*ECN*000TL SERIES USER MANUAL*

CNC MILLING CONTROLLER



www.hybridservo.com

©2016 shenzhen ECON technology co.,ltd

This user manual describes all items concerning the operation of this CNC system in

detail. However, it is impossible to give particular descriptions for all unnecessary or unallowable operations due to length limitation and products application conditions；Therefore, the items not presented herein should be considered impractical or unallowable.

Copyright is reserved to Shenzhen ECON technology Co.,Ltd. It is illegal for any organization or individual to publish or reprint this manual. Shenzhen ECON technology Co.,Ltd. reserves the right to ascertain their legal liability.

Preface

Yours Excellency,

We are honored by your purchase of products from Shenzhen ECON technology Co.,Ltd.
This manual introduces programming, operation and connection of ECN*000M series CNC Milling Machine in detail. To ensure safe and efficient work, please read this manual carefully before installation and operation.

Warning and Precaution

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

Please read this manual carefully before operation!

Special caution:The power supply fixed on/in the cabinet is exclusively used for the CNC system made by Mei Too.
It can't be applied to other purposes, or else it may cause serious danger.This manual is subject to change without further notice.This manual is reserved by end user

Ver.1.0

CAUTIONS

■ Transportation and Storage

1) Packing box over 6 layers in pile is not allowed.

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

3)Do not move or drag the products by the cables connected to it.

4) Forbid collision or scratch to the panel and display screen.

5) Avoid dampness, insolation and drenching.

■ Open-package Inspection

1) Confirm that the products are the required ones.

2) Check that the products are not damaged in delivery.

3) Confirm that the parts in packing box are in accordance with the packing list.

4)Contact us in time if any inconsistence, shortage or damage is found.

■ Wiring

● Only qualified personnel can connect the system or check the connection.

● The system must be earthed, and the earth resistance must be less than 0.1Ω. The earth wire cannot be replaced by a neutral wire (zero wire).

● The connection must be correct and firm to avoid any fault or unexpected consequence.

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

● Switch off power supply before plugging out or opening electric cabinet.

■ Troubleshooting

● Cut off the power supply before troubleshooting or component replacement.

● Check for fault when short circuit or overload occurs. Restart can only be done after troubleshooting.

● Frequent switching on/off of the power is forbidden, and the interval time should be at least 1 minute.

ANNOUNCEMENT!

●This manual describes various possibilities as much as possible. However, operations allowable or unallowable cannot be explained one by one due to so many possibilities that may involve with,

so the contents that are not specially stated in this manual shall be considered as unallowable.

WARNING!

- please read this manual and a manual from machine tool builder carefully before installation, programming and operation, and strictly observe the requirements. Otherwise, products and machine may be damaged, workpiece be scrapped or the user be injured.

NOTE!

- Functions, technical indexes (such as precision and speed) described in this user manual are only for this system. Actual function deployment and technical performance of a machine tool with this CNC system are determined by machine tool builder's design, so functions and technical indexes are subject to the user manual from machine tool builder.
- Though this system is employed with integrated operator panel, the functions of the key on the panel are defined by PLC program (ladder diagram). It should be noted that the key functions described herein are for the standard PLC program (ladder diagram).
- Refer to the user manual from machine tool builder for function and meaning of key on control panel.

Volume Ⅰ PROGRAMMING

Introduces product specification, types, command codes and format of programs.

Volume Ⅱ OPERATION

Describe the operation methods of ECN*000M series CNC Milling Controller.

VolumeⅢ INSTALLATION

Describe the methods for installation, connection andseting of ECN*000M series CNC Milling Controller.

APPENDIX

Describe appearance dimensions, rigid tapping and alarm messages of ECN*000M series CNC Milling Controller.

Safety Responsibility

Manufacturer Responsibility

——Be responsible for the danger which should be eliminated and/or controlled on

design and configuration of the provided CNC systems and accessories.

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

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

User Responsibility

——Be trained with the safety operation of CNC system and familiar with the safetyoperation procedures.

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

—— Be responsible for the failure to observe the provisions for operation, adjustment,maintenance, installation and storage in the manual.

This manual is reserved by end user.

We are full of heartfelt gratitude to you for supporting us in the use of ECON'sproducts.

Content

Volume 1 Programming

CHAPTER 1 PROGRAMMING FUNDMENTALS.....15
1.1 Introduction.....15
1.1.1 Product brief.....15
1.1.2 Specifications.....16
1.1.3 Adaptation of climate, enviroument.....21
1.1.4 Power supply adaptive capacity.....21
1.1.5 Defend.....21
1.2 Operation of program.....22

1.2.1 Sequence of program operation....22
1.2.2 The execution order of codes in one program segment....23
CHAPTER 2 MST Code....23
2.1 M Codes (Miscellaneous Function)....23
2.1.1 End of program M02....24
2.1.2 End of run M30....24
2.1.3 Subprogram call M98....25
2.1.4 Return from Subprogram M99....25
2.1.6 Program Stop M00....26
2.1.7 Program Choice Stop M01....26
2.1.8 CCW, CW and Spindle stop control M03、M04 and M05....27
2.1.9 Cooling control M08、M09....27
2.1.10 Tool control M16、M17....27
2.2.2 Spindle Speed Analog Voltage Control....29
2.2.3 Spindle Override....30
2.3 Tool Function....30
CHAPTER 3 G COMMAND....31
3.1 Varities of preparation functions G code....31
3.2 Simple G code....35
3.2.1 Rapid Positioning G00....35
3.2.2 Linear interpolation G01....37
3.2.7 System parameter's online change (G10)....47
3.2.8 Workpiece coordinate G54 ~ G59....48
3.2.9 Additional workpiece coordinate system....50
3.2.10 Choose machine tool coordinate system G53....51
3.2.11 Float coordinate system G92....51
3.2.14 Scaling G51/G50....56
3.2.16 Skip function G31....65
3.2.17 Conversion of Inch and metric G20/G21....67
3.2.18 Any angle chamfer/corner circle arc....68
3.3 Reference point G code....69
3.3.1 Return reference point G28....70
3.4 Fixed cycle G code....73
3.4.2 Complete circle internal groove milling cycle G24/G25....83
3.4.5 Rectangle groove internal milling cycle G35/G36....90
3.4.6 Rectangle external groove milling cycle G37/G38....92
3.4.7 High speed deep hole processing cycle G73....94
3.4.8 Drilling cycle, spot drilling cycle G81....96
3.4.9 Drilling cycle, spot drilling cycle G82....98
3.4.10 Chip removal and drilling cycle G83....100
3.4.11 Right-handed tapping cycle G84....102
3.4.13 Precision boring cycle G76....106
3.4.14 Boring cycle G85....108
3.4.16 Hole boring cycle, back hole boring cycle G87....112

Valume 2 Operation explanation

Chapter 1 Operation mode and display interface..... 178
1.1 Panel divide.....178
1.1.3 Machine panel.....180
1.2 Operation mode summarize.....183
1.3 Display interface.....183
1.3.1 Position interface.....183
1.3.2 Programming interface.....187
1.3.3 Tool offset wear, macro variable interface.....189
1.3.4 Alarm interface.....191
1.3.5 Set the Visual Design.....192
1.3.6 State parameters, data, and the parameters of the screw filling.....196
1.3.7 CNC diagnosis, machine tools, PLC signal soft panel, help information, version information interface.....198
Chapter 2 Start-up, shutdown and safety protection.....201
2.1 Starting up.....202
2.2 Power off.....202
2.3 Distance protection.....202
2.3.1 Hardware distance protection.....202
2.3.2 Distance protection of software.....203
2.4 Emergency operation.....203
2.4.1 Reset.....203
2.4.2 Scram.....204
2.4.3 Feed hold.....204
2.4.4 Power cut.....204
Chapter 3 manual manipulation.....204
3.1 Axis mobile.....205
3.1.1 Manual feed.....205
3.1.2 Manually move quickly.....205
3.1.3 Speed repair adjust.....206
3.2 Other manual.....206
3.2.1 Turn counterclockwise, turn clockwise, stop control.....206
3.2.2 Spindle point move.....206
3.2.3 Coolant control.....207
3.2.4 Lubrication control.....207
3.2.5 Tool changing.....208
3.2.6 Axis ratio of callback.....208
Chapter 4 The handwheel/a single step operation.....208
4.1 Single step feed.....208
4.1.1 The choice of incremental.....209
4.1.2 Mobile direction choice.....209
4.2 The handwheel (hand pulser) feed.....209

4.2.1 The choice of incremental..... 209
4.2.2 Axis and the direction of choice..... 209
4.2.3 Other Actions..... 210
4.2.4 Important notes..... 211
Chapter 5 Input operations..... 211
5.1 The input code word..... 211
5.2 The implementation of the code word..... 212
5.3 Preferences..... 212
5.4 modifications of data..... 212
5.5 Other Actions..... 213
Chapter 6 Program editing and management..... 214
6.1 The establishment of the program..... 214
6.1.1 The generation of program segment number..... 214
6.1.2 Program content input..... 215
6.1.3 The movement of the cursor..... 216
6.1.4 Words, the line number of retrieval..... 217
6.1.5 Word of insert..... 218
6.1.6 The deletion of the word..... 218
6.1.7 Word modify..... 218
6.1.8 The remove of the procedures section..... 218
6.1.9 To delete multiple procedures section..... 218
6.1.10block delete..... 219
6.1.11 The copy of the procedures section..... 219
6.1.12 More than the copy of the procedures section..... 219
6.1.13 Block replication..... 219
6.1.14 Block of the paste..... 220
6.2 The deletion of the program..... 220
6.2.1 To delete a single program..... 220
6.2.2 Delete all program..... 220
6.3 The selection of the program..... 220
6.3.1 Retrieval method..... 220
6.3.2 Scanning method..... 221
6.3.3 The cursor recognition method..... 221
6.4 Program name..... 222
6.5 Copy of program..... 222
6.6 Program management..... 222
6.6.1 Program directory..... 222
6.6.2 The number of stored procedures..... 223
6.6.3 Memory capacity..... 223
CHAPTER 7 AUTO OPERATIONS..... 223
7.1 Auto run..... 223
7.1.1 Selection of the program to be run..... 223
7.1.2 Program Start..... 224
7.1.3 Stop of the Auto Run..... 224

7.1.4 Auto run from an arbitrary block....226
7.1.5 Adjustment of the feedrate override, rapid override....226
7.1.6 Spindle override adjustment....227
7.2 DNC Running....227
7.3 Running State....227
7.3.1 Single Block Execution....227
7.3.2 Dry Running....228
7.3.3 Machine Lock....228
7.3.4 MST lock....228
7.3.5 Block Skip....229
7.4 Other Operations....229
CHAPTER 8 MACHINE ZERO RETURN OPERATION....230
8.1 Machine Zero....230
8.1.1 Machine zero point....230
8.1.2 Machine Zero Ruturn Steps....230
8.2 Other operations of Machine zero....231
CHAPTER 9 DATA SETTING, BACKUP and RESTORE....232
9.1 Data setting....232
9.1.1 Switch setting....232
9.1.2 Graphic setting....233
9.1.3 Parameter setting....234
9.2 Restore and backup of data....235
9.3 The pemission setting and alteration....236
9.3.1 Entry of the operation level....237
9.3.2 Alteration of the Password....238
9.3.3Lower Level Set....238
9.4 Operation of U-disk....239
9.4.1 File content page....239
9.4.2 File copy....239

Volume 3 Installation and connection

CHAPTER 1 INSTALLATION LAYOUT....241
1.1 ECN*000M series Connection....241
1.1.1 ECN*000M series Back Cover Interface Layout....241
1.1.2 Interface Explanation....241
1.2 ECN*000M series Installation....242
1.2.1 External Dimensions....242
1.2.2 Installation Conditions of the Cabinet....242
1.2.3 Protection Methods against Interference....242
Chapter 2 EFINITION&CONNECTION OF INTERFACE SIGNALS....244
2.1 Connection to Drive Unit....245

2.1.1 Drive Interface Definition..... 245
2.1.2 Command Pulse and Direction Signals..... 245
2.1.3 Drive Unit Alarm Signal..... 245
2.1.4 Axis Enable Signal ENn..... 246
2.1.5 Pulse Disable Signal SET..... 246
2.1.6Zero Signal nPC..... 246
2.1.7 Connection to Drive Unit..... 248
2.2 Connection of spindle axis encoder..... 249
2.2.1 Spindle axis encoder interface definition..... 249
2.2.2 Signal explanation..... 249
2.2.3 Connection of spindle encoder port..... 249
2.3 Connection to Handwheel..... 250
2.3.1 Handwheel Interface Definition..... 250
2.3.2 Signal explanation..... 250
2.4 Connection to Spindle Encoder..... 251
2.4.1 Spindle Encoder Interface Definition..... 251
2.4.2 Connection of gerrnal inverter..... 252
2.5 Connection of ECN*000M series to PC serial port..... 252
2.5.1 Communication Interface Definition..... 252
2.5.2 Communication Interface Connection..... 253
2.7 Connection of Power Interface..... 253
2.7 I/O Interface Definitions..... 254
2.7.1 Input signal..... 257
2.7.2 Output signal..... 258
2.8 I/O function and connection..... 259
2.8.1 Travel limit and scram..... 260
2.8.2 Machine zero..... 262
2.8.3 Spindle control..... 266
2.8.4 Spindle rotate speed switching value control..... 269
2.8.5 Spindle auto shift gears control..... 269
2.8.6 External cycle start and feeding hold..... 272
2.8.7 Cooling pump control..... 273
2.8.8 Lubrication control..... 274
2.8.9 Protective door..... 276
2.8.10 CNC Marco variable..... 276
2.8.11 Tricolour light..... 277
2.8.12 External MPG..... 277
2.9 Electric diagram common symbols comparison..... 278
Chapter 3 Machine Debugging methods and steps..... 279
3.1 Emergency stop and strock limit..... 279
3.2 Drive unit setting..... 280
3.3 Gear Ratio Adjustment..... 280
3.4 Acceleration&deceleration Characteristic Adjustment..... 281
3.5 Machine zero adjust..... 282

3.6 Spindle Adjustment..... 285
3.6.1 Spindle Encoder.....285
3.6.2 Switch Volume Control of Spindle Speed..... 286
3.6.3 Analog Voltage Control for Spindle Speed.....286
3.7 Backlash Offset.....286
3.8 Step/MPG Adjustment.....287
3.9 Other Adjustment.....288
CHAPTER 4 DIAGNOSIS MESSAGE.....289
4.1 CNC Diagnosis..... 289
4.1.1 I/O Fixed address diagnose message.....289
4.1.2 Axes Moving State and Data Diagnosis Signal of CNC.....289
4.1.3 Key diagnosis.....290
4.1.4 Others.....290
4.2 PLC state.....290
4.2.1 F signal.....290
4.2.2 G Sign..... 300
4.2.3 An address (message displays request signal, standard PLC definition).....307
CHAPTER 5 MEMORIZING SCREW-PITCHES ERROR COMPENSATION FUNCTION.....308
5.1 Function Explanation.....308
5.2 Specifications.....308
5.3 Parameter Setting.....308
5.3.1 Screw-Pitch Compensation..... 309
5.3.2 Pitch error compensation the origin..... 309
5.3.3 Offset interval..... 309
5.3.4 Compensation dosage.....309
5.4 Precaution of offset value setting..... 309
5.5 Example of the compensation parameters setting.....310

Appendix

Appendix 1 Outline Dimension of MST-81 series.....310
Appendix 2 Dimensions of Additional Panel.....311
Appendix 3 Common operations list.....312

VOLUME I PROGRAMMING

CHAPTER 1 PROGRAMMING FUNDMENTALS

1.1 Introduction

1.1.1 Product brief

ECN*000M series Milling controller is a new generation of CNC system developed by Shenzhen ECON technology Co.,Ltd.As the upgraded version, it supports milling, boring and drilling cycle. It employs 32 bits high-capability CPU and very large scale programmable device FPGA, applies real-time multi-task control technology and hardware interpolation technology, and is able to perform μm level precisionmotion control and PLC logic control. ECN*000M series controller is the optimum choice for upgrading CNC milling machine.



Characteristics:

●Five axes control; 5 axes linkage; 2 analog spindles

●1ms interpolation cycle, 0.1μm control precision.

●Two channels -10V ~ 10V analog voltage output, support double spindle control

●Linear type, index type and S type etc. acceleration and deceleration types can be chosen.

●Embedded with PLC programming, the current running PLC programming can be chosen

●PLC displayed online, real time monitoring, signal real time tracing

●Supports statement type macro programming, supports macro codes with parameters calls.

●Flexible tapping and rigid tapping can be set by parameters

● Includes spiral, scaling, polar coordinate, fixed circle and a variety of slotting compound cycle function.

●Includes historical alarm and resumen operation functions, it's convenient for users to operate and maintenance.

● Support multi-level passwords for equipment maintenance and management

● 36 point general input/36 point general output

●Support standard RS232 and USB port, support U disk documents operation, system configuration and software updated, it can meet the functions that documents transfer, serial ports DNC machining and USB online machining.

1.1.2 Specifications

Control axis

● Control axis: 5 axis (X, Y, Z, 4th, 5th)

● Linkage axis: 5 axis

Feeding axis functions

- Position command range:

  Metric input (G21) -9999.9999mm ~9999.9999mm, the mininum command unit: 0.0001mm

  Inch input (G20): -999.9999inch ~999.9999inch, the mininum command unit: 0.0001inch
- Electronic gear ration: command multiplier 1 ~ 65536, command multiplier1 ~65536
- Rapid traverse speed: maximum 60m/min
- Rapid traverse override: F0, 25%, 50%, 100% four levels real-time tuning
- Cutting feedrate: maximum feeding speed: maximum 15m/min (G94) or 500.00mm/r (G95)
- Feedrate override: 0 ~ 150% sixteen-level real-time tuning
- Manual feedrate: 0 ~ 150% sixteen-level real time tuning
- MPG feeding: 0.001mm, 0.01mm, 0.1mm, 1mm four gears
- Single step feeding: 0.001mm, 0.01mm, 0.1mm, 1mm four gears
- Interpolation mode: linear interpolation, arc interpolation, spiral interpolation and ragid rapping
- Automatic chamfering function

Acceleration/deceleration types:

- Cutting feeding: before the acceleration and deceleration linear type, before the acceleration and deceleration S type, after the acceleration and deceleration linear type and after the acceleration and deceleration index type.
- Fast moving: before the acceleration and deceleration linear type, before the acceleration and deceleration S type, after the acceleration and deceleration linear type and after the acceleration and deceleration index type.
- System with preview function, it can read 15 NC programming, make small line interpolation in high speed and smooth, it's suit for components machine toolings processing.
- The original speed of acceleration and deceleration, end speed and acceleration&deceleration speed set by parameters.
- Manual mode, handwheel mode is after acceleration and deceleration control, handwheel mode can choose emergency mode or full operation mode.
- Rapid positioning can choose linear or broken line.

Spindle axis functions:

- Two channels -10V ~ 10V analog voltage output, support double spindle control
- Spindle encoder: encoder lines can be set (100 p/r ~ 5000p/r)

● Encoder and spindle's transmission ratio: (1 ~ 255):(1 ~ 255)

● Spindle ration: 50% ~ 120% eight stages real-time tunning in total

● Spindle axis constant line speed control

● Tapping circle, rigid tapping

Tool functions:

● Tool length compensation

● Tool point radius compensation (C type)

Precision compensation:

●Thread pitch error compenstation: Compensation points, compensation interval, compensate original position can be set.

●Backlash compensation: can set reverse interval to interpolation machine tool by fixed frequency or lifting speed mode.

PLC functions:

● Two levels PLC programming, operation speed is 1.5μs/ step basic instruction; 4700 steps mostly, the cycle of the first level programming is 8ms.

● Support PLC warning and PLC alarm.

● Support multiply PLC programming (mostly 20), the current PLC programming can be chosen.

● 45 kinds of commands (10 kinds of basic commands, 35 kinds of function commands)

● I/O unit input/output: 36/36

Display interface:

● 7.0 inch wide-screen LCD, 800×480 lattice

● 2D tool path displays.

● Real-time clock

Operation mode:

● Operation modes: edit, automatic, input, machine tool reset, handwheel/single-step, manual, DNC.

● Multiple operateauthority management.

● Alarm logging

Program edit:

● Program volumn: 56MB, it can store 400 programmings mostly (includes sub-programming, Marco

programming)

● Edit function: Programming/programming segment/word retrieval, amend, delete, copy, and stick

●Programming format: ISO code,supports statement macro program, supports relative coordinate, absolute coordinate and polar coordinate.

● Programming calls: supports maco with parameters calls, level 4 subprogram nesting.

Communication function:

● RS232: parts program, parameters and some documentation two-way transmission, supports PLC program, system software serial ports update.

● USB: U disk documentation operatiom, U disk processing directly, Support PLC program, system software U disk update.

● Support serial port DNC processing function and USB online processing function, reliability and safty

Relability and safty function

● Emergency stop

● Handware route limit

● Software routet inspect

● Data backups and Data recover.

G code table:

Code	Function	Code	Function
G00	Positioning (rapid traverse)	G51	Scaling
G01	Linear interpolation (cutting feed)	G53	Mechanical coordinate system
G02	Circular interpolation CW (clockwise)	G54	Workpiece coordinate system1
G03	Circular interpolation CCW(anticlockwise)	G55	Workpiece coordinate system2
G04	Delay, exact stop	G56	Workpiece coordinate system3
G12	Store rount detection function connected	G57	Workpiece coordinate system4
G13	Store rount detection function broken	G58	Workpiece coordinate system5
G15	Cancel polar coordinate instruction	G59	Workpiece coordinate system6
G16	Polar coordinate instruction	G54.1-G 54.50	Additional workpiece coordinate system
G17	XY plane selection	G60	Single direction positioning
G18	ZX plane selection	G61	Orientation mode
G19	YZ plane selection	G62	Automatic corner ratio
G20	Inch input	G63	Tapping mode
G21	Metric input	G64	Cutting mode
G22	Anticlockwise circle internel groove milling	G65	Marco instruction

G23	Clockwise circle interner groove milling	G68	Coordination rotation
G24	Anticlockwise entire circle finish-groove milling	G69	Coordination rotation cancel
G25	Clockwise entire circle finish-groove milling	G73	High speed deep hole processing cycle
G26	Anticlockwise circle external finish-groove milling	G74	Left rotation tapping cycle
G27	Reference position return checking	G76	Fine boring cycle
G28	Return to reference position	G80	Fixed cycle cancel
G29	Return from reference point	G81	Drilling cycle (spot drilling cycle)
G30	Return to 2, 3, 4 reference point	G82	Drilling cycle, point boring cycle
G31	Skip function	G83	Chip removal drilling cycle
G32	Clockwise external circle external finish-groove milling	G84	Right rotation tapping cycle
G33	Clockwise tangle groove milling	G85	Boring cycle
G34	Anticlockwise tangle groove milling	G86	Boring cycle
G35	Anticlockwise tangle internal finish-groove milling	G87	Back boring cycle
G36	Clockwise tangle internal finish-groove milling	G88	Boring cycle
G37	Anticlockwise tangle external finish groove milling	G89	Boring cycle
G38	Clockwise tangle external finish groove milling	G90	Absolute programing
G39	Corner offset circle interpolation	G91	Relative programing
G40	Tool radius offset cancel	G92	floating coordinate system setting
G41	Left tool radius offset	G94	Feeding/min
G42	Right tool radius offset	G95	Feeding/rpm
G43	Positive tool radius offset	G96	Constant speed control (cutting speed)
G44	Negative tool radius offset	G97	Constant speed control cancel (cutting speed)
G49	Tool length offset cancle	G98	Return original plane
G50	Ratio zoom cancel	G99	Return R point plane

PLC instruction form

Instruction code	Function	Instruction code	Function	Instruction code	Function
RD	Read normaL open contactor	SPE	Sub-program finished	DIFU	Rising edge detection
RD.NOT	Read normal close contactor	SET	Set	DIFD	Falling edge detection
WRT	Output winding	RST	Reset	COMP	Binary number comparation
WRT.NOT	Output winding negation	JMPB	Label skip	COIN	Consistency comparation
AND	Normally opened cotactors connect in	LBL	Label	MOVN	Data transfer

	series				
AND.NOT	Normally closed cotactors connect in series	TMR	Timer	MOVB	1 bit transfer
OR	Normally opened cotactors connect in parallel	TMRB	Fixed timer	MOVW	2 bits transfer
OR.NOT	Normally closed cotactors connect in parallel	TMRC	Any address timer	XMOV	Binary address change data transfer
OR.STK	Circuit block in parallel	CTR	Binary system counter	DSCH	Binary data research
AND.STK	Circuit block in series	DEC	Binary system code	ADD	Binary system plus
END1	The 1st level sequence program ended	COD	Binary system code conversion	SUB	Binary system minus
END2	The 2nd level sequence program ended	COM	Public line control	ANDF	Logical AND
CALL	Call sub-program	COME	Public line control stop	ORF	Logical OR
CALLU	Non-conditional call sub-program	ROT	Binary system rotation control	NOT	Logical NOT
SP	Sub-program	SFT	Register shift	EOR	XOR

1.1.3 Adaptation of climate, enviroument

ECN*000M series controller storage, transpotation, working environment as below:

Item	Work climate conditional	Store transpotation climate conditional
Environment temperature	0℃～ 45℃	-40℃～ +70℃
Relative humidity	≤ 90%（non condensation ofmoisture）	≤ 95%（40℃）
Atmosphere pressure	86 kPa ～ 106 kPa	86 kPa ～ 106 kPa
Altitute	≤ 1000m	≤ 1000m

1.1.4 Power supply adaptive capacity

ECN*000M series controller Under the condition of the following input AC power supply can work normally.

