In this instruction manual, we will endeavor to describe various matters related to the operation of the GSK25iMc/GSK25iTc Series Bus-type Milling/Turning CNC System. It is not possible to describe in detail all the operations that need not be done and/or cannot be done in the CNC system due to length constraints and specific product usage. Therefore, anything not specifically indicated in this instruction manual shall be deemed to be the operation that is "impossible" or "not permitted".

CNCmakers Limited
Add: No.168, Xiadu Road, Haizhu District, Guangzhou, China 510300 Tel:86-139-25042139 Fax:86-20-84185336 www.CNCmakers.com
Online shop: www.paycnc.com Email: paycnc@paycnc.com cnc@CNCmakers.com WhatsApp: https://wa.me/+8613925042139

Preface

Dear Customers:

We feel honored and grateful for your choice of GSK25iMc/GSK25iTc Series Bus-type Milling/Turning CNC System.

This Manual describes in detail the PLC programming and functions of the GSK25iMc/GSK25iTc Series Bus-type Milling/Turning CNC System.

Please read this manual carefully before installing and using the series products in order to ensure product safety and normal and effective operation.

Safety Warnings



Improper operation will lead to accidents, thus personnel with appropriate qualifications are required to operate the system.

Special tip: The power supply installed on (in) the chassis is specially provided for the CNC Systems manufactured by our company.

Users are not allowed to use this power supply for other purposes. Otherwise, great danger will be caused!

Precautions

■ Transport and storage

- ☻ **Product packing boxes shall not be stacked more than six layers**
- ☻ **Do not climb, stand or place heavy objects on product packing boxes**
- ☻ **Do not use cables connected to the product to drag or carry the products**
- ☻ **Do not collide or scratch the panel and display screen**
- ☻ **Product packing boxes shall be protected from dampness, exposure to the sun and rain**

■ Unpacking inspection

- ☻ **Please confirm whether it is the product you purchased after opening the package**
- ☻ **Check whether the product is damaged in transit**
- ☻ **Confirm whether the components are complete and damaged according to the list**
- ☻ **Please contact our company in time in case of product model inconsistency, missing accessories or transport damage**

■ Wiring

- ☻ **Participants in wiring and inspection must be competent professionals**
- ☻ **The product must be grounded reliably, with the grounding resistance no greater than 0.1 Ω. Neutral wire shall not be used instead of grounding wire**
- ☻ **The wiring must be correct and firm so as not to cause product failure or unintended consequences**
- ☻ **Surge absorbing diodes must be connected to the product in the specified direction; otherwise the product will be damaged**
- ☻ **The product must be powered off before plugging or unplugging or opening the product chassis**

■ Servicing

- ☻ **Power supply must be cut off before repairing or replacing components**
- ☻ **The failure must be checked in case of short circuit or overload, and it can be restarted after troubleshooting**
- ☻ **The power to the product shall not be frequently turned on or off. The interval shall be at least 1 min if re-energization is required after power-off**

Statement

- This manual describes various contents as much as possible. However, due to too many possibilities involved, it is impossible to describe all the operations that can or cannot be performed one by one. Therefore, contents with no special description in this manual can be considered unusable.

Warnings

- Before installation and connection, programming and operation of the product, you must read the Product Manual and the Operation Instructions of the machine manufacturer in detail, and perform related operations in strict accordance with the requirements of the Manual and the Instructions, otherwise, it may cause damage to the product and machine, workpiece scrapping or even personal injury.

Caution

- The product functions and technical indicators (such as accuracy, speed, etc.) described in this manual are only for this product. The actual function configuration and technical performance of the CNC machine are determined by the design of the machine manufacturer, which are subject to the instruction manual of the machine manufacturer.
- Although the system is equipped with a standard machine operation panel, the function of each key on the standard machine panel is defined by the PLC program (ladder diagram). Please note that the functions of the keys on the machine panel in this manual are described for the standard PLC program.
- Please refer to the Operating Manual provided by the machine manufacturer for the function and meaning of keys on the machine panel.

All the contents of this manual are subject to change without prior notice.

Part I Programming

Introduction to the Command Code and Program Format

Part II Operation

Introduction to the Operation and Usage Method of the GSK25iMc/GSK25iTc Series Bus-type Milling/Turning CNC System

Appendix

Introduction to GSK25iMc/GSK25iTc Series Bus-type Milling/Turning CNC System G Code Programming Rules, Alarm Information Table, etc.

Safety Responsibility

Manufacturer's Safety Responsibilities

——**The manufacturer shall be responsible for the hazard of the supplied CNC system and accompanying accessories that have been eliminated and/or controlled in design and structure.**

——**The manufacturer shall be responsible for the safety of the supplied CNC system and accompanying accessories.**

——**The manufacturer shall be responsible for the use information and advice provided for the user.**

Users' responsibilities for safety

——**Users shall be familiar with and master the safe operation through learning and training of safe operation of CNC system.**

——**Users shall be responsible for the safety and hazards as a result of adding, changing or modifying the original CNC system and accessories by themselves.**

——**Users shall be responsible for any hazard caused by failure to operate, adjust, maintain, install, store and transport the products in accordance with the Manual.**

This manual shall be kept by final users.

Thank you very much for using the products of GSK CNC Equipment Co., Ltd., and your friendly support to our company!

Contents

Part I Programming

Chapter I Fundamentals of CNC Machine Programming 3
1.1 Definition of CNC programming 3
1.2 Programming characteristics and requirements of CNC machine 3
1.2.1 Programming characteristics and requirements of CNC milling machine/machining center 3
1.2.2 Programming characteristics and requirements of CNC lathe/turning center 4
1.3 Machine coordinate system 5
1.3.1 Definition of machine coordinate system 5
1.3.2 Specification in machine coordinate system 5
1.3.3 Direction of machine coordinate system 6
1.3.4 Machine origin and reference point 7
1.4 Workpiece coordinate system 8
1.4.1 Workpiece coordinate system 8
1.4.2 Workpiece coordinate system origin 9
1.5 Structure of program 9
1.5.1 Program name 10
1.5.2 Content of program 10
1.5.3 End of program 10
1.6 Composition of program block 11
1.6.1 Basic format of program block 11
1.6.2 Composition of program block 13
1.6.3 Slash block skip of program 13
1.6.4 Annotation for program block 14
1.6.5 System macro program folder 14
Chapter II Preparation of Function Command Codes 15
2.1 Modal and non-modal 15
2.2 Coordinate function 15
2.3 List of preparatory function command G code 15
2.3.1 Milling preparatory function G code 15
2.3.2 Turning preparatory function G code 18
Chapter III Position Command 25
3.1 Absolute value and incremental value programming (G90, G91) 25
3.2 Imperial/metric conversion (G20, G21) 27
3.3 Decimal point input 28
3.4 Diameter and radius programming 30
Chapter IV Interpolation Function 33
4.1 Fast moving (G00) 33
4.2 Linear interpolation (G01) 35
4.3 Plane selection (G17, G18, G19) 38
4.4 Circular interpolation (G02, G03) 38
4.5 Helical interpolation (G02, G03) 43
4.6 Helical interpolation and conic interpolation (G02, G03) 44
4.7 Thread turning (G32, G34) 47
4.7.1 Equal-pitch thread (G32) 47
4.7.2 Variable-pitch conic thread (G34) 50
4.8 Thread cutting (G33) 52
4.9 One-way positioning (G60) 53

4.10 Cylindrical interpolation (G07.1) 55
4.11 Polar coordinate interpolation (G12.1) 60
4.12 Polar coordinate command (G16/G15) 67
4.13 High-speed high-precision machining (G05 P3) 72
4.13.1 Relevant parameters of algorithm of high-speed machining 73
4.14 Preset function of the pre-read program block (G08) 74
Chapter V Simplified Programming 77
5.1 Linear angle commands (G01 X_ /Z_ , A_ and G01 U_ /W_ ,A_) 77
5.2 Chamfering/smoothing (,C/,R) 78
5.2.1 Chamfering (G01/G02/G03 X_ Z_ ,C_) 78
5.2.2 Smoothing (G01/G02/G03 X_ Z_ ,R_) 81
5.3 Direct programming of pattern size 84
Chapter VI Feed Functions 87
6.1 Dwell (G04) 87
6.2 Rapid feed speed 88
6.3 Cutting feed speed 88
6.4 Feed per minute/per revolution (G94 (G98) and G95 (G99)) 91
6.5 Accurate positioninf and inspection (G09) 92
6.6 Accurate positioning and inspection mode (G61) 93
6.6.1 Automatic internal corner override (G62) 94
6.6.2 Automatic internal arc cutting speed override 96
6.7 Tapping mode (G63) 97
6.8 Cutting mode (G64) 97
Chapter VII Auxiliary Functions 99
7.1 Auxiliary functions 99
7.2 2nd auxiliary function 101
7.3 Indexing of rotary worktable 102
Chapter VIII Conversion of Programming Coordinates 105
8.1 Scaling (G50/G51) 105
8.2 Program coordinate rotation (G68/G69) 108
8.3 Programmable mirror (G50.1/G51.1) 113
8.4 Center point control of 5th axis machining tool (G43.4) 116
8.5 Inclined surface machining (G68.2) 125
8.6 Normal direction control (G40.1/G41.1/G42.1) 133
Chapter IX Fixed Cycle of Milling 139
9.1 Standard fixed cycle (G80~G89/G73/G74/G76) 139
9.2 High-speed deep hole drilling cycle (G73) 142
9.3 Left-hand tapping cycle (G74) 144
9.4 Fine boring cycle (G76) 146
9.5 Fixed cycle cancelled (G80) 148
9.6 Drilling cycle (G81) 148
9.7 Drilling cycle and boring cycle (G82) 150
9.8 Deep hole chip-removal drilling cycle (G83) 151
9.9 Right-hand tapping cycle (G84) 153
9.10 Boring cycle (G85) 155
9.11 Boring cycle (G86) 156
9.12 Back boring cycle (G87) 157
9.13 Boring cycle (G88) 159
9.14 Boring cycle (G89) 161
9.15 Left-hand rigid tapping Cycle (G74) 163
9.16 Right-hand rigid tapping cycle (G84) 164
9.17 Chip removing rigid tapping cycle (G84, G74) 166
Chapter X Fixed Cycle of Turning 171

10.1 Inner and outer circles single fixed cycle (G90, G94) 171
10.1.1 Inner and outer circles cutting cycle (G90) 171
10.1.2 End face cutting cycle (G94) 172
10.2 Longitudinal, end face, profiling rough turning cycle and fine turning cycle (G71, G72, G73, G70) .. 175
10.2.1 Longitudinal rough turning cycle (G71) 175
10.2.1.1 Type Ⅰ 176
10.2.1.2 Type II 177
10.2.2 End face Rough Turning Cycle (G72) 181
10.2.3 Profiling Rough Turning Cycle (G73) 184
10.2.4 Precautions to fixed cycles G71, G72, G73 and G70 187
10.3 Grooving fixed cycle (G74, G75) 188
10.3.1 Radial grooving cycle (G75) 188
10.3.2 End face grooving cycle (G74) 190
10.3.3 Precautions to fixed cycles commands G74 and G75 193
10.4 Thread cutting fixed cycle (G92, G76) 193
10.4.1 Thread cutting single fixed cycle (G92) 193
10.4.2 Threading composite fixed cycle command (G76) 196
10.4.3 Comprehensive programming examples 198
10.5 Overview of boring fixed cycle 200
10.6 End face deep hole drilling cycle, longitudinal deep hole drilling cycle (G83, G87) 202
10.7 End face tapping cycle, longitudinal tapping cycle (G84, G88) 203
10.7.1 Selection of rigid/non-rigid Tapping 205
10.7.2 Release rigid tapping selection 205
10.7.3 Feed Per Round of Rigid Tapping 205
10.8 End face boring cycle, longitudinal boring cycle (G85, G89) 205
10.9 Cancel of drilling fixed cycle (G80) 206
10.10 Precautions to use of drilling fixed cycle 206
10.11 Initial point and R point standard return (G98, G99) 207

Chapter XI Subprogram Programming 209

11.1 Subprogram control (M98, M99, M198) 209
11.2 Programming examples 211

Chapter XII Variable Command 215

Chapter XIII Macro Program 217

13.1 User macro program 217
13.2 Macro program call 218
13.2.1 Non-modal call (G65) 218
13.2.2 Modal calling A (G66) 220
13.2.3 G code macro program calling 220
13.2.4 Auxiliary command macro program calling (M, T Code Macroprogram Calling) 221
13.2.5 Detail explanation of macro program calling command 222
13.3 Variable 223
13.4 Types of variables 224
13.4.1 Covariate 224
13.4.2 Local variables (#1~#33) 225
13.4.3 Macro program interface input output (#1000~#1035, #1100~#1135) 227
13.4.4 Tool offset 230
13.4.5 Workpiece coordinate offset (#2500~#2906) 231
13.4.6 Alarm (#3000) 232
13.4.7 Time information (#3001, #3002, #3011, #3012) 233
13.4.8 Suppression of single block stop, auxiliary function end signal waiting (#3003) 234
13.4.9 Validness and invalidness of fFeed hold, feed speed override and G09 (#3004) 234
13.4.10 Information display and stop (#3006) 234
13.4.11 G Command Mode (#4001~#4130, #4201~#4330) 235
13.4.12 Position information (#5001~#5140+n) 237
13.4.13 Workpiece machining number (#3901, #3902) 239

13.5 Operation command 239
13.6 Control command 244
13.6.1 Branch 244
13.6.2 Repeat 245
13.7 Precautions 247
Chapter XIV Spindle Speed Function 249
14.1 Function of the spindle 249
14.2 Spindle function S2 digit gear control 249
14.3 Spindle function S5 digit numerical control 250
14.4 Constant cutting linear speed control (G96, G97) 250
14.5 Spindle clamping speed setting (G50) 252
14.6 Spindle speed fluctuation detection function (G25/G26) 253
14.7 Spindle/C-axis control 254
Chapter XV Tool Function 257
15.1 Tool selection function 257
Chapter XVI Milling Tool Offset Function 259
16.1 Tool compensation 259
16.2 Tool length compensation/cancellation (G43, G44/G49) 260
16.3 Tool radius compensation (G39/G40/G41, G42) 262
16.3.1 Tool radius compensation action 263
16.3.2 Other commands and actions in tool radius compensation 271
16.3.3 General precautions for tool radius compensation 277
16.3.4 Change of compensation number in compensation mode 217 278
16.3.5 Interference check 280
16.3.6 Compensation amount diameter specifying 282
16.4 Programmable compensation input (G10 L2, L10, L11, L12, L13) 283
Chapter XVII Turning Tool Offset Function 285
17.1 Tool compensation 285
17.2 Tool length compensation 285
17.3 Tool nose wear compensation 287
17.4 Tool nose R compensation (G40/G41, G42) 288
17.4.1 Tool nose point and compensation direction 289
17.4.2 Tool nose R compensation action 290
17.4.3 Canceling tool nose R compensation 295
17.4.4 Cancellation action of the tool nose R compensation 295
17.4.5 Other actions of tool nose R compensation 298
17.4.6 General precautions for tool nose R compensation 304
17.4.7 Interference check 304
17.5 Programmable compensation input (G10 L2/L10/L11) 305
Chapter XVIII Coordinate System Setting Function 309
18.1 Coordinate system and control axis 309
18.2 Basic machine coordinate system, workpiece coordinate system and local coordinate system 311
18.3 Machine origin and second reference point 311
18.4 Automatic coordinate system setting 312
18.5 Basic machine coordinate system selection (G53) 313
18.6 Coordinate system shift (G92 (G50)) 314
18.7 Reference point return (G28, G29) 316
18.8 Return to the second, third and fourth reference points (G30) 318
18.9 Reference point check (G27) 320
18.10 Workpiece coordinate system setting and workpiece coordinate system offset (G54 - G59 (G54.1)) 321
18.11 Local coordinate system setting (G52) 325
18.12 Workpiece coordinate system restoration (G92.1) 326
Chapter XIX Protection Functions 329

19.1 Stored stroke limit (G22/G23) 329
Chapter XX Measurement Functions 331
20.1 Automatic tool length measurement (G37) 331
20.2 Jump function (G31) 334
20.3 Torque limit jump function (G31) 338
Chapter XXI Programmable Data Input 341
21.1 Programmable parameters input (G10 L50/G11) 341
21.2 Programmable pitch compensation data input (G10 L51) 342

Part II Operation343

Chapter I Operation Panel 345
1.1 Panel partition 345
1.2 Panel function description 346
1.2.1 LCD display area 346
1.2.2 Edit keyboard area 346
1.2.3 Page display mode area 347
1.2.4 Machine control area 348
Chapter II System Power-on, Shutdown, and Safe Operation 351
2.1 System power-on 351
2.2 Switching off 351
2.3 Safe Operation 351
2.3.1 Reset operation 351
2.3.2 Emergency stop 352
2.3.3 Feed hold 352
2.4 Cycle start and feed hold 352
2.5 Over-travel protection 352
2.5.1 Hardware over-travel protection 352
2.5.2 Software over-travel protection 353
2.5.3 Stored travel inspection 353
2.5.4 Chuck tailstock barrier 357
Chapter III Interface Display and Operation 363
3.1 Position interfaces 363
3.1.1 Absolute mode 363
3.1.2 Relative mode 363
3.1.3 Comprehensive interface 364
3.1.4 Monitoring mode 365
3.1.5 Load meter 366
3.1.6 MPG 366
3.2 Program interfaces 367
3.2.1 File list display 368
3.2.2 Transfer program from USB flash drive 368
3.2.3 Program display 371
3.2.4 Program creation 371
3.2.5 Program editing 375
3.2.6 MDI input display 391
3.2.7 Detection interface 393
3.2.8 Data display 393
3.2.9 Introduction to background editing 394
3.2.10 Program restart 396
3.2.11 Scheduling 397
3.3 Offset Interface 399
3.3.1 [Offset] interface operation instructions 399

3.3.2 [Workpiece System] interface operation instructions 400
3.3.3 [Additional Workpiece System] interface operation instructions 400
3.3.4 [Macro Variable] interface operation instructions 401
3.3.5 [Coordinate Centering] interface operation instructions 401
3.4 System interface 402
3.4.1 [System Parameters] display interface 403
3.4.2 [Parameter Classification] display interface 403
3.4.3 [Diagnosis] display interface 404
3.4.4 [Wiring Diagnosis] display interface 404
3.4.5 [DB List] display interface 405
3.4.6 [File Management] display interface 407
3.4.7 [Pitch Compensation] display interface 408
3.5 Setting interface 410
3.5.1 [Login] interface operation instructions 410
3.5.2 [User Privileges] interface operation instructions 411
3.5.2 [Date and Time] interface display 412
3.6 Graphic display 412
3.6.1 Graphic parameters 413
3.6.2 Path 416
3.7 Ladder diagram interface 416
3.7.1 Ladder diagram display interface 417
3.7.2 PLC diagnosis 417
3.7.3 Signal Tracking 418
3.7.4 K parameters 419
3.7.5 Timer 419
3.7.6 Counter 419
3.7.7 PLC list 420
3.7.8 Information check 421
3.7.9 Data table 421
3.8 User defined interface 422
3.9 Information interface display 423
3.9.1 Current alarm interface 423
3.9.2 Alarm history 424
3.9.3 Operation history 424
3.9.4 Machining history 425
3.9.5 Maintenance records 425
3.9.6 Periodic maintenance 426
3.9.7 System information 426
3.9.8 Running information 427
3.10 Help interface display 427
3.10.1 Alarm interface 428
3.10.2 Wizard interface 428
3.10.3 G-code list interface 429
3.10.4 Macro-command interface 430
3.10.5 PLC address interface 430
3.10.6 Calculator interface 431
3.10.7 Parameter table interface 431
3.10.8 Servo parameter interface 432
3.10.9 Servo motor interface 432
3.10.10 Spindle Motor Interface 433

Chapter IV Manual Operation 435

4.1 Coordinate axis movement 435
4.1.1 Manual feed 435
4.1.2 Manual rapid movement 435
4.1.3 Manual feed rate and manual fast moving speed selection 435
4.1.4 Manual feed per revolution 436
4.2 Spindle control 436

4.2.1 Spindle clockwise rotation (M03) 436
4.2.2 Spindle counterclockwise rotation (M04) 436
4.2.3 Spindle stop (M05) 436
4.2.4 Spindle pause 436
4.3 Other manual operation 436
4.3.1 Coolant control 436
4.3.2 Lubrication control 437
4.3.3 Chip removal control 437
Chapter V Single-step Operation 439
5.1 Single-step feed 439
5.1.1 Selection of movement amount 439
5.1.2 Selection of moving axis and moving direction 439
5.2 Auxiliary control during the single-step operation 439
Chapter VI MPG Operation 441
6.1 MPG feed 441
6.1.1 Selection of MPG control mode 441
6.2 Auxiliary control during MPG operation 442
Chapter VII Automatic Operation 443
7.1 Autorun 443
7.1.1 Operating steps for autorun program 443
7.1.2 Launch of autorun 443
7.1.3 Stop of autorun 443
7.1.4 Spindle speed control during autorun 444
7.1.5 Speed control in autorun 444
7.1.6 Dry run 445
7.1.7 Single block operation 445
7.1.8 Full axis function lock operation 445
7.1.9 Auxiliary function lock operation 446
7.1.10 Manual intervention 446
7.1.11 Insertion of the MPG 447
7.2 MDI operation 448
7.2.1 MDI program editing 448
7.2.2 MDI command block execution and stop 449
7.2.3 Switch among various operation modes 449
Chapter VIII Homing Operation 451
8.1 Machine reference position concept 451
8.2 Operating steps for machine homing 451
8.3 Homing debug method 451
Chapter IX System Communication 455
9.1 Network communications 455
9.1.1 Network connection diagnosis 455
9.1.2 Use of a helper 456
9.1.2.1 Connection of the CNC system 456
9.1.2.2 Network DNC 457
9.1.3 File transmission 458
9.2 U disk communication 459
9.2.1 USB flash drive DNC machining 459
9.2.2 Transmission between USB flash drive and CNC 459
9.2.3 Transmission of ladder diagram files between USB flash drive and CNC 461
Chapter X Workpiece Coordinate System Setting 463
10.1 Coordinate system chain 463
10.2 Workpiece coordinate system origin setting 463
10.3 Offset of workpiece coordinate system 464

10.4 Tool offset measurement 465

Appendix

Appendix I G code programming rules 469

Appendix Ⅱ System Alarm List 471

2.1 System alarm (PS alarm) 471
2.2 Servo and position control alarm (PV alarm) 498
2.3 Spindle alarm (PD alarm) 502

Part I Programming

Chapter I Fundamentals of CNC Machine Programming

1.1 Definition of CNC programming

Function and purpose

In CNC machining machine, the program that is pre-programmed is used to control the machine and work. When machining a part on a CNC machine, the tool path and other machining conditions must be programmed (programmed), and the programmed program is called a part program. In order to enable the CNC machine to act according to the requirements of parts processing, these requirements must be informed to the CNC system in the form of commands that the CNC system of machine can recognize. The commands that can be recognized by the CNC system are called programs, and the process of making programs is called CNC programming.

The process of CNC programming not only refers to the process of writing CNC machining commands, but also includes the whole process from part analysis to writing machining commands, then to making control medium and checking program.

Before programming, it is necessary to analyze the machining process of the part first to determine the machining process route, process parameters, tool motion trajectory, displacement, cutting parameters (cutting speed, feed, back-cut amount) and various auxiliary functions (changeover). Tool, spindle forward and reverse rotation, cutting fluid switch, etc.); then, write a processing program list according to the commands and program format specified by the CNC machine; then record the content of this program list on the control medium (such as mobile memory, hard disk), After checking it is correct, it is input into the numerical control device of the numerical control machine by manual input mode or computer transmission mode, so as to control the machine processing parts.

1.2 Programming characteristics and requirements of CNC machine

1.2.1 Programming characteristics and requirements of CNC milling machine/machining center

Detailed description

1. Programming characteristics of CNC milling machine/machining center

(1) In order to facilitate numerical calculation in programming, tool radius compensation is widely used in the programming of CNC milling machine/machining center.

(2) In order to meet the machining needs of CNC milling machine, the common boring, drilling and cutting operations can be achieved by using the fixed cycle function of CNC system itself, in order to simplify programming.

(3) Most CNC milling machines have special commands such as mirror machining, scaling and polar coordinate programming command to improve programming efficiency and simplify the program.

2. Programming requirements of CNC milling machine/machining center

(1) There are many procedures and a complex variety of tools used for parts processing on the machining center. All the processes of roughing, semi-finishing and finishing can be completed in one clamping, so it is necessary to carry out reasonable process analysis and carefully arrange the processing sequence of each process before programming the machining center to improve processing efficiency

and processing accuracy.

(2) The use of automatic or manual tool changing can be determined according to the size of the processing batch. Manual tool changing is generally adopted for single piece or small batch of workpiece machining. However, automatic tool changing is generally adopted for machining workpiece with more than 10 pieces in a batch and frequent tool changing.

(3) Attention shall be paid to the reasonable selection of the position of the automatic tool changing point in the program, and the collision accident of the tool, workpiece and fixture shall be avoided in the process of tool retracting and automatic tool changing.

(4) Use off-machine tool setting as much as possible in the tool setting process, and fill in the measured size on the tool card, so that the operator can modify the tool compensation parameters in time before running the program, thereby improving the efficiency of the machine.

(5) The programs of different working procedures shall be arranged into different subprograms as much as possible, so as to debug each independent working procedure individually and readjust the machining procedures because of the unreasonable machining sequence. In the main program, focus is taken on completing tool changing and subprogram call.

1.2.2 Programming characteristics and requirements of CNC lathe/turning center

Detailed description

According to the characteristics of CNC lathe/turning center, its programming has the following characteristics:

(1) Mixed programming: In a program block, programming can be done in absolute or incremental mode according to the dimensions indicated on the drawing. Mixed programming can also be adopted. The absolute and incremental dimensions are specified by G90/G91 command in B and C series G code, while the incremental values on X-axis and Z-axis are specified directly by address symbols U and W in A series G code.

(2) Radial dimension expressed by diameter: The radial dimension of the turned parts is expressed by the diameter dimension in the drawing and measurement. Therefore, X (U) is usually expressed in diameter when programming is done in the diameter direction. If it is expressed in radius, the parameters will usually be modified or assigned in the program with relevant commands.

(3) Fixed cycles to simplify the programming: In order to simplify the programming, the CNC system adopts different forms of fixed cycles to facilitate repeated cyclic cutting because the bars or forgings are often used as blanks in turning and there is a large working allowance.

(4) Tool nose arc radius compensation: When NC programming, the tool nose of turning tool is often regarded as a point, while the actual tool nose is usually an arc with a small radius. In order to improve the machining accuracy of the workpiece, G41 or G42 command is often used to compensate the tool nose arc radius of the turning tool when the machining program with the circular arc turning tool is prepared.

(5) Using tool position compensation: The tool setting operation of CNC lathes and the setting of the workpiece coordinate system are usually carried out by means of tool position compensation.

1.3 Machine coordinate system

Function and purpose

1.3.1 Definition of machine coordinate system

When machining parts on CNC machine, the movement of the machine is controlled by the commands issued by the CNC system. In order to determine the movement direction and distance of the machine, it is necessary to establish a coordinate system on the machine, and this coordinate system is called the machine coordinate system, also known as the standard coordinate system.

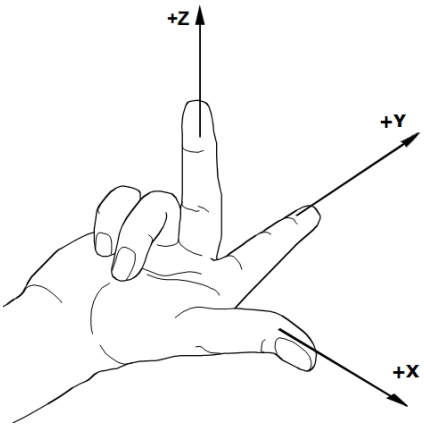
1.3.2 Specification in machine coordinate system

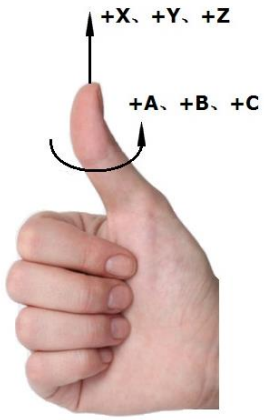
The machining action of CNC machine is mainly divided into two parts: the motion of tool and workpiece. Therefore, it is stipulated that the tool is always assumed to be moving relative to the stationary workpiece in determining the direction of the machine coordinate system.

For the direction of machine coordinate, it is uniformly stipulated that the direction of increasing the distance between the workpiece and the tool is the forward direction.

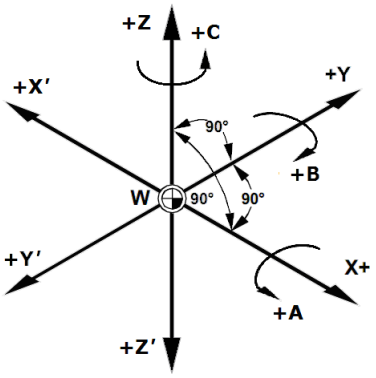
The coordinate system of CNC machine adopts Cartesian coordinate system conforming to the right-hand rule. Right-hand Cartesian coordinate system assigns the direction of the thumb as the forward direction of the X-axis, the index finger pointing to the forward direction of Y-axis and the middle finger pointing to the forward direction of the Z-axis. The right-hand screw rule assigns the forward direction in which A-axis, B-axis and C-axis rotate. The forward direction (+ C') of the spindle (e.g., lathe spindle) that rotates the workpiece is opposite to +C direction.

For all types of front-mounted tool-holder lathes, the present term "forward rotation" shall be the reverse rotation (+C') according to the standard, and "forward rotation" refers to the customary common term

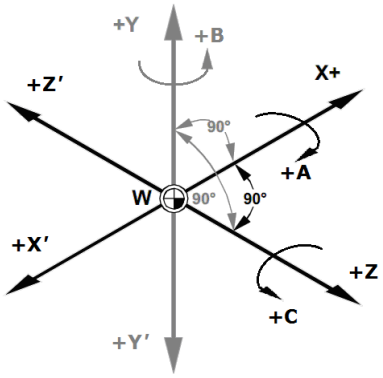




Milling:



Turning:



Right-hand Cartesian coordinate system

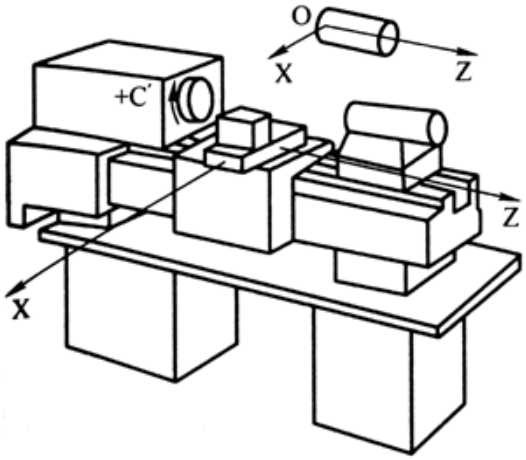
1.3.3 Direction of machine coordinate system

When determining the coordinate axes of the coordinate system, the spindle is always used to determine Z-axis, then X-axis and finally Y-axis.

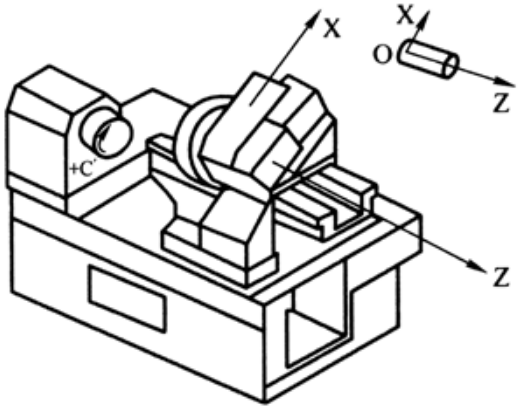
(1) Z coordinate direction: The motion of Z coordinate is determined by the spindle which mainly transmits the cutting power. In the machine with spindle, the coordinate axis parallel to the spindle axis is Z-axis. According to the principle of determining the forward direction in the coordinate system, the direction of the tool away from the workpiece is the forward direction of the axis.

(2) X coordinate direction: The X coordinate direction is generally horizontal and perpendicular to Z-axis. For the machine (such as lathe) that rotates the workpiece, X coordinate direction is defined in the radial direction of the workpiece and is parallel to the cross rail of the lathe. It is also stipulated that the direction of the tool away from the workpiece is the forward direction of X-axis.

When determining X coordinate direction, special attention shall be paid to the difference between the front-mounted tool-holder CNC lathe and the rear-mounted tool-holder CNC lathe

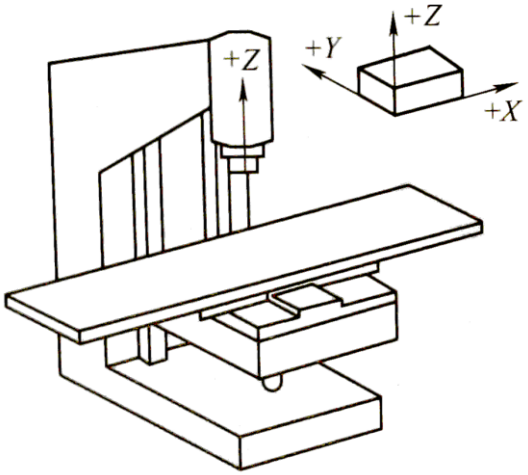


The coordinate system of the CNC lathe with the front turret on the horizontal bed

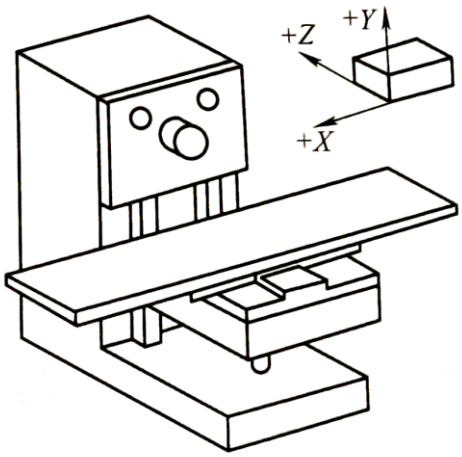


The coordinate system of the CNC lathe with the rear turret on the inclined bed

For a vertical milling machine, the Z-direction is vertical, and in front of the stationmaster's worktable, looking from the tool spindle to the column, the horizontal right direction is the positive direction of X-axis. For Z-axis being horizontal, the direction to the right is the forward direction of X-axis when it is viewed from the spindle to the workpiece (that is, from the machine back to the workpiece).



Vertical knee type milling machine



Horizontal knee type milling machine

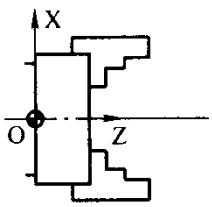
(3) Y coordinate direction: The Y coordinate is perpendicular to the X and Z coordinate axes and is determined according to the right-hand Cartesian coordinate system.

When determining the coordinate axes of the coordinate system, the spindle is always used to determine Z-axis, then X-axis and finally Y-axis

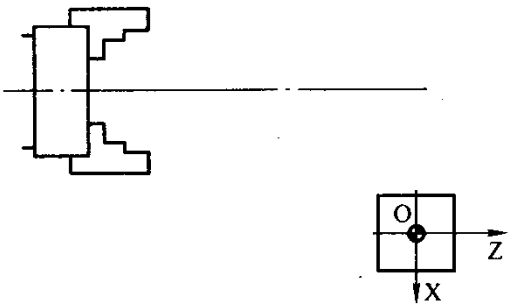
(4) Rotation axis direction: The rotating coordinates A, B, C indicates that their axes are parallel to the rotating coordinates of X, Y, Z coordinate axes respectively. The forward directions of A, B, C coordinates are defined in the directions along X, Y, Z coordinates respectively and the right-hand thread forward direction.

1.3.4 Machine origin and reference point

(1) Machine origin: The machine origin (also known as machine zero) is a fixed point set on the machine, that is, the origin of the machine coordinate system. It has been adjusted well when the machine is assembled and commissioned. Usually, users are not allowed to make any changes, thus it is a fixed point. The machine origin is the datum point of machining movement of CNC machine. Some CNC machines set the machine origin on a certain feature point, for example: the machine origin of milling machine (machining center) is set in the center of the worktable, and that of CNC lathe is set in the center of the chuck. Some CNC machines also set the machine origin at the forward limit point of the tool or tool holder displacement.



The machine origin is at the center of the chuck



The machine origin is at the positive limit point of the tool holder

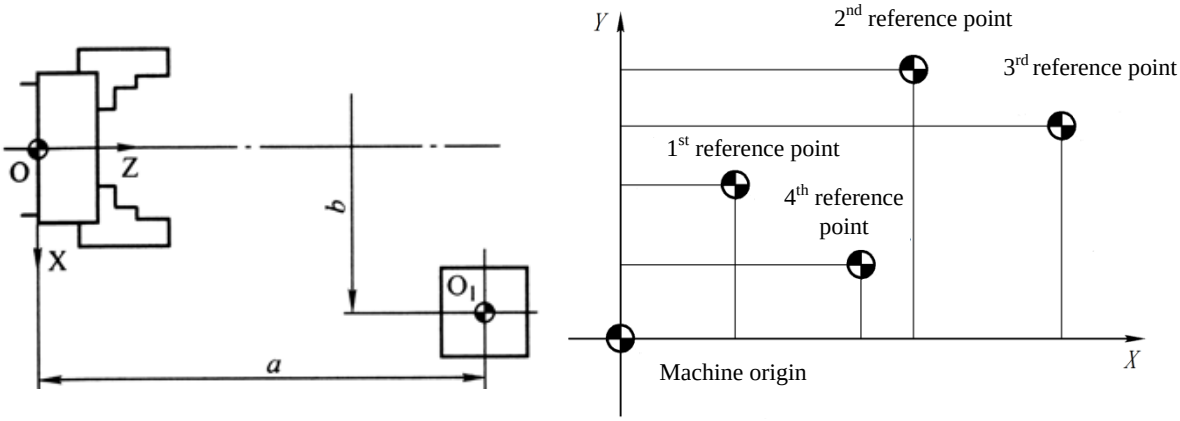
(2) Machine reference point: The machine reference point is a special physical position on the CNC machine.

Generally, the first reference point of CNC machine is located at the limit point of forward movement of the tool (tool holder), and its specific position is determined by the mechanical document block. The distance between the machine reference point and the machine origin is set by the system parameter #1050, and its value can be zero. If the value is zero, it means that the machine reference point and the machine zero coincide.

For CNC machine with incremental encoder, the first step in starting the machine is always to make the machine return to reference point (the so-called machine back-to-zero). When the machine is at the reference point, the machine coordinate system on the system display displays the values set in the system (i.e., the distance between the reference point and the machine origin). The purpose of starting up the machine for homing is to establish the machine coordinate system, that is, the machine origin position is deducted through the current position of the reference point and the distance between the reference point and the machine origin set in the system parameter. Once the machine coordinate system is established, as long as the machine is continuously powered, it will remain unchanged forever and cannot be changed through programming.

For CNC machine with absolute encoder, the machine coordinate system can be read at startup, and the step of machine reference point return can be omitted. The machine origin is set by the system parameter. Once it is set, the machine coordinate system is established. Even if the machine is powered off, the machine coordinate system will not be lost.

The machine origin is O, the machine reference point is O1, a is the distance parameter value in the Z-direction, and b is the distance parameter value in the X-direction. In addition to the reference points established on the machine, parameters can also be used to set the 2nd, 3rd, and 4th reference points. The purpose of establishing these reference points is to establish a fixed point at which the CNC machine can perform some special operations such as tool change.



Machine origin and reference point

1.4 Workpiece coordinate system

Function and purpose

1.4.1 Workpiece coordinate system

The establishment of the machine coordinate system ensures the correct movement of the tool on the machine. However, machining programs are usually written for a workpiece and carried out based on part drawing. In order to facilitate dimensioning and inspection, the coordinate origin of the machining program is generally consistent with the dimensional datum of the part drawing as far as possible. This kind of coordinate system is called workpiece coordinate system (also called programming coordinate system) which is established for a

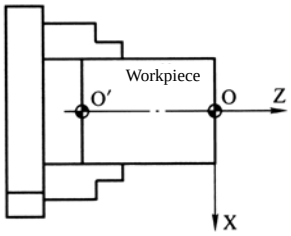
workpiece and based on the part drawing.

1.4.2 Workpiece coordinate system origin

The workpiece coordinate system origin is also called programming origin, which means that after the workpiece is assembled and clamped, a certain point on the workpiece is selected as the datum point for programming or workpiece machining. The origin of the workpiece coordinate system is represented by the symbol "◔" in the figure.

The origin of CNC lathe workpiece coordinate system in the X-direction is generally selected at the workpiece rotation center, while that in the Z-direction is generally selected at the right (point O) or left end face (point O') of machining workpiece.

Using the left end face as the workpiece origin in the Z-direction is beneficial to ensure the total length of the workpiece. However, using the right end face as the workpiece origin in the Z-direction is beneficial to set the tool



1.5 Structure of program

Function and purpose

In CNC machining, the composing forms of part machining program vary slightly with the strength of CNC system functions, and the CNC system machining programs with stronger functions can be divided into main program and subprogram.

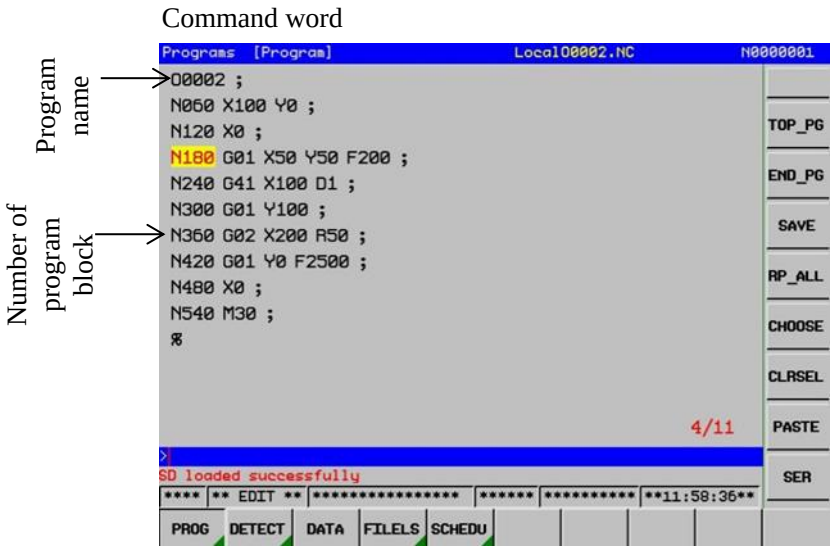
Structure form of main program and subprogram

Main Program	**Subprogram**
O2001; N10 G21 G40 G00; N20 G28 X0 Z0; N30 S2000 M03; N40 T0101; …… N70 M98 P2002 L3; or M98 P032002; N80 G00; …… N110 M09; N120 G91 G28 Z0; M30	O2002; N10 G90 X12 Z-12 R3 F200; N20 X10 R3; N30 X8 R3; …… N50 M99;

Detailed description

Whether it is main program or subprogram, a complete program consists of three parts: program name, program content and program end. The content of the program is composed of several program blocks. The program block is composed of a number of words, while each word is composed of letters and numbers. That is, letters and numbers constitute word, words constitute program block, and program blocks constitute program.

Note: Full-shaped and English punctuation must be used for input method.



Structure of program

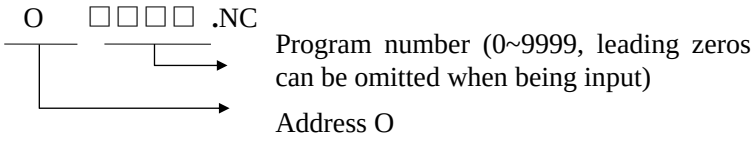
1.5.1 Program name

In the system, multiple programs can be stored in the system memory. To distinguish programs in memory, a program name composed of a sequence of letters and numbers can be used with the suffix .NC.

Such as: YePian1.NC

Ball123.NC

When it is used for called program (e.g., M98), address O and the next four digits shall be used to constitute the program name.



Composition of program name

Note: Chinese program names are not supported.

1.5.2 Content of program

The content of program is the single core of the whole program and is composed of many program blocks, and each program block is composed of one or more commands, representing all the actions to be completed by the CNC machine.

1.5.3 End of program

The end of program is achieved through M command, which must be written at the end of the program. M commands that can be used as program end tag include M02 and M30, representing the end of the part machining

main program. To ensure proper execution of the final program block, it is usually required that M02 (M30) must also have an exclusive line. In addition, the end of the subprogram has a special end tag, and the system use M99 to indicate that subprogram returns to the main program after the end.

1.6 Composition of program block

1.6.1 Basic format of program block

Program blocks are the essential components of a program, each program block is composed of several data words, and the data words are composed of English letters, special characters and numbers representing the address. Such as X30.0, G50, etc.

A program block contains all the data required to execute a process. When there are many commands in the program block, the sequence of word-address program block format is as follows:

N_ G_ X_ Y_ Z_ I_ J_ K_ P_ Q_ R_ A_ B_ C_ F_ S_ T_ M_ ;

Program Section No.	Preparation Function	Dimension Word	Feed Function	Spindle Function	Tool Function	Auxiliary Function	End Label

The commands contained in the above program block are not mandatory in every program block of the machining program, but are programmed with appropriate command according to the specific function of each program block.

Such as: N20 G01 X35.2 Z-46.8 F120;

List of Meanings of English Letters in Addresses

Address	Value Range	Functional Meaning
O	0～9999	Program name
N	0～999999	Sequence number
G	0～999	Preparatory function
X	±999999.9999(mm) Diameter programming: ±1999999.9998(mm)	X-direction coordinate address
	0～9999.999(s)	Dwell time specification
Y	±999999.9999(mm)	Y-direction coordinate address
Z	±999999.9999(mm)	Z-direction coordinate address
U	±999999.9999(mm)	Increment coordinate value of X-axis
	0～9999.999(s)	Dwell time
	±99.9999(mm)	X-axis fine machining allowance in G71, G72, G73 commands
	0.001～99.9999(mm)	Cutting depth in G71
	±999999.9999(mm)	Tool retracting distance for X-axis in G73
V	±999999.9999(mm)	Increment coordinate value of Y-axis
W	±999999.9999(mm)	Increment coordinate value of Z-axis
	±99.9999(mm)	Z-axis fine machining allowance in G71, G72,

Address	Value Range	Functional Meaning
		G73 commands
	0.001～99.9999(mm)	Cutting depth in G72
	±999999.9999(mm)	Tool retracting distance for Z-axis in G73
R	±999999.9999(deg)	Angular displacement amount
	±999999.9999(mm)	Arc radius/R plane in fixed cycle
	0~99999.999(mm)	G71, G72 cyclic tool retraction amount
	1～99999	G73 rough turning cycle stratification times
	0~99.9999(mm)	G74, G75 cutting retract amount
	0~999999.999(mm)	G74, G75 cutting bottom tool retraction amount
	0~99.999(mm)	Fine machining allowance in G76
	±999999.9999(mm)	Taper in G90, G92, G94, G76
	±999999.9999(mm)	Screw pitch increment in G34 thread cutting
I	±999999.9999(mm)	Relative start point of the arc center in the X-axis vector
J	±999999.9999(mm)	Relative start point of the arc center in the Y-axis vector
K	±999999.9999(mm)	Relative start point of the arc center in the Z-axis vector
	0～9999	Number of repeats in a fixed cycle
A	±999999.9999(deg)	Rotate around the X-axis
B	±999999.9999(deg)	Rotate around the Y-axis
C	±999999.9999(deg)	Rotate around the Z-axis
E	0.00005~999999.9999(Tooth/Inch)	Number of imperial thread teeth
F	0.01～200000(mm/min)	Feed speed per minute
	0.001～1000 (mm/r)	Feed speed per revolution
	0.01～500 (mm/r)	Screw pitch
S	0～99999.9999(r/min)	Spindle rotational speed specification
	0～06	Multi-spindle output
T	0～9999	Tool function
M	0～999	Auxiliary function output, program execution flow, subprogram call
P	0～999999(ms)	Dwell time
	0～9999	Call subprogram number
	0～999	Number of subprogram calls
	1～99999999(0.001mm) 0～99999.999(mm)	X-axis tool infeed amount in G74, G75
	1～999999	Height of thread in G76
	0~999999	Start program block number of combined cycle command fine machining program block
Q	±999999.9999(mm)	Cutting depth or hole bottom offset in the fixed cycle
	0~999999	Start program block number of combined cycle command fine machining program block

Address	Value Range	Functional Meaning
	0~9999999 (0.001mm)	Z-axis tool infeed amount in G74, G75
	0~99999 (0.001mm)	Minimum cut depth in G76
	0~999999 (0.001mm)	First cut depth in G76
H	0～400	Length offset number
	±999999.9999(mm)	Increment coordinate value of C-axis
D	0～400	Radius offset number
L	0～999	Number of repeats of subprogram

Please note that the value scope shown in the table are the limit for CNC device, except for machine. Therefore, when programming, the specification provided by machine manufacturer shall be referred in addition to referring to this manual, with programming carried out in a good understanding of the limitations of programming.

1.6.2 Composition of program block

(1) Program block number. The program block number begins with the address character "N" followed by several numbers.

In the system, the program block number is only used as a target position indication for "Jump" or "Program Retrieval". Therefore, its size and order can be reversed or omitted. The program blocks are arranged in order of input in memory, and the programs are executed strictly in order of sequence in the memory of the information, that is to say, the sequence of program execution in the system is independent of the sequence number of the program block. However, when a program block sequence number is omitted, the program block will not be regarded as the target program block for "Jump" or "Program Retrieval".

It is generally not recommended to use N0.

The program block number can also be automatically generated by the CNC system, and the incremental value of the block number can be set through "machine parameters", generally, the incremental value can be set to 10.

(2) Content of program block. The content of the program block lies in the middle part of the program block. The program content shall have 6 basic elements, namely, preparatory function word, dimension function word, feed function word, spindle function word, tool function word and auxiliary function word.

(3) End of program block. The program block ends with the end tag ";".

1.6.3 Slash block skip of program

Sometimes the program block is preceded by a "/" symbol called a slash block skip symbol, and the program block is called a block-skip program block. Such as following program blocks:

/N10 G00 X100.0;

For such program blocks, the operator can control the program block and execution. When the "skip program block" signal of the system is activated by operating the machine, these program blocks will be skipped when the program is executed; when the "skip program block" function is invalid, the block executed as usual will be identical to the program block without the "/" sign.

Caution:

Slash (/) must be specified at the beginning of the program block. If the slash (/) is placed at the first place of the non-program block, the system alarm "macro program expression format error" will

appear.

1.6.4 Annotation for program block

In order to check and read the CNC program conveniently, it is allowed to comment on the program in the CNC system. The comment can be displayed on the screen as a prompt to the operator, but the comment has no influence on the machine action.

The annotation for the program block shall be placed at the end of the program and is not allowed to be inserted between addresses and numbers. The program annotation of the system shall be enclosed with "()".

Note: Full-shaped method must be used for inputting Chinese annotation.

1.6.5 System macro program folder

In order to facilitate the management of the processing program, the system macro program (SYSNC), the machine factory macro program (OEMNC), and the user program are classified by means of folders. The function of creating a new folder can be performed on the program list screen, and various machining programs can be operated in this folder.

The programs stored in the system macro program (SYSNC) are O9000~O9999.

The programs stored in the system macro program of the machine factory (OEMNC) are O8000~O8999.

User programs can be placed in a custom folder or on the first-level screen. The default program list screen at power-on is the currently loaded folder.

Remarks:

1. Import and export system macro program (SYSNC) needs to open parameter 1610 (NE9).
2. Importing and exporting machine macro programs (OEMNC) requires opening parameter 1610 (NE8) and setting the interface to log in to the NC8xxx password.
3. When importing a folder, the content format of the folder is automatically checked, and it cannot contain non-suffixed .NC and files larger than O8000.NC (automatically ignore files that do not meet the conditions or generate a prompt message).
4. The fixed path gcode/SYSNC/... of the subprogram calling O9000~O9999.
5. The fixed path gcode/OEMNC/... of the subprogram calling O8000~O8999.
6. The fixed path for calling subprograms below O8000 is the current folder (that is, the folder where the main program is located).
7. Programs O8000~O9999 cannot be created in the part program folder.
8. In DNC mode, call subprograms (O0000~O7999) with fixed path gcode/....

Chapter II Preparation of Function Command Codes

2.1 Modal and non-modal

Function and purpose

(1) G code is a command that assigns the action mode of each program block within the program.
(2) G code is divided into modal command and non-modal command.
(3) Modal command usually specifies 1 G code as 1 NC action mode in G code within a group. This action mode shall be kept until the command is cancelled or other G codes within the same group are re-specified.
(4) It is NC action mode command only when non-modal command is specified. It is invalid for next program block.

2.2 Coordinate function

Function and purpose

Coordinate function word (also called dimension function word) is used to set the displacement amount of each coordinate of machine. Addresses including X, Y, Z, U, V, W, P, Q, R (for specifying rectilinear coordinate dimension) and A, B, C, D, E (for specifying angle coordinate) and I, J, K (for specifying center coordinate point position dimension) are used at the beginning, with address character preceded by a string of numbers with "+" or "-".

2.3 List of preparatory function command G code

2.3.1 Milling preparatory function G code

List of Milling Preparation Function G Code

G Code	Group	Program Format and Description	Function
G00▲	01	G00 X_Y_Z_	Fast point positioning
G01▲		G01 X_Y_Z_F_	Linear interpolation
G02		G02 X_Y_ R_ F_;	Clockwise circular interpolation
G03		G03 X_Y_ I_J_ F_;	Counterclockwise circular interpolation
G04	00	G04 P_ Or G04 X_	Dwell; orientation
G05		G05 P_	High-speed and high-precision contour control
G07.1 (G107)	00	G07.1 X_Y_Z_r	Cylindrical interpolation
G08		G08 Q_	Pre-read program block preset function
G09		G09	Accurate stop
G10		G10 L_N_P_R_	Programmable data input

G Code	Group	Program Format and Description			Function
G11▲		G11			Programmable data input cancelled
G15▲	14	G15			Polar coordinate cancelled
G16		G16			Polar coordinate command
G17▲	02	G17			Select XY plane
G18▲		G18			Select ZX plane
G19▲		G19			Select YZ plane
G20	06	G20			Inch input
G21		G21			Millimeter input
G22	04	G22 X_Y_Z_I_J_K_			Storage travel detection function is valid
G23▲		G23			Storage travel detection function is invalid
G27	00	G27	X_Y_Z_		Reference point return detection
G28		G28			Reference point return
G29		G29			Homing from reference point
G30		G30 Pn			Return to the 2nd, 3rd and 4th reference points
G31		G31			Jump function
G33	01	G33	Z_ (X_ Y_ α_) F/E_ Q_		Thread cutting
G37	00	G37	X_Y_Z_		Automatic tool length measurement
G39	00	G39 X_Y_			Corner switching
G40▲	07	G17	G40 G41 G42	X_Y_	Tool radius compensation cancelled
G41		G18		X_Z_	Tool radius compensation on the left side
G42		G19		Y_Z_	Tool radius compensation on the right side
G43	08	G43		Z_	Tool length compensation in positive direction
G44		G44			Tool length compensation in negative direction
G49▲		G49			Tool length compensation cancelled
G43.4		G43.4 X_Y_Z_α_β_ H_;			Tool center point control
G45	00	G45		X_Y_Z_D_	Tool offset value increase
G46		G46			Tool offset value decrease
G47		G47			2 times tool offset value increase
G48		G48			2 times tool offset value decrease
G50▲	11	G50			Scaling cancelled
G51		G51 X_ Y_ Z_ P_			Scalling

G Code	Group	Program Format and Description	Function
G50.1	21	G50.1 X_Y_Z_	Programmable mirror cancelled
G51.1		G51.1 X_Y_Z_	Programmable mirror valid
G52	00	G52 X_Y_Z_	Set local coordinate system
G53		G53	Select machine coordinate system
G53.1		G53.1	Tool axis direction control
G54▲	17	G54	Workpiece coordinate system 1
G55		G55	Workpiece coordinate system 2
G56		G56	Workpiece coordinate system 3
G57		G57	Workpiece coordinate system 4
G58		G58	Workpiece coordinate system 5
G59		G59	Workpiece coordinate system 6
G54.1		G54.1 P_	Additional workpiece coordinate system
G60	00/01	G60 X_Y_Z_	Unidirectional positioning
G61	15	G61	Orientation mode
G62		G62	Automatic corner override
G63		G63	Tapping mode
G64▲		G64	Cutting mode
G65	00	G65 P_L_＜IV assignment＞	Macro program call
G66	12	G66 P_L_＜IV assignment＞	Macro program modal call
G67		G67	Macro program modal call cancelled
G68	16	G68 X_ Y_ R_	Coordinate rotation
G68.2		G68.2 X_ Y_Z_I_J_K_	Inclined surface machining
G69▲		G69	Coordinate rotation cancelled
G73	09	G73 X_Y_Z_R_Q_F_	Deep hole drilling cycle
G74		G74 X_Y_Z_R_P_F_	Left-hand tapping cycle
G76		G76 X_Y_Z_Q_R_P_ F_K_	Fine boring cycle
G80▲		G80	Fixed cycle cancelled
G81		G81 X_Y_Z_R_F_	Drilling cycle (spot drill cycle)
G82		G82 X_Y_Z_R_P_F_	Drilling cycle (boring ladder hole cycle)
G83		G83 X_Y_Z_R_Q_F	Deep hole drilling cycle
G84		G84 X_Y_Z_R_P_F_	Tapping cycle
G85		G85 X_Y_Z_R_F_	Boring cycle
G86		G86 X_Y_Z_R_F_	Boring cycle
G87		G87 X_Y_Z_R_Q_P_F_	Reverse boring cycle

G Code	Group	Program Format and Description	Function
G88		G88 X_Y_Z_R_P_F_	Boring cycle
G89		G89 X_Y_Z_R_P_F_	Boring cycle
G90▲	03	G90	Absolute value programming
G91▲		G91	Incremental value programming
G92	00	G92 X_Y_Z_α_β_	Coordinate system setting
G92.1		G92.1 X0Y0Z0α0β0	Workpiece coordinate system recovery
G94▲	05	G94	Feed per minute (FRM)
G95		G95	Feed per rotation
G98▲	10	G98	Return to the initial plane in the fixed cycle
G99		G99	Return to R point (in the fixed cycle)

Precautions

(1) The code identified by the symbol "▲" in the table is the default code when the power is turned on.
(2) When specifying a G code that is not in the list. the system displays the "Incorrect G Code" alarm.
(3) Different groups of multiple G codes can be commanded in the same program block. If multiple G codes in the same group are commanded in the same program block, only the last specified G code can be executed.
(4) When the fixed cycle G code and G code of Group 01 are commanded in the same block (excluding the modal G60 command), the fixed cycle will be cancelled and become the G80 state if the G code of Group 01 is commanded in the fixed cycle state (not in the same block).
(5) G code is represented by group number according to different types.
(6) Programming precautions are detailed in: Appendix Table 1 G Code Programming Rules.

2.3.2 Turning preparatory function G code

There are three G code systems: A, B, and C.

List of Turning Preparation Function G Code

G Code			Group	Function
A	B	C		
G00▲	G00▲	G00▲	01	Positioning (quick)
G01▲	G01▲	G01▲		Linear interpolation (cutting feed)
G02	G02	G02		Clockwise circular interpolation
G03	G03	G03		Counterclockwise circular interpolation
G04	G04	G04	00	Dwell
G08	G08	G08		Pre-read program block preset function
G10	G10	G10		Programmable data input
G11	G11	G11		Programmable data input mode cancelled
G18▲	G18▲	G18▲	16	ZX plane selection

G Code			Group	Function
A	B	C		
G20	G20	G70	06	Inch input
G21	G21	G71		Millimeter input
G22	G22	G22	09	Check stored travel is on
G23▲	G23▲	G23▲		Check stored travel is off
G27	G27	G27	00	Reference point return check
G28	G28	G28		Reference point return position
G30	G30	G30		Return to the 2nd, 3rd and 4th reference points
G31	G31	G31		Jump function
G32	G33	G33	01	Thread cutting
G34	G34	G34		Thread cutting with variable screw pitch
G40▲	G40▲	G40▲	07	Tool nose radius compensation cancelled
G41	G41	G41		Tool nose radius compensation on the left side
G42	G42	G42		Tool nose radius compensation on the right side
G50	G92	G92	00	Coordinate system displacement element or maximum spindle speed setting
G92.1	G92.1	G92.1		Workpiece coordinate system recovery
G52	G52	G52	00	Set local coordinate system
G53	G53	G53		Set machine coordinate system
G54▲	G54▲	G54▲	14	Select workpiece coordinate system 1
G55	G55	G55		Select workpiece coordinate system 2
G56	G56	G56		Select workpiece coordinate system 3
G57	G57	G57		Select workpiece coordinate system 4
G58	G58	G58		Select workpiece coordinate system 5
G59	G59	G59		Select workpiece coordinate system 6
G65	G65	G65	00	Macro program call
G66	G66	G66	12	Macro program modal call
G67▲	G67▲	G67▲		Macro program modal call cancelled
G70	G70	G72	00	Finish machining cycle
G71	G71	G73		Axial rough turning cycle
G72	G72	G74		Radial rough turning cycle
G73	G73	G75		Copying turning cycle
G74	G74	G76		Radial cutting cycle
G75	G75	G77		End face cutting cycle
G76	G76	G78		Multiple thread cycle
G80▲	G80▲	G80▲	10	Fixed cycle cancelled
G83	G83	G83		Drilling cycle
G84	G84	G84		Tapping cycle

G Code			Group	Function
A	B	C		
G85	G85	G85		Front boring cycle
G87	G87	G87		Side drilling cycle
G88	G88	G88		Side tapping cycle
G89	G89	G89		Side boring cycle
G90	G77	G20	01	Single cycle for outer/inner diameter turning
G92	G78	G21		Single cycle for thread cutting
G94	G79	G24		Single cycle for end face turning
G96	G96	G96	02	Constant surface cutting speed control
G97▲	G97▲	G97▲		Constant surface cutting speed control cancelled
G98▲	G94▲	G94▲	05	Feed per minute
G99	G95	G95		Feed per rotation
—	G90▲	G90▲	03	Absolute value programming
—	G91	G91		Incremental value programming
—	G98	G98	11	Return to the initial plane
—	G99	G99		Return to R plane

A series of preparatory function commands are commonly used in turning.

List of Series A Preparation Functions

G Code	Group	Function	Program Format and Description
G00▲	01	Fast point positioning	G00 X_Z_
G01▲		Linear interpolation	G01 X_Z_F_
G02		Clockwise circular interpolation	G02 X_Z_R_F_ G02 X_Z_I_K_F_
G03		Counterclockwise circular interpolation	
G04	00	Dwell	G04 X1.5; or G04 U1.5; or G04 P1500
G07.1 (G107)		Cylindrical interpolation	G07.1 IPr (valid); G07.1 IP0 (cancelled)
G10		Programmable data input	G10 P_X_Z_R_Q_
G11▲		Programmable data input cancelled	G11
G12.1 (G112)	21	Polar coordinate command	G12.1 G112
G13.1▲ (G113)		Polar coordinate cancelled	G13.1 G113
G17▲	16	Select XY plane	G17
G18▲		Select ZX plane	G18
G19▲		Select YZ plane	G19
G20	06	Inch input	G20

G Code	Group	Function	Program Format and Description
G21		Millimeter input	G21
G22	09	Storage travel detection is on	G22 X_Z_I_K_
G23		Storage travel detection is off	G23
G25▲	08	Spindle speed fluctuation detection is off	G25
G26		Spindle speed fluctuation detection is on	G26 PpQqRr
G27	00	Reference point return detection	G27 X_Z_
G28		Reference point return	G28 X_Z_
G30		Return to the 2nd, 3rd and 4th reference points	G30 P3 X_Z_
G31		Jump function	G31 IP
G32	01	Thread cutting	G32 X_Z_F_ (F is lead)
G34		Thread cutting with variable screw pitch	G34 X_Z_F_R_
G37	00	Automatic tool length measurement	G37 X/Z_
G40▲	07	Tool nose radius compensation cancelled	G40
G41		Tool nose radius compensation on the left	G41 G01 X_Z_
G42		Tool nose radius compensation on the right	G42 G01 X_Z_
G50	00	Coordinate system shift or maximum speed limit	G50 X_Z_; or G50 S_
G92.1		Workpiece coordinate system recovery	G92.1 IPO
G50.2 (G250)	20	Polygon turning cancelled	G50.2; G250 (Optional function)
G51.2 (G251)		Polygon turning	G51.2 P_Q_; G251 P_Q_ (Optional function)
G52	00	Set local coordinate system	G52 X_Z_
G53		Select machine coordinate system	G53 X_Z_
G54▲	14	Select workpiece coordinate system 1	G54
G55		Select workpiece coordinate system 2	G55
G56		Select workpiece coordinate system 3	G56

G Code	Group	Function	Program Format and Description
G57		Select workpiece coordinate system 4	G57
G58		Select workpiece coordinate system 5	G58
G59		Select workpiece coordinate system 6	G59
G65	00	Macro program non-modal call	<Independent Variable Specification>
G66	12	Macro program modal call 1	<Independent Variable Specification>
G67▲		Macro program modal call cancelled	G67
G70	00	Finish machining cycle	G70 P_Q_
G71		Axial rough turning cycle	G71 U_R_ G71 P_Q_U_W_F_
G72		Radial rough turning cycle	G72 W_R_ G72 P_Q_U_W_F_
G73		Copying turning cycle	G73 U_W_R_ G73 P_Q_U_W_F_
G74		End face cutting cycle	G74 R_ G74 X(U)_Z(W)_P_Q_R_F_
G75		Cylindrical grooving cycle	G75 R_ G75 X(U)_Z(W)_P_Q_R_F_
G76		Multiple thread cutting cycles	G76 Pmra Q_R_ G76 X(U)_Z(W)_P_Q_R_F_L _
G80▲	10	Fixed cycle cancelled	G80
G83		Drilling cycle	G83 X_C_Z_R_Q_P_F_M_
G84		Tapping cycle	G84 X_C_Z_R_P_F_K_M_
G85		Front boring cycle	G85 X_C_Z_R_P_F_K_M_
G87		Side drilling cycle	G87 Z_C_X_R_Q_P_F_M_
G88		Side tapping cycle	G88 Z_C_X_R_F_K_M_
G89		Side boring cycle	G89 Z_C_X_R_P_F_K_M_
G90	01	Single cycle for inner and outer diameter turning	G90 X_Z_F_ G90 X_Z_R_F_
G92		Single cycle for thread cutting	G92 X_Z_F_ G92 X_Z_R_F_
G94		Single cycle for end face turning	G94 X_Z_F_ G94 X_Z_R_F_
G96	02	Constant linear speed	G96 S200 (200m/min)
G97▲		RPM	G97 S800 (800r/min)
G98▲	05	Feed per minute (FRM)	G98 F100 (100mm/min)
G99		Feed per rotation	G99 F0.1 (0.1mm/r)

Precautions

(1) The code identified by the symbol "▲" in the table is the default code for CNC when the power is turned on.
(2) The G codes of 00 groups other than G10 and G11 are non-modal G codes.
(3) When specifying G code that is not in the list, the system displays the "Incorrect G Code" alarm.
(4) Different groups of multiple G codes can be commanded in the same program block. If multiple G codes in the same group are commanded in the same program block, only the last specified G code can be executed.
(5) If the G code of Group 01 is commanded in the fixed cycle, the fixed cycle will be cancelled, which is the same as command G80.
(6) G code is displayed by group number.

Chapter III Position Command

3.1 Absolute value and incremental value programming (G90, G91)

Function and purpose

According to G90, G91 commands or address words X, Z, U, W, a coordinate command may be specified as an absolute value or incremental value command, but the arc radius specified by R or the circular arc center specified by I, J, K is usually increment specification.

Milling G Code System	
Command method	G90,G91

Turning G code system	A	B or C
Command method	Address word	G90,G91

Command format

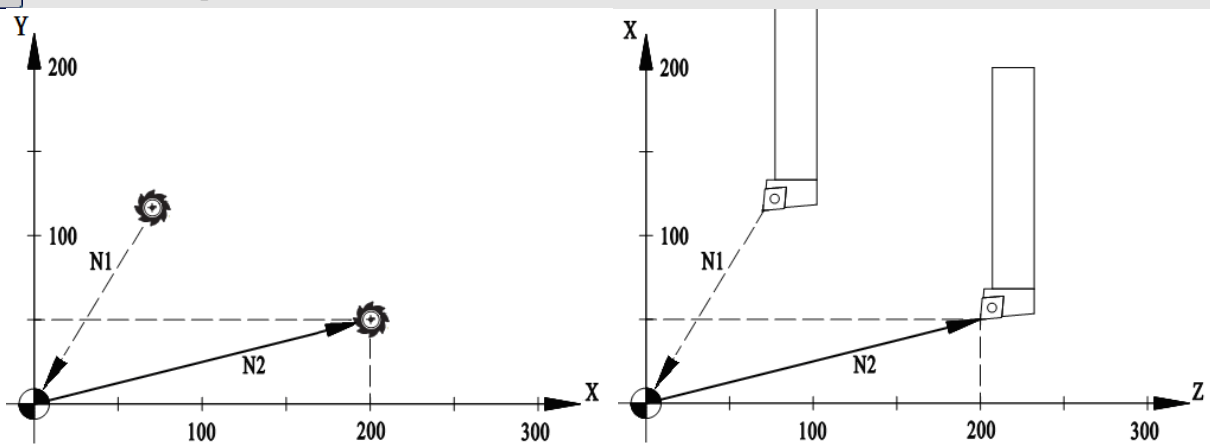
Turning G code system A

	Absolute Value Command	Increment Command
X-axis movement command	X	U
Z-axis movement command	Z	W
Y-axis movement command	Y	V
C-axis movement command	C	H

Milling G code system and turning G code system B or C

G90/G91 X_{X1} Y_{Y1} Z_{Z1} $\alpha_{\alpha 1}$;
G90 : Absolute value command
G91 : Incremental value command
α : Additional axis

Detailed description



(1) In absolute value mode, the movement is independent of the current position and is based only on the position of the workpiece coordinate system specified by the program.

Milling G code system	Turning G code system	
	A	B or C
N1 G90 G00 X0 Y0	N1 G00 X0 Z0	N1 G90 G00 X0 Z0

In incremental mode, it moves according to the relative value specified by the program with the current position as the start point.

Milling G code system	Turning G code system	
	A	B or C
N2 G90 G01 X200 Y50 F100	N2 G01 X50 Z200	N2 G90 G01 X50 Z200 F100
N2 G91 G01 X200 Y50 F100	N2 G01 U50 W200	N2 G91 G01 X50 Z200 F100

The command is issued from the origin point of the workpiece coordinate system, and the coordinate command value is the same regardless of absolute value or incremental value mode.

(2) For the movement of the next program block, the G90 or G91 mode of the preceding program block is determined in the milling G code system and turning G code system B or C. In System A of turning G code, it is determined based on the address word of the current block.

(G90)

Milling G code system	Turning G code system	
	A	B or C
N3 X100 Y100	N3 X100 Z100	N3 X100 Z100

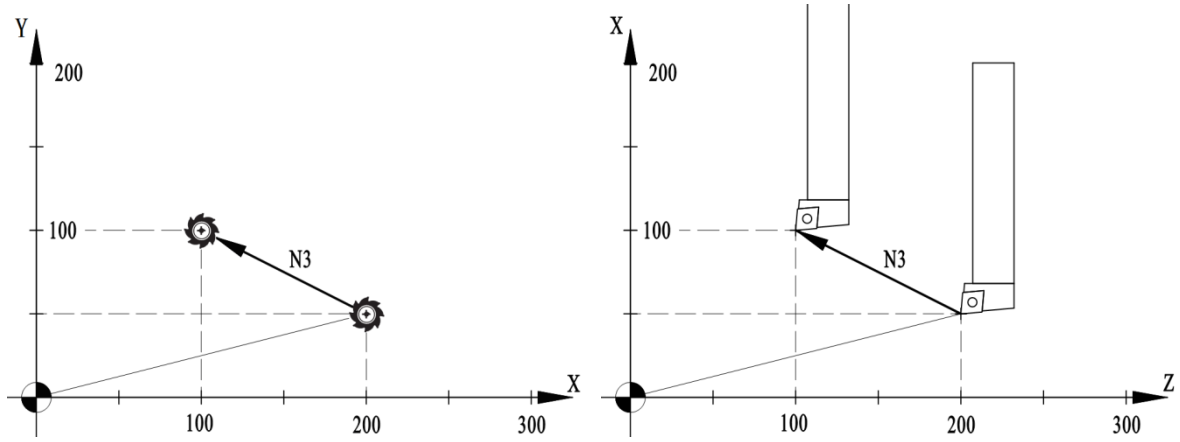
The horizontal axis and the vertical axis of workpiece coordinate system move to 100 mm respectively.

(G91)

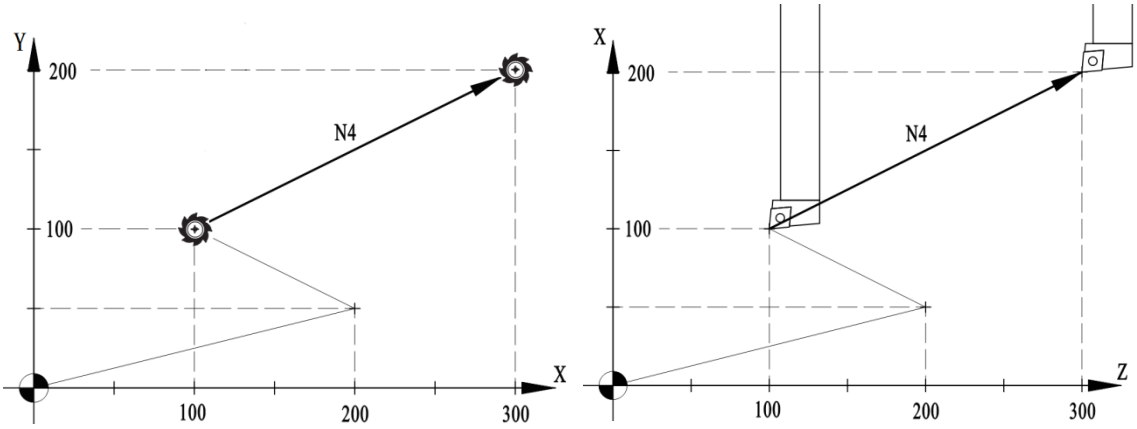
Milling G code system	Turning G code system	
	A	B or C
N3 X-100 Y50	N3 U50 W-100	N3 X50 Z-100

When it is the incremental value, the abscissa axis moves to -100 mm and the vertical axis moves to +50

mm.



(3) In the same program, a specific position can be specified by using a mixture of absolute value and incremental value.



Milling G Code System	Turning G Code System	
	A	B or C
N4 G91X200 G90Y200 (Optional function)	N4 X200 W200	N4 G90X200 G91Z200 (Optional function)

The vertical axis moves to the position of 200 mm in the workpiece coordinate system in an absolute value mode; the abscissa axis moves in incremental value mode in the direction of +200 mm, obtaining a final result of 300 mm. In G code system of turning B or C and G code system of milling, the position command mode is G91 increment mode after this program block is executed.

(4) When the power is turned on, the initial state is absolute value or incremental value mode, which is determined depending on the parameter setting.

(5) G90/G91 mode continues to be valid from this program block when the operation is entered in input mode (MDI).

3.2 Imperial/metric conversion (G20, G21)

Function and purpose

Imperial command and metric command can be switched according to G command.

Command format

G20/G21; G20 : Imperial command G21 : Metric command

Detailed description

The switching of G20 and G21 corresponds only to the linear axis rather than the rotation axis.

G20 and G21 will switch programming command unit, tool offset, position display, coordinate system unit and value. System parameter and screw pitch compensation setting unit are still metric in operation. Please note the set value.

Enter the relationship between the command unit and G20/G21

Axis	Command Example	G21	G20
X	X0.1	0.1mm	2.54mm
Y	Y0.1	0.1mm	2.54mm
Z	Z0.1	0.1mm	2.54mm
C	C1.0	1.0 º	1.0 º

About output unit, command unit and setting unit element

Item	G21	G20
Moving speed command	Metric	Imperial
Counting display	Metric	Imperial
Speed display	Metric	Imperial
User parameter setting and display	Metric	Metric
Workpiece, tool offset setting and display	Metric	Imperial
MPG feed command	Metric	Imperial

Precautions

When inch is commanded, the parameters are executed in internal unit mm.

3.3 Decimal point input

Function and purpose

This function provides decimal command input, which can be used in imperial or metric units to define tool path, distance, speed and machining program data.

Parameter "#1800 bit0" allows you to choose whether to use the smallest movement unit without decimal data as the minimum input command unit (type 1) or to use mm/sec as the real number for the command unit (type 2).

Detailed description

(1) Decimal point commands are only valid for processing program data such as distance, angle, time, speed, and scaling factor.

(2) For the number of significant digits in decimal place, 3 decimal places are valid when parameter #1000/bit1=0, while 4 decimal places are valid when parameter #1000/bit1=1.

(3) Addresses of decimal point command include X, Y, Z, U, V, W, A, B, C, D, E, F, P, Q, R, etc. However, P is only valid at scaling factor. Please refer to the list for details.

(4) Decimal point command is also valid for variable data definition of subprogram.

(5) Decimal point command for an invalid decimal point address means that data after the decimal point is ignored, but only the integer part is valid and the invalid decimal point address is as follows:\[D, H, L, M, N, O, S, T]. Variable commands, however, are all processed as data with decimal points.

Decimal point command is valid or invalid

In the following list, when a command that does not use a decimal point is issued at an address where the decimal point command is valid, the decimal point input I and II act as follows. When a command that use the decimal point is issued, the decimal point input I and II do the same act.

(1) Decimal point input I

The lowest bit of command data is consistent with the command unit.

(Example) When "X1" is specified in the system, it is the same as the command "X0.001" specified.

(2) Decimal point input II

The lowest bit of command data is consistent with the position of the decimal point.

(Example) When "X1" is assigned in the system, it is the same as the command "X1.".

Validity or invalidity of decimal point

Address	Decimal Point	Use	Address	Decimal Point	Use
A	Yes	Coordinate value	L	No	Number of repetitions of subprogram execution
	No	Auxiliary function code of rotation axis	M	No	Auxiliary function code
	Yes	Angle value	N	No	Sequence number
	No	Data setting, axis number	O	No	Program number
B	Yes	Coordinate value	P	No	Dwell time
	No	Auxiliary function code of rotation axis		No	Subprogram call number
C	Yes	Coordinate value	Q	Yes	Cutting amount of deep hole drilling
	No	Auxiliary function code of rotation axis		Yes	Displacement of reverse boring
D	No	Correction amount (tool position, diameter)		Yes	Displacement of fine boring
	Yes	Tool automatic length measurement, deceleration area		No	Minimum limiting speed of spindle
	No	Data settings		Yes	Displacement angle of start point of lathe teeth

Address	Decimal Point	Use	Address	Decimal Point	Use
E	Yes	Imperial thread	R	Yes	Fixed cycle R point
F	Yes	Feed rate		Yes	R specifies arc radius
	Yes	Thread lead		Yes	Arc radius of fillet
G	Yes	Preparatory function code	S	No	Spindle function code
H	No	Correction amount		No	Maximum limiting speed of spindle
	No	Selection of reference spindle		No	Linear speed control
I	Yes	Circular arc center coordinate	T	No	Tool function code
	Yes	Tool radial offset vector component	U	Yes	Coordinate value
	Yes	Vector of tool axis	V	Yes	Coordinate value
J	Yes	Circular arc center coordinate	W	Yes	Coordinate value
	Yes	Tool radial offset vector component	X	Yes	Coordinate value
	Yes	Vector of tool axis	Y	Yes	Coordinate value
K	Yes	Circular arc center coordinate	Z	Yes	Coordinate value
	Yes	Tool radial offset vector component			
	Yes	Vector of tool axis			

Note: The independent variable and decimal point of macro program are all valid.

3.4 Diameter and radius programming

Function and purpose

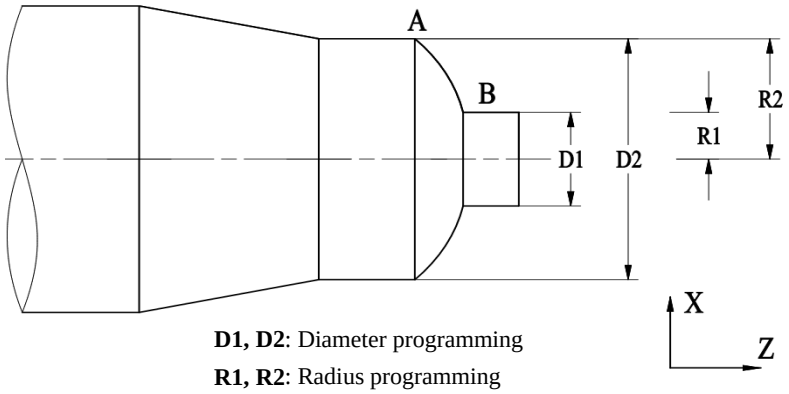
The cross-section of the workpiece is generally circular in the CNC lathe control program, thus its dimension can be commanded in two ways: diameter and radius

When using diameter command, it is called diameter programming; when using radius command, it is called radius programming.

Detailed description

Radius programming or diameter programming is set by the 3rd bit (DIA) of parameter #3001. When using diameter programming, the diameter programming is converted into a radius value within the system, and the pulse equivalent is not halved.

The description of the X-axis travel range in this manual shall be subject to radius programming mode.



D1, D2: Diameter programming

R1, R2: Radius programming

Precautions on command diameter

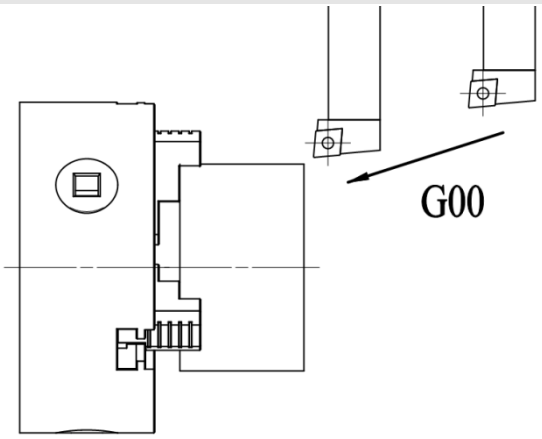
Item	Description
X-axis command	Specified with diameter value
Increment command	Specified with diameter value. In the above figure, the tool path B to A is assigned by D2 minus D1
Coordinate system shifting (G50)	Specify coordinate value with diameter value
Fixed cycle parameters, such as cutting-in along the X-axis (R)	Radius value specification
Radius in circular interpolation	Radius value specification
Feed speed along the axis	Change in specified radius/rotation or change in radius/minute
Axis position display	Displayed by diameter value

Chapter IV Interpolation Function

4.1 Fast moving (G00)

Function and purpose

This command is accompanied by a coordinate command, with the current point as the start point, the coordinate command value as the end point and a linear or non-linear path as positioning.



In the non-cutting state of CNC machine, the movement of the motion axis is usually realized by the fast moving command G00. Under this command, the moving parts (worktable, tool holder, etc.) move quickly at the speed set by the machine, reducing the auxiliary time of non-cutting and improving the effective cutting efficiency of the machine.

Command format

Milling

G00 X_ Y_Z_ α_;(α is the additional axis)

X,Y,Z,α	Represent a coordinate value. At this time, with the state of G90/G91, it appears as an absolute position or an increment position

Turning

G00 X(U)_ Z(W)_ α_;(α is the additional axis)

X(U)_	X-axis end point coordinate (X is the absolute value of the workpiece coordinate system, U is the incremental value from the current position)
Z(W)_	Z-axis end point coordinate (Z is the absolute value of the workpiece coordinate system, W is the incremental value from the current position)

Detailed description

(1) When G00 command is used, the feed speed does not need to be programmed but is specified by machine parameter. G00 is a modal command with a persistent validity function until being specified by G01, G02, or G03 command of Group 01. When the G00 modal is valid, subsequent G00 command only needs to specify coordinates.

(2) Coordinate addresses can be omitted in part or in whole, and omitted addresses indicate that the coordinate value of the start point is consistent with that of end point of the axis. The latter is valid when

X and U, Z and W are in the same block.

(3) The tool path of turning system has linear or non-linear distinction and is selected according to parameter #1401.0_LRP, but the positioning time remains unchanged. The milling system has no non-linear positioning mode.

① Linear positioning: The path is the same as the linear interpolation mode (G01), and the speed is limited by the fast feed speed of each axis.

② The turning system has non-linear positioning: Each axis is independent, moving at maximum speed for positioning.

(4) G00 command fast-moving speed is set by the machine manufacturer in parameter #1226 for each axis.

(5) When G00 command is in the non-linear path, the actual path of the tool between two points is not always linear because of the number of axes programmed, the actual moving length of each axis and the fast-moving speed of each axis. Therefore, attention shall be paid to tool interfering with the workpiece and fixture when G00 command is used.

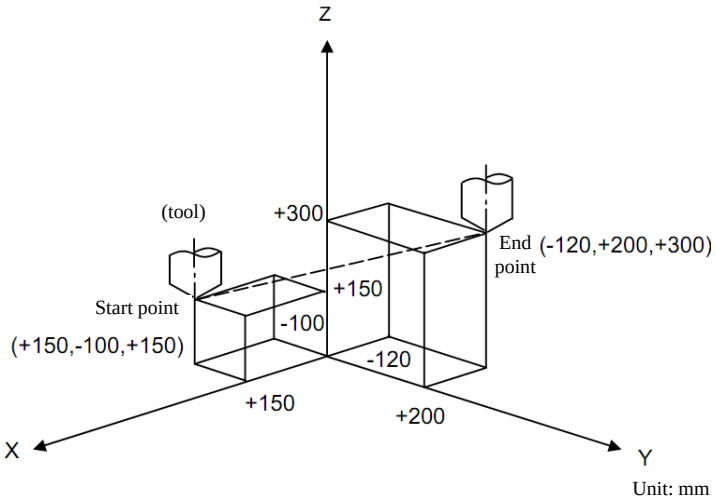
Programming examples

Milling

When the following programs are executed, the tool movement path is shown in the figure.

G92 X150 Y-100 Z150;

G91 G00X-270 Y300 Z150;

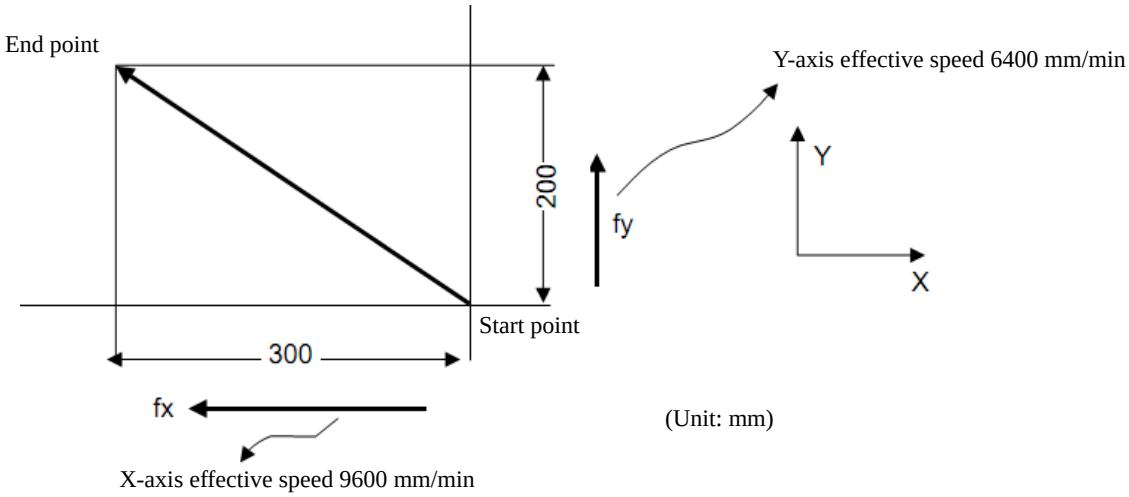


The tool movement path is the shortest path connecting the start point and end point. The positioning speed is calculated automatically to ensure the shortest allocation time, provided that the speed of each axis at the specified positioning speed does not exceed the fast feed speed.

For example, when the fast feed speed of X-axis and Y-axis is 9600 mm/min

G91 G00 X-300 Y200;

The program is edited, and the tool moves along the path shown in the figure below.



Turning non-linear path

As shown in the figure, if the X-axis fast feed speed is 3000 mm/min, and that of Z-axis is 6000 mm/min, the tool is moved from the start point A to the point C using the fast-moving command, with the program as follows.

G50 X80.0 Z222.0: Shift of workpiece coordinate system

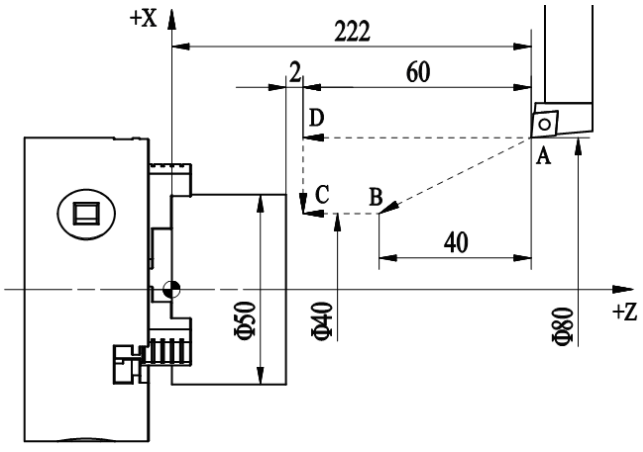
G00 X40.0 Z162.0: (or U-40. 0 W-60. 0) Tool moving path is A → B → C

When the above program block is executed, the actual movement path of the tool is not a straight line, but a polyline. First, the tool moves to point B at a fast resultant speed, and then moves to point C along the Z-axis. In practical application, the programming method of single axis moving separately is often adopted, with the program as follows.

G50 X80.0 Z222.0: Shift of workpiece coordinate system

G00 Z162.0: (or W-60. 0) The tool moves along Z-axis A→D

G00 X40.0: (or U-40. 0) The tool moves along Z-axis D→C



4.2 Linear interpolation (G01)

Function and purpose

Linear interpolation command G01 commands the tool to make arbitrary linear motion between two coordinate points in the way of interpolation linkage at the specified feed speed. This command is a modal

command.

Command format

Milling

G01 X_ Y_Z_ α_ F_; (α is the additional axis)

X,Y,Z,α	Represent a coordinate value. At this time, with the state of G90/G91, it appears as an absolute position or an increment position
F	Feed speed (mm/min or ℊmin)

Turning

G01 X(U)_ Z(W)_ α_ F_;

X,U,Z,W,α	Target point coordinate of tool movement (α is an additional axis)
F	Feed speed (mm/min or ℊmin)

Detailed description

(1) Once this command is specified, it will remain until other G commands in group 01 are specified, and then the mode of G01 will be changed. If the next command is still G01 and the feed speed is the same, then only the coordinates need to be specified.

(2) If no F command is given in the program block where G01 command first appears, the speed is set by parameter #1211, or a program error alarm will be given.

(3) F command is also modal command, and the unit of F is determined by the time feed rate or rotation feed rate command.

(4) For simultaneous control of the two axis coordinates of the feed mode per minute, the feed speed along each axis is as follows:

G01α_{α} β_{β} Ff;

Feed speed in α coordinate direction: $F\alpha=\frac{\alpha}{L}\times f$

Feed speed in β coordinate direction: $F\beta=\frac{\beta}{L}\times f$

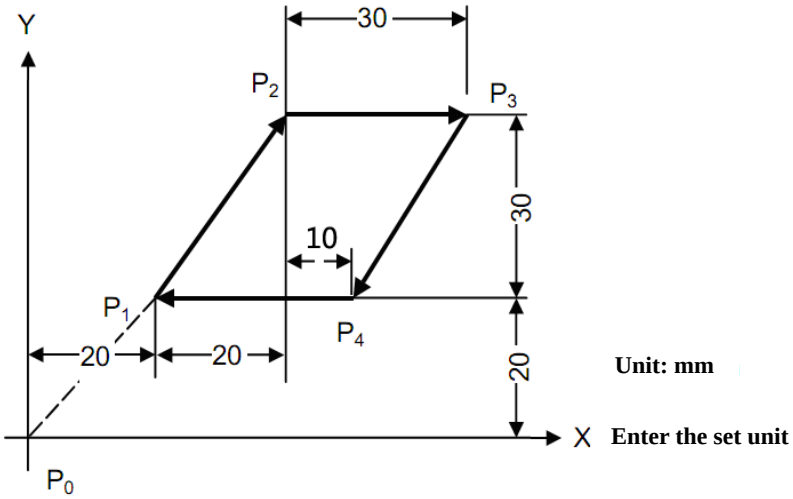
$L=\sqrt{\alpha^2+\beta^2}$

Programming examples

Milling

Example 1: Cutting P1 → P2 → P3 → P4 → P1 at a feed speed of 300 mm/min

P 0 → P1 is tool positioning

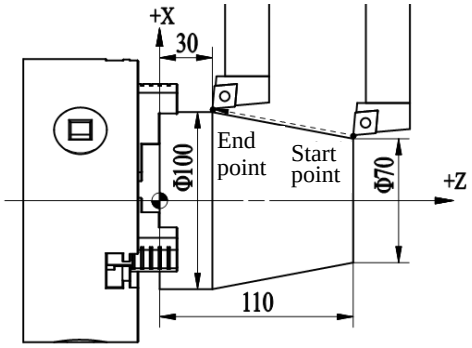


G91 G00 X20 Y20;	P0→P1
G01 X20 Y30 F300;	P1→P2
X30;	P2→P3
X-20 Y-30;	P3→P4
X-30;	P4→P1

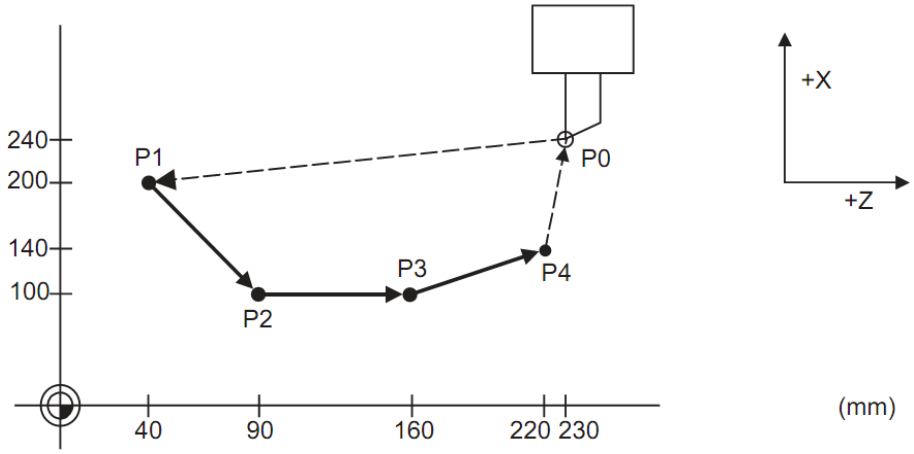
Turning

Example 2: As shown in the figure, the tool is cut from Φ70 to Φ100.

<Diameter Programming>
G01 X100 Z30 (Absolute value programming)
G01 U30 W-80 (Relative value programming)
G01 U30 Z30 (Hybrid programming)
G01 X100 W-80 (Hybrid programming)



Example 3: Cutting P1 → P2 → P3 → P4 at a feed speed of 300 mm/min, but P0 → P1, P4 → P0 are tool positioning.



G00 X200 Z40;	P0 → P1
G01 X100 Z90 F300;	P1 → P2
Z160	P2 → P3
X140 Z220;	P3 → P4
G00 X240 Z230;	P4 → P0

4.3 Plane selection (G17, G18, G19)

Function and purpose

G code is used to select the control plane or the plane where the circular arc is located.

Planes determined by any 2 non-parallel axes can be selected by setting 3 basic axes and the corresponding parallel axes as parameters. If the rotation axis is set to be a parallel axis, and a plane including the rotation axis may be selected.

Plane selection can be used for selecting:

– Plane performing circular arc compensation

– Plane on which the tool nose R compensation is performed

Command format

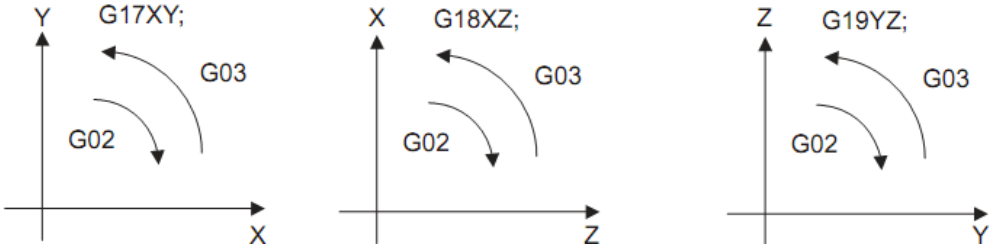
G17;XY plane selection
G18;ZX plane selection
G19;YZ plane selection

Detailed description

If an axis address is omitted in the plane selection program block, the address of two basic axes is considered to be omitted.

When power it turned on, the plane set by parameter "#1801bit1 or #801bit2" is selected.

Plane selection by (G17, G18, G19)



4.4 Circular interpolation (G02, G03)

Function and purpose

The circular interpolation command specifies the tool to perform circular arc motion at a given feed speed F in an specified plane to machine circular arc elements.

Command format

G17 {G02 / G03} X_Y_ {I_J_ / R_} F_;

G18 {G02 / G03} X_Z_ {I_K_ / R_} F_;Or turning A series is G18X {G02 / G03} (U)_Z(W)_F_ {I_K_ / R_};

G19 {G02 / G03} Y_Z_ {J_K_ / R_} F_;

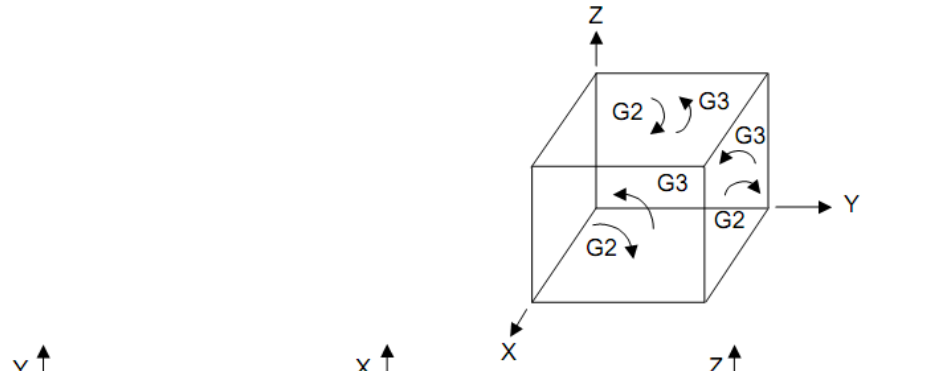
G17	Specify XY plane selection
G18	Specify ZX plane selection
G19	Specify YZ plane selection
G02	Clockwise circular interpolation
G03	Counterclockwise circular interpolation
X_	X-axis arc end point coordinate
Y_	Y-axis arc end point coordinate
Z_	Z-axis arc end point coordinate
X(U)_	X-axis arc end point coordinate (X is the absolute value of the workpiece coordinate system, U is the incremental value from the current position)
Z(W)_	Z-axis arc end point coordinate (Z is the absolute value of the workpiece coordinate system, W is the incremental value away from the current position)
I_	Incremental value of the X-coordinate from the start point to the arc center
J_	Incremental value of the Y-coordinate from the start point to the arc center
K_	Incremental value of the Z-coordinate from the start point to the arc center
R_	Arc radius
F_	Feed speed

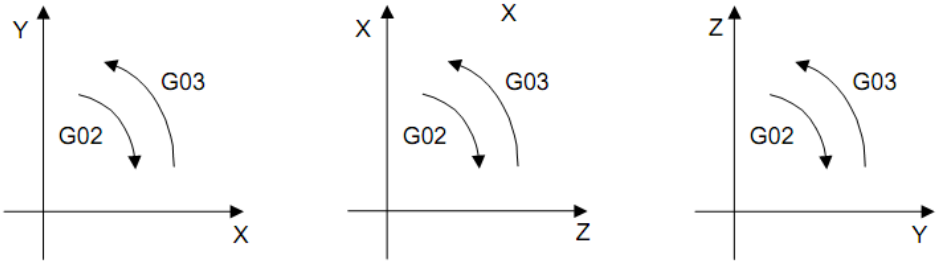
Detailed description

(1) Judgment on clockwise/counterclockwise of the arc

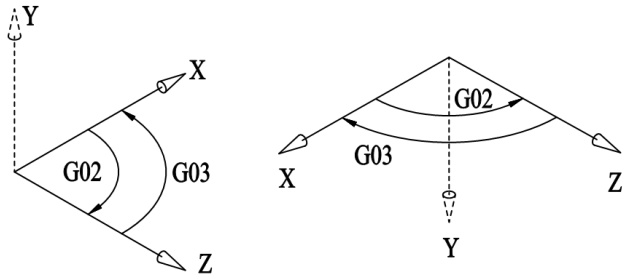
The commands of circular interpolation can be classified into the clockwise (G02) and counterclockwise ones (G03).

Judgment on clockwise/counterclockwise of the circular interpolation direction: if looking from the negative direction of the vertical coordinate axis on the plane where the arc is located at, G02 is clockwise and G03 is counterclockwise (see the figure below).





G17 (X-Y) plane G18 (Z-X) plane G19 (Y-Z) plane



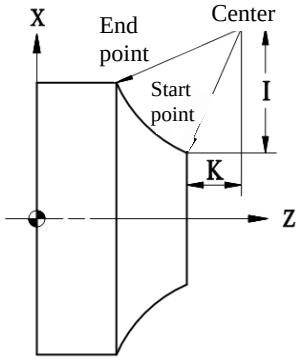
The coordinates of the rear tool post of the lathe

The coordinates of the front tool post of the lathe

(2) Movement distance on the arc

Specify the end point of the arc with the address X, Y or Z and use the absolute value or incremental value according to G90 or G91; for turning A series, use the address X (U) or Z (W) to specify the arc with the absolute value or incremental value end. Incremental value is the distance of the command looking from the arc start point to the arc end point.

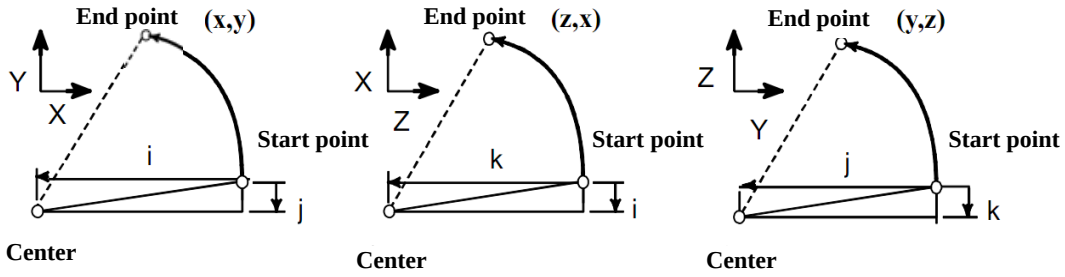
(3) Distance from the start point to the arc center



The arc center is specified by the address I, J and K (corresponding to the X/Y/Z coordinate). However, the figures behind I, J or K are vector components looking from the arc start point to the arc center, and are always specified with incremental va lue (as shown in the figure).

I0, J0 and K0 can be ignored.

An alarm will occur if the difference between the radius of the start point and that of the end point exceeds the value in parameter #1810.



(4) Programming of the entire arc

When the command at the programming position is ignored, or when the position of the end point and the start point at the command is identical, and the arc center is specified by I/J/K, one 360 °arc (entire arc) is specified.

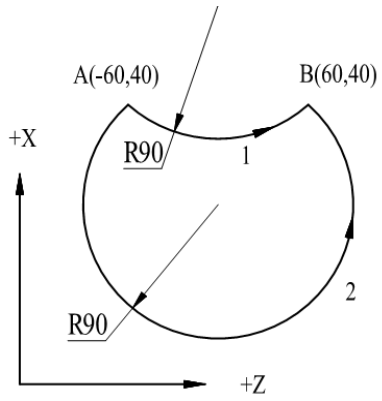
(5) Arc radius

The distance between the arc and the arc center may be specified with the radius R command of such arc rather than I/J/K. The radius of the arc can be classified into positive and negative ones. Once the arc center is less than or equal to 180 °(AB1 of the arc shown in the figure), R in the program is positive. Once the radius angle of the arc is larger than 180 °but is less than 360 °(AB2 of the arc shown in the figure), R is negative.

For instance, for the path AB shown in the figure, the program block prepared in the format of the R command is as follows:

AB1 G18 G03 X60.0 Z40.0 R90.0 F100;

AB2 G18 G03 X60.0 Z40.0 R-90.0 F100;



(6) Feed speed

The feed speed of circular interpolation is that specified by the F code and the feed speed (tangential feed speed of the arc) of the arc is controlled as the specified one.

Limits

(1) Simultaneous specification of R and I/J/K

If I/J/K and R are specified simultaneously, the arc specified by R is preferential and others can be ignored.

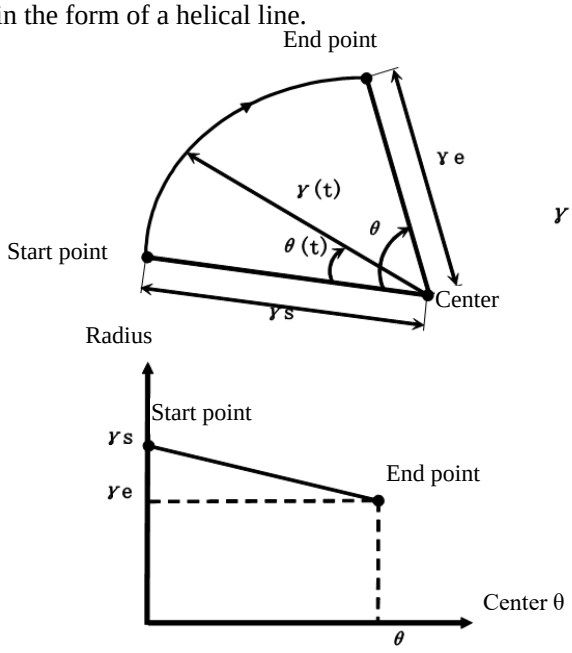
(2) When the axis beyond the specified plane is specified

An alarm will occur if the axis beyond the specified plane is specified.

(3) Radius difference between the start point and the end point

An alarm will occur if the radius difference between the start point and the end point of the arc exceeds the value set in parameter #1810. If the end point does not lies on the arc, the tool will arrive at the end

point in the form of a helical line.



$$\gamma(t) = \gamma s + \frac{(\gamma e - \gamma s)\ \theta(t)}{\theta}$$

(4) Designate the semi-arc with R

If one arc whose central angle is close to 180 °is specified by R, an error will occur in the calculation results of the central coordinate. In this case, specify the arc center with I/J/K.

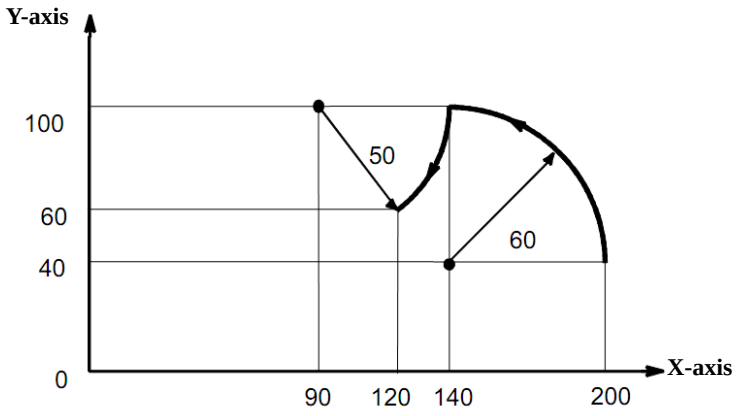
Programming examples

Milling

Attempt to prepare the program of the path above through absolute values and Incremental value.

(1) Absolute value

G92 X200.0 Y40.0 Z0 ;
G90 G03 X140.0 Y100.0 R60.0 F300. ;
G02 X120.0 Y60.0 R50.0 ;
Or
G92 X200.0 Y40.0 Z0 ;
G90 G03 X140.0 Y100.0 I-60.0 F300. ;
G02 X120.0 Y60.0 I-50.0 ;

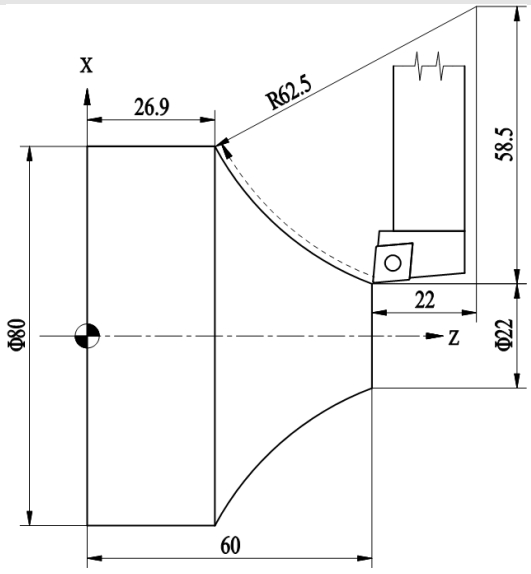


(2) Incremental value

G91 G03 X-60.0 Y60.0 R60.0 F300. ;
G02 X-20.0 Y-40.0 R50.0 ;
Or
G91 G03 X-60.0 Y60.0 I-60.0 F300. ;
G02 X-20.0 Y-40.0 I-50.0 ;

Turning

<Diameter Programming>
G02 X80 Z26.9 I58.5 K22 F0.3; Or
G02 U58 W-33.1 I58.5 K22 F0.3; Or
G02 X80 Z26.9 R62.5 F0.3; Or
G02 U58 W-33.1 R62.5 F0.3;



4.5 Helical interpolation (G02, G03)

Function and purpose

If the moving Axis 1/2 beyond the specified plane is specified during the specification of the circular interpolation, helical interpolation realizing helical movement of the tool can be performed.

Command format

$$G17\begin{Bmatrix}G02\\G03\end{Bmatrix}X_Y_\begin{Bmatrix}I_J_\\R_\end{Bmatrix}\alpha_\beta_F_;$$

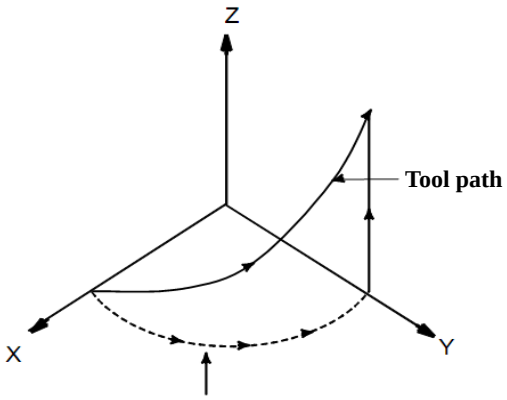
$$G17\begin{Bmatrix}G02\\G03\end{Bmatrix}X_Z_\begin{Bmatrix}I_K_\\R_\end{Bmatrix}\alpha_\beta_F_;$$

$$G17\begin{Bmatrix}G02\\G03\end{Bmatrix}X_Z_\begin{Bmatrix}J_K_\\R_\end{Bmatrix}\alpha_\beta_F_;$$

G17	Specify XY plane selection
G18	Specify ZX plane selection
G19	Specify YZ plane selection
G02	Clockwise circular interpolation
G03	Counterclockwise circular interpolation
X_	X-axis arc end point coordinate
Y_	Y-axis arc end point coordinate
Z_	Z-axis arc end point coordinate
α_ β_	Any axis (two at most) except for the circular interpolation arc can be specified
I_	Incremental value of the X-coordinate from the start point to the arc center
J_	Incremental value of the Y-coordinate from the start point to the arc center

K_	Incremental value of the Z-coordinate from the start point to the arc center
R_	Arc radius
F_	Feed speed

Detailed description



The rate along the circumference of two circular interpolation axes is the specified rate.

F command is the specified feed speed along the arc of the plane. Therefore, the rate of the linear axis is:

$$F \times \frac{\text{length of linear axis}}{\text{length of arc}}$$

Determine the feed speed and notice that the speed on the linear axis shall not exceed any extreme value.

Limits

(1) Only the arc will be subject to tool radius compensation.

(2) F is the tangential speed of the arc and the rate of the axis under synchronous movement is the follow-up rate. The system only limits the arc speed within parameter #1224 and the synchronous movement axis speed within parameter #1225, and the final resultant speed may be greater than the value of parameter #1224.

4.6 Helical interpolation and conic interpolation (G02, G03)

Function and purpose

Based on the commands of circular interpolation, helical interpolation can be made only by specifying the rotation times or the radius increase/decrease per revolution.

Conic interpolation can be carried out by specifying the increase/decrease value of such axis per revolution if adding movement commands of one axis to the commands of helical interpolation.

Command format

Helical interpolation

$$G17\begin{Bmatrix}G02\\G03\end{Bmatrix}X_Y_I_J_Q_L_F_;$$

$$G17\begin{Bmatrix}G02\\G03\end{Bmatrix}X_Z_K_I_Q_L_F_;$$

G17 {G02 / G03} Y_Z_J_K_Q_L_F_;

X_ Y_ Z_	End point coordinate
L_	Times (🔊1)
Q_	Radius increase/decrease per helical revolution (🔊1)
I_ J_ K_	Signed distance from the start point to the center (the same as circular interpolation)
F_	Feed speed

(🔊1) Either the times L or the radius increase/decrease Q can be ignored.

If L is ignored, the times can be calculated automatically as per the distance between the current position and the center, the position of the endpoint and the radius increase/decrease.

If Q is ignored, the radius increase/decrease can be calculated automatically as per the distance between the current position and the center, the position of the endpoint and the times.

If L and Q are to be specified simultaneously, Q is preferential in case of any contradiction between both. In general, either L or Q is specified.

Moreover, since L is a positive value without decimal points, if the mode of "4 arcs + 90" is to be specified, "L5" shall be specified.

Conic interpolation

G17 {G02 / G03} X_ Y_ I_ J_ Z_ Q_ L_ F_;

G18 {G02 / G03} X_ Z_ K_ I_ Y_ Q_ L_ F_;

G19 {G02 / G03} Y_ Z_ J_ K_ X_ Q_ L_ F_;

X_ Y_ Z_	End point coordinate
L_	Times (🔊1)
Q_	Radius increase/decrease per helical revolution (🔊1)
I_ J_ K_	Arc center and Z-axis (K is the incremental value of the Z coordinate from the start point to the arc center)
F_	Feed speed

(🔊1) Either the times L or the radius increase/decrease Q can be ignored.

If L is ignored, the times can be calculated automatically as per the distance between the current position and the center, the position of the endpoint and the radius increase/decrease.

If Q is ignored, the radius increase/decrease can be calculated automatically as per the distance between the current position and the center, the position of the endpoint and the times.

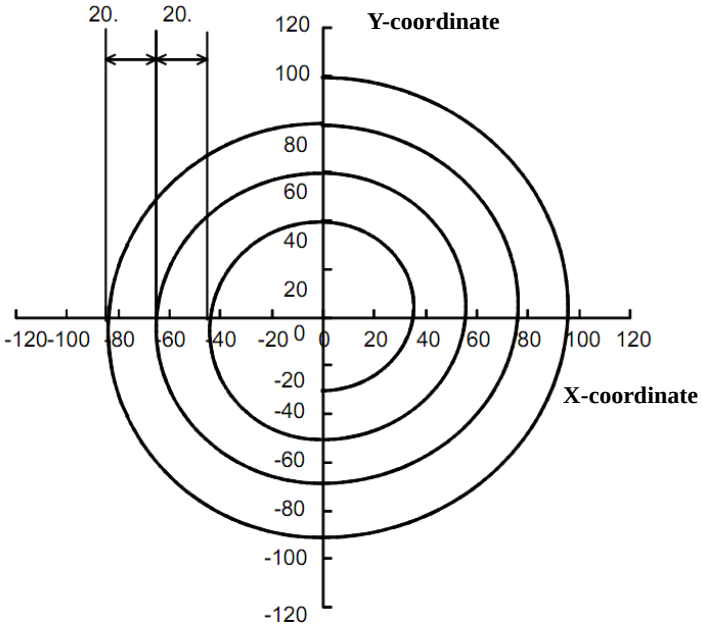
If L and Q are to be specified simultaneously, Q is preferential in case of any contradiction between both. In general, either L or Q is specified.

Moreover, since L is a positive value without decimal points, if the mode of "4 arcs + 90" is to be specified, "L5" shall be specified.

Programming examples

Helical interpolation

If the path of the figure below is specified with absolute values and incremental values, the results are as follows:



Start point: (0, 100.0)

End point (X, Y): (0, -30.0)

Distance to the center (I, J): (0, -100.0)

Radius increase/decrease value (Q): -20.0

Times (L): 4

(1) Specified with absolute value:

G90 G02 X0 Y-30.0 I0 J-100.0 { Q-20.0 / L4 } F300 ;

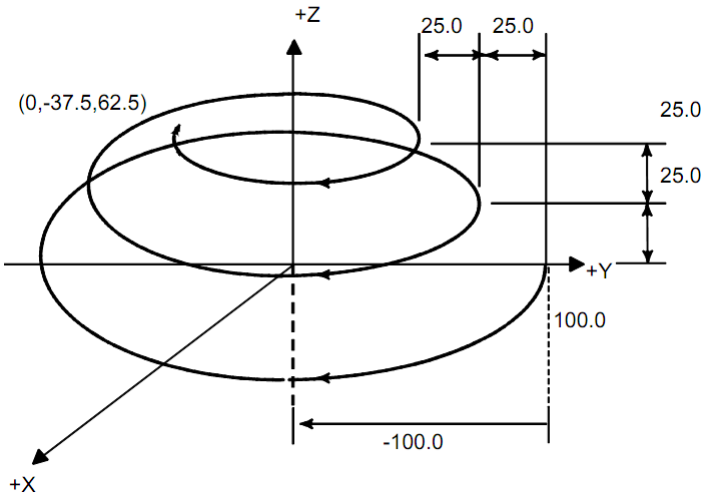
(2) Designate with incremental value:

G91 G02 X0 Y-130.0 I0 J-100.0 { Q-20.0 / L4 } F300 ;

(Either Q or L can be ignored)

Conic interpolation

If the path of the figure below is specified with absolute value and incremental value, the results are as follows:



Start point: (0,100.0,0)
End point (X, Y, Z): (0,-37.5,62.5)
Distance to center (I, J) : (0,-100.0)
Radius increase or decrease value (Q): -25.0
Height increase or decrease value (K): 25.0
Times (L): 3

(1) Designate with absolute value

G90 G02 X0 Y-37.5 Z62.5 I0 J-100.0 { K25.0 / Q-25.0 / L3 } F300;

(2) Specify with incremental value

G91 G02 X0 Y-137.5 Z62.5 I0 J-100.0 { K25.0 / Q-25.0 / L3 } F300 ;

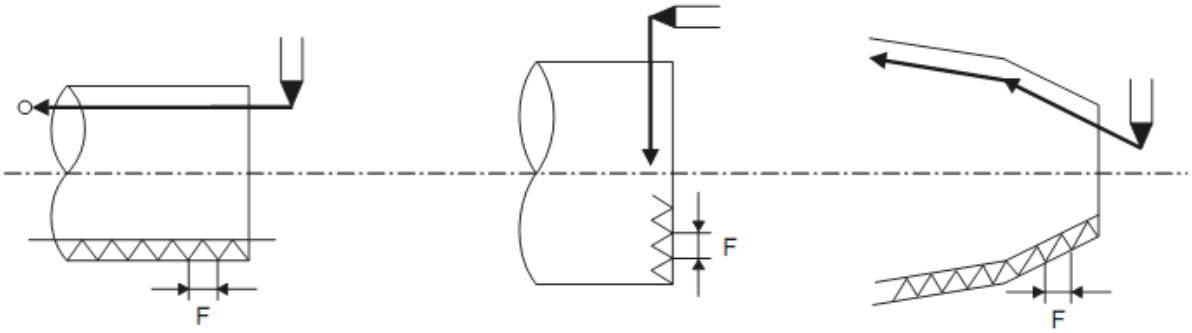
(Either Q or L can be ignored)

4.7 Thread turning (G32, G34)

4.7.1 Equal-pitch thread (G32)

Function and purpose

Feed control of the tool under synchronous rotation with the spindle shall be made through G32 command. Linear and continuous thread cutting with a fixed lead can be made.



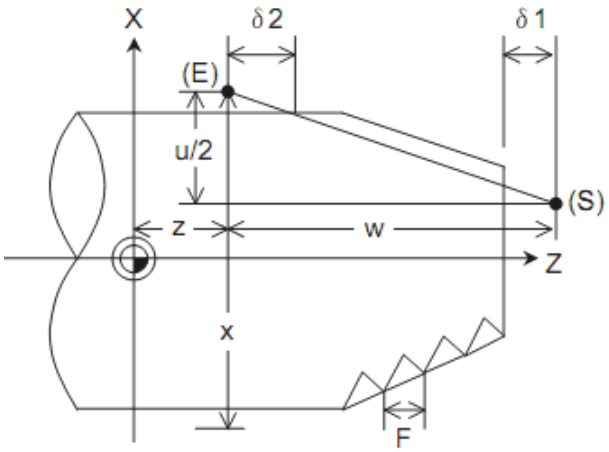
Cylindrical thread End face thread Continuous thread

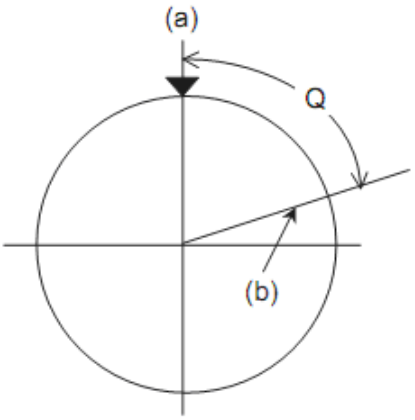
Command format

G32 X(U)_ Z(W)_ Q_ F/E_;

X(U)_ Z(W)_	Coordinate of the end point of the thread
F_	Lead of the thread in the direction of the long axis Pitch of the thread for the single thread
E_	Threads per inch in the direction of the long axis
Q_	Thread start angle. The value is a non-modal value without a decimal point, in unit of 0.001 º. When the value is not specified, it defaults to 0

In command format, if there is only the data word Z (W)_ of the Z-direction, the equal-pitch cylindrical thread is machined; if there is only the data word X (U)_ of X-direction, the equal-pitch end thread is machined.





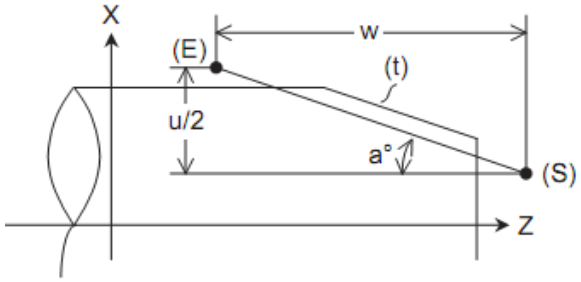
δ1: Incorrect lead during acceleration when thread cutting is started

δ2: Incorrect lead during deceleration when thread cutting is finished

(S) Start point (E) End point

(a) 0 degree of the encoder (b) The position where thread cutting starts

As for the lead F of conic threads, the lead is the Z-axis direction when a≤45º, and the X-axis direction when a≥45º.



(t) Conic thread (E) End point (S) Start point

When a<45 º, the lead is in the Z-axis direction

When a>45 º, the lead is in the X-axis direction

Both Z-axis and X-axis are allowed if a=45 º

Selection of lead of conic threads

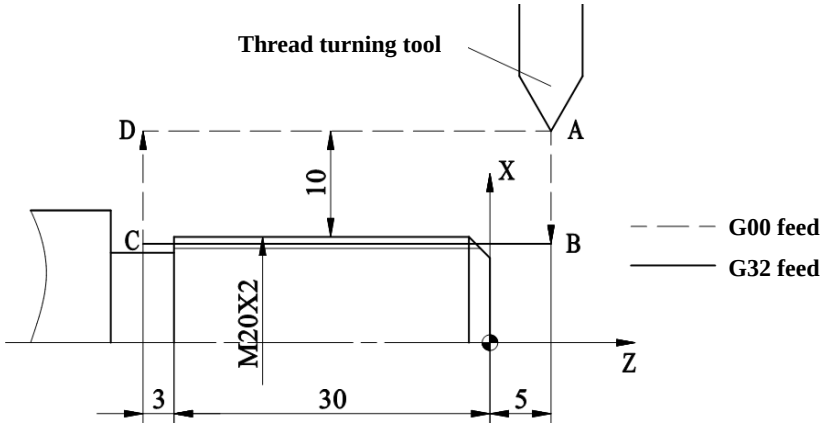
Program examples

G32 W-30.0 F4.0;

Detailed description

The path of G32 is similar to that of G01, and the tool will move from point B to point C at the speed of one lead/pitch per revolution. Both infeed before cutting and tool retracting after cutting shall be realized through other programs (as shown in AB/CD/DA program block in the figure).

During machining of the equal-pitch cylindrical thread and other threads other than end threads, the installation methods (forward/reverse) of the threading tool and the rotation direction of the spindle shall match with the configuration methods (front/rear) of the lathe carriage. If the rear carriage is used for turning the right thread, the threading tool must be installed in a reverse manner (the front tool face downwards) and the rotation direction of the lathe spindle must be designated with M04 command. If the threading tool is installed in a forward manner, such direction of the spindle must be designated with M03. The start point shall also be changed as D point.



Movement path and programming example of G32 cylindrical thread

Programming examples

Write the thread machining program with G32 command (The thread cutting lead-in distance δ1 is 5 mm, and the lead-out distance δ2 is 3 mm. The total cutting depth of the thread is scheduled to be 1.3 mm, and the cutting is divided into three times. The amount of back cutting is 0.8 mm, 0.4 mm and 0.1 mm.) The specific procedures are as follows:

```
O0402;
    ……
G00 X40.0 Z5.0;
    X19.2;
G32 Z-33.0 Q0 F2.0; Machining of the first helical line and the starting angle of threads is 0 º
G00 X40.0;
    Z5.0;
    X18.8;
G32 Z-33.0 Q0 F2.0; Machining of the first helical line is completed
G00 X40.0
    Z5.0;
    X18.7;
G32 Z-33.0 Q0 F2.0; Machining of the first helical line is completed
```

```
G00 X40.0;
    Z5.0;
    X19.2;
G32 Z-33.0 Q180000 F2.0;
                    Repetitive cutting with multiple tools until machining of the 2nd helical line is completed
M30;
```

Other purposes of G32 command

G32 command can also be used for machining the following threads in addition to the threads above:

(1) Conic threads: Conic threads will be machined in case of incremental movement both in the X and Z-directions in the thread commands.

(2) Multiple thread: During preparation of the program for machining multiple threads, only the "1st revolution" signal of the spindle and the deviation angle at the start point of thread cutting need to be specified with Q.

(3) End face thread: During execution of the program block of end threads, the tool will feed within the specified distance of thread cutting along the X-direction at a speed of F per revolution rather than the Z-direction.

4.7.2 Variable-pitch conic thread (G34)

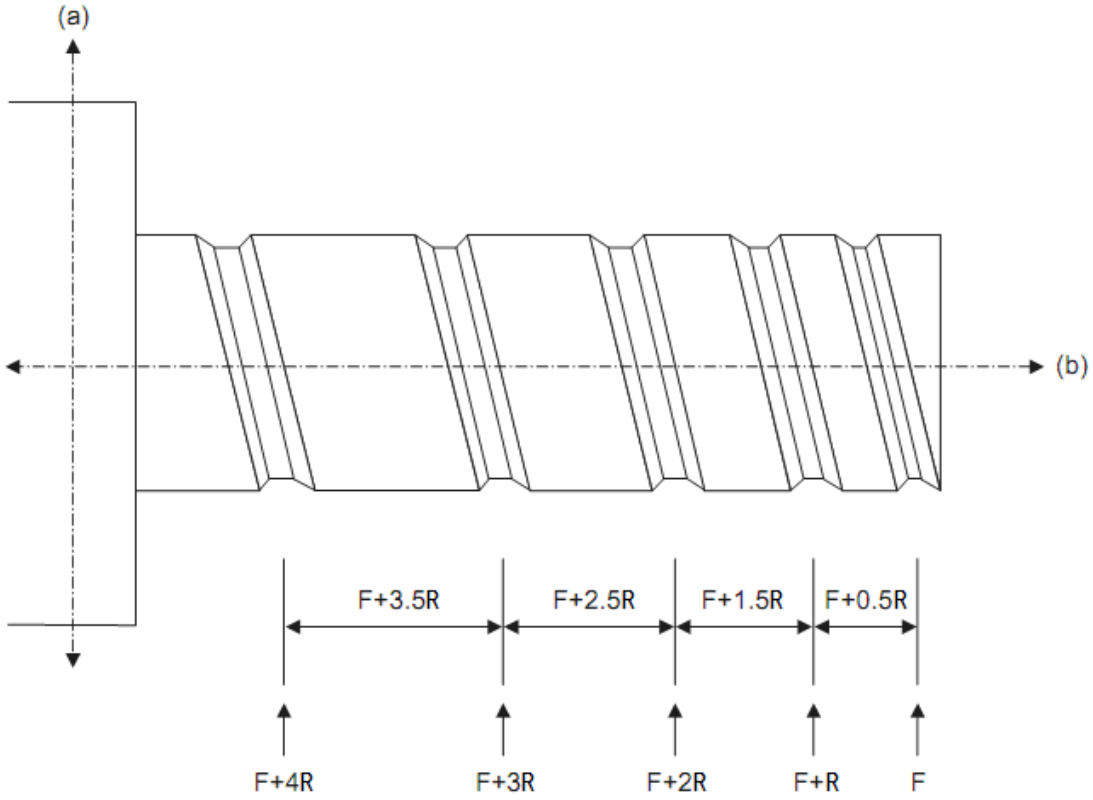
Function and purpose

Mainly used for cutting the variable-pitch cylindrical thread and conic threads.

Command format

G34 X(U)_ Z(W)_ Q_ F/E_ R_;

X(U)_ Z(W)_	Coordinate of the end point of the thread
F_	Basic lead of threads in the direction of the long axis
E_	Basic threads per inch in the direction of the long axis
Q_	Thread start angle. The value is a non-modal value without a decimal point, in unit of 0.001 º. When this value is not specified, it defaults to 0
R_	Increment (positive) or decrement (negative) of the "1st revolution" lead of threads



Unit of R:

Metric input:mm/rev ±v

Imperial input: inch/rev ±v

Program examples

G34 W-30.0 F4.0 R0.1;

Detailed description

During execution of G34, other actions and paths are identical to those of G32 command except for increment in pitch per revolution.

Precautions of G32 and G34

(1) The feed speed override is invalid during thread cutting.

(2) Feed hold is invalid during thread cutting. If pressing down the "Feed Hold" button during thread cutting, the tool will stop after performing the program for non-thread cutting.

(3) The spindle speed override is invalid during thread cutting.

(4) The constant line speed control should not be adopted during thread cutting, and the constant-speed control is relatively proper.

(5) The spindle speed must be kept constant from rough turning to finish turning, or otherwise the thread lead will be incorrect.

(6) In general, lagging of the servo system will give rise to incorrect lead at the starting and end point of thread cutting. To make up for this, the specified thread length shall be longer than the thread length required.

(7) If F and E are specified simultaneously, E is invalid and F is valid.

4.8 Thread cutting (G33)

Function and purpose

The G33 command enables tool feed control synchronized with the spindle rotation, so thread cutting with a fixed lead can be performed. Moreover, machining of multiple threads can also be made through the specified starting angle of thread cutting.

Command format

```
G33 Z_ (X_ Y_ α_) F/E_ Q_;
```

Z_ (X_ Y_ α_)	Thread end point
F_	Lead of the thread in the direction of the long axis Pitch of the thread for the single thread
E_	Threads per inch in the direction of the long axis
Q_	Thread start angle. The value is a non-modal value without a decimal point, in unit of 0.001 º. When the value is not specified, it defaults to 0

Detailed description

(1) The commands of thread cutting start as per the "1st" synchronizing signal of the rotary encoder.

(2) The spindle speed shall be kept constant from rough machining to finish machining.

(3) If feed stops due to feed hold during thread cutting, the threads may be damaged, so feed hold shall not be performed in thread cutting. However, in the process from execution of thread cutting to axis movement, the signal for feed hold is valid.

If pressing down the "Feed Hold" switch during thread cutting, stop at the end point of the next program block upon completion of thread cutting (non-G33 mode).

(4) The converted cutting feed speed is compared with the maximum cutting feed speed when starting thread cutting, and an operation error occurs if the speed exceeds the maximum speed.

(5) In the course of thread cutting, to guarantee the lead, the cutting feed speed converted may exceed the highest cutting feed speed.

(6) Such lead may be incorrect due to lagging of the servo system generally before/after thread cutting. To make up for this, the specified thread length shall be longer than the thread length required.

(7) The spindle speed is limited by the followings:

$1 \leqq R \leqq$ the highest feed speed/thread lead

But $R \leqq$ allowable speed of the encoder (r/min)

R: Spindle speed (r/min)

(8) Dry run is valid during thread cutting. However, the feed speed cannot synchronize with the rotation of the spindle due to dry run.

(9) The command for thread cutting is that for feed per revolution even during G94.

(10) In thread cutting, finishing of the spindle override and the cutting feed override is invalid, turning as 100%-constant.

(11) During execution of G33, when the original mode is switched to other modes, autorun will be stopped after the program block for non-thread cutting of the next program block is performed.

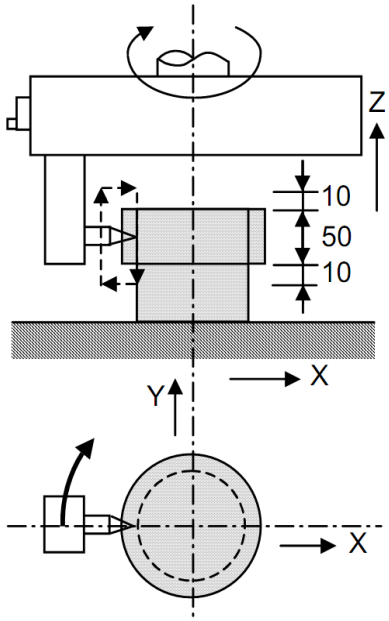
(12) During thread cutting, interruption of the hand wheel under autorun mode is valid.

(13) The starting angle of threads is non-modal. If Q command is not specified in G33, it will be deemed as

"Q0".

(14) For G33, only one thread is processed. Please change Q value in case of cutting of another thread of multiple threads, and the same command shall be specified.

Program examples



N110 G90 G0 X-200. Y-200. S50 M3;	The center of the spindle is oriented at the workpiece center and the spindle is under forward rotation.
N111 Z110. ;	
N112 G33 Z40. F6.0 ;	Perform the 1st thread cutting. Thread lead = 6.0 mm
N113 M19;	The spindle is positioned through M19 command.
N114 G0 X-210. ;	Retract the tool in the X-axis direction.
N115 Z110. M0;	Return the tool to the workpiece and stop the program through M00. Adjust the tool as needed.
N116 X-200. M3 ;	Prepare for the 2nd thread cutting.
N117 G04 X5.0 ;	Delay the command as needed, to stabilize the spindle speed.
N118 G33 Z40. ;	Conduct the 2nd thread cutting.

4.9 One-way positioning (G60)

Function and purpose

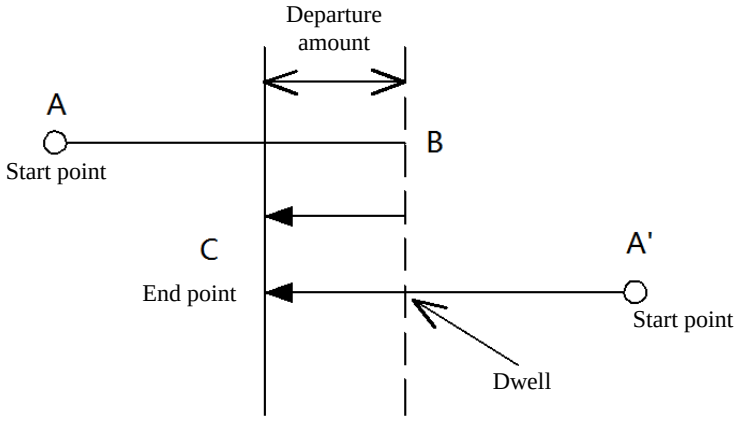
In case precise positioning is required to eliminate the backlash of the machine, accurate positioning can be made in one direction through G60.

Command format

G60 X_ Y_ Z_ α_;

α_	Auxiliary axis

Detailed description



Overshoot can be set through the parameter #1880, and the positioning direction can be determined through the positive/negative overshoot set.

Program examples

G90 G00 X-10 Y10;
G60 X20 Y25; ①

If parameter #1880 is set to (-8, 5, 0, 0, 0), then for the ① statement, the tool path is AB→virtual point B (X28 Y20) and after a slight pause for about 0.5s→BC (X20 Y25). When A' reaches the end point of C in the same direction, it will pause for a while when it reaches the virtual point B.
The system program interface prompts changes: ***Linear interpolation***------**********-------***Linear interpolation***. Don't show countdown

G60 is a non-modal G code, if parameter #1870.0 (MDL) is set to 1, it becomes the modal G code of group 01, but the G60 set to modal cannot be used to cancel the fixed cycle. This setting can avoid specifying the G60 command once for each program block. The modal G60 can be cancelled with other G codes of the 01 group. The rest designation methods are the same as those of the non-modal G60 command.

Example:

<table>
<tr><td colspan="2">Non-modal G60 command (#1870.0=0)</td><td colspan="2">Modal G60 command (#1870.0=1)</td></tr>
<tr><td colspan="2">G90</td><td>G90 G60</td><td>One-way positioning mode</td></tr>
<tr><td>G60 X10 Y10</td><td rowspan="3">One-way positioning is valid</td><td>X10 Y10</td><td rowspan="4">One-way positioning is valid</td></tr>
<tr><td>G60 X30 Y30</td><td>X30 Y30</td></tr>
<tr><td>G60 X80 Y80</td><td>X80 Y80</td></tr>
<tr><td>X50 Y50</td><td>One-way positioning cancelled</td><td>X50 Y50</td></tr>
</table>

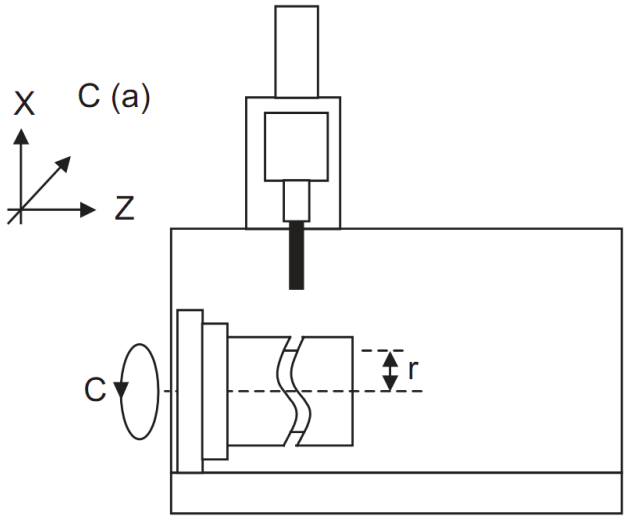
Precautions

(1) In the fixed cycle of drilling, the one-way positioning shall not be made for the Z-axis.
(2) One-way positioning shall not be made for the axis for which overshoot is not specified.
(3) The system will not execute the one-way positioning when the command for movement distance is 0.

4.10 Cylindrical interpolation (G07.1)

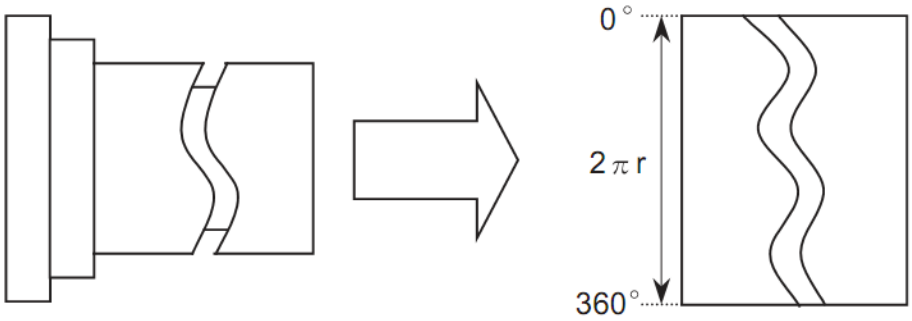
Function and purpose

This function expands the cylindrical surface (the shape on the cylindrical coordinate system) into a plane, and issues a program command with the expanded plane coordinates. During machining, the linear axis and the rotation axis are converted into cylindrical coordinates to perform contour control.



(a) Assumed C-axis

Programming can be made as per the plane after the cylindrical plane is spread, so the function is effective for machining of the cylindrical cam. Program commands are sent out through the rotation axis and the orthogonal axis, and grooving is made on the cylindrical surface.



Command format

G07.1 C(r); Cylindrical interpolation started/cancelled

C	Address of the rotation axis. Corresponding axis address can be input as per the name of the practical rotation axis
r	Cylinder radius value -Radius≠0: cylindrical interpolation started -Radius=0: cylindrical interpolation cancelled

Detailed description

(1) The coordinate commands within the section from starting to cancelation of the cylindrical interpolation are the cylindrical coordinate system.

G07.1 C cylinder radius value; start of cylindrical interpolation mode (start cylindrical interpolation)

:

: (the coordinate commands within the section are the cylindrical coordinate system)

:

G07.1 C0 ; Cylindrical interpolation cancelled (end the cylindrical interpolation)

(2) In place of G07.1 with G107.

(3) It is recommended to specify G07.1 at a separate program block.

(4) Create the rotation axis through the angle.

(5) In the mode of cylindrical interpolation, the commands for linear interpolation or circular interpolation can be issued. However, please specify plane selection commands in front of G07.1 program block.

(6) The coordinate commands can also be absolute commands, or increment commands.

(7) The tool radius compensation can be attached to the program commands. Cylindrical interpolation can be made for the path subjected to tool radius compensation.

(8) As for the feed speed, the tangential speed in the course of cylindrical spreading shall be specified through F (mm/min or inch/min).

Precision of cylindrical interpolation

In the cylindrical interpolation mode, the movement amount of the rotation axis specified by the angle is converted into the distance on the circumference, and after performing linear and circular interpolation operations with other axes, it is converted into an angle again.

Therefore, the practical movement amount may be inconsistent with the specified value when the cylinder radius is small. However, the cumulative error shall not be calculated in this case.

$$\text{Practical movement distance} = \left[\frac{REV\ \text{movement}}{2\times 2\pi R} \times \left[\text{Specified value} \times \frac{2\times 2\pi R}{REV\ \text{movement}} \right] \right]$$

REV movement: The movement amount per revolution of the rotation axis (generally 360°) can be set through parameter #1068 in the system.

R: Cylinder radius

[ ]: Rounding off the smallest unit set

Relevant parameters

Attributes of each axis in the #1024 basic axis coordinate system.

Set Value	Meaning
0	Neither the basic 3-axis nor the parallel axis
1	X-axis of the basic 3-axis
2	Y-axis of the basic 3-axis
3	Z-axis of the basic 3-axis
5	Coordinate axis of the rotation axis is parallel to X-axis
6	Coordinate axis of the rotation axis is parallel to Y-axis
7	Coordinate axis of the rotation axis is parallel to Z-axis

Plane selection

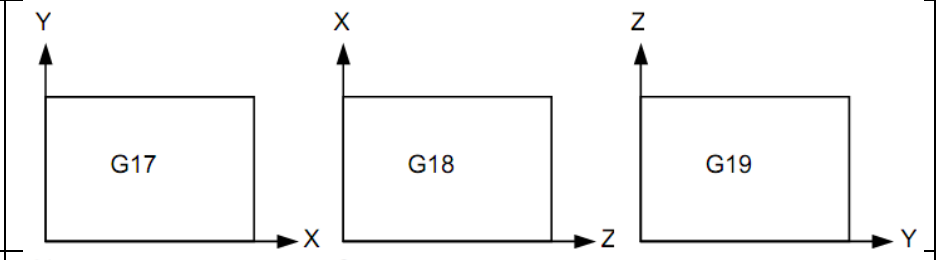
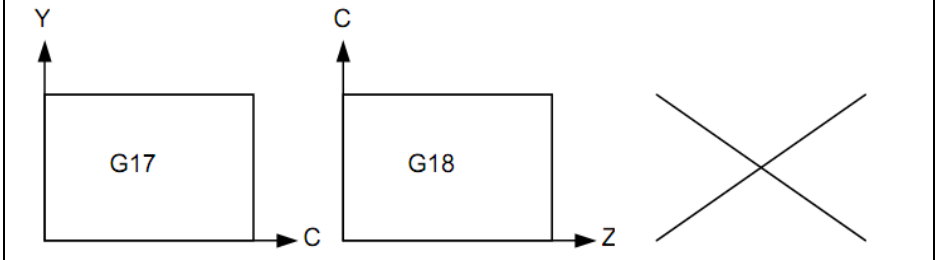
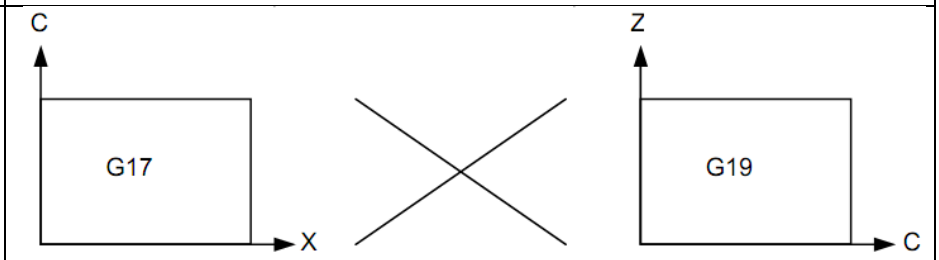
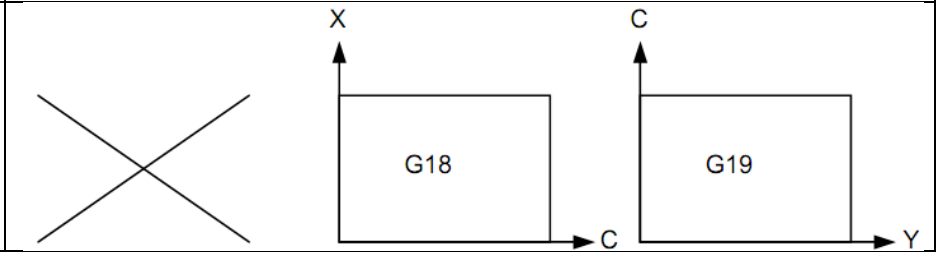
The axis performing cylindrical interpolation shall be set through the plane selection command.

The axis to which the coordinate axis of the rotation axis is parallel shall be set through parameter #1024=5/6/7.

In such plane, circular interpolation/tool nose radius compensation can be designated.

The plane selection command shall be set in front of or after G07.1. If such command is not set, an error may occur in the program in case of movement commands.

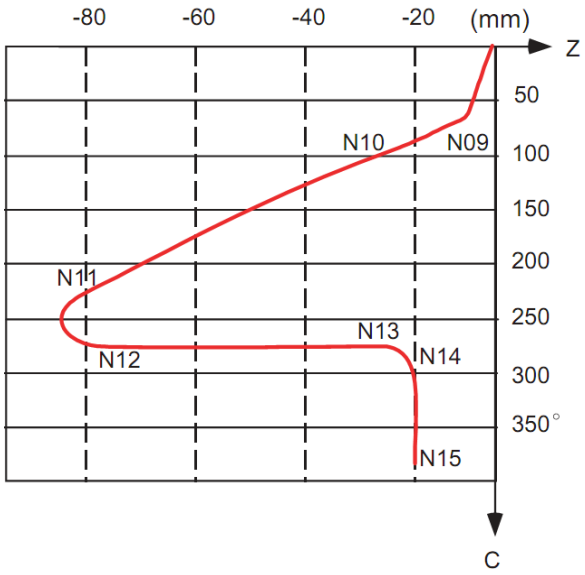
(Example)
G19 Z0. C0. ;
G07.1 C100. ;
:
G07.1 C0 ;

Fundamental coordinate system X,Y,Z	
Cylindrical coordinate system C ,Y ,Z (The rotation axis is the parallel axis of X-axis) #1024 C=5	
Cylindrical coordinate system X ,C ,Z (The rotation axis is the parallel axis of Y-axis) #1024 C=6	
Cylindrical coordinate system X ,Y ,C (The rotation axis is the parallel axis of Z-axis) #1024 C=7	

Programming examples

<Program>
N01 G28 …… ;
N02 T020 ;

```
N03 S100 M03 ;
N04 G00 X50. Z0. ;
N05 G01 X40. F100. ;
N06 G19 C0 Z0 ;  ..... The plane selection command performing cylindrical interpolation and the command of the Axis 2 performing interpolation
N07 G07.1 C20. ;  ..... Cylindrical interpolation started
N08 G41 ;
N09 G01 Z-10. C80. F150 ;
N10 Z-25. C90. ;
N11 Z-80. C225 ;
N12 G03 Z-75.C270. R55. ;
N13 G01 Z-25 ;
N14 G02 Z-20.C280. R80. ;
N15 G01 C360. ;
N16 G40 ;
N17 G07.1 C0 ;  ..... Cylindrical interpolation cancelled
N18 G01 X50. ;
N19 G0 X100. Z100. ;
N20 M05 ;
N21 M30 ;
<Parameter>
#1024 C=6
```



Relationships with other functions

Circular interpolation

(1) In the mode of cylindrical compensation, circular interpolation can be made between the rotation axis and the linear axis.

(2) The R command can be specified in circular interpolation. (When specifying I, J, K commands after specifying the R command, by default, R is valid, but the other commands are invalid)

Tool radius compensation

In the mode of cylindrical interpolation, tool radius compensation can be carried out.

(1) Please start/cancel under the mode of cylindrical interpolation during tool radius compensation.
(2) When G07.1 command is issued from tool radius compensation, an alarm concerning program error will occur.
(3) If G07.1 command is issued without movement commands after tool radius compensation is cancelled, the follow-up actions shall be performed by taking the axis position of the G07.1 command program block as the position after the tool radius compensation is cancelled.

Feed per minute of cutting

(1) Enter the mode of feed per minute in a forced manner by starting the mode of cylindrical interpolation.
(2) The mode of feed per revolution will go back to the state before cylindrical interpolation is started by cancelling cylindrical interpolation.

Constant line speed control

(1) If G07.1 command is issued from the constant line speed control mode (G96), an error may occur in the program.

Auxiliary functions

(1) Auxiliary functions (M) may be specified in the mode of cylindrical interpolation.
(2) The speed of rotary tools is specified by the S command in cylindrical interpolation rather than the spindle speed.
(3) T command shall be specified before cylindrical interpolation. An error will occur in the program if performing T command in cylindrical interpolation.

```
    :
    :
T1212 ;
G0 X100. Z0. ;          ... T command before executing cylindrical compensation →Program correct
G19 Z C ;
G07.1 C100. ;
    :
T1200 ;                 ... T command in cylindrical compensation mode →Program incorrect
    :
G07.1 C0 ;
```

(4) Please complete tool compensation before cylindrical interpolation (movement of the tool length and wear compensation).

F command in cylindrical interpolation

For F command in cylindrical interpolation, whether previous F command shall be used depends on the previous commands for feed per minute (G94) and feed per revolution (G95).

(1) G94 G95 in front of G07.1

If F command is not specified in cylindrical interpolation, keep using the feed speed of the previous F command.

The feed speed after cylindrical interpolation is cancelled will be recovered to the feed speed before cylindrical interpolation or those of F command finally set in cylindrical interpolation.

Limitations and precautions

(1) In the mode of cylindrical interpolation, the G code commands which can be used are as follows:

G Code	Content
G00	Positioning
G01	Linear interpolation
G02	Circular interpolation (CW)
G03	Circular interpolation (CCW)
G09	Accurate positioning inspection
G22/G23	Inspection on stored travel ON/OFF
G40-G42	Tool radius compensation
G61	Accurate stop
G64	Cutting
G90/G91	Absolute value/increment
G94	Feed per minute (FRM)
G98	Fixed cycle/return of initial point
G99	Fixed cycle/return of R point

(2) In the mode of cylindrical interpolation, cylindrical interpolation cannot be reset. The cylindrical interpolation can only be cleared through G07.1 C0 command, system re-start/emergency stop.

(3) In case of the axis not homed in the command axis of cylindrical interpolation, an error may occur in the program.

(4) The tool radius compensation needs to be cancelled if cylindrical compensation is to be cancelled.

(5) Switch to the turning mode and return to the plane selected before cylindrical interpolation by cancelling cylindrical interpolation.

(6) Corresponding program of the program block in cylindrical interpolation cannot be re-started (by the program).

(7) An error may occur in the program while specifying the command for cylindrical interpolation in the mirror.

(8) Decelerated inspection shall be carried out when cylindrical interpolation is started/cancelled.

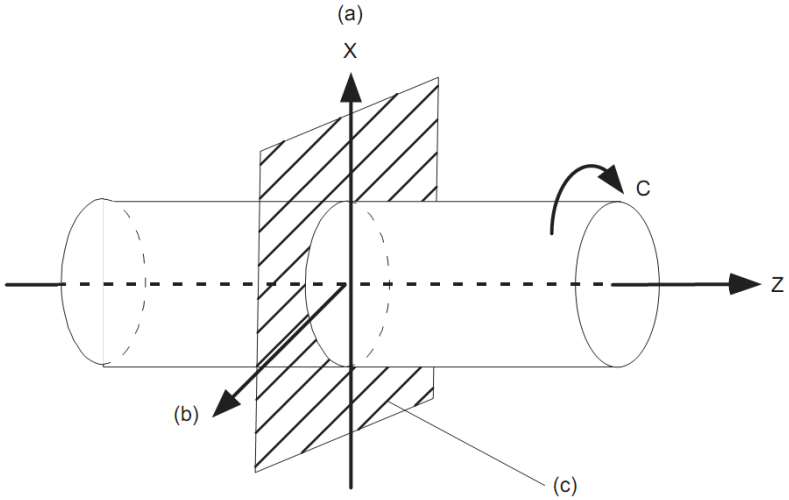
(9) An error may occur in the program if specifying cylindrical interpolation and polar coordinate interpolation in the mode of cylindrical interpolation.

4.11 Polar coordinate interpolation (G12.1)

Function and purpose

The program command will be converted into movement (tools) of the linear axis and of the rotation axis (workpieces) in the orthogonal coordinate shaft through the function for contour control.

The linear axis shall serve as the orthogonal axis of the 1st axis of the plane, and the assumed orthogonal axis shall serve as the plane of the 2nd axis ("plane of polar coordinate interpolation"). Polar coordinate interpolation shall be made on the plane. The point of origin of the workpiece coordinate system shall serve as that of the coordinate system in the polar coordinate interpolation.



(a) Linear axis (b) Rotation axis (imaginary axis) (c) Polar coordinate interpolation plane (G17 plane)

The function is applicable to cutting at the cut on the line at the outer diameter of the workpiece as well as cutting of the camshaft.

Command format

G12.1;	Polar coordinate interpolation started

G13.1;	Polar coordinate interpolation cancelled

Detailed description

(1) The coordinate command is polar coordinate interpolation within the section from starting to cancellation of polar coordinate interpolation.

G12.1 ; Polar coordinate interpolation mode start (start polar coordinate interpolation)
:
: (the coordinate command within the section is polar coordinate interpolation)
:
G13.1 ; Polar coordinate interpolation mode cancel (end polar coordinate interpolation)

(2) G12.1 and G13.1 can also be replaced by G112 and G113.

(3) G12.1/G13.1 command shall be specified at a separate program block, and an error may occur in the program if being specified at the same program block with other G codes.

(4) The command for linear interpolation or circular interpolation can be sent out under the mode of polar coordinate interpolation.

(5) The coordinate commands can also be absolute commands, or increment commands.

(6) The tool radius compensation can be attached to the program commands. For the path after tool radius compensation, polar coordinate interpolation can be made.

(7) For the feed speed, the tangential speed on the polar coordinate interpolation plane (orthogonal coordinate system) can be specified through. F is in unit of mm/min or inch/min.

(8) When G12.1/G13.1 command is delivered, decelerated inspection shall be performed.

Plane selection

The linear axis and the rotation axis performing polar coordinate interpolation shall be set in the parameters

in advance.

(1) The plane performing the polar coordinate interpolation is determined by the linear axis parameter (#3147) performing the polar coordinate interpolation.

Set Value of #3147	Plane
1	G17 (XY plane)
2	G19 (YZ plane)
3	G18 (ZX plane)
Blank (not set)	G17 (XY plane)

(2) An error may occur in the program if performing the plane selection command (G17~G19) under the mode of polar coordinate interpolation.

Note: Parameter #3147 may not exist according to the model and version. In this case, the action the same as that when parameter #3147 is blank (not set) shall be performed.

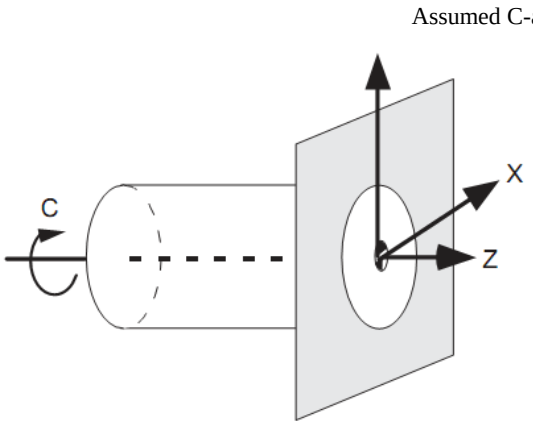
Relevant parameters

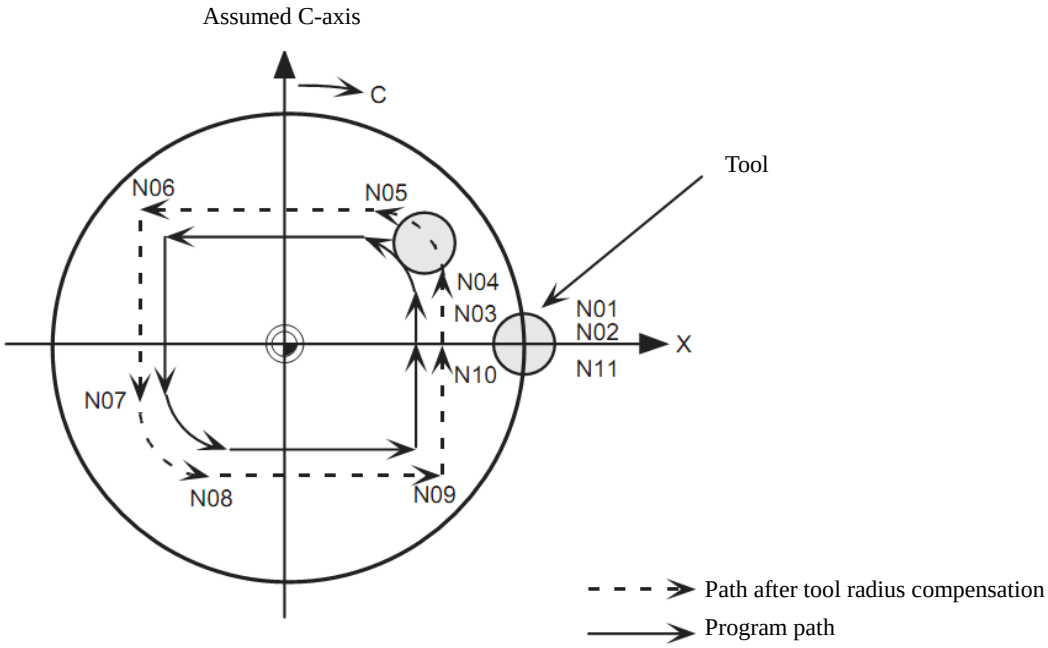
#3147 linear axis of polar coordinate interpolation

#3148 rotation axis of polar coordinate interpolation

#3149 highest cutting feed speed of polar coordinate interpolation

Program examples





<Program>

:

:

```
N00 T0101;
  :
  :
N01 G17 G90 G01 X40.0 C0 Z0;          Determine the starting position
N02 G12.1;                             Polar coordinate interpolation: Start
N03 G1 G42 X20.0 F2000;                Start practical machining
N04 C10.0;
N05 G3 X10.0 C20.0 R10.0;
N06 G1 X-20.0;                         Shape program
N07 C-10.0;
N08 G3 X-10.0 C-20.0 I10.0 J0;         (According to orthogonal coordinates on the plane of
                                        the assumed X-C axis)
N09 G1 X20.0;
N10 C0;
N11 G40 X40.0;
N12 G13.1;                             Polar coordinate interpolation: Cancel
  :
  :
M30 ;
```

Relationships with other functions

Program command of polar coordinate interpolation

(1) On the plane of polar coordinate interpolation, the program command in polar coordinate interpolation shall be specified through the orthogonal coordinates of the linear axis and the rotation axis (assumed axis). The address of the rotation axis (C) shall be specified through that in the command of Axis 2 (assumed axis) of the plane. The command unit is that (mm or inch) delivered through the address of the

1st axis (linear axis) of the plane rather than deg (°).

