CSS	5.12.2.1
F2.3	Thread cutting signal	THRD	5.15.1
F2.7	Dry run check signal	MDRN	5.9.4.2
F3.0	Step mode check signal	MINC	5.6.4.2
F3.1	MPG mode check signal	MH	5.6.4.2
F3.2	Manual mode check signal	MJ	5.6.4.2
F3.3	MDI mode check signal	MMDI	5.6.4.2
F3.4	DNC mode check signal	MRMT	5.6.4.2
F3.5	Auto mode check signal	MMEM	5.6.4.2
F3.6	Edit mode check signal	MEDT	5.6.4.2
F4.0	Skip optional block check signal	MBDT1	5.9.6.2

Address	Function	Symbol	Chapter index
F4.1	All axes machine locked check signal	MMLK	5.9.3.2
F4.2	Manual absolute check signal	MABSM	5.9.7.2
F4.3	Single block check signal	MSBK	5.9.5.2
F4.4	Auxiliary function locked check signal	MAFL	5.11.5.2
F4.5	Machine zero return mode check signal	MREF	5.6.4.2
F7.0	Miscellaneous function strobe signal	MF	5.11.1.1
F7.2	Spindle speed function strobe signal	SF	5.11.2
F7.3	Too function strobe signal	TF	5.11.3.1
F8.4	2M miscellaneous function strobe signal	MF2	5.11.1.3
F8.5	3M miscellaneous function strobe signal	MF3	5.11.1.3
F9.4	M decoding signal	DM30	5.11.1.2
F9.5		DM02	5.11.1.2
F9.6		DM01	5.11.1.2
F9.7		DM00	5.11.1.2
F10~F13	Miscellaneous function code signal	M00~M99	5.11.1.1
F14~F15	2M miscellaneous function code signal	M200~M299	5.11.1.3
F16~F17	3M miscellaneous function code signal	M300~M399	5.11.1.3
F22~F25	Spindle speed code signal	S00~S31	5.11.2
F26~F29	Tool function code signal	T00~T31	5.11.3.1
F38.2	The 2 nd spindle enabling signal	ENB2	5.12.4
F62.7	Target workpiece count arrival signal	PRTSF	5.16.1
F94.0~ F94.4	Machine zero return end signal	ZP1~ZP5	5.8.1.1
F96.0~ F96.4	Machine zero return end signal of the 2 nd reference point	ZP21~ZP25	5.8.1.1
F98.0~ F98.4	Machine zero return end signal of	ZP31~ZP35	5.8.1.1

Address	Function	Symbol	Chapter index
	the 3 rd reference point		
F100.0~ F100.4	Machine zero return end signal of the 4 th reference point	ZP41~ZP45	5.8.1.1
F102.0~ F102.4	Axis movement signal	MV1~MV5	5.5.1.1
F106.0~ F106.4	Axis motion direction signal	MVD1~MVD5	5.5.1.2
F120.0~ F120.4	Reference point establishment signal	ZRF1~ZRF5	5.8.1.2