Voltage changes: in the range of (0.85 ~ 1.1) *rated AC input voltage (AC220V); Frequency change: 49Hz ~ 51Hz continuous change.

1.1.5 Defend

ECN*000M series controller defend level can't be lower than IP20.

1.2 Operation of program

1.2.1 Sequence of program operation

The current opened program can only be operated in automatic operation mode;ECN*000M series controller can not open 2 or more programs in the same time. The cursor at the beginning of a program segment when open a program, the cursor can be moved in edit operation mode. To operate stop state in automatic operate mode, use cycle start signal(the key of machine tool panel or external cycle start signal) to start program from program segment which the current cursor in, usually it executes for each program segment according to the compile sequence of program segment, the program stop running until execute M02 and M30 code. Cursor moved along with the programming's running. The running sequence of program will be changed in the following situations:

* The program running will stop if you press  key or emergency stop key.
* The program running will stop when program runs with CNC alarm or PLC alarm;
* The operation modes will be changed to input, edit operation way, program runs period stop (program runs stop when finish the current program period), switch to automatic operation mode, start program from the program segment where the cursor is, when press key or external cycle start signal connected.
* Operation mode switchs to other mode when program runs, then program stop running;
* Press key or external pause signal broke when program runs, program running paused. Signal will run from the stop position when press key or connect external cycle start signal.
* When open SBK switch, program pause after each program segment end of running; it needs re-press key  or connect external cycle start signal to continue the running of next program segment;
* Open the BDT of program segment, the program segment starts with "/" will be jumped or no executed;
* Turn to target program segment when execute G65 skip code;
* Invoke corresponding sub-program or Marco when execute M98 code; Sub-program or Marco run is complete, return the program segment next to the invoke program segment (if M99 code specified target program segment number, then turn to it) when execute M99 code;
* When execute M99 code in major program (The program run is not started by other programs call), turn to the first program segment and keep to run, the current program will keep on running repeatly.

1.2.2 The execution order of codes in one program segment

A program segment has G, X, Z, F, R, M, S, T etc multiple codes, most of the M, S, T codes explained by NC and send to processing by PLC, NC will process other codes. M98, M99 and S code which ends with r/min and m/min and defined speed to spindle axis also to be processing by NC.

When G code and M00, M01, M02 and M30 in the same program segment, it can only execute M code after NC completes G code, and send the corresponding M signal to PLC do processing.

When G code and M98, M99 in the same program segment, NC will execute these M code (wouldn't send M signal to PLC) after complete G code.

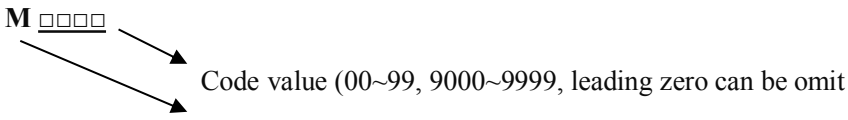
When G code in the same program segment with M, S, and T which process by PLC, PLC program (ladder diagram) decide M, S, T code and G code execute in the same time, or execute M, S, and T code after complete G code, about the code execute order,subject to machine tool manufacturer.

M00, M01, M02, M30will execute after other program segment complete.

CHAPTER 2 M CODE

2.1 M Codes (Miscellaneous Function)

The M codes are composed by code address M and 1~2 or 4 digits after the codes M is used for controlling the program execution or outputting M code to PLC.



Address

NC deal with M98, M99 independently, it wouldn't output M code to PLC.

The M02 and M03 are defined as program END codes by NC, meanwhile it also outputs M codes to PLC for the I/O control (spindle OFF, cooling OFF control etc.).

M98, M99 as program invoke code; M02, M30 as program end code, PLC program can't change the meaning of above codes. All the other M code can be output to PLC,define code functions by PLC program, please refer machine manufacturer's manuals.

Only can be one M code in a program segment, the CNC will alarm when there are two or more M codes in a program segment.

Form 2-1 the execute procedure of control program M code list

Codes	Functions
M02	End-of Run
M30	End-of Run
M98	Subprogram call
M99	Return from the subprogram; If the code M99 is used for main program ending (namely the current program is not called by other programs), the program will be repeated executed.

2.1.1 End of program M02

Format:M02 or M2

Function: The M02 code is executed in the Auto mode. The automatic run is ended after the other codes of current segment are executed, and the workpieces number is added by one, they system cancels the tool nose radius compensation and cursor returns to the beginning of program (whether the cursor returns to the head of the program is determined by parameters)

2.1.2 End of run M30

Format: M30

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

The cursor does not return to be beginning of the program when the BIT4 of parameter No.005 is set to 0; when it set to 1, the cursor returns to the beginning of the program as soon as the program execution is finished.

2.1.3 Subprogram call M98

Format: M98 P ◇◇◇◇□□□□

Repeated call times

The called sub-program No.

Function: In Auto mode, when the M98 is executed, the subprogram specified by P is called after the execution of other codes in the current segment; the subprogram can be performed 9999 times at most. M98 can not be performed in MDI, or an alarm will occur.

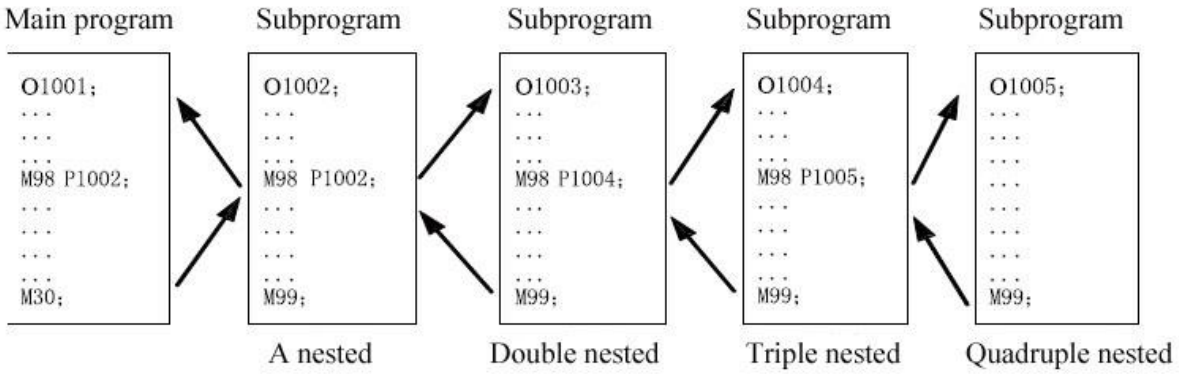
2.1.4 Return from Subprogram M99

Format:M99 P ◇◇◇◇

The segmentnumber (0000~9999) when return to main program is executed, the leading zero can be omitted

Function:(in subprogram) as the other commands of current segment are executed, the segment specifiedby P is performed continuously when the main program is returned. The nextsegment is performed continuously by calling current subprogram of M98 command whenreturning to the main program; because of the P is not given. If the main program is endedby using the M99 (namely, the current program is not called by other programs forexecution), the current program will be run circularly. So, the M99 command is disabled inMDI.

ECN*000M series controller can call quadruple subprogram, namely, the other subprogram can be calledfrom the subprogram. (See Picture 2-3)



Picture 2-3 Subprogram nestifications

2.1.5 M command Defined by Standard PLC Ladder Diagram

The M commands other than the following mentioned commands (M02, M30, M98,and M99) are defined by PLC. The M commands are defined by standard PLC here in after. This ECN*000M series milling machine controller is used for machine control. About the function,meaning, control time sequence and logic etc. of the M command,please refer to the manual issued by the machine manufacturer.

M command specified by standard PLC ladder diagram

Command	Function	Remark
M00	Program pause	
M01	Program option stop	
M03	Spindle CCW	Function interlock, status hold
M04	Spindle CW	
*M05	Spindle stop	
M08	Cooling on	Function interlock, status hold
*M09	Cooling off	
M16	spindle tool umclamp	Function interlock, status hold
M17	spindle tool clamp	
M18	Cancel spindle orientation	Function interlock, status hold
M19	Spindle orentation	
M28	Cancle rigid tapping	Function interlock, status hold
M29	Rigid tapping	
M32	Lubricating on	Function interlock, status hold
*M33	Lubricating off	
*M41, M42, M43, M44	Spindle gears change automatically	Function interlock, status hold

Note: The command with "*" specified by standard PLC is valid when the power is on.

2.1.6 Program Stop M00

Format: M00 or M0

Function: the program is stopped after executing the M00 command, the "pause" is displayed; the program will continue when the key of Cycle Start is pressed.

2.1.7 Program option Stop M01

Format: M01 or M1

Function: it is available in AUTO and EDIT mode, press OPT to make its light on, it means enter option stop status, the program will be stoped at once when execute M01 code and show “pause”, the program will go on when press cycle start key; but if didn’t open OPT, the program wouldn’t be paused even run M01 code.

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

Format: M03 or M3;

M04 or M4;

M05 or M5;

Function: M03: CCW;

M04: CW;

M05: Spindle stop.

Note: The control time sequence and logic of M03 and M04, M05 are specified by standard PLC program; refer tothe VOLUMN 3 Installation and connection of this manual.

2.1.9 Cooling control M08、M09

Format: M08 or M8;

M09 or M9;

Function: M08: cooling on;

M09: cooling off.

Note: The control time sequence and logic of M08 and M09 are specified by standard PLC program; refer tothe VOLUMN 3 Installation and connection of this manual.

2.1.10 Tool control M16、M17

Format: M16;

M17;

Function: M16: Loose spindle tool;

M17: Clamp spindle tool

2.1.11 Spindle orientation M18, M19

Format: M18;

M19;

Function: M18: Cancel spindle orientation;

M19: Spindle orientation.

2.1.12 Rigid Tapping M28, M29

Format: M28;

M29;

Function: M28: Cancel rigid tapping;

M29: Rigid tapping.

2.1.13 Lubricating Control M32, M33

Format: M32;

M33;

Function: M32: Lubricating on;

M33: Lubricating off.

Note: The control time sequence and logic of M32 and M33 are specified by standard PLC program,refer to the VOLUMN 3 Installation and connection of this manual.

2.1.14 Spindle shift automatically M41、M42、M43、M44

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

Function: Spindle switch to the n gear when execute M4n

Note: The control time sequence and logic of M41、M42、M43and M44 are specified by standard PLC program, refer tothe VOLUMN 3 Installation and connection of this manual.

2.2 Spindle function

The spindle speed is controlled by S command, there are two ways to control spindle speed for ECN*000M series controller.Spindle speed switching value control mode: the S□□ (2-digit command value) command is processed by PLC program for exporting the switching value signal to machine, so that the step speed change of the spindle is achieved.

Spindle speed analog voltage control mode: the actual spindle speed is specified by the S□□□□ (4-digit command value), the NC outputs the 0~10V analog voltage signal to the spindle servo device or inverter for achieving the stepless speed regulating of the spindle

2.2.1 Spindle Speed Switch Value Control

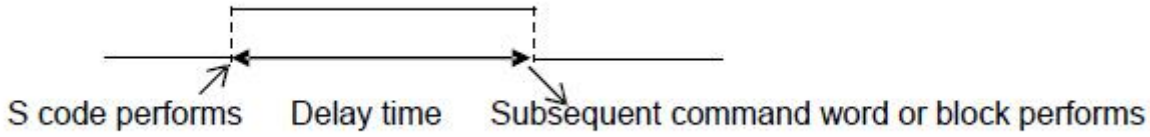
The spindle speed is on switching value control when the BIT4 of bit parameter NO.001 is set to0. One segment only has one S command. The CNC alarm occurs when there are two or more S commands displayed in segment. When the S command shares the same segment with the command, the performance sequence is defined by PLC program. For details, refer to the manual issued by the machine manufacturer.

This ECN*000M series milling machine controller is used for machining control when the spindle speed switching value is controlled. The time sequence and logic for S command should be referred by the manual issued by the machine manufacturer. The following S command is defined by ECN*000M series standard PLC, for reference only.

Format: S □□

00~04 (leading zero can be omitted):1~4 gears spindle speed switching value control

In spindle speed switching value control mode, the FIN signage is returned after the set time is delayed after the code signage of S command is set to PLC. Now the time is called execution time of S code.



The S01, S02, S03 and S04 output status are invariable when the CNC is reset.

The S1~S4 commands are ineffective output when the CNC is switched on. An arbitrary command is performed from S01, S02, S03 and S04, the corresponding S signal output is effective and held on, at the same time the other 3 S signal output are cancelled. The S1~S4 output are cancelled when performing the S00 command, only one of S1~S4 is effective in the meantime.

2.2.2 Spindle Speed Analog Voltage Control

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

Format: S □□□□

0000~9999(leading zero can be omitted) spindle analog voltage control

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

When spindle speed analog voltage control function is valid, there are two ways of spindle speed input: S code set spindle fixed speed (r/min), spindle speed wouldn't be changed when S code value keeps the same, called constant speed control (G97 modality); S code sets the tool relative speed of excircle tangent line (m/min), called constant linear speed control (G96 modality), under linear control mode, the spindle rotate speed changes along with the programming track X axis' absolute when cutting. Please check 2.2.3 of this section for details.

CNC has four gears spindle mechanical gear function, when execute S code, calculate to constant speed the coore sponding analog voltage value according to the highest spindle speed (output analog is 10V) set value (cooresponding data parameter NO. 210 ~NO.213) of current spindle gear, then output to spindle servo or inverter, control the spindle speed at the same speed of request.

The analog voltage output is 0V when power on, the output analog voltage keep the same value after execute S code (except stay in constant linear speed control cutting feed state and the absolute value of X axis absolute coordinate changed). Analog voltage output is 0V after execute S0; analog voltage output is constant when CNC reset and emergency stop.

2.2.3 Spindle Override

The spindle actual speed can be modified by spindle override switch when spindle speed analog voltage control is effective,but the actual speed limited by the current spindle gear's maximum. speed, and also it is controlled by the lowest spindle limitation value and the top spindle limitation value in constant linear speed control mode.

This NC offers 8-level spindle override (50%~120%, the change is 10% per level). The actual level and the modificative mode of the spindle override are defined by PLC ladder diagram. Refer to the manual issued by the machine manufacturer when attempting to use it. The following description is ECN*000M series standard PLC ladder diagram function, for reference only.

The spindle override defined by ECN*000M series standard PLC ladder diagram has 8 levels. The spindle actual real-time speed can be adjusted by using the spindle override switch in the command speed range of 50%~120%, the spindle override will be memorized when thepower is turned off. Refer to the OPERATION of this manual for modification operation of thespindle override.

2.3 Tool Function

Specify value after address T (up to 8) to select the tool.

In principle, it can't specify two or more T codes in the same program segment, but it wouldn't alarm if set the same group codes in one sigment.

Subject to the following T codes, about the T address and the machine tool movement, please refer the machine manufacturer's manual.

When movement codes and T code in the same program segment, code execution with the following ways:

1. Movement code and T code execution at the same time.
2. It starts to execure T code after finish movment code.
3. When T code and tools change code M06 at the same segment, execute T code first then execute tool change code; when T code and tools change code M06 in the different segments, M06 execute the T code which specified by last program.
4. As the following program;

   O00010;

   N10 T2M6; the tool on spindle is T2 tool;

   N20 M6T3; the tool on spindle is T3 tool;

   N30 T4; the tool on spindle is T3 tool;

   N40 M6; the tool on spindle is T4 tool;

   N50 T5; the tool on spindle is T4 tool;

   N60 M30;

   %;
5. The tool on spindle axis is T4 tool after execute tools change program.

CHAPTER 3 G COMMAND

3.1 Varieties of preparation functions G code

Preparation function expressed by G code and the following number, defined the meanings of the program segment it is. G code with the following types:

Form 3-1-1

Types	Meanings
Non-modal G code	It's vaild only in the commanded program segment
Modal G code	It's always vaild in front of other same group G codes

(For example) G01 and G00 are modal G code of the same group.

G01 X __;

Z _____; G01 is valid

X _____; G01 is valid

G00 Z ____; G00 is valid

Notice: the detailed system parameters please refer system parameters form.

Form 3-1-2 G command and function

G code	Group	Command type	Function
*G00	01	G00 X_Y_Z_	Positionning (fast moving)
G01		G01 X_Y_Z_F_	Linear interpolation (cutting feed)
G02		G02 R_ X_Y_ F_	Circular interpolation (CW)
G03		G03 I_J_	Circular interpolation (CCW)
G04	00	G04 P_ or G04 X_	Pause, Orientation
G10		G10 L_N_P_R_	Programmable data input
*G11		G11	Programmable data input type cancel
*G12	16	G12 XY_Z_I_J_K_	Store route measuring ability function is connected
*G13		G13	Store route measuring ability function is broke
*G15	11	G15	Cancel polar coordinate command
G16		G16	Polar coordinate command

G code	Group	Command type	Function
*G17	02	Write in program block, used in circle interpolation and tool radius compensation	XY plane selection
G18			ZX plane selection
G19			YZ plane selection
G20	06	It must be specified in the beginning of program and before coordinate setting, in independent program block.	British data input
*G21			Metric data input

G22	09	G22 X_Y_Z_I_L_W_Q_V_D_F_K			Anticlockwise internal circle groove milling
G23		G23 X_Y_Z_I_L_W_Q_V_D_F_K			Clockwise internal circle groove milling
G24		G24 X_Y_Z_R_I_J_D_F_K			Anticlockwise entire circle internal groove milling cycle
G25		G25 X_Y_Z_R_I_J_D_F_K			Clockwise entire circle internal groove milling cycle
G26		G26 X_Y_Z_R_I_J_D_F_K			Anticlockwise entire external milling cycle
G27	00	G27	X_Y_Z		Return to reference point detection
G28		G28			Return to reference point
G29		G29			Return from reference point
G30		G30Pn			Return 2, 3, 4 reference point
G31		G31			Skip function
G32	09	G32 X_Y_Z_R_I_J_D_F_K_			Clockwise external circle finish milling cycle
G33		G33 X_Y_R_I_J_L_W_Q_V_U_D_F_K			Anticlockwise tangle groove milling cycle
G34		G33 X_Y_R_I_J_L_W_Q_V_U_D_F_K			Clockwise tangle groove milling cycle
G35		G35 X_Y_Z_R_I_J_L_ U_D_F_K_			Anticlockwise tangle groove finish milling cycle
G36		G36 X_Y_Z_R_I_J_L_ U_D_F_K_			Clockwise tangle groove finish milling cycle
G37		G37 X_Y_Z_R_I_J_L_ U_D_F_K_			Anticlockwise tangle external finish milling cycle
G38		G38 X_Y_Z_R_I_J_L_ U_D_F_K_			Clockwise tangle external finish milling cycle
G39	00	G39 I_J_; I_K_; J_K_ or G39			Corner offset circle interpolation
*G40	07	G17	G40	X_Y_	Tool radius compensation cancel
G41		G18	G41	X_Y_	Left tool radius compensation
G42		G19	G42	X_Y_	Right tool radius compensation
G43	08	G43	H_Z_		Positive direction tool length compensation
G44		G44			Negative direction tool length compensation
*G49		G49			Tool length compensation cancel
*G50	12	G51			Scale cancel
G51		G51 X_Y_Z_P_			Scale
G53	00	Write in program			Choose machine coordinate system

*G54	05	Write in program and normally put in the beginning of program	Work piece coordinate system 1
G55			Work piece coordinate system 2
G56			Work piece coordinate system 3
G57			Work piece coordinate system 4
G58			Work piece coordinate system 5
G59			Work piece coordinate system 6
G60	00	G60 X_Y_Z_	Single direction positional

G code	Group	Command type	Function
G61	14	G61	Scram type
G62		G62	Automatic corner ratio
G63		G63	Tapping type
*G64		G64	Cutting type
G65	00	G65 H_P# I Q# j R# k	Marco command
G68	13	G68 X_ Y_ R_	Coordinate rotation
* G69		G69	Coordinate rotation cancel
G73	09	G73 X_Y_Z_R_Q_F_;	High speed deep hole processing cycle
G74		G74 X_Y_Z_R_Q_F_;	Left tapping cycle
G76		G76 X_Y_Z_R_Q_F_;	Right boring cycle
*G80		Write in program block with other program	Fixed cycle cancel
G81		G81 X_Y_Z_R_F_;	Drilling cycle (Spot drilling cycle)
G82		G82 X_Y_Z_R_P_F_;	Drilling cycle (Dimpling cycle)
G83		G83 X_Y_Z_R_Q_F_;	Chip removal drilling cycle
G84		G84 X_Y_Z_R_P_F_;	Right 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_;	Back boring cycle
G88		G88 X_Y_Z_R_P_F_;	Boring cycle

G89		G89 X_Y_Z_R_P_F_;	Boring cycle
*G90	03	Write in program block	Absolute value program
G91			Increment value program
G92	00	G92 X_Y_Z_	Float coordinate system setting
*G94	04	G94	Feeding per minute
G95		G95	Feeding per rotation
G96	15	G96 S_	Constant speed control (cutting speed)
* G97		G97 S_	Constant speed control cancel (cutting speed)
* G98	10	Write in program block	Return to the original plane from fixed cycle
G99			Return to the R point plane from fixed cycle

Remark:

1. **If modal instruction and non-modal instruction in the different segment, then priority with non-modal instruction;**
2. **When power on, system in the state of this G code which with "*";**
3. **All the G codes of 00 group are non-modal G codes except G10, G11 and G92;**
4. **It will alarm when use the G codes which don't be listed in the G code list, or command the function which G code has no it;**
5. **It can command severalsame group G codes at one program segment; in principle, it can't command two or more same group G codes at one program segment; when set same group commands don't alarm in the same segment, then subject to the following G code;**
6. **When group 01 and 09 G code in the same segment, then subject to 01; In fixed cycle modal, fixed cycle will be canceled and changed to G80 state if command code of group 01;**
7. **G codes show in group number according to the different types; if reset and emergency stop clean groups G codes set by bit parameterNO: 46#1 ~ 7 and NO: 47#0 ~ 7;**
8. **Rotating zoom command and group 01 or 09 commands in the same segment, gourp 01 or 09 modalwill be changed in the same time; the system will alarm when rotating zoomcommand and group 00 in the same segment.**

3.2 Simple G code

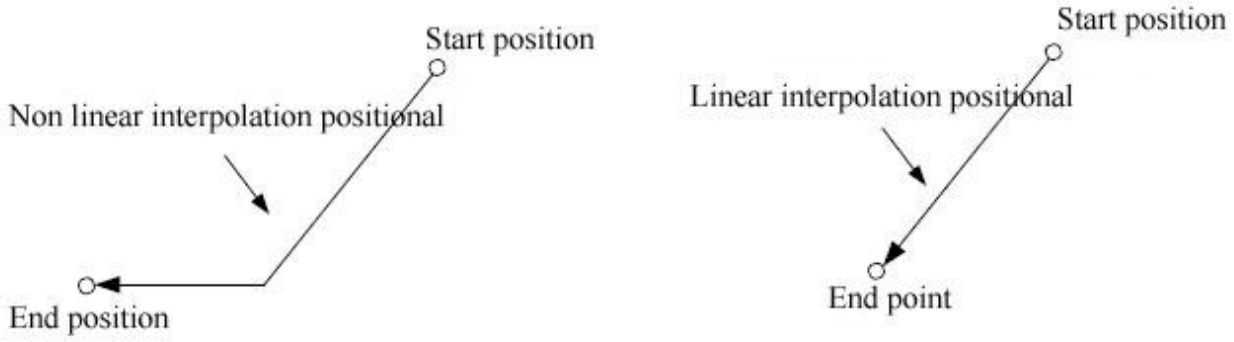
3.2.1 Rapid Positioning G00

Format:G00 X_Y_Z_

Function: G00 command, tool move to absolute instruct fastly or Increment instruct specify the position of coordinate, Use bit parameter N0: 14#0 to set it, choose one of the following two tool paths (as picture 3-2-1-1)

1. Linear interpolation position: tool path and linear interpolation (G01) is the same, tool moved at the speed lower than the speepd of each axis and positioning in shortest time.

2. Non-linear interpolation positioning: tool is positioning fastly subject to the move speed of each axis, tool paths usually are not linear (as it is effective in positioning).



Picture 3-2-1-1

Explanation:

1. System changed the modal of current tool move mode to G00 mode after execute G00.Through change system bit parameters value of N0: 48#0, it can set that system default modal is G00 or G01 (parameter value is 1) when connect power supply.
2. The positioning parameter tool doesn't move, system only change the current tool's move mode to G00.The positionning parameter tool which didn't specified wouldn't move, system only changed the move modal of the current tool to be G00.
3. G00 and G0 are equivalent format;
4. The G0 speeds of X, Y and Z axis are set by parameters P90 ~ P92.

Limitation:

Fast moving speed set by parameter, such as set F speed in G0 instruct to be the cutting feed speed for the following processing speed.

For example:

G0 X0 Y10 F800; adopt the speed which set by system parameters to feed fastly.