(2) Assume that the axis coordinate is "0" while specifying G12.1. In other words, polar coordinate interpolation shall be started by regarding the angle at the position where G12.1 is specified as 0.

Circular interpolation on the polar coordinate plane

The arc radius address depends on parameter #3147 of the linear axis during circular interpolation under the mode of polar coordinate interpolation.

Set Value of #3147	Specified Address at the Center
1	I, J (regard the polar coordinate plane as the XY plane)
2	J, K (regard the polar coordinate plane as the YZ plane)
3	K, I (regard the polar coordinate plane as the ZX plane)
Blank (not set)	I, J (regard the polar coordinate plane as the XY plane)

The circular radius can be specified through R.

Note: Parameter #3147 may not exist according to the model and version. In this case, the action the same as that when parameter #3147 is blank (not set) shall be performed.

Tool radius compensation

Tool radius compensation can be made under the mode of polar coordinate interpolation.

(1) Please specify plane selection (the same as polar coordinate interpolation).

Please start/cancel tool radius compensation under the mode of polar coordinate interpolation.

(2) An error may occur in the program if performing polar coordinate interpolation in tool radius compensation.

(3) If G12.1/G13.1 command is issued without movement commands after tool radius compensation is cancelled, the follow-up actions shall be performed by taking the axis position of the G12.1/G13.1 command program block as the position after the tool compensation is cancelled.

Feed per minute of cutting

(1) Enter the mode of feed per minute in a forced manner when polar coordinate interpolation is started.

(2) The synchronous mode will be recovered to the state before polar coordinate interpolation by cancelling polar coordinate interpolation.

(3) If G12.1 command is issued from the constant speed control mode (G96), an error may occur in the program.

Auxiliary functions

(1) Auxiliary functions (M) and the 2nd auxiliary function may be specified in the mode of polar coordinate interpolation.

(2) The S command of polar coordinate interpolation is used for designating the speed of rotary tools rather than that of the spindle.

(3) Please issue the T command before polar coordinate interpolation. An error (PS0108) may occur in the program if performing the T command in the mode of polar coordinate interpolation.

```
:
T1212 ;  .....  Perform the T command before polar coordinate interpolation  → Yes
G1 X100. Z0. ;
```

G12.1 ;
:
T1200 ; ..... Perform the T command in the mode of polar coordinate interpolation → Error in the program
:
G13.1 ;

(4) Please finish the action (movement of the tool length and wear compensation) of tool compensation before polar coordinate interpolation.
If such action is not finished yet when execution of the command of polar coordinate interpolation is started, an error (PS0108) may occur in the program.

Limitations and precautions

(1) The G code commands which can be used are as follows in the mode of polar coordinate interpolation:

G Code	Content
G01	Linear interpolation
G02	Circular interpolation (CW)
G03	Circular interpolation (CCW)
G04	Dwell
G40~G42	Tool radius compensation
G65	User macro program (non-modal call)
G66	User macro program (modal call)
G67	User macro program cancelled (modal call cancelled)
G90/G91	Absolute value/increment
G98	Fixed cycle/return of initial point
G99	Fixed cycle/return of R point

In polar coordinate interpolation, an error may occur in the program if the G codes other than above are issued.

(2) For the program block in polar coordinate interpolation, the program cannot be re-started.

(3) Please set the workpiece coordinate system before designating the polar coordinate interpolation, making the center of the rotation axis become the point of origin of the coordinate system. Please do not change the coordinate system in the mode of polar coordinate interpolation. (G50, G52, G53, resetting of relative coordinates, G54~G59 etc.)

(4) The feed speed in polar coordinate interpolation is the interpolation speed on the polar coordinate interpolation plane (orthogonal coordinate system). (Change with the relative speed of the tool by converting the polar coordinate)
On the polar coordinate interpolation plane (orthogonal coordinate system), the feed speed on the side of the rotation axis after polar coordinate interpolation will be increased significantly while passing through the center of the rotation axis.

(5) The movement command of the axis beyond the polar coordinate interpolation plane is unrelated to polar coordinate interpolation.

(6) The current position of polar coordinate interpolation indicates all the practical coordinates. Only the "Surplus movement amount" indicates the movement amount on the input plane of the polar coordinate.

(7) Power-on and resetting indicate cancellation of polar coordinate interpolation.

(8) An error may occur in the program in case of the axis not homed in the command axis of polar coordinate interpolation.

(9) Tool radius compensation must be cancelled when polar coordinate interpolation is cancelled.

(10) Return to the plane selected before polar coordinate interpolation by switching from polar coordinate interpolation to turning.

(11) An error may occur in the program while issuing the command for polar coordinate interpolation in the mirror.

(12) An error may occur in the program while issuing the commands for cylindrical interpolation/polar coordinate interpolation in the mode of polar coordinate interpolation.

(13) Rigid tapping cycle of G84/G88 cannot be used in the mode of polar coordinate interpolation. Flexible tapping can be used in such mode. However, please do not execute the command for rigid tapping.

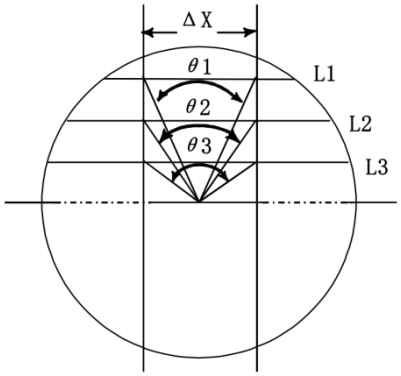
(14) For the program block in G12.1, the program cannot be re-started.

(15) Cutting feed speed of the rotation axis:

Polar coordinate interpolation transforms the shape programmed in the Cartesian coordinate system into tool movement in the rotation axis (C-axis) and linear axis (X-axis), however, when the tool moves near the workpiece center, the velocity component of the C-axis becomes larger and exceeds the maximum cutting feed speed of the C-axis (the setting value of parameter #1225), the system will automatically reduce the feed speed to clamp the speed.

Warning!!

Considering straight lines L1, L2 and L3, inside the Cartesian coordinate system, assume that the movement amount per unit time of the tool is ΔX under the feed speed of the F command. When the tool moves from L1 to L2 and to L3 and is close to the center, the movement amount of the C-axis corresponding to the movement amount per unit time of ΔX in the Cartesian coordinate system is increased from θ1 to θ2 and to θ3. In other words, the movement amount of the C-axis per unit time is increased with approaching of the tool close to the C-axis at the workpiece center. The tool movement in the Cartesian coordinate system is transformed into the movement of the C-axis and X-axis. As a result, the velocity component of the C-axis may exceed the maximum cutting feed speed of the C-axis.



L: The distance between the tool center and the workpiece center when the tool center is close to the workpiece center (mm)

R: The highest cutting feed speed of the C-axis (deg/min)

Under the assumption above, the speed of the F command in polar coordinate interpolation can be obtained through the following formula: If such speed goes beyond the highest cutting feed rate of the C-axis, the speed will be controlled automatically through polar coordinate interpolation.

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

(16) The alarm will be sent out from the decoder and the conditions for starting/cancelling polar coordinate interpolation shall mainly be checked.

Alarm Number	Information	Notes
PS0108	Illegal condition in polar coordinate interpolation	The condition is incorrect when polar coordinate interpolation starts or is cancelled: 1) G12.1/G13.1 command is not specified at a separate program block. 2) G12.1/G13.1 command is not cancelled under the mode of tool nose radius compensation (G40). 3) The G code limited by G12.1/G13.1 is used. 4) The plane selection command is executed under the mode of G12.1. 5) Incorrect definition of parameters #3147 and #3148. 6) The T command is executed in the mode of polar coordinate interpolation and such command must be issued before polar coordinate interpolation.

4.12 Polar coordinate command (G16/G15)

Function and purpose

The function aims at designating the coordinates at the end point as those of radius and angle.

Command format

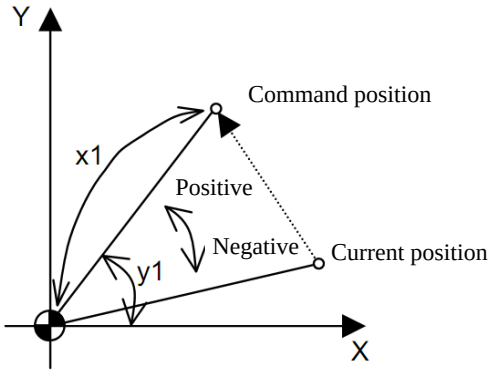
G16	Polar coordinate command ON

G15	Polar coordinate command OFF

Detailed description

(1) The coordinate command is polar coordinate interpolation within the section from starting to cancellation of polar coordinate interpolation.

```
G1x ;                          Plane selection of polar coordinate command (G17/G18/G19)
G16 ;                          Polar coordinate command  ON
G9x G01 Xx1 Yy1 F2000 ;        Polar coordinate command
   :
G15 ;                          Polar coordinate command  OFF
```



G90/G17 (X-Y plane)

G9x: Selection of the center of polar coordinate commands (G90/G91)

G90: Workpiece coordinate system origin is the center of the polar coordinate

G91: Current position is the center of the polar coordinate

x1: The 1st axis on the plane…radius of the polar coordinate of the command

y1: The 2nd axis on the plane…angle of the polar coordinate of the command

(2) Plane selection in the mode of polar coordinate command shall be executed through G17/G18/G19.

(3) The polar coordinate command is modal. The polar coordinate command mode at power-on and reset is OFF (G15).

(4) The command shall be executed with the 1st axis on the plane selected in the mode of polar coordinate command as radius and the 2nd axis as angle. For instance, if X-Y plane is selected, the command shall be executed with "X" as radius and "Y" as angle.

(5) The angle is positive when CCW direction of the plane is selected, and the angle is negative when CW direction is selected.

(6) Both the radius and angle can be designated through the absolute value/incremental value command (G90/G91).

(7) If the radius is the command of absolute value, the command shall be performed as per the distance from the point of origin of the workpiece coordinate system (the local coordinate system if the local coordinate system has been set).

(8) If the radius is a command for incremental values, the end point of the last program block shall serve as the center of the polar coordinate, and the incremental values behind it shall be designated. The command shall be performed with angle as the incremental value for that when the last program block starts.

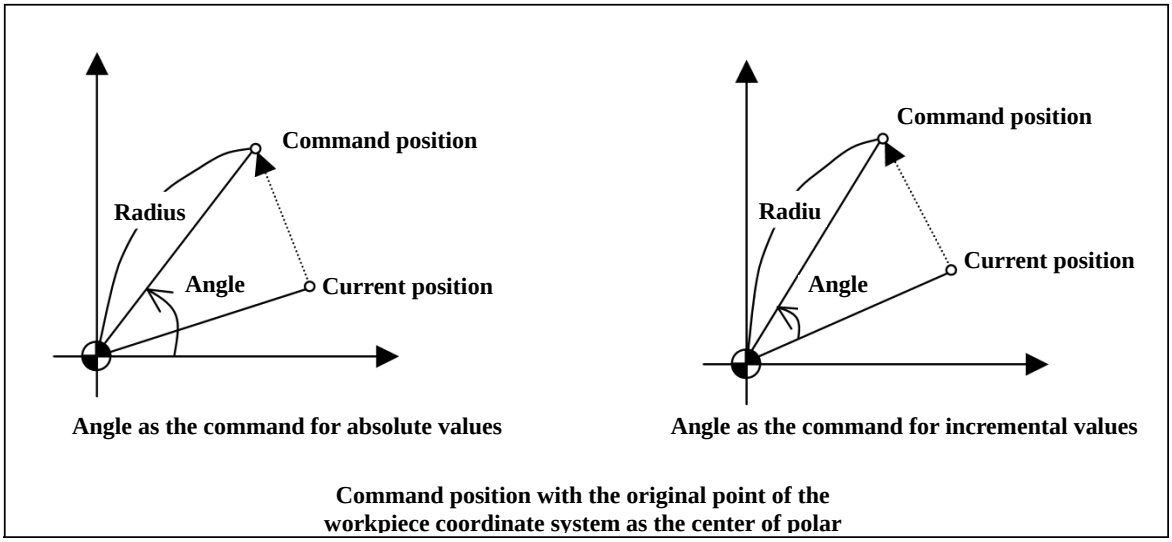
(9) If the radius of the command is negative, the command value of radius shall be changed as absolute values, and the same action shall be performed by adding 180 °to the command value of angle.

Command position

(1) When the point of origin of the workpiece coordinate system is the center of the polar coordinate

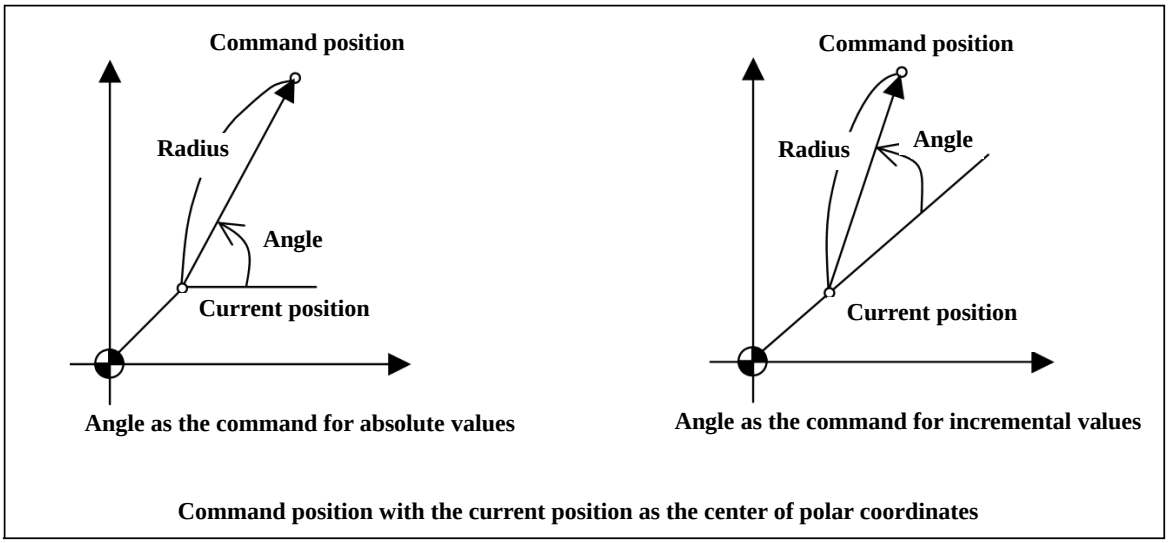
The radius shall be designated through absolute values, and the point of origin of the workpiece coordinate system serves as the center of the polar coordinate.

However, the point of origin of the local coordinate system is the center of the polar coordinate only when the local coordinate system (G52) is used.



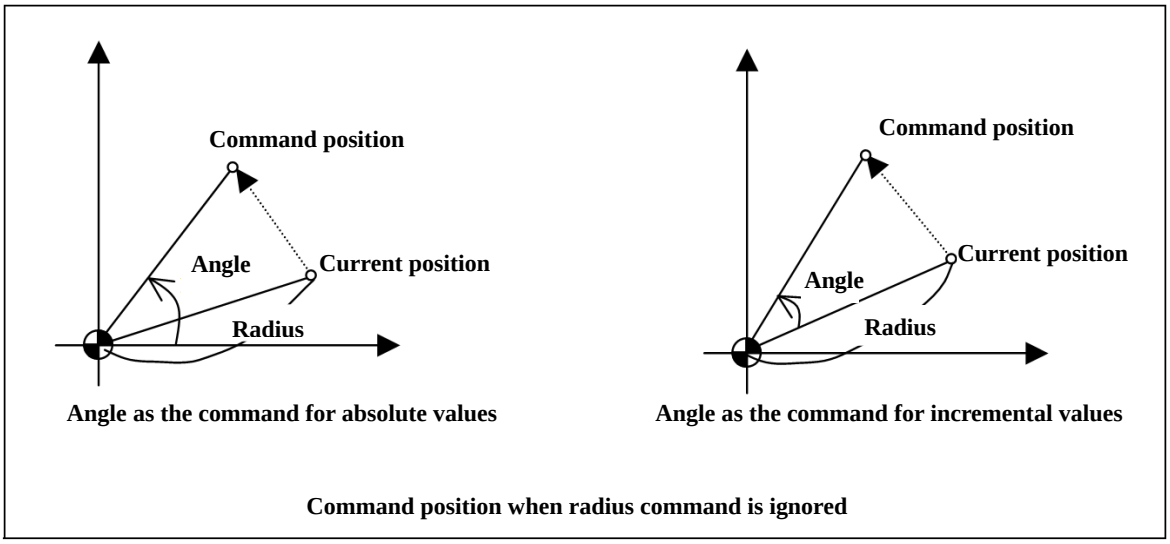
(2) When the current position is the center of the polar coordinate

The radius shall be designated through incremental values, and the current position serves as the center of the polar coordinate.



(3) When the radius command is ignored

When the radius command is ignored, the point of origin of the workpiece coordinate system shall serve as the center of the polar coordinate. The distance from the center of the polar coordinate to the current position is the radius. The point of origin of the local coordinate system is the center of the polar coordinate only when it is used as the local coordinate system (G52).

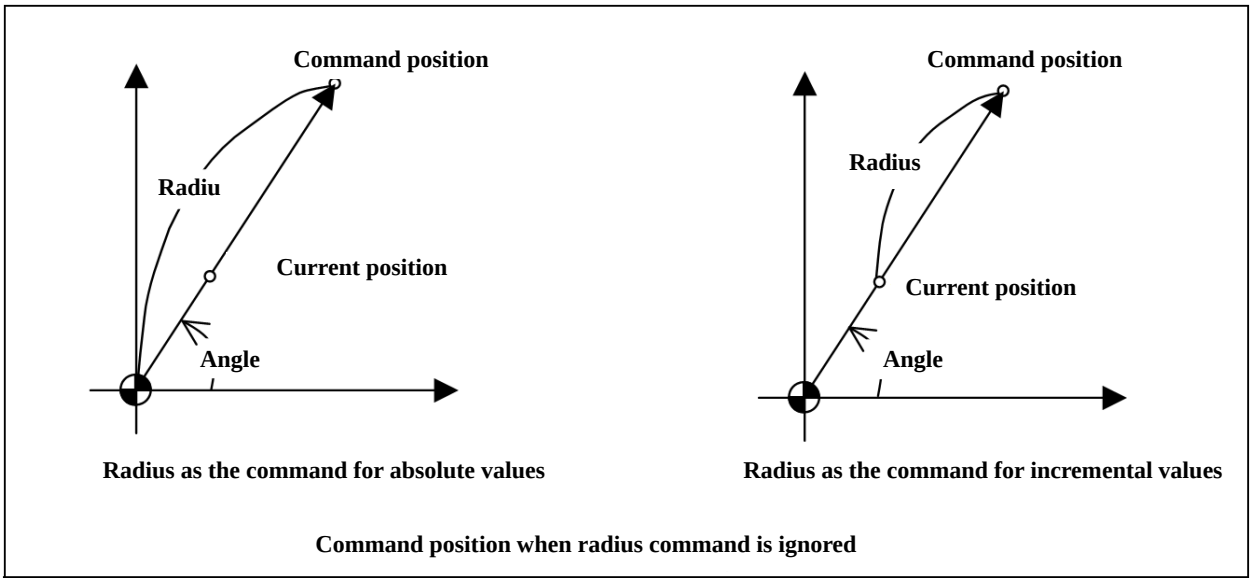


(4) When angle command is ignored

When the angle command is ignored, the angle at the current position of the workpiece coordinate system is the angle command.

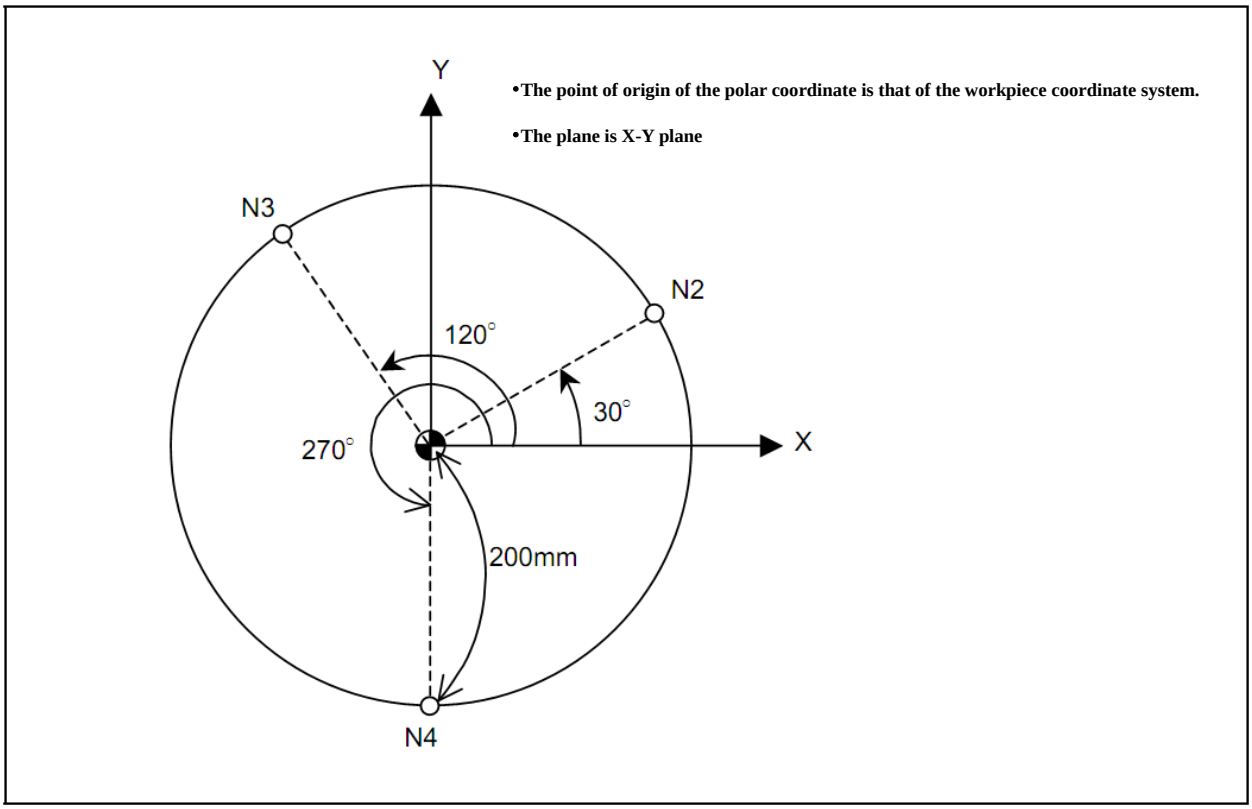
If the radius is designated through absolute values, the point of origin of the workpiece coordinate system is the center of the polar coordinate. The point of origin of the local coordinate system is the center of the polar coordinate only when the local coordinate system (G52) is used.

On the contrary, if the radius is designated through incremental values, the current position is the center of the polar coordinate.



Program examples

When the point of origin of the workpiece coordinate system is that of the polar coordinate,



(1) Both the radius and angle are the command for absolute values

N1 G17 G90 G16 ;	Polar coordinate command, selection of the X-Y plane The point of origin of the polar coordinate is that of the workpiece coordinate system
N2 G85 X200. Y30. Z-20. F200. ;	Radius 200 mm; angle 30 °
N3 Y120. ;	Radius 200 mm; angle 120 °
N4 Y270. ;	Radius 200 mm; angle 270 °
N5 G15 G80 ;	Polar coordinate command cancelled

(2) When radius serves as the command for absolute values and angle as that for incremental values

N1 G17 G90 G16 ;	Polar coordinate command, selection of the X-Y plane The point of origin of the polar coordinate is that of the workpiece coordinate system
N2 G85 X200. Y30. Z-20. F200. ;	Radius 200 mm; angle 30 °
N3 G91 Y90. ;	Radius 200 mm, angle + 90 °
N4 Y150. ;	Radius 200 mm, angle +150 °
N5 G15 G80 ;	Polar coordinate command cancelled

4.13 High-speed high-precision machining (G05 P3)

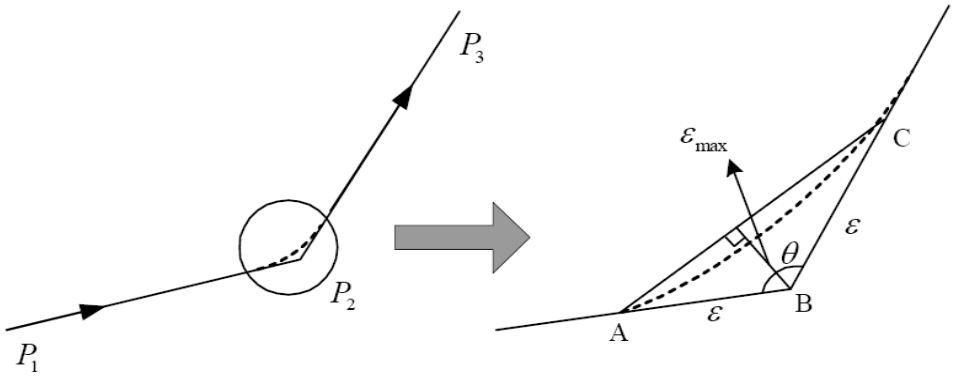
Function and purpose

As for the high-speed and high-precision machining of the tiny program block with the free-form surface, the cubic B-spine is used for smoothing transition, to guarantee second-order or above smooth connection of the tool path, thus realizing continuous acceleration of servo axes of the machine, more stable operation of the machine and higher machining quality.

Command format

G05 P3; B-spline transition mode on G05 P0; B-spline transition mode off

Detailed description



(1) The contour error of the maximum value is determined by parameter #1473.

(2) Curve transition of the B-spine shall not be operated in the program which has 3-axis and above. High-speed machining is invalid during 5-axis machining.

(3) It is recommended that ON (G05 P3)/OFF (G05 P0) of curve transition of the spline be made at a separate program block.

(4) Certain error may occur during single-block orientation if such function is adopted. Please use G05P0 if single block orientation is required.

Program examples

```
G91 G28 Z0.0
G00 G90 G54 X0 Y0 S3000 M03
Z20.
G05 P3                          (B-spline transition mode on)
G01 X0.1 Y0.1 Z0.1 F1000
 ⋮
X0.23 Y0.23 Z0.23
X0.45 Y0.45 Z0.45
 ⋮
G05 P0                          (B-spline transition mode off)
M30
```

4.13.1 Relevant parameters of algorithm of high-speed machining

Parameters	Notes	Remark
#1200.7	G0 positioning does not transition: 0: Invalid 1: Valid	When the parameter is set to 0, under the G05P3 algorithm, G0 performs B-spline curve transition, that is, the G0 positioning adopts the G05P3 algorithm; When the parameter is set to 1, under the G05P3 algorithm, G0 does not perform B-spline curve transition, that is, G0 positioning uses the G05P0 algorithm. Whether this parameter is set to 0 or 1, under the G05P3 algorithm, G1 performs B-spline transition.
#1400.6	Acceleration/deceleration mode before interpolation: 0: linear type; 1: S type;	S type can realize more stable operation of the machine and better machining effect.
#1209	Default high-speed mode of system power-on: 0: G05P0 3: G05P3	If 3 is selected, for high-speed machining, it is unnecessary to add the command of G05P3 in the program.
#1409	Number of the program block of far-sighted machining	For the tiny program block, the higher the number of those subjected to far-sighted machining is, the more stable the speed will be and the better the machining efficiency and effect will be. (The highest set value is 45)
#1440	Maximum acceleration	Only effective for control over linear acceleration/deceleration; the lower the acceleration is, the lower the efficiency will be, but the better the machining effect will be (valid when #1400.6=0).
#1224	Maximum cutting feed resultant speed (common to all axes)	The highest feed speed shall be limited as the speed set by the parameter if the feed speed of machining given is larger than this speed.
#1225	Maximum cutting feed speed for each axis in automatic mode	When #1400.6=1, the acceleration/deceleration mode before cutting feed is of S type; under the precondition that Vm is determined, the higher T1/T2 is, the lower the efficiency will be, but the better the machining effect will be. (For related content, please refer to "4.4 Feed speed parameters (1200～1399)" in Part 3 Chapter 4 of *Installation and Commissioning Manual of GSK25iMc/GSK25iTc Series Bus-Type Milling/Turning Machining Center CNC System*)
#1480	Acceleration and deceleration S-type time constant T1 before cutting feed	
#1481	Acceleration and deceleration S-type time constant T2 before cutting feed	

Parameters	Notes	Remark
#1226	Fast moving speed of each axis in automatic mode	When #1400.6=1, the acceleration/deceleration mode before rapid feed is of S type; under the precondition that Vm is determined, the lower T1/T2 is, the higher the rapid positioning efficiency will be. (For related content, please refer to "4.4 Feed speed parameters (1200~1399)" in Part 3 Chapter 4 of *Installation and Commissioning Manual of GSK25iMc/GSK25iTc Series Bus-Type Milling/Turning Machining Center CNC System*)
#1410	Acceleration/deceleration S-type acceleration/deceleration time constant T1 before fast feed	
#1411	Time constant of S-type acceleration/deceleration of rapid feed T2	
#1473	Maximum contour error of system	Curve transition of G5P2/G5P3 will be affected directly; the lower the contour error is, the higher the precision will be, but the lower the efficiency will be. A proper contour error will realize more stable operation of the machine and better efficiency.
#1442	Highest feed acceleration of circular interpolation;	The engagement speed of G5P2/G5P3 will be affected directly; the higher the acceleration is, the higher the engagement speed will be, but the intenser the vibration of the machine will be; a proper highest acceleration of circular interpolation will realize more stable operation of the machine and better efficiency.
#1501	Allowable variation of the highest speed	At the engagement position of blocks, allowing for the change in the vector, even though the resultant speed is changing continuously, the speed of the split axis is under step change. Such parameter is used to limit the largest step (such parameter is universal for each axis) of each axis.

4.14 Preset function of the pre-read program block (G08)

Function and purpose

Number of pre-read program blocks of the NC command in operation

Command format

G08 Q_; Enable the preset function at the pre-reading block
G08 Q0; Cancel the preset function at the pre-reading block and restore to the maximum number of blocks

Q_	Number of pre-read program block; Q value 1000 at most

Chapter V Simplified Programming

5.1 Linear angle commands (G01 X_ /Z_ , A_ and G01 U_ /W_ ,A_)

Only applicable to the turning system

Function and purpose

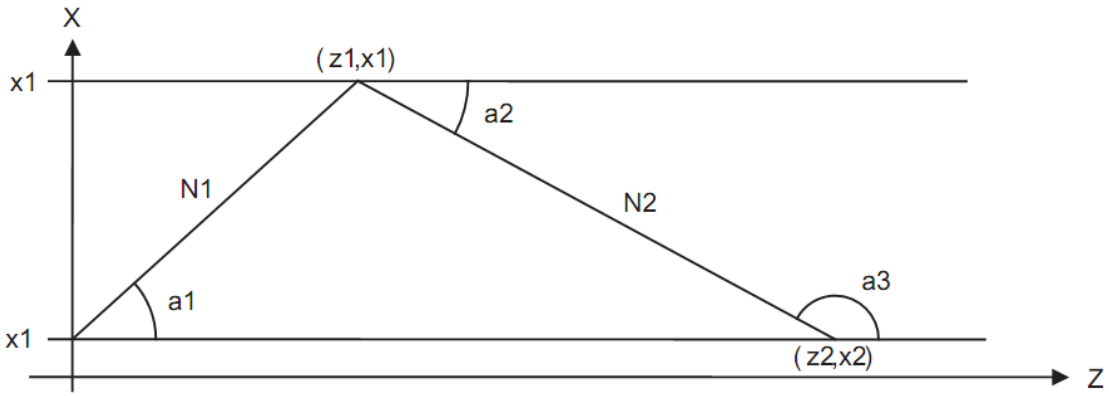
The coordinates of the end point is calculated automatically by designating the angle of the straight line and the coordinates of the end point of any axis.

Command format

```
N1 G01 Xx1(Zz1) ,Aa1;
N2 G01 Xx2(Zz2) ,A-a2;
```

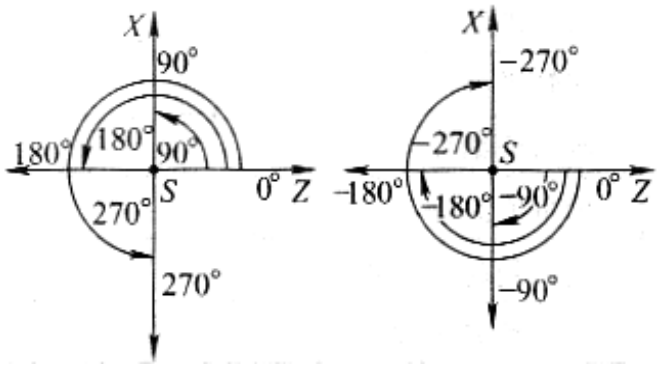
The angle and coordinates of the X-axis or the Z-axis shall be specified.

The command plane shall be selected through the G17~G19.



Detailed description

(1) The angle of "horizontal axis + direction" of the plane selected shall be adopted, i.e. + for the counterclockwise direction (CCW) and – for the clockwise direction (CW).



(2) Any axis on the plane selected shall be specified as the end point.

(3) When both the angle and coordinates of both axes are specified, ignore the former.

(4) If only the angle is specified, it shall be deemed as geometrical.

(5) Any angle of the start point (a1) and the end point (a2) can be used.

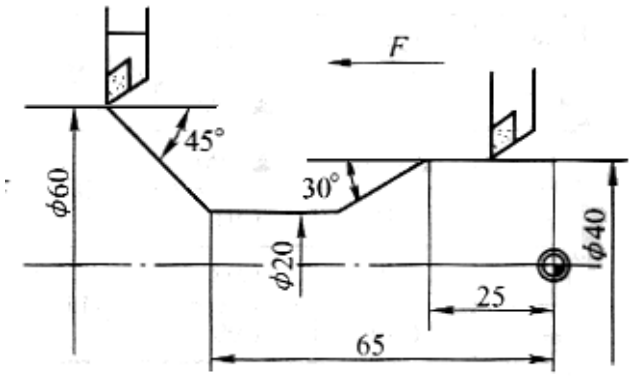
(6) The function is only valid for G01 command and is invalid for other interpolations or position.

(7) The slope a is -360.000≤a≤360.000. Once the command goes beyond such scope, designate with the remainder after being divided by 360.

Program examples

For instance, for the workpiece in the figure below, the tool is moved from right to left and the programming is as follows:

G00 X40 Z0;
G01 Z-25.0 F0.2;
G01 X20.0 ,A210.0 F0.1;
G01 Z-65.0 F0.2;
G01 X60.0 ,A135.0;



5.2 Chamfering/smoothing (,C/,R)

Function and purpose

In the program block designating the corner constituted by continuous straight lines or arc of any angle, chamfering and smoothing R shall be performed by attaching ",C" or ",R" to the program block specified first.

5.2.1 Chamfering (G01/G02/G03 X_ Z_ ,C_)

Function and purpose

For two program blocks comprising continuous straight lines or arc, chamfering shall be performed by designating "C" at the 1st program block.

Command format

N100 G01 X_ Y_ ,C_; N200 G01 X_ Y_ ;	N100 G03 X_ Z_ I_ K_ ,C_ ; N200 G01 X_ Z_ ;

,C_	The length from the assumed corner to the start point of chamfering or the end point of smoothing

Chamfering is performed at the intersection between N100 and N200.

Detailed description

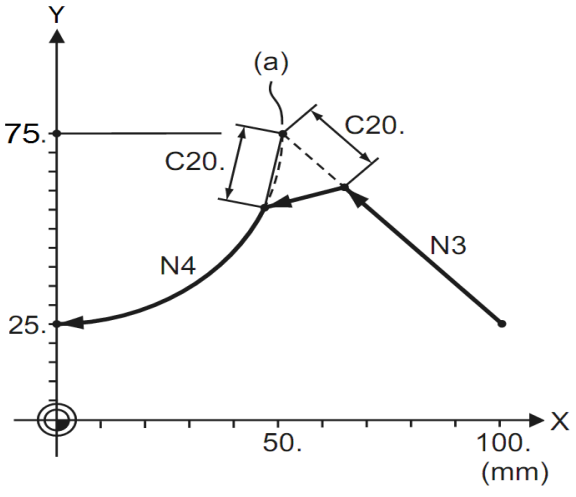
(1) The start point at the next program block of chamfering is the assumed intersection of corner.
(2) If numerous chamfering commands are specified at the same program block, that specified finally shall be valid.
(3) In case of two chamfering/smoothing R at the same program block, the latter shall be valid.
(4) Tool offset shall be calculated for the shape chamfered.
(5) In case of positioning or thread cutting commands at the program block for chamfering or the next program block, an error may occur in the program.
(6) In case of G command or other commands other than group 01 at the next program block of chamfering, an error may occur in the program.
(7) At the program block where chamfering is specified, an error may occur in the program once the movement amount is less than the chamfering amount.
(8) At the next program block where chamfering is specified, an error may occur in the program once the movement amount is less than the chamfering amount.
(9) Chamfering is also the command value of radius even for the diameter command.
(10) At the next program block of chamfering command, an error may occur in the program if without movement commands.

Programming examples

(1) Straight line - straight line

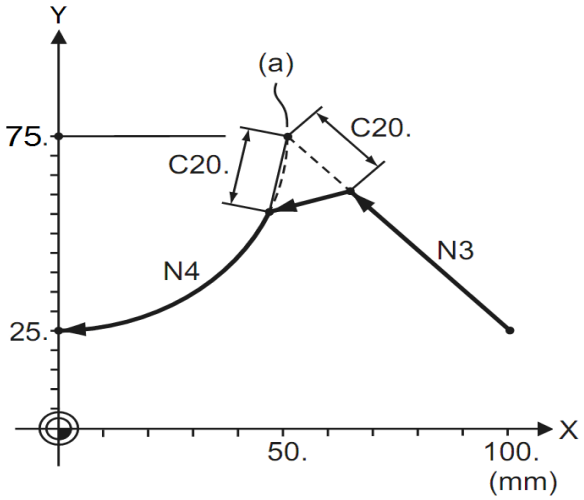
Milling

G91 X100. ,C10 F100;
X100. Y140.;



Turning

G01 W100. ,C10 F100;
U280. W100.;



(a) Start point of chamfering (b) Imaginary corner intersection (c) End point of chamfering

(2) Straight line - arc

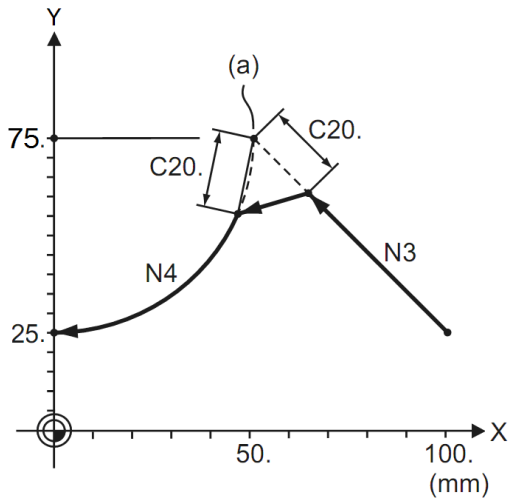
Milling

Absolute value command

N1 G91 G28 X0 Y0 ;
N2 G90 G00 X100. Y25.;
N3 G01 X50. Y75.,C20. F100 ;
N4 G02 X0 Y25. I-50. J0 ;

Relative value command

N1 G91 G28 X0 Y0 ;
N2 G00 X100. Y25.;
N3 G01 X-50. Y50. ,C20. F100 ;
N4 G02 X-50. Y-50. I-50. J0;



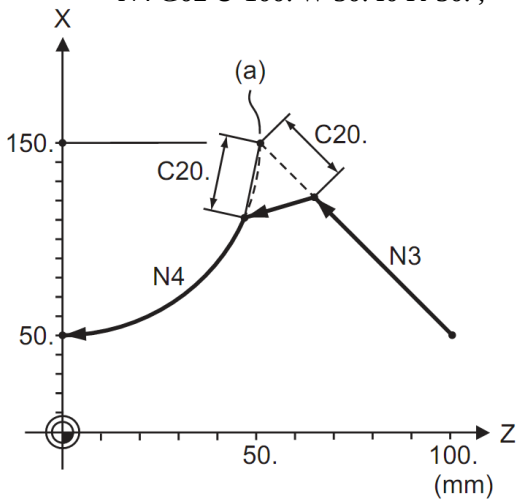
Turning

Absolute value command

N1 G28 U0 W0 ;
N2 G00 X50. Z100. ;
N3 G01 X150. Z50. ,C20. F100 ;
N4 G02 X50. Z0 I0 K-50. ;

Relative value command

N1 G28 U0 W0 ;
N2 G00 U50. W100. ;
N3 G01 U100. W-50. ,C20. F100 ;
N4 G02 U-100. W-50. I0 K-50. ;



(a) Assumed intersection of corner

(3) Arc - arc

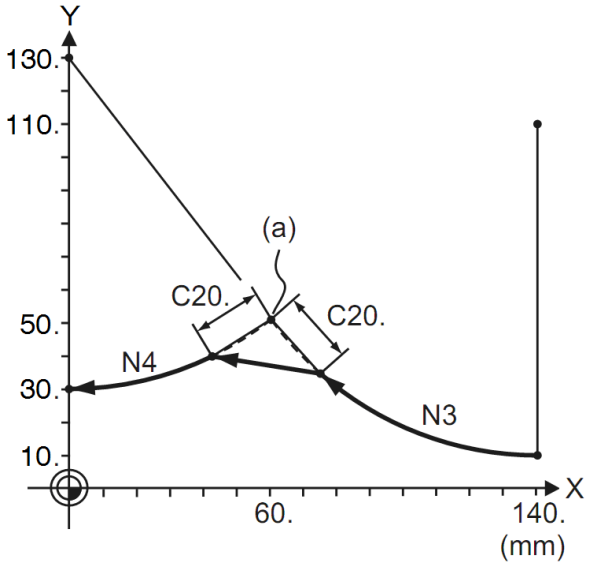
Milling

Absolute value command

N1 G91 G28 X0 Y0;
N2 G90 G00 X140. Y10.;
N3 G02 X60. Y50. I0. J100.,C20. F100 ;
N4 X0 Y30. I-60. J80.;

Relative value command

N1 G91 G28 X0 Y0 ;
N2 G00 Y10. X140. ;
N3 G02 X-80. Y40. R100. ,C20. F100 ;
N4 X-60. Y-20. I-60. J80.;



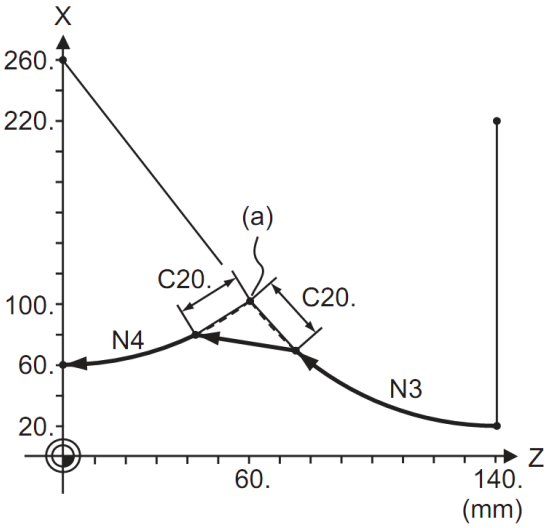
Turning

Absolute value command

N1 G28 U0 W0;
N2 G00 X20. Z140. ;
N3 G02 X100. Z60. I100. K0. ,C20. F100 ;
N4 X60. Z0 I80. K-60. ;

Relative value command

N1 G28 U0 W0 ;
N2 G00 U20. W140. ;
N3 G02 U80. W-80. R100. ,C20. F100 ;
N4 U-40. W-60. I80. K-60.;



(a) Assumed intersection of corner

5.2.2 Smoothing (G01/G02/G03 X_ Z_ ,R_)

Function and purpose

Smoothing R shall be performed through the radius arc specified by ",R_" in front of and behind the virtual corner to which smoothing R is assumed not to be made.

Command format

N100 G01 X_ Z_ ,R_ ; N200 G01 X_ Z_ ;	N100 G03 X_ Z_ I_ K_ ,R_ ; N200 G01 X_ Z_ ;

R_	Arc radius of smoothing R

Smoothing R shall be performed at the intersection between N100 and N200.

Detailed description

(1) The start point of the next program block of smoothing R is the assumed intersection of corner.
(2) In case of chamfering and smoothing R at the same program block, the latter command is valid.
(3) Tool offset is the calculation made for shape after smoothing R.

(4) In case of positioning or thread cutting commands at the program block for smoothing R or the next program block, an error may occur in the program.

(5) In case of G command or other commands other than the group 01 at the next program block of smoothing R, an error may occur in the program.

(6) At the program block where the command of smoothing R is issued, an error may occur in the program once the movement amount is less than R amount.

(7) At the next program block where the command of smoothing R is issued, an error may occur in the program once the movement amount is less than R amount.

(8) Smoothing R is also the command value of radius even for the diameter command.

(9) At the next program block of smoothing R, an error may occur in the program if without movement commands.

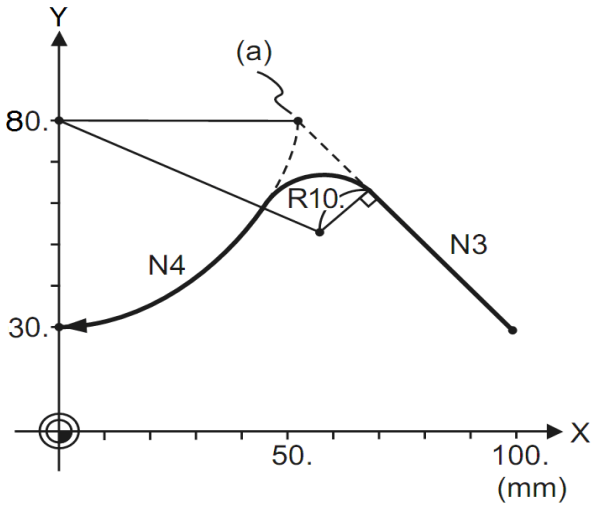
(10) At the program block of arc commands, smoothing R cannot be specified through "R". "R" is the command for arc radius.

Programming examples

(1) Straight line - straight line

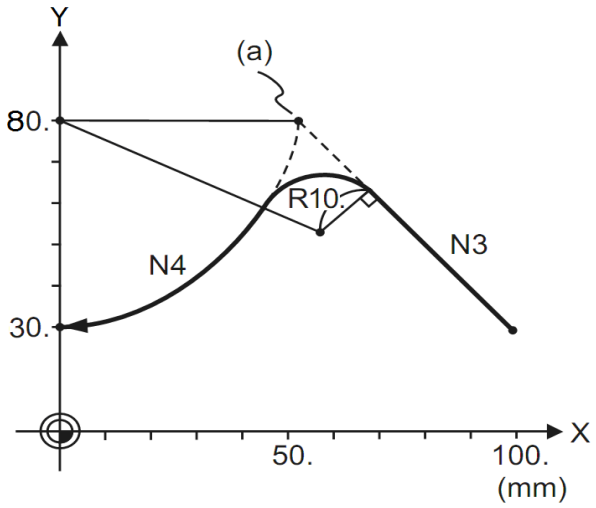
Milling

G91 G01 X100. ,R10. F100;
X100. Y140.;



Turning

G01 W100. ,R10. F100;
U280. W100.;



(a) Start point of rounding R (b) End point of rounding R (c) Imaginary corner intersection

(2) Straight line - arc

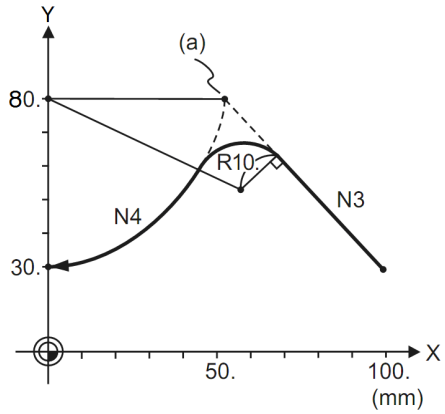
Milling

Absolute value command

N1 G91 G28 X0 Y0 ;
N2 G90 G00 X100. Y30.;
N3 G01 X50. Y80. ,R10. F100 ;
N4 G02 X0 Y30. I-50 J0. ;

Relative value command

N1 G91 G28 X0 Y0 ;
N2 G00 X100. Y30.;
N3 G01 X-50. Y50. ,R10. F100 ;
N4 G02 X-50. Y-50. I-50. J0;



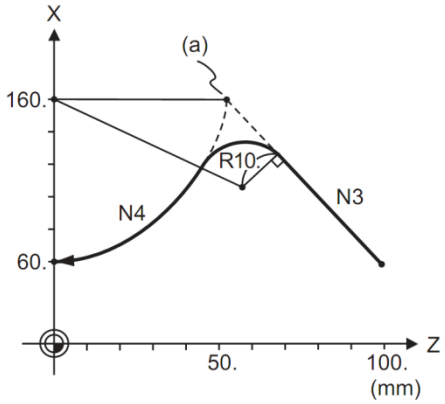
Turning

Absolute value command

N1 G28 U0 W0 ;
N2 G00 X60. Z100. ;
N3 G01 X160. Z50. ,R10. F100 ;
N4 G02 X60. Z0 I0 K-50. ;

Relative value command

N1 G28 U0 W0 ;
N2 G00 U60. W100. ;
N3 G01 U100. W-50. ,R10. F100 ;
N4 G02 U-100. W-50. I0 K-50. ;



(a) Assumed intersection of corner

(3) Arc - arc

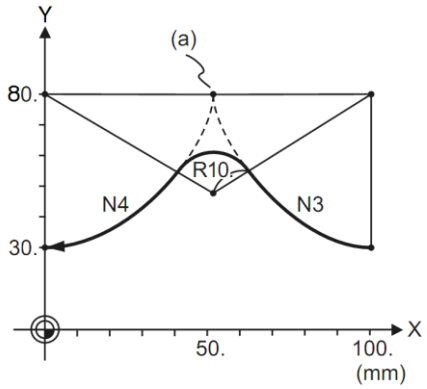
Milling

Absolute value command

N1 G91 G28 X0 Y0;
N2 G90 G00 X100. Y30.;
N3 G02 X50. Y80. R50 ,R10. F100 ;
N4 X0 Y30. R50. ;

Relative value command

N1 G91 G28 X0 Y0 ;
N2 G00 X100. Y30.;
N3 G02 X-50. Y50. I0 J50. ,R10. F100 ;
N4 X-50. Y-50. I-50. J0. ;



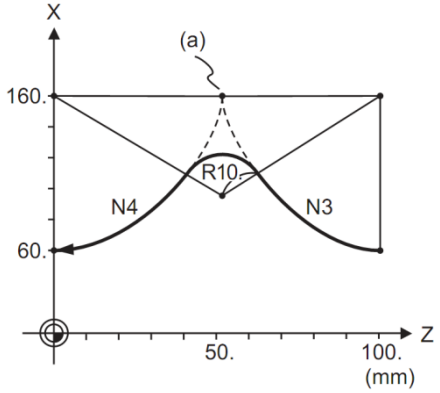
Turning

Absolute value command

N1 G28 U0 W0;
N2 G00 X60. Z100. ;
N3 G02 X160. Z50. R50 ,R10. F100 ;
N4 X60. Z0 R50. ;

Relative value command

N1 G28 U0 W0 ;
N2 G00 U60. W100. ;
N3 G02 U100. W-50. I50. K0 ,R10. F100 ;
N4 U-100. W-50. I0. K-50. ;



(a) Assumed intersection of corner

5.3 Direct programming of pattern size

Only applicable to the turning system

Function and purpose

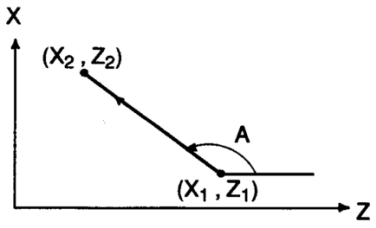
Contour graphs are used for programming directly as per the graph of workpieces. The intersection of straight lines may be input through coordinates or angles during programming and multi-point connections can be made as needed. Such programming mode can improve the reliability of the program and the programming efficiency, thus simplifying the programming commands.

In the direct pattern dimension programming mode, the connection programming of multiple straight lines (or arcs) can be completed through one program block, and the connection between straight lines can be smoothly transitioned by either sharp corners, or smoothed by arcs, straight line chamfering, etc. Chamfering size at the bottom of straight lines/arc is defined through direct sizes, and calculations at the transition point are completed automatically through CNC.

Command format

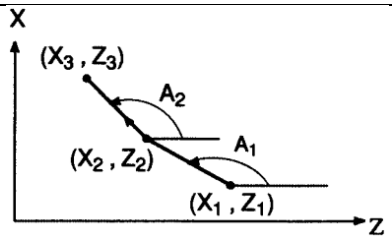
(1) One straight line (angle programming)

```
G01 X2_(Z2_) , A_;
```



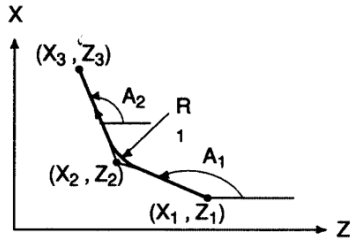
(2) Two straight lines

```
G01 ,A1_;
    X3_ Z3_ ,A2_;
```



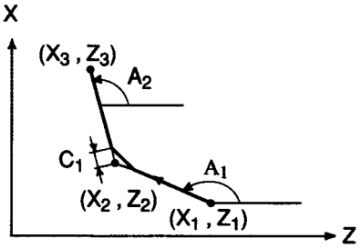
(3) Two straight lines and transition arc

```
G01 X2_ Z2_ ,R1_;                    Or    G01 ,A1_ ,R1_;
    X3_ Z3_;                                 X3_ Z3_ ,A2_;
```



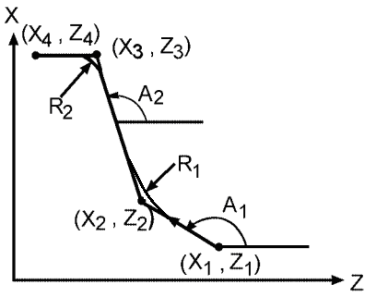
(4) Two straight lines and chamfer angle

G01 X2_ Z2_ ,C1_;	Or	G01 ,A1_ ,C1_;
X3_ Z3_;		X3_ Z3_ ,A2_;



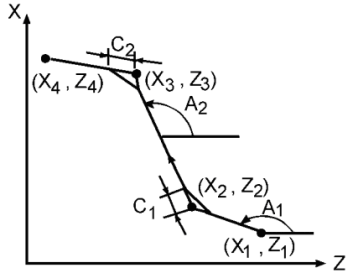
(5) Three straight lines and two transition arcs

G01 X2_ Z2_ ,R1_;	Or	G01 ,A1_ ,R1_;
X3_ Z3_ ,R2_;		X3_ Z3_ ,A2_ ,R2_;
X4_ Z4_;		X4_ Z4_;



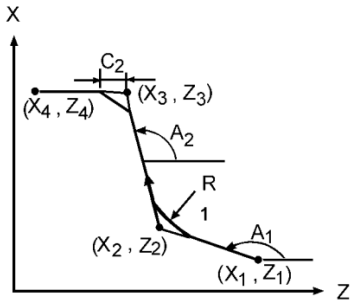
(6) Three straight lines and two chamfer angles

G01 X2_ Z2_ ,C1_;	Or	G01 ,A1_ ,C1_;
X3_ Z3_ ,C2_;		X3_ Z3_ ,A2_ ,C2_;
X4_ Z4_;		X4_ Z4_;



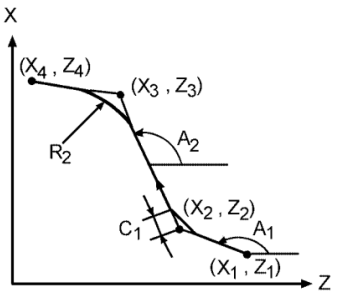
(7) Three straight lines, one transition arc and one chamfer angle

G01 X2_ Z2_ ,R1_;	Or	G01 ,A1_ ,R1_;
X3_ Z3_ ,C2_;		X3_ Z3_ ,A2_ ,C2_;
X4_ Z4_;		X4_ Z4_;



(8) Three straight lines, one chamfer angle and one transition arc

G01 X2_ Z2_ ,C1_;	Or	G01 ,A1_ ,C1_;
X3_ Z3_ ,R2_;		X3_ Z3_ ,A2_ ,R2_;
X4_ Z4_;		X4_ Z4_;



Precautions

(1) The dip angle A of straight lines, the chamfer angle C and the smooth angle R must be separated with ",".

(2) The following G codes cannot be specified in the program block in which pattern size is programmed directly, or among the program blocks in which the pattern size is specified directly of defined graphs.

① Group 00: G code (except for G04).

② Group 01: G02, G03, G90, G92, G94.

(3) Arc transition of the corner cannot be inserted into the program block for thread cutting.

(4) The angle difference for calculation of the intersection of the program shall be more than ±1 º on the ground that the movement distance obtained in the calculation is too large.

(5) If the angle difference of two straight lines is within ±1 º during the calculation of the intersection, an alarm will be given.

(6) If the angle difference of two straight lines is within ±1 º, ignore the chamfer or smooth angle.

(7) The angle command must be designated after the size command (programming of absolute values).

Chapter VI Feed Functions

Function and purpose

Used to specify the speed of the tool relative to movement of workpieces and is made up of F and the figures behind it. Feed can be classified into feed per minute and that per revolution as per the machining needs.

(1) For feed per minute, the unit of linear movement is in mm/min; if the spindle is a rotating axis, the unit is in °/min. Feed per minute is specified by preparing function words G94 (G98), and the value is a constant larger than zero, such as:
G94 (G98) G01 X20.0 F100; the feed speed is 100 mm/min.

(2) Feed per revolution: in the process of thread/boring, the feed speed is generally specified by feed per revolution (in mm/r) by preparing function words G95 (G99), such as:
G95 (G99) G01 X20.0 F0.2; the feed speed is 0.2 mm/r.

The feed speed shall not be negative during programming, and feed stop cannot be controlled through F0 generally. However, in the actual operation process, the feed speed value can be corrected through the feed override switch on the machine operation panel. Therefore, the feed speed value can be controlled to 0 through the override switch.

6.1 Dwell (G04)

Function and purpose

The function is used for pause movement of the machine through the program command, entering the state of "Time Wait". Therefore, starting of the next program block can be postponed.

Command format

G04 X_; or G04 U_; or G04 P_;

X_	Specify time (decimal points allowed)
U_	Specify time (decimal points allowed) (for A series of turning system)
P_	Specify time (decimal points not allowed)

Detailed description

Dwell G04 and perform the next program block as per the time delay designated.

Scope of command value of dwell time (X command)

Scope of Command Value	Unit of Dwell Time
0～9999.999	Seconds

Scope of command value of dwell time (P command)

Scope of Command Value	Unit of Dwell Time
0～9999999	0.001 s

Program examples

For instance: Dwell for 3.8s

G04 X3.8 or G04 P3800; or G4 U3.8

Precautions

(1) As a non-modal command, G04 is only valid in the current line.

(2) Non-to feed cutting is specified in G04 and perform the next program block after delay for the specified time. Moreover, in the mode of cutting (G64), non-to feed cutting can be specified for accurate stop for inspection.

(3) When P and X are specified simultaneously, perform through P.

(4) When X and U are specified simultaneously, perform through the latter. No alarm will occur if the former is negative.

(5) If X/P is specified as negative, an alarm will be given from the system.

(6) When X/P is specified as zero, the system will perform orientation.

6.2 Rapid feed speed

Function and purpose

Rapid feed speed can be set independently for each axis. The speed which can be set is 0 mm/min~999999.9999 mm/min, is limited by the specifications of the machine and the upper limit of speed. Please refer to the specifications of the machine for the set value of rapid feed speed. The path during positioning includes linear interpolation made at the start point and the end point as well as non-linear positioning (only applicable to the turning system) moving at the highest speed of each axis. Selectable via parameter #1401.0. The positioning times are all the same.

Note: Rapid feed override

For rapid manual and automatic feed, the override can be specified through the external input signal. There are two types and which type shall be selected depends on the specifications of PLC.

Type 1: There are 4 override levels, including F0, 25%, 50% and 100%. F0: It is set through parameter #1231.

Type 2: 0%~100% (override with a step of 10%).

6.3 Cutting feed speed

Function and purpose

As for specification of the feed speed of cutting commands, the feed per revolution or per minute of the spindle shall be specified. Such value will be memorized as a modal value once being designated. Clearing of the modal value of feed speed is only valid during power-on. The highest cutting feed speed is limited by the highest cutting feed speed (the set range is identical to the cutting feed speed) set by the parameter. Specify the cutting feed speed by address F and numbers. The cutting feed speed is valid for G01, G02, G03, G32, G33, G34 commands.

If F is not designed in the initial cutting command (G01/G02/G03) after being powered-on, operate as per the value set by parameter #1211.

Note: Cutting feed override

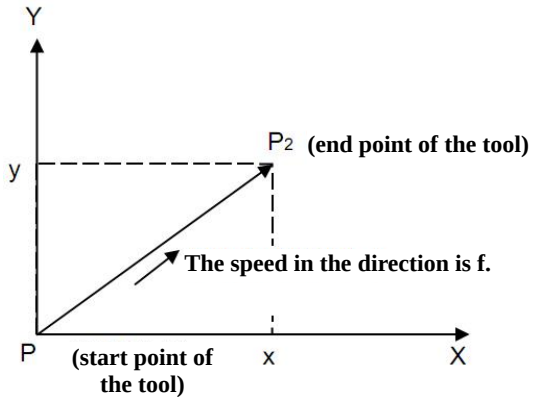
For manual and automatic cutting feed, the rate can be specified through the external input signal. The range of override is 0%~200% (with a step of 10%).

Detailed description

Control over linear axis

No matter how many axes are controlled by the machine, the feed speed specified through F will serve as the linear speed in the forward direction of the tool.

(Example) The feed speed is specified as f for control over the linear axis (X-axis and Z-axis)



$$\text{Feed speed of X-axis} = f \times \frac{x}{\sqrt{x^2+y^2}}$$

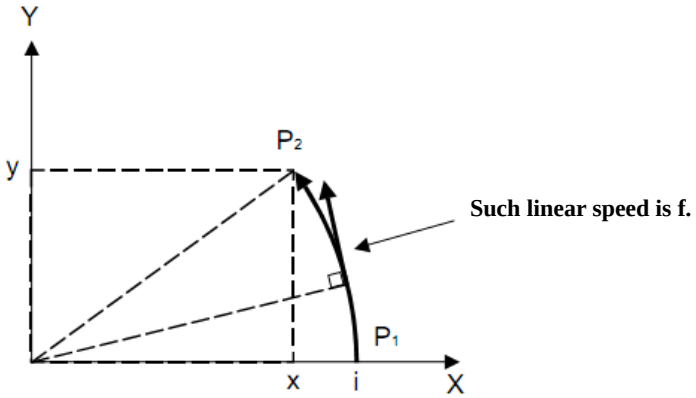
$$\text{Feed speed of Y-axis} = f \times \frac{y}{\sqrt{x^2+y^2}}$$

When only the linear axis is controlled, only the cutting speed needs to be specified in the program.

The feed speed specified is that of each axis after being composed into the corresponding component of the movement amount.

Note: When the tool is moved along the circumference through the linear control axis by virtue of circular interpolation, the tangential speed in the forward direction of the tool is the feed speed specified for the program.

(Example) When the linear axis (X-axis and Z-axis) is controlled through circular interpolation with the feed speed specified as f, the feed speed of the X-axis and Z-axis will vary with the movement of the tool. However, the resultant speed will still be kept as a fixed value f.

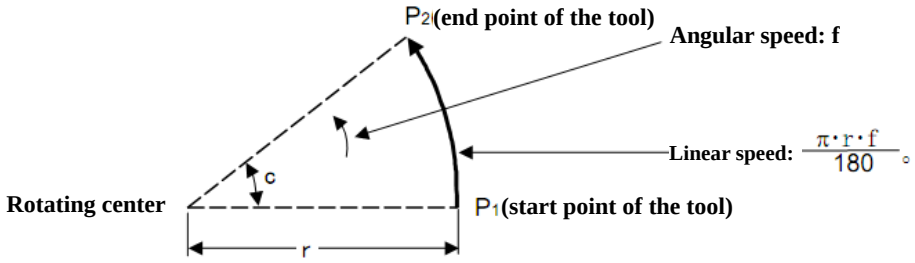


Control over rotation axis

During control over the rotation axis, the feed speed specified serves as the speed of the rotary speed, i.e. the angular speed. Therefore, the cutting speed in the forward direction of the tool is linear speed and will vary with the distance between the rotation center and the tool. For the speed specified by the program, such distance must be considered.

(Example) When the rotation axis (C-axis) is controlled with the feed speed specified as f

(f: in °/min)



In this case, the cutting speed (linear speed) in the forward direction of the tool is f_c

$$fc=f\times\frac{\pi\bullet r}{180}$$

Therefore, the feed speed specified by the program must be that shown in the figure below.

$$f=f\,c\times\frac{180}{\pi\bullet r}$$

Simultaneous control over the linear axis and the rotation axis

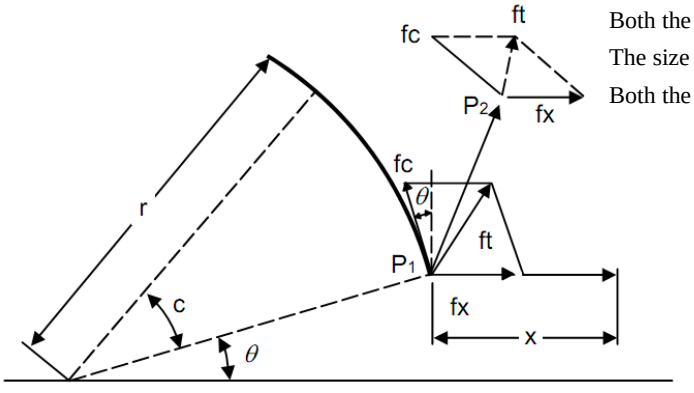
Control over the linear axis and the rotation axis is identical.

During control over the latter, all the values set through the feed speed (F) are valid as the linear speed with the value set through the coordinate statement (A, B, C) as angle. In other words, 1 ° of the rotation axis is equivalent to 1 mm of the linear axis.

Therefore, during simultaneous control over the linear axis and the rotation axis, the component of each axis corresponding to the value set through F is identical to that set during control over the linear axis. However, in this case, since both the size and direction of the speed component will not be changed in control over linear axis, the direction of the speed component will vary (the size remains unchanged) with the movement of the tool in control over the rotation axis, the feed speed in the forward direction of the tool will vary with the movement of the tool.

(Example) When the feed speed is specified through f and the linear axis (X-axis) and the rotation axis (C-axis) are controlled simultaneously

If the command value for increments of the X-axis is x and that for increments of the C-axis is c, the followings can be concluded:



Both the size and direction of Fx are constant
The size of Fc is constant, with variable direction
Both the size and direction of Ft are constant

Rotating center

The feed speed (linear speed) fx of the X-axis and that (angular speed) ω of the C-axis are

$$fx=f\times\frac{x}{\sqrt{x^2+c^2}}\text{ -------- (1)}$$

$$\omega = f \times \frac{c}{\sqrt{x^2+c^2}} \quad \text{-------} \quad (2)$$

The linear speed fc controlled by the C-axis is

$$fc = \omega \times \frac{\pi \times r}{180} \quad \text{-------} \quad (3)$$

If the speed in the forward direction of the tool at the start point P1 is set as ft and the component speed in the direction of the X-axis and Y-axis as ftx and fty respectively, the followings can be concluded

$$ftx = -r\sin(\frac{\pi}{180}\theta) \times \frac{\pi}{180}\omega + fx \quad \text{-------} \quad (4)$$

$$fty = -r\cos(\frac{\pi}{180}\theta) \times \frac{\pi}{180}\omega \quad \text{-------} \quad (5)$$

Where: R - distance between rotation center and the tool (unit: mm)
θ - angle between rotation center P1 and the X-axis (°)
According to the formulas (1), (2), (3), (4) and (5), the resultant speed ft is:

$$ft = \sqrt{ftx^2 + fty^2}$$

$$= f \times \frac{\sqrt{x^2 - x \times c \times r\sin(\frac{\pi}{180}\theta)\frac{\pi}{90} + (\frac{\pi \times r \times c}{180})^2}}{x^2+c^2} \quad \text{-------} \quad (6)$$

Therefore, the feed speed f specified in the program needs to be

$$f = ft \times \frac{x^2+c^2}{\sqrt{x^2 - x \times c \times r\sin(\frac{\pi}{180}\theta)\frac{\pi}{90} + (\frac{\pi \times r \times c}{180})^2}} \quad \text{-------} \quad (7)$$

However, since ft in the Formula (6) is the speed on P1 and the C-axis is under rotary advance, θ is constantly changing and ft also varies correspondingly. Therefore, to keep the cutting speed ft constant as much as possible, the rotation angle specified in a single program block must be as small as possible, to shorten the change variation of θ.

6.4 Feed per minute/per revolution (G94 (G98) and G95 (G99))

Feed per minute (G98) and that per revolution (G99) are adopted for A series of turning.

Function and purpose

Feed per minute

Follow-up values of F shall be specified directly by regarding the command issued from such program block as the feed speed (mm/min, inch/min) per minute by designating G94.

Feed per rotation

Follow-up values of F shall be specified directly by regarding the command sent out from such program block as the feed speed (mm/r, inch/r) per revolution of the spindle by designating G95.

A rotary encoder must be installed for the spindle while using such command.

Command format

G94(G98);	Feed per minute (mm/min)

G95(G99);	Feed per revolution (mm/r)

Detailed description

G94/G95 is a modal command.

(Example) G95 command is always valid after being designated until the next G94 command occurs.

(1) The command scope determined by the F code is as follows:

Command mode	Feed per minute	Feed per revolution
Command address	F(mm/min)	F(mm/r)
Minimum command unit	0.01	0.001
Command range	200000	1000

(2) The execution speed (actual movement speed of the machine) during feed per revolution is shown in the formula (Formula 1) below.

FC = F×N×OVR ..... (Formula 1)

FC : Execution speed (mm/min, inch/in)

F : Command speed (mm/r, inch/r)

N : Spindle speed (r/min)

OVR : Cutting feed override

Act in the vector direction of such command when the execution speed FC obtained through Formula 1 is used for specifying numerous axes simultaneously.

Precautions

(1) When the execution speed above exceeds the clamping speed of cutting feed, it will be limited by such clamping speed.

(2) If the spindle speed is zero during feed per revolution, an operation alarm (PS0243) will occur.

(3) Feed per minute shall be carried out during dry run, and the action shall be performed at the speed of dry run.

(4) Feed per minute (G94) shall be performed by default during power-on and

(5) F value will be cleared to zero when feed per minute (G94) is switched to feed per revolution (G95).

6.5 Accurate positioninf and inspection (G09)

Function and purpose

In case of sharp change in the feed speed of the tool, to mitigate collision of the machine and to prevent circular bead during corner cutting, the command of the next program block shall be started after the orientation state is confirmed or after decelerated inspection on time after the machine is stopped. Accurate positioning and inspection aim at preventing such circumstance.

G09 command is a non-modal command. The deceleration check is performed only when G09 (exact positioning check) command is issued in the current program block.

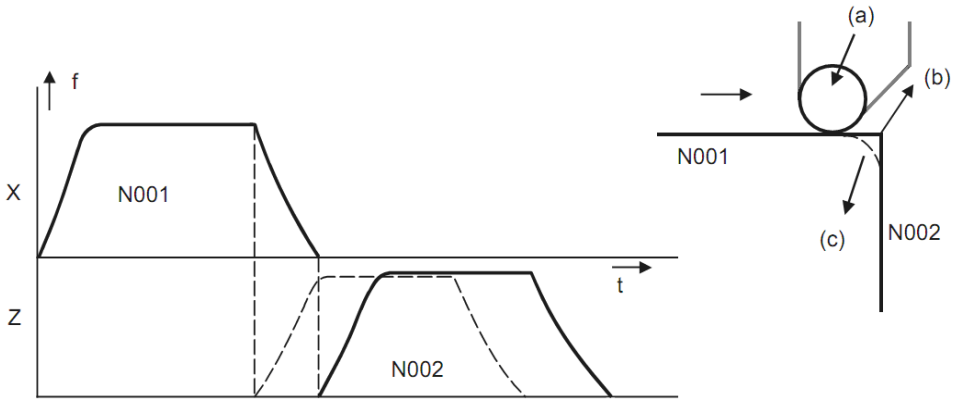
Command format

G09 G01 (G02,G03);	Accurate positioning inspection

Accurate positioning and inspection (G09) are only valid for the cutting command (G01~G03) at the program block where the command is located at.

Program examples

N001 G09 G01 X100.000 F150 ;After confirming the positioning status after deceleration stop, execute the next program block.
N002 Z100.000 ;
Effect of accurate positioning and inspection



(a) Tool (b) With G09 (c) Without G09 f: Command speed t: Time

Solid line: The speed pattern with G09
Dotted line: The speed pattern without G09

6.6 Accurate positioning and inspection mode (G61)

Function and purpose

If the accurate positioning and inspection are carried out through G09, the positioning state shall only be confirmed at such program block, with G61 as a modal action. Therefore, the cutting commands (G01~G03) after G61 shall all be subject to inspection on deceleration and positioning state at various program blocks.

The modality shall be cancelled through the following commands.

G62 ..... automatic corner override
G63 ..... tapping mode
G64 ..... cutting mode

Command format

G61;	Accurate positioning and inspection mode

When G61 is selected, the positioning check is performed. Then, before releasing the check mode, perform the positioning check at the end point of the cutting command program block.

6.6.1 Automatic internal corner override (G62)

Function and purpose

When performing cutting with the tool radius compensation, in order to prevent the increase of cutting load and the inclination of the machined surface, the cutting amount of the inside corner or automatic corner R should not be increased as much as possible within a certain period of time. The override command is automatically issued to the cutting feed speed.

Automatic corner override is valid before specifying the followings: tool radius compensation cancelled (G40); accurate positioning and inspection mode (G61); tapping mode (G63); cutting mode (G64).

Command format

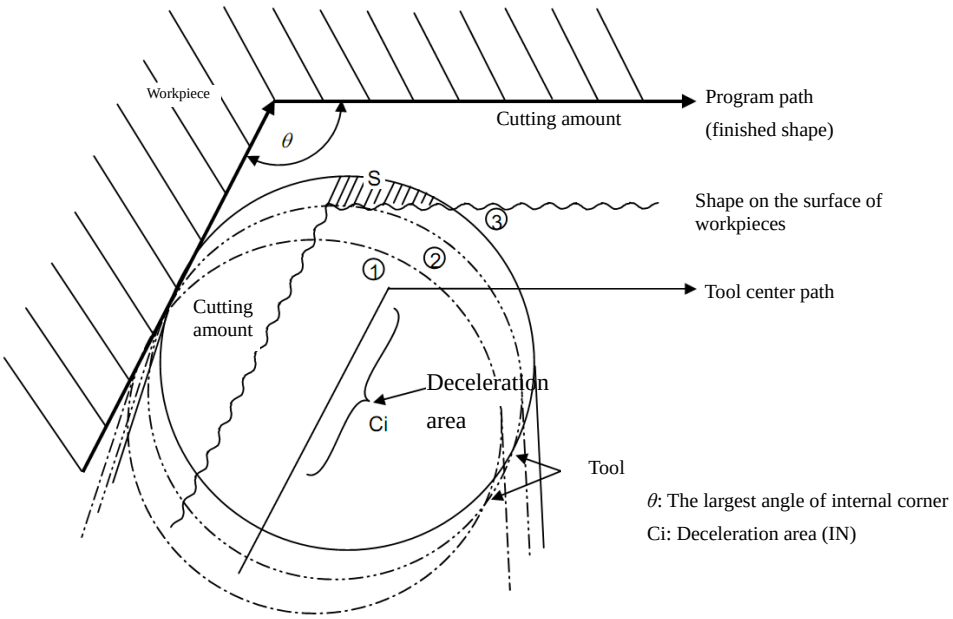
G62;

Detailed description

Internal corner

The load of the tool will be increased if the cutting amount is increased during cutting at the internal corner. Therefore, the override shall be adjusted automatically within the set range of corner, to lower the feed speed and to restrict the load increase, thus realizing favorable cutting.

The function is applicable to programming the shape subjected to finish machining.



(1) Action

(a) When automatic corner override is not made

When the tool moves in the sequence of ①→②→③, the load on the tool will increase, because ③ has more cutting amount than ② concerning the shaded area S.

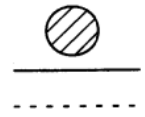
(b) When automatic corner override is adopted.

If the angle θ of the internal corner is less than that set in the parameter, the override set by the parameter

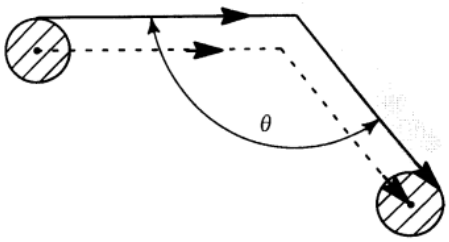
shall be adjusted automatically inside the deceleration area Ci.

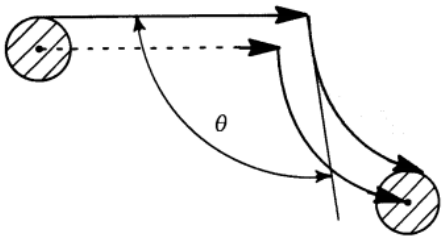
(2) There are four types of internal corners.

1. Straight line-straight line

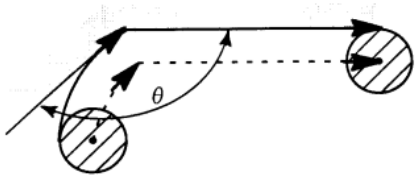


2. Straight line-arc

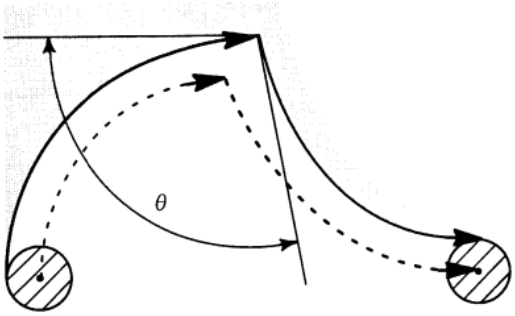




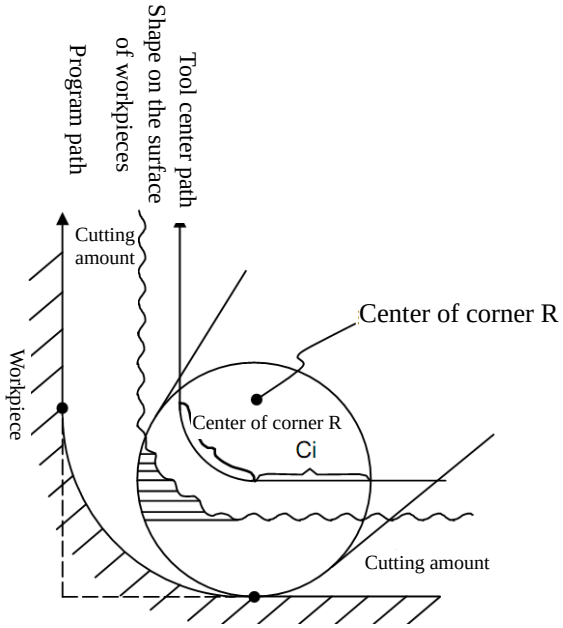
3. Arc-straight line



4. Arc-arc



Automatic corner R



(1) During the internal compensation at the automatic corner R, the override set in the parameter will be adjusted automatically in the deceleration area Ci and at the corner R. (Without inspection on angle)

Precautions

(1) Automatic corner override is only valid in G01/G02/G03 and is invalid in G00. When the corner is

switched from G00 to G01 (G02, G03) (vice versa), such corner will not perform automatic corner override at the program block of G00.

(2) Automatic corner override will not be performed before tool nose R compensation even under the mode of automatic corner override.

(3) For the corner with started/cancelled tool nose R compensation, automatic corner override shall not be performed.

(4) For the corner of I/K vector command with tool nose R compensation, automatic corner override shall not be performed.

(5) Automatic corner override shall not be performed if intersection operations are not carried out.

Such operations shall not be made under the followings:

(a) More than 4 discontinuous program blocks for movement commands

(b) The deceleration area during arc commands is arc length.

(c) The angle of the internal corner is that on the program path in parameter setting.

(d) Automatic corner override will not be made if setting the maximum angle of the parameter to 0 or 180.

(e) Automatic corner override will not be made if setting the override of the parameter to 0 or 100.

6.6.2 Automatic internal arc cutting speed override

Function and purpose

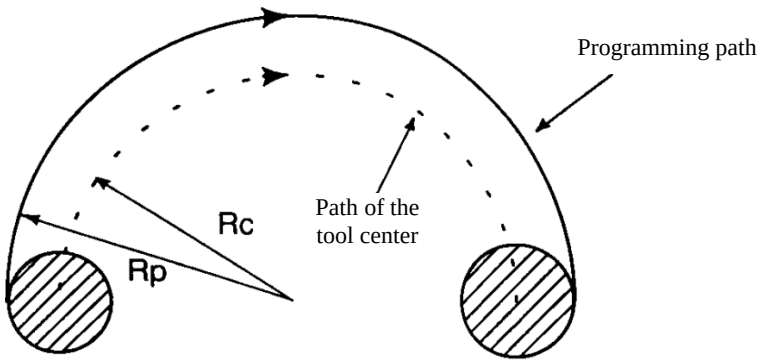
For the arc cutting subjected to internal offset, the feed speed on the programming path shall be determined as per the programmed feed speed F of arc cutting and the arc radius. Such speed is valid in tool radius compensation and is unrelated to G62.

$$F \times \frac{Rc}{Rp}$$

Rc: Radius of the path of tool center

Rp: Programmed radius

Also valid for dry run



Change in the feed speed of internal arc cutting

If Rc is far less than Rp, i.e. Rc/Rp≈0, the tool will be stopped. . When Rc/Rp≤MDR, the feed speed of the tool is (F×MDR)

Precautions

When internal arc cutting and internal corner override are executed simultaneously, the feed speed of the tool is as follows:

$$F \times \frac{Rc}{Rp} \times (\text{inner corner override}) \times (\text{feed speed override})$$

6.7 Tapping mode (G63)

Function and purpose

Enter the following control mode applicable to tapping through G63.

(1) Cutting override (100% fixed)

(2) Continuous deceleration commands among program blocks invalid

(3) Feed hold invalid

(4) Single block invalid

(5) Signal output in tapping mode

Cancel G63 through the accurate positioning and inspection mode (G61), automatic corner override (G62) or cutting mode (G64).

Enter the cutting mode during power-on.

Command format

```
G63;
```

6.8 Cutting mode (G64)

Function and purpose

Enter the cutting mode obtaining a smooth cutting surface through G64. In this mode, opposite to the accurate positioning and inspection mode (G61), the next program block shall be performed continuously rather than stop among program blocks for cutting feed.

Cancel G64 through the accurate positioning and inspection mode (G61), automatic corner override (G62) or tapping mode (G63).

Enter the cutting mode during power-on and resetting.

Command format

```
G64;
```

Chapter VII Auxiliary Functions

7.1 Auxiliary functions

Function and purpose

The auxiliary function is also called as M function and is used to designate the commands of auxiliary functions of the machine such as forward/reverse rotation and stop of the spindle and ON/OFF of the coolant etc.

Detailed description

In the control device, 3 groups can be specified at most within a single program block. The number which can be specified within the same program block depends on parameter #1803.6 M3B.

M00, M01, M02, M30, M96, M97, M98, M99, M198 are nine auxiliary commands that are used for specific purposes and not assigned as ordinary auxiliary commands.

Please refer to the instructions issued by the machine factory for specific values and corresponding functions.

Since pre-reading is prohibited for relevant matters of M00/M01/M02/M30, the next program block will not be pre-read to the buffer memory.

When M function and the movement commands are specified within the same program block, the execution sequence of commands may be classified into two following cases. Whichever is applicable depends on the specifications of the machine.

(1) Execute M function after movement is completed.

(2) Execute movement commands and M function simultaneously.

Except for M98/M99/M198, or the subprogram (set by parameters #6071~#6079) called or the macroprogram (set by parameters #6080~#6089) called, process and finish all M commands through PLC.

Code of M Command Controlled by CNC

S/N	Code	Function	S/N	Code	Function
1	M00	Program dwell	6	M97	Interruption of user macro program invalid
2	M01	Program optional stop	7	M98	Subprogram call
3	M02	End of program	8	M99	Return to main program
4	M30	End of program	9	M198	Call of the external subprogram
5	M96	Interruption of user macro program valid			

Standard Code of M Command Controlled by PLC

S/N	Code	Function	S/N	Code	Function
1	M03	Spindle turning clockwise	20	M36	Stop chip removal
2	M04	Spindle turning counter-clockwise	21	M37	Probe blowing on
3	M05	Stop rotation of spindle	22	M38	Probe blowing off
4	M06	Automatic tool changing	23	M39	Rigid tapping cancelled
5	M07	Blowing of workpieces	24	M41	Spindle 1st gear

S/N	Code	Function	S/N	Code	Function
6	M08	Cutting fluid on	25	M42	Spindle 2nd gear
7	M09	Cutting fluid off	26	M43	Spindle 3rd gear
8	M10	Clamping	27	M44	Spindle 4th gear
9	M11	Releasing	28	M54	Spindle tool releasing
10	M16	Tool exchange	29	M55	Spindle tool clamping
11	M18	Spindle positioning	30	M60	Rotation of tool carrier
12	M19	Spindle orientation	31	M61	Detection on conditions of tool changing
13	M20	5th axis clamping/tailstock forward	32	M65	Tool magazine forward/tool-case fall
14	M21	5th axis releasing/tailstock reverse	33	M66	Tool magazine reverse/tool-case rise
15	M22	Switching of CS axis	34	M76	Automatic tool gang
16	M24	Quit of CS axis	35	M81	Switch of spindle 1
17	M29	Rigid tapping	36	M82	Switch of spindle 2
18	M34	Chip removal reverse	37	M88	Data acquisition started
19	M35	Chip removal forward rotation	38	M89	Data acquisition finished

For the code of M command controlled by PLC, those defined by the machine factory shall prevail finally.

Program stop (M00)

After executing the M00 command, the autorun stops so as to perform some manual operations, such as accuracy detection, etc., press the cycle start button again, and then continue to execute the program after the M00 command. Such command is commonly used for dwell of precision detection between rough machining and finish machining. All modal information shall remain unchanged when the program is stopped.

Program optional stop (M01)

The execution process of M01 is similar to that of M00. The sole difference is that such command is only valid after the "Optional Stop" switch on the control panel of the machine is pressed down, or otherwise the machine will keep executing the follow-up program. Such command is commonly used to check some critical size of the workpiece.

Program end (M02)

After the M02 program end command is executed, it means that all the contents in this machining program have been completed, but after the program ends, the execution cursor on the machine display screen does not return to the program start block.

Program end (M30)

The execution process is similar to M02, but the difference lies in that when the program content ends, all machine actions such as spindle and cutting fluid are turned off immediately, and the execution cursor on the machine display screen returns to the beginning of the program to prepare for machining the next workpiece.

Spindle function (M03/M04/M05)

M03 command is used for clockwise rotation of the spindle. When the spindle is started, rotate as per the

direction in which the right-hand thread enters the workpiece.

M04 command is used for counterclockwise rotation of the spindle. When the spindle is started, rotate as per the direction in which the right-hand thread leaves the workpiece.

M05 command indicates suspended rotation of the spindle.

ON/OFF of cutting fluid (M08/M09)

ON of cutting fluid is expressed in M08 and OFF in M09.

Subprogram call commands (M98/M99/M198)

M98 is a subprogram calling command, and M99 command is used when calling the subprogram and returning to its main program. M198 is the command for calling external subprogram.

Clamping and releasing (M10/M11)

Applicable to slide of the machine, workpieces, clamping and spindle etc.

To guarantee that the program is performed in a correct sequence, some M commands (such as M30/M02/M98) should be programmed at a separate program block.

7.2 2nd auxiliary function

Function and purpose

It is used to designate the position of the indexing worktable etc. In the control device, any number within the range of 0~30000 can be specified through the 5 digits after A/B/C/U/V/W. Corresponding position of each code varies with the specifications of the machine.

Detailed description

When parameter #1803.0 is set to 1, start the 2nd auxiliary function. Only one group of the 2nd auxiliary function can be specified within a single program block.

The address (any of A/B/C/U/V/W) of the 2nd auxiliary function shall be set by parameter #1822. In the turning system using A-series G codes, even though parameter #1822 is set to 85~87, B shall still be used as the address of the 2nd auxiliary function.

Command address	A	B	C	U	V	W
Set value	65	66	67	85	86	87

When the command for the 2nd auxiliary function and the movement command are specified within the same program block, the execution sequence of the command may be classified into the following two types. Which type is applicable depends on the specifications of the machine.

(1) Execute the 2nd auxiliary function after the movement command is completed.

(2) Execute the movement command and the 2nd auxiliary function simultaneously.

Process and end all the 2nd auxiliary functions through PLC.

The same address shall not be used for the name of the additional axis and the 2nd auxiliary function.

Precautions

For the command with decimal points, round such figure for output. The figures behind the decimal points shall be ignored.

If the 2nd auxiliary function is commanded through a negative value, such command shall be ignored.

7.3 Indexing of rotary worktable

Function and purpose

Indexing of rotary worktable can be performed by setting the indexing axis.

The indexing command is only designed for the indexing of the rotary worktable, and the indexing angle of the shaft command shall be set. Clamping for the worktable is unnecessary to be designated and the special M code of such clamping shall be cancelled, to facilitate programming.

Detailed description

Indexing of rotary worktable can realize the following actions.

Example of command format

(Example) G00 B90;

└The axis specified by the indexing axis of the rotary table (parameter #1932)

(1) Set the "Selection of Indexing Axis" parameter (#1932) performing the indexing axis of the rotary worktable as "0" or "4".

(2) Perform the movement command (absolute values/increments) of the axis selected as per the program commands.

(3) Cancel the clamping action before movement of the axis.

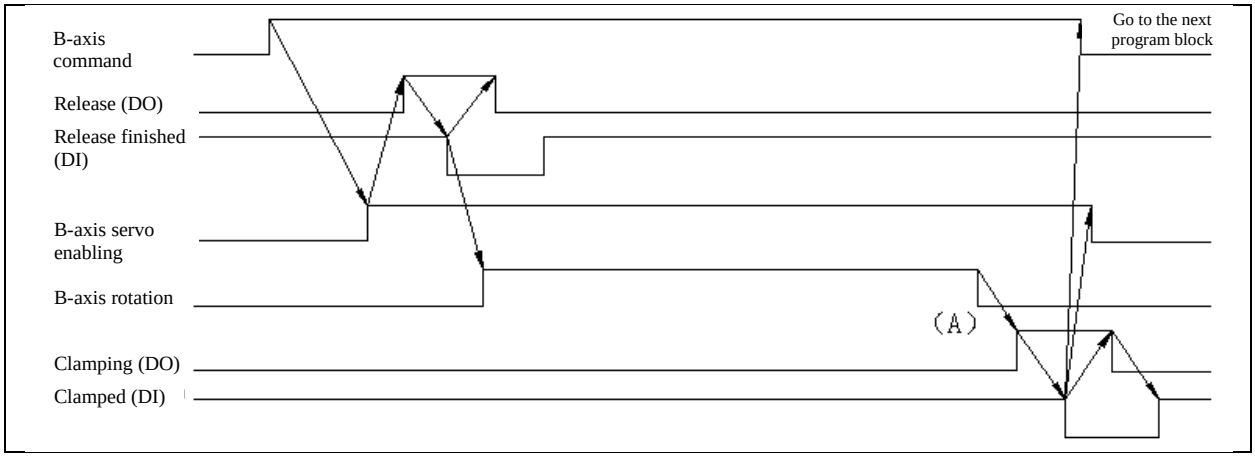
(4) Start movement of the specified axis after cancellation of clamping.

(5) Perform the clamping action after movement.

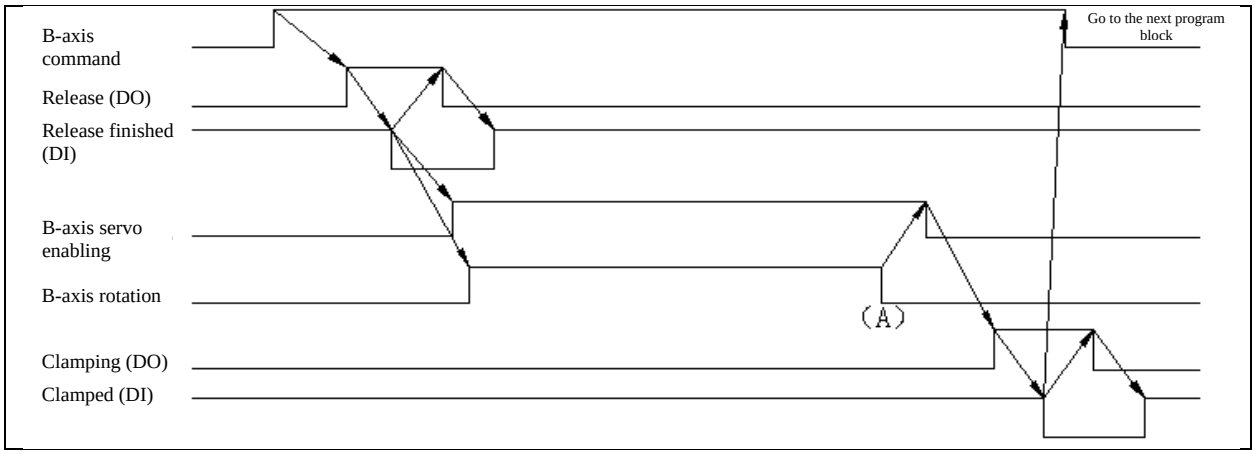
(6) Perform the action of the next section after cancellation of clamping.

Indexing sequence

Type A (#1030.6=0)



Type B (#1030.6=1)



Precautions

(1) Activate the indexing of rotary worktable and set parameter #1030.7 ITI as 1.
(2) The minimum angle of indexing shall be set through parameter #1931.
(3) Nearby movement can be set for rotation of the indexing axis of the rotary worktable and set parameter #1023.3 RHAY as 0.
(4) The movement speed of the indexing axis of the rotary worktable is unrelated to the feed speed (rapid movement speed generally) of this mode (G0/G1).
(5) If both the indexing axis and other shafts of the indexing axis of the rotary worktable are designated in the same program block, the command for cancelling clamping of such shaft is performed. Therefore, movement of other shafts in the same program block shall not be performed before such cancellation.
(6) The indexing axis of the rotary worktable is generally used for the rotation axis, and the action cancelling clamping can also be executed through the function even on the linear axis.
(7) In automatic operation, during movement of the indexing axis of the rotary worktable, if the command for cancelling clamping is OFF, stop the indexing axis of such table under the state of clamping cancelled. Moreover, when the indexing axis is the same as other axes specified in the program block, the system gives an alarm of "illegal axis".
(8) During movement of the indexing axis of the rotary worktable, if movement of the axis is interrupted due to interlocking etc., the state of clamping cancelled shall be maintained.
(9) In case of continuous commands for movement of the indexing axis of the rotary worktable, the action cancelling clamping shall not be performed. However, during synchronous operation of the program block, the action cancelling clamping shall also be performed even if continuous commands for movement are given.
(10) Please note that the position where clamping is impossible shall not be specified.
(11) The indexing axis command is not an integral multiple of the minimum indexing angle. When the indexing axis command is executed, a program error alarm "PS0135 illegal angle command" will be generated.

Chapter VIII Conversion of Programming Coordinates

8.1 Scaling (G50/G51)

Function and purpose

It is only applicable to the milling system.

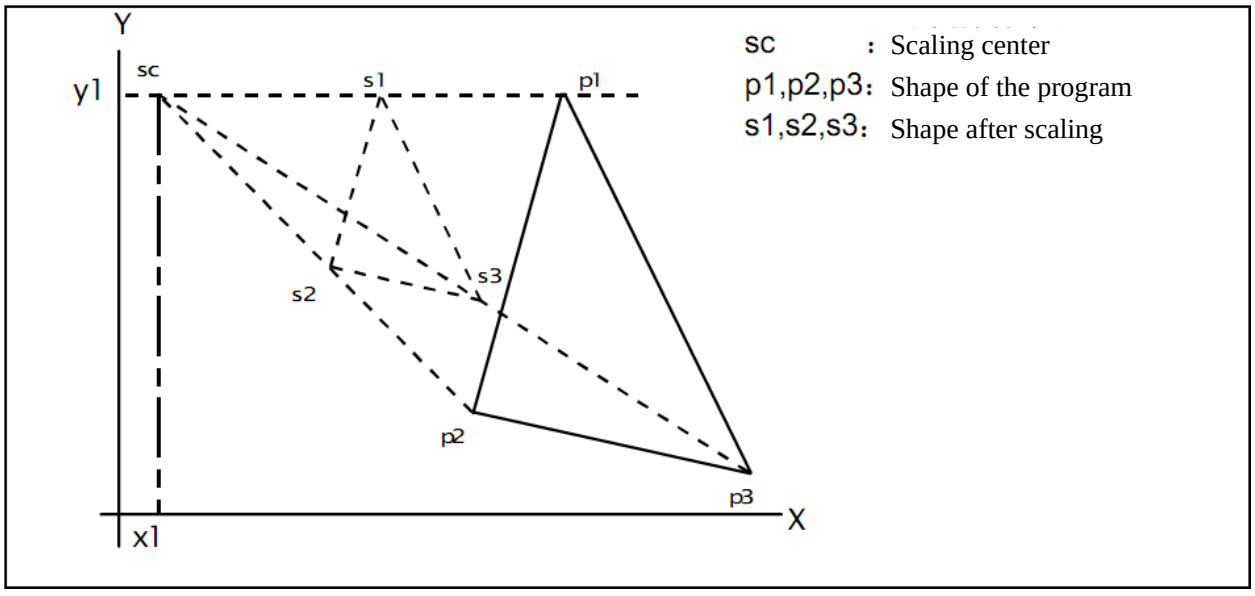
For the movement command of the axis within the scope specified by such command, the shape specified in the program can be magnified or narrowed to the size required by multiplying by the override.

Command format

G51 X_ Y_ Z_ P_;	Scaling is on, all axes are scaled by the same ratio
G51 X_ Y_ Z_ I_ J_ K_;	Scaling is on, each axis is scaled by a separately specified ratio
G50	Scaling cancelled

X_ Y_ Z_	Coordinates in the scaling center
P_	Scaling factor
I_ J_ K_	Corresponding scaling factor of X/Y/Z-axis

Specification must be made within a separate program block.



Detailed description

(1) Setting of scaling parameters

Whether scaling is valid shall be determined through parameter #1850.3.

Whether the scaling factor of each axis is valid shall be determined through parameter #1850.6.

(2) Specification of the scaling shaft and center and its factor

It will go to the scaling mode after G51 is issued. G51 is only used to specify the scaling shaft and center

and its factor, without any movement.

(a) Scaling center

- X/Y/Z is used to specify the scaling center. The current point of the tool shall serve as the scaling center if X/Y/Z is ignored.
- The scaling center is always specified in the mode of absolute values (G90) even under the mode of increments (G91).
- Under the mode of the polar coordinate (G16), the coordinate command in the program block of G51 is expressed with the right angle system.

(b) Scaling factor

- The same scaling factor is used for each axis and is specified through P.
- Different scaling factors are used for each axis, and are specified through corresponding I/J/K. Mirroring can be applied by designating a negative rate. In this case, the object axis of mirroring becomes the same position as the scaling center.
- Scope of command:
  P:0～999999.9999
  I,J,K:±999999.9999
- If the rate is not specified within the same program block with G51, the rate set through parameter (scaling factor of #1861 or #1862) will be used.
  When parameter #1850.6=0, the rate set by parameter #1861 shall be used.
  When parameter #1850.6=1, the rate set by parameter #1862 shall be used.
- Change of such parameter will be invalid in the mode of scaling. Scaling shall be made as per the set value when G51 is specified.
- Scaling cannot be used for offset, such as tool radius compensation, tool length compensation and tool position offset.

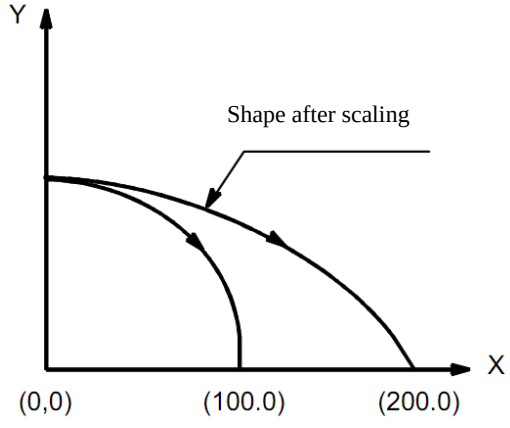
(3) Scaling cancelled

- Scaling will be cancelled after G50 is specified.

(4) Scaling of circular interpolation

- For circular interpolation, even though different scaling ratios are used for each axis, the path of interpolation is not an ellipse.
- Circular interpolation is programmed with radius R. See the figure below for the interpolation graph when different scaling rates are used for each axis.

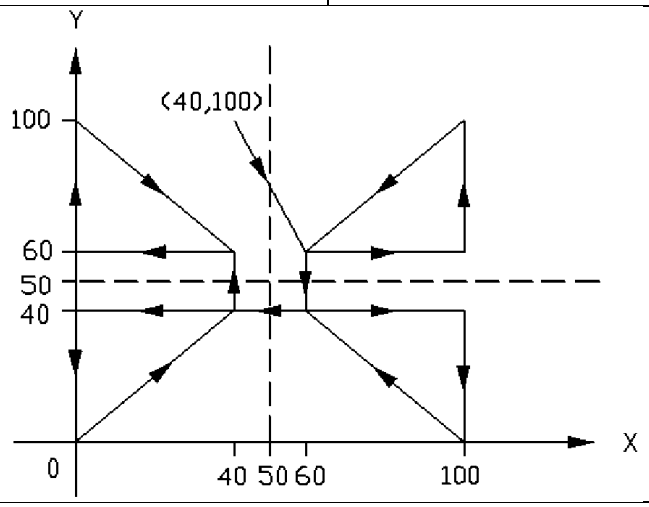
G90 G00 X0.0 Y100.0 ;
G51 X0.0 Y0.0 Z0.0 I2 J1;(2 times scaled up in the X-direction, 1 time scaled up in the Y-direction)
G02 X100.0 Y0.0 R100.0 F500 ;
The commands above are equivalent to those below.
G90 G00 X0.0 Y100.0 Z0.0 ;
G02 X200.0 Y0.0 R200.0 F500 ;
The rate of radius R depends on I or J, whichever is larger.



Program examples

Mirroring program

Main program	Subprogram
G00 G90 G50 ; G92 X40 Y100 ; M98 P9000 ; G51 X50.0 Y50.0 I1 J - 1 ; M98 P9000 ; G51 X50.0 Y50.0 I - 1 J - 1 ; M98 P9000 ; G51 X50.0 Y50.0 I - 1 J1 ; M98 P9000 ; G50 ;	O9000 G00 G90 X60.0 Y60.0 ; G01 X100.0 F100 ; G01 Y100 ; G01 X60.0 Y60.0 ; M99 ;



Precautions

(1) For tool radius compensation, tool position offset and tool length compensation etc., scaling shall not be performed. (Calculation of compensation will be made for the shape after scaling.)

(2) Any two of G51, G51.1 and G68 cannot be on the same block, otherwise an alarm will be given.

(3) The position is displayed as the coordinate after scaling.

(4) It will go to the "Cancellation" mode if M02/M30 commands are issued in the mode of scaling, or NC resetting is performed.

(5) If offset (G92/G52 commands) is made for the coordinates under the state of scaling, the scaling center will offset as per the difference value.

(6) Scaling is invalid for manual operation, but valid for DNC, Automatic or MDI operations.

(7) Scaling is invalid for the following movement in the fixed cycle.

- Z and R positioning points in the fixed cycle.
- Cutting depth Q and retraction amount of deep hole drilling cycle (G83/G73).
- Offset Q of the X-axis and the Y-axis under finish boring (G76) and back boring (G87).

8.2 Program coordinate rotation (G68/G69)

Function and purpose

It is only applicable to the milling system.

If there is a complex shape in the position after the coordinate system is rotated, when machining it, you can program the shape before the rotation in the local coordinate system, and use the program coordinate rotation command to specify the rotation angle and process the rotated shape.

Command format

G68 X_ Y_ R_;	Coordinate rotation turning on
G69	Coordinate rotation cancelled

X_ Y_ Z_	Coordinates in the rotation center. Specify the 2nd axis corresponding to the plane selected in X/Y/Z with absolute position.
R_	Rotation angle. The counterclockwise direction is +. As for setting of parameter #1850.0, the absolute values can be adopted for designation at any moment, or such designation can be made as per the mode of G90/G91.

Detailed description

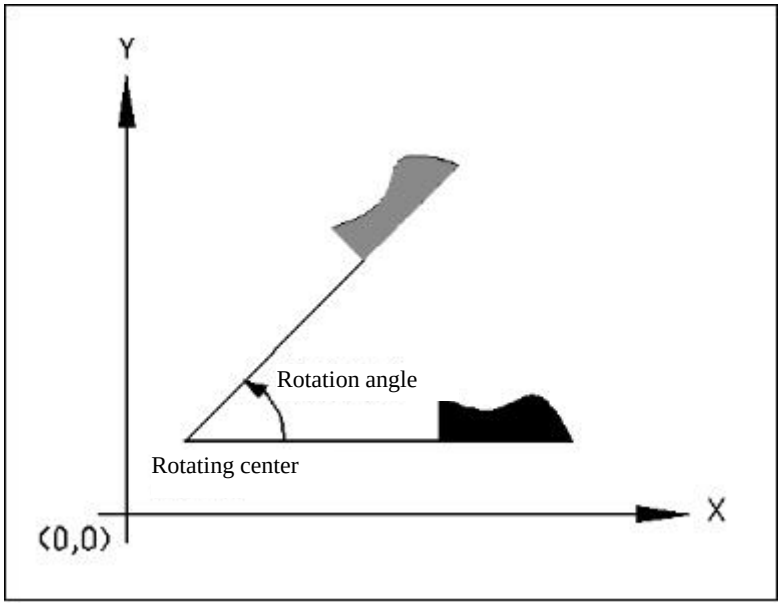
The programmed shape can be rotated. When the workpiece is composed of many graphics of the same shape, the coordinate rotation function can be used for programming. It is only necessary to program the subprogram of the graphics unit, and then call the subprogram through the rotation function.

Command format: G17 G68 X_ Y_ R_

Or G18 G68 X_ Z_ R_

Or 19 G68 Y_ Z_R_

G69



Function: G68 can rotate the shape programmed on the plane with the specified center as the point of origin; G69 can be used to cancel the rotation of the coordinate system.

Meanings of commands:

G17 (G18 or G19): Plane selection; the rotation shape is comprised on the plane.

X_Y_Z_: Coordinates of two axes in X_, Y_or Z_ corresponding to the command plane (G17/G18/G19). The rotation center shall be specified behind G68.

R_ : Rotation angle; the positive value indicates counterclockwise rotation. For parameter No:1850#0, absolute values are specified for the rotation angle, and the absolute values or increments shall be determined as per the specified G code (G90 or G91).

Minimum input unit of increments: 0.0001deg

Valid range of data: -360.0000 ~360.0000

Description:

1. It is recommended to designate G17/G18/G19 for plane selection in front of the program block of G68.
2. If G68 is specified below G91, the tool position when G68 is specified will be deemed as the rotation center of the system, and the command specified is invalid.
3. The tool position when G68 is specified will become the rotation center if the rotation center is not specified.
4. If the rotation angle R is ignored, the value set in parameter No1860 will be taken as the rotation angle of the system.
5. Perform tool radius compensation, tool length compensation and other compensations after the rotation of the coordinate system.
6. If the coordinate system rotation command is executed in the scaling mode (G51), the coordinate value of the rotation center is also scaled, but the rotation angle is not scaled. When a movement command is issued, the scaling is performed first, and then the coordinates are rotated.
7. Any two of G51, G51.1 and G68 cannot be on the same block, otherwise an alarm will be given.

Under the mode of coordinate rotation, the command position shall be calculated first as per the coordinate system before rotation, to make the coordinate rotate.

Example:N01 G92 G90 G0 X0 Y0 ;

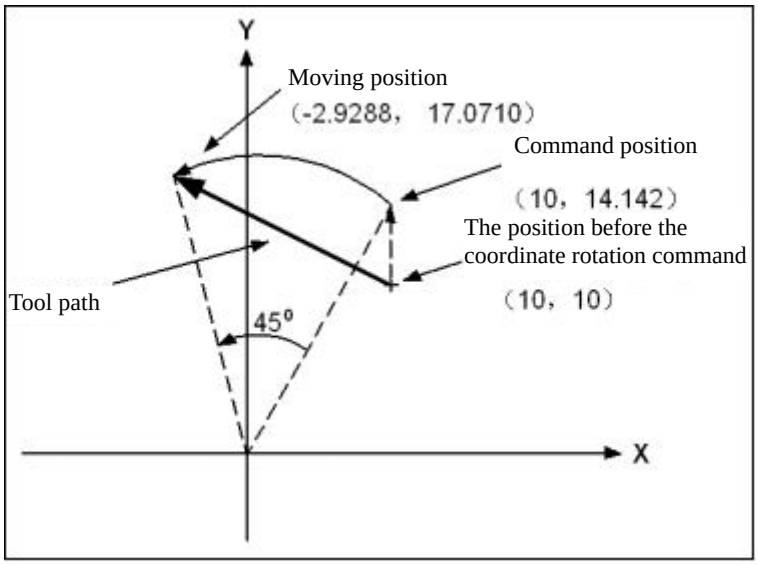
N02 G01 X10 Y10 F6000 ;

N03 G68 X0 Y0 R45. ;

N04 Y14.142 ;

N05 G69 ;

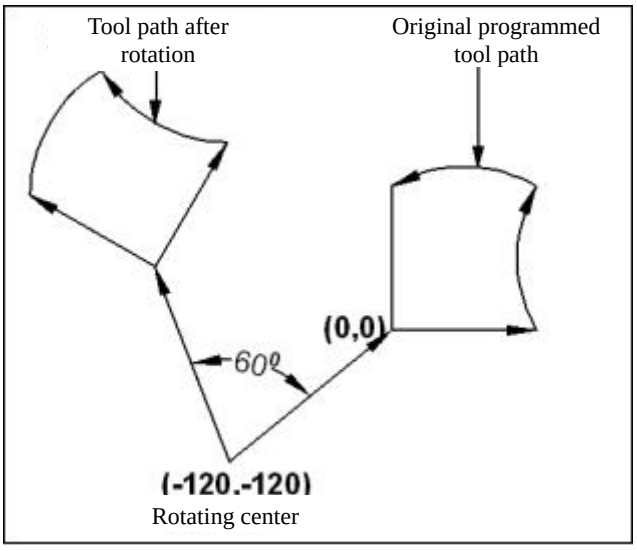
The command position shall be calculated as per the coordinate system (X/Y) before rotation, to make the coordinate rotate. Therefore, the position of X which has not been designated in N04 is X10 and the command position becomes (X10, Y14.142). Afterwards, move it to a position (X-2.9288, Y17.0710) realizing 45 °rotation.



Program examples

Example 1: Rotation

```
G92 X-120 Y-120 G69 G17;
G68 X-120Y-120 R60;
G90 G01 X0 Y0 F200;
G91 X100;
G02 Y100 R100;
G3 X-100 I-50 J-86.603;
G01 Y-100;
G69 ;
```



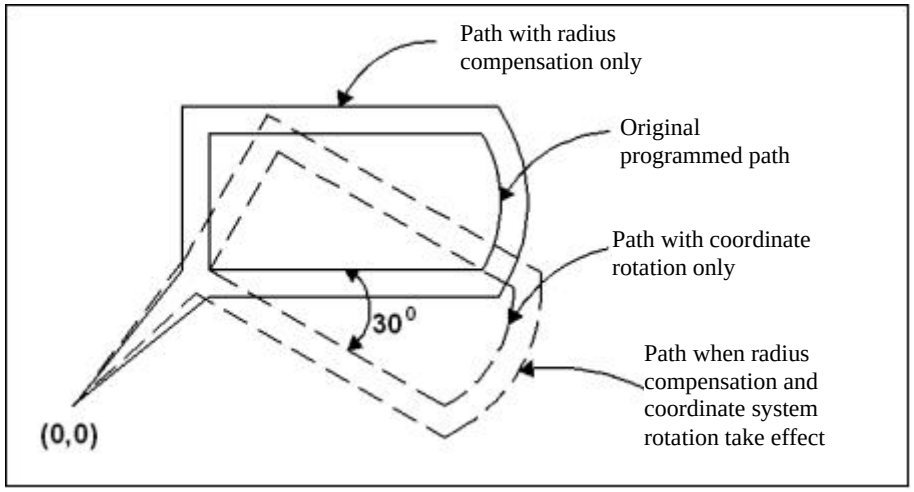
Example 2: Tool radius compensation and coordinate rotation

G68 and G69 can be specified under the mode of tool compensation; the rotation plane must be in line with the plane of tool radius compensation.

Example:

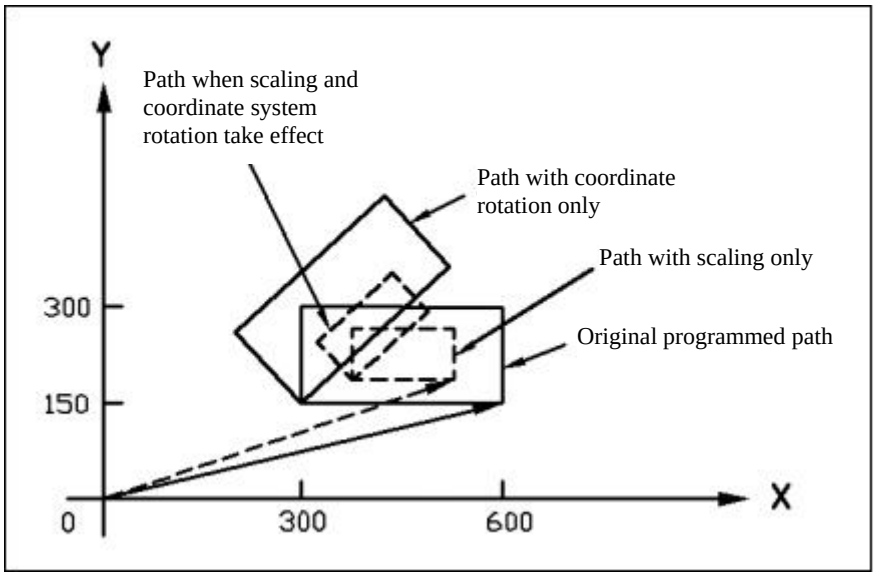
```
G92 X0 Y0 G69 G01;
```

G42 G90 X10 Y10 F1000 D1;
G68 R-30;
G91 X30;
G03 Y10 R15;
G01 X-50;
Y-15;
G69 G40 G90 X0 Y0;
M30;



Example 3: Scaling and rotation

G92 G90 X0 Y0
G51 X450 Y200P0.5;
G68 X300Y150R45;
G01 G90 X600 Y150
G91 Y150
X-300;
Y-150
X300;
G69 G50;



Example 4: Repetitive use of G68

One program can be saved as a subprogram, which can be called repetitively by changing the angle.

Base program (main program)

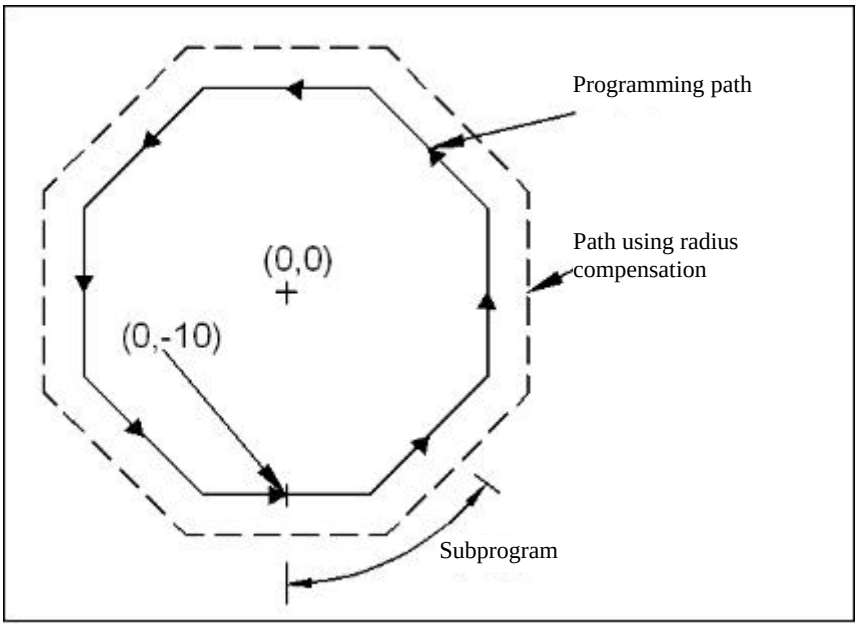
```
G92 X0 Y0 Z0 G69 G17 ;
G01 F200;
M98 P3155 ;                    (subprogram call)
M98 P073166 ;          (call 7 times)
G00 G90 X0 Y0 Z0 ;
M30 ;
```

Subprogram 3166

```
O3166 ;
G68 X0 Y0 G91 R45.0 ;          (relative rotation angle)
G90 ;
M98 P3155 ;                    (subprogram O3166 calls subprogram O3155)
M99 ;
```

Subprogram 3155

```
O3155;
G90 G01 G42 X0 Y-10 D1 ; (tool compensation on the right established)
X4.142 ;
X7.071 Y-7.071 ;
M99 ;
```



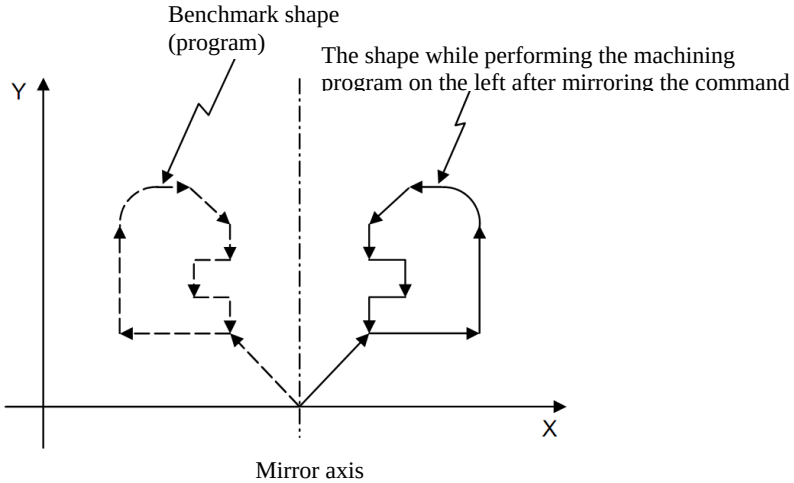
8.3 Programmable mirror (G50.1/G51.1)

Function and purpose

It is only applicable to the milling system.

During cutting of a symmetrical shape, the shape on the opposite side can be machined only by programming the left or right side, thus saving the time required for programming. In this case, the most effective function is mirroring.

For instance, as shown in the figure below, in case of the program for machining of the shape on the left, the shape symmetrical to the left can be completed on the right by performing mirroring for such program.



Command format

G51.1 Xx1 Yy1 Zz1;	Mirroring turning on
G50.1 Xx2 Yy2 Zz2 ;	Mirroring cancelled

x1, y1, z1	Coordinate in the mirroring center
x2, y2, z2	Mirroring axis closed (ignore the values of x2, y2, z2)

Detailed description

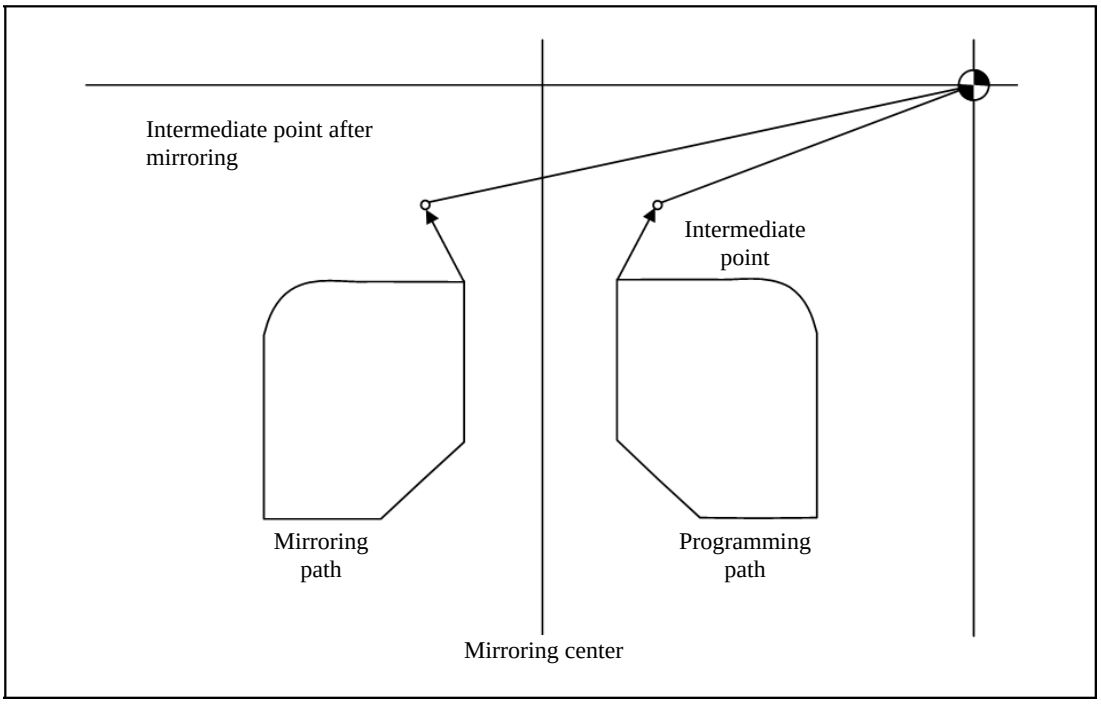
(1) For the coordinate statement and coordinates in G51.1, the axis for mirroring command and the coordinates in the mirroring center are designated through absolute values or incremental values.

(2) The coordinate statements in G50.1 indicate the axis with invalid mirroring, and the coordinate values are ignored.

(3) If the mirroring is only specified on Axis 1 on the specified plane, both rotation and compensation will be made in the opposite direction in arc and tool radius compensation, and coordinate rotation etc.

(4) Since the function is used on the local coordinate system, the mirroring center will move due to presetting of the counter and change in the workpiece coordinate.

(5) When the homing command (G28/G30) is executed in mirroring, the mirroring is valid in the action before arriving at the intermediate point, but invalid in the action from the intermediate point to the reference point.



(6) Return from the point of origin in mirroring

If the command (G29) returning from the point of origin is executed in mirroring, it is valid for the mirroring at the intermediate point.

(7) The mirroring of G53 command is invalid.

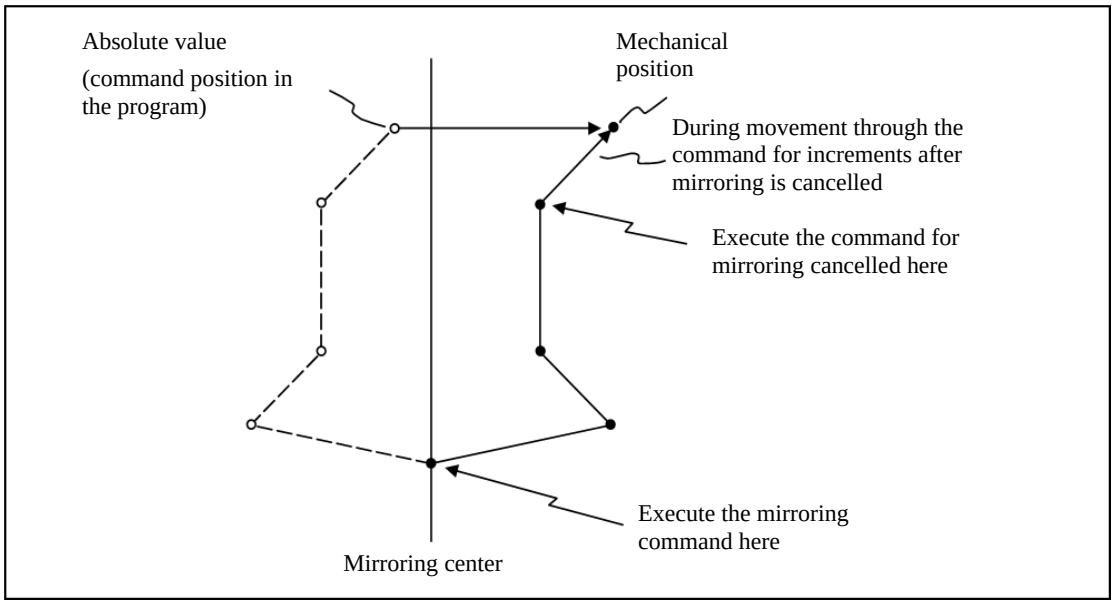
Precautions

Please turn ON/OFF the mirroring in the mirroring center.

If the mirroring is cancelled beyond the mirroring center, it will enter the state of "mechanical position deviating from the absolute value" as shown in the figure below. (Such state will last before executing the command for absolute values [Position under the G90 mode] in the program, or before homing through G28/G30). Since the mirroring center is set through absolute values, if it is re-designated under such a state, the center may

be set at an unforeseen position.

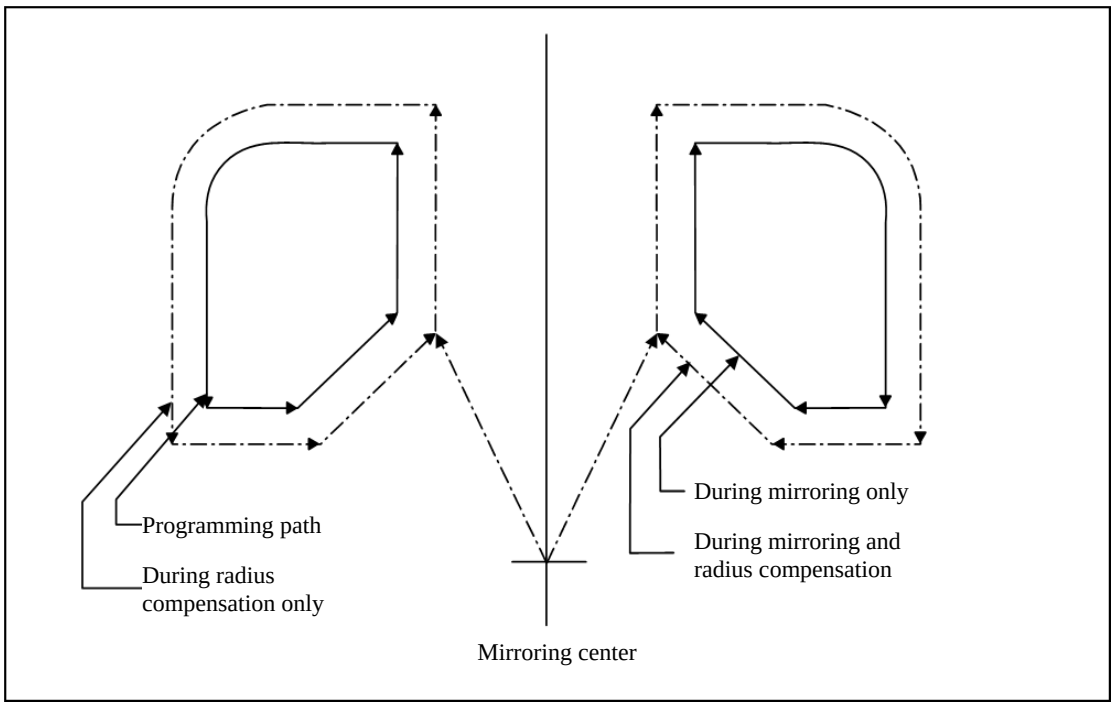
Please cancel the mirroring in the mirroring center, or conduct position through the command for absolute values after such cancellation.



Combination with other functions

(1) Combination with radius compensation

The following cutting shall be made if the mirroring (G51.1) is carried out after radius compensation (G41/G42).



8.4 Center point control of 5th axis machining tool (G43.4)

Function and purpose

The control point shall be controlled by CNC during interpolation for the rotation axis, so as to realize linear movement at the center point of the tool relative to the worktable (workpieces). The end point at the center point of the tool is the coordinates in the programming coordinate system.

Command format

G43.4 X_ Y_ Z_ α_ β_ H_;Tool center point control turned on G49; Tool center point control turned off

α_ β_	Space angle. Any two of the A/B/C rotation axes
H_	Offset number of the tool

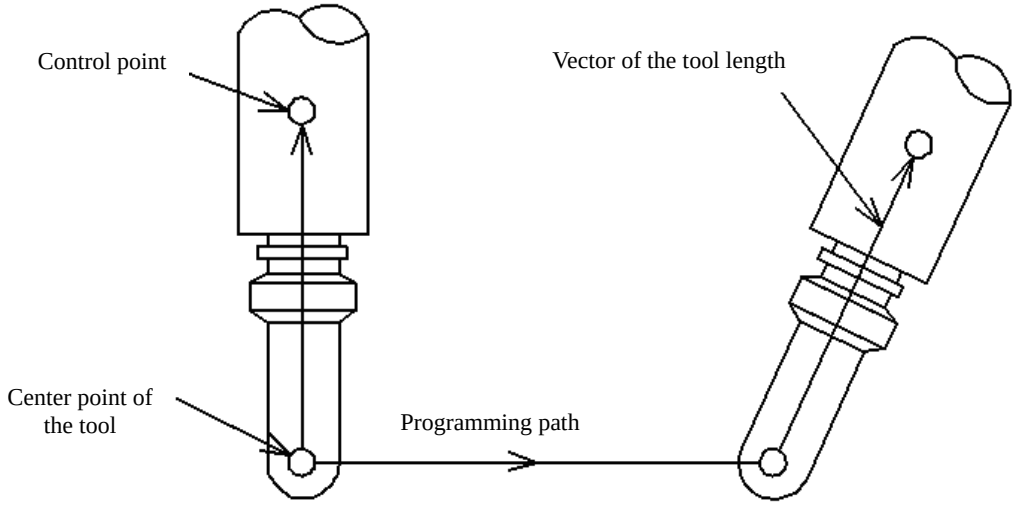
Detailed description

About this function, in the 5-axis machine comprising two rotation axes (the tool and the worktable) and three orthogonal axes (X, Y, Z), the tool length compensation is made while changing the gesture of the tool. For the tool relative to the workpiece, even though the tool axis direction is changed, the center point of the tool will still move among the specified path.

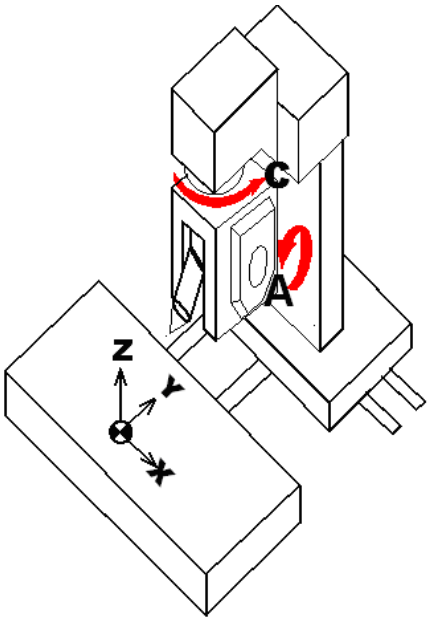
In the center point control function of the tool, the coordinate system used is called the programming coordinate system. The coordinate system fixed onto the worktable is used as the programming coordinate system, thus simplifying programming of CAM.

The 5-axis machine can be classified into the following three types: ① spindle-tilting; ② table-tilting; ③ table/spindle-tilting.

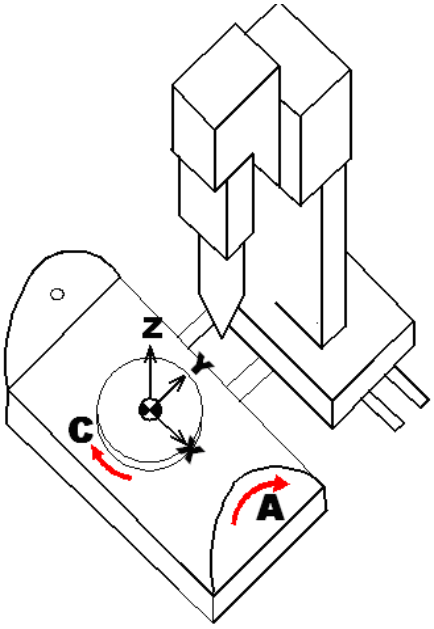
The function is used for the 5-axis machine comprising three orthogonal axes (X/Y/Z) and the tool or the worktable.



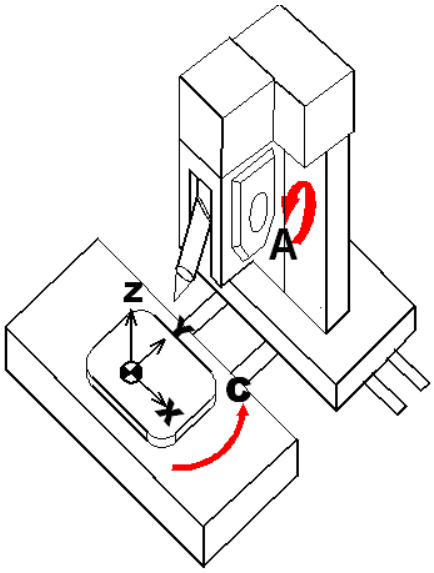
(1) Spindle-tilting machine tool



(2) Table-tilting machine tool

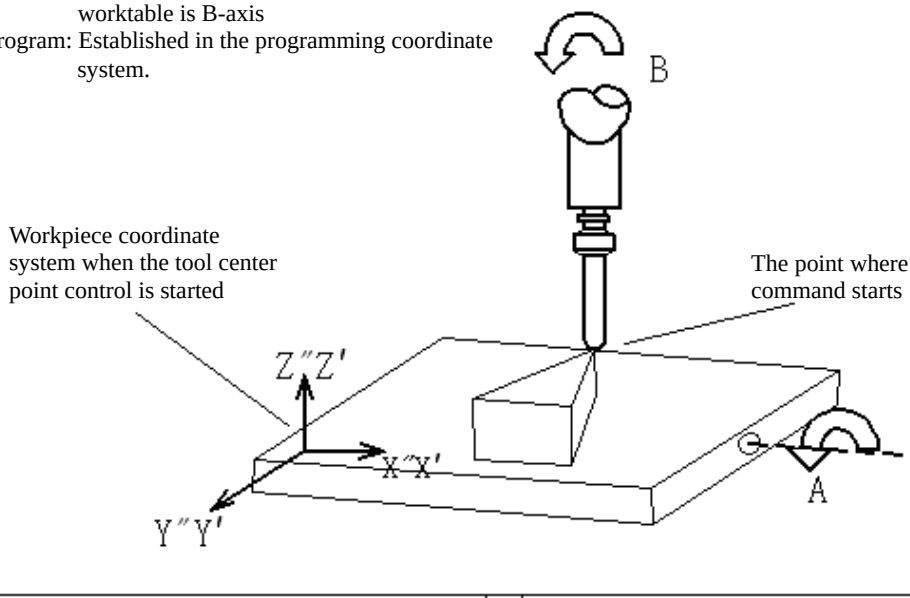


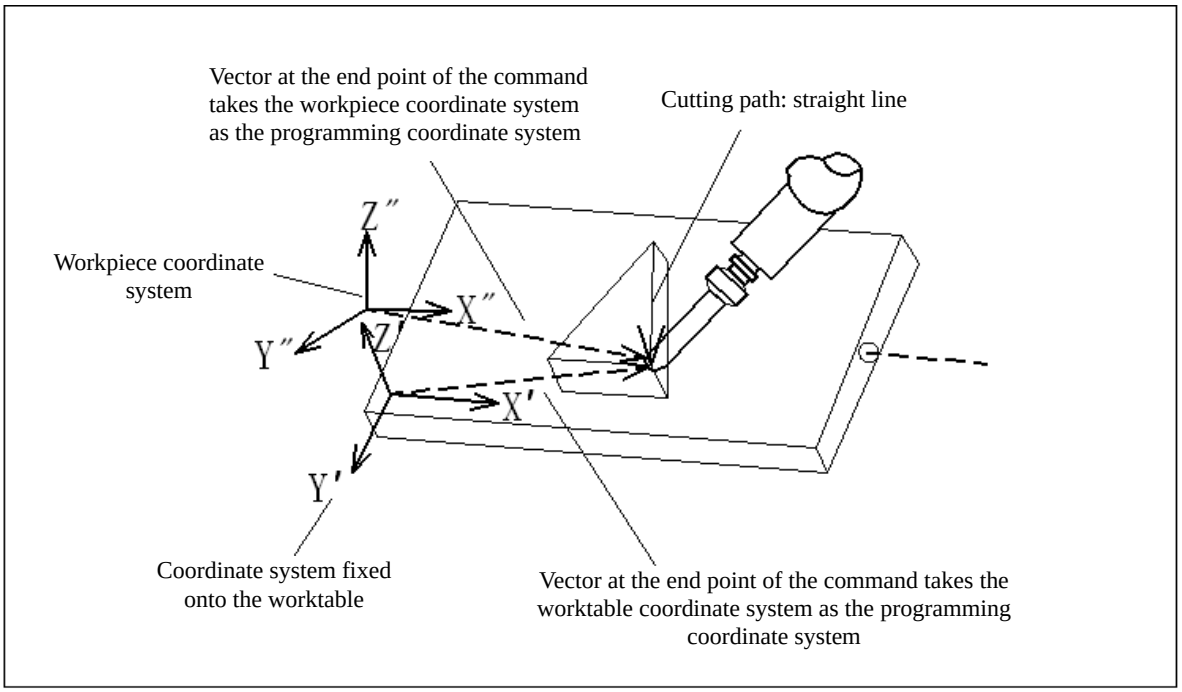
(3) Table/spindle-tilting machine tool



When the coordinate system fixed on the worktable is used as the programming coordinate system, the position and direction of the workpiece fixed on it are changing due to the rotation of the worktable, but the programming coordinate system is static relative to the worktable, so the work can be ignored. The program is run by the rotation of the table, and when a straight line is commanded, the path of the tool center point is in a straight line with the workpiece according to the command.

Example:
Composition of the machine:
The axis controlling rotation of the tool is A-axis
The axis controlling rotation of the worktable is B-axis
Program: Established in the programming coordinate system.



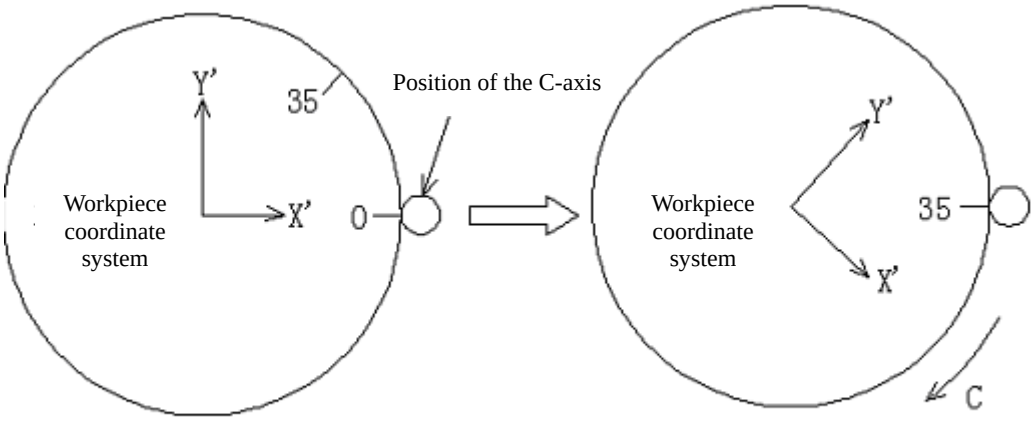


(1) Programming coordinate system

The programming coordinate system is the coordinate system used for tool center point control. The workpiece coordinate system fixed onto the worktable is used as the programming coordinate system by designating G43.4. When the coordinate of the rotation axis is zero, the programming coordinate system, and the workpiece coordinate system will be overlapped. Afterwards, the programming coordinate system will rotate with the rotation of the worktable, but will not rotate with the tool. The X/Y/Z commands after G43.4 will be regarded as the coordinate values in the programming coordinate system.

When G43.4 command is specified, the benchmark state of the programming coordinate system of the rotation axis of such worktable is set through offset (G54-G59).

In the descriptions below, the coordinate system fixed onto the worktable is expressed with X' Y' Z'.



G43. 4 command, C=0 degree tool center point control

(2) Display of the current position in the tool center point control

The position of the control point in the workpiece coordinate system will be displayed in the machine coordinates in the tool center point control.

(3) Commands for the tool center point control

In the tool center point control, the position at the end point of each program block seen at the programming coordinate system shall be specified.

For the rotation axis, the coordinate at the end point of each program block shall be specified.

Moreover, for the feed speed, the tangential speed (relative speed of the workpiece and the tool) relative to the workpiece shall be specified by F.

(4) Commands specified in the tool center point control

The commands which can be specified in the tool center point control include linear interpolation (G01) and positioning (G00).

When the linear interpolation (G01) is specified in the tool center point control, the speed at the center point of the tool shall be controlled and transformed to be the specified speed.

(5) Resetting the tool center point control

The mode will be cancelled (as if G49 is performed) if resetting under the mode of G43.4.

(6) Mode switching

When the "Tool Center Point Control" is enabled, the mode switching will cancel the modal (as if G49 was executed). If the "Tool Center Point" control is enabled in the automatic and MDI modes, and the operation mode is switched after entering the G43.4 mode, it means that the "Tool Center Point" control is cancelled. Then, if you switch back to automatic and MDI, it is necessary to re-enter the G43.4 modal before operation, otherwise there will be processing errors and dangers.

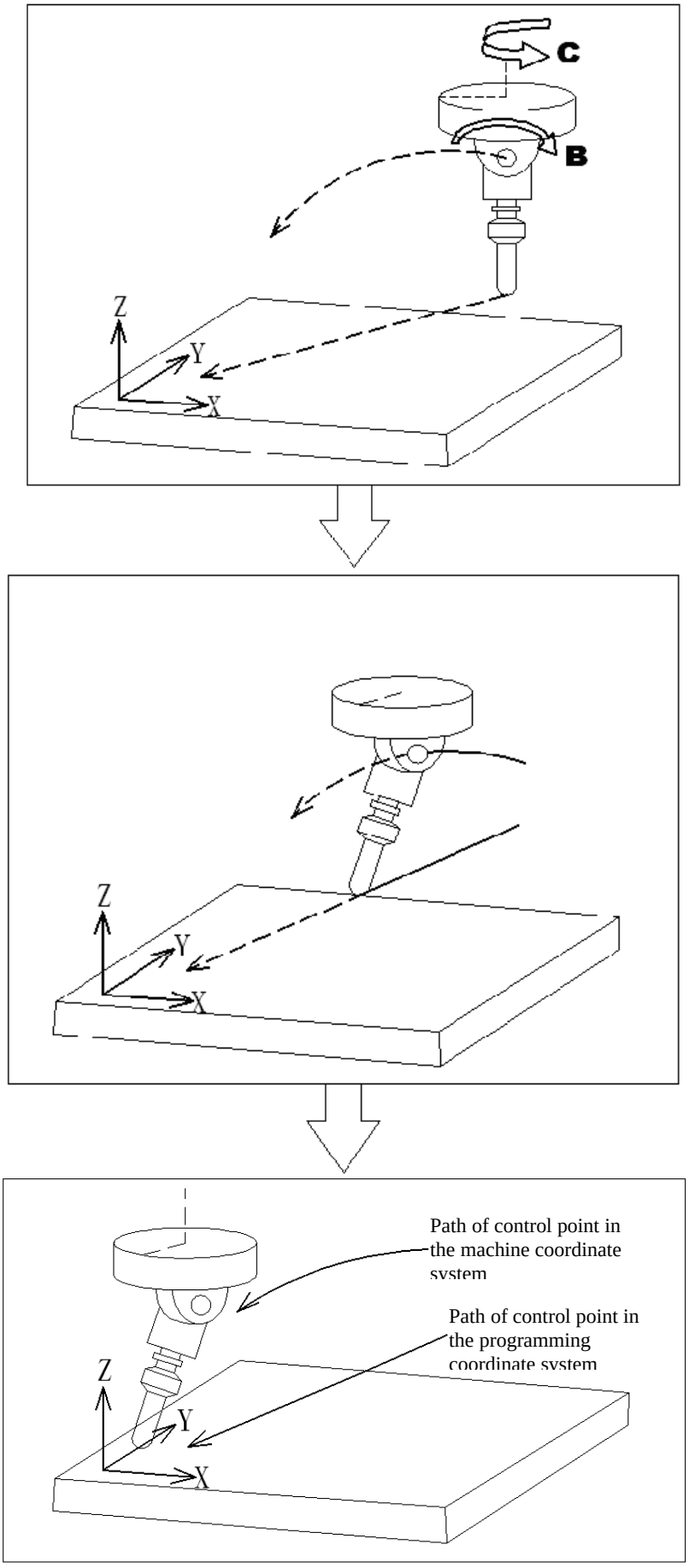
Program examples

(1) Use of spindle-tilting machine tool

If the linear interpolation of X/Y/Z is specified by taking the workpiece coordinate system as the programming coordinate system, the center point of the tool shall linearly move towards the worktable (workpiece) through control during the tool rotation. The center point of the tool shall reach the specified speed relative to the worktable (workpiece) through the speed control.

For this machine type, when the tool rotation axis rotates, the worktable does not rotate relative to the workpiece coordinate system, so the programming coordinate system always coincides with the workpiece coordinate system.

```
N1 G00 G90 B0 C0 ;
N2 G54 ;                                  Prepare the programming coordinate system
N3  G43.4  H01  ;                         Start the tool center point control H01 is the tool
                                          compensation number
N4 G00 X200.0 Y150.0 Z20.0 ;              Move towards the start point
N5 G01 X5.0 Y5.0 Z5.0 C60.0 B45.0 F500 ;  Linear interpolation
N6 G49;                                   Cancel the tool center point control
N7 M30;
```


C
B
Z
Y
X
Z
Y
X
Path of control point in the machine coordinate system
Path of control point in the programming coordinate system
Z
Y
X

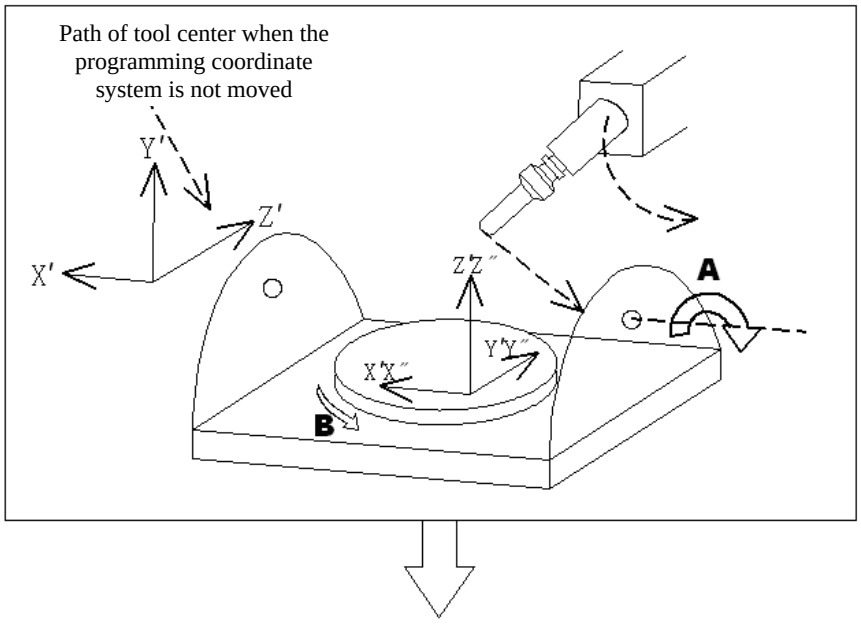
(2) Use of table-tilting machine tool

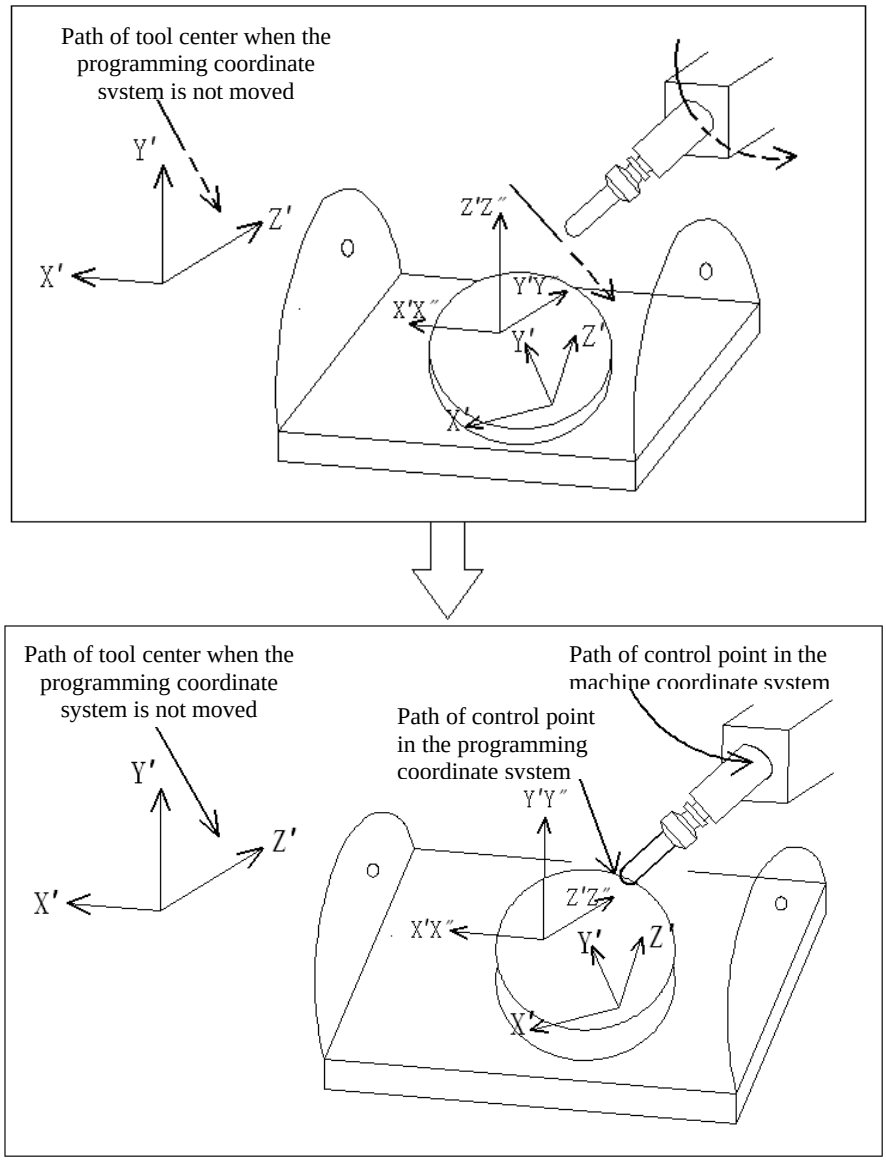
If the linear interpolation of X/Y/Z is specified in the programming coordinate system, the center point of the tool linearly moves towards the worktable (workpiece) through control during rotation of the rotary table.

Meanwhile, the center point of the tool shall move as per the specified speed towards the worktable (workpiece) through the speed control.

In this type of machine, the rotation of any rotation axis will cause the table to rotate. Then, the workpiece coordinate system will not change, but the programming coordinate system fixed on the worktable will rotate with it.

N1 G00 G90 A0 B0 ;	
N2 G54 ;	Prepare the programming coordinate system
N3 G43.4 H01 ;	Start the tool center point control H01 is the tool compensation number
N4 G00 X20.0 Y100.0 Z0 ;	Move towards the start point
N5 G01 X10.0 Y20.0 Z30.0 A60.0 B45.0 F500 ;	Linear interpolation
N6 G49;	Cancel the tool center point control
N7 M30;	



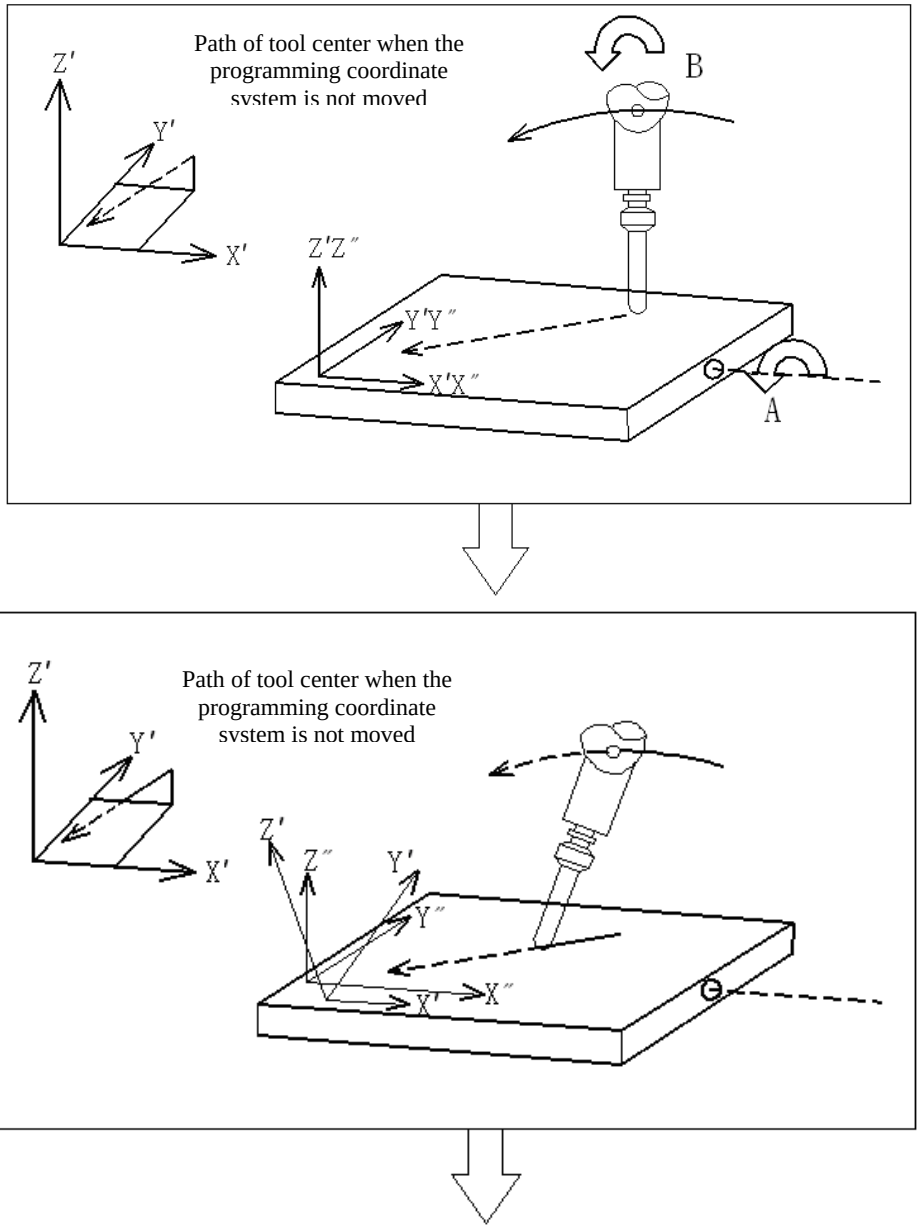


(3) Use of table/spindle-tilting machine tool

If the linear interpolation of X/Y/Z is specified in the programming coordinate system, the center point of the tool linearly moves towards the worktable (workpiece) through control during rotation of the rotary table and the tool. Meanwhile, the center point of the tool shall move as per the specified speed towards the worktable (workpiece) through the speed control.

In this type of machine, the programming coordinate system will remain unchanged if the tool rotates but the worktable does not rotate. However, the programming coordinate system will rotate correspondingly if the worktable rotates.

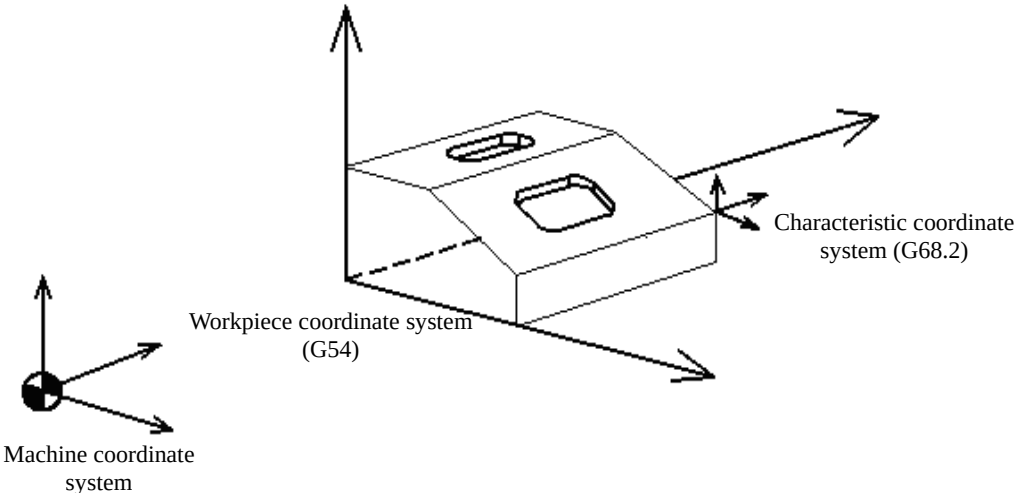
N1 G00 G90 A0 B0;	
N2 G54;	Prepare the programming coordinate system
N3 G43.4 H01;	Start the tool center point control H01 is the tool compensation number
N4 G00 X200.0 Y150.0 Z20.0;	Move towards the start point
N5 G01 X5.0 Y5.0 Z5.0 A60.0 B45.0 F500;	Linear interpolation
N6 G49;	Cancel the tool center point control
N7 M30;	


Path of tool center when the programming coordinate system is not moved
Z′
Y′
X′
B
Z′Z″
Y′Y″
X′X″
A
Path of tool center when the programming coordinate system is not moved
Z′
Y′
X′
Z″
Y″
X″
Path of tool center when the programming coordinate system is not moved
Path of control point in the machine coordinate system
Path of control point in the programming coordinate system

8.5 Inclined surface machining (G68.2)

Function and purpose

When drilling, concave surface milling and other operations are performed on an inclined plane relative to the workpiece reference plane, if these commands can be relative to a coordinate system fixed on the inclined plane, the programming will become very simple. This function provides the ability to specify machining commands relative to the "Characteristic Coordinate System". The characteristic coordinate system is defined in the workpiece coordinate system. See the figure below for the relationship between the "Characteristic Coordinate System" and the workpiece coordinate system.