G1 X20 Y50; adopt the feeding speed of F800

Fast feeding speed adjusted by the key of operation panel (as the picture 3-2-1-2) F0, 25, 50, 100%; the coordinate speed of F0 set by data parameter P85, be in common use in each axis.



Picture 3-2-1-2 Fast speeding rate key

Notice: please notice the workbench and workpiece position, in case tools hit.

3.2.2 Linear interpolation G01

Format:G01 X_ Y_ Z_ F_

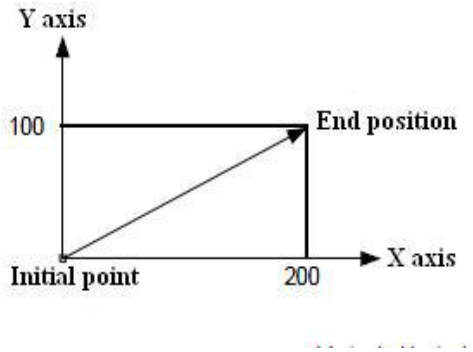
Function: Tool's movement path is a straight line to target position and the feeding speed is specified by parameter F.

Command explanation:

1) X_ Y_ Z_ are end coordinate value.

2)The feeding speed which F specified is valid until new F value specified.

The specified feeding speed is valid until new F value is specified. The feeding speed which specified by F code calculated by the interpolation along with linear paths, if F code didn't specified in the program, feeding speed adopts the acquiescent F value when power on. (Please refer data parameters P83)



G01 X200 Y100 F200;

Notice: The speed of each axis as below:

G01 Xα Yβ ZγFf;

In this program segment::

The speed of X axis direction: $F_X = \frac{\alpha}{L} \times f$

The speed of Y axis direction: $F_Y = \frac{\beta}{L} \times f$

The speed of Z axis direction: $F_Z = \frac{\gamma}{L} \times f$

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

Picture 3-2-2-1

Note:

1) All the parameters are positionning parameters except F. Data parameter P86 can set the upper limite value of

cutting speed F. The real cutting speed (the feeding speed after use ratio) will be upper limite value if exceed it, as unit mm/min. Data parameter P87 can set the lower limite value of cutting speed F. The real cutting speed (the feeding speed after use ratio) will be lower limite value if lower than it, as unit mm/min.

2) The tool wouldn't move until it specified positioning parameter after G01, system only change the modal G01 of current tool's move mode. When power on, it can set the default modal is G00 (parameter is 0) or G01 (parameter is 1) by change the system bit parameter N0: 48#0.

3.2.3 Arc and Helical Interpolation G02/G03

A、Circular interpolation G02/G03

G02 and G03'srules：

Plane circular interpolation is that finish the specified rotation and radius circular paths (or centre of the circular) in the specified plane. As the known start point and end point can't confirm the circular paths, so it needs to give the following parameters:

* The rotation direction of circle arc (G02, G03)
* The plane of circular interpolation (G17, G18 and G19)
* Circular coordinate or radius, then lead to two types codes instructions from this, circle coordinate I, J, K or radius R programming.

It can only do interpolation operation in the coordinate after the above three points confirmed.

It can do circular interpolation by the following instructions, tool moves along with circular as below:

Arc interpolation in XY plane

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

Arc interpolation in ZX planeZX

$$G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X_Z_ \begin{Bmatrix} R_ \\ I_K_ \end{Bmatrix} F_;$$

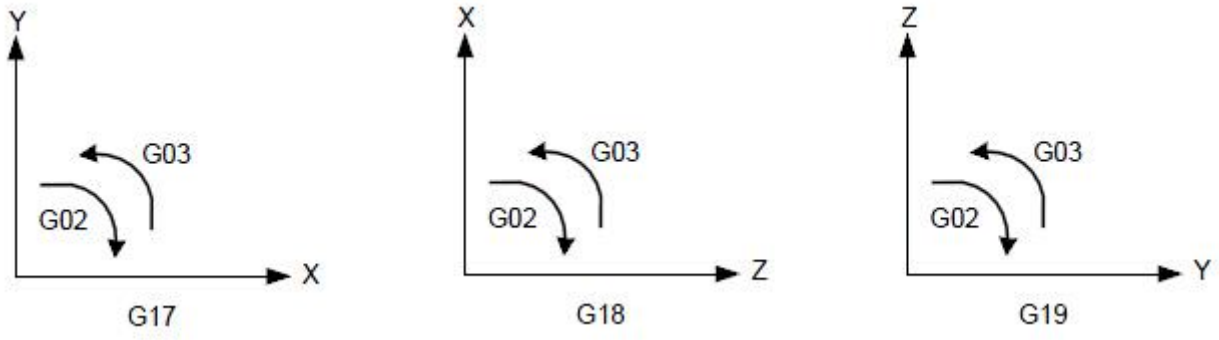
Arc interpolation in YZ plane

$$G19\begin{Bmatrix}G02\\G03\end{Bmatrix}Y_Z_\begin{Bmatrix}R_\\J_K_\end{Bmatrix}F_;$$

Table 3-2-3-1

Item	Specified content		Command	Significance
1	Plane specification		G17	Specifying XY plane arc
			G18	Specifying ZY plane arc
			G19	Specifying YZ plane arc
2	Rotating direction		G02	CW
			G03	CCW
3	End point	G90 mode	Two axis of X, Y and Z	End point in the part coordinate system
		G91 mode	Two axis of X, Y and Z	Distance from start to end points
4	Distance from start point to circle center point		Two axis of I, J and K	Z axis distance from start point to the center point (with sign)
	Arc radius		R	Circular radius
5	Feedrate		F	Feedrate along the arc

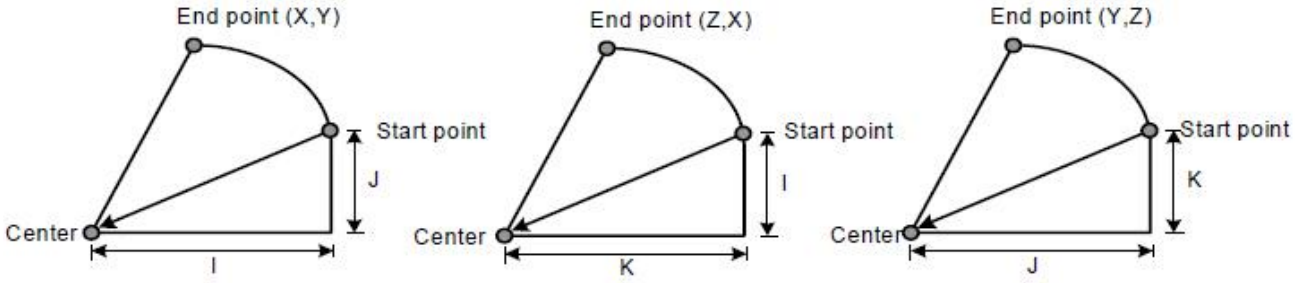
"Clockwise" and "Counterclockwise" are defined when XY plane (ZX plane, YZplane) is viewed in the positive-to-negative direction of the Z axis (Y axis, X axis) in the Cartesian coordinate system (as the picture 3-2-3-1)



Picture 3-2-3-1

Set bit parameter N0: 48#1, #2, #3 can specify plane modal information which is default when starts up.

Specify circular's end-point by parameter X, Y or Z. The instruction correspond to G90 shows in absolute value, correspond to G91 shows in increment value, Increment value is the coordinate that end-point relative to original-point. Circular centre specify by parameters I, J and K, they correspond to X, Y and Z. No matter in absolute mode G90 or relative mode G91, parameters I, J and K are the coordinates (it can be interpreted as that make start point is the original point temporarily, and circle centre is the coordinate) which circle centre relative to original point, all are the Increment with symbolic (Picture 3-2-3-2)



Picture 3-2-3-2

I, J and K have signs according to the direction. The circular center also can be specified by radius R other than I, J and K, as follows:

G02 X_ Y_ R_;

G03 X_ Y_ R_;

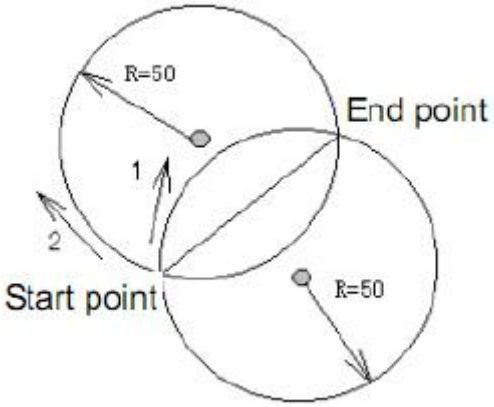
Now, the following two arcs can be described, one arc is more than 180°, the other is less than180°. The arc radius which is more than 180° is specified by the negative value.

(As picture 3-2-3-3) Arc ① less than 180°

G91 G02 X60 Y20 R50 F300;

Arc ②more than 180°

G91 G02 X60 Y20 R-50 F300;



Picture 3-2-3-3

2. The arc which is equal to 180°can be programming by I, J, K and R:

Example G90 G0 X0 Y0; G2 X20 I10 F100

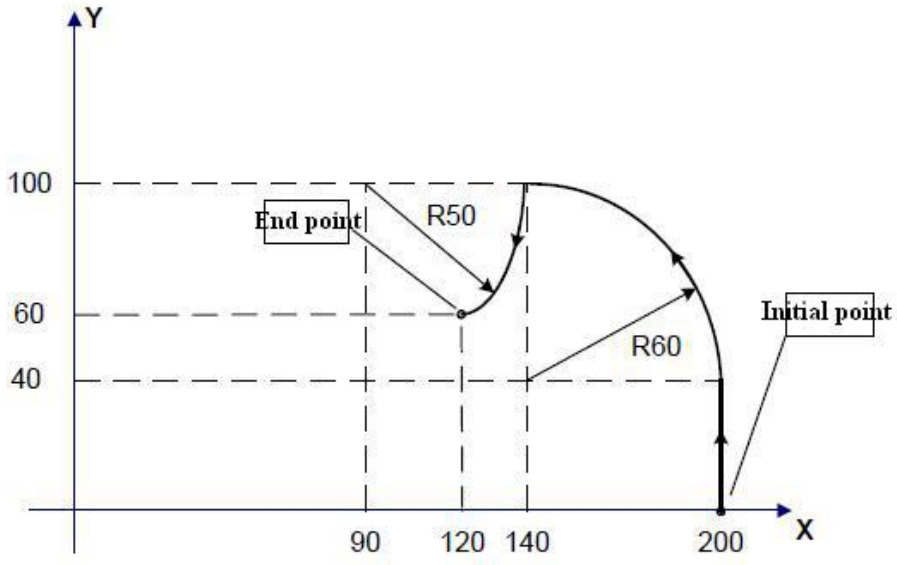
Equal to G90 G0 X0 Y0; G2 X20 R10 F100

Or G90 G0 X0 Y0; G2 X20 R-10 F100

Note: The positive and negative value of Arc 180° wouldn't affect the movement path.

3. The arc which is equal to 360° can only be programming by I, J, K.

(Examples of program):



Picture 3-2-3-4

The tool path program of picture 3-2-3-4 as below:

To program the picture 3-2-3-4 tool movement paths using the absolute mode and Incremental mode respectively:

1) Absolute mode

G90 G0 X200 Y40 Z0;

G3 X140 Y100 R60 F300;

G2 X120 Y60 R50;

Or

G0 X200 Y40 Z0;

G90 G3 X140 Y100 I-60 F300;

G2 X120 Y60 I-50;

2) Incremental mode

G0 G90 X200 Y40 Z0;

G91 G3 X-60 Y60 R60 F3000;

G2 X-20 Y-40 R50;

Or

G0 G90 X200 Y40 Z0;

G91 G3 X-60 Y60 I-60 F300;

G2 X-20 Y-40 I-50;

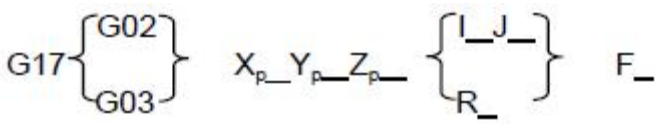
Limitation:

1. The R is effective when address I, J and K are commanded with the R, but I, J and K are disable data one time.
2. The system will alarm if circular radius and the value from start-point to centre is not specified.
3. If entire circle interpolation, it can only through the parameterI, J and K which specified from start-point to circle centre, the form that adopts specified R is not vaild.
4. Please notice the coordinate plane when do circular interpolation.
5. The X, Y and Z can be omitted simultaneously when the end and start points share same position. The tool wouldn't move when R is specified (for example: G02R50)

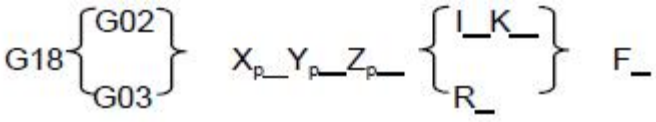
B. Spiral line interpolation

Command format: G02/G03

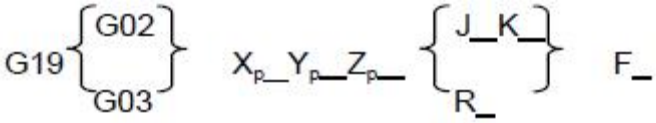
Arc in XY plane



Arc in ZX plane



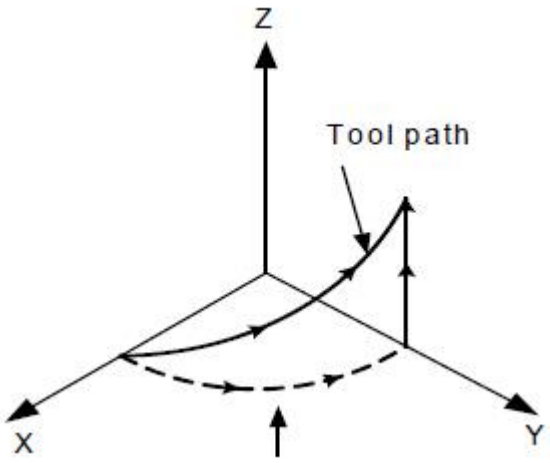
Arc inYZ plane



Picture 3-2-3-5

Function: make tool do spiral path move to specified position from current points with the speed which parameter F specified.

Explanation:



The feedrate a long the circumference of two circularinterpolated axes is the specified feedrate

Picture 3-2-3-6

The first two of instruction parameter are positioning parameter. Parameters are the two axis' number (X, Y or Z) of current plane. The place which this two positioning tool should move at of current plane. The third parameter is linear axis; the parameter value is the height of spiral. Other instruction parameters' specific meaning and limit are the same to circle arc interpolation.

System will reurn wrong information if it can't process circle according to specified instruction parameters. System

changes the current tool move mode's modal to G02/G03 after execute it.

The feeding speed is specified

Instruction just adds a move axis whitch do not do circular interpolation .F instruction specified feeding speed along circular. So the feeding speeds of linear axis as below:

FC=F* the length of linear axis/ the length of circular

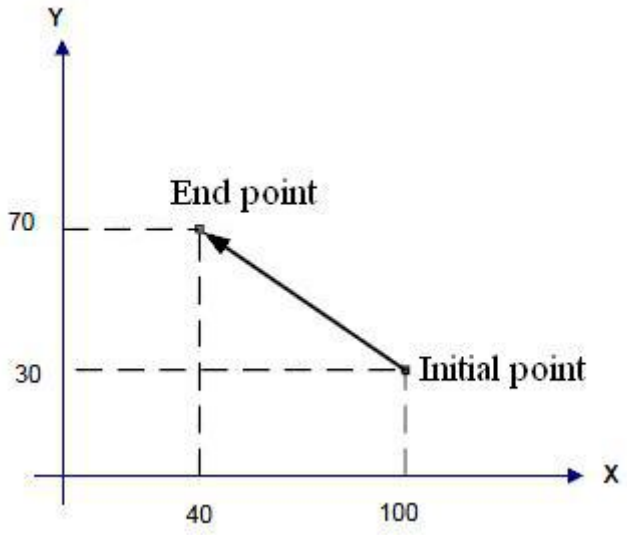
Be sure that linear axis' feeding speed not exceeds any limit value.

Limite: please notice the set of coordinate plane when doing spiral interpolation.

3.2.4 Absolute value/ Increment programming G90/G91

Instruction format: G90/G91

Function: As the axis movement instruction, it has two methods : absolute value and Increment value instructions. Absolute value instruction is the programming method that adopts end-point coordinate value of axis movement. Increment value instruction is the programming method that adopts axis relative movemen amount.Increment value is irrelevant with the coordinate which located movement direction and distance from end point to original point. Absolute value instruction and increment are G90 and G91.



Picture 3-2-4-1

The movement from original point to end point in picture 3-2-4-1, programing by absolute value instruction G90 and increment value instruction G91 as below:

G90 G0 X40 Y70;

Or G91 G0 X - 60 Y40;

The two methods can finish the same actions; operater can choose each of them flexibly.

Explanations:

*Non-instruction parameter. It can be written into program segment along with other instructions.

*G90 and G91 are the same group modal value, it is G90 mode (default mode) when specifiedG90 and before specified G91, G91 is always vaild before canceled G90 mode.

System parameter:

Set bit parameter N0: 48#4 can specified the default positioning parameter is G90 mode (when parameter is 1)

3.2.5 Pause (G04)

Format: G04 X_ or P_

Function: G04 execute pause operation, according to the specified time delays to execute next program segment. In addition, G64 can be suspended for doing corrent stop inspect.

Form 3-2-5-1

G04	X	0 ~ 9999.999	X corresponding to second
	P	0 ~ 99999.9999	P corresponding to millinsecond

Explanation:

1. G04, which is a non-modal G-command;

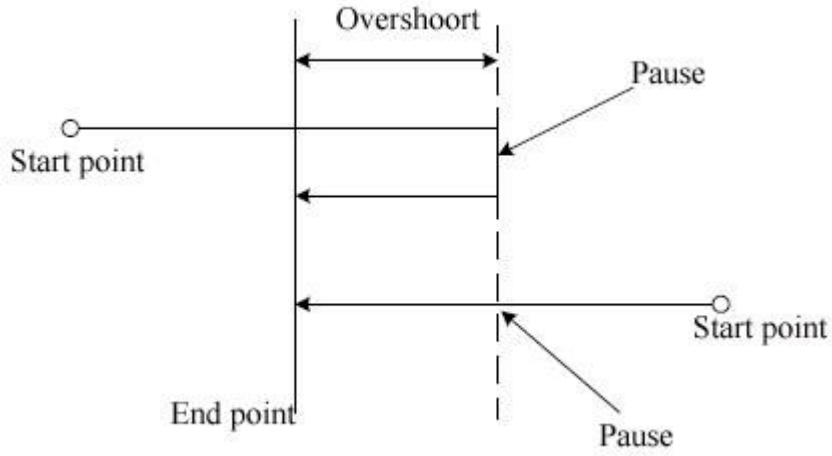
2. The P is effective when the P and X are in the same segment;

3. It will alarm when X, P in negative value;

4. System wouldn't execute pause when X and P are not specified.

3.2.6 Single direction positioning (G60)

Instruction format:G60 X_ Y_ Z_



Picture 3-2-6-1

Function: In order to elimnate machine ballscrew's backlash to accurate positioning, it can use G60 to positioning from one direction.

Explanations:

G60 is non-modal G code (it can set to be modal value or not by bit parameter NO:52#2), it is only valid in specified program segment.

Parameters X, Y and Z are end-point's coordinate in absolute programming, and are the movement distance of tool in increment programming. The positioning path is the tool compensation path when use single direction positioning under the tool offset.

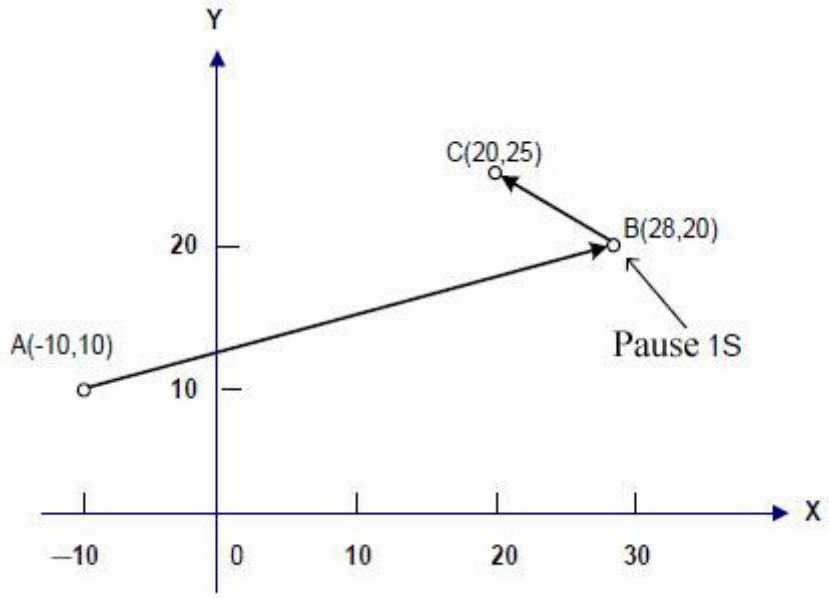
In the above picture, marked overshoot value can be set tought parameters P351, P352, P353, P354 and P355, pause can be set by P350, positioning direction can be ensure by over travel value's plus or minus, please refer system parameters for specific circumstance.

Example 1:

G90 G00 X-10 Y10;

G60 X20 Y25; (1)

When system is at parametersP350=1, P351=-8, P352=5; to sentence (1), tool path is AB →pause 1s → BC



Picture 3-2-6-2

System parameter:

Form 3-2-6-1

P350	Pause time of single direction positioning (unit: s)
P351	X axis single positioning direction and over travel value (unit: mm)
P352	Yaxis single positioning direction and over travel value (unit: mm)
P353	Z axis single positioning direction and over travel value (unit: mm)
P354	4th axis single positioning direction and over travel value (unit: mm)
P355	5th axis single positioning direction and over travel value (unit: mm)

Notice: 1) The symbol of parameter P351 ~ P354 means the direction of single direction positioning, parameter value means over travel value.

2) Over travel value >0, positioning direction is positive direction;

3) Over travel value <0, positioning direction is negative direction;

4) Over travel value =0, will not do single direction positioning.

3.2.7 System parameter's online change (G10)

Function description: This function uses in program for setting or amending screw pitch error offset, tool radius, length offset, external zero point offset, work piece zero offset, additional workpiece zero point offset, data parameter and bit parameter etc.

Instruction format:

G10 L50 N_P _R_; set or change bit parameter

G10 L51 N_ R_; set or change data parameter

G11; cancel parameter's input mode

Parameters definition:

N: Parameters number. Serial number of parameter needs to modify.

P: Parameters bit number. Bit number of parameter needs to modify.

R: Modified value. Which used for spefified the value after modify.

Also can amend specified value through the following instructions, please refer relative chapters for details:

G10 L2 P_X_Y_Z_A_B_;set or amend external zero-point offset or workpiece end-point offset.

G10 L10 P_R_;set or amend length offset

G10 L11 P_R_;set or amend length abrasion value

G10 L12 P_R_;set or amend radius offset

G10 L13 P_R_;set or amend radius abrasion value

G10 L20 P_ X_Y_Z_A_B_; set or amend additional workpiece zero-point offset

Note: 1) Specify other NC statement is not allowed under parameter input mode, except annotation etc.

2) G10 program segment must be commanded sperately, otherwise it will alarm, please remember to use G11 cancel parameter input mode after use G10, and in case affect program.

3) Parameter value which modifie by G10 must meet system parameter's range, or it will alarm;

4) It must cancel fixed cycle's modal instruction before run G10, otherwise it will alarm;

5) For some parameters modification, controller should reboot. These parameters can not modify by G10.

3.2.8 Workpiece coordinate G54 ~ G59

Instruction format: G54 ~ G59

Function: specified current workpiece coordinate, specifies workpiece coordinate by G code

Explanations:

1) No instruction parameter;

2) System can set 6 workpiece coordinates by itself, it can choose any coordinate from G54~G59.

G54 ---------- workpiece coordinate1

G55 ---------- workpiece coordinate2

G56 ---------- workpiece coordinate3

G57 ---------- workpiece coordinate4

G58 ---------- workpiece coordinate5

G59 ---------- workpiece coordinate6

3) When startup, system shows the coordinate G54 ~59 or additional workpiece which executed before power off;

4) The instruction movement axis will be positioning new workpiece coordinate point when program segment call different workpiece coordinate; the axis whitch with no instruction movement, coordinate will skip to the coordinate value relatives to new workpiece, while real machine tool place wouldn't be changed.