Command format

G68.2 Xx0 Yy0 Zz0 Iα Jβ Kγ;Setting of characteristic coordinate system
G53.1; Tool axis direction control
…
G69; Characteristic coordinate system cancelled

x0,y0,z0	The point of origin of the characteristic coordinate system is the absolute coordinate in the workpiece coordinate system
α,β,γ	The Euler angle is used to specify the orientation of the characteristic coordinate system

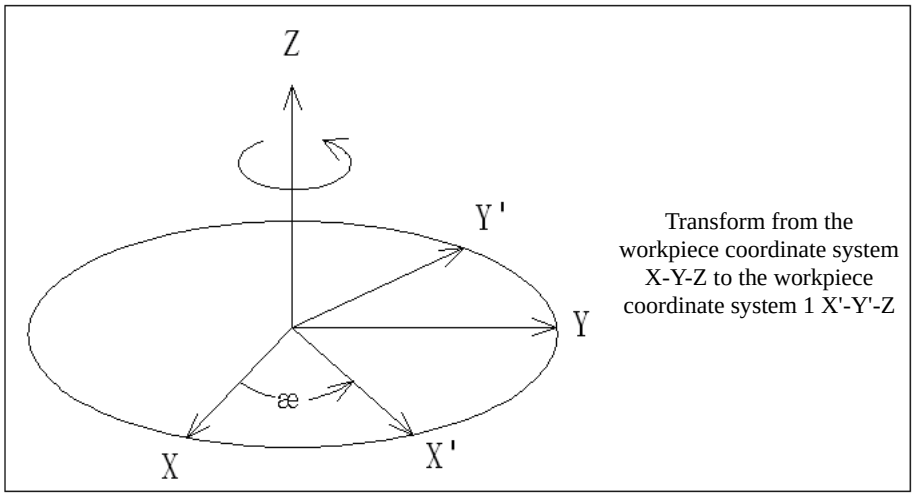
(1) Tool axis direction control (G53.1)

1. G53.1 must be specified in the command program after G68.2 is specified, and an alarm will occur for G53.1 if G68.2 hasn't been specified first.
2. G53.1 must be specified separately.
3. Specification of G53.1 will be accompanied by rotation of the rotation axis generally. In this case, the movement speed of the rotation axis is the highest rapid movement speed during rapid movement, and is the specified speed under the mode of cutting feed.

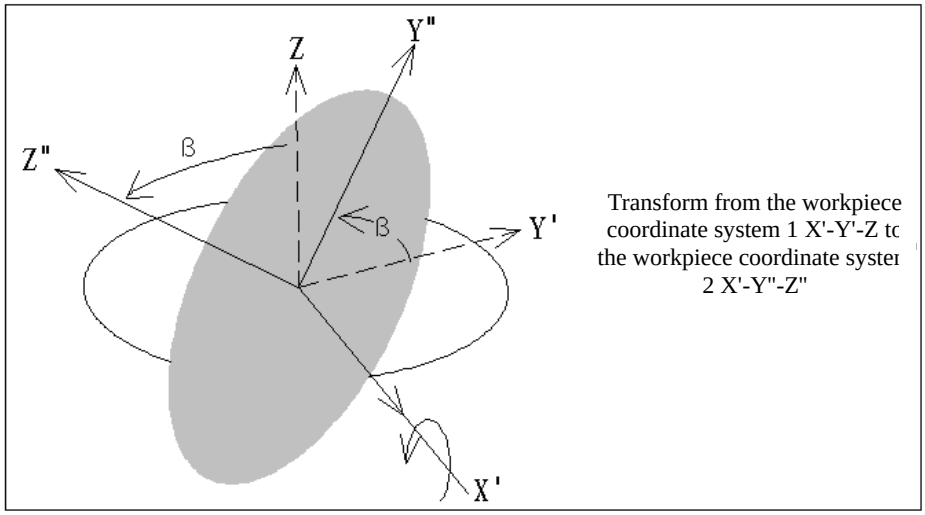
(2) Setting of the coordinate system of Euler angle

In the command, the point of origin centering the workpiece coordinate system is rotated successively to determine the characteristic coordinate system. Firstly, set the coordinate system rotating at an angle of α centering Z-axis of the "Workpiece Coordinate System" as "Coordinate System 1" and set that rotating at

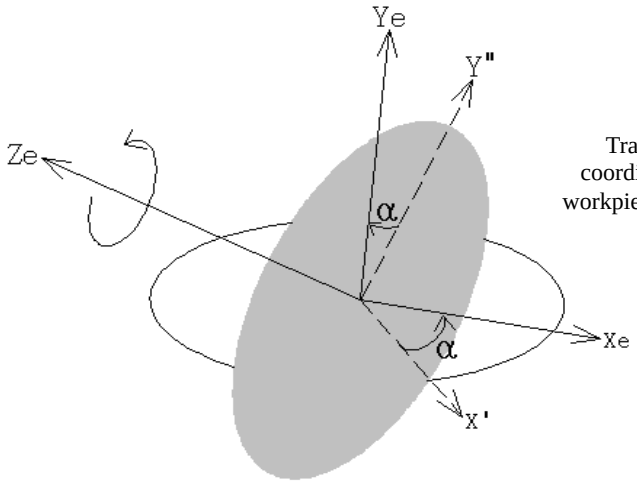
an angle of β centering X-axis of the "Coordinate System 1" as "Coordinate System 2", then translate the coordinate system rotating at an angle of γ centering Z-axis of the "Coordinate System 2"" to (Xo, Yo, Zo) from the point of origin of the workpiece coordinate, and finally the coordinate system obtained is the "Characteristic Coordinate System". See the figure below for the conversion between the "Workpiece Coordinate System" and the "Characteristic Coordinate System":



Transform from the workpiece coordinate system X-Y-Z to the workpiece coordinate system 1 X'-Y'-Z



Transform from the workpiece coordinate system 1 X'-Y'-Z tc the workpiece coordinate syster 2 X'-Y"-Z"



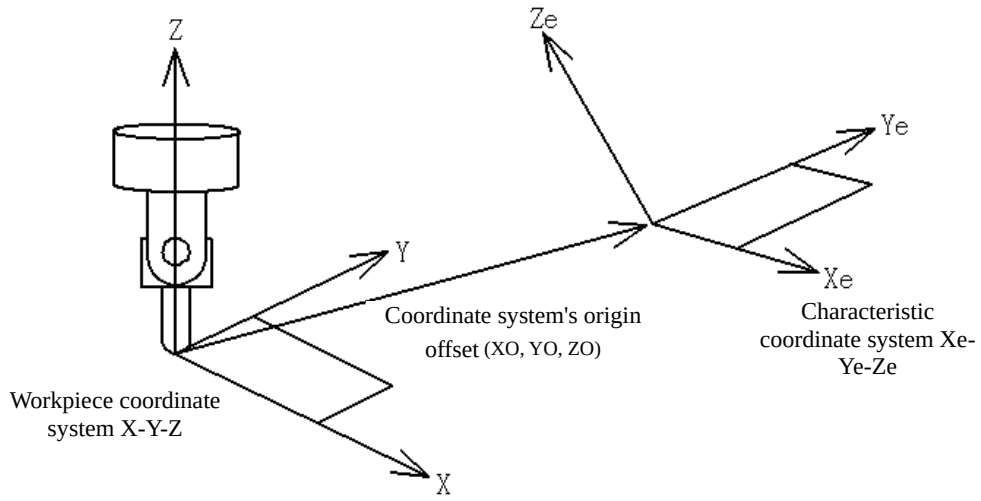
Transform from the workpiece coordinate system 2 X'-Y"-Z" to the workpiece coordinate system Xe-Ye-Ze

Detailed description

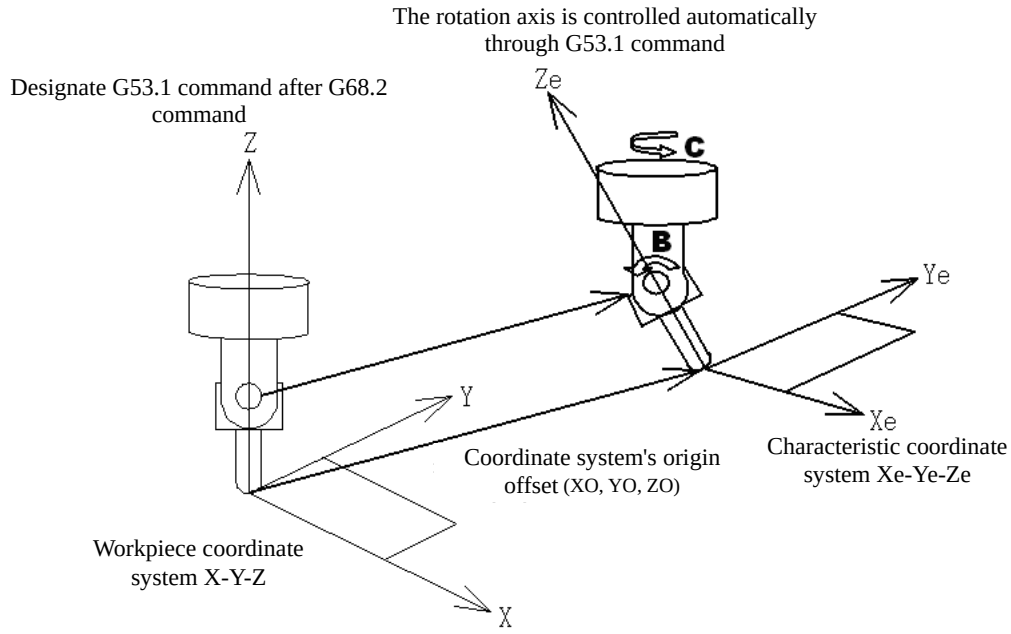
For G68.2 command, move the coordinate system of the machining program to "characteristic coordinate

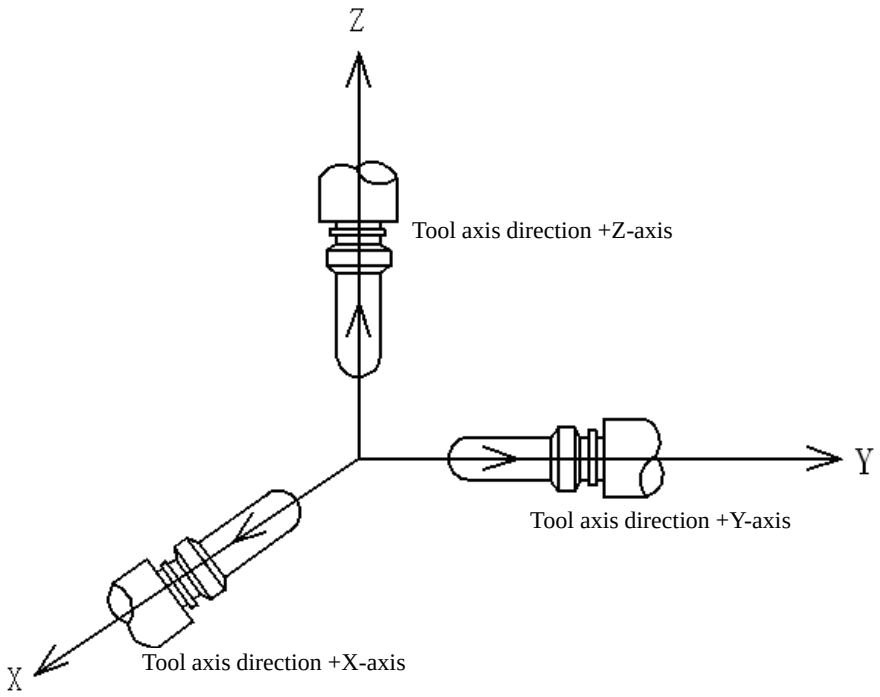
system" and the command coordinates at the follow-up program block are relative to the "characteristic coordinate system", until G69 appears. If the relationship between the characteristic coordinate system and the workpiece coordinate system is pre-set through G68.2 and the angle of the rotation axis is not specified, the +Z-direction of the characteristic coordinate system will be specified as the tool axis direction by G53.1. See the figure below for the relationship of the two commands:

C68.2 command



The vector direction of the tool axis is the direction pointed from tool nose to the handle (as shown in the figure below):





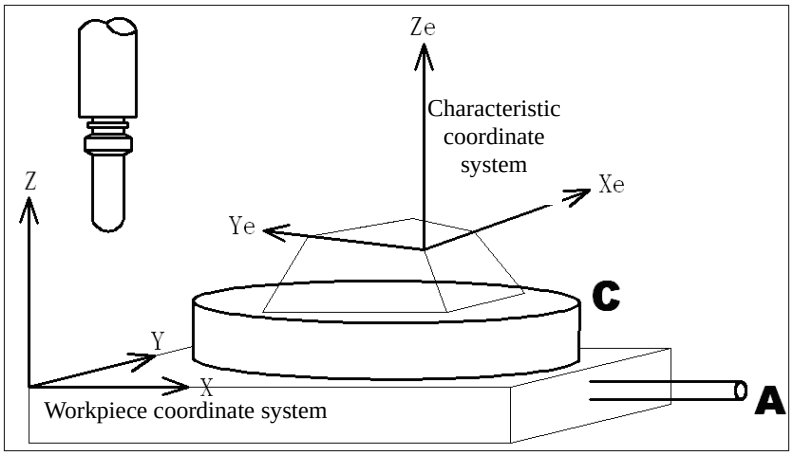
The function is applicable to three types of 5-axis machine:

1. The table-tilting 5-axis machine tool whose worktable is controlled by two rotation axes;
2. The spindle-tilting 5-axis machine whose tool axis is controlled by two rotation axes;
3. The table/spindle-tilting 5-axis machine tool whose tool axis is controlled by one rotation axis and worktable is controlled by one rotation axis.

(1) Table-tilting machine tool

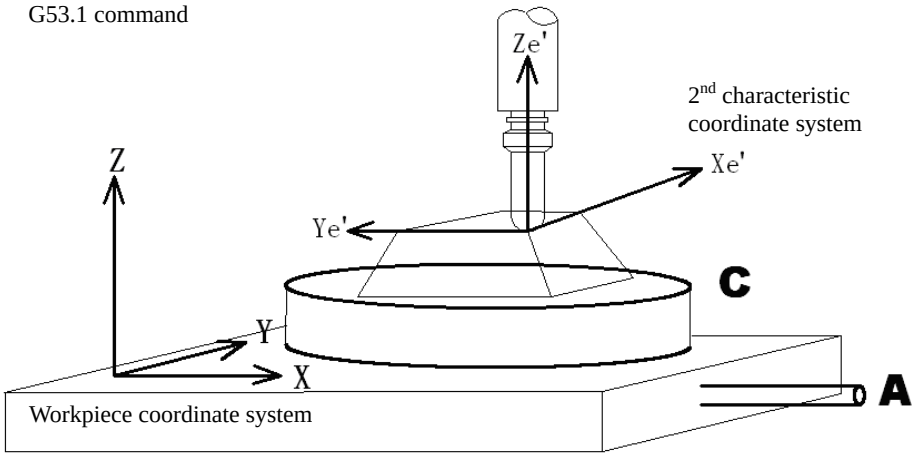
When the function is used for the rotary machine with two rotation axes worktables, the "Characteristic Coordinate System Xe-Ye-Ze" shall be set first through G68.2 command on the "Workpiece Coordinate System" through the point of origin (x0, y0, z0) of the coordinate system and the Euler angle. Afterwards, movement of the rotation axis will be calculated and controlled by G53.1 command, turning the tool axis direction into +Z-direction of the "Characteristic Coordinate System".

Taking A and C types table-tilting as an example, G68.2 is used to set the characteristic coordinate system as shown in the figure below. At this time, only the characteristic coordinate system is set, without any action.



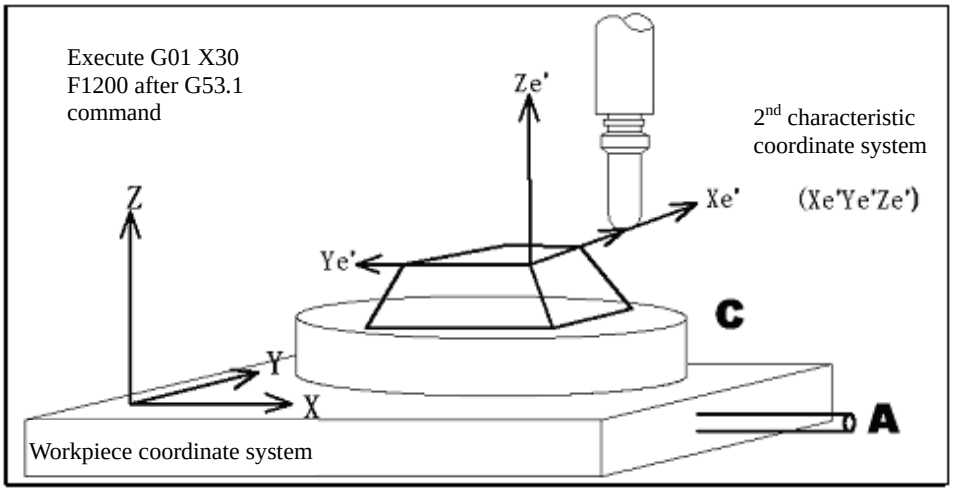
After the characteristic coordinate system (the characteristic coordinate system at this time is called as

the 1st characteristic coordinate system) is set with G68.2, if G53.1 is used to control movement of the rotation axis, CNC will control movement of two rotating axes, turning the tool axis direction into +Z-axis direction of the "Characteristic Coordinate System". If maintaining the position of the characteristic coordinate system relative to the workpiece, the characteristic coordinate system will vary with the movement of the rotation axis. The characteristic coordinate system at this time is called as the 2nd characteristic coordinate system. The follow-up coordinates for movement commands are relative to the 2nd characteristic coordinate system. As shown in the following figure:



(2) Spindle-tilting machine tool

When this function is used in a spindle-tiltingmachine, the G68.2 command sets the characteristic coordinate system, and the G53.1 command controls the two tool rotation axes, so that the tool axis direction coincides with the +Z-direction of the characteristic coordinate system, and the tool axis rotates on the tool axis. Afterwards, G43 can be used for tool length compensation to move the control point to the tool center point.



(3) Table/spindle-tilting machine tool

This function can also be used in the table/spindle-tilting machine tool, which uses the tool rotation axis to rotate the tool and the table rotation axis to rotate the worktable. The "Characteristic Coordinate System Xe-Ye-Ze" determined as per (x0, y0, z0) and the Euler angle is set on the "Workpiece Coordinate System" through G68.2. The tool axis direction is turned as +Z-direction of the "Characteristic Coordinate System" through control over the rotation axis with G53.1. For the table/spindle-tilting machine tool similar to the double-table machine, to maintain the relative position of the workpiece and the characteristic coordinate system, rotation of the worktable will turn the "Characteristic Coordinate System" (the 1st characteristic coordinate system) set through G68.2 as a new "Characteristic Coordinate

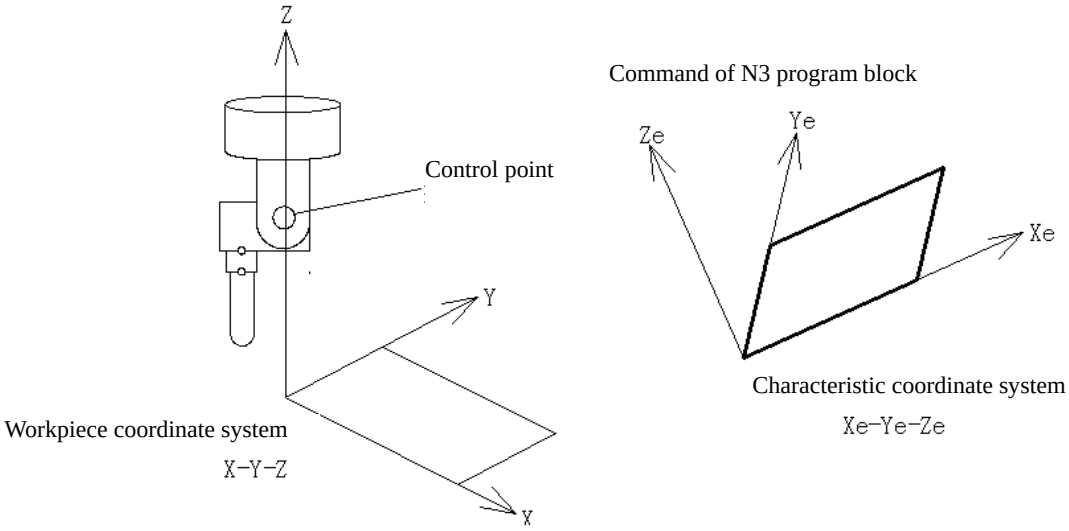
System" (the 2nd characteristic coordinate system). The tool axis direction is actually +Z-direction of the "Second Characteristic Coordinate System". Similar to the spindle-tilting machine tool, for the table/spindle-tilting machine tool, the position of the control point and the length compensation for offset vector when the tool axis intersects with the rotation axis can be considered in the characteristic coordinate system through G43 after G68.2.

Program examples

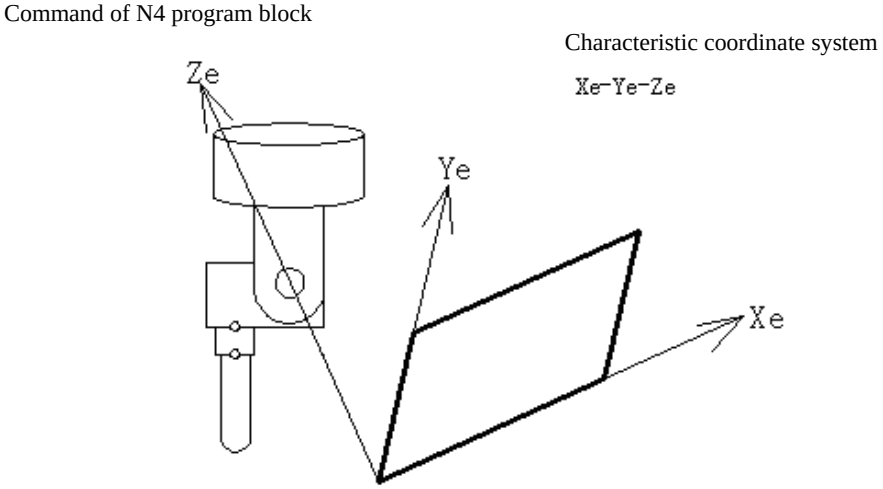
For instance, if the tool axis not intersects with the rotation axis of the spindle-tilting machine:

```
O1000
N1 G54 G43 H01;
N2 G90 G01 X0 Y0 Z100.0 F1000;
N3 G68.2 X100.0 Y100.0 Z50.0 I30J15.0 K20.0;
N4 G01 X0 Y0 Z30.0 F1000;
N5 G53.1;
N6 G01 X0 Y0 Z0;
```

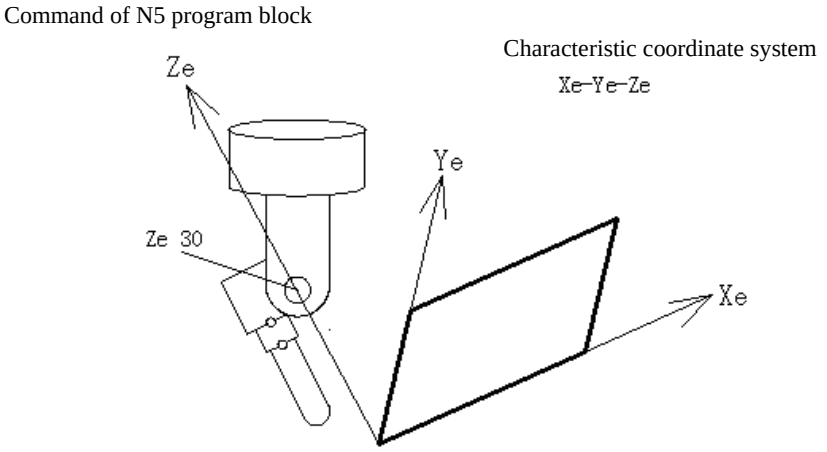
(1) The characteristic coordinate system is set through G68.2 command and the machine does not act at this moment.



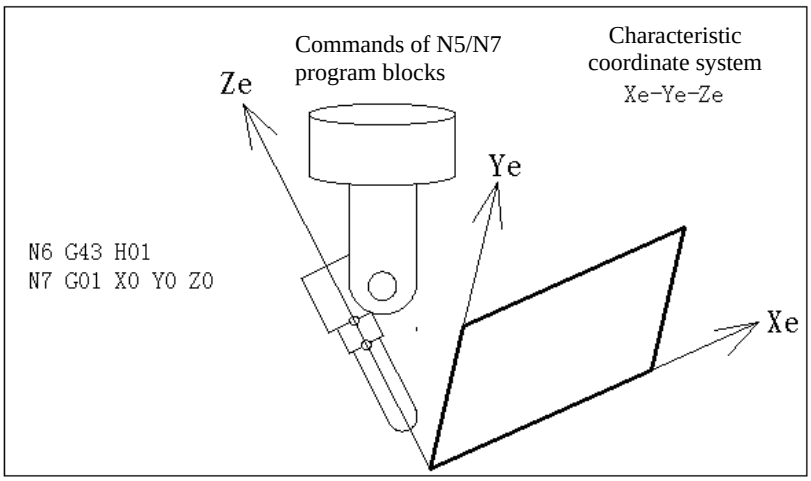
(2) Since the coordinate after G68.2 is designed by reference to the characteristic coordinate system, the follow-up movement is within the characteristic coordinate system.



(3) Rotation of the tool axis is controlled through G53.1, so that the tool axis direction can overlap with +Z-direction of the characteristic coordinate system.



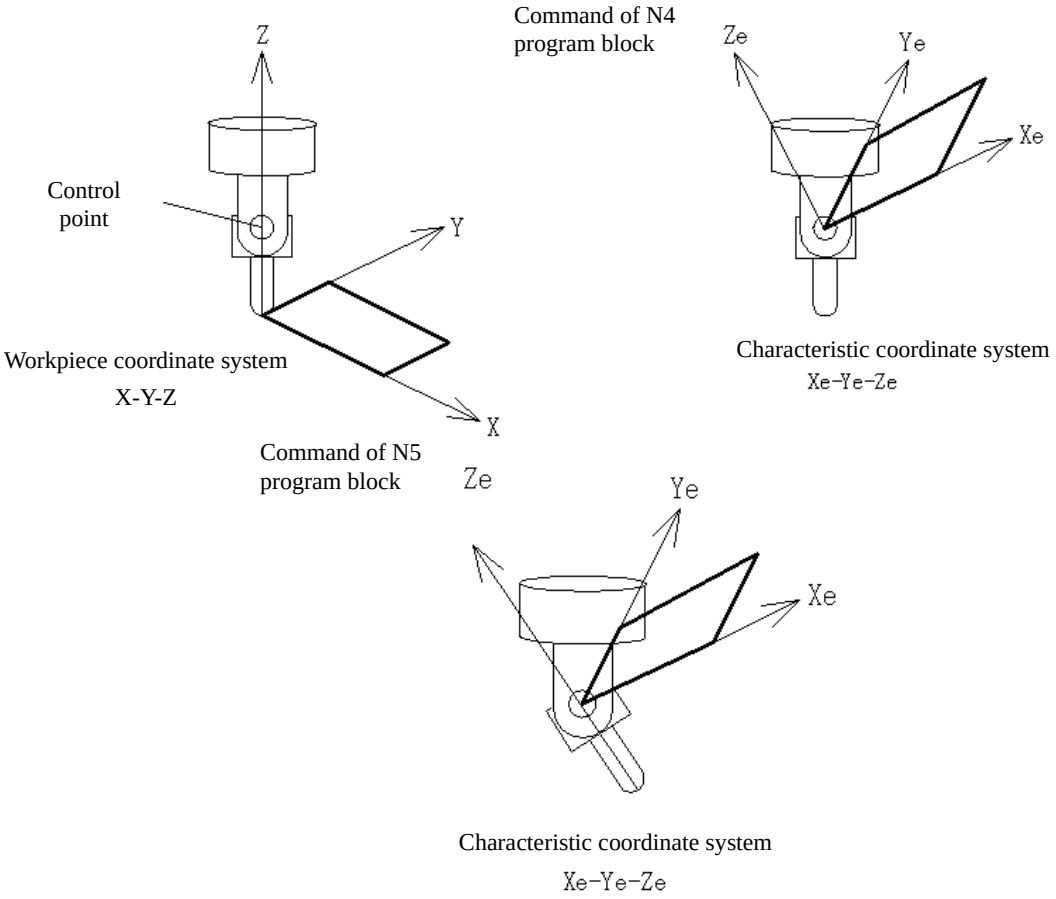
(4) The tool length and the offset vector when the tool axis intersects with the rotation axis are considered by G43 command, for length compensation for the command coordinate after G43. G43 itself will not trigger movement, and length compensation is only valid in case of movement commands after G43.



(5) The action with G53.1 only without G43

Length compensation shall not be made if without G43 and the coordinates of the program are the practical coordinates of the control point. The sample program is as follows:

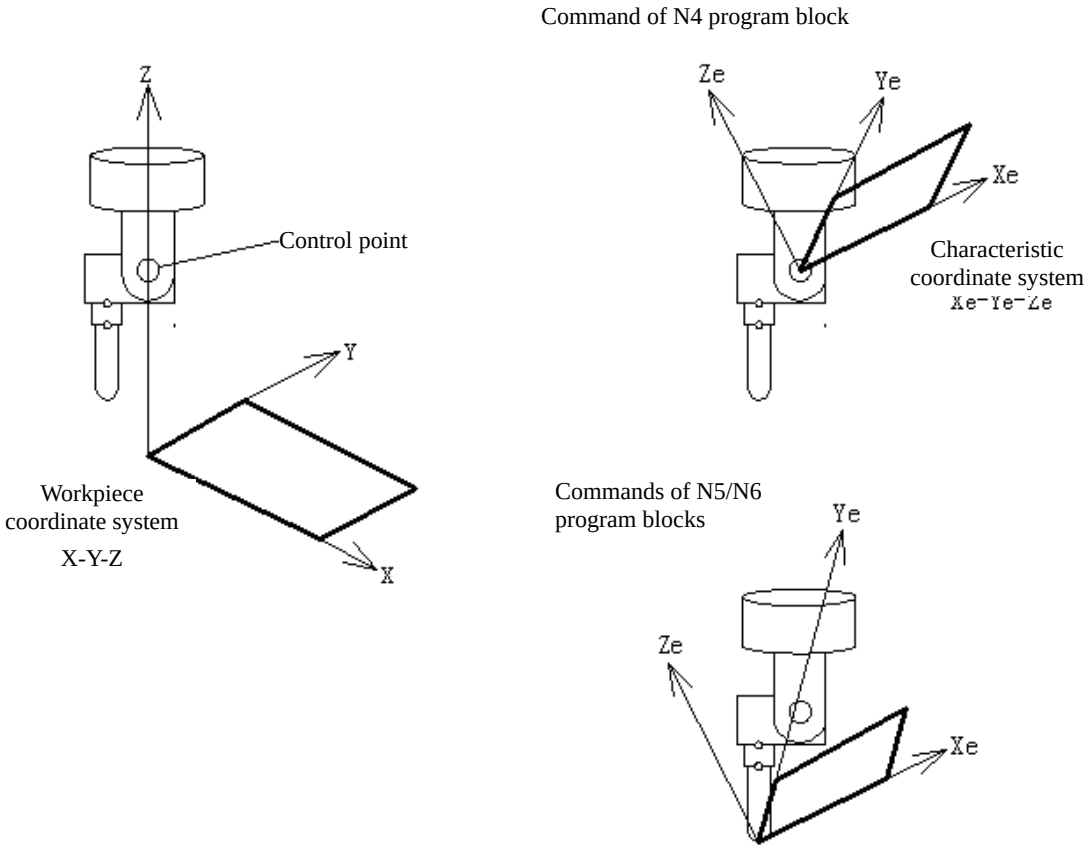
```
O200;
N1 G54;
N2 G90 G01 X0 Y0 Z30.0 F1000;
N3 G68.2 X100.0 Y100.0 Z50.0 I30.0 J15.0 K20.0;
N4 G01 X0 Y0 Z0 F1000;
N5 G53.1;
```



(6) The action with G43 only without G53.1

When there is only G43, the tool axis should not be rotated. Considering the cross offset vector and tool length of the tool and the rotation axis, the length compensation is performed in the characteristic coordinate system. The G43 command itself does not act, but the compensation is performed in the motion command after G43. The sample program is as follows:

N1 G54 G43 H01;
N2 G90 G01 X0 Y0 Z30.0 F1000;
N3 G68.2 X100.0 Y100.0 Z50.0 I30.0 J15.0 K20.0;
N4 G01 X0 Y0 Z0 F1000;
N5 G01 X0 Y0 Z0 F1000;

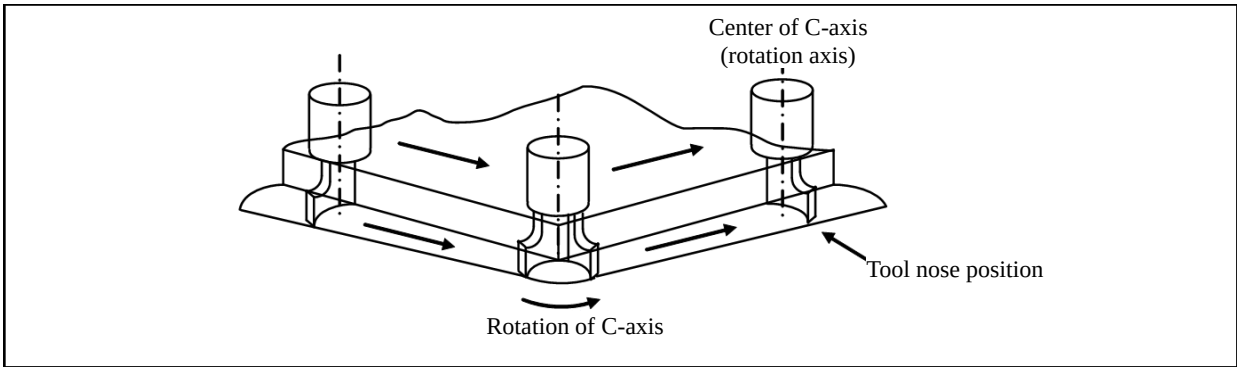


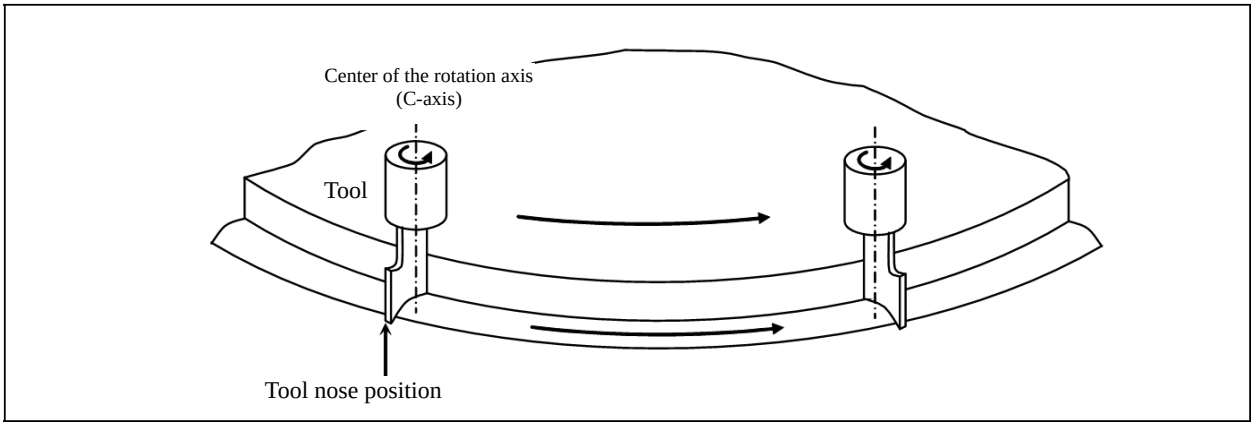
8.6 Normal direction control (G40.1/G41.1/G42.1)

Function and purpose

During machining, when the tool with rotation axis (C-axis) is moving on the XY plane, the tool can be controlled through such function as normal direction control, so that C-axis can always be perpendicular to the tool path.

Under the state of circular interpolation, rotation of the rotation axis is controlled, so that it can synchronize with the action of circular interpolation.





Command format

G40.1 X_ Y_ F_;	Normal control cancelled
G41.1 X_ Y_ F_;	Normal control open on the left
G42.1 X_ Y_ F_;	Normal control open on the right

X_ Y_	End point coordinate
F_	Feed speed

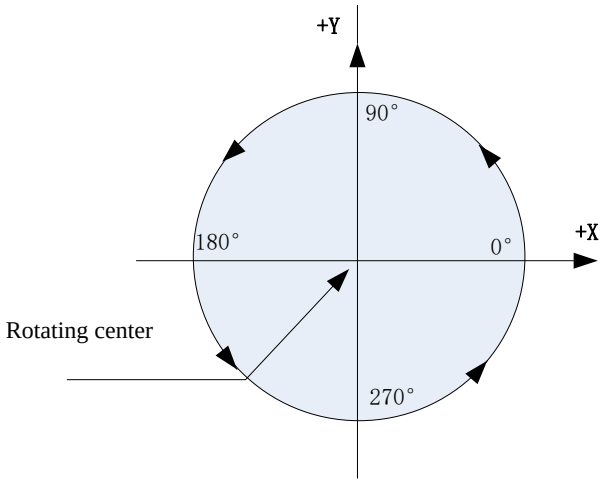
Detailed description

1. Axial direction and angle of the rotation axis

Setting of the normal direction control uses parameter #7120 of the rotation axis; #7120=65 indicates that A-axis controls the normal direction of the tool, while #7120=66 and 67 respectively correspond to B-axis and C-axis.

The direction of the rotation axis is controlled by the plane selection command; G17 corresponds to the axial direction taking Z-axis as the rotation axis; G18 takes Y-axis as the axial direction and G19 takes X-axis as the axial direction.

The tool axis direction at 0 °can be set through parameter #7130. In general, it is the tool axis direction with the positive direction of the coordinate axis as 0 °. For example, when the rotation axis is at 0 °under G17, the tool axis direction is the positive direction of X-axis; the positive direction of Y-axis is at 90 °, the negative direction of X-axis is at 180 °and the negative direction of Y-axis is at 270 °. In this case, X=1, Y=0 and Z=0 in #7130. Any direction can be set as the tool axis and X/Y/Z in parameter #7130 has no need to be set as the unit vector.



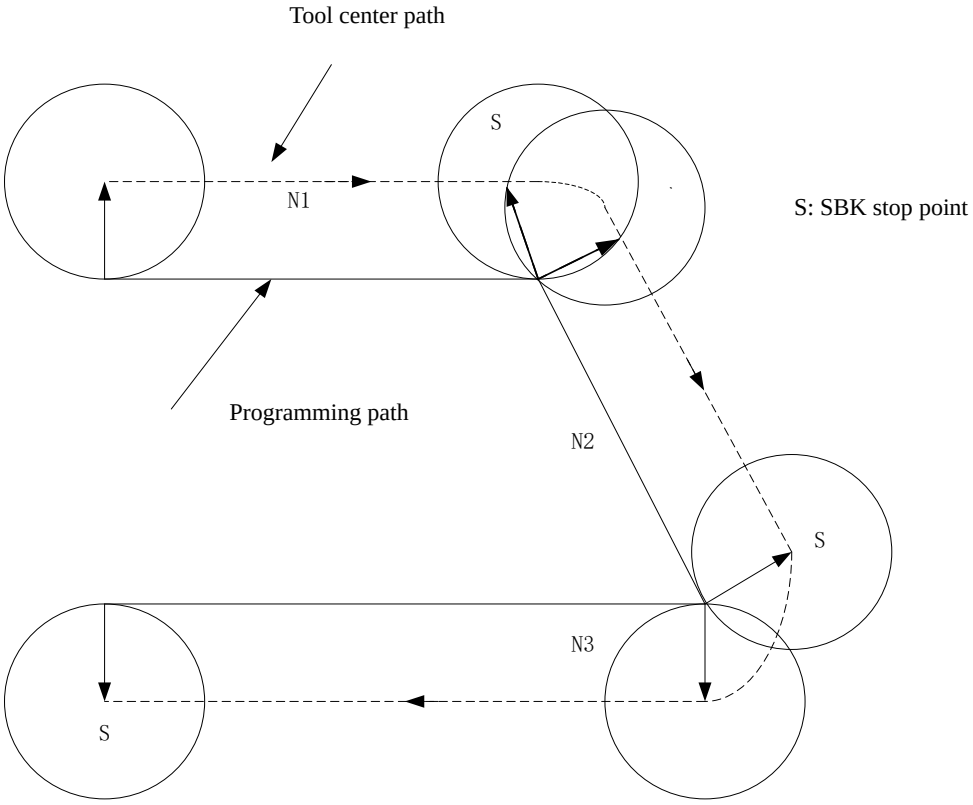
2. Normal direction control of C-axis

While switching from cancellation to normal direction control, the C-axis at the G41.1/G42.1 program block is perpendicular to the tool path.

The command for movement of the tool shall be inserted automatically at the area where the program block of normal direction control is switched, so C-axis is perpendicular to the tool path at the area where each program block starts. The tool shall be oriented first, to make C-axis perpendicular to the tool path specified by the movement commands, and then move along the X-axis and the Y-axis.

In the mode of tool radius compensation, the tool shall be oriented first, to make C-axis perpendicular to the tool path established after compensation.

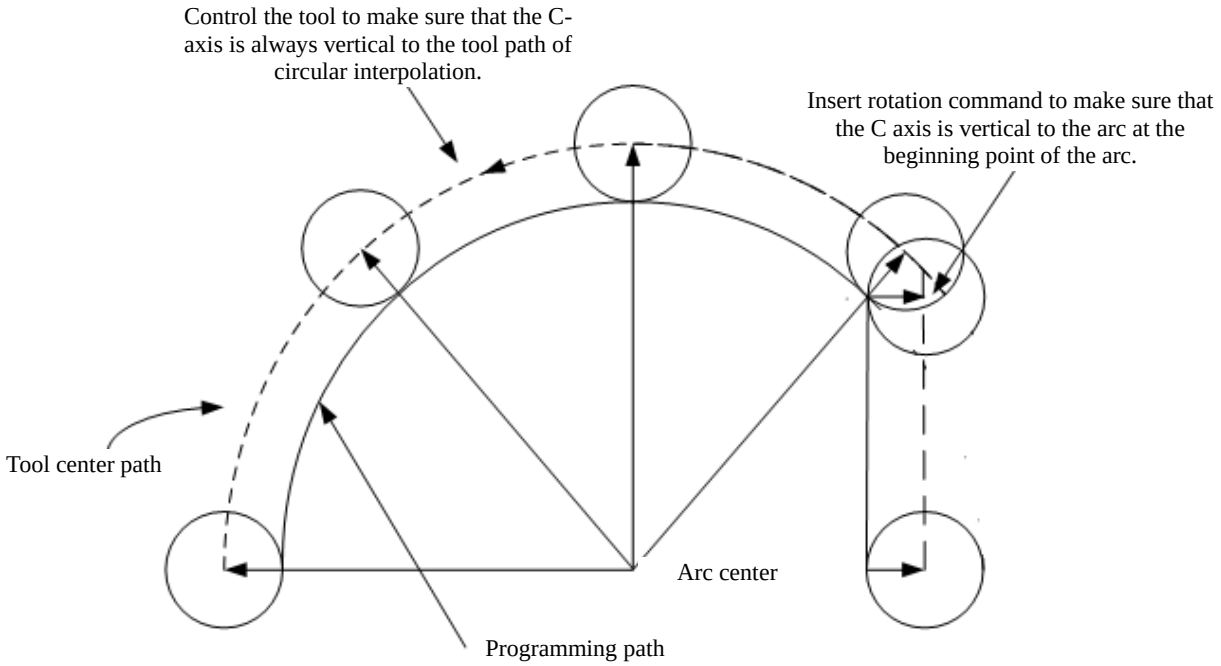
During single block operation, the tool shall not stop between the rotation command and the movement command along the X-axis and Y-axis. Single program block stop always occurs after the tool is moved along the X-axis and Y-axis.



The C-axis rotates before circular interpolation begins, making C-axis vertical to the arc at the start point

of the arc.

Control the tool to make sure that the C-axis is always vertical to the tool path of circular interpolation during circular interpolation.



Note:

The rotation angle of C-axis is less than 180° during normal direction control. That is, no matter what direction it rotates, it always moves by a shorter path.

3. C-axis feed speed

For the tool movement inserted at the area where each program block starts, the feed speed is controlled by F. If the machine is at dry run mode, it moves at the speed of dry run. The tool will move along the X-axis and the Y-axis at a rapid movement speed under the mode of G00.

The feed speed of C-axis during circular interpolation can be calculated as per the following formula:

$$F \times \frac{\text{Movement amount of C-axis}}{\text{Arc length (}mm\text{ or }inch\text{)}} \quad (^\circ\text{/minute})$$

F: Feed speed specified by corresponding arc program block (mm/min or inch/min)

Movement amount of C-axis: The angle difference between the start point of the program block and the end point.

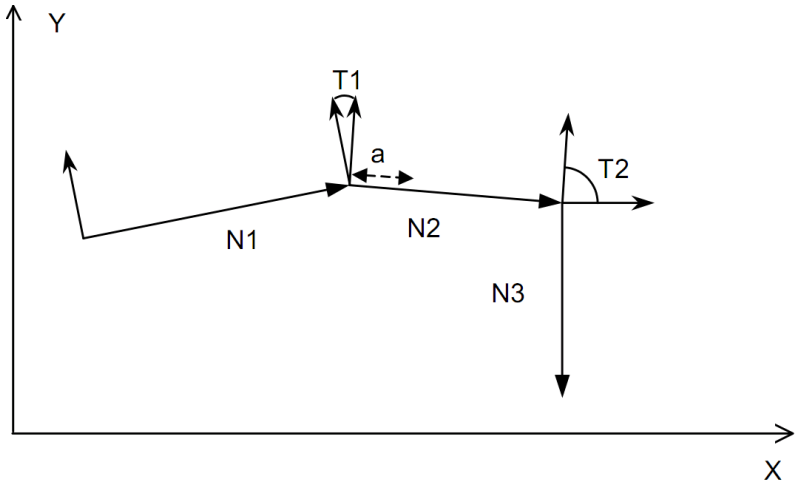
Note:

If the feed speed of C-axis exceeds the highest cutting speed of C-axis specified by #1224, the speed of each axis is limited, to make the feed speed of C-axis lower than the allowable maximum value.

4. Linkage length

To improve the machining speed and to reduce the speed fluctuation, the linkage length of normal direction control can be set through parameter #7121.

As shown in the figure, instead of rotating to the position before executing the N2 block, the C-axis finishes this action at the end of a linkage movement length a.



When length of the program block (arc length) is less than twice of the linkage movement length, the C-axis finishes the angle adjustment at the end of the whole section.

Precautions

1. During normal direction control, no command can be specified to the C-axis. Any command specified at this time shall be ignored.
2. Before processing, the coordinate of C-axis of the workpiece must be linked with its practical position of C-axis on the machine with G92 or other coordinate systems.
3. Helical cutting of CNC is required to be selected for such function. But screw cutting cannot be commanded during normal control mode.
4. Normal direction control cannot be performed with the movement command of G53.
5. The C-axis must be a rotation axis.
6. Normal vector of the tool at 0 °. When #7130 = 1,0,0 and B=0, set the tool at this position with tool nose.

Chapter IX Fixed Cycle of Milling

9.1 Standard fixed cycle (G80~G89/G73/G74/G76)

Function and purpose

The functions of positioning and drilling, boring, tapping and other machining programs are executed according to the pre-determined operation sequence through the command of a single program block. The machining sequence includes the following contents.

See the figure below for the summary on fixed cycle of the control system.

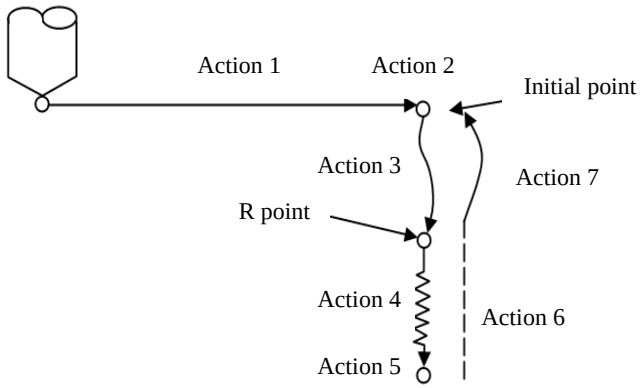
Summary on Fixed Cycle

G Code	Drilling (-Z-Direction)	Action at Hole Bottom	Rollback Action (+Z-Direction)	Use
G73	Interrupted feed	——	Rapid feeding	High-speed deep hole drilling
G74	Cutting feed	Spindle forward	Cutting feed	Reverse tapping
G76	Cutting feed	Orientation of the spindle stopped	Rapid feeding	Fine boring cycle
G80	——	——	——	Fixed cycle cancelled
G81	Cutting feed	——	Rapid feeding	Drilling cycle (oriented drilling)
G82	Cutting feed	Dwell	Rapid feeding	Drilling cycle (counter boring)
G83	Interrupted feed	——	Rapid feeding	Deep hole drilling
G84	Cutting feed	Spindle reverse	Cutting feed	Tapping
G85	Cutting feed	——	Cutting feed	Boring cycle
G86	Cutting feed	Spindle stop	Fast moving	Boring cycle
G87	Cutting feed	Dwell	Rapid operation	Boring cycle (reversal boring)
G88	Cutting feed	Dwell; the spindle stopped	Manual operation or rapid operation	Boring cycle
G89	Cutting feed	Dwell	Cutting feed	Boring cycle

When G80 or G command (G00/G01/G02/G03) of the 01 group is received, the mode of fixed cycle will be cancelled. Meanwhile, the data are cleared.

General process of a fixed cycle

The fixed cycle is made up of the action in 7 sequences (as shown in the figure below):



Action Sequence of Fixed Cycle

- Action 1: Execute the plane axis positioning with G00.
- Action 2: When G87 is designated in the action after the position is finished (initial point), M19 is output from the mechanical end of the control device. Perform M command and start the next action when the finishing signal (FIN) is delivered to the control device. Moreover, when the stop switch for a single block is turned on, the single block stops upon completion of positioning.
- Action 3: Position to R point through rapid feed.
- Action 4: Conduct boring through cutting feed.
- Action 5: The action made at the hole and vary with the mode of the fixed cycle, including spindle stop (M05), spindle reverse (M04), spindle forward (M03), time delay and tool deviation etc.
- Action 6: Retract the tool to R point and cutting feed or rapid feed may be adopted as per the mode of fixed cycle.
- Action 7: Return to the initial point through rapid feed.

However, regardless of whether the action finishing fixed cycle is action 6 or action 7, it can be switched through G98/G99 commands.

Moreover, ignore the data concerning boring if any of X, Y, Z, R command is absent.

(1) Positioning plane

The positioning plane depends on the plane selection code G17/G18/G19.

The positioning axis is the axis except for the drilling axis.

(2) Drilling axis

Even though the fixed cycle includes tapping, boring and drilling, the term drilling will still be used to illustrate the action performing fixed cycle in this chapter. The drilling axis is the basic axis (X/Y/Z) not used to define the position plane or the axis parallel to the basic axis.

The drilling axis shall be determined as per the axis address (basic axis or the parallel axis) designated in the program block of the G code (G73~G89).

If the axis address is not specified for the drilling axis, the basic axis can be deemed as the drilling axis.

Positioning plane and drilling axis

G Code	Positioning Plane	Drilling Axis
G17	Xp-Yp plane	Zp
G18	Zp-Xp plane	Yp
G19	Yp-Zp plane	Xp

Xp: X-axis or its parallel axis

Yp: Y-axis or its parallel axis

Zp: Z-axis or its parallel axis

If U/V/W axis is parallel to X-axis/Y/Z respectively, The parallel axis is set through parameter #1024.

G17G81--------Z_ _: Z-axis as the drilling axis

G17G81--------W_ _: W axis as the drilling axis

G18G81--------Y_ _: Y-axis as the drilling axis

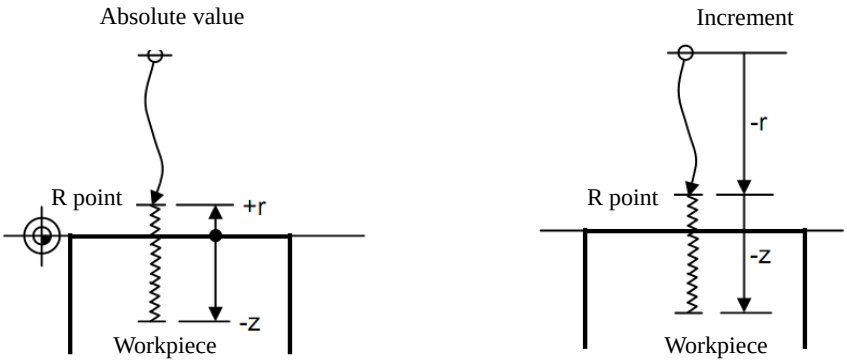
G18G81--------V_ _: V axis as the drilling axis

G19G81--------X_ _: X-axis as the drilling axis

G19G81--------U_ _: U axis as the drilling axis

G17~G19 may not be specified in the G73~G89 program blocks.

(3) Difference of the absolute value command G90 and the increment command G91



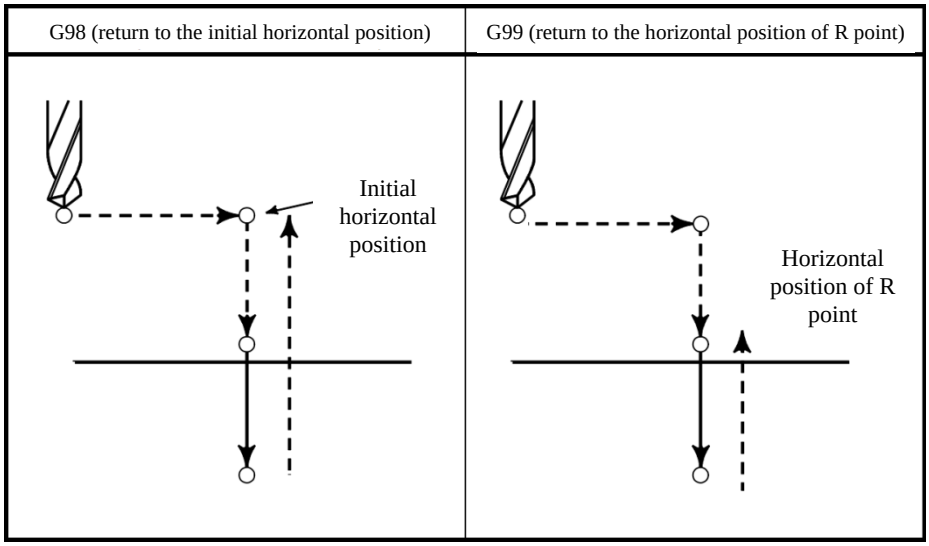
(4) Return to the plane

G98--------return to the initial plane

G99--------return to the plane of R point

When the tool reaches the hole bottom, the tool can return to the plane of R point or the initial plane (specified by G98 and G99).

These commands are modal, and the mode of G98 will always last before designating G99 at the next time after G98 is specified. The default mode of the system is G98.



(5) Number of repeats

K_ shall be used to designate the number of repeats during equidistant drilling.

K is only valid within the program block specified and

The position of the first hole is specified through increments (G91).

If the command (G90) for absolute values is used, holes shall be drilled repetitively at the same position.

Number of repeats K	Largest command value =9999

If K0 is specified, the drilling data will be saved but not be performed.

(6) Cancel

Fixed cycle can be cancelled through G80 or the G code of the 01 group.

G code of the 01 group: G00/G01/G02/G03.

9.2 High-speed deep hole drilling cycle (G73)

Function and purpose

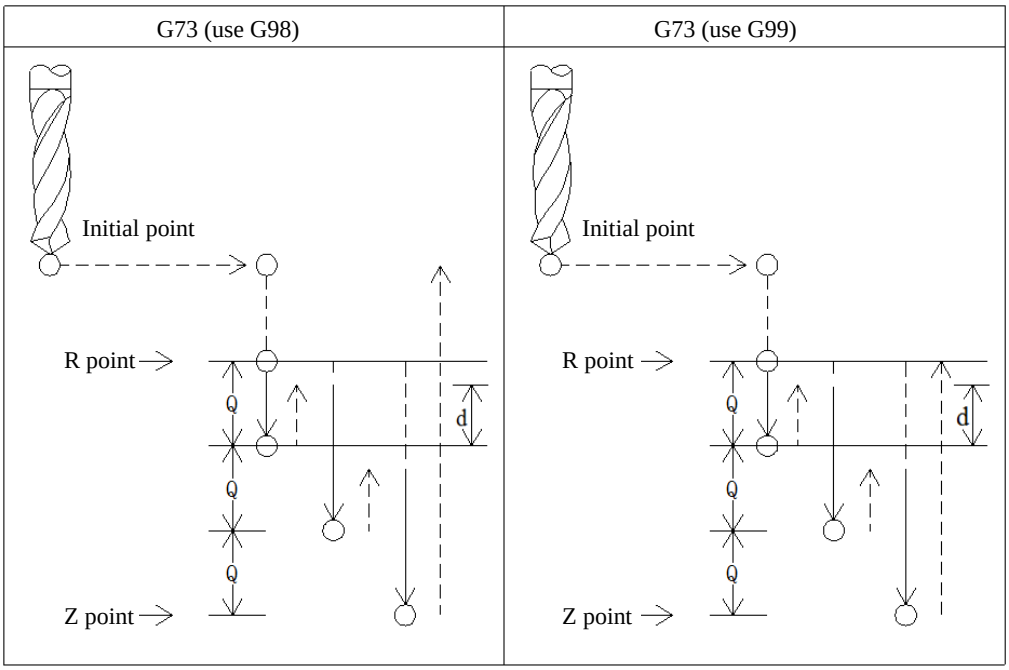
To execute the high-speed deep hole drilling, perform the intermittent cutting feed to the bottom of the hole, and discharge chips from the hole while feeding.

Command format

```
G73 X_Y_Z_R_Q_F_K_;
```

X_ Y_	Positioning data of the hole
Z_	Increment programming indicates the distance between the specified R point and the hole bottom; absolute programming indicates the absolute coordinates at the hole bottom
R_	Increment programming indicates the distance between the plane at the initial point and R point ; absolute programming indicates the absolute coordinates at the R point
Q_	Cutting depth of each cutting feed
F_	Cutting feed speed
K_	Number of repeats

Detailed description



For the high-speed deep hole drilling cycle, the tool shall be fed interruptedly along the Z-axis, so that metal filings can be removed from the hole easily. A small value can be set for the tool retraction amount, so that effective drilling can be realized. For the tool retraction amount d set through parameter #2010, the tool will retract at a rapid movement speed.

Before specifying G73, rotate the spindle with the M code of auxiliary functions.

If tool length compensation is specified in such fixed cycle, such offset shall be maintained during positioning at the R point.

The fixed cycle can be cancelled through G80/G00/G01/G02/G03.

The tool radius offset in fixed cycle is ignored.

Program examples

M3 S1500;	Spindle starts rotating
G90 G99 G73 X0 Y0 Z-150. R-120.Q5 F120;	Position; bore Hole 1 and then return to R point
Y-50.;	Position; drill Hole 2 and then return to R point
Y-80.;	Position; drill Hole 3 and then return to R point
X10.;	Position; drill Hole 4 and then return to R point
Y10.;	Position; drill Hole 5 and then return to R point
G98 Y75.;	G98 Y75.; Positioning; drilling Hole 6 and then return to the initial plane
G80;	
G28 G91 X0 Y0 Z0;	Return to the reference point
M5;	Spindle stops
M30;	End of program

During machining of the Hole 2 – Hole 6 in the example above, Q is ignored and chips will be removed simultaneously.

Precautions

(1) Please cancel the fixed cycle before switching of the drilling axis, or otherwise an alarm will occur.

(2) Drilling shall not be performed in case of no commands for X, Y, Z, R or any other axis in the program block.

(3) Q is specified in the program block performing drilling. If being specified in a program block not performing drilling, it cannot be saved as modal data.

9.3 Left-hand tapping cycle (G74)

Function and purpose

If left-hand tapping is performed, the spindle will rotate clockwise while reaching the hole bottom.

Command format

```
G74 X_ Y_ Z_ R_ P_ F_ K_;
```

X_ Y_	Positioning data of the hole
Z_	Increment programming indicates the distance between the specified R point and the hole bottom; absolute programming indicates the absolute coordinates at the hole bottom
R_	Increment programming indicates the distance between the plane at the initial point and R point ; absolute programming indicates the absolute coordinates at the R point
P_	Dwell time
F_	Cutting feed speed
K_	Number of repeats

Detailed description

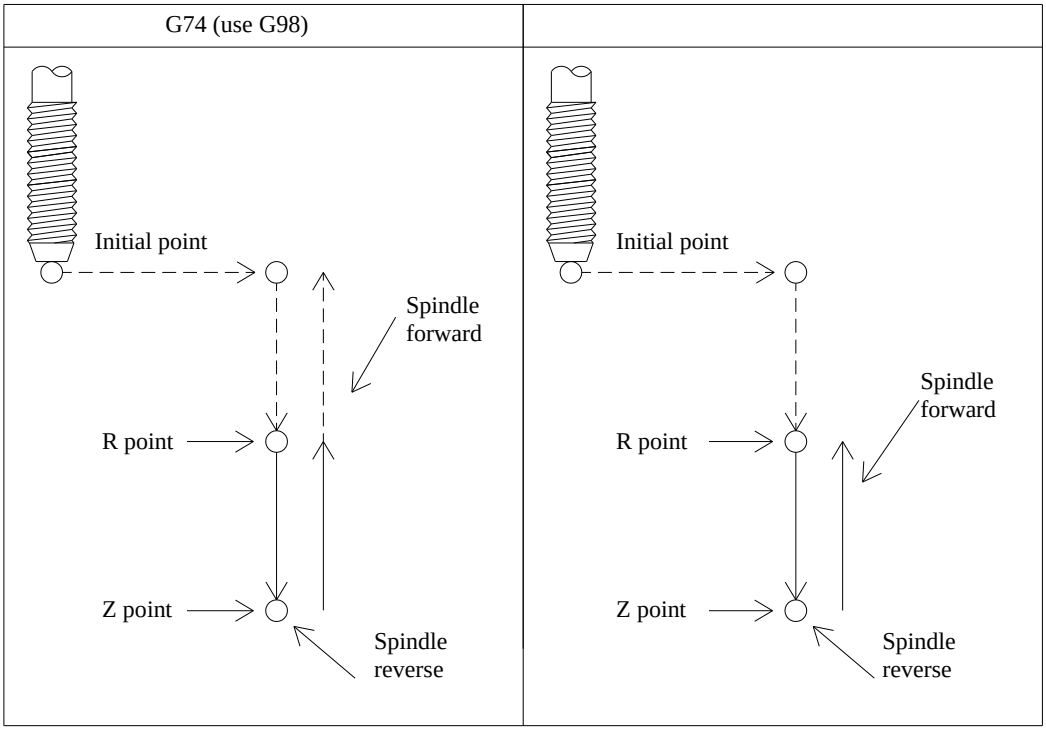
The spindle rotates counterclockwise to perform tapping. When it reaches the bottom of the hole, in order to retract the spindle, it rotates in the opposite direction. This process generates threads. During tapping, the feed override is ignored, and the feed holding will not stop the machine until the return action finishes.

Before designating G74, rotate the spindle through the M code of auxiliary functions. If counterclockwise rotation of the spindle is not designated, the system will be adjusted as counterclockwise rotation on the R plane automatically as per the current command speed of the spindle.

When tool length compensation is specified in such fixed cycle, such offset shall be maintained when positioning to the R point.

The fixed cycle can be cancelled through G80/G00/G01/G02/G03.

The tool radius offset in fixed cycle is ignored.



Program examples

```
M4 S100;                                    Spindle starts rotating
G90 G99 G74 X30. Y-25. Z-150. R-120. P300 F1
50;     Position; tap Hole 1 and then returning to R point
Y-55.;                                      Position; tap Hole 2 and then returning to R point
Y-75.;                                      Position; tap Hole 3 and then returning to R point
X100.;                                      Position; tap Hole 4 and then returning to R point
Y-55.;                                      Position; tap Hole 5 and then returning to R point
G98 Y-75.;                                  Positioning; tapping Hole 6 and then return to the
                                             initial plane
G80;
G28 G91 X0 Y0 Z0 ;                          Return to the reference point
M5;                                         Spindle stops
M30;                                        End of program
```

Precautions

(1) Please cancel the fixed cycle before switching the drilling axis, or otherwise an alarm will occur.

(2) In case any command for X, Y, Z, R or any other axis is not provided in the program block, start performing the program at the position located last time in default for X/Y, and start performing the program at the position located by Z-axis last time in default for R.

(3) Specify P in the program block performing drilling. If being specified in the program block not performing drilling, it cannot be saved as modal data.

9.4 Fine boring cycle (G76)

Function and purpose

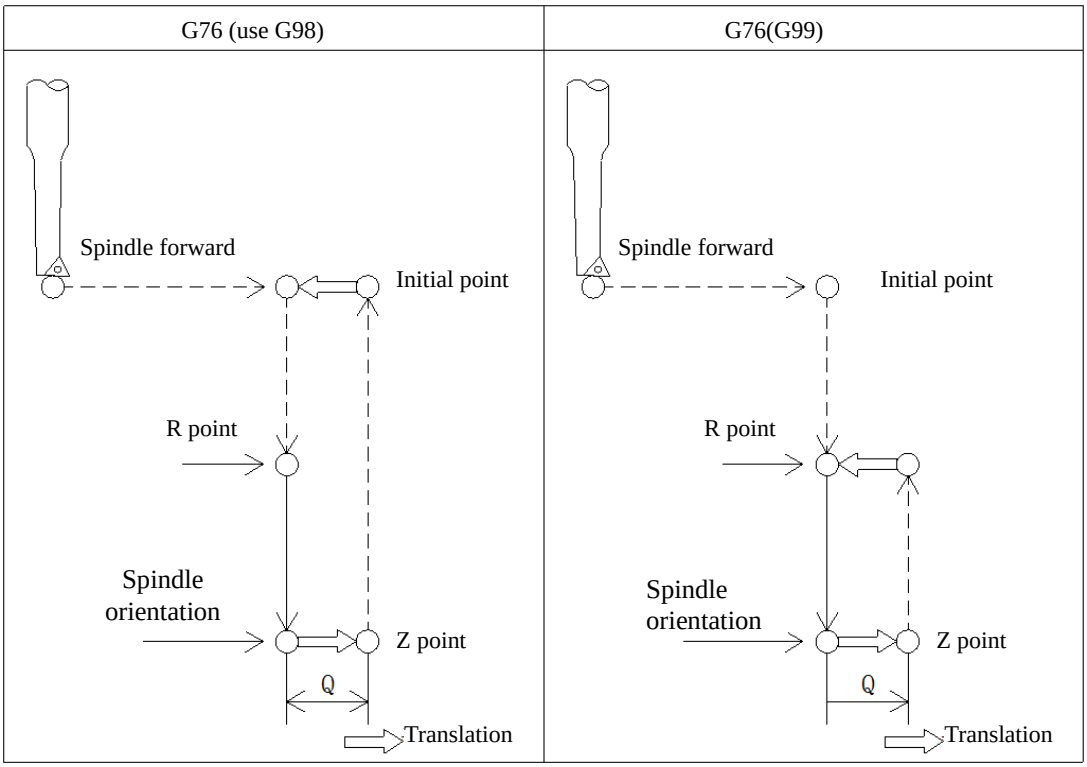
To perform a precise boring operation, when the bottom of the hole is reached, the spindle stops and oriented, and the tool leaves the machined surface of the workpiece and returns. Applicable to the machine with positioning function of spindle.

Command format

G76 X_ Y_ Z_ R_ P_ { Q_ / I_ J_ } F_ K_;

X_ Y_	Positioning data of the hole
Z_	Increment programming indicates the distance between the specified R point and the hole bottom; absolute programming indicates the absolute coordinates at the hole bottom
R_	Increment programming indicates the distance between the plane at the initial point and R point ; absolute programming indicates the absolute coordinates at the R point
P_	Dwell time
Q_	Offset at the hole bottom; the offset direction set by the parameter shall be used
I_J_	Offset at the hole bottom; offset as per the given value
F_	Cutting feed speed
K_	Number of repeats

Detailed description



When the tool reaches the hole bottom, the spindle will stop at a fixed orientation location and the tool will retract by moving towards the opposite direction of the tool nose, so that the machining surface will not be damaged, realizing precision and effective boring. Parameter Q specifies the tool retracting distance. The offset direction is set by the bit parameters 2000.4 and 2000.5 (RD1, RD2). Q must be positive, and the symbol will be ignored even it is negative. The offset of Q at the hole bottom is a modal value saved inside the fixed cycle and must be specified carefully on the ground that it is also used for cutting depth of G73 and G83.

The offset shall be specified as per the following modes if I/J is used.

G17: I represents offset in the X direction, and J represents offset in the Z-direction

G18: I represents offset in the X direction, and J represents offset in the Z-direction

G19: I represents offset in the Y direction, and J represents offset in the Z-direction

Before designating G76, rotate the spindle with the M code of auxiliary functions.

When tool length compensation is specified in the fixed cycle, the offset is maintained before positioning to the R point.

The fixed cycle can be cancelled through G80/G00/G01/G02/G03.

The tool radius offset in fixed cycle is ignored.

Program examples

M3 S500;	Spindle starts rotating
G90 G99 G76 X30.Y-25 Z-150. R-120.Q5 P1000 F120.;	Positioning; boring Hole 1; suspend for 1s at the hole bottom; move for 5 mm after position at the hole bottom; return to R point finally
Y-55.;	Position; bore Hole 2 and then return to R point
Y-75.;	Position; bore Hole 3 and then return to R point
X100.;	Position; bore Hole 4 and then return to R point
Y-55.;	Position; bore Hole 5 and then return to R point
G98 Y-75.;	Position; bore Hole 6 and then return to the initial plane
G80 G28 G91 X0 Y0 Z0;	Return to the reference point
M5;	Spindle stops

Precautions

(1) Please cancel the fixed cycle before switching the drilling axis, or otherwise an alarm will occur.

(2) Drilling shall not be performed in case of no commands for X, Y, Z, R or any other axis in the program block.

(3) Specify P and Q in the program block performing drilling. If being specified in the program block not performing drilling, it cannot be saved as modal data.

(4) If I, J and Q are used for G76/G87 simultaneously, Q is preferred.

(5) When the special fixed cycles G121~G125 are on the same program block as G87/G76, Q must be specified, or otherwise an alarm (PS0170) will occur.

9.5 Fixed cycle cancelled (G80)

Function and purpose

Cancel fixed cycle.

Command format

G80;

Detailed description

Cancel all fixed cycles and perform the normal operation. R point and Z point are also cancelled. Other data concerning drilling and boring will also be cleared.

Program examples

M3 S100;	Spindle starts rotating
G90 G99 G88 X30. Y-25. Z-150. R-120. F120;	Position; bore Hole 1 and then return to R point
Y-55.;	Position; bore Hole 2 and then return to R point
Y-75.;	Position; bore Hole 3 and then return to R point
X100.;	Position; bore Hole 4 and then return to R point
Y-55.;	Position; bore Hole 5 and then return to R point
G98 Y-75.;	Position; bore Hole 6 and then return to the initial plane
G80;	Fixed cycle cancelled
G28 G91 X0 Y0 Z0 ;	Return to the reference point to cancel the fixed cycle
M5;	Spindle stops

9.6 Drilling cycle (G81)

Function and purpose

It is used for drilling operations, performing cutting infeeds to the bottom of the hole, and retracting the tool quickly from the bottom of the hole.

Command format

G81 X_ Y_ Z_ R_ F_ K_;

X_ Y_	Positioning data of the hole
Z_	Increment programming indicates the distance between the specified R point and the hole bottom; absolute programming indicates the absolute coordinates at the hole bottom
R_	Increment programming indicates the distance between the plane at the initial point and the R point; absolute programming indicates the absolute coordinates at R point
F_	Cutting feed speed

K_	Number of repeats

Detailed description

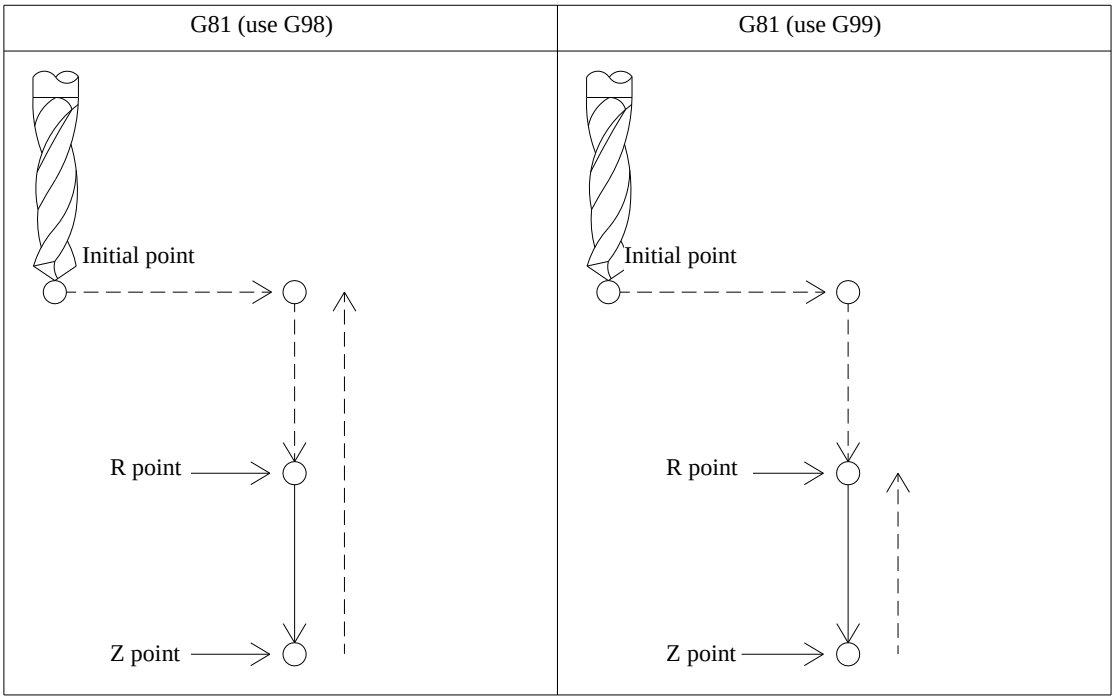
Move to the horizontal position of R point rapidly after positioning along the X-axis and the Y-axis. Drill from the horizontal position of R point to Z point, and then retract the tool through rapid movement.

Rotate the spindle with the M code of auxiliary functions before designating G81.

If tool length compensation is specified in such fixed cycle, such offset shall be maintained during positioning at the R point.

The fixed cycle can be cancelled through G80/G00/G01/G02/G03.

The tool radius offset in fixed cycle is ignored.



Program examples

M3 S2000;	Spindle starts rotating
G90 G99 G81 X30. Y-25. Z-150. R-120. F120;	Position; bore Hole 1 and then return to R point
Y-55.;	Position; drill Hole 2 and then return to R point
Y-75.;	Position; drill Hole 3 and then return to R point
X100.;	Position; drill Hole 4 and then return to R point
Y-55.;	Position; drill Hole 5 and then return to R point
G98 Y-75.;	G98 Y75.; Positioning; drilling Hole 6 and then return to the initial plane
G80;	
G28 G91 X0 Y0 Z0 ;	Return to the reference point
M5;	Spindle stops
M30;	

Precautions

(1) Please cancel the fixed cycle before switching the drilling axis, or otherwise an alarm will occur.
(2) Drilling shall not be performed in case of no commands for X, Y, Z, R or any other axis in the program block.

9.7 Drilling cycle and boring cycle (G82)

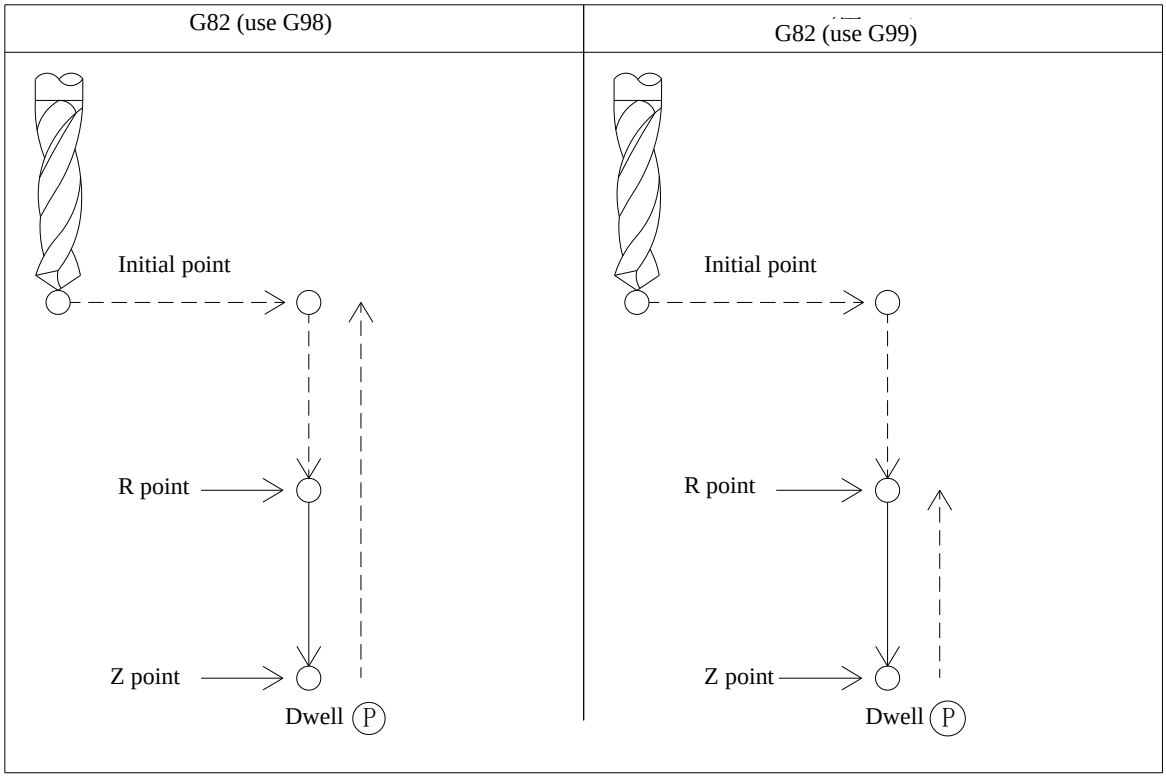
Function and purpose

It is used for drilling operations, performing cutting feeds to the bottom of the hole, holding, and then retracting the tool quickly from the bottom of the hole. Such cycle can improve the precision of hole depth.

Command format

G82 X_ Y_ Z_ R_ P_ F_ K_;	
X_ Y_	Positioning data of the hole
Z_	Increment programming indicates the distance between the specified R point and the hole bottom; absolute programming indicates the absolute coordinates at the hole bottom
R_	Increment programming indicates the distance between the plane at the initial point and the R point; absolute programming indicates the absolute coordinates at R point
P_	Dwell time
F_	Cutting feed speed
K_	Number of repeats

Detailed description



After positioning along the X-axis and the Y-axis, move to R point rapidly, and drill from R point to Z point. Such drilling shall be dwelled while reaching the hole bottom, and then the tool will retract rapidly through rapid movement.

Rotate the spindle through the M code of auxiliary functions before specifying G82.

If tool length compensation is specified in such fixed cycle, such offset shall be maintained during positioning at the R point.

The fixed cycle can be cancelled through G80/G00/G01/G02/G03.

The tool radius offset in fixed cycle is ignored.

Program examples

M3 S2000;	Spindle starts rotating
G90 G99 G82 X30. Y-25. Z-150. R-120. P1000 F120;	Position, bore Hole 1, hold for 1 second at the hole bottom, then return to R point
Y-55;	Position, bore Hole 2, hold for 1 second at the hole bottom, then return to R point
Y-75;	Position, bore Hole 3, hold for 1 second at the hole bottom, then returning to R point
X100.;	Position, bore Hole 4, hold for 1 second at the hole bottom, then returning to R point
Y-55;	Position, bore Hole 5, hold for 1 second at the hole bottom, then returning to R point
G98 Y-75;	Positioning, boring Hole 6, holding for 1 second at the hole bottom, then return to the initial plane
G80;	Fixed cycle cancelled
G28 G91 X0 Y0 Z0 ;	Return to the reference point
M5;	Spindle stops
M30;	

Precautions

(1) Please cancel the fixed cycle before switching the drilling axis, or otherwise an alarm will occur.

(2) Drilling shall not be performed in case of no commands for X, Y, Z, R or any other axis in the program block.

(3) Specify P in the program block performing drilling. If being specified in the program block not performing drilling, it cannot be saved as modal data.

9.8 Deep hole chip-removal drilling cycle (G83)

Function and purpose

It is used for deep hole drilling operations, performing the intermittent cutting feeds to the bottom of the hole, and removing metal chips from the hole during drilling.

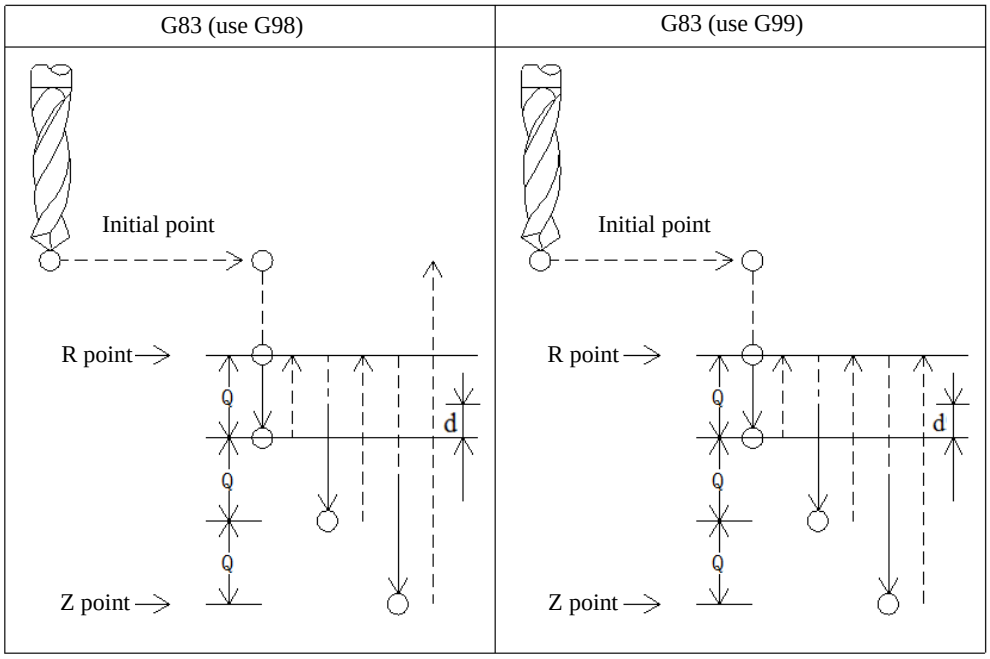
Command format

G83 X_ Y_ Z_ R_ Q_ F_ K_;

X_ Y_	Positioning data of the hole

Z_	Increment programming indicates the distance between the specified R point and the hole bottom; absolute programming indicates the absolute coordinates at the hole bottom
R_	Increment programming indicates the distance between the plane at the initial point and the R point; absolute programming indicates the absolute coordinates at R point
Q_	Cutting depth of each cutting feed
F_	Cutting feed speed
K_	Number of repeats

Detailed description



Q indicates the cutting depth of cutting feed each time and must be expressed with increment. When the first feed arrives at R point rapidly, cut towards the Q distance and return to R point rapidly. In the second and follow-up cutting feed, perform rapid movement towards the point at a distance of d from the last drilling and then re-perform cutting feed d+Q and then return to R point rapidly. Value d can be set through parameter #2011.

A positive value must be specified in Q and the negative sign is ignored and the system is still processing in a positive value.

Before specifying G83, rotate the spindle with the M code of auxiliary functions.

If tool length compensation is specified in such fixed cycle, such offset shall be maintained during positioning at the R point.

The fixed cycle can be cancelled through G80/G00/G01/G02/G03.

The tool radius offset in fixed cycle is ignored.

Program examples

M3 S2000;	Spindle starts rotating
G90 G99 G83 X30. Y-25. Z-150. R-120. Q15 F120;	Position; bore Hole 1 and then return to R point
Y-55;	Position; drill Hole 2 and then return to R point
Y-75;	Position; drill Hole 3 and then return to R point

X100;	Position; drill Hole 4 and then return to R point
Y-55;	Position; drill Hole 5 and then return to R point
G98 Y-75;	G98 Y75.; Positioning; drilling Hole 6 and then return to the initial plane
G80;	
G28 G91 X0 Y0 Z0 ;	Return to the reference point
M5;	Spindle stops
M30;	

Precautions

(1) Please cancel the fixed cycle before switching the drilling axis, or otherwise an alarm will occur.

(2) Drilling shall not be performed in case of no commands for X, Y, Z, R or any other axis in the program block.

(3) Q is specified in the program block performing drilling. If being specified in a program block not performing drilling, it cannot be saved as modal data.

9.9 Right-hand tapping cycle (G84)

Function and purpose

If right-hand tapping is performed, the spindle will rotate counterclockwise while reaching the hole bottom.

Command format

```
G84 X_ Y_ Z_ R_ P_ F_ K_;
```

X_ Y_	Positioning data of the hole
Z_	Increment programming indicates the distance between the specified R point and the hole bottom; absolute programming indicates the absolute coordinates at the hole bottom
R_	Increment programming indicates the distance between the plane at the initial point and the R point; absolute programming indicates the absolute coordinates at R point
P_	Dwell time
F_	Cutting feed speed
K_	Number of repeats

Detailed description

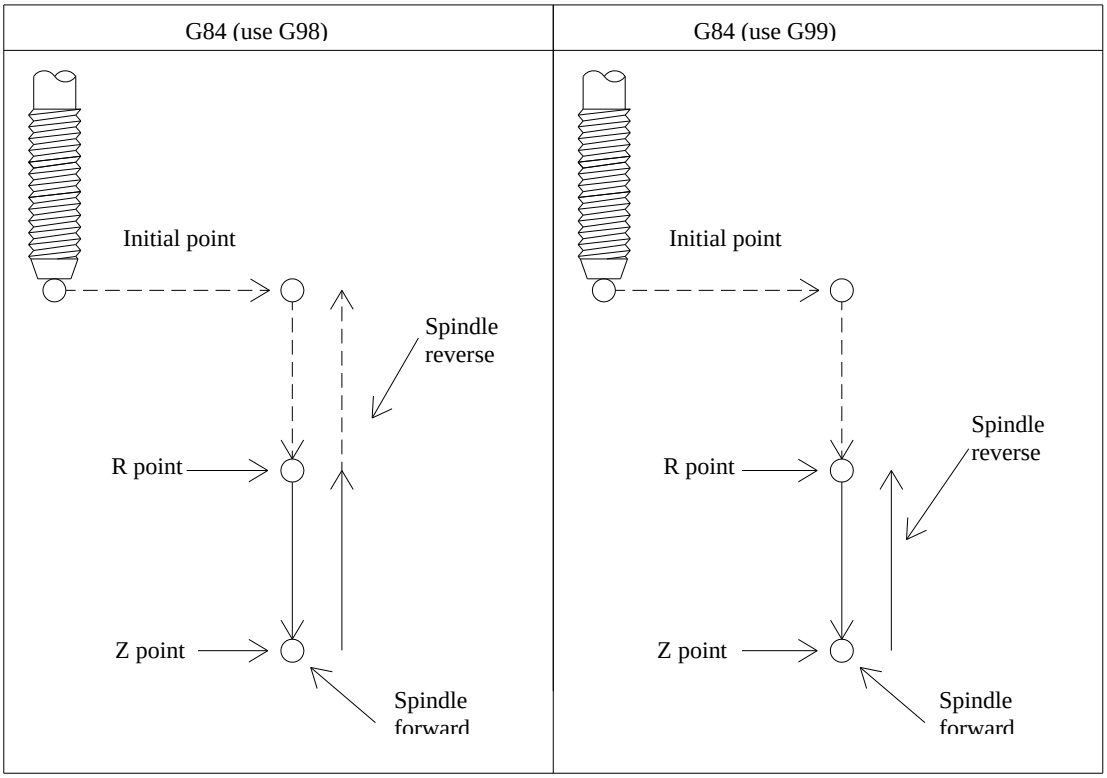
Tapping is performed when the spindle is rotated clockwise, and rotate in the opposite direction to retract the spindle while reaching the hole bottom. The feed override is ignored during tapping, and feed hold will not stop the machine, until the return action is completed.

Before specifying G84, rotate the spindle through the M code of auxiliary functions. If clockwise rotation of the spindle is not designated, the system will be adjusted as clockwise rotation on the R plane automatically as per the current command speed of the spindle.

If tool length compensation is specified in such fixed cycle, such offset shall be maintained during positioning at the R point.

The fixed cycle can be cancelled through G80/G00/G01/G02/G03.

The tool radius offset in fixed cycle is ignored.



Program examples

M3 S100;	Spindle starts rotating
G90 G99 G84 X30. Y-25. Z-150. R-120 P3000F120;	Position; tap Hole 1 and then returning to R point
Y-55.;	Position; tap Hole 2 and then returning to R point
Y-75.;	Position; tap Hole 3 and then returning to R point
X100.;	Position; tap Hole 4 and then returning to R point
Y-55.;	Position; tap Hole 5 and then returning to R point
G98 Y-75.;	Positioning; tapping Hole 6 and then return to the initial plane
G80;	
G28 G91 X0 Y0 Z0 ;	Return to the reference point
M5;	Spindle stops
M30;	

Precautions

(1) Please cancel the fixed cycle before switching the drilling axis, or otherwise an alarm will occur.

(2) Drilling shall not be performed in case of no commands for X, Y, Z, R or any other axis in the program block.

(3) Specify P in the program block performing drilling. If being specified in the program block not

performing drilling, it cannot be saved as modal data.

9.10 Boring cycle (G85)

Function and purpose

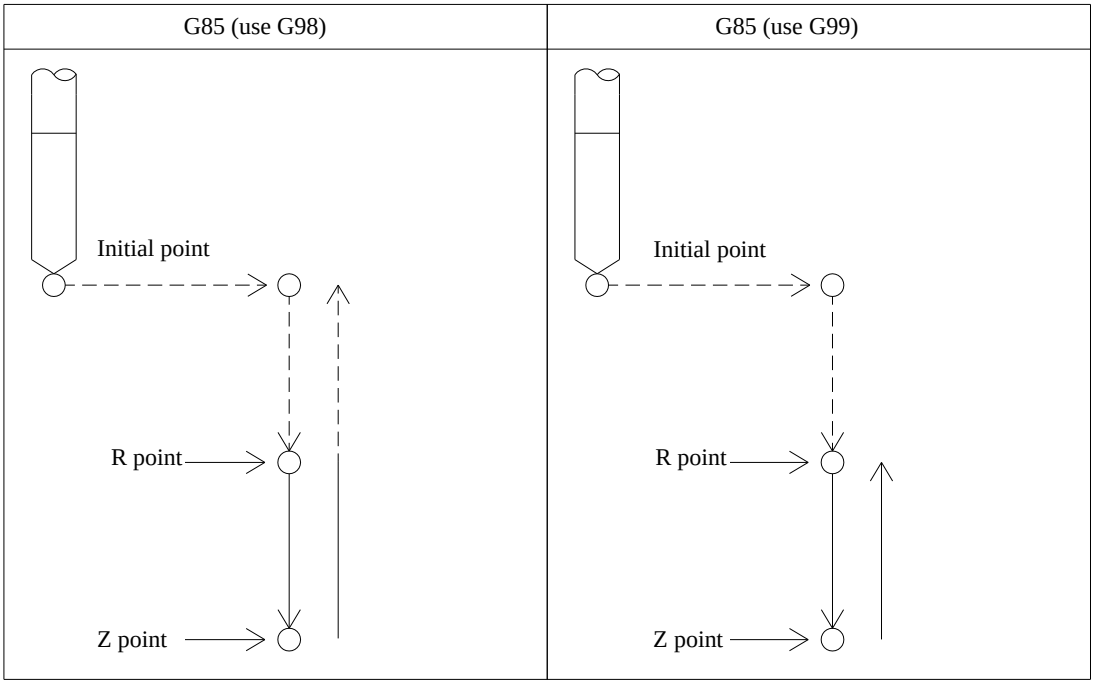
Used for boring. Applicable to the machine with or without positioning function of spindle.

Command format

```
G85 X_ Y_ Z_ R_ F_ K_;
```

X_ Y_	Positioning data of the hole
Z_	Increment programming indicates the distance between the specified R point and the hole bottom; absolute programming indicates the absolute coordinates at the hole bottom
R_	Increment programming indicates the distance between the plane at the initial point and the R point; absolute programming indicates the absolute coordinates at R point
F_	Cutting feed speed
K_	Number of repeats

Detailed description



After positioning along the X- and Y-axis, move quickly to R point, and then perform boring from R point to Z point. When reaching the bottom of the hole, perform cutting feed and return to R point.

Before specifying G85, rotate the spindle with the M code of auxiliary functions.

If tool length compensation is specified in such fixed cycle, such offset shall be maintained during positioning at the R point.

The fixed cycle can be cancelled through G80/G00/G01/G02/G03.

The tool radius offset in fixed cycle is ignored.

Program examples

```
M3 S100;                                Spindle starts rotating
G90 G99 G85 X30. Y-25. Z-150. R-120. F120;  Position; bore Hole 1 and then return to R point
Y-55.;                                  Position; bore Hole 2 and then return to R point
Y-75.;                                  Position; bore Hole 3 and then return to R point
X100.;                                  Position; bore Hole 4 and then return to R point
Y-55.;                                  Position; bore Hole 5 and then return to R point
G98 Y-75.;                              Position; bore Hole 6 and then return to the initial plane
G80;
G28 G91 X0 Y0 Z0 ;                      Return to the reference point
M5;                                     Spindle stops
M30;
```

Precautions

(1) Please cancel the fixed cycle before switching the drilling axis, or otherwise an alarm will occur.

(2) Drilling shall not be performed in case of no commands for X, Y, Z, R or any other axis in the program block.

9.11 Boring cycle (G86)

Function and purpose

Used for boring. Applicable to the machine with or without positioning function of spindle.

Command format

G86 X_ Y_ Z_ R_ F_ K_;

X_ Y_	Positioning data of the hole
Z_	Increment programming indicates the distance between the specified R point and the hole bottom; absolute programming indicates the absolute coordinates at the hole bottom
R_	Increment programming indicates the distance between the plane at the initial point and the R point; absolute programming indicates the absolute coordinates at R point
F_	Cutting feed speed
K_	Number of repeats

Detailed description

After positioning along the X-axis and the Y-axis, move to R point rapidly, and bore from R point to Z point. The tool will retract through rapid movement after the spindle stops rotating at the hole bottom.

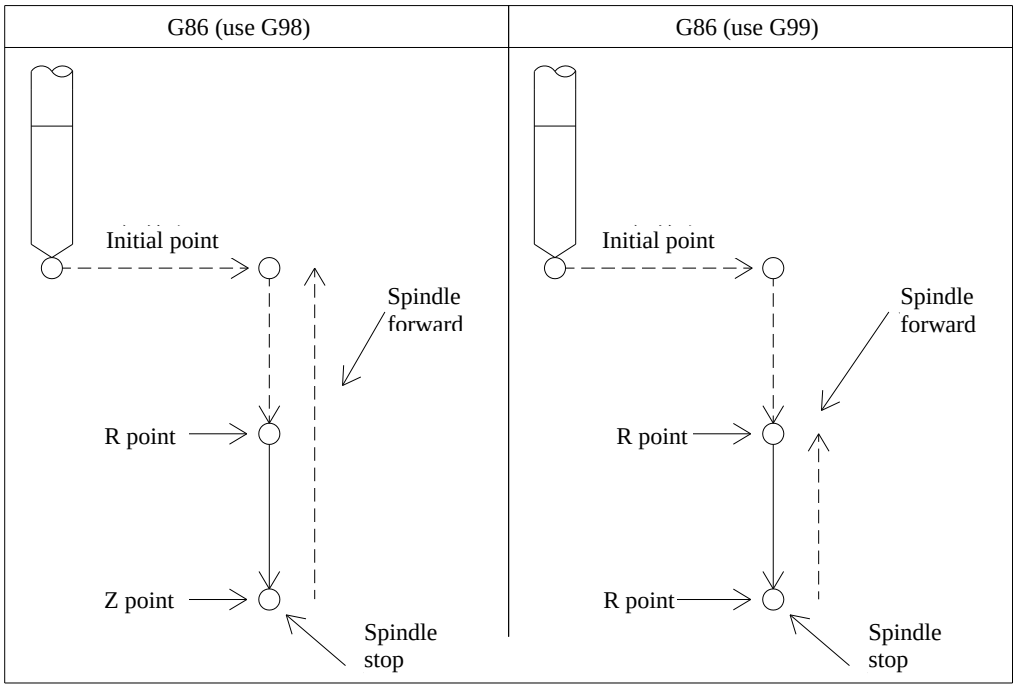
Before specifying G86, rotate the spindle with the M code of auxiliary functions.

If tool length compensation is specified in such fixed cycle, such offset shall be maintained during

positioning at the R point.

The fixed cycle can be cancelled through G80/G00/G01/G02/G03.

The tool radius offset in fixed cycle is ignored.



Program examples

```
M3 S2000;                                Spindle starts rotating
G90 G99 G86 X30. Y-25. Z-150. R-120. F120;  Position; bore Hole 1 and then return to R point
Y-55.;                                    Position; bore Hole 2 and then return to R point
Y-75.;                                    Position; bore Hole 3 and then return to R point
X100.;                                   Position; bore Hole 4 and then return to R point
Y-55.;                                    Position; bore Hole 5 and then return to R point
G98 Y-75.;                                Position; bore Hole 6 and then return to the initial plane
G80;
G28 G91 X0 Y0 Z0 ;                       Return to the reference point
M5;                                       Spindle stops
M30;
```

Precautions

(1) Please cancel the fixed cycle before switching the drilling axis, or otherwise an alarm will occur.

(2) Drilling shall not be performed in case of no commands for X, Y, Z, R or any other axis in the program block.

9.12 Back boring cycle (G87)

Function and purpose

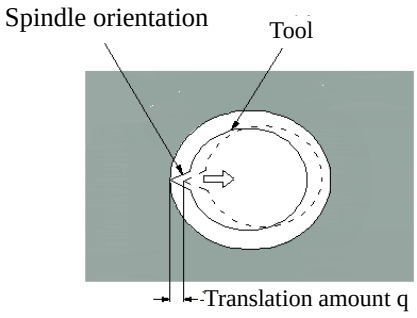
Used for high-precision boring. Applicable to the machine with positioning function of spindle.

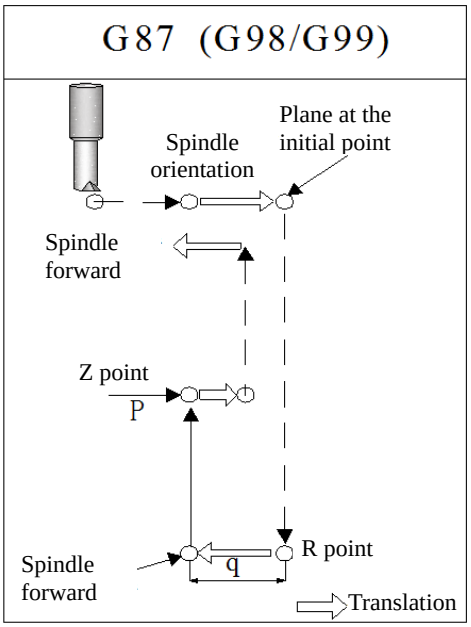
Command format

G87 X_ Y_ Z_ R_ { Q_ / I_ J_ } P_ F_ K_;

X_ Y_	Positioning data of the hole
Z_	Increment programming indicates the distance between the specified R point and the hole bottom; absolute programming indicates the absolute coordinates at the hole bottom
R_	Increment programming indicates the distance between the plane at the initial point and the R point; absolute programming indicates the absolute coordinates at R point
Q_	Offset at the hole bottom; the offset direction set by the parameter shall be used
I_J_	Offset at the hole bottom; offset as per the given value
P_	Dwell time
F_	Cutting feed speed
K_	Number of repeats

Detailed description





After the X- and Y-axis are positioned, the spindle stops exactly, the tool is offset in the opposite direction of the tool nose and quickly positioned at the bottom of the hole (R point). The tool is offset in the direction of the tool nose and returns, and the spindle rotates clockwise, and is processed along the Z-axis to the Z point in the positive direction. The spindle stops accurately again, and then offsets and exits in the opposite direction of the tool nose, and after returning to the initial horizontal plane quickly, the tool returns to the initial point along the direction of the tool nose, the spindle will rotate in a forward manner and shift to the next program block. The offset and direction of the X-axis/Y is completely identical to those of G76 (the direction set is universal with G76). The offset of Q at the bottom of the hole is a modal value stored within the fixed cycle and must be

specified carefully. Because it is also used as the cutting depth for G73 and G83.

The offset shall be specified as per the following modes if I/J is used.

G17: I represents offset in the X direction, and J represents offset in the Z-direction

G18: I represents offset in the X direction, and J represents offset in the Z-direction

G19: I represents offset in the Y direction, and J represents offset in the Z-direction

Before specifying G87, rotate the spindle with the M code of auxiliary functions.

If tool length compensation is specified in such fixed cycle, such offset shall be maintained during positioning at the R point.

The fixed cycle can be cancelled through G80/G00/G01/G02/G03.

The tool radius offset in fixed cycle is ignored.

Program examples

M3 S500;	Spindle starts rotating
G90 G99 G87 X30. Y-25. Z-120. R-150. Q5. P1000 F120;	Positioning; boring Hole 1; offset for 5mm after positioning at the initial position 5 mm stop for 1s at Z point
Y-55;	Position; bore Hole 2 and then return to R point
Y-75;	Position; bore Hole 3 and then return to R point
X100.;	Position; bore Hole 4 and then return to R point
Y-55.;	Position; bore Hole 5 and then return to R point
G98 Y-75;	Position; bore Hole 6 and then return to the initial plane
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference point
M5;	Spindle stops
M30;	

Precautions

(1) Please cancel the fixed cycle before switching the drilling axis, or otherwise an alarm will occur.

(2) Drilling shall not be performed in case of no commands for X, Y, Z, R or any other axis in the program block.

(3) Specify P and Q in the program block performing drilling. If being specified in the program block not performing drilling, it cannot be saved as modal data.

9.13 Boring cycle (G88)

Function and purpose

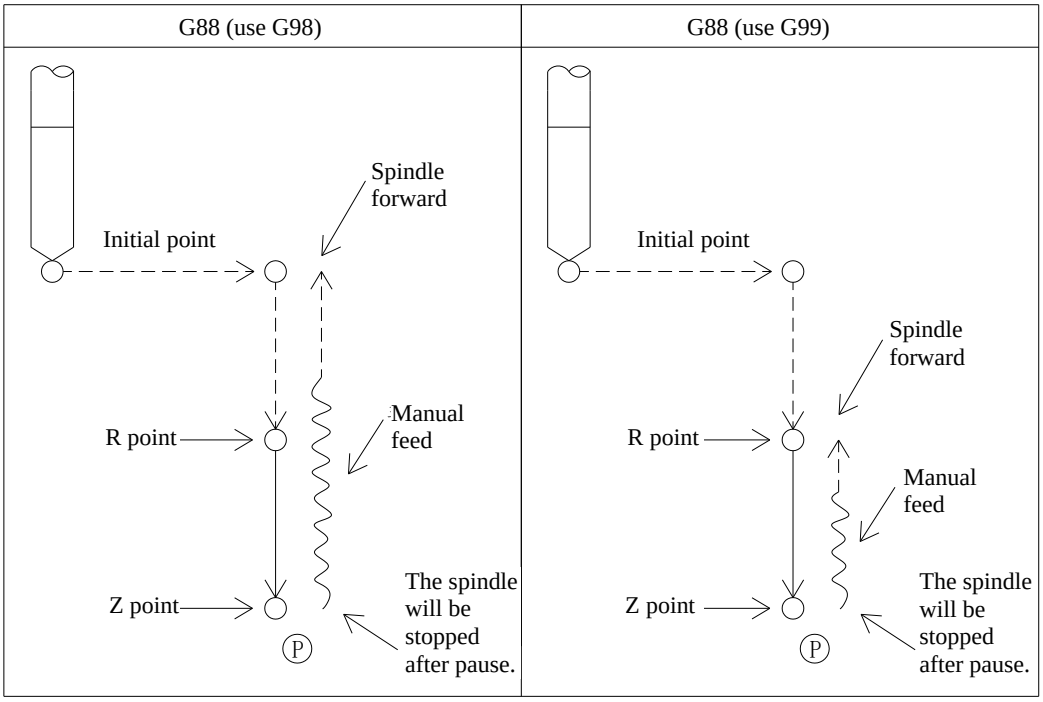
Used for boring. Applicable to the machine with or without positioning function of spindle.

Command format

G88 X_ Y_ Z_ R_ P_ F_ K_;

X_ Y_	Positioning data of the hole
Z_	Increment programming indicates the distance between the specified R point and the hole bottom; absolute programming indicates the absolute coordinates at the hole bottom
R_	Increment programming indicates the distance between the plane at the initial point and the R point; absolute programming indicates the absolute coordinates at R point
P_	Dwell time
F_	Cutting feed speed
K_	Number of repeats

Detailed description



After positioning along the X- and Y-axis, move to R point rapidly, and bore from R point to Z point. Pause at the hole bottom after boring, and the spindle will decelerate and stop rotating when such suspension starts; enter the state of feed hold, and turn the indicator of feed hold ON. In this case, switch to manual mode and retract the tool manually from the hole bottom to the horizontal position of R point. (It is safer if operating after the tool is removed). Press down "Cycle Start" and start performing the next program block after such returning. There is also the possibility that, when cycle start is directly pressed instead of using the manual return while cutting to the hole bottom, the tool automatically returns to the initial plane or the R plane.

Before designating G88, rotate the spindle with the M code of auxiliary functions. If tool length compensation is specified in such fixed cycle, such offset shall be maintained during positioning at the R point.

The fixed cycle can be cancelled through G80/G00/G01/G02/G03.

The tool radius offset in fixed cycle is ignored.

Program examples

M3 S2000;	Spindle starts rotating
G90 G99 G88 X30. Y-25. Z-150. R-120. P1000 F120;	Position; bore Hole 1 and then return to R point
Y-55.;	Position; bore Hole 2 and then return to R point
Y-75.;	Position; bore Hole 3 and then return to R point
X100.;	Position; bore Hole 4 and then return to R point
Y-55.;	Position; bore Hole 5 and then return to R point
G98 Y-75.;	Position; bore Hole 6 and then return to the initial plane
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference point
M5;	Spindle stops

Precautions

(1) The system performs the manual return operation after the first hole bottom hold time; Z-axis returns to the initial plane after clicking the [Cycle Start]; the action of next boring cycle is executed after clicking the [Cycle Start] once, and the boring cycle is executed according to the K value.

(2) During G88 hole circular machining, if the mode is not switched to manual, no action can be executed after clicking the cycle start (e.g., switched to MDI mode); and in general, operation shall be executed in strict accordance with G88. Otherwise, the tool may be damaged.

9.14 Boring cycle (G89)

Function and purpose

Used for boring. Applicable to the machine with or without positioning function of spindle.

Command format

```
G89 X_ Y_ Z_ R_ P_ F_ K_;
```

X_ Y_	Positioning data of the hole
Z_	Increment programming indicates the distance between the specified R point and the hole bottom; absolute programming indicates the absolute coordinates at the hole bottom
R_	Increment programming indicates the distance between the plane at the initial point and the R point; absolute programming indicates the absolute coordinates at R point
P_	Dwell time
F_	Cutting feed speed
K_	Number of repeats

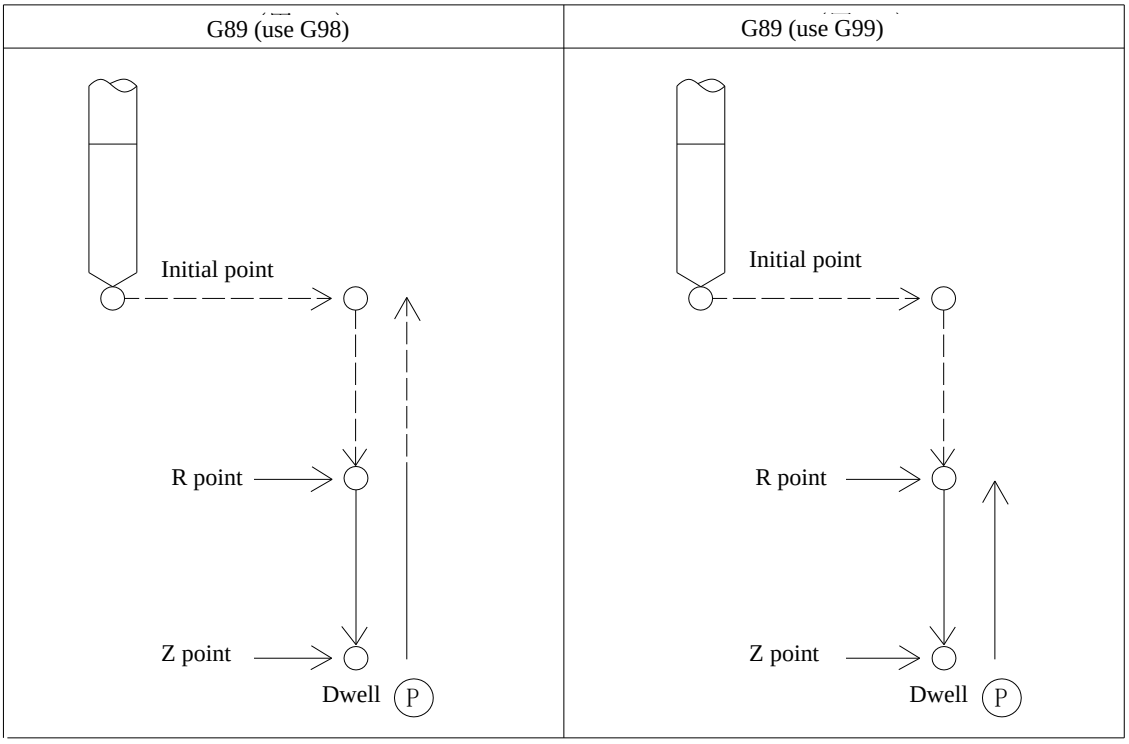
Detailed description

This cycle is almost the same as G85, but it performs the hold at the hole bottom.

Before specifying G89, the spindle rotates with auxiliary function code M.

If tool length compensation is specified in such fixed cycle, such offset shall be maintained during positioning at the R point.

The fixed cycle can be cancelled through G80/G00/G01/G02/G03.
The tool radius offset in fixed cycle is ignored.



Program examples

M3 S100;	Spindle starts rotating
G90 G99 G89 X30 Y-25 Z-150 R-120 P1000 F120;	Position, bore Hole 1, holing for 1 second at hole bottom, then return to R point
Y-55.;	Position; bore Hole 2 and then return to R point
Y-75.;	Position; bore Hole 3 and then return to R point
X100.;	Position; bore Hole 4 and then return to R point
Y-55.;	Position; bore Hole 5 and then return to R point
G98 Y-75.;	Position; bore Hole 6 and then return to the initial plane
G80;	
G28 G91 X0 Y0 Z0 ;	Return to the reference point
M5;	Spindle stops

Precautions

(1) Please cancel the fixed cycle before switching the drilling axis, or otherwise an alarm will occur.
(2) Drilling shall not be performed in case of no commands for X, Y, Z, R or any other axis in the program block.
(3) Specify P in the program block performing drilling. If being specified in the program block not performing drilling, it cannot be saved as modal data.

9.15 Left-hand rigid tapping Cycle (G74)

Function and purpose

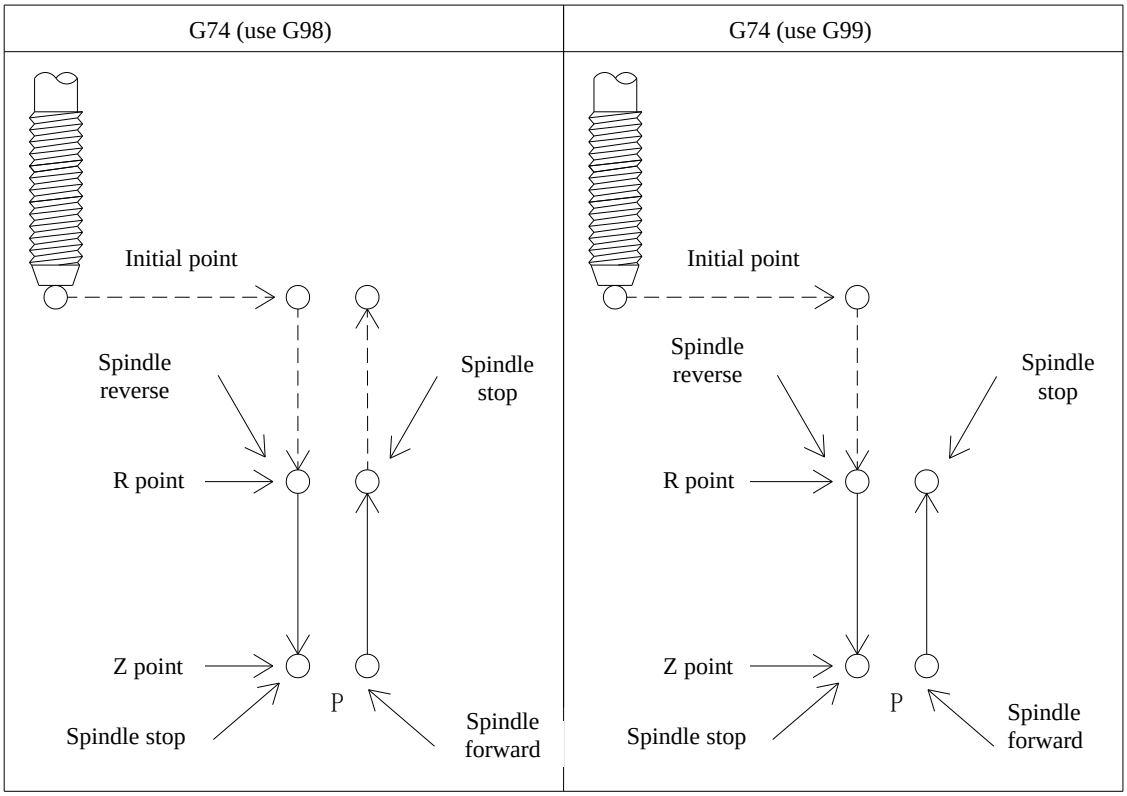
In the rigid mode, the working spindle motor is a servo motor, which is able to realize the high-speed high-precision tapping.

Command format

G74 X_ Y_ Z_ R_ P_ F_ K_;

X_ Y_	Positioning data of the hole
Z_	Increment programming indicates the distance between the specified R point and the hole bottom; absolute programming indicates the absolute coordinates at the hole bottom
R_	Increment programming indicates the distance between the plane at the initial point and the R point; absolute programming indicates the absolute coordinates at R point
P_	Dwell time
F_	Cutting feed speed
K_	Number of repeats

Detailed description



After positioning along the X- and Y-axis, the Z-axis moves rapidly to the R point, executes G74, the spindle starts to reverse, and performs tapping from the R point to the Z point. When the tapping is completed, the spindle stops and performs a pause, and then the spindle rotates in the opposite direction. The rotary tool retracts to R

point, the spindle stops, and then executes a rapid traverse to the initial position. In the tapping progress the feed speed override and the spindle override are considered to be 100%.

If tool length compensation is specified in such fixed cycle, such offset shall be maintained during positioning at the R point.

The lead of screw thread is expressed as: feed speed ÷spindle speed.

Z-axis feed speed F = spindle speed ×lead of screw thread.

The fixed cycle can be cancelled through G80/G00/G01/G02/G03.

The tool radius offset in fixed cycle is ignored.

Rigid mode

The rigid mode can be specified by any of the following methods:

(1) Specify M29 S***** before tapping command block

(2) Specify M29 S***** in the program block with tapping command

Program examples

Spindle speed 1000 rpm

Lead of screw thread 1.0 mm

Then Z-axis feed speed F = 1000 ×1 = 1000 mm/min

```
G00 X120 Y100;              Positioning
M29 S1000                   Specify the rigid mode
G74 Z-100 R-20 F1000;    .Rigid tapping
```

Precautions

(1) Please cancel the fixed cycle before switching the drilling axis, or otherwise an alarm will occur.

(2) Drilling shall not be performed in case of no commands for X, Y, Z, R or any other axis in the program block.

(3) Specify P in the program block performing drilling. If being specified in the program block not performing drilling, it cannot be saved as modal data.

F: When the F value exceeds the upper limit of cutting feed speed, the operation will be performed in accordance with the upper limit of feed speed.

S: When the speed is higher than the maximum speed of specified level, alarm will be given. The speed levels are configured by parameters #2140~#2142.

(4) Program restart: The program restart is invalid during rigid tapping.

9.16 Right-hand rigid tapping cycle (G84)

Function and purpose

In rigid mode the control of spindle motor is a servo motor, which is able to realize the high-speed high-precision tapping.

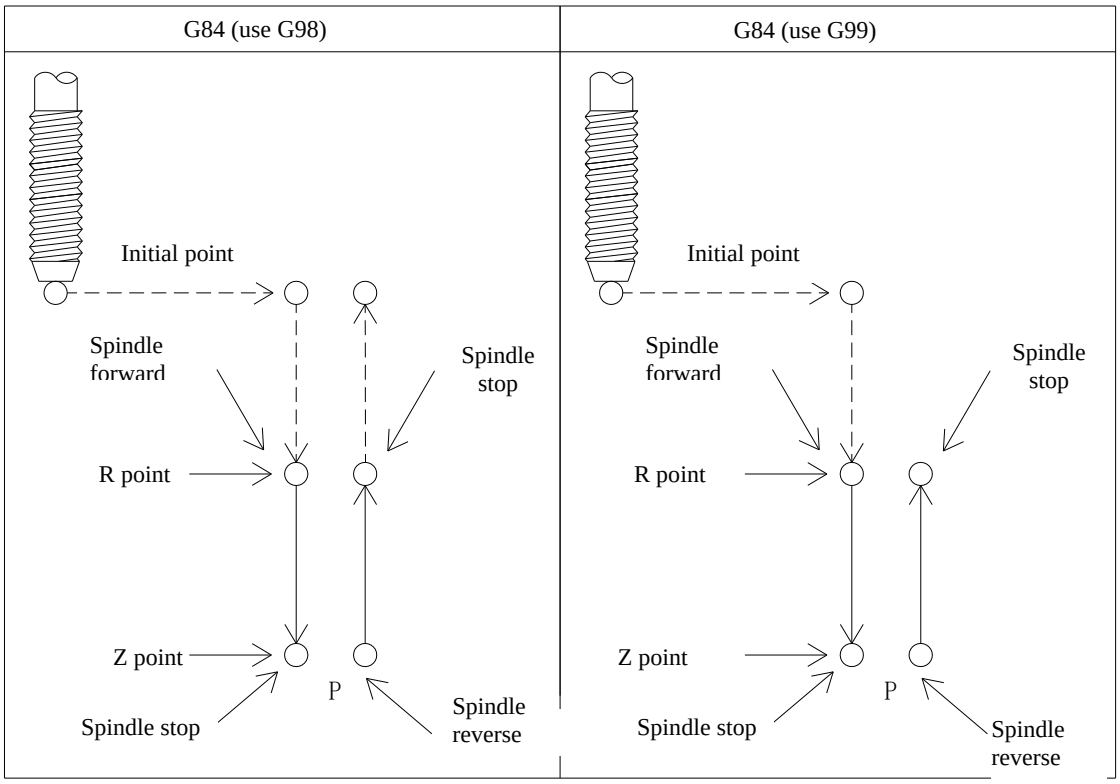
Command format

G84 X_ Y_ Z_ R_ P_ F_ K_;

X_ Y_	Positioning data of the hole
Z_	Increment programming indicates the distance between the specified R point and the

	hole bottom; absolute programming indicates the absolute coordinates at the hole bottom
R_	Increment programming indicates the distance between the plane at the initial point and the R point; absolute programming indicates the absolute coordinates at R point
P_	Dwell time
F_	Cutting feed speed
K_	Number of repeats

Detailed description



After positioning along the X- and Y-axis, the Z-axis executes rapid traverse to R point, executes G84 and the spindle starts to rotate forward, and performs tapping from R point to Z point. When tapping is completed, the spindle stops and executes a pause, and then the spindle rotates in the opposite direction. Rotate the tool in the direction, return to R point, and stop the spindle. Then it moves rapidly to the initial position.

In the tapping progress the feed speed override and the spindle override are considered to be 100%.

If tool length compensation is specified in such fixed cycle, such offset shall be maintained during positioning at the R point.

In feed per minute mode, the lead of screw thread is expressed as: feed speed ÷spindle speed.

Z-axis feed speed = spindle speed ×lead of screw thread.

The fixed cycle can be cancelled through G80/G00/G01/G02/G03.

The tool radius offset in fixed cycle is ignored.

Rigid mode:

The rigid mode can be specified by any of the following methods:

(1) Specify M29 S***** before tapping command block

(2) Specify M29 S***** in the program block with tapping command

Program examples

Spindle speed 1000 rpm
Lead of screw thread 1.0 mm
Then Z-axis feed speed F = 1000 × 1 = 1000 mm/min
G00 X120 Y100; Positioning
M29 S1000 Specify the rigid mode
G84 Z-100 R-20 F1000; .Rigid tapping

Precautions

(1) Please cancel the fixed cycle before switching the drilling axis, or otherwise an alarm will occur.
(2) Drilling shall not be performed in case of no commands for X, Y, Z, R or any other axis in the program block.
(3) Specify P in the program block performing drilling. If being specified in the program block not performing drilling, it cannot be saved as modal data.
F: When the F value exceeds the upper limit of cutting feed speed, the operation will be performed in accordance with the upper limit of feed speed.
S: When the speed is higher than the maximum speed of specified level, alarm will be given. The speed levels are configured by parameters #2140~#2142.
(4) Program restart: The program restart is invalid during rigid tapping.

9.17 Chip removing rigid tapping cycle (G84, G74)

Function and purpose

In this cycle, several infeeds are performed until the bottom of the hole, two of which are selected by parameter PCP #2000.7.

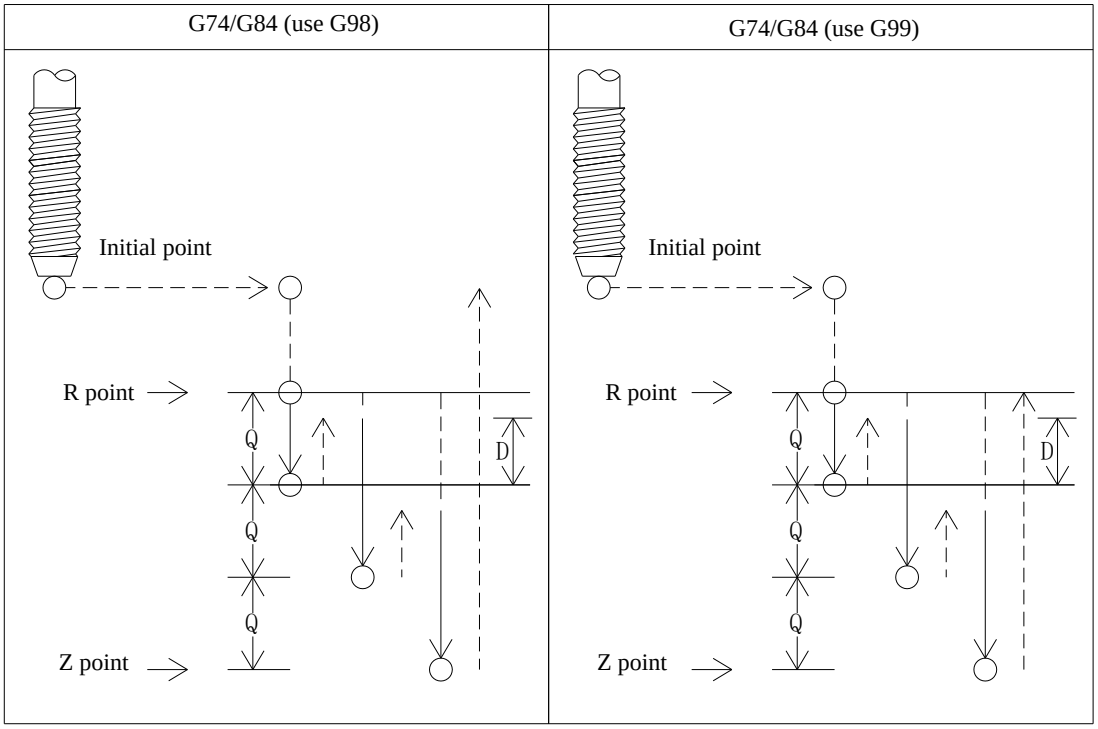
Command format

G84/G74 X_ Y_ Z_ R_ P_ F_ K_;

X_ Y_	Positioning data of the hole
Z_	Increment programming indicates the distance between the specified R point and the hole bottom; absolute programming indicates the absolute coordinates at the hole bottom
R_	Increment programming indicates the distance between the plane at the initial point and the R point; absolute programming indicates the absolute coordinates at R point
P_	Dwell time
Q_	Cutting depth of each cutting feed
F_	Cutting feed speed
K_	Number of repeats

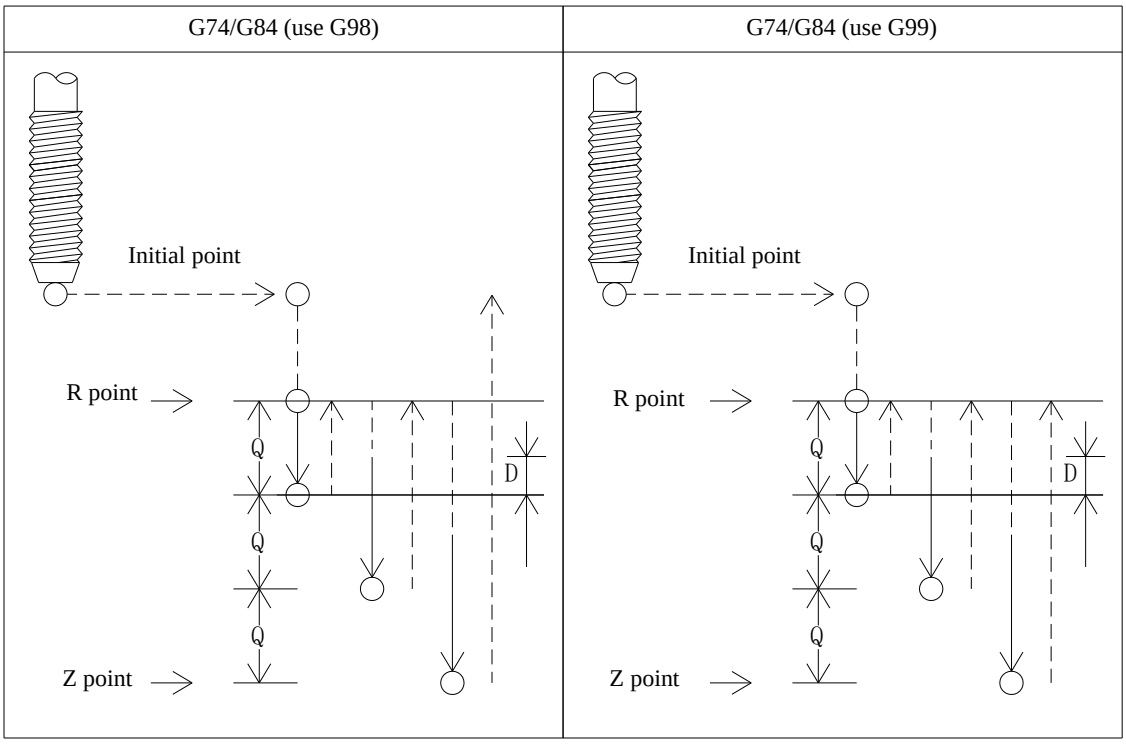
Detailed description

High-speed deep-hole tapping cycle (parameter PCP #2000.7 = 0)



After positioning along X-axis and Y-axis, Z-axis moves rapidly to R point to perform the cutting with the cut depth Q from R point; then the tool return by a distance d; the spindle stops when reaching Z point at the hole bottom; then it rotates in opposite direction and retracts.

Deep-hole tapping cycle (parameter PCP #2000.7 = 1, non-high-speed tapping)



After positioning along the X- and Y-axis, the Z-axis performs rapid traverse to R point, and from R point, the cutting is performed with the infeed depth Q, and then the tool returns to R point. The position from R point to

the last cutting end point is d. At the starting position, when reaching the bottom of the hole at Z point, the spindle stops, and then rotates back in the opposite direction.

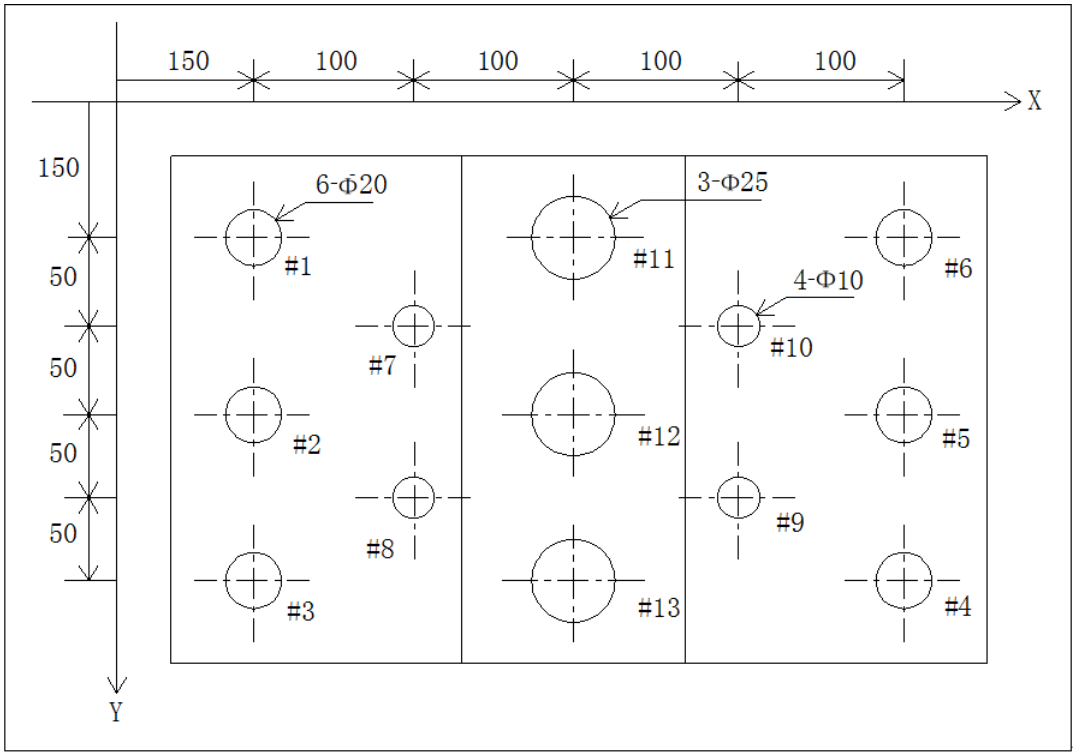
In the tapping progress the feed speed override and the spindle override are considered to be 100%.

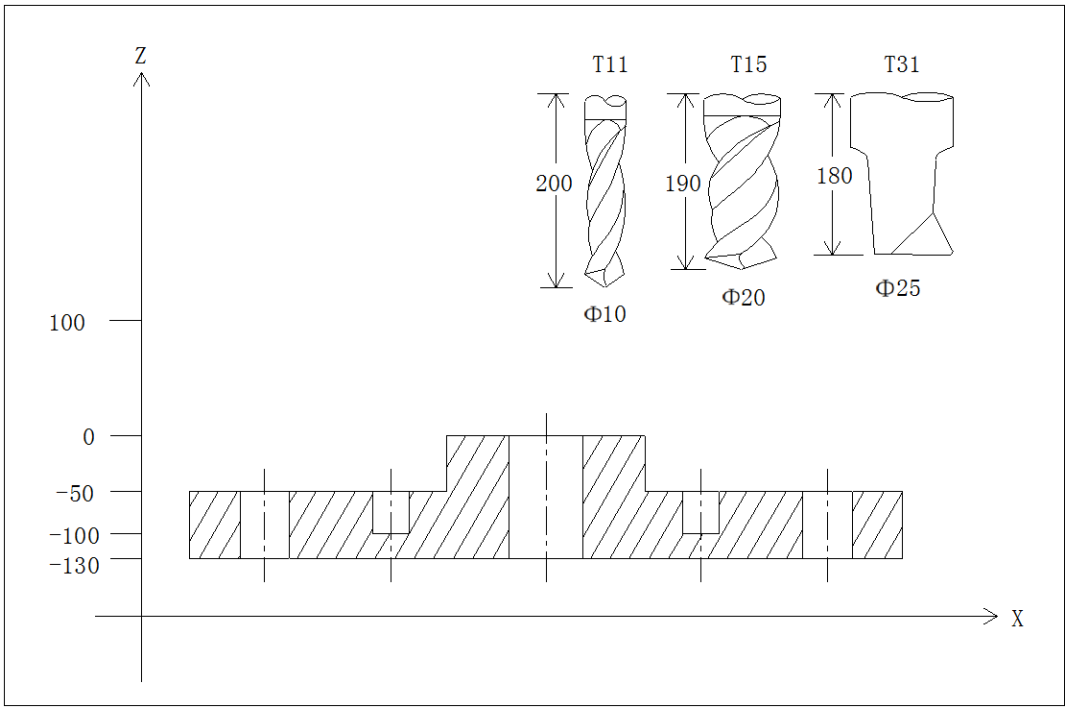
The fixed cycle can be cancelled through G80/G00/G01/G02/G03.

The tool radius offset in fixed cycle is ignored.

Program examples

The application of fixed cycle is comprehensively explained below by using the tool length compensation.





Properly set workpiece coordinate system and tool length offset. Material: Aluminum part, machining

procedure as below:

N001 G90 G54 G00 G17 G40 G80 G49 ;	Initialize the program.
N002 M06 T15 ;	Change the tool.
N003 G43 H15 Z100 ;	Enable H15 tool length compensation.
N004 S5500 M3 ;	Start the spindle.
N005 G98 G83 X150 Y-150 Z-140 R-45 Q8 F300 ;	Process hole #1 after positioning.
N006 G91 Y-100 K2 ;	Process holes #2 and #3 after positioning, return to the initial point plane.
N007 G90 X550 ;	Process hole #4 after positioning, return to the initial point plane.
N008 G91 Y100 K2 ;	Process holes #5 and #6 after positioning, return to the initial point plane.
N011 G90 G49 Z100 M5 ;	Cancel the tool length compensation, stop the spindle.
N012 M06 T11 ;	Change the tool.
N013 G43 H11 Z100 ;	Enable H11 tool length compensation.
N014 S800 M3 ;	Start the spindle.
N015 G98 G83 X250 Y-200 Z-140 R-45 Q5 F150 ;	Process hole #7 after positioning, return to the initial point plane.
N016 G91 Y-100 ;	Process hole #8 after positioning, return to the initial point plane.
N017X200 ;	Process hole #9 after positioning, return to the initial point plane.
N018 Y100 ;	Process hole #10 after positioning, return to the initial point plane.
N017 G90 X350 Y-150 ;	Rough hole #11 after positioning, return to the initial point plane.
N018 G91 Y-100 K2 ;	Rough holes #12 and #13 after positioning, return to the initial point plane.
N011 G90 G49 Z100 M5 ;	Cancel the tool length compensation, stop the spindle.
N020 M06 T31 ;	Cancel the tool length compensation, change the tool.
N021 G43 H31 Z100 ;	Enable H31 tool length compensation.
N022 S600 M3 ;	Start the spindle.
N023 G81 G99 X350 Y-150 Z-140 R5 F100 ;	Process #11 after positioning, return to R point plane.
N024 G91 Y-100 K2 ;	Process hole #12 and #13 after positioning, return to R point plane.
N011 G49 G80 Z100 M5 ;	Cancel the tool length compensation, stop the spindle.
N011 G91 G28 Z0 ;	Z-axis returns to zero.
N027 M30 ;	End the program.

Precautions

(1) Please cancel the fixed cycle before switching the drilling axis, or otherwise an alarm will occur.

(2) Drilling shall not be performed in case of no commands for X, Y, Z, R or any other axis in the program block.

(3) Specify P and Q in the program block performing drilling. If being specified in the program block not performing drilling, it cannot be saved as modal data.

F: When the F value exceeds the upper limit of cutting feed speed, the operation will be performed in accordance with the upper limit of feed speed.

S: When the speed is higher than the maximum speed of specified level, alarm will be given. The speed levels are configured by parameters #2140~#2142.

(4) Program restart: The program restart is invalid during rigid tapping.

Chapter X Fixed Cycle of Turning

10.1 Inner and outer circles single fixed cycle (G90, G94)

10.1.1 Inner and outer circles cutting cycle (G90)

Function and purpose

The tool infeeds in radial direction from the start point, cuts the cylindrical or conical surface at the speed expressed as F, and returns to the start point.

Command format

G90 X(U)_Z(W)_F_;	Cylindrical surface cutting cycle, the R value is 0 and can be omitted
G90 X(U)_Z(W)_R_ F_;	Conical surface cutting cycle

X(U)_Z(W)_	Coordinate at circular cutting end point
R	X coordinate at conical surface cutting start point minus one half of the value of X coordinate at end point
F	Feed speed (mm/min)

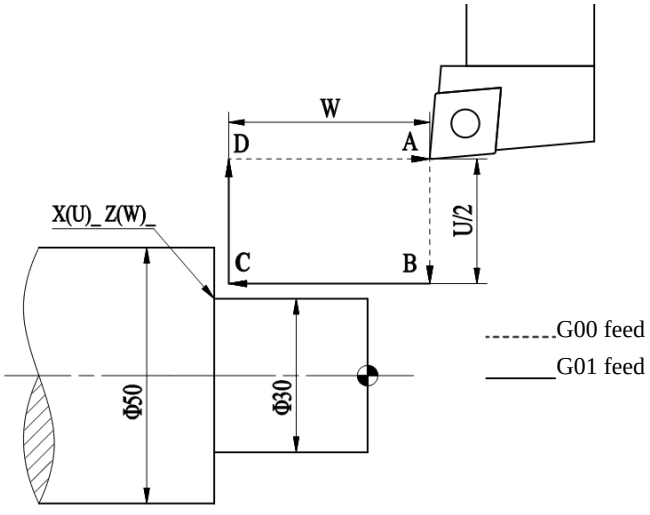
Program examples

Example 1: G90 X30.0 Z-30.0 F100;

Example 2: G90 X30.0 Z-30.0 R-5.0 F100;

Detailed description

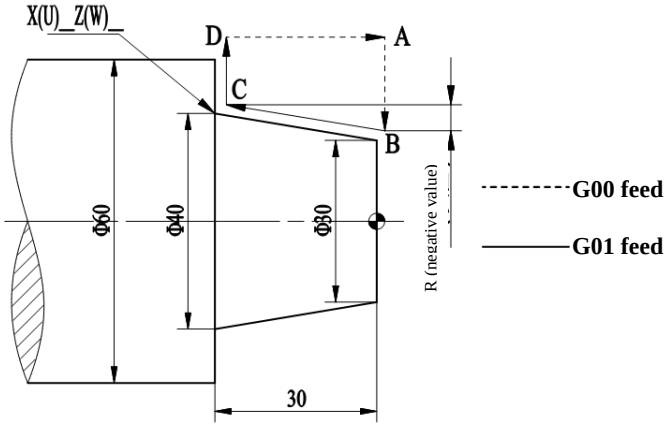
This command can be used for either the cylindrical cutting cycle or the conical surface cutting cycle.



Cylindrical surface cutting cycle

The tool movement path of the cylindrical surface cutting cycle (i.e., the rectangular cycle) starts from the start point A of the program and moves radially to the point B at the X coordinate in the command in G00 mode, and then cuts along the axial direction in the mode of G01 to the end point coordinate C. point, and then retreat to point D at the X coordinate where the cycle starts, and finally return to the cycle start point A in the way of G00 to prepare for the next action.

The process of conical surface cutting cycle is similar to that of the cylindrical surface cutting cycle. The R value in conical surface cutting cycle command has positive and negative values; when the radius at the cutting start point is less than the radius at the end point, the R value is negative; otherwise, it is positive.



Conical surface cutting cycle

Relative to simple programed commands (e.g., G00, G01), G90 has programmed the commands of four straight lines AB, BC, CD, and DA as one command, for the purpose of simplifying the programming.

Precautions

For all cycle commands of CNC lathe, special attentions shall be paid to select the correct position of program cycle start point, as such point is both the start point and the end point of the program cycle; and the point is generally 1 mm~2 mm from the surface of work blank.

10.1.2 End face cutting cycle (G94)

Function and purpose

The tool infeeds axially from the start point, cuts the end face or conical end face at the speed expressed as F, and returns to the start point.

Command format

```
G94 X(U)_Z(W)_F_;
G94 X(U)_Z(W)_R_ F_;
```

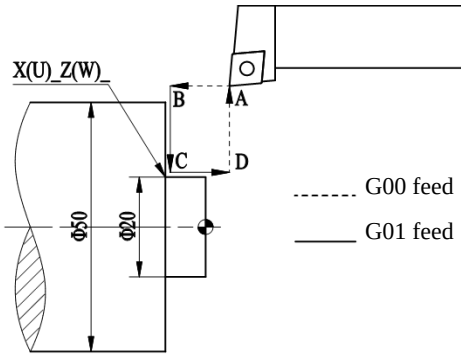
X(U)_Z(W)_	Coordinate at circular cutting end point
R	Z coordinate at conical surface cutting start point minus Z coordinate value at the end point
F	Feed speed (mm/min)

Program examples

Example: G94 X10.0 Z-20.0 F100;

Detailed description

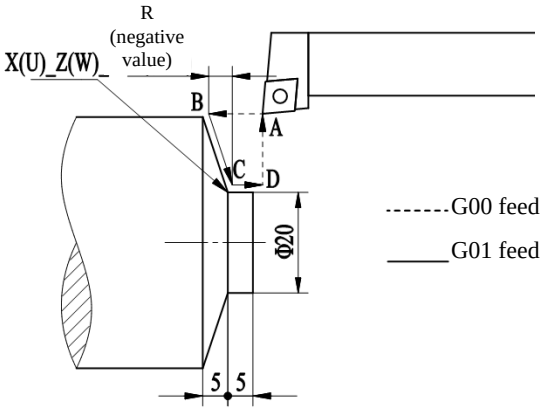
In plane cutting cycle, the tool starts from the program start point A in G00 mode, moves rapidly to point B at the Z coordinate in the command, feeds to point C at coordinate of the end point in G01 mode, and returns to point D at Z coordinate where the cycle starts, and then returns to the cycle start point A in G00 mode to prepare the next action.



Path of flat end face cutting cycle

The path of conical end face cutting cycle is similar to that of the flat end face cutting cycle.

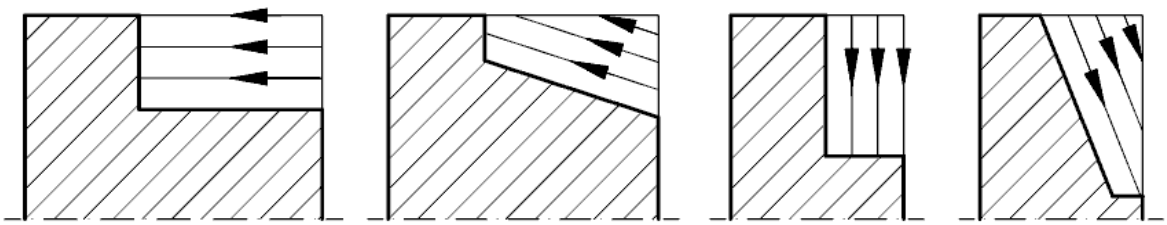
When machining with G94 command, the cutting feed speed and the back engagement of cutting edge shall be slightly smaller, in order to reduce the tool vibration during cutting.



Path of conical end face cutting cycle

Precautions to the use of single fixed cycles (G90, G94)

(1) The use of G90 or G94 shall be appropriately selected according to the shape of work blank and the machining profile of work piece; in general, it shall be selected as indicated in the diagram below.



a) b) c) d)

Selection of fixed cycle

(2) As the numeric values of X(U), Z(W) and R are modal in the fixed cycle, if X(U), Z(W) and R are not otherwise specified, the previously specified data shall remain valid.

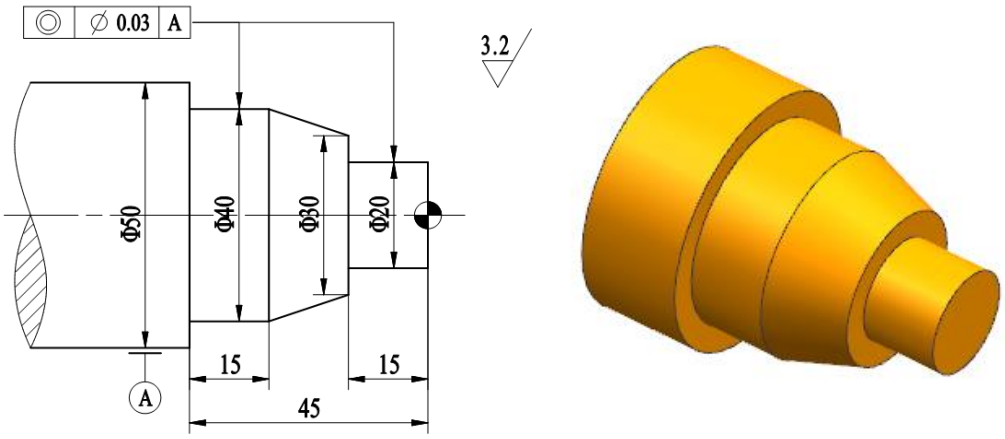
(3) If EOB or zero movement command is given in the program block using fixed cycle, the same fixed cycle shall not be executed.

(4) If M, S and T functions are assigned simultaneously in fixed cycle mode, the fixed cycle and the M, S and T functions shall be completed at the same time.

(5) If cycle is to be executed in single block operation mode, every cycle shall be implemented in 4 stages, and the Cycle Start button shall be pressed 4 times during implementation.

Programming examples

Try to compile CNC lathe machining program based on system specifications.



Examples of single fixed cycle

List of programs

Tool	Tool #1, 93 ºHard Alloy Outer Circle Turning Tool	
Program Block No.	System program	Program description
	O0010	Program number
N10	G98 G40 G21;	Beginning of program
N20	T0101;	
N30	G00 X100.0 Z100.0;	
N40	M03 S600;	
N50	G00 X52.0 Z2.0;	
N60	G90 X46.0 Z-45.0 F100;	G90 rough machining of Φ40 outer circle, fine machining allowance 0.5 mm
N70	X42.0;	
N80	X40.5;	
N90	X40.0 F50.0;	Fine machining of Φ40 outer circle
N100	X36.0 Z-15.0 F100;	G90 rough machining of Φ20 outer circle, fine machining allowance 0.5 mm
N110	X32.0;	
N120	X28.0;	
N130	X24.0;	

N140	X20.5;	
N150	X20.0 F50.0;	Fine machining of Φ40 outer circle
N160	G00 X46.0 Z-12.0;	Quick positioning
N170	G90 X46.0 Z-30.0 R-5.667 F100;	Rough machining of conical surface, infeed the tool along the cone, note the value of R
N180	X42.0 R-5.667;	
N190	X40.5 R-5.667;	
N200	X40.0 R-5.667 F50	Fine machining of conical surface
N210	G00 X100.0 Z100.0;	End part of program
N220	M30;	

10.2 Longitudinal, end face, profiling rough turning cycle and fine turning cycle (G71, G72, G73, G70)

10.2.1 Longitudinal rough turning cycle (G71)

Function and purpose

This function is a fine machining program, which automatically computes the path of axially feed rough machining tool.

Command format

```
G71 U(Δd) R(e) H_;
G71 P(ns) Q(nf) U(Δu) W(Δw) F_ S_ T_; Rough combined cycle
G70 P(ns) Q(nf);                      Finishing combined cycle
```

Different meanings of two U values in G71 command shall be noted

When using the parameter setting value, the first program block can be omitted.

Δd	X-direction back engagement of cutting edge (determined by radius amount) is a modal value without symbol; parameter "#3132 cut depth" can be configured
e	Tool retraction amount, modal value, set parameter "#3133 tool retraction amount"
ns	Block number of the first program block of fine turning program
nf	Block number of the last program block of fine turning program
Δu	Magnitude and direction of X-direction fine turning allowance, determined by diameter amount. This machining allowance is of directionality, i.e., the machining allowance of outer circuit is positive, the machining allowance of inner bore is negative
Δw	Magnitude and direction of Z-direction fine turning allowance
H	Machining type, can be configured by reversible parameter "#3112 G71/G72 type of machining path" When H is 1, the machining mode of Type II will be executed When H is omitted or is not 1, the machining mode of Type I will be executed
F,S,T	They are respectively the feed speed, the spindle speed and the tool function in rough machining cycle

G70 is the finishing combined cycle; this command cannot be used alone; it shall be used after the rough combined cycle commands G71, G72, or G73.

Note: Reversible parameter means that no program command is issued, setting value of parameters is used, and parameter of parameter value is rewritten by program command.

Program examples

G71 U1.5 R0.5;
G71 P100 Q200 U0.3 W0.05 F150;
G70 P100 Q200;

Detailed description

10.2.1.1 Type I

Machining path

Tool of G71 rough turning cycle starts from the cycle start point (point C). The tool starts from the cycle start point (point C), and quickly retracts to point D, and the tool retraction amount is determined by the values of Δw and Δu/2; then rapidly infeeds along the X-direction by Δd (radius value) to point E; then press G01 to infeed. After reaching point G, quickly retract the tool in the 45° direction to point H (the tool retraction amount in the X-direction is determined by the e value); quickly retract the tool in the Z-direction to the Z value of the cycle start point (point I); feed the tool in the X-direction again to the point J (the feed amount is e+Δd) for semi-finishing parallel to the finishing surface (now the tool leaves Δw and Δu machining allowances along the infeed surface respectively); after the semi-finishing is completed, it rapidly returns to the cycle start point, and ends all actions of the rough turning cycle.

F and S values in the command are the F and S values in rough machining cycle; all F and S values between program block numbers "ns" and "nf" are invalid in the cycle. In addition, the value may not be specified; instead, the F value in previous program block can be followed, even in the program after the rough and fine machining.

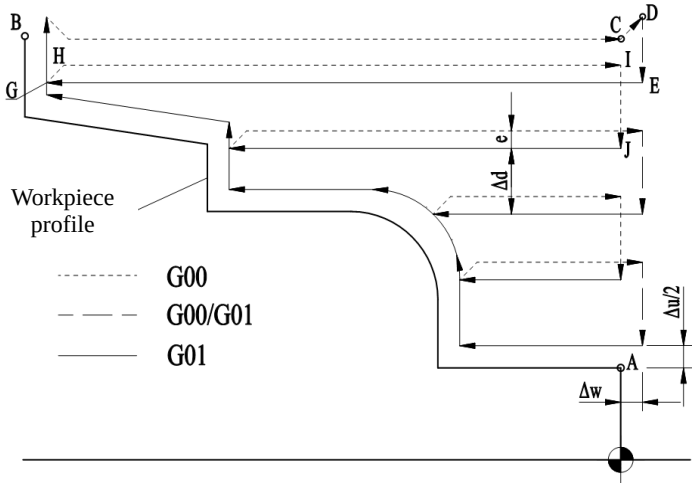


Diagram of the Path of Type I Rough Turning Cycle

Rough machining direction

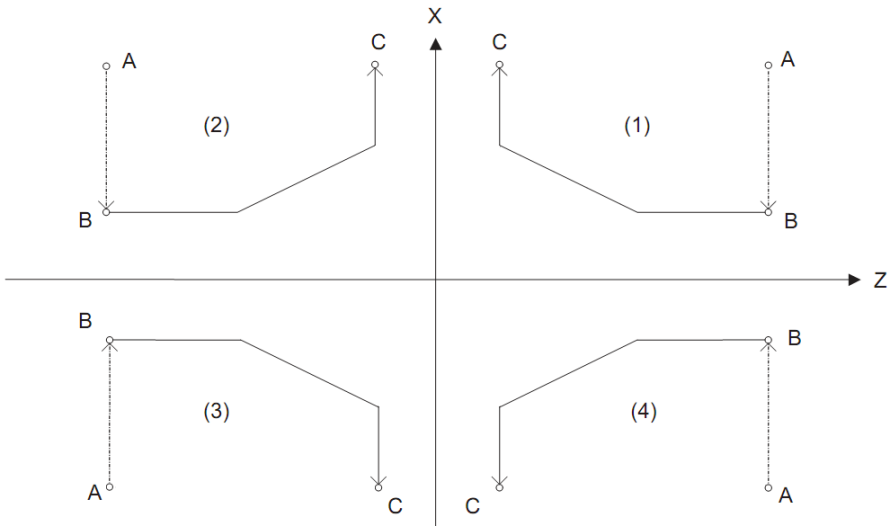
The rough machining direction is decided by the following method according to the machining path.

X-axis of C (machining path end program block) > X-axis of B (machining path start program block	(1) or (2)
X-axis of C (machining path end program block) < X-axis of B (machining path start program block	(3) or (4)

Z-axis of C (machining path end program block) > Z-axis of B (machining path start program block	(2) or (3)
Z-axis of C (machining path end program block) < Z-axis of B (machining path start program block	(1) or (4)

X-axis of C (machining path end program block) = X-axis of B (machining path start program block), or Z-axis of C (machining path end program block) = Z-axis of B (machining path start program block), program error alarm is given.

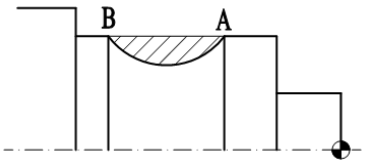
The direction of fine machining amount depends on the symbol of fine machining amount specified in program.



A: G71 cycle command point | B: Machining path start program block | C: Machining path end program block

The machining path program is A→B→C.

Usually, in all rough machining cycles of type I, the profile must be in the monotone increasing or monotone decreasing form; otherwise the concave profile will be cut in one time instead of being cut by layers. When machining the inner arc section AB shown in the diagram, the shadow part is cut in one time during semi-fine turning, for the layered cutting is inoperable because both increasing and decreasing in X-direction exist for machining allowance of rough turning cycle.



Rough turning of groove

In G71 Type I cycle, the tool must be infed along X-direction at program block of number "ns", and movement command of Z-axis cannot appear; otherwise, program alarm occurs.

N100 G01 X30.0; Correct "ns" program block

N100 G01 X30.0 Z2.0; Wrong "ns" block, the Z coordinate word appears in the program block

When performing G70 cycle, the tool cuts along the actual path of workpiece, as the workpiece profile A~B in the diagram above. The tool returns to the cycle start point after the cycle is finished. F and S values during G70 process are determined by the F and S values between block numbers "ns" and "nf".

10.2.1.2 Type II

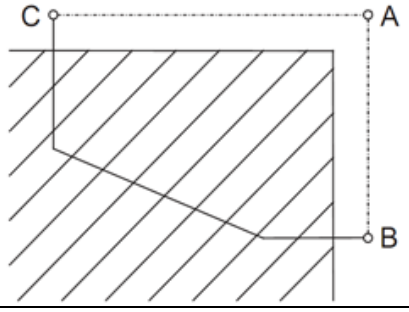
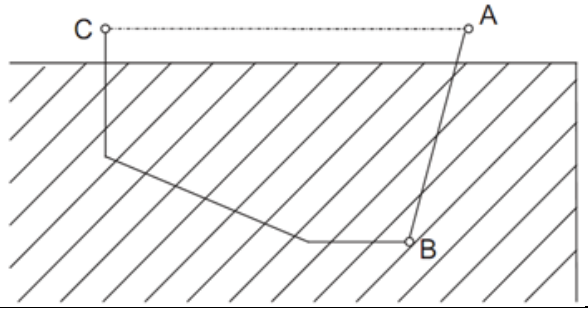
Machining of open part and middle part

In rough turning cycle, machining includes the machining where the workpiece side is open part, and the

machining start from the middle of workpiece.

Machining of open part ..... Type I/type II can be used

Machining of middle part ..... Only type II can be used

Open part	Middle part
	

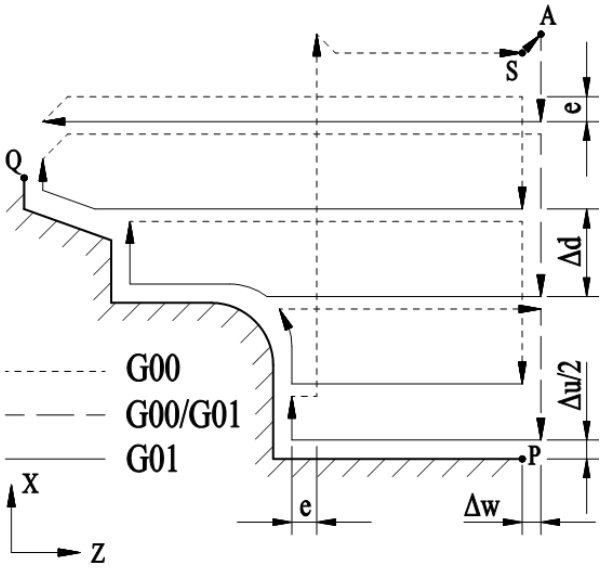
A: Rough machining start point　B: Machining path start program block　C: Machining path end program block

Machining of open part

```
G00 X80.0 Z75.0 T0101;          ←(1)
G71 U7 R2 H1;                   ←(2)
G71 P10 Q20 U3 W1.5 F500 S1500;←(3)
N10 G00 X15.0;
G01 Z55. F450;
G01 X30.0;
G03 X40.0 Z50.0 R5.0;
G01 Z42.0;
G01 X50.0;
G01 X55.0 Z35.0;
N20 G01 X60.0;
G70 P10 Q20;                    ←(4)
```

Program execution sequence (1), (2), (3), (4)

(N10-N20　4 fine machining program)



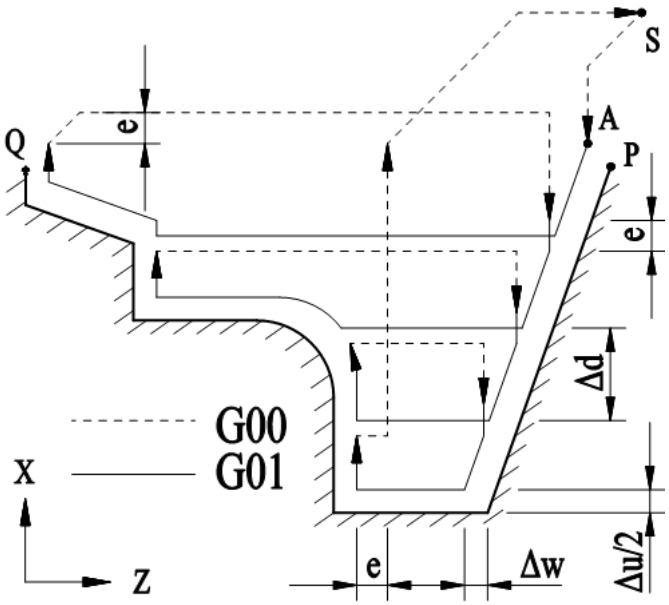
S: Cycle command point　A: Rough machining start point

Machining of middle part

```
G00 X80.0 Z75.0 T0101;          ← (1)
G71 U6 R2 H1;                   ← (2)
G71 P10 Q20 U3 W1.5 F500 S1500;←(3)
N10 G00 X60.0 Z73.0;
G01 X15.0 Z65.0;
G01 Z55. F450;
G01 X30.0;
G03 X40.0 Z50.0 R5.0;
G01 Z42.0;
G01 X50.0;
G01 X55.0 Z35.0;
N20 G01 X60.0;
G70 P10 Q20;                    ← (4)
```

Program execution sequence (1), (2), (3), (4)

(N10-N20 fine machining program)



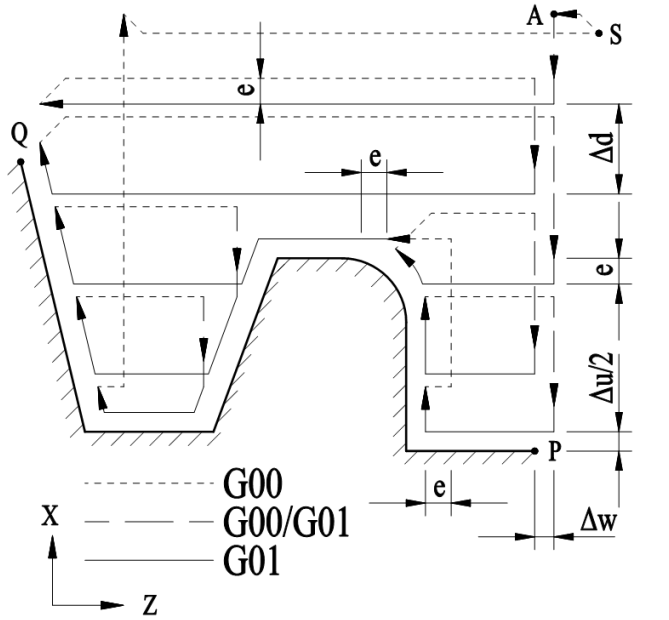
S: Cycle command point A: Rough machining start point

Joint machining

```
G00 X80.0 Z75.0 T0101;          ← (1)
G71 U7 R2 H1;                   ← (2)
G71 P10 Q20 U3 W1.5 F500 S1500;←(3)
N10 G00 X15 Z70;
G01 Z60. F450;
G01 X35.0;
G03 X45.0 Z55.0 R5.0;
G01 Z50.0;
G01 X18.0 Z45.0;
G01 Z35.0
N20 G01 X60.0 Z30.0;
G70 P10 Q20;                    ← (4)
```

Program execution sequence (1), (2), (3), (4)

(N10-N20 fine machining program)



S: Cycle command point A: Rough machining start point

In G71 type II cycle, movement command of Z-axis may occur at program block number "ns", without causing the program alarm. The movement path shall refer to the path from point S to point A in examples of the machining of middle part and the joint machining.