For example: The machine tool coordinate is (10, 10, and 10) relatives to G54 coordinate zero point;

The machine tool coordinate is (30, 30, and 30) relatives to G55 coordinate zero point;

Execute program in sequence, the end-point coordinate and machine tool coordinate as below:

Form 3-2-8-1

Program	Absolute coordinate	Machine tool coordinate
G0 G54 X50 Y50 Z50	50, 50, 50	60, 60, 60
G55 X100 Y100	100, 100, 30	130, 130, 60
X120 Z80	120, 100, 80	150, 130, 110

5) It can change external workpiece offset or workpiece zero-point offset by G 10, the method as below:

Use instructions G10 L2 Pp X_Y_Z_

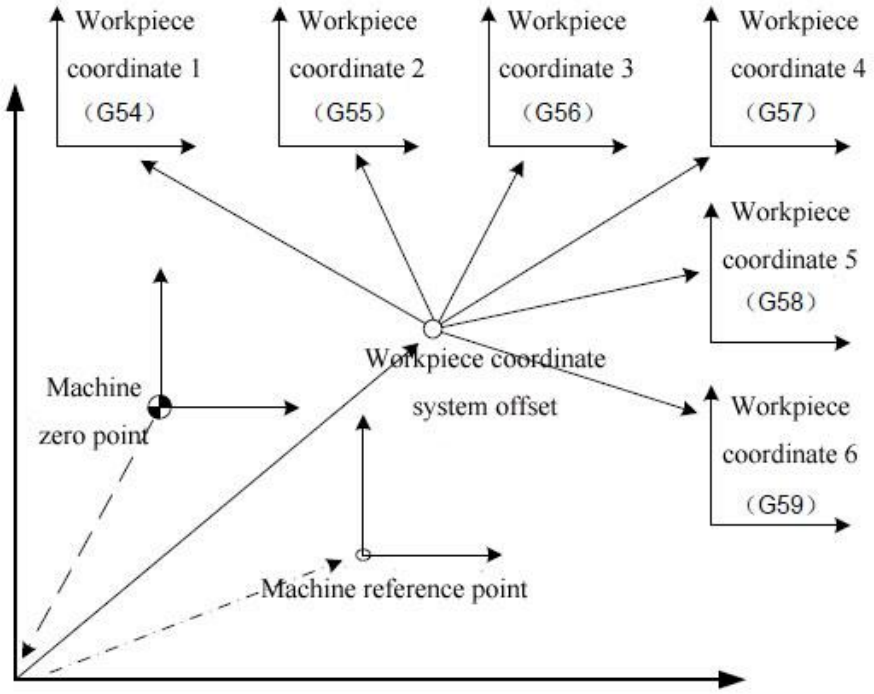
P=0: external workpiece zero-point offset (based offset);

P=1 to 6: workpiece zero-point offset of workpiece coordinate 1 to 6;

X_Y_Z_: absolute instruction is the workpiece zero-point offset value.

Increment instruction G91 is the added workpiece offset of each axis (the added result is the new workpiece zero-point offset)

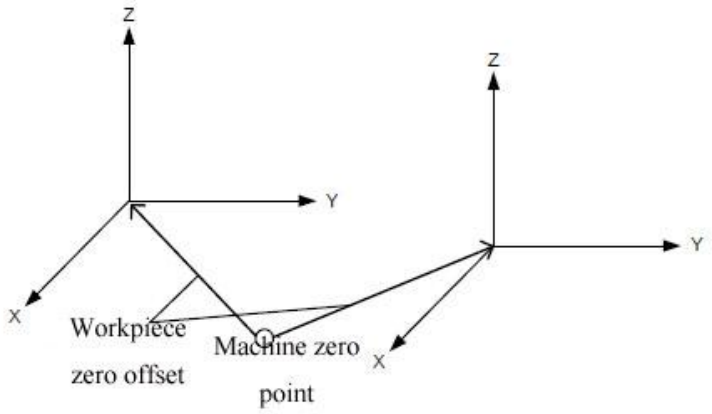
Workpiece coordinate can be changed respectively by G10 instruction.



Picture 3-2-8-1

As the above picture 3-2-8-1 shows, manual reset to machine zero point after machine tool startup, machine zero

point builds machine tool coordinate system and then generate machine tool's reference point and confirm workpiece coordinate. The values workpiece coordinate offset data parameter P260~264 are 6 workpieces' complete offset. It can specify 6 workpiece coordinate's zero point by input coordinate offset or set data parameter P265~P294, the 6 workpieces is setting according to the distance from mechanical zero point to each coordinate zero point.



Picture 3-2-8-2

For example: N10 G55 G90 G00 X100 Y20;

N20 G56 X80.5 Z25.5;

In above examples, positioning to workpiece coordinate G55 (X=100, Y=200) fastly when execute N10 program segment. Positioning to workpiece G56 fastly and absolute coordinate value change to coordinate (X=80.5, Z=25.5) of G56 coordinate system automatically when execute N20 program segment.

3.2.9 Additional workpiece coordinate system

System can also use 50 additional workpieces coordinate system, except 6 workpieces coordinate system (G54 toG59)

Instruction format: G54 Pn

Pn: specify additional workpiece coordinate code. The range of Pn is 1 ~ 50.

The seting and limit of Additional workpiece coordinate are the same with G54 ~ G59.

It can set workpiece zero offset value by G10 in additional workpiece coordinate. The method as below:

Instruction: G10 L20 Pn X_Y_Z_;

N=1 to 50: additional workpiece system codes;

X_Y_Z_: set the axis address and offset value of workpiece zero offset;

For absolute value instruction (G90), specified value is the new offset value.

For increment value instruction (G91), specified value plus current offset value to a new offset value;

Each workpiece coordinate can be changed separately by G10;

When additional workpiece coordinate address P and other instruction which has address P in the same segment, it will use one address P.

3.2.10 Choose mechanical coordinate system G53

Instruction format: G53 X_ Y_ Z_

Function: make the tool to positioning in the coordinate of machine tool's coordinate system.

Explanations:

1. When G53 in program, its instruction coordinate should be the coordinate value of machine tool and machine tool will be positioning to specified place fastly.

2. It is only valid in current segment when G53 is not modal instruction, and wouldn't affect defined coordinate system.

Restriction:

Select mechanical coordinate system G53

When a position on a machine coordinate system is specified, the tool moves to the position by rapid traverse. G53, which is used to select a mechanical coordinate system, is a non-modal G code. It is valid only in the block that specifies the machine coordinate system. It should specified G90 for G53, and will be igored when specify G53 and G91 in Increment value mode (G53 will continue G90 mode but not change G91 modal). When the tool moves to the specific position, for example: tool exchange position, travel program on the machine coordinate system should be edited by G53.

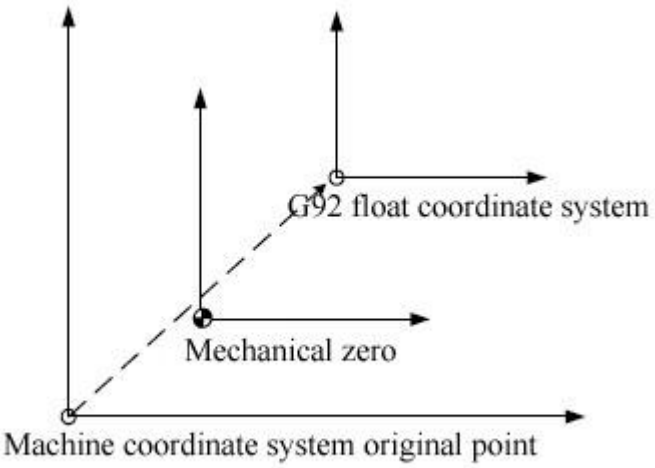
Notice: When the G53 command is specified, cancel the radius compensation and tool length offset, then recovery in the next program segment.

3.2.11 Float coordinate system G92

Instruction format: G92 X_ Y_ Z_

Function: Set float workpiece coordinate. 3 instruction parameters specify current tool'sabsolute coordinate value in new float workpiece coordinate. This instruction wouldn't generate movement of motion axis.

Explanations:



Picture 3-2-11-1

1. As picture 3-2-11-1 shows, the zero point of G92 float coordinate is the value of machine tool coordinate system, it has nothing to do with the workpiece coordinate system, it can only build after machine reset.

   For G92's seting availability is valid in the following circumstance:

   1) System power off

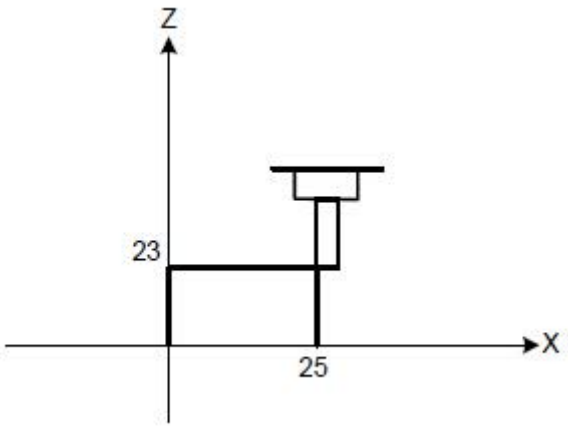
   2) Before call workpiece coordinate

   3) Before machine reset operation

   G92 float coordinate system usually used for temporary workpiece processing alignment, it will lose after power off. Normally, G92 instruction runs in the start of program or before automatic program of MDI mode.

2. There are two methods for confirm the float coordinate:

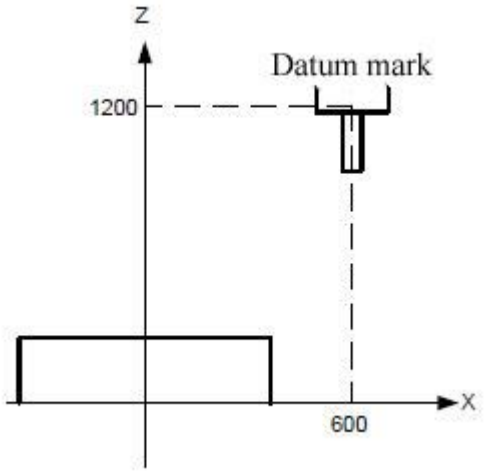
   1) Decide coordinate by tool nose:



Picture 3-2-11-2

As picture 3-2-11-2 display, G92 X25 Z23 regards the place of tool nose as float coordinate system (X25, Z23).

2) Decide coordinate system based on a fixed point of knife handle:



Picture 3-2-11-3

As picture 3-2-11-3 shows, set coordinate system (based on one datum point of handle as the cutting point) by instructions G92 X600 and Z1200. Regards one datum point of tool handle as starting point, datum point moves to instruction place if moves according to absolute value instruction, it must add tool length compensation and its value as the distance from datum point to tool nose.

Notice: 1. if set G92 coordinate system in tool offset, then it didn't add tool offset for tool length compensation.

2. For tool radius compensation, tool offset should be cancelled when use G92 command.

Restriction:

The frist fixed cycle instruction must be complete format after setting float coordinate, or the tool runs wrong.

3.2.12 Plane selection G17/G18/G19

Instruction format:G17/G18/G19

Function: The plane of circular interpolation and tool radius compensation are chosen by G17/G18/G19.

Explantion:no instruction parameters, the default is G17 plance when startup, and also can set bit parameters N0: 48#1, #2, #3 to decide the default plane when startup. The corresponding relations of instruction and plane:

G17----------XY plane

G18----------ZX plane

G19----------YZ plane

G17, G18, G19, the plane will not be changed when a block without any command inside

For example: G18 X_ Z_; ZXplane

G0 X_ Y_;plane wouldn't be changed (ZX plane)

What's more, movement instruction has nothing to do with plane selection. For example, in the following instruction, Y axis is not at ZX plane and Y axis' movement has nothing to do with ZX plane.

G18Y_;

Reminder: Currently, it only supports the fixed cycle in G17 plane, it's better to point out the plane when programming, especially at the circumstance that many people use one system, in order to avoid abnormal or accident which caused by programing.

3.2.13 Polar coordinate start/cancel G16/G15

Instruction format: G16/G15

Function:

G16 Start the polar coordinate mode

G15 Specify polar coordinates of the location parameters as the beginning of the presentation

G16 Specify polar coordinates of the location parameters as the cancel of the presentation

Explantion:

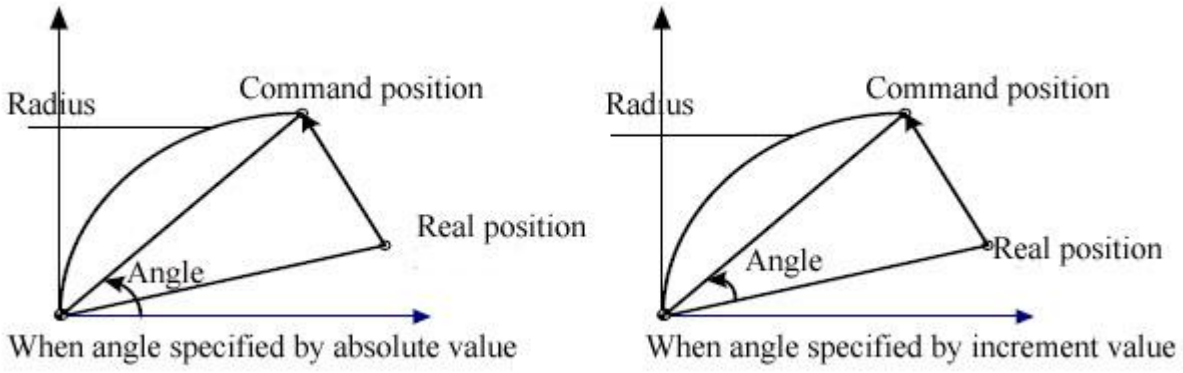
No instruction parameter.

Set G16 can use coordinate radius and angle input. When the specified angle is positive, the polar coordinate rotates anticlockwise of the 1st axis's positive direction in the currently selected plane, and the polar coordinate rotates clockwise when it is negative. Absolute command or incremental command (G90, G91) can be used in polar coordinate radius and angle

The positioning parameter first axis of tool movement instruction signifies the polar radius of polar coordinate system and second axis signifies polar angle after G16 appears. It can cancel polar coordinate mode and make coordinate value return to input by right angle coordinate by setting G15.

For the rule of polar coordinate zero point:

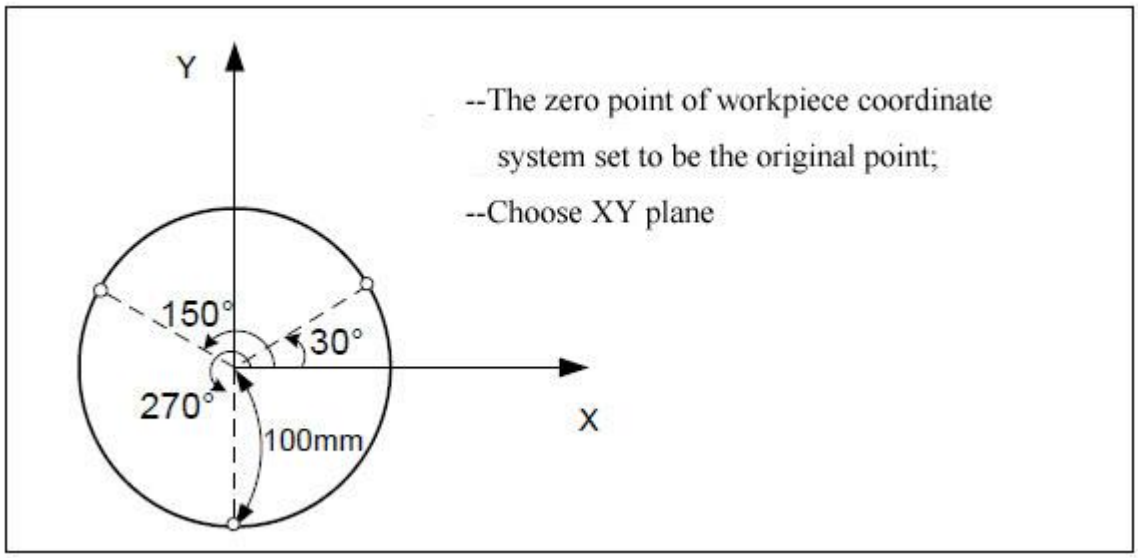
1.In G90 absolute mode, the workpiece coordinate system zero point is set to be the polar coordinate original specify by G16.



Picture 3-2-13-1

2. In G91 increment mode, the current point is set to be polar coordinate original point by G16.

For example: bolt hold circle (the zero point of workpiece coordinate set to be the zero point of polar coordinate, select X-Y plane)



Picture 3-2-13-2

Specify angle and radius by absolute value

G17 G90 G16; specify polar coordinate instruction and selection XY plane, set the zero point of the workpiece coordinate system as the origin of the polar coordinate.

G81 X100 Y30 Z-20 R -5 F200; specify 100mm as distance and 30° as angle;

Y150; specify 100mm as distance and 150° as angle;

Y270; specify 100mm as distance and 270° as angle;

G15 G80; cancel polar coordinate instruction.

Specify angle with increment, specify polar radius with absolute value.

G17 G90 G16; specify polar coordinate instruction and selection XY plane, set the zero point of the workpiece coordinate system as the origin of the polar coordinate.

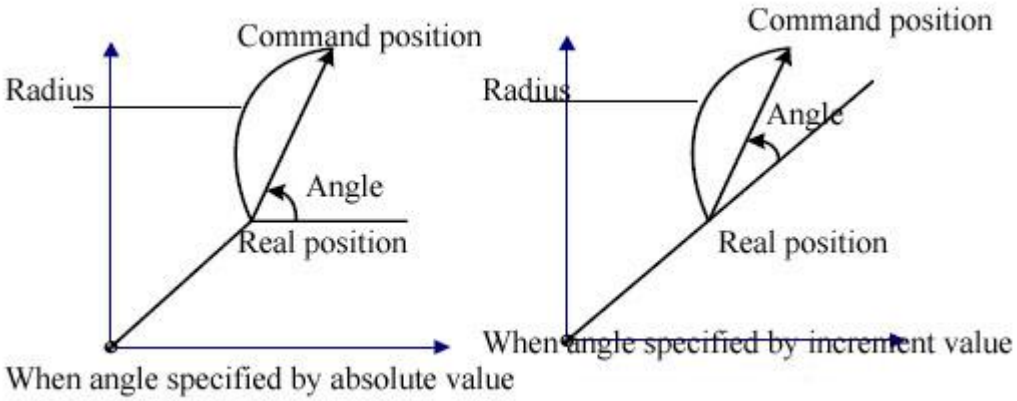
G81 X100 Y30 Z-20 R -5 F200; specify 100mm as distance and 30° as angle;

G91 Y120;specify 100mm as distance and 150° as angle;

Y120; specify 100mm as distance and 270° as angle;

G15 G80;cancel polar coordinate instruction.

In addition, please notice the seting of current coordinate plane when programing polar coordinate. Polar coordinate plane relatives to current coordinate plane, for example: in G91, the current tool place X, Y axis components as original point if the current coordinate place is G17; the current tool place X, Y axis components as original point if the current coordinate place is G18.



Picture 3-2-13-3

The first hole positioning parameters of loop command if not specified after G16, The system tool current location as hole loop positioning parameters by default. After the polar coordinates the first fixed cycle instructions must be complete, otherwise, the feeding is not correct.

In addition to the hole cycle, cutting tool mobile command positioning parameters and digital related to specific plane selection mode after G15. After using the G15 commands to cancel the polar, followed by mobile command, the default current, the starting point of the cutter location for this move.

3.2.14 Scaling G51/G50

Instruction format:

G51 X_ Y_ Z_ P_ (X.Y.Z: absolute command for center coordinate value of scaling; P: same scaling rate)

… Scaling processing program segment

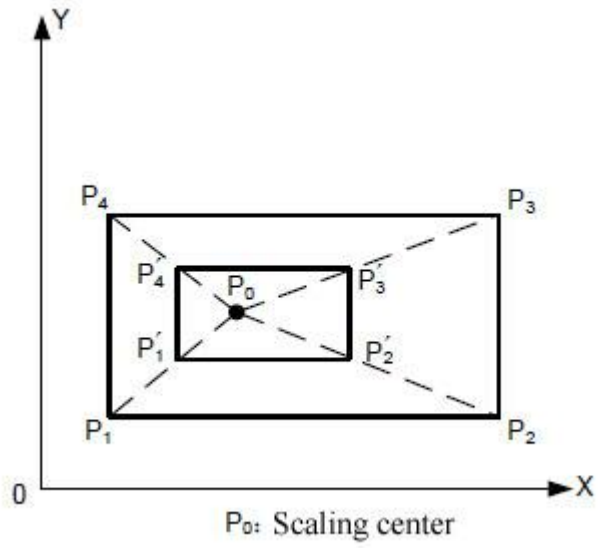
G50 Cancel scaling rate

Or G51 X_ Y_Z_ I_ J_ K_(each axis had different scaling rate(I, J, K) to do scaling)

… Scaling processing program segment

G50 Cancel scaling rate

Function: G51 makes specify place as the centre of the programing shape, amplification or scaling same rate. It should be pointed out that G51 should be specified by single program segment (or it will occur unexpect situation that workpiece damage or people injuries) and cancel G50.



Picture 3-2-14-1 Scaling rate (P1P2P3P4 → P1'P2'P3'P4')

Explanation:

1. Zoom centre: G51 can take 3 positioning parameters X_Y_Z as selection parameter. System will regard the tools' current place as rate zoom centre if didn't specify positioning parameter. No matter the current positioning mode is absolute mode or increment mode, the zoom centrespecified by absolute positioning mode. In addition, parameters of G51 also in rectangular coordinate system under polar coordinate G16 method

   For example:G17 G91 G54 G0 X10 Y10;

   G51 X40 Y40 P2; Increment mode, scaling centre is absolute coordinate (40, 40) of G54 coordinate system.

G1 Y90; Parameter Y adopts increment mode.

2. Scaling rate: No matter the current positioning mode is G90 mode or G91 mode, the scaling rate shows in absolute mode.

Scaling rate can be specified in program and also be set in parameter, data parameter P345~347 correspoding to scaling rate X, Y, Z respectively, data parameter P344 seting value starts to zoom if there is no scaling rate instruction.

If specified parameter P or I, J and K' parameters are negative values, then the corresponding axis to running mirror image.

3. Scaling seting: bit parameter N0: 43#2 set if X axis scaling is vaild or not, bit parameter N0: 43#3 set if Y axis scaling is vaild or not, bit parameter N0: 43#4 set if Z axis scaling is vaild or not, bit parameter N0: 43#7 sets scaling ration specify mode of each axis (0: each axis uses P instruction; 1: each axis use I, J and K instruction)

4. Scaling canceled: when cancel rated scaling by G50 and follow movement instruction closely, then make the tool location as original point of movement instruction.

5. Scaling status: can not specify the G code (G27~G30) which returnsreference point and G code (G53~G59, G92 etc) of instruction coordinate. If these G code must be specified, it should be specified after cancel scaling function.

6. Even though specify different scaling rates to circle interpolation and exis, tool wouldn't draw ellipse path.

When magnification for axis are different, and the circular interpolation is programmed with radius R, its figure is as picture3-2-14-2 shows (magnification 2 is applied to X-axis and magnification 1 applied to Y axis)

G90 G0 X0 Y100;

G51 X0 Y0 Z0 I2 J1;

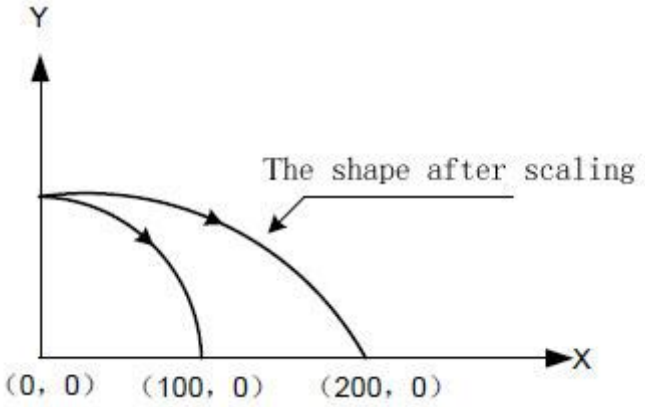
G02 X100 Y0 R100 F500;

Above commands are equivalent to the following commands:

G90 G0 X0 Y0 Z0;

G02 X200 Y0 R200 F500;

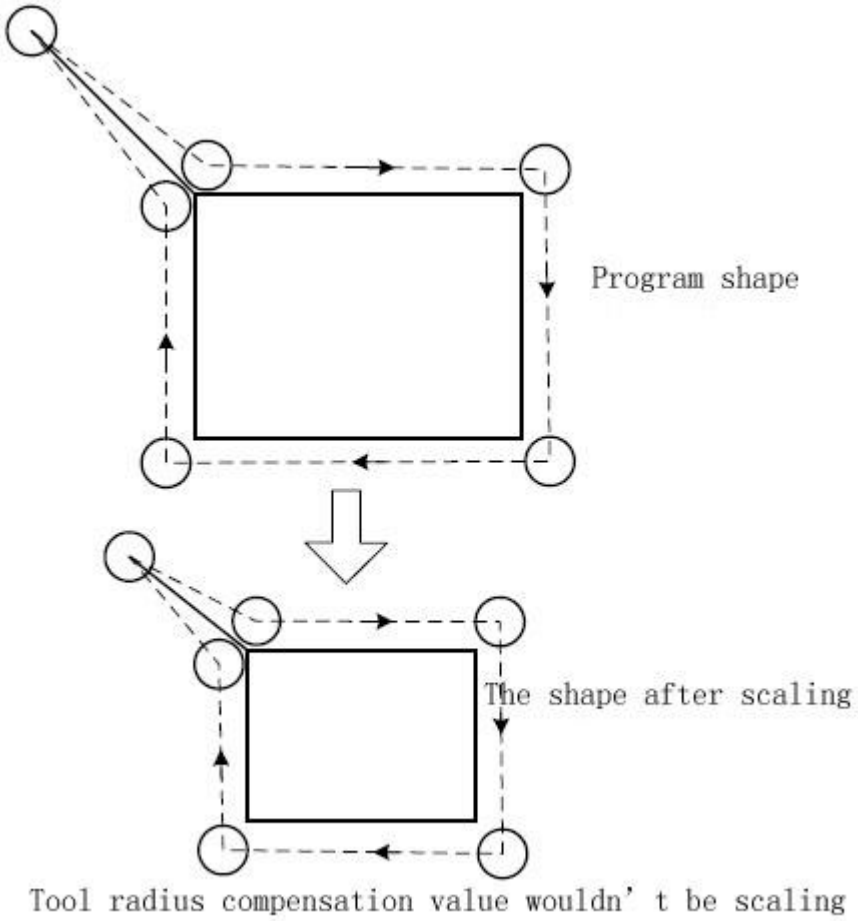
Magnification of radius R is depends on I or J whichever is larger



Picture 2-14-2 the scaling of circular interpolation 1

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

7. Rated scaling is invalid for tool radius compensation value, tool length compensation and tool offset value, as picture 3-2-14-3



Picture 3-2-14-3 rated scaling of tool radius compensation

Examples of mirror image program:

Main program

G00 G90;

M98 P9000;

G51 X50.0 Y50.0 I-1 J1;

M98 P9000;

G51 X50.0 Y50.0 I-1 J-1;

M98 P9000;

G51 X50.0 Y50.0 I1 J-1;

M98 P9000;

G50;

M30;

Sub-program

O9000;

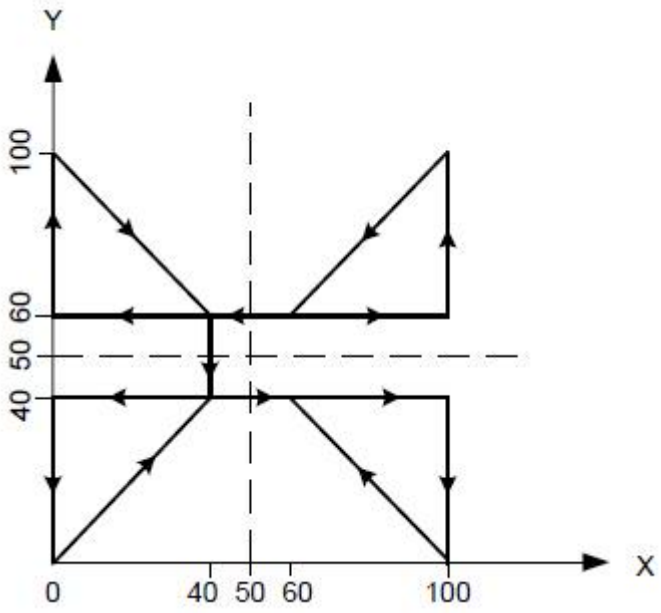
G00 G90 X60.0 Y60.0;

G01 X100.0 F100;

G01 Y100;

G01 X60.0 Y60.0;

M99;



Picture 3-2-14-4

Restriction:

1. Z xis movement scaling is invalid in the following fixed cycle:

1) Value of cut point Q and return value D of deep hole drilling cycle (G83, G73).

2) Fine boring cycle (G76)

3) Offset value Q of Back to boring (G87)' X axis and Y axis.

2. Movement distance can't use scaling function when runs manually

Notice: 1. location displays the coordinate value after rated scaling.

2. The results as below when specified plane has an axis running mirror image:

1) Arc instruction ---------- reverse rotation

2) Tool radius compensation C ---------- forward rotation

3) Coordinate rotation ---------- reverse rotation angle

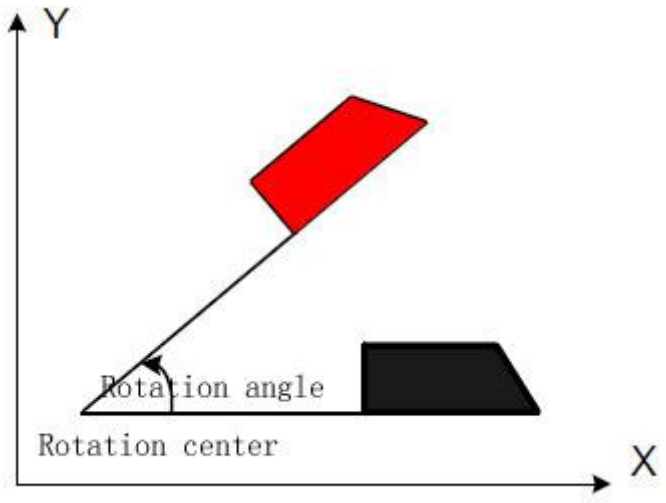
3.2.15 Coordinate rotation G68/G69

The programmed shape can be rotated. By using this function (rotation command), a workpiece can be rotated with a specific angle. If the pattern of the workpiece comprising some identical shapes,the time required for programming and the length of the program can be reduced by editing a subprogram and calling it with the rotation command of the main program. The function is as follows: **Instruction format:** G17 G68 X_ Y_ R_;

or G18 G68 X_ Z_ R_;

or G19 G68 Y_ Z_R_ ;

G69



Picture 3-2-15-1

Function: G68 specify one point as the rotation centre of programming shape in the plane. G69 uses for cancel coordinate rotation.

Explaination:

1. G68 can take 2 positioning parameter as selection parameter. Positioning parameter use for specify the rotation centre. System will rotate around the tool's current location if don't specify rotation centre. Positioning parameter is related to current coordinate plane, choose X and Y in G17, choose Z and X in G18, choose Y and Z in G19.

2. No matter the current positioning mode is absolute mode or relative mode; rotation centre can only be specified by rectangular coordinate absolute positioning mode. G68 can also take a command parameter R, the parameter value is the rotation angle and positive value means anticlockwise rotation, unit as degree.

In coordinate rotation, the rotation angle set by parameter P342 when there is no rotation angle command.

3. In G91, system regards the tool's current location as the rotation centre; whether the rotation angle execute increment or not, the parameter NO: 42#3 (G68 coordinate rotation angle, 0: absolute instruction; 1: G90.G91 instruction) sets it.

4. Plane selection execute is not allowed when system in rotation mode, or it will alarm. Please notice it when programming.

5. In coordinate rotation mode, the G codes that return to reference point (G27 ~ G30 etc) and instruction coordinate system (G53 ~ G59, G92 etc) can be specified. If those G codes must be specified, then cancel the rotation function first.

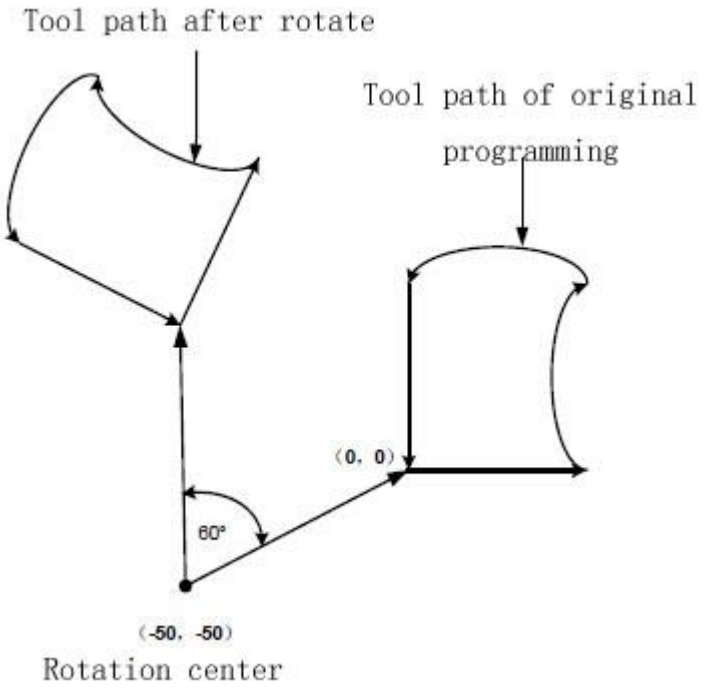
6. Tool radius compensation, tool length compensation, tool offset and other compensation operation can be

executed after coordinate system rotation.

7. Execute coordinate system rotation instructon in ration scaling mode (G51), the coordinate value in the centre of rotation will also be scaling, except the rotation angle; ration scaling executed first then coordinate rotation when send out movement command.

For example 1: Rotation:

```
G92 X-50 Y-50 G69 G17;
G68 X-50Y-50 R60;
G90 G01 X0 Y0 F200;
G91 X100;
G02 Y100 R100;
G3 X-100 I-50 J-50;
G01 Y-100;
G69;
M30;
```



Picture 3-2-15-2

For example 2: Rotation scaling and rotation:

```
G51 X300 Y150 P0.5;
```

G68 X200 Y100 R45;

G01 G90 X400 Y100;

G91 Y100;

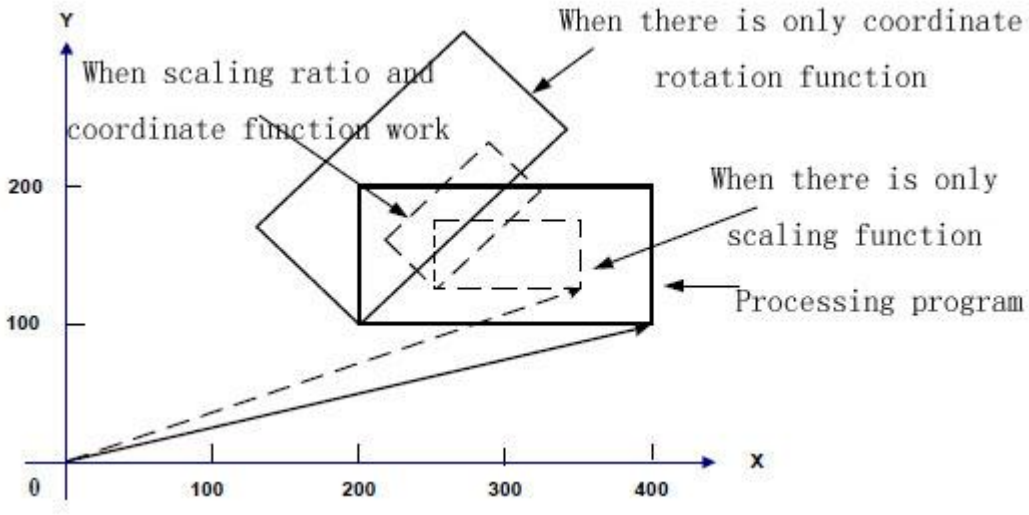
X-200;

Y-100;

X200;

G69 G50;

M30;



Picture 3-2-15-3

For example 3: use G68 repeatly

According program (main program)

G92 X0 Y0 Z20 G69G17;

M3 S1000;

G0 Z2;

G51 X0Y0 I1.2 J1.2;

G42 D01; (tool offset)

M98 P2100(P02100); (call sub-program)

M98 P2200L7;(call 7 times)

G40;

G50;

G0 G90 Z20;

X0Y0;

M30;

Sub-program 2200

O2200

G91

G68 X0 Y0 R45.0; (relative rotation angle)

G90;

M98 P2100; (Sub-program O2200 call sub-programO2100)

M99;

Sub-program 2100

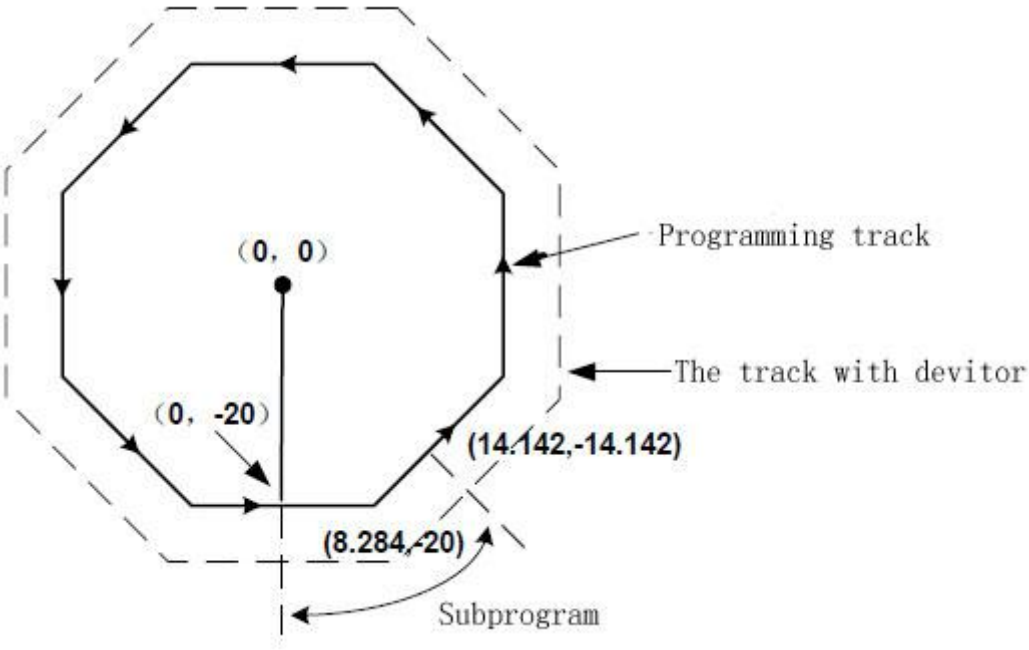
O2100 G90 G0 X0 Y-20; (build in right tool compensation mode)

G01Z － 2 F200;

X8.284;

X14.142 Y-14.142;

M99;



Picture 3-2-15-4

3.2.16 Skip function G31

Instruction format: G31 X_Y_Z_

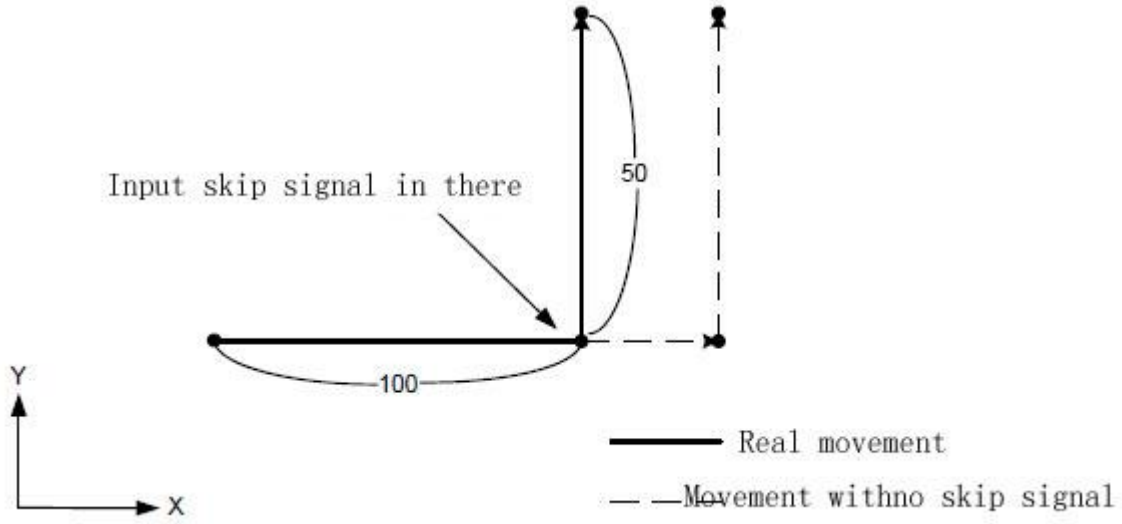
Function: As G01 linear interpolation is performed after G31, if an external SKIP signal is valid during execution of this command, execution of this command is interrupted and the next block is executed. The skip function is used when the end of machining is not programmed but specified with a signal from the machine, for example, in grinding. It is used also for measuring the dimensions of a workpiece.

Explanation:

1. G31, which is a non-modal G-code, it is effective only in the block in which it is specified.

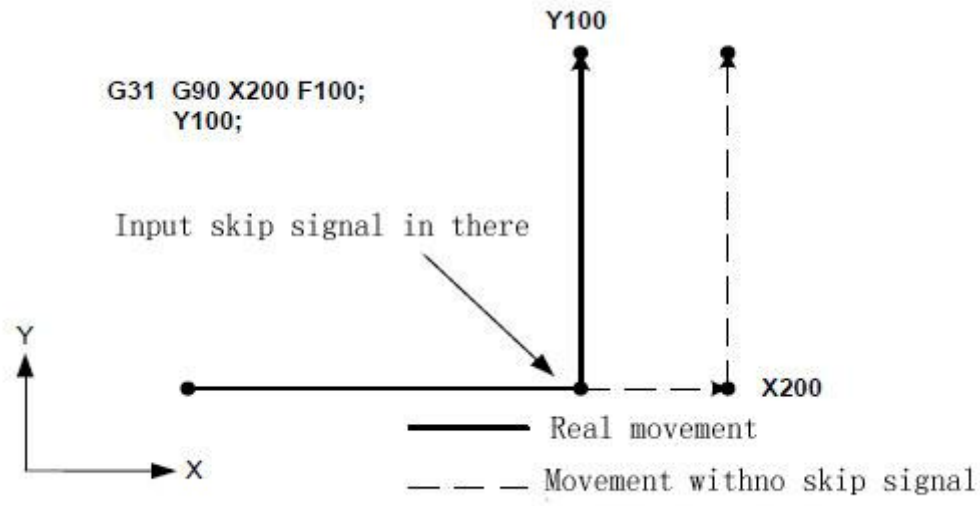
2. G31 can not be specified in the tool compensation or the alarm will be generated. It is necessary to cancel the tool compensation.

For example: the next block to G31 is increment command's single axis movement, as picture 3-2-16-1 shows:



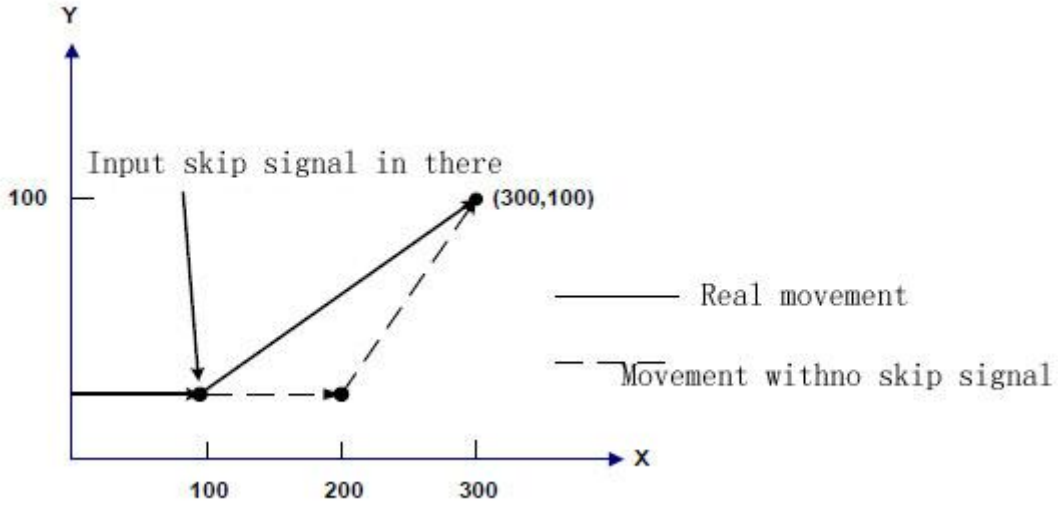
Picture 3-2-16-1 next block is increment command's single axis movement

Next block to G31 is increment command's single axis movement, as picture 3-2-16-2 shows:



Picture 3-2-16-2next block is increment command's single axis movement

Next block to G31 is increment command's two axis movement, as picture 3-2-16-2 shows:



Picture 3-2-16-3next block is increment command's two axis movement

Notice: DNC and MDI single block mode doesn't support program skip.

3.2.17 Conversion of Inch and metric G20/G21

Instruction format: G20: Inch input

G21: metric input

Function: The input unit either inch or metric is achieved.

Explanation:

The following value will be changed after conversion inch and metric:

The feeding speed, position command, workpiece zero point offset value, tool compensation value, the scale unit of manual pulse generator and movement distance of increment feeding.

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

Notice:

1. Inch and metric can be switched when execute program.

2. Inch switched to metric or opposite, tool compensation should be preset according to the minimum input increment unit.

3. Inch switched to metric or opposite, the first G28 instruction runs from middle point is same with manual return reference point.

4. When the minimum input increment unit and minimum instruction increment unit are different, the maxium error is half of the minimum instruction unit, and this error wouldn't be accumulated.

5. Set program input inch/metric by parameter N0: 01#0.

6. Set program input inch/metric by parameter N0: 04#7.

3.2.18 Any angle chamfer/corner circle arc

Instruction format: ,L_: chamfer

, R_: corner circle arc transition

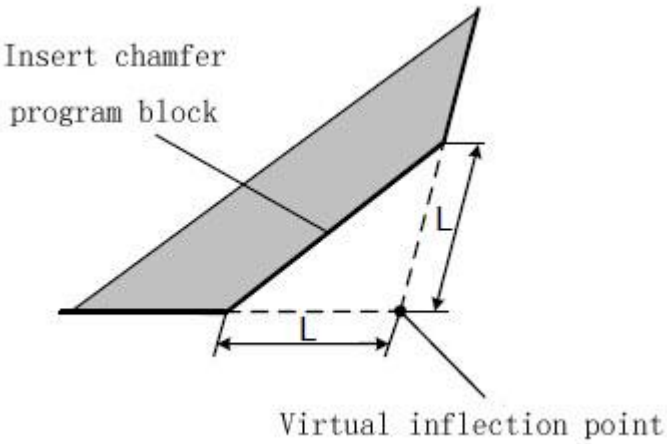
Function: The above instruction should be added at the end of program linear interpolation (G01) or circle arc interpolation (G02, G03), chamfer or transition circle arc will be added to corner during processing. The program blocks of chamfer or corner circle arc transition can be specified continuous.

Explaination:

1. Chamfering after L, specify the distance from virtual corner to start point and end point of corner, if virtual corner is assumption inexecution chamfer, the real corner point as below picture:

(1) G91 G01 X100 ,L10;

(2) X100 Y100;

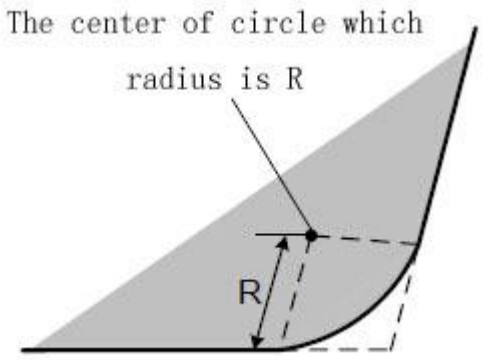


Picture 3-2-18-1

2. Corner circle arc transition after R, specified corner circle arc's radius, as below picture:

(1) G91 G01 X100,R10;

(2) X100 Y100;



Picture 3-2-18-2

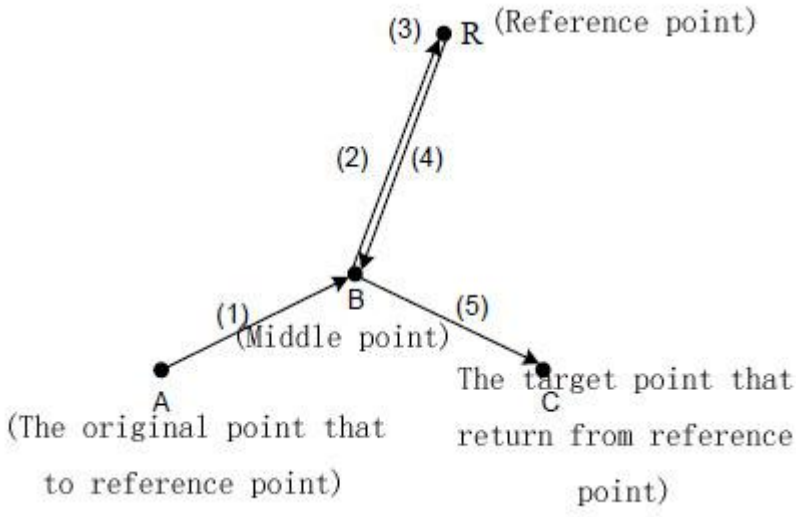
Restriction:

1. Chamfer and corner circle arc can only operated in the specified plane, plane axis can not execute these functions.
2. If inserted chamfer or circle arc transitional program segment cause tool exceed original interpolation movement range, it will alarm.
3. Corner circle arc transition can't be specified in screw thread programing block.
4. System adopts its absolute valuewhen instruct chamfer value or corner value is negative.

3.3 Reference point G code

Reference point is a fixed point on machine tool, use reference point return function, tool can move to this location easily.

For reference point, there are three instruction operation methods, through G28 as picture 3-3-1, can through G28, can use tool through middle point and along specified axis move to reference point automatically; through G29, can use tool through middle point from reference pointm along specified axis move to specified point automatically.