In G71 type II cycle, movement command of Z-axis is set by default at program block number "ns". The movement path is similar to G71 type I; please refer to the path from point S to point A in example of the machining of open part.

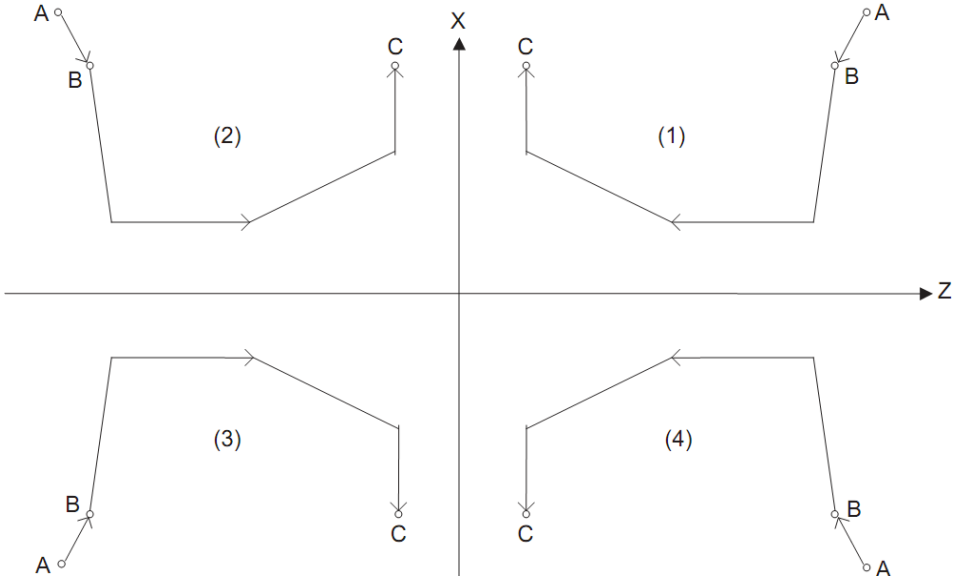
Rough machining direction

The rough machining direction is decided by the following method according to the machining path.

X-axis of A (G71 cycle command point) > X-axis of B (machining path start program block)	(1) or (2)
X-axis of A (G71 cycle command point) < X-axis of B (machining path start program block)	(3) or (4)
Z-axis of C (machining path end program block) > Z-axis of B (machining path start program block)	(2) or (3)
Z-axis of C (machining path end program block) > Z-axis of B (machining path start program block)	(1) or (4)

※ Z-axis of C (machining path end program block) = Z-axis of B (machining path start program block), program error alarm occurs.

※ When X-axis of point B is taken as the bottom; it is the same as "Type I".



A: G71 cycle command point

B: Machining path start program block

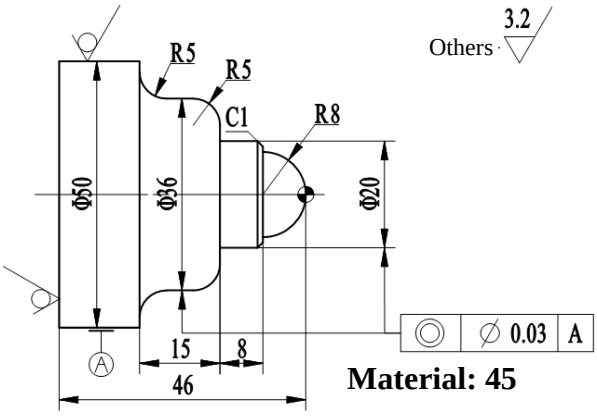
C: Machining path end program block

The machining path program is A→B→C.

The direction of fine machining amount depends on the symbol of fine machining amount specified in program.

Programming examples

Example: Try to compile the rough and fine machining programs of workpiece by composite fixed cycle command.



Example of rough combined cycle

```
O0205;
    G98 G40 G21;
    T0101;
    G00 X100.0 Z100.0;
    M03 S600;
    G00 X52.0 Z2.0;
    G71 U1.0 R0.3;
    G71 P100 Q200 U0.3 W0.0 F150;
N100 G00 X0.0 S1000;
    G01 Z0.0 F80;
    G03 X16.0 Z-8.0 R8.0;
    G01 X18.0;
        X20.0 Z-9.0;
        Z-16.0;
        X26.0;
    G03 X36.0 Z-21.0 R5.0;
    G01 Z-26.0;
    G02 X46.0 Z-31.0 R5.0;
N200 G01 X52.0;
    G00 X100.0 Z100.0;
    T0202;
    G00 X52.0 Z2.0;
    G70 P100 Q200;
    G00 X100.0 Z100.0;
    M30;
```

10.2.2 End face Rough Turning Cycle (G72)

Function and purpose

This function is a fine machining program, which automatically computes the path of axially feed rough machining tool.

Command format

G72 W(Δd) R(e) H_; G72 P(ns) Q(nf) U(Δu) W(Δw) F_ S_ T_;

The fixed cycle shall be specified by 2 program blocks.

However, the first program block can be omitted when using the parameter setting values.

Δd	Z-direction back engagement of cutting edge is a modal value without symbol; parameter "#3132 cut depth" can be configured
e	Tool retraction amount, modal value, set parameter "#3133 tool retraction amount"
ns	Block number of the first program block of fine turning program
nf	Block number of the last program block of fine turning program
Δu	Magnitude and direction of X-direction fine turning allowance, determined by diameter amount. This machining allowance is of directionality, i.e., the machining allowance of outer circuit is positive, the machining allowance of inner bore is negative
Δw	Magnitude and direction of Z-direction fine turning allowance
H	Machining type, can be configured by reversible parameter "#3112 G71/G72 type of machining path" When H is 1, the machining mode of Type II will be executed When H is omitted or is not 1, the machining mode of Type I will be executed
F,S,T	They are respectively the feed speed, the spindle speed and the tool function in rough machining cycle

Note: Reversible parameter means that no program command is issued, setting value of parameters is used, and parameter of parameter value is rewritten by program command.

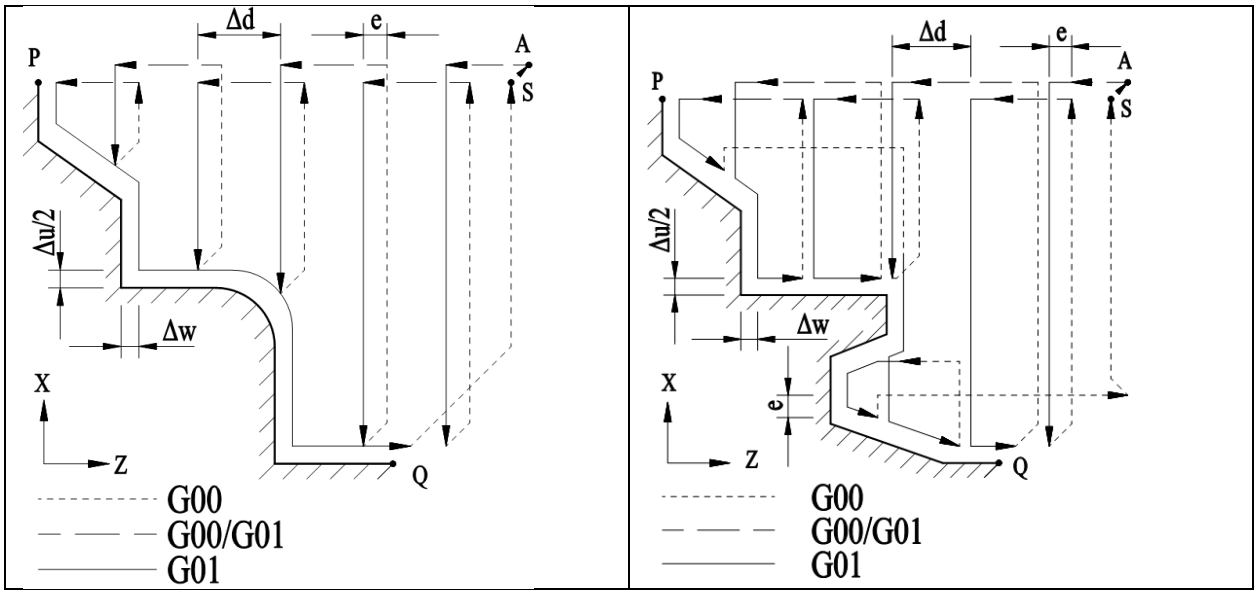
Program examples

G72 W1.5 R0.5;
G72 P100 Q200 U0.3 W0.05 F150;

Detailed description

The machining path of G72 cycle is similar to that of G71, but the cycle cuts layer by layer along Z-direction. Detail functions are the same as the "longitudinal rough turning cycle".

Example of Type I machining path	Example of Type II machining path

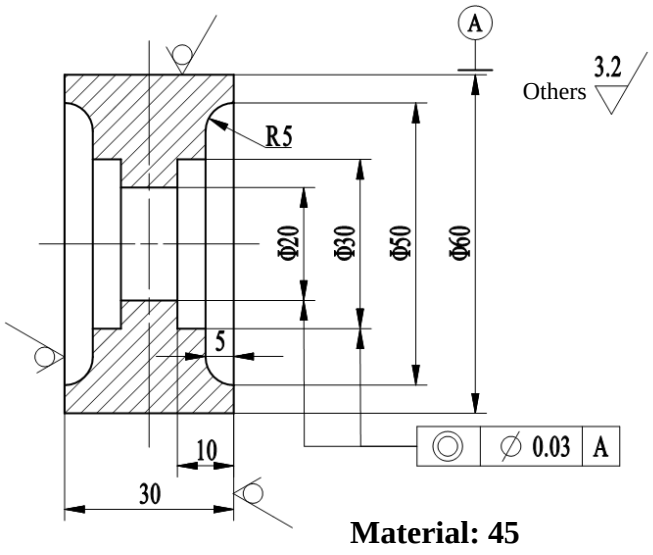


S: Cycle command point A: Rough machining start point

Diagram of the Path of End Face Rough Turning Cycle

Programming examples

Example: Try to compile the machining program for the inner contour shown in the diagram (where the Φ20mm hole is properly bored) with G72 and G70 commands.



Demonstration piece of end face rough turning cycle

```
O0207;
   ……
    G00 X19.0 Z1.0;
    G72 W1.0 R0.3;
    G72 P100 Q200 U-0.05 W0.3 F100;
N100 G00 Z-10.0 S1000;
    G01 X30.0 F50;
        Z-5.0;
        X40.0;
    G02 X50.0 Z0.0 R5.0;
```

When using G71, G72 or G73, the fine machining allowance "U" for inner hole in the command is a negative value

```
N200 G01 Z1.0;
    G70 P100 Q200;          No tool change, fine turning cycle
    G00 X100.0 Z100.0;
    M30;
```

10.2.3 Profiling Rough Turning Cycle (G73)

Function and purpose

This function is to translate the fine machining profile data to the rough machining tool path for effectively cutting the forged or roughly machined workpiece.

Command format

```
G73 U(Δi) W(Δk) R(d);
G73 P(ns) Q(nf) U(Δu) W(Δw) F_ S_ T_ ;
```

Δi	The magnitude and direction (to be specified by the radius) of tool retraction amount in X-axis direction; the value is a modal value; parameter "#3135" can be configured
Δk	The magnitude and direction of tool retraction amount in Z-axis direction; the value is a modal value; parameter "#3136" can be configured
d	Number of layering (repeat times of rough machining); parameter "#3137" can be configured

Other parameters shall refer to G71 command.

It shall be noted that the "R (d)" here is different from the "R(d)" in G71 and G72 commands

Program examples

```
G73 U3.0 W0.5 R3.0;
G73 P100 Q200 U0.3 W0.05 F150;
```

Detailed description

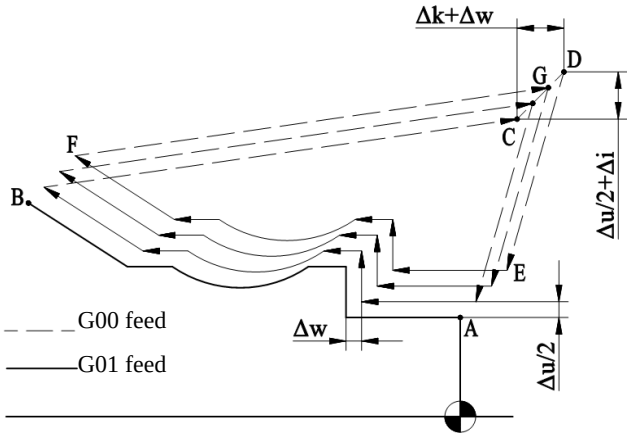


Diagram of the Path of Multiple Combined Cycle

The machining path of the G73 combined cycle starts from the cycle start point (point C), and the tool rapidly retracts to point D (the retraction amount in the X-direction is Δu/2+Δi, and the tool retraction amount in

the Z-direction is Δw+Δk); rapidly infeed to point E (the coordinate value of point E is determined by the coordinates of point A, finishing allowance, tool retraction amount Δi and Δk, and number of rough cuts); cut to point F after a certain value of circular offset along the contour; return to point G quickly to prepare the second layer of cycle cutting; in this way, the layering (the number of layers is determined by the parameter d in the cycle program) cuts to the end of the cycle, and then quickly returns to the cycle start point (point C).

G73 cycle is mainly used for turning the profile of a fixed path. Such combined cycle is able to effectively cut the cast, forged or roughly machined workpiece. For workpiece without such forming conditions, the adoption of G73 for programming and machining will increase the idle running of tool during cutting, which is adverse to the calculation of the rough turning allowance.

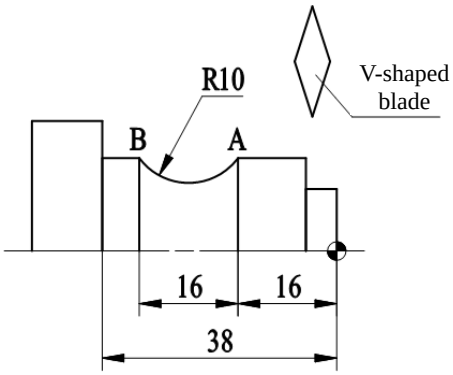
In the G73 program block, the "ns" program block can infeed in any direction in X-axis or Z-axis.

The profile to be machine by G73 cycle is free from limitations of monotone increasing or monotone decreasing.

Programming examples

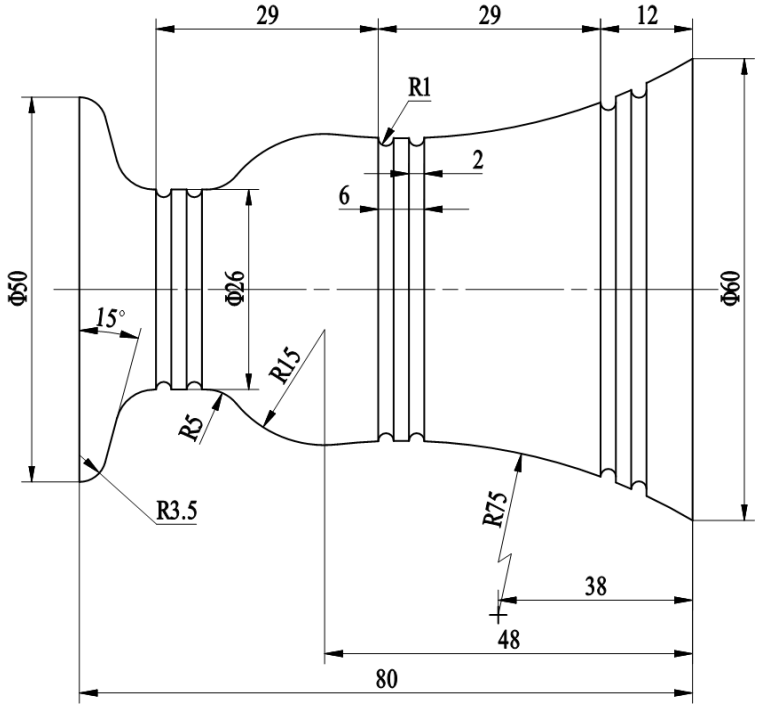
Example: Try to compile the machining program of inner arc workpiece (profile other than inner arc has been shaped by G71 and G70 commands; V-shaped blade (exchangeable turning tool) is used as the tool) with G73 command.

```
O0408
……
    G00 X32.0 Z-16.0;
    G73 U1.0 W0 R2;
    G73 P100 Q200 U0.5 W0 F100;
N100 G01 X30.0 F50 S1000;
    G02 Z-32.0 R10;
N200 G01 X32.0
    G70 P100 Q200;
    G00 X100.0 Z100.0;
    M30
```



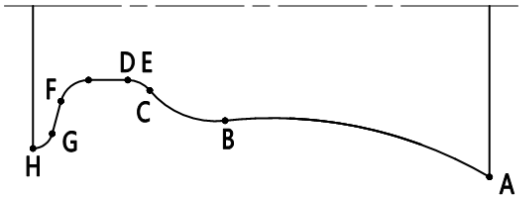
Programming example of multiple combined cycle

Programming examples



Programming example of profiling composite fixed cycle

Example: Try to compile CNC lathe machining program for workpiece with profiling rough and fine turning composite fixed cycles command (grooving program omitted). Suitable tool shall be selected for machining of this subject.



A (60.0 , 0) ; B (45.336 , -45.491) ;
C (30.924 , -60.923) ; D (26.0 , -65.231) ;
E (26.0 , -70.128) ; F (33.412 , -74.958) ;
G (45.812 , -76.619) ; H (50.0 , -80.0) .

Benchmark coordinate

Machining reference program

Tool	#1 Tool, V-Shaped Blade (Tool Nose Angle 35 °) Exchangeable Turning Tool	
Program Block No.	System program	Program description
	O0060	Program number
N10	G98 G40 G21	Beginning of program
N20	T0101	
N30	G00 X100 Z100	
N40	M3 S600	
N50	G00 X64 Z2	Quickly locate to the cycle start point
N60	G73 U17 W0 R12	Profiling combined cycle, rough turning

N70	G73 P80 Q160 U0.5 W0 F150	repeated 12 cycles, maximum allowance in X-direction 34 mm (diameter amount)
N80	G01 X60 Z0 S1000 F80	Fine machining program block infeeds along X-direction
N90	G02 X45.336 Z-45.691 R75	Fine machining path
N100	G03 X30.924 Z-60.923 R15	
N110	G02 X26.0 Z-65.231 R5.0	
N120	G01 Z-70.128	
N130	G02 X33.412 Z-74.958 R5.0	
N140	G01 X45.812 Z-76.619	
N150	G03 X50.0 Z-80.0 R3.5	
N160	G01 Z-82.0	
N170	G70 P80 Q160	Fine machining program
N180	G00 X100.0 Z100.0	End part of program
N190	M30	

10.2.4 Precautions to fixed cycles G71, G72, G73 and G70

Precautions to G71, G72, G73 and G70

(1) Inner and outer circle composite fixed cycle shall be selected according to the blank shape, workpiece machining profile and the machining requirements.

① G71 fixed cycle is mainly used as the rough turning cycle for workblank with higher radial dimension requirements as well as axial cutting dimension greater than radial cutting dimension. In programming, the value of fine machining allowance in X-direction is generally greater than that in Z-direction.

② G72 fixed cycle is mainly used as the rough turning cycle for workblank with higher end face precision requirements as well as radial cutting dimension greater than axial cutting dimension. In programming, the value of fine turning allowance in Z-direction is generally greater than that in X-direction.

③ G73 fixed cycle is mainly used as the rough turning cycle for shaped workpiece. Its fine turning allowance shall be determined according to specific machining requirements and machining shape.

(2) When programming with inner and outer circle composite fixed cycle, the program blocks between ns and nf shall not contain the following commands:

① Fixed cycle command.

② Reference point return command.

③ Thread cutting command.

④ Macro program or subprogram calling command.

(3) While executing G71, G72 or G73 cycle, the F, S, and T are only valid in program blocks of G71, G72 or G73; all F, S and T functions programmed between the called program blocks ns~nf will be ignored. On the contrary, while executing G70 fine turning cycle, the F, S and T functions in G71, G72 or G73

program blocks are invalid; in such case the F, S and T values rely on the F, S and T functions programmed between program blocks ns～nf.

(4) In G71, G72 and G73 program blocks, Δd(Δi) and Δu are both specified by address character U, while Δk and Δw are both specified by address character W; the system distinguishesΔd(Δi), Δu and Δk, Δw by judging whether P and Q are specified in G71, G72 and G73 program blocks. When P and Q are not specified in program block, U and W in the program block respectively refer to Δd (Δi) and Δk; when P and Q are specified in program block, U and W in the program block respectively refer to Δu and Δw.

(5) Δw and Δu in G71, G72 and G73 program blocks refer to the fine machining allowance; the value may be either positive or negative according to the direction of allowance. Furthermore, Δi and Δk values in G73 may either be positive or negative, as per the tool position and the tool feed-retracting mode.

(6) When composite fixed cycle G71, G72, or G73 command and M02 command are in the same block, the system directly executes M02 command instead of the common program block, until the end of program operation.

(7) When executing Type II of G71, G72 cycles, the allowable maximum groove number shall not exceed 38.

(8) At most 100 program blocks can be written between program blocks ns～nf called by G71, G72 and G73 cycles.

10.3 Grooving fixed cycle (G74, G75)

10.3.1 Radial grooving cycle (G75)

Function and purpose

The command is used for machining radial circular groove, cylindrical surface and radial boring (ignore Z, W and Q in the second line in such case); it is able to cut intermittently in radial direction to break off and remove chips.

Command format

G75 R(e); G75 X(U)_ Z(W)_ P(Δi) Q(Δk) R(Δd) F_;	
e	Retract amount; its value is a modal value; parameter "#3139 retract amount" can be configured
X(U)　Z(W)	Coordinate at grooving end point
Δi	Depth of every cut in X-direction, expressed by radius amount without symbol; its unit is set by parameter #3112.5
Δk	Offset of tool in Z-direction after finished a radial cutting, expressed by value without symbol; its unit is set by parameter #3112.5
Δd	Tool retraction amount in Z-direction when tool is cutting the bottom; it can be omitted when unnecessary
F	Feed speed during radial cutting

The last cutting depth and the last Z-direction offset are both calculated by the system itself

Program examples

G75 R0.5;
G75 U6.0 W5.0 P1500 Q2000 F60;

Detailed description

G75 cycle rectangular path:

(1) The tool starts from the cycle start point (point A), infeeds by amount Δi along radial direction and reaches point C.

(2) The tool retracts by amount e (chip break-off) and reaches point D.

(3) The tool cuts progressively by the cycle to the X coordinate at the end point.

(4) The tool returns to the radial start point to complete a cutting cycle.

(5) It deviates axially by amount Δk to point F to execute the second layer cutting cycle.

(6) It circulates sequentially until the tool cuts to the radial tool retract start point (point G) at the program end coordinate (B); then the tool retracts axially to the tool start point (point A) to complete the whole grooving cycle.

Z(W) value in G75 program block can be omitted or set to 0; when it is 0, the tool only feeds in X-direction without deviating in Z-direction while executing the cycle.

For Δi, Δk values in program block, programming unit (0:0.001mm/1:1mm) is set by parameter #3112bit5 in the system; when the value is 0, P1500 means the depth of every radial cut is 1.5mm.

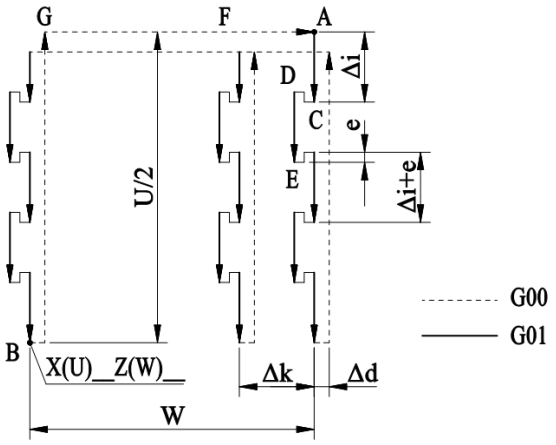
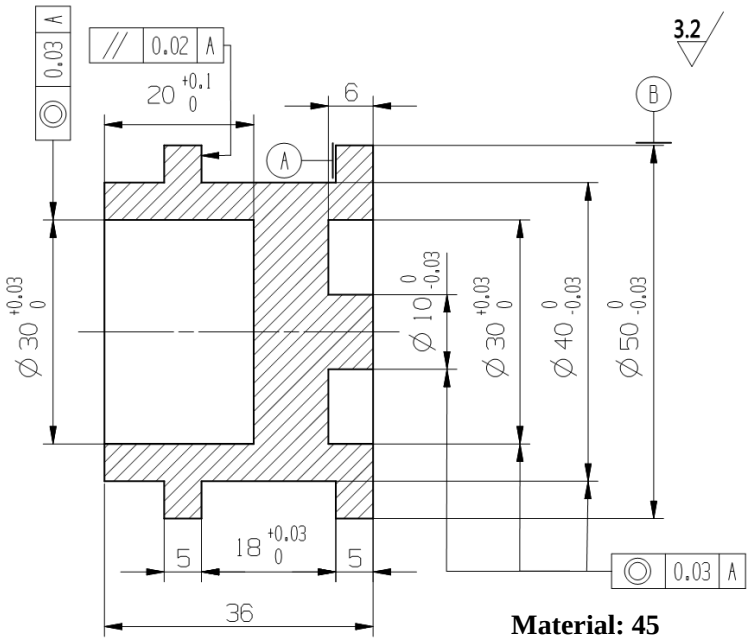


Diagram of the Path of Radial Rectangular Grooving Cycle

Programming examples

Example: Try to compile the CNC machine machining program for outer circle groove of workpiece shown in the diagram with the grooving cycle command.



Programming example of grooving fixed cycle

Its CNC machine machining program shall refer to the list of reference programs for outer circle grooving. When machining the outer circle groove, special attention shall be paid to the relationship between the Z-direction coordinate at cycle start point and the tool width.

List of reference programs for outer circle grooving

Tool	**#1 Tool, Outer Circle Grooving Tool; #2 Tool, End Face Grooving Tool**	
Program Block No.	System program	Program description
	O0060	Program number
N30	……	Beginning of program
N40	T0101	Change outer circle grooving tool
N50	G00 X52.0 Z-8.0	Quickly position to cycle start point, tool width 3 mm
N60	G75 R0.5	Cylindrical grooving cycle
N70	G75 X40.0 Z-23.0 P2000 Q2500 F60	
N80	G28 U0 W0	End part of program
N140	M30	

10.3.2 End face grooving cycle (G74)

Function and purpose

The command is used to machining of axial circular groove, end face and axial bore (X, U and P can be ignored in such case); it is able to cut intermittently in axial direction, in order to break off and remove chips.

Command format

```
G74 R(e);
G74 X(U)_ Z(W)_ P(Δi) Q(Δk) R(Δd) F_;
```

e	Retract amount; its value is a modal value; parameter "#3139 retract amount" can be configured
X(U)_ Z(W)_	Coordinate at grooving end point
Δi	Offset of tool in X-direction after finished a radial cut, expressed by value without symbol
Δk	Depth of every cut in Z-direction, expressed by value without symbol
Δd	Tool retraction amount in X-direction when tool is cutting the bottom; it can be omitted when unnecessary
F_	Feed speed during axial cutting

Program examples

G74 R0.5;
G74U6.0 W5.0 P1500 Q2000 F60;

Detailed description

The rectangular path of G74 cycle is the same as the rectangular path of G75 cycle. The difference is that, the tool starts from the cycle start point A, then cuts deep in the axial direction, and then translates in the radial direction, cycling in sequence until the action is finished.

X(U) value in G74 cycle command can be omitted or set to 0; when it is 0, the tool only feeds in Z-direction without offset in X-direction while executing G74 cycle; in such case the command can be used for end face pecking type deep hole boring cycle.

When G74 command is used for end face pecking type deep hole boring cycle command, the tool mounted on common tool holder has to be precisely located to the rotation center of workpiece.

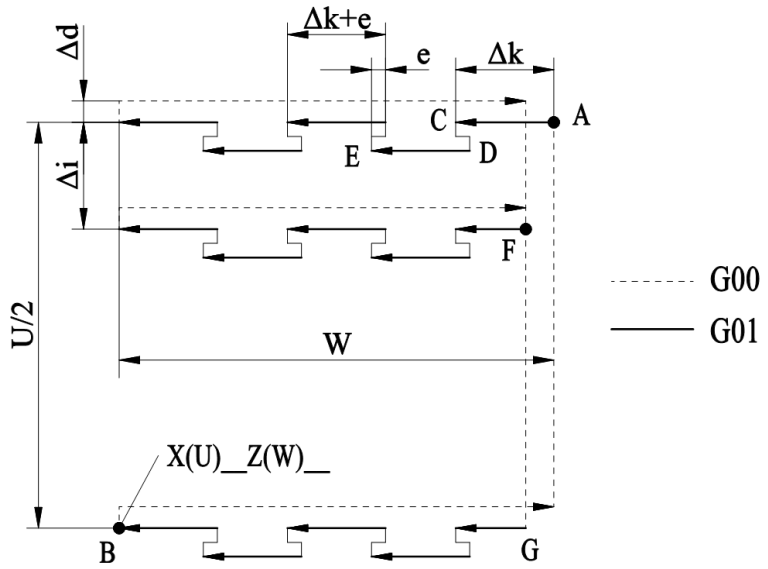
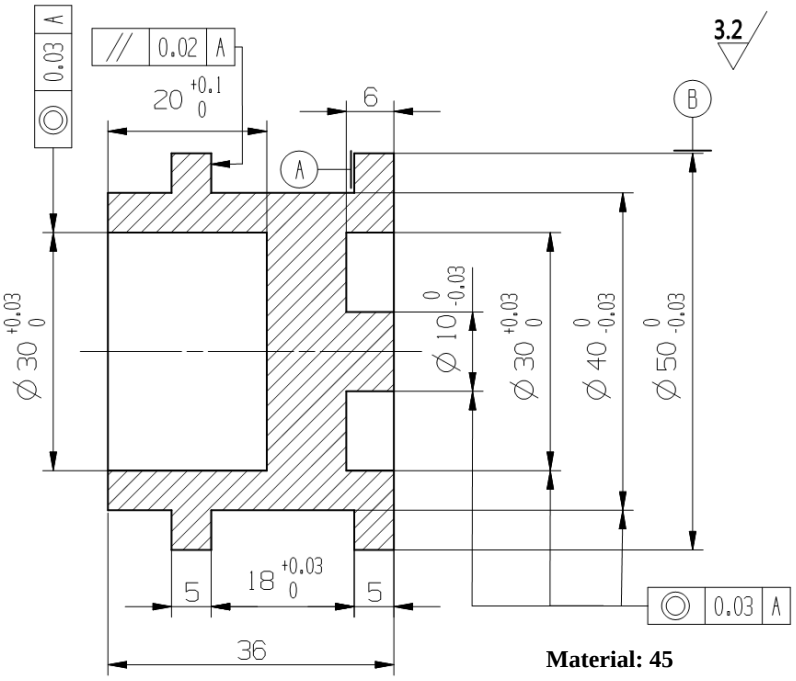


Diagram of the Path of End Face Rectangular Grooving Cycle

Programming examples

Example: Try to compile CNC machine machining program for workpiece end face groove with the grooving cycle command.



Programming example of grooving fixed cycle

Its CNC machine programs shall refer to the list of reference programs for end face grooving. While machining the end face groove, to prevent the turning tool from collision with large arc surface of workpiece groove, the end relief angle surface at tool nose shall be ground to arc shape according to the bottom size of end face groove arc, and certain relief angle shall be ensured.

List of reference programs for end face groove

Tool	**#1 Tool, Outer Circle Grooving Tool; #2 Tool, End Face Grooving Tool**	
Program Block No.	System program	Program description
	O0060	Program number
N40	……	Beginning of program
N90	T0202	Change end face grooving tool, tool width 3 mm
N100	G00 X24.0 Z2.0	Fast point positioning
N110	G74 R0.5	End face cutting cycle
N120	G74 X10.0 Z-6.0 P2500 Q3000 F60	
N130	G28 U0 W0	End part of program
N140	M30	

The width of end face grooving tool is 3 mm, but when calculating the cycle start point, the diameter of tool width is measured as 6 mm. Thus the start point of cycle X is X24.0.

10.3.3 Precautions to fixed cycles commands G74 and G75

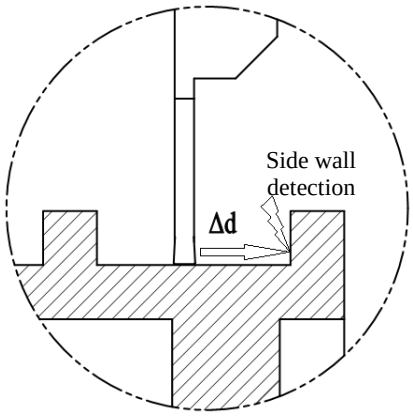
Precautions to G74 and G75

(1) In the system, program alarm appears when executing the grooving composite fixed cycle command in one of the following cases.
① X(U) or Z(W) is specified, while Δi or Δk value is not specified or specified as 0.
② Δk value is set to negative.
③ Δi value is set to negative.
④ Tool retraction amount is greater than infeed amount, i.e., e value is greater than the depth of every cut Δi or Δk.

(2) As Δi and Δk are values without symbol, the tool offset direction after finished the cut depth is automatically judged by the system according to the coordinate at tool start point and grooving end point.

(3) During the grooving process, the tool or workpiece is subjected to a large unidirectional cutting force, which is prone to vibration during the cutting process. Therefore, the value of the feed speed F in the grooving process should be slightly smaller (especially in the end face grooving process), and usually take 0.05 mm/rev ~ 0.1 mm/rev.

(4) When parameter #3112bit6 is set to 1, grooving Δd tool retracting side wall detection is enabled. When the tool is interfered by the side wall, the length of Δd will be shortened automatically, such that Δd will be limited in the range of groove width.



(5) When parameter #3112bit7 is set to 0, Δd tool retracting speed is G00. When it is set to 1, Δd tool retracting speed is G01.

10.4 Thread cutting fixed cycle (G92, G76)

10.4.1 Thread cutting single fixed cycle (G92)

Function and purpose

This function is able to realize the continuous actions of cylindrical or conical thread turning.

Command format

```
G92 X(U)_ Z(W)_ F/E_ R_ Q_ L_;
```

X(U)_ Z(W)_	Coordinate at threading end point; symbol of numeric value after U and W depends on the direction of path.
R	X coordinate at conical thread cutting start point minus ½ of X coordinate value at its end point (program end point). When R value is zero, it can be omitted in the program, and in such case the thread is a cylindrical thread.
F/E	Lead of screw thread in long axis direction/thread numbers per inch.
Q	Thread start angle
L	Number of multiple thread, the range of the value is 1～99; it is a modal parameter; when L is omitted, it is a single thread by default

Program examples

G92 X30.0 Z-30.0 F2.0;
G92 X30.0 Z-30.0 F2.0 R-5.0 L2;

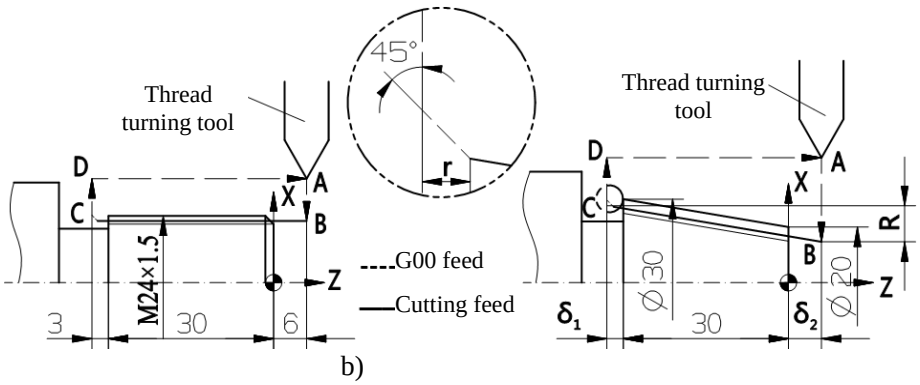
Detailed description

Path of G92 cylindrical thread cutting is as shown in Figure a); similar to the G90 cycle, the movement path is also a rectangular path. The tool moves rapidly from the cycle start point A to point B along X-direction, then cuts to point C by the feed speed of lead/round, and then retracts to point D rapidly in X-direction, and finally returns to the cycle start point A to prepare the next cycle.

In G92 cycle programming, attentions shall still be paid to the correct selection of cycle start point. Usually, the start point of the X-direction cycle is 1 mm to 2 mm (diameter) from the outer surface, and the start point of the Z-direction cycle is selected according to the imported value.

The path of G92 conical thread cutting cycle is as shown in Figure b); the path is similar to the path of G92 straight threading cycle (i.e., the original horizontal straight line BC is changed to an inclined straight line).

For R value in conical threading, in addition to distinguishing it from positive and negative during programming, the R value shall also be determined according to different length. As shown in Figure b), the length for determining R value is 30+δ1+δ2, thus the magnitude of R value shall be calculated according to this length, in order to guarantee the correctness of the thread taper.



a) b)

Diagram of the Path of Thread Cutting Single Fixed Cycle

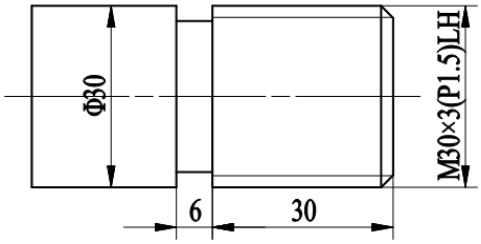
Programming examples

Example 1: Try to compile the conical threading program as shown in Figure b) with G92 command (thread cutting lead-in distance δ1 is 6mm, lead-out distance δ2 is 3, Z-direction thread pitch is 1.5mm)

```
O0305;
G98 G40 G21;
T0202;
M03 S600;
G00 X31.0 Z6.0;
G92 X28.9 Z-33.0 F1.5 R-6.5;
     X28.4;
     X28.15;
     X28.05;
G00 X100.0 Z100.0;
M30;
```

It should be noted that, R value in the command is negative; "R" is used as a modal code

Example 2: Try to compile the machining program for double left-hand thread as shown in the diagram below with G92 command. Before thread machining, the thread outer circle diameter has been machined to Φ29.8mm.



When machining external thread on steel piece, the outer circle is usually machine 0.1~0.3 mm smaller

Left-hand double thread

```
O0006;
G98 G40 G21;
T0202;
M03 S600;
G00 X31.0 Z-34.0;
G92 X28.9 Z3.0 F3.0;
     X28.4;
     X28.15;
     X28.05;
G01 Z-32.5 F200;          Translate one pitch in the Z-direction
G92 X28.9 Z4.5 F3.0;      Machining the second helix line
     X28.4;
     X28.15;
     X28.05;
G00 X100.0 Z100.0;
M30;
```

Precautions

(1) During thread cutting, the tool retracts immediately by an inclined line after the feed hold button is pressed, then returns to the start point of X-axis, and then returns to the start point of Z-axis.

(2) To execute G92 cycle in single block mode, the Cycle Start button has to be pressed 4 times after every execution of the cycle.

(3) G92 command is a modal command; when the movement amount of Z-axis is unchanged, the fixed cycle actions can be repeated only by giving the movement command to X-axis.

(4) When executing G92 cycle, the tool retracts obliquely in nearly 45 ºdirection at the back end of threading due to the delay of servo system, and the Z-direction tool retracting distance r=0.1S~12.7S (lead of screw); the value is set by system parameter #3130.

(5) During execution of G92 command, the feed speed override and the spindle speed override adjustment ratio are invalid.

(6) If F and E are specified simultaneously, E is invalid and F is valid.

10.4.2 Threading composite fixed cycle command (G76)

Function and purpose

It is able to realize the rough and fine threading, and complete the thread machining of assigned height.

Command format

G76 P(m)(r)(a) Q(Δdmin) R(d); G76 X(U)_ Z(W)_ R(i) P(k) Q(Δd) F/E_ L_;

m	The number of repeats of fine machining is 01~99; parameter "#3142" can be configured
r	Chamfering amount, the Z-direction (45 º) tool retracting distance at back end of thread cutting. When the lead of screw (screw pitch) is expressed by S, it can be configured within the range of 0.1S~9.9S, with unit of 0.1S (two digits: 00~99); parameter "#3130" can be configured
a	Tool nose angle (thread profile angle). Any of six angles, namely 80 º, 60 º, 55 º, 30 º, 29 º and 0 º, may be available for selection. The value is assigned by 2 digits; parameter "#3143" can be configured
Δdmin	Minium cut depth, expressed by radius value without decimal point; parameter "#3140" can be configured
d	Fine machining allowance, expressed by radius value without symbol; parameter "#3141" can be configured
X(U)_ Z(W)_	Coordinate at thread cutting end point
i	Thread radius difference. If i=0, cylindrical thread cutting will be executed
k	Thread form programming height. The value is expressed by radius value without decimal point, with unit of 0.001 mm
Δd	Depth of first cut, expressed by radius value without decimal point, with unit of 0.001 mm
F/E_	Lead of screw thread in long axis direction/thread numbers per inch.
L_(optional function)	Number of multiple thread, the range of the value is 1～99; it is a modal parameter; when L is omitted, it is a single thread by default

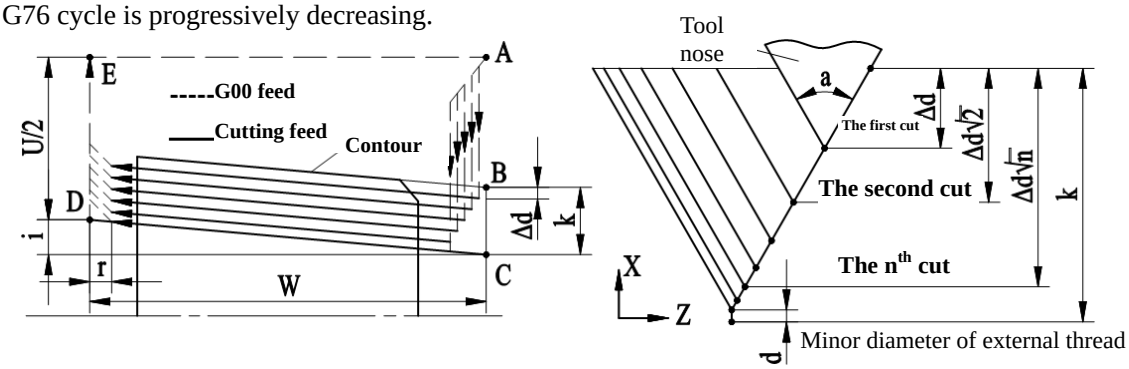
Program examples

G76 P011030 Q50 R0.05;
G76 X27.6 Z-30.0 R0 P1200 Q400 F2.0;

Detailed description

The movement path of G76 thread cutting combined cycle is as shown in Figure (a) Taking a cylindrical external thread (i value is zero) as an example, the tool feeds from the cycle start point A along the X-direction to the X coordinate of the thread crest (point B, the X coordinate value of this point = minor diameter + 2k), then feeds in a direction parallel to one side of the basic tooth profile, (as shown in Figure b), with the amount of back cutting in the X-direction being Δd, next cuts to the distance r from the end point in the Z-direction by thread cutting, retracts chamfering to D point, and the tool in the X-direction to point E, and finally returns to point A to prepare for the second tool cutting cycle. When necessary, threading shall repeat several cuts until the cycle is finished.

The back engagement of the first cut is Δd (Figure b), the back engagement of the second cut is $(\sqrt{2}-1)$ △d, and the back engagement of the nth cut is $(\sqrt{n}-\sqrt{n-1})$△d. Therefore, the back engagement of the cutting edge in G76 cycle is progressively decreasing.



(a) (b)

Movement path and tool feed path of G76 cycle

The infeed of G76 command is as shown in Figure b; the threading infeeds toward the depth direction and along the direction parallel to the basic thread form, so as to ensure that one cutting edge is used throughout the rough threading process, thereby reducing the cutting resistance, improving the tool life and ensuring the fine turning quality.

In G76 cycle command, m, r and a are specified by address P and the two digits thereafter; the front 0 in every two digits cannot be omitted. Specific meanings and specifying methods of these digits are as below:

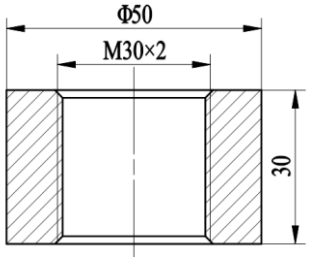
Example P011560

The detail meaning of the example is that: fine machining time "01", i.e., m=1; chamfering amount "15", i.e., r=15×0.1S=1.5S (S is lead of screw); thread form angle "60", i.e., a=60°.

Programming examples

Example: Try to compile the machining program for internal thread shown in the diagram with G76 command on CNC lathe (without considering dimensional tolerance of each diameter).

```
O0406;
G98 G40 G21;
……
T0404;
M03 S400;
G00 X26.0 Z6.0;
G76 P021060 Q50 R0.08;
```



Example workpiece of internal thread machining

```
G76 X30.0 Z-30.0 P1300 Q300 F2.0;
G00 X100.0 Z100.0;
M30;
```

For internal thread machining, the fine machining allowance "R(d)" shall also be a positive value

Precautions

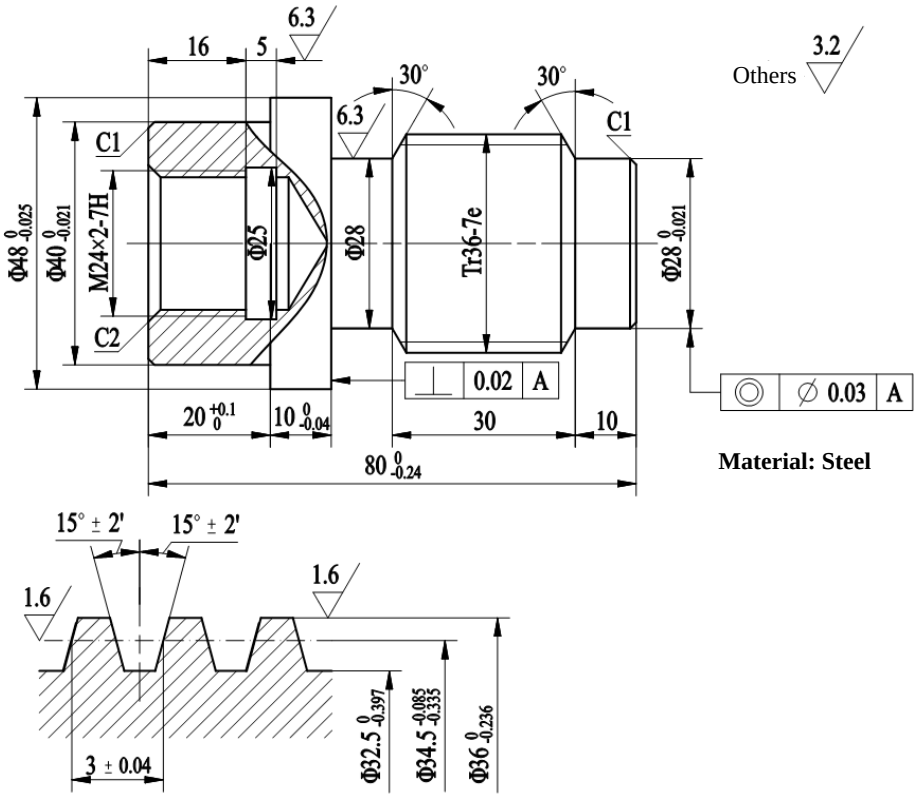
(1) G76 can be used in MDI mode.
(2) When executing G76 cycle, the tool retracts by an inclined line immediately after the feed hold button is pressed; it returns to the start point of X-axis and then returns to the start point of Z-axis.
(3) G76 command is a non-modal command, thus it has to be specified every time.
(4) While executing G76, the tool shall be returned to the cycle operation stop position prior to carrying out manual operation. If the cycle operation is restarted before the tool returns to the cycle stop position, the displacement of manual operation will be superposed on the position of the program block when it stops, thus the tool will move further by the displacement amount of the manual operation.
(5) If F and E are designated simultaneously, E is invalid and F is valid.

10.4.3 Comprehensive programming examples

Comprehensive example

Try to compile the CNC lathe programs for external trapezoidal thread and internal triangular thread shown in the diagram with thread cutting cycle commands.

In the example, the inner and the outer contours are programmed and machined by G71 and G75 commands (machining programs omitted). When machining the trapezoidal thread, direct feed method cannot be used because of greater threading depth. Therefore, oblique feed method by G76 command shall be adopted for programming of machining of trapezoidal thread. The threading depth of triangular thread is not great. Therefore, in addition to G76 command, direct feed method by G92 command can also be adopted for programming and machining. The CNC lathe program for thread machining of this subject shall refer to the reference program list of CNC lathe for thread processing



Example of thread cutting programming

Reference program list of CNC lathe for thread processing

Tool	**#1 tool, outer circle grooving tool; #2 tool, end face grooving tool**	
Program Block No.	System program	Program description
	O0060	Internal thread machining program
N40	……	Beginning of program
N50	G00 X21.0 Z2.0;	Quickly locate to the cycle start point
N60	G92 X23.0 Z-18.0 F2.0;	Internal thread is cut by five layers with G92 command
N70	X23.6;	
N80	X24.0;	
N90	X24.1;	Tolerance of internal thread is considered
N100	X24.18;	
N110	G28 U0 W0;	End part of program
N120	M30;	
N10	O0080	Machining of trapezoidal thread
N20	……	Initialize the program
N30	T0404;	Change the thread turning tool
N40	G00 X37.0 Z-4.0;	Quickly locate to the cycle start point
N50	G76 P020530 Q50 R0.08;	Trapezoidal thread machining composite fixed cycle
N60	G76X32.3 Z-43.5 P1750 Q500 F3.0;	
N70	G00 X150.0 Z30.0;	Note the nose when retracting the tool
N80	M30;	End of program

10.5 Overview of boring fixed cycle

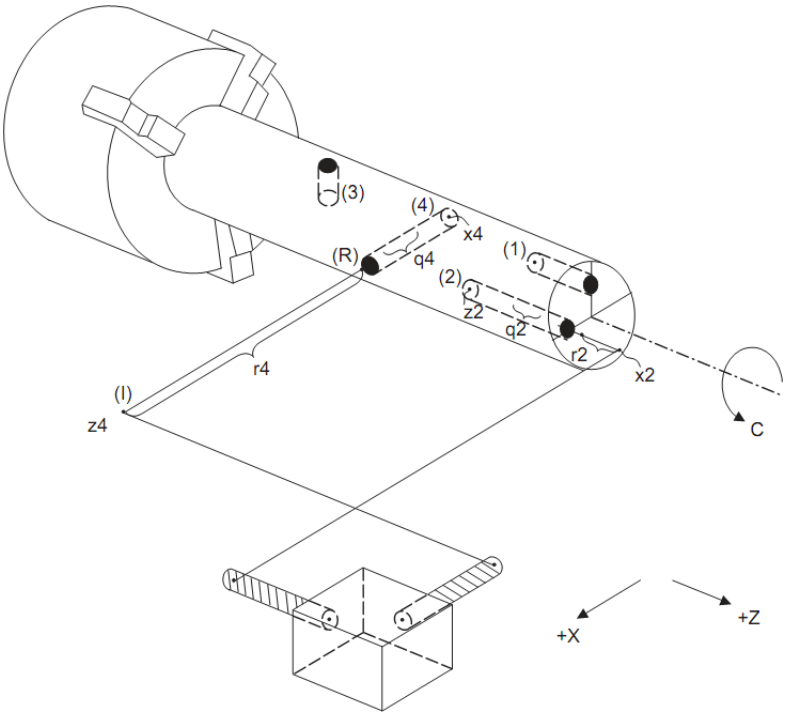
Function and purpose

Boring fixed cycle is the function which performs the common machining programs such as positioning and drilling, boring and tapping by command from a single program block in accordance with the predetermined operation sequence. When same operation needs to be repeated, only axis position command shall be executed. The types of fixed cycle are as follows.

Note: Longitudinal boring fixed cycle appoints X-axis as the boring axis; the longitudinal boring function cannot be used to appoint Y-axis as the boring axis.

G Code	Drilling Axis	Commencement of Drilling Operation	Action at Hole Bottom	Return Action	Use
G80	-	-	-	-	Cancel
G83	Z	Cutting feed Interrupted feed	Dwell	Rapid feeding	End face deep hole drilling cycle
G84	Z	Cutting feed	Dwell Spindle reverse	Cutting feed	End face tapping cycle
G85	Z	Cutting feed	Dwell	Cutting feed	End face boring cycle
G87	X	Cutting feed Interrupted feed	Delay	Rapid feeding	Longitudinal deep hole drilling cycle
G88	X	Cutting feed	Dwell Spindle reverse	Cutting feed	Longitudinal tapping cycle
G89	X	Cutting feed	Dwell	Cutting feed	Longitudinal boring cycle

Fixed cycle mode is cancelled by G80 command, other boring mode or G command of group 01; the data will be wiped at the same time.



(I) Initial point (R) R point

It means the sketch diagram of drilling axis, positioning, etc. of boring fixed cycle

(1) G83 Xx1 Cc1 Zz1 Rr1 Qq1 Pp1 Ff1 Kk1 ; End face deep hole drilling cycle

(2) G83 Xx2 Cc2 Zz2 Rr2 Qq2 Pp2 Ff2 Kk2 ;

(3) G87 Zz3 Cc3 Xx3 Rr3 Qq3 Pp3 Ff3 Kk3 ; Longitudinal deep hole drilling cycle

(4) G87 Zz4 Cc4 Xx4 Rr4 Qq4 Pp4 Ff4 Kk4 ;

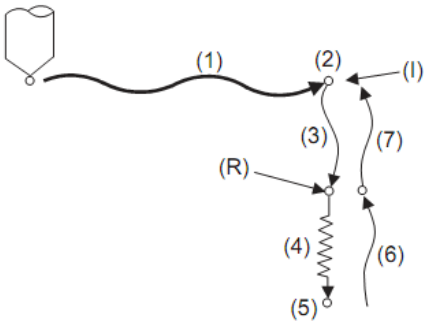
During drilling cycle, C-axis (spindle) is clamped and cannot move. (Except low-end machines)

M03, M04 and M05 (forward, backward, stop) are used to rotate the tool.

Detailed description

- Basic operations of drilling the fixed cycle

There are 7 types of actual actions.



(I) Initial point (R) R point

(1) Locate (quick feed) to initial point of C-axis, X (Z)

When specifying the "locating axis in-place width", in-place check shall be implemented when the program block is finished.

(2) If code M for clamping C-axis is issued, output will be executed.

(3) Locate (quick feed) to point R.

(4) Drilling operation is performed by cut feeding.

When specifying the "drilling axis in-place width", in-place check shall be implemented when the program block is finished. But for deep hole drilling cycles 1 and 2, in-place check shall not be implemented on the boring in process; the in-place check shall be implemented at the specified hole bottom (final hole machining process).

(5) Actions at the hole bottom are different from the fixed cycle mode; the actions include tool backward (M04), tool forward (M03) dwell, etc.

(6) Return to R point.

(7) Return to the initial point by quick feeding.

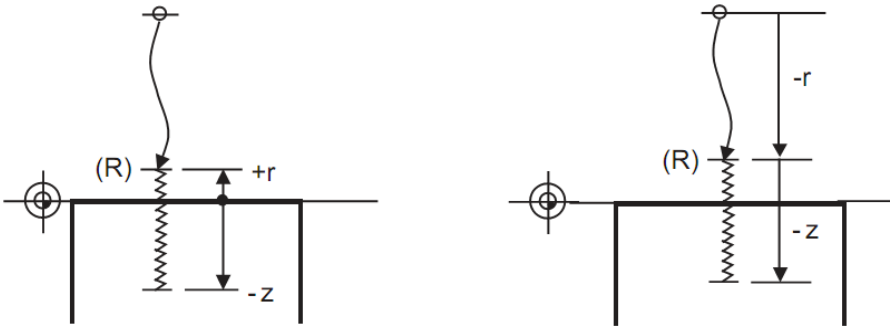
Caution: (6) and (7) may become one action due to the fixed cycle mode.

Note 1: In the case of rigid tapping command, the in-place position executed may be different according to the parameter setting.

Note 2: The fixed cycle end position can be switched to (6) or (7) by G98/G99 commands.

- Difference between absolute value command and incremental value command

At absolute value At incremental value



10.6 End face deep hole drilling cycle, longitudinal deep hole drilling cycle (G83, G87)

Command format

G83(G87) X/U(Z/W)_ C/H_ Z/W(X/U)_ Rr Qq Pp Ff Kk Mm ;

G83(G87)	G83 end face deep drilling cycle mode 1 G87 longitudinal deep drilling cycle mode 1
X/U(Z/W) C/H	Appointment of hole position initial point (absolute/increment value) ... Locate data of X-axis (Z) C
Z/W(X/U)	Specify the hole bottom position (incremental value from absolute value/R point)
Rr	Specify R point (#3101.0=0 incremental value from initial point, #3101.0=1 absolute value)
Qq	Specify the cut depth of G83 (G87). Radius value (symbol ignored) is usually specified by incremental value
Pp	Specify the dwell time at the bole bottom point; parameter #3111 can be configured Relation between time and specified value is the same as that specified by G04
Ff	Specify the feed speed for cutting
Kk	Specify the number of repeats 0~9999 (standard value=1)
Mm	Specify auxiliary commands

Note 1: In longitudinal deep hole drilling cycle (G87), Z/W command is issued at the hole position initial point, and X/U command is given at the hole bottom position.

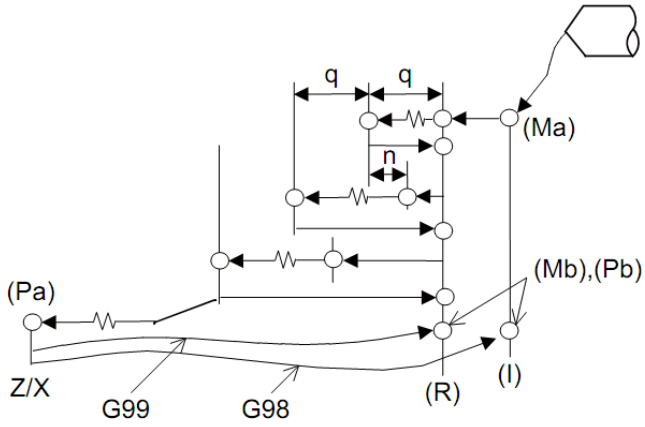
Note 2: The command of hole position initial point is non-modal. To continuously issue G83 (G87), please specify the hole position initial point in every program block.

Note 3: Q command is non-modal. Please give command in every program block.

Note 4: K command is non-modal. When there is no K command, it shall be considered as K1. When K0 is specified, the boring data will be remembered, but the boring operation will not be performed.

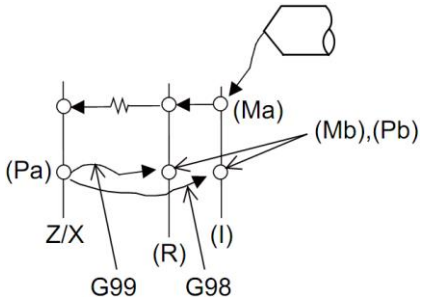
Detailed description

- When there is a Q command (deep hole drilling)



(1) Return amount n is configured by parameter (#3114 G83 retract distance). Return is performed by quick feeding.
(2) (Ma) when code M command for clamping C-axis (Mm) is assigned, output its M code (Mm).
(3) (Mb) when code M command for clamping C-axis (Mm) is assigned, output M code for releasing C-axis (M code for clamping C-axis + 1, i.e., Mm+1).
(4) (Pa) Dwell for only the time specified by P.
(5) (Pb) after output of M code for releasing C-axis (Mm+1), dwell for only the time configured by parameter.

- When there is no Q command (deep hole drilling)



G83 (G87) X(z)_ C_ Z(x)_ Rr Pp Ff Kk Mm;
(Ma), (Mb), (Pa) and (Pb) are the same as those in "With Q command (Deep Hole Drilling)".

Precautions

Please refer to the precautions to use of drilling fixed cycle.

10.7 End face tapping cycle, longitudinal tapping cycle (G84, G88)

Command format

G84 X/U_ C/H_ Z/W_ Rr Pp Ff Kk Ss Mm ; ... End face tapping cycle (end face reverse tapping cycle)

G84	G84 end face tapping cycle mode
X/U C/H	Appointment of hole position initial point (absolute/increment value) Locate data of X-axis, C
Z/W	Specify the hole bottom position (incremental value from absolute value/R point)
Rr	Specify R point (#3101.0=0 incremental value from initial point, #3101.0=1 absolute value)
Pp	Specify the dwell time at the hole bottom. Relation between time and specified value is the same as that specified by G04
Ff	Specify the feed speed for cutting
Kk	Specify the number of repeats 0~9999 (standard value=1)
Ss	Specify the spindle speed (During rigid tapping, the S command of command format with spindle number (example: " S2=2000") shall be ignored.)
Mm	Specify auxiliary commands

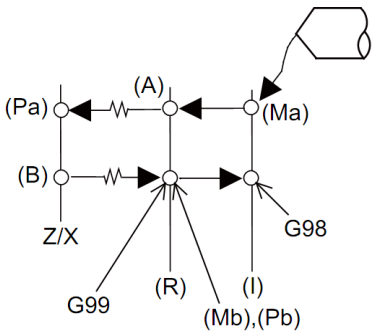
G88 Z/W_ C/H_ X/U_ Rr Pp Ff Kk Ss Mm ; ... Longitudinal tapping cycle (longitudinal reverse tapping cycle)

G88	G88 longitudinal tapping cycle mode
Z/W C/H	Specify the hole position initial point (absolute value/incremental value), locate data of Z-axis, C
X/U	Specify the hole bottom position (incremental value from absolute value/R point)
Rr～Mm	Follow-up operations are the same as the end face tapping cycle. Specify R point (#3101.0=0 incremental value from initial point, #3101.0=1 absolute value)

Note 1: Command of the hole position initial point is non-modal. To continuously issue tapping cycle commands, please specify the hole position initial point in every program block.

Note 2: K command is non-modal. When there is no K command, it shall be considered as K1. When K0 is specified, the boring data will be remembered, but the boring operation will not be performed.

Detailed description



In reverse tapping cycle, the tapping direction is reversed.

(1) (Ma), (Mb), (Pa) and (Pb) are the same as those in the "G83 End Face Deep Hole Drilling Cycle 1".

(2) When executing G84 (G88), the override is cancelled; the override is automatically set to 100%.

(3) During dry run, it is valid on locating command. When executing G84 (G88), the program block stops upon completion of the return action after the feed hold button is pressed.

(4) During single block operation, it does not stop at the return position of tapping cycle.

(5) In G84 (G88) mode, "NC" outputs the output signal of "Tapping in Progress".

(6) In G84 (G88) rigid tapping mode, there is no output of signals such as M3, M4 and S codes.
(7) Hold at R point, output tool forward signal. ((A) in the figure)
(8) Tool revolving at the hole bottom is reversed to perform the tapping. ((B) in the figure)
(9) Before output of tool backward (M04) or tool forward (M03) signal, if the tool has to be stopped (M05), please edit the fixed cycle subprogram.

10.7.1 Selection of rigid/non-rigid Tapping

Select by M function
(Example)

M29;(M command rigid tapping)
G84 Z50. R20. F2;

Or

G84 Z50. R20. F2. M29;

10.7.2 Release rigid tapping selection

Rigid tapping selection is released by reset, G80 (boring fixed cycle cancelled), G code of group 01, and G code of other cycles.

10.7.3 Feed Per Round of Rigid Tapping

Feed per round mode is adopted for tapping. G99 in the G code system A. G95 in the G code system B or C.

Precautions

Please refer to the precautions to use of drilling fixed cycle.

10.8 End face boring cycle, longitudinal boring cycle (G85, G89)

Command format

G85(G89)X/U(Z/W)_ C/H_ Z/W(X/U)_ Rr Pp Ff Kk Mm;

G85 (G89)	G85 end face boring cycle mode G89 longitudinal boring cycle mode
X/U(Z/W)_C/H_	Specify the hole position initial point (absolute value/incremental value)
Z/W(X/U)_	Specify the hole bottom position (incremental value from absolute value/R point)
Rr	Specify R point (#3101.0=0 incremental value from initial point, #3101.0=1 absolute value)
Pp	Specify the delay time at the hole bottom. Relation between time and command constant is the same as that specified by G04
Ff	Specify the feed speed for cutting
Kk	Specify the number of repeats 0~9999 (standard value=1)
Mm	Specify auxiliary commands

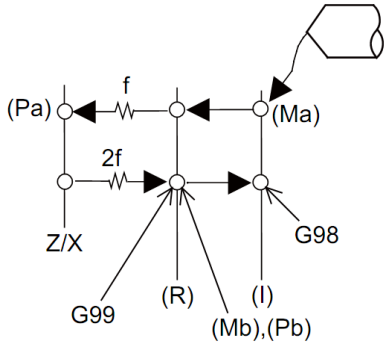
Note 1: In longitudinal boring cycle (G89), specify the hole position initial point as Z/W, and specify the hole bottom position

as X/U.

Note 2: The command of hole position initial point is non-modal. To continuously given G85 (G89), please specify the hole position initial point in every program block.

Note 3: K command is non-modal. When there is no K command, it shall be considered as K1. When K0 is specified, the boring data will be remembered, but the boring operation will not be performed.

Detailed description



(1) (Ma), (Mb), (Pa) and (Pb) are the same as those in the "G83 End Face Deep Hole Drilling Cycle 1".

(2) For return before R point, actions are executed at the cut feeding speed twice of the specified feed speed. But it cannot exceed the maximum cut feeding speed.

Precautions

Please refer to the precautions to use of drilling fixed cycle.

10.9 Cancel of drilling fixed cycle (G80)

Detailed description

Cancel the drilling fixed cycle. It cancels the drilling mode and the boring data at the same time.

10.10 Precautions to use of drilling fixed cycle

Precautions

(1) To specify G84, G88 fixed cycles, the revolving tool shall be rotated to the specified direction in advance by auxiliary functions (M3, M4).

(2) In fixed cycle mode, if the program block contains basic axis, additional axis or R data, drilling actions will be executed; if there is no data, drilling actions will not be executed.

However, if the program block is a dwell time command (G04), i.e., the X-axis data, the drilling actions will not be executed.

(3) Drilling data must be specified in the program block executing the drilling actions (program block containing basic axis, additional axis or R data).

(4)If reset is performed during execution of G85 (G89), F mode may be changed.

(5) In addition to G80, G code in group 01 can also cancel the drilling fixed cycle. If fixed cycles are specified in the same program block, the fixed cycle shall be ignored.

m = code of the group 01, n = code of the drilling fixed cycle

Gm Gn X C Z R Q P K F ;

(Example)

G01 G83 X100. C30. Z50. R-10. Q10. P1 F100.;

G83 G01 X100. C30. Z50. R-10. Q10. P1 F100.;

Both execute G01 X100. C30. Z50. F100..

(6) If auxiliary function and fixed cycle command are specified in the same program block, output shall be given at the initial locating stage.

However, if the C-axis clamped M code set in the parameter is specified in the same program block, the M code will be output after the positioning action.

When specifying the repeating times, the control above shall be executed at only the first time, except the C-axis clamped M code. C-axis clamping/releasing code M running in the mode shall be output every time before the C-axis clamping/releasing code M is cancelled by the fixed cycle cancel command.

(7) In drilling fixed cycle mode, if tool length offset command (T function) is specified; actions will be executed according to the tool length offset function.

(8) If drilling fixed cycle is specified in tool nose R compensation, program error appears.

(9) For G code series A, the standard is the fixed return initial point. The standard cannot be changed by G98/G99. Please be advised that, other functions will be executed if G98/G99 is specified.

10.11 Initial point and R point standard return (G98, G99)

Function and purpose

In fixed cycle, it may choose to finally return to R point or the initial point.

In G code system A, the tool returns from the hole bottom to the initial point. In G code system B or C, the tool returns from the hole bottom to the initial point when G98 is assigned, and returns from the hole bottom to R point when G99 is assigned. In general, the initial boring process uses G99, and the last drilling process uses G98.

Command format

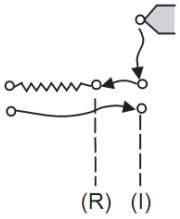
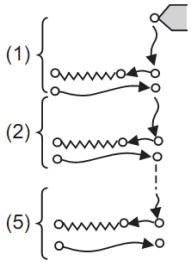
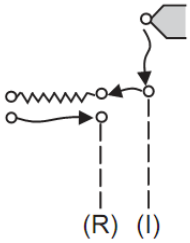
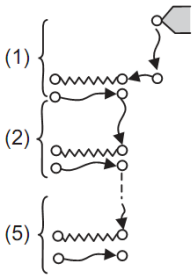
G98;... Initial point standard return

G99; ... Standard return from R point

Detailed description

Relations between G98/G99 mode and the number of repeats are as below:

Drilling times	Executed only once	Executed twice or more
Program example	G90 G83 X100. Z-50. R25 F1000;	G90 G83 X100.Z-50. R25 K5 F1000;

G98	 Initial point standard return.	 All initial point standard return.
G99	 Point R standard return.	

Chapter XI Subprogram Programming

11.1 Subprogram control (M98, M99, M198)

Concept of subprogram

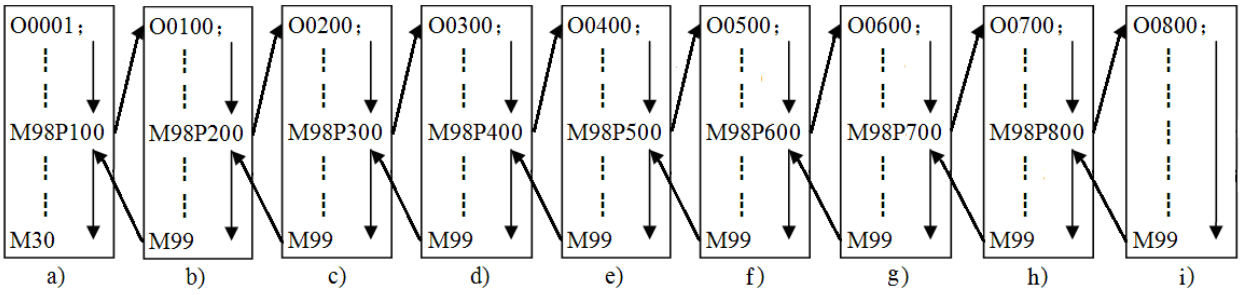
(1) Definition of subprogram: machine machining comprises two types, the main program and the subprogram. Main program is a complete parts machining program, or the principle part of a parts machining program. It is in one-to-one correspondence to the machined parts or the machining requirements; different parts or different machining requirements are corresponding to the unique main program.

While compiling the machining program, some group of program blocks may appear several times in one program, or may be used in several programs. The typical machining program can be made into a fixed program and named separately; and the group of program blocks is referred to as the subprogram.

In general, subprogram cannot be used as independent machining program; it has to be called through the main program to realize local machining actions. When completed, subprogram will automatically return to the main program that called it.

(2) Nesting of subprogram: To further simplify the machining program, the subprogram is allowed to nest with another subprogram, and this function is referred to as the program nesting.

When a main program calls a subprogram, the subprogram is deemed as the primary subprogram; subprograms in the system are allowed for 8 levels of nesting.



a) Main program; b) Level one nesting; c) Level two nesting; d) Level three nesting; e) Level four nesting;

f) Level five nesting; g) Level six nesting; h) Level seven nesting; i) Level eight nesting

Call of subprogram

(1) Subprogram format: There is no essential difference between subprogram and main program. Subprogram and main program are basically the same regarding program number and program content, with only different end tags. End of main program is expressed by M02 or M30, while end of subprogram is expressed by M99, which realizes the function of automatically returning to main program, as the subprogram below.

O0401;

```
G01 X-1.0 Z0;
……
G28 X0 Z0;
M99;
```

The subprogram end command M99 has not certainly to be written in a separate line, as in the subprogram above, the last two paragraphs can be written as "G28 X0 Z0 M99".

(2) Calling of subprogram: local subprogram can be called by auxiliary function M98; subprogram in root directory of external storage medium (USB flash drive) loaded to the system can be called by auxiliary function M198. Meanwhile, the program number address of subprogram shall be changed to P in the calling format; there are two types of subprogram calling format commonly used:

Command format I

```
M98(M198) P××××L××××;
```

The 4 digits after address character P indicate the subprogram number; the digits after address L indicate the times of repeated calling; 0 before subprogram number and calling times can be omitted. If the subprogram is to be called once, address L and digits before or after it can be omitted.

Command format II

```
M98(M198) P××××××××;
```

Among the 8 digits after address P, the first 4 digits indicate the calling times, and the last 4 digits indicate the subprogram number; when this calling format is adopted, 0 before the calling times can be omitted, but 0 before the subprogram number cannot.

In CNC lathe system, the second type of subprogram calling format is used more.

Example of subprogram executing process

```
Main program:                    Subprogram:
O0402;                           O0100;
N10 ……;                          ……
N20 M98 P0100;                   M99;
N30 ……;
    ……
    ……
N60 M98 P20200;                  O0200;
    ……                               ……
N100 M30;                        M99;
```

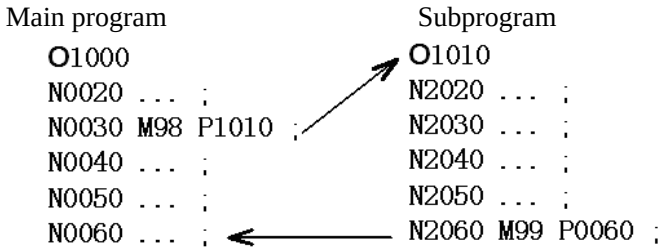
Special usage of subprogram call

(1) Return subprogram to certain block of main program: if Pn commad is added to the subprogram return

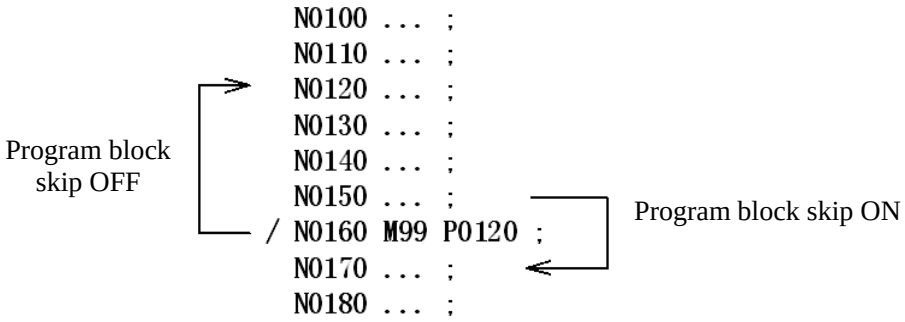
command, the subprogram will return to the program block with number of "Nn" in the main program, instead of directly returning to the main program. The program format is as below:

M99 Pn;

M99 P60; return to program block N60



(2) Automatically return to program start block: if M99 is executed in the main program, the program will return to the start program block of the main program and continues to execute the main program. M99 Pn can also be inserted in the main program to return to the specified program block. To execute follow-up program, character "/" is usually added before the command, such that the program block can be skipped by the machining tool program block skipping function when unnecessary return is to be executed.



Precautions

(1) When compiling a subprogram, the absolute coordinate method is the best way for programming, in order to prevent mistakes.

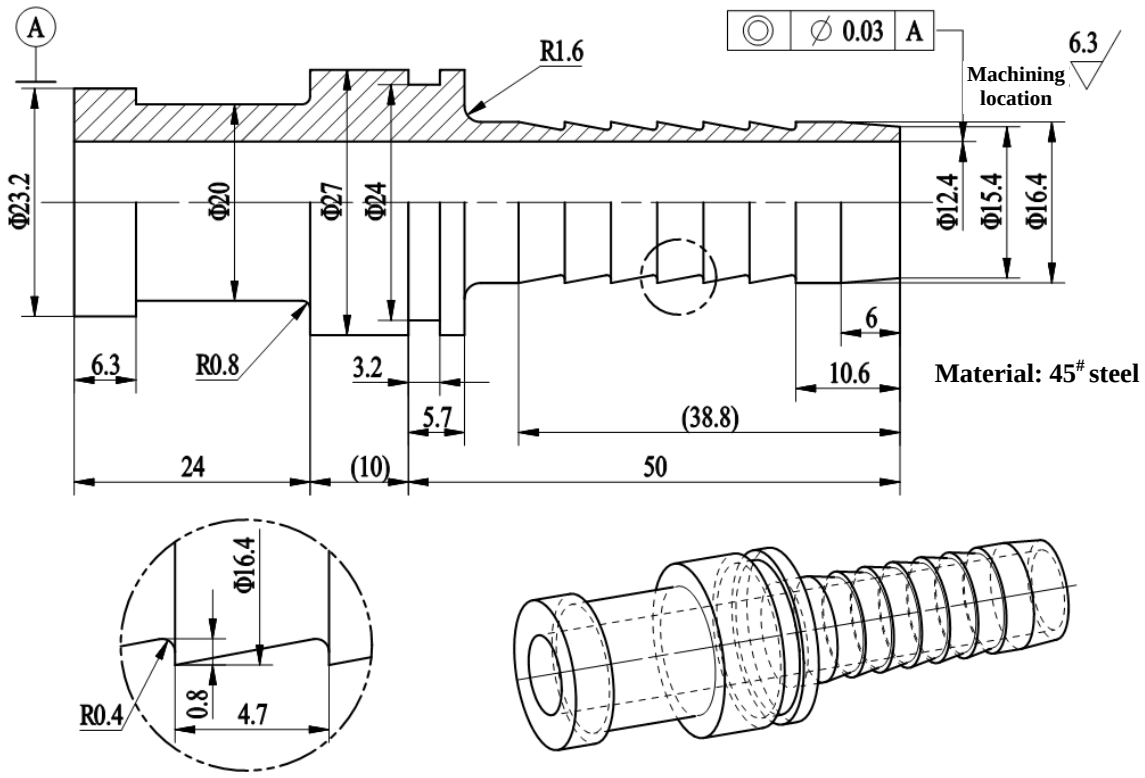
(2) When the program number of M198P_ command is **O**8000~**O**9999, the program **O**8000~**O**9999 in the system disk will be called.

(3) When the name of main program is the same as the name of subprogram called from USB flash drive by M198P_, the system gives alarm.

11.2 Programming examples

Example of turning programming

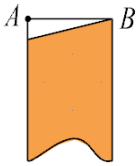
Try to compile the CNC machining program for workpiece shown in the diagram below with the subprogram programming method.



Subprogram programming examples

When compiling a fine machining program, the workpiece contour consists of several similar shapes. Therefore, programming by subprogram method can simplify the programming process. The reference program is shown in the reference program list of the example CNC lathe.

In case of fine machining of workpiece groove, special grooving tool (tool width 3 mm) shall be adopted; if assumed point A is taken as the tool location point, when machining the first groove, the Z-direction coordinate of tool location point = -10.6-3 (tool width) = -13.6.



Special tool

Sample reference program list of CNC lathe

Tool	**#1 tool, 93 °hard alloy outer circle turning tool; 2# tool, customized grooving tool**	
Program Block No.	System program	Program description
	O0010;	Main program and outer circle machining program
N10	G98 G40 G21 G18;	Beginning of program
N20	T0101;	
N30	G00 X100.0 Z100.0;	
N40	M03 S800;	
N50	G00 X28.0 Z2.0;	Quickly locate to the cycle start point
N60	G71 U1.5 R0.3;	Rough turning of outer circle surface
N70	G71 P80 Q130 U0.3 W0.0 F150;	

Tool	#1 tool, 93 °hard alloy outer circle turning tool; 2# tool, customized grooving tool	
N80	G00 X15.4 S1600;	Fine machining program block infeeds along X-direction
N90	G01 Z0.0 F60;	Fine machining path
N100	X16.4 Z-6.0;	
N110	Z-42.7;	
N120	G02 X19.6 Z-44.3 R1.6;	
N130	G01 X28.0;	
N140	G70 P80 Q130;	Fine machining of outer circle
N150	G00 X100.0 Z100.0;	Locate to tool change point
N160	T0202;	Change to grooving tool
N170	M03 S1600;	Spindle revolving forward at 1600 r/min
N180	G00 X17.4 Z-13.6;	Quick positioning
N190	M98 P60020;	Call the subprogram 6 times
N200	G00 X100.0 Z100.0;	End part of program
N210	M30;	
N10	O0020;	Grooving subprogram
N20	G01 U-1.8 F100;	Increment coordinates are suitable for subprogram
N30	G02 U-0.78 W-0.47 R0.4;	
N40	G01 U1.58 W-4.23;	
N50	U1.0;	
N60	M99;	Return to main program

Chapter XII Variable Command

Function and purpose

For a certain address in program, when direct numeric value is replaced by variable, the variable can be valued at any time as the case may be during execution of the program, in order to make the program scalable and universal.

Command format

#△△△=○○○○○○○○ ;

#△△△= [equation]　;

Detailed description

1. Expression of variable

#m	m is a numeric value consisting of 0~9.	#100
#[f]	f is an equation which indicates the following content.	#[-#120]
	Numberic value m	123
	Variable	#543
	Expression	#110+#119
	negative) equation	-#120
	[equation]	[#119]
	Function [equation]	SIN[#110]

Note 1: Standard expression comprises 4 types, i.e. +, -, *, / .
Note 2: When variable number is negative, error occurs.
Note 3: Example of incorrect variable is as below.

Error	Correct
#6/2	# [6/2] (#6/2 is construed as [#6]/2.)
#--5	#[-[-5]]
#-[#1]	#[-#1]

2. Types of variables

Variables comprise local variable, common variable (global variable) and system variable three types.

(1) Local variable (#1~#33): Local variable refers to the variable locally used in macro program. When macro program 1 calls macro program 2, and both of them have variable #1, as the variable #1 serves different parts, the #1 in program 1 and the #1 in program 2 are not the same variable, thus they can be given different values, without interfering with each other.

When the system is powered on, local variables are initialized to blank. When reset is pressed or M30 program is completed, parameter 6001 #1 can determine whether to initialize the variables to blank. When calling a macro program, a value is assigned to the local variable, and the user can use the

unassigned local variable arbitrarily. (For the correspondence between local variables and independent variables, please refer to "13.2 Macro program call" in Part 1 Chapter 13 of *Programming Operation Manual of GSK25iMc/GSK25iTc Series Bus Milling/Turning Machining Center CNC System*.)

(2) Common variables (#100~#199, #500~#999): They spread throughout the program. Likewise, when macroprogram 1 calls macroprogram 2, and both of them have variable #100, as #100 is a global variable, the #100 in program 1 and the #100 in program 2 are the same variable.

When the system is powered on, variables #100~#199 will be initialized to blank. When reset is pressed or M30 program is completed, parameter 6001#2 can determine whether to initialie the variables to blank. Data of variables #500~#999 are remembered, and they will not lose after power-off.

(3) System variables (#1000~): System variables refer to the variables with fixed purposes; and their values decide the system status. System variables comprise: tool offset variable, interface input and interface output signal variables, and position signal variable, etc.

According to attributes, system variables can be divided into read only, write only and read-write three types.

3. Variable reference

Except O, N and / (backslash), they can be used for all addresses.

(1) Use variables directly

X#1.....use the value of #1 as the value of X.

(2) Use complement of variable value

X- #2.....use the value of #2 after the symbol change as the value of X.

(3) Define variables

#3 = #5 .....use the value of #5 (equivalent to variable #3)

#1 = 1000 .....use value 1000 equivalent to variable #1 (#1 is assigned as 1000).

(4) Define variable expression

#1 = #3 + #2-100 .....#1 use the operation result value of #3 + #2-100.

X[#1 + #3 + 1000] ..... X use the operation result of #1 + #3 + 1000

Note 1: Variable of address program block cannot be defined. Please define it separately.

Error		Correct
X#1 = #3 + 100 ;	→	#1 = #3 + 100 ; X#1 ;

Note 2: At most 5 layers of [ ] can be used.

#543 = -[[[[[#120]/2+15.]*3-#100]/#520+#125+#128]*#130+#132]

Note 3: In variable definition, there is no limitation on variable number and character number.

Note 4: Please control the variable value within the range of 0~±99999999.

Beyond this range, correct operation may not be resulted.

Note 5: Variable definition becomes valid when started defining.

#1 = 100 ; .....#1 = 100

#1 = 200; ....#1 = 200

#2 = #1 + 200 ; ... .#2 = 400

#3 = #1 + 300 ; ....#3 = 500

Note 6: Reference of variable is always deemed as the numeric value with decimal point.

When #100 = 10

X#100; is X10.

Chapter XIII Macro Program

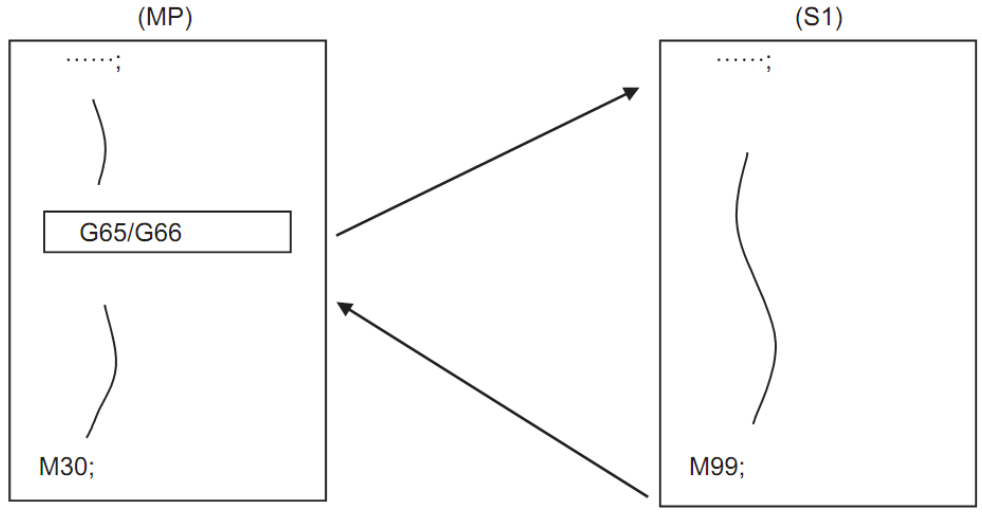
13.1 User macro program

Function and purpose

To aggregate multiple functions as one, one group of control commands and operating commands are logged in and used as macro program.

Macro program is the program that is compiled by using variables, operating commands and control commands to realize special control function.

Through combinations of variable commands, several commands such as macro program calling, various operations, data input and output with PLC, control, judgment, and divergence can be implemented, and measurement can be executed, and so on.



(MP) main program (S1) macro program (subprogram)

In a main program, macro calling commands can be called and these special control functions (macro programs) can be used when needed.

G code	Function
G65	User macro program non-modal call
G66	User macro program modal call A (movement command call)
G67	User macro program modal call cancelled

Detailed description

(1) Input G66, specified user macro program will be called before inputting G67 (Cancel) and after execution of the program block with movement command, or after execution of every program block.

(2) In the same program, G66 and G67 commands must appear in pairs.

13.2 Macro program call

Function and purpose

Macro program cal commands include non-modal call and modal call.

13.2.1 Non-modal call (G65)



(MP) main program (S1) subprogram end of user macro program subprogram is M99.

Command format

G65 P_ L_ independent variable; non-modal call

P	Program number
L	Number of repeats.

Detailed description

In user macro program subprogram, when independent variables need to be transferred as local variables, please specify actual values after address.

In such case, the independent variables are irrelative to the address and symbol and decimal point can be used; and the independent variables can be classified into the following two types.

(1) Independent variable specified I

Format: A_ B_ C_ .....X_ Y_ Z_

1) Except G, L, N, O and P, all addresses can be used to specify independent variables.

2) I, J and K must be specified in alphabetic order.

I_ J_ K_...yes

J_ I_ K_...no

3) Others than I, J and K shall not be specified in alphabetic order.

4) Addresses unnecessary to be specified can be omitted.

5) In independent variable specification I, the matchup between available address and variable number in user macro program is as shown in the table below.

Address	Variable Number	Address	Variable Number	Address	Variable Number
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

(2) Independent variable specification II

Format: A_ B_C_ I_ J_ K_ I_ J_ K_ ...

1) Except addresses A, B and C, addresses I, J and K can be used as 1 group of independent variables; at most 10 groups can be specified.

2) When same addresses repeat, please specify them in specific order.

3) Addresses unnecessary to be specified can be omitted.

4) In independent variable specification II, the matchup between available address and variable number in user macro program is as shown in the table below.

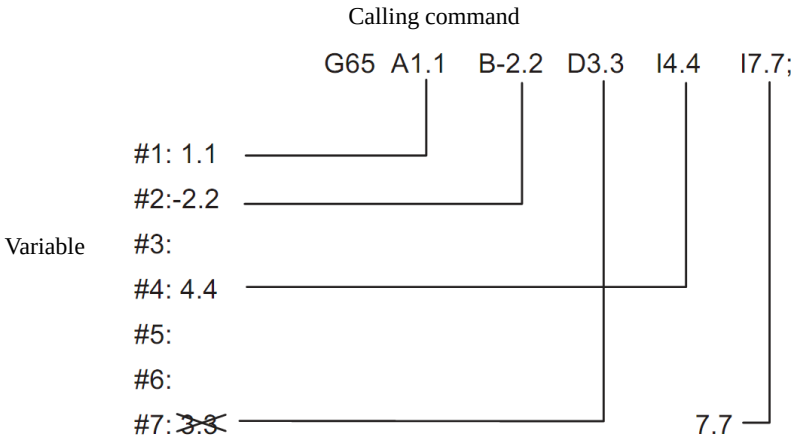
Address	Variable Number	Address	Variable Number	Address	Variable Number
A	#1	K3	#12	J7	#23
B	#2	I4	#13	K7	#24
C	#3	J4	#14	Il8	#25
I1	#4	K4	#15	J8	#26
J1	#5	I5	#16	K8	#27
K1	#6	J5	#17	I9	#28
I2	#7	K5	#18	J9	#29
J2	#8	I6	#19	K9	#30
K2	#9	J6	#20	I10	#31
I3	#10	K6	#21	J10	#32
J3	#11	I7	#22	K10	#33

Note: 1~10 added to I, J and K refer to the order of the command group; they are not required in the actual command.

(3) Independent variable specification I, II coexist

When types I and II are used for specifying independent variables, if the addresses corresponding to the same variable are commanded, the later address shall take effect.

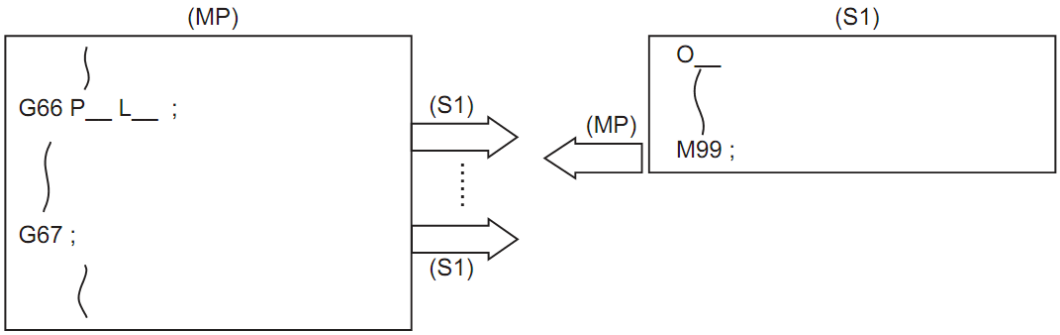
Example:



In the example above, when two independent variables D3.3 and I7.7 are specified to variable #7, the later specified I7.7 takes effect.

13.2.2 Modal calling A (G66)

Function and purpose



(MP) main program (S1) subprogram

If the movement command program block between G66 and G67 is specified, the specified user macro program subprogram shall be executed after the execution of the movement command.

The executing times of user macro program subprogram shall be specified by L.

Calling of independent variable is the same as non-modal calling.

Command format

G66 P_ L_ independent variable; modal calling A

P	Program number
L	Number of repeats.

Detailed description

(1) After input of G66 command and before input of G67 (cancel) the program block with movement command shall call the specified user macro program subprogram after execution of the movement command.

(2) In the same program, G66 and G67 commands must appear in pairs.

13.2.3 G code macro program calling

Function and purpose

The user macro program subprogram with specific program number can be called by only specific G code.

Command format

G** independent variable; G code macroprogram calling

G**	G code for macroprogram calling

Detailed description

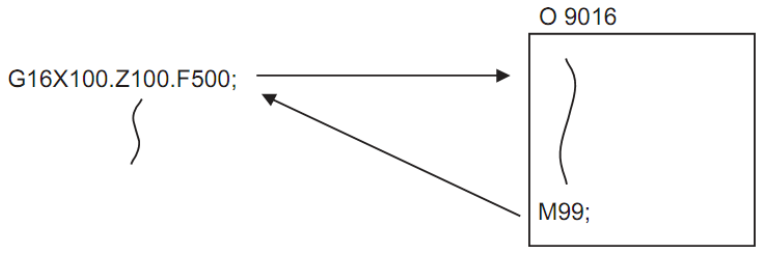
(1) The command above and the command below execute the same action; but G codes shall be respectively configured by parameters to decide which command is matched with the action.

a : M98 P △△△△ ;

b: G65 P △△△△ independent variable;

(2) To execute macro program calling, the number P△△△△ of macro program to be called shall be configured in the parameter for matchup.

(3) It cannot be specified in the user macro program subprogram called by G code.



13.2.4 Auxiliary command macro program calling (M, T Code Macroprogram Calling)

Function and purpose

Specifying only M (or T) code can call the user macro program subprogram with specific program number. (M is code already logged in; S, T and B are objects of all codes.)

Command format

M**	M code for macro program call

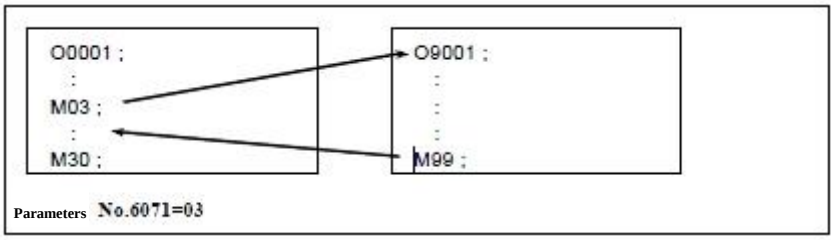
T**	T code for macro program call

Detailed description

1. Call macro program by M code

Through parameter settings, subprogram can be called by M code; M code is not sent to PLC when calling subprogram by M code.

The calling method is the same as M98.



Matchup between subprogram called by code M×× and the program number △△△△ of the called subprogram is configured by parameters.

Parameter number	Program number
6071	O9001
6072	O9002
6073	O9003
6074	O9004
6075	O9005
6076	O9006
6077	O9007
6078	O9008
6079	O9009

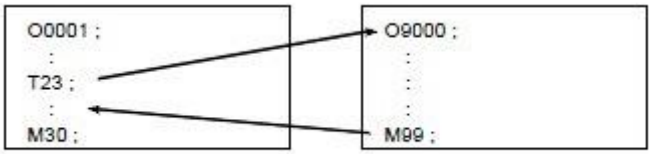
Example 1: call automatic tool change program through M06

Set the following parameter: No6071=6

Run code M06 to call the subprogram O9001.

2: Call macro program by T code

To call subprogram or macroprogram by T code in parameters, call the macroprogram once the T code appears in machining program every time.



Parameters No. 6001#0=1

Set the following parameter

No6001 TCS #0=1 call subprogram O9000 by T code.

Function of T code for calling subprogram O9000 is the same as that of M98 P9000.

13.2.5 Detail explanation of macro program calling command

Detailed description

(1) Difference between M98 command and G65 command

1) G65 can specify independent variables. But M98 cannot.
2) M98 can specify program numbers, but G65 cannot.
3) M98 executes subprogram after execution of other commands than M, P, H and L in program blocks of M98. But M65 enters subprogram without any processing.
4) When other addresses than O, N, P, H and L contained in program blocks of M98, single block stops; but in G65 single block does not stop.
5) Standard of local variables in M98 is fixed. But in G65 the standard changes along with nesting levels. For example, "#1" before and after M98 are of the same meaning. But in G65 they have different meanings.

(2) Nesting of macro program calling commands

When calling macro program subprogram through pure calling or modal calling, at most 8 levels can be nested.

Independent variables of macro program calling command are valid only in the macro program calling level.

At most 8 levels of macro program calling can be nested; in every macro program calling, independent variables can be used as local variables for the program.

Note: If G65 is executed, the G code macro program calling or auxiliary command macro program calling is deemed as Level 1 nesting, and the standard of local variables is increased by 1 level.

13.3 Variable

Function and purpose

The variable used in the custom macro program needs to use the user macro program specification.

Among the variables in the CNC device, the values of common variables #500 to #999 and system variables are retained even after the power is turned off.

Detailed description

(1) Nesting of variables

For using the variables of user macro program specification, the variable number can be set by using variable (nesting), or the variable number can be replaced by <Expression>.

Example 1: Variable nesting

```
#1=10;
#10=20;
#20=30;
#5=# [#[#1]] ;                    By #1=10 becomes #[#[#1]]=#[#10].
Change to #[#10]=#20 through #10=20.
Therefore, #5=#20 is #5=30.
```

```
#1=10;
#10=20;
#20=30;
#5=1000;
#[#[#1]]=#5;                       By #1=10 becomes #[#[#1]]=#[#10].
                                    Change to #[#10]=#20 through #10=20.
                                    Therefore, #20=#5 is #20=1000.
```

Example 2: A specified example of variable nesting

```
#10=5;                             Usage of <expression>##10=100; is the same as #[#10]=100;
##10=100;                          Change to #5=100.
```

Example 3: Replace variable number with <expression>.

```
#10=5 ;
#[#10 + 1] = 1000; change to #6 = 1000
#[#10 - 1] = -1000; change to #4 = -1000
#[#10 * 3] = 100; change to #15 = 100
#[#10/2] = -100; change to #2 = -100 (decimal fraction automatically rounded).
```

(2) Undefined variable

After power-on, among the variables of user macro program specification, unused variables and local variables as independent variables in G65 and G66 can be used as <blank>. And the variables can be forcibly made <blank>.

Variable #0 is always used as <blank> variable, which cannot define the left.

1) Expression

#1 = #0; ..... #1 = <empty>

#2 = #0 + 1; ..... #2 = 1
#3 = 1 + #0; ..... #3 = 1
#4 = #0 * 10; ..... #4 = 0
#5 = #0 + #0; ..... #5 = 0

In the expression, <blank> has the same effect as 0; please pay attention to this during use.

<Blank> + <blank> = 0

<Blank> + <blank> = constant

<Constant> + <blank> = constant

2) Variable reference

When an undefined variable is referenced, the address will be ignored.

When #1 = <blank>

and G0 X#1 Z1000 ; ..... G0 Z1000; equivalent.

and G0 X[#1+10] Z1000; ..... G0 X10 Z1000; equivalent.

(3) Conditional expression

When there are only EQ and NE, <blank> is different from 0. (#0 is <blank>)

When #101 = <blank>	#101 = 0
#101EQ#0 <Blank> = <blank> is satisfied	#101EQ#0 0 = <blank> is unsatisfied
#101NE0 <Blank>≠0 is satisfied	#101NE0 0≠0 Unsatisfied
#101GE#0 <Blank>≥<blank> is satisfied	#101GE#0 0≥<blank> is satisfied
#101GT0 <Blank>＞0 is unsatisfied	#101GT0 0＞0 Unsatisfied
#101LE#0 <Blank>≤<blank> is satisfied	#101LE#0 0≤<blank> is satisfied
#101LT0 < Blank >＜0 is unsatisfied	#101LT0 0＜0 Unsatisfied

13.4 Types of variables

13.4.1 Covariate

Detailed description

Meanings of common variable are the same in different macro programs.

When power is off, variables #100~#499 are initialized to blank. When reset is pressed or M30 program is completed, the parameter 6001#2 can determine whether to initialie the variables to blank.

Data of variables #500~#999 will be saved, and will not lose even after power-off.

Setting of variable name

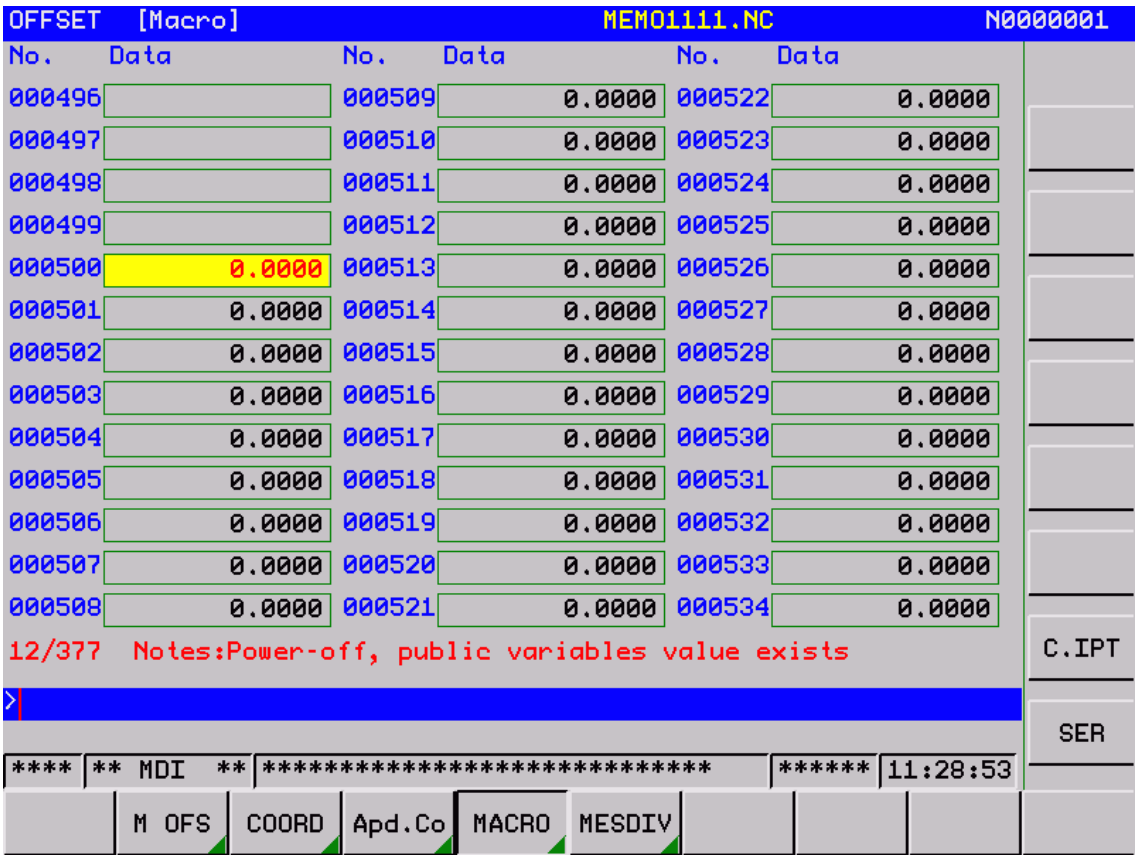
Common variables #100~#499, #500~#999 named at will (variable name). Variable name shall be limited to 64 characters (or 32 Chinese characters).

SETVNn [NAME] ;

n	Number of named variable

NAME	Name of #n (variable name)

(1) Configured variable name will lose after power-off.
(2) Variable name is displayed in macro variable comment information on offset setting interface.
(Example 1)
Program . .... SETVN503 [4LZB];



13.4.2 Local variables (#1~#33)

Detailed description

In addition to being defined as <independent variable> for calling subprogram in macro program, local variables can also be used in main program and subprogram. Local variables are independent in each program, and they can repeatedly appear. (At most 8 levels)

G65 P__ L__ <independent variable>

P	Program number
L	Number of repeats

<Independent variable> is Aa1 Bb1 Cc1 ..... Zz1.
[Independent variable specification I]
Matchup between address specified by <independent variable> and local variable number used in user macro program is as shown in the table below.

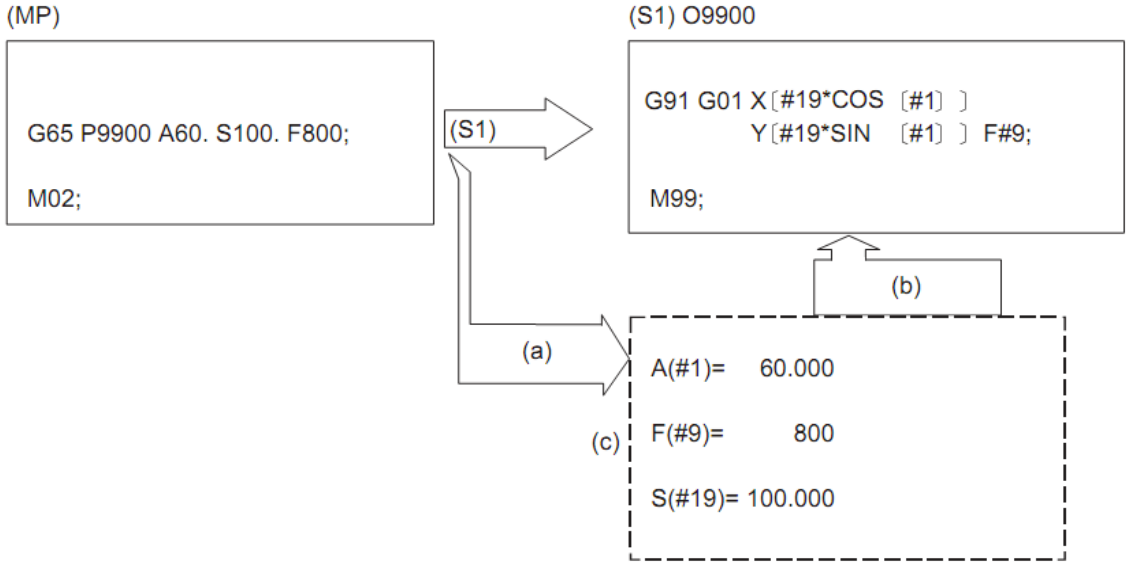
Address	Variable Number	Address	Variable Number	Address	Variable Number
A	#1	I	#4	T	#20
B	#2	J	#5	U	#21
C	#3	K	#6	V	#22
D	#7	M	#13	W	#23
E	#8	Q	#17	X	#24
F	#9	R	#18	Y	#25
H	#11	S	#19	Z	#26

[Independent variable specification II]

Address	Variable Number	Address	Variable Number	Address	Variable Number
A	#1	K_3	#12	J_7	#23
B	#2	I_4	#13	K_7	#24
C	#3	J_4	#14	I_8	#25
I_1	#4	K_4	#15	J_8	#26
J_1	#5	I_5	#16	K_8	#27
K_1	#6	J_5	#17	I_9	#28
I_2	#7	K_5	#18	J_9	#29
J_2	#8	I_6	#19	K_9	#30
K_2	#9	J_6	#20	I_{10}	#31
I_3	#10	K_6	#21	J_{10}	#32
J_3	#11	I_7	#22	K_{10}	#33

Note: 1~10 added to I, J and K refer to the order of command groups, and they are not required in actual command.

(1) When calling macro program, local variable of subprogram can be defined by specifying <independent variable>. (Local variable can be freely used in the subprogram.)

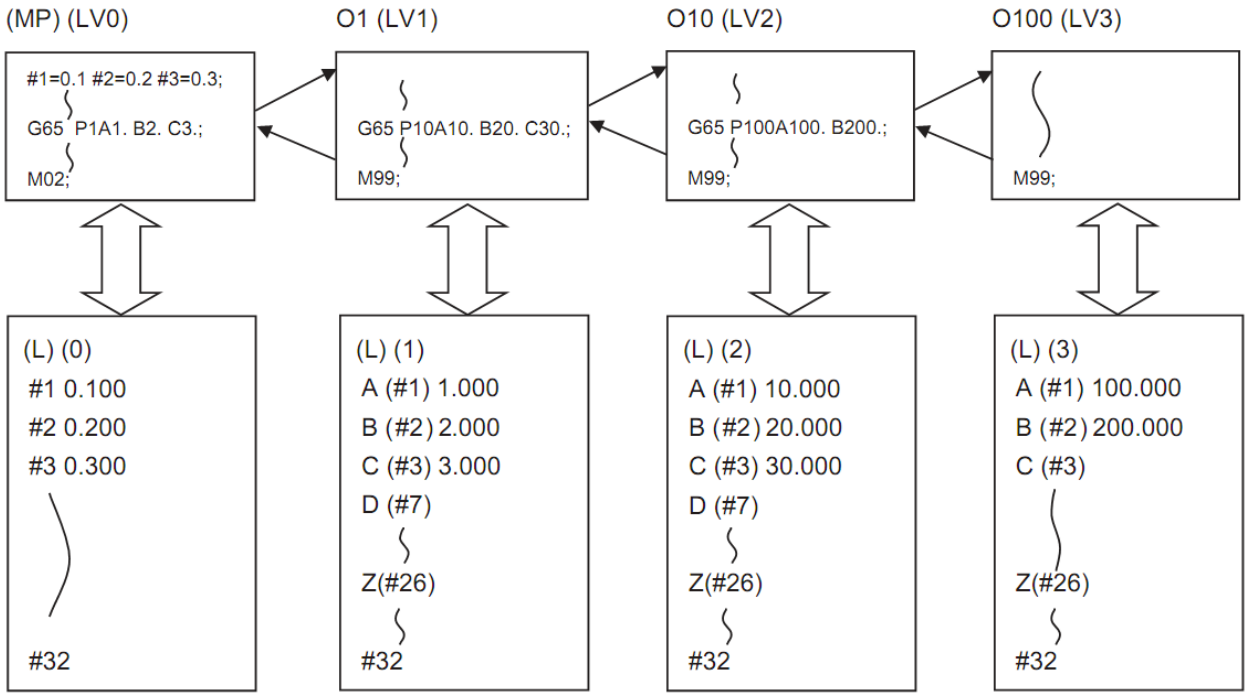


(MP) main program (S1) subprogram

(a) Set local variable by independent variable (b) Control actions like movement with reference to

local variable (c) Local variable data sheet

(2) Levels of local variable to macro program calling (8 levels) can be separately used. Local variable can also be set separately in main program (macro program level 0). Independent variable cannot be used in level 0 local variable.



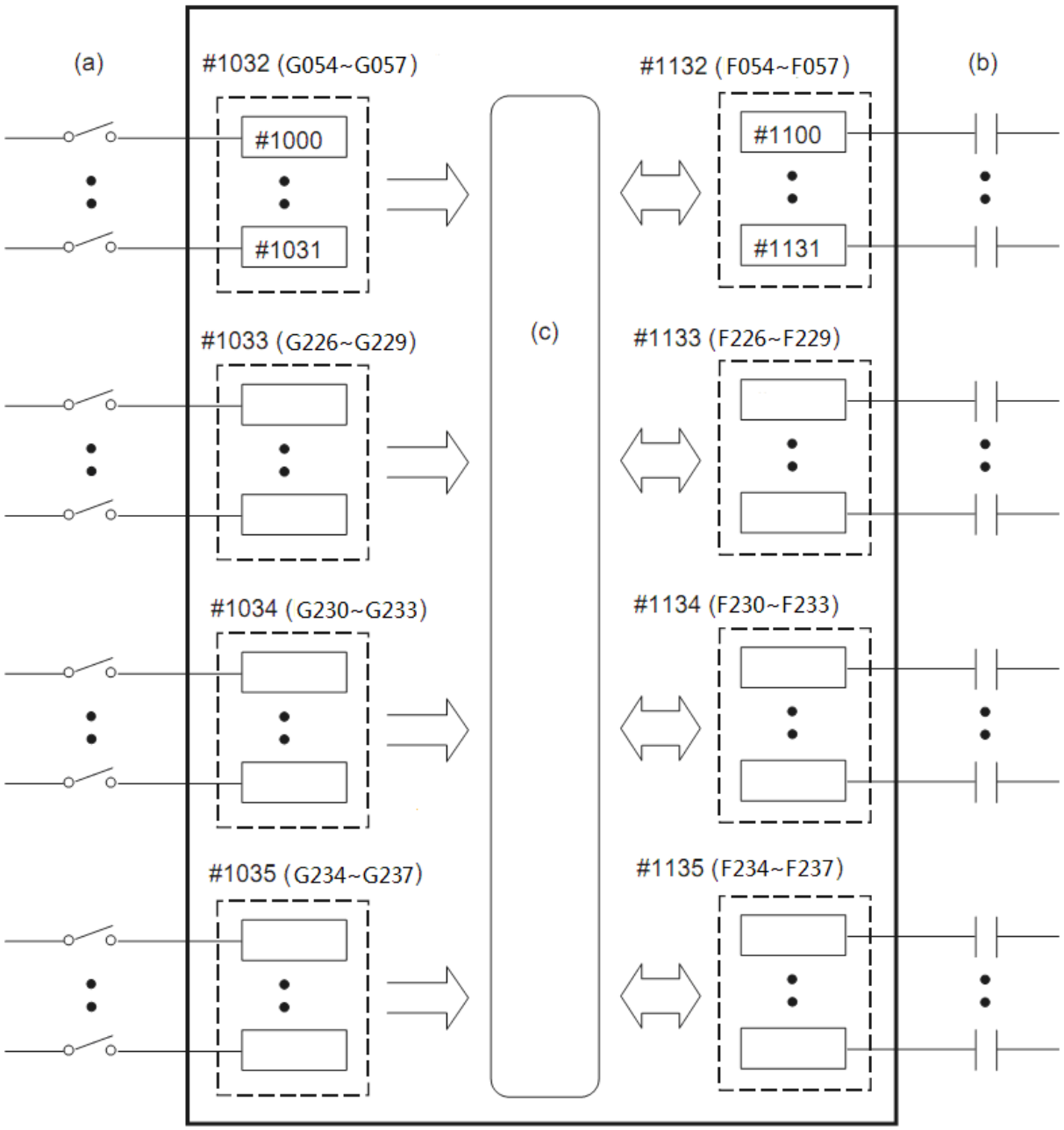
(MP) main program (1) local variable (LV0-3) macroprogram level 0~3

Condition of local variable is displayed on setting display device.

13.4.3 Macro program interface input output (#1000~#1035, #1100~#1135)

Function and purpose

Status of interface input signal can be read by reading the values of variables number #1000~#1035.
Interface output signal can be transmitted by bringing the values in variables number #1100~#1135.



(a) Input signal (b) Output signal (c) Macro program command

Detailed description

(1) Macroprogram interface input (#1000~#1035): PLC→NC

The read values are only 1 and 0 (1 = interface disabled, 0 = interface enabled). All input signals of #1000~#1031 can be read at one time y reading the value of variable number #1032. The 32-bit signal of PLC input can be read respectively by reading values of variables number #1033~#1035.

Read only the #1000~#1035, it cannot be placed on the left of expression.

System Variables	Number of Points	Interface Input Signal	System Variables	Number of Points	Interface Input Signal
#1000	1	Bit0 of address G054	#1016	1	Bit0 of address G056
#1001	1	〃 1	#1017	1	〃 1
#1002	1	〃 2	#1018	1	〃 2
#1003	1	〃 3	#1019	1	〃 3

#1004	1	〃	4	#1020	1	〃	4
#1005	1	〃	5	#1021	1	〃	5
#1006	1	〃	6	#1022	1	〃	6
#1007	1	〃	7	#1023	1	〃	7
#1008	1	Bit0 of address G055		#1024	1	Bit0 of address G057	
#1009	1	〃	1	#1025	1	〃	1
#1010	1	〃	2	#1026	1	〃	2
#1011	1	〃	3	#1027	1	〃	3
#1012	1	〃	4	#1028	1	〃	4
#1013	1	〃	5	#1029	1	〃	5
#1014	1	〃	6	#1030	1	〃	6
#1015	1	〃	7	#1031	1	〃	7

System Variables	Number of Points	Interface Input Signal
#1032	32	G054,G055,G056,G057
#1033	32	G226,G227,G228,G229
#1034	32	G230,G231,G232,G233
#1035	32	G234,G235,G236,G237

(2) Macro program interface output (#1100~#1135): NC→PLC

Output signals include 0 and 1.

All output numbers from #1100 to #1131 can be output at once by substituting the value in variable number #1132. By substituting values into variable numbers #1133 to #1135, a 32-bit output signal can be output. Output signals of #1100~#1135 can write and read the output signal status.

The output herein means the output from NC side.

System Variables	Number of Points	Interface Input Signal		System Variables	Number of Points	Interface Input Signal	
#1100	1	Bit0 of address F054		#1116	1	Bit0 of address F056	
#1101	1	〃	1	#1117	1	〃	1
#1102	1	〃	2	#1118	1	〃	2
#1103	1	〃	3	#1119	1	〃	3
#1104	1	〃	4	#1120	1	〃	4
#1105	1	〃	5	#1121	1	〃	5
#1106	1	〃	6	#1122	1	〃	6
#1107	1	〃	7	#1123	1	〃	7
#1108	1	Bit 0 of address G055		#1124	1	Bit0 of address F057	
#1109	1	〃	1	#1125	1	〃	1
#1110	1	〃	2	#1126	1	〃	2
#1111	1	〃	3	#1127	1	〃	3
#1112	1	〃	4	#1128	1	〃	4
#1113	1	〃	5	#1129	1	〃	5
#1114	1	〃	6	#1130	1	〃	6
#1115	1	〃	7	#1131	1	〃	7

System Variables	Number of Points	Interface Input Signal
#1132	32	F054,F055,F056,F057
#1133	32	F226,F227,F228,F229
#1134	32	F230,F231,F232,F233
#1135	32	F234,F235,F236,F237

Note: The final output values of system variables #1100~#1135 are 1 or 0, which will be saved. (They cannot be cleared by resetting.)

13.4.4 Tool offset

Detailed description

Using variable number can read and set tool offset data.

System variable of milling system tool compensation value

Compensation Number	Attributes	Tool Length Compensation (H)		Tool Radius Compensation (D)	
		Shape Compensation	Wear Compensation	Shape Compensation	Wear Compensation
1 ⋮ 200 ⋮ 400	Read/Write	#11001(#2201) ⋮ #11200(#2400) ⋮ #11400	#10001(#2001) ⋮ #10200(#2200) ⋮ #10400	#13001 ⋮ ⋮ ⋮ #13400	#12001 ⋮ ⋮ ⋮ #12400

System variable of turning system tool compensation value

Contents	Range of variable number
X shape compensation amount	#10001~#10000+n
X wear compensation amount	#11001~#11000+n
Additional axis shape compensation amount	#12001~#12000+n
Additional axis wear compensation amount	#13001~#13000+n
Z shape compensation amount	#14001~#14000+n
Z wear compensation amount	#15001~#15000+n
R shape compensation amount	#16001~#16000+n
R wear compensation amount	#17001~#17000+n
Tool nose position	#18001~#18000+n

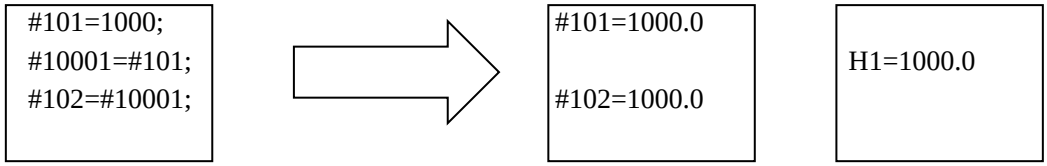
In the table, n is corresponding to the tool number. The maximum value of n is the number of tool offset groups.

The tool offset data is the same as other variables, which is the data with decimal fraction.

If the program is #10001=1000, the tool offset data shall be set to 1000.000.

Tool compensation of additional axis can use only one of the 3rd axis/4th axis.

Programming examples Covariate Tool offset data



Example 1

Measurement of tool offset data

G28X0 T0101; Reference point return

M06; Tool change (T0101)

#1=#5001; Start point memory

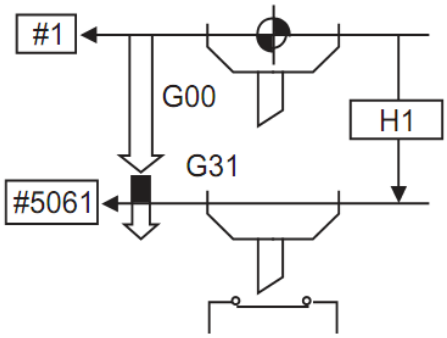
G00 X-200. ; Rapid feed to a safe position

G31 X-50.F100 ; Jump count

#10001=#5061-#1; Measurement distance calculation and tool offset data setting

The example does not consider the signal delay of jump sensor.

Here, #5001 is the rotation start position of X-axis, #5061 is the jump coordinate of X-axis, indicating the position of jump signal inputted during execution of G31.



13.4.5 Workpiece coordinate offset (#2500~#2906)

Detailed description

Using variables number #2500~#2906 can read the workpiece coordinate system off set data or bring the value therein.

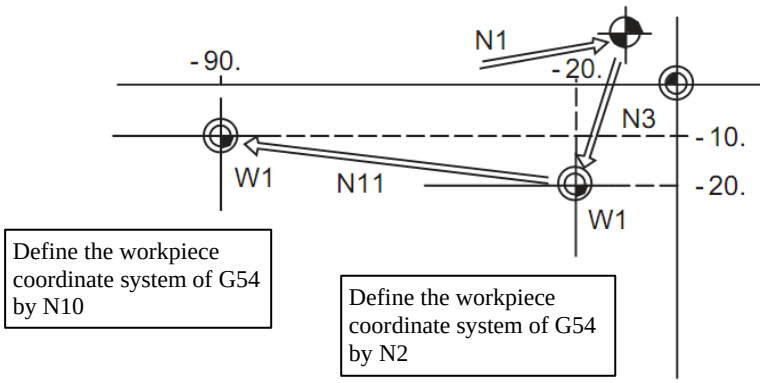
Note: The number of controllable axes changes with the specification.

The digit 1 bit lower than the variable number is corresponding to the control axis number.

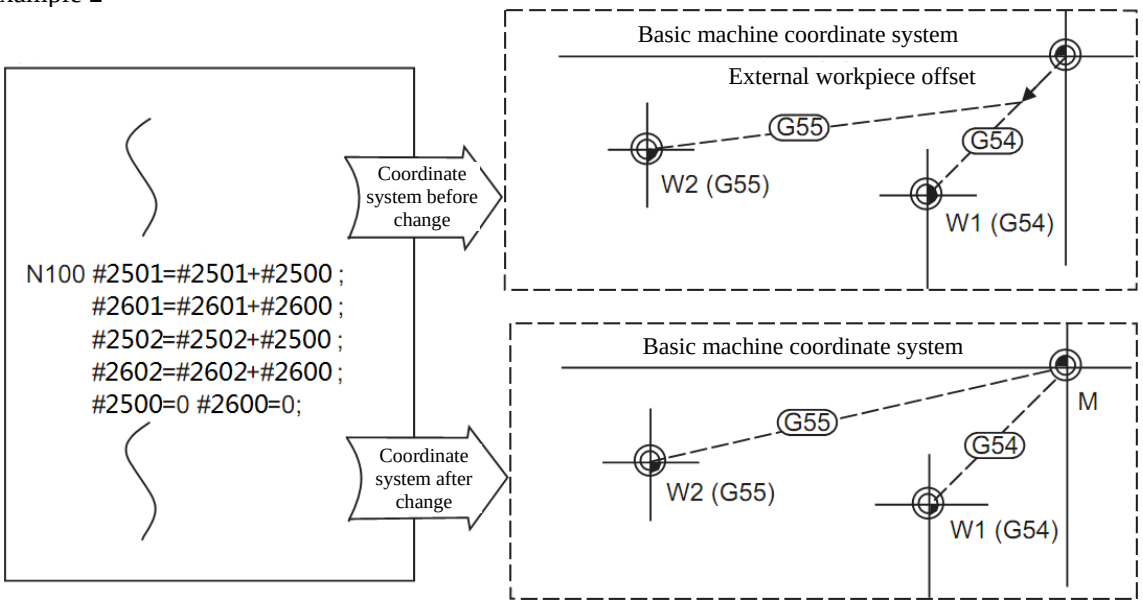
Coordinate Name	1st Axis	2nd Axis	3rd Axis	4th Axis	5th Axis	Remarks
External workpiece offset	#2500	#2600	#2700	#2800	#2900	External workpiece offset data required
G54	#2501	#2601	#2701	#2801	#2901	Workpiece coordinate system offset data required
G55	#2502	#2602	#2702	#2802	#2902	
G56	#2503	#2603	#2703	#2803	#2903	
G57	#2504	#2604	#2704	#2804	#2904	
G58	#2505	#2605	#2705	#2805	#2905	
G59	#2506	#2606	#2706	#2806	#2906	

Example 1

```
N1 G28 X0  Z0 ;
N2 #2501=20. #2601= - 20. ;
N3 G00 G54 X0 Z0 ;
      ≀
N10 #2501=10. #2601= - 90. ;
N11 G00 G54 X0Z0 ;
      ≀
M02 ;
```



Example 2



Without changing the workpiece coordinate system position, example of external workpiece offset value is added on the basis of each workpiece coordinate (G54, G55) system offset value.

13.4.6 Alarm (#3000)

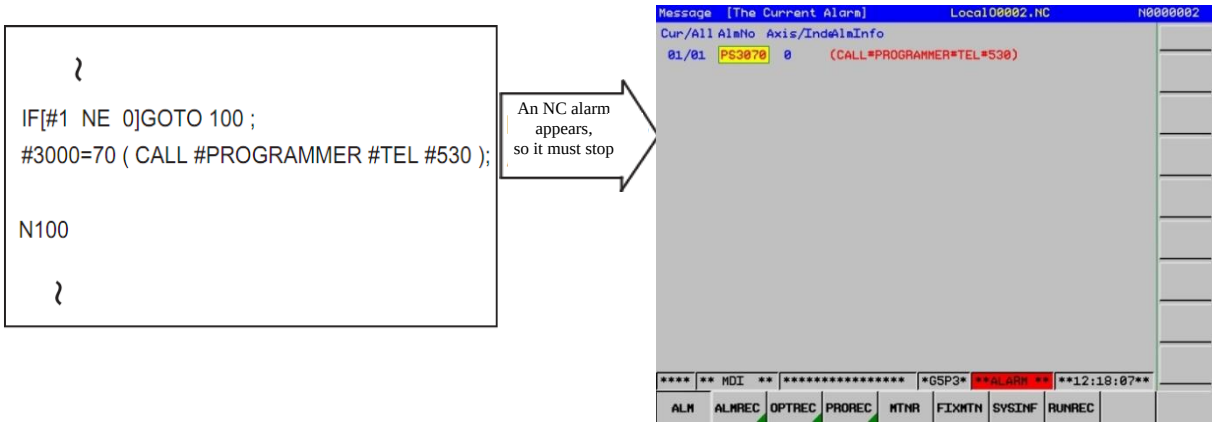
Detailed description

Using variable number #3000 can forcibly enter the alarm status.

#3000= 70(CALL #PROGRAMMER #TEL #530) ;

3070	Alarm number
CALL #PROGRAMMER #TEL #530	Alarm messages

Range of alarm numbers is 0~200.
Please contain the alarm information within 59 characters.
Output NC alarm signal.
The <Current Alarm> column on "Message Interface" shows the macro program alarm message, as well as alarm number 3070 and alarm information(CALL#PROGRAMMER#TEL#530).
Example (alarm occurred when #1=0)



Note 1: Alarm number is not displayed when it is "blank". And digits more than 3200 cannot be displayed.
Note 3: When Whenediting on PC, the alarm information shall be put in full-width brackets "( )".

13.4.7 Time information (#3001, #3002, #3011, #3012)

Detailed description

(1) Using variables number #3001 and #3002 can read the cumulative time in power-up or automatic startup, and bring the values into variables.

(2) Current date (#3011) and current time (#3012) can be read and written by system variables #3011 and #3012 of time information.

Variable Number	Contents
#3001	The unit is in millisecond.
#3002	Cumulative time can be read and value can be brought in during automatic startup. The unit is hour.
#3011	The current date is erasable. YYXX year MM month DD day is read as a numeric value YYXXMMDD. XX year MM month and DD day (2 bits lower display for year) shall be configured for reading and writing YYXXMMDD numeric value. Command setting range of year month day: Year (YYXX): 2000~2099 Month (MM): 1~12 Day (DD): Maximum days in months from 1 to
#3012	Current time is erasable. HH hour MM miute SS second is read as numeric value HHMMSS. HH hour MM minute SS second shall be configured for reading and writing HHMMSS numeric value. Command setting range of time Hour (HH): 0~23 (24h system) Minute (MM): 0~59 Second (SS): 0~59

(3) The accumulated time is about 2.44×1011ms and returns to zero.

(4) In settings of cumulative time, if the specified value is negative or exceeds 9544.371767 hours, the value of timer returns to 0.

(5) When command setting date and time, if the specified value exceeds the setting range, program error occurs.

(6)In settings of date and time, month/day/hour/minute/second must be 2-digit values.

When the value is 1 digit only, 0 must be added as required. (February 14, 2001→#3011=010214; etc.)

13.4.8 Suppression of single block stop, auxiliary function end signal waiting (#3003)

Detailed description

Bringing the values below in variable number #3003 can suppress the single block stop in follow-up program blocks, and enter the next program block without waiting for the end signal (FIN) of auxiliary functions (M, S, T, and B).

#3003	Single Block Stop	End Signal for Auxiliary Function
0	Unsuppressed	Waiting
1	Suppressing	Waiting
2	Unsuppressed	Not wait
3	Suppressing	Not wait

Note: #3003 can be zeroed by resetting.

13.4.9 Validness and invalidness of fFeed hold, feed speed override and G09 (#3004)

Detailed description

Bringing the values below in variable number #3004 can enable or disable the feed hold, feed speed override and G09 in follow-up program blocks.

#3004 (Value)	Feed Hold	Feed Speed Override	G09
0	Yes	Yes	Yes
1	No	Yes	Yes
2	Yes	No	Yes
3	No	No	Yes
4	Yes	Yes	No
5	No	Yes	No
6	Yes	No	No
7	No	No	No

Note 1: #3004 can be zeroed by resetting.

Note 2: When the feed hold is invalid in #3004, the following actions will be executed by pressing the feed hold button.

—— During thread cutting, the program block stops at the end of the next program block after the thread cutting is completed.

—— During tapping actions in the tapping cycle, the program block stops after the returning action to R point is completed.

—— Except the scenarios above, the program stops after the program block in progress is finished.

13.4.10 Information display and stop (#3006)

Detailed description

Using variable number #3006 can stop program block after operation of the last program block. When information display data is specified, the information will be displayed in information tip column.

#3006 = 1(TAKE FIVE);

1~9999	Stop number (set as value ranged in 1~9999.)
TAKE FIVE	Information (no display when there is no specific information.)

Information shall be contained in 31 characters and put in brackets "( )".

13.4.11 G Command Mode (#4001~#4130, #4201~#4330)

Detailed description

Using the variable numbers #4201~#4330 can read modal commands assigned in previous program blocks. Modes of program blocks in execution can also be read in #4001~#4130.

System Variables	Attributes	Modal Information	Group
#4001	Read only	G00, G01, G02, G03, G6.2	01
#4002	Read only	G17, G18, G19	02
#4003	Read only	G90, G91	03
#4004	Read only	G22, G23	04
#4005	Read only	G94, G95	05
#4006	Read only	G20, G21	06
#4007	Read only	G40, G41, G42	07
#4008	Read only	G43, G44, G49	08
#4009	Read only	G73, G74, G76, G80~G89	09
#4010	Read only	G98, G99	10
#4011	Read only	G50, G51	11
#4012	Read only	G66, G67	12
#4013	Read only	G97	13
#4014	Read only	G16, G15	14
#4015	Read only	G61~G64	15
#4016	Read only	G68,G69	16
#4017	Read only	G54~G59	17
#4019	Read only	G150,G151,G152	19
#4020	Read only	G160,G161	20
#4021	Read only	G50.1, G51.1	21
…		…	…
#4107	Read only	D code	
#4109	Read only	F code	
#4111	Read only	H code	
#4113	Read only	M code	
#4114	Read only	N serial number	

#4119	Read only	S code	
#4120	Read only	T code	
#4130	Read only	Additional workpiece coordinate system P	
#4201	Pre-read only	G00, G01, G02, G03, G6.2	01
#4202	Pre-read only	G17, G18, G19	02
#4203	Pre-read only	G90, G91	03
#4204	Pre-read only	G22, G23	04
#4205	Pre-read only	G94, G95	05
#4206	Pre-read only	G20, G21	06
#4207	Pre-read only	G40, G41, G42	07
#4208	Pre-read only	G43, G44, G49	08
#4209	Pre-read only	G73, G74, G76, G80~G89	09
#4210	Pre-read only	G98, G99	10
#4211	Pre-read only	G50, G51	11
#4212	Pre-read only	G66, G67	12
#4213	Pre-read only	G97	13
#4214	Pre-read only	G16, G15	14
#4215	Pre-read only	G61~G64	15
#4216	Pre-read only	G68,G69	16
#4217	Pre-read only	G54~G59	17
#4219	Pre-read only	G150,G151,G152	19
#4220	Pre-read only	G160,G161	20
#4221	Pre-read only	G50.1, G51.1	21
…		…	…
#4307	Pre-read only	D code	
#4309	Pre-read only	F code	
#4311	Pre-read only	H code	
#4313	Pre-read only	M code	
#4314	Pre-read only	N serial number	
#4315	Pre-read only	Program number	
#4319	Pre-read only	S code	
#4320	Pre-read only	T code	

Example:

```
O300
G0 G90 G54 X0 Y0 Z0
G01 X10 F1000
#1 = #4201;      = →Group 01G mode (preread) #1 = 2.0
#2= #4001;       = →Group 01G mode (in execution) #2 = 1
G02 X20 Y20 R5
```

```
G01 X#1 Y#2 ;
M99 ;
%
```

13.4.12 Position information (#5001~#5140+n)

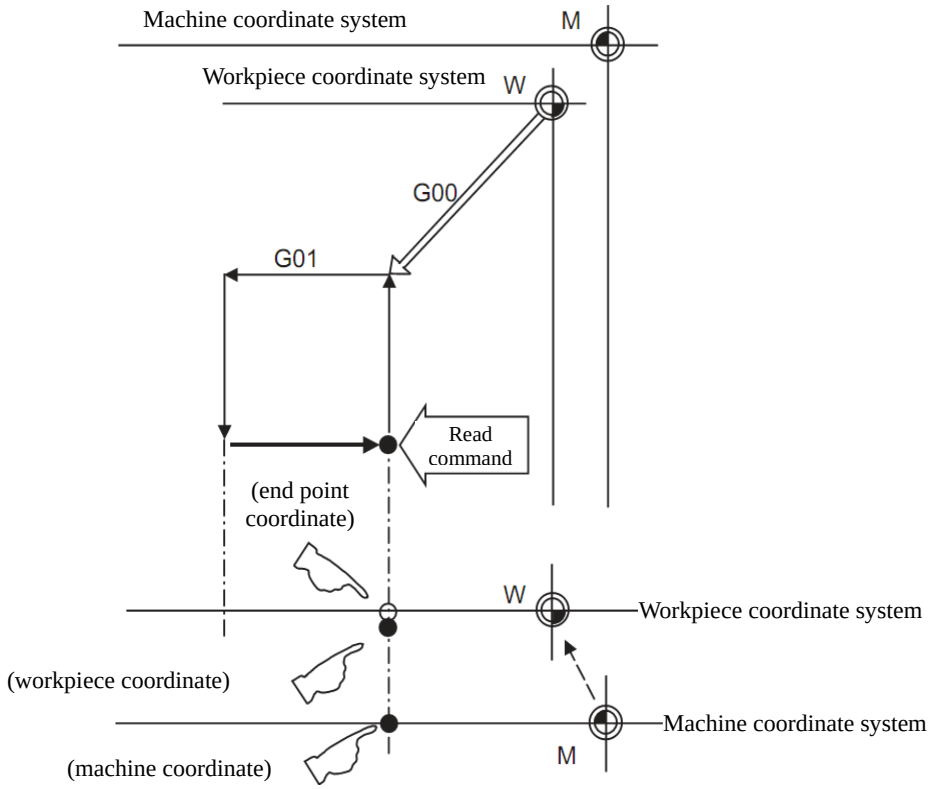
Detailed description

The variable number #5001~#5140+n can be used to read the end point coordinate value of the curren program block, current machine coordinate value, current workpiece coordinate value, jump coordinate value, tool position offset amount and servo offset amount.

Location	Axis No.						Tool Compensation Value	Moving Read
	1	2	3	4	…	n		
End point coordinate of the current program block	#5001	#5002	#5003	#5004	…	#5000+n	Not included	Yes
Current machine coordinate	#5021	#5022	#5023	#5024	…	#5020+n	Included	Yes
Current workpiece coordinate	#5041	#5042	#5043	#5044	…	#5040+n	Included	Yes
Jump coordinate	#5061	#5062	#5063	#5064	…	#5060+n	Included	Yes
Tool position compensation amount	#5081	#5082	#5083	#5084	…	#5080+n		Yes
Servo offset amount	#5101	#5002	#5103	#5104	…	#5100+n		Yes
External displacement	#5201	#5202	#5203	#5204	…	#5200+n		No
G50G92 displacement	#5221	#5222	#5223	#5224	…	#5220+n		No
Workpiece displacement	#5241	#5242	#5243	#5244	…	#5240+n		No
Local displacement	#5261	#5262	#5263	#5264	…	#5260+n		No

Note: Number of controllable axis changes with specifications.

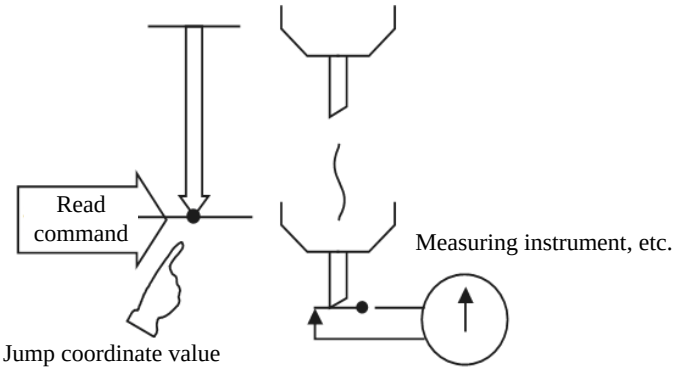
The digit 1 bit lower than the variable number is corresponding to the control axis number.



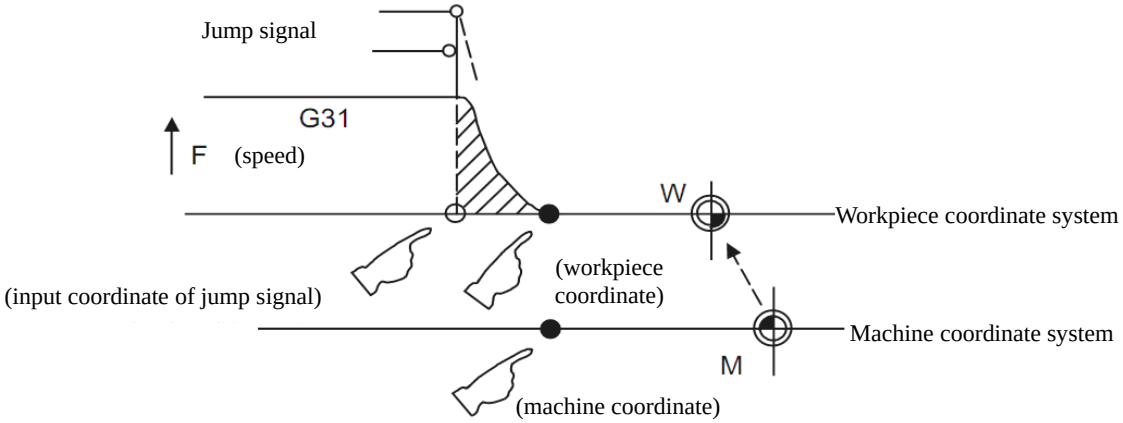
(1) End point coordinate and jump coordinate are positions in workpiece coordinate system.

(2) End point coordinate, jump coordinate and servo offset amount can be read even in movement. However, when movement is stopped, please read after confirmed the mechanical coordinate and the workpiece coordinate.

(3) Jump coordinate table means the position with jump signal enabled in G31 program block. When jump is disabled, it is the position of end point.



(4) End point coordinate refers to the tool nose position regardless of factors such as tool offset. Machinery coordinate, workpiece coordinate, and jump coordinate refer to the tool datum point position in consideration of the tool offset.



● Read after confirming that it stops.

○ It can also be read while moving.

Input coordinate value of the jump signal is a position in workpiece coordinate system. Coordinate values of #5061~#5064 are memorized at the moment when the jump input signal is input during machine movement, so they can be read at any time thereafter. For details, please refer to "20.2 Jump function (G31)" in Chapter 20 of this part.

13.4.13 Workpiece machining number (#3901, #3902)

Detailed description

Workpiece machining number can be read by using variable number #3901.

Maximum value of workpiece can be changed by substituting the value into the variable number #3902.

Type	Variable Number	Data Setting Range
Workpiece machining number	#3901	0～65535
Maximum value of workpiece	#3902	

Note: The value substituted in the workpiece machining number must be the correct value.

13.5 Operation command

Function and purpose

Various operations can be performed among variables.

Command format

#i = <formula>;

<Formula> is composed of constants, variables, functions or operators.

Constants can also be used to substitute #j, #k described below.

(1) Definition and replacement of variables	#i = #j	Definition, replacement
(2) Addition	#i = #j + #k	Addition

	#i = #j - #k	Subtraction
(3) Multiplication	#i = #j * #k	Multiplication
	#i = #j / #k	Division
(4) Remainder	#i = #j MOD #k	Remainder
(5) Logical operation	#i = #j OR #k	Logical sum (each bit of 32bit)
	#i = #j XOR #k	Exclusive OR
	#i = #j AND #k	Logical AND (each bit of 32bit)
(6) Functions	#i = SIN [#k]	Sine
	#i = COS [#k]	Cosine
	#i = TAN [#k]	Tangent tanθ equal to sinθ /cosθ
	#i = ASIN [#k]	Arc sine
	#i = ATAN [#k]	Arctangent
	#i = ACOS [#k]	Arc cosine
	#i = SQRT [#k]	Square root
	#i = ABS [#k]	Absolute value
	#i = BIN [#k]	Convert from BCD to BINARY
	#i = BCD [#k]	Convert from BINARY to BCD
	#i = ROUND[#k]	rounding
	#i = FIX [#k]	Discard below the decimal point
	#i = FUP [#k]	Round below decimal point
	#i = LN [#k]	Natural logarithm
	#i = EXP [#k]	Exponent to base e (e=2.718...)

Note 1: Any value without a decimal point is basically treated as a decimal point at the end (1=1.000).

Note 2: Data such as the offset amount starting from #10001 and the workpiece coordinate system offset value starting from #5201 are data with decimal points. Therefore, even if data without decimal point is defined to these variable numbers, it becomes data with decimal point.

(Example)

Operation command	Covariate after command execution
#101 =10 ; #10001 =#101 ; #102 =#10001 ;	#10001 10.000 #102 10.000

Note 3: The <formula> after the function must be enclosed in [ ].

Detailed description

1. Sequence of operations

(1) The order of operations from (a) to (c) is function, multiplication, and addition.

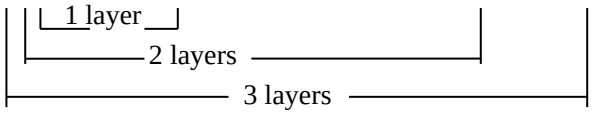
#101=#111+#112*SIN[#113]

(a) Function
(b) Multiplication
(c) Addition

(2) Use [ ] to enclose the part that you want to operate with priority. [ ], including the [ ] of functions, is

up to 5 layers.

#101=SQRT [[[#111- #112] * SIN[#113] + #114] * #115] ;



2. Examples of operation commands

(1) Specify the macro program and independent variables	G65 P100 A10 B20. ; #101=100.000 #102=200.000 ;	#1 10.000 #2 20.000 #101 100.000 #102 200.000	
(2) Define and substitute =	#1 = 1000 #2 = 1000.	#1 1000.000 #2 1000.000	
	#3 = #101 #4 = #102	#3 100.000 #4 200.000	According to covariates
	#5 = #10001 (#10001 = -10.)	#5 -10.000	According to tool offset
(3) Add and subtract + -	#11 = #1 + 1000 #12 = #2 - 50. #13 = #101 + #1 #14 = #10001 - 3. (#10001= -10.) #15 = #10001 + #102	#11 2000.000 #12 950.000 #13 1100.000 #14 -13.000 #15 190.000	
(4) Multiply and divide * /	#21 = 100 * 100 #22 = 100. * 100 #23 = 100 * 100. #24 = 100. * 100. #25 = 100 / 100 #26 = 100. / 100 #27 = 100 / 100. #28 = 100. / 100. #29=#10001*#101(#10001=-10.) #30 = #10001 / #102	#21 10000.000 #22 10000.000 #23 10000.000 #24 10000.000 #25 1.000 #26 1.000 #27 1.000 #28 1.000 #29 -1000.000 #30 -0.050	
(5) Remaining part MOD	#19 = 48 #20 = 9 #31 = #19 MOD #20	#19/#20 = 48/9 = 5 remainder 3 #31 = 3	
(6) Logical sum OR	#3 = 100 #4 = #3 OR 14	#3 = 01100100 (binary) 14 = 00001110 (binary)	
		#4 = 01101110 = 110	
(7) XOR XOR	#3 = 100 #4 = #3 XOR 14	#3 = 01100100 (binary) 14 = 00001110 (binary)	
		#4 = 01101010 = 106	

(8) Logical product AND	#9 = 100 #10 = #9 AND 15	#9 = 01100100 (binary) 15 = 00001111 (binary) #10 = 00000100 = 4
(9) Sine SIN	#501 = SIN [60] #502 = SIN [60.] #503 = 1000 * SIN [60] #504 = 1000 * SIN [60.] #505 = 1000. * SIN [60] #506 = 1000. * SIN [60.] (Note) SIN [60] is equivalent to SIN [60.].	#501 0.866 #502 0.866 #503 866.025 #504 866.025 #505 866.025 #506 866.025
(10) Cosine COS	#541 = COS [45] #542 = COS [45.] #543 = 1000 * COS [45] #544 = 1000 * COS [45.] #545 = 1000. * COS [45] #546 = 1000. * COS [45.] (Note) COS [45] is equivalent to COS [45.].	#541 0.707 #542 0.707 #543 707.107 #544 707.107 #545 707.107 #546 707.107
(11) Tangent TAN	#551 = TAN [60] #552 = TAN [60.] #553 = 1000 * TAN [60] #554 = 1000 * TAN [60.] #555 = 1000. * TAN [60] #556 = 1000. * TAN [60.] (Note) TAN [60] is equivalent to TAN [60.].	#551 1.732 #552 1.732 #553 1732.051 #554 1732.051 #555 1732.051 #556 1732.051
(12) Arc sine ASIN	#531 = ASIN［100.500 / 201.］ #532 = ASIN［100.500 / 201］ #533 = ASIN［0.500］ #534 = ASIN［-0.500］	#531 30.000 #532 30.000 #533 30.000 #534 -30.000
(13) Arc tangent ATAN	#561 = ATAN [173205 / 100000] #562 = ATAN [173205 / 100000.] #563 = ATAN [173.205 / 100] #564 = ATAN [173.205 / 100.] #565 = ATAN [1.73205]	#561 60.000 #562 60.000 #563 60.000 #564 60.000 #565 60.000
(14) Arc cosine ACOS	#521 = ACOS [100 / 141.421] #522 = ACOS [100. / 141.421]	#521 45.000 #522 45.000

(15) Square root SQRT	#571 = SQRT [1000] #572 = SQRT [1000.] #573 = SQRT [10.*10.+ 20.*20] **Note:** In order to improve accuracy, please do the operation in [ ] as much as possible.	#571 #572 #573	31.623 31.623 22.360
(16) Absolute value ABS	#576 = -1000 #577 = ABS [#576] #3 = 70. #4 = -50. #580 = ABS [#4 - #3]	#576 #577 #580	-1000.000 1000.000 120.000
(17) BIN BCD	#1 = 100 #11 = BIN [#1] #12 = BCD [#1]	 #11 #12	 64 256
(18) Rounding ROUND	#21 = ROUND [14 / 3] #22 = ROUND [14. / 3] #23 = ROUND [14 / 3.] #24 = ROUND [14. / 3.] #25 = ROUND [-14 / 3] #26 = ROUND [-14. / 3] #27 = ROUND [-14 / 3.] #28 = ROUND [-14. / 3.]	#21 #22 #23 #24 #25 #26 #27 #28	5.000 5.000 5.000 5.000 -5.000 -5.000 -5.000 -5.000
(19) Discard anything below the decimal point FIX	#21 = FIX [14 / 3] #22 = FIX [14. / 3] #23 = FIX [14 / 3.] #24 = FIX [14. / 3.] #25 = FIX [-14 / 3] #26 = FIX [-14. / 3] #27 = FIX [-14 / 3.] #28 = FIX [-14. / 3.]	#21 #22 #23 #24 #25 #26 #27 #28	4.000 4.000 4.000 4.000 -4.000 -4.000 -4.000 -4.000
(20) Rounding up below the decimal point FUP	#21 = FUP [14 / 3] #22 = FUP [14. / 3] #23 = FUP [14 / 3.] #24 = FUP [14. / 3.] #25 = FUP [-14 / 3] #26 = FUP [-14. / 3] #27 = FUP [-14 / 3.] #28 = FUP [-14. / 3.]	#21 #22 #23 #24 #25 #26 #27 #28	5.000 5.000 5.000 5.000 -5.000 -5.000 -5.000 -5.000
(21) Natural logarithm LN	#10 = LN [5] #102 = LN [0.5] #103 = LN [-5]	#101 #102 Error	1.609 -0.693 Warning

(22) Exponent EXP	#104 = EXP [2] #105 = EXP [1] #106 = EXP [-2]	#104 #105 #106	7.389 2.718 0.135

Precautions

Precautions to logical operations:

The same calculations as addition and subtraction are also performed for EQ, NE, GT, LT, GE, LE, so be careful of the error. For example, when judging whether #10 and #20 are equivalent, it is impossible to judge whether it is correct or not because of the error in the following formula.

According to IF [#10 EQ #20] ,

As shown in the following formula, if the error between #10 and #20 is within the range, please perform equivalent processing of #10 and #20.

IF [ABS [#10 - #20] LT 200000]

13.6 Control command

Function and purpose

The flow of the program can be controlled by IF~GOTO~ and WHILE~DO~END.

Detailed description

13.6.1 Branch

IF [conditional expression] GOTO n; (n is the sequence number in the program)
IF [[conditional expression] conditional operator [conditional expression]] GOTO n; (conditional operators comprise EQ, NE, GT, LT, GE, LE)
IF [[conditional expression] operator [conditional expression]] GOTO n; (operators comprise +, -, *, /, AND, OR, XOR)

When the condition is satisfied, jump to n, if not, execute the next program block.

IF[conditional expression] can be omitted, in this case jump to n unconditionally.

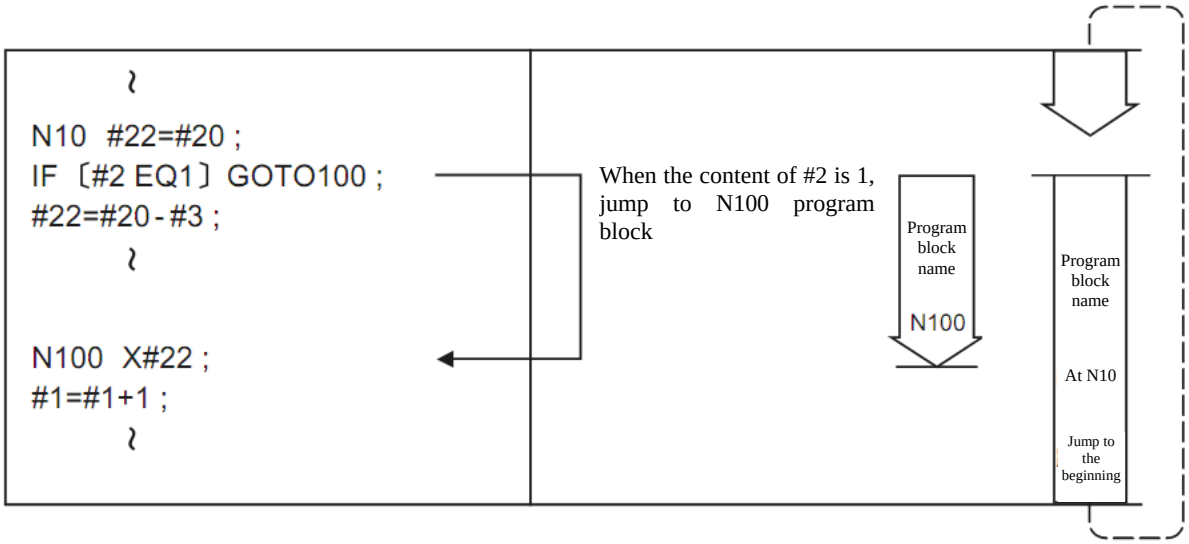
The types of [conditional expressions] are listed as follows.

#i EQ #j	= when #i is equal to #j
#i NE #j	≠ when #i and #j are not equal
#i GT #j	> when #i is greater than #j
#i LT #j	< when #i is less than #j
#i GE #j	≧ when #i is greater than or equal to #j
#i LE #j	≦ when #i is less than or equal to #j

n of GOTO n must be within the same program. Otherwise, a program error will occur. Expressions or variables can be used in place of #i, #j, n.

In the block of sequence number n executed after GOTO n, the sequence number Nn must be at the beginning of the program block. Otherwise, a program error will occur.

However, when the beginning of the program block is "/" and there is Nn after it, it can jump to the sequence number.



Note: When searching for the sequence number of the branch address, start from the next program block of IF.....; and search to the end of the program (% code), otherwise it will search from the beginning of the program to IF.....; previous program block. Therefore, the reverse search branch to the program flow takes longer to execute than the forward branch search.

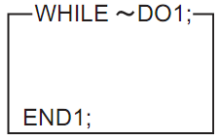
13.6.2 Repeat

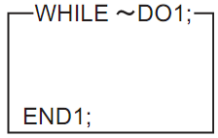
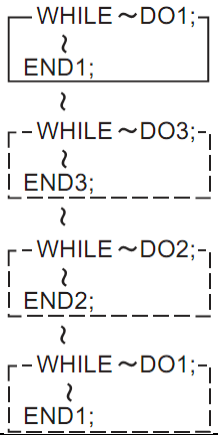
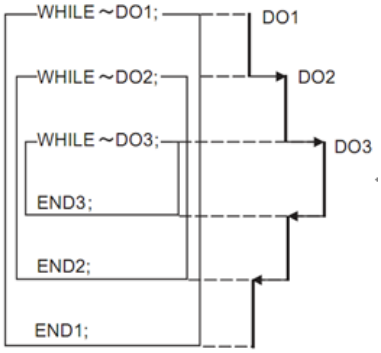
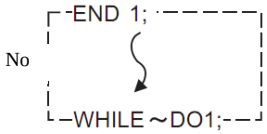
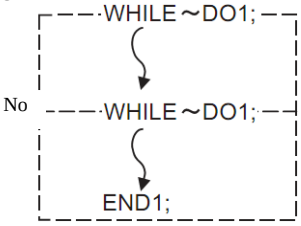
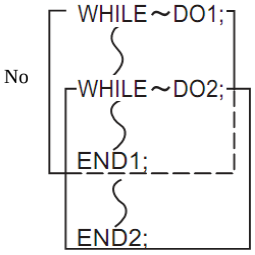
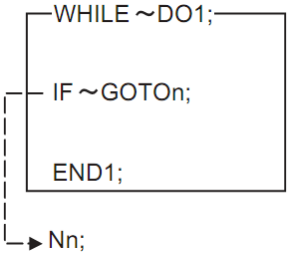
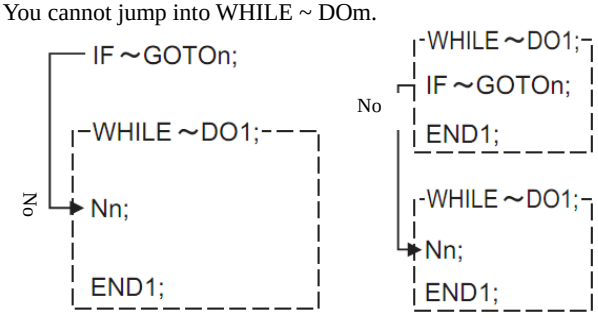
```
WHILE [conditional expression] DOm ; (m = 1, 2, 3)
    ⋮
END m ;
```

When the conditional expression is established, the ENDm program block will be repeated from the next program block, and if it is not established, the next program block of ENDm will be executed. It doesn't matter that DOm lies before WHILE.

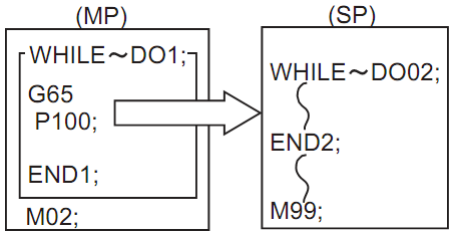
WHILE[conditional expression] DOm and ENDm should be used in pairs. If WHILE[conditional expression] is omitted, DOm～ENDm will be repeated infinitely. The duplicate identification numbers are 1~3. (DO1, DO2, DO3), if a number other than 1, 2, and 3 is used as the identification number, a program error will occur.

Identification numbers 1~3 can be used multiple times in the DO~END loop.

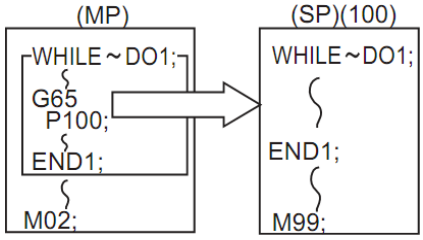
<table>
<tr>
<td>Use the same identification number multiple times.
 


</td>
<td>The identification number of WHILE ~ DOm can be freely selected.
 
</td>
</tr>
<tr>
<td>The nesting depth of WHILE ~ DOm is at most 3 layers.
 
</td>
<td>You must specify WHILE ~ DOm first, and then specify ENDm.
 
</td>
</tr>
<tr>
<td>WHILE ~ DOm and ENDm must appear in pairs in the same program.
 
</td>
<td>Two WHILE ~ DOm cannot be used interchangeably.
 
</td>
</tr>
<tr>
<td>You can jump out of the range of WHILE ~ DOm.
 
</td>
<td>You cannot jump into WHILE ~ DOm.
 
</td>
</tr>
</table>

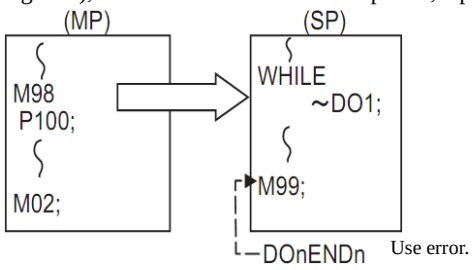
Between WHILE ~ DOm, subprograms can be called through M98, G65 and G66.



Between WHILE ~ DOm, the macro program call is executed through G65 and G66, and can be specified from 1. The main program and subprograms can be nested up to 3 layers.



In subprograms (including macro programs), if WHILE and END are not paired, a program error will occur due to M99.



(MP) main program (SP) subprogram

Note: When calling a fixed cycle containing WHILE, the number of nesting levels is also accumulated.

13.7 Precautions

Precautions

By using the user macro program commands, a machining program can be created by combining control commands such as previous movement commands and MST commands with macro commands such as operation, judgment, and branching. If the former is an execution statement and the latter is a macro program statement, the processing of the macro program statement has nothing to do with the direct machine control, so the processing can be executed in the shortest possible time, which can effectively shorten the processing time.

Program example

```
N1 G91 G28 X0 Z0 ;                    .....(1)
N2 G50 X0 Z0 ;                        .....(2)
N3 G00 X-100. Z-100. ;                .....(3)
N4 #101 = 100. * COS[210.] ;          .....(4)(4), (5) Macro program statements
N5 #103 = 100. * SIN[210.] ;          .....(5)
N6 G01 X#101 Z#103 F800 ;             .....(6)
```

Macro program statements refer to the following statements:

(a) Operation command (program blocks containing =)

(b) Control commands (program blocks containing GOTO, DO~END, etc.)

(c) It includes the macro program call or cancel command (G65, G66, G67) executed by macro call command (G code, etc.), and the execution statements refer to those other than macro program statements.

Chapter XIV Spindle Speed Function

14.1 Function of the spindle

Function and purpose

The function used to control the spindle speed, also known as the S function, consists of the address S and its suffix number. According to the processing needs, the rotational speed of the spindle is divided into two types: the linear speed *v* and the rotational speed S.

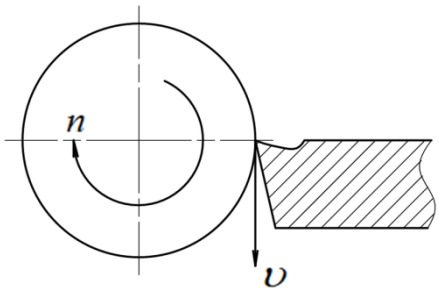
(1) Rotational speed S: The unit of rotational speed S is r/min, which is specified by the preparatory function G97, and its value is a constant greater than 0, such as: G97 S1000; the spindle speed is 1000 r/min.

(2) Constant linear speed *v:* In order to ensure the machining quality of the workpiece surface during processing, the rotational speed is specified by the constant linear speed. The unit of the constant linear speed is m/min, which is specified by the preparatory function G96.

When programming with the constant linear speed, in order to prevent the accident caused by the high speed S, it is set with the maximum speed limit command "G50 S_;".

G96 S100; the spindle speed is 100 m/min.

The linear speed *v* and the rotational speed S can be converted to each other, and the conversion relationship is shown in the figure.



$v = \pi Dn/1000$

n=1000v/πD

In the formula: v——cutting linear speed, in unit of m/min;

D - diameter of the tool, in unit of mm;

n - spindle speed, in unit of r/min.

During programming, the spindle speed is not allowed to be represented by a negative value, and S0 is allowed to stop the speed, but it is generally not used.

In the actual operation process, the spindle speed value can be corrected through the spindle override switch on the machine operation panel. Generally, the adjustment range is 50%~120%.

(3) Spindle start and stop: In the program, the clockwise rotation, counterclockwise rotation and stop of the spindle are controlled by auxiliary functions M03/M04/M05. Among them, M3 means the spindle rotates forward, M04 means the spindle rotates reversely, and M05 means the spindle stops.

Example: G97 M03 S300; The spindle rotates clockwise, and the speed is 300r/min

M05; spindle stops

14.2 Spindle function S2 digit gear control

Function and purpose

In the numerical control device, the address S and the continuous 2-digit numbers, any group from 0 to 99 (100 groups in total), can be used to specify the spindle speed. However, the number of groups that can be used in these 100 groups and the actual corresponding values are determined by the mechanical specifications. When

the number of digits exceeds 2, only the last 2 digits are valid.

The S function can be specified at the same time with other commands. When it is specified in the same program block as the movement command, the execution sequence of the command is as follows:

After the movement is completed, the S function is executed.

The S function is executed at the same time as the movement command.

Which method to use depends on the mechanical specifications.

The S commands from S00~S99 all need to be processed by sequential control.

14.3 Spindle function S5 digit numerical control

Function and purpose

When the S5-digit function is valid, the spindle speed can be specified by commanding 5 digits after S. For other functions, refer to the S2 function.

This function outputs the rotational speed signal and start signal corresponding to the commanded speed of the spindle according to the S code and the numerical command of 5 continuous digits, with the appropriate gear signal, etc.

All S commands must be processed by sequential control.

During the execution of the S command, when the gear shift is manually operated, the set rotational speed corresponding to the gear position is compared with the previous command rotational speed to obtain an appropriate rotational speed signal.

There are four gear positions at most, with various preset parameters. Proper gear position selection signal will be sent by NC according to the S command.

(1) Relevant parameters of each gear stage...limit speed, maximum speed, gear shift speed, positioning speed and tapping maximum speed.

(2) Parameters corresponding to all gears...minimum speed.

1. Analog voltage output speed signal

The system converts the rotational speed into a corresponding analog voltage to the spindle drive controller.

The specifications of the analog signal are as follows:

(1) Output voltage……0~10V

(2) Resolution……1/4096 (2^{12})

(3) Load condition……10KΩ

(4) Output impedance……220Ω

2. Ethernet bus output speed signal

The system communicates directly with the spindle drive controller to transmit digital information on the spindle speed.

14.4 Constant cutting linear speed control (G96, G97)

Function and purpose

For cutting in the radial direction, the number of revolutions of the spindle is automatically controlled with the change of the radius coordinate, so that the speed of the cutting point is kept constant during processing.

Command format

G96 S; constant cutting linear speed is valid S : cutting linear speed (S0~S9999, m/min)

G97; cancel the constant cutting linear speed S : the given spindle speed (S0~S9999, rpm)

Program examples

G50 S2000;(the maximum spindle speed is limited to 2000 rpm)
M03 G96 S200;(the constant surface cutting speed control is valid, and the surface speed is 200 m/min)
G97 S1000;(cancel the constant surface cutting speed, and specify the spindle speed as 1000 rpm)

Detailed description

(1) G96 and G97 are modal values of the same group. G97 is the initial state command, G97 is valid when the system is powered on.

(2) In G96 mode, the S value is the cutting linear speed. The system calculates and executes the spindle speed according to the absolute value of the X-axis absolute coordinate value of the programmed path (ignoring tool length compensation), and keeps the cutting linear speed as the S value specified by G96. The formula is as follows:

$$\text{Spindle rotational speed} = \frac{\text{Cutting linear speed} \times 1000}{|X| \times \pi}(rpm)$$

Where:

$|X|$: Absolute value (diameter value) of the X-axis absolute coordinate value, in unit of mm.

Cutting linear speed: S value specified by G96, in unit of m/min.

(3) For the constant surface cutting speed control, the spindle speed is changed only during the cutting feed (interpolation) process with the change of the absolute value of the X-axis absolute coordinate value of the programmed path. For G00 rapid movement, since no actual cutting is performed, G00 is in the process of execution, the spindle speed remains unchanged, and the spindle speed now is calculated according to the linear speed at the end of the program block.

(4) For the constant surface cutting speed control, the Z-axis of the workpiece coordinate system must coincide with the spindle axis (workpiece rotation axis).

(5) For the thread cutting, the constant surface cutting speed control function is still valid, but in order to ensure the thread machining accuracy, thread cutting should be performed in the G97 state.

(6) When the G97 program block executed in the G96 mode has no S code (rpm), the current speed is used as the S code in the G97 state, that is, the spindle speed remains unchanged now.

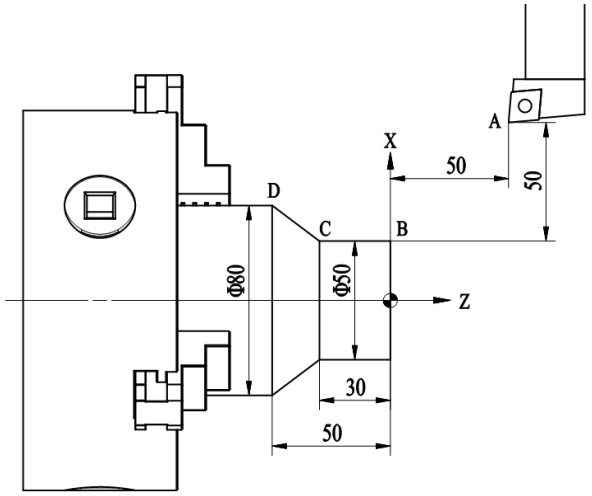
In the G96 state, the commanded S value is kept even in the G97 state. When returning to the G96 state, its value is restored; for example:

G96 S50; (Surface cutting speed 50 m/min)
G97 S1000;(Spindle speed 1000 r/min)
G96 X3000;(Cutting linear speed 50 m/min)

Programming examples

Example: As shown in the figure, the path of the tool cutting workpiece is A→B→C→D. Try the constant

surface cutting speed control function G96 to write the machining program, the surface cutting speed is 300 m/min, and the maximum spindle speed is 3000 r/min.



Machining parts diagram

```
N10  G50 S3000;          (The maximum spindle speed is limited to 3000 r/min)
N20  M3 G96 S300;        (The rotating spindle and constant surface cutting speed control is valid, and
                          the surface speed is 300 m/min)
N30  G0 X100 Z50;        (Quickly move to point A, the spindle speed during the movement is 955
                          r/min)
N40  G0 X50 Z0;          (Quickly move to point B, the spindle speed during the movement is 1910
                          r/min)
N50  G1 W-30 F200;       (Cutting from point B to point C, the spindle speed is always 1910r/min
                          during cutting)
N60  X80 W-20 F150;      (Cut from point C to point D, and the spindle speed changes linearly from
                          1910 r/min to 1194 r/min)
N70  G0 X100 Z50;        (Quickly return to point A, and the spindle speed during the movement is 955
                          r/min)
N80  G97 S1500;          (Cancel the constant surface cutting speed control, and specify the spindle
                          speed as 1500r/min)
N90  M30;                (The program ends, turn off the spindle and coolant)
%
```

14.5 Spindle clamping speed setting (G50)

Function and purpose

Address S following G50 specifies the maximum clamping speed of the spindle, and address Q specifies the minimum clamping speed of the spindle.

Command format

```
G50 S   Q ;
S       : Maximum clamping speed
```

Q	: Minimum clamping speed

Detailed description

(1) The corresponding gear change is used between spindle and spindle motor, and there are at most four parameters in the speed range, which are set in units of 1 rpm.

(2) Setting of the speed range parameter and G50 S_ Q_; in the speed range between the two, the upper limit is valid according to the lower one, and the lower limit is valid according to the higher one.

(3) When the spindle speed is higher than the S value specified by G50, the spindle speed is limited to this value; when the spindle speed is lower than the spindle safety limit speed set by parameter #5118, the actual spindle speed is limited to this value.

(4) When the system is powered on, the maximum spindle speed limit value is not set, and the spindle maximum speed limit function is invalid. The maximum speed limit value defined by G50 S_ is maintained before re-specification. The maximum speed limit function is valid in G96 state. In G97 state, the maximum spindle speed set by G50 S_ does not have the limiting effects, but the maximum spindle speed limit value is still kept.

14.6 Spindle speed fluctuation detection function (G25/G26)

Function and purpose

Due to the influence of the conditions of the machine, the actual speed of the spindle may be lower or higher than the commanded speed. When the system detects that the fluctuation exceeds a certain range, an abnormal spindle speed alarm will be triggered.

Command format

G26;	The spindle speed fluctuation detection function is valid
G25;	The spindle speed fluctuation detection function is cancelled

Program examples

G26;
G25;

Detailed description

Use parameters to set the speed fluctuation detection as valid and the G code control.

Spindle fluctuation alarm signal SPAL<F35.0> in PLC

This signal indicates that the actual spindle speed has exceeded the specified speed allowable range.

Position encoder selection signal PC2SLC in PLC <G028#7>

The fluctuation detection selection signal for multi-spindle control.

Parameter setting:

Press the 

key to go to the offset and setting interface, find [Login] on this interface, and obtain permission on the login interface by pressing [Login]. Press the 

key to go to the system interface. In

this interface, find [Parameters], and enter the parameter interface by pressing [Parameters]--->[Operations]--->[All Parameters] keys. Enter the data you want to search in the input field, press [Search] to find the corresponding parameter, move the cursor to the relative position by moving the [Up and Down] keys, input data in the input field, and press the 

key to modify the parameter or parameter bit.

Spindle speed fluctuation detection valid flag: 5001#0

Spindle speed display selection (command speed/actual speed): 5001#3

Set the spindle speed range q for starting the spindle speed fluctuation detection: 5010

Allowable spindle speed fluctuation rate r of the spindle speed fluctuation detection: 5011

Allowable spindle speed fluctuation value d of the spindle speed fluctuation detection: 5012

Time (p) from spindle speed command change to spindle speed fluctuation detection start: 5013

Note: Alarm: 112 The spindle speed is abnormal.

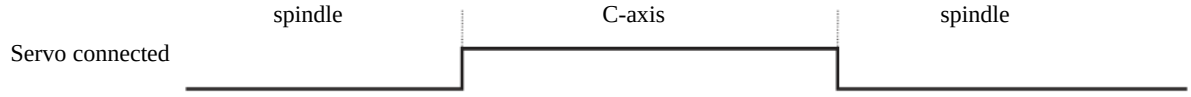
14.7 Spindle/C-axis control

Function and purpose

This function is that the spindle adopts the spindle servo controller with position control function to turn the spindle into a C-axis control axis (rotation axis) through an external signal.

Detailed description

1. Spindle/C-axis switching



When the servo is OFF ..... Spindle (C-axis control cannot be executed)

When the servo is connected ..... C-axis (Spindle control cannot be executed)

The C-axis is in the incomplete state of reference point return.

(1) Status of reference point return

When the Z phase fails, it is the status of reference point return uncompleted.

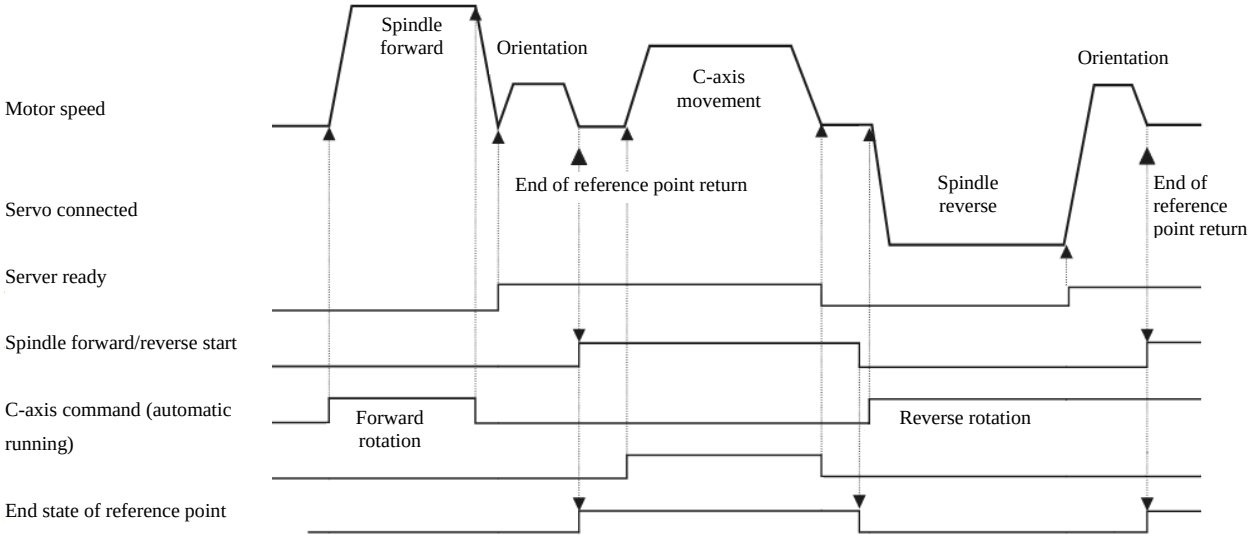
When the Z-phase passes, it is the status of reference point return completed.

(2) C-axis position data

Even if the spindle is rotated during spindle control, the position data of the C-axis in the NC is still updated.

The coordinate value counter of the C-axis is held during the spindle control, and the moving part of the spindle control is updated when the C-axis servo is ready. (When the servo is connected, the C-axis position will be different from the position before the last servo off)

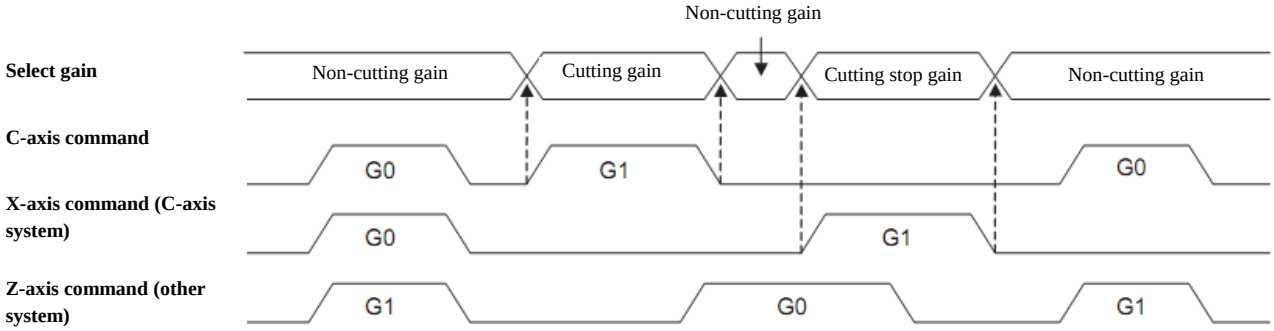
2. Switching timing diagram



Note: A program error occurs when the C-axis servo start command and the C-axis command are continuously executed because the reference point return completion is checked during the calculation of the axis command.
In such a case, the following two processes need to be performed by the user's PC:
- Input the recalculation request signal through the servo connected command input;
- Before the C-axis enters the servo ready state, wait for the servo connected command.

3. C-axis gain switching selection

Switch the C-axis gain (select the optimum gain) according to the cutting conditions of the C-axis. Select the cutting gain when the C-axis cutting feed is used, select the cutting stop gain when another axis performs the cutting feed (C-axis face cutting), and select the non-cutting gain in other cases.



Note 1: The cutting feed of other systems is not affected by the C-axis gain selection.
Note 2: The cutting gain includes the 1st to the 3rd, which can be selected by the ladder diagram.

Precautions and restrictions

(1) If the detector (encoder or other) has no Z-phase signal specification, the positioning function cannot be performed, so the reference point return operation cannot be performed.
(2) A program error alarm will be generated when the C-axis command is executed while the servo is off or in the reference point return positioning mode.
(3) Do not run the axis with the C-axis command, and execute the servo-off operation.
Because in the C-axis control mode, if the servo signal is turned off, the C-axis will stop feeding and switch to the spindle control mode. The remaining movement amount of the C-axis command will be cleared when the servo is turned on.
(4) If the servo-on operation is performed during the spindle rotation, the spindle rotation will stop immediately, and the operation mode will be switched to the C-axis control mode.

Chapter XV Tool Function

15.1 Tool selection function

Function and purpose

The tool selection function refers to the function command for the system to select or change the tool, also known as the T function.

The milling system is used to specify the tool number, the command is issued through the 4-digit value (0~9999) after the address T, the 4-digit value is used as the tool number, and the leading zero can be omitted.

The turning system is used to specify the tool number and tool compensation number, the command is issued through the 4-digit value (0~9999) after the address T, the upper 2 digits are used as the tool number, and the lower 2 digits are used as the compensation number.

Command format

Milling

```
Toooo;
  ↑
  └──── Tool selection
```

Turning

```
Too oo;
  ↑  ↑
  │  └──Offset number of the tool
  └──── Tool selection
```

Detailed description

Only one T code can be specified in a program block. For the number of digits that can be commanded after the T address and the correspondence between T codes and machine operations, see the machine manufacturer's commands. When the movement command and the T code are commanded in the same program block, the command is executed according to one of the following two methods. Which one to choose depends on the machine manufacturer's specifications. Refer to the machine manufacturer's instructions for details.

(1) Execute the movement command and the T function command at the same time.

(2) The T function command is executed when the execution of the movement command is completed.

All T commands must be processed and completed by the PLC.

Program examples

Turning example

```
N1 G00 X1000 Z1400;
N2 T0313;(Select the offset values for the tools No. 3 and No. 13)
N3 X400 Z1050;
```

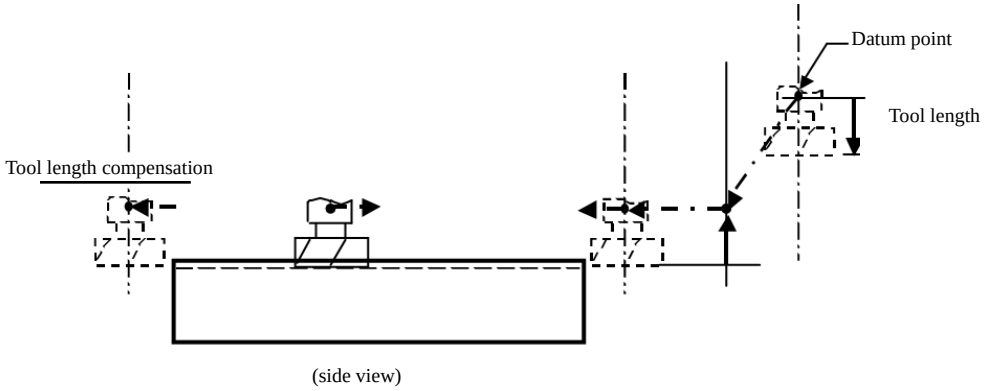
Some machines use one-digit commands to select tools.

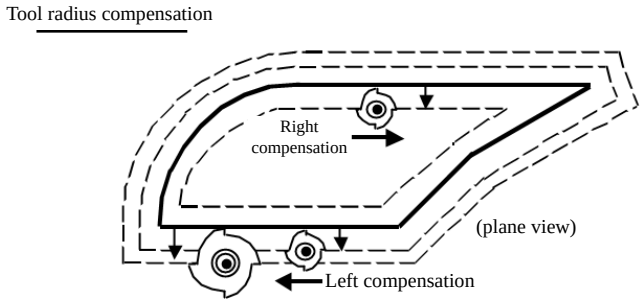
Chapter XVI Milling Tool Offset Function

16.1 Tool compensation

Function and purpose

As shown in the figure below, the basic tool compensation functions include tool length compensation and tool radius compensation, and each compensation amount is specified according to the tool compensation number. Each compensation amount is input from the setting offset interface.





Tool compensation data

As shown in the table below, the shape compensation amount related to tool length, wear compensation amount, shape compensation amount related to tool radius, and wear compensation amount corresponding to one compensation number can be set independently.

H is used to select the tool length compensation amount, and D to select the tool radius compensation amount.

(H1)=b1+c1,(D1)=d1+e1

(H2)=b2+c2,(D2)=d2+e2

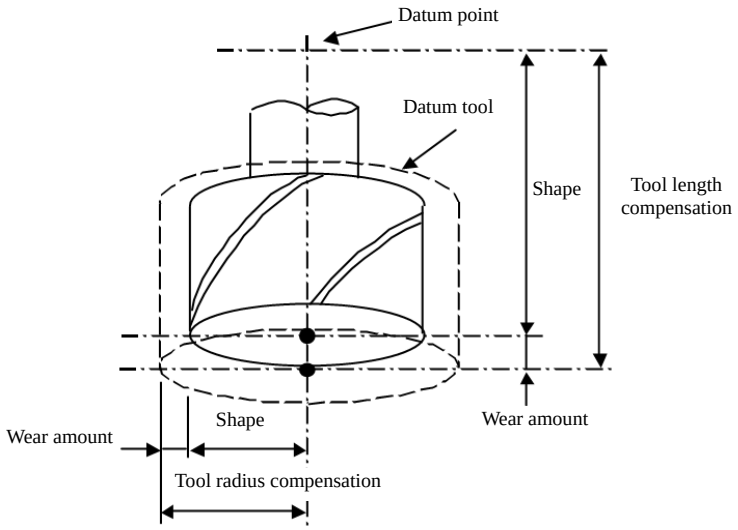
…

…

(Hn)=bn+cn,(Dn)=dn+en

Compensation Number	Tool Length (H)		Tool Radius (D)/(Position Compensation)	
	Shape Compensation Amount	Wear Compensation Amount	Shape Compensation Amount	Wear Compensation Amount
1	b1	c1	d1	e1

2	b2	c2	d2	e2
3	b3	c3	d3	e3
*	*	*	*	*
*	*	*	*	*
n	bn	cn	dn	en



Tool offset number (H/D)

Specify the address of the tool offset number.

(1) H is used for tool length compensation, and D is used for tool position compensation and tool radius compensation.

(2) After specifying once, the tool compensation number will not change until a new H or D command is specified.

(3) The compensation amount corresponding to each compensation number is set in advance through the setting offset interface.

16.2 Tool length compensation/cancellation (G43, G44/G49)

Function and purpose

With this command, the end point position of each axis movement command can be pre-compensated according to the set compensation amount. When programming using this function, by setting the deviation between the considered tool length value and the actual value as the compensation amount, the versatility of the program can be improved.

Command format

Tool length compensation A

G43 Z_ H_;	Tool length compensation + start
G44 Z_ H_;	Tool length compensation - start

Tool length compensation B

G17 G43 Z_ H_;	Tool length compensation in the Z positive direction starts
G17 G44 Z_ H_;	Tool length compensation in the Z negative direction starts
G18 G43 Y_ H_;	Tool length compensation in the Y positive direction starts
G18 G44 Y_ H_;	Tool length compensation in the Y negative direction starts
G19 G43 X_ H_;	Tool length compensation in the X positive direction starts
G19 G44 X_ H_;	Tool length compensation in the X negative direction starts

Tool length compensation mode cancelled

G49 or H0;	Tool length compensation cancelled

Use parameter #2600.1_TLB to set tool length offset method A or B.

Detailed description

(1) Movement amount of the tool length compensation

When the tool length compensation start command of G43 or G44 and the tool length compensation cancel command of G49 are executed, the movement amount is calculated according to the following formula.

Z-axis movement amount

G43 Zz Hh1; z +(h1) .The tool compensation amount is only compensated in the + direction

G44 Zz Hh1; z -(h1) The tool compensation amount is only compensated in the - direction

G49 Zz; z -(+)(h1) Cancellation of compensation

h1: Compensation amount of compensation number h1

As shown in the above operation, regardless of whether an absolute value command or an incremental value command is used, the actual end point is the coordinate value of the end point coordinate of the programmed movement command after the specified compensation amount is compensated.

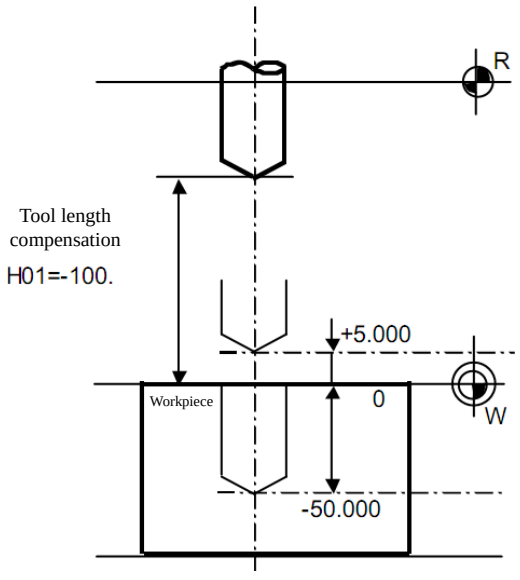
The G49 (tool length compensation cancel) mode is entered when the power is turned on.

(Example 1) In the case of absolute value commandH01=-100.

```
N1  G28  Z0  T01  M06;
N2  G90  G92  Z0;
N3  G43  Z5.  H01;
N4  G01  Z-50.  F500;
```

(Example 2) In the case of incremental value commandH01=-100.

```
N1  G28  Z0  T01  M06;
N2  G91  G92  Z0;
N3  G43  Z5.  H01;
N4  G01  Z-55.  F500;
```



(2) Compensation number

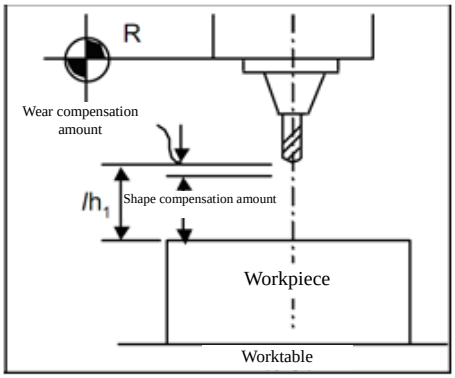
(a) Compensation amount.

G43 Hh1;

When commanded, the compensation amount /h1 commanded with the compensation number h1

/h1=shape compensation amount+wear compensation amount.

The wear compensation amount range is set by parameter #2611. If the value is 100, the actual range of wear compensation is: -100~100. The range of shape compensation amount is -9999999.9999~9999999.9999.



(b) The valid range of compensation numbers depends on the specification (number of compensation groups).

(c) A program error occurs when the commanded compensation number exceeds the specification range.

(d) When H0 is specified, the tool length will be cancelled.

(e) The compensation number specified in the same program block as G43 or G44 is valid as the following modal.

(3) Valid axes for tool length compensation

(a) When parameter "#2600.1_TLB" is "0", the tool length compensation is always valid for the Z-axis.

(b) When parameter "#2600.1_TLB" is "1", it depends on the axis perpendicular to the current work plane (G17, G18, G19).

(Example 3)

```
G19 G43 Xx1 Hh1;      ……+ compensation for the X-axis
    ⋮
G49 Xx2;
    ⋮
G18 G44 Yy1 Hh2;      ……- compensation for the Y-axis
    ⋮
G49 Yy2;
    ⋮
G17 G43 Xx3 Yy3 Zz3;      ……Compensation for the Z-axis.
    ⋮
G49;
```

16.3 Tool radius compensation (G39/G40/G41, G42)

Function and purpose

The tool radius compensation function can compensate the tool radius amount selected by G commands (G39~G42) and D commands in any vector direction.

Command format

G40 X__Y__;	Tool radius compensation cancelled
G41 X__Y__;	Tool radius compensation (left)
G42 X__Y__;	Tool radius compensation (right)
G39 X__Y__;	Corner switch Only available in the radius compensation mode

Detailed description

In the tool radius compensation, H command is ignored, and only D command is valid.

Moreover, the plane selection for tool radius compensation can be compensated according to the G code or the plane with the specified plane position of the 2-axis, and the axis command that is not in the selected plane will not be compensated. For the method of plane selection by G code, please refer to "4.3 Plane selection (G17, G18, G19)" in Chapter 4 of this part.

16.3.1 Tool radius compensation action

Tool radius compensation cancellation state

Under any of the following conditions, the tool radius compensation becomes the compensation cancellation mode.

(1) After turning on the power

(2) After pressing the reset button of the setting display device

(3) After executing M02 and M30 with reset function

(4) After executing the compensation cancellation command (G40)

In compensation cancellation mode, the compensation vector is 0, and the tool center path is consistent with the program path.

Be sure to end the program including tool radius compensation in the compensation cancellation state.

Start of tool radius compensation (start)

In the compensation cancellation state, the tool radius compensation starts when all the following conditions are met.

(1) Movement command after G41 or G42 command.

(2) The compensation number of tool radius compensation is 0 < D ≤ maximum compensation number.

(3) Positioning (G00) or linear interpolation (G01) movement command.

When starting the compensation, no matter whether it is continuous operation or single block operation, the command with the movement amount must be read before the compensation can be established.

Also in the compensation mode, up to 3 blocks are pre-read, and the compensation calculation is performed.

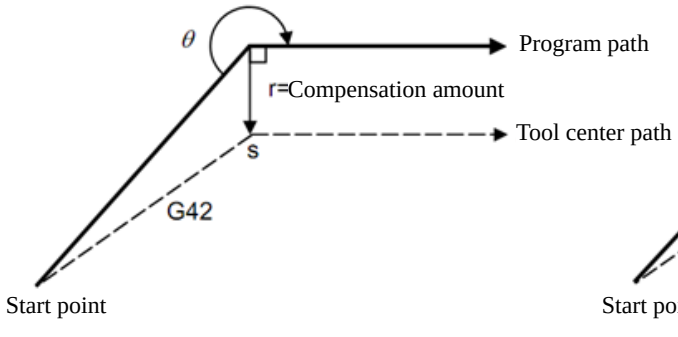
Compensation start action, including type A and type B. Select by setting parameter #2600.0. This parameter is common to the type of compensation cancellation action.

In the following explanatory diagrams, S represents a stop point when a single-block stop occurs.

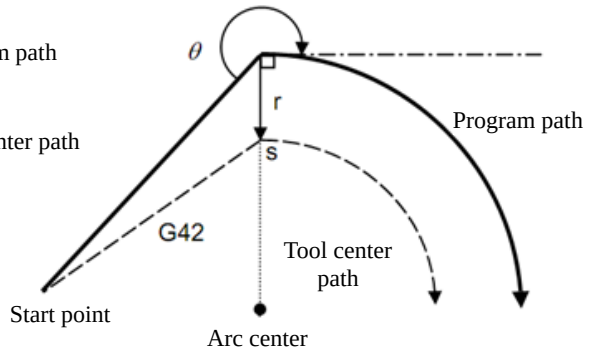
Start action of tool radius compensation

(1) When inside the corner

Straight line→ straight line

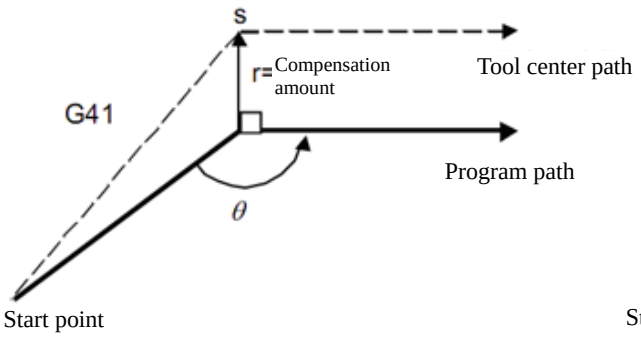


Straight line → arc

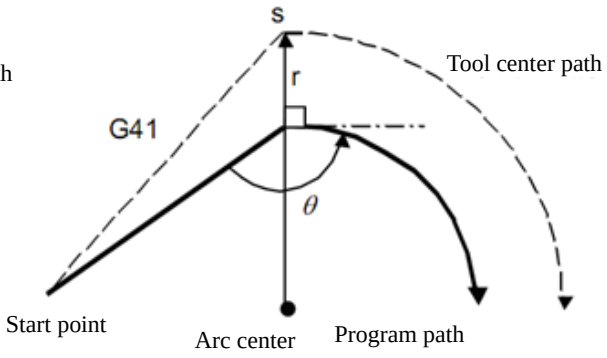


(2) Outside the corner (obtuse angle) [90≤θ＜180 °]

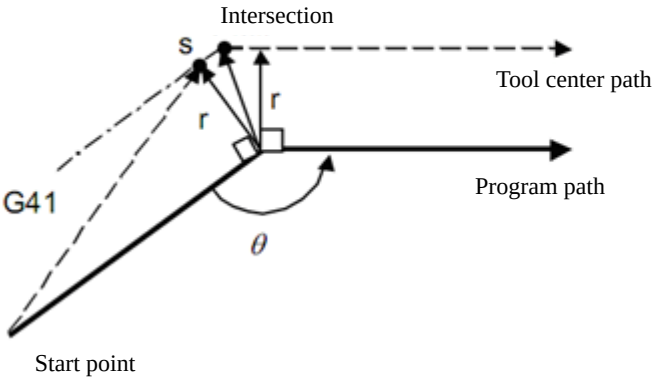
Straight line → straight line (type A)



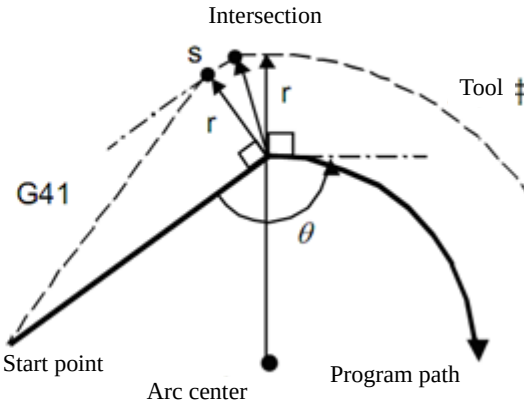
Straight line → arc (type A)



Straight line → straight line (type B)

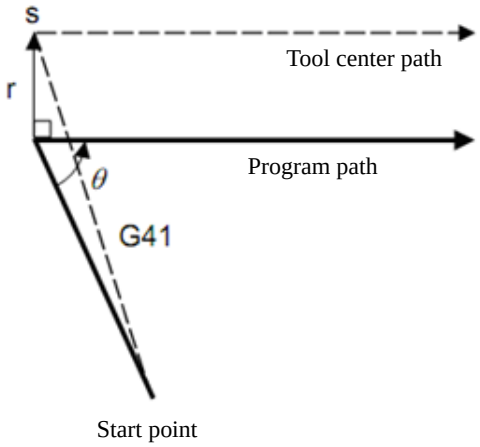


Straight line → arc (type B)

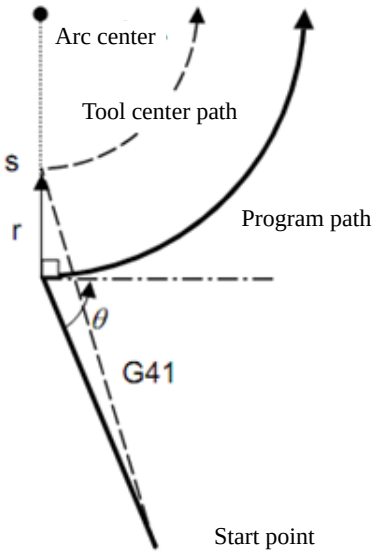


(3) Outside the corner (acute angle) [θ<90°]

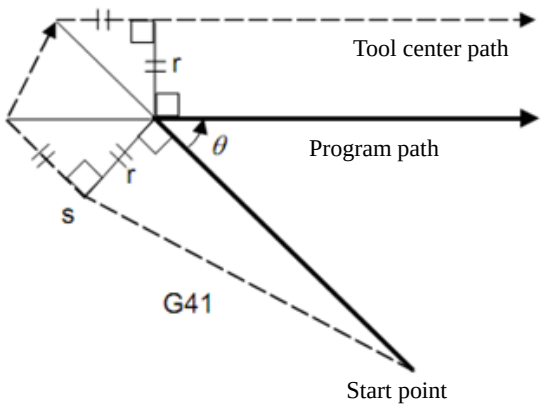
Straight line → straight line (type A)



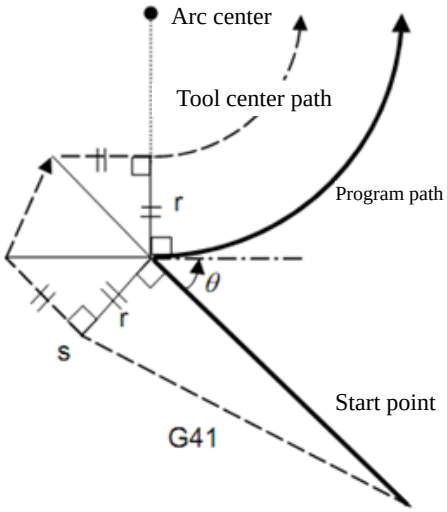
Straight line → arc (type A)



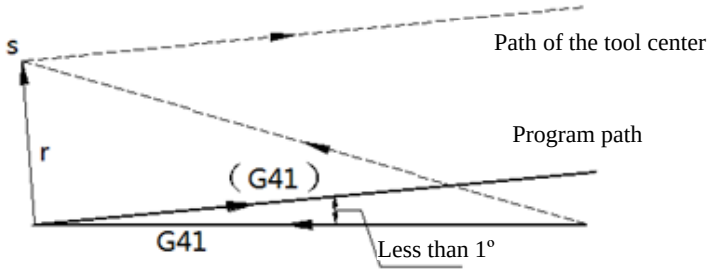
Straight line → straight line (type B)



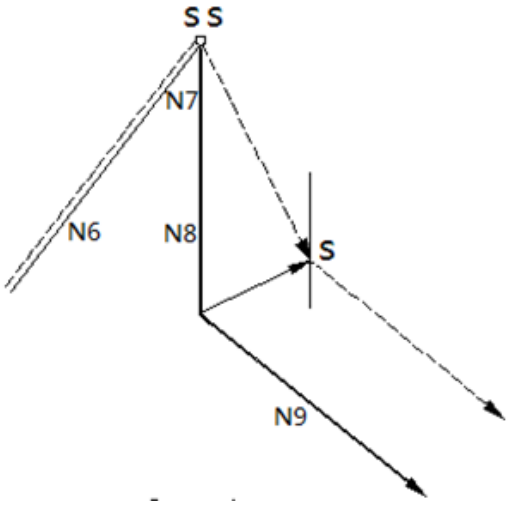
Straight line → arc (type B)



(4) When moving the outside tool around an acute angle less than 1 degree [θ < 1°]
Straight line→ straight line



(5) There is no movement command when the tool compensation is established




```
G04 G91……
⋮
N6 X100.0 Y100.0;
N7 G41 D1 X0;
N8 Y-100.0;
N9 X100.0 Y-100.0;
```

Actions in the tool radius compensation mode

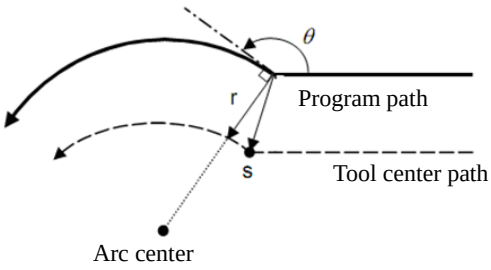
For the programmed paths (G00, G01, G02, G03), the tool path is calculated from the straight line/arc and compensated.

In the compensation mode, even if the same compensation command (G41/G42) is commanded, it will be

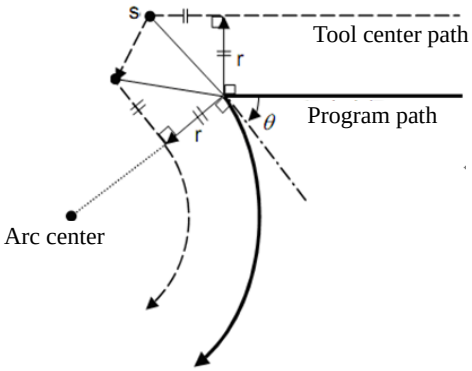
ignored.

(1) When rotating outside the corner

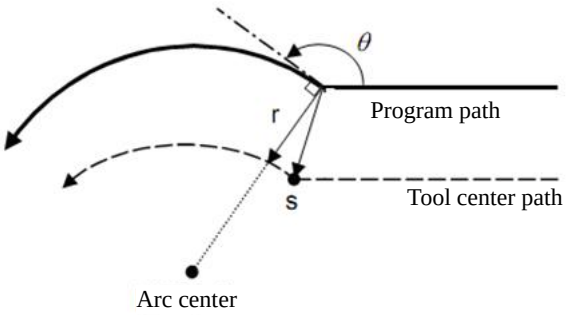
Straight line → arc (90° ≦θ < 180°)



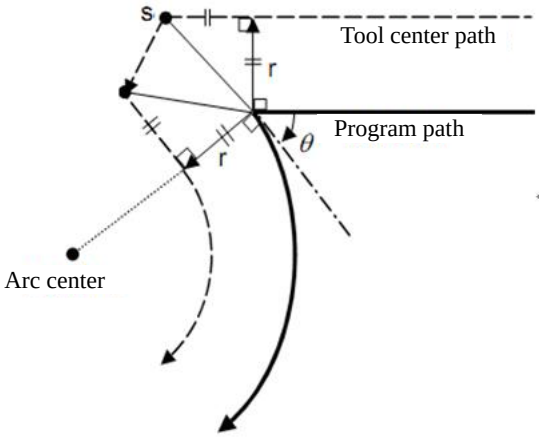
Straight line → arc (0° < θ < 90°)



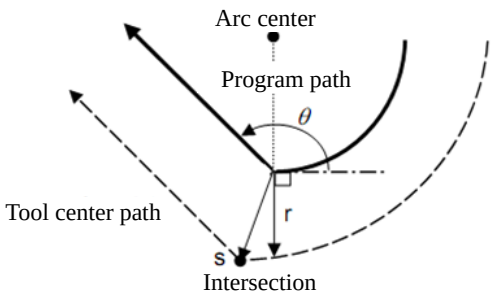
Straight line → arc (90° ≦θ < 180°)



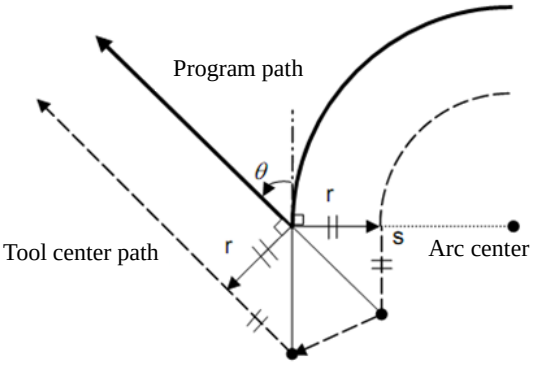
Straight line → arc (0° < θ < 90°)



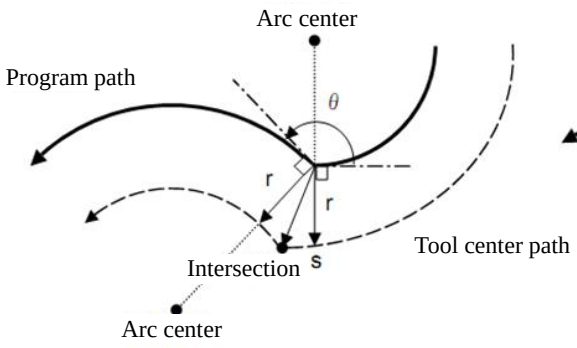
Arc→ straight line (90° ≦ θ < 180°)



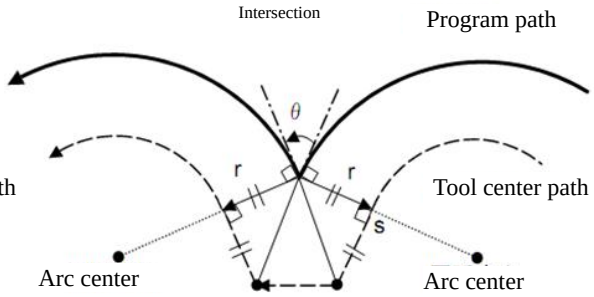
Arc→ straight line (0° < θ < 90°)



Arc → arc (90° ≦ θ < 180°)

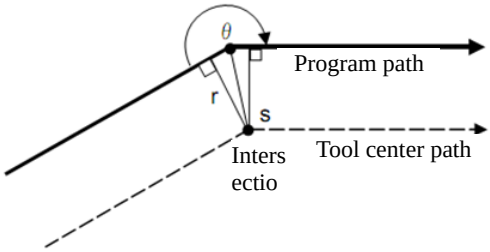


Arc → arc (0° < θ < 90°)

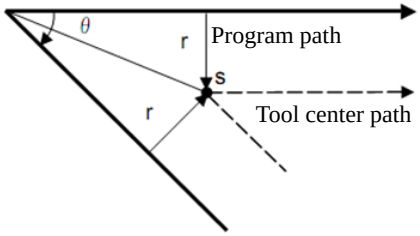


(2) When turning inside the corner

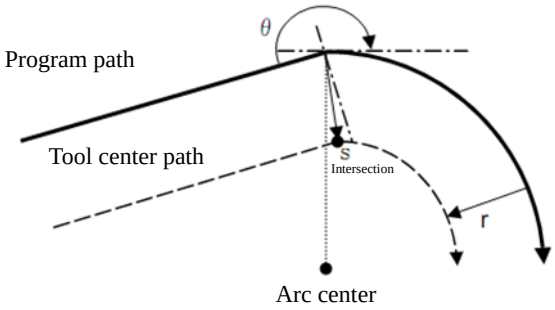
Straight line → straight line (obtuse angle)



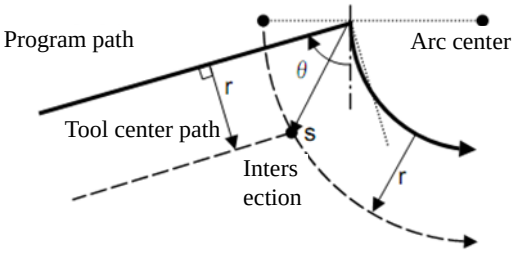
Straight line → straight line (acute angle)



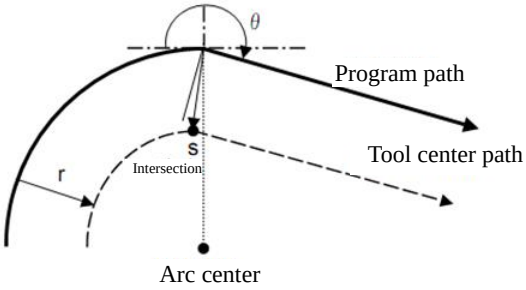
Straight line → arc (obtuse angle)



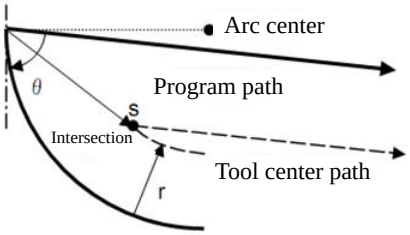
Straight line → arc (acute angle)



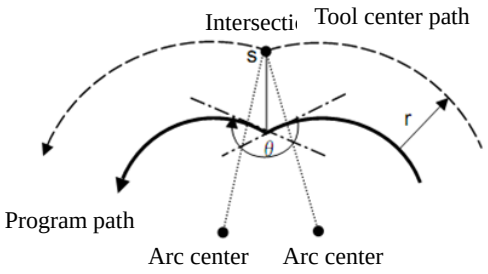
Arc→ straight line (obtuse angle)



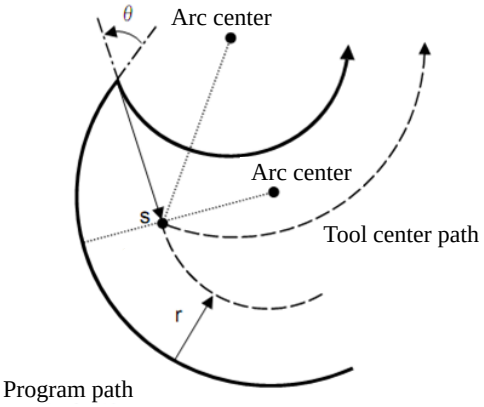
Arc→ straight line (acute angle)



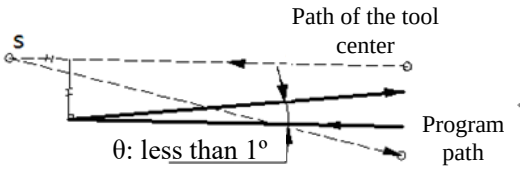
Arc→ arc (obtuse angle)



Arc→ arc (acute angle)



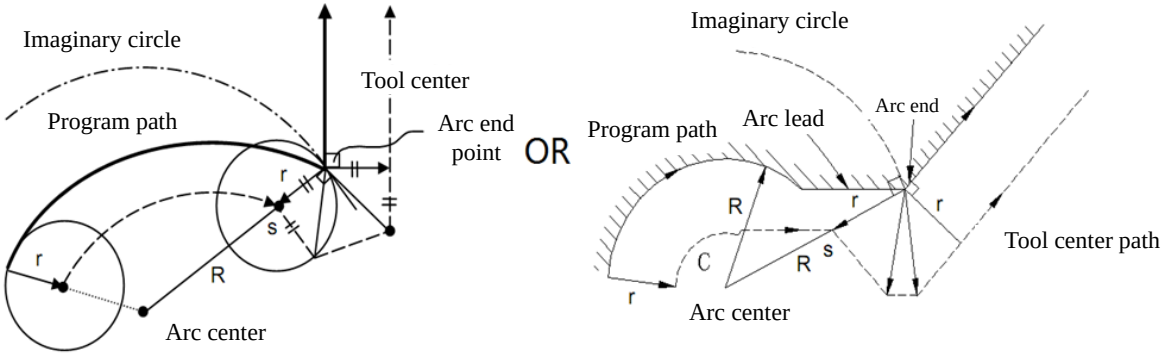
(3) The tool moves around the inside of the super-long vector. (θ＜1°)



(4) When the end point of the arc is not on the arc

When a helical arc is commanded... Interpolate with a helical arc from start point to end point of the arc.

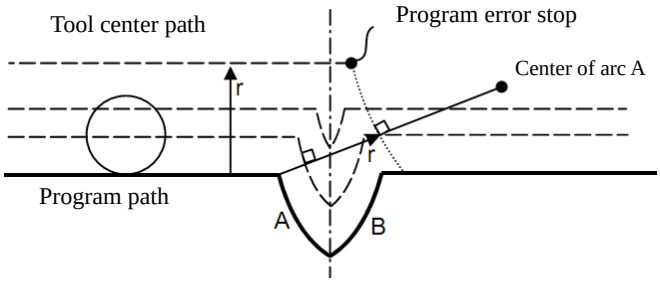
In the case of a normal arc command... If an error after compensation is within the range of parameter #1810, the interpolation will be performed with a helical arc. If the error range is exceeded, an arc lead may be interpolated.



(5) When there is no internal intersection point

As shown in the figure below, depending on the compensation amount, the intersection point of arc A and arc B may not exist.

Then, the program error alarm PS0041 is displayed at the end point of the previous program block, and the program stops running.



Cancellation of tool radius compensation

In the tool radius compensation mode, when any of the following conditions are met, the tool radius compensation is cancelled. However, in this case, there must be a movement command other than the arc command.

(1) Execute the G40 command.

(2) Execute the compensation number D00.

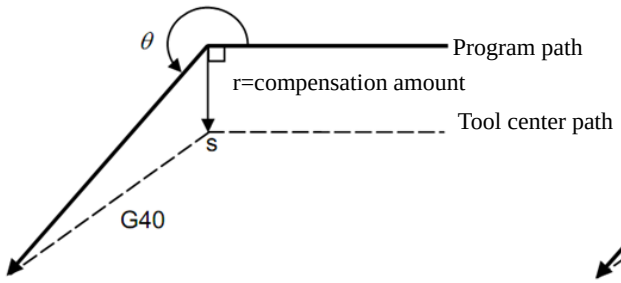
After reading the compensation cancellation command, enter the cancellation mode.

If the cancellation compensation is commanded in the arc command, a program error will occur.

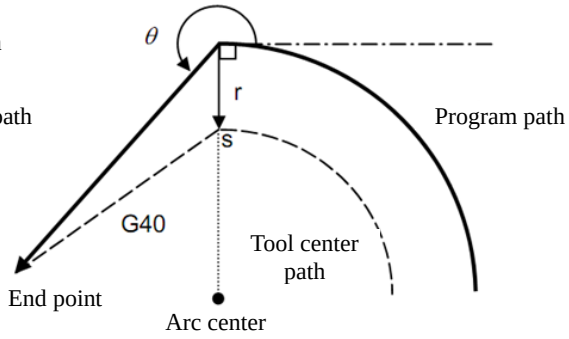
Cancellation of tool radius compensation

(1) When the inside of the corner

Straight line→ straight line

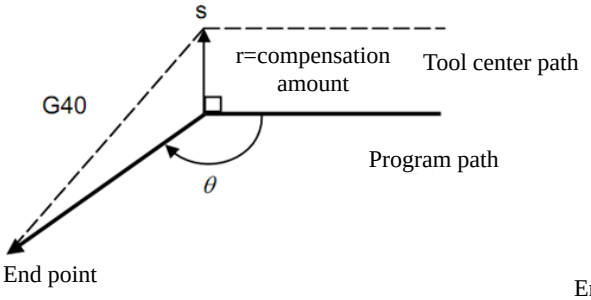


Arc→ straight line

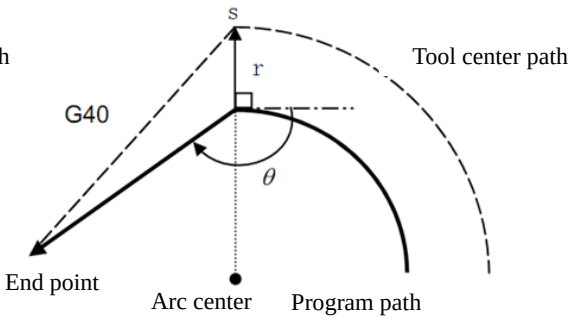


(2) When the outside of the corner (obtuse angle)

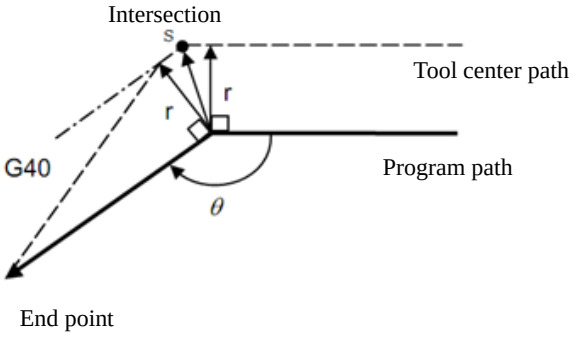
Straight line → straight line (type A)



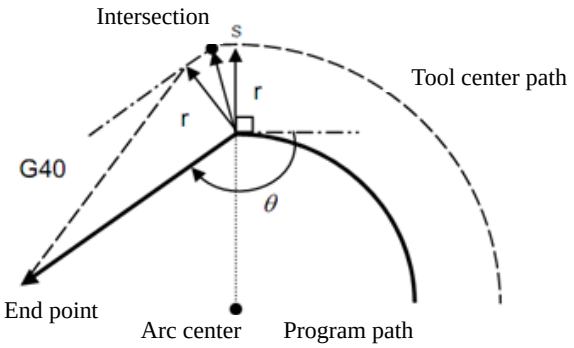
Arc→ straight line (type A)



Straight line → straight line (type B)

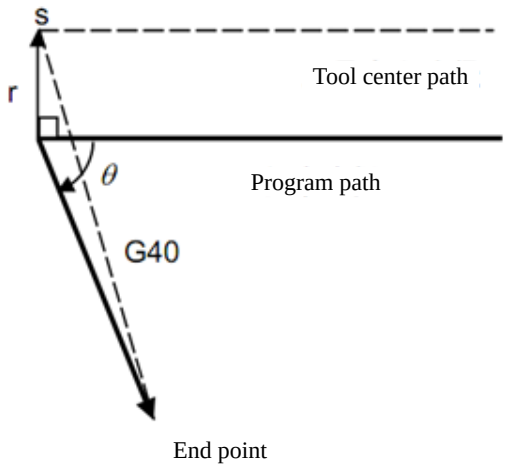


Arc→ straight line (type B)

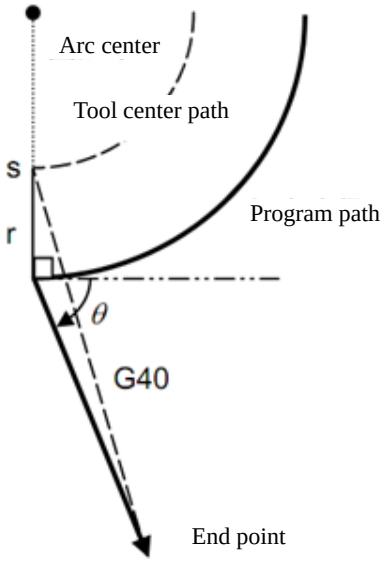


(3) When the corner is outside (acute angle)

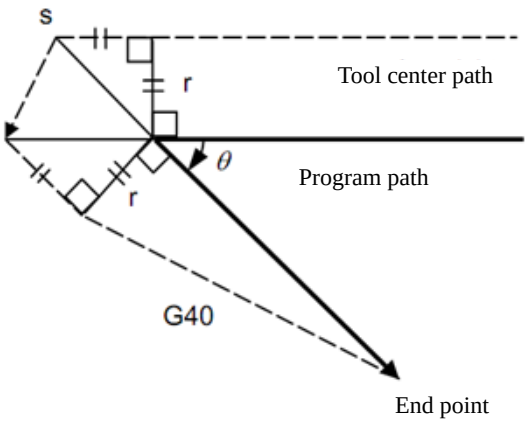
Straight line → straight line (type A)



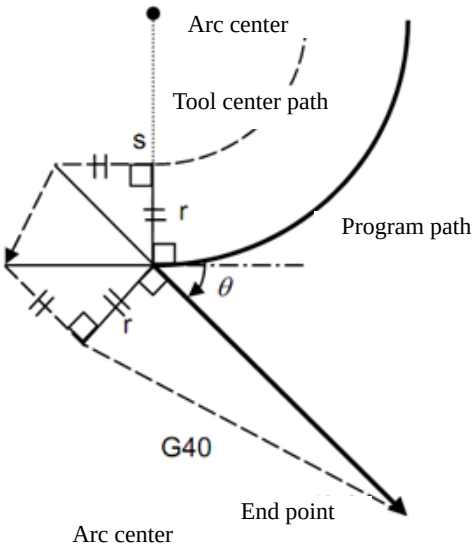
Arc→ straight line (type A)



Straight line → straight line (type B)

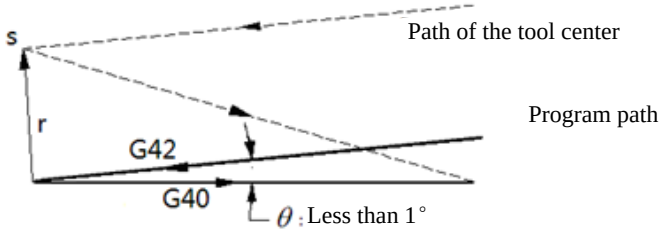


Arc→ straight line (type B)



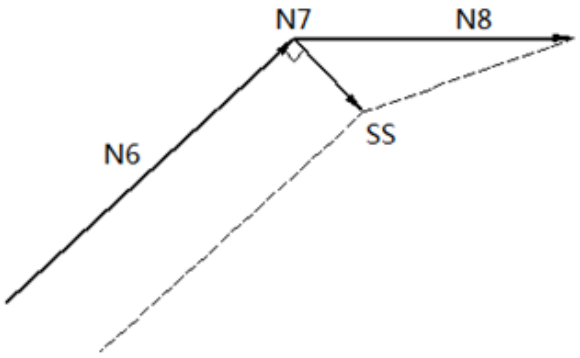
(4) The tool moves around the outside of the acute-angle corner is less than 1 degree, (θ<1°)

Straight line→ straight line



(5) There is no movement command when the tool radius compensation is cancelled

When the tool compensation is cancelled, a program block without tool movement is specified, then the vector is at right angles to the moving direction of the previous program block, and the magnitude is equal to the offset, and the vector is cancelled in the next movement command.

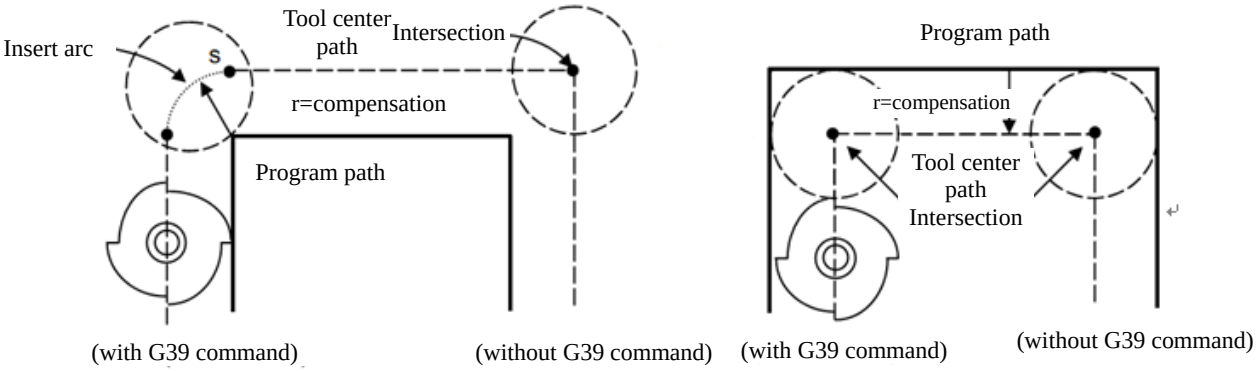


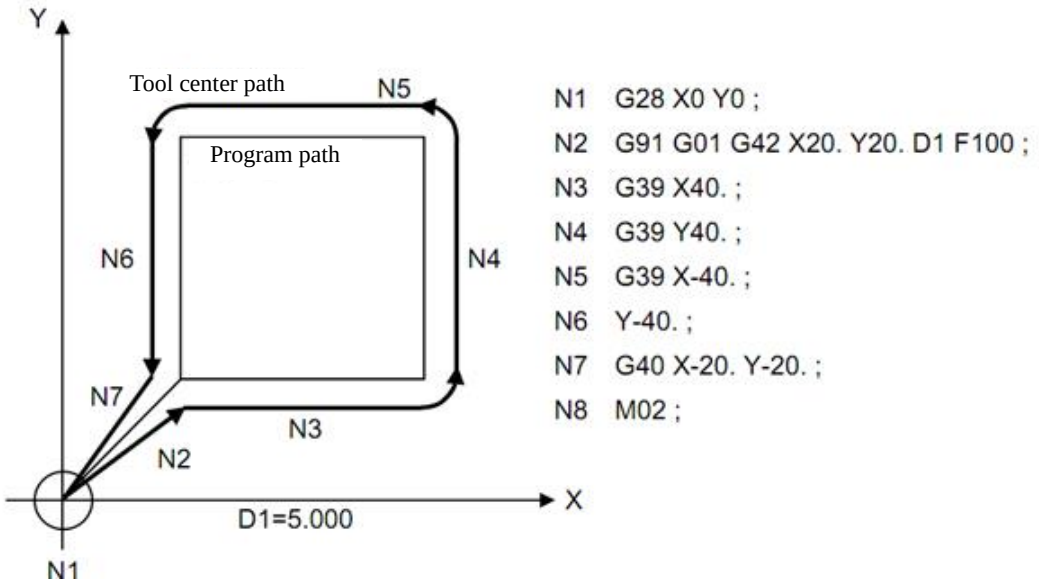

```
N6  G91  X100.0  Y100.0;
N7  G40;
N8  X50.0;
```

16.3.2 Other commands and actions in tool radius compensation

Insertion of corner arcs

When G39 (corner arc) is commanded, the intersection point at the workpiece corner is not calculated, but an arc whose radius is the compensation amount is inserted.





Change of compensation direction in the tool radius compensation

The compensation direction depends on the tool radius compensation command (G41, G42) and the symbol of the compensation amount.

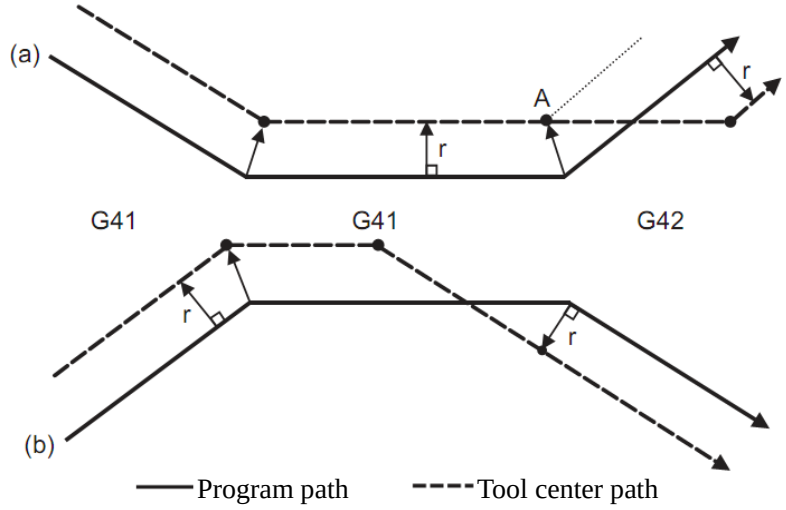
Offset amount symbol / G code	+	−
G41	Left offset	Right offset
G42	Right offset	Left offset

In the compensation mode, the compensation direction can be changed only by changing the compensation command without the compensation cancellation command. However, changes cannot be made between the compensation start block and the next program block.

(1) Straight line→ straight line

(a) There is an intersection point when the compensation direction is changed (A in the figure)

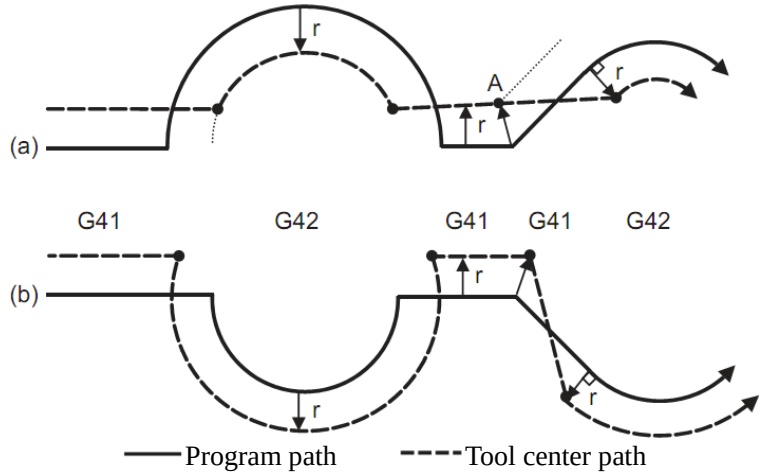
(b) There is no intersection point when the compensation direction is changed



(2) Straight line ←→ arc

(a) There is an intersection point when the compensation direction is changed (A in the figure)

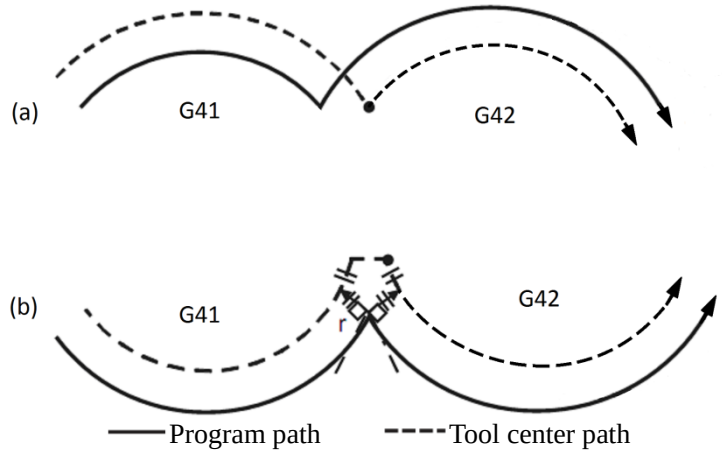
(b) There is no intersection point when the compensation direction is changed



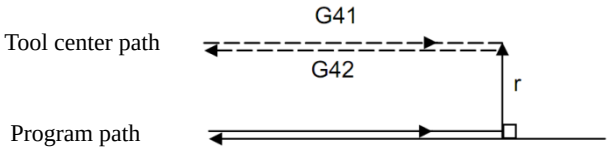
(3) Arc → arc

(a) There is an intersection point when the compensation direction is changed

(b) There is no intersection point when the compensation direction is changed



(4) When going back and forth in a straight line

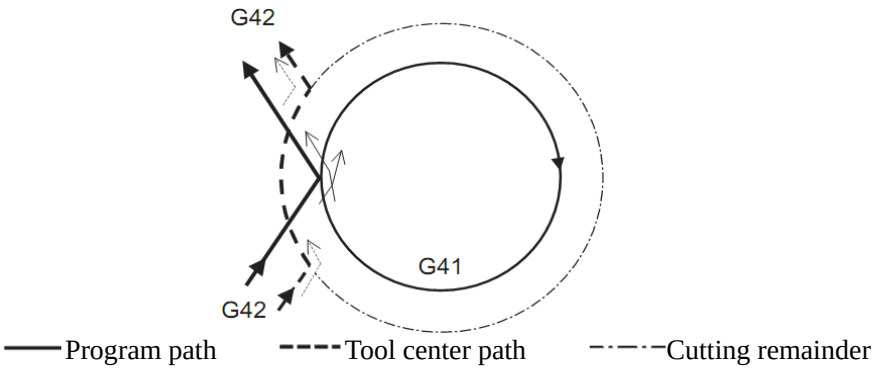


(5) When the arc is larger than 360 °according to the compensation method

In the following cases, the arc may reach more than 360 °.

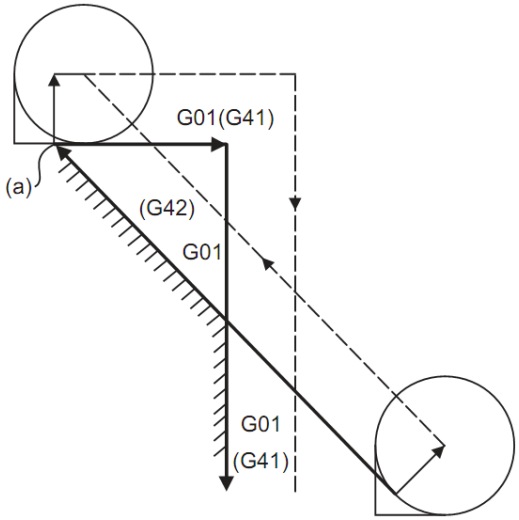
By switching between G41/G42, the compensation direction is switched.

Then, the tool center will pass through the overlapping part of the arc due to compensation. As a result, the cutting remainders are produced.

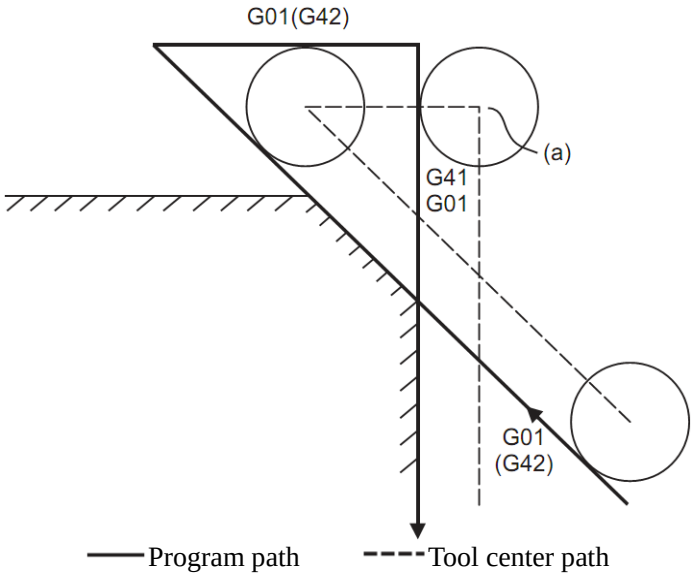


Tool radius compensation for the closed paths with G41/G42

1) G42→G41 commanded action (G41 command is issued at the position (a) in the figure)



2) G42→G41 commanded action [G41 command is issued at the position (a) in the figure]



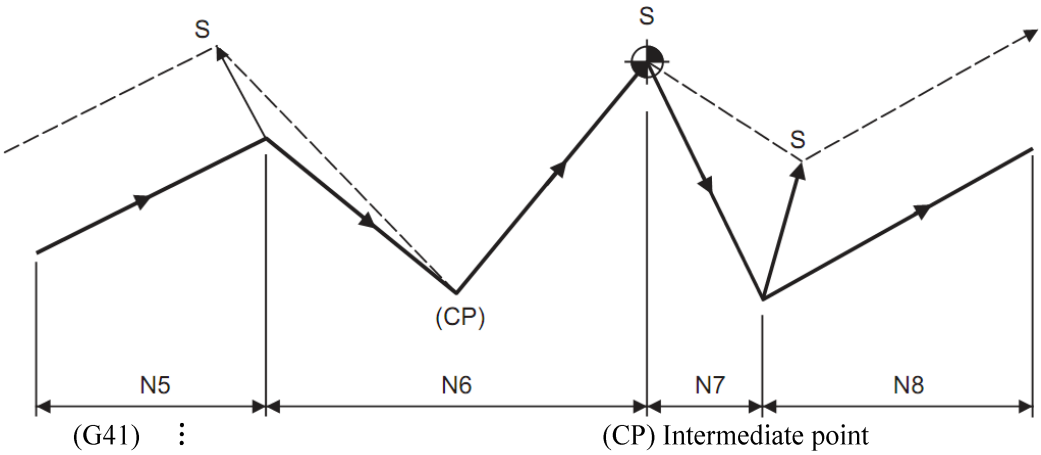
Temporary cancellation of the compensation vector command

In the compensation mode, the following command is issued to temporarily cancel the compensation vector, and then automatically return to the compensation mode.

Then, the compensation operation is not cancelled, and immediately after the intersection point vector, it moves to the vectorless point. That is, the command point of the program. When returning to the compensation mode, it also moves to the next intersection point.

(1) Reference point return command

At the intermediate point (if there is no intermediate point, it is the reference point), the temporary compensation vector is 0.



N5 G91 G01 X60. Y30. ;
N6 G28 X50. Y-40. ;
N7 X30. Y-60. ;
N8 X70. Y40. ;
⋮

(2) In the basic machine coordinate system selection (G53), the compensation vector is temporarily cancelled.

Note: In the coordinate system shift (G50) command, the compensation vector does not change.

Non-moving blocks

The program blocks shown below are called non-moving blocks.

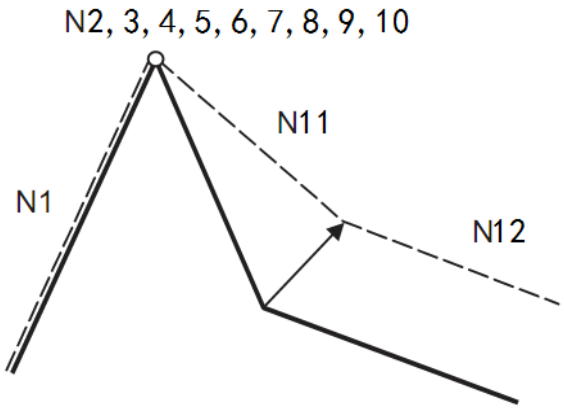
M03	M command
S12	S command
T45	T command
G04 X500	Dwell
G22 X200 Y150 Z100	Processing prohibited area setting
G10 L10 P01 R50	Compensation amount setting
G92 X600 Y400 Z500	Coordinate system determination
(G17)Z40.	Movement outside the compensation plane
G00	G code only
G91 X0	Movement amount is 0

(1) Specify at the start of compensation

Compensation vectors are not created when four or more continuous blocks are non-moving program blocks and the pre-reading prohibition M command is issued.

N1 U60.W30.T0101 ;
N2 G41;
N3 G4 X1000;
N4 G95;
N5 S500;
N6 M3;
N7 M8;
N8 G98;
N9 F100;
N10 G1;
N11 X20. Y- 50.;
N12 X50. Y- 20.;

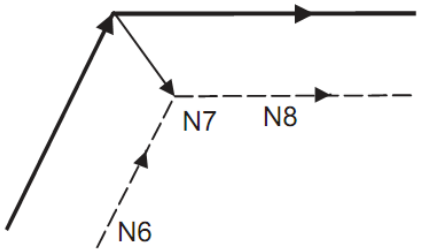
Non-moving program blocks



(2) When the compensation mode is specified

In the compensation mode, the blocks within 8 continuous program blocks are non-moving program blocks, and a usual intersection point vector is created.

N6 U200. W100. ;
N7 G04X P1000 ; Non-moving program blocks
N8 W200. ;



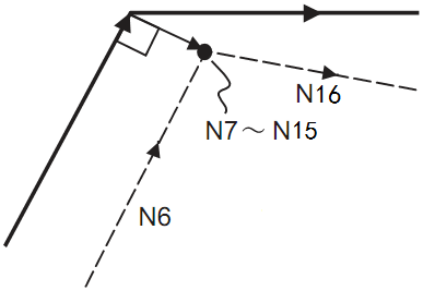
N7 in the figure represents the execution of the N7 program block.

When eight or more continuous program blocks are non-moving program blocks, a vertical compensation vector is created at the end point of the previous program block.

At this point, a cut-in may occur.

N6 X100. Y200. ;
N7 G4 X1000 ;
N8 F100 ;
⋮
blocks
N14 S500 ;
N15 M4 ;
N16 X100. ;

Non-moving program

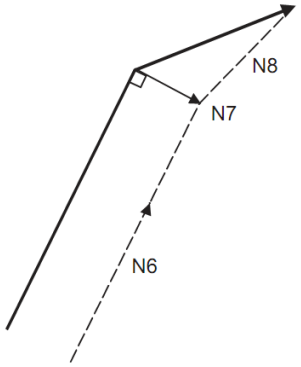


Parameter 2612 can correct the number of non-moving program blocks by 8. When the set number of non-moving blocks is smaller than the non-moving program blocks in the actual program, the intersection point cannot be calculated because the moving blocks of the next block cannot be found, and the program will automatically cancel the compensation at this time. Note that this condition may result in an overcut or undercut.

(3) Issue command at the same time as compensation cancellation

When a program block without movement is specified at the same time as G40, only the compensation vector is cancelled.

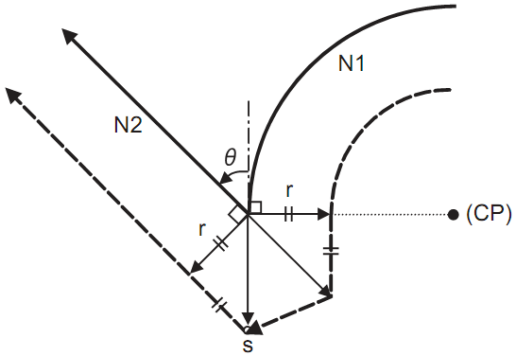
N6 X100. Y200.;
N7 G40 M5 ;
N8 X100. Y50.;



Corner movement

When there are multiple compensation vectors at the junction of the movement command program blocks, the movement is performed between the vectors in a straight line. Such an action is called corner movement.

Actions in single-block operation are executed with the previous program block and the corner movement as one block, and the feed speed is the speed specified by the N1 block. In the next startup, the remaining movement and the next program block are executed as one block, and the feed speed is the speed specified by the N2 block.



(CP) Arc center
(r) Tool radius
(s) Single segment stop point

Program path Tool center path

16.3.3 General precautions for tool radius compensation

Precautions

(1) Designation of compensation amount

Designate the compensation amount by specifying the compensation amount number with the D code. Once a D code is specified, it is valid until another D code is specified.

(2) Change of compensation amount

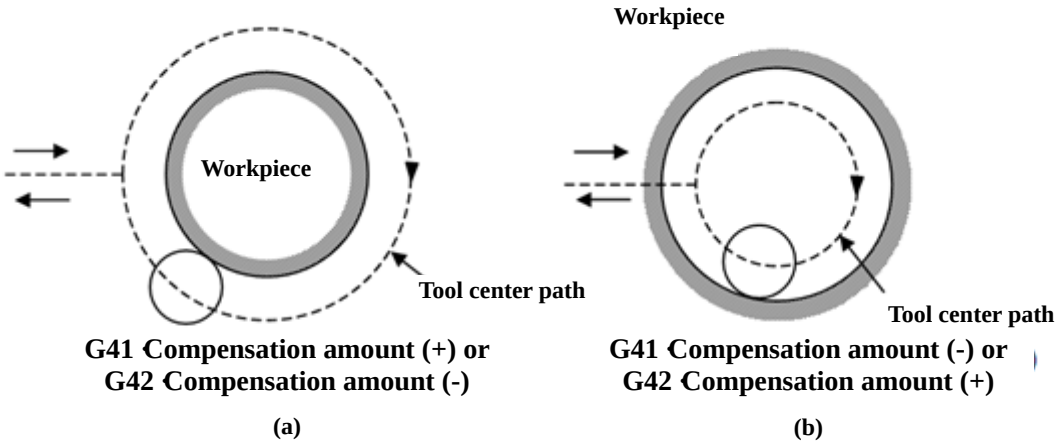
The compensation amount is usually changed by selecting another tool in the radius compensation cancellation mode. When the compensation amount is changed in the compensation mode, the vector at the end point of the program block is calculated using the compensation amount specified in the block.

(3) Symbols of compensation amount and the tool center path

If the compensation amount is set to negative (-), it is equivalent to the tool path after swapping G41 and G42. Therefore, the processing performed from the front around the outside of the workpiece becomes the processing around the inner side of the workpiece, and the processing performed from the front around the inner side of the workpiece becomes the processing performed around the outer side of the workpiece. An example is given in the image below. Usually, the compensation amount is set to positive (+) and programmed. When the tool center path is programmed as shown in (a), if the compensation amount is set

to minus (-), the operation shown in (b) is performed. Conversely, if the compensation amount is set to minus (-) when it is programmed according to (b), it will operate according to (a). Therefore, it is possible to cut and machine both male and female shapes using one program, and the tolerance between the two can be arbitrarily determined by selecting an appropriate compensation amount.

(However, when a compensation is started and cancelled, the circle is divided into two equal parts by the type A)



16.3.4 Change of compensation number in compensation mode 217

Function and purpose

In the compensation mode, in principle, it is best not to change the compensation number. If it must be changed, operate as shown in the figure below.

When changing the compensation number (compensation amount)

G41 G01-------- Dr1 ;

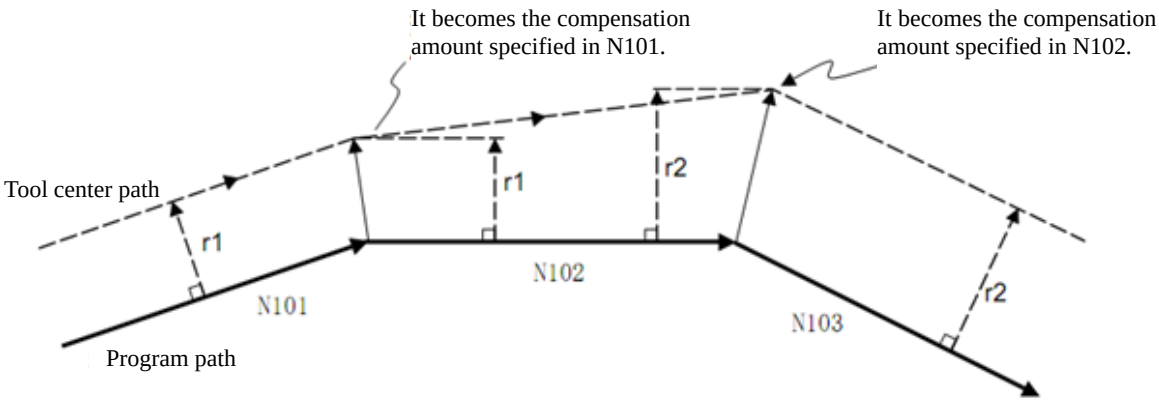
(α=0,1,2,3)

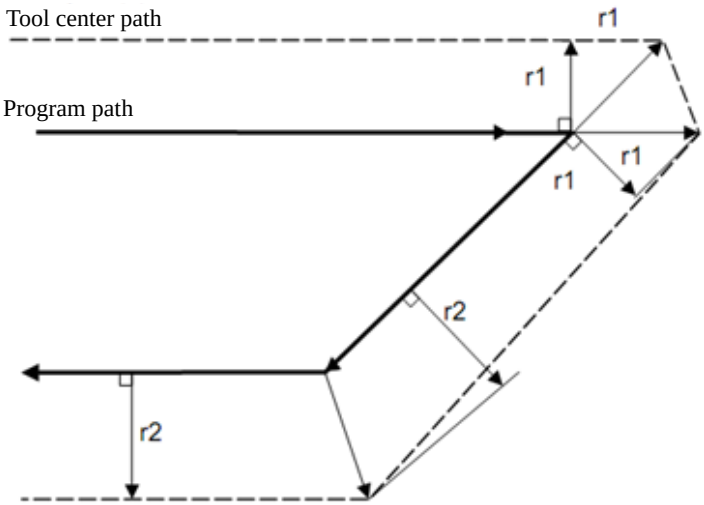
N101 G0α Xx1 Yy1;

N102 G0α Xx2 Yy2 Dr2 ; -------- Change the compensation number

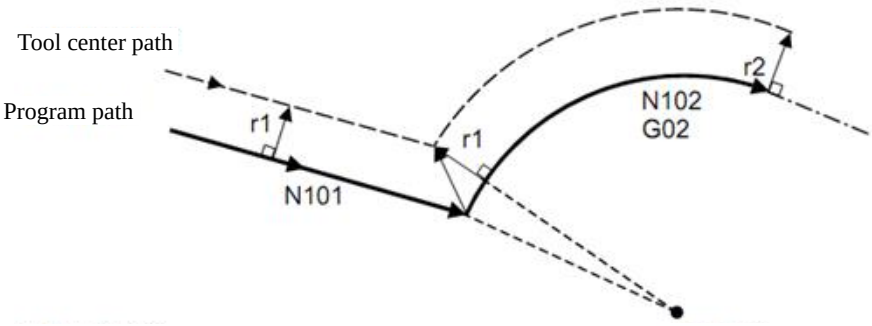
N103 Xx3 Yy3;

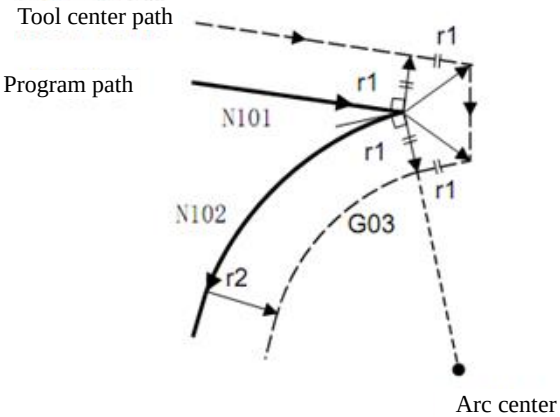
(1) Straight line – straight line



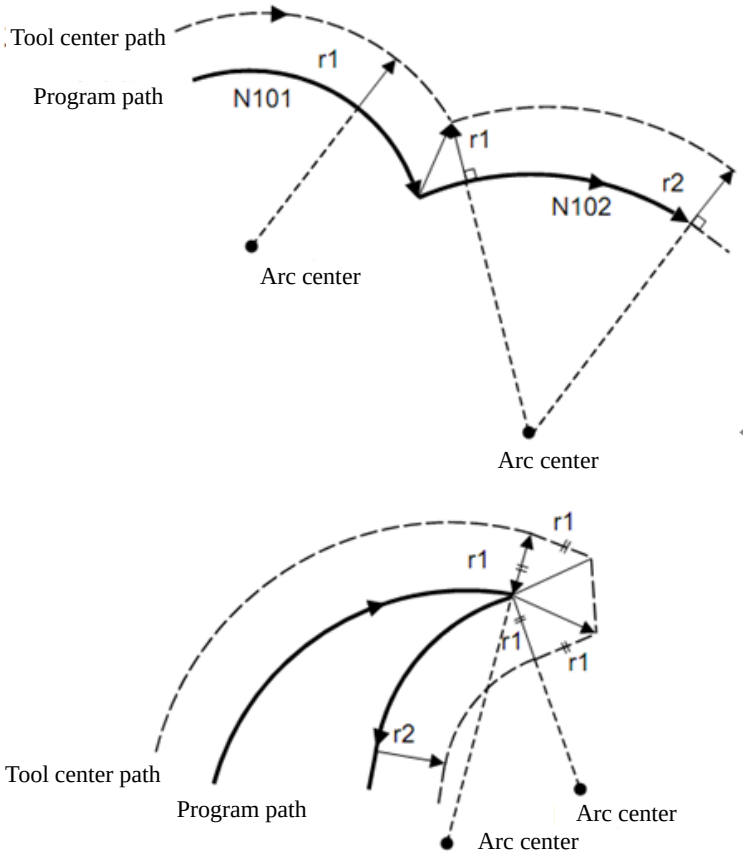


(2) Straight line → arc





(3) Arc → arc



16.3.5 Interference check

Function and purpose

Usually, according to the two pre-read program blocks, the tool nose compensated in the tool compensation may cut into the workpiece.

This phenomenon is called interference, and the function to avoid interference is called interference check.

The interference check has the following two capabilities. It can be selected by a parameter.

	Function	Parameter #2602bit1	Action
(1)	Perform the interference check	0	A program error causes a stop before the program block where the cut-in occurs
(2)	Interference check is not performed	1	Continue cutting even if the cut-in occurs

Detailed description

(Example)

(G41)

N1 G90 G1 X50. Y-100.

N2 X70 Y-100;

N3 X120. Y0.;

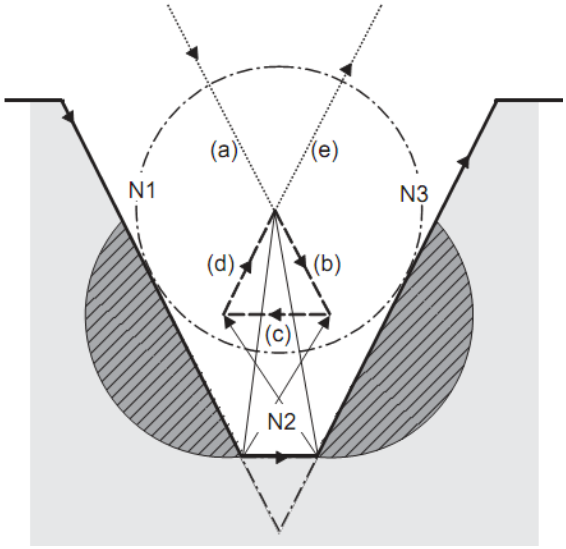
When interference check functions effectively

An alarm is given when N1 is executed.

When the interference check functions ineffectively

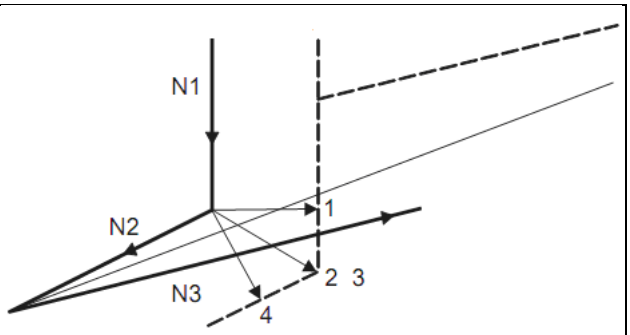
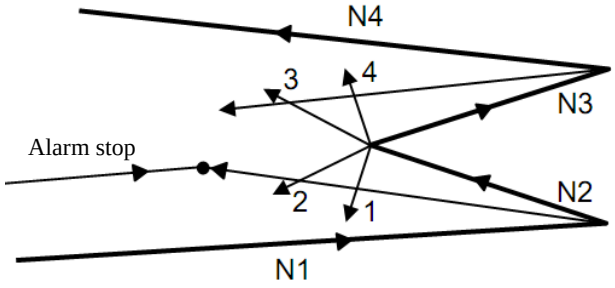
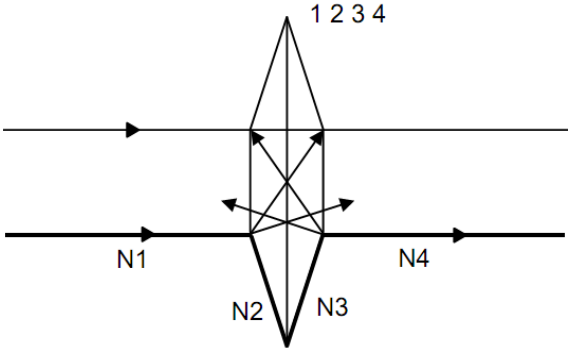
Pass through the straight lines N1 and N3 while cutting into them.

The center path of the tool nose R is (a)→(b)→(c)→(d)→(e).

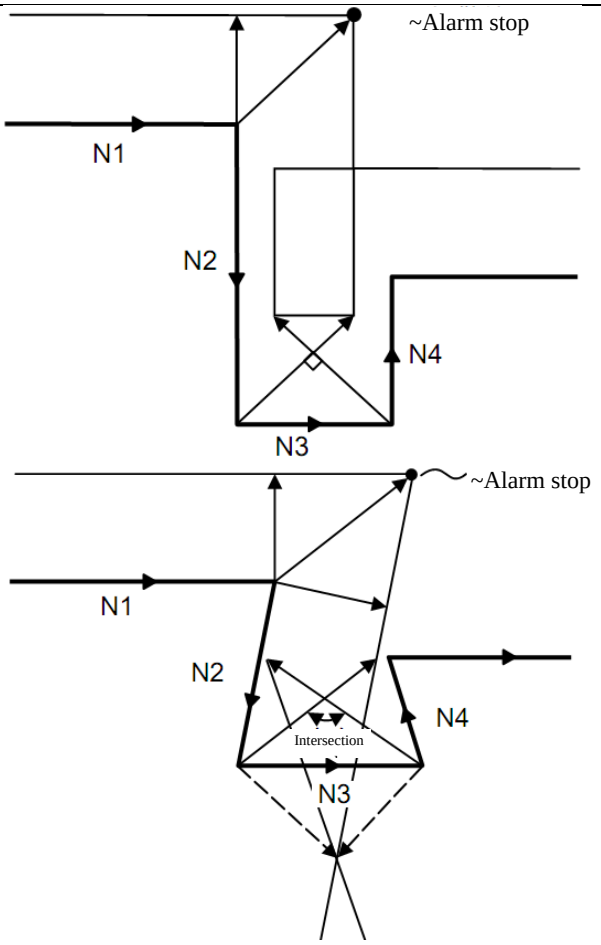


Interference check alarm

When the interference check alarm function is selected, an interference check alarm occurs under the following conditions.

(1) As illustrated in the figure on the right, when all the vectors 1 to 4 at the end point of the N1 program block are cancelled, a program error occurs before N1 is executed, causing a stop.	
(2) In the figure on the right, if the interference check of N2 is performed, all the end-point vectors of N2 will be deleted. At this time, a program error will occur at the end-point of N1.	
(3) As illustrated in the figure on the right, the moving direction is reversed at N2. Then, a program error occurs before N1 is executed.	

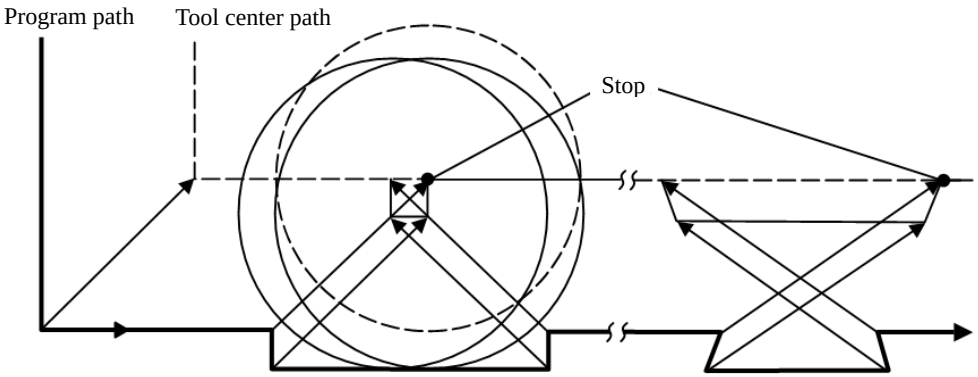
(4) As illustrated in the figure on the right, the vector interferes with N3. Therefore, when the vector intersection angle is greater than 90 °, a program error will occur at the end point of N1



(5) When the program forward direction is opposite to that after compensation

In the following cases, probably an interference may be deemed to have occurred even if no interference has occurred.

When programming a groove that is smaller than the tool diameter, parallel or with a wider bottom.



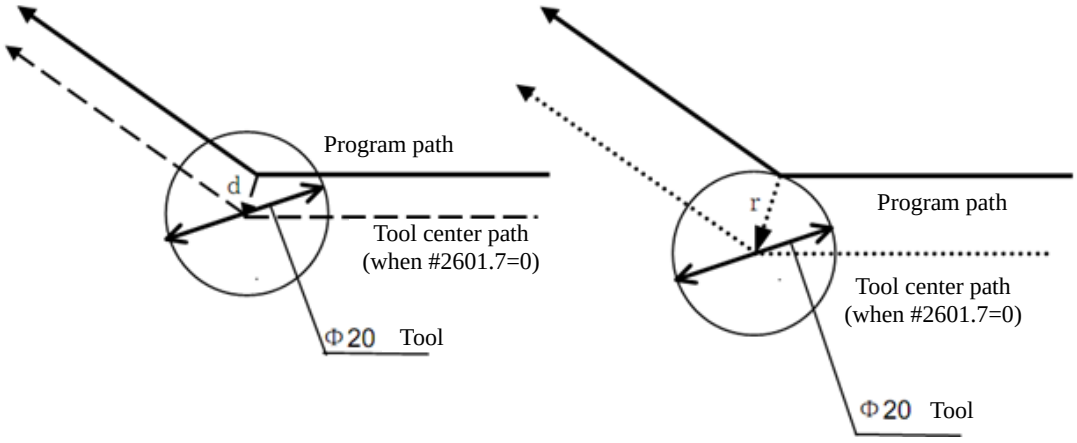
16.3.6 Compensation amount diameter specifying

Function and purpose

This function specifies the tool radius compensation by tool diameter. Turn on the control parameter #2601.7 and set the tool radius compensation to enable the diameter. The compensation amount set to the commanded tool number is regarded as the diameter compensation amount, and is converted into the radius compensation amount while compensating.

Actions when the diameter is specified for the tool radius compensation amount

When the command tool radius compensation D=10.0 and parameter #2601.7=0, the tool radius compensation d=5.0. (when parameter #2601.7=1, the tool radius compensation amount r=10.0)



16.4 Programmable compensation input (G10 L2, L10, L11, L12, L13)

Function and purpose

The tool compensation and workpiece offset can be set/changed with the G10 command. In the absolute value (G90) mode, the commanded compensation amount is a new compensation amount, while in the incremental value (G91) mode, the currently set compensation amount plus the commanded compensation amount is used as a new compensation amount.

Command format

(1) Workpiece offset input

```
G91/G90   G10   L2   P__X__Y__Z__;
```

```
P: 0 external workpiece
     1   G54
     2   G55
     3   G56
     4   G57
     5   G58
     6   G59
```

Note: In G91 mode, the offset is an incremental value, which is accumulated every time the program is executed. In order to avoid such mistakes, please try to command G90 or G91 before G10, so as to call attention.

(2) Tool compensation input

```
G10   L10   P__R__;      Length offset compensation H
G10   L11   P__R__;      Length wear compensation H
G10   L12   P__R__;      Radius shape compensation D
G10   L13   P__R__;      Radius wear compensation D
```

Detailed description

(1) G10 does not include movement, so please do not use it together with G commands other than G54~G59,

G90, and G91.

(2) Please do not place the fixed cycle and subprogram call commands in the same program block as the G10 command. Otherwise, a false action or program error will occur.

(3) Please do not command the workpiece offset input (L2 or L20) and the tool compensation input (L10) in the same program block.

(4) If an incorrect L number or compensation number is commanded, a program error will occur respectively. Also, when the compensation amount exceeds the maximum command value, a program error will occur.

(5) The compensation amount is valid if it is a decimal point input.

(6) In the compensation amount of the external workpiece coordinate system and the workpiece coordinate system, the distance from the point of origin of the basic machine coordinate system is commanded.

(7) The updated workpiece coordinate system is input through the workpiece coordinate offset, with the previous modal (G54~G59) or the modal of the same segment (G54~G59) being used.

(8) If there is no L address in G10 command, the system will generate a PS0014 alarm.

(9) If the P command is omitted from the workpiece offset input, the program error "PS0031 Illegal P command in G10" occurs.

Chapter XVII Turning Tool Offset Function

17.1 Tool compensation

Function and purpose

A tool compensation is performed by the T function, and a command is issued with the 4-digit value following the address T. The tool compensation includes tool length compensation and tool nose wear compensation. The tool length compensation and tool nose wear compensation are specified by the lower 2 digits of the T command.

Only one B code can be specified in a program block.

Command format

T○○ ○○;

Tool length compensation + tool nose wear compensation

Tool selection

Detailed description

(1) Tool selection: The tool selection is to specify the T code corresponding to the tool number. For the relationship between the tool selection number and the tool, refer to the machine manufacturer's instructions.

(2) Offset number: The tool offset number has two meanings, which are used to start the offset function and specify the offset distance corresponding to the number. The tool offset number 0 or 00 means that the offset is 0 or the offset is cancelled.

(3) Offset: There are two types of offset. One is the tool wear offset, and the other is the tool geometry offset.

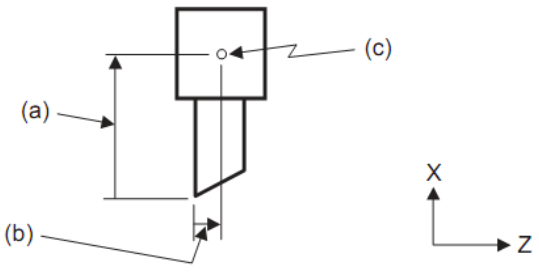
17.2 Tool length compensation

Function and purpose

1. Set the tool length compensation

The tool length compensation is executed for the reference position of the program. The program reference position is usually located at the center of the turret or the nose position of the reference tool.

(1) At the center of the turret

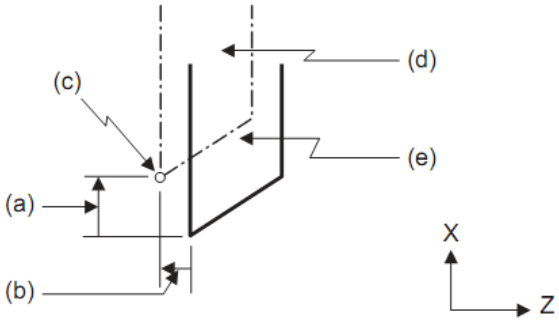


(a) Tool length compensation amount in the X-axis

(b) Tool length compensation amount in the Z-axis

(c) Datum position (datum point)

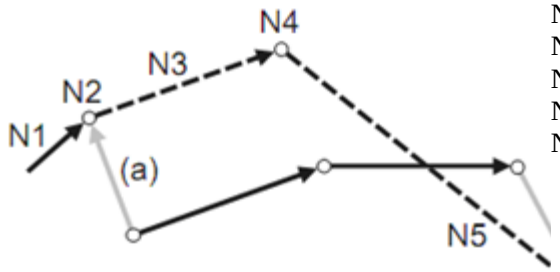
(2) At the nose position of the datum tool



(a) Tool length compensation amount in the X-axis
(b) Tool length compensation amount in the Z-axis
(c) Datum point
(d) Datum tool
(e) Tools for machining

2. Change the tool length compensation number

When the tool number is changed, the tool length compensation amount corresponding to the new tool number is added to the movement amount of the machining program.



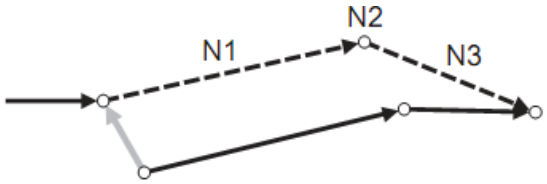
N1 G1 X10.0 Z10.0 F100 ;
N2 T0101;
N3 G1 X13.0 Z15.0 F20.0 ;
N4 T0202 ;
N5 G1 X13.0 Z20.0 F25.0 ;

Machining program path
Compensation path
(a) Compensation amount (old)
(b) Compensation amount (new)

3. Cancel the tool length compensation

(1) When the compensation number 0 is specified

When the tool length compensation number in the T command is 0, the compensation is cancelled.

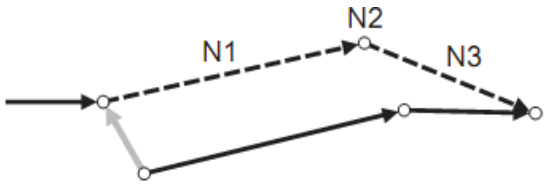


N1 X10.0 Z10.0 F10 ;
N2 T0000 ;
N3 G1 X10.0 Z20.0 ;
When the compensation operation is executed in the program block with the movement command.

Machining program path
Compensation path
Compensation amount

(2) When the specified compensation amount is 0

When the compensation amount of the tool length compensation number in the T command is 0, the compensation is cancelled.



N1 G1 X10.0 Z10.0 F10 ;
N2 T0100 ;
N3 G1 X10.0 Z20.0 ;
When the compensation operation is executed in the program block with the movement command.

Machining program path
Compensation path
Compensation amount

Precautions

(1) G28, G29, or G30 commands are issued to temporarily cancel the compensation. Therefore, although the machine moves to the position where the compensation is cancelled, the compensation amount is still memorized, so it moves by the next movement command to the position after compensation.

(2) When G28, G29, G30 and compensation cancellation command are issued in the same program block, although the machine moves to the position where compensation is cancelled, the compensation amount is still memorized. Therefore, the displayed coordinates may contain a compensation amount. If the compensation amount is not memorized, please issue a command in another program block.

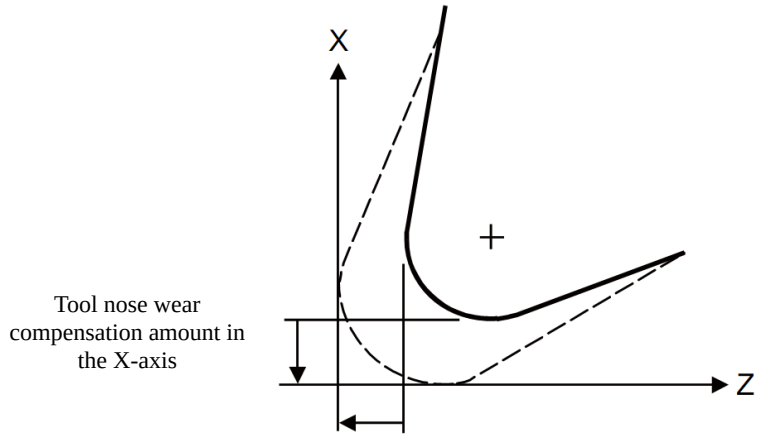
(3) During the autorun, even if the compensation amount of the currently selected compensation number is changed through MDI, etc., the changed compensation amount will not take effect as long as the T command with the same number is not executed again.

(4) The tool length compensation and tool nose wear compensation amount may be cleared by reset or emergency stop. It can be kept by parameter "#2601bit6".

17.3 Tool nose wear compensation

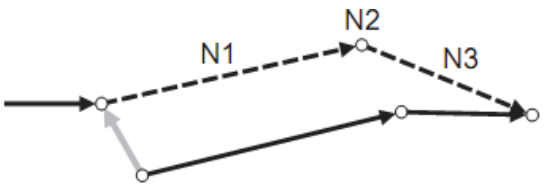
1. Set the tool nose wear compensation

The tool nose can be compensated if it is worn.



2. Cancel the tool nose wear compensation

When the tool nose wear compensation number is 0, the compensation is cancelled.



N1 G1 X10.0 Z10.0 F10 ;
N2 T0100 ;
N3 G1 X10.0 Z20.0 ;
When the compensation operation is executed in the program block with the movement command.

Machining program path
Compensation path
Compensation amount

Precautions

(1) G28, G29, or G30 commands are issued to temporarily cancel the compensation. Therefore, although the machine moves to the position where the compensation is cancelled, the compensation amount is still memorized, and the machine moves by the next movement command to the position after compensation.

(2) When G28, G29, G30 and compensation cancellation command are issued in the same program block, although the machine moves to the position where compensation is cancelled, the compensation amount is still memorized. If the compensation amount is not memorized, please issue a command in another program block.

(3) During the autorun, even if the compensation amount of the currently selected compensation number is

changed through MDI, etc., the changed compensation amount will not take effect as long as the T command with the same number is not executed again.

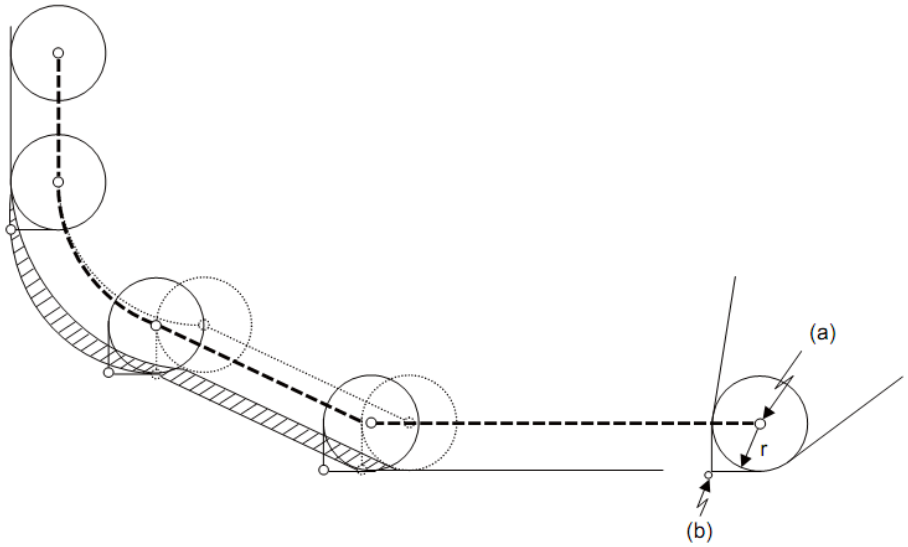
(4) The tool length compensation and tool nose wear compensation amount may be cleared by reset or emergency stop. It can be kept by parameter "#2601bit6".

17.4 Tool nose R compensation (G40/G41, G42)

Function and purpose

Since the tool nose is generally curved, the imaginary tool nose point is regarded as the tool nose for programming. When performing tapered cutting and arc cutting, an error due to the radian of the tool nose occurs between the programmed shape and the cutting shape. The tool nose R compensation is a function that automatically calculates and compensates the error by setting the tool nose R value.

Through the command code, you can choose to fix or automatically identify the compensation direction.



(a) Tool nose center (b) Imaginary tool nose point (r) Tool nose R

..... Tool nose center path when the tool nose R compensation is not performed (the oblique line is the cutting shape error)

▬▬▬ Tool nose center path when the tool nose R compensation is executed

Command format

G40 X(U) Z(W) ;	Tool nose R compensation cancelled
G41 X(U) Z(W) ;	Tool nose R compensation on the left
G42 X(U) Z(W) ;	Tool nose R compensation on the right

X(U)	X-axis end point coordinate (X is the absolute value of the workpiece coordinate system, U is the incremental value from the current position)
Z(W)	Z-axis end point coordinate (Z is the absolute value of the workpiece coordinate system, W is the incremental value from the current position)

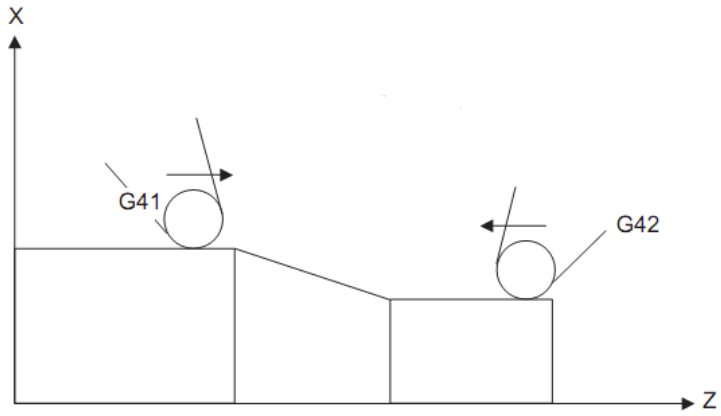
Detailed description

(1) G41 performs the tool nose R compensation for the tool on the left side of the workpiece in the forward

direction.

G42 performs the tool nose R compensation for the tool on the right side of the workpiece in the forward direction.

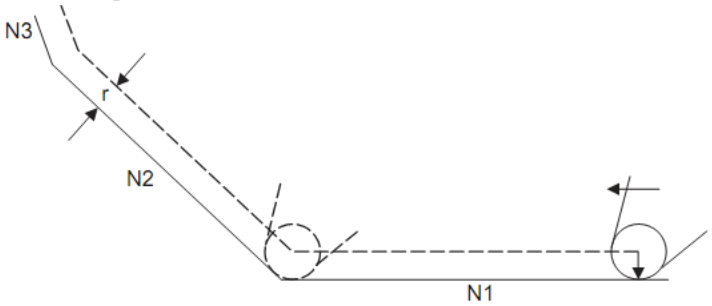
G40 is to cancel the tool nose R compensation mode.



(2) Tool nose R compensation pre-reads the block data of the following two movement commands (if there is no movement command, pre-reads up to 5 program blocks), and controls the path of the tool nose R and the center of the tool nose at a distance from the intersection point calculation formula. The program path is offset on the path after the tool nose R radius.

In the figure, r is the tool nose R compensation amount (tool nose R radius).

The tool nose R compensation amount corresponds to the tool length number, and should be pre-set together with the tool nose point.



(3) A program error will occur when there is no movement amount for 8 continuous program blocks.

(4) The tool nose R compensation is valid for fixed cycles (G77~G79) and for roughing cycles (G70, G71, G72, G73).

(5) The thread cutting command is temporarily cancelled in the previous program block.

(6) The compensation plane, moving axis, and the next forward direction vector depend on the plane selection commands specified by G17 to G19.

G17... XY plane

G18... ZX plane

G19... YZ plane

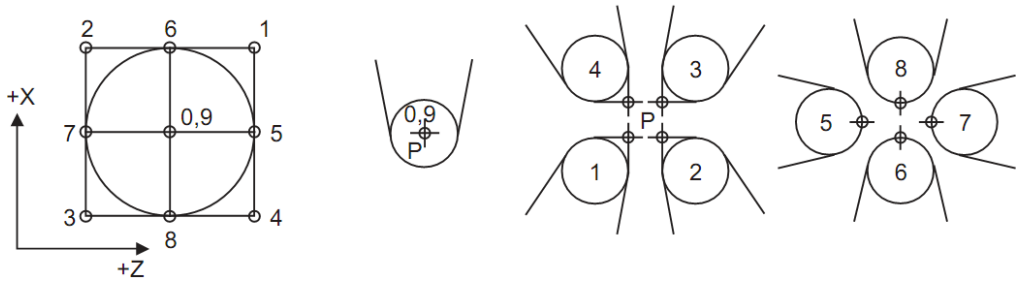
17.4.1 Tool nose point and compensation direction

Detailed description

(1) Tool nose point

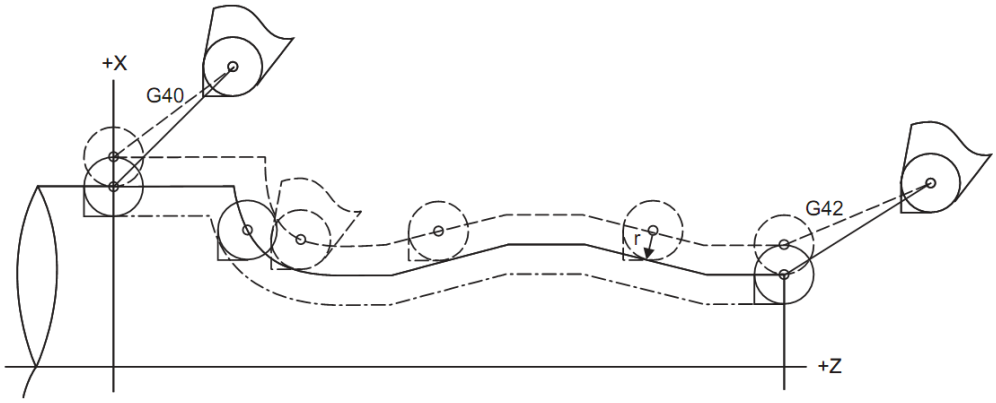
Since the tool nose is generally arc-shaped, the position of the tool nose in the program is illustrated in the figure below, aiming at the point P.

In the tool nose R compensation, select one point for each tool length number in the figure below, and preset the positional relationship in the T column of the system offset setting interface.



(2) Tool nose point and compensation action

When machining is performed with the center of the tool nose R aligned with the machining start position (with the tool nose point 0 or 9)



Machining shape with the programmed path or the tool nose R compensation

Tool nose R center path with the nose R compensation

Machining shape without the nose R compensation

17.4.2 Tool nose R compensation action

Detailed description

(1) Tool nose R compensation cancellation state

The tool nose R compensation proceeds to the compensation cancellation mode under the following conditions.

1) After power on

2) After pressing the reset button of the setting display device

3) After executing M02 and M30 with the reset function

4) After executing the compensation cancellation command (G40)

5) After selecting the tool number 0 (execute T00)

In compensation cancel mode, the compensation vector is 0, and the tool nose point path is consistent with the program path.

Be sure to end the program including tool nose R compensation while the compensation is cancelled.

(2) Start the nose R compensation (start)

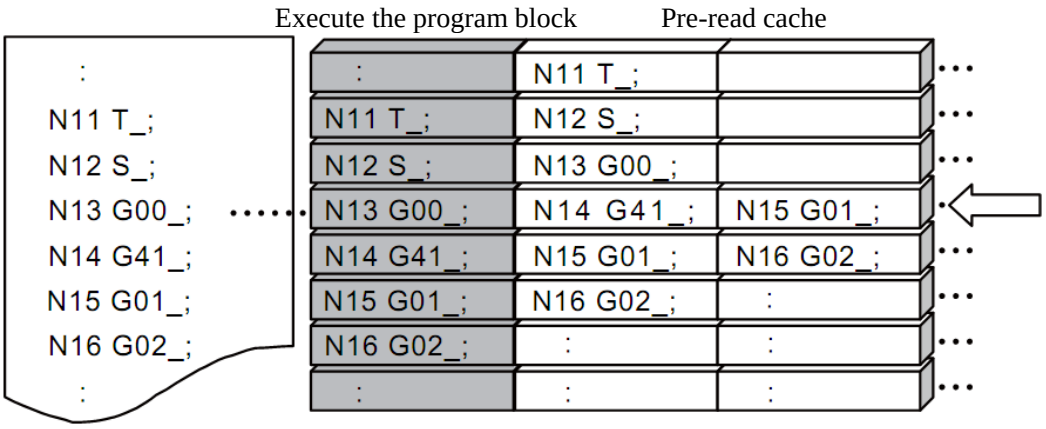
In the compensation cancellation state, the tool nose R compensation starts when all the following conditions are met.

1) There is a movement command after specifying G41 and G42.

2) There is a movement command other than the arc command.

At the start of compensation, regardless of whether it is continuous operation or singleblock operation, after 2 to 8 program blocks are continuously read due to the intersection operation, the tool nose R compensation is started. (In the movement command, if there are no movement commands in two continuous blocks, up to 8 program blocks are pre-read.)

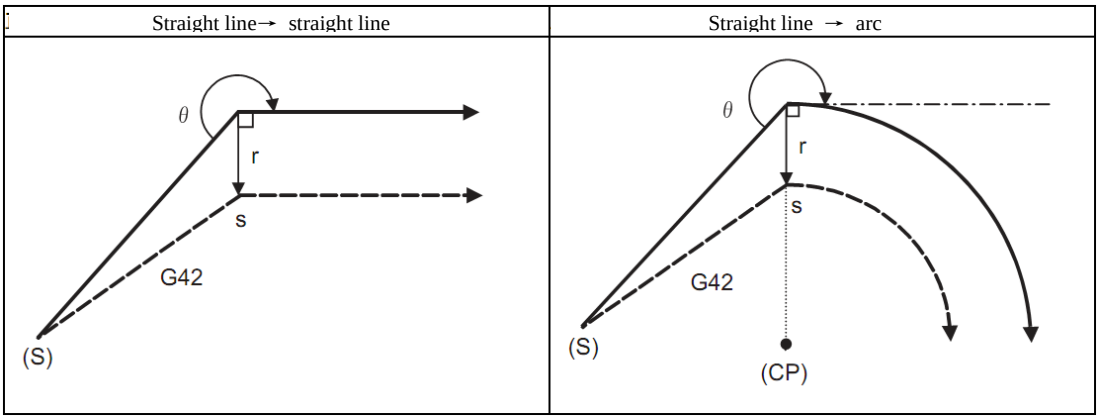
And in the compensation mode, it also pre-reads up to 8 blocks to execute the compensation operation.
[Control state diagram]



In the tool nose R compensation and radius compensation, when the start and cancellation commands are executed, the command program block and the next command program block are executed to perform the intersection calculation processing.

(3) Start action of the tool nose R compensation

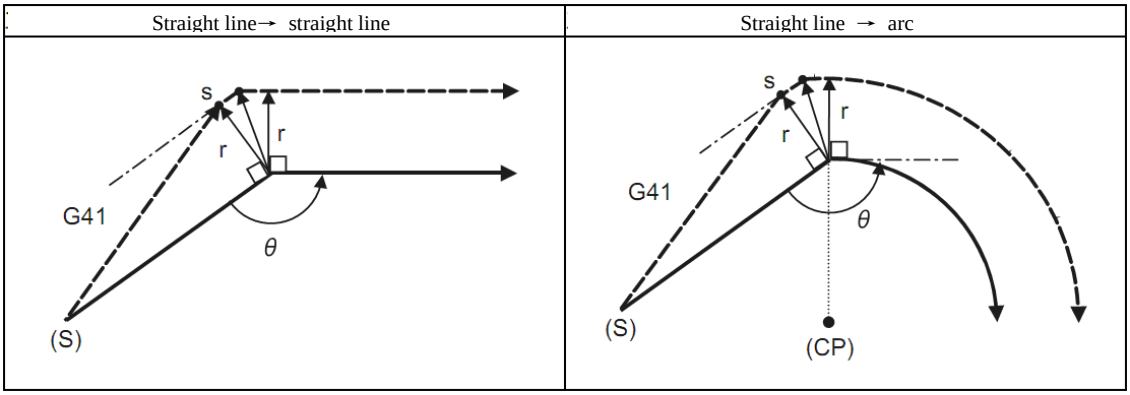
1) Inside the corner



(S): Start point (CP): Arc center r: Tool nose R radius s: Single-block stop point

——Program path - - - -Tool nose R center path

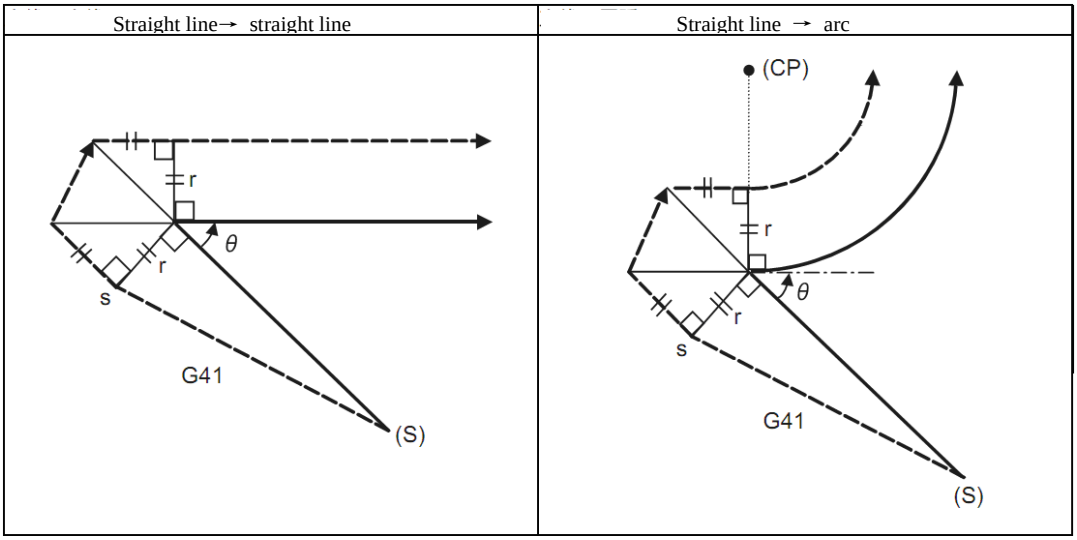
2) Outside the corner (obtuse angle), [90º≤θ＜180 º]



(S): Start point (CP): Arc center r: Tool nose R radius s: Single-block stop point

——Program path - - - -Tool nose R center path

3) Outside the corner (acute angle), [θ＜90 º]



(S): Start point (CP): Arc center r: Tool nose R radius s: Single-block stop point

——Program path - - - -Tool nose R center path

Note: When there is no axis movement command in the program block where G41 or G42 is located, the compensation action will be executed in the vertical direction of the next program block.

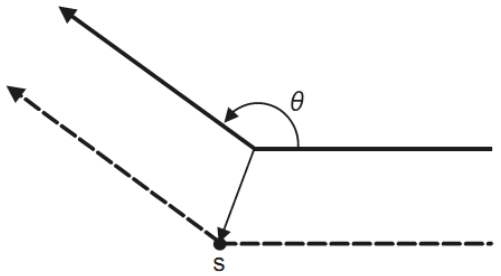
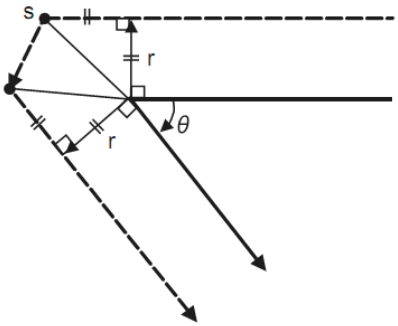
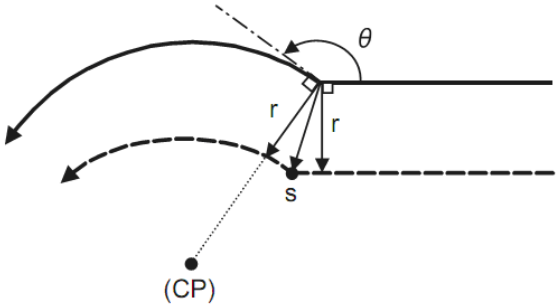
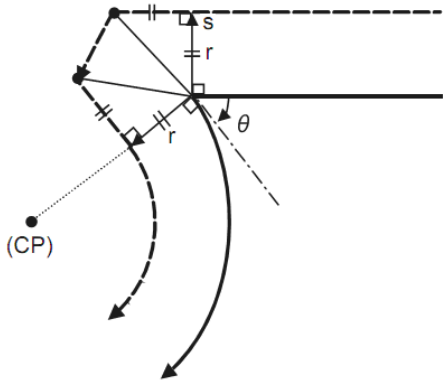
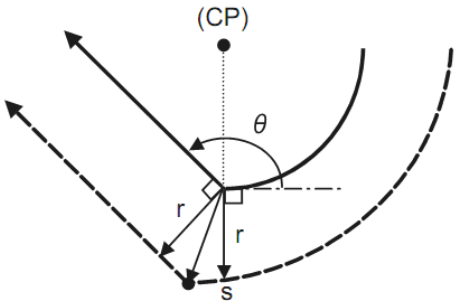
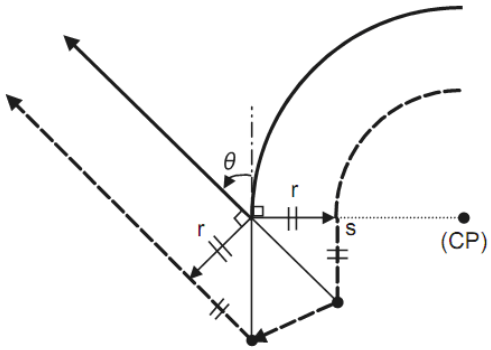
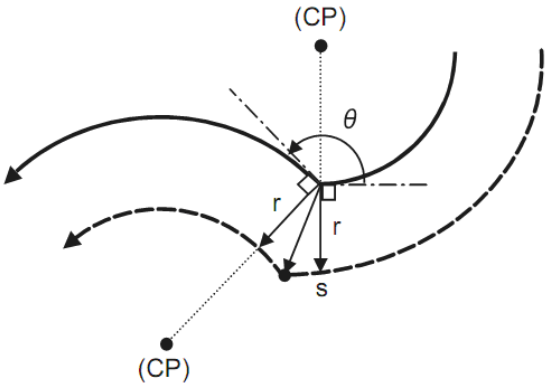
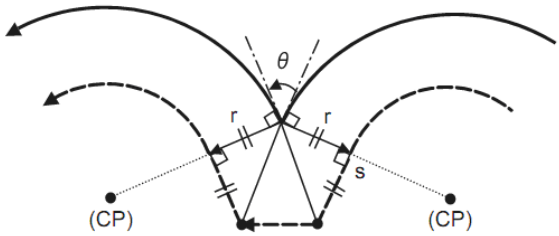
(4) Action in the compensation mode

For the program paths (G00, G01, G02, G03), calculate the tool center path according to the straight line/arc, and execute the compensation.

In the tool nose R compensation (G41, G42) mode, even if the same tool nose R compensation command (G41, G42) is specified, it will be ignored.

If more than 8 non-moving program block commands are continuously specified in the compensation mode, overcut or undercut will occur.

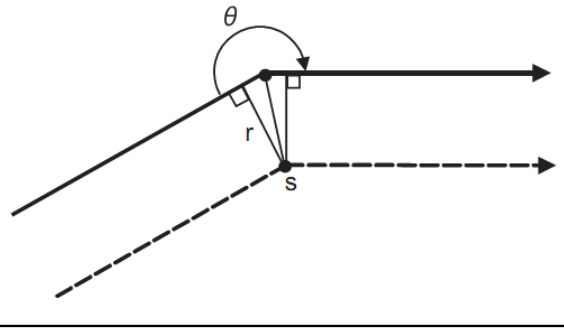
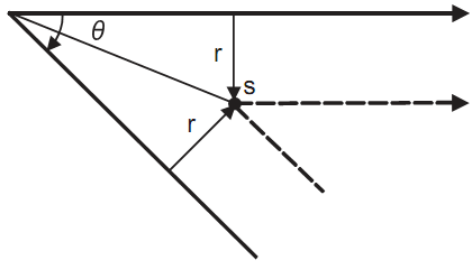
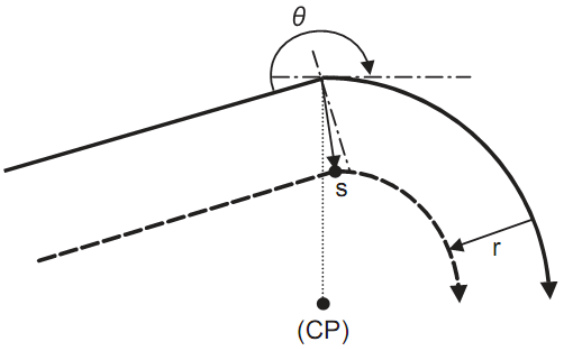
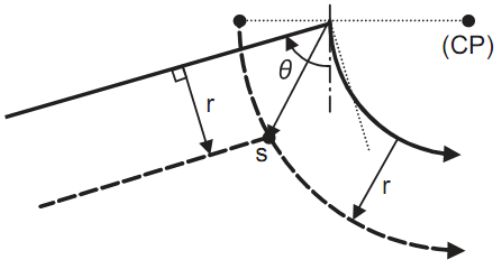
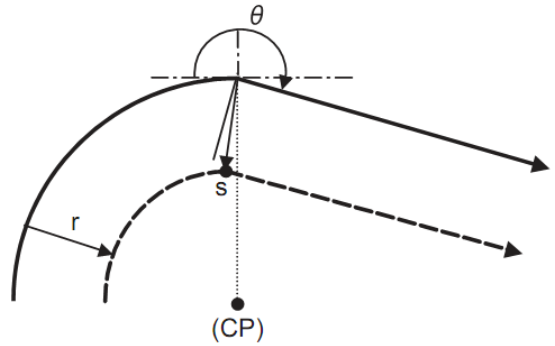
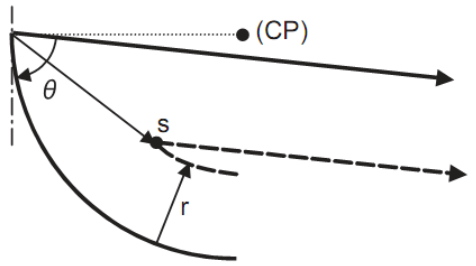
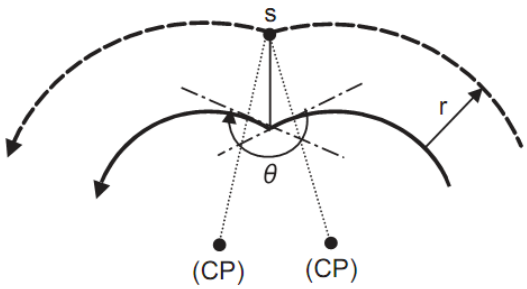
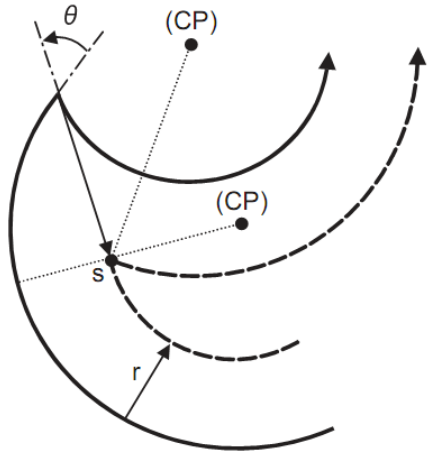
1) When rotating outside the corner

Straight line→ straight line(90º≤θ＜180º)	Straight line→ straight line(0º＜θ＜90º)
	
Straight line → arc(90º≤θ＜180º)	Straight line → arc(0º＜θ＜90º)
	
Arc→ straight line(90º≤θ＜180º)	Arc→ straight line(0º＜θ＜90º)
	
Arc → arc(90º≤θ＜180º)	Arc → arc(0º＜θ＜90º)
	

(S): Start point (CP): Arc center r: Tool nose R radius s: Single-block stop point

——Program path - - - -Tool nose R center path

2) When rotating inside the corner

Straight line→straight line (obtuse angle)	Straight line→straight line (acute angle)
	
Straight line→arc (obtuse angle)	Straight line→arc (acute angle)
	
Arc→straight line (obtuse angle)	Arc→straight line (acute angle)
	
Arc→arc (obtuse angle)	Arc→arc (acute angle)
	

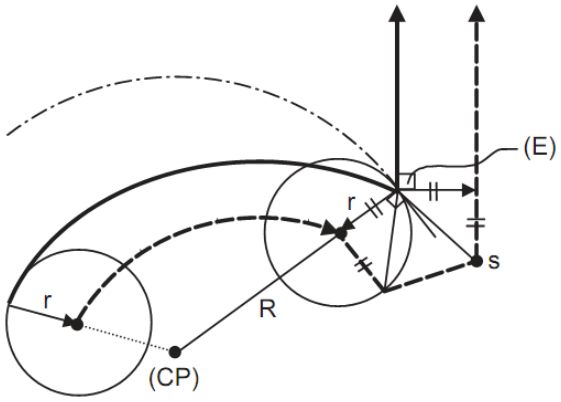
(S): Start point (CP): Arc center r: Tool nose R radius s: Single-block stop point

——Program path - - - -Tool nose R center path

3) When the arc end point is not on the arc

If the error after compensation is within the range set by parameter #1810, the spiral arc is used to

perform interpolation between the arc start point and end point.



(E) End point of arc
(CP) Arc center
r: Tool nose R radius

Program path
Tool nose center path

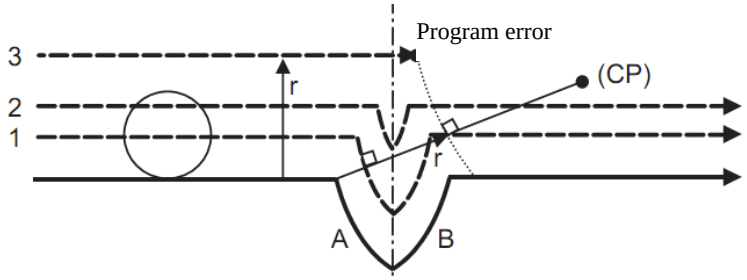
4) When there is no inner intersection point

As illustrated in the figure below, depending on the compensation amount, there may be no intersection point between arc A and arc B.

Then, at the end point of the previous program block, a program error is displayed, and the operation is stopped.

Patterns 1 and 2 in the figure are smaller than the tool nose R radius r, so there is an intersection point, and machining can be performed.

Pattern 3 is larger than the tool nose R radius r, so there is no intersection point, and a program error arises.



(CP) Center of arc A
r: tool nose R radius

Program path
Tool nose center path

17.4.3 Canceling tool nose R compensation

In the tool nose R compensation mode, the tool nose R compensation is cancelled if any of the following conditions are met.

However, now it must be a movement command other than the arc command.

A program error will arise if the compensation cancellation command is specified in the arc command.

(1) The G40 command is executed.

(2) The tool number T00 is executed.

After reading the compensation cancellation command, enter the cancellation mode, stop pre-reading 5 program blocks, and pre-read 1 block.

17.4.4 Cancellation action of the tool nose R compensation

When the tool nose R cancellation command is issued, the following operations are performed.

(1) When the tool nose R compensation is completed, only if G40 is specified and G00 is set before G40, the tool nose R compensation is temporarily cancelled, and it proceeds to the tool nose R compensation cancellation state.

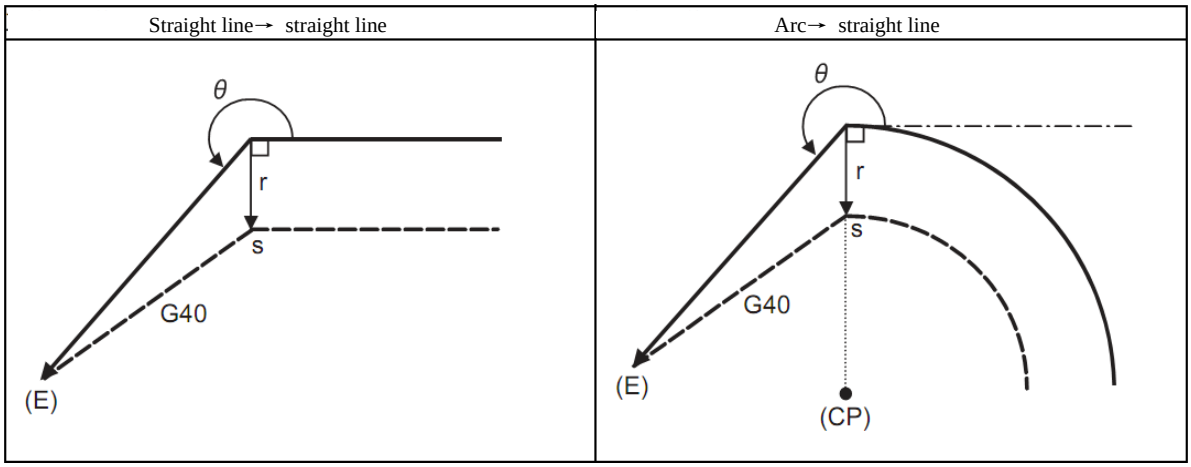
(2) When the tool nose R compensation is completed, the tool nose R compensation is not cancelled and the tool nose R center is stopped at the vertical position only when the G40 is specified and the interpolation

command comes before G40. The tool nose R compensation is cancelled by the first axis movement command after G40. Even if there is an axis command, the tool nose R compensation will not be cancelled without a movement command. There is no axis movement command after G40, and the tool nose compensation state is maintained when the program ends with the M0 command. The tool nose R compensation is cancelled upon reset, but without a cancellation operation.

(3) When the T00 command is issued alone, it goes to the tool nose R cancellation mode in this program block, and moves to the tool nose R cancellation position.

(4) The relationship between the inside/outside the corner and the cancellation:

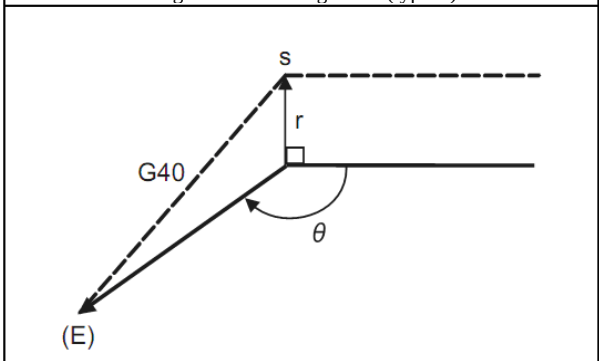
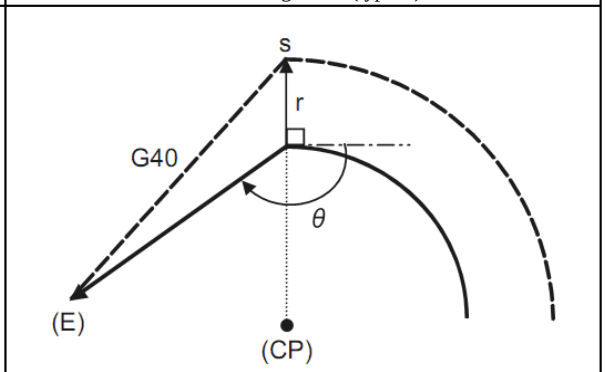
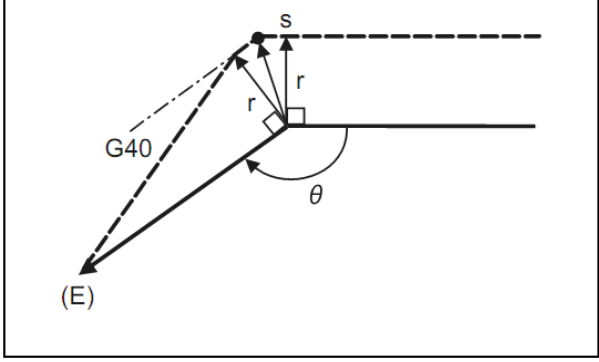
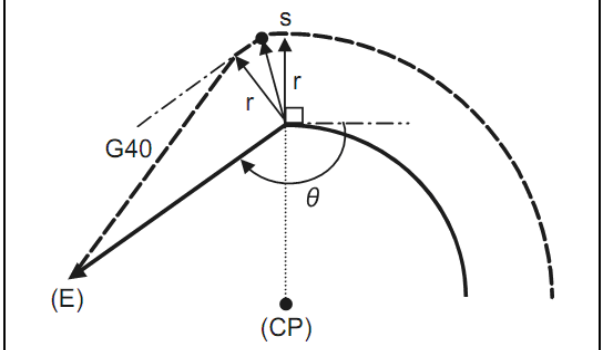
1) Inside the corner



(E): End point (CP): Arc center r: Tool nose R radius s: Single-block stop point

——Program path - - - -Tool nose R center path

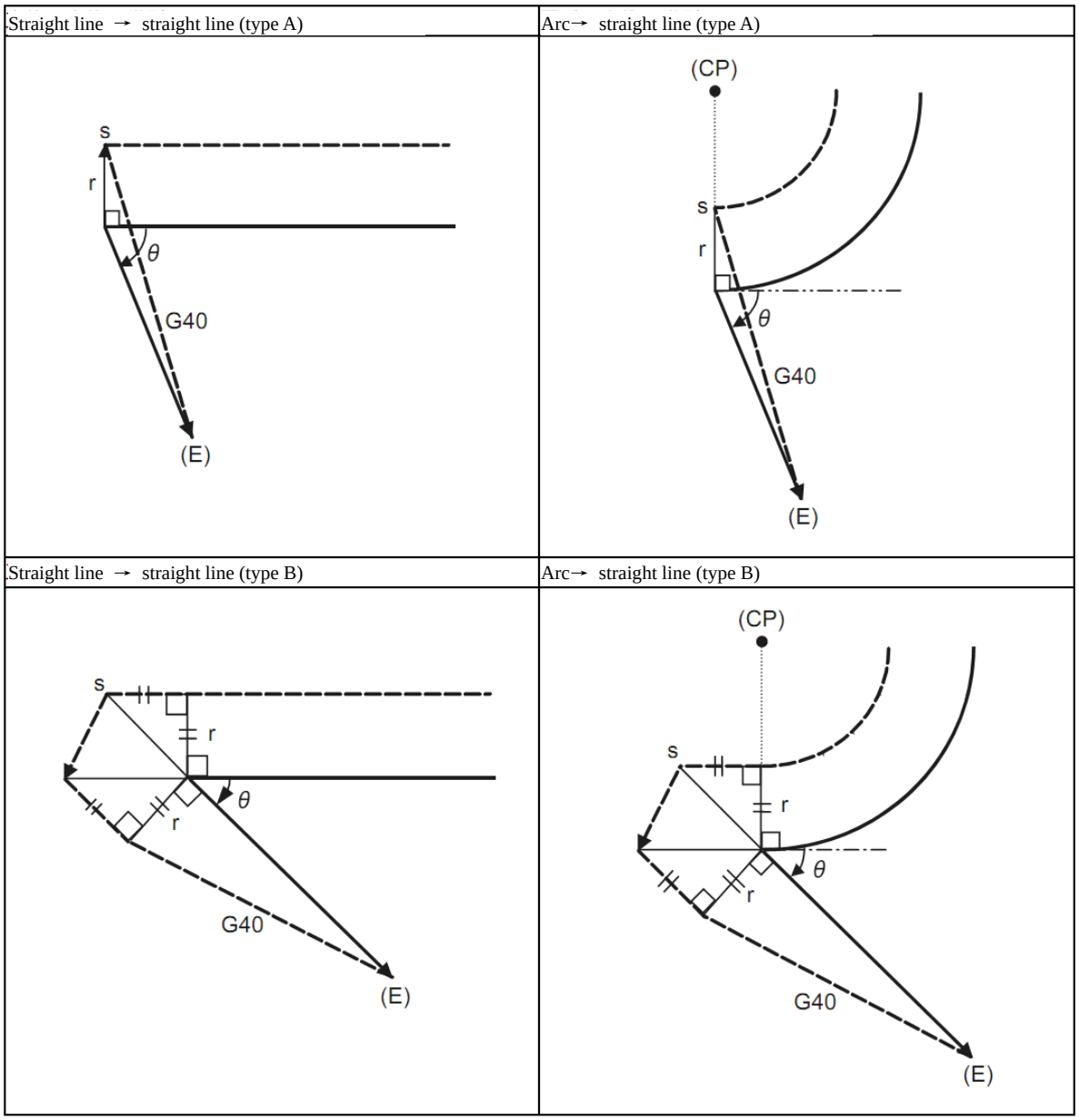
2) Outside the corner (obtuse angle)

Straight line → straight line (type A)	Arc→ straight line (type A)
	
Straight line → straight line (type B)	**Arc→ straight line (type B)**
	

(E): End point (CP): Arc center r: Tool nose R radius s: Single-block stop point

——— Program path - - - - Tool nose R center path

3) Outside the corner (acute angle)



(E): End point (CP): Arc center r: Tool nose R radius s: Single-block stop point

——Program path - - - -Tool nose R center path

17.4.5 Other actions of tool nose R compensation

Detailed description

(1) Change of the compensation direction during the tool nose R compensation

The compensation direction relies on the nose R compensation command (G41, G42).

G code	Compensation direction
G41	Compensation on the left side
G42	Compensation on the right side

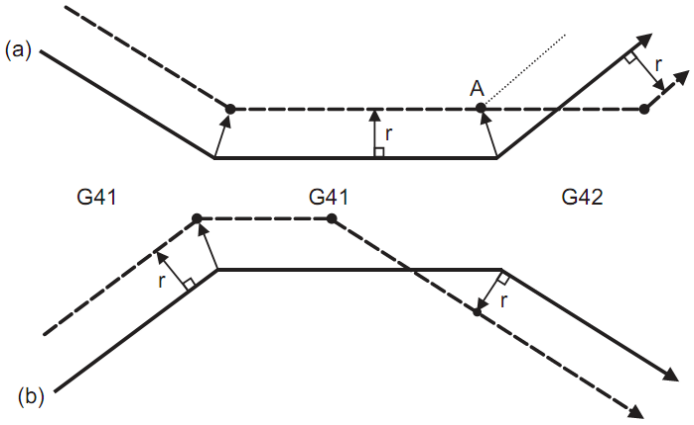
In the compensation mode, the compensation direction can be changed by changing the compensation

command without issuing a compensation cancellation command. However, the compensation start program block and the next block cannot be changed.

1) Straight line→straight line

(a) There is an intersection point when the compensation direction is changed (A in the figure)

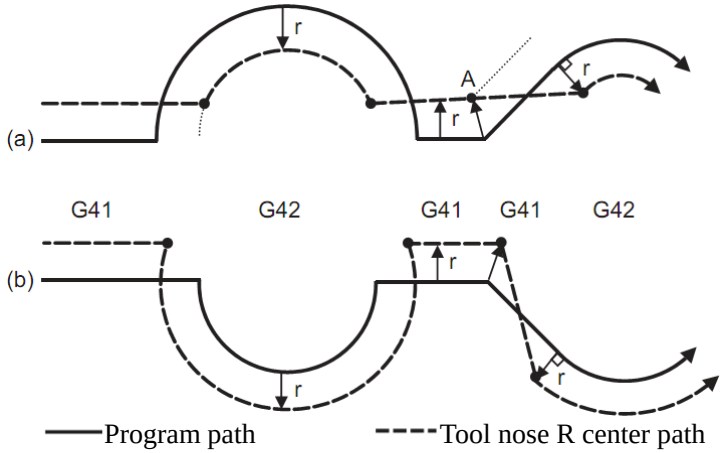
(b) There is no intersection point when the compensation direction is changed



2) Line←→arc

(a) There is an intersection point when the compensation direction is changed (A in the figure)

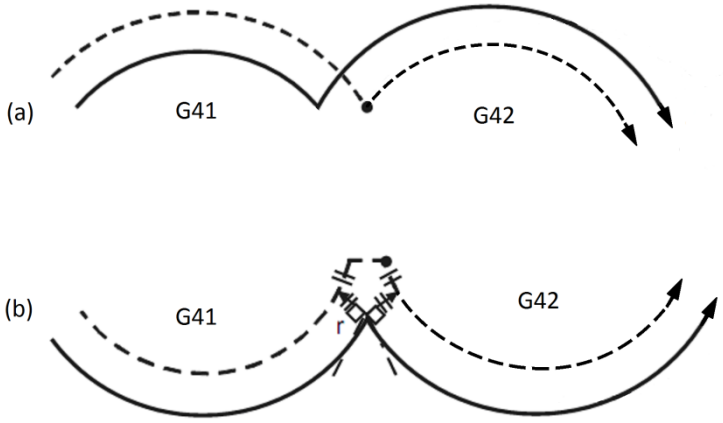
(b) There is no intersection point when the compensation direction is changed



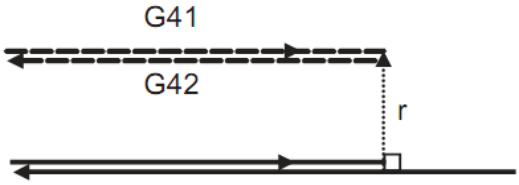
3) Arc→arc

(a) There is an intersection point when the compensation direction is changed

(b) There is no intersection point when the compensation direction is changed



4) When going back and forth in a straight line

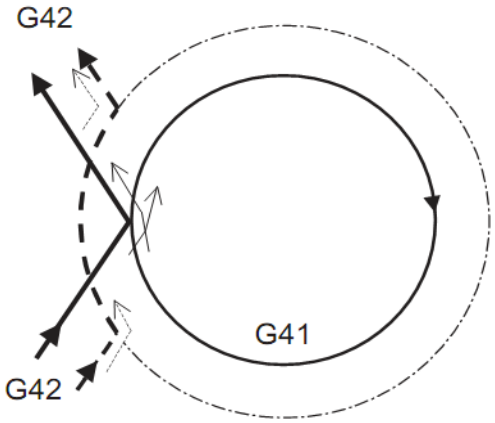


5) According to the compensation method, when the arc is larger than 360 °

In the following cases, the arc may be larger than 360 °.

a. The compensation direction is switched by switching G41/G42.

When the arc is larger than 360 °, the compensation illustrated in the figure is performed to generate the remaining part of the cutting.

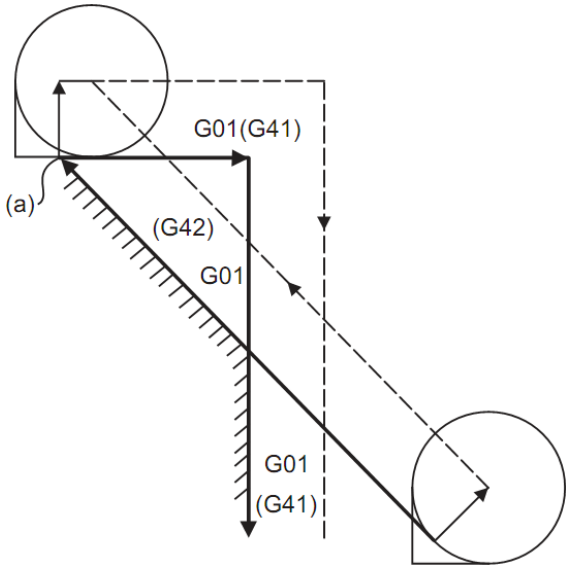


⋮
G42 G01 X_ Z_;
G41 G02 X_ Z_ I_ K_;
G42 G01 X_ Z_;
⋮

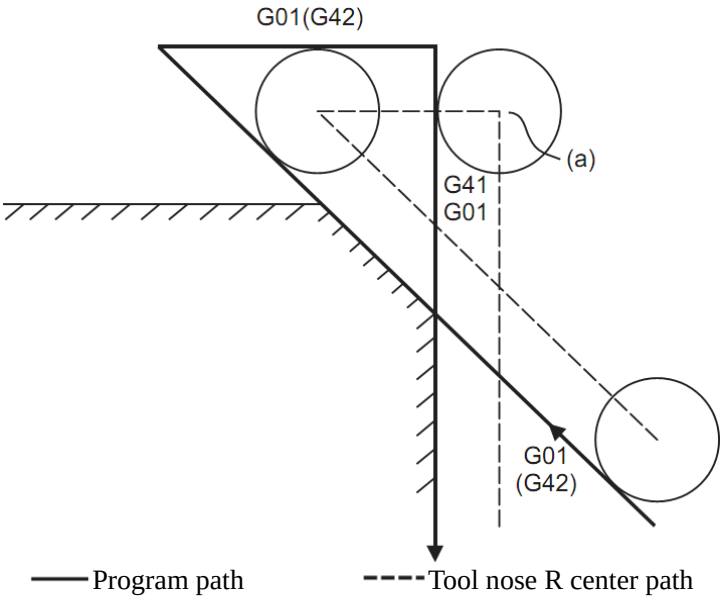
Program path Tool nose R center path Remainder of cutting

(2) Tool nose R compensation for the closed path performed by G41/G42

1) G42→G41 command action [G41 command is issued at the position (a) in the figure]



2) G42→G41 commanded action [G41 command is issued at the position (a) in the figure]



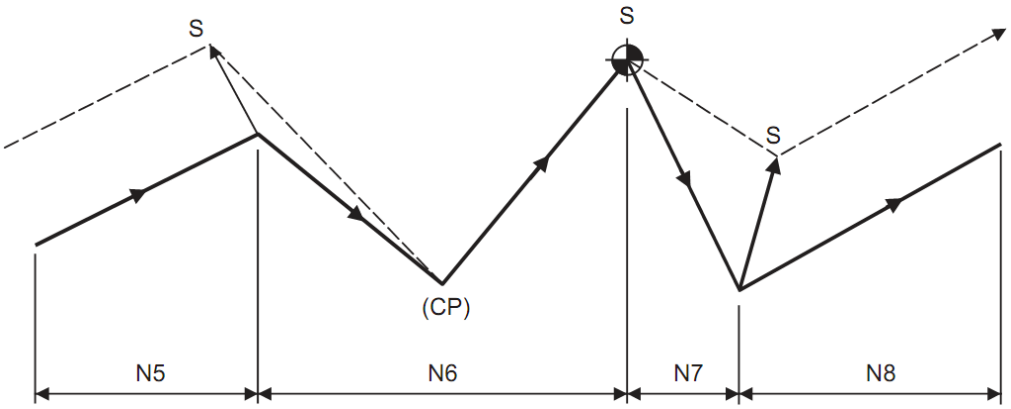
(3) Temporarily cancel the compensation vector command

In the compensation mode, the following command is issued to temporarily cancel the compensation vector, and then automatically return to the compensation mode.