Picture 3-3-1

3.3.1 Return reference point G28

Instruction format: G28 X_Y_Z

Function: G28 instruction used for execute middle point

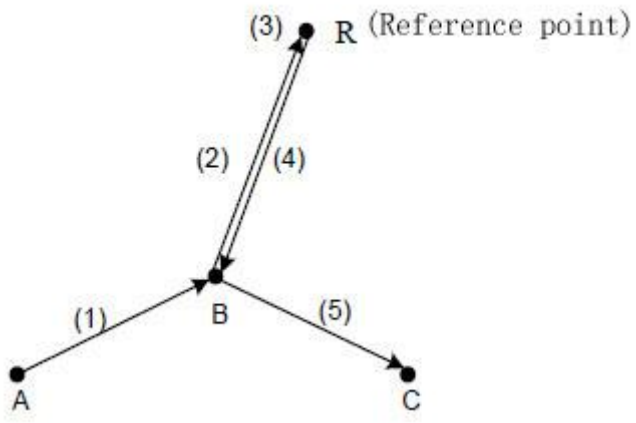
Explanation:

Middle point:

Middle point is specified by the instruction parameter which through G28, it can express by absolute instruction or increment instruction. In this execute program segment, it also stores the middle point coordinate of instruction axis to supply usage for G29 (return from reference point)

Notice:

The coordinate of middle point is stored in CNC, but it only stores G28 coordinate value of instruction axis, other non instruction axis uses coordinate value which G28 used. So when users use G28 instructions, it's better to specify each axis if you are not sure the default middle point. Please consider it combine the N5 program segment of following example 1.



Picture 3-3-1-1

1. The G28 program segment's decomposition of movement as below (please refer picture 3-3-1-1):

   (1) Fast positioning from current position to middle point of instruction axis (A point – B point)

   (2) Fast positioning from middle point to reference point (B point – R point)

2. G28 is non-modal instruction and only works for current segment.

3. Support the combination of single axis or multiple axis returns to reference point, system stores coordinate of middle point when change coordinate of workpiece.

Example:

N1 G90 G54 X0 Y10;

N2 G28 X40; set the middle point of X axis is X40 of G54 workpiece coordinate, return to reference point through point (40, 10), namely X axis returns to reference point alone.

N3 G29 X30; from reference point (40, 10) return to point (30, 10), namely X axis returns to target point alone.

N4 G01 X20;

N5 G28 Y60; middle point (X40, Y60), X axis has no instruction, so replaced by X40 which instructed by G28 mentioned before. Notice: middle point is not (20, 60).

N6 G55;workpiece coordinate system changes, then middle point changes to G55's coordinate (40, 60) by G54's coordinate (40, 60).

N7 G29 X60 Y20; reference point through of G55 workpiece coordinate's middle point (40, 60) return to point (60, 20).

G28 can cancel tool compensation automatically. But this instruction usually used in automatic tool exchang (namely tool exchange in reference point after return to reference point), so the tool radius compensation and tool

length compensation should be canceled when use this instruction. Please refer data parameter P50 ~ P54.

3.3.2 Return 2, 3, 4 reference point G30

Set 4 reference points in machine tool coordinate system, but in the system which has no absolute location detector, only in, Set four reference point in the machine tool coordinate system, But there is no absolute position detector in the system, Only in returns reference (G28) on an automatic or manual after return reference point, Can use return 2, 3, 4 reference point function.

Instruction format:

G30 P2 X_ Y_ Z_; return the second 2 reference point (P2 can be elliptical)

G30 P3 X_ Y_ Z_; return the third reference point

G30 P4 X_ Y_ Z_; return the forth reference point

Function: G30 execute the operation from middle point to reference point which specified by G30.

Explanation:

1. X_Y_Z; the instruction which specified middle location (absolute value/increment value instruction)
2. G30 instruction seting and restriction are the same with G28, the seting of the second, third and forth reference please refer the data parameter P55~69.
3. G30 instruction can be used with G29 (return from reference point), seting and restriction are the same with G28.

3.3.3 Return G29 from reference point

Instruction format: G29 X_ Y_ Z_

Function: G29 execute the operation from reference point (current point) through the middle point specified by G28, G30, and then return to specified point.

Explanation:

1. The movement of G29 program segment can be broke as below (details please refer 3-3-1-1):
   (1) Positioning from reference point (current point) to the middle point (R point –B point) which definied by G28, G30 in fast movement speed.
   (2) Positioning from middle point to specified point (B point – C point) in fast movement speed.
2. G29 is non-modal information and only valid in current segment. It must specify the instruction that return from reference point after G28 and G30.
3. The selection parameters in G29 instruction format use for specify the target point which returns from reference point (namely the C point of picture 3-3-1-1); it can show in absolute value or increment value. For increment

programing, instruction value specifies the increment value which left middle point. When doesn't specify some axis, it means there is no movement value relative to middle point. The instruction that there is only on axis after G29 is return, other axis wouldn't do any motions.

For example:

G90 G0 X10 Y10;

G91 G28 X20 Y20;return to reference point through middle point (30, 30)

G29 X30;return to (60, 30) from reference and through middle point (30, 30), please notice it is in increment programing mode;the component of X axis is 60.

G29 instruction's middle point assigned by G28 and G30 instructions. Please refer G28 instruction for definition, standard and default of middle point.

3.3.4 Return reference point detection G27

Instructin format: G27 X_ Y_ Z_

Function: G27 execute return reference detection,X_ Y_ Z_ specify the instruction of reference point (absolut evalue/ increment value instruction)

Explanation:

1. G27 instruction, tool positioning in fast speed.If tool reached reference point, then the instruction light is on; but if the reached place is not reference point, it alarm.

2. In tool latching state, tool alrealdy returns to reference point even though specify G27 instruction, the instruction light is not on.

3. In offset mode the place which G27 specified is the place tool reached and added offset place, so the added place is not reference place, then the instruction light is not on and alarm. Normally, the tool offset should be canceled when use G27.

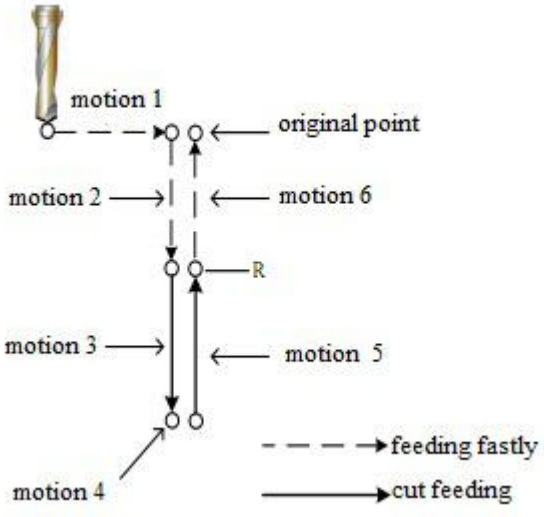
4. The X, Y, Z coordinate points is the place in tool coordinate system.

3.4 Fixed cycle G code

Fixed cycle is that use a program segment which includes G function to achieve a processing motion that can only achieved by multi program segment instructions, simpilify the program. Make programmer programming become easiler (the system only includes the fixed cycle of G17 plane)

General procedure of fixed cycle:

Fixed cycle is consititute of 6 sequece motion, as picture 3-4-1:



Picture 3-4-1

Motion 1: the positioning of X axis and Y axis (also can includes another axis)

Motion 2: move to R point fastly

Motion 3: spot facing work

Motion 4: the motion under hole-bottom

Motion 5: return to R point

Motion 6: move to original point fastly

Positioning in XY plane, hole machining in Z axis. Stipulate each fixed cycle decided by three motions. They are specified by G code.

1) Data form

G90 absolute mode; G91 increment value mode

2)Return point plane

G98 original point plane; G99 R point plane

3)Slot machining mode

G22, G23, G24, G25, G26, G32, G33, G34, G35, G36, G37, G38

4)Hole machining mode

G73, G74, G76, G81 ~ G89

Original point Z plane and R point plane

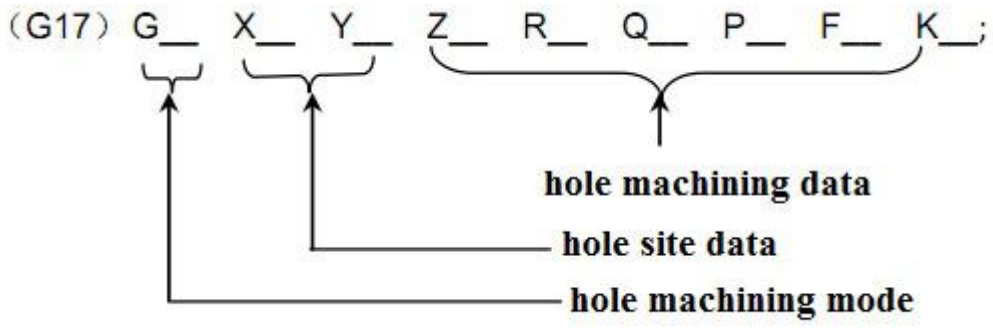
Original point plane: the Z axis absolute location of tool before start fixed cycle

R point plane: also called safe plane, it is the Z axis direction place when fastly turn into of fixed cycle, it usually fixed in a place at a distance from workpiece surface, in case tool hits workpiece and make sure the distance is enough for acceleration.

G73/G74/G76/G81 ~ G89 specified all the data of fixed cycle (hole site data, hole machining data, repeat times), makes it consititute a program segment.

Z, R: hole-bottom parameter Z and parameter R will miss one when execute the first drilling, system will only change the modal, don't execute Z axis motion.

The format of hole machining mode as below:



The basic meaning of hole site data and hole machining data show as form 3-4-1:

Form 3-4-1

Specified the content	Parameter code	Explanation
Hole machining mode	G	Please refer form 3-4-3, and notice above limitation
Hole site data	X, Y	Specified the site of hole by absolute value or increment value, control is the same with G00 positioning
Hole machining data	Z	As picture 3-4-2 (A), specifiy the distance from R point to hole-bottom or specified the coordinate of hole-bottom. Feeding speed in motion 3 is specified by F; in motion 5, it is fast feeding or the speed specified by F code according to the different hole machining
	R	Specify the original point plane to R point distance of picture 3-4-2 (B), or specified the coordinate of R point by absolute value. Feeding speeds are fast in motion 2 and motion 6.

	Q	Specify each penetration value of G73, G83 or transmition quantity (increment value) of G76 and G87
	P	Specified the paused time in hole-bottom. Fixed cycle instructions with parameter P_; in parameter value P_, execute pause operation after tool reach Z plane, unit is ms. Parameter's minimum is set by data parameter P336, parameter's maximum is set by parameter P337
	F	Specify cutting feed speed
	K	Specify repeat times in parameter value K_, K is only valid in specified program segment. It can be omitted and default is once. The max drilling times is 99999, when specify negative value, it will excute its absolute value; it wouldn't exete when it is zero, only the modal changed

Restriction:

Fixed cycle G is modal instruction, it's vaild until cancel fixed cycle G code.

Cancel fixed cycle G code; there are G80 and 01 group G codes.

Once the machining data specified in fixed cycle, then continuously until cancel fixed cycle, it's better specify all the necessary hole machining datas at the beginning of fixed cycle, and only specify changed datas in the following fixed cycle.

Notice:

1.The cutting speed of F instruction will be kept even cancel fixed cycle.

2. Notice: Z axis (cutting axis direction) zoom function is invalid in fixed cycle.

3. Fixed cycle adopts 3 processing methods in single mode, positioning-R plane-original plane

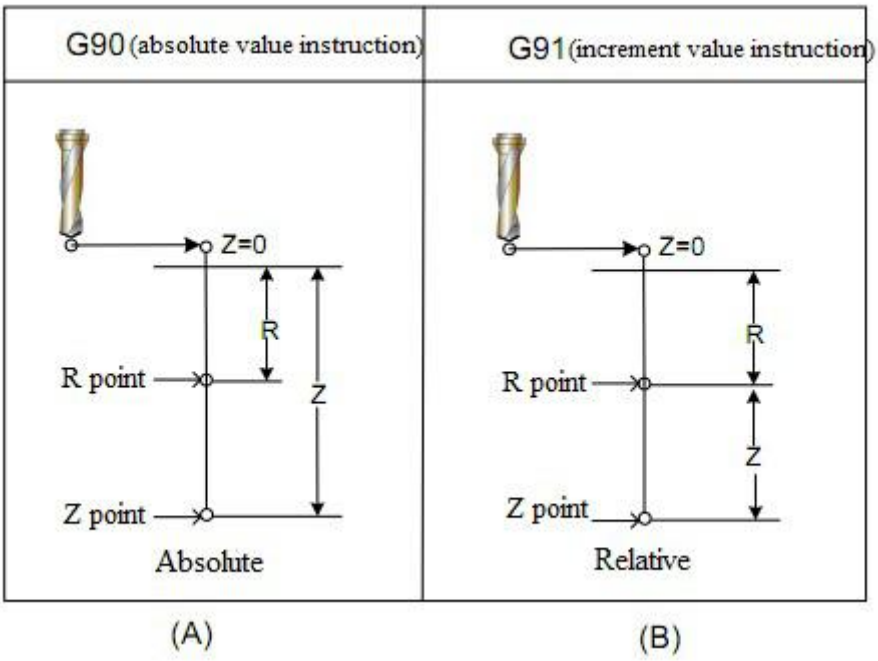
4. In fixed cycle, the hold machining data and hole's position data will be deleted if reset. The above mentioned examples of keep data and delete data as the below form shows:

Form 3-4-2

Sequence	Specified data	Explanation
①	G00X_M3	
②	G81X_Y_Z_R_F_	As it is beginning, specified the needed value for Z, R and F.
③	Y_	As it is the same with hole machining mode and hole machining data of hole②, Z-R-F- can be omitted in G81. Hole's position move in Y, do hole machining one more time.
④	G82X_P_	It is can only move in X axis relatives to hole ③. Machining in mode G82, use Z, R, F which specified by ② and make P which specified by ④as the hole machining data to do hole machining.
⑤	G80X_ Y_	No hole machining. Cancel the entire hole machining data.
⑥	G85X_Z_R_P_	As cancel the entire data in ⑤, So Z, R need to specify again. F is the same with the specified data in ②, so it can be omitted. No need P in this program segment and just retore it.
⑦	X_Z_	The hole machining is just different from ⑥ in Z value, and hole position just move in X axis.
⑧	G89X_Y_	Make the Z specified in ⑦, R specified in ⑥, P and F specified in ② as the hole machining data to doing hole machining in G89 mode.
⑨	G01X_Y_	Remove hole machining mode and hole machining data.

A. The absolute value instruction and increment value instruction of fixed cycle G90/G91

The change of movement distance along with drilling axis relative to G90 and G91 as picture 3-4-2 shows (normally programming by G90, or the Z and R will be negative value if programming by G91)



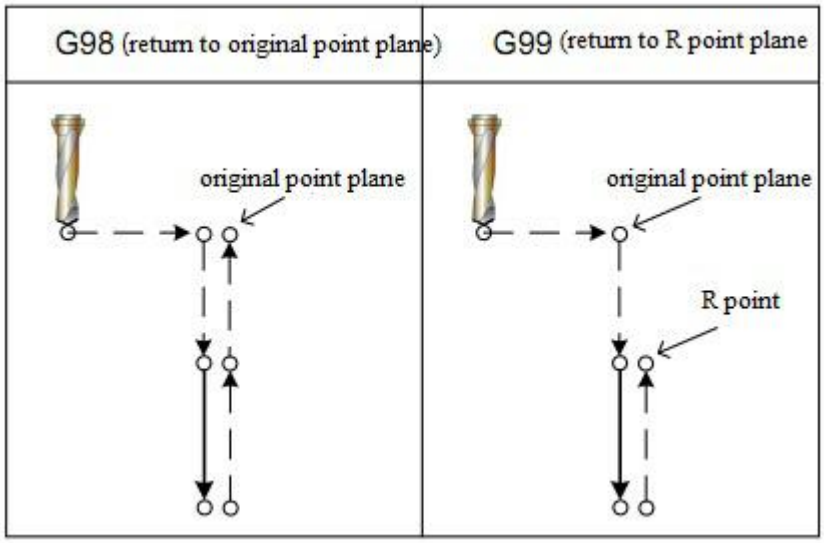
Picture 3-4-2

B. Return plane instruction G98/G99 of fixed cycle

Tool can return to R point plane or original position plane when tool reaches hole-bottom. Tool can return to original point plane or R point plane according to the different G98 and G99.

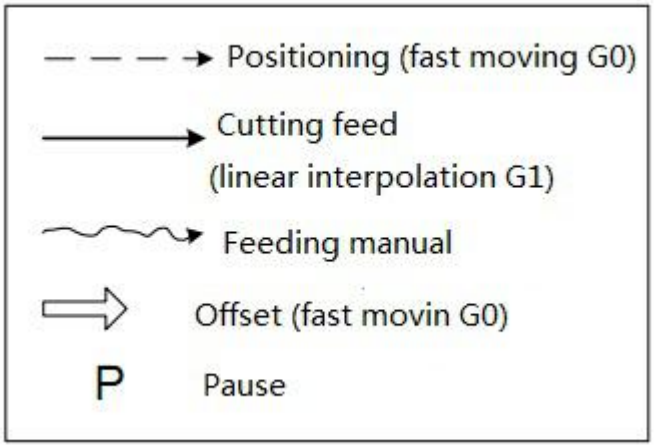
Normally, G99 used for the first drilling and G98 used for the last drilling. Even use G99 state to machining hole, the original plane will not be changed. The motions specified by G98 and G99 as below picture:

System default is G98.



Picture 3-4-3

Each fixed cycle key drawing as below:



Picture 3-4-4

Fixed cycle comparation form (G22 ~ G89)

Form 3-4-3

G code	Drilling (-Z direction)	Hole-bottom motion	Tool retract motion (+Z direction)	Application
G22	Cutting feed		Fast feeding	Anticlockwise circle internal groove rough milling
G23	Cutting feed		Fast feeding	Clockwise circle internal groove rough milling
G24	Cutting feed		Fast feeding	Anticlockwise direction complete circle internal groove cycle
G25	Cutting feed		Fast feeding	Clockwise complete circle internal groove cycle
G26	Cutting feed		Fast feeding	Anticlockwise outer circle groove cycle
G32	Cutting feed		Fast feeding	Clockwise outer circle groove cycle
G33	Cutting feed		Fast feeding	Anticlockwise rectangle groove milling
G34	Cutting feed		Fast feeding	Clockwise rectangle groove milling
G35	Cutting feed		Fast feeding	Anticlockwise tangle internal groove finish milling
G36	Cutting feed		Fast feeding	Clockwise tangle internal groove finish milling
G37	Cutting feed		Fast feeding	Anticlockwise tangle external finish milling

G38	Cutting feed		Fast feeding	Clockwise tangle external finish milling
G73	Pick feed		Fast feeding	High speed deep hole machining
G74	Cutting feed	Pause spindle axis foreward	Cutting feed	Anti tapping cycle
G76	Cutting feed	Spindle axis directional stop	Fast feeding	Precision boring
G80				Cancel
G81	Cutting feed		Fast feeding	Drilling, point drilling
G82	Cutting feed	Tool stop	Fast feeding	Drilling, Ladder boring
G83	Pick feed		Fast feeding	Deep hole machining cycle
G84	Cutting feed		Cutting feed	Tapping
G85	Cutting feed		Cutting feed	Boring
G86	Cutting feed	Spindle axis stop	Fast feeding	Boring
G87	Cutting feed	Spindle axis foreward	Fast feeding	Boring
G88	Cutting feed	Tool stop-spindle axis foreward	Manual operation	Boring
G89	Cutting feed	Pause	Cutting feed	Boring

Restriction:

In fixed cycle positioning, tool radius offset (D) will be omitted.

3.4.1 Circle internal groove rough milling G22/G23

Instruction format:

G22

G98/G99 X_Y_ Z_ R_ I_ L_ W_ Q_ V_ D_ F_ K_

G23

Function: Start from circle centre, do circle interpolation in spiral way until machining programming size circle groove.

Explanation:

G22: anticlockwise circle internal groove rough milling

G23: clockwise circle internal groove rough milling

X, Y: the original point place of X, Y plane

Z: deep machining, it is absolute positionin G90, it is the based level relatives to R in G91.

R: R based level position. It is absolute position in G90;it is the position relatives to the start point of this program segment in G91.

I: circle internal groove radius, I should be larger than the radius of current tool.

L: The cutting's width increment in XY plane should smaller than tool radius and greater than 0.

W: Z axis direction cutting-in at first time, it is the distance down from the R based plane and should be greater than 0 (if cutting-in exceeds the bottom of slot at the first time, then process at bottom of slot)

Q: The cut depth of cutting feed in each time

V: The distance from machining face is greater than 0 when fast cutting

D: tool compensation number, data range is 0 ~ 256, D0 defaults to 0. Take out the current tool radius value according to specified serial number

K: Repeat times

Cycle process:

(1) Positioning to XY plane fastly

(2) Drop to R point plane fastly

(3) Down-cutting in cut speed to W distance depth

(4) Center every time according to the L value increasing outward spiral milling circular surface of the radius of the I.

(5) Z axis return to R base plane fastly

(6) XY axis positioning to circle fastly

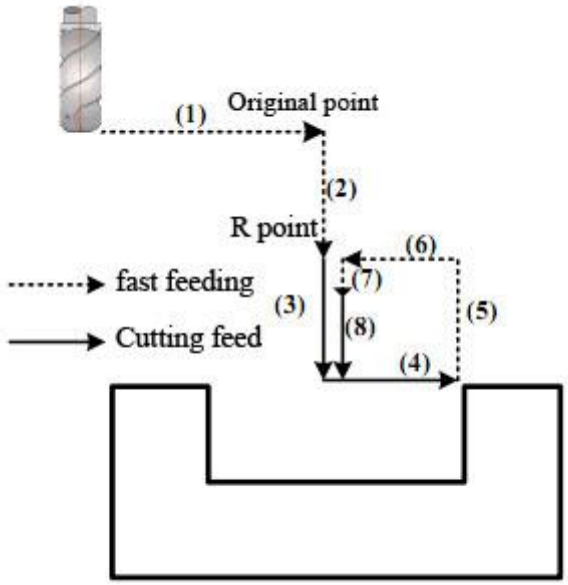
(7) The distance that Z axis rapid decline in un-processing plane V

(8) Down-cutting in cut speed to (Q+V) distance depth

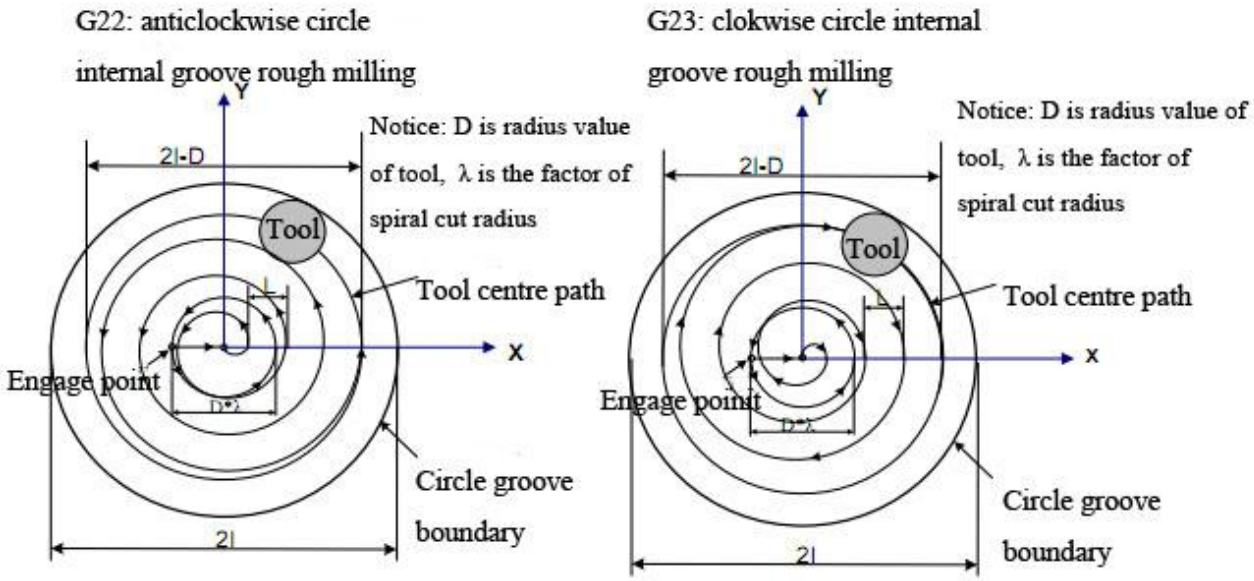
(9) The motion of cycle (4) ~ (8), until machining complete until finish processing cut deep circle face.

(10) Return to original point plane or R point plane according to the difference of G98 or G99

Instruction path:



Picture 3-4-1-1



Picture3-4-1-2

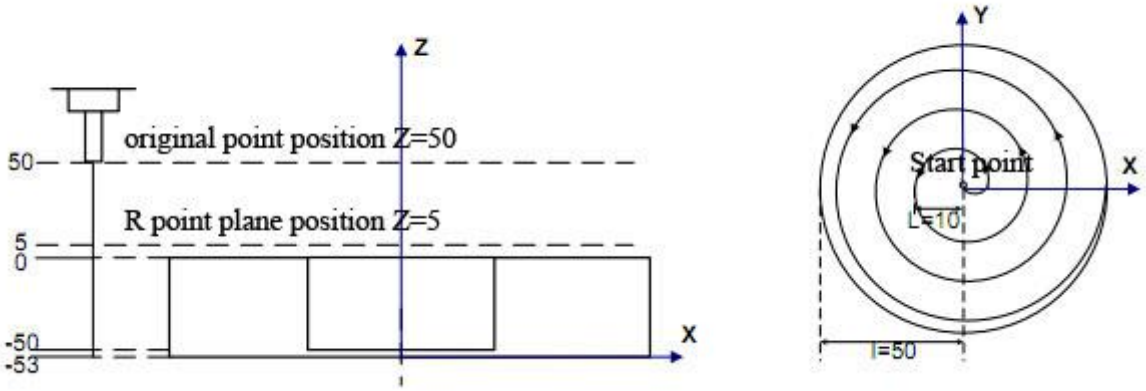
Attention:

1. NO：14#1 should be changed to 1 when use this instruction.

2. Spiral cutter radius should be greater than 0 when in groove cycle, cut down in the feeding speed of F15 when its speed of pograming is greater F15, cut down in the feeding speed of programing speed when is

smaller than F15. Spiral cut-down radius is specified by P333.