Then, the compensation operation is not cancelled, and immediately after the intersection point vector, it moves to the vectorless point. That is, the command point of the program. When returning to the compensation mode, it also moves to the next intersection point.

1) Reference point return command.

At the intermediate point (if there is no intermediate point, it is the reference point), the temporary compensation vector is 0.



(CP) Intermediate point

```
(G41)    ⋮
N5 G01 U 30. W 60. ;
N6 G28 U-40. W 50. ;
N7       U-60. W 30. ;
N8       U 40. W 70. ;
         ⋮
```

2) In the basic machine coordinate system selection (G53), temporarily cancel the compensation vector.

Note: In the coordinate system shift (G50) command, the compensation vector does not change.

3) The fixed cycle for the combined turning

When a compound turning fixed cycle (G70, G71, G72, G73) is specified, the tool nose R compensation is temporarily cancelled. The cutting is performed with the finished shape in which the

tool nose R compensation has been cancelled, and it returns to the compensation mode automatically after completion.

(4) No moving program block

The program blocks shown below are called non-moving blocks.

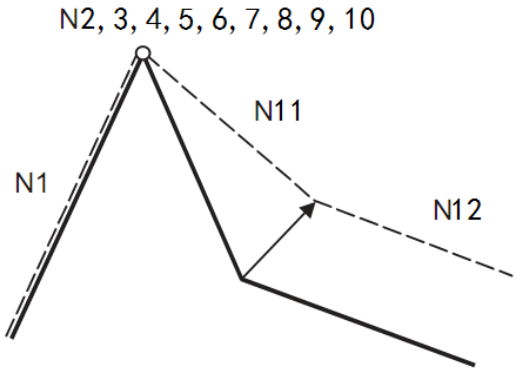
M03	M command
S12	S command
T0101	T command
G04 X500	Dwell
G10 L10 P01 R50	Compensation amount setting
G50 X600 Z500	Coordinate system determination
Y40.	Movement outside the compensation plane
G00	G code only
U0	Movement amount is 0

1) Specify at the start of compensation

Compensation vectors are not created when four or more continuous blocks are non-moving program blocks and the pre-reading prohibition M command is issued.

```
N1 U60.W30.T0101 ;
N2 G41;
N3 G4 X1000;
N4 G95;
N5 S500;
N6 M3;
N7 M8;
N8 G98;
N9 F100;
N10 G1;
N11 U- 50.W20. ;
N12 U- 20.W50. ;
```

Non-moving program blocks

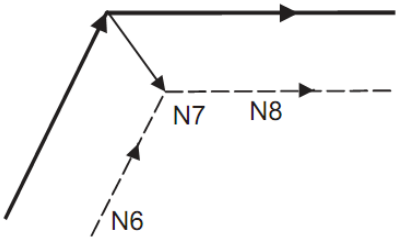


2) When the compensation mode is specified

In the compensation mode, the blocks within 8 continuous program blocks are non-moving program blocks, and a usual intersection point vector is created.

```
N6 U200. W100. ;
N7 G04X P1000 ;
N8 W200. ;
```

Non-moving program blocks

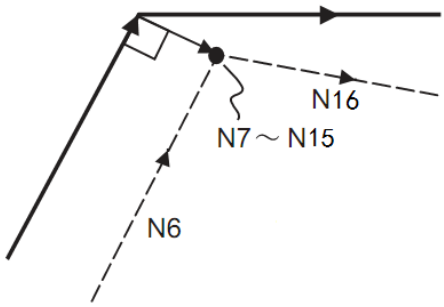


N7 in the figure represents the execution of the N7 program block.

When eight or more continuous program blocks are non-moving program blocks, a vertical compensation vector is created at the end point of the previous program block.

At this point, a cut-in may occur.

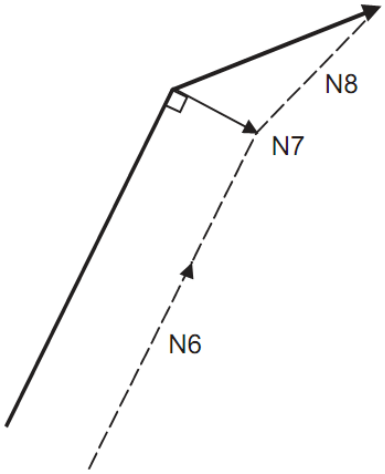
N6 U200. W100. ;
N7 G4 X1000 ;
N8 F100 ;
⋮ Non-moving program blocks
N14 S500 ;
N15 M4 ;
N16 W100. ;



3) Issue command at the same time as compensation cancellation

When a program block without movement is specified at the same time as G40, only the compensation vector is cancelled.

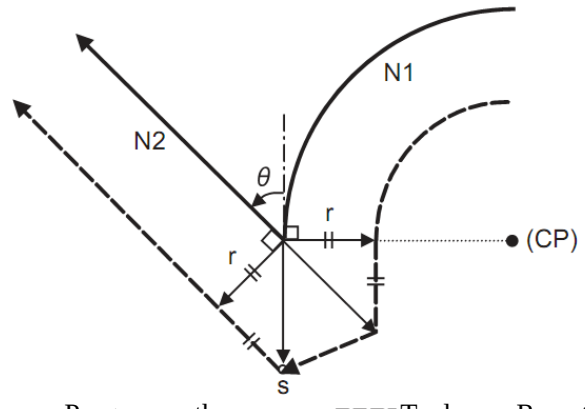
N6 U200. W100. ;
N7 G40 M5 ;
N8 U50. W100. ;



(5) Movement of corners

When there are multiple compensation vectors at the junction of the movement command program blocks, the movement is performed between the vectors in a straight line. Such an action is called corner movement. When the vectors do not coincide, the movement for rotating the corner is performed.

For actions in a single block operation, the previous program block and the corner movement are executed as one block. At the next start, the remaining movement and the next program block are executed as one program block.



(CP) Arc center
(r) Tool nose R radius
(s) Single-block stop point

Program path ---- Tool nose R center path

17.4.6 General precautions for tool nose R compensation

Precautions

(1) Compensation amount

1) Usually, the compensation amount number is specified by the lower 2 digits of the T code.

Once a T code is specified, that T code is valid until another T code is specified.

The T code is used to specify the tool length offset amount, in addition to the tool nose R compensation amount.

2) Usually, the compensation amount is changed when another tool is selected in the compensation cancellation mode. When the compensation mode is changed, the vector of the end point of the program block is calculated using the compensation amount specified in the program block.

(2) Error of the tool nose R compensation

1) An error will arise if a program containing the following commands is created during the tool nose R compensation.

G17, G18, G19 (when a plane different from the compensation plane is specified)

2) If there is an arc command in the start program block or end program block of the tool nose R compensation action, an error will arise.

3) When executing the tool nose R compensation, if the intersection point when skipping one program block is not calculated during the interference program block processing, a program error will arise.

4) When the tool nose R compensation is executed, if the pre-read program block is wrong, a program error will arise.

5) When the tool nose R compensation is executed, if interference occurs without interference avoidance, a program error will arise.

6) A program error will arise if a tool nose R compensation command is issued from a device that cannot satisfy the tool nose R compensation specification.

17.4.7 Interference check

Function and purpose

Usually, the tool nose compensated in the tool nose R compensation may cut into workpiece by pre-reading 2 program blocks.

This phenomenon is called interference, and the function to avoid interference is called interference check.

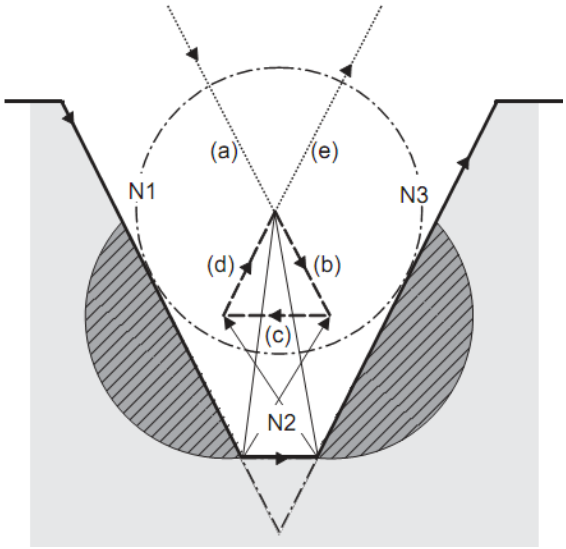
The interference check has the following two functions. It can be selected by a parameter.

	Function	Parameter #2602bit1	Action
(1)	Perform the interference check	0	A program error arises before the program block where the cut-in occurs, thus resulting in a stop.
(2)	Interference check is not performed	1	The cutting may be continued even if the cut-in occurs.

Detailed description

(Example)

(G41)
N1 G1 X100. Z50.
N2 Z70;
N3 X300. Z120.;



(1) During the alarm function
An alarm is given when N1 is executed.
(2) When the interference check cannot function effectively
Pass through the straight lines N1 and N3 while cutting into them.
The center path of the tool nose R is (a)→(b)→(c)→(d)→(e).
(3) Interference check alarm
Please refer to "16.3.5 Interference check" in Chapter 16 of this part.

17.5 Programmable compensation input (G10 L2/L10/L11)

Function and purpose

The tool and workpiece offset amounts can be set/changed by the G10 command. When a command is issued with the absolute value (X, Z, R), the compensation amount is a new amount, and when a command is issued with the incremental value (U, W, C), the currently set compensation amount is accumulated after the specified compensation amount, and used as a new compensation amount.

Command format

G10 L2 P_ X(U)_ Z(W)_;	Workpiece offset input (L2)

P	Compensation number
X	X-axis compensation amount (absolute)
U	X-axis compensation amount (increment)
Z	Z-axis compensation amount (absolute)
W	Z-axis compensation amount (increment)

G10 L10 P_ X(U)_ Z(W)_ R(I)_ Q_ ;	Tool length compensation input (L10)

P	Compensation number

X	X-axis compensation amount (absolute)
U	X-axis compensation amount (increment)
Z	Z-axis compensation amount (absolute)
W	Z-axis compensation amount (increment)
R	Tool nose R compensation amount (absolute)
I	Tool nose R compensation amount (increment)
Q	Imaginary tool nose point

G10 L11 P_ X(U)_ Z(W)_ R(I)_ Q_ ; Tool nose wear compensation input (L11)

P	Compensation number
X	X-axis compensation amount (absolute)
U	X-axis compensation amount (increment)
Z	Z-axis compensation amount (absolute)
W	Z-axis compensation amount (increment)
R	Tool nose R compensation amount (absolute)
I	Tool nose R compensation amount (increment)
Q	Imaginary tool nose point

A program error occurs when there is no L command in the tool length compensation input (L10) or tool nose wear compensation input (L11).

Detailed description

(1) The setting range of compensation number and imaginary tool nose point is briefed as follows.

Address	Address Meaning	Predetermined Area		
		L2	L10	L11
P	Compensation number	0: external workpiece offset	1～maximum number of compensation groups	1～maximum number of compensation groups
		1: G54 workpiece offset		
		2: G55 workpiece offset		
		3: G56 workpiece offset		
		4: G57 workpiece offset		
		5: G58 workpiece offset		
		6: G59 workpiece offset		
Q	Imaginary tool nose point		0～9	

Note: In the tool compensation input (L10/L11), the maximum number of compensation groups for P (compensation number) is up to 80 groups in total (the number of groups varies according to the model, please confirm the specifications).

(2) The set unit of compensation amount is specified as follows.

After changing the unit of the command value, if there is a value that does not match the value in the table, a program error will arise.

If it is an incremental value command, the set range of compensation amount is the sum of the current set

value and the command value.

The maximum wear compensation amount is set by parameter 2611.

Tool Length Compensation		Wear Compensation Amount	
Metric system	Imperial system	Metric system	Imperial system
±9999.9999(mm)	±9999.9999(inch)	±100(mm)	±100(inch)

Precautions

(1) Check the set range of compensation amount
In the command of the maximum value and incremental value of the wear compensation amount, the maximum value and the maximum incremental value of the wear data checked by the wear compensation input are given priority in each compensation amount. When the wear compensation amount is larger than this value, a program error will arise.

(2) G10 L2/L10/L11 are non-modal and valid only for the specified program block. The exception is the G10 L50.

(3) A compensation input is also available for the 3rd axis. Even if the C-axis is specified as the 3rd axis, in L10 and L11, the address I can be used as the incremental value command of the tool nose R.

(4) A program error will arise if an incorrect L number or tool offset number is specified.

(5) If there is no L address in G10 command, the system will generate PS0014 alarm.

(6) A program error will arise when the P command is omitted from the workpiece offset input.

(7) A program error will arise when the compensation amount exceeds the set range.

(8) When X, Z and U, W are mixed in a single program block. However, when the same compensation input address such as X, U or Z, W is specified, the address input later becomes effective.

(9) A compensation input is executed as long as one address after G10L (2/10/11) P_ is specified. If not specified, a program error will arise.

(Example) G10 L10 P3 Z50. ;

Result of the execution:

[Offset setting interface]

NO.	Z
3	50.000

(10) The compensation amount is valid if it is a decimal point.

(11) When G40~G42 and G10 are specified in the same program block, G40~G42 will be memorized.

(12) Do not specify the fixed cycle, subprogram call commands and G10 in the same program block. Otherwise, a false action or program error may arise.

(13) When G10 and T commands are specified in the same program block, a compensation is performed in the next block.

Chapter XVIII Coordinate System Setting Function

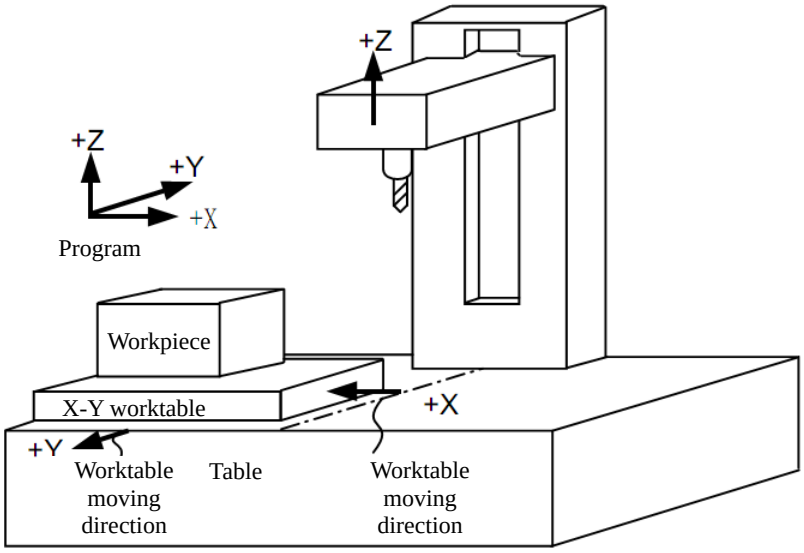
18.1 Coordinate system and control axis

Function and purpose

Milling

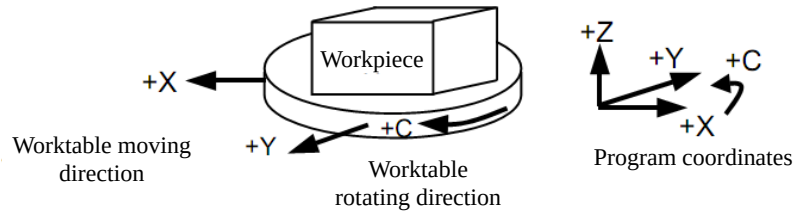
The number of control axes in the standard specification is 3-axis. Up to 8 axes can be controlled by adding auxiliary axes. The specification of each machining direction uses its corresponding coordinate language.

When X-Y worktable



When X-Y worktable

When X-Y & rotary worktable



Turning

For a lathe, define the axis name (coordinate system) and direction as follows.

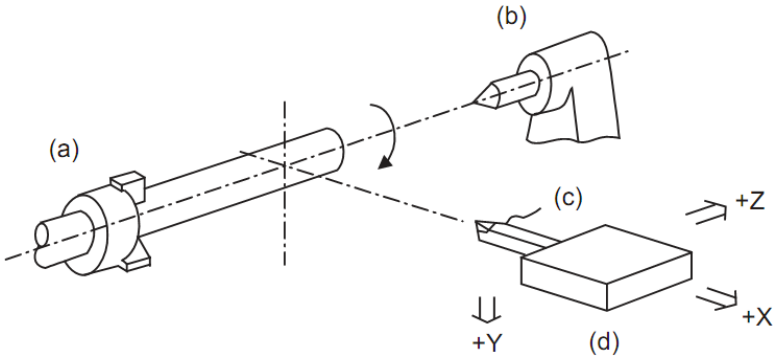
Axis perpendicular to the spindle

Axis name: X-axis

Axis parallel to the spindle

Axis name: Z-axis

(1) Coordinate axis and motion direction

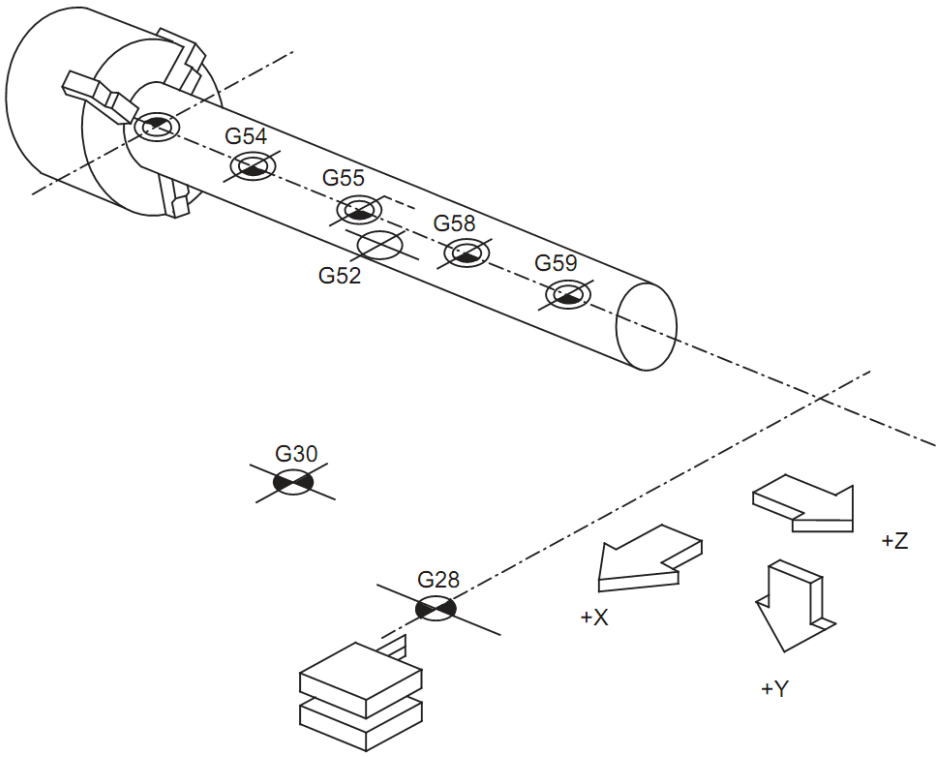


(a) Chuck
(b) Tailstock
(c) Tool
(d) Tool holder

The lathe uses a right-handed coordinate system. As illustrated in the figure above, the Y-axis, which is perpendicular to the X- and Z-axis, takes the downward direction as the positive direction in the figure. Viewed from the positive direction of the Y-axis, the arcs on the X and Z planes are embodied by clockwise and counter-clockwise rotations, please be careful.

Refer to "4.4 circular interpolation (G02, G03)" in Chapter 4 of this part.

(2) Relationships between the coordinates



Reference point

Origin of the workpiece coordinate

Origin of the machine coordinate

Origin of the local coordinate

18.2 Basic machine coordinate system, workpiece coordinate system and local coordinate system

Function and purpose

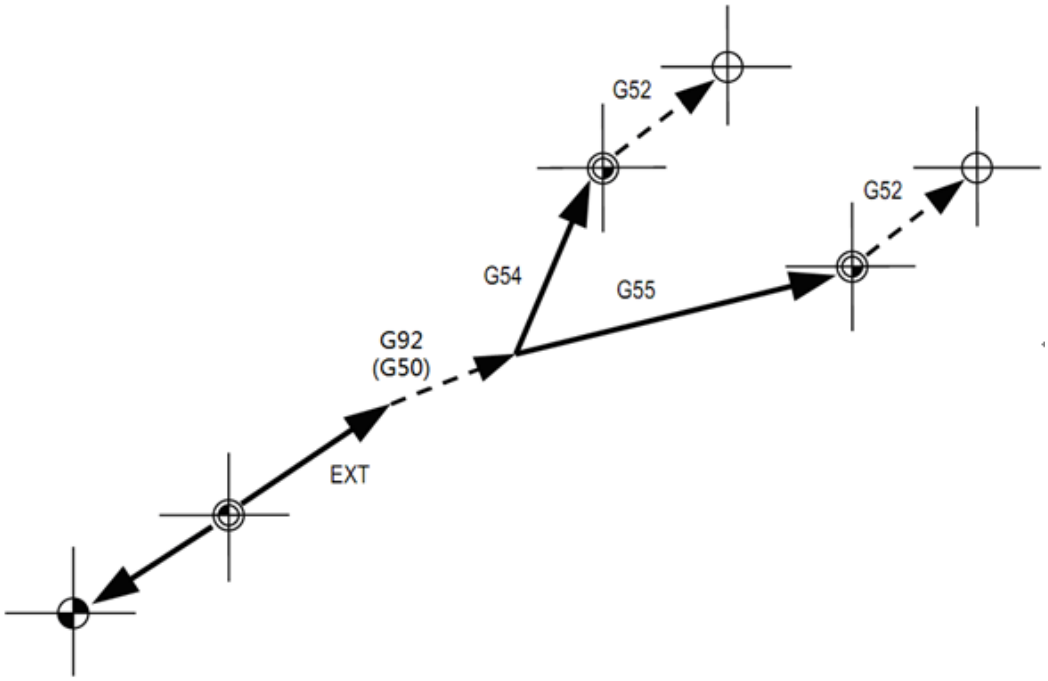
The basic machine coordinate is a machine-specific coordinate system that shows the machine-specific position.

The workpiece coordinate system is a coordinate system used by the programmer when programming, which is set with the datum point on the workpiece as the point of origin of the coordinate.

The local coordinate system is a coordinate system created on the workpiece coordinate system, with the aim to facilitate the creation of some machining programs.

When the basic machine coordinate system and workpiece coordinate system (G54~G59) complete the reference point return, the reference parameters will be automatically set.

Then, the basic machine coordinate system is set so that the first reference point is moved from the origin point of the basic machine coordinate (machine zero point) to the position specified by the parameter.



Reference point
Origin point of the machine coordinates
Origin point of the workpiece coordinate system
Origin point of the local coordinate system

Offset set in the parameter

Offset set in the program
(0 when power on)

G52 local coordinate system offset (*1)
G54 workpiece coordinate system (G54) offset (*1)
G55 workpiece coordinate system (G55) offset
G92 (G50). shift of the coordinate system
EXT external workpiece coordinate offset
(*1) Each of G54~G59 has a G52 offset independently.

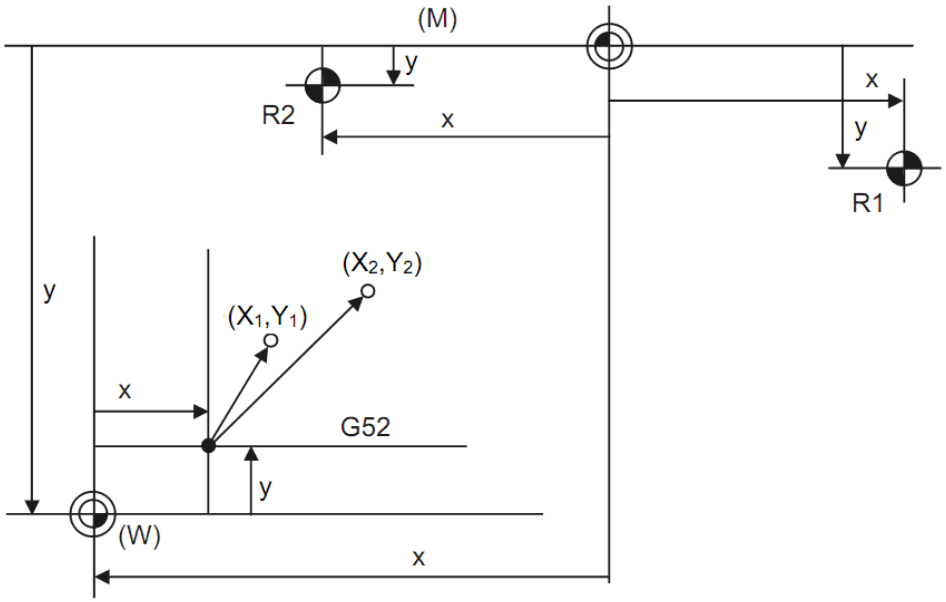
18.3 Machine origin and second reference point

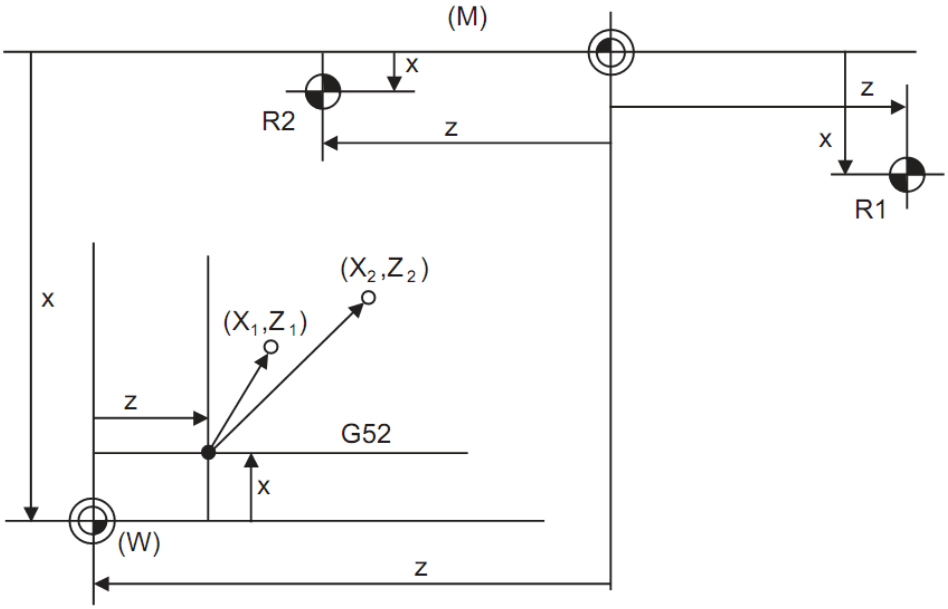
Function and purpose

The machine origin is the datum point of the basic machine coordinate system, which is a point unique to the

machine determined by the first reference point, and is set by parameter #1050.

The second reference point is the origin determined from the basic machine coordinate system, and the position of the coordinate value set in advance by the parameter.





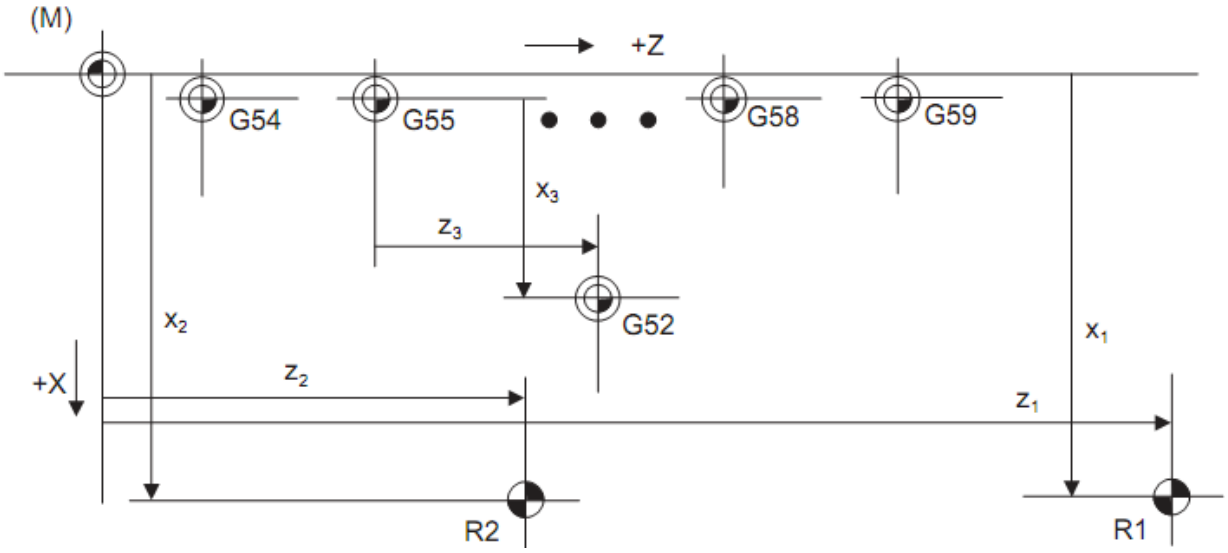
(M) Basic machine coordinate system. (G52) Local coordinate system (R1) 1st reference point
(R2) Second reference point (W) Workpiece coordinate system (G54~G59)

18.4 Automatic coordinate system setting

Function and purpose

This function creates each coordinate system according to the parameter values input through the setting and display device in advance when the reference point is returned through the dog-type reference point after the NC is powered on.

It edits the actual machining program in the coordinate system set as above.



(M) Basic machine coordinate system	(R1) 1st reference point	(R2) 2nd reference point	(G54) Workpiece coordinate system 1
(G55) Workpiece coordinate system 2	(G58) Workpiece coordinate system 5	(G59) Workpiece coordinate system 6	(G52) Local coordinate system

Detailed description

(1) The coordinate system created by this function is briefed as follows.
  (a) Basic machine coordinate system
  (b) Workpiece coordinate system (G54~G59)
    The local coordinate system (G52) is cancelled.

(2) All the parameters related to the coordinate systems are all assigned by the distance from the origin point of the basic machine coordinate system. Therefore, it is necessary to set the origin position of the workpiece coordinate system after obtaining the position of the first reference point in the basic machine coordinate system.

(3) When the automatic setting function of the coordinate system is executed, the workpiece coordinate system shift by G92 (G50), local coordinate system setting by G52, workpiece coordinate system shift by the initial setting, and workpiece coordinate system shifted by manual inserting are cancelled.

(4) At the first manual or automatic reference point return after power-on, the dog-type reference point return is performed. On the other hand, when "dog type" is selected through parameter, the dog type reference position return will be executed at the second or subsequent manual or automatic reference position return.

Precautions

If the workpiece coordinate offset is changed during autorun (including single program block operation), the command will take effect from the next program block or after multiple blocks.

18.5 Basic machine coordinate system selection (G53)

Function and purpose

The tool is moved through the G53 command to the commanded position on the basic machine coordinate

system by the coordinate command after the rapid feed mode.

Command format

Milling

(G90/G91) G53 X_ Y_ Z_ α_;

α	Additional axis

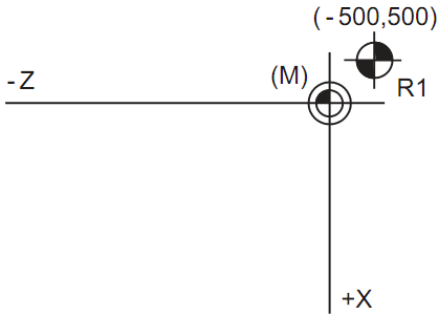
Turning

G53 X(U)_ Z(W)_ α(β)_;

α	Additional axis
β	Incremental value command of the additional axis

Detailed description

(1) The basic machine coordinate system is automatically set based on the specified reference point return position through automatic or manual reference point return when power on.
(2) The basic machine coordinate system cannot be changed by the G50 command.
(3) The G53 command is only valid for the specified program block.
(4) The first reference point coordinate value indicates the distance from the origin of the basic machine coordinate system to the reference point return position.
(5) The G53 command executes the movement by rapid feed.



(M) Basic machine coordinate system
(R1) 1st reference point

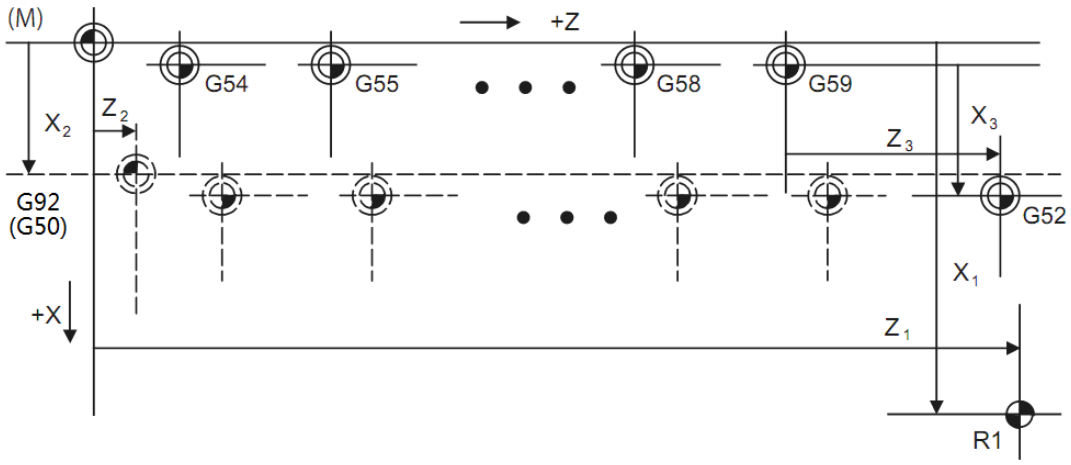
The first reference point coordinate value X=-500 Z=+500.

18.6 Coordinate system shift (G92 (G50))

Function and purpose

The tool is positioned to an arbitrary position, and the coordinate system is set by issuing the G92 command for coordinate system shift at that position.

The coordinate system can be set arbitrarily, but usually the X-axis workpiece origin is set on the workpiece axis line, and the Z-axis workpiece origin is set on the workpiece end face.



(M) Basic machine coordinate system
(G92) Imaginary machine coordinate system (shift by G92)
(G54) Workpiece coordinate system 1
(G55) Workpiece coordinate system 2
(G58) Workpiece coordinate system 5
(G59) Workpiece coordinate system 6
(G52) Local coordinate system
(R1) 1st reference point

Command format

Milling

(G90/G91) G92 X_ Y_ Z_ α_;

α	Additional axis

Turning

G50 X(U)_ Z(W)_ α(β)_;

α(β)	Additional axis

Detailed description

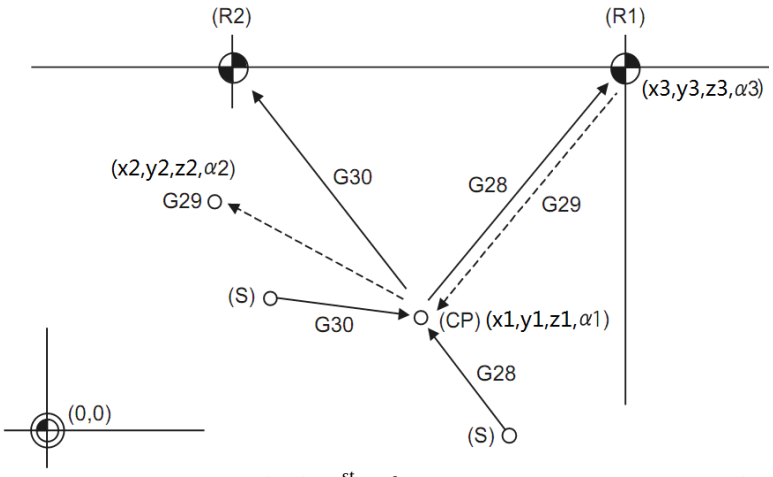
(1) Specify the G92 (G50) command to move the workpiece coordinate system (choose code from G54~G59) and set a new workpiece coordinate system, in which the current tool position is consistent with the specified coordinate.
(2) Move the basic machine coordinate system to create an imaginary machine coordinate system by the G92 (G50) command, and move the workpiece coordinate systems 1~6 simultaneously.
(3) The workpiece coordinate system offset by the G92 (G50) command is not saved when the system is powered off, and it is 0 when power on.
(4) If the G92 (G50) and S or Q command are issued at the same time, the spindle clamp speed is set. For details, please refer to "14.5 Spindle clamping speed setting" in Chapter 14 of this part.

18.7 Reference point return (G28, G29)

Function and purpose

The function can return each commanded axis to the 1st reference point by rapid feed after issuing the G28 command and specifying the axis positioning by G0.

By issuing the G29 command, each axis is positioned independently and at high speed to the intermediate point of G28 or G30, and then the positioning is performed at the commanded position by G0.



(CP) Intermediate point (R1) 1st reference point (R2) 2nd reference point
(S) Start point

Command format

G28 Xx1 Yy1 Zz1 αα1 ;	Reference point return
G29 Xx2 Yy2 Zz2 αα2 ;	Homing from reference point

αα1/αα2	Additional axis

Detailed description

(1) The G28 command is equivalent to the following command.
G00 Xx1 Yy1 Zz1 αα1 ;
G00 Xx3 Yy3 Zz3 αα3 ;
Here x3, y3, z3, α3 are the coordinate values of the reference point. Set the distance from the origin of the basic machine coordinate system with parameter "#1050".

(2) After power-on, the axis that is not manually returned to the reference position is the same as the manual dog-type reference position return. Now, the return direction is regarded as the direction of the command symbol, and for the second time, it returns at high speed to the reference point memorized for the first time.

(3) When the reference point return is completed, the output origin reaches the output signal.

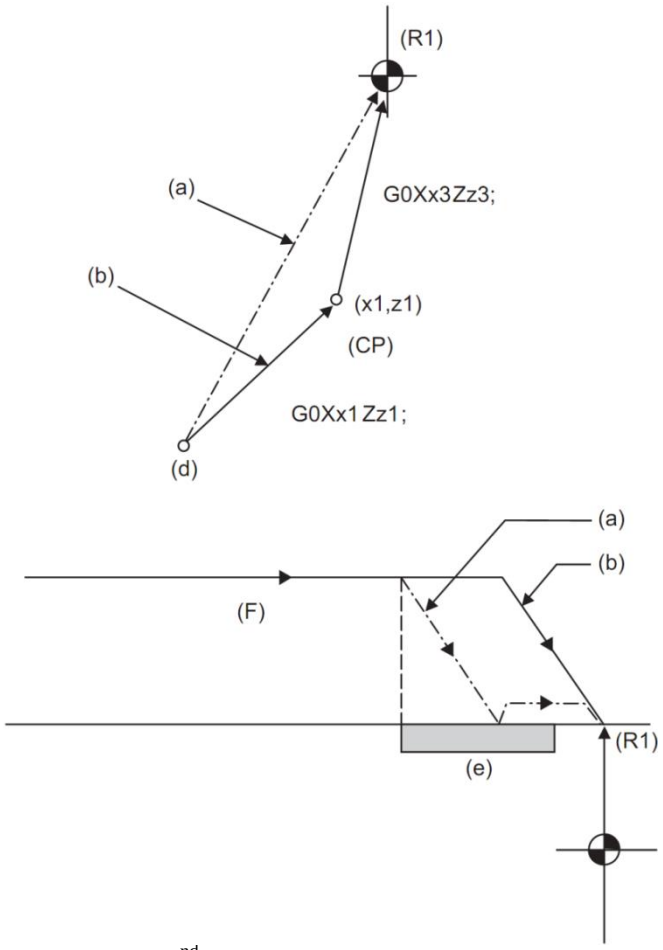
(4) The G29 command is equivalent to the following command.
G00 Xx1 Yy1 Zz1 αα1 ;
G00 Xx2 Yy2 Zz2 αα2 ;
Independent rapid feed of each axis (non-interpolation type).
Here x1, y1, z1, α1 are the coordinate values of the intermediate point of G28 or that of G30.

(5) Specify the intermediate point coordinate value (x1, y1, z1, α1) of the positioning point by absolute value/incremental value.

(6) G29 is valid for both G28 and G30. However, after returning to the latest intermediate point, the positioning of the specified axis is performed.
(7) When returning to the reference point, the tool offset is not cancelled, the intermediate point is the position after compensation, and the compensation is temporarily cancelled when returning to the reference point from the intermediate point.
(8) When the machine is locked, the path of returning from the reference point is the same as that of returning to the reference point when the machine lock is turned off. The next program will not be executed until the "G28" command completes the return to the reference point.
(9) The reference point return is executed in the mirror, but the mirror at the intermediate point is invalid.

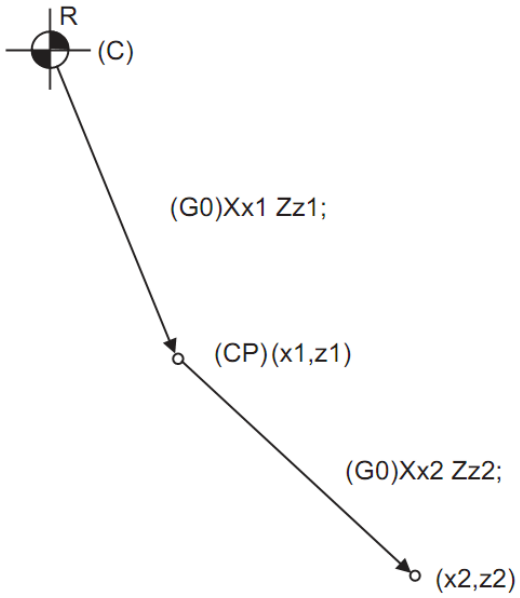
Program examples

Example 1 G28 Xx1 Zz1;



(a) 1st operation after power-on (b) 2nd and subsequent operations (d) Return to start position
(e) Proximity dog (F) Rapid feed speed (CP) Intermediate point (R1) Reference point position (#1)

Example 2 G29 Xx2, Zz2;



(C) Current position (CP) G28, intermediate point of G30

Example 3 G28 Xx1 Zz1 ;

⋮ (from point A to 1st reference point)

⋮

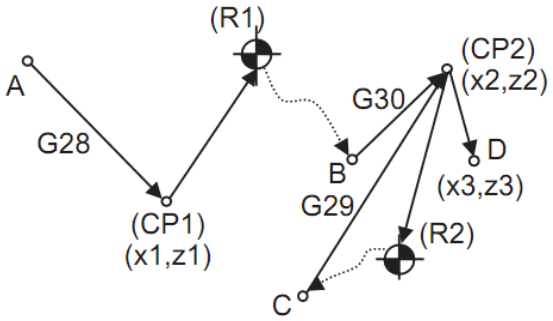
G30 Xx2 Zz2 ;

⋮ (from point B to 2nd reference point)

⋮

G29 Xx3 Zz3 ;

(from point C to point D)



(CP1) Old intermediate point

(CP2) New intermediate point

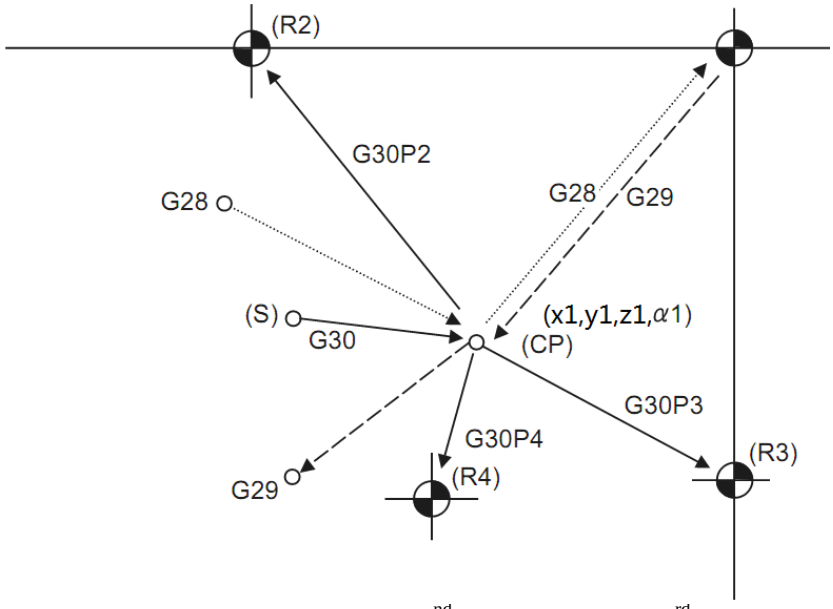
(R1) Point of reference point (#1)

(R2) Position of 2nd reference point (#2)

18.8 Return to the second, third and fourth reference points (G30)

Function and purpose

It can return to the 2nd, 3rd or 4th reference point positions by issuing the G30 P2 (P3, P4) command.



(S) Start point (CP) Intermediate point (R2) 2nd reference point (R3) 3rd reference point (R4) 4th reference point

Command format

G30 P2(P3,P4) X_{X1} Y_{Y1} Z_{Z1} $\alpha_{\alpha1}$;

$\alpha_{\alpha1}$	Additional axis

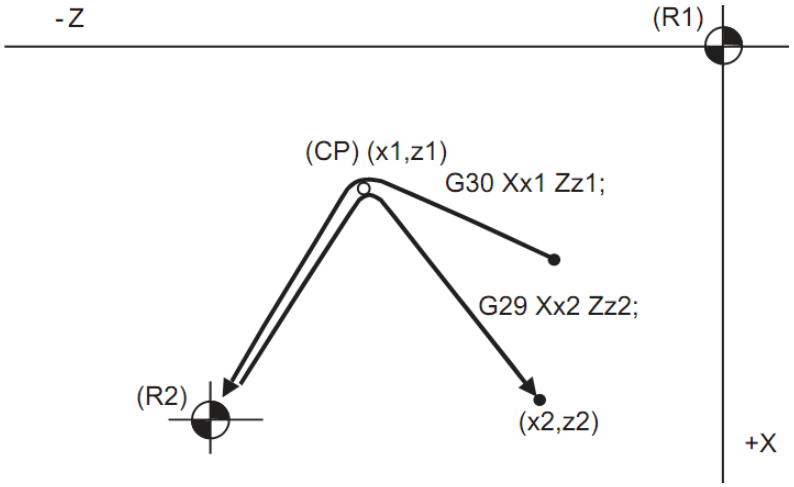
Detailed description

(1) The 2nd, 3rd or 4th reference point return are specified by P2, P3 or P4.
When there is no P command or other designation methods are used, the alarm P command is illegal.

(2) The 2nd, 3rd or 4th reference point return is the same as the 1st reference point return. It may return to the 2nd, 3rd or 4th reference point positions after passing through the intermediate point specified by G30.

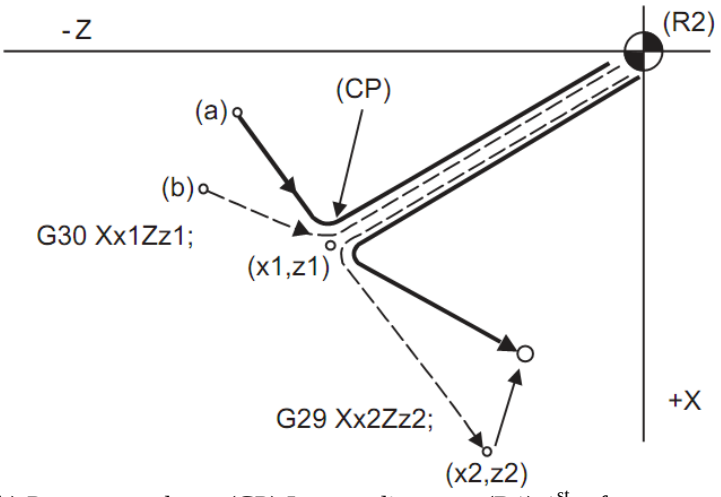
(3) The position coordinates of the 2nd, 3rd, and 4th reference points are machine-specific positions. This can be confirmed by the setting display device.

(4) After executing the 2nd, 3rd, and 4th reference point return, when G29 is specified, the intermediate point position during G29 return is the intermediate point position of the last executed reference point return.



(CP) Intermediate point (R1) 1st reference point (R2) 2nd reference point

(5) The reference point return of the compensation mid-plane from the middle point to the reference point is the path without tool nose R compensation (the compensation is 0). In the subsequent G29 command, the movement from the reference point to the intermediate point is performed without tool nose R compensation, and the movement from the intermediate point to G29 is performed with tool nose R compensation.



(a) Tool center path (b) Program path (CP) Intermediate point (R1) 1st reference point (R2) 2nd reference point

(6) After the 2nd, 3rd, and 4th reference point return, the tool offset amount of the axis is temporarily cancelled.

(7) The execution of the 2nd, 3rd, and 4th reference point return under the machine lock state is consistent with the path of turning off the machine lock and returning to the reference point. The next program will not be executed until the "G30" command completes the reference point return.

18.9 Reference point check (G27)

Function and purpose

This command is to locate the position specified by the program, and the positioning point is the 1st reference point, which is the same as G28, and outputs the reference point arrival signal to the machine. Therefore, when starting from the 1st reference point and then returning to the machining program of the 1st reference point, you can check whether the reference point has been returned after executing the program.

Command format

G27 X_ Y_ Z_ α_ ;	Check commands

X Y Z α	Return to the control axis

Detailed description

(1) The number of axes that can perform reference point checks at the same time is the number of axes controlled simultaneously.

(2) When the reference point is not reached after the command is completed, an alarm arises.

18.10 Workpiece coordinate system setting and workpiece coordinate system offset (G54 - G59 (G54.1))

Function and purpose

(1) The workpiece coordinate system takes the workpiece datum point to be processed as the origin point, and is used to simplify the coordinate system for workpiece programming.

(2) This command can be used to move to the position in the workpiece coordinate system.

Command format

G54～G59	Workpiece coordinate system selection

G54.1 Pn ;(or G54 Pn;)	Workpiece coordinate system selection (P1~P48)

G10 L20 Pn IP_ ;	Setting of workpiece coordinate system offset (P1~P48)

Detailed description

1. Workpiece coordinate system G54~G59

Command format: G54~G59

Function: The workpiece coordinate system is established relative to the machine coordinate system. The CNC system usually has 6 standard workpiece coordinate systems (G54~G59), which are convenient for users to set. Once the workpiece coordinate system is established, the subsequent programs will be executed in the selected coordinate system. Each coordinate system is determined by the distance from each axis from the reference point (fixed point on the machine) to their respective coordinate origin (offset of workpiece origin). See Figure 18-10-1 for details:

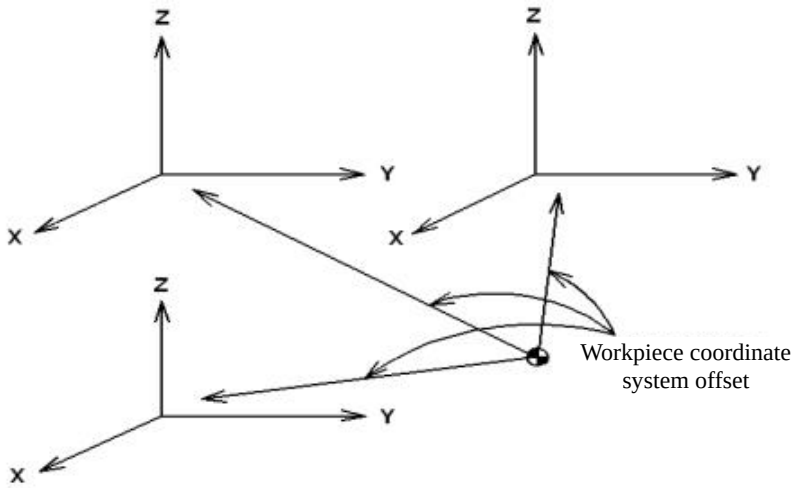


Figure 18-10-1

When using the above workpiece coordinate system, it is not necessary to use the G92 command to establish the coordinate system. If G54~G59 and G92 are used at the same time, the coordinate system established by G54~G59 will be replaced. So usually G92 is not used together with G54~G59.

Description:

1. There is no command parameter.
2. The system itself can set six workpiece coordinate systems, and any one of them can be selected by the commands G54~G59.
   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
3. When a different workpiece coordinate system is called in the program block, the axis that is commanded to move will be positioned to the coordinate point under the new workpiece coordinate system;for the axis without command to move, the coordinate will jump to the corresponding coordinate value in the new workpiece coordinate system, but the actual machine position will not change.
   Example: The machine coordinate corresponding to the origin of the coordinate system of G54 is (20, 20, 20)
   The machine coordinate corresponding to the origin of the coordinate system of G55 is (30, 30, 30)
   When the program is executed sequentially, the absolute coordinates of the end point and the machine coordinates are displayed as follows:

Program	Absolute Coordinate	Machine Coordinate
G0 G54 X50 Y50 Z50	50,50,50	70,70,70
G55 X100 Y100	100,100,30	130,130,60
X120 Z80	120,100,80	150,130,110

4. The external workpiece zero offset value or the workpiece zero offset value can be changed with G10. The specific methods are as follows:
   With command G10 L2 Pp X_Y_Z_
   P=0 : External workpiece zero offset value.
   P=1～6: Workpiece zero offset value in the workpiece coordinate systems 1 to 6.
   X_Y_Z_ : For the absolute value command (G90), it is the workpiece zero offset value for each axis.
   For the incremental value command (G91), the offset amount is added to the set workpiece zero point for each axis
   (The added result is the zero offset of new workpiece).
   G10 is used to change the external workpiece zero offset value or workpiece zero offset value. If you want to take effect immediately, please re-specify the workpiece coordinate system command.
   When the current workpiece is modified manually or through G10, both absolute coordinate and relative coordinate cannot be refreshed immediately; they can only be refreshed during execution.
   With G10 command, each workpiece coordinate system can be changed separately.

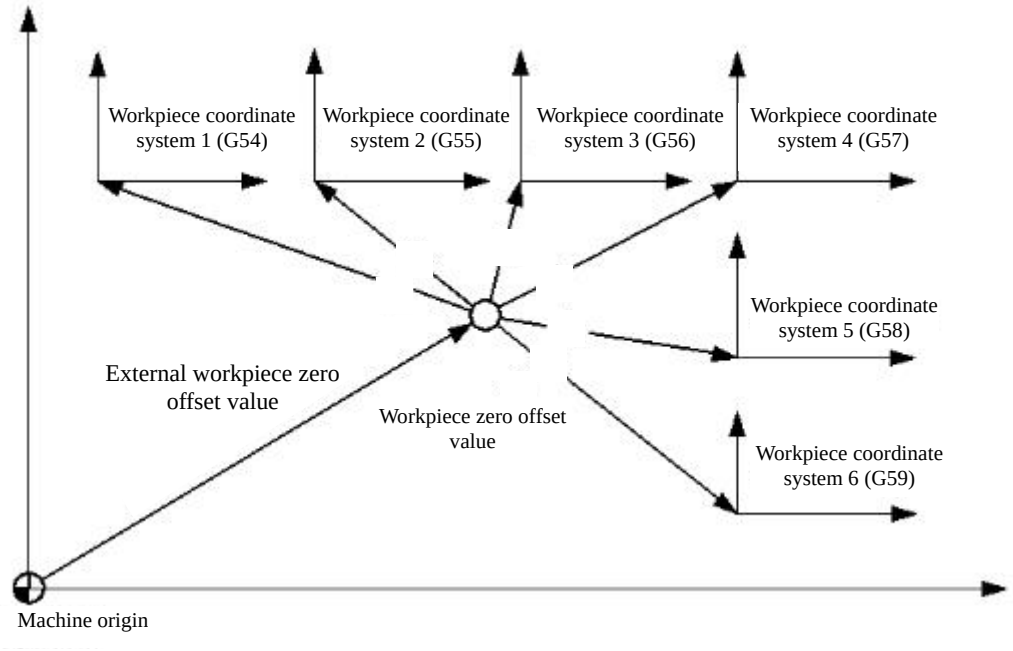


Figure 18-10-2

As illustrated in Figure 18-10-2, after the machine tool is turned on, it is manually returned to the machine zero point, and the machine coordinate system is established by the machine zero point, thereby generating the machine reference point and determining the workpiece coordinate system. The offset data parameters No.1040~1046 of the workpiece coordinate system correspond to the total offset amount of the six workpiece coordinate systems. The origin point of six workpiece coordinate systems can be specified by inputting the coordinate offset in the input mode or setting the data parameters: 1040~1046. The six workpiece coordinate systems are set according to the distance from the machine zero point to the zero point of the respective coordinate system.

Example:

```
N10 G55 G90 G00 X100 Y20;
N20 G56 X80.5 Z25.5;
```

In the above example, when the N10 program block starts to execute, it quickly locates to the position of the workpiece coordinate system G55 (X=100, Y=20). When the N20 program block starts to execute, the absolute coordinate value automatically becomes the coordinate value (X=80.5, Z=25.5) for rapid positioning to the G56 workpiece coordinate system.

2. Additional workpiece coordinate system (G54.1Pn)

In addition to the 6 workpiece coordinate systems (standard workpiece coordinate systems) G54 to G59, 48 additional workpiece coordinate systems (additional workpiece coordinate systems) can be used.

Command format: G54.1 Pn; or G54 Pn;

Pn: Code specifying additional workpiece coordinate system

n :1~48

G54.1 P1 … Additional workpiece coordinate system 1

G54.1 P2 … Additional workpiece coordinate system 2

⋮

G54.1 P48 …Additional workpiece coordinate system 48

Use G10 to change the zero offset value of the additional workpiece coordinate system:

G10 L20 Pn IP_;

Pn: Code specifying the workpiece zero offset value in the workpiece coordinate system

n :1~48

IP_: Set the axis address and offset value of workpiece zero offset

Description:

1. Select the additional workpiece coordinate system
   (1) When the P code and G54.1 (G54) are specified together, select the corresponding coordinate system from the additional workpiece coordinate systems (1~48).
   (2) Once the workpiece coordinate system is commanded, it is valid until another workpiece coordinate system is commanded.
   (3) When the power is turned on, the standard workpiece coordinate system 1 (G54) is used by default.
   (4) When G10 is used to change the zero offset value of the additional workpiece coordinate system, if you want it to take effect immediately, please re-designate the workpiece system command.
2. Set the workpiece zero offset value in the additional workpiece coordinate system
   Similar to the standard workpiece coordinate system, the workpiece zero offset can be performed in the additional workpiece coordinate system by:
   (1) Use the 

   function key to display and set the workpiece zero offset value.
   (2) The G10 function makes the workpiece zero offset value valid by program setting
   (3) The user macro program allows the workpiece zero offset value to be handled as a system variable.
   (4) The workpiece zero offset data can be input or output as external data.
   (5) When the workpiece origin offset is specified with an absolute value, the specified value is the new offset value. When specified with an incremental value, the specified value is added to the current offset value to become the new offset value.

Caution:

After G54.1, the P code must be specified. If there is no code after G54.1 in the same program block, it is considered as additional workpiece coordinate system 1 (G54.1P1). If the specified value in the P code is not within the specified range, alarm No.030: Illegal compensation number will be issued. P codes other than the workpiece offset number cannot be specified in the G54.1 program block.

Example: G54.1(G54)P1000 G04

3. The offset of the extended workpiece coordinate system will be assigned to the variable numbers shown in the table below.

Variable Number Table of the Extended Workpiece Coordinate Offset System

	1-axis to n-axis		**1-axis to n-axis**
P1	#7001～#700n	P25	#7481～#748n
P2	#7021～#702n	P26	#7501～#750n
P3	#7041～#704n	P27	#7521～#752n
P4	#7061～#706n	P28	#7541～#754n
P5	#7081～#708n	P29	#7561～#756n
P6	#7101～#7010n	P30	#7581～#758n

P7	#7121～#712n	P31	#7601～#760n
P8	#7141～#714n	P32	#7621～#762n
P9	#7161～#716n	P33	#7641～#764n
P10	#7181～#718n	P34	#7661～#766n
P11	#7201～#720n	P35	#7681～#768n
P12	#7221～#722n	P36	#7701～#770n
P13	#7241～#724n	P37	#7721～#772n
P14	#7261～#726n	P38	#7741～#774n
P15	#7281～#728n	P39	#7761～#776n
P16	#7301～#730n	P40	#7781～#778n
P17	#7321～#732n	P41	#7801～#780n
P18	#7341～#734n	P42	#7821～#782n
P19	#7361～#736n	P43	#7841～#784n
P20	#7381～#738n	P44	#7861～#786n
P21	#7401～#740n	P45	#7881～#788n
P22	#7421～#742n	P46	#7901～#790n
P23	#7441～#744n	P47	#7921～#792n
P24	#7461～#746n	P48	#7941～#794n

18.11 Local coordinate system setting (G52)

Function and purpose

By issuing the G52 command, the local coordinate system can be independently set on each workpiece coordinate system of G54~G59 to ensure that the commanded position is the program origin. The G52 command can also be used instead of the G50 command to change the offset between the origin of the machining program and the origin of the workpiece.

Command format

```
G54(G54～G59) G52 X_ Y_ Z_ α_;
```

Detailed description

(1) Before issuing a new G52 command, the G52 command remains valid, without any movement. The G52 command does not change the origin position of the workpiece coordinate system (G54~G59), so it is convenient to use another coordinate system.

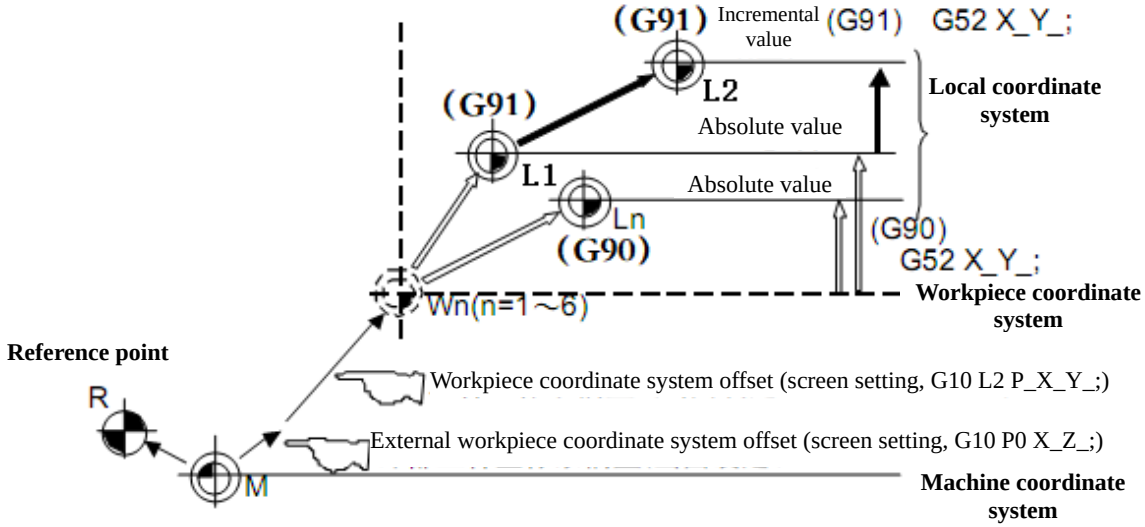
(2) After power-on, the local coordinate system offset is cleared.

(3) The local coordinate system is cancelled through G52 X0 Y0 Z0. Moreover, in case of reset or emergency stop of G54~G59, G54.1Pn, M30, M02, G92.1, the local coordinate system will also be cleared.

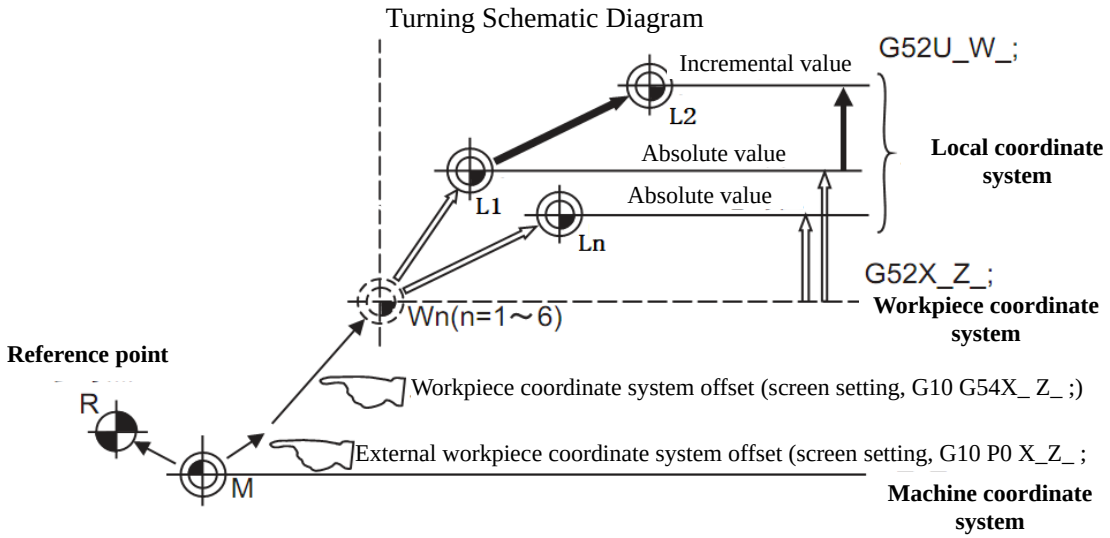
(4) The coordinate command in the absolute value mode indicates the movement to the position in the local

coordinate system.

Milling Schematic Diagram



Turning Schematic Diagram



Note: If the program is executed repeatedly, the workpiece coordinate system will be offset each time, so please specify the reference point return action at the end of the program.

18.12 Workpiece coordinate system restoration (G92.1)

Function and purpose

This function restores the offset workpiece coordinate system generated by manual and program commands to the basic workpiece coordinate system (such as EXT.(0), G54~G59 coordinate system) through program commands (G92.1). When the following operations or program commands are executed, the workpiece coordinate system will be shifted from the machine coordinate system.

—— Set the local coordinate system through G52;

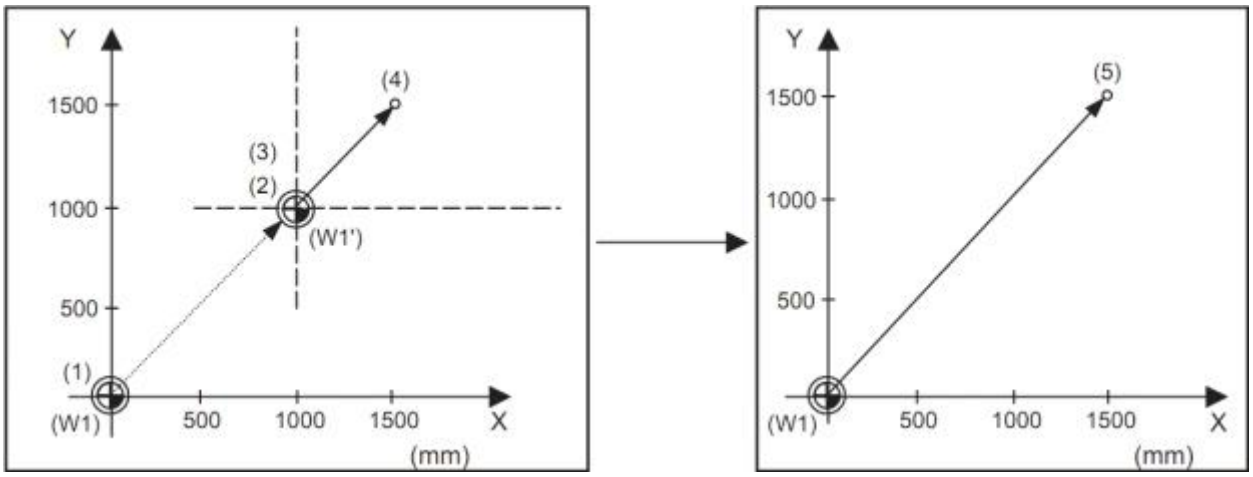
—— Move the workpiece coordinate system by G92 (G50).

Command format

G92.1 ;

Program examples

Restore the workpiece coordinate system moved by G92 (G50) through G92.1.



(W1) Origin point of the workpiece coordinate
(W1') Origin point of the workpiece coordinate after G92 (G50) command

(Example)

```
G28 X0 Y0 ;                    ... (1)
G00 G90 X1000 Y1000 ;          ... (2)
G92 X0 Y0                      .... (3)
G00 X500 Y500 ;                ... (4)
G92.1 ;                        ... (5)
```

Chapter XIX Protection Functions

19.1 Stored stroke limit (G22/G23)

Function and purpose

The stored stroke limit is a function to prevent tool collision by setting the tool forbidden area.

The stored stroke limit II function can change the forbidden area (parameter) and turn on/off this function by program designation.

Command format

G22 X_ Y_ Z_ I_ J_ K_ ; (all axes)	Parameter change or turning ON of Stored stroke limit II function
G23;	Stored stroke limit II function OFF (all axes)

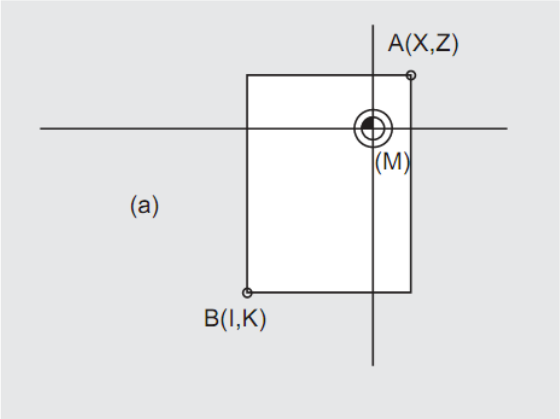
X,Y,Z	Stored stroke limit + side designation
I,J,K	Stored stroke limit - side designation

Detailed description

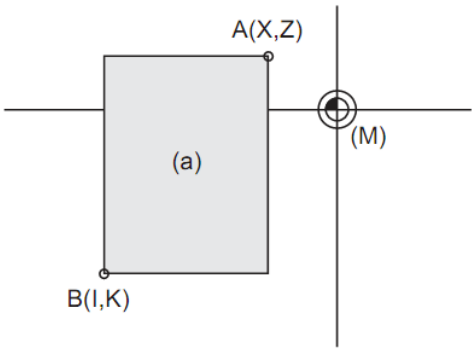
Please issue the G22 and G23 commands in separate program blocks.

Whether to set the inside as the forbidden area or the outside as the forbidden area depends on parameter (No.1070#0).

The stroke limits are stored in parameters #1082 and #1083.



[When the forbidden area is outside]



[When the forbidden area is inside]

(a) Forbidden area (M) Machine coordinate system

For more details, please refer to "2.5.3 Stored stroke inspection" in Chapter 2 of Part 2.

Chapter XX Measurement Functions

20.1 Automatic tool length measurement (G37)

Function and purpose

A command value is sent out from the measurement start position to the measurement position, allowing the tool to move in the direction of the measurement position, and stop when the tool reaches the sensor. The difference between the coordinate value now and the measurement position coordinate value in the command is automatically calculated as compensation amount for this tool.

When a tool compensation has been performed, it moves toward the measurement position in the state where the compensation has been performed. When the result of measurement and calculation indicates that further compensation is required, the current wear compensation amount is further compensated.

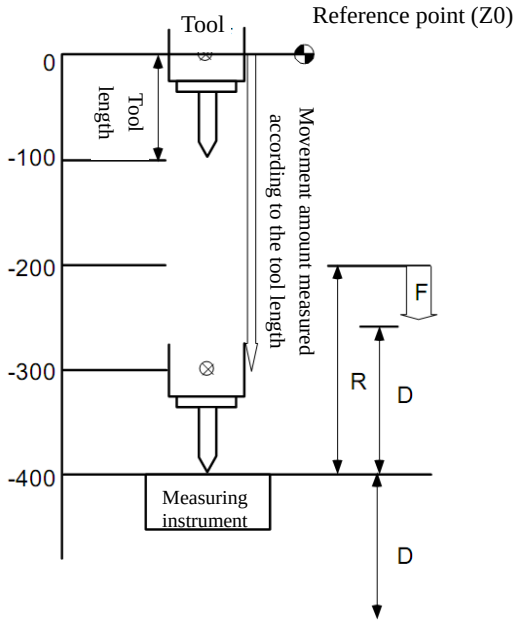
Command format

G37 α_ R_ D_ F_ ; ... Automatic tool length measurement command

α	It is the coordinate value of measurement axis address and measurement position.
R	Specify the distance from the point that starts moving at the measurement speed to the measurement position. (Radius value fixed/incremental value)
D	Specify the range where the tool should stop. (Radius value fixed/incremental value)
F	Specify the measurement speed. When R_, D_, and F_ are omitted, the parameter setting value is used. #2651 Measurement speed .0～60000 [mm/min] #2652 Deceleration area r 0 to 99999.999 [mm] #2653 Measurement area d 0 to 99999.999 [mm]

Program examples

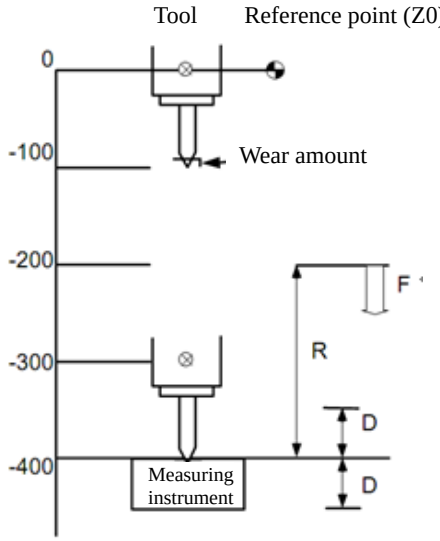
(1) When the current tool length compensation amount is 0, it is regarded as the measurement of tool compensation amount.



G28 Z0;
T01;
M06 T02;
G90 G00 G43 Z0 H01;
G37 Z-400 R200 D150 F1;

Coordinates when it arrives at the measuring position = -300
-300-(-400)=100
0+100=100
H01 compensation amount=100.

(2) When the current tool length compensation amount is not 0, it is regarded as the measurement of tool wear amount.

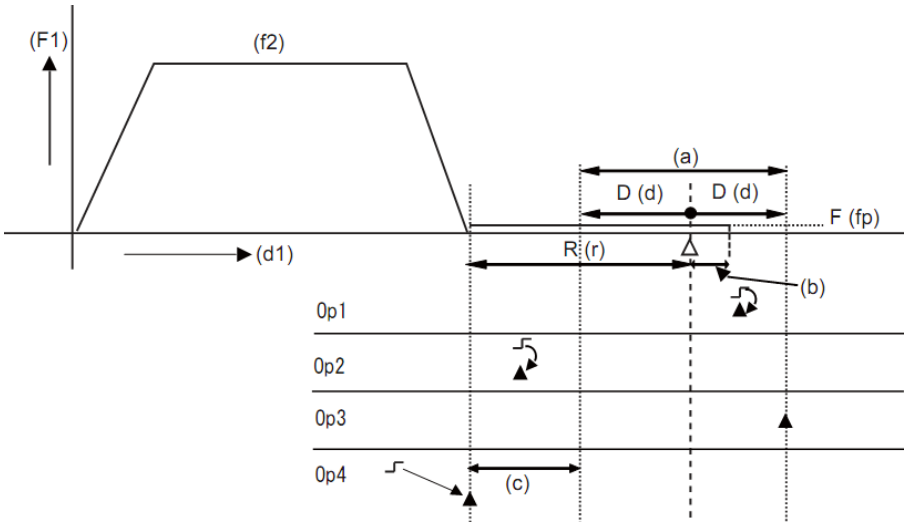


G28 Z0;
T01;
M06 T02;
G43 G00 Z0 H01;
G37 Z-400. R200. D50. F10;

H01 compensation amount=100.
Coordinates when it arrives at the measuring position = -305
-305-(-400)-100= -5
H01 wear amount = -5.

Detailed description

(1) Operation commanded by G37
(2) The sensor signal (measurement position arrival signal) parameter #2413 sets the I/O access point.
(3) In the synchronous feed mode, move at the synchronous feed [mm/rev].
(4) The updated compensation amount takes effect from the next T command of the G37 command.
(5) The delay and error in the sensor signal processing are removed on the PLC side, which is 0~0.2ms. Therefore, the following measurement errors arise.
Maximum measurement error [mm] = measurement speed [mm/min] * (1/60) * (0.2[ms]/1000)
(6) According to the detection of the sensor signal, read the current machine position coordinates. However, the machine stops after passing by an approximate servo offset amount.
Maximum overrun [mm] = calculated speed [mm/min] * (1/60) * (1/position loop gain [1/s])
The standard position loop gain is 33[1/s].



Op1: It is within the measurement allowable range, so it ends normally
Op2: The alarm is stopped because the measurement is outside the allowable range
Op3: The sensor is not detected, so the alarm stops
Op4: When the measurement is outside the allowable range, the alarm stops. But when there is no (c) scope, it ends normally

(a) Measurement allowable range (b) Compensation amount (d1) Distance
(F1) Speed (f2) Feed speed (d) Measurement area (r) Deceleration area
△Measurement position ▲Stop point ⌐¬Sensor output

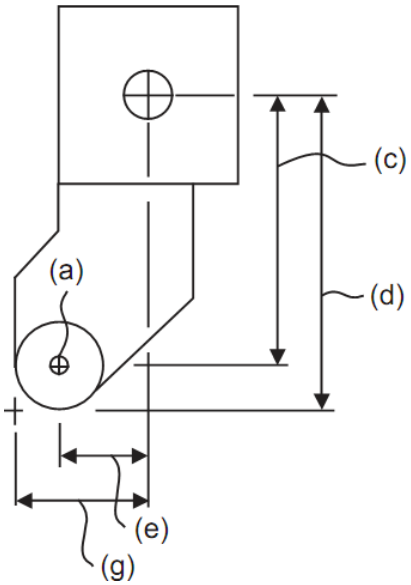
Precautions

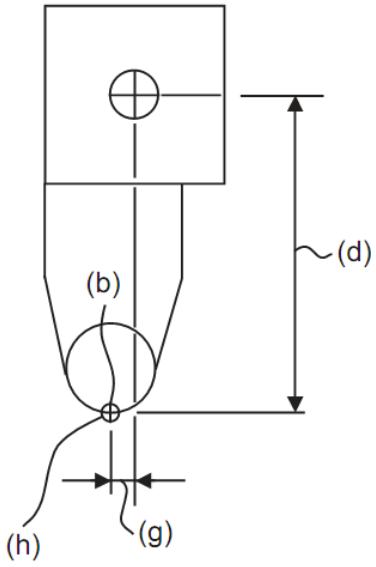
(1) A program error will arise if the G37 command is issued to a model without the automatic tool length measurement function.
(2) A program error will arise when no axis is specified in the G37 program block or when two or more axes are commanded.
(3) A program error will arise when no T code is specified in the G37 program block. But when the lower two bits of T are 0, the error (4) will arise.
(4) If the T code is not specified before the G37program block, a program error will arise. Even if T is specified, a program error will arise when the lower two bits of T are 0.
(5) If the manual insertion is performed during movement at the measurement speed, it is necessary to return to the insertion position and then restart.
(6) Make sure that the data specified by G37 or the parameter setting data satisfies the following conditions.
|Measuring point - start point| > R command or parameter r > D command or parameter d
(7) In the above (6), when the D address and parameter d are 0, only the specified measurement point and the sensor signal detection point will be ended normally. Otherwise, a program error will arise.
(8) In the above (6), if the R address, D address, parameter r, and parameter d are all 0, a program error will arise regardless of whether there is a sensor signal after positioning at the specified measurement point.
(9) When the measurement command distance is less than the measurement allowable range, it is all the measurement allowable ranges.
(10) When the measurement command distance is less than the movement distance at the measurement speed, all moves at the measurement speed.
(11) When the measurement allowable range is greater than the movement distance at the measurement speed, it moves within the measurement allowable range at the measurement speed.
(12) Before issuing the G37 command, be sure to cancel the tool radius (tool nose R) compensation.
(13) The tool length compensation is calculated without considering tool nose R value and tool nose point

number even if the tool nose R compensation option is provided.

When the tool nose point number is desired to be 0, the value of tool nose R will be subtracted from the measured tool length compensation amount.

When the tool nose point number (tool nose shape) is 5, 6, 7, or 8, the tool length will be measured at the tool nose.





(a) Tool nose point 0

(b) Tool nose point 8

(c) X-axis tool length compensation amount after subtracting the tool nose R value

(d) Measured X-axis tool length compensation amount

(e) Z-axis tool length compensation amount after subtracting the tool nose R value

(g) Measured Z-axis tool length compensation amount

(h) Tool nose

20.2 Jump function (G31)

Function and purpose

In the process of linear interpolation by G31 command, if a jump signal is input from outside, the machine feed will be stopped immediately, the remaining distance will be discarded, and the command of the next block will be executed.

This jump uses a high-speed jump signal (directly connected to NC; without being processed by PLC). When the high-speed jump signal input function is being used, the above value is less than 0.1ms, and in this way it can achieve high-precision measurement. At most 8 jump signals can be connected.

For details, refer to the relevant manual of the machine manufacturer.

Command format

G31 X(U)_ Z(W)_ F_ P_;

X,Z,U,W	Coordinate values of each axis. Specify through an absolute value or an incremental value.
F	Feed speed (mm/min)

P	Within the range of 1~8, P1 can be omitted. It corresponds to bit0~bit7 of the I/O address set by parameter #2412

Detailed description

(1) If Ff is specified, the feed speed will be the commanded speed f. If Ff is not specified, the feed speed will be the value set by parameter "#1211". In either case, the F modal will not be updated.

(2) The automatic acceleration and deceleration will not be performed in the G31 block.

(3) When parameter #1940.7=0, the G31 command is executed, and the override is invalid and fixed at 100%. Dry run is also invalid. However, the stop conditions (feed hold, override 0, interlock, stroke end) are valid. External deceleration also works.

(4) The G31 command, being non-modal, needs to be specified here.

(5) If a jump signal is input when the G31 command begins, the G31 command ends immediately.
And before the end of G31 program block, if no jump signal is input, the G31 command ends after the end of the movement command.

(6) A program error will arise if the G31 command is issued during the tool nose R compensation.

(7) When the machine is locked, the absolute coordinates will run to the position specified by the program value, and the machine coordinates will remain unchanged. When a jump signal is input during the running process, the absolute coordinates will return to the position where the machine is locked at the speed of G00, and then the next program block runs.

1. Read the jump coordinates

The coordinates of the input jump signal are stored in system variables #5061 (1st axis) to #506n (nth axis), so they can be used in user macro programs.

G00 X-100. ;

G31 X-200. F60 ; (jump command)

#101=#5061 ; Read the jump signal input coordinate value (workpiece coordinate system) to variable #101.

2. G31 inertial movement amount

In the G31 command, the inertial movement amount from the input of the jump signal to the stop varies according to parameter (#1211) or the F command in G31.

The time from the start of the jump signal response to the deceleration stop is short, so the inertial movement amount is small, and the precision is high may stop. The inertial movement amount is calculated by the following formula.

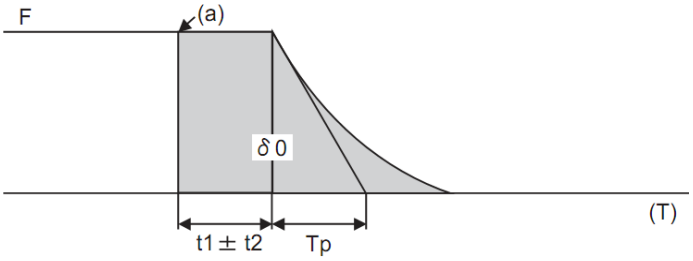
$$\delta 0 = \frac{F}{60} \times Tp + \frac{F}{60} \times (t1 \pm t2)$$
$$= \underbrace{\frac{F}{60} \times (Tp + t1)}_{\delta 1} \pm \underbrace{\frac{F}{60} \times t2}_{\delta 2}$$

δ0: inertial movement amount (mm)
F: G31 jump speed (mm/min)
Tp.: Time constant of position loop (s) = (position loop gain) -1
t1: Response delay time (s) = (the time it takes to reach the control device through the PC after detecting the jump signal)
t2: Response error time 0.001 (s)

When the G31 command is used during measurement, the δ1 measurement value in the above formula may

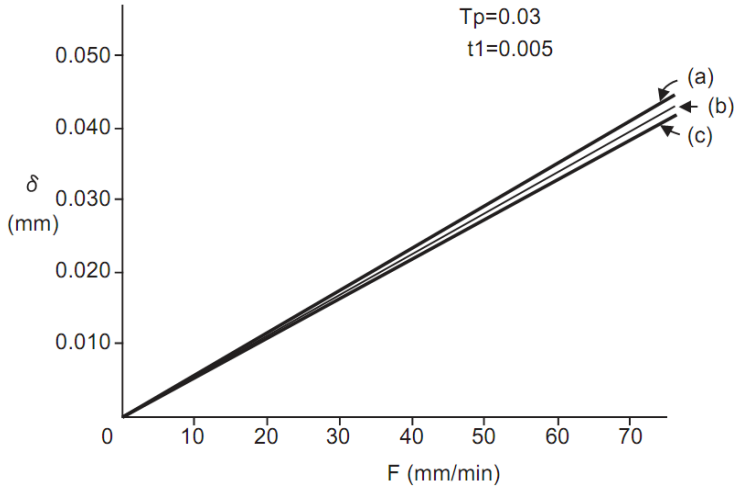
be compensated. However, δ2 is a measurement error.

The stop curve upon the jump signal input is illustrated in the figure below.



(a) Jump signal input
(T) Time

The relationship between the speed and the inertial movement amount when Tp=30ms and t1=5ms is illustrated in the figure below.



(a) Maximum value
(b) Average value
(c) Minimum value
F feed speed
δ inertial movement amount

3. Reading error of jump coordinates in mm

(1) Read the input coordinates of the jump signal

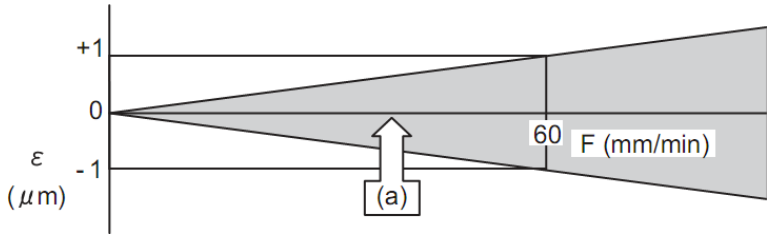
The coordinate value of the jump signal input does not include the inertial movement amount due to the position loop time constant Tp and the cutting feed time constant Ts.

Therefore, the workpiece coordinate value upon the jump signal input can be read within the error range of the following formula, and the coordinate value can be input as the jump signal. However, since the inertial movement amount due to the response delay time t1 is included in the measurement error, a compensation must be performed.

$\varepsilon = \pm(F/60)\times t\,2$

εε: Reading error
F: Feed speed
t2: Response error time 0.001 (s)

(a) measurement value



Reading error of jump signal input coordinates

The reading error at the feed speed of 60 mm/min is shown below, and the measured value is within the reading error range of ±1 μm.

ε= ±(60/60) ×0.001 = ±0.001(mm)

(2) Read coordinates other than the jump signal input coordinates

The read coordinate value includes the inertial movement amount. Therefore, if you need to input the coordinate value of the jump signal, please refer to the G31 inertial movement amount section to make compensation. However, when it is (1), the inertial movement amount due to the response error time t2 cannot be calculated, so it becomes a measurement error.

4. Compensation example of the inertial movement amount

(1) Compensation of the jump signal input coordinate value

```
⋮
G31 X100.F100 ;          Jump command
G04 ;                    Confirm the machine stop
#101=#5061 ;             Read the jump signal input coordinate value
#102=#110*#111/60 ;      Inertia movement amount due to the response delay time
#105=#101-#102 ;         Input coordinate of the jump signal
⋮
#110=jump feed speed; #111=response delay time t1;
```

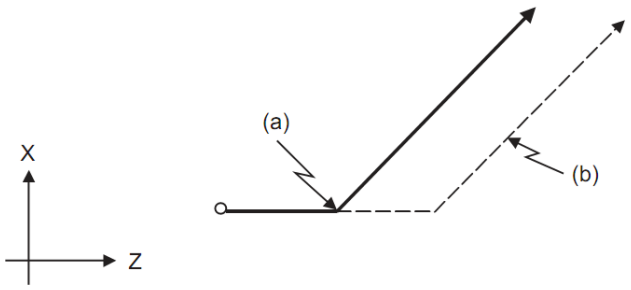
(2) Compensation of the workpiece coordinate value

```
⋮
G31 X100.F100 ;          Jump command
G04 ;                    Confirm the machine stop
#101=#5061 ;             Read the jump signal input coordinate value
#102=#110*#111/60 ;      Inertia movement amount due to the response delay time
#103=#110*#112/60 ;      Inertial movement amount due to the position loop time constant
#105=#101-#102-#103 ;    Input coordinate of the jump signal
⋮
#110 = jump feed speed ; #111 = response delay time t1 ; #112 = position loop time constant Tp ;
```

Operation examples

Example 1 The following block is an incremental value command

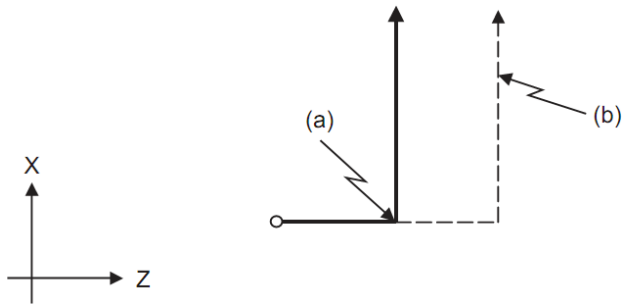
```
G31 Z1000 F100;
G01 U2000 W1000
```



(a) Operation with an external signal input
(b) Operation without any external signal input

Example 2 The following program block is an absolute value of single-axis movement command

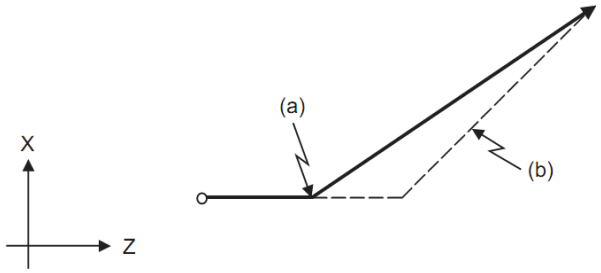
```
G31 Z1000 F100;
G01 X1000;
```



(a) Operation with an external signal input
(b) Operation without any external signal input

Example 3 The following program block is an absolute value of movement command of two axes

G31 Z1000 F100;
G01 X1000 Z2000;



(a) Operation with an external signal input
(b) Operation without any external signal input

20.3 Torque limit jump function (G31)

Function and purpose

The function that can change the torque limit value of the servo axis to an arbitrary value in the program can be used for ejecting the workpiece. The torque limit value may be specified by limiting the ratio of the torque to the rated torque.

Please execute the torque limit command in the PLC window in advance. If the torque limit override is not set in advance, a program error will arise. In addition, an alarm will be issued when an out-of-range command is executed.

For details, refer to the relevant manual of the machine manufacturer.

Command format

G31 P98 α_ F_ ; G31 P99 α_ F_ ;

P98	When the torque of servo motor reaches its limit value, a skip action is executed.
P99	The torque of servo motor reaches its limit value, or the skip operation is performed when the skip signal is input.
α	Axis address of any one axis
F	Feed speed (mm/min)

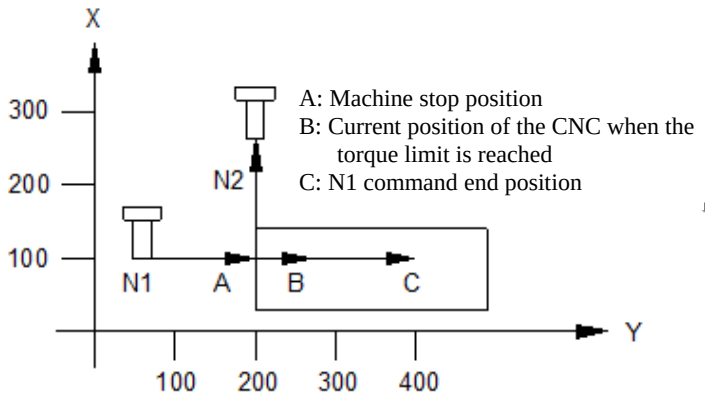
Detailed description

1. Conditions for executing jumps

Condition	Command	
	G31 P98	G31 P99
When the torque limit is reached	Execute	Execute
When a jump signal is input	Do not execute	Execute

2. Torque limit jump action

N1 G31 P99 Y400.0 F100
N2 G01 X300.0 F500



In terms of the torque limit jump, when the commanded servo axis collides with the rigid part during the running process, the servo motor torque can reach the given limit value and execute the jump action. There is no need to use the sensor unit to output the jump signal as the normal jump function does.

(1) At point A, the machine comes into contact with the measuring object and stops. Now, since the given torque limit value has not yet been reached, the jump action will not be executed, but the movement command will continue to be output, and the current position of the CNC will be updated.

(2) Although the movement command is output, the machine remains in a stop state, an offset (deviation amount) arises between the current position of the CNC and the machine position, and a torque is applied to the servomotor.

(3) When the torque limit value is reached, the jump action is executed from point A at the machine stop position, and the N2 command is executed. If the current position of the CNC when the torque limit value is reached is taken as point B, the deviation amount when the torque limit is jumped becomes (A-B).

Example

```
O1111 ;
⋮
M□□(the torque limit is commanded via the window through the PLC);
⋮
G31 P99 X200 .F100(torque limit jump command);
⋮
G01 X100    F500(the movement command in the torque limit state is applied);
⋮
```

M△△(release the torque limit through PLC);

⋮

M30;

3. User macro program variables

The coordinates of the input jump signal are stored in system variables #5061 (1st axis) to #506n (nth axis), so they can be used in user macro programs. There is a deviation from the delay amount of the servo system between the machine position when the jump operation is actually executed and the current position of the CNC.

Chapter XXI Programmable Data Input

21.1 Programmable parameters input (G10 L50/G11)

Function and purpose

The set parameters can be changed online through the machining program.

Command format

G10 L50;	Parameter input method start command
N_ R_;	Non-axis type parameter
N_ P_ R_;	Axis type parameter

N	Parameter number
R	Parameter setting value (leading zero can be omitted)
P	Axis number: 1~5 (axis type parameter) Bit number: 0~7 (bit type parameter) (if it is an axis bit type parameter: 0~39)

G11;Parameter input mode end command

Detailed description

(1) Parameter setting value (R_): When it is a bit parameter, R can only be 1 or 0.

(2) Axis number (P): Specify the axis number (P_) from 1 to 5 (maximum 5 axes) for the axis type parameter. If P is not specified, it means to modify the parameters of the 1st axis. The control axes are numbered in the order displayed by the CNC. Bit-type parameters, from 0 to 7, from low to high, and axis-type parameters, from 0 to 39, from low to high according to the control axis (0~7, 8~15, 16~23, 24~31, 32~39).

For example, specify P2 for the control axis which is the 2nd in the display order.

Precautions

(1) It is recommended that the G10 program block be commanded separately. After using G10, remember to use G11 to cancel the parameter input mode (G10 is modal only under L50) so as not to affect the normal use of the program.

(2) The parameter value modified by G10 must be within the range of system parameters, otherwise a program error will arise.

(3) G10 is for parameters that are not authorized to be modified. For parameters that cannot be modified, a program error will arise.

(4) The fixed cycle mode must be cancelled before parameter input. If not cancelled, the system will give an alarm.

Modal and non-modal

(1) G10 is modal only in the case of L50, but non-modal for L2, L10, L11, L12, L13, L20, and L51.
(2) When using G10 and L2, L10, L11, L12, L13, L20, L51, the command format should be completely specified, and G10 and L should be specified for each line.
(3) When G10 is in L50 mode, G11, L2, L10, L11, L12, L13, L20, L51 and non-modal G codes in the same group can cancel their modes.

Example:

(1) Set bit 5 (SEQ) of the bit type parameter #0001

```
G10 L50 N1 P5 R1;          SEQ is set to 1
G11;                       Cancel the parameter input mode
(2). G10 L50 N1810 R0.03;  Modify the allowable error of the arc radius
N1862 P3 R10;              Modify the Z-axis scaling factor
G11;                       Cancel the parameter input mode
```

21.2 Programmable pitch compensation data input (G10 L51)

Function and purpose

Change the set pitch compensation data online with the machining program.

Command format

G10 L51 N_ R_ P_; Pitch compensation data input

N	Pitch compensation position number
R	Pitch compensation setting value
P	Screw compensation positive and negative: 0 forward 1 reverse

Detailed description

(1) Pitch compensation data setting value (R_): The setting range is -1024～+1024. If it exceeds the setting range, a program error (PS0024) will arise. If there is a fractional part, it will be discarded. The data unit is 0.001 mm, and the system will multiply the override set by parameter #2813 as the final compensation data.
(2) Pitch compensation positive and negative (P): The default P is regarded as the input forward pitch compensation data. P can only be 1 or 0.

Part II Operation

Chapter I Operation Panel

1.1 Panel partition

The GSK25iMc/GSK25iTc series CNC system has an integrated operation panel, which can be divided into LCD (liquid crystal display) area, editing keyboard area, page display mode area and machine control area. The GSK25iMc-H10 CNC system is illustrated in Figure 1-1-1.

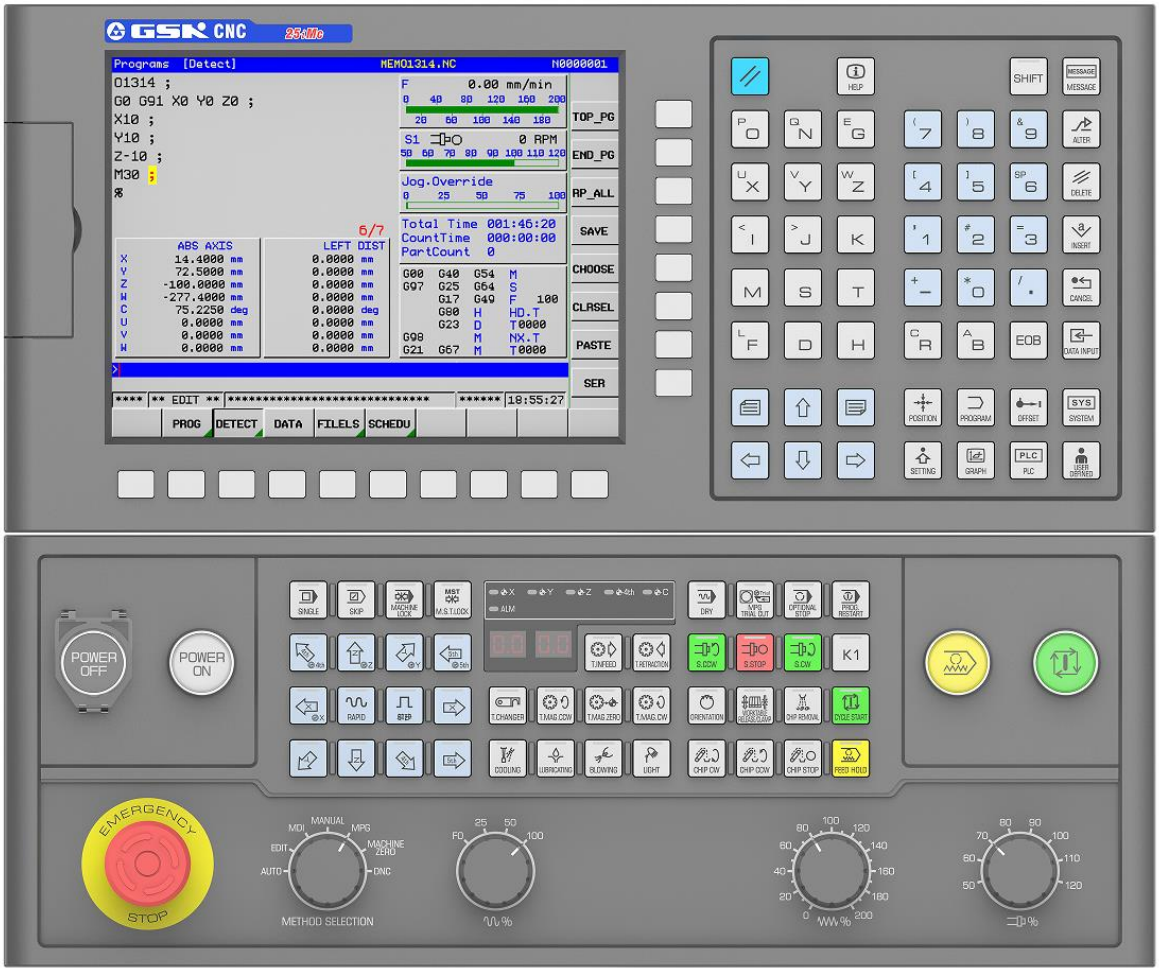


Figure 1-1-1

1.2 Panel function description

1.2.1 LCD display area

The display area of this system adopts a high-resolution color LCD (see Figure 1-2-1).

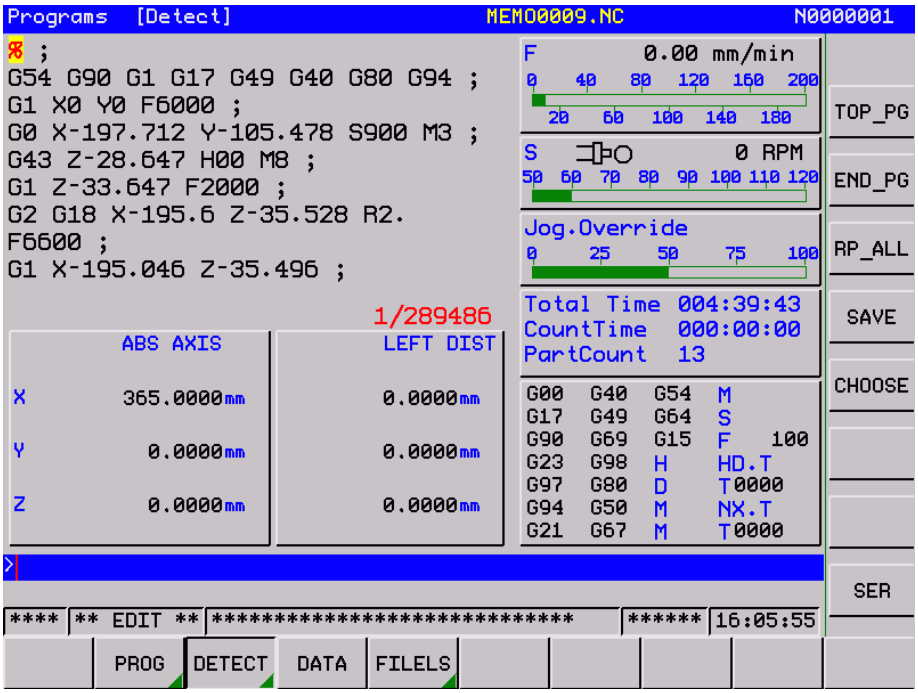


Figure 1-2-1

1.2.2 Edit keyboard area

The editing keyboard area is mainly composed of reset key, English alphabet keys, number keys, cancel key, input key, help key, screen operation keys, page turning key and cursor movement key (see Figure 1-2-2).

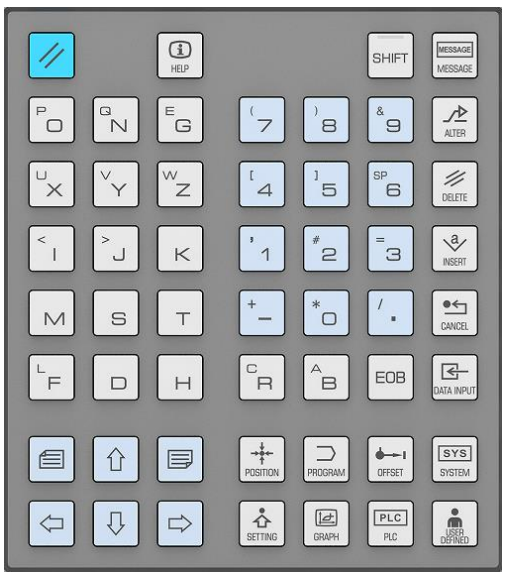


Figure 1-2-2

The instructions for use of the main keys in the editing keyboard area are briefed as follows:

S/N	Name	Function Description
1	Reset key	System reset, feed, output, stop
2	English alphabet keys	Entry of addresses
3	Number keys	Entry of numbers
4	Cancel key	Delete input characters (characters not stored in the buffer)
5	Enter key	After a number key or address key is pressed, data will be input into the buffer by pressing this key.
6	Help key	Used to view help contents and operation guides
7	Screen operation keys	Press any of the keys to enter the corresponding interface display (detailed below)
8	Page key	Used for page change and program page turning under the same display mode
9	Cursor movement key	Move the cursor up, down, left and right

1.2.3 Page display mode area

The system has the following page display keys on the operation panel:





















Name	Function Description	Remarks
Position page	Go to the position page	Display absolute, relative, comprehensive, monitoring, load meter, MPG and roundness analysis pages through soft key switching
Program Page	Go to the program page	Display program, MDI, test, data, file list, scheduling pages by soft key switching
Offset page	Go to the offset page	Display offset, workpiece system, additional workpiece system, macro variable, coordinate centering page by soft key switching
System page	Go to the system page	Display system parameters, parameter classification, diagnosis, wiring diagnosis, DB list, file management, pitch compensation, FTP page through soft key switching
Settings page	Enter the settings page	Display login, user permissions, date and time pages through soft key transitions
Graphics page	Go to the graphics page	Display graphic parameters and path pages through soft key switching

Ladder diagram page	Go to the ladder diagram page	Display ladder diagram, PLC diagnosis, signal tracking, K parameter, timer, counter, PLC list, information check, data table page by soft key switching
User-define page	Go to the user-define page	Display MENU01, MENU02, MENU03, MENU04, MENU05, MENU06, MENU07, MENU08 pages by soft key switching
Help page	Go to the help page	Display alarm, wizard, G code table, macro command B, PLC address, calculator, parameter table, servo parameter, servo motor, spindle motor page through soft key switching
Information page	Go to the information page	Display current alarm, alarm history, operation history, machining history, maintenance record, regular maintenance, system information, and operation information page through soft key switching

Note: The above soft key switching interface can also be realized by pressing the corresponding function key continuously. For the detailed description of each page, please refer to Chapter 3 of this part.

1.2.4 Machine control area

The keys in the machine control area are mainly composed of power switch, emergency stop, mode selection, feed hold, rapid override, cycle start and other functional operation keys (see Figure 1-2-3).



Figure 1-2-3

Key	Selection Mode	Function Description	Remarks and Operation Instructions
Single block	Single-block switch	Program single block/continuous running state switching When turned on, the indicator light is on	Automatic mode, input mode, DNC mode
Skip	Optical program block skip switch	No matter whether the program block with the "/" symbol at the program block header is skipped, when it is turned on, the indicator light is on	Automatic mode, input mode, DNC mode
Machine lock	Machine lock switch	When the machine lock is open, the indicator light is on, and the axis action output is invalid.	Automatic mode, input mode, reference point return, MPG mode, single-step mode, manual mode, DNC
Auxiliary lock	Auxiliary function Switch	When the auxiliary function is locked, the auxiliary function lock indicator light is on, and	Automatic mode, input mode, DNC mode

Key	Selection Mode	Function Description	Remarks and Operation Instructions
		the M, S, T function output is invalid	
+X/-X/+Y/-Y/+Z/-Z/+4/-4/+5/-5	Manual feed key	Manual, single-step mode X, Y, Z, 4, 5 axes moves towards the positive/negative direction, the positive direction is the MPG axis selection	Reference point return, single-step mode, manual mode, MPG mode
Fast moving	Rapid feed key	Rapid feed on/off	Any mode
Single-step	Manual single-step mode selection key	Go to the manual single-step operation mode	This method is only useful in manual mode, but other modes are not affected
Tool magazine forward (reverse) Tool magazine reverse (backward)	Tool magazine action key	Tool magazine movement forward/backward	Manual mode, MPG mode
Tool changer	Manual tool change	Change the tool manually	Manual mode, MPG mode
Tool selection counterclockwise	Tool change action key	Tool selection in counterclockwise, return-to-zero or clockwise mode	Manual mode, MPG mode
Tool magazine return to zero			Return to reference point
Tool selection clockwise			Manual mode, MPG mode
Cool down	Coolant switch key	Coolant switch	Any mode
Lubricate	Lubrication switch key	Machine's lubrication on/off	Any mode
Air blowing	Air blow key	Air blow switch	Any mode
Work light	Work light switch of machine	Work light on/off of machine	Any mode
Dry running	Dry run switch	When dry run is valid, the indicator light is on	Automatic mode, input mode, DNC mode
MPG trial cutting	MPG trial cut key	Go to the MPG trial cut function	Automatic mode, input mode, DNC mode
Optional stop	Stop on/off key	Does M01 in the program stop	Automatic mode, input mode, DNC
Program restart	Program restart key	Program restart	Auto mode
Turn counter-clockwise Spindle stop Turn clockwise Spindle orientation	Spindle control key	Spindle turning counter-clockwise Stop rotation of spindle Spindle turning clockwise Spindle orientation	MPG mode, single-step mode, manual mode
K1	Custom key	User-define function keys	Any mode
Worktable loose/tight	Worktable action key	Worktable loose/tight	Any mode

Key	Selection Mode	Function Description	Remarks and Operation Instructions
Chip flushing	Chip flushing action key	Chip flushing on/off	Any mode
Cycle start	Cycle Start key	Press this key, the program will run automatically	Automatic mode, input mode, DNC
Forward rotation of chip conveyor Chip conveyor stop Reverse rotation of chip conveyor	Chip removal action	Chip removal on/off	Any mode
Feed hold	Feed hold key	Press this key to automatically stop the system	Automatic mode, input mode, DNC
Operation mode selection	Auto mode	Go to the automatic operation mode	When the automatic mode is selected, the system selects the internal memory program
	Editing mode	Go to the editing operation mode	Switch to the editing mode during autorun, and then to the past after the system finishes running the current block and stops.
	Enter mode	Go to the entry mode	Switch to the input mode during autorun, and the system will switch to the past after running the current block and stopping.
	Manual mode	Go to the manual operation mode	Switch to the manual mode during autorun, the system will decelerate and stop immediately
	MPG mode	Go to the MPG operation mode	Switch to MPG mode during autorun, and the system will decelerate and stop immediately
	Reference point return mode	Go to the reference point return operation mode	Switch to reference point return mode during autorun, and the system will decelerate and stop immediately
	DNC mode	Go to the DNC operation mode	Switch to the DNC mode during autorun, and the system will switch to the past after it finishes running the current block and stops
Speed override	F0, 25%, 50%, 100%	Quick override, manual single-step, MPG override selection keys	Automatic mode, input mode, back-to-zero, MPG mode, single-step mode, manual mode, DNC
Feed rate	F0-F200%	Feed override selection key	Automatic mode, input mode, DNC
Spindle override	F50%-F120%	Spindle override selection key	Automatic mode, input mode, DNC

Note: The specific key functions shall follow the design of the machine factory.

Chapter II System Power-on, Shutdown, and Safe Operation

2.1 System power-on

Before powering on the GSK 25iMc system, make sure that:

1. The state of the machine is normal.
2. The power supply voltage meets the requirements.
3. The wiring connections are correct and firm.
4. After the system is powered on, a series of initialization operations will be performed, make sure not to disconnect power supply.
5. After the initialization is completed, it goes to the MDI mode and displays the current position (absolute coordinates) page.
6. When the connection of each slave station is normal for the first time, there will be a process of creating database. Please wait for about 1 minute before performing other operations.

2.2 Switching off

Before GSK988TA is powered off, you should confirm:

1. The X, Y, Z, 4, and 5 axes of the CNC are stopped.
2. Auxiliary functions (such as spindle, water pump, etc.) are closed.
3. Cut off the power supply of the CNC first, then the power supply of the machine.

When turning off the power, the following checks should be made:

1. Check whether the LED on the operation panel indicates that the cycle start should be in the stop state.
2. Check whether all moving parts of the CNC machine are stopped.
3. Press the POWER OFF button to shut down.

Power off in case of emergency:

During the operation, the power supply of the machine can be cut off immediately in case of emergency to prevent accidents. However, it must be noted that there may be deviations between the system coordinates and the actual position after the power is turned off, and operations such as returning to the reference point and tool setting must be performed.

Note: For the cutting off operation of the power supply of the machine, please refer to the machine instruction manual of the machine manufacturer.

2.3 Safe Operation

2.3.1 Reset operation

After pressing the  key, the system is in a reset state:

1. All axis movement stops;
2. M, S, T functions stop
3. In the milling machine mode, modify parameter #2601.6 to set the tool length offset, and when resetting, 0: not clear, 1: clear.

Note: In the reset state, the system cannot perform program editing operations.

2.3.2 Emergency stop



In case of emergency, press the emergency stop button, the movement of all axes of the machine will immediately stop, and the spindle rotation and coolant, etc. are also closed. Moreover, the button is always blocked at the stop position.

The button is released by different methods with respect to different machine manufacturers, and usually by pressing the button and turning it clockwise.

Note 1: Press this button, the motor enable is cut off, and whether the motor power is cut off depends on the circuit design of the machine manufacturer.

Note 2: The control unit is in the reset state.

Note 3: Troubleshoot before releasing the button.

Note 4: After releasing the button, the machine using the incremental encoder should return to the reference point by manual operation.

Note 5: In the emergency stop state, the system cannot conduct the program loading operation.

Generally, the emergency stop signal is a normally closed contact signal. When the contact is disconnected, the system enters the emergency stop state and urgently stops the machine .

2.3.3 Feed hold

When the machine is running, you can press the 

key (or the button ) to pause the operation, and the feed hold light will be on at the same time. It should be noted that when executing rigid tapping, cycle command, and single-block command, it will dwell after running the current command.

2.4 Cycle start and feed hold

The [Cycle Start] key and the [Feed Hold] key in the system panel are used to start and dwell the program in the automatic mode, input mode and DNC mode.

2.5 Over-travel protection

In order to avoid damage to the machine due to over-travel of the feed axis, the machine must take over-travel protection measures.

2.5.1 Hardware over-travel protection

Install the travel limit switches at the positive and negative maximum strokes of each feed axis of the machine. When an overtravel occurs, the feed axis will decelerate after hitting the limit switch and finally stop, and the system will prompt a hard overtravel alarm message.

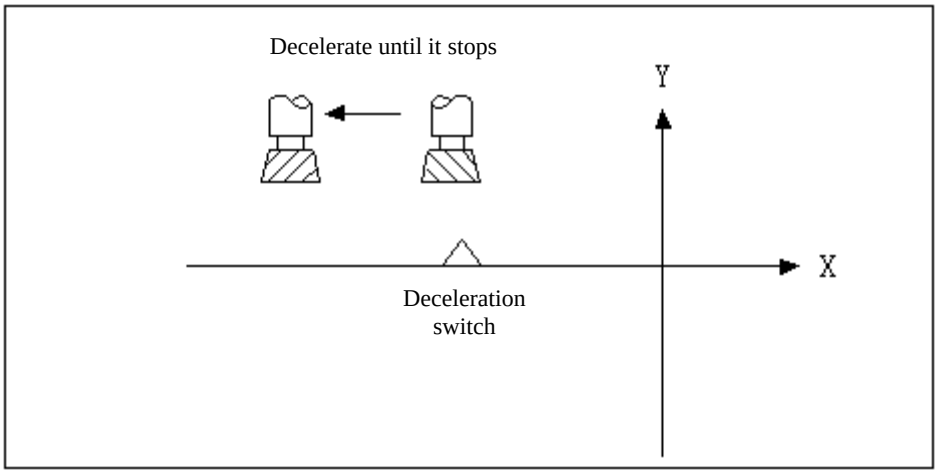


Figure 2-5-1

Detailed description

(1) Overtravel during autorun

During autorun, when a certain feed axis moves and touches the limit switch, the movement of all feed axes will decelerate and finally stop, and an overtravel alarm will be displayed at the same time, and the program will stop at the overtravel program block.

(2) Overtravel during manual operation

During manual operation, only the axis that touches the limit switch decelerates to stop and alarm, but does not affect the operation of other axes.

(3) Release of hardware overtravel

The method of releasing overtravel alarm is: in manual mode, press the "overtravel release" key and move the axis in the opposite direction (if it is positive overtravel, move out in the negative direction; if it is negative overtravel, move out in the positive direction).

After the limit alarm occurs, even if the limit switch returns to the inactive state, the axis cannot move in this direction until the limit alarm is cleared.

Due to the inconsistent handling of overtravel by different machine manufacturers, please refer to the instructions provided by the machine manufacturer for the specific method of overtravel alarm cancellation.

2.5.2 Software over-travel protection

The software travel range I s set by parameters #1080~#1087, with the machine coordinate value as the reference value. If the movement of the tool coordinate position reaches or exceeds the range of the software limit stroke, an overtravel alarm will arise.

When the software limit is over-travel, it will give an alarm after the over-travel. Move the axis in the opposite direction in [Manual] or [MPG] mode, and automatically cancel the overtravel alarm after moving out of the overtravel range.

2.5.3 Stored travel inspection

The tool cannot enter the area specified by the stored stroke limit checks 1, 2, and 3.

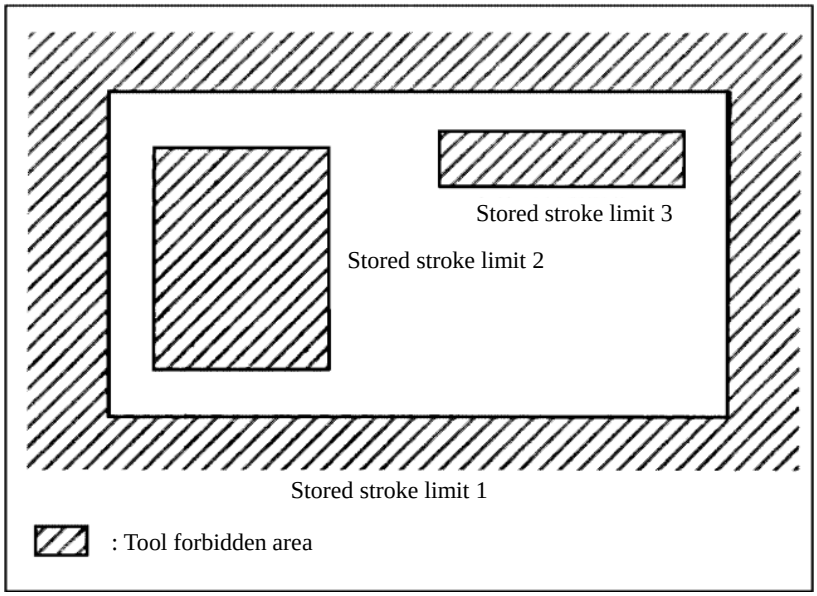


Figure 2-5-2

When the tool exceeds the stored stroke limit, an alarm is displayed and the feed axis decelerates and stops.

When the tool invades the forbidden area and triggers an alarm, it can move in the direction opposite to the invasion.

Detailed description

1. Stored stroke check 1

   Parameters (#1080, #1081) may set the boundary. Outside this area is the forbidden area. Machine manufacturers often set this area as the maximum travel of the machine.

2. Stored stroke check 2

   Parameters (#1082, #1083) or commands (G22, G23) can set this boundary, and the area inside or outside the boundary can be set as a forbidden area. Parameter #1070.0 may set the inside or outside of the area as the forbidden area.

   Parameter OT2 (#1070.2) may set whether to check the stored stroke detection 2 (0: not check/1: check). When this parameter is set to 0 (no check), the G22 command is invalid; when this parameter is set to 1 (check), G22 is valid.

   When using the program command, G22 prohibits the tool from entering the forbidden area. G23 allows the tool to enter the forbidden area. Each G22 and G23 in the program must be commanded with an independent program block.

   There are two ways to create or change the forbidden area:

   ① Establish or change the forbidden area through the program, as shown in Figure 2-5-3.

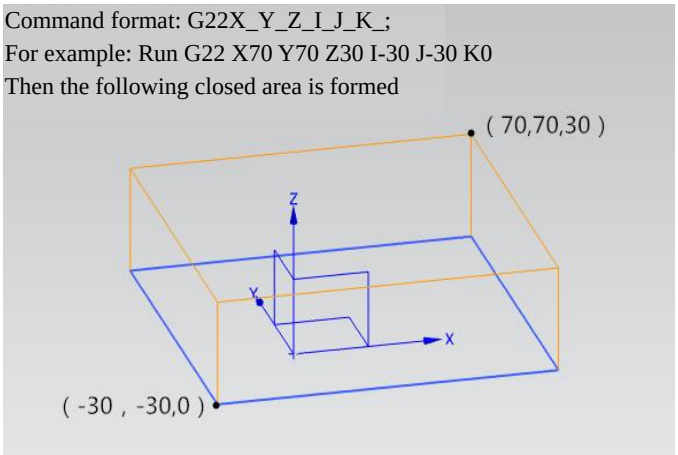


Figure 2-5-3

② Establish or change the forbidden area through parameter setting, as shown in Figure 2-5-4.

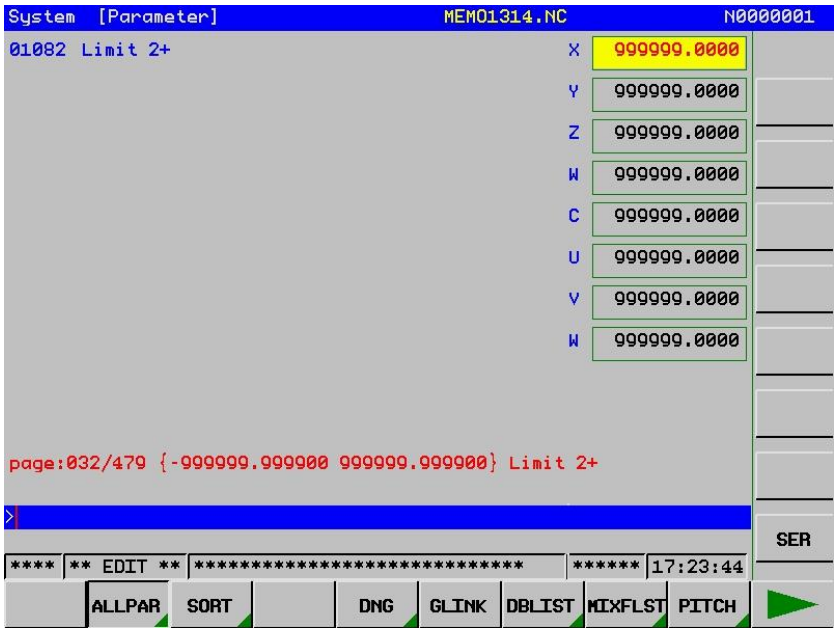


Figure 2-5-4

When setting the forbidden area through parameters #1082 and #1083, the X boundary value in the positive direction set by #1082 must be greater than the X boundary value in the negative direction set by #1083, and the difference between the two must be greater than the minimum movement amount of the machine. The boundary values of the Y and Z axes are set in the same way.

When set by G22 X_Y_Z_I_J_K_, X must be greater than I, and the difference between X and I must be greater than the minimum movement amount of the machine. The same is true for the settings of Y and J, Z and K. And after executing the G22 X_Y_Z_I_J_K_ command, the system will automatically input and save the values of X, Y, and Z in parameter #1082, and input and save the values of I, J, and K in parameter #1083.

3. Stored stroke check 3

The boundary is set with parameters #1084 and #1085. The inner boundary of this area is a forbidden area.

4. Checkpoints in the forbidden area

The set parameter value or programmed value depends on which part of the tool is checked to intrude into the forbidden area: the tool nose or the tool shank. Before programming, determine the inspection location (tool nose or tool chuck).

If you want to check point C (tool nose) in Figure 2-5-5, the distance "C" will be set as data of the stored stroke limit function.

If you want to check point D (tool chuck), the distance "d" must be set.

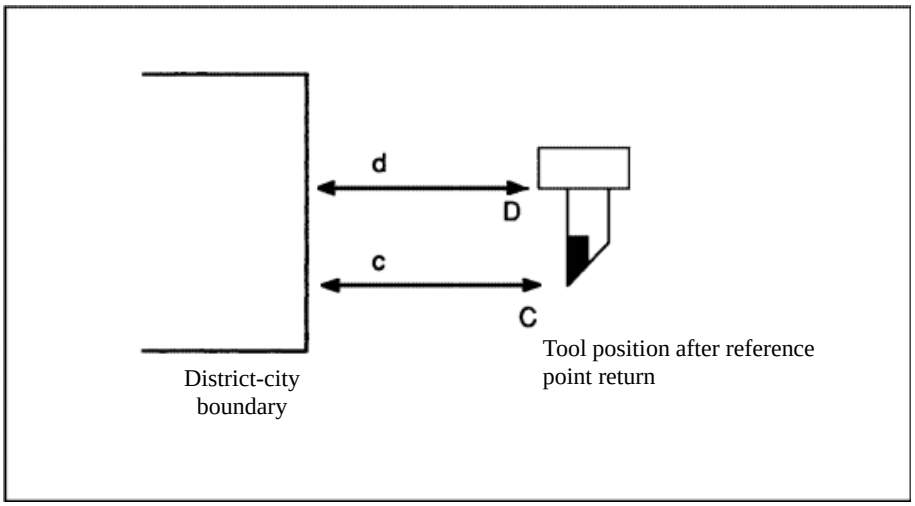


Figure 2-5-5 Setting the forbidden area

5. Overlapping of tool forbidden areas

The forbidden areas can overlap.

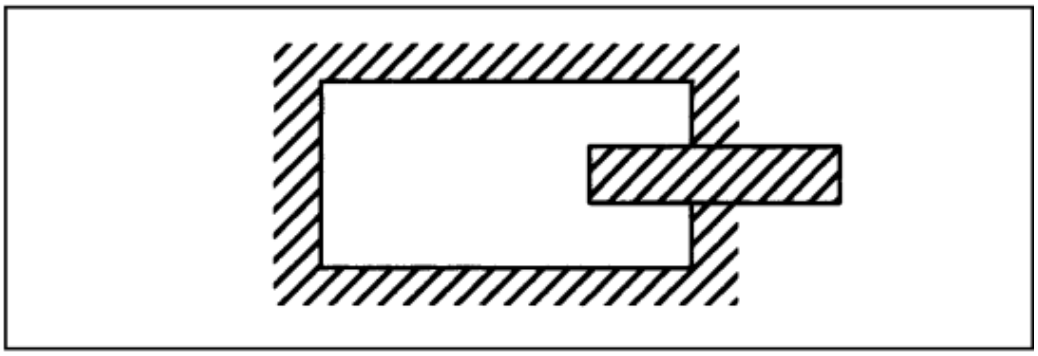


Figure 2-5-6 Setting overlapping forbidden areas

It is not necessary to set limits outside the machine stroke.

6. Effective time of the forbidden area

After the machine coordinate system is established, the forbidden area takes effect (the G22 mode is required to stored stroke limit 2).

Before the machine coordinate system is established, when parameter #1070.6=0, the forbidden area is invalid; when parameter #1070.6=1, the forbidden area is valid (the G22 mode is required for the stored stroke limit 2).

7. Clear the alarm

If the tool enters the forbidden area of the outside stored stroke and triggers an alarm, it can only move towards the opposite direction. To clear the alarm, move the tool in the opposite direction until it exits the forbidden area and reset the system. After the alarm is cleared, the tool can be moved forward or backward.

If the tool enters the forbidden area of the inner stored stroke and triggers an alarm, the tool needs to move out of the forbidden area according to the actual operation situation, and resets the system to clear the alarm.

8. Change from G23 to G22 in the forbidden area

When switching from G23 to G22 in the forbidden area, an alarm will be promptly given.

9. Allowable overtravel of the stored stroke limit

If the maximum rapid movement speed is F (mm/min), the maximum overtravel amount L (mm) of the stored stroke limit can be calculated by the following formula: L (mm)=F/7500

The tool shall not enter the specified forbidden area more than L (mm).

2.5.4 Chuck tailstock barrier

Chuck tailstock barrier function is the function used to detect the interference between chuck and tailstock and tool nose to prevent damage to the machine. Depending on the shape of the chuck and tailstock, a tool entering prohibited area can be pre-set by using a dedicated setting screen. This function stops the tool from shifting and displays an alarm message if the tool nose enters the prohibited area during processing. The tool can be removed from the prohibited area in the opposite direction of the feed.

Detailed description

1. Steps for entering the interface

First, press 

to enter the login interface (see Figure 2-5-7), and enter the password of the corresponding level.

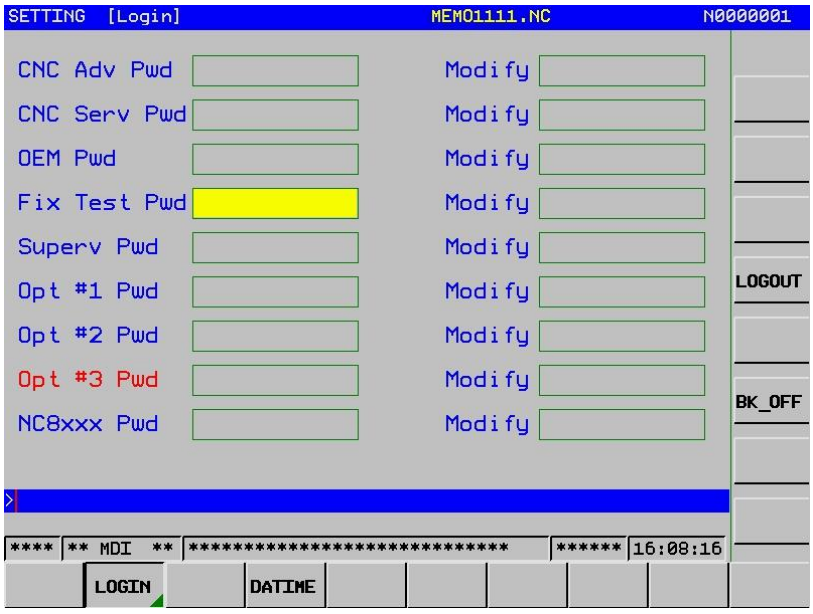


Figure 2-5-7

After logging in, modify parameter #3000.0 GMT=1, restart to take effect and enter the lathe mode (see Figure 2-5-8).

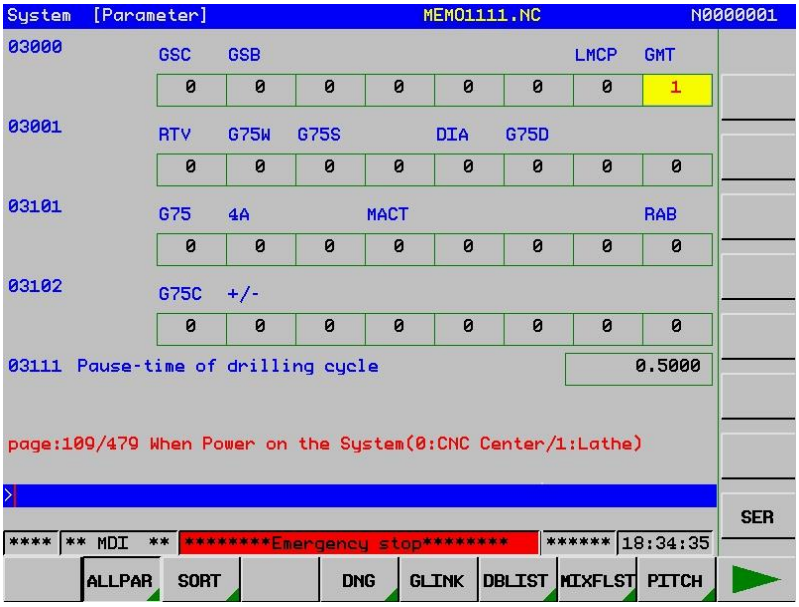


Figure 2-5-8

After restarting, press the SETTING key ➔ [Login], and after logging in, the soft keys [Chuck] and [Tailstock] will be displayed (see Figure 2-5-9).

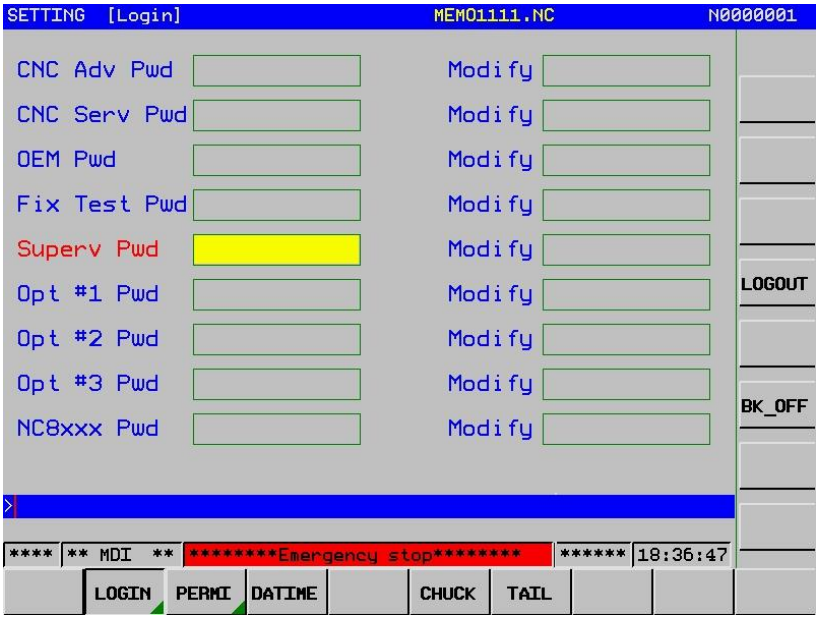


Figure 2-5-9 Login interface diagram

2. Setting of prohibited area of the chuck

Press the SETTING key ➔ **CHUCK** soft key, to display the chuck interface (see Figure 2-5-10) and the detailed introduction of the chuck (see Figure 2-5-11).

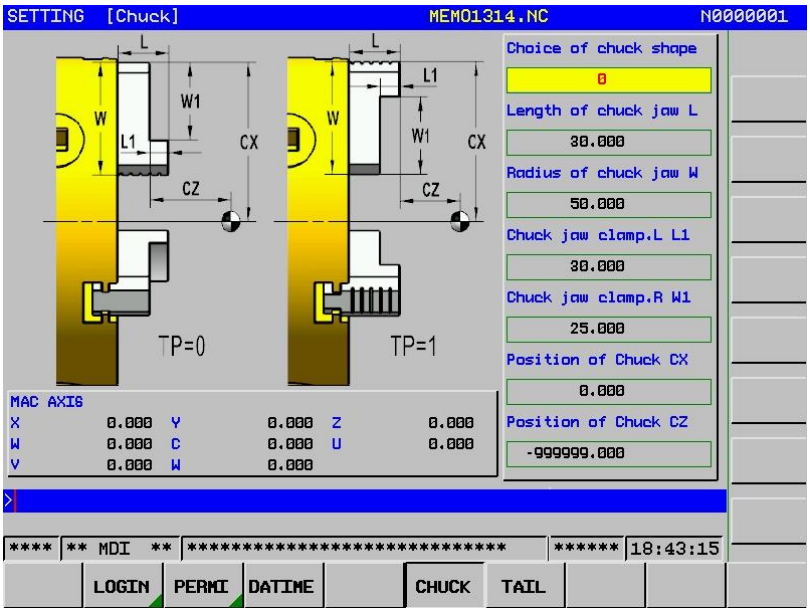


Figure 2-5-10 Chuck interface diagram

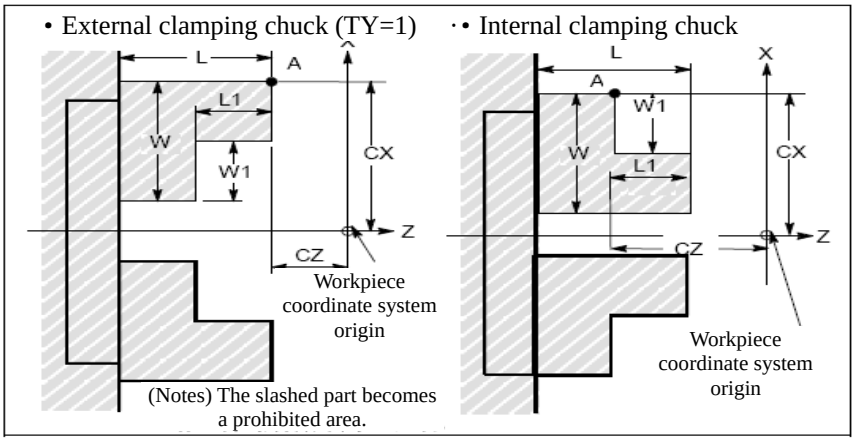


Figure 2-5-11 Detailed introduction to the chuck

Mark	Notes	Parameters
TY	Chuck shape selection (TP=0: internal clamping/TP=1: external clamping)	#1101
L	Length L of chuck jaw	#1102
W	Radius W of chuck jaw	#1103
L1	Clamping length L1 of chuck jaw	#1104
W1	Clamping mouth radius W1 of chuck jaw	#1105
CX	Chuck position CX	#1106
CZ	Chuck position CZ	#1107

TY: Select chuck shape. The internal clamping chuck is selected if 0 is specified, while the external clamping is selected if 1 is specified. It is assumed that the chuck is symmetrical around Z-axis.

CX, CZ: Set the chuck position through the coordinate value in the workpiece coordinate system. It is not the coordinate value in the machine coordinate system.

3. Setting of the prohibited area of the tailstock

Press the Tailstock soft key of the 

key ➔ to display the **TAIL** interface (see Figure 2-5-12), and the detailed introduction of the tailstock (see Figure 2-5-13).

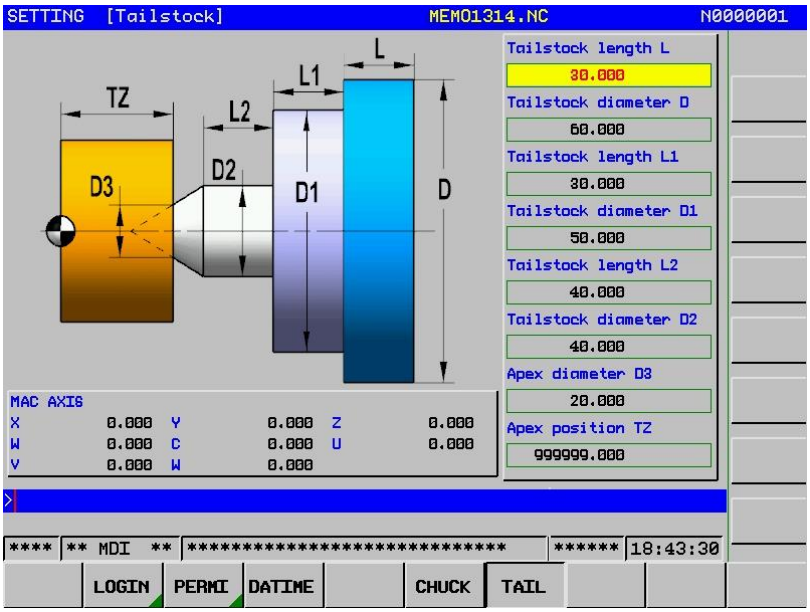


Figure 2-5-12 Tailstock interface diagram

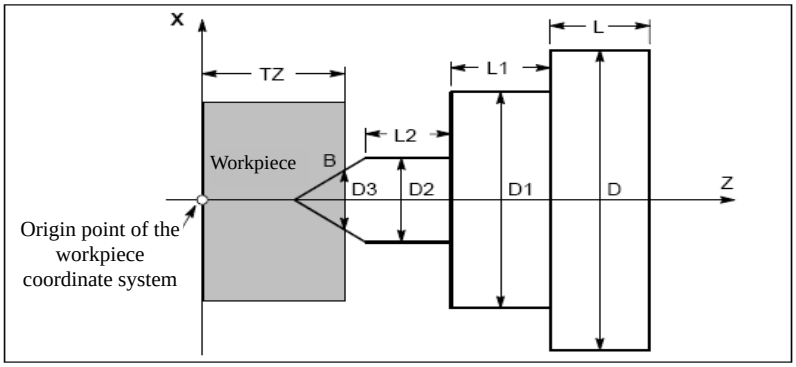


Figure 2-5-13 Detailed introduction of tailstock

Mark	Notes	Parameters
L	Length L of tailstock	#1111
D	Diameter D of tailstock	#1112
L1	Length L1 of tailstock	#1113
D1	Diameter D1 of tailstock	#1114
L2	Length L2 of tailstock	#1115
D2	Diameter D2 of tailstock	#1116
D3	Diameter D3 of tailstock top	#1117
TZ	Position TZ of tailstock top	#1118

TZ: set the position (point B in the figure above) of tailstock through the coordinate value in the workpiece coordinate system. It is not the coordinate value in the machine coordinate system. It is assumed that the

tailstock is symmetrical around the Z-axis.

Precautions

(1) Correct setting for entering the prohibited area

The prohibited area may be invalid if the setting for entering the prohibited area is incorrect. Incorrect settings refer to the following situations.

- In setting of the Chuck Shape, L<L1 or W<W1.
- In setting of the Tailstock Shape, D2<D3.
- The positions of the chuck and the tailstock are overlapped.

(2) Tool retracting while entering the prohibited area

If the tool enters the prohibited area and an alarm is sent out from the system, retract the tool manually after the switch is switched to manual mode. Afterwards, reset the system to cancel such alarm.

At this time, in manual mode, tool can be moved in the opposite direction to that when entering prohibited area. Thus, the tool can no longer move along the direction (the direction further entering the prohibited area) identical to that entering the prohibited area.

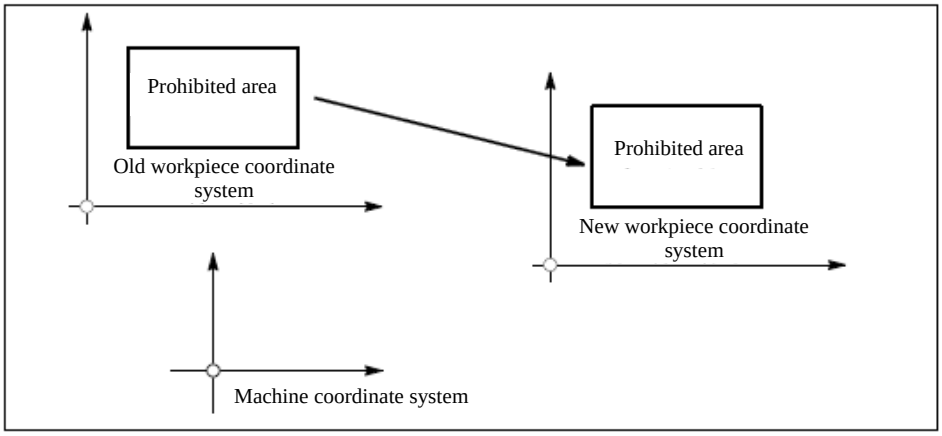
If the prohibited area of the chuck tailstock is valid and the tool is positioned in such prohibited area, an alarm will be sent out once the tool is moved.

If such alarm is sent out when the tool is moved and it is impossible to retract the tool from the prohibited area, change the setting of the prohibited area where the tool is located at, to make such tool beyond such area. Afterwards, reset the system to cancel such alarm, and retract the tool. Finally, recover the original setting.

(3) Coordinate system

Since the prohibited area is defined through the workpiece coordinate system, the followings shall be noticed:

While executing the commands or operations which may move the workpiece coordinate system, the prohibited area will also move for the same value.



The workpiece coordinate system may be displaced through the following commands and operations.

Commands: G54~G59, G52, G50 (G92 in the G code systems B or C)

Operation: interruption of the manual-control hand wheel; change of the offset of the reference point of the workpiece; change of the offset (tool shape compensation) of the tool position; operation by virtue of the locked machine; manual operation when the absolute signal of the machine is OFF.

When the tool enters the prohibited area while in autorun and the tool is to be exited from the prohibited area by manual operation, please set manual absolute signal *ABSM<Gn006.2> to "0" (manual absolute

signal is ON). If it is set to "1", even if the tool is moved in the manual operation mode, its movement amount will not be reflected in the workpiece coordinate value, and the tool cannot be withdrawn from the prohibited area.

Chapter III Interface Display and Operation

3.1 Position interfaces

Press the POSITION key to enter the position page display. The position display page has six interfaces: [Absolute], [Relative], [Comprehensive], [Monitor], [Load Meter], [MPG], which can be viewed through the corresponding soft keys, or by pressing the POSITION key to switch view. If there is a triangle mark in the lower right corner of the soft key, such as REL , it means the soft key with an extended function. On the right side of the [Relative] interface, the sub-soft keys [Preset], [Return to Zero], [Centering], and [Piece Number Reset] can be displayed for related operations on this interface. The following is an introduction of the interfaces:

3.1.1 Absolute mode

Press ABS soft key to display the current tool position in the current coordinate system (hereinafter referred to as "absolute coordinate", see Figure 3-1-1).

The left side of the figure shows the absolute coordinate values in the coordinate system. The first progress bar F on the right side represents the feed speed, which can be adjusted by the feed override knob; and S1 and the fast override progress bar below can be adjusted by the spindle override knob and fast override key.

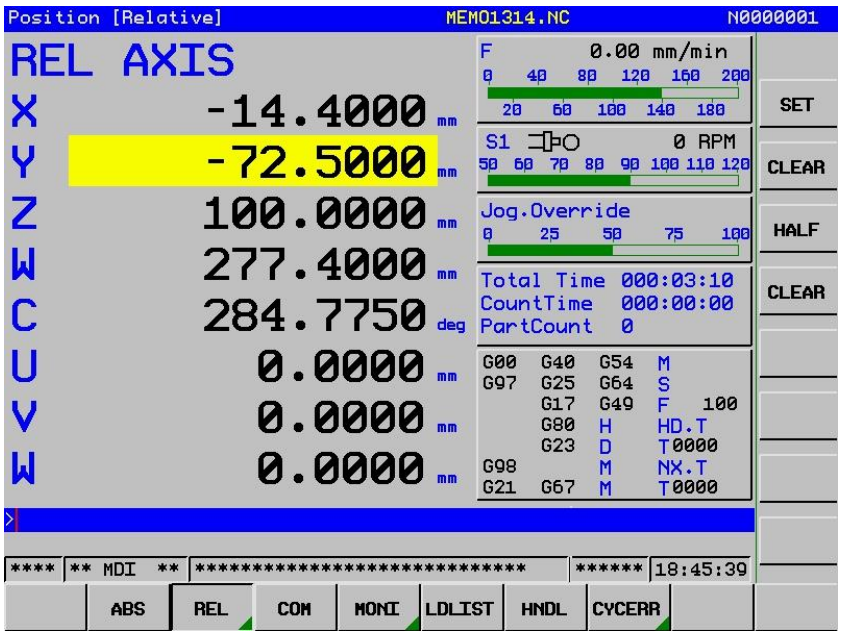


Figure 3-1-1 Absolute coordinate

3.1.2 Relative mode

Press the REL soft key to display the position of the current tool in the relative coordinate system (hereinafter referred to as "relative coordinate"). On the right side of the relative coordinate display interface,

there are four sub-soft keys [Preset], [Return to Zero], [Centering], and [Piece Number Reset] (see Figure 3-1-2) for related operations on this interface. .

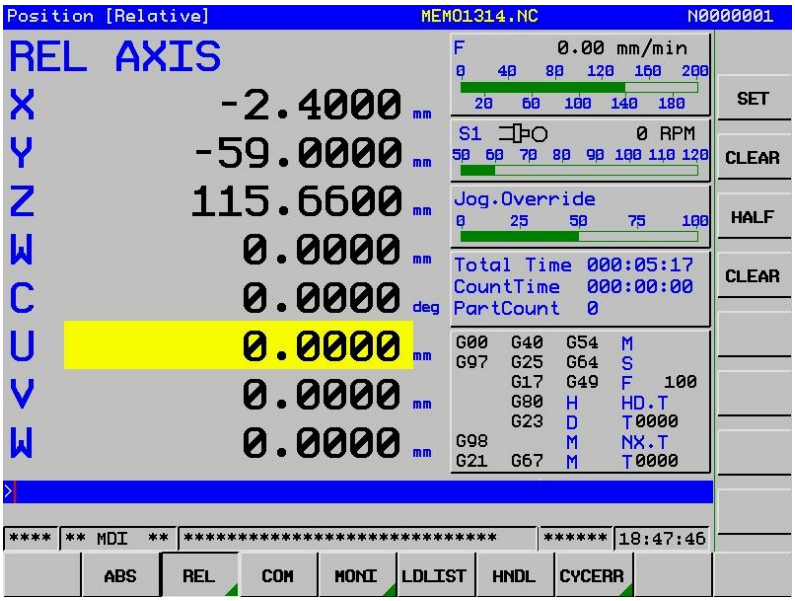


Figure 3-1-2 Relative coordinate

(1) Steps of the presetting operation in the relative coordinate system: Select the axis to be modified by ⇧ ⇩ keys, and the selected area is highlighted by yellow color. Enter the data to be set, press the [Preset] key, the data is entered in the corresponding coordinate, and the cursor jumps to the next axis.

(2) Steps of the zeroing operation in the relative coordinate system: Select the intended axis by ⇧ ⇩ keys, then press soft key to [Return to Zero], and the coordinate of the axis is cleared.

(3) Steps of the centering operation in the relative coordinate system: Select the intended axis by ⇧ ⇩ keys, then press soft key to [Centering], and the data of the axis is halved.

(4) Steps of the workpiece number reset in the relative coordinate system: When the quantity of workpieces is not zero, press soft key to make "Number of Workpieces Zero", and the quantity of workpieces is cleared.

Note: Set X, Y, and Z coordinate values to 999999.9999 or -999999.9999 in the relative coordinate interface, then operate manually to move incrementally in the same direction, and once the movement is executed, only the axis number not the value will be displayed for the coordinate.

3.1.3 Comprehensive interface

Press COM soft key to enter the comprehensive coordinate interface. The detailed page is illustrated below (see Figure 3-1-3).

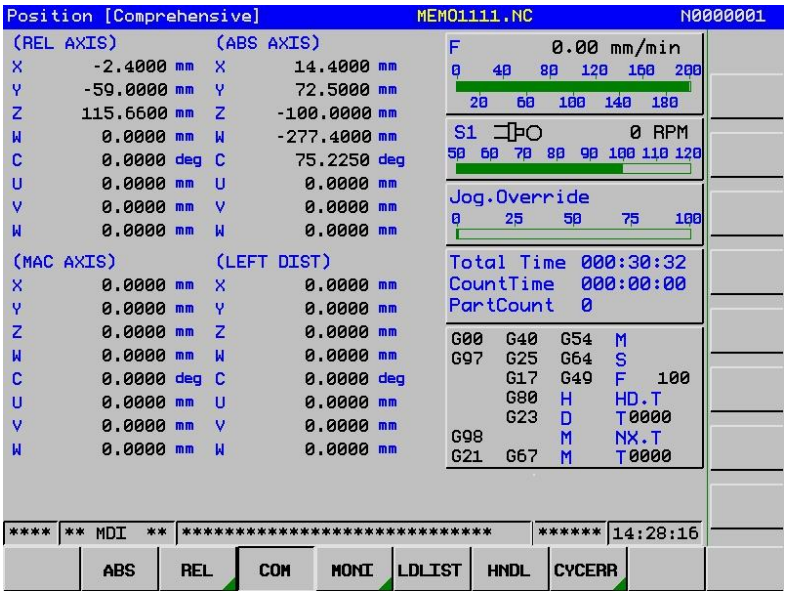


Figure 3-1-3 Comprehensive interface

The comprehensive interface can simultaneously display position coordinate values in the following coordinate systems: Position in the relative coordinate system, position in the absolute coordinate system, position in the machine coordinate system, remaining distance and various other information (including feed speed, spindle speed, fast override, running time, number of machined parts, current modal, etc.).

3.1.4 Monitoring mode

Press MONI soft key to enter the monitoring interface (see Figure 3-1-4).

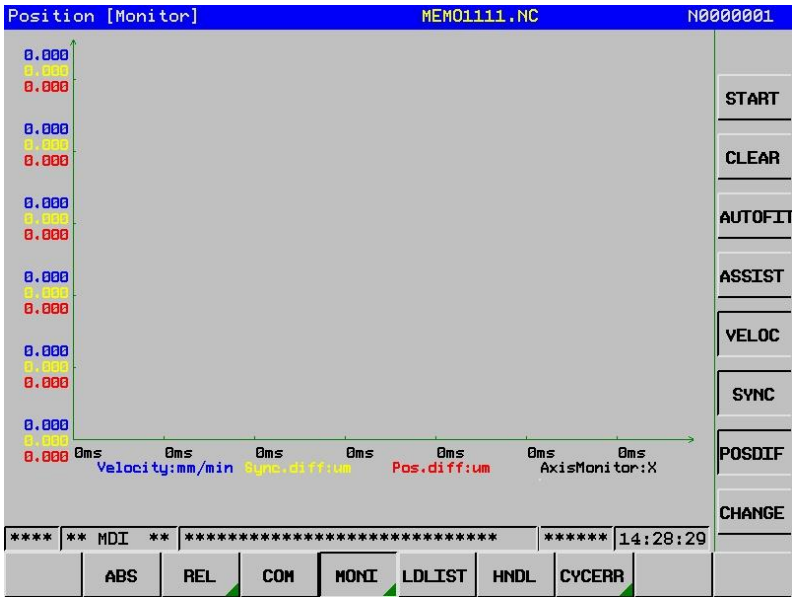


Figure 3-1-4

(1) Press the [Speed] key to turn on or off the speed monitoring, and the corresponding speed waveform will be displayed;

(2) Press the [Synchronization] key to turn on or off the synchronization monitoring, and the corresponding synchronization waveform will be displayed;

(3) Press the [Displacement Deviation] key to turn on or off the displacement deviation monitoring, and the corresponding displacement deviation waveform will be displayed;

(4) In non-startup mode, press the [Switch] key to switch monitoring axes, but this operation can't be performed in the startup mode;

(5) In non-startup mode, press the [Auxiliary Line] key to switch monitoring axes, but this operation can't be performed in the startup mode.

The operation of each axis can be viewed by modifying the axis selection parameters: Modify the waveform selection for viewing the waveforms of speed, synchronization and displacement deviation; The ratio of the two axes are changed to modify the display scale of the waveform, where the horizontal axis represents the timescale axis with each interval representing the time period of inputting, the vertical axis represents the distance scale axis with each interval represents the input distance block.

In the monitoring interface, the absolute coordinate value for the current operation and some simple modal information are also displayed.

3.1.5 Load meter

Press LDLIST soft key to enter the load meter interface (See Figure 3-1-5).

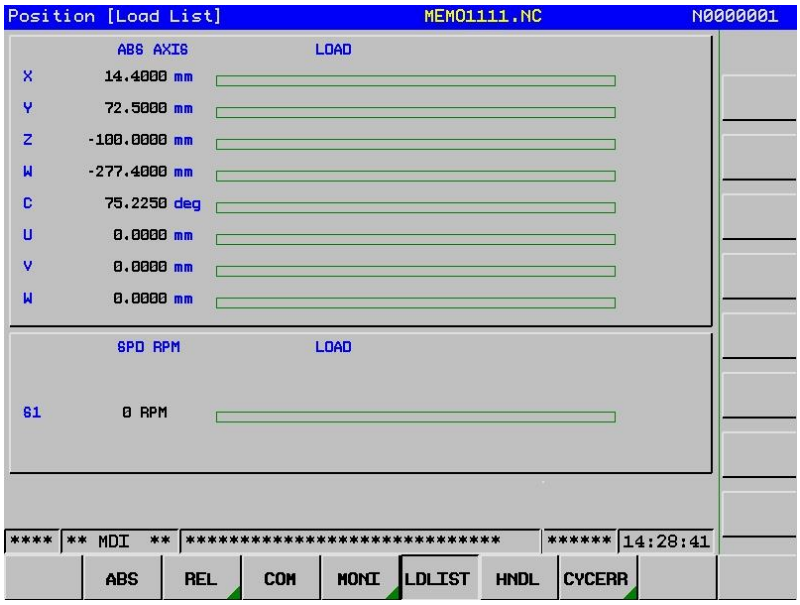


Figure 3-1-5

3.1.6 MPG

Press the HNDL soft key to enter the MPG insertion interface (See Figure 3-1-6).

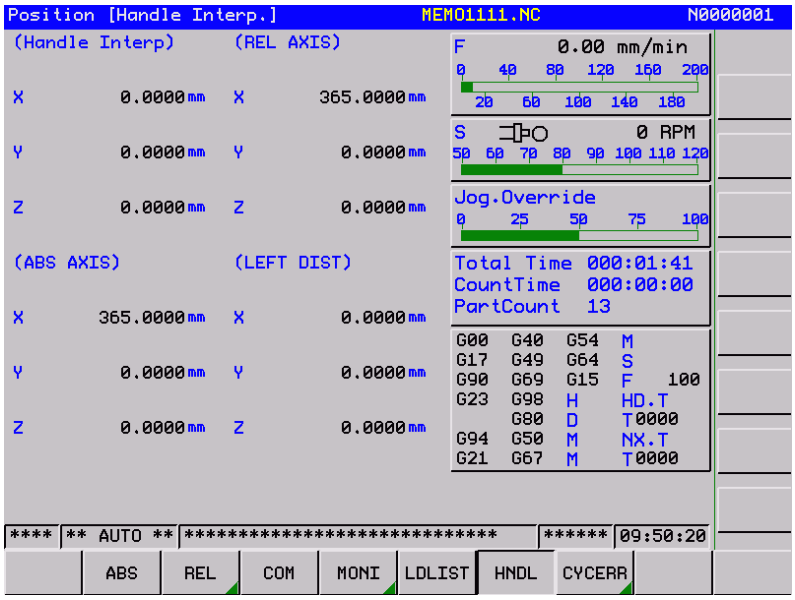


Figure 3-1-6

Entering the MPG interface display, you can monitor the MPG insertion amount of the servo axis. Refer to "7.1.11 MPG insertion" in Chapter 7 of this part for the usage of MPG insertion.

3.2 Program interfaces

Press the 

key to enter the program page display, and on the program display page there are five interfaces: [Program], [Test], [Data], [File List], and [Scheduling]. When the operation is in the entry mode, the [Detection] interface shifts to the [MDI] interface, and each interface can be viewed and modified through the corresponding soft keys (see Figure 3-2-1).

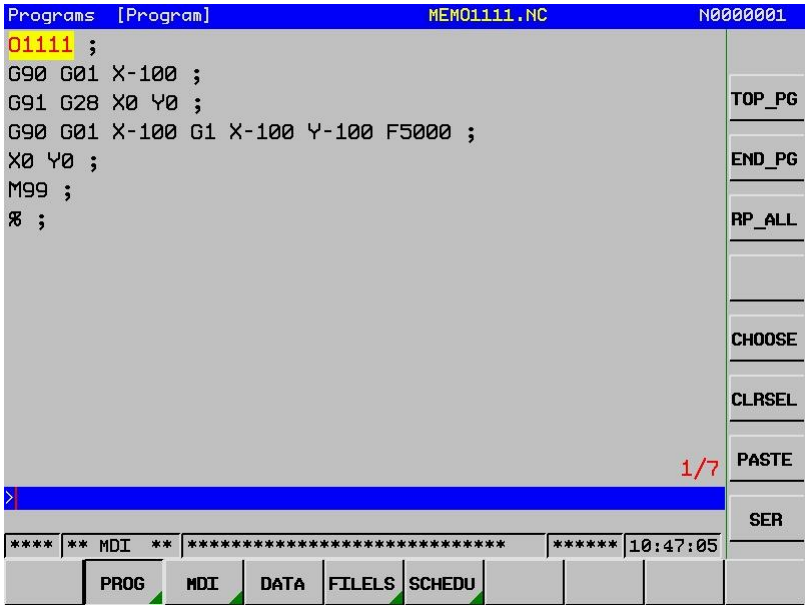


Figure 3-2-1

3.2.1 File list display

Press the PROGRAM key ➔ FILELS on the panel to enter the file list display interface (see Figure 3-2-2).