For example: Use a fixed cycle G22 to rough circle internal groove as below picture:



Picture 3-4-1-3

G90 G00 X50 Y50 Z50; (G00 fast positioning)

G99 G22 X25 Y25 Z-50 R5 I50 L10 W20 Q10 V10 D1 F800; (do circle internal groove rough milling cycle)

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

M30;

Cancel:

G code in group 01, modal G code (bit parameter NO: 52#2 set to be 1) of G60 and G22/G23 can't be specified in the same program segment, or G22/G23 will be canceled.

Tool offset:

Tool radius offset is omitted during fixed cycle positioning.

3.4.2 Complete circle internal groove milling cycle G24/G25

Instruction format:

G24

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

G25

Function: Tool finish milling a entire circle inside the circle at specified radius I, then turn back after milling.

Explanation:

G24: anticlockwise direction inside finish milling cycle inside entire circle;

G25: clockwise direction inside finish milling cycle inside entire circle;

X, Y: start position of X, Yplane;

Z: processing depth, G90 is absolute position, G91 is the position relative to R based plane;

R: R based plane, G90 is absolute position, and G91 is the position relative to start point of this program segment;

I: finish milling circle radius, value range is 0.0001 ~ 99999.9999mm, adopts it absolute value when it is negative value;

J: the distance from milling start point to circle centre, value range is 0.0001 ~ 99999.9999mm, adopts absolute value when it is negative value;

D: tool compensation number, value range is 0 ~ 256, D0 defaults is 0. Take out current tool's radiusvalue according to given sequence number;

K: repeat times.

Cycle processing:

(1) Positioning to XY plane rapidly;

(2) Fall to R point plane rapidly;

(3) Cutting feed to hole-bottom

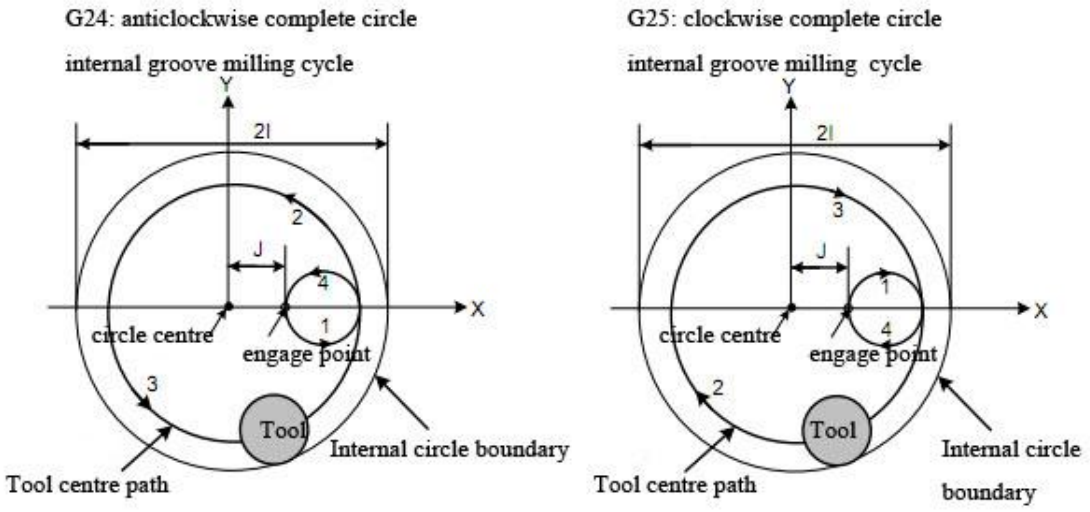
(4) The circle arc interpolation from start point and makes the transition arc 1 as path;

(5) The entire circle interpolation makes the finish milling internal circle as path;

(6) The circle arc interpolation returns to start point and make the transition arc 4 as path;

(7) Return to original point plane or R point plane according to the difference of specified G98 or G99.

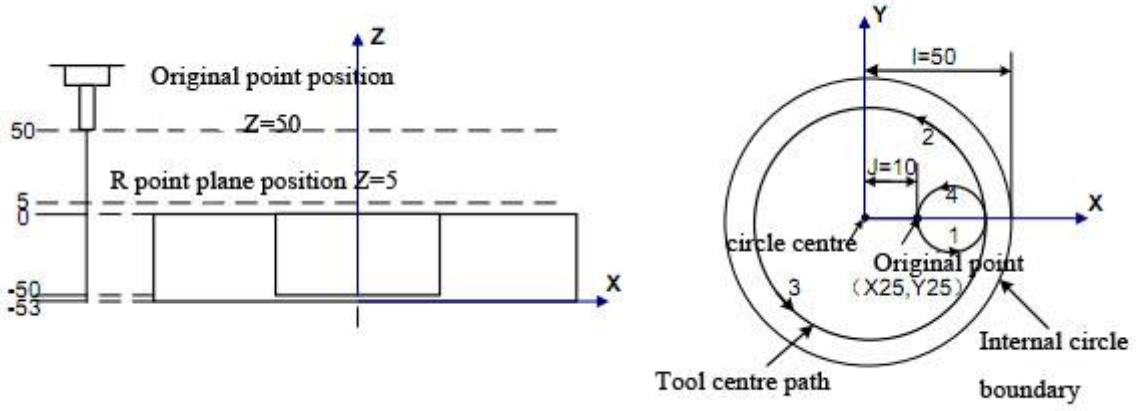
Instruction path:



Picture 3-4-2-1

Attention:NO：14#0 should be changed to 1 when use this instruction.

For example: finish milling the rough milling circle groove as below adopts G24 fixed cycle.



Picture 3-4-2-2

G90 G00 X50 Y50 Z50;(G00 fast positioning)

G99 G24 X25 Y25 Z-50 R5 I50 J10 D1F800; (Start fixed cycle, down to hole-botton and do circle internal finish-milling cycle)

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

M30;

Cancel:

G code (G00 TO G03) in group 01, G60 as modal G code (bit parameter NO: 52#2 set to be 1) and G24/G25 can't be specified in the same program segment, or G24/G25 will be canceled.

Tool offset:

Tool radius offset is omitted during fixed cycle positioning.

3.4.3 External circle groove milling cycle G26/G32

Instruction format:

G26

G98/G99 X_ Y_ Z_R_ I_ J_ D_ F_ K_

G32

Function:tool finish milling an entire circle external circle as the specified radius and direction, then go back.

Explanation:

G26: anticlockwise external circle rough groove milling cycle

G32: clockwise external groove rough milling cycle

X, Y: the start position of X, Y plane

Z: processing depth, G90 is absolute position, G91 relatives to R based plane.

R: R based plane, G90 is absolute position, G91 relatives to start point of this program segment.

I: finish-milling circle radius, value range is 0.0001 ~ 99999.9999mm, adopts absolute value when it is negative value.

J: the distance from finish-milling start point to round edge of finish milling circle, value range 0.0001 ~ 99999.9999mm, adopts it's absolute value when it is negative value.

D: tool compensation number, value range is 0 ~ 256, D0 default is0, adops the radius value of current tool according to specified sequence number;

K: repeat times

Cycle processing:

(1) Positioning to XY plane rapidly;

(2) Fall to R point plane rapidly;

(3) Cutting feed to hole-bottom

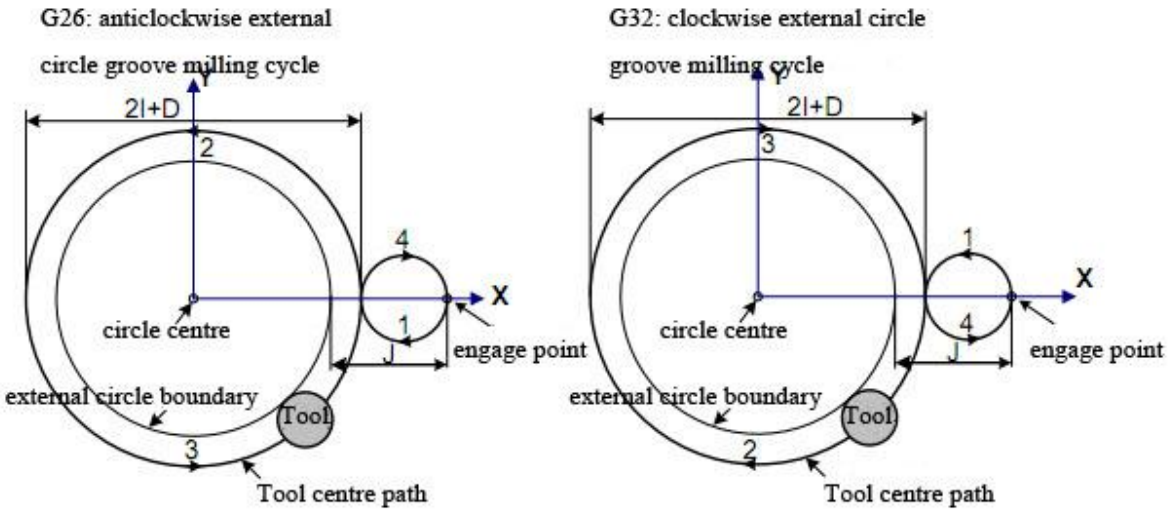
(4) The circle arc interpolation from start point and makes the transition arc 1 as path;

(5) The entire circle interpolation makes arc2 and arc3 as path;

(6) The circle arc interpolation returns to start point and make the transition arc 4 as path;

(7) Return to original point plane or R point plane according to the difference of specified G98 or G99.

Instruction path:

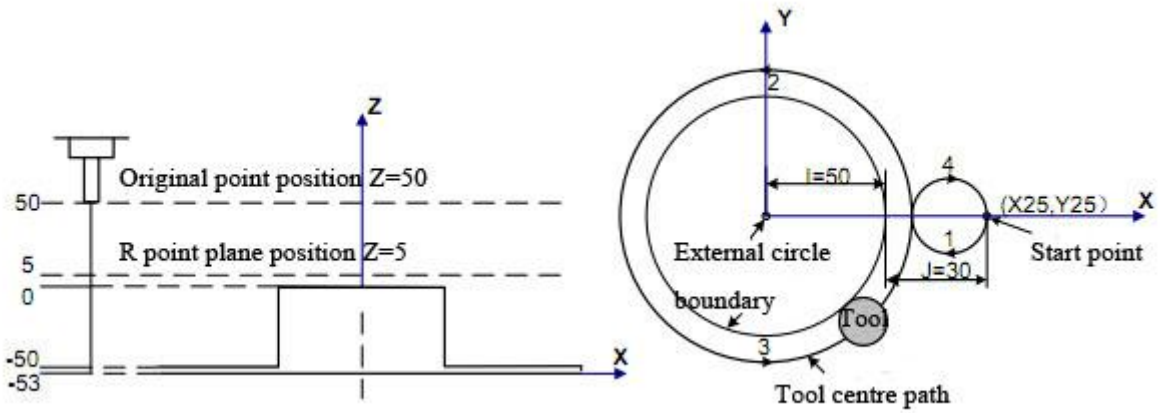


Picture 3-4-3-1

Instructions:

The interpolation of transition arc and finish-milling arc are different when external circle finish-milling, the interpolation direction of instruction explanation is finish-milling arc interpolation direction.

For example: finish milling the rough milling circle groove as below adopts G26 fixed cycle.



Picture 3-4-3-2

G90 G00 X50 Y50 Z50;(G00 fast positioning)

G99 G26 X25 Y25 Z-50 R5 I50 J30 D1 F800;(Start fixed cycle, down to hole-botton and do circleinternal finish-milling cycle)

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

M30;

Cancel:

G code (G00 TO G03) in group 01, G60 as modal G code (bit parameter NO: 52#2 set to be 1) and G26/G32 can't be specified in the same program segment, or G26/G32 will be canceled.

Tool offset:

Tool radius offset is omitted during fixed cycle positioning.

3.4.4 Rectangle groove rough milling G33/G34

Instruction format:

G33

G98/G99 X_ Y_ Z_ R_ I_ J_ L_ W_ Q_ V_ U_ D_ F_ K_

G34

Function: Start from rectangle center, make the specified parameter as the linear cutting cycle until process the

rectangle groove in programing size.

Explanation:

G33: anticlockwise internal rectangle rough groove milling cycle;

G32: clockwise rectangle rough groove milling cycle;

X, Y: the start position of X, Y plane;

Z: processing depth, G90 is absolute position, G91 relatives to R based plane.

R: R based plane, G90 is absolute position, G91 relatives to start point of this program segment.

I: the width of rectangle groove in X axis direction. I should be larger than tool radius. Spiral cut-down radius should be smaller than half of I.

J: the width of rectangle groove in Y axis direction. J should be larger than tool radius. Spiral cut-down radius should be smaller than half of J

L: the width increment of cutting should smaller than tool radius and larger than 0 in specified planes;

W: cut-in Z axis direction for the first time, it's the distance down from R based plane and should be greater than 0 (if the frist time cut-in exceeds groove bottom, then process at the groove bottom directly)

Q: the cutting depth of each feeding;

V: the distance from the unprocessing plane should be greater than 0 when cut rapidly;

U: Corner circle arc radius, it means corner circle arc transition when omitted. U should be larger or equal to tool radius;

D: tool compensation number, value range is 0 ~ 256, D0 default is 0, adops the radius value of current tool according to specified sequence number;

K: repeat times

Cycle processing:

(1) Positioning to spiral plunge cut start point of XY plane rapidly;

(2) Fall to R point plane rapidly;

(3) The value multiply radius compensation value by data parameter 269 is the W distance of diameter spiral plunge cut;

(4) Feeding to rectangle centre X0Y0;

(5) Outward from center finish milling rectangle plane according to L value progressive increase;

(6) Z axis return to R based plane;

(7) Positioning to spiral plunge cut start point of XY plane rapidly;

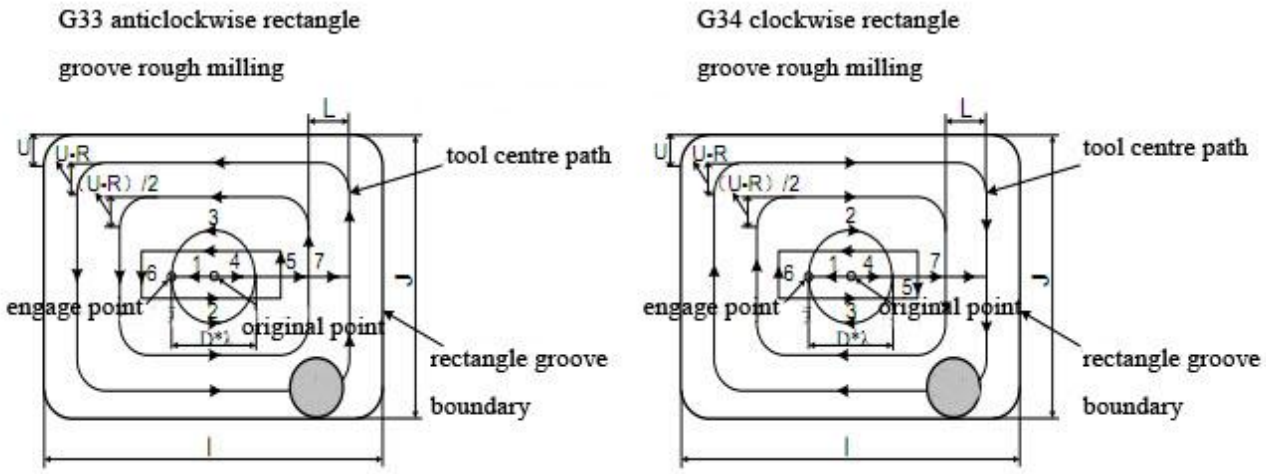
(8) The distance of Z axis down to no processing plane V;

(9) The cut-down (Q+V) depth of Z axis

(10) Repeat motion (4) ~ (9) until finish cutting depth rectangle plane;

(11) Return to original point plane or R point plane according to the difference of specified G98 or G99;

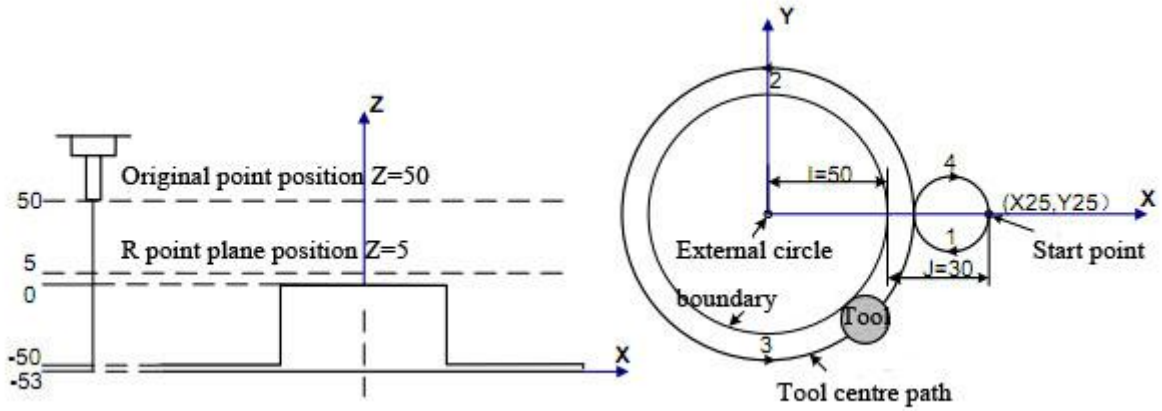
Insruction path:



Picture 3-4-4-1

Attention:NO：14#0 should be changed to 1 when use this instruction.

For example: finish milling the rough milling circle groove as below adopts G33 fixed cycle.



Picture 3-4-4-2

G90 G00 X50 Y50 Z50; (G00 fast positioning)

G99 G26 X25 Y25 Z-50 R5 I50 J30 D1 F800; (doing rectangle internal groove rough milling cycle)

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

M30;

Cancel:

G code (G00 TO G03) in group 01, G60 as modal G code (bit parameter NO: 52#2 set to be 1) and G33/G34 can't be specified in the same program segment, or G33/G34 will be canceled.

Tool offset:

Tool radius offset is omitted during fixed cycle positioning.

3.4.5 Rectangle groove internal milling cycle G35/G36

Instruction format:

G35

G98/G99 X_ Y_Z_ R_ I_ J_ L_ U_ D_ F_ K_

G36

Function:tool finish-milling inside of rectangle as the specified width and direction, then return.

Explanation:

G35: anticlockwise internal rectangle groove milling cycle;

G32: clockwise internal rectangle groove milling cycle;

X, Y: the start position of X, Y plane;

Z: processing depth, G90 is absolute position, G91 relatives to R based plane.

R: R based plane, G90 is absolute position, G91 relatives to start point of this program segment.

I: The width of X axis direction, value range of tool radius is~ 99999.9999mm;

J: The width of Y axis direction, value range of tool radius is~ 99999.9999mm.

L: The distance between finish-milling start point and rectangle edge X axis direction, value range of tool radius is ~99999.9999mm;

U: Corner circle arc radius, it means corner circle arc transition when omitted. It is alarm when 0<U<tool radius;

D: tool compensation number, value range is 0 ~ 256, D0 default is0, adops the radius value of current tool according to specified sequence number;

K: repeat times

Cycle processing:

(1) Positioning to XY plane rapidly;

(2) Fall to R point plane rapidly;

(3) Cutting feed to hole-bottom

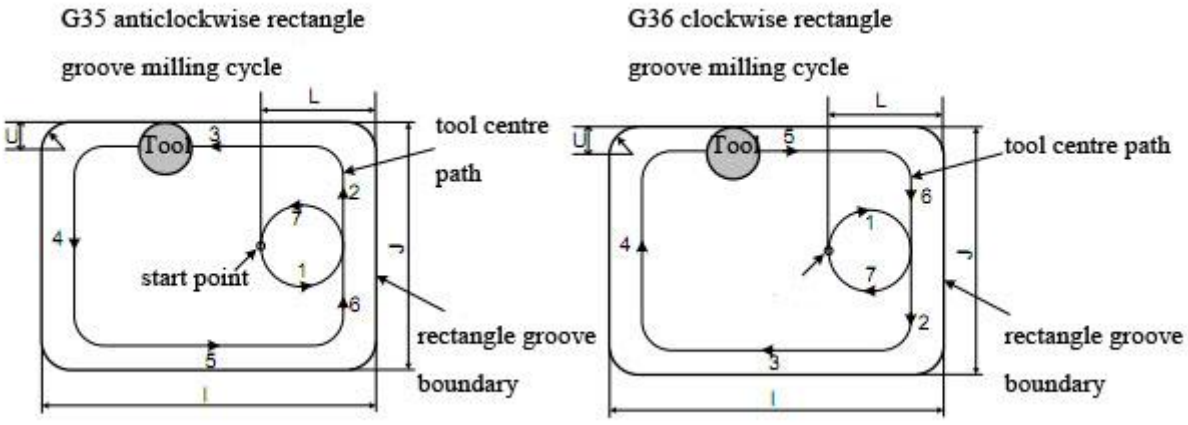
(4) The circle arc interpolation from start point and makes the transition arc1 as path;

(5) The linear and circle arc interpolation makes arc 2-3-4-5-6 as path;

(6) The circle arc interpolation returns to start point and make the transition arc 7 as path;

(7) Return to original point plane or R point plane according to the difference of specified G98 or G99.

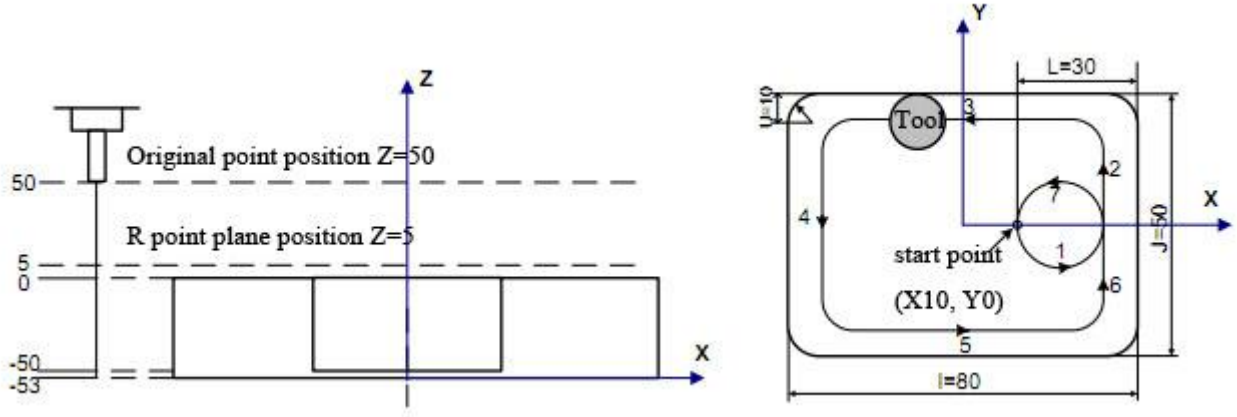
Instruction path:



Picture 3-4-5-1

Attention:NO: 14#0 should be changed to 1 when use this instruction.

For example: finish milling the rough milling circle groove as below adopts G35 fixed cycle.



Picture 3-4-5-2

G90 G00 X50 Y50 Z50; (G00 fast positioning)

G99 G26 X25 Y25 Z-50 R5 I50 J30 D1 F800; (doing rectangle internal groove milling cycle under hole-bottom)

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

M30;

Cancel:

G code (G00 TO G03) in group 01, G60 as modal G code (bit parameter NO: 52#2 set to be 1) and G35/G36 can't be specified in the same program segment, or G35/G36 will be canceled.

Tool offset:

Tool radius offset is omitted during fixed cycle positioning.

3.4.6 Rectangle external groove milling cycle G37/G38

Instruction format:

G37

G98/G99 X_ Y_Z_ R_ I_ J_ L_ U_ D_ F_ K_

G38

Function:tool finish-milling inside of rectangle as the specified width and direction, then return.

Explanation:

G37: anticlockwise internal rectangle groove milling cycle;

G38: clockwise internal rectangle groove milling cycle;。

X, Y: the start position of X, Y plane;

Z: processing depth, G90 is absolute position, G91 relatives to R based plane;

R: R based plane, G90 is absolute position, G91 relatives to start point of this program segment;

I: The width of X axis direction, value range of tool radius is0 ~ 99999.9999mm;

J: The width of Y axis direction, value range of tool radius is 0~ 99999.9999mm;

L: The distance between finish-milling start point and rectangle edge X axis direction, value range of tool radius is 0 ~99999.9999mm;

U: Corner circle arc radius, it means corner circle arc transition when omitted.