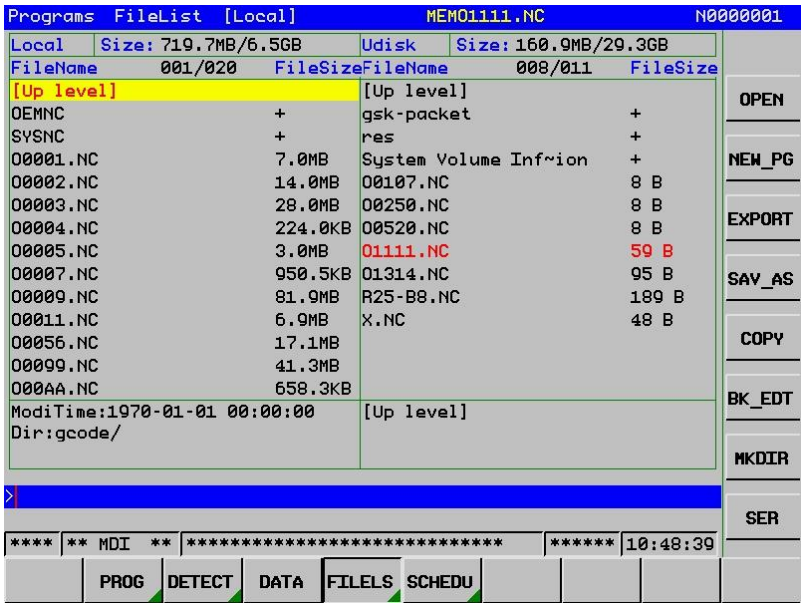


Figure 3-2-2

On the left of [File List] display interface is the local file list display interface, and the right side is the USB flash drive file list display interface. The left and right keys can be used to shift the selection area.

(1) Local size: as shown in the figure, 11.8 MB is the storage capacity occupied by the stored programs; 511.5 MB is the total storage capacity space.

(2) USB flash drive size: as shown in the figure, 2.1 GB is the storage capacity occupied by the stored programs; 7.4 GB is the total program storage capacity space.

Note:

1. This system can only load programs up to 8 MB in the editing mode. For a program larger than 8MB, please use the network DNC or USB flash drive DNC in machining.
2. The program directory display page can display up to 10 CNC program names at a time. If there are more than 10 CNC programs, they will not be fully displayed in one page. Then, you can press the page key on the page. Press the page turning key repeatedly, the LCD displays all CNC program names in a rotating manner.
3. The system only displays file names ended with *.NC in the file list. ? ? ? , directory names are displayed for English directory; and no directory names longer than 31 characters will be displayed.
4. The file name is not allowed to be written in Chinese characters, otherwise it will be displayed as "? ? ? " and copy errors will arise. When a file name larger than or equal to 28 characters is copied into the system, the file won't be displayed on the file list.
5. USB device hot swap is allowed, but please wait for a USB device to read and write operations before unplugging the USB device, otherwise an error may be resulted.
6. USB devices are not allowed to be plugged or unplugged while in loading, importing, and exporting files. Import and export operations must be performed in the editing mode.
7. Subprograms and macro programs invoked are limited to NC programs in the system memory and NC programs in USB devices can't be called.

3.2.2 Transfer program from USB flash drive

1. Step of transferring programs from USB flash drive to NC:

(1) Rotate the , select the editing mode ➔ PROGRAM key ➔ FILELS , and enter the file list display interface. Insert the USB flash drive, and allow it to be loaded in the system. After successful

loading, the folder and *.NC file name information in the USB flash drive will be displayed, and the prompt "SD card loaded successfully" will appear. The prompt disappears after pressing the 

key on the panel or proceeding to the next operation. You can shift between "local" and "USB flash drive" by pressing the left and right keys (see Figure 3-2-3).

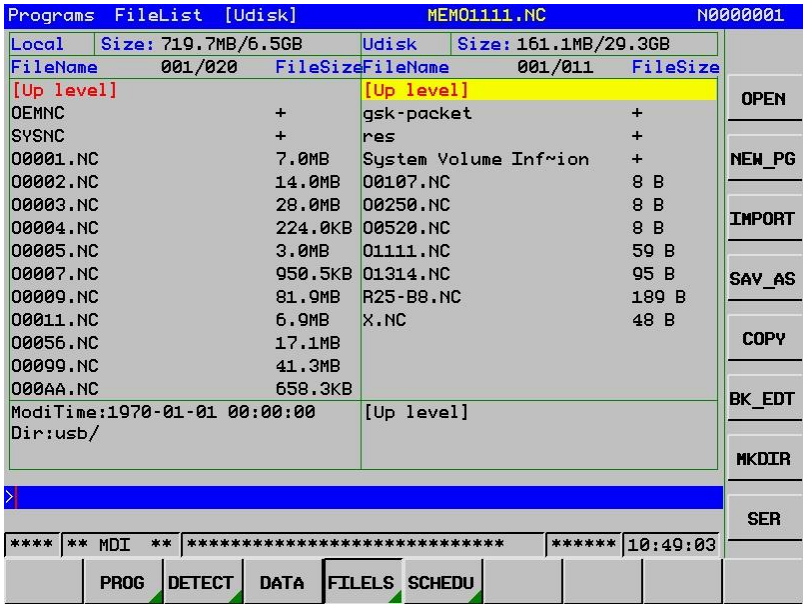


Figure 3-2-3

(2) Press   page turning keys or press   cursor keys to select the program to be imported. Press the [Import File] sub-soft key on the right side of the interface, and the selected program will be transferred to local memory.

Example: file name X.NC (see Figure 3-2-4).

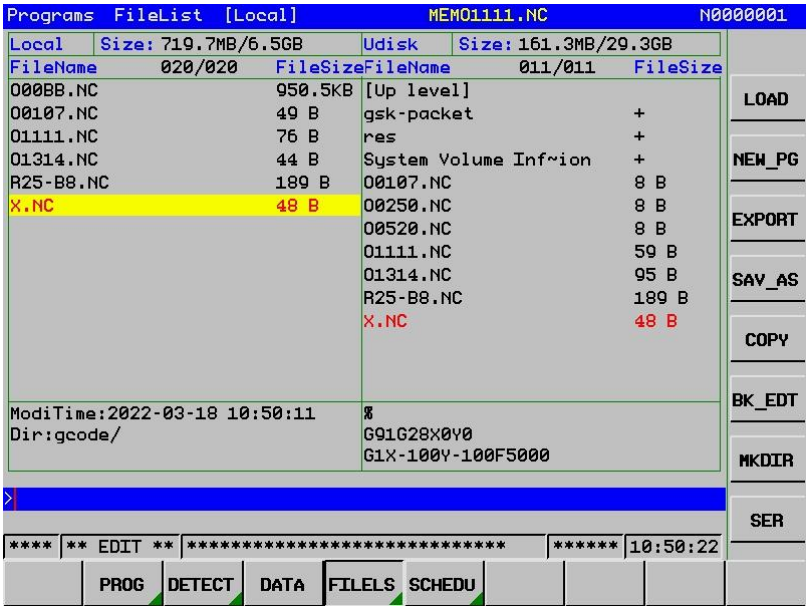


Figure 3-2-4

2. Steps of transferring programs from memory to USB flash drive:

(1) According to the steps of transferring programs from USB flash drive to NC, go to the following screen. Press  ,  page turning keys or press  ,  cursor keys to select the program to be exported.

Example: file name O1111.NC (see Figure 3-2-5).

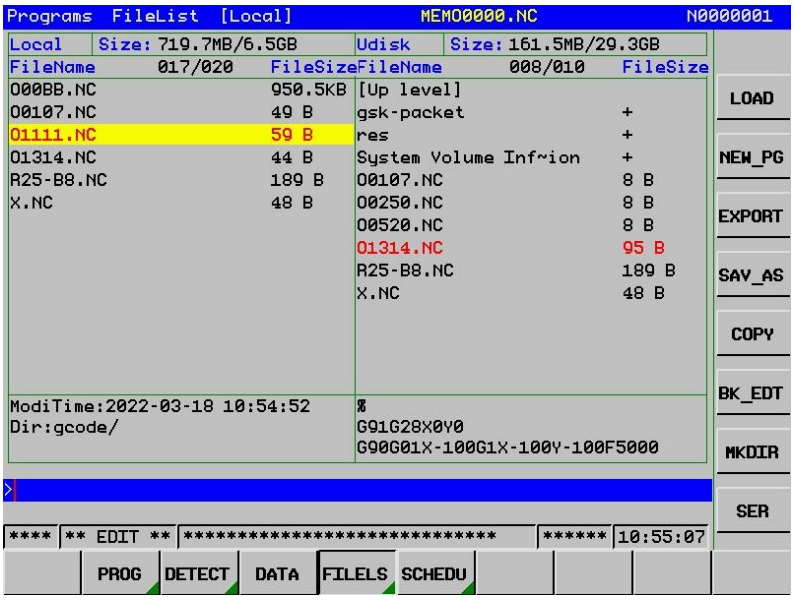


Figure 3-2-5

(2) Press the [Export File] sub-soft key (see Figure 3-2-6).

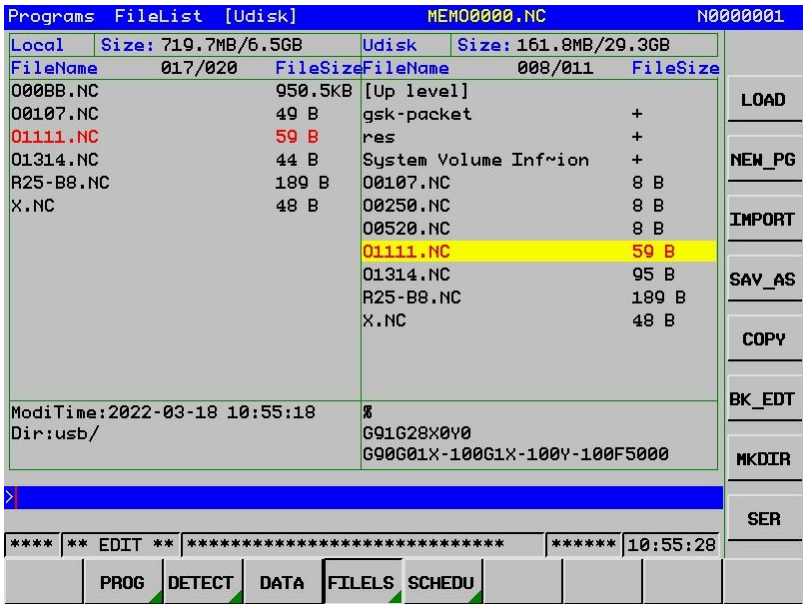


Figure 3-2-6

The program is exported to the current folder in the USB flash drive, press the panel  key to enter the USB flash drive file list, so that the USB flash drive folder can be refreshed to show the transferred program O1111.NC. After related operations have finished, directly unplug the USB flash drive, and prompt "SD card has been successfully downloaded" appears. The prompt disappears after

pressing the 

key on the panel or proceeding to the next operation.

3.2.3 Program display

Press the 

key program ➔ PROG on the panel to enter the Program display interface. This interface will display the G code being executed or waiting to be executed in the memory (see Figure 3-2-7). Here, "local O1111.NC" represents the program name, "N0000001" means the number of lines of running codes for the currently executed codes, 1 of "1/6" is the number of lines currently running, and 6 denotes the total number of lines.

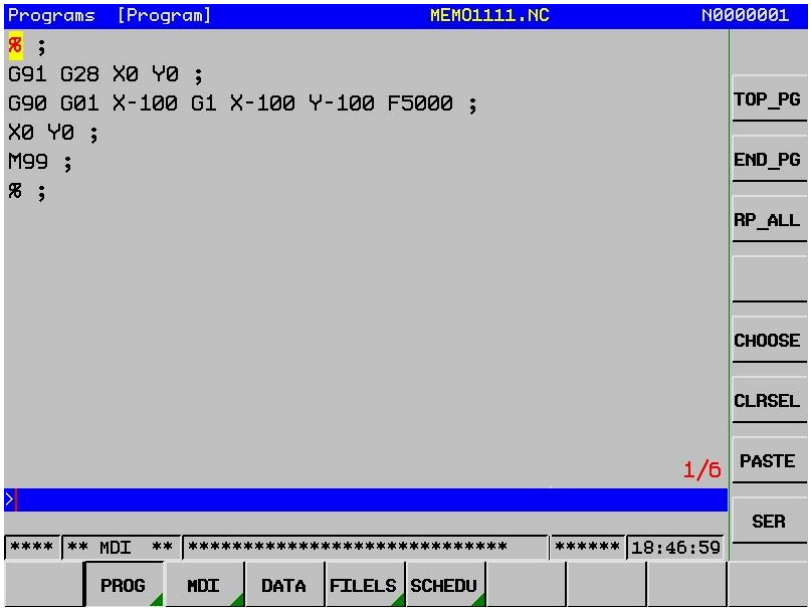


Figure 3-2-7

3.2.4 Program creation

This function describes in detail when editing the program, including the new creation of the program, character insertion, modification, deletion, copy, paste, replacement. It also includes the function of deleting the complete program and inserting the automatic sequence number.

New programs can be created on the [Program], [Detection], or [File List] pages. The specific operation steps are as follows:

1. New program creation on the PROG page and its operation steps:

Rotate the  , select the editing mode ➔ 

key ➔ PROG , and go to the program interface. Enter the program name such as O0520 in the input field (see Figure 3-2-8).

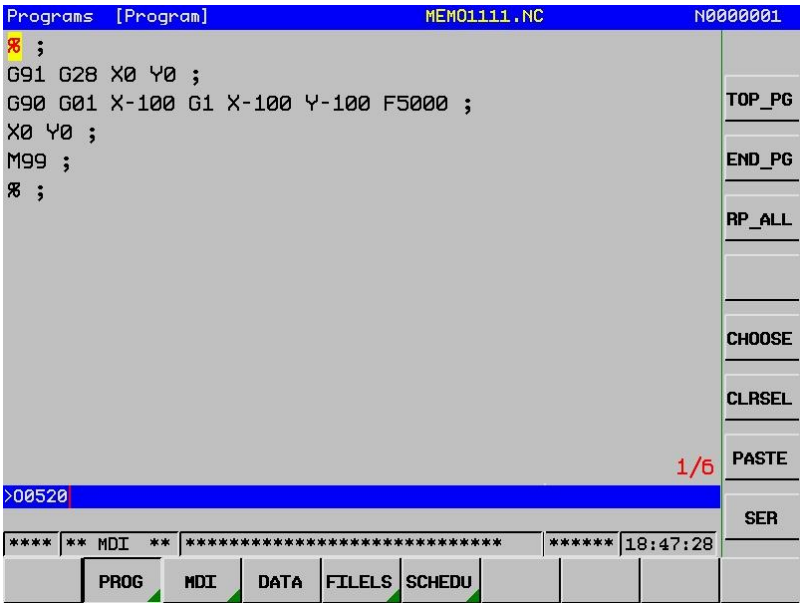


Figure 3-2-8

Press the INSERT key on the panel to create a new program and update the current interface to be a new program interface (see Figure 3-2-9).

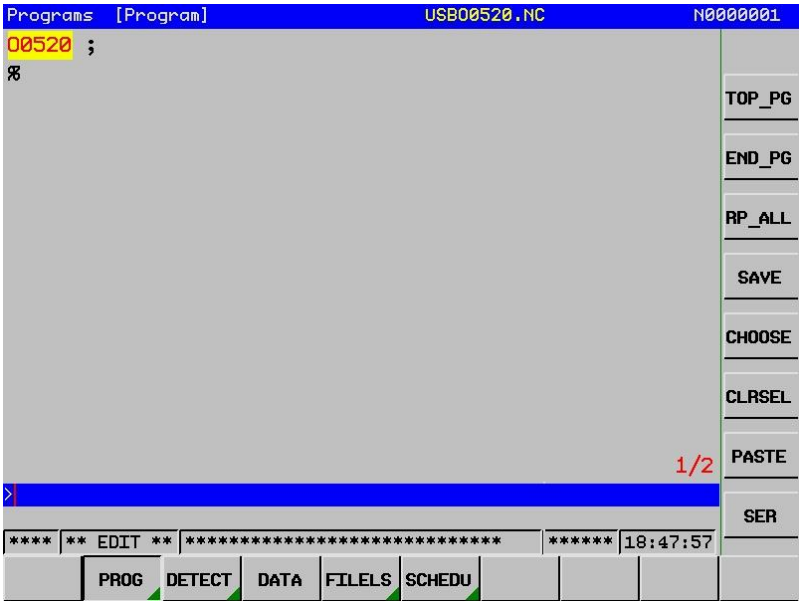


Figure 3-2-9

2. New program creation on the DETECT page and its operation steps:

Rotate the , select the editing mode ➔ PROGRAM ➔ DETECT , and go to the detection interface.

Enter the program name in the input field, such as O1314, and press the INSERT key to create a new program. The display interface is similar to that shown in Figure 3-2-8 and Figure 3-2-9.

3. New program creation on the FILELS page and its operation steps:

Rotate the , select the editing mode ➔ PROGRAM ➔ FILELS , and go to the file list interface.

Enter the program name O1314 in the input field (see Figure 3-2-10).

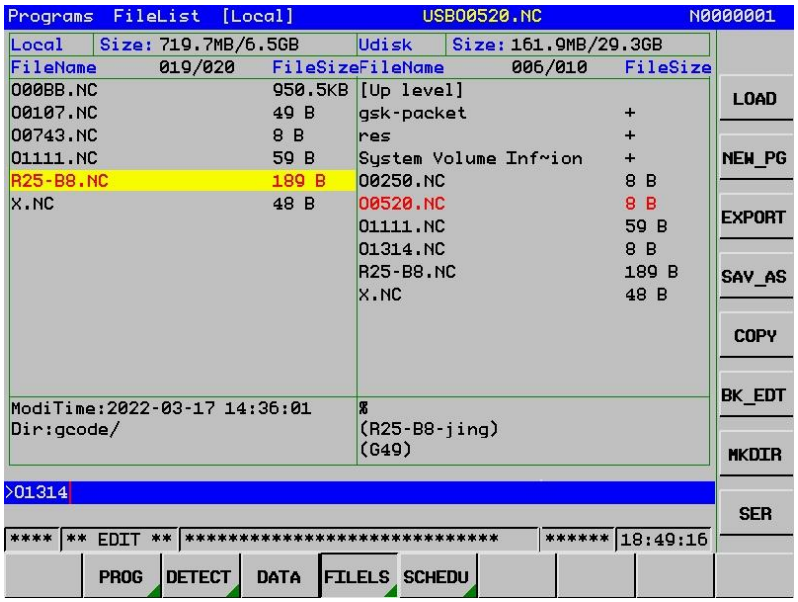


Figure 3-2-10

Press the 

key or the NEW_PG sub-soft key on the panel to create a new program (see Figure 3-2-11).

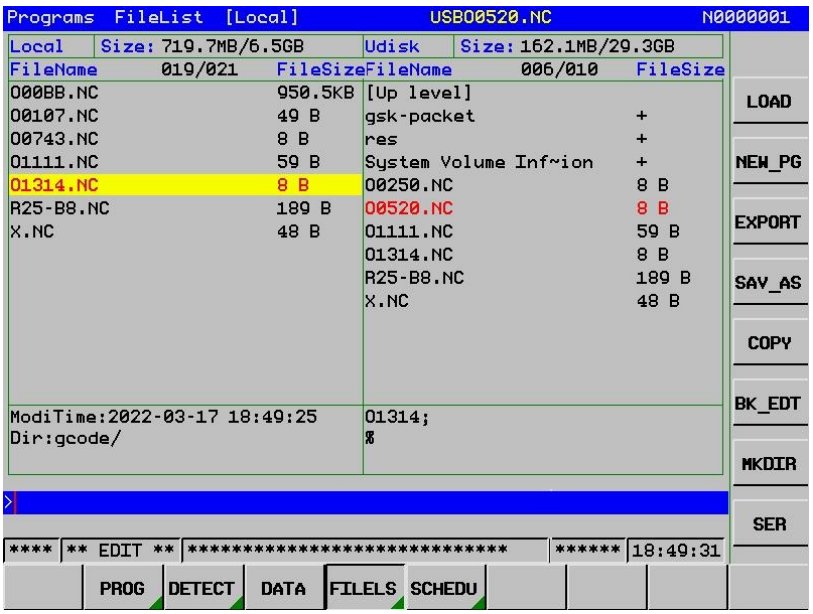


Figure 3-2-11

The new program will be loaded if pressing the LOAD or 

key (see Figure 3-2-12).

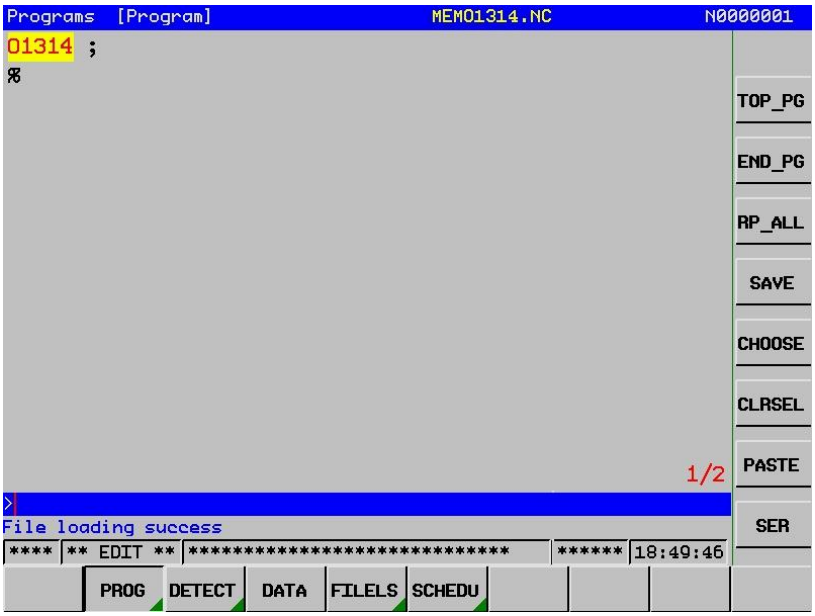


Figure 3-2-12

Note:

1. In the editing mode, only NC programs within 8M are allowed to be loaded directly. NC programs larger than 8M should be loaded in the DNC mode.
2. Only the program name starting with the letter O plus a maximum of four digits can be written during its creation. For example, for other letters or directly write-in numbers, if the program name does not conform to the rules, the system will remind of invalid input, or the input size or length being out of bounds.
3. Whether the edited file under the current editing interface is automatically saved is determined by parameter #100.3 when a new file is loaded.
4. When a program is edited, the system will prompt invalid input, input size or length out of bounds if more than 60 characters are entered into the input field.
5. If the NC file name exceeds 17 characters, the first 17 letters are captured and automatically changed to ~. NC after the letters are displayed.
   For example, the program name O12345678912345678912345. NC is displayed on the system as O1234567891234567 ~. NC.

4. Save as of the program and its operation steps:

On the FILELS page, select the program file to be saved as, and enter the new program file name in the input field, for example: file name O0123 (see Figure 3-2-13).

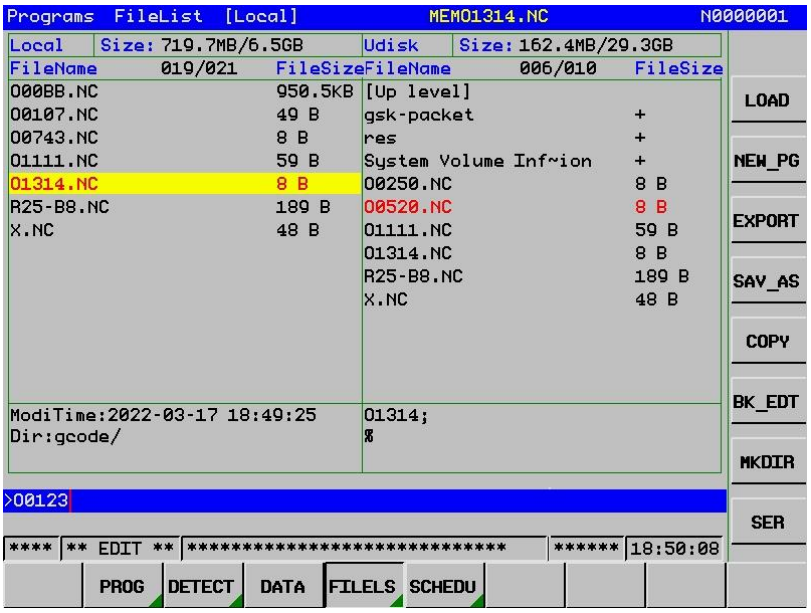


Figure 3-2-13

Press the soft key SAV_AS as to copy the currently selected program file as the O0123.nc program file (see Figure 3-2-14).

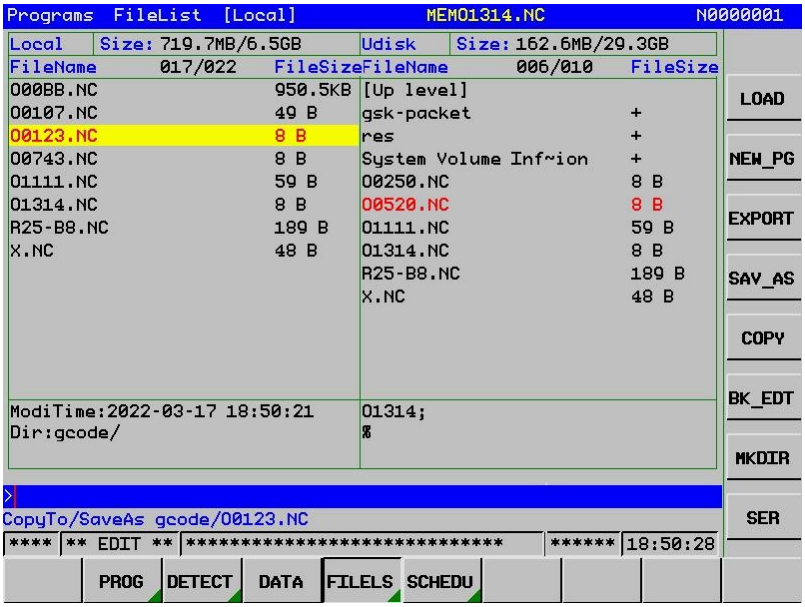


Figure 3-2-14

3.2.5 Program editing

The system can edit, select or delete part programs from the newly created program through the operation panel.

When the program is edited, whether the system automatically inserts the sequence number between the program blocks is set by parameter #0001.5, and the incremental value of sequence numbers is set by parameter #1621. Users can set these two parameters as needed. Please refer to the instruction manual for details about the parameters.

1. Operation method of editing a program:

(1) The program editing needs to be carried out in the [Edit] mode. Rotate the  to select the editing mode. Follow the operation steps of program creation, and go to the new program interface (see Figure 3-2-15).

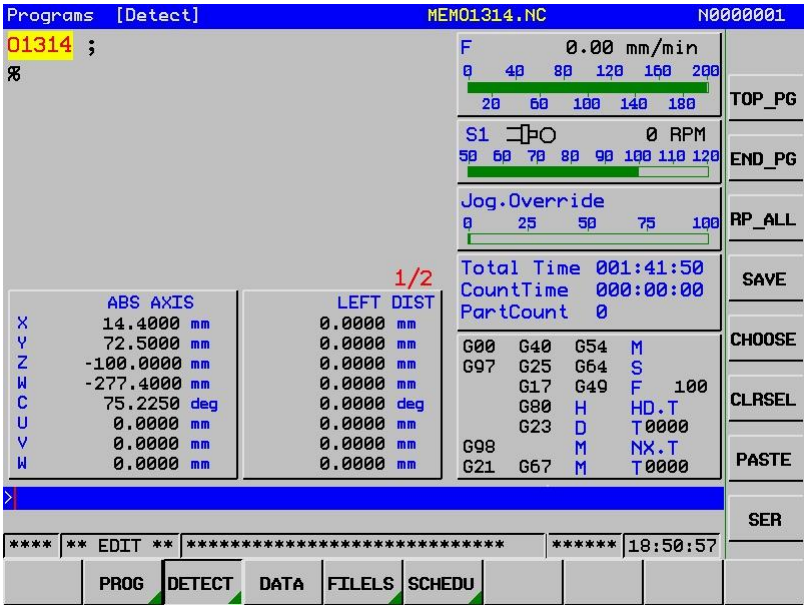


Figure 3-2-15

(2) Enter the command code or movement command of each axis in the input field. Press the 

key on the operation panel to divide the command into lines, and finally press the 

key to complete the program editing (see Figure 3-2-16 and Figure 3-2-17).

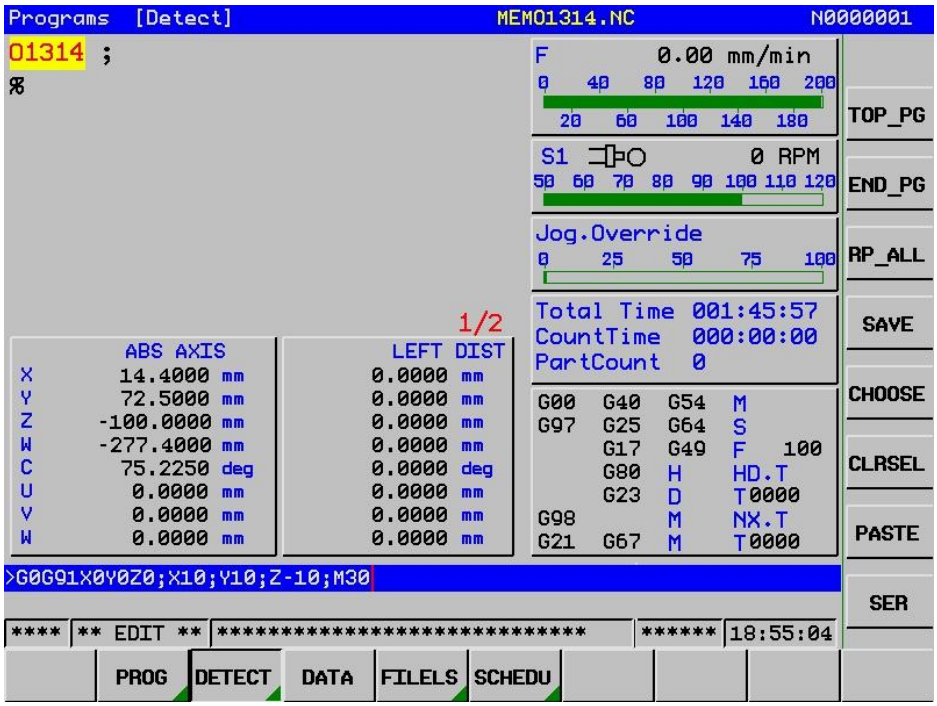


Figure 3-2-16

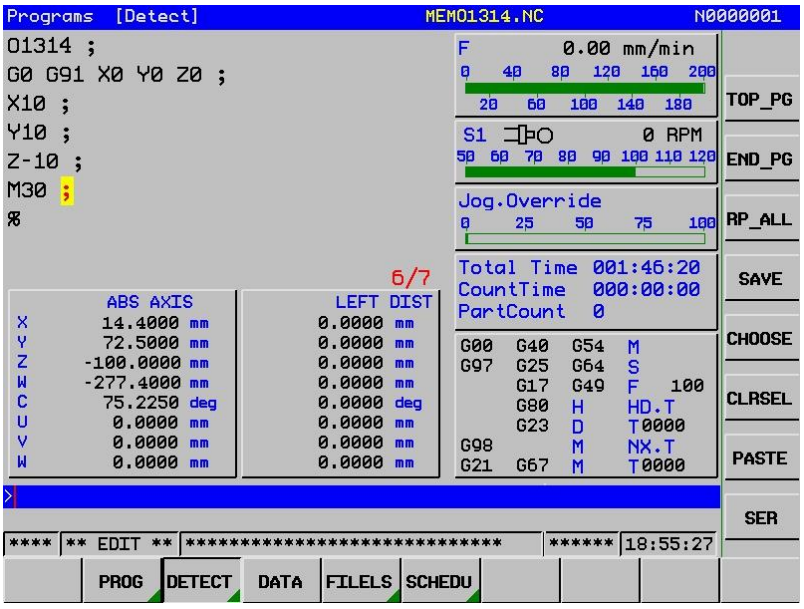


Figure 3-2-17

The system provides the sub-soft keys to TOP_PG , END_PG , RP_ALL , SAVE , CHOOSE , CLRSEL , PASTE , and SER , which can provide convenient for customers to edit programs.

1) Use of [to program header] and [to program end]:

In the program editing screen, press the sub-soft key to TOP_PG or to END_PG , and the program interface immediately switches to the beginning or end of the program.

2) Use of [Replace All]: Select the command that needs to be replaced in the program editing screen, such as F3000, and then enter the replacement content in the input field, such as F1000, and press the soft key RP_ALL . All commands in the program that are the same as the selected command are replaced. For example: if F3000 is selected in the program, F1000 is entered in the input field and press the sub-soft key RP_ALL , then all F3000 in the program will be replaced with F1000, while other F values in the program, such as F800, F1500, will not be replaced.

3) [Search] has two functions: search code and search line number. The specific use method is detailed as follows:

A: Search Code

①Write a code or movement command for each axis in the input field, press the sub-soft key SER continuously, and the content to be searched will be searched circularly on the program interface.

② Write a code or movement command for each axis in the input field, press the ⇧ cursor key to search up until the program header is found, press the ⇩ cursor key to search down until the program end is reached, and a prompt appears after reaching the program end in one direction: "The search content does not exist".

For example: Search code S1000

Enter S1000 in the input field

Press the ⇩ cursor keys to perform a search from the program block where the cursor stays to the direction of the program end. Press the ⇧ cursor keys to perform a search from the program block where the cursor stays to the direction of the program header.

B: Search Line Number

[Search line number] is used to search a program block in the whole program. For example, if you want to search the 55th line of the program, you can write 55 in the input field, and press the soft key SER to search, the system will automatically search until the 55th line of the program block.

4) Usage of [Select]:

In the program editing screen, move the cursor to the line where the program needs to be modified, press the soft key CHOOSE to select, and the selected program will be shown in yellow. Select a multi-line program by pressing ⇩ ⇧ cursor keys.

5) Usage of [Save]:

When the input program is less than 500K, the modified program will be automatically saved (parameter # 100.3 bits must be opened); when the input program is equal to or greater than 500K, the system prompts "Please save the program manually" and the program will only be saved by pressing the soft key SAVE. (See Figure 3-2-18)

If the program over 500K is not saved after editing, the content of the program can be restored by reloading. When loading other programs, whether the original program is saved is determined by parameter #100.3 ASF.

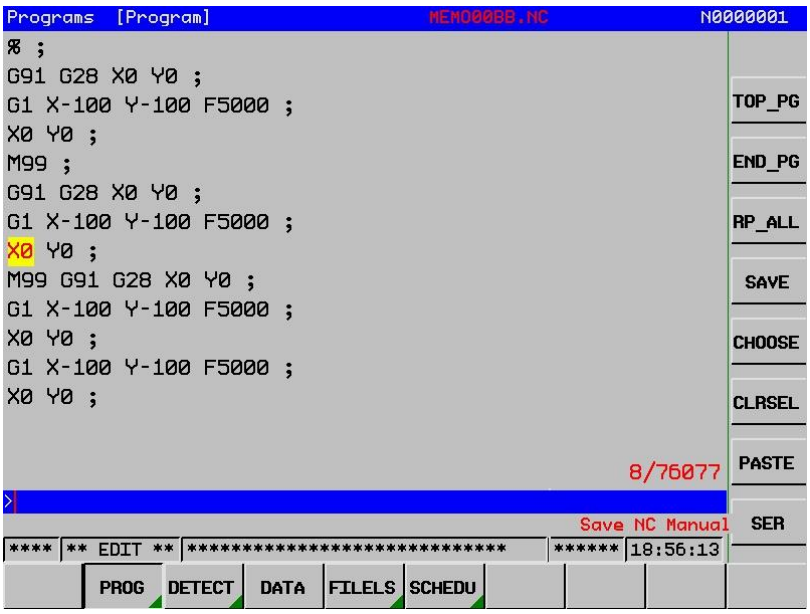


Figure 3-2-18

2. Insertion, substitution and deletion of characters

This section describes how to insert, replace and delete characters from programs that have been entered into the memory.

Insert operation steps:

(1) Rotate the  , select the editing mode, and then press the 

key.

(If the program to be edited has not been loaded, start from step (2); if the program is loaded, enter the program editing interface and start from step (3).)

(2) Go to the file list interface, press  ,  keys to move the cursor, select the program to be edited, and press the soft key 

to load the program or the panel 

key.

(3) Retrieve characters that need to be modified. This can be done by performing a [Search].

(4) Inserts can be divided into the insertion of characters and the insertion of program blocks.

1) Retrieve the address field to be inserted, press the  ,  keys on the panel to select it, enter the character in the input field, press the 

key on the panel, and insert the characters into the specified program block.

For example: if you want to insert the character S100 after Z0, move the cursor to Z0 first, then enter S100 in the input field, and press the 

key to complete (see Figure 3-2-19).

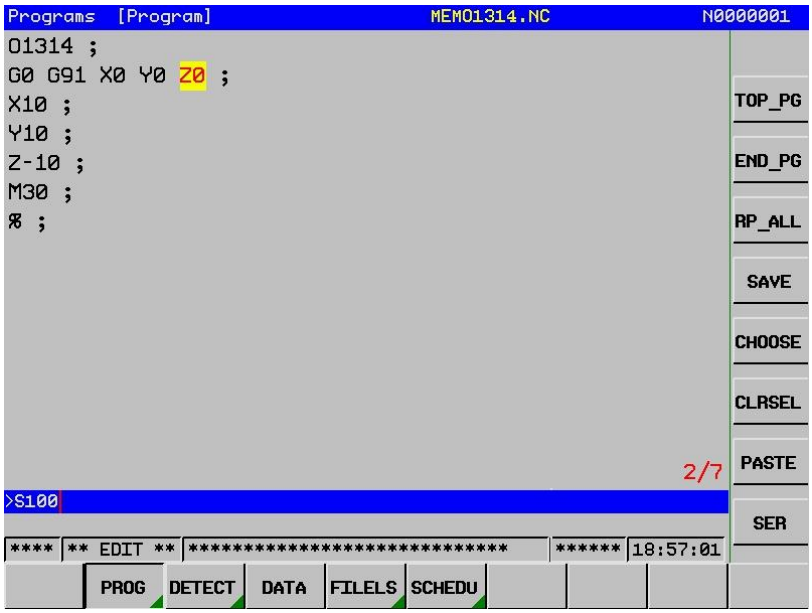


Figure 3-2-19

After insertion, it will be shown as Figure 3-2-20.

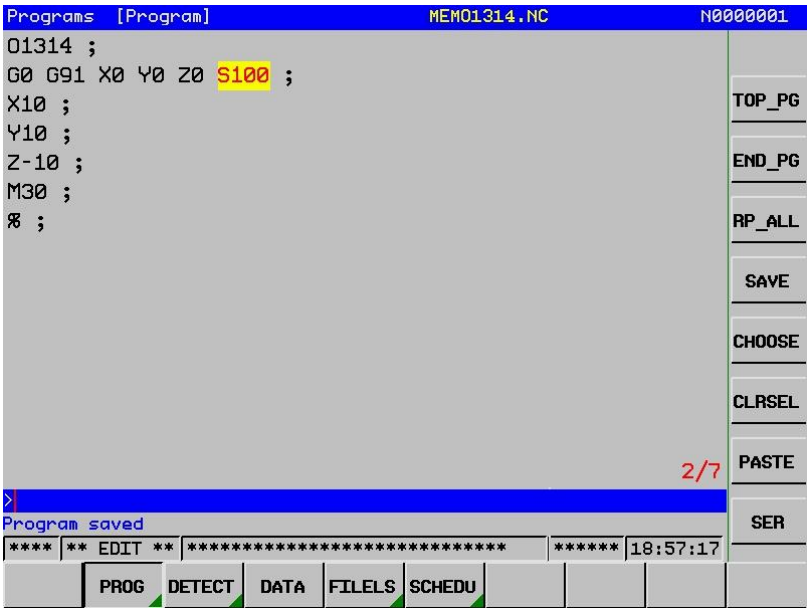


Figure 3-2-20

2) Retrieve the address block to be inserted, press the panel "  ,  " cursor keys, move the cursor to the semicolon of the previous program, edit the block to be inserted in the input field, press the  ➔  key, and insert the program block automatically into the program.

For example: if you want to insert the program block M3 S100 between the block G0 G91 X0 Y0 Z0 S100 and the program degree X10, first move the cursor to the semicolon after S100, then enter M3 S100 in the input field and press the  ➔  key on the panel to complete, as shown in Figure 3-2-21.



Figure 3-2-21

The picture after insertion is shown as Figure 3-2-22.

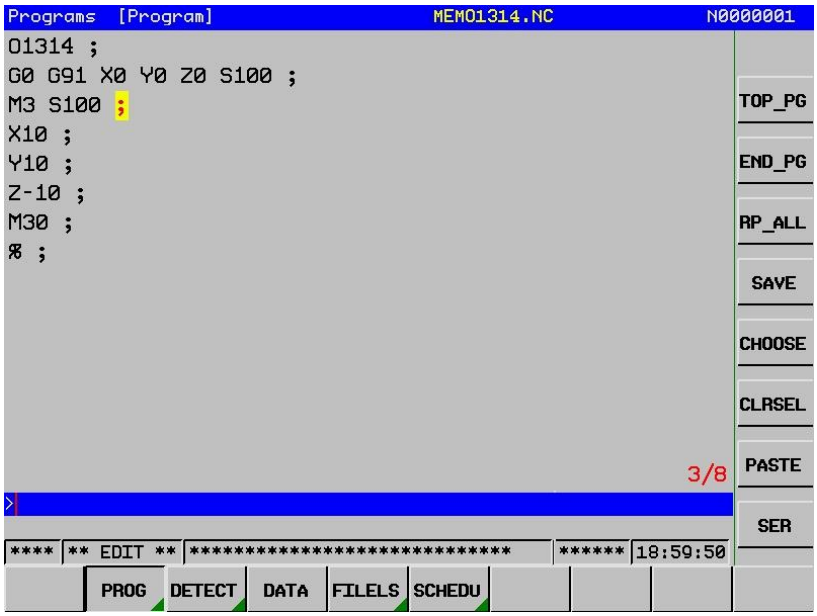


Figure 3-2-22

3. Operation steps of replacement:

(1) Search the program editing screen for characters to be replaced.

(2) Enter the characters to be replaced in the input field.

(3) Press the 

key on the panel to replace the original characters.

For example: if you want to replace M3 with M04, first move the cursor to M3, then enter M04 in the input field and press the 

key on the panel to complete, as shown in Figure 3-2-23.

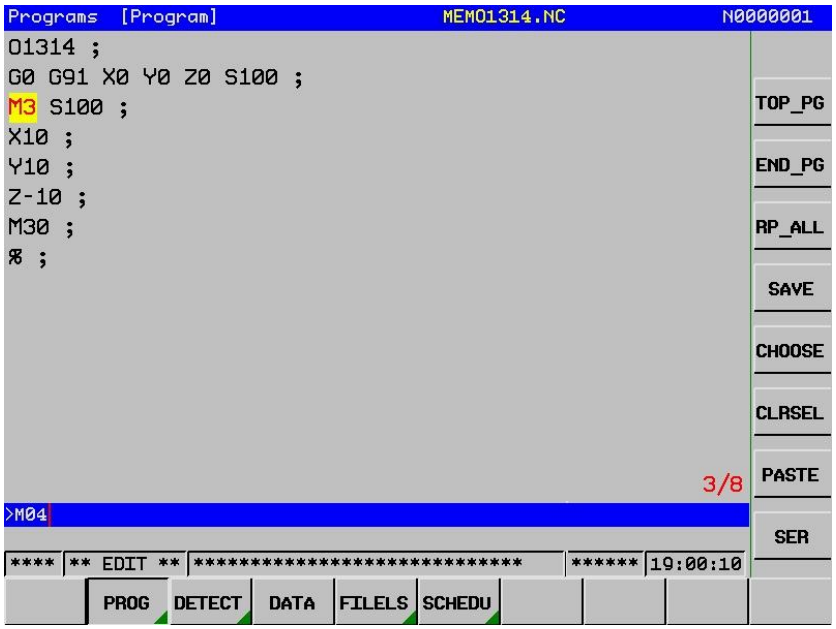


Figure 3-2-23

Figure 3-2-24 shows the scene after replacement.

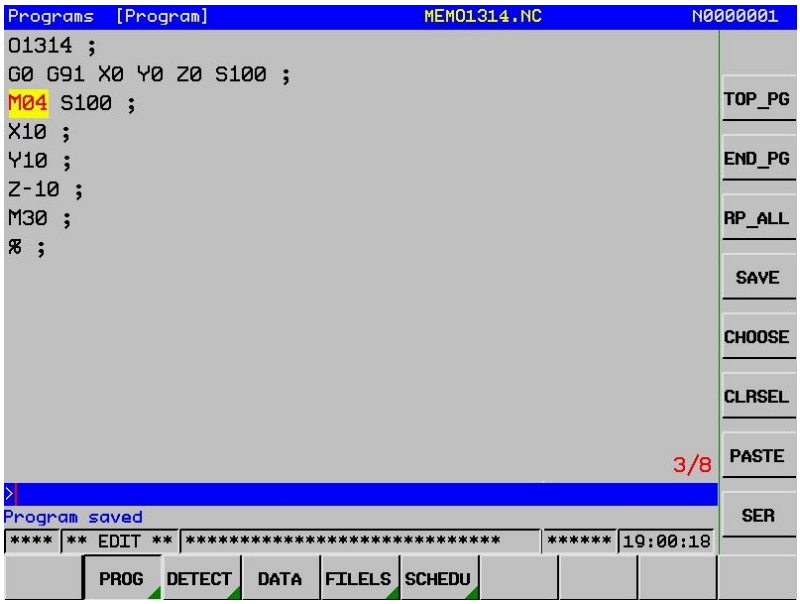


Figure 3-2-24

4. Operation steps of deletion:

(1) Rotate the  , select the editing mode, and enter the program editing screen.

(2)Press the cursor movement keys  ,  ,  ,  or the page turning keys  ,  to move the cursor to the place of character deleting.

(3) Press 

on the panel to delete.

For example, if you want to delete the character M04, first move the cursor to M04, and then press the 

key on the panel to complete, as shown in Figure 3-2-25.

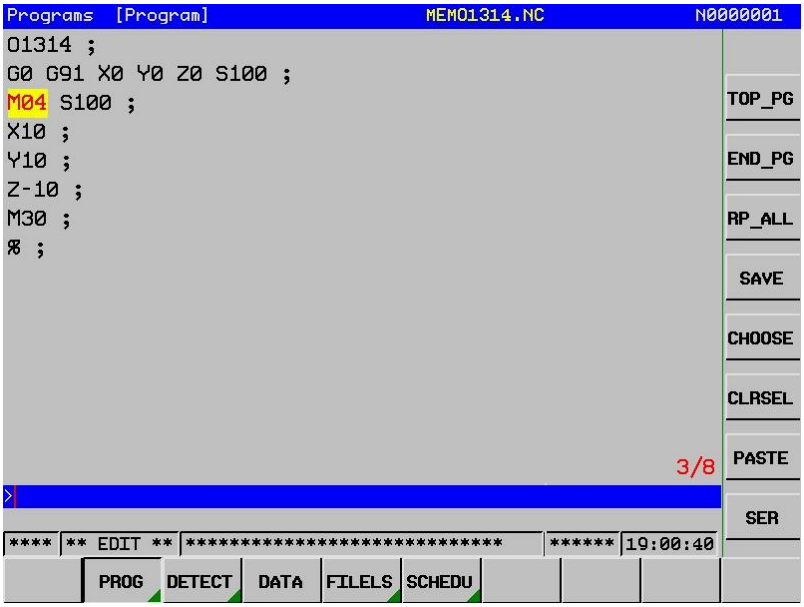


Figure 3-2-25

Figure 3-2-26 shows the scene after deletion.

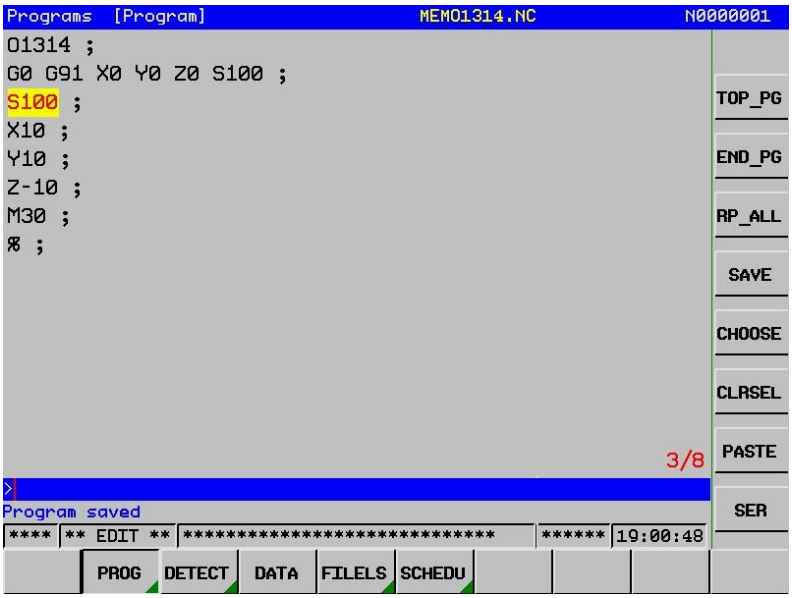


Figure 3-2-26

5. Copy, clip, paste, delete and replace in the extended editing.

Use the editing in extended program and perform the following actions:

(1) A complete program or part of a program is completely copied or moved to another program.

(2) A program can be merged with any location of another program.

(3) The character or address specified in a program can be replaced by another character or address.

6. Steps of copying multiline programs:

(1) Load the program O1314 in the [Edit] mode.

(2) Press the page turning keys  ,  or the cursor keys  ,  to move the cursor to the starting line of the copy target, as shown in Figure 3-2-27.

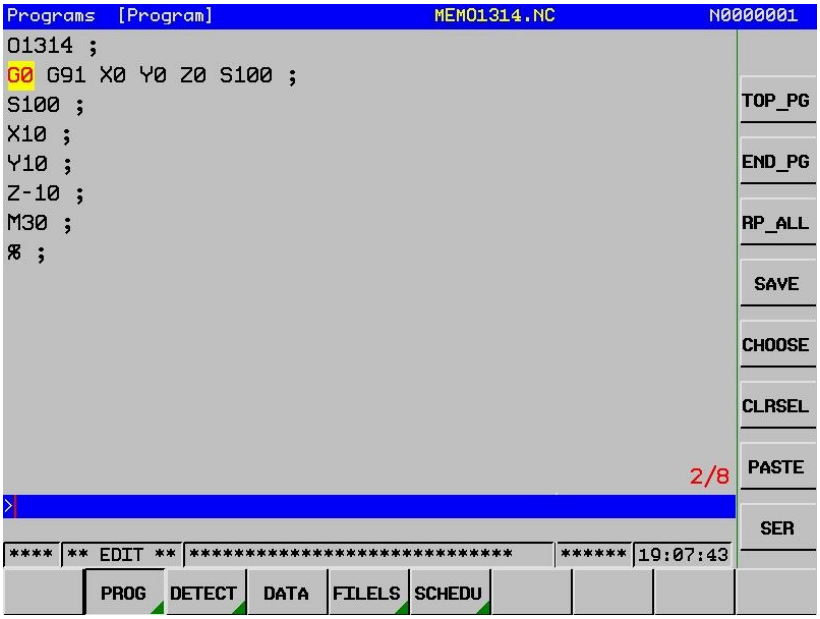


Figure 3-2-27

(3) Press the sub-soft key CHOOSE to select, and then press the  key or the page turning key



to move to the final copy program line (see Figure 3-2-28).

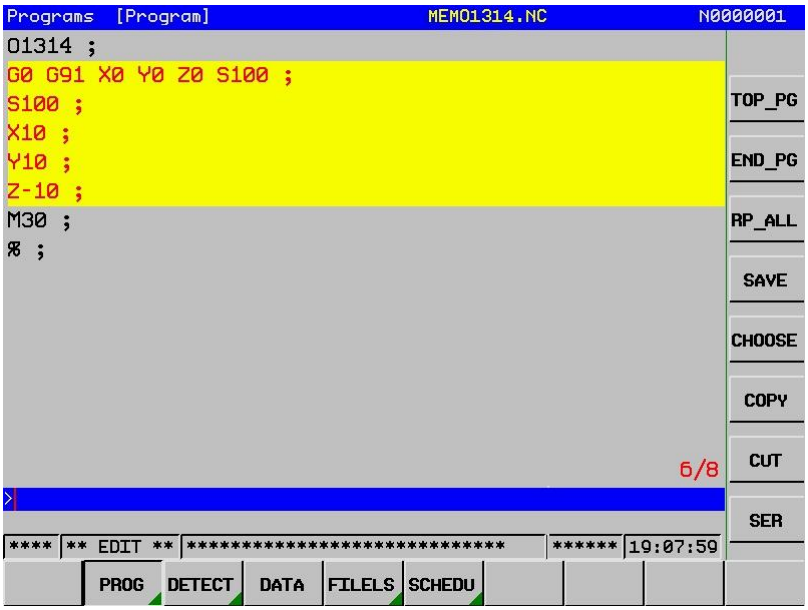


Figure 3-2-28

(4) Press the sub-soft key COPY to copy, move the cursor, and then press the sub-soft key PASTE to paste. The copied content will be pasted in the previous line of the cursor line, and then the copying procedure is completed, as shown in Figure 3-2-29.

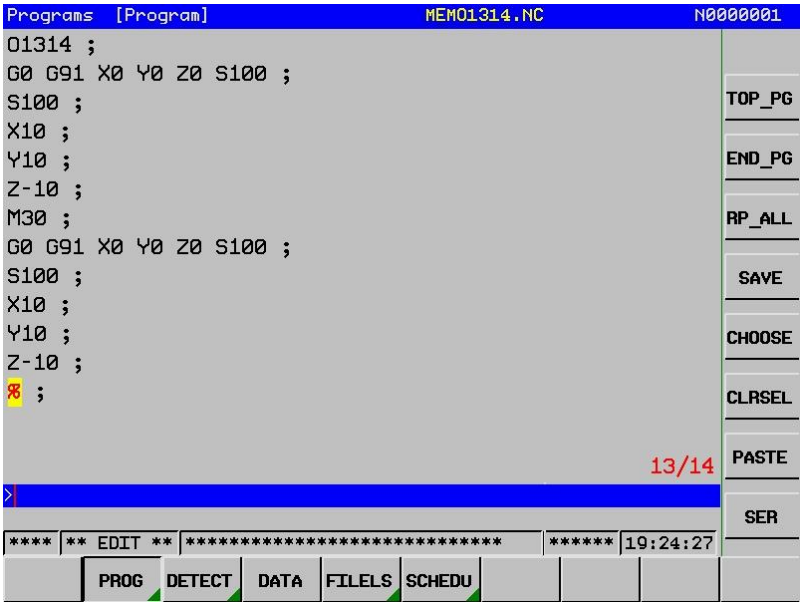


Figure 3-2-29

Note:

(1) When copying a program, the selected program content cannot contain the first line (program name) and the last line (%), otherwise the system cannot successfully copy.

(2) For the copy of multi-line programs, the system stipulates that a maximum of 5000 lines can be copied.

(3) Only in the selection state will the sub-soft keys COPY and CUT be displayed, and the soft key PASTE

will be displayed when the COPY or CUT keys are pressed.

7. Operation steps of clipping program:

(1) Follow the steps 1~3 to copy a multiline program, and select the multiline program block to be clipped, as shown in Figure 3-2-30.

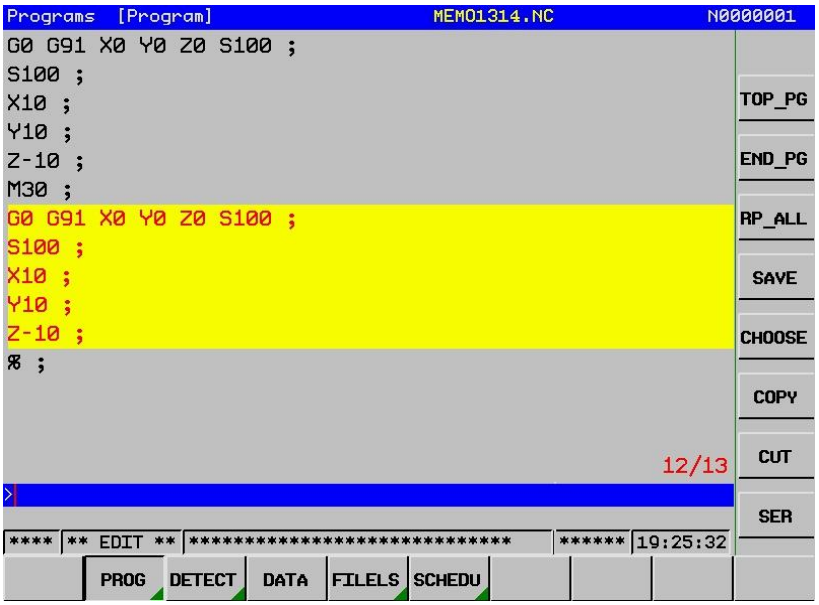


Figure 3-2-30

(2) Press the sub-soft key CUT , press the FILELS soft key to enter the file list interface, create a new program and press the sub-soft key LOAD to load the program, press the sub-soft key PASTE, and the program clip step is completed, as shown in Figures 3-2-31 and 3-2-32.

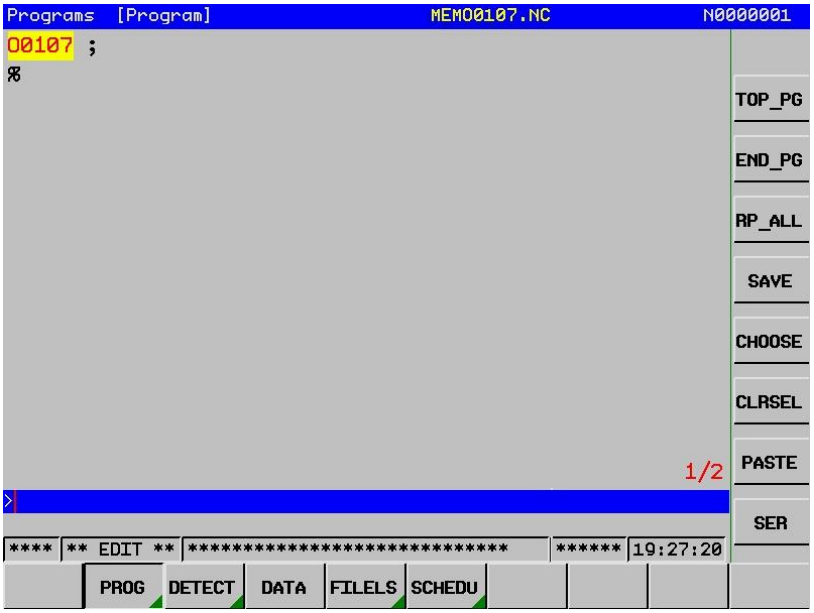


Figure 3-2-31

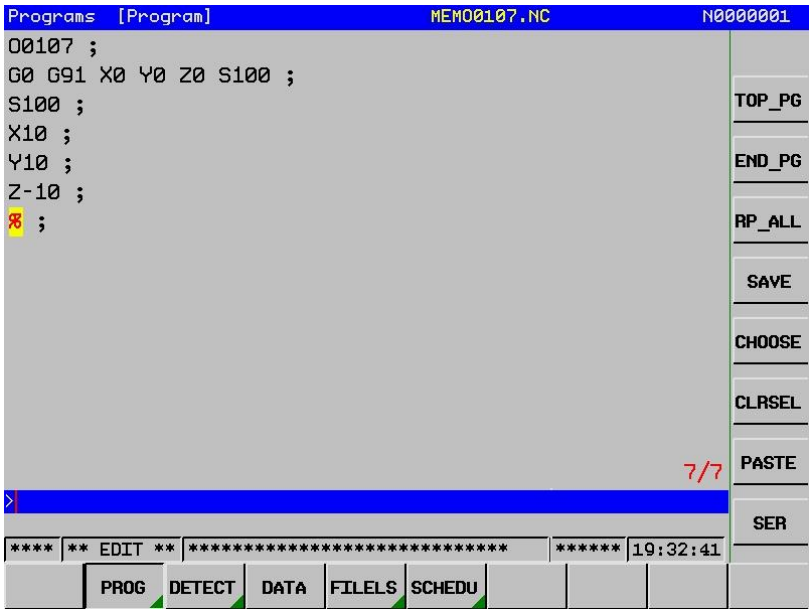


Figure 3-2-32

8. Operation steps of pasting program:

(1) Follow the steps described above for copying and clipping programs to copy selected program blocks.

(2) Copy will be programmed according to the specified location.

Note:

(1) Only when the copy and clipping functions are selected, the PASTE soft key will appear in the new program, otherwise the PASTE soft key will not appear.

(2) When copying the first line of the program, the copy is invalid because the first line (program name) copy is invalid.

(3) After the copy, click the PASTE soft key and the copied content is pasted on the previous line of the cursor line after the paste operation has been done.

9. Operation steps of replacing program:

(1) Press the , page turning keys or the , cursor keys, and move to the object code. Or press the SER soft key to find the code directly after entering the object code, as shown in Figure 3-2-33.

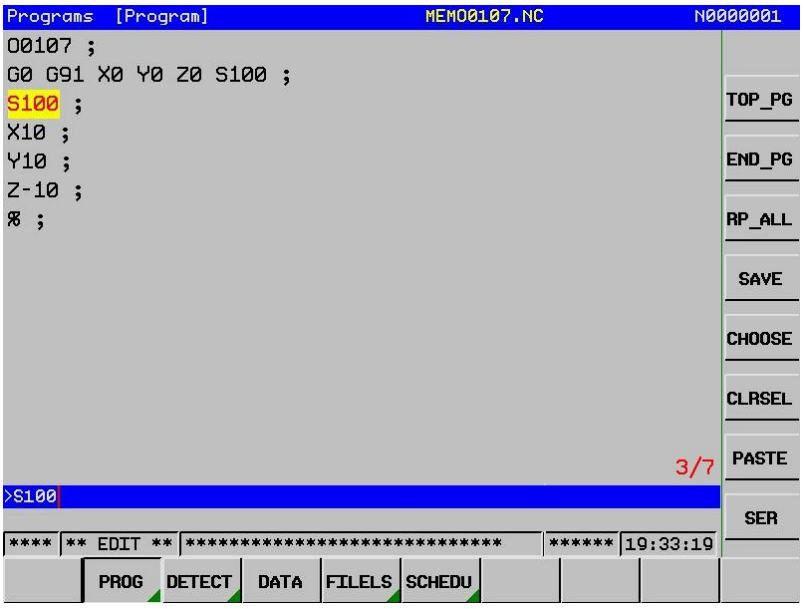


Figure 3-2-33

Now, if the **SER** sub-soft key is continuously pressed, the cursor will continue to search for the next object code, which can be executed until the end of the program and can return to the beginning of the program to search in a loop, without requiring the user to enter the same object code again.

Note: The system will automatically retain the search record of the last input to facilitate users to achieve continuous search.

(2) After finding the object code, enter the content that the user wants to replace, and then press the ALTER key on the panel, and replace the content at the cursor with the new content. For example: enter S200, then press the ALTER key on the panel, the content at the cursor is replaced with S200, as shown in Figures 3-2-34 and 3-2-35.

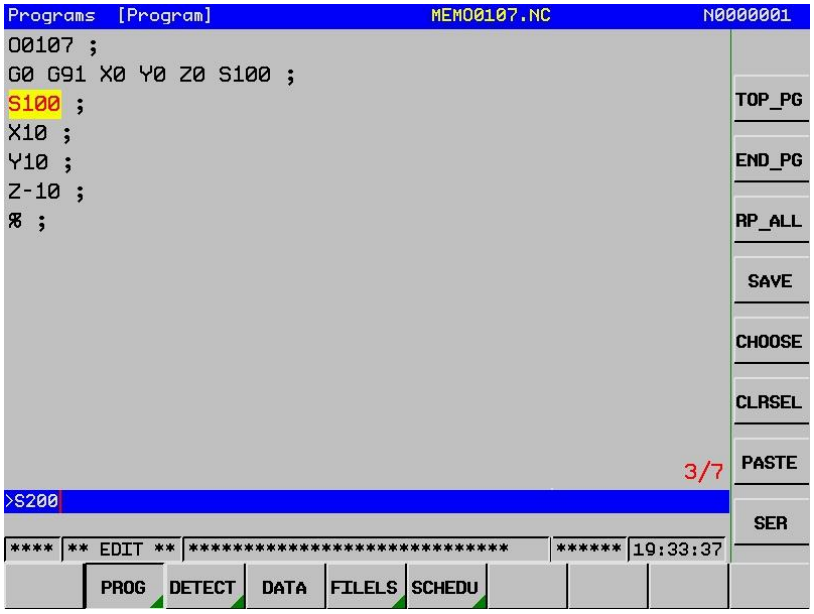


Figure 3-2-34

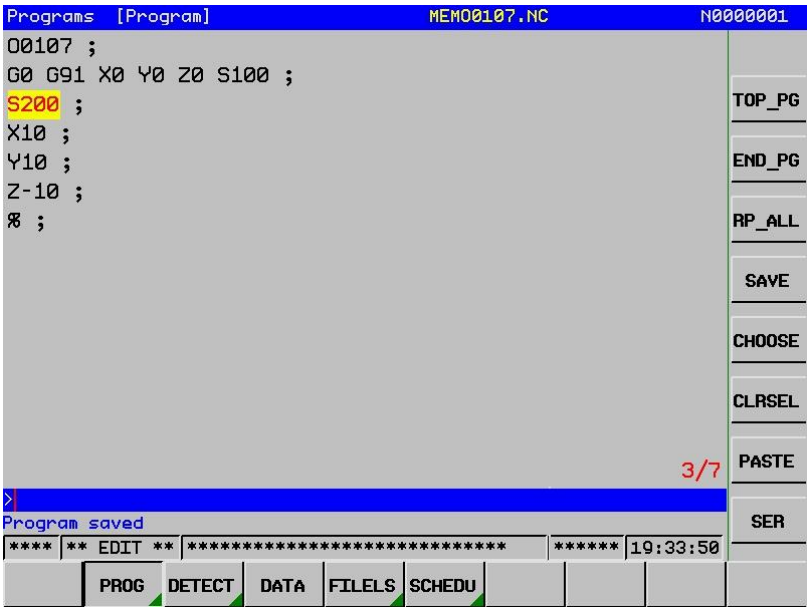


Figure 3-2-35

(3)If all of the same characters or addresses are replaced in the program, the following should be done:

Press the ⇧ , ⇩ cursor keys (or follow the search method described earlier) to find the content to be replaced, and enter the replaced content in the input field. Press the soft key RP_ALL , and the system will replace all the contents of the program that are the same as those replaced. For example: first move the cursor to one S100 and enter S300 in the input field to replace all S100 in the program O0107 with S300, as shown in Figure 3-2-26.

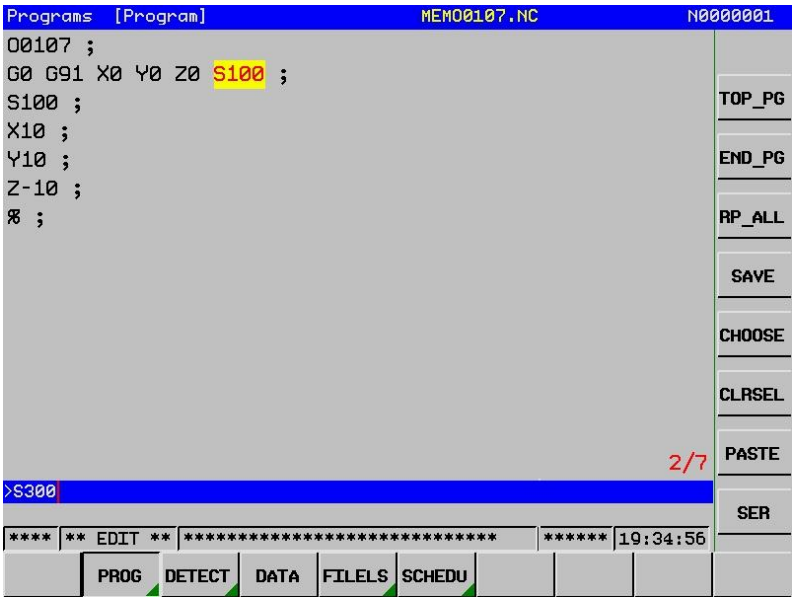


Figure 3-2-36

Press the RP_ALL sub-soft key again, and all S100 in the program will be replaced with S300, as shown in Figure 3-2-37.



Figure 3-2-37

10. Operation steps of deletion procedures:

Programs stored in memory can be deleted program by program or multiple programs can be deleted at the same time.

Note: At present, the system deletes multiple programs at the same time, which is limited to programs starting with the letter O.

(1) Operation steps for deleting one procedure:

1) Rotate the  , select the editing mode ➔  ➔  , and enter the [File List] interface (see Figure 3-2-38).

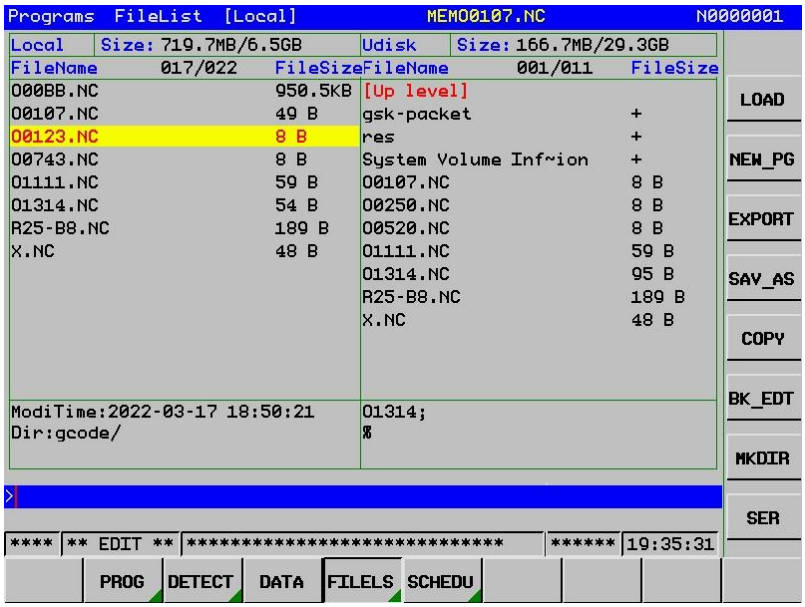


Figure 3-2-38

2) Press the  ,  page turning keys or the  ,  cursor keys, and select the program to be deleted. As shown in Figure 3-2-38, select O0123.NC.

3) Press the 

key on the panel, then it reminds that "Do you want to delete the file", press the [OK] sub-soft key on the right side of the interface to delete the currently selected file (see Figure 3-2-39).

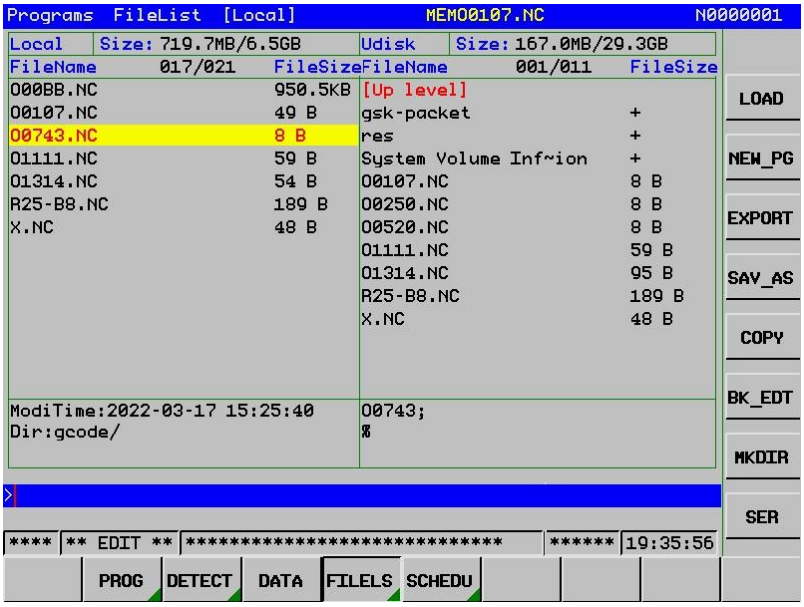


Figure 3-2-39

(2) Operation steps for deleting multiple programs:

After entering the [File List] interface according to the operation steps of deleting a program mentioned above, when all programs of O0740~O0743 need to be deleted, enter O0740-0743 or O740-743 in the input field (where the leading zero of the numeric part of the program name may be omitted), as shown in Figure 3-2-40.

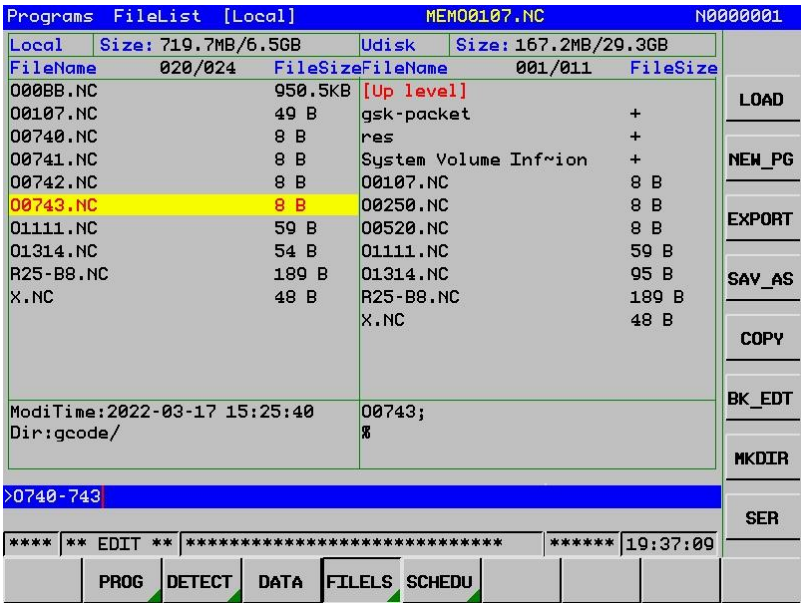


Figure 3-2-40

Press the 

key on the panel, then it reminds that "Do you want to delete the file", press the

OK soft key, and the system will delete all the programs in the range of O0740-0743, as shown in Figure 3-2-41.

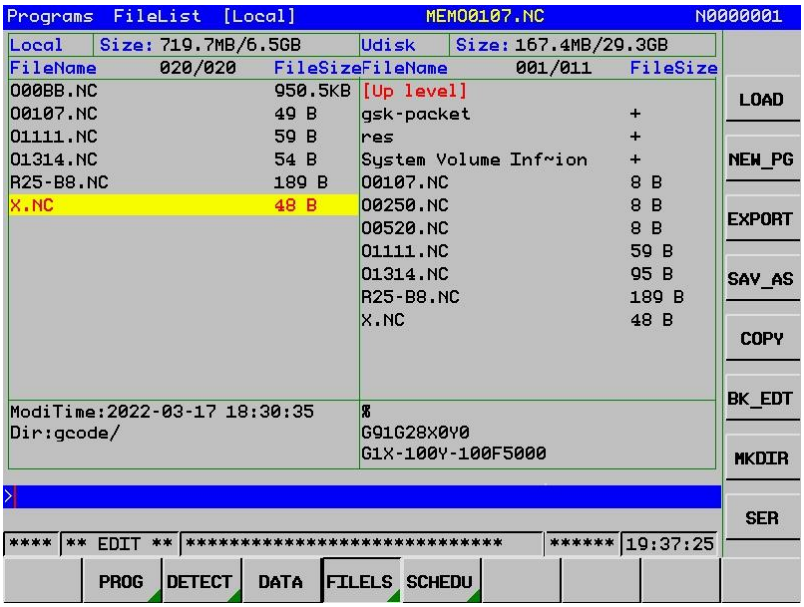


Figure 3-2-41

Note:

(1) In order to delete all the programs of O8000~O8999, you only need to enter O-8000 to delete them all. (But you can't delete files with the same name as the foreground load)

(2) When parameter #1610.0 and parameter #1610.4 are 1, the program of 8000-9999 cannot be deleted, even if the program has been loaded, but the parameters are still 1 and cannot be deleted. For detailed instructions, please refer to "4.6 Display edit parameters (1600~1799)" in Chapter 4 of Part 3 of the *Installation and Commissioning Manual of GSK25iMc/GSK25iTc Series Bus-Type Milling/Turning Machining Center CNC System*.

3.2.6 MDI input display

Rotate the  , select the input mode ➔  ➔, and enter the MDI [MDI] display interface (see Figure 3-2-42). On this interface, you can input and execute a single command or a single- or multi-block program, and the program format is the same as that of editing program. MDI is suitable for single command or short program block operation.

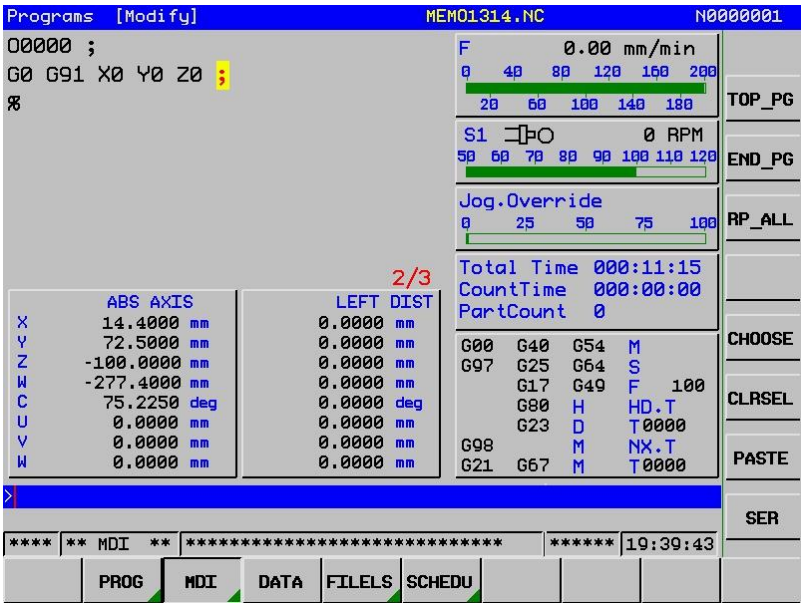


Figure 3-2-42

The main points of operation of MDI are as follows:

(1) Rotate the  to select the input mode.

(2) Press the  button on the editing panel and press the soft key MDI to enter the MDI interface.

After entering, the system will automatically add the program number O0000.

(3) Input the command program to be executed and move the cursor to the line segment to start running, click the  to run the edited program. By programming in MDI mode, you can insert, modify, delete, search for line numbers, and so on. (Specific operation refers to the introduction of editing interface)

(4) To stop or end the MDI operations halfway, please follow the following steps:

a. Stop the MDI operation

Press the feed hold switch on the operation panel. The feed hold indicator light is on and the cycle start indicator light is off. The machine responds as follows:

(i) When the machine is in motion, the feed operation decelerates and stops.

(ii) Stop the dwell when the machine is performing a dwell.

(iii) When the M, S or T command is executed, the operation stops after the M, S and T are executed.

When the Cycle Start button on the operation panel is pressed again, the system continues to perform subsequent actions.

b. Finish the MDI operation

Press the  key on the editing keyboard area to end autorun, and enter the reset state. When a reset command is executed during machine movement, the motion decelerates and stops.

Note:

1. Programs programmed in MDI mode cannot be stored.
2. It is not allowed to modify parameters during operation in MDI mode.
3. To completely delete a program written in MDI mode, refer to the operation of the editing interface.
4. In MDI mode, you can input and execute a single command, one-line or multi-line program block. After inputting a multi-line program block, you must move the cursor to the program block where the execution starts before the cycle starts. Otherwise, only the line where the cursor is located, and the subsequent program block will be executed, the preceding ones are ignored.

3.2.7 Detection interface

Rotate the  to select the editing mode. Press the DETECT soft key to enter the detection interface in the [Edit] mode. In the detection interface, the running process of the whole code, absolute position coordinates and remaining distance, spindle speed, feed speed, tool number, mode and other information can be viewed in a real-time manner. On this interface, the corresponding buttons on the operation panel can also be used to modify various overrides, as shown in Figure 3-2-43.

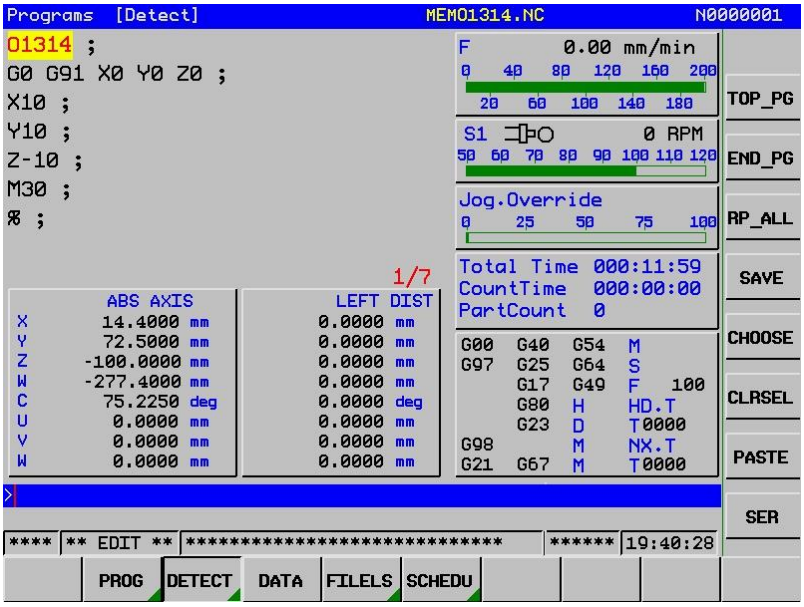


Figure 3-2-43

3.2.8 Data display

Press the  ➔ DATA soft key to enter the data display interface and display the command value of the program block being executed, the next command value and the current modal value, as shown in Figure 3-2-44.

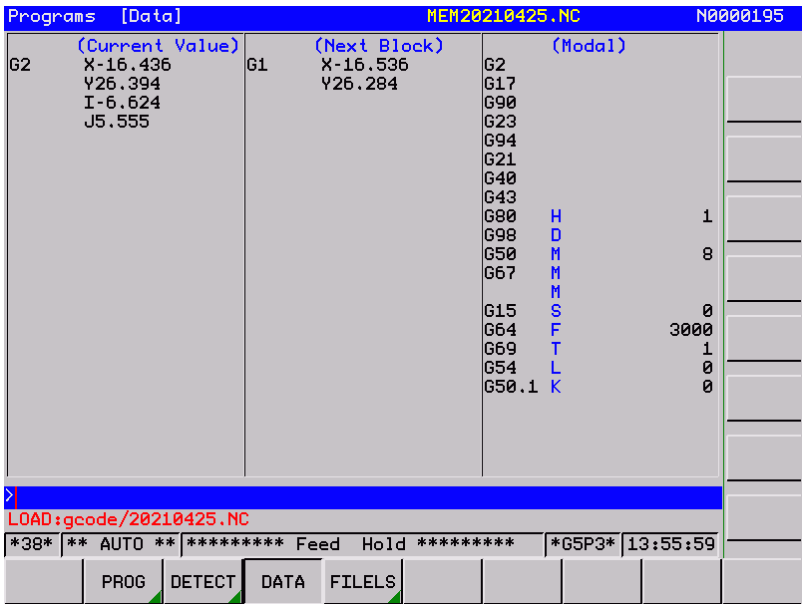


Figure 3-2-44

3.2.9 Introduction to background editing

Background editing refers to loading the program into the background of the system for editing, which can edit the program in any way, while the front-end editing can only edit the program in the editing mode or enter/MDI mode. In background editing, other processing programs can be edited in the process of executing the processing program, and the same editing operation as the front editing can be carried out.

(1) Background editing can be done in the automatic, enter/MDI, DNC or editing modes.

(2)The number of programs that can be edited in the background is limited to 1, and the current loaded program cannot be edited in the background. The sum of the size of the program edited in the background and the size of the program edited in the front cannot be greater than 8M.

Operation steps for background editing:

1) Rotate the  to select one of Automatic, Edit, Enter or DNC modes.

2) The 

➔ FILELS can go to the following operation interface (see Figure 3-2-45).

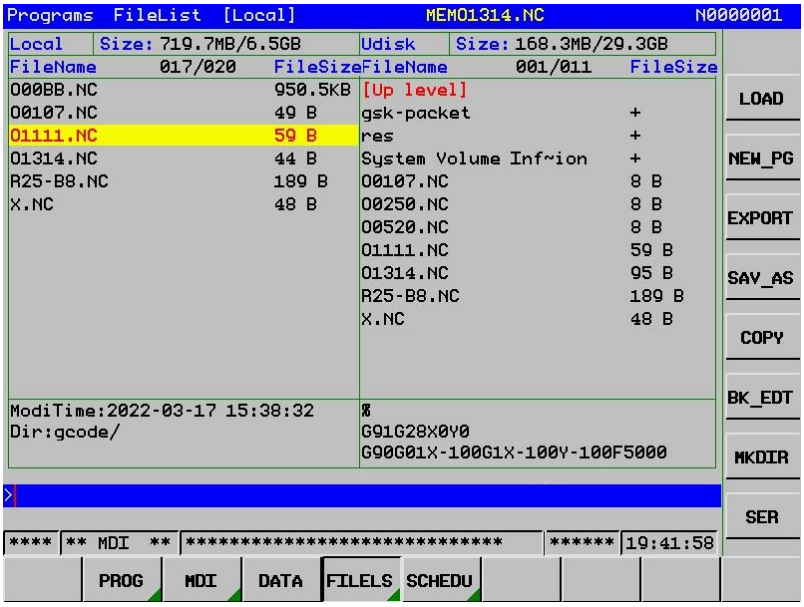


Figure 3-2-45

3) Press the page turning keys  ,  or the cursor keys  ,  to select the program to be edited, and then press the BK_EDT sub-soft key to enter the background editing screen as shown in Figure 3-2-46.

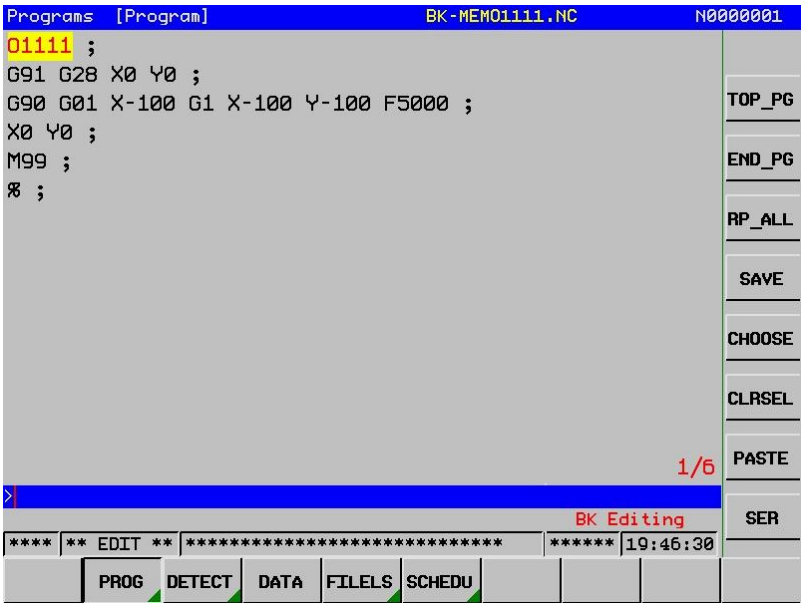


Figure 3-2-46

4) For program editing, input programs in the input field such as: G90G01X-100, press the 

key on the panel to complete the editing (see Figure 3-2-47).

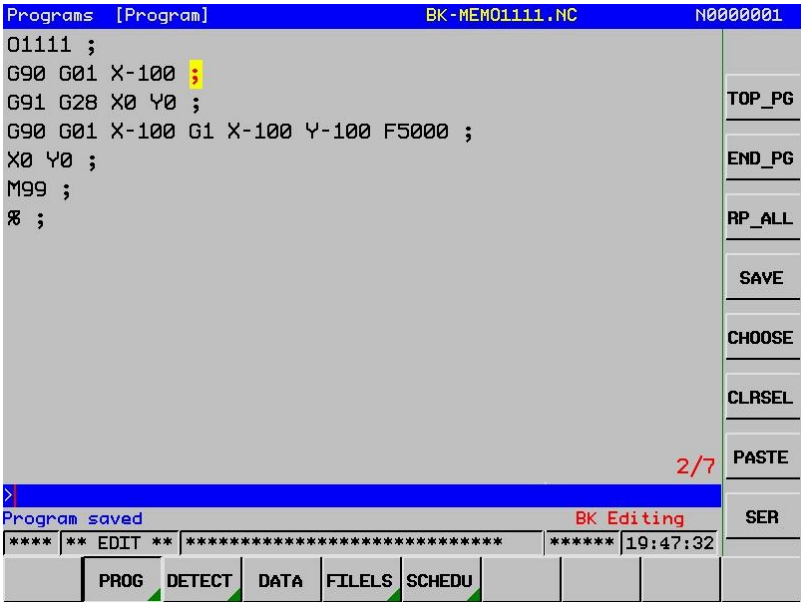


Figure 3-2-47

5) Press the soft key 

to enter the following interface (see Figure 3-2-48).

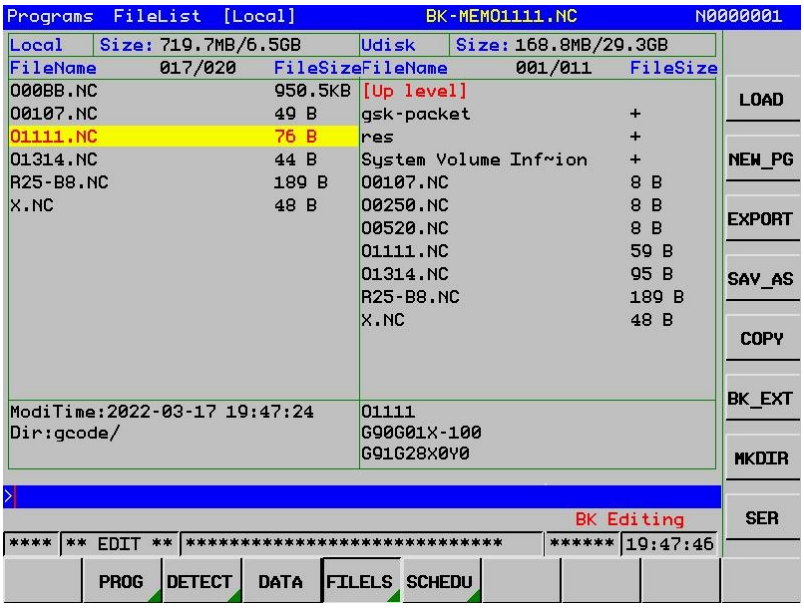


Figure 3-2-48

6) Press the BK_EXT sub-soft key, the background editing is completed, the edited program will be automatically saved, and the interface shown in Figure 3-2-49 will be displayed after exiting.

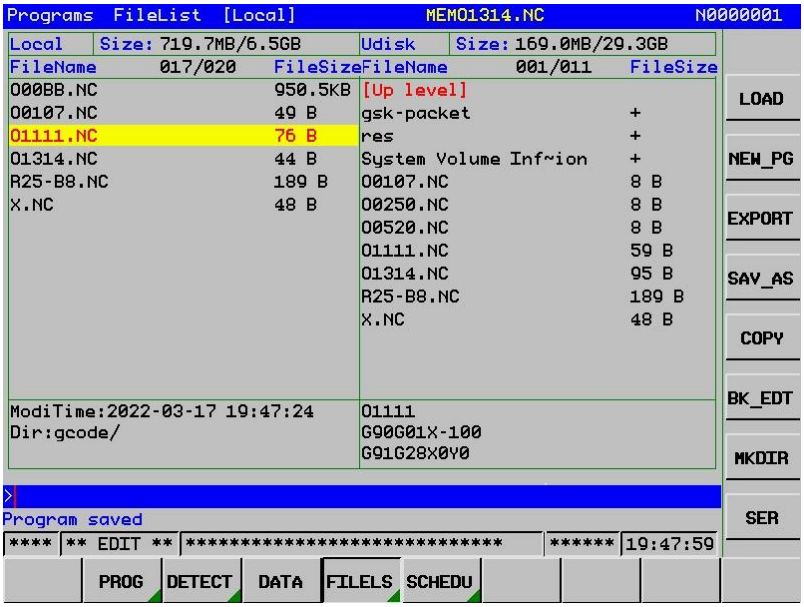


Figure 3-2-49

Note: In the background editing, the editing program is the same as the front-end editing program.

3.2.10 Program restart

Function and purpose

This function is mainly used for accidents during program running, such as tool breakage, system power off and restart, etc. After the accident is eliminated, the system returns to the breakpoint through the program restart function to continue executing the program.

There are two types of program restart: P type and Q type.

P type indicates that the program is started by the program block number specified by the operator.

The Q type automatically starts the program from the program block number before the last power failure,

and is valid when the system is restarted after the power is turned off.

Operation steps

The specific steps are as follows:

1. Confirm that the current program file is to be processed or that it was processed during the last power cut.
2. Confirm that it is currently in the automatic mode.
3. Press the  key on the machine operation panel.
4. Enter the number of running lines you want to restart, press the reset key, and then press the P_TYPE sub-soft key to automatically jump to the program block you desire. Pressing the Q_TYPE sub-soft key does not need to input the number of running lines, because the system will automatically retrieve the number of running lines when the power was off last time. If the processed files are inconsistent with those before power failure, the system will start running from the beginning of the program after pressing the Q_TYPE sub-soft key directly.
5. Press the  key, the tool first moves to the starting position at the dry run speed of G01 (controlled by the feed override), and then runs according to the program content. Attention should be paid here to confirm whether the tool may encounter a workpiece or other object on its way to the restart position. If this is possible, move the tool manually to a position where it will not touch the workpiece.

Precautions

1. Whether the M, S, T and B functions are to be executed will be determined according to the actual processing needs, which is set by parameter #1950.7.
2. The return order of each axis is determined according to the actual machining needs, and set by parameter #1960.
3. Please follow the instructions carefully and do not attempt unconventional operations, otherwise there will be unpredictable danger.

3.2.11 Scheduling

Function and purpose

The job scheduling function allows you to select files in the local directory or USB flash drive directory in the [Edit] mode, specify their execution order and the number of repeats, and execute them in [DNC] mode (the scheduling function enable parameter is #1602.2).

Operation steps

1. Go to the [Scheduling] interface (see Figure 3-2-50).



Select the [Edit] method



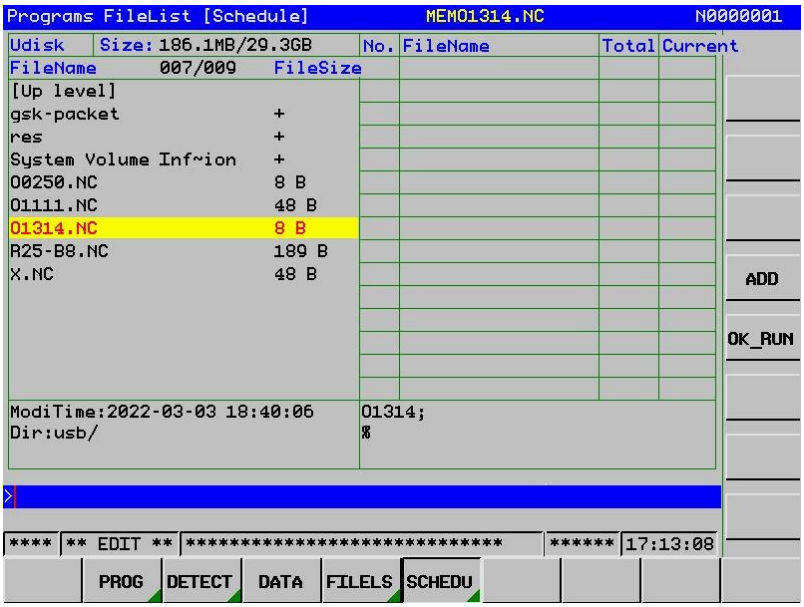


Figure 3-2-50

2. Scheduling file operations

(1) Move the yellow cursor to the right window, when the USB flash drive is loaded into the system, the file list in the right window can be passed and switched between the local file list and the USB flash drive file list through  ,  , as shown in Figure 3-2-51.

(2) Select the *.NC file as needed, and press  to add the scheduling files one by one.

(3) When the yellow cursor is moved to the left side, the scheduling files to be added can be arranged, and the file may be moved  or  .

(4) When moving the cursor to the data box of the number of repeats, you can enter a number and press the  key to modify the repeat execution times of the program.

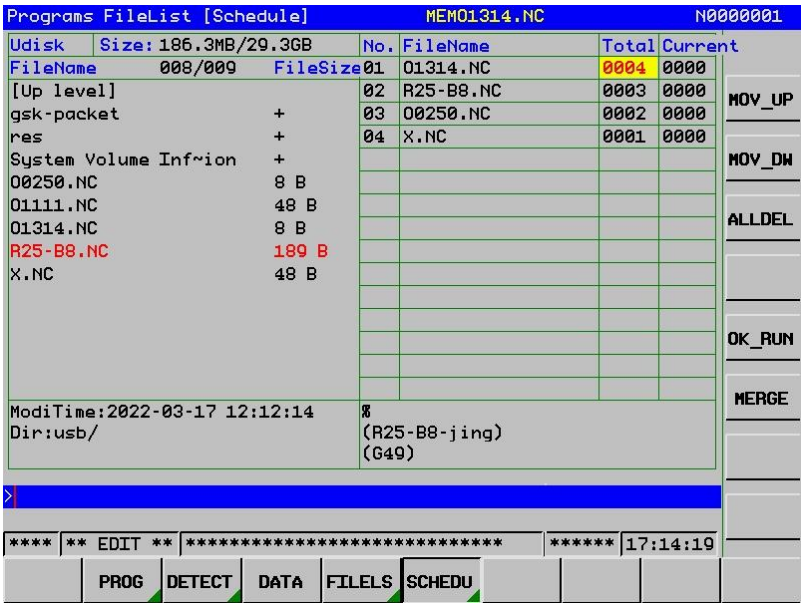


Figure 3-2-51

(5) Delete a single scheduling file, move the cursor to the file and press  to delete.

(6) Delete all scheduling files by clicking  .

3. Execution of scheduling file

The schedule file can only be executed if the PLC signal G43.5 is switched on.

Rotate the  to select the DNC mode ➔ The PLC signal G43.5 is switched on (subject to the PLC specified key of the machine manufacturer) ➔ OK_RUN ➔ 

3.3 Offset Interface

Press the 

key to enter the offset interface, in which there are five interfaces: [Offset], [Workpiece System], [Additional Workpiece System], [Macro Variable], and [Coordinate Centering]. View or modify it with the corresponding soft keys, or press 

to switch between the various interfaces.

3.3.1 [Offset] interface operation instructions

Press the function key 

➔ M offse to enter the offset interface (see Figure 3-3-1).

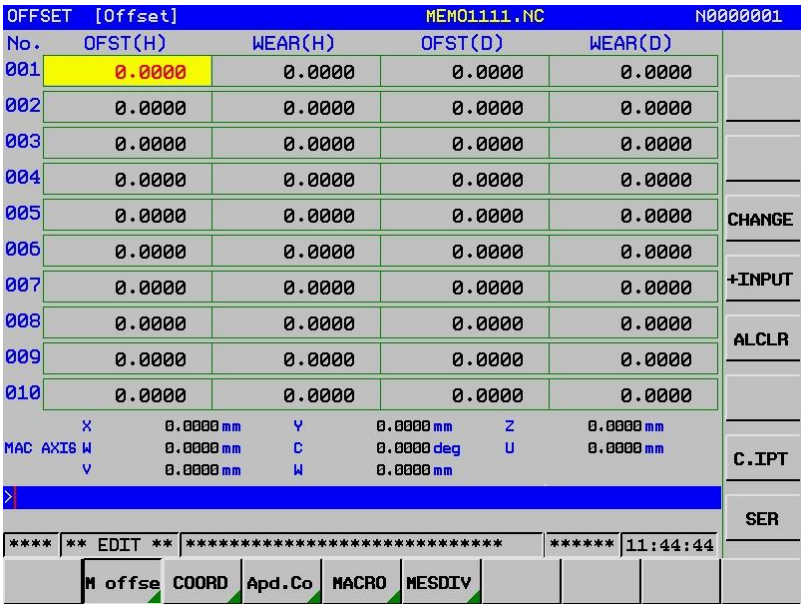


Figure 3-3-1

Move the cursor to the place where you want to set and change the compensation value by the page-turning keys and cursor keys, or enter the compensation number and press the sub-soft key SER to find the compensation number.

Set the compensation value: Enter a value and press key 

on the panel to set the compensation value. +INPUT can increase or decrease the tool compensation value. If the current X is 102.1, change to 100. It can be realized by two methods: a) write 100 directly, and then press the panel key 

to Input; b) write -2.1, and

then press the sub-soft key +INPUT . CHANGE can switch the machine coordinates, relative coordinates and absolute coordinates to make it easy for users to view the coordinates in time. The sub-soft key C.IPT will calculate the tool offset value based on the current machine coordinate, workpiece coordinate system and entered value.

3.3.2 [Workpiece System] interface operation instructions

In the operation mode of <Enter> or <Edit>, press the soft key COORD to enter the workpiece interface (see Figure 3-3-2).

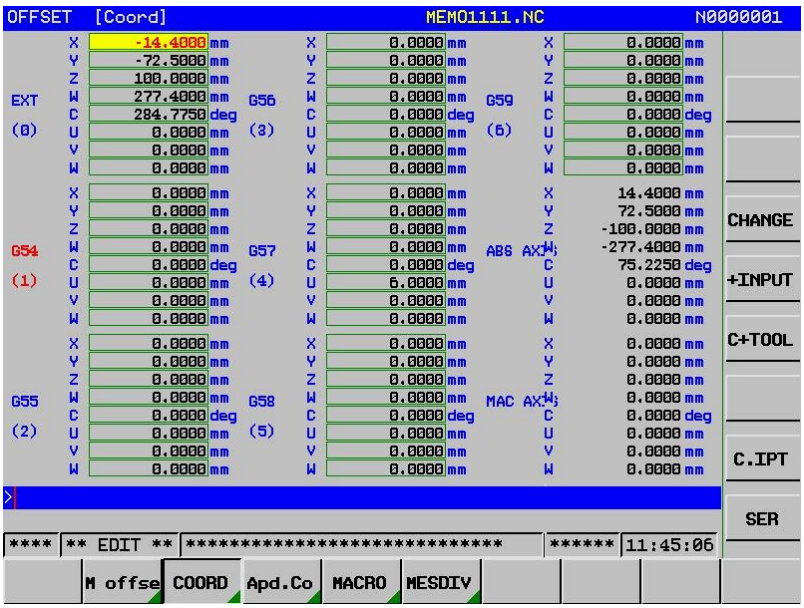


Figure 3-3-2

CHANGE : Users can switch between absolute coordinates and relative coordinate display by using this function for ease of viewing.

EXT. is a basic offset. Users can set up by using the DATA INPUT key or the +INPUT soft key. The operations of G54-G59 are similar.

C.IPT : The measurement result will be calculated according to the coordinate system chain.

Users can search according to the number in parentheses in each workpiece coordinate system. As shown in Figure 3-3-2, for G57 (4), the user can enter 4 directly and press the SER sub-soft key, the cursor will automatically jump to the content you are searching for. When searching for the additional workpiece coordinate system, add a P must be added in front (the input range of workpiece coordinate system is 0～6, and the additional workpiece coordinate system is P1~P48).

When pressing the CHANGE sub-soft key, the user can freely switch the absolute coordinates and relative coordinates display for ease of viewing.

3.3.3 [Additional Workpiece System] interface operation instructions

Press the soft key Apd.Co to enter the additional workpiece system interface (see Figure 3-3-3).

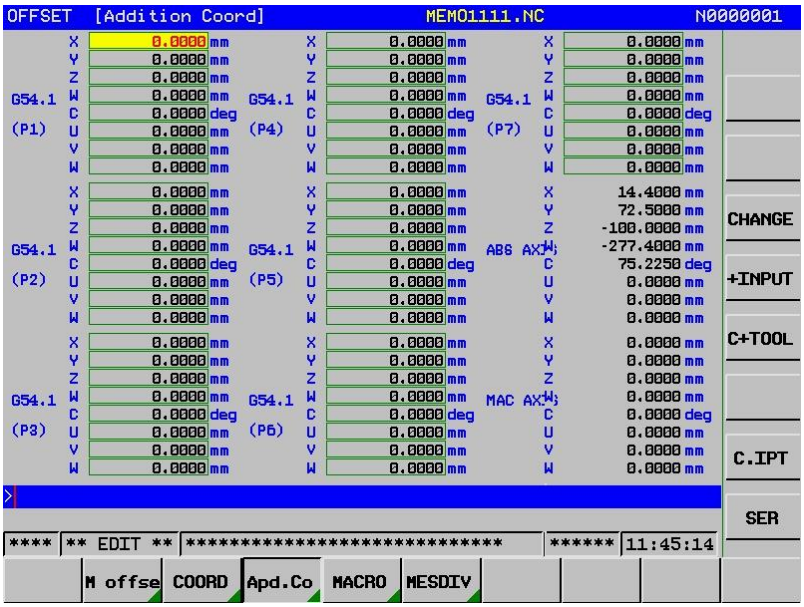


Figure 3-3-3

3.3.4 [Macro Variable] interface operation instructions

Press the soft key MACRO to enter the macro variable interface (see Figure 3-3-4).

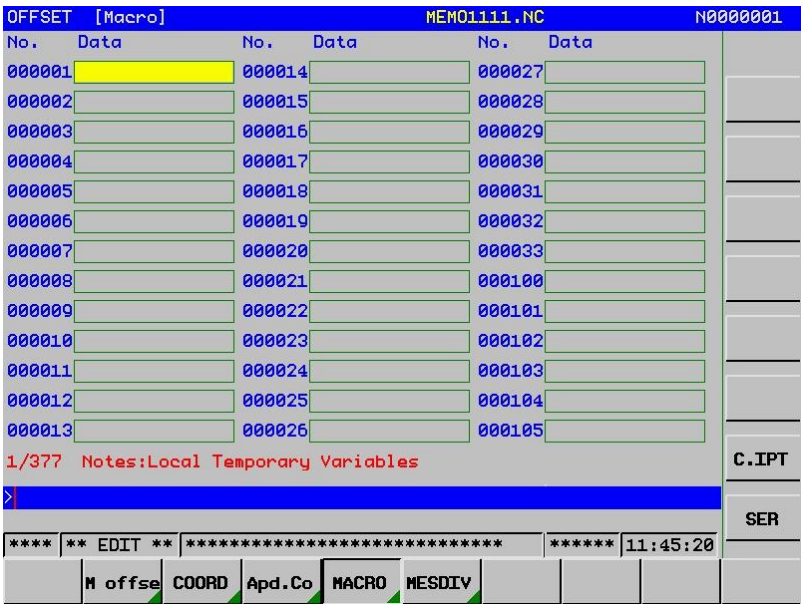


Figure 3-3-4

C.IPT : Enter the axis address (such as X, Y or Z) in the input field, and click the C.IPT soft key. The system effectively inputs the corresponding machine coordinate value to the macro variable.

3.3.5 [Coordinate Centering] interface operation instructions

Press the soft key MESDIV to enter the coordinate centering interface (see Figure 3-3-5).

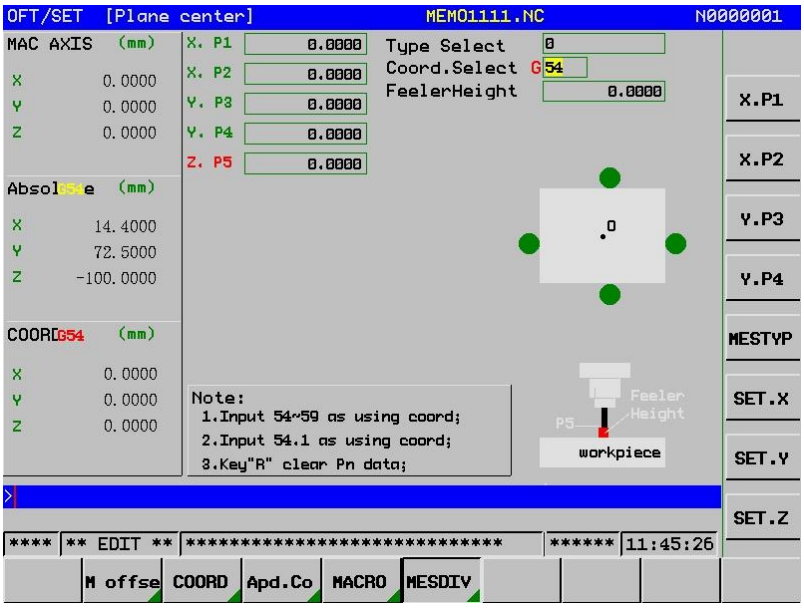


Figure 3-3-5

Enter the [Coordinate Centering] interface, press the sub-soft key [Centering Type], according to the shape of the workpiece, determine whether to find the center point of the workpiece by two-point centering or three-point circle setting.

3.4 System interface

Press the 

key to enter the display page (see Figure 3-4-1), there are eight display pages: [System Parameters], [Parameter Classification], [Diagnosis], [Wiring Diagnosis], [DB List], [File Management], [Screw Compensation] and [FTP] can be switched by the corresponding soft keys. If there is a soft key with the extension mark [] in the lower right corner of the soft key, such as 

soft key, the corresponding sub-soft key will appear on the right side of the interface. The sub-soft key displayed on the right side of the system parameter interface is [Search]. For soft keys without extended symbols, the right side of the interface is blank.

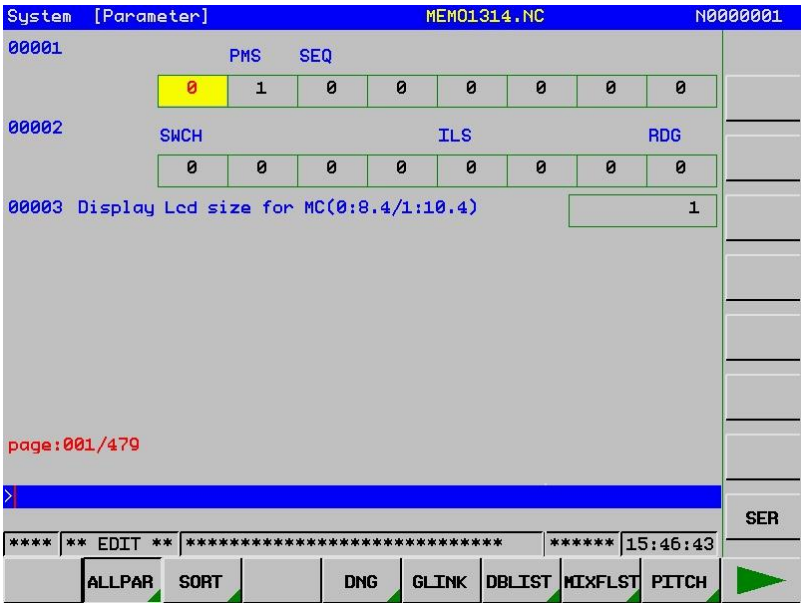


Figure 3-4-1

3.4.1 [System Parameters] display interface

Press the ALLPAR soft key to enter the parameter display interface. The steps of parameter display and settings are detailed as follows:

(1) First press the SETTING key on the edit panel, go to the [Log In] setting interface, enter the corresponding password in the "Installation and Debugging Password" or higher permission to log in successfully;

(2) Press the SYS SYSTEM key on the editing panel to enter the system interface;

(3) Press the soft key ALLPAR to display the parameter interface;

(4) Use the following method to move the cursor to the parameter number to be written or displayed. The specific method is detailed as follows:

a) Enter the parameter number and press the [Search] sub-soft key.

b) Use the page up and down keys and , and direction keys, , , , to move the cursor to the parameter number.

(5) Enter a value in the input field and press the DATA INPUT key on the bottom panel to set the parameters.

3.4.2 [Parameter Classification] display interface

Press the soft key [Parameter Classification] to enter the parameter classification display interface (see Figure 3-4-2). The [Parameter Classification] page mainly classifies the parameters that are often used in system debugging.

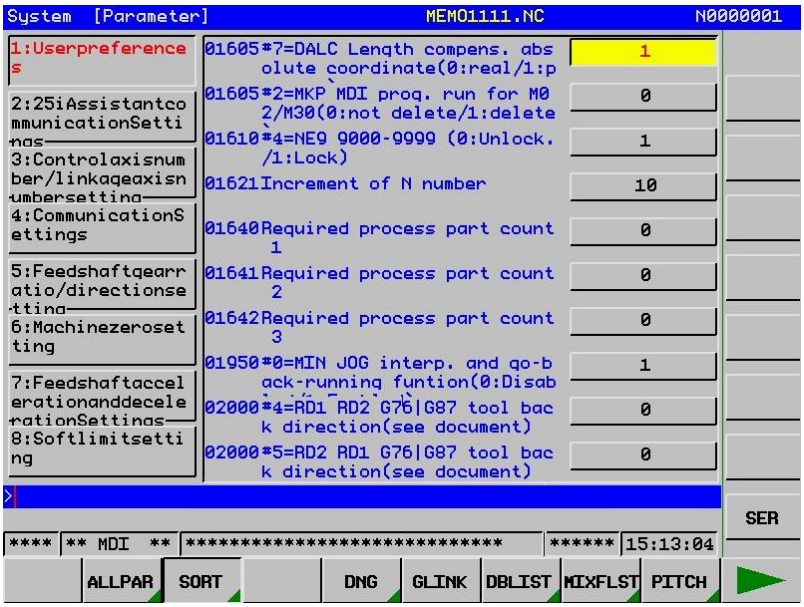


Figure 3-4-2

3.4.3 [Diagnosis] display interface

Press the DNG soft key, input the diagnosis number in the input field, and press the [Search] subsoft key to directly search for the required diagnosis number (see Figure 3-4-3).

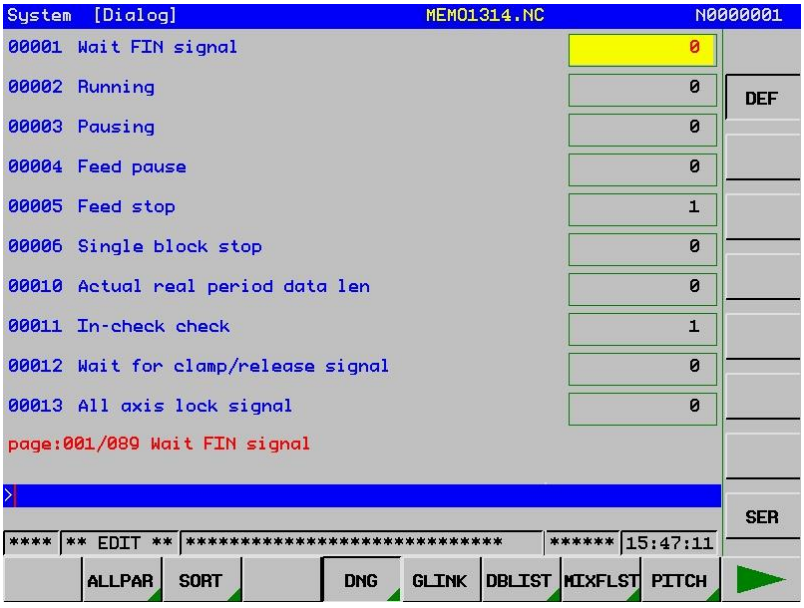


Figure 3-4-3

Note: The three sub-soft keys [NC], [H2N] and [N2H] are not open to users and only for internal use.

3.4.4 [Wiring Diagnosis] display interface

Press the GLINK soft key to enter the wiring diagnosis display interface (see Figure 3-4-4).

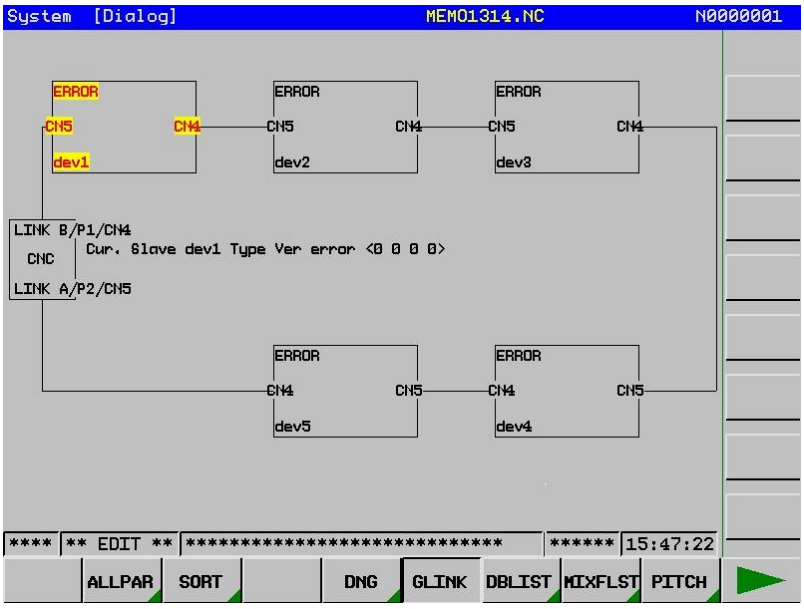


Figure 3-4-4

When the hardware device is abnormally connected, the line connection between all devices will be disconnected, and an alarm message will be prompted: abnormal hardware connection, disconnection or reverse connection of the bus, as shown in Figure 3-4-5.

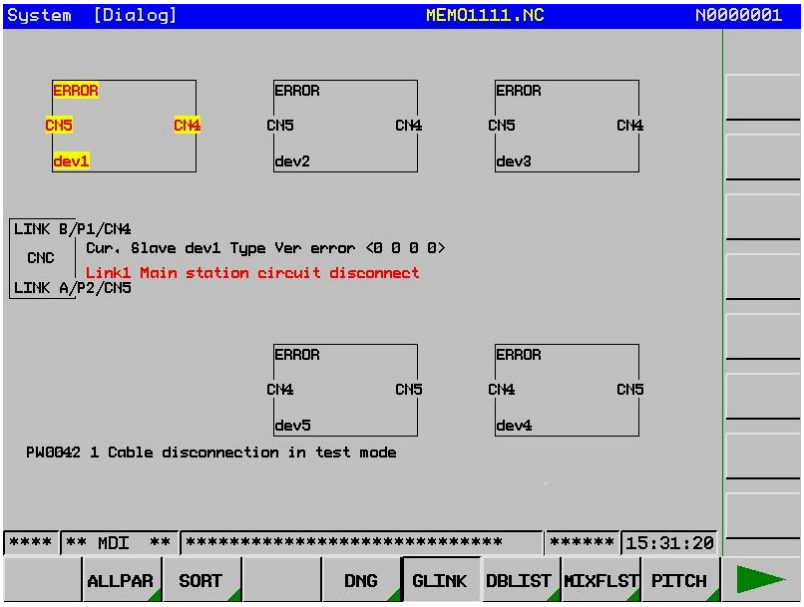


Figure 3-4-5

3.4.5 [DB List] display interface

Press the DBLIST soft key to enter the display interface (see Figure 3-4-6). When the login authority of the system lies at the machine manufacturer's password or above, the cursor is displayed in the local directory. The right side of the interface displays seven sub-soft keys: [Local Backup], [Local Recovery], [Export File], [One-Key Backup], [Configuration Output], [Export All], and [Restore to Factory]. When the cursor moves to [Previous Level] or the file name of the folder type, the sub-soft key [Open Directory] will also be displayed.

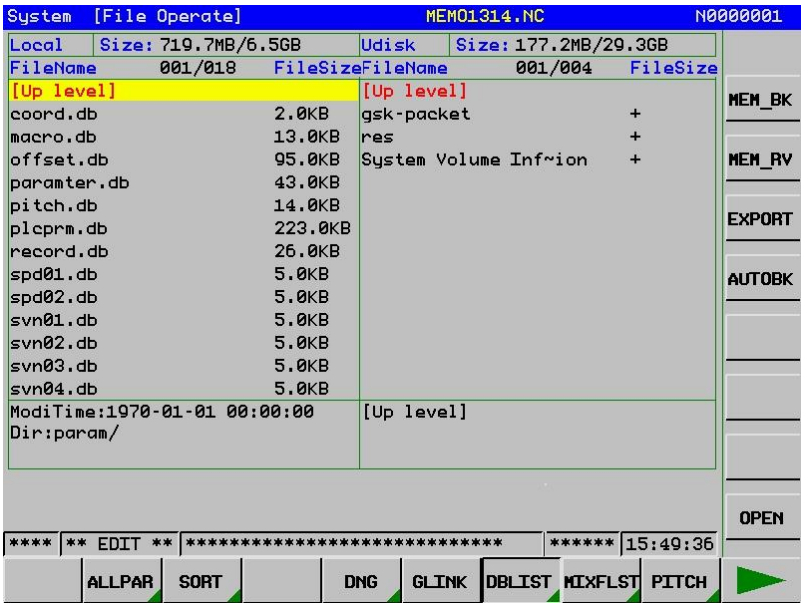


Figure 3-4-6

When the system recognizes the USB flash drive and the loading is successful, the cursor moves to the [Previous Level] position in the file display area of the USB flash drive by default, and switches from the sub-soft key [Restore to Factory] to the sub-soft key [Import File], and the [One-key Recovery] sub-soft key will also be displayed (see Figure 3-4-7).

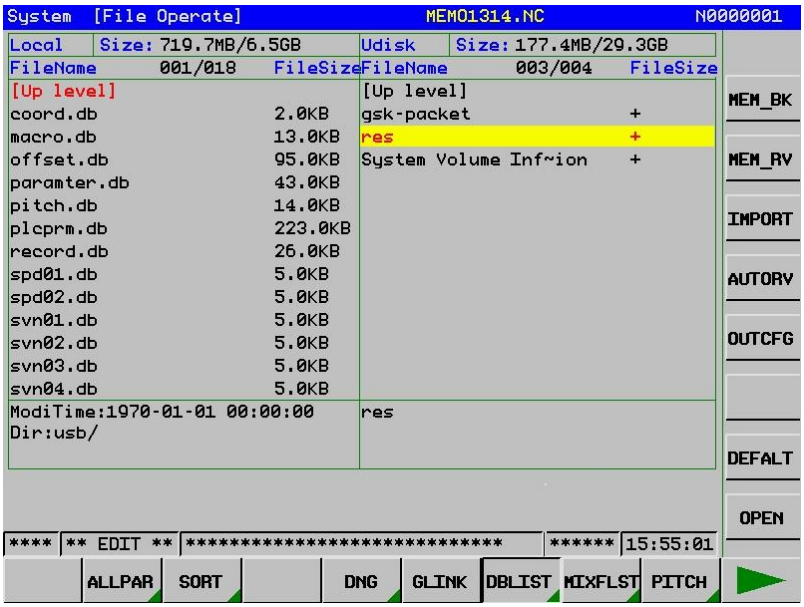


Figure 3-4-7

When the cursor is moved to the local disk file display area, the [One-key Recovery] sub-soft key will be automatically switched to [One-key Backup] sub-soft key (see Figure 3-4-8).

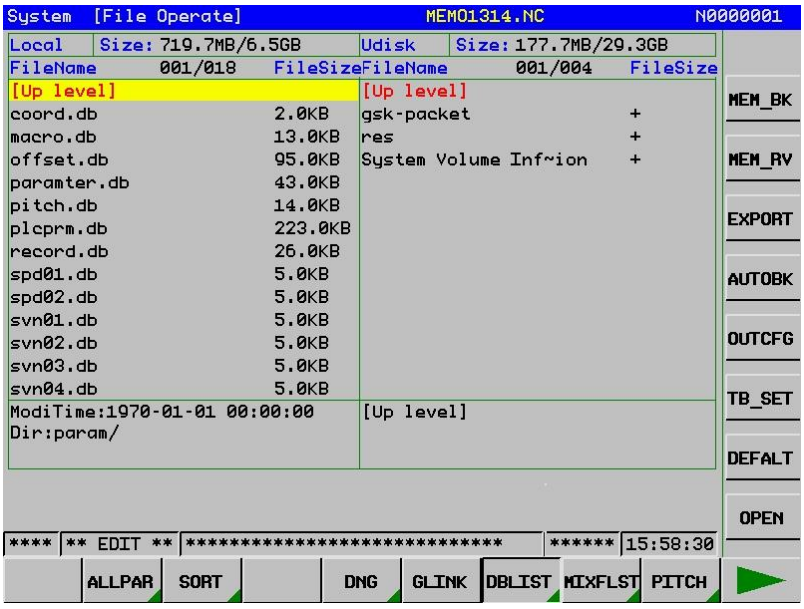


Figure 3-4-8

[Local Backup] sub-soft key function -It can back up the current controller's parameters and provide them to the [Local Recovery] function;

[Local Recovery] sub-soft key function -It can recover the current controller's backup parameters;

[Export File] sub-soft key function -It can export parameter files from a local disk to a USB flash drive;

[Import File] sub-soft key function -It can import parameter files from a USB flash drive to a local disk;

[One-key Recovery] sub-soft key function -It is used to restore the relevant files used by end users from a USB flash drive to the controller after [One-key Backup]; after the parameters are recovered successfully, the controller will display the message of successful recovery in the information field; if the name of the recovered folder is inconsistent with the system number, the controller will prompt the message "The file does not exist";

[One-key Backup] sub-soft key function -It is used to back up the relevant files of end users to the gsk-packet directory under the root directory of USB disk through USB interface and storing all files in folders named by system numbers;

[Export All] sub-soft key function -It is used to export all ".db" files from the controller to a USB disk.

[Restore to Factory] sub-soft key function -It is used to restore the original factory data.

3.4.6 [File Management] display interface

Press the MIXFLST soft key to enter the file list display interface. The right side of the interface displays the [Import] \ [Export] or [Open Directory] sub-soft key (see Figure 3-4-9).

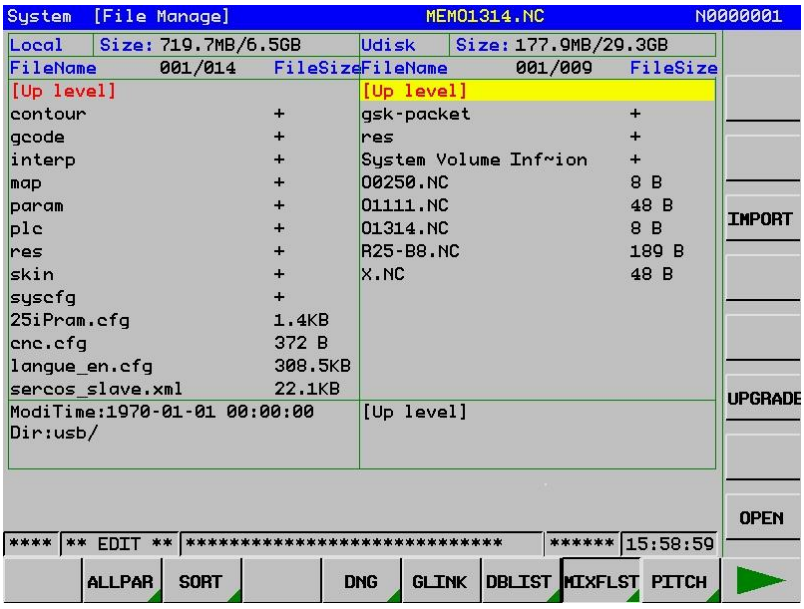


Figure 3-4-9

3.4.7 [Pitch Compensation] display interface

Press the PITCH soft key to enter the pitch compensation display interface (see Figure 3-4-10). The right side of the interface displays the sub-soft keys [Pitch Compensation Display] and [Pitch Compensation Insertion], and you can press the corresponding soft keys to switch between the sub-interfaces.

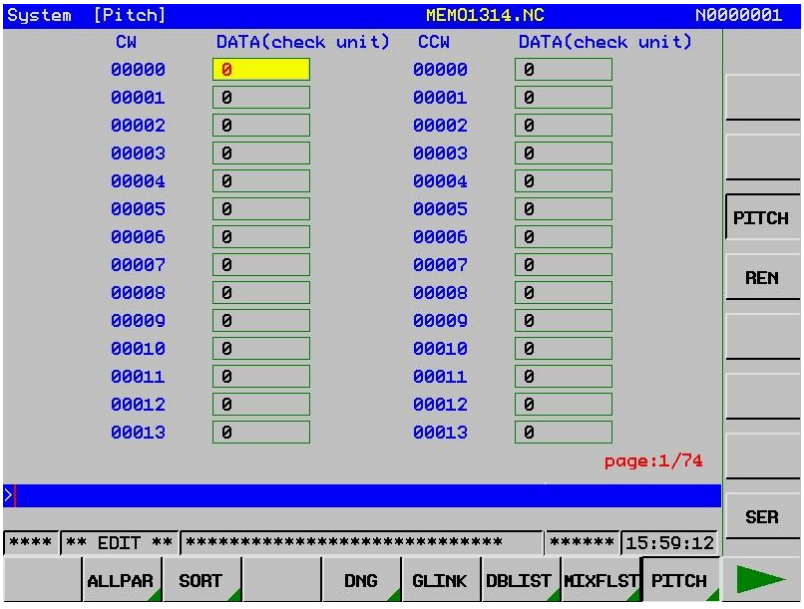


Figure 3-4-10

Press the soft key [Pitch Compensation Import] to enter the pitch compensation import display interface (Figure 3-4-11). You can perform operations related to [Incremental Loading], [Switch Axis], and [Load REN] on this interface.

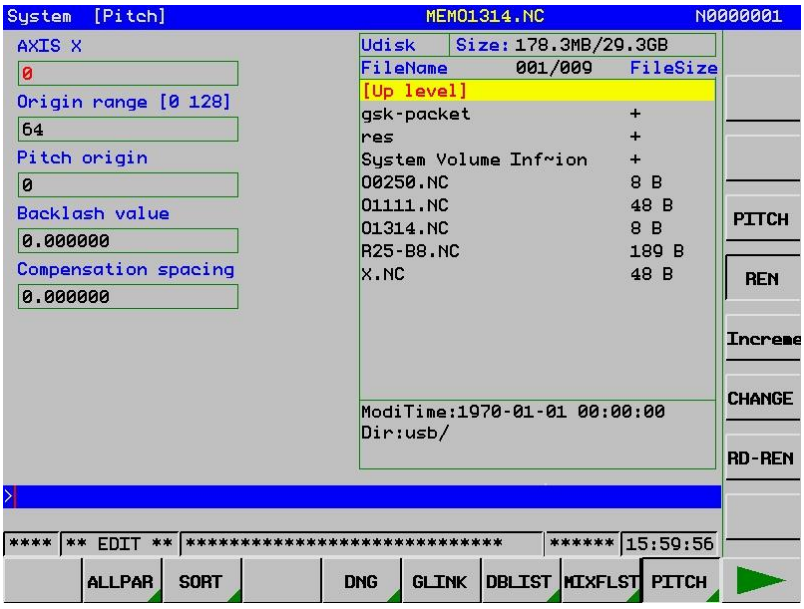


Figure 3-4-11

Setting pitch error compensation data can compensate pitch error of each axis. The unit of compensation value is the testing unit.

The compensation points of each axis are set at a certain distance interval, and the compensation amount is set for each point. The origin of compensation is the reference point of tool return.

The compensation data should be set according to the machine characteristics. Different machines are different in compensation data. Changing these data will reduce the accuracy of machine. In principle, end users cannot change these data.

The pitch compensation parameters to be set are described below. These parameters set the compensation values for each compensation point.

Parameter Number	Description
2800.0	Pitch compensation: 0: not performed 1: proceed
2800.1	Bidirectional pitch error compensation: 1: Valid/0: Invalid
2806	Pitch error compensation value for reference point return
2810	Pitch error compensation point number corresponding to the machine reference position
2811	Farthest pitch error compensation point number on the reverse side when moving in the forward direction
2812	Farthest pitch error compensation point number on the forward side when moving in the forward direction
2813	Ratio of compensation values
2814	Spacing of compensation points
1068	Gyration amount per revolution of gyration axis

3.5 Setting interface

Press the setting button on the editing panel to enter the 

interface, in which there are three display pages: [Login], [User Privileges], and [Date and Time].

3.5.1 [Login] interface operation instructions

In order to prevent malicious modification of the machining program, CNC parameters, etc., the GSK25i system provides the permission setting functions, and the password has 8 levels from high to low, including Level 0 (system high level), Level 1 (system service level), Level 2 (machine manufacturer level), Level 3 (installation and commissioning level), Level 4 (terminal management level), Level 5 (operator level 1), Level 6 (operator level 2) and Level 7 (operator level 3) respectively (see Figure 3.3-10).

Level 0: The highest authority is reserved by the developer. Clear alarm/operation message.

Level 1: Used for system manufacturer service and various data of the system can be modified.

Level 2: Can modify PLC program, PLC annotation editing and pitch compensation; start/stop PLC operation; introduce and outgoing PLC and screw complement files; modify/introduce and issue User Customization Interface Authority.

Level 3: Can modify parameters and PLC resource data; introduce and issue NC parameter and PLC parameter files; can upgrade system, interpolation and position control maintenance software.

Level 4: Can modify the program, tool offset and set various function settings of the interface, workpiece coordinate system values, macro variable values; has authority to modify the programmer password.

Level 5, Level 6, Level 7: End-user administrators authorize the operation authority of the appropriate personnel by means of a bit parameter. Level 7 is the lowest level in the system by default, without the need of entering passwords.

Note: To modify the login password of the interface, first enter the original password to log in to the corresponding user, and then enter the password to be modified at the corresponding place (entered twice). The passwords of lower-level users can also be modified through advanced users, but users at the same level cannot modify passwords to each other.

Press the edit panel 

key ➔ to enter the LOGIN interface (see Figure 3-5-1).

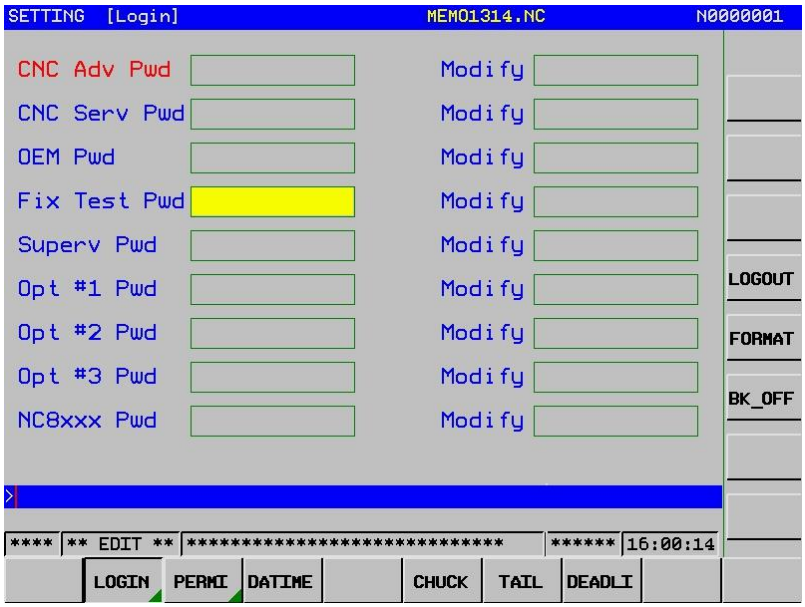


Figure 3-5-1

Enter the password of the corresponding level and press the panel 

key. If it is correct, the system will not give any prompt; if it is incorrect, the system will prompt "Password Error". Press the [Logout] sub-soft key to log out directly, and the password will be logged out immediately (see Figure 3-5-2).

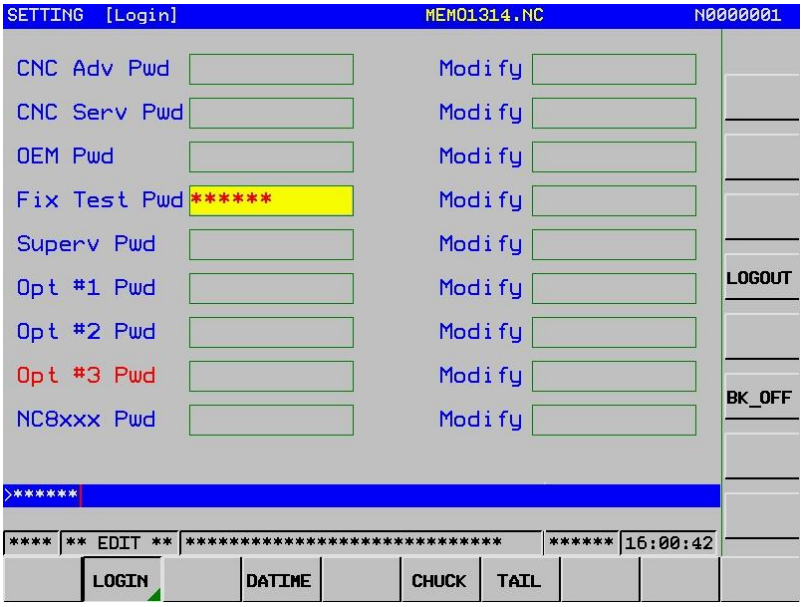


Figure 3-5-2

Note: The password will be cancelled automatically after power failure.

3.5.2 [User Privileges] interface operation instructions

Press the 

key ➔ on the editing panel to enter the PERMI interface (see Figure 3-5-3).

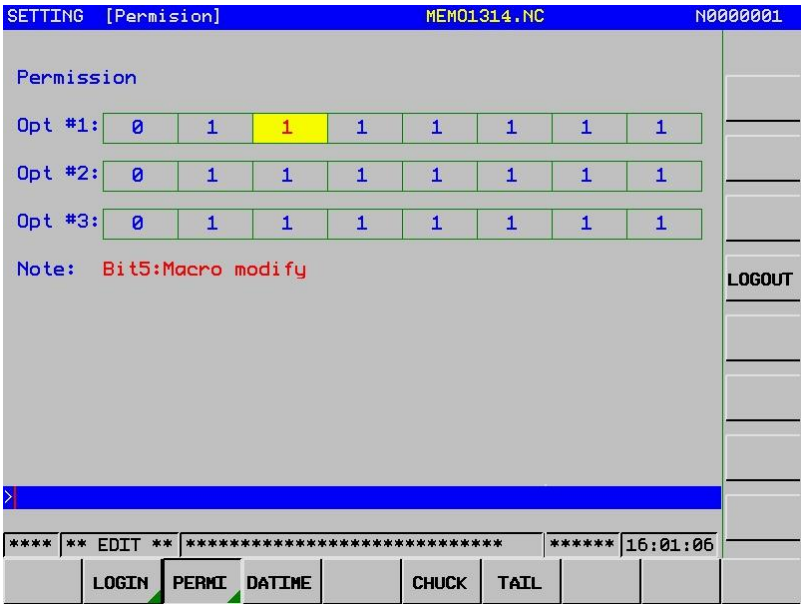


Figure 3-5-3

The definitions of bit parameter authorized by the end user's administrator are presented in the table below:

Bit	Meaning	Notes
0	G-code modification right	1 permitted
1	Tool offset modification permission	1 permitted
2	Wear modification permission	1 permitted
3	Setting modification permission	1 permitted
4	Coordinate system modification permission	1 permitted
5	Macro modification permission	1 permitted
6	Permission to copy G-code to USB flash disk	1 permitted
7	Retained	

3.5.2 [Date and Time] interface display

Press the 

key➔ DATIME on the editing panel to enter the date and time display interface (see Figure 3-5-4).

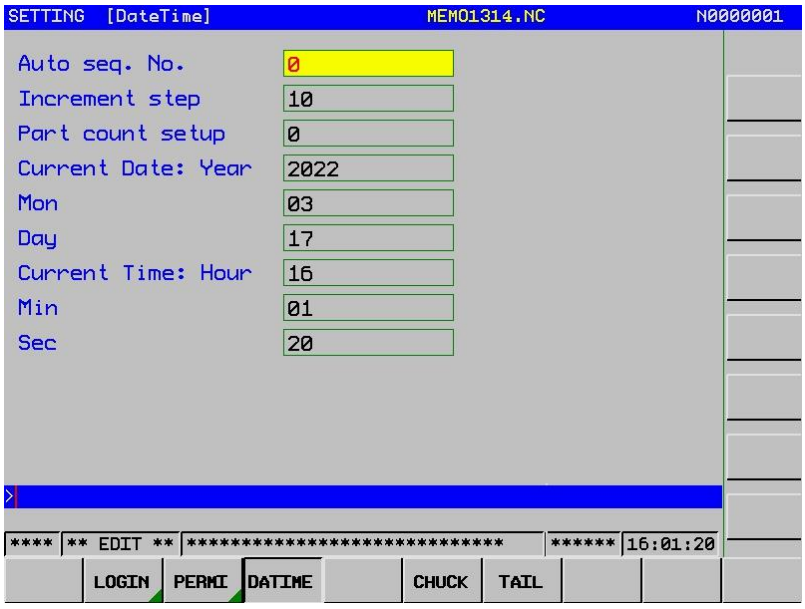


Figure 3-5-4

3.6 Graphic display

The tool path of the program can be drawn on the screen. By observing the path on the screen, the machining process of "graphic display" can be checked. The displayed graphic can be enlarged/reduced. Before drawing, the graphic parameters must be set.

Press the 

key on the editing panel to enter the graphic interface, there are two display pages: [Graphic Parameters] and [Path], which can be switched by the corresponding soft keys.

3.6.1 Graphic parameters

Press the GRAPH soft key to enter the graphical interface, as shown in Figure 3-5-5.

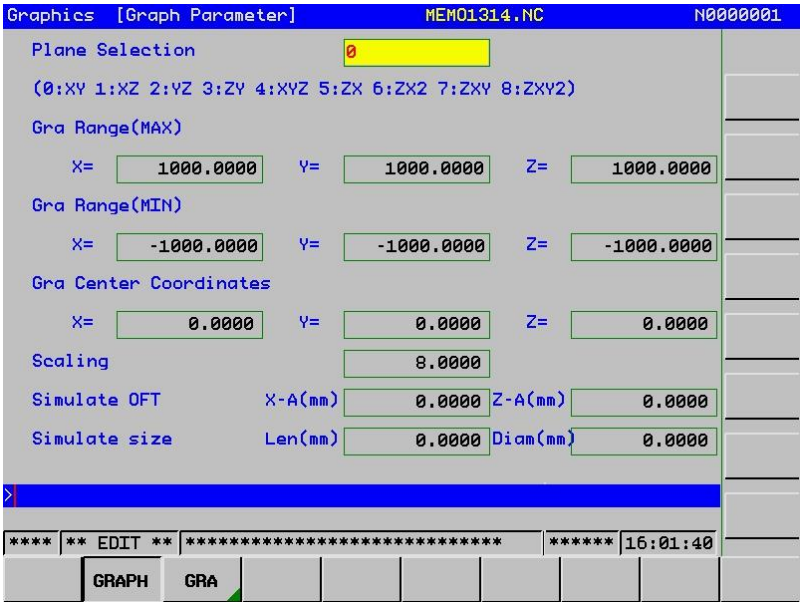


Figure 3-6-1

1. Meanings of graphical parameters

   Plane selection: Set the drawing plane, there are 9 kinds of coordinate plane selection, as shown in the second line.

   Graphic range (Maximum and Minimum): Set the processing trajectory space effectively depicted by the graph.

   Graph Center Coordinate: Set the workpiece coordinate value corresponding to the LCD center in the workpiece coordinate system.

   Scaling ratio: Set the drawing ratio.

   Rotate (horizontal, vertical): Set the graphic display.

2. Graphic parameter setting methods

   1) Move cursor to those parameters to be set, type the corresponding values according to actual requirements, and confirm by pressing the DATA INPUT key on the panel.

   2) Start autorun, the machine starts to move; and the tool path is drawn and displayed on the LCD display.

Detailed description

- Range (actual graphic range) Display the size of the graphic screen (see Figure 3-6-2), the maximum graphic range is 144 mm (width) × 90 mm (height).

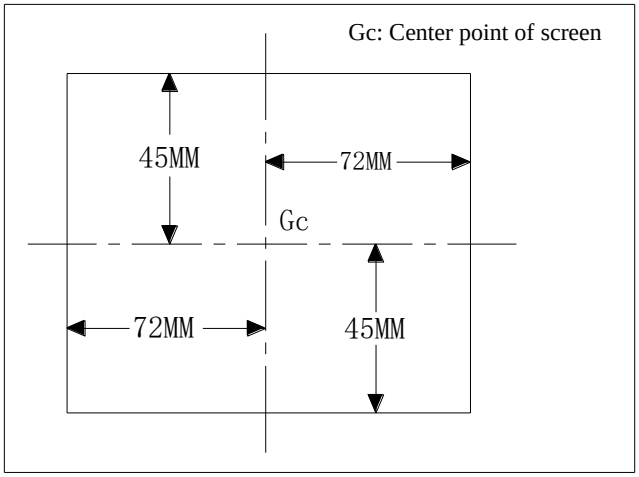


Figure 3-6-2

- Set the graphic range: To draw a part of the program within the actual graphic range, and set the graphic range by any of the following methods:
  1) Set the center point coordinates and magnification of the range.
  2)Set the maximum and minimum coordinate values of the program.
  Whether to use 1 or 2 depends on the setting of parameters. The range of the designed graphics remains after the power is turned off.

3. Set the graphic magnification of the center point coordinate within the graphic range

Set to the center of the graphics range at the center of the screen. If the drawing range of the program is within the actual drawing range mentioned above, the magnification can be set to 1 (the actual value is 100).

When the drawing range is much larger or smaller than that of the screen, the scaling ratio should be modified, that is, the scaling ratio of the graphic parameters should be modified. It may be decided as follows:

Graphic magnification = the smallest of both graphic magnification (H) and graphic magnification (V) Q

Graphic magnification H = α/ (length of horizontal axis in program)

Graphic magnification V = β/ (length of longitudinal axis in program)

α:144mm

β:90mm

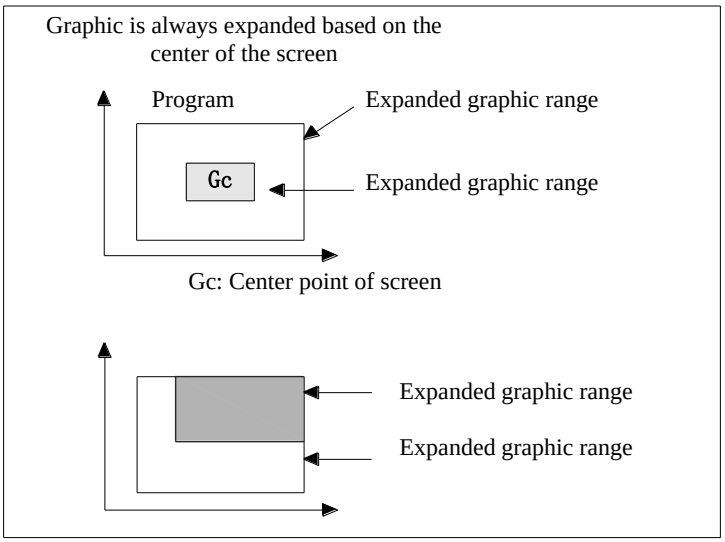
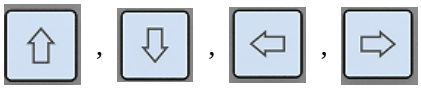


Figure 3-6-3 Use of magnification (examples of magnification)

Supplementary to the enlargement and reduction of Figure 3-6-3:

1) Graphic rotation: Graphics can be rotated by four keys on the operation panel [F]/[D]/[H]/[B]. Large capacity graphics responds more slowly.

2) Graphic scaling: The graphic zooming can be controlled by the page up and down keys.

3) The arrow keys  can be used to change the coordinates of the drawing origin and the graphical center.

● Even if the origin of the workpiece coordinate system changes, the position of the origin and the center of the drawing will not change.

In other words, the origin point of the workpiece coordinate system is always consistent with the origin point of the graph (see Figure 3-6-4).

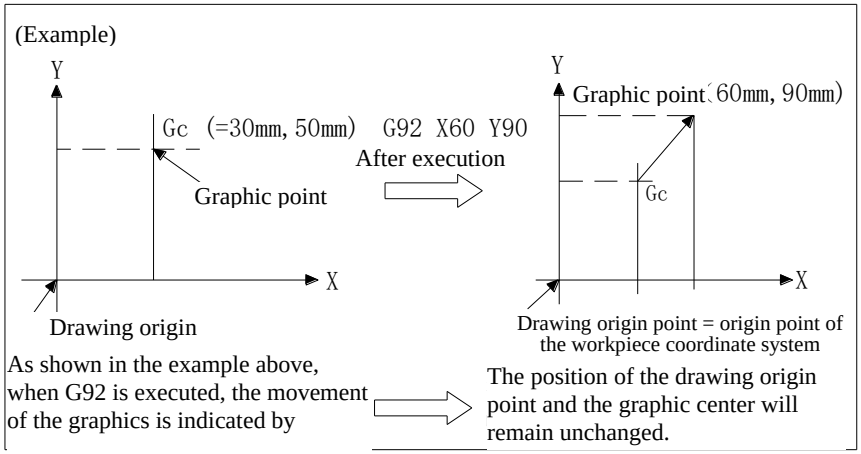


Figure 3-6-4 Origin point of the workpiece coordinate system and drawing origin point

Effective range of graphic parameter axis: 0~±9999999

Caution:

The unit is 0.001 mm or 0.0001 inch. The maximum value of each axis must be greater than the minimum value.

When the maximum and minimum drawing parameters are used to set the image range, the magnification and screen center coordinates cannot be set again. The last set parameter is valid.

● Scale

Set the magnification of the drawing to modify the zoom ratio of the graph parameters.

● Graphic center point

X=_

Y=_

Z=_

B=_

C=_

Set the coordinate value on the workpiece coordinate system to the center of the drawing.

Note:

1.When the maximum and minimum values of the range are set, once the drawing is performed, the values of the drawing points will be automatically calculated.

2. When the drawing range is set by magnification and screen center coordinate parameters, it is not necessary to set the maximum and minimum values of the drawing range. If the parameter set last time is set, it is valid.

4. Graphical execution

During autorun, when the coordinate values are updated, the graphics are drawn. Therefore, drawing can only be done in autorun mode. In order to execute drawing without moving the machine, the machine should be locked.

5. Deletion of drawn graphics

Press the [Clear] soft key to delete the previously drawn graphic at any time.

3.6.2 Path

Press the GRA soft key to enter the path display interface (see Figure 3-6-5).

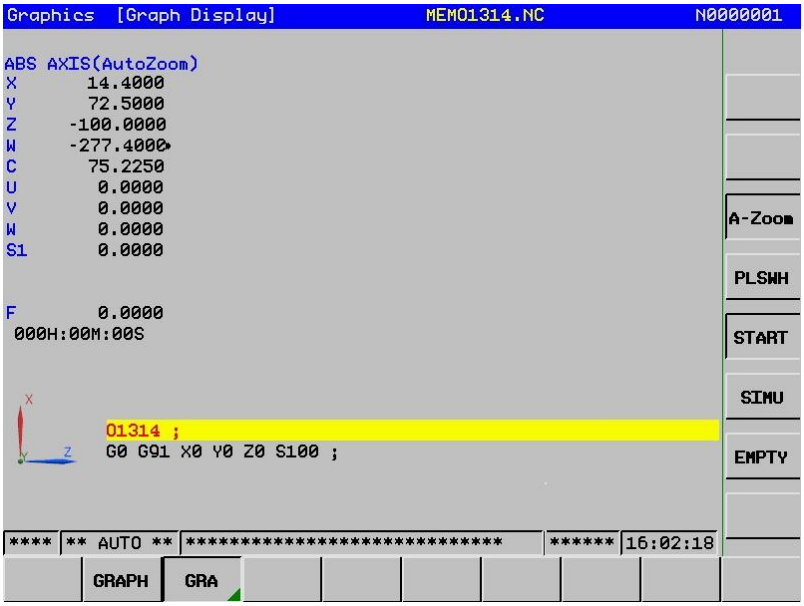


Figure 3-6-5

1. On the graphics page, the machining path of the running program is being monitored.
   A. Press the [Start] soft key to start drawing. If the [Start] soft key can display the selected state, you can see the movement of the tool nose.
   B. Press the [Automatic Scaling] sub-soft key to rotate, zoom in and zoom out the graphics.
   C. Every time the [Plane Switch] soft key is pressed, the graphics are switched and displayed in the corresponding coordinate system of 0~8.
   D. Press the [Clear] soft key to clear the drawn graphics.
2. On the graphics page, the motion trajectory of the program is simulated.

   Rotate the  to select the manual mode. In the manual mode, press the [Simulate] sub-soft key, the currently loaded program is simulated, and the path is displayed. When the simulation ends at the end of the program, the simulation stops automatically.

3.7 Ladder diagram interface

Press the edit panel 

key to enter the ladder diagram display interface. There are nine display pages: [Ladder Diagram], [PLC Diagnosis], [Signal Tracking], [K Parameter], [Timer], [Counter], [PLC List], [Information Check], and [Data Table].

3.7.1 Ladder diagram display interface

Press the PLC ➔ soft key on the edit panel to enter the TODISP display interface. The sub-soft keys displayed on the right side of the interface: [Save], [Replace All], [Select], [Copy], [Cut], [Stop], and [Search], which indicate the related operations that can be performed on this interface (see Figure 3-7-1).

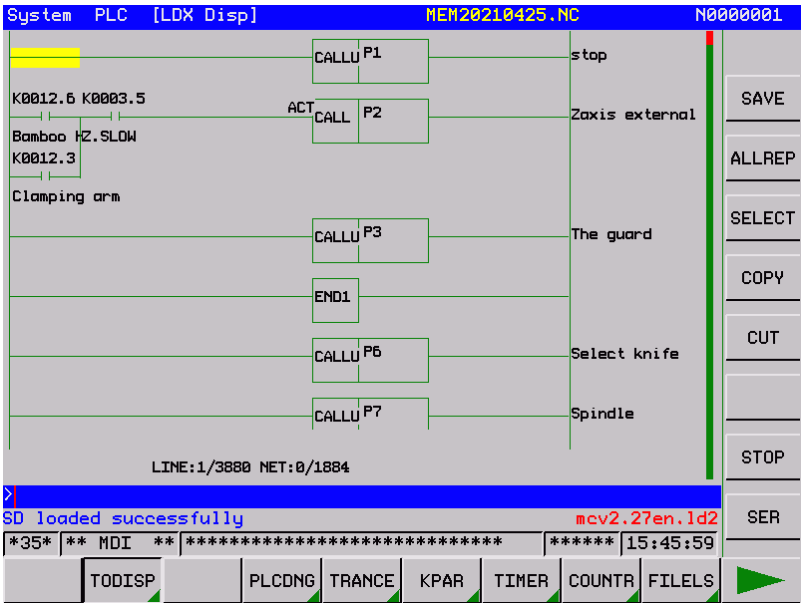


Figure 3-7-1

3.7.2 PLC diagnosis

Press the soft key [PLC Diagnosis] to enter the PLC diagnosis interface (see Figure 3-7-2). You can use the page turning key to check the status of each address.

This interface has six sub-interfaces: [F resource], [G resource], [X resource], [Y resource], [R resource], [A resource]. For the corresponding meaning and setting method of each diagnosis number address, please refer to the *Installation and Commissioning Manual of GSK25iMc/GSK25iTc Series Bus-type Milling/Turning CNC System*.

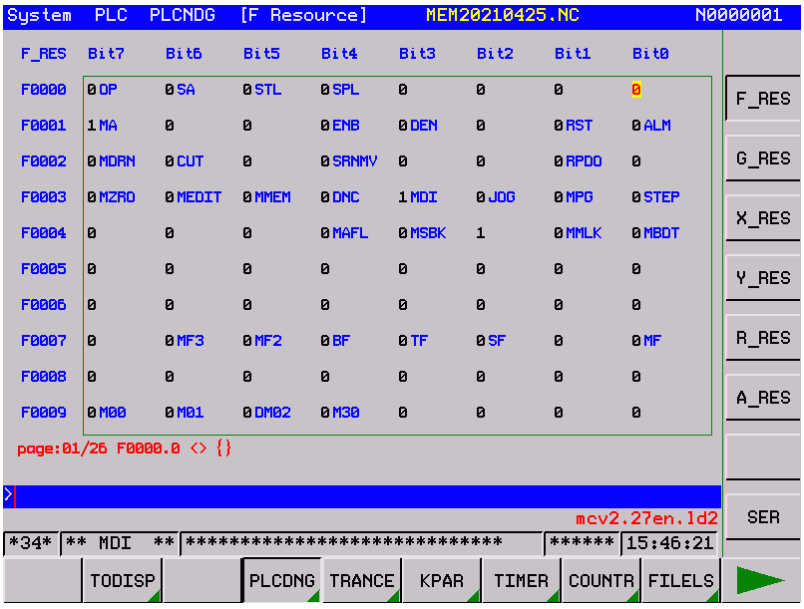


Figure 3-7-2

Part II Operation

3.7.3 Signal Tracking

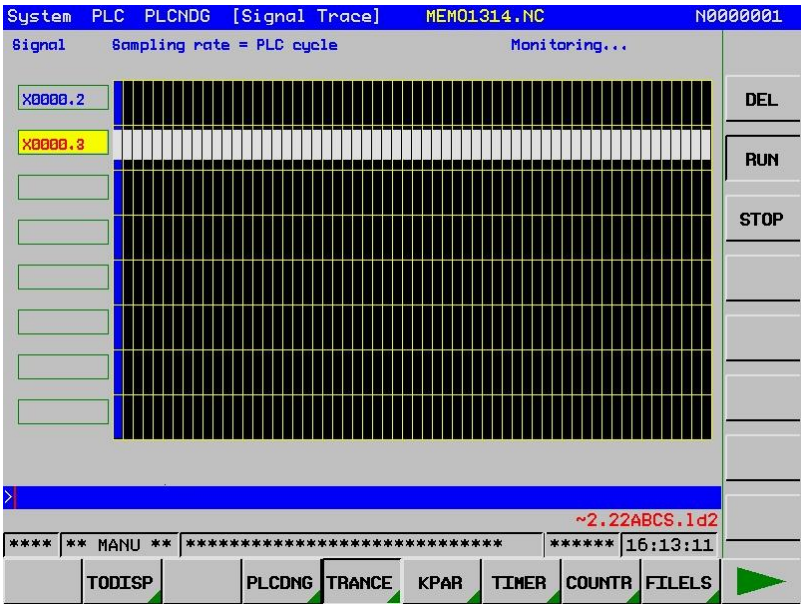


Figure 3-7-3

Specific operation is as follows:

a) Press the soft key TRANCE to enter the PLC signal tracking interface.

b) Enter the signal address in any operating modes such as X0000.3 and so on as shown above.

c) Press the [Run] sub-soft key. When this signal is executed, the block diagram in Figure 3-7-3 will appear. If no signal is transmitted, it will be displayed as a straight line.

d) Press the [Stop] sub-soft key to stop the tracking operation of the signal;

e) In the state of tracking stop, the cursor selects the signal address in the signal source, and press the [Delete] sub-soft key to delete the signal address selected by the cursor from the list.

3.7.4 K parameters

Press the KPAR soft key to enter the specific parameter modification interface (see Figure 3-7-4).

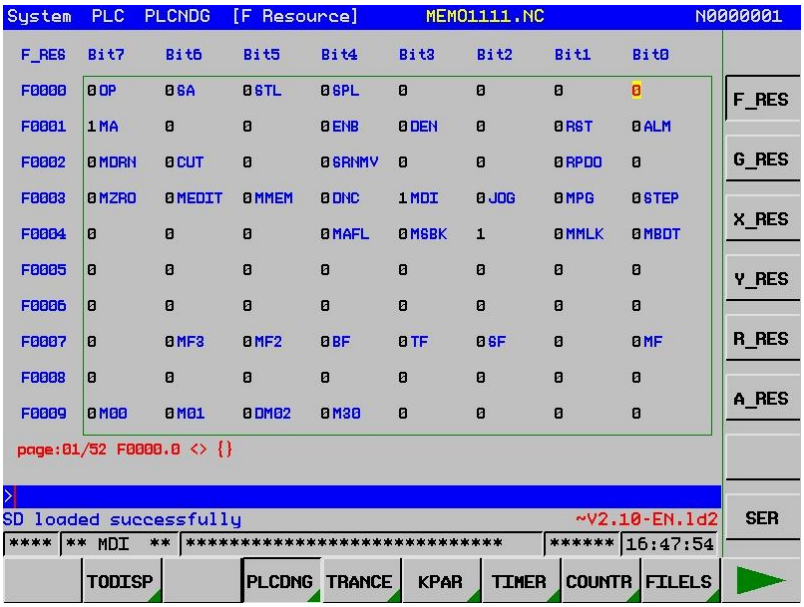


Figure 3-7-4

3.7.5 Timer

Press the TIMER soft key to enter the timer display interface (see Figure 3-7-5).

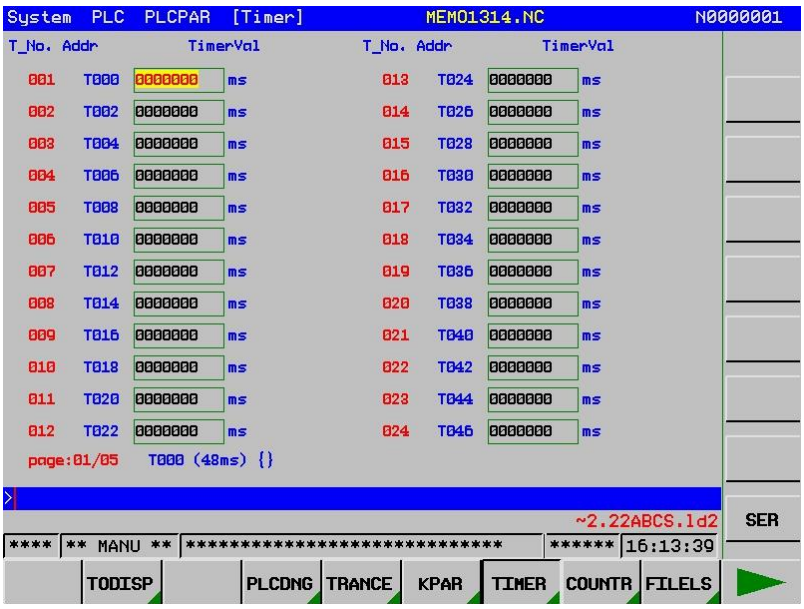


Figure 3-7-5

3.7.6 Counter

Press the COUNTR soft key to enter the timer display interface (see Figure 3-7-6).

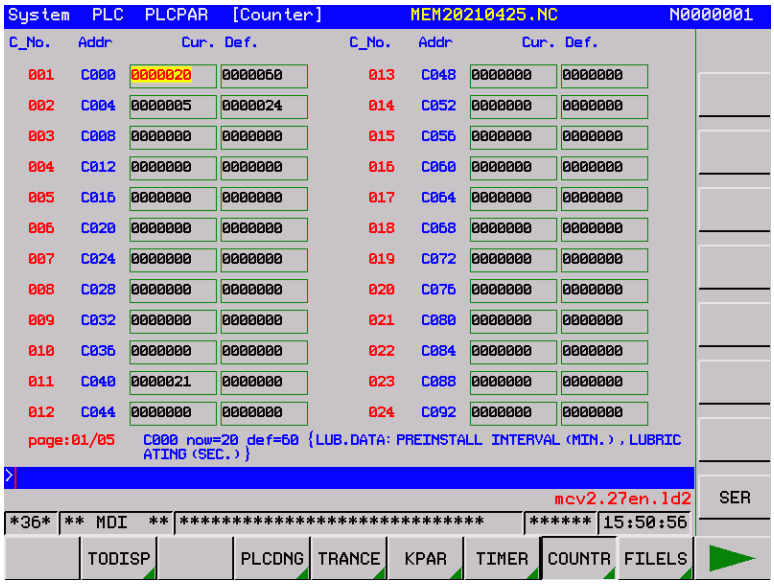


Figure 3-7-6

3.7.7 PLC list

Press the FILELS soft key to enter the PLC file list operation interface (see Figure 3-7-7).

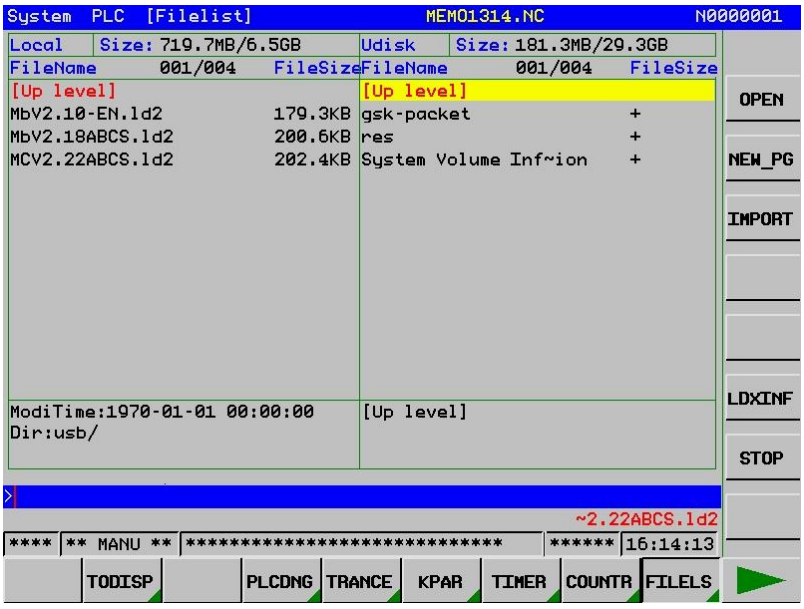


Figure 3-7-7

On the PLC file list interface, when the controller has identified that there is a USB disk, the size of the USB disk and the contents of the file in the USB disk will be displayed on the right side of the interface.

The cursor's left and right move keys can be used to switch the cursor's location in the USB flash drive directory or in the local disk directory, and the cursor's up and down move keys can be used to select different files or folders under the disk pointed by the cursor.

The file name with "+" on the right side of the file name indicates the folder type. After the cursor selects the file name with the "+" sign, press the DATA INPUT key or the [Open Directory] sub-soft key to open the folder; you can move the cursor to the folder. Go to the "[Previous Page]" position, press the DATA INPUT key or the [Open

Directory] sub-soft key to return to the previous directory of the current folder.

On the right side of the interface, there are five sub-soft keys: [Open Directory], [New Ladder Diagram], [Import File], [Ladder Diagram Information], and [Stop]. When the cursor is displayed on the local disk location and it is recognized that a USB disk has been successfully loaded, the [Export File] sub-soft key will be displayed.

Among them, when there is enough operation authority, press the [Stop] sub-soft key, the controller will stop the currently running PLC program, and display the [Stop] sub-soft key as the [Run] sub-soft key, indicating that it is used as an operation key when the PLC should enter the running state, that is, you can use the PLC to change from the stop state to the running state.

The [New Program] sub-soft key is used to create a new PLC file, with the default suffix name being ".LD2".

The [Export File] sub-soft key is used to transfer the PLC file from the local disk to the USB flash drive.

The [Export File] sub-soft key is used to transfer the PLC file from the USB flash drive to the local disk.

The [Ladder Diagram Information] soft key is used to display the attribute information of the PLC file selected by the cursor.

3.7.8 Information check

Press the CHKINF soft key to enter the information check display page (see Figure 3-7-8).

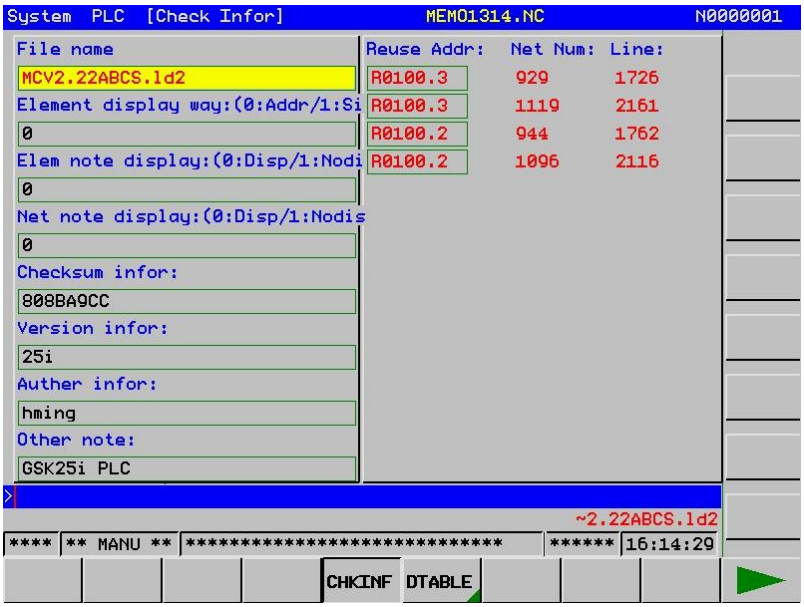


Figure 3-7-8

3.7.9 Data table

Press the DTABLE soft key to enter the data table display page (see Figure 3-7-9), and the right side of the interface displays four sub-soft keys [Table Number Setting], [Table Display], [Data Display], and [Search].

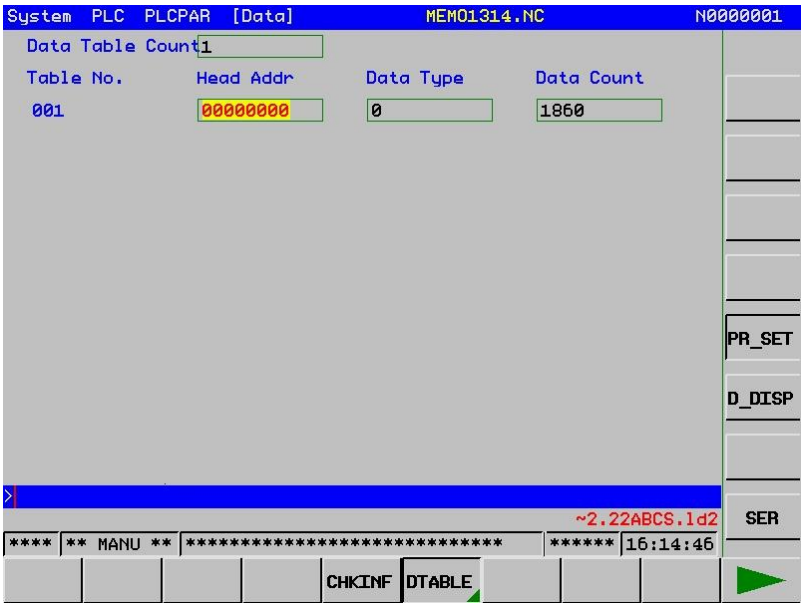


Figure 3-7-9

3.8 User defined interface

Press the 

key on the editing panel to enter the user-defined display interface. Figure 3-8-1 shows an undefined blank interface, and Figure 3-8-2 shows a user-defined interface.

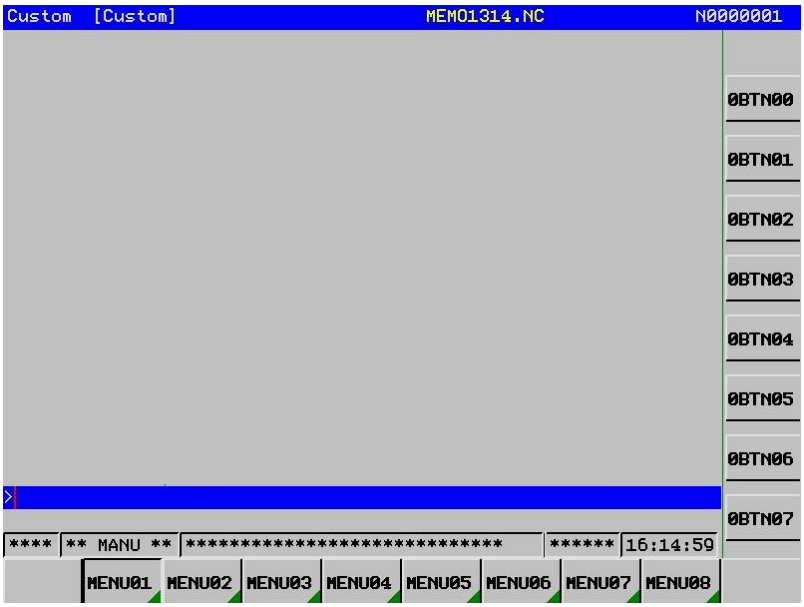


Figure 3-8-1

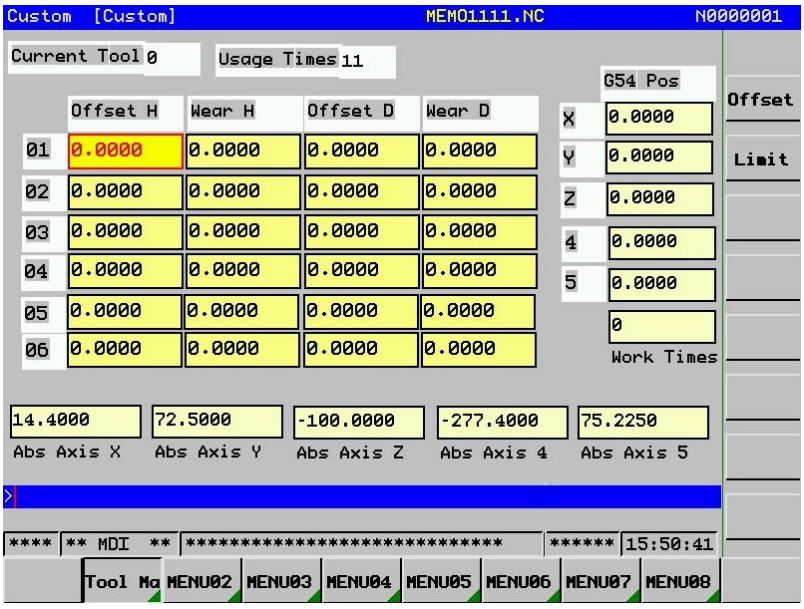


Figure 3-8-2

3.9 Information interface display

Press the MESSAGE key on the editing panel to display the information page. It includes eight interfaces: [Current Alarm], [Alarm History], [Operation History], [Machining History], [Maintenance Record], [Periodic Maintenance], [System Information], and [Running Information], whose functions may be viewed by the corresponding soft keys.

3.9.1 Current alarm interface

When a system error alarm occurs, the "Alarm" message flashes on the bottom line of the LCD. Press the ALM soft key to enter the current alarm interface and view the current alarm information (see Figure 3-9-1).

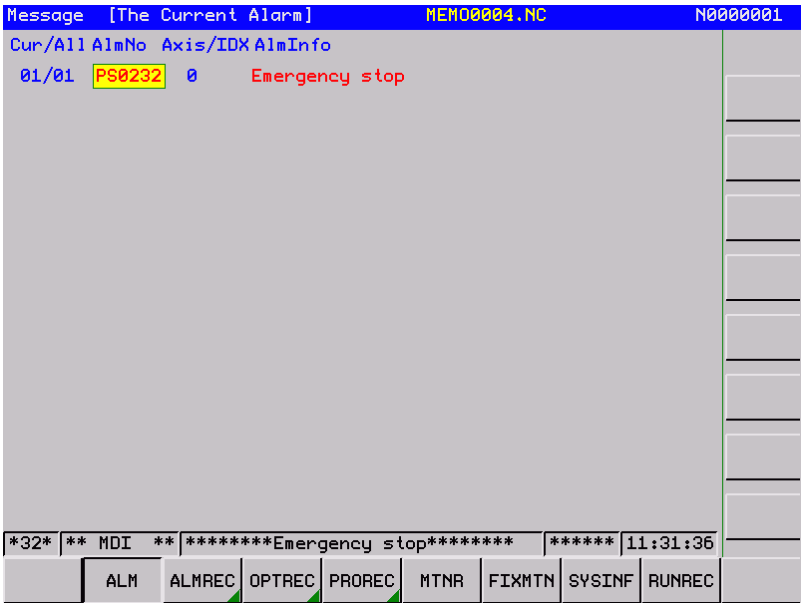


Figure 3-9-1

The detailed content of the current P/S alarm number is displayed on the alarm display screen, with the specific alarm content shown in Appendix II.

3.9.2 Alarm history

Press the MESSAGE ➔ ALMREC to enter the alarm history display interface (see Figure 3-9-2).

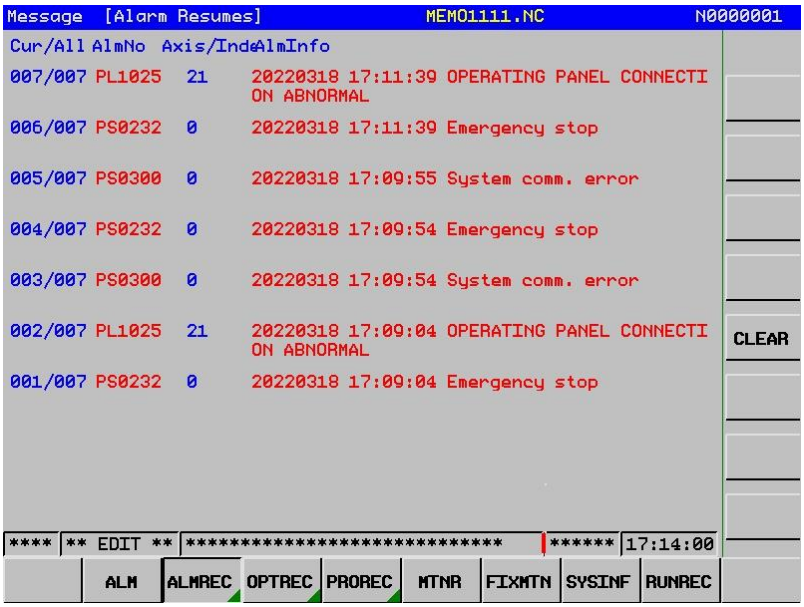


Figure 3-9-2

On this interface, the time is arranged from near to far in order for the user to view.

The operation steps of displaying the alarm history are as follows:

1) Press the function key MESSAGE .

2) Press the ALMREC soft key to display the following information items.

a) Alarm number

b) Axis number

c) Alarm information

d) Alarm time

3) Pages may be changed by the page turning key.

4) Press the [Clear] sub-soft key to delete the recorded information.

3.9.3 Operation history

On the information interface, press the OPTREC soft key to enter the operation history interface (see Figure 3-9-3).

The content displayed on the operation history interface is the specific modification information of the system parameters and the ladder diagram, such as the modification content and the modification time.

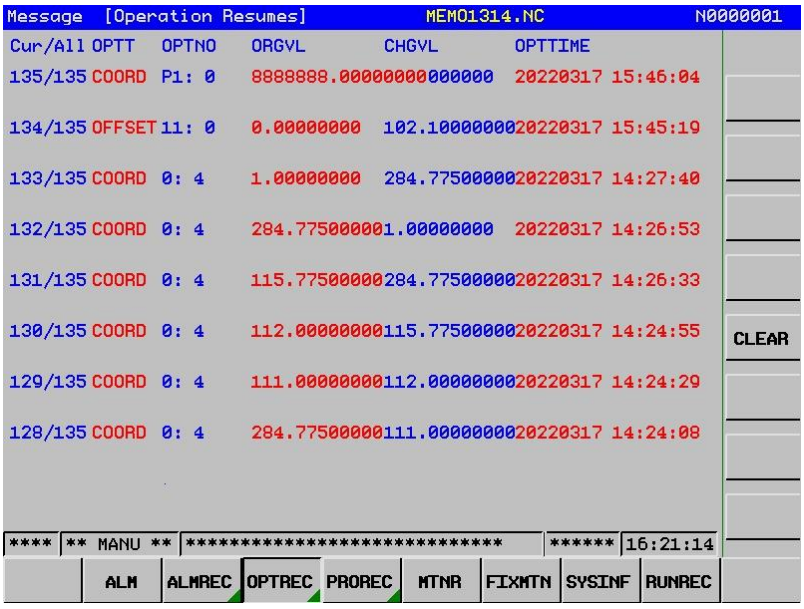


Figure 3-9-3

Use the page key to view the operation history, and delete it by pressing the [Clear] sub-soft key.

Note: After logging in the password and obtaining the highest authority, the system can perform the [Clear] operation.

3.9.4 Machining history

Press the PROREC soft key on the information interface to enter the machining history interface (see Figure 3-9-4). The content displayed on the machining history interface includes the machining program name, machining time and end time. Press the [Clear] sub-soft key to delete the recorded information.

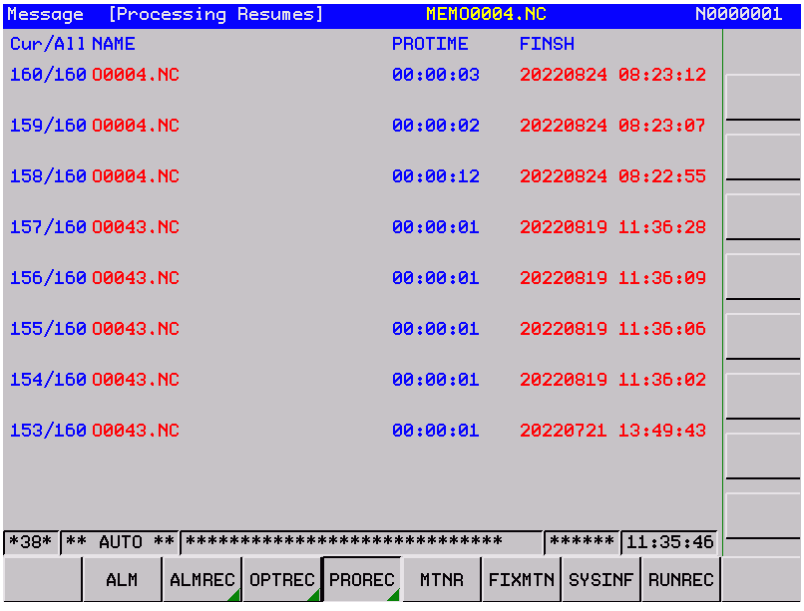


Figure 3-9-4

3.9.5 Maintenance records

Press the MTNR soft key to enter the maintenance record display page (see Figure 3-9-5).

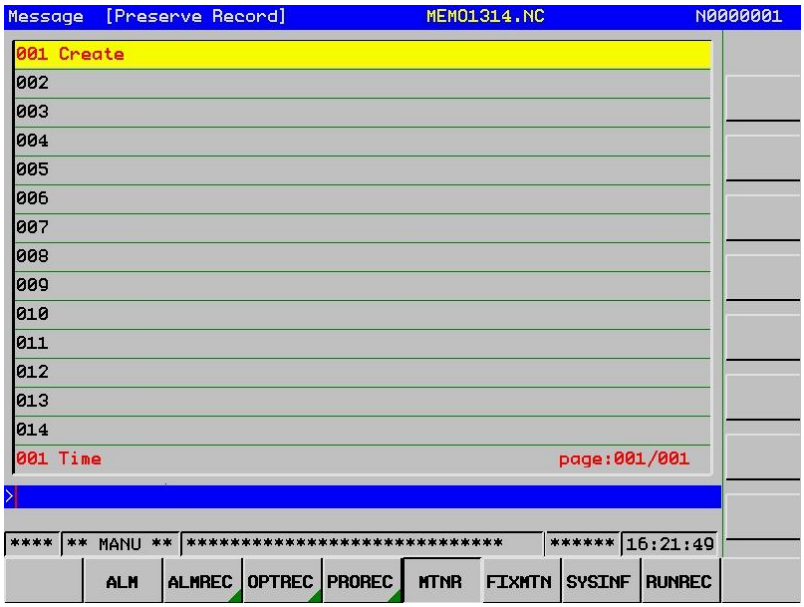


Figure 3-9-5

3.9.6 Periodic maintenance

Press the FIXMTN soft key to enter the periodic maintenance message interface, where you can view the remaining life of the drive unit battery and system backlight (see Figure 3-9-6).

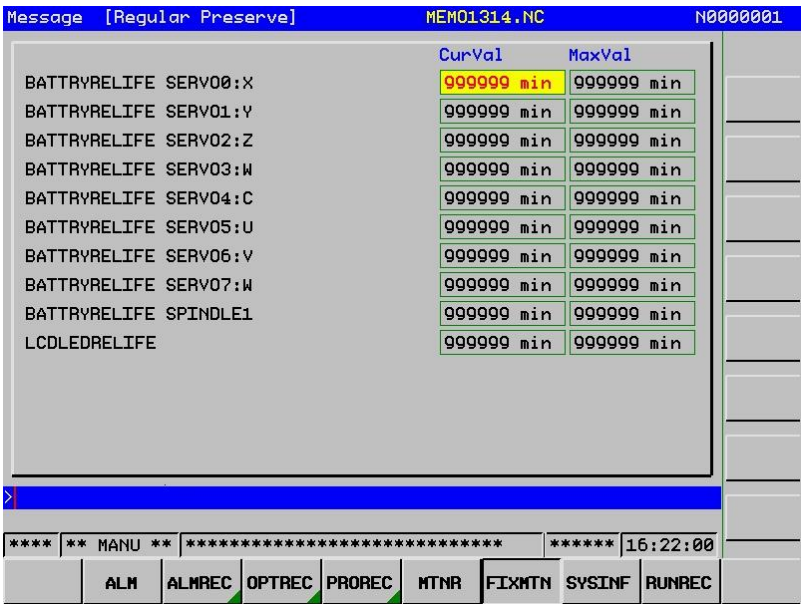


Figure 3-9-6

3.9.7 System information

Press the soft key SYSINF to enter the system information display interface (see Figure 3-9-7).

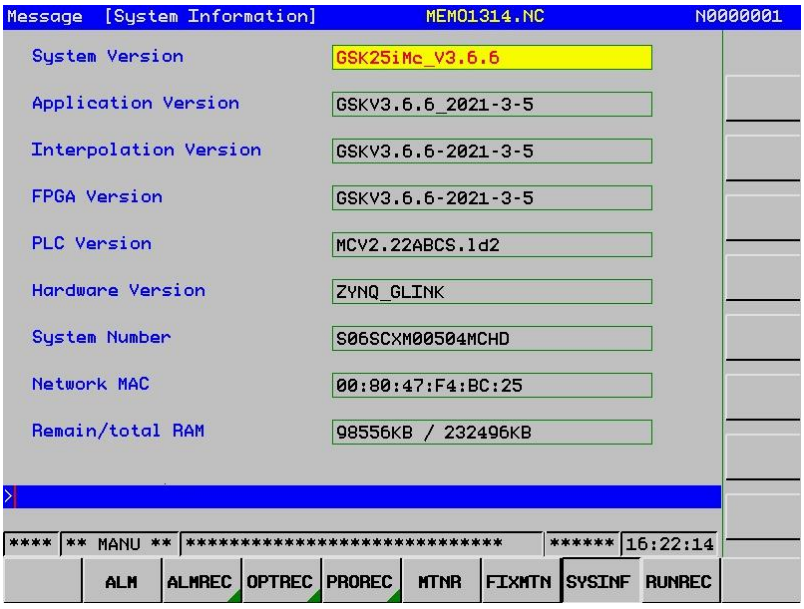


Figure 3-9-7

3.9.8 Running information

Press the soft key RUNREC to enter the operation information display interface (see Figure 3-9-8).

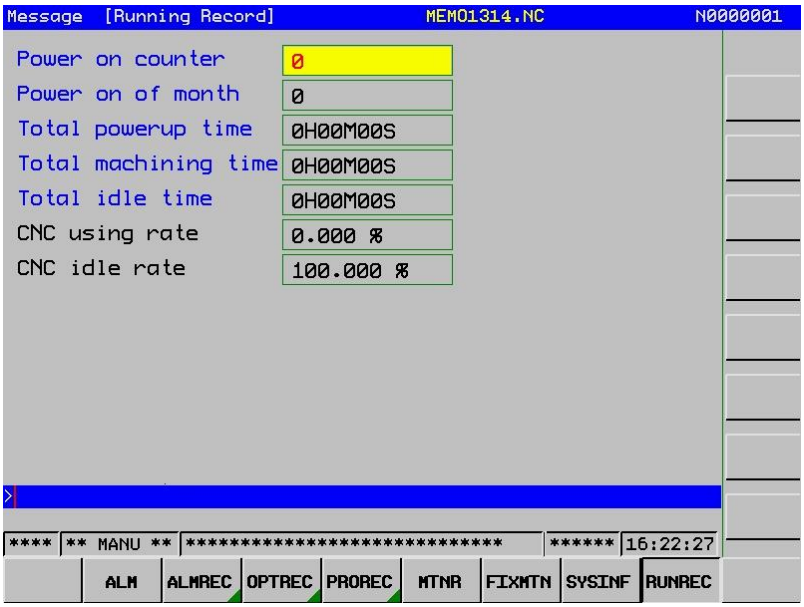


Figure 3-9-8

3.10 Help interface display

Press the HELP button on the editing panel to enter the help display interface. This interface includes the following ten display interfaces: [Alarm], [Wizard], [G Code Table], [Macro Command B], [PLC Address], [Calculator], [Parameter Table], [Servo Parameters], [Servo Motor], [Spindle Motor], which can be switched and viewed through the corresponding soft keys.

3.10.1 Alarm interface

Press the HELP key ➔ ALM soft key on the editing panel to enter the alarm display interface (see Figure 3-10-1).

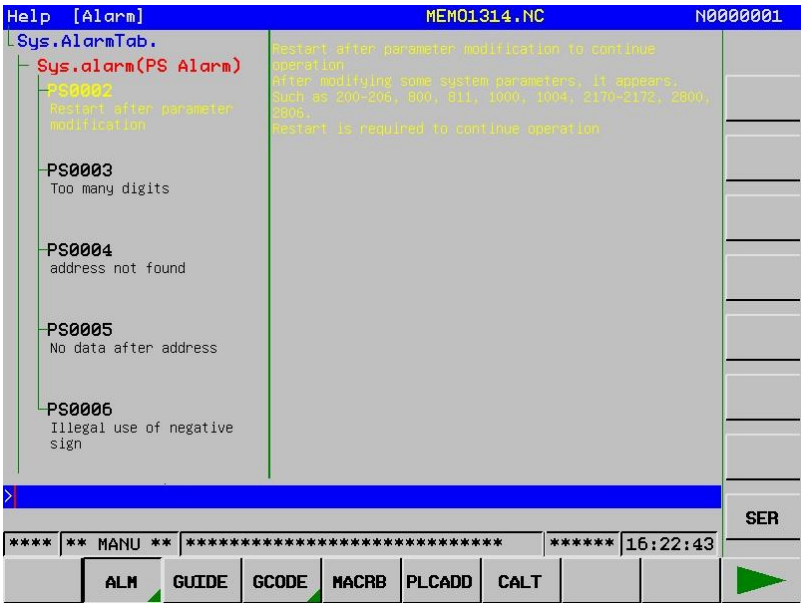


Figure 3-10-1

This interface introduces the meanings and handling methods of various alarm codes in detail. It is possible to check the alarm information and annotation of each alarm code using direction keys ⇧ and ⇩. It is also possible to enter the alarm code to the input field and press the [Search] and DATA INPUT key on the editing panel to search and check the corresponding contents.

3.10.2 Wizard interface

Press the GUIDE soft key on the help interface to enter the operation interface (see Figure 3-10-2).

The [Guidance] interface briefly introduces manual operation steps and methods of each interface, and any question about operations can be searched and solved here. Please use the direction keys ⇧, ⇩, ⇦, ⇨ to select the corresponding items and check relevant operations.

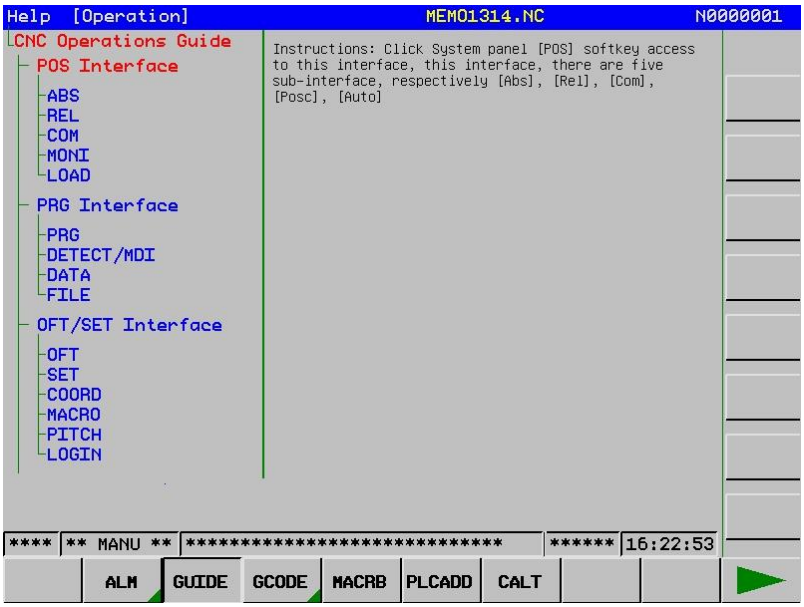


Figure 3-10-2

3.10.3 G-code list interface

Press the GCODE Table soft key on the help interface to enter the G code display interface (see Figure 3-10-3).

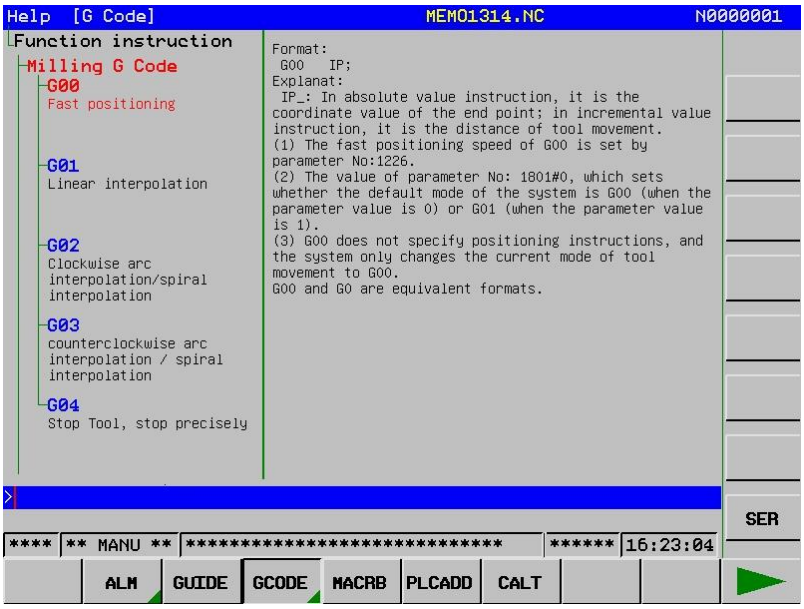


Figure 3-10-3

The G-code interfaces introduce definitions of each G code adopted in the system and use cursor  ,  ,  ,  to select the G code for viewing, with its definition showing at the bottom of the interface. The format, usage and other relevant information of a G code can be obtained by simply selecting it with cursor. This interface introduces the format, function and definition of each command in detail, and any question about commands can be searched and solved here.

3.10.4 Macro-command interface

Press the MACRB soft key on the help interface to enter the macro command display interface (see Figure 3-10-4).

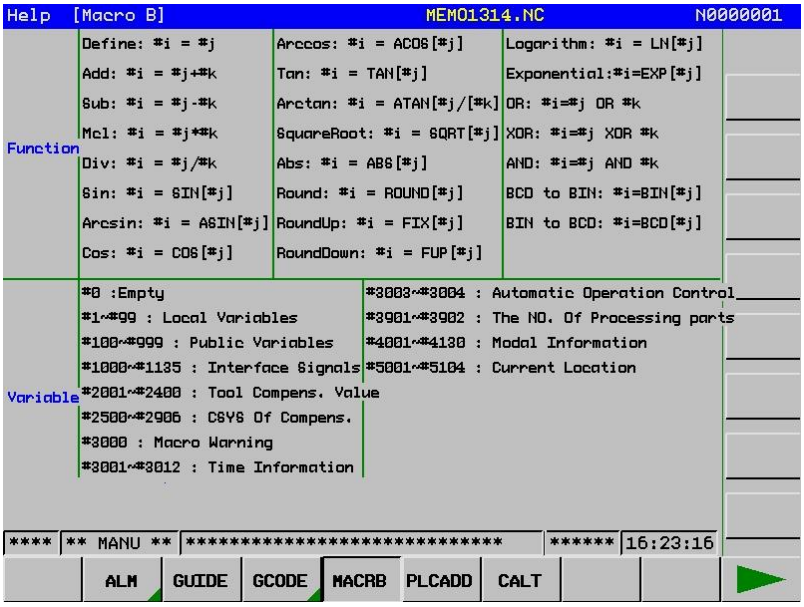


Figure 3-10-4

This interface introduces format of the macro-command and various operational commands and provides the setting range of local variables, general variables and system variables. If you are unfamiliar with or unclear about the macro command operation, search it and make comparison in this interface.

3.10.5 PLC address interface

Press the PLCADD soft key on the help interface to enter the PLC address interface (see Figure 3-10-5).

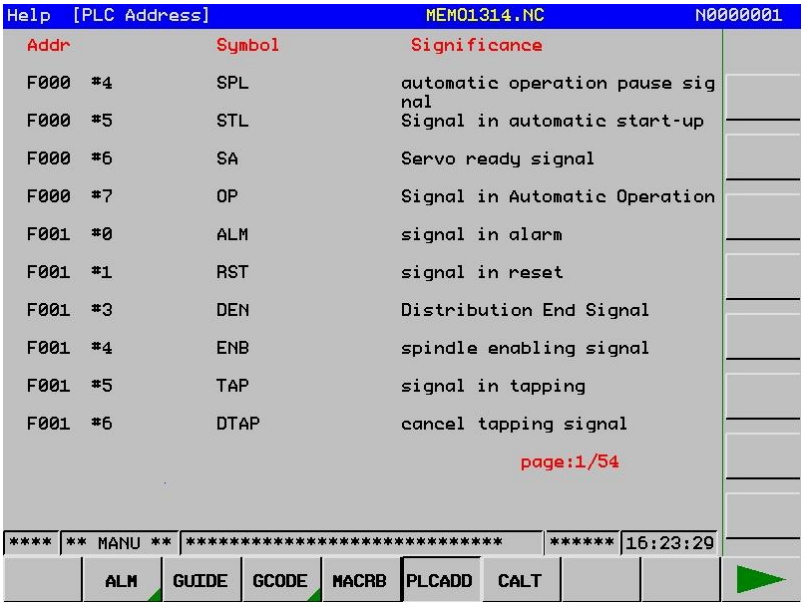


Figure 3-10-5

This interface introduces PLC address, symbols and meaning in details, and any question about PLC address

can be searched and solved on this interface. There are a total of 17 pages; please use the page turning keys to view.

3.10.6 Calculator interface

Press the soft key CALT on the help interface to enter the calculator address interface (see Figure 3-10-6).

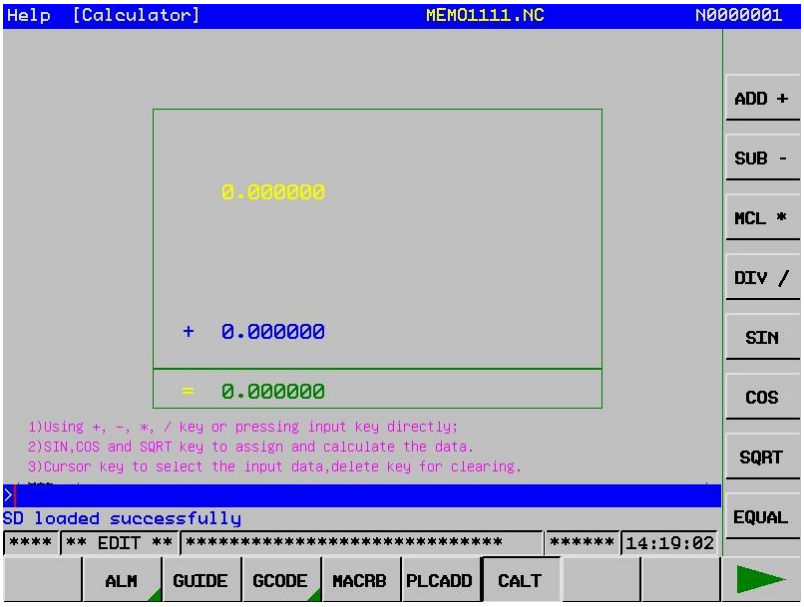


Figure 3-10-6

On this interface, the system provides the operations of add, subtract, multiply, divide, sine, cosine and square root, which may be operated by the following two methods:

(1) The operation method of "add, subtract, multiply and divide" is: Entering data → Short pressing the computing format soft key → Entering data → Short pressing [Equal] soft key → Obtaining the result.

(2) The operation method of "sine, cosine and square root" is: Entering data → Short pressing the computing format soft key → Obtaining the result.

3.10.7 Parameter table interface

On the second page of the help interface, press the soft key PAR_TB to enter the parameter table display interface (see Figure 3-10-7). This interface introduces the range of parameter number of each function's parameters in details.

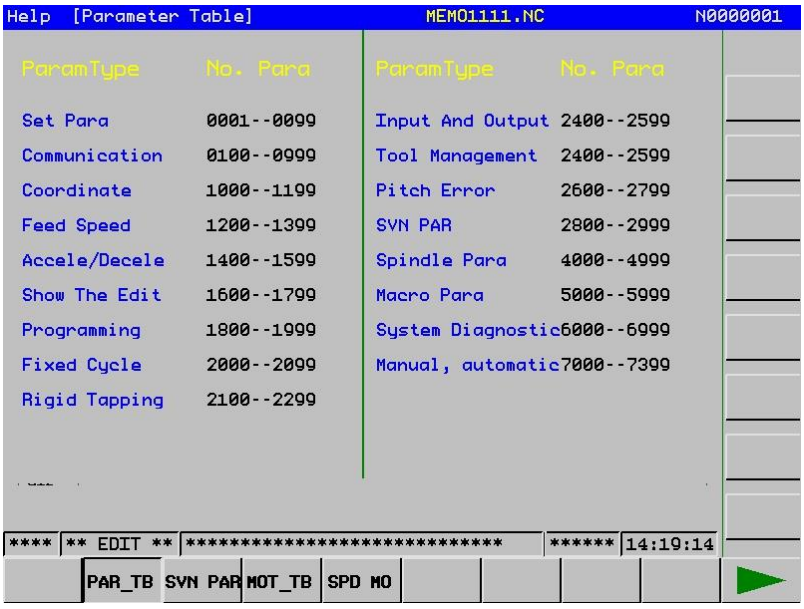


Figure 3-10-7

3.10.8 Servo parameter interface

Press the soft key SVN PAR on the second page of the help interface to enter the servo parameter display interface (see Figure 3-10-8). This interface mainly displays some technical parameters related to the servo motor. You can view the parameter number by turning the page up and down keys , .

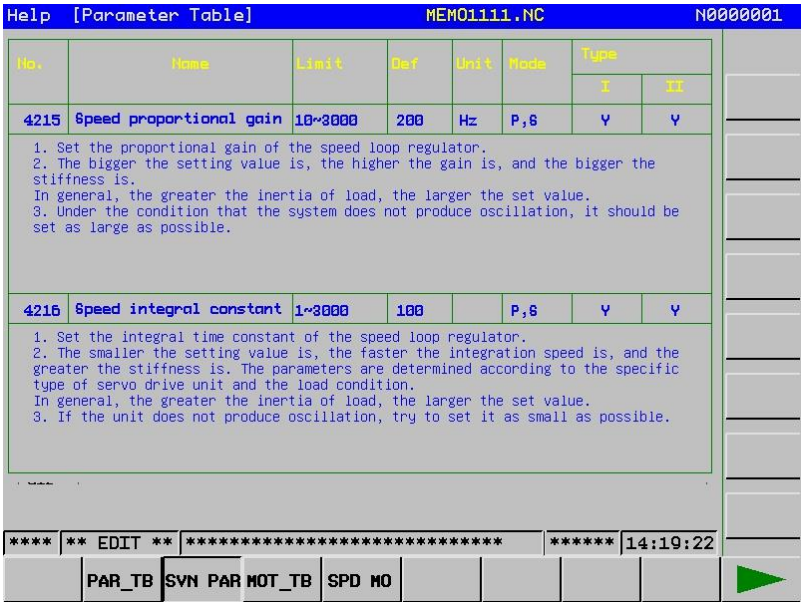


Figure 3-10-8

3.10.9 Servo motor interface

Press the soft key MOT_TB on the second page of the help interface to enter the servo motor display page (see Figure 3-10-9). You can use the page up and down keys , to view the model of each servo motor.

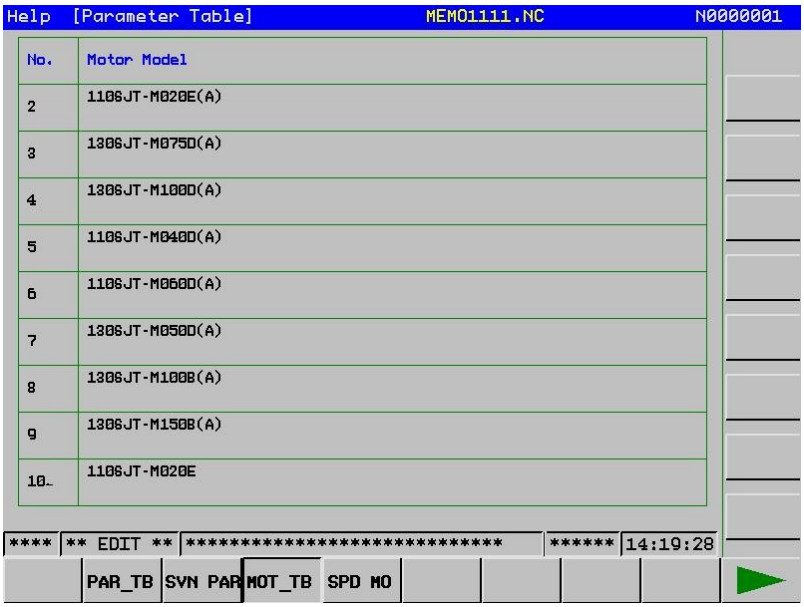


Figure 3-10-9

3.10.10 Spindle Motor Interface

On the second page of the help interface, press the soft key SPD MO to enter the display interface of the spindle motor (see Figure 3-10-10). You can view the model and related technical parameters of each spindle motor through the page up and down keys , .

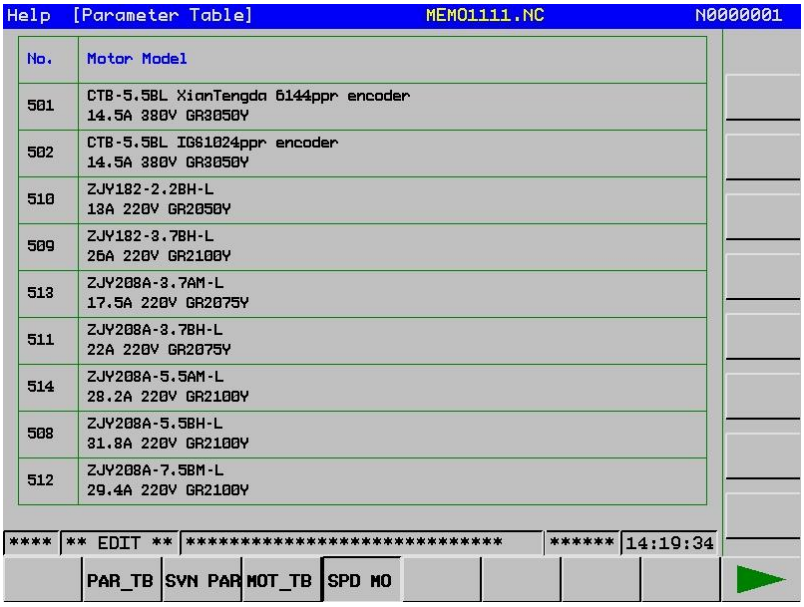


Figure 3-10-10

Chapter IV Manual Operation

Rotate the  to select the manual operation mode, mainly including manual feed, spindle control and machine panel control, etc.

4.1 Coordinate axis movement

Under manual operation mode, each axis moves at manual feed speed or manual fast moving speed.

4.1.1 Manual feed

Press  or  , it can move the X-axis positively or negatively, and release the key, it can stop the axis movement. Also, the feed speed can be adjusted by changing the feed override. Other axis moves in the same way. This system supports simultaneous movement of five axes as well as simultaneous homing.

Note: The manual feed speed of each axis is decided by #1232 parameter; manual fast moving speed is decided by #1233 parameter.

4.1.2 Manual rapid movement

Press the 

key, the indicator light on the 

key is on, indicating that it enters the manual rapid movement state. Then, press the manual feed axis key, each axis moves at the rapid running speed. The manual fast moving is invalid at the time of manual single-step (here, the "manual single-step" means to enter the single-step mode by pressing the 

key).

Note 1: The manual fast moving speed is set by #1233.

Note 2: Place parameter #1200.0 can decide whether manual fast moving is valid before reference point return after switching on.

4.1.3 Manual feed rate and manual fast moving speed selection

If the key indicator of the 

key is not on, the machine is in the manual feed state and the axis movement speed can be adjusted using the feed override knob. When the 

key is pressed and the indicator light is on, it is in the manual rapid movement state. The rapid movement speed of the axis can be adjusted by rotating the rapid override band switch  . Fast override has four levels: Fo, 25%, 50% and 100%. (Manual fast moving speed is controlled by #1233 and Fo speed is decided by data parameter #1231). Without pressing the rapid movement key, the movement speed can be adjusted using feed override knob.

Note: Fast override selection can be valid for the following movement speed.

(1) G00 fast feed

(2) Fast feed in fixed cycle

(3) Fast feed in G28

(4) Manual fast feed

For example: When the fast feed speed is 6000 mm/min, if the override is 50%, the speed is 3000 mm/min.

4.1.4 Manual feed per revolution

Parameter controlled manual feed mode: feed per minute/ feed per revolution.

Note:

(1) Manual mode selection is controlled by #1400.

(2) Under the mode of manual feed per revolution, alarm PS243 (spindle speed: 0) will be given when the spindle is stopped.

(3) Manual fast moving mode will be performed on minute basis.

(4) Feed speed is affected by feed override switch.

4.2 Spindle control

4.2.1 Spindle clockwise rotation (M03)



: Set the S rotation speed in the entry mode, and the spindle rotates clockwise when pressing this key under the manual/ MPG/ single-step mode.

4.2.2 Spindle counterclockwise rotation (M04)



: Set the S rotation speed in the entry mode, and the spindle rotates counterclockwise when pressing this key under the manual/ MPG/ single-step mode.

4.2.3 Spindle stop (M05)



: Spindle stops when pressing this key under the manual/ MPG/single-step mode.

4.2.4 Spindle pause



: The spindle pauses at a fixed angle and stops rotating when pressing this key under the manual/ MPG/ single step mode; either pressing the  key or executing spindle off and spindle rotation may discharge the spindle pause.

4.3 Other manual operation

4.3.1 Coolant control



: Compound key, for switching between coolant on and coolant off. Indicator is on for activated or off for disabled.

Note:

1. The initial cooling state is: cool without output.
2. The panel cooling key can always be correctly executed in the modes of editing, auto, MDI, homing, MPG, single-step, manual, returning to program zero and program run, or under the valid functions of single block, block skip, machine lock, auxiliary lock, dry run and optional stop.
3. When the panel cooling key is used together with M08/ M09 codes and the panel cooling key, the output of the cooling signal is subject to the last operation.

4.3.2 Lubrication control



: It connects immediately after pressing and automatically disconnects with time delay. It connects when the indicator is on and disconnects when it is off.

4.3.3 Chip removal control



: Compound key, for switching between chip removal on and off. Indicator is on for activated or off for disabled.

Chapter V Single-step Operation

5.1 Single-step feed

Under manual mode, press the 

key to enter the single-step mode; in the single-step feed mode, the machine moves at a system-defined step length each time.

5.1.1 Selection of movement amount

Unit of movement on the keys: Metric: mm Imperial: 0.1 inch

For example: For the rotation  , select 50%, and the single-step length selected by the metric system is: 0.100; therefore, the corresponding axis of the machine moves 0.1 mm when pressing the movement key once.

The single-step length for the imperial system: 0.100×0.1=0.0100; therefore, the corresponding axis of the machine moves 0.01 inch when pressing the movement key once.

5.1.2 Selection of moving axis and moving direction

Press the  or  key to control the X-axis to move positively or negatively, and the axis can move a single-step distance defined by the system upon each pressing; the single-step movement of other axis may be operated in this way. The manual feed speed of each axis is determined by parameter #1232. Since the movement distance is short, it is not limited by the feed override and fast moving.

5.2 Auxiliary control during the single-step operation

It is the same as manual operation mode; see 4.2 and 4.3 of this part for more details.

Chapter VI MPG Operation

6.1 MPG feed

Rotate the  to select the MPG mode; in the MPG feed mode, MPG is used to control the machine movement and carry out the precise adjustment of the machine.

MPG movement steps:

The "mode selection" switch is set at the "MPG" position.

(1) Select the movement axis

(2) Rotate the handle of the manual pulse generator

Clockwise rotation + direction

Counterclockwise rotation - direction

(The direction varies according to different machine manufacturers.)

(3) Movement amount: Some MPGs are equipped with a movement amount selection knob. Of which, ×1 means to multiply movement amount by 1, ×10 means to multiply movement amount by 10, and ×100 means to multiply movement amount by 100

Input System	Movement Amount Per Grid		
	×1	**×10**	**×100**
Metric input	0.001mm	0.01mm	0.1mm
Imperial input	0.0001inch	0.001inch	0.01inch

(4) The relationship between MPG scale and machine movement amount is as the following table:

	Movement Amount of Each Notch on MPG		
MPG increment (mm)	0.001	0.01	0.1
Machine movement amount (mm)	0.001	0.01	0.1

Numerical value in the above table may vary according to different mechanical transmission, so please refer to instructions of the machine manufacturer for details.

Note: If MPG rotates at a speed higher than 5 rps, difference will be appear between MPG rotation amount and machine movement amount. Therefore, revolution of MPG should not be too fast.

6.1.1 Selection of MPG control mode

MPG control mode is decided by parameter #1401.4 (0: adopting reservoir mode, 1: adopting real-time mode).

When the parameter is set to 0, the system uses the reservoir method to control the movement of the MPG, that is, after the MPG stops, the system will run the accumulated pulses that have not finished running; when the parameter is set to 1, the system uses the real-time control, that is, after the MPG stops, the movement stops either.

In the reservoir mode, pulses from MPG will not be lost but kept before being converted to actual motion; in

the real-time mode, pulses will be lost, without being kept if not converted to actual motion. The real-time mode is generally recommended.

6.2 Auxiliary control during MPG operation

It is the same as manual operation mode; see 4.2 and 4.3 of this part for more details.

Chapter VII Automatic Operation

7.1 Autorun

7.1.1 Operating steps for autorun program

It is possible to load programs under autorun and editing modes.

(a) Rotate the  to select the autorun or editing mode;

(b) Press the 

[File List] soft key to enter file list page, then move cursor to find the object program name;

(c) Press the [Load Program] sub-soft key, the system will load the object program;

(d) Rotate the  to select the automatic operation mode. It is also possible to move cursor, select a specific line in a program for execution, and then enter autorun.

Note: Before operating the program, confirm whether the current coordinate point is the end position of the previous program block before the target program block. if the target program block is tool changing or other actions, it is recommended to confirm the current position to avoid interface and collision with the workpiece or other objects, so as to prevent from machine damage and human injury.

7.1.2 Launch of autorun

After selecting the program to launch according to the method specified in Section 7.1.1, press the 

key to launch the autorun program; it is possible to switch to <Position>, <Detection>, <Graphics> and other interfaces to observe the program running status.

The program running starts from the cursor line, so it is recommended to check the cursor position before pressing the autorun key; if it should start from the initial line, but the cursor is not there, press the  key to return the cursor to the initial line, then press the 

key to run the program from the initial line.

7.1.3 Stop of autorun

There are six methods in the system for stopping an autorun program:

1. Program stop (M00)

After executing a program block containing M00, the program dwells and all modal information is saved. Press the 

key, and the program continues to execute.

2. Program optional stop (M01)

Before program run, if the 

key is pressed, the program execution will dwell when coming to the program block containing M01, and all modal information will be saved. Press the 

key, and the

program continues to execute. If the 

key is not pressed and the selection stop key indicator light is off, it will be deemed as without or no need to execute M01.

3. Press the 

key

When pressing the 

key during autorun, the machine will display the following state:

1) Machine feed deceleration stops;
2) When executing dwell command (G04), it will continue to pause;
3) Other modal information is saved;
4) Press the 

key, and the program continues to execute.

4. Press the 
key

In the MDI or editing mode, press the reset key to stop the cursor at the program header; the reset is only valid when the interface displays the word [Reset]. The program will execute from the header after pressing the 

key. When system parameter 1031.0=0, the reset key is pressed during continuous run under the autorun mode, and the cursor rests at the current line. The program will start running from the current line after pressing the 

key. The program will be eliminated when pressing the reset key under the DNC mode.

5. Press the <Emergency Stop> 
button

6. Mode switching methods

When executing programs under MDI interface of automatic mode, DNC mode and entry mode, the machine can also be stopped when switching to other modes. Specific operation is as follows:

1) The machine will decelerate and stop when switching to the manual, MPG or homing mode.
2) The machine will stop after running the current program block under the single block mode.
3) The machine will stop after executing the current program block when switching to each other or to the editing mode under the automatic, DNC or entry mode.

7.1.4 Spindle speed control during autorun

During autorun, when the spindle speed is controlled by selecting the analog quantity, it can be trimmed by the analog voltage.

During autorun, the spindle override knob can adjust spindle override to change its speed, and the spindle override adjustment is real time with eight levels, from 50% to 120%.

Actual speed of the spindle = Program command speed × Spindle override. Maximum spindle speed is decided by data parameter #5116. Rotate at this speed when exceeding this value.

7.1.5 Speed control in autorun

It is possible to control the moving speed of operation during autorun by trimming the feed override.

The knob may be rotated to change the feed override, which can achieve real-time adjustment at 21 levels within the range of 0%~200%.

Note 1: Feed override trims the programming speed set by F in the program.

Actual feed speed = Value set by F × Feed override

When the PLC parameter K2.2 is set to 0 (default value), turn the  during autorun to select the rapid traverse override, and the rapid override can realize the four-speed adjustment of Fo, 25%, 50%, and 100%.

When the PLC parameter K2.2 is set to 1, autorun moves at the fast moving speed by adjusting the rate knob, and the fast moving rate can switch among 11 levels from 0% to 100%.

Note 2: Final value of the fast moving speed obtained by trimming the fast rate can be calculated as follows:

The fast feed resultant speed is the minimal value of all resultant rates decided by each axis.

Resultant speed decided by each axis = Maximum fast moving speed of corresponding axis (#1226) ÷ (Component of movement amount of corresponding axis ÷ Resultant movement amount)

The final-trimmed fast moving speed value = Fast feed resultant speed × Fast feed override

For example:

G00 Xx Yy Zz

Resultant speed decided by the X-axis: Fx = Vx / (x/L)

Resultant speed decided by the Y-axis: Fy = Vy / (y/L)

Resultant speed decided by the Z-axis: Fz = Vz / (z/L)

L = $\sqrt{x^2 + y^2 + z^2}$

Fast feed resultant speed: F = MIN (Fx, Fy, Fz)

Note 3: The fast feed override is F0, if #1200.4=1, the deceleration stops;

If #1200.4=0, the fast feed speed is set by #1231.

7.1.6 Dry run

Before the program autorun, "Dry Run" can be used to verify the program.

Rotate the  to select the automatic operation mode. Press the 

key to (when the dry run indicator light shines in the status indicating area, it enters the dry run mode).

During fast feed, the program speed adopts dry run speed

During cutting feed, the program speed adopts dry run speed

Note:

(1) Dry run speed is decided by data parameter #1210 (universal for all axes).

(2) Validity of dry run during fast feed is controlled by place parameter #1200.6.

7.1.7 Single block operation

Before autorun, please select single block operation to test program performance.

Rotate the  to enter the automatic operation mode. Press the 

key (when the single block operation indicator light shines in the status indicating area, it enters the single block operation mode). During the single block operation, the system stops after running each program block; now press the 

key to continue the execution until the entire program ends.

Note: In G28, the intermediate point also adopts single program block stop.

7.1.8 Full axis function lock operation

In the automatic operation mode, press the 

key (when the machine lock indicator light shines in the

status indicating area, it enters the machine lock mode). At this moment, the machine stops but display of position coordinates is the same as that during machine motion; it is possible to observe current operation status from [Monitoring] interface and M, S and T are all ready to run; this function is used for program checkout.

Although the machine does not move when locked and displayed machine coordinates stop updating, CNC will continue to execute storage travel limit detection inside according to machine coordinate value calculated by movement command.

Precautions

1. If machine lock key is pressed during machine operation, the machine will stop after finishing the current program in tapping mode; in other modes, it will just decelerate and stop.
2. Only the feed axis has moved when the machine lock is valid; the machine will discharge an alarm and prompt "Machine Lock Canceled" after cancelling the lock, or no alarm message will be prompt.

7.1.9 Auxiliary function lock operation

In the automatic operation mode, press the 

key (when the auxiliary function lock indicator light shines in the status indicating area, it enters the auxiliary function lock mode). At this moment, M, S and T commands are not executed and together with machine lock function, they are all designed for program checkout.

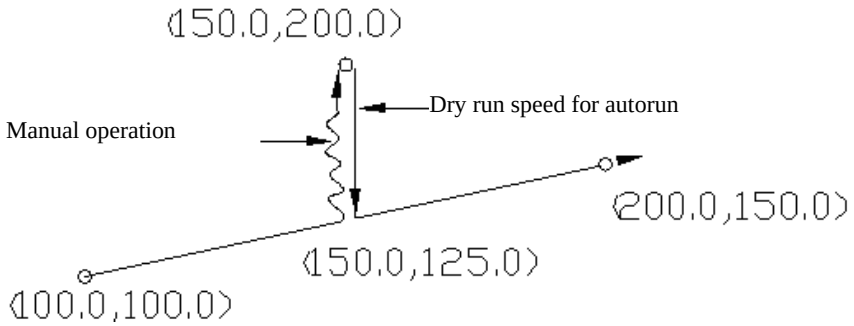
Note: M00, M01, M02, M30, M98 and M99 are executed as normal.

7.1.10 Manual intervention

During autorun, carry out manual stop and withdraw the feed axis. There are two process modes for cycle start after manual operation.

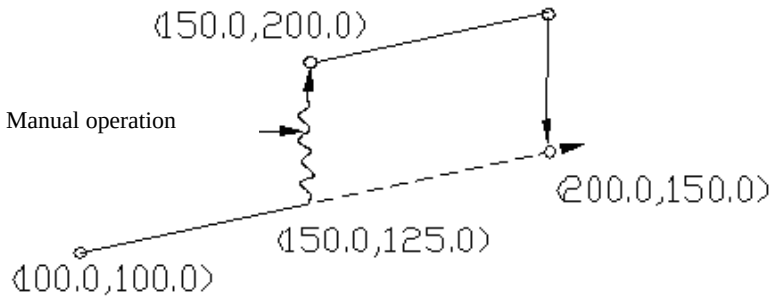
1. When parameter #1950.0 =1, the manual intervention return is valid.

   For the cycle start after manual operation, the machine moves in the order set by parameter #1960, executes the linear interpolation until the automatic stop position and continues the subsequent programs, as shown in the following figure:

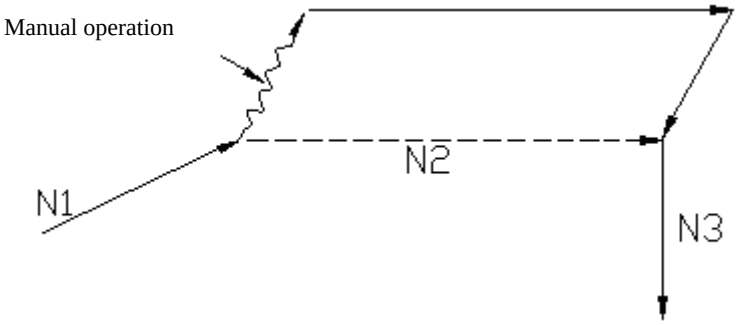


2. When parameter #1950.0 = 0, the manual intervention return is invalid.

   For the cycle start after manual operation, the machine finishes the current program block first, then executes the interpolation at the current speed until the end position of the current program block and continues the subsequent programs; and then it moves to the intermediate processing as shown in the following figure:



After executing the current program block under the single block mode, its processing is shown as follows:



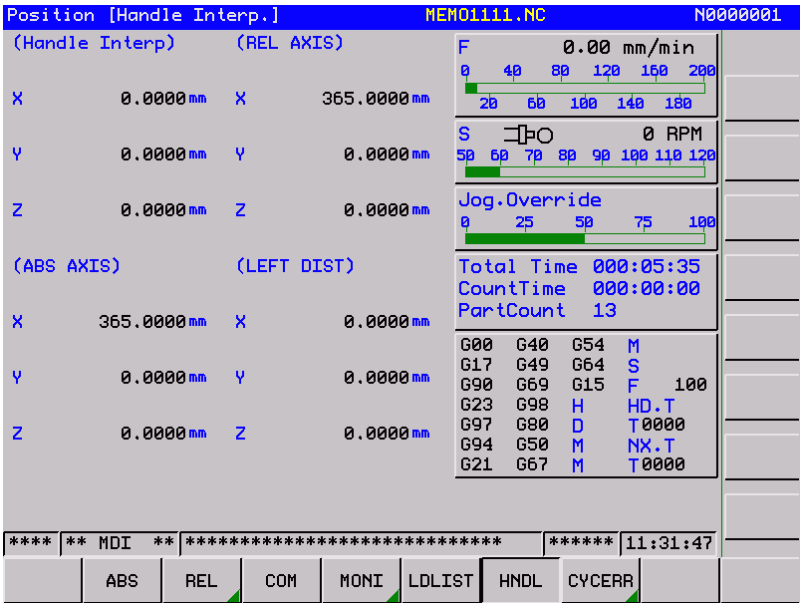
Precautions

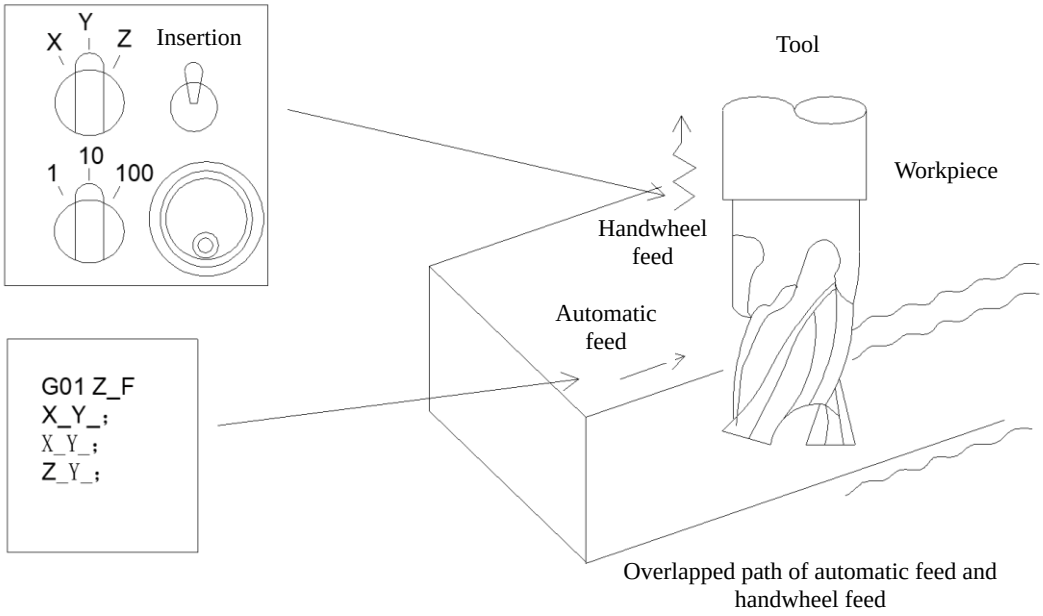
1. When the dry run speed is utilized during the return operation, both manual feed override and single block functions are valid.
2. In case of reset or alarm during manual intervention or return operation, this function is cancelled.
3. The return order of each axis is determined according to the actual machining needs, and set by parameter #1960.
4. For manual intervention operation while executing the RTCP function, force the processing as the manual intervention returns.
5. It is strictly prohibited to carry out manual operation again before the manual intervention startup is finished.
6. Please follow the instructions carefully and do not attempt any unconventional operations, otherwise there will be unpredictable danger.

7.1.11 Insertion of the MPG

Under the mode of autorun (or enter, DNC), the movement overlapping autorun is inserted by rotating the manual pulse generator. The axis for interrupting the MPG is selected through the selection signal of the axis for interrupting the MPG. The feed override for inserting the MPG is identical to that of ordinary MPG (MP1, MP2<G019.4, 5>). This function is commonly used for adjusting while machining if the machining allowance is uncertain during programming. The insertion value of MPG will not change the workpiece coordinate system.

The value inserted by the MPG may be displayed on the position screen.





Cancel the MPG inserting value with reset key.

Note: It is strictly prohibited to conduct this operation under RTCP mode.

7.2 MDI operation

The system provides MDI operation function, which can help to input command code directly.

7.2.1 MDI program editing

Just as the editing mode, it is possible to search codes and line number, select, copy, cut and paste under MDI mode. Before pressing 

key, it is possible to cancel the input character command in the input area by press



key; press 

key can replay content at the cursor position.

7.2.2 MDI command block execution and stop

Press [Cycle Start] key to enter MDI operation after inputting the command and press [Feed Hold] key can stop the operation; if there is no command M30 or M02 codes added to the end of the program, the cursor will not return to the program header after executing the program.

Note: MDI operation can only proceed under enter operation mode.

7.2.3 Switch among various operation modes

When switching to manual, MPG or machine homing mode during autorun, the machine will immediately stop, and the feed indicator keeps on. In order to switch to enter, DNC and editing mode under autorun state, it is only valid after executing the current line.

When switching to manual, MPG or machine homing mode from enter mode, the machine will immediately stop and the feed indicator keeps on. When switching to automatic mode, DNC mode or editing mode, the machine also needs to execute the current line before responding.

When switching to manual, MPG or machine homing mode from DNC mode, the machine will immediately stop, and the feed indicator keeps on. When switching to automatic mode, MDI mode or editing mode, the machine also needs to execute the current line before responding.

Chapter VIII Homing Operation

8.1 Machine reference position concept

For the concepts of machine reference position and machine origin, please refer to Section 1.3.4 of Chapter 1 in the first part of this manual.

8.2 Operating steps for machine homing

Manual machine homing:

1) Rotate the  to select the return-to-zero mode, and enter the machine homing operation mode.

2) Select the X-axis, Y-axis, Z-axis, 4^{th} axis or 5^{th} axis for machine homing and the homing direction is controlled by place parameter #1004.5. (This system supports simultaneous homing of five axes).

3) The machine moves along the direction of the machine zero point. Before arriving at the deceleration point, the machine moves rapidly (the moving speed is set by the data parameter #1235). When the deceleration switch is reached, the machine moves at the speed of FL (which is set by the data parameter #1234) until the deceleration switch is disengaged, and then moves to the machine reference point at the second FL speed (which is set by the data parameter #1236). During machine homing, the coordinate axis stops moving and the reference position indicator is on.

Note 1: If your CNC machine is not equipped with machine reference point, please do not adopt machine homing operation.

Note 2: During machine homing, the corresponding axis indicator is on.

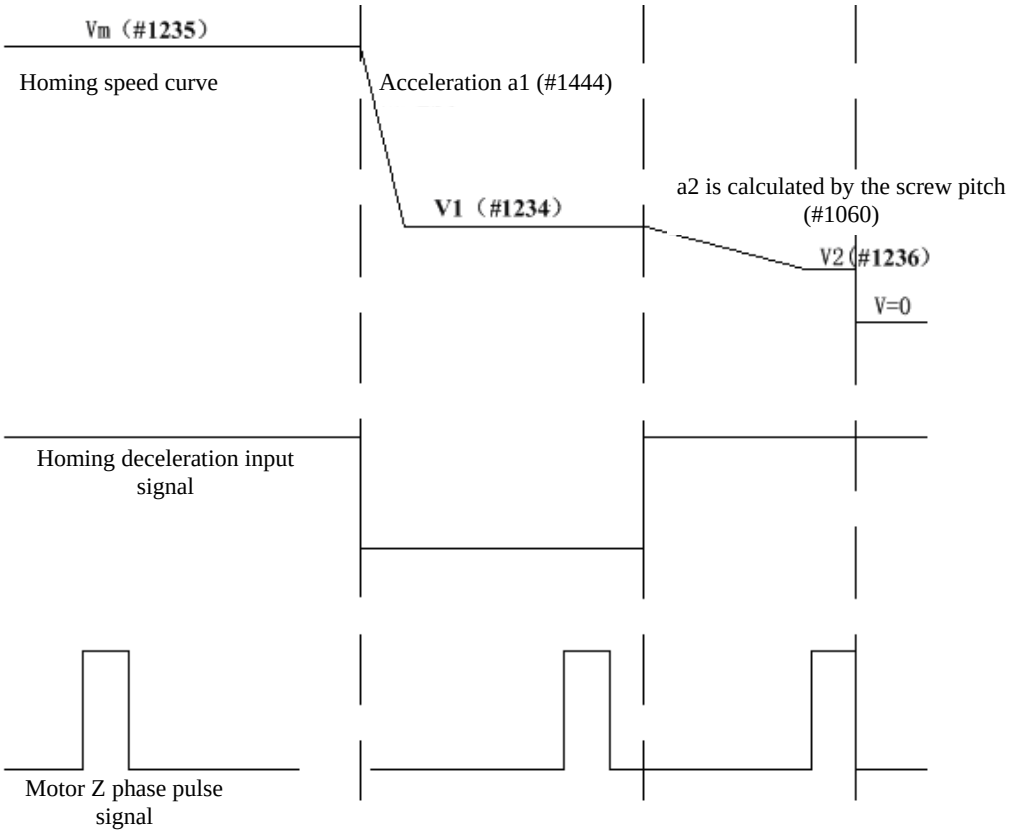
Note 3: After machine homing, the indicator goes out in the following case: When operator moves corresponding axis out of the machine reference point.

Note 4: The machine homing operation may vary from different manufacturers, so please refer to Operating Manual of the machine manufacturer.

8.3 Homing debug method

1) Relevant homing parameters:
   Homing direction setting (#1004)
   Movement amount per revolution for each axis (#1060)
   FL speed for reference point return of each axis (#1234)
   Speed for reference point return of each axis (#1235)
   Second FL speed for reference point return of each axis (#1236)
   Mechanical homing acceleration (#1444)

2) Schematic diagram for homing process:



3) Homing parameter adjustment step:

a) Confirm the homing direction according to the machine status (#1004)

b) Confirm the movement amount per revolution for each axis according to the machine status (#1060)

c) Set the homing speed and acceleration

Reference point return speed for each axis (#1235) (default value: 10000 for linear axis, 8000 for rotation axis).

FL speed (#1234) for each axis to return to the reference point (default value: 300 for linear axis, 75 for rotation axis).

Second FL speed for each axis to return to the reference point (#1236) (default value: 7 for linear axis, 7 for rotation axis)

Machine homing acceleration (#1444) (default value: 0.139 for linear axis, 80 for rotation axis).

d) Confirm whether the reference point return deceleration input signal of each axis is normal

Enter the PLC diagnosis interface and select X resource.

Move the machine manually; when the machine passes file block of the reference position, check that whether there are corresponding changes of the reference point return acceleration signal (X9.0 X9.4) input.

e) Conduct the separate homing operation for each axis

Observe reference point return speed (#1235), FL speed (#1234), and second FL speed (#1236) of each axis and confirm whether such operation has maintained for over two seconds.

If the holding time is less than 2 seconds in the reference point return speed (#1235) stage, you need to manually move the axis away from the reference point in the opposite direction of the reference point return direction. The reference point return speed (#1444) parameter needs to be increased if it reaches the moving limit.

If the reference point return FL speed (#1234) has maintained for less than two seconds, the homing acceleration (#1444) parameter should be increased. If the increase of acceleration does not work, the stopper may be too short and the reference point return speed (#1235) and FL speed (#1234) should be

decreased.

If the reference point return second FL speed (#1236) has maintained for less than two seconds, the movement amount per revolution of motor should be confirmed first. If it is correct, the Z-phase signal may have gone wrong.

Chapter IX System Communication

This system can communicate with PC via Ethernet, but it can also read USB equipment. The specific operations are detailed as follows:

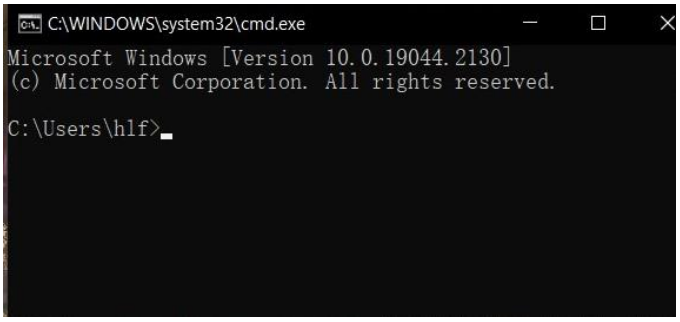
9.1 Network communications

The GSK 25iMc system can achieve the remote DNC and document transmission function via Ethernet. This software is designed for Windows system and is compatible with Win98, WinMe, WinXP, Win2000, Win7 and Win8.

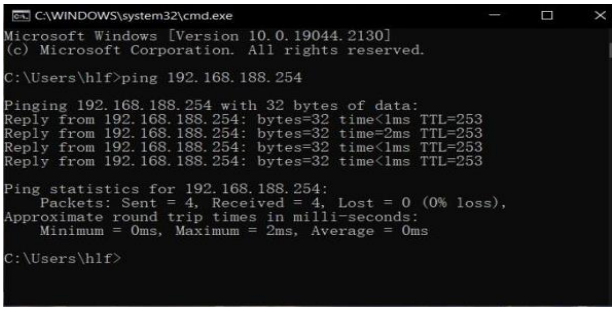
9.1.1 Network connection diagnosis

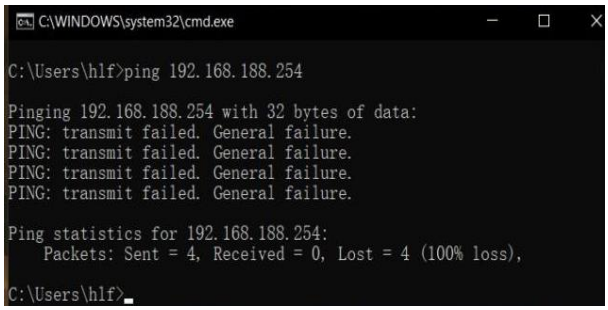
To check whether the network communication between the Windows PC and the CNC is smooth, the user may make judgement and resolve problems according to following steps.

1. Click the "Start" key, then the "Accessory" menu, and next the "Command Prompt". The pop-up window is similar to the following figure.



2. Input the ping command + space + equipment IP address + enter to test whether network communication is smooth. For example, after entering ping 192.168.188.254, it is correct if the following figure on the left side appears, and incorrect if the following right figure on the right side appears.





3. If the ping fails

(1) First, check whether the two indicator lights on the network interface of the CNC front panel shine or flash; if not, check whether the network cable is connected and make sure that it is connected, then ping again after two indicator lights shine;

(2) Log onto the machine manufacturer permissions on the CNC and modify the fourth segment of the IP address or the #130 parameter. For instance, change 192.168.188.123 to be 192.168.188.124, and ping the new IP address again;

(3) Close the Windows firewall and ping again; the firewall must be closed during the network communication of the dedicated CNC software, then it can be restored after that communication;

(4) Close or quit any antivirus, security guard and network management software installed in the Windows system and ping again; such kind of software must be closed during the network communication of the dedicated CNC software, then they can be restored after that communication;

(5) If the PC and the CNC are not connected point-to-point, that is, they are connected in a distributed manner through multiple switches or other devices, please ask the IT department which builds up the network to check the network routing settings or network behavior management. The relevant IT personnel must open the corresponding IP segment and network communication with the port, and then ping again. If the dedicated CNC software is no longer used for network communication, the relevant network settings can be restored.

9.1.2 Use of a helper

Launch the "25i Helper" shortcut on the desktop directly, and after startup, the software interface is illustrated in Figure 9-1-1.

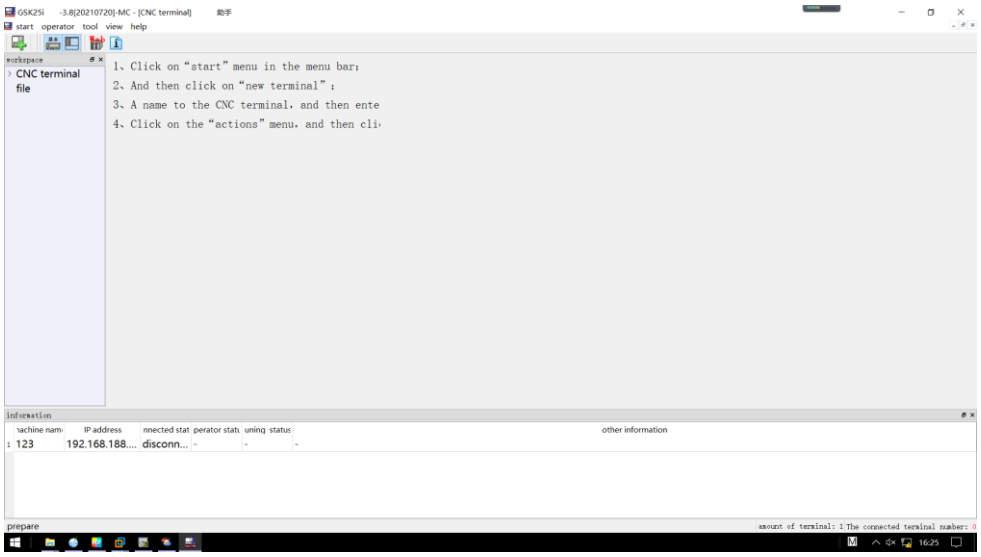


Figure 9-1-1

9.1.2.1 Connection of the CNC system

1. Connect the GSK 25i Mc/GSK25iTc CNC system and the PC with a network cable.
2. Set the correct CNC parameters and IP addresses (the IP addresses of the CNC #127~#130 parameters must be the same as those set in the transmission software; also, the IP of the CNC system much be located in the same network segment with that of the PC, i.e. the former three IP segments must be the same, and only the last segment may be different).
3. For the first use, select "Create New Terminal" from the system menu to connect to CNC; after selecting the corresponding terminal, press "Connect to Terminal" to connect to the CNC system.

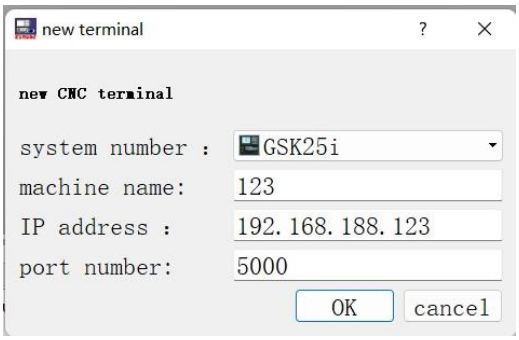


Figure 9-1-2

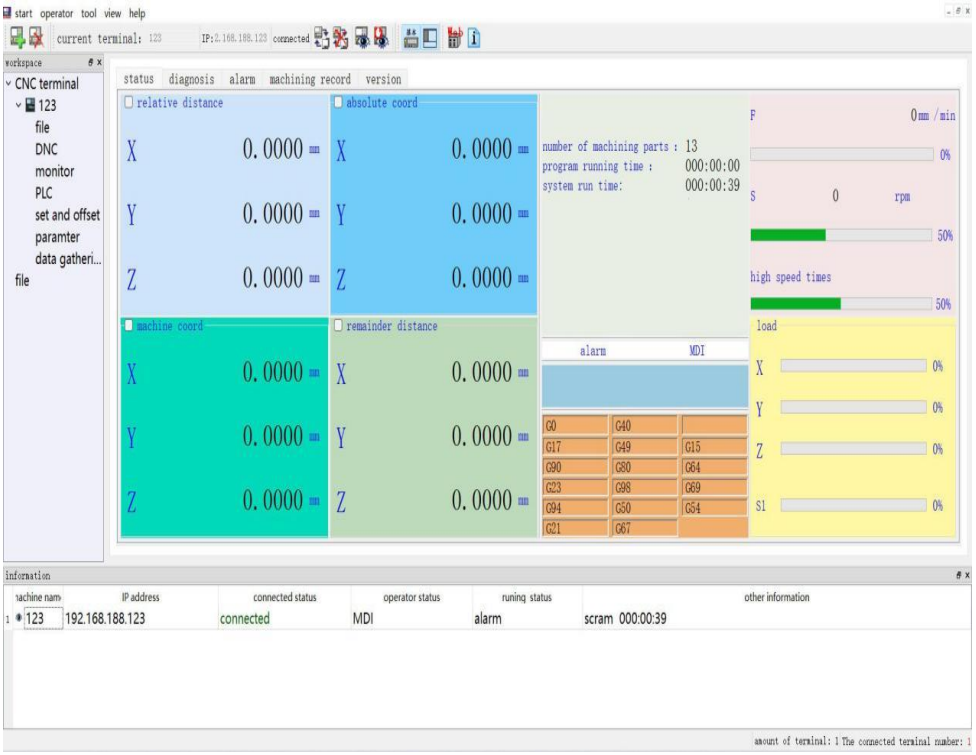


Figure 9-1-3

9.1.2.2 Network DNC

1. Select DNC on the PC software, click "File" button and select the NC file for DNC machining. Make sure the file content is read only and displayed in the software. The file path will be displayed on the left of the button.
2. Click "Send" to begin the data network transmission. The transmission can only be launched when current operation mode is DNC without alarm, or a message shall be prompted.
3. Press the "Cycle Start" button on the CNC panel, and the system will run the program and start machining.

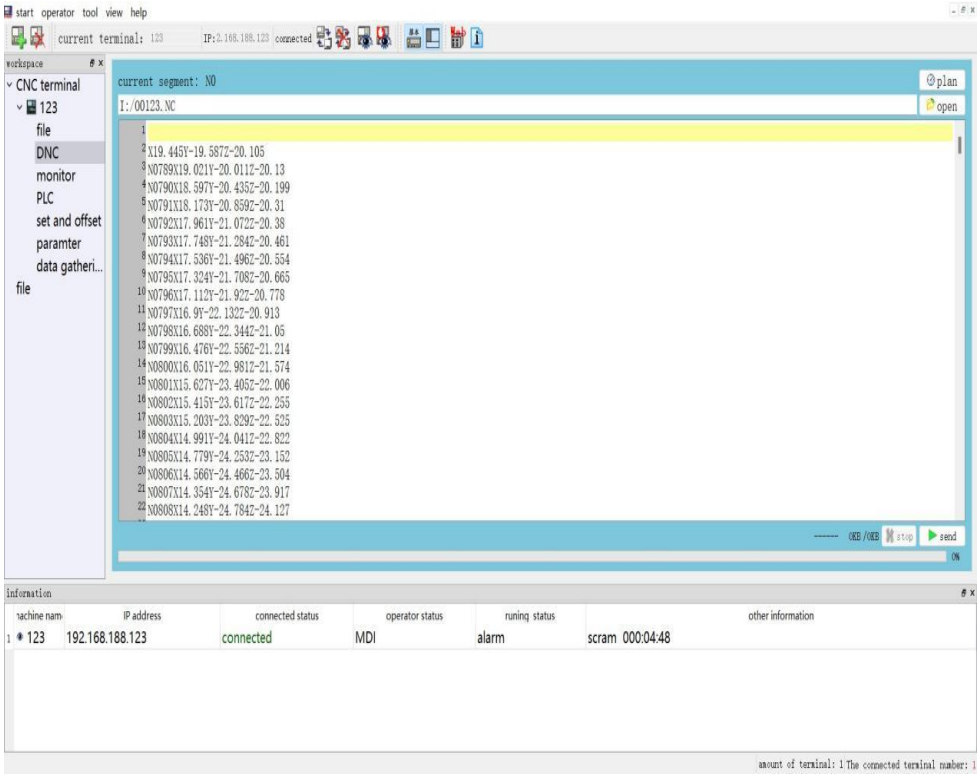


Figure 9-1-4

Precautions

(1) If G code loaded on the CNC is deleted from the transmission software, the PC software has no response and the system prompts "Current File Cannot Be Deleted" alarm.

(2) During the network DNC, #10 and #130 parameters cannot be modified.

(3) After stopping the DNC during the network DNC machining, the machine must be reset before transmitting other files.

(4) During the network DNC machining, M99 and M30 codes will erase the display after DNC operation finishes.

(5) If the main program of DNC calls a subprogram, but the subprogram contains M99P, an alarm message will be prompted.

(6) During the download or upload of files on the transmission software, the network DNC and program deletion are invalid.

9.1.3 File transmission

After selecting the "File" on the PC software and then the corresponding program, PLC and parameters, operate according to the corresponding upload, download, deletion and refresh function keys.

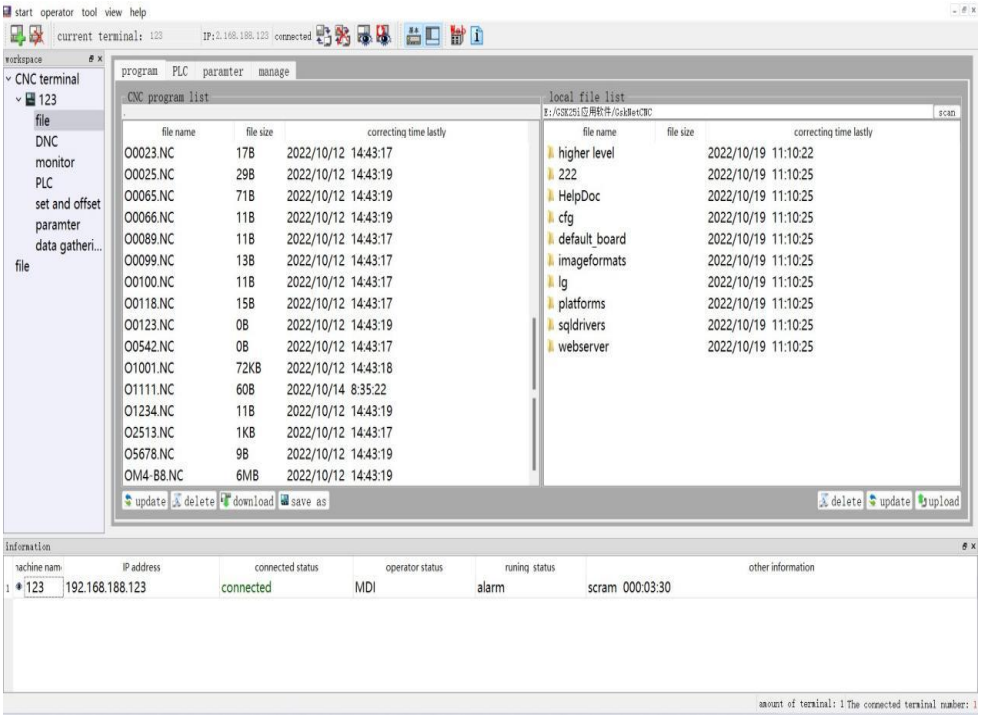


Figure 9-1-5

9.2 U disk communication

9.2.1 USB flash drive DNC machining

Under the DNC operation mode, it is possible to launch programs stored in the USB flash drive or the local directory for machining.

Precautions

(1) The difference between DNC and automatic operation is: automatic operation adopts single loading and the maximum size of a NC program is with 8 M due to system resource limit; on the other hand, network DNC can execute up to 200M NC programs.
(2) During DNC machining, jump command is prohibited.
(3) Please do not pull out the USB equipment during writing or reading to avoid serious consequences.

9.2.2 Transmission between USB flash drive and CNC

For relevant permission requirements and detailed operation instructions for transmission between a USB flash drive and the CNC, please refer to Table 9-2-1.

Table 9-2-1

Transmission Conditions		Associated Operation	
		CNC to USB Flash Drive	USB Flash Drive to CNC
Transmitting a single program (program name smaller than O8000)	Permission requirements	For terminal management password or higher levels, the operator should refer to the sixth digit permission setting	For terminal management password or higher levels, the operator should refer to the sixth digit permission setting
	Current CNC operating mode	Editing mode	Editing mode
	USB flash drive transmission requirements	USB flash drive must be loaded	USB flash drive must be loaded
	Programs without overlapping names	Direct transmission	Direct transmission
	Programs with overlapping names	Pop-up message for replacement or cancel	Pop-up message for replacement or cancel
Transmission of a single program (program name is O8000~O8999)	Permission requirements	Terminal management password or higher levels	Terminal management password or higher levels
	Current CNC operating mode	Editing mode	Editing mode
	Relevant parameter settings	Parameter 1610.0=0	Parameter 1610.0=0
	USB flash drive transmission requirements	USB flash drive must be loaded	USB flash drive must be loaded
	Programs without overlapping names	Direct transmission	Direct transmission
	Programs with overlapping names	Pop-up message for replacement or cancel	Pop-up message for replacement or cancel
Transmission of single program (program name is O9000-O9999)	Permission requirements	Terminal management password or higher levels	Terminal management password or higher levels
	Current CNC operating mode	Editing mode	Editing mode
	Relevant parameter settings	Parameter 1610.4=0	Parameter 1610.4=0
	USB flash drive transmission requirements	USB flash drive must be loaded	USB flash drive must be loaded
	Programs without overlapping names	Direct transmission	Direct transmission
	Programs with overlapping names	Pop-up message for replacement or cancel	Pop-up message for replacement or cancel

Table 9-2-1 has listed all parts programs for transmission in CNC; however, if their names are larger than or equal to 8000, the relevant parameters must be modified before transmission. If there is any program with a overlapping name during transmission, the system will prompt message for replacement or cancel.

Notes: (1) It supports hot plug equipment, but USB flash drives with larger capacity require longer reading access time.

(2) During loading, reading and wring data, it is not normal operation to plug or unplug the USB flash drive, which

may affect the normal operation of system.

9.2.3 Transmission of ladder diagram files between USB flash drive and CNC

The transmission of ladder diagram files between the current directory of USB flash drive and that of the CNC is shown in Table 9-2-2.

Table 9-2-2

Transmission Conditions		Associated Operation	
		CNC to USB Flash Drive	USB Flash Drive to CNC
Permission requirements		Machine manufacturer password or higher level	Machine manufacturer password or higher level
Current CNC operating mode		Under the editing mode	If it shares the same name as the current PLC file, PLC must stop; other files can handled at any mode
USB flash drive transmission requirements		USB flash drive must be loaded	USB flash drive must be loaded
To transmit ladder diagram files	Files without overlapping names	Direct transmission	Direct transmission
	Files with overlapping names	Overwrite directly, without prompt message	Overwrite directly, without prompt message

Table 9-2-2 shows that in order to transmit ladder diagram files between the current directory of a USB flash drive and that of the CNC, the permission level should be the machine manufacturer password or higher level.

If programs have no overlapping names, the ladder diagram transmission will be carried out directly; if there are overlapping names, the system will overwrite directly, without any prompt message.

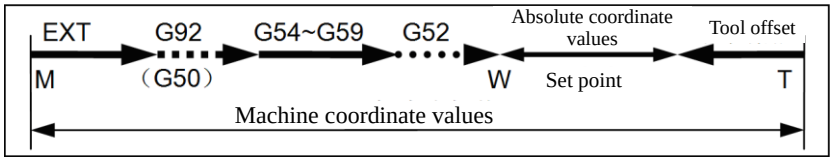
Note:

(1) If the USB flash drive is not correctly uninstalled, previous USB flash drive operation may be invalid.

(2) During loading, reading and wring data, it is not normal operation to plug or unplug the USB flash drive, which may affect the normal operation of system.

Chapter X Workpiece Coordinate System Setting

10.1 Coordinate system chain



Machine coordinate value = EXT+G92 (G50) offset + Workpiece coordinate system origin + G52 offset + Absolute coordinate value + Tool compensation

Note: G50 is adopted by G codes, A series in turning system.

10.2 Workpiece coordinate system origin setting

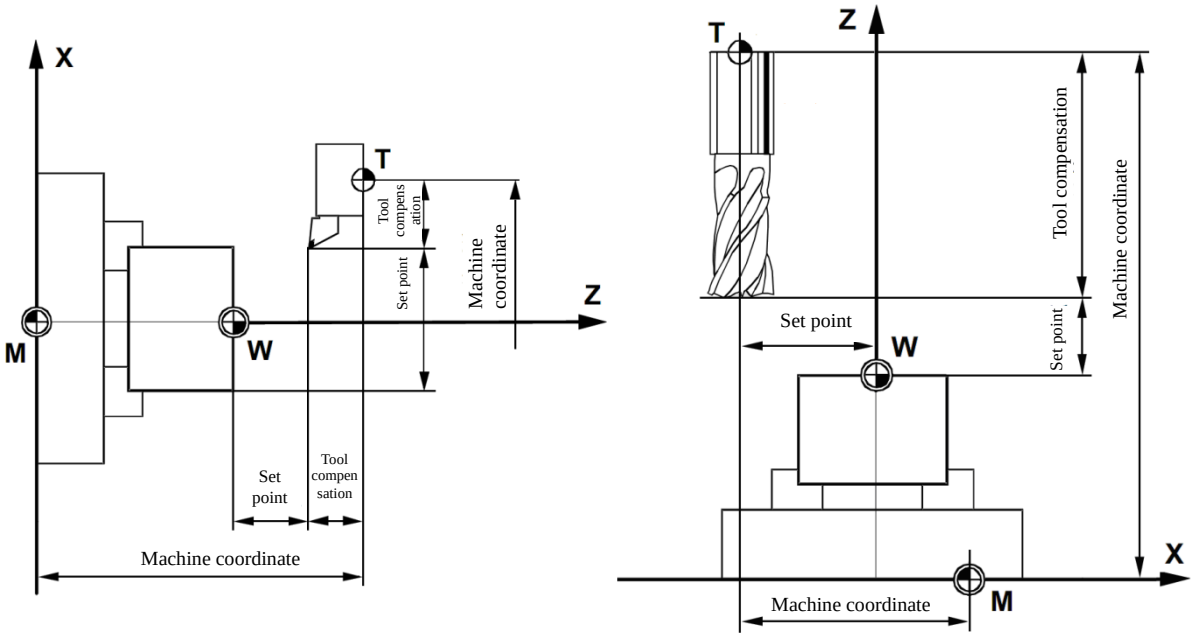
If the origin point of the workpiece system is set at point W, the tool nose should be moved to a measurable value setting point; choose a workpiece system from the list page, input the setting value and press the [Measure] key ([Measure with Tool] key for the milling system).



➔ COORD : Move the yellow cursor to the selected workpiece system coordinate axis ➔ Input the setting point value in the input field ➔ C.IPT ([Measure with Tool] key for the milling system)

For example: Setting the origin point of the G54 workpiece coordinate.

Move the tool nose to the diameter of the workpiece; assume that its diameter measured is Φ50mm, then move the cursor on the workpiece system page to the G54 X field, enter 50 in the input field and press the [Measure] key to complete the measurement of the origin point of the G54 X workpiece coordinate system. Measurement of G54 Z coordinate system origin shares the same procedure.



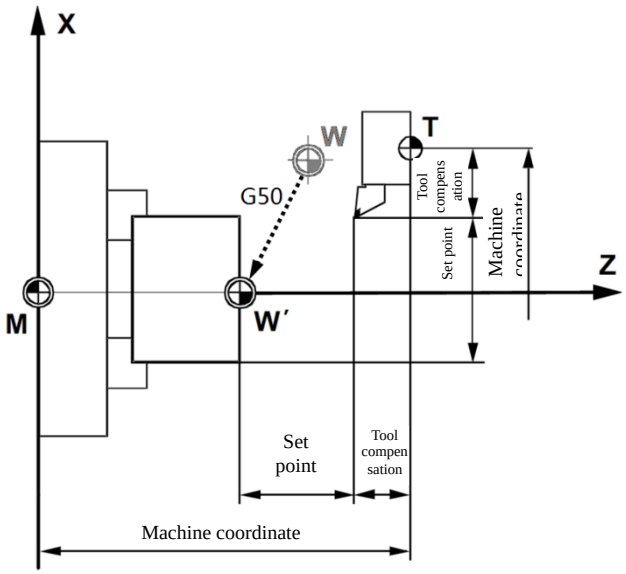
10.3 Offset of workpiece coordinate system

When the origin point of the workpiece coordinate system deviates from the original point W to a new point W', the tool nose may be moved to a measurable value setting point in the W' coordinate system. Obtain the coordinate value of setting point in the W' coordinate system and execute through G92 (G50) X_Z_ command in the system. The system will calculate the G92 (G50) offset according to the coordinate system chain and execute the offset, so as to deviate workpiece coordinate system origin from W to W'.

The workpiece origin is deviated from point W to point W', and the offset will be maintained until the system is powered off; the G92 (G50) offset will be reset to 0 when the system is powered on again.

Turning system example:

Move the tool nose to some diameter of the workpiece; assume that the diameter is measured as Φ50 mm and the distance away from the end face of workpiece is 50 mm in the Z-direction, input the program G50 X50 Z50 in MDI and press the [Cycle Start] key to execute the program block, and the origin point of the workpiece coordinate system will be deviated from point W to point W'. This program can be divided into two blocks, G50 X50 in the front and followed by G50 Z50. Of course, the program block can also be input under the editing mode and executed in automatic mode.



10.4 Tool offset measurement

Move the tool nose to a measurable value setting point and enter the corresponding tool compensation number on the offset page; then input the setting point value, press the [Measure] sub-soft key and the system will calculate the tool offset compensation value based on the coordinate system chain.

Press the ➔ [Offset] soft key of the 

function key, move the yellow cursor to the input field in the corresponding offset table, ➔ enter the value of the setting point, and ➔ press the [Measure] sub-soft key.

For example: Set No.1 tool offset.

Move the tool nose to some diameter of the workpiece; assume that its diameter is measured as Φ50 mm, move the cursor on the workpiece system page to the X offset field of No.1 tool, enter 50 in the input field and then press the [Measure] key to complete the measurement of No.1 tool's X offset. The Z offset of No.1 tool may be measured with the same method.

Appendix

Appendix

Appendix I G code programming rules

1. It is possible to command multiple G codes from different groups in the same program block. If the same program block has commanded multiple G codes from the same group, only the last G code is executed.
   For example: N10 G00 G01 X10.; G01 is valid
   N11 G00 G02 I5.; G02 is valid
   N12 G90 X10. G91 X20.; G91 valid
2. If the modal G code and non-modal G code are in the same block, the non-modal G code command shall be given priority.
3. In the G02 G03 arc commands, if the addresses I, J and K are commanded at the same time, the arc specified by the address R shall be given priority to.
   For example: G02 I10. R20.; R is valid, I is invalid
4. If a single program block contains multiple X, Y, Z, A, B, C, I, J, K, R, U, V, W, P, L, H, D, Q, E, O, N and S similar address commands, only the last command is valid.
   For example: N12 G90 X10. G91 X20.; G91 is valid, X20 is valid, and X10 in the front is invalid
5. The corresponding drilling data should be specified when the fixed cycle begins; if it is not specified, the machine will only position to the initial point without drilling.
   For example: G73 X100. Y200.; Z R Q, there is no drilling data specified, so the machine will only position to (100. 200.) without drilling.
   G80;
6. The creation and cancellation of radius compensation are only valid for moving command blocks (G00/ G01); creating or cancelling tool compensation on G02/ G03 will trigger a PS0034 alarm.
7. The macro program jump command and circle command should occupy a single line each, or it will cause PS0129 alarm.
8. When M codes (M00, M01, M05, M09, M30, M02) are executed together with other codes, other commands will be executed first.
   For example: G83 X100. Z-50. R5. Q2. M5 M9; execute M05 M09 after the punching action in the current block
9. In the automatic mode, M30 returns to the starting position of the program while M02 does not return to the starting position; but in the MDI mode, both of them will return to the starting position. % indicates that it does not return to the starting position at the end of the program.
10. When the GSK25iMc/GSK25iTc system alarms, the cursor will stop at the current block regardless of whether the current block has finished running.
11. When the system is running a machining program, if you manually modify shape, wear value, workpiece system, macro variables or some parameters, the system may not take effect immediately, so it should be noted that when these values are modified, it is better to reset the cursor. Move to the corresponding G-code and rerun.
    When modifying the current workpiece manually or through G10, both absolute coordinate and relative coordinate cannot be refreshed immediately; they can only be refreshed during execution.
12. If the calculation result of macro variables overflows, it displays the empty status. But the alarm will prompt "Operation Data Overflow". The macro variables 500-999 are global variables that will be initialized as 0 when the system starts.
13. For the skip symbol "/", the skip signal must be lighted up in the first five blocks of the skip line,

otherwise the signal may not be captured.

14. When calling the tool changing program, it is recommended to cancel G16, G68, G51 and G51.1.

Appendix II System Alarm List

2.1 System alarm (PS alarm)

Alarm number	PS0002	Alarm messages	Need to restart after modifying the parameters to continue the operation
Analysis	Appear after modifying some system parameters. For example, 200-206, 800, 811, 1000, 1004, 2170-2172, 2800, 2806.		
Handling method	Need to restart to continue the operation.		
Alarm number	PS0003	Alarm messages	Too many digits
Analysis	Program data input exceed the scope of system allowable value, for example: coordinate value X 999999.9999 is the maximum value in the machining program, and the system will give an alarm when this value is exceeded.		
Handling method	Check the data in the program to change that with too long digit to the appropriate value.		
Alarm number	PS0004	Alarm messages	Address not found
Analysis	Fail to enter the correct character address at the beginning of program block. Instead, enter number or illegal character.		
Handling method	Check the modifying program of the appropriate program block.		
Alarm number	PS0005	Alarm messages	No data after address
Analysis	No corresponding data is written after the address/command in the program, for example, command G, X, Y, Z, M appear separately.		
Handling method	Check the program addresses/commands and complete the data with commands that omit to write data.		
Alarm number	PS0006	Alarm messages	Illegal use of minus sign
Analysis	A minus sign is entered or multiple minus signs are entered where a minus sign cannot be used.		
Handling method	Modify the program		
Alarm number	PS0007	Alarm messages	Illegal use of decimal point
Analysis	There are extra decimal points in the program or decimal points are used where they cannot be used.		
Handling method	Check the program to remove the extra decimal point.		
Alarm number	PS0009	Alarm messages	Enter illegal character address
Analysis	There are unusable character addresses in the program.		

Handling method	Check the program and modify the appropriate address.		
Alarm number	PS0010	Alarm messages	Incorrect G-code
Analysis	G-code that the system does not have or that cannot be used is assigned in the program.		
Handling method	Check the program to change the wrong G-code to be a correct one.		
Alarm number	PS0011	Alarm messages	No feed speed command
Analysis	Feed speed is not specified in the cutting feed or is improper.		
Handling method	Modify the program		
Alarm number	PS0012	Alarm messages	Address P is reused, with program modified
Analysis	Two or more addresses P appear in the same program block.		
Handling method	Separate the commands that can be used for address P into different program blocks.		
Alarm number	PS0013	Alarm messages	Tool compensation method cannot specify unidirectional positioning
Analysis	There is also unidirectional positioning command G60 in the program block for establishing the tool compensation.		
Handling method	Modify the program, and G60 cannot be written in the same line with the tool compensation command.		
Alarm number	PS0014	Alarm messages	G10 command format error
Analysis	Online modification of system parameter (G10), programming format error.		
Handling method	Modify the program to correct the G10 format.		
Alarm number	PS0015	Alarm messages	Too many axes commanded
Analysis	Axis that does not exist or is not set by the system is specified in the program.		
Handling method	Check program, delete redundant axis commands or check system parameter 1020 (whether the system parameter 800 is set correctly), and check whether the corresponding axes name is correct.		
Alarm number	PS0016	Alarm messages	G10 does not allow the modification of such parameter
Analysis	The parameters that can be modified through G10 are: 1, 130, 1020-1021, 1031-1053, 1605-1642, 1801-1930, 1933-2034, 2112, 2113, 2600-2653. An alarm will be given if other parameters are specified.		
Handling method	Modify the program to delete parameter numbers other than those that can be modified by G10 command.		
Alarm number	PS0018	Alarm messages	Wrong plane selection when commanding planar circular arc in length compensation
Analysis	When the length compensation is established in the program, the plane where the arc path is located is incorrectly selected.		

Handling method	Modify the program to select the correct plane to use the circular arc command.		
Alarm number	PS0019	Alarm messages	Too many linkage movement axes commanded
Analysis	The number of linkage movement axes commanded in the same program exceeds the limit.		
Handling method	Modify the program		
Alarm number	PS0020	Alarm messages	Out of radius tolerance
Analysis	In circular interpolation, the distance from the start point to the end point cannot form a circular arc at the given radius R or the distance from the beginning to the arc center is different from that from the end point to the circular arc center.		
Handling method	Modify the program		
Alarm number	PS0021	Alarm messages	Illegal plane axis commanded
Analysis	Axis not in the (G17 G18 G19) plane is specified in the circular interpolation.		
Handling method	Modify the program		
Alarm number	PS0022	Alarm messages	No arc radius
Analysis	The program does not specify a arc radius where the arc radius command is required.		
Handling method	Add the arc radius command to the program.		
Alarm number	PS0024	Alarm messages	Input data out of range assigned by the system
Analysis	The data entered by G10 is out of system range.		
Handling method	Modify the program		
Alarm number	PS0025	Alarm messages	The pitch compensation number is illegal or does not exist or the pitch compensation value is not specified
Analysis	When using G10 to enter pitch compensation, the specified pitch compensation number does not exist or is illegal, and the pitch compensation value is not specified.		
Handling method	Modify the program		
Alarm number	PS0026	Alarm messages	G51 cannot be in the same block with other commands
Analysis	G51 is not allowed to be in the same block with G10 and G65.		
Handling method	Modify the program, and write it line by line.		
Alarm number	PS0027	Alarm messages	The parameter number modified by G10 is illegal or does not exist
Analysis	A parameter that does not exist in the system or an incorrect parameter number is specified in G10.		

Handling method	Modify the program		
Alarm number	PS0028	Alarm messages	Illegal plane selection
Analysis	Plane switching is not possible in fixed cycle modal.		
Handling method	To modify the program, please cancel the fixed cycle, then switch planes or modify # 2000.0 parameter.		
Alarm number	PS0029	Alarm messages	Illegal offset value
Analysis	The compensation value specified by H is either too large or does not exist.		
Handling method	Modify the program		
Alarm number	PS0030	Alarm messages	Illegal compensation number
Analysis	The compensation number specified by D/H code is too large or does not exist. In addition, the value specified by the additional workpiece coordinate system number specified by P code exceeds the value specified by the system.		
Handling method	Modify the program		
Alarm number	PS0031	Alarm messages	Illegal P commanded in G10
Analysis	The data specified by address P in G10 command is incorrect.		
Handling method	Modify the program to check whether the relevant data number to be modified online is correct.		
Alarm number	PS0032	Alarm messages	Illegal compensation value in G10
Analysis	When the offset value is set by G10 or written by the system variable, the offset value exceeds the range assigned by the system.		
Handling method	Modify the program		
Alarm number	PS0033	Alarm messages	Tool compensation does not find the inter-block point
Analysis	In tool compensation, the end point coordinate of the previous program block is not on-line in the next one.		
Handling method	Modify the program or check the tool compensation value.		
Alarm number	PS0034	Alarm messages	Cannot lift tool or cancel tool compensation in circular arc command
Analysis	Cannot establish or cancel tool compensation in circular arc command.		
Handling method	Modify the program		
Alarm number	PS0036	Alarm messages	Tool compensation mode cannot command G31 and G37
Analysis	G31 and G37 commands are not allowed in tool compensation.		
Handling	Check whether there are G31 and G37 commands in the program of tool compensation, and		

method	modify the program.		
Alarm number	PS0037	Alarm messages	Switching plane in tool compensation
Analysis	Switching plane (G17 G18 G19) is not allowed in tool compensation.		
Handling method	Check whether the program in the tool compensation switches planes, and modify the program.		
Alarm number	PS0038	Alarm messages	There is interference in the circular arc program block
Analysis	Overcutting occurs in the tool radius compensation method because the circular arc start point or endpoint is the same as the circular arc center.		
Handling method	Modify the program		
Alarm number	PS0039	Alarm messages	The arc end point is not on the circular arc after the tool compensation
Analysis	The arc command is after the tool compensation, and #1810 parameter determines whether an alarm is given if the end point is not on the circular arc.		
Handling method	Modify the program		
Alarm number	PS0040	Alarm messages	The tool compensation is changed in circular interpolation
Analysis	It is not allowed to change the radius compensation in the tool radius compensation mode.		
Handling method	Check whether there is D value in the program that modifies the compensation.		
Alarm number	PS0041	Alarm messages	There is interference in CRC
Analysis	Overcutting occurs in the tool radius compensation mode.		
Handling method	Modify the program		
Alarm number	PS0042	Alarm messages	Commanding G45-G48 is not allowed in the tool compensation
Analysis	Command G45-G48 is used in tool compensation.		
Handling method	Modify the program to delete G45-G48 command.		
Alarm number	PS0043	Alarm messages	Radius value out-of-tolerance
Analysis	#1810 parameter determines whether an alarm is given if the end point is not on the arc.		
Handling method	Modify the program		
Alarm number	PS0044	Alarm messages	Commanding G27-G30 G53 is not allowed in the fixed cycle
Analysis	There are G27-G30 G53 commands in the fixed cycle.		
Handling method	Modify the program		
Alarm number	PS0045	Alarm messages	Address Q is not found in the fixed cycle

Analysis	In the fixed cycle G73/G83, no cutting-in (Q) is specified each time.		
Handling method	Modify the program		
Alarm number	PS0046	Alarm messages	Illegal homing command
Analysis	In the 2nd, 3rd and 4th reference point returning commands, commands other than P2, P3, and P4 are given.		
Handling method	Modify the program		
Alarm number	PS0047	Alarm messages	G10 and the fixed cycle cannot be used at the same time
Analysis	G10 command is used in the fixed cycle.		
Handling method	G10 command in the fixed cycle is deleted or the fixed cycle must be cancelled before G10 is executed.		
Alarm number	PS0048	Alarm messages	Metric-imperial conversion command format is not correct
Analysis	Metric-imperial conversion can only be switched at the program header. Switching during program execution or subprogram is not allowed.		
Handling method	Modify the program		
Alarm number	PS0051	Alarm messages	Moving error after CHF/CNR
Analysis	The filleting or chamfering conditions do not exist or the amount of movement is uncertain.		
Handling method	Modify the program		
Alarm number	PS0053	Alarm messages	This condition cannot be determined or does not exist in the transfer type determination
Analysis	The four types (L_L, L_C, C_L, C_C) assigned by the system cannot be determined.		
Handling method	Modify the program		
Alarm number	PS0055	Alarm messages	The number of movement commands is less than 2 from the establishment to the cancellation of the tool compensation
Analysis	After establishing tool compensation, the tool compensation is cancelled due to the specified movement command less than two blocks.		
Handling method	Modify the program		
Alarm number	PS0056	Alarm messages	Wrong definition of parallel axis command
Analysis	Parallel axis specified by the parallel axis function exceeds the maximum axis number		
Handling method	Modify program		
Alarm number	PS0057	Alarm messages	Wrong definition of linkage axis command
Analysis	Linkage axis designated in the linkage axis function exceeds the maximized axis number		

Handling method	Modify program		
Alarm number	PS0058	Alarm messages	Illegal command of linkage axis
Analysis	Linkage axis and synchronic axis are conflict in the command of linkage function		
Handling method	Modify program		
Alarm number	PS0059	Alarm messages	Program number not found
Analysis	The specified program number is not found in the external program number retrieval or external workpiece number retrieval. Either the specified program is edited in the background or no non-modal macro program calls the program in memory.		
Handling method	Check program number and external sequence number.		
Alarm number	PS0063	Alarm messages	The S-command is zero or illegal
Analysis	S code command value exceeds the range assigned by the system		
Handling method	Check the program		
Alarm number	PS0064	Alarm messages	Illegal M-code command
Analysis	The assigned M-code is out of range or too many M-codes are assigned in the same block.		
Handling method	Modify the program		
Alarm number	PS0065	Alarm messages	Program block is too long
Analysis	Program block exceeds 32 units at maximum.		
Handling method	Modify the program		
Alarm number	PS0066	Alarm messages	A unit string is too long
Analysis	The number of unit characters exceeds the maximum number of characters.		
Handling method	Modify the program		
Alarm number	PS0067	Alarm messages	Illegal sequence number
Analysis	N sequence number is out of system range.		
Handling method	Modify the program		
Alarm number	PS0068	Alarm messages	P/X dwell time is illegal or times out
Analysis	Dwell time is out of system range.		
Handling method	Modify the program		
Alarm	PS0069	Alarm	Incompatible NC command

number		messages	
Analysis	Coordinate values conflict.		
Handling method	Check the program, and modify the coordinate value.		
Alarm number	PS0070	Alarm messages	Insufficient memory capacity
Analysis	The system storage capacity is full or the copied program files exceed the remaining capacity of the system.		
Handling method	Delete unnecessary program files or reduce the number of program files.		
Alarm number	PS0071	Alarm messages	Data not found
Analysis	No addresses to be searched is found. Or the specified program is not found in the program retrieval.		
Handling method	Check data.		
Alarm number	PS0072	Alarm messages	Program call error or M99 jump error, and modify the code
Analysis	M99 is used under commands or DNC that call multiple subprograms in the same block.		
Handling method	Modify the program		
Alarm number	PS0073	Alarm messages	Program number already in use
Analysis	There is program name that needs to be created in the program storage directory.		
Handling method	Change program name.		
Alarm number	PS0074	Alarm messages	Illegal program number
Analysis	Program number initial is not 'O' or exceeds the maximum range.		
Handling method	Modify the program		
Alarm number	PS0075	Alarm messages	Operation fails, without authority
Analysis	Insufficient authority to modify system parameters. Please enter password.		
Handling method	Enter the password		
Alarm number	PS0076	Alarm messages	Address P is not defined or illegal
Analysis	Address P is lacked in the program block of M98, G65 and G66.		
Handling method	Modify the program		
Alarm number	PS0077	Alarm messages	Subprogram nesting error
Analysis	Subprogram calling nest exceeds 8 loops.		

Handling method	Modify the program		
Alarm number	PS0078	Alarm messages	No corresponding program number or sequence number is found
Analysis	The program number and sequence number specified by address P are not found in the program block of M98, M99, G65, or G66. The sequence number specified by GOTO statement is not found or the called program is edited in background editing.		
Handling method	Modify the program or terminate background editing.		
Alarm number	PS0079	Alarm messages	Lathe and milling machine cannot be switched directly
Analysis	Lathe and milling machine can be switched only when the system is in a reset state.		
Handling method	Modify the program		
Alarm number	PS0080	Alarm messages	The automatic measurement start position of the tool is in the deceleration position
Analysis	The start position of the tool measurement is incorrect.		
Handling method	Modify the automatic measurement start position of the tool in the program.		
Alarm number	PS0081	Alarm messages	No compensation number is found in G37
Analysis	Tool compensation number is not specified prior to execution of G37 command.		
Handling method	Modify the program		
Alarm number	PS0082	Alarm messages	Commanding H code is not allowed in G37
Analysis	H code and G37 command cannot be specified on the same line.		
Handling method	Modify the program		
Alarm number	PS0083	Alarm messages	Illegal axis command in G37
Analysis	In automatic tool length measurement, an invalid axis is specified or the command is an incremental value.		
Handling method	Modify the program		
Alarm number	PS0084	Alarm messages	G37 without arrival signal
Analysis	In the automatic tool compensation function, the measurement position arrival signal is not output within the range assigned by the parameter.		
Handling method	Perform the correct setting and operation.		
Alarm number	PS0085	Alarm messages	USB error
Analysis	Incorrect input/output equipment		
Handling	Check setting of communication software or line		

method			
Alarm number	PS0086	Alarm messages	The second storage travel detection boundary assigned by G22 is incorrect
Analysis	The negative limit value is greater than the positive limit value in the second travel detection.		
Handling method	Modify the program		
Alarm number	PS0087	Alarm messages	Spiral or conical interpolation command is incorrect
Analysis	L or Q or H parameter in interpolation command is incorrect.		
Handling method	Check the program and modify the appropriate parameter value.		
Alarm number	PS0090	Alarm messages	Illegal G107 command
Analysis	The condition to create or cancel cylindrical interpolation is incorrect.		
Handling method	Modify the program		
Alarm number	PS0091	Alarm messages	Code that is not allowed is assigned in G107
Analysis	Code that is not allowed is commanded in cylindrical interpolation mode.		
	Please refer to 4.10 G07.1 Command "Limitations and precautions" in Part I of this manual for available G codes.		
Handling method	Modify the program		
Alarm number	PS0092	Alarm messages	G107 does not end normally or the tool compensation is not cancelled normally
Analysis	G107 cylindrical interpolation does not end normally.		
Handling method	Modify the program		
Alarm number	PS0093	Alarm messages	Fixed cycle command cannot be specified under G05
Analysis	Fixed cycle command is used in the program that G05 command takes effect.		
Handling method	Modify the program		
Alarm number	PS0094	Alarm messages	Another program call is not allowed in G66 modal
Analysis	Another subprogram call is not allowed in G66 modal.		
Handling method	Modify the program		
Alarm number	PS0095	Alarm messages	The polygon turning tool axis needs to be set to the rotation axis
Analysis	The polygon turning tool axis needs to be set to the rotation axis.		
Handling method	The axis number set by parameter 7610 must be set to 1 in parameter 1023.1.		
Alarm	PS0096	Alarm	The polygon cutting command is incorrect

number		messages	
Analysis	In the polygon turning mode, G51.2 is repeatedly commanded.		
Handling method	Modify the program		
Alarm number	PS0097	Alarm messages	G51.2 has no P/Q or command value is out of range
Analysis	There is no P/Q command or command value is out of range in G52.1.		
Handling method	Both P and Q are commanded in G51.2, and it is ensured that the spindle command speed*Q/P is less than parameter #7621		
Alarm number	PS0098	Alarm messages	Defined speed exceeds the valid range
Analysis	F speed specified by the feed/min or feed/rotate exceeds the specified range		
Handling method	Modify program		
Alarm number	PS0100	Alarm messages	The positioning spindle address and other axis movement commands are in the same program block
Analysis	The positioning spindle address set by the parameter cannot be in the same program block with other axis movement addresses.		
Handling method	Modify the program		
Alarm number	PS0101	Alarm messages	The specified data is out of valid range
Analysis	The relevant data is out of range assigned by the system.		
Handling method	Modify the program		
Alarm number	PS0102	Alarm messages	The withdrawal length at the end of thread is greater than the thread machining length of the long axis
Analysis	The withdrawal length at the end is greater than the thread machining length.		
Handling method	Modify the program		
Alarm number	PS0103	Alarm messages	Illegal thread command
Analysis	The number of imperial thread teeth is 0 or too large.		
Handling method	Modify the program		
Alarm number	PS0104	Alarm messages	There is interference at the start point of the program
Analysis	There is interference in the machining path.		
Handling method	Modify the program		
Alarm number	PS0105	Alarm messages	The fixed cycle specifies the same contour end point as the start point
Analysis	Fixed cycles G90, G92, G94 cannot have the same contour end point as the start point.		
Handling	Modify the program		

method			
Alarm number	PS0106	Alarm messages	R specified by fixed cycle does not constitute enclosure with the end point and start point
Analysis	The programming path does not form enclosure.		
Handling method	Modify the program		
Alarm number	PS0107	Alarm messages	Direct drawing dimension programming command format is incorrect
Analysis	Graphic dimension programming format is incorrect.		
Handling method	Modify the program		
Alarm number	PS0108	Alarm messages	Illegal condition in polar coordinate interpolation
Analysis	The condition is incorrect when polar coordinate interpolation starts or is cancelled:		
	1) The G12.1/G13.1 commands are not specified in a separate program block.		
	2) The G12.1/G13.1 commands are not cancelled under the mode of tool nose radius compensation (G40).		
	3) The G code limited by G12.1/G13.1 is used.		
	4) The plane selection command is executed under the mode of G12.1.		
	5) Parameters No. 5460 and No. 5461 are defined incorrectly.		
	6) The T command is executed in the polar coordinate interpolation mode and must be issued before the polar coordinate interpolation begins.		
Handling method	Check the above conditions and modify the program or parameter value.		
Alarm number	PS0109	Alarm messages	G161 command format error
Analysis	G161 format or parameter is incorrect.		
Handling method	Modify format or parameter value.		
Alarm number	PS0111	Alarm messages	Operational data overflow
Analysis	The results of the calculation are out of range allowed by the system.		
Handling method	Modify the program		
Alarm number	PS0112	Alarm messages	Divided by zero
Analysis	The specified divisor is zero (including tan 90°).		
Handling method	Modify the program		
Alarm number	PS0113	Alarm messages	Incorrect macro command
Analysis	Function commands that cannot be used are specified in the user macro program.		
Handling method	Modify the program		

Alarm number	PS0114	Alarm messages	Macro program expression format error
Analysis	Macro program expression format is incorrect.		
Handling method	Modify the program		
Alarm number	PS0115	Alarm messages	Illegal variable number is specified
Analysis	The macro variable 3000 assignment expression is out of range.		
Handling method	Modify the program		
Alarm number	PS0116	Alarm messages	There are no operands in the macro program expression
Analysis	Valid operands are required to be filled in macro program expression.		
Handling method	Modify the program		
Alarm number	PS0118	Alarm messages	Macro program bracket nesting is incorrect
Analysis	The bracket nesting in macro program exceeds the upper limit.		
Handling method	Modify the program		
Alarm number	PS0119	Alarm messages	Illegal macro variable number
Analysis	Value that cannot be used as variable number is used in the user macro program.		
Handling method	Modify the program		
Alarm number	PS0123	Alarm messages	Macro command is used in DNC
Analysis	There is macro command in the program that performs DNC machining.		
Handling method	Modify the program		
Alarm number	PS0124	Alarm messages	DO-END is not corresponded one to one
Analysis	In the macro program statement, DO-END is not corresponded one to one.		
Handling method	Modify the program		
Alarm number	PS0126	Alarm messages	Number of illegal cycles
Analysis	The number of cycles after DO-END is incorrect.		
Handling method	Modify the program		
Alarm number	PS0127	Alarm messages	NC command and macro program command are in the same block
Analysis	NC command and macro program command statement coexist.		
Handling	Modify the program		

method			
Alarm number	PS0128	Alarm messages	Illegal macro program sequence number
Analysis	The sequence number in the branch command is out of range assigned by the system or not retrieved.		
Handling method	Modify the program		
Alarm number	PS0129	Alarm messages	The macro program jump command cannot be in the same line as other commands
Analysis	The macro program jump command and other commands are in the same program block.		
Handling method	Modify the program		
Alarm number	PS0135	Alarm messages	Illegal angle command
Analysis	The indexing angle of the indexing worktable is not a multiple of the angle unit.		
Handling method	Modify the program		
Alarm number	PS0136	Alarm messages	Illegal axis command
Analysis	In the indexing of indexing worktable, the other axis is commanded together with B-axis.		
Handling method	Modify the program		
Alarm number	PS0140	Alarm messages	Incorrect program block address is specified in multiple cycles
Analysis	The starting program block number is greater than the ending program block number or there is no specified program block number.		
Handling method	Modify the program		
Alarm number	PS0141	Alarm messages	Incorrect G (or M) code is specified in multiple cycles
Analysis	G-code that is not allowed is specified in multiple cycles.		
Handling method	Modify the program		
Alarm number	PS0142	Alarm messages	Too many program blocks are specified in multiple cycles
Analysis	The number of program blocks specified in the multiple cycles is out of the range specified by the system.		
Handling method	Modify the program		
Alarm number	PS0143	Alarm messages	Coordinate monotonicity in multiple cycles is incorrect
Analysis	X or Z monotonicity (unidirectional increase or decrease) in multiple cycles is incorrect.		
Handling method	Modify the program		
Alarm	PS0144	Alarm	The first program block format of multiple cycles is

number		messages	incorrect
Analysis	Illegal code is specified in the starting program block of multiple cycles.		
Handling method	Modify the program		
Alarm number	PS0145	Alarm messages	Code format in multiple cycles is incorrect
Analysis	The machining path enters an abnormal state in multiple cycles.		
Handling method	Modify the program		
Alarm number	PS0146	Alarm messages	Machining path shape incorrect
Analysis	The machining path shape is incorrect in multiple cycles.		
Handling method	Modify the program		
Alarm number	PS0147	Alarm messages	The specified circular arc cannot be roughed
Analysis	Circular arc command is not allowed in rough machining.		
Handling method	Modify the program		
Alarm number	PS0148	Alarm messages	P/Q value is not specified (or specified as 0)
Analysis	Specified P/Q is illegal.		
Handling method	Modify the program		
Alarm number	PS0149	Alarm messages	The tool retraction amount is greater than infeed amount
Analysis	The tool retraction amount is greater than infeed amount.		
Handling method	Modify the program		
Alarm number	PS0150	Alarm messages	Illegal tool group number
Analysis	No tool number given or illegal tool number obtained.		
Handling method	Check whether the tool given by the program is correct.		
Alarm number	PS0154	Alarm messages	G51.1, G51, G68 cannot be in the same block
Analysis	G51.1, G51, G68 cannot be in the same program block.		
Handling method	Modify the program		
Alarm number	PS0155	Alarm messages	G51 can only take P parameter in G110-G137 modal
Analysis	G51 command can only take P parameter in G110-G137 special fixed cycle modal.		
Handling method	Modify the program		

Alarm number	PS0157	Alarm messages	The plane vector of the characteristic coordinate system is 0
Analysis	The plane vector of the characteristic coordinate system is 0.		
Handling method	Modify the program		
Alarm number	PS0158	Alarm messages	Machine type is not supported
Analysis	Machine type is not supported.		
Handling method	Modify parameter 8010.		
Alarm number	PS0160	Alarm messages	Level specification is incorrect
Analysis	Level for NURBS interpolation is incorrect.		
Handling method	Modify P value.		
Alarm number	PS0161	Alarm messages	Node has not been specified
Analysis	NURBS interpolation does not specify node.		
Handling method	Modify the program		
Alarm number	PS0162	Alarm messages	Node specification is incorrect
Analysis	Node range specified by NURBS interpolation is incorrect.		
Handling method	Modify the program		
Alarm number	PS0163	Alarm messages	Too many axes specified
Analysis	Extra axes are specified in the NURBS interpolation.		
Handling method	Modify the program		
Alarm number	PS0164	Alarm messages	The interpolation method that cannot be used simultaneously in NURBS interpolation method is specified
Analysis	The interpolation method that cannot be used simultaneously is specified in NURBS interpolation.		
Handling method	Modify the program		
Alarm number	PS0165	Alarm messages	G68.2 not found
Analysis	When specifying G53.1, it is found that it is not G68.2, G68.3, G68.4 modals.		
Handling method	Modify the program		
Alarm number	PS0166	Alarm messages	E, K in G124/G125 are undefined or 0
Analysis	The number of holes E on side 1, 3 and that of holes K on side 2, 4 in G124/G125 are		

	undefined or 0.		
Handling method	Modify the program		
Alarm number	PS0167	Alarm messages	The program code in the same block is unreasonable. Please write it in blocks
Analysis	Program codes that cannot coexist is used in the same program block.		
Handling method	Modify the program		
Alarm number	PS0170	Alarm messages	Fixed cycle drilling G73-G89 needs to be defined
Analysis	No fixed cycle drilling method G73-G89 is defined in G120-G125 continuous drilling cycles.		
Handling method	Fixed cycle drilling method G73-G89 is added in G120-G125 continuous drilling cycles.		
Alarm number	PS0171	Alarm messages	I is not defined or I is 0
Analysis	"I" is undefined or 0 in G110-G137 special fixed cycle.		
Handling method	Modify the program to add I or make I not 0.		
Alarm number	PS0172	Alarm messages	J is undefined or 0
Analysis	J is undefined or 0 in G110-G137 special fixed cycle.		
Handling method	Modify the program to add J or make J not 0.		
Alarm number	PS0173	Alarm messages	W is too small or undefined
Analysis	W is undefined or 0 in the rough milling cycle.		
Handling method	Modify the program to add W or make W not 0.		
Alarm number	PS0174	Alarm messages	Q is too small or undefined
Analysis	Q is undefined or 0 in the rough milling cycle.		
Handling method	Modify the program to add Q or make Q not 0.		
Alarm number	PS0175	Alarm messages	L is too small or undefined
Analysis	L is undefined or too small in the G110-G137 special fixed cycle and G126 plane milling.		
Handling method	Modify the program to add L or increase the value of L.		
Alarm number	PS0176	Alarm messages	V is too small or undefined
Analysis	V is undefined or 0 in the rough milling cycle.		
Handling method	Modify the program to add V or make V not 0.		
Alarm	PS0178	Alarm	I and J are too small or tool radius is too large

number		messages	
Analysis	I and J are too small or tool radius is too big in G110-G137 special fixed cycle.		
Handling method	Modify the program so that I and J increase or tool is changed.		
Alarm number	PS0179	Alarm messages	L is too big
Analysis	L is too big in rough milling cycle.		
Handling method	Modify the program to reduce L.		
Alarm number	PS0180	Alarm messages	U is smaller than D
Analysis	U is greater than D in the rectangular rough milling cycle.		
Handling method	Modify the program so that U is smaller than or equal to D.		
Alarm number	PS0181	Alarm messages	The number of holes in special fixed cycle is zero or does not exist
Analysis	The number of holes is zero or does not exist in G120-G125 continuous drilling cycle.		
Handling method	Modify the program to add the number of holes parameter or make it not 0.		
Alarm number	PS0182	Alarm messages	Parameters of main station and of slave station are inconsistent
Analysis	There are difference between parameters of main stations and of slave stations.		
Handling method	Modify parameters of main stations or of slave stations		
Alarm number	PS0183	Alarm messages	Special fixed cycle can only be performed in G17 plane
Analysis	G110-G137 special fixed cycle and G120-G125 continuous drilling cycle can only be performed in G17 plane.		
Handling method	Modify the program to G17 plane.		
Alarm number	PS0184	Alarm messages	Alarm file could not be opened or memory is inadequate, please modify the parameter
Analysis	Operating memory of CNC system is inadequate		
Handling method	Please adjust Parameter 101 and 102		
Alarm number	PS0185	Alarm messages	Corner radius U is too big or I and J are too small
Analysis	U is too big or both I and J are too small in G130-G137 rectangular milling groove cycle.		
Handling method	Modify the program so that U decreases or I and J increase.		
Alarm number	PS0186	Alarm messages	U value is less than tool radius
Analysis	U value is less than tool radius in rectangle inner precision milling cycle.		
Handling method	Modify the program so that U increases or tool is changed.		

Alarm number	PS0187	Alarm messages	I and J are too small or L is too big
Analysis	I and J are too small or L is too big in G110-G137 special fixed cycle.		
Handling method	Modify the program so that I and J increase or L decreases.		
Alarm number	PS0188	Alarm messages	The same name within NC/PLC file has been modified, please check up and restart.
Analysis	Rename the same names within NC/PLC file when updating CNC system		
Handling method	Restart		
Alarm number	PS0189	Alarm messages	Special fixed cycle cannot specify G68G51G16G51.1
Analysis	Specification of G68 G51 G16 G51.1 is not allowed in special fixed cycle.		
Handling method	Modify the program		
Alarm number	PS0190	Alarm messages	cnc.db is damaged, please contact the manufacturer
Analysis	cnc.db file is damaged		
Handling method	Re-introduce completed file		
Alarm number	PS0191	Alarm messages	macro.db is damaged, please introduce it again
Analysis	macro.db file is damaged		
Handling method	Re-introduce completed file		
Alarm number	PS0192	Alarm messages	offset.db is damaged, please introduce it again
Analysis	offset.db file is damaged		
Handling method	Re-introduce completed file		
Alarm number	PS0193	Alarm messages	paramter.db is damaged, please introduce it again
Analysis	paramter.db file is damaged		
Handling method	Re-introduce completed file		
Alarm number	PS0194	Alarm messages	plcprm.db is damaged, please introduce it again
Analysis	plcprm.db file is damaged		
Handling method	Re-introduce completed file		
Alarm number	PS0195	Alarm messages	skin.db is damaged, please introduce it again
Analysis	skin.db file is damaged		
Handling method	Re-introduce completed file		
Alarm number	PS0196	Alarm messages	Installment is due, please contact the manufacturer
Analysis	Installment is due		

Handling method	Input and pay the installment in order to unlock		
Alarm number	PS0197	Alarm messages	Please input the password to unlock after paying the installment
Analysis	Installment is due		
Handling method	Input and pay the installment in order to unlock		
Alarm number	PS0198	Alarm messages	Setting of system time is wrong, please contact the manufacturer
Analysis	No authority to set the system time		
Handling method	Please contact the manufacturer		
Alarm number	PS0199	Alarm messages	Please select to load PLC file or modify Parameter 102
Analysis	It is required to update PLC file		
Handling method	Reload PLC file		
Alarm number	PS0200	Alarm messages	Algorithm protection error 0
Analysis	Forward-looking structure NC code is full.		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0201	Alarm messages	Algorithm protection error 1
Analysis	Greater than the maximum speed change in a cycle. The amount of speed change exceeds the set value of parameter 1501.		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0202	Alarm messages	Algorithm protection error 2
Analysis	Greater than the speed of the last cycle in the acceleration phase.		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0203	Alarm messages	Algorithm protection error 3
Analysis	Greater than the speed of the first cycle in the deceleration phase.		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0204	Alarm messages	Machine structure parameter setting error
Analysis	Such machine mechanism does not exist or is temporarily not supported by the system		
Handling method	Please refer to 5-axis machining parameter of the instruction, and fill in the machine mechanism parameters		
Alarm number	PS0207	Alarm messages	Algorithm protection error 7
Analysis	Greater than program block length.		

Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0210	Alarm messages	Algorithm protection error 10
Analysis	The number of adjusted acceleration cycles is greater than that of original acceleration cycles.		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0211	Alarm messages	Algorithm protection error 11
Analysis	The number of adjusted deceleration cycles is greater than that of original deceleration cycles.		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0212	Alarm messages	Algorithm protection error 12
Analysis	Total number of adjusted cycles ≤ 0		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0215	Alarm messages	Algorithm protection error 15
Analysis	Greater than joining speed		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0216	Alarm messages	Algorithm protection error 16
Analysis	Greater than maximum allowable speed		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0217	Alarm messages	Algorithm protection error 17
Analysis	Greater than block head speed		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0218	Alarm messages	Algorithm protection error 18
Analysis	Calculation error		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0219	Alarm messages	Algorithm protection error 19
Analysis	During the exit phase, the condition is not met		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0221	Alarm messages	G-code transmission error

Analysis	System protective alarm.		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0222	Alarm messages	Transmission plane error
Analysis	System protective alarm.		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0224	Alarm messages	Algorithm protection error 24
Analysis	Analysis		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0225	Alarm messages	Algorithm protection error 25
Analysis	Total interpolation period error		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0226	Alarm messages	No reference point return
Analysis	This alarm is given when running the program because the system zero is lost in the machine with the stop homing after shut-down or there is an emergency stop and servo alarm.		
Handling method	Reference point return again.		
Alarm number	PS0230	Alarm messages	Rigid tapping VPO signal is invalid
Analysis	In the rigid tapping process, the spindle servo position mode is cancelled.		
Handling method	Check spindle servo.		
Alarm number	PS0231	Alarm messages	Machine lock cancelled
Analysis	In the machine lock state, the program is run or the axis is moved, and then the machine lock is cancelled.		
Handling method	The machine with block homing needs to home, while that without stop homing needs to be directly reset.		
Alarm number	PS0232	Alarm messages	Emergency stop alarm
Analysis	In case of emergency, the emergency stop button is pressed to prohibit the machine from running.		
Handling method	Release the emergency stop button after troubleshooting.		
Alarm number	PS0233	Alarm messages	Performing positioning error
Analysis	System protective alarm.		

Handling method	Rerun after reset cancellation alarm or contact support personnel.		
Alarm number	PS0234	Alarm messages	Decoding and interpolation communication error
Analysis	System protective alarm.		
Handling method	Rerun after reset cancellation alarm or contact support personnel.		
Alarm number	PS0235	Alarm messages	Algorithm protection error 35
Analysis	Number of Nurbs blocks exceeds the maximum limit		
Handling method	Rerun after reset cancellation alarm or contact support personnel.		
Alarm number	PS0236	Alarm messages	Tapping spindle speed is zero
Analysis	Spindle speed is not set before tapping.		
Handling method	Check whether the spindle speed is set in the program.		
Alarm number	PS0237	Alarm messages	Algorithm protection error 37
Analysis	Nurbs interpolation calculation error		
Handling method	Rerun after reset cancellation alarm or contact support personnel.		
Alarm number	PS0238	Alarm messages	G27 reference point return detection error
Analysis	Reference point return detection position is not at the reference point position.		
Handling method	Check whether the program command position is consistent with the reference point position.		
Alarm number	PS0239	Alarm messages	Running too long, out of range of system counter
Analysis	System protective alarm.		
Handling method	Rerun after reset cancellation alarm or contact support personnel.		
Alarm number	PS0240	Alarm messages	Second travel detection forbidden area
Analysis	The current position of the system indicates the second travel detection forbidden area.		
Handling method	Check whether the program commands entry into the second travel detection forbidden area.		
Alarm number	PS0241	Alarm messages	Third travel detection forbidden area
Analysis	The current position of the system indicates the third travel detection forbidden area.		
Handling method	Check whether the program commands entry into the third travel detection forbidden area.		
Alarm number	PS0243	Alarm messages	Spindle speed is zero

Analysis	The spindle speed is not commanded in per rotation feed modal.		
Handling method	Check whether the spindle speed is set in the program.		
Alarm number	PS0244	Alarm messages	Performing an illegal operation in G88 boring
Analysis	In G88 modal, there is no operating mode returning to the original after switching to operating mode.		
Handling method	Operating again after returning to the original operating mode.		
Alarm number	PS0245	Alarm messages	Arithmetic protection error 45
Analysis	The speed less than 0 interpolated		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0246	Alarm messages	Algorithm protection error 46
Analysis	The remaining time less than 0 interpolated		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0247	Alarm messages	Algorithm protection error 47
Analysis	The distance less than 0 interpolated		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0248	Alarm messages	Algorithm protection error 48
Analysis	The interpolation distance is inconsistent with the remaining distance		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0249	Alarm messages	Algorithm protection error 49
Analysis	Interpolation super-linkage speed		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0250	Alarm messages	Algorithm protection error 50
Analysis	The actual interpolation position is greater than the interpolation distance		
Handling method	Rerun after reset cancellation alarm; or contact support personnel.		
Alarm number	PS0252	Alarm messages	The thread cutting speed is greater than the maximum allowable speed
Analysis	The resultant speed of thread turning is greater than the set value of parameter #1224[maximum cutting feed resultant speed]		
Handling	Modify the program to reduce thread pitch or spindle speed; or raise the set value of parameter		

method	#1224.		
Alarm number	PS0253	Alarm messages	Thread cutting spindle speed is zero
Analysis	Thread machining is not allowed when the spindle is not rotating		
Handling method	Modify the program to command spindle rotation before thread cutting.		
Alarm number	PS0254	Alarm messages	coord.db is damaged, please re-import
Analysis	The coord.db file is damaged.		
Handling method	Re-introduce completed file		
Alarm number	PS0255	Alarm messages	pitch.db is damaged, please introduce it again
Analysis	pitch.db is damaged		
Handling method	Re-introduce completed file		
Alarm number	PS0256	Alarm messages	record.db is damaged, please introduce it again
Analysis	record.db is damaged		
Handling method	Re-introduce completed file		
Alarm number	PS0257	Alarm messages	Varying pitch could not be switched as dry running under processing
Analysis	Varying pitch could not be switched as dry running under processing		
Handling method	Rerun after reset cancellation alarm		
Alarm number	PS0258	Alarm messages	G05P3 has the linear axis moving command beyond the basic 3-axis
Analysis	At present, there is a rotation axis command under G05P3.		
Handling method	Modify the program		
Alarm number	PS0260	Alarm messages	NC axis operation occurs in PLC axis
Analysis	Processing program is written wrong, and current axis is controlled by PLC axis		
Handling method	Modify processing program		
Alarm number	PS0261	Alarm messages	No change to PLC control axis
Analysis	PLC axis could not be changed to NC axis control when moving.		
Handling method	Check PLC program		
Alarm number	PS0262	Alarm messages	Axis-selecting signal is connected when PLC axis is being switched to NC axis
Analysis	Axis-selecting signal is connected when PLC axis is being switched to NC axis		
Handling method	Check PLC program		

Alarm number	PS0263	Alarm messages	PLC axis operation exists in NC axis
Analysis	PLC operates on such axis under NC axis control mode		
Handling method	Check PLC program		
Alarm number	PS0264	Alarm messages	Error 1 on improving efficiency of slow stop
Analysis	Error 1 on calculating speed by system		
Handling method	Stop the function of improving slow stop and set #1401.2＝0		
Alarm number	PS0265	Alarm messages	Error 2 on improving efficiency of slow stop
Analysis	Error 2 on calculating speed by system		
Handling method	Stop the function of improving slow stop and set #1401.2＝0		
Alarm number	PS0266	Alarm messages	Error 3 on improving efficiency of slow stop
Analysis	Error 3 on calculating speed by system		
Handling method	Stop the function of improving slow stop and set #1401.2＝0		
Alarm number	PS0267	Alarm messages	No RTCP function
Analysis	Such function could not be used in system		
Handling method	Modify processing program		
Alarm number	PS0268	Alarm messages	Error in detecting G code communication port
Analysis	System communication error		
Handling method	Please contact the manufacturer		
Alarm number	PS0269	Alarm messages	Error in detecting algorithm parameter transmission
Analysis	System communication error		
Handling method	Please contact the manufacturer		
Alarm number	PS0270	Alarm messages	Wrong parameters setting for the shape parameter of the chuck
Analysis	An alarm will be given when the length of the chuck jaw is less than or equal to the clamping length or the radius of the chuck jaw is less than or equal to the clamping height difference.		
Handling method	Modify the parameters to ensure that #1102 is greater than #1104 and #1103 is greater than #1105.		
Alarm number	PS0271	Alarm messages	Wrong parameters setting for the shape parameter of the tailstock
Analysis	The diameter (2) of the tailstock is smaller than or equal to its aperture (3), with an alarm given.		
Handling method	Modify the parameters to ensure that #116 is greater than #1117.		
Alarm	PS0272	Alarm	Overlapped position of the chuck and the nose

number		messages	
Analysis	The position of the chuck is greater than that of the tailstock, causing the chuck to overlap with the top position.		
Handling method	Modify the parameters to ensure that #1107 is smaller than or equal to #1118.		
Alarm number	PS0276	Alarm messages	P4 algorithm only support speed increase and decrease before S type
Analysis	Parameter cannot be set as straight speed increase and decrease when using G05P4		
Handling method	Modify processing program without G05P4 or modify parameter #1400.6＝1		
Alarm number	PS0278	Alarm messages	Maximum speed of super spindle
Analysis	In rigid tapping, the maximum speed set by the spindle speed over-parameter (#2140~#2143).		
Handling method	Reduce the spindle speed during tapping or modify the parameter value (#2140~#2143).		
Alarm number	PS0279	Alarm messages	The positioning spindle cannot be in the same block with other movement commands
Analysis	In the spindle positioning, the other spindles cannot have the movement command at the same time.		
Handling method	Spindle positioning and other axis movement commands are separated into two blocks.		
Alarm number	PS0280	Alarm messages	Waiting M code mismatched
Analysis	Waiting M-codes specified by the two channels are not equal.		
Handling method	Modify the program so that the waiting M-codes specified by the two channels are between parameters #920 and #921 and equal.		
Alarm number	PS0283	Alarm messages	P2 algorithm is not allowed.
Analysis	G05P2 algorithm is not allowed		
Handling method	Modify the program		
Alarm number	PS0300.0	Alarm messages	Error in initiated communication alarm
Analysis	Setting of parameter 221-228 is not consistent with actual connection or with error, and the hardware port exists problem		
Handling method	Set parameter correctly or connect the line correctly, or replace hardware		
Alarm number	PS0300.1	Alarm messages	Error in initiated communication alarm
Analysis	Alarm appears when communication is valid in normal communication.		
Handling method	Check inter-connection is reliable or not and improve the capability of anti-interference		
Alarm number	PS0301	Alarm messages	Slave station alarm
Analysis	A slave station gives an alarm. Please refer to the instruction manual of the GR series drive units for the specific handling method.		
Handling method	Cut the power and then restart after releasing the slave station alarm		

Alarm number	PS0311.1	Alarm messages	Error in system communication figuration parameter
Analysis	Error in detection code of communication parameter read initially.		
Handling method	Error in detection code of communication parameter read initially. Restart or replace hardware.		
Alarm number	PS0311.2	Alarm messages	Error in system communication figuration parameter
Analysis	Error in communication parameter data, alarm for any error in communication cycle/number of slave stations/number of servers/number of spindles/number of IO, modify parameter		
Handling method	Modify parameter 221-228		
Alarm number	PS0312.1	Alarm messages	Error in system communication figuration parameter
Analysis	Error in detection code of data communication of parameter from interface to control, hardware failure		
Handling method	Replace mainboard if failure still occurs after restart.		
Alarm number	PS0312.2	Alarm messages	Error in data communication detection
Analysis	Error in detection code of data communication of parameter from interface to control, hardware failure		
Handling method	Replace mainboard		
Alarm number	PS0313	Alarm messages	Error in reading real time data initially.
Analysis	Error in reading real time data/core data initially. Restart when such error appears after the first upgrade of system. It is a hardware failure.		
Handling method	Replace mainboard if failure still occurs after restart.		
Alarm number	PS0314.1	Alarm messages	It is overtime for system to get the site initiation
Analysis	Server is not ready		
Handling method	Check the server		
Alarm number	PS0314.2	Alarm messages	It is overtime for system to get the site initiation
Analysis	Feedback site is not reasonable		
Handling method	Check the feedback site of server is normal or not.		
Alarm number	PS0314.3	Alarm messages	It is overtime for system to get the site initiation
Analysis	Final feedback stop site is over the value set by Parameter N4111.		
Handling method	Check feedback stop site is normal or not.		

2.2 Servo and position control alarm (PV alarm)

The alarms before PV0~109 are drive alarms, please refer to the instruction manual of the GR series drive unit.

Alarm number	PV110	Sub-alarm signal		Alarm messages	Servo speed command is abnormal.
Analysis	PID regulation speed command exceeds the allowable set value, and the diagnosis 102 displays the maximum command speed value.				
Solutions	1. Confirm whether the position gain setting parameter 4009 is too big. 2. Confirm whether parameter 4254 is set correctly. 3. Confirm whether the command of position loop and bit 5 and 6 of 4003 in feedback direction are correct.				
Alarm number	PV112	Sub-alarm signal		Alarm messages	Servo axis does not exist
Analysis	An alarm is given when invalid servo axis moves, and diagnosis 133 displays whether the servo axis is valid.				
Solutions	1. Set 4000#0 to 1 to ignore this alarm. 2. Confirm whether parameter 225/226 is set correctly.				
Alarm number	PV113	Sub-alarm signal		Alarm messages	Coordinate update handshake timeout
Analysis	An error occurs in the system's internal data interaction reply				
Solutions	Power off and restart.				
Alarm number	PV114	Sub-alarm signal		Alarm messages	Synchronous mode command is illegal
Analysis	1. Shield the slave axis under synchronous control. 2. Change the synchronous control mode in the movement process. 3. Move the slave axis in manual, MPG and return-to-zero mode. 4. PLC axis control and synchronous axis control are selected at the same time.				
Solutions	Operate correctly				
Alarm number	PV115	Sub-alarm signal		Alarm messages	Synchronization error for positional deviation is too large
Analysis	The difference between the master and slave position deviations controlled synchronously is too large. Parameter 4023 is the set allowable value and is invalid when being set to 0. Diagnosis 141 displays the actual deviation error value.				
Solutions	Correctly set parameters and solve mechanical problems				
Alarm number	PV116	Sub-alarm signal		Alarm messages	Synchronization error of machine coordinate is too large
Analysis	The difference between master and slave machine coordinates controlled synchronously is too large. Parameter 4022 is the set allowable value and is invalid when being set to 0. Diagnosis 140 displays the actual deviation error value.				
Solutions	Correctly set parameters and solve mechanical problems				
Alarm number	PV119	Sub-alarm signal		Alarm messages	Synchronous torque difference limit alarm
Analysis	The difference between master and slave torques controlled synchronously is too large. Parameter 4025 is the set allowable value and is invalid when being set to 0. Diagnosis 142 displays the actual error value.				
Solutions	Correctly set parameters and solve mechanical problems				
Alarm number	PV120	Sub-alarm signal		Alarm messages	NC and PLC axis control commands competed
Analysis	Temporarily ineffective				

Solutions					
Alarm number	PV121	Sub-alarm signal		Alarm messages	PLC control axis cannot be changed
Analysis	An alarm will be given when it is switched to NC axis in PLC axis control operation.				
Solutions	Operate correctly				
Alarm number	PV123	Sub-alarm signal		Alarm messages	System position command is illegal
Analysis	An alarm will be given when the system command position cycle difference is greater than the value set by 4112 parameter. Diagnosis 128 displays the maximum command difference				
Solutions	Power off and restart.				
Alarm number	PV124	Sub-alarm signal		Alarm messages	Increment raster axis not reference point return error
Analysis	An alarm will be given when the axis moves without manually homing after increment raster control setting is valid.				
Solutions	Do manual home first				
Alarm number	PV125	Sub-alarm signal		Alarm messages	This raster scale equipment does not exist
Analysis	Set the 200 cycle length parameter to 16.				
Solutions	Correctly set parameters				
Alarm number	PV126	Sub-alarm signal		Alarm messages	The difference between the raster coordinate and the encoder coordinate exceeds the allowable value
Analysis	Parameter 4115 is the allowable setting value. When it is set to 0, the alarm is ignored (for safety reasons, it is generally not set to 0); the diagnosis 145 is the coordinate value of the feedback grating, the diagnosis 146 is the difference of the coordinate values of grating and code disk, and the diagnosis 147 is the original grating feedback value.				
Solutions	Set parameters correctly and solve mechanical problems				
Alarm number	PV0127	Sub-alarm signal		Alarm messages	Overtime update of system initiative site
Analysis	Check sub-alarm code 314: if it is 1, server might be not ready; if 2, feedback site might be not reasonable; if 3, the final feedback stop site exceeds the value set by N4111 parameter.				
Handling method	Check the value of sub-alarm code 314 and deal with it.				
Alarm number	PV128	Sub-alarm signal		Alarm messages	System servo data error, and reset zero
Analysis	An alarm will be given when single-turn/multi-turn zero offset value is greater than or equal to the set value. Diagnosis 115/116 is a multi-turn/single-turn value, and diagnosis 151/152 is a current multi-turn/single-turn offset value.				
Solutions	Reset 4001#APZ machine zero resolution.				
Alarm number	PV129	Sub-alarm signal		Alarm messages	Alarm for axis operation when servo is disconnected
Analysis	In the case of disconnected servo axis, an alarm will be given for axis operation. Diagnosis 16 shows whether the servo axis is disconnected.				
Solutions	Operate correctly				

Alarm number	PV0130	Sub-alarm signal		Alarm messages	Alarm in moment coupling sub-control axis movement
Analysis	The moment coupling sub-control axis is regarded as spindle for command movement under its server control.				
Handling method	Invalid moment coupling sub-control axis or modify processing program				
Alarm number	PV131	Sub-alarm signal		Alarm messages	Motion position out-of-tolerance
Analysis	When the position difference between the command and the feedback exceeds the maximum value set by parameter 4112, appropriately increase the value of parameter #4112; the alarm under abnormal conditions needs to observe the diagnostic parameters so as to make a comprehensive judgment, which may be caused by the unreasonable setting of other parameters.				
Solutions	Set parameters correctly and solve mechanical problems				
Alarm number	PV132	Sub-alarm signal		Alarm messages	Over-travel of site in static status
Analysis	When the position difference between the command and the feedback exceeds the maximum value set by parameter 4111, appropriately increase the value of parameter #4111; the alarm under abnormal conditions needs to observe the diagnostic parameters so as to make a comprehensive judgment, which may be caused by the unreasonable setting of other parameters.				
Solutions	Set parameters correctly and solve mechanical problems				
Alarm number	PV133	Sub-alarm signal		Alarm messages	Torque deviation is too large under servo torque control
Analysis	Torque deviation is too large under servo torque control				
Solutions	Set parameters correctly and solve mechanical problems				
Alarm number	PV134	Sub-alarm signal		Alarm messages	The current homing position of the increment raster is illegal
Analysis	Homing position is incorrect				
Solutions	Homing away from block				
Alarm number	PV135	Sub-alarm signal		Alarm messages	Servo feedback position is illegal
Analysis	Encoder data feedback is abnormal, resulting in excessive error in two adjacent cycles				
Solutions	Replace hardware				
Alarm number	PV0140	Sub-alarm signal		Alarm messages	Homing is abnormal
Analysis	The nth axis cannot be returned to the exact position during homing.				
	Enter the nth axis pitch parameter correctly or adjust homing block position.				
Solutions	Correctly set parameters,				
Alarm number	PV0141	Sub-alarm signal		Alarm messages	Positive soft position limit
Analysis	The machine coordinate exceeds or equals to the set value of the n^{th} axis soft limit in system parameter #1081. Move the nth axis in the reverse direction away from the position set by parameter #1081.				
Solutions	Move the nth axis in the reverse direction away from the position set by parameter #1081.				

Alarm number	PV0142	Sub-alarm signal		Alarm messages	Negative soft position limit
Analysis	The machine coordinate exceeds or equals to the set value of the n^{th} axis soft limit in system parameter #1081.				
Solutions	Move the nth axis in the forward direction away from the position set by parameter #1081.				
Alarm number	PV0143	Sub-alarm signal		Alarm messages	Forward hard over-travel
Analysis	The n^{th} axis forward limit switch is pressed to the limit block.				
Solutions	The nth axis moves in the reverse direction away from the limit block position				
Alarm number	PV0144	Sub-alarm signal		Alarm messages	Reverse hard over-travel
Analysis	The n^{th} axis reverse limit switch is pressed to the limit block				
Solutions	The nth axis moves in the forward direction away from the limit block position				

2.3 Spindle alarm (PD alarm)

The alarms before PD0~109 are drive alarms. For details, please refer to the instruction manual of the GR series drive unit.

Alarm number	PD110	Sub-alarm signal		Alarm messages	Spindle closed loop out-of-tolerance alarm
Analysis	Under analog spindle closed loop function and rigid tapping, the difference between command and feedback position exceeds the tapping difference set.				
Solutions	Correct the allowable value of static and dynamic out-of-tolerance of rigid tapping and check whether the setting parameters of spindle closed loop are appropriate.				
Alarm number	PD111	Sub-alarm signal		Alarm messages	Spindle servo alarm
Analysis	It occurs when the spindle alarm function is valid and the spindle gives an alarm.				
Solutions	Identify the cause of the spindle alarm or whether the spindle alarm level parameters are appropriately.				
Alarm number	PD112	Sub-alarm signal		Alarm messages	Spindle speed is abnormal
Analysis	When the spindle speed fluctuation check function is valid, the spindle speed fluctuation range exceeds the set value.				
Solutions	Check whether the set speed fluctuation detection parameters are appropriate and identify the cause of the spindle speed fluctuation.				
Alarm number	PD113	Sub-alarm signal		Alarm messages	Broad allowable range of rigid tapping synchronization error is exceeded
Analysis	Under the valid analog spindle closed loop and rigid tapping, the synchronization error between spindle and tapping shaft exceeds the set allowable range.				
Solutions	Correct the synchronization error setting range or identify whether the setting parameters for the spindle and tapping shaft are appropriate.				

CNCmakers Limited
Add: No.168, Xiadu Road, Haizhu District, Guangzhou, China 510300 Tel:86-139-25042139 Fax:86-20-84185336 www.CNCmakers.com
Online shop: www.paycnc.com Email: paycnc@paycnc.com cnc@CNCmakers.com WhatsApp: https://wa.me/+8613925042139