D: tool compensation number, value range is 0 ~ 256, D0 default is0, adops the radius value of current tool according to specified sequence number;

K: repeat times

Cycle processing:

(1) Positioning to XY plane rapidly;

(2) Fall to R point plane rapidly;

(3) Cutting feed to hole-bottom

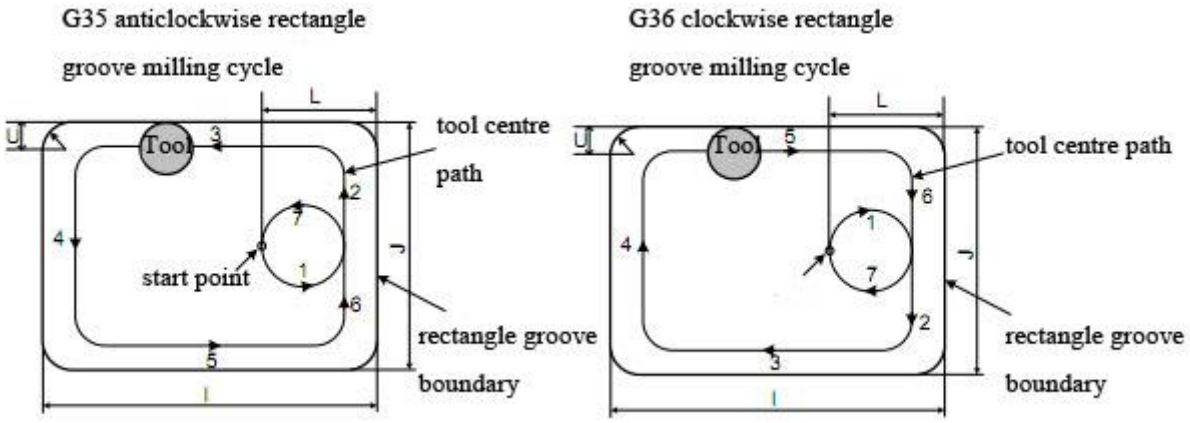
(4) The circle arc interpolation from start point and makes the transition arc1 as path;

(5) The linear and circle arc interpolation makes arc 2-3-4-5-6 as path;

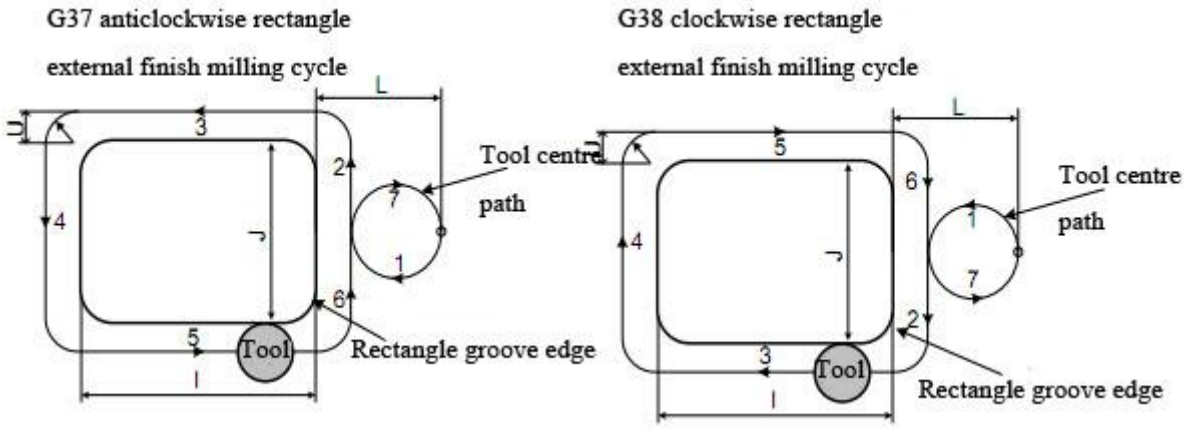
(6) The circle arc interpolation returns to start point and make the transition arc 7 as path;

(7) Return to original point plane or R point plane according to the difference of specified G98 or G99.

Instruction path:



Instruction path:



Picture 3-4-6-1

Relative explanation:

The direction of transition arc and finish milling arc are different when rectangle external finish milling, the interpolation direction of instruction explanation belongs to finish milling arc.

For example: do rectangle external finish milling by fixed cycle G37 instruction.

G90 G00 X50 Y50 Z50; (G00 positioning fastly)

G99 G37 X25 Y25 Z-50 R5 I80 J50 L30 U10 D1F800; (fixed cycle hole-bottm does rectangle external milling)

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

M30;

Cancel: can't specify G code in 01 group in the same program segment (G00 to G03), G60 as modal G code (bit parameter NO: 52#2 set to be 1) and G37/G38 will be canceled.

Tool offset: tool radius offset will be omitted during fixed cycle positioning.

3.4.7 High speed deep hole processing cycle G73

Instruction format:G73 X_Y_Z_R_Q_F_K_

Function:this cycle specially sets for execute high speed drilling setting, it executes interaval cutting feed until hole-bottom, retract rapidly from the hole at the same time with feeding and output chips. Action sketch as the picture 3-4-7-1 shows.

Explanation:

X_Y_: hole positioning data;

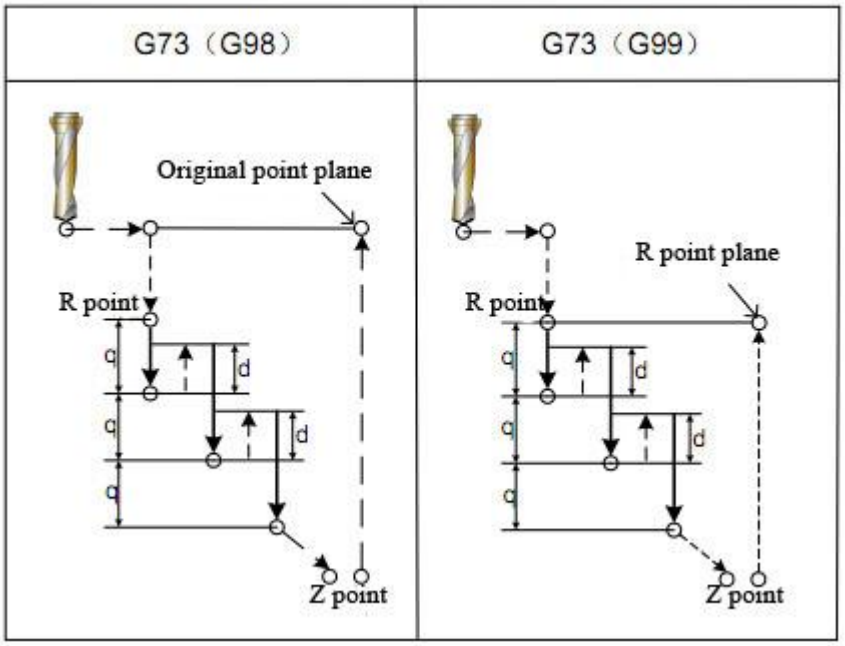
Z_: increment programming shows the distance from specified R point to hole-bottom; absolute programming means hold-bottom absolute coordinate;

R_: increment programming shows the distance from original point plane to R point; absolute programming shows the absolute coordinate of R point;

Q_: the cut depth of each cutting feed;

F_: cutting feed speed;

K_: repeat times.



Picture 3-4-7-1

Z, R: hole-bottom paramenter Z and parameter R misses on in random when execute the first drilling hole, system will only change the modal, not execute Z axis motion;

Q: it does the interval feeding as the above picture shows when specify instruction parameter Q. System will return by retract tool amount d (as picture 3-4-7-1) set by parameter P334, tool feed interval execute rapidly move return at the distance is d.

When G73 instruction and M instruction in the same program segment, the first hold positioning and M code execution are at the same time, then system processes the next drilling motion.

When specifies repeat time is K, it only executes M code of the first hole.

Notice:

1.System will give alarm reminding: address Q didn't be found or Q value is 0 (G73/G83) if didn't specify instruction parameter Q. System will do interval feeding by its absolute value if Q is negative.

2. In fixed cycle, when tool length offset (G43, G44 OR G49), added offset or cancel offset at the same time that positioning to R point.

Cancel: G code of 01 group, G60 is modal G code (bit parameter NO: 52#2 sets to be 1) and G73 can be specified in the same program segment, or G73 will be canceled.

Tool offset: tool radius offset will be omitted during fixed cycle positioning.

For example 1:

M3 S1500; spindle axis starts to rotate.

G90 G99 G73 X0 Y0 Z-15 R-10 Q5 F120; positioning, drill the frist hole and then return to R point.

Y-50; positioning, drill the 2nd holes and then return to R point.

Y-80;positioning, drillthe 3rd holes and then return to R point.

X10;positioning, drillthe 4th holes and then returns to R point.

Y10;positioning, drillthe 5th holes and then returns to R point.

G98 Y75;positioning, drill the 6th holes and then returns to original point plane.

G80;

G28 G91 X0 Y0 Z0;return to reference point

M5; spindle stop rotation

M30;

Notice: processing the 2nd~6th holes although omit Q, it also outputs chips.

3.4.8 Drilling cycle, spot drilling cycle G81

Instruction format:G81 X_ Y_ Z_ R_ F_ K_

Function:this cycle uses for normally drill cutting feed, and execute to hole-bottom, then tool goes back rapidly from hole-bottom.

Explanation:

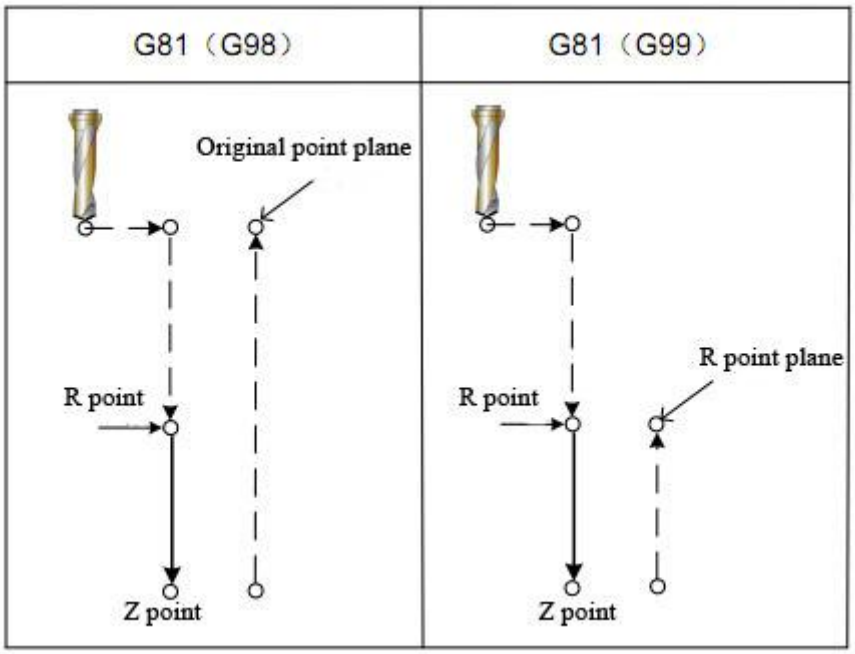
X_Y_: hole positioning data;

Z_: increment program shows the distance from R point to hole-bottom; absolute programming shows the absolute coordinate value of hole-bottom;

R_: increment programming shows the distance from original point plane to R point plane; absolute programming means the R point absolute coordinate value;

F_: cutting feed speed;

K_: repeat times (if it is necessary)



Picture 3-4-8-1

Z, R: hole-bottom paramenter Z and parameter R misses on in random when execute the first drilling hole, system will only change the modal, not execute Z axis motion; move to R point rapidly along with X, Y axis, execute drill processing from R point to Z point, then tool turns back rapidly;

Rotate spindle axis by auxiliary function M code before specify G81;

When G81 instruction and M instruction in the same program segment, the first hold positioning and M code execution are at the same time, then system handle the next drilling motion.

When specifies repeat time is K, it only executes M code to the first hole.

When specify tool length offset G43, G44 or G49 in fixed cycle, add or cancel offset at the same time when positioning to R point.

For example:

M3 S2000 spindle axis starts to rotate

G90 G99 G81 X300.Y-250.Z-150.R-10. F120. Positioning, drill the first holes and then return to R point.

Y-550; positioning, drill the 2nd holes and then return to R point.

Y-750; positioning, drill the 3rd holes and then return to R point.

X1000; positioning, drill the 4th holes and then returns to R point.

Y-550; positioning, drill the 5th holes and then returns to R point.

G98 Y-75; positioning, drill the 6th holes and then returns to original point plane.

G80;

G28 G91 X0 Y0 Z0; return to reference point

M5; spindle stop rotation

M30;

Notice: processing the 2nd~6th holes although omit Q, it also outputs chips.

Cancel: G code (G00 to G03) of 01 group, G60 is modal G code (bit parameter NO: 52#2 sets to be 1) and G81 can be specified in the same program segment, or G81 will be canceled.

Tool offset: tool radius offset will be omitted during fixed cycle positioning.

3.4.9 Drilling cycle, spot drilling cycle G82

Instruction format:G82 X_ Y_ Z_ R_ P_ F_ K_;

Function:this cycle uses for normally drill cutting feed, and execute to hole-bottom, then tool goes back rapidly from hole-bottom.

Explanation:

X_Y_: hole positioning data;

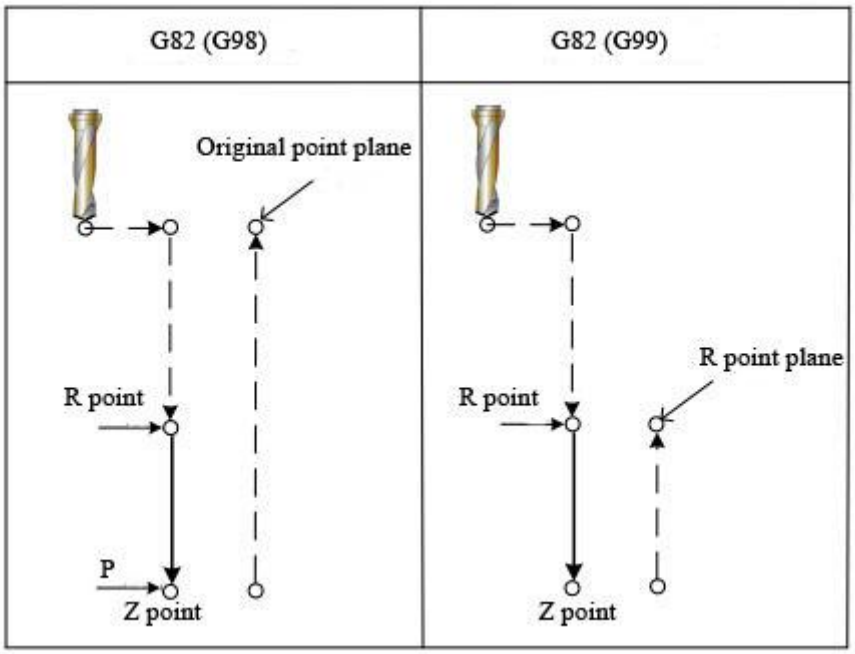
Z_: increment program shows the distance from R point to hole-bottom; absolute programming shows the absolute coordinate value of hole-bottom;

R_: increment programming shows the distance from original point plane to R point plane; absolute programming means the R point is absolute coordinate value;

F_: cutting feed speed;

P_: pause time;

K_: repeat times (if it is necessary)



Picture 3-4-9-1

Move to R point rapidly after positioning along with X, Y axis, execute drill processing from R point to Z point. Excute pause and then tool move rapidly when reach hole-bottom;

Rotate spindle axis by auxiliary function M code before specify G82;

When G82 instruction and M instruction in the same program segment, the first hold positioning and M code execution are at the same time, then system processes the next drilling motion;

It only execute M code when specify repeat times is K;

When specify tool length offset G43, G44 or G49 in fixed cycle, add or cancel offset at the same time when positioning to R point.

P is modal code instruction, parameter minimum value sets by parameter P336, parameter maximumvalue sets by P337. Run in the minimum value when P value is smaller than P336 set value; Run in the maxinum value when P value is greater than P337 set value. P can't be modal data to be stored if not execute drill processing

For example:

M3 S2000 spindle axis starts to rotate

G90 G99 G81 X300.Y-250.Z-150.R-10. F120. Positioning, drill the first holes and then return to R point.

Y-550;positioning, drill the 2nd holes and pause 1 second, then return to R point.

Y-750; positioning, drill the 3rd holes and pause 1 second, then return to R point.

X1000; positioning, drill the 4th holes and pause 1 second, then returns to R point.

Y-550; positioning, drill the 5th holes and pause 1 second, then returns to R point.

G98 Y-75; positioning, drill the 6th holes and pause 1 second, then returns to original point plane.

G80; cancel fixed cycle.

G28 G91 X0 Y0 Z0;return to reference point.

M5; spindle stop rotation

M30;

Cancel: G code (G00 to G03) of 01 group, G60 is modal G code (bit parameter NO: 52#2 sets to be 1) and G82 can be specified in the same program segment, or G82 will be canceled.

Tool offset: tool radius offset will be omitted during fixed cycle positioning.

3.4.10 Chip removal and drilling cycle G83

Instruction format:G83 X_ Y_ Z_ R_ Q_ F_ K_

Function: the cycle uses for depth drilling. Excute interval cutting feed to hole bottom, remove chips from hole during drilling.

Explanation:

X_Y_: hole positioning data;

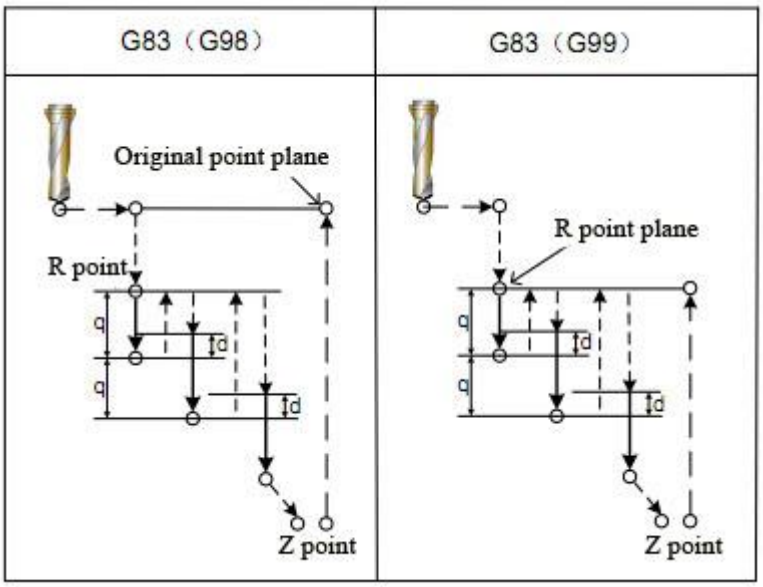
Z_: increment program shows the distance from R point to hole-bottom; absolute programming shows the absolute coordinate value of hole-bottom;

R_: increment programming shows the distance from original point plane to R point plane; absolute programming means the R point absolute coordinate value;

F_: cutting feed speed;

P_: pause time;

K_: repeat times (if it is necessary)



Picture 3-4-10-1

Q: It means the depth of each cutting feed; it must be incremental value. In the second or the following cutting feed, execute fast moving to the point which distance is d from the drilling hole of last time, and execute cutting feed again, the value of d set by parameter P335. As picture 3-4-10-1 shows.

It has to specify positive value in Q, negative sign is omitted, system continues to process in positive value.

Specify Q in execute drilling hole program segment, or Q can't be stored as modal data.

Rotate spindle axis by auxiliary function M code before specify G83;

When G83 instruction and M instruction in the same program segment, the first hold positioning execute M code at the same time, then system handle the next drilling motion.

When specifies repeat time is K, it only executes M code to the first hole.

When specify tool length offset G43, G44 or G49 in fixed cycle, add or cancel offset at the same time when positioning to R point.

For example:

M3 S2000; spindle axis starts to rotate

G90 G99 G83 X300 -250 -150 -100 Q15 F120;Positioning, drill the first holes and then return to R point.

Y-550; positioning, drill the 2nd holes and then return to R point.

Y-750; positioning, drill the 3rd holes and then return to R point.

X1000; positioning, drill the 4th holes and then returns to R point.

Y-550; positioning, drill the 5th holes and then returns to R point.

G98 Y-750; positioning, drill the 6th holes and then returns to original point plane.

G80;

G28 G91 X0 Y0 Z0;return to reference point

M5; spindle stop rotation

M30;

Cancel: G code (G00 to G03) of 01 group, G60 is modal G code (bit parameter NO: 52#2 sets to be 1) and G81 can be specified in the same program segment, or G81 will be canceled.

Tool offset: tool radius offset will be omitted during fixed cycle positioning.

3.4.11 Right-handed tapping cycle G84

Instruction format:G84 X_ Y_ Z_ R_ P_ F_

Function: the cycle executes tapping. In this tapping cycle, spindle change rotating direction when reaches hole-bottom.

Explanation:

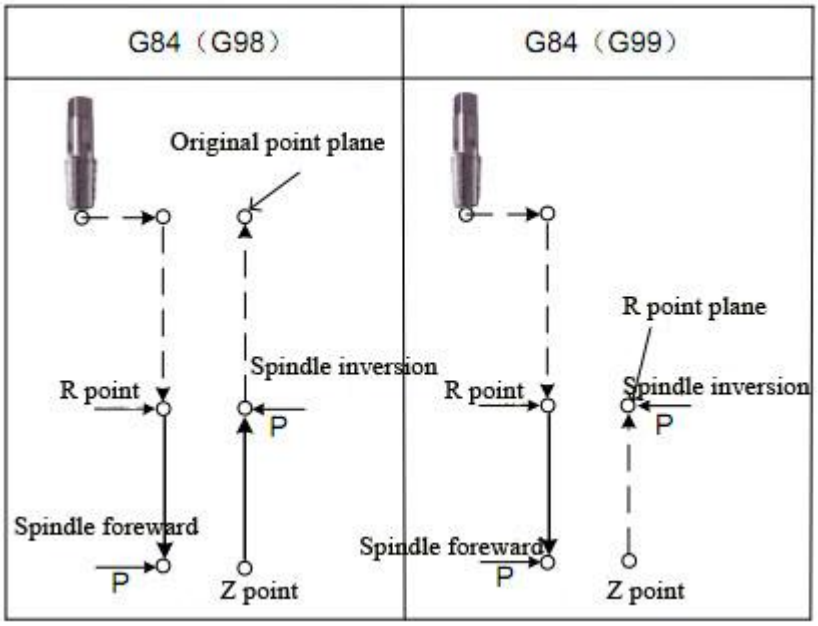
X_Y_: hole positioning data;

Z_: increment program shows the distance from R point to hole-bottom; absolute programming shows the absolute coordinate value of hole-bottom;

R_: increment programming shows the distance from original point plane to R point plane; absolute programming means the R point absolute coordinate value;

P_: pause time;

F_: cutting feed speed;



Picture 3-4-11-1

Spindle executes tapping in clockwise direction, and rotates in opposite direction in order to return spindle axis at hole-bottom and generates thread.

Feeding rate is omitted during tapping. Machine tool wouldn't stop when speeding paused until finish returning motions.

Assistant function M code makes the spindle rotate before specify G84. Spindle axis rotates clockwise if there are no commands, system adjust for clockwise rotation according to current spindle instruction rotation in R plane.

When G84 and M instruction in the same program segment, execute M code at the same time with the first hole-positioning, then system processes the next drilling motion.

Only execute M code to the first hole when specify repeat times K.

P is modal instruction, minimum value is set by data parameter P336, maximum value is set by parameter P337. It runs in the minimum value when P is smaller than P336, and runs in the max value when the set P is larger than P337. If specify in the program segment which not execute hole-drilling processing, P can't be stored as modal data.

When specify tool length offset G43, G44 or G49 in fixed cycle, add or cancel offset at the same time when positioning to R point.

In each feeding mode, the relationships of screw lead, feeding speed and spindle speed:

Feeding speed F = tapping pitch * spindle speed S

For examples: tapping M12*15 screw hole in workpiece, parameter as below:

S500 = 500 r/min; F=1.5×500=750mm/min;

When it is multi-head screw, multiply by head number can be F value.

For example:

M3 S100; spindle axis starts to rotate

G90 G99 G84 X300 -250 Z-150 -120 P300 F120;Positioning, drill the first holes and then return to R point.

Y-550; positioning, drill the 2nd holes and then return to R point.

Y-750; positioning, drill the 3rd holes and then return to R point.

X1000; positioning, drill the 4th holes and then returns to R point.

Y-550; positioning, drill the 5th holes and then returns to R point.

G98 Y-750; positioning, drill the 6th holes and then returns to original point plane.

G80;

G28 G91 X0 Y0 Z0; return to reference point

M5; spindle stop rotation

M30;

Cancel: G code (G00 to G03) of 01 group, G60 is modal G code (bit parameter NO: 52#2 sets to be 1) and G84 can be specified in the same program segment, or G84 will be canceled.

Tool offset: tool radius offset will be omitted during fixed cycle positioning.

3.4.12 Left-handed tapping cycle G74

Instruction format:G74 X_ Y_ Z_ R_ P_ F_

Function:the cycle executes tapping. In this tapping cycle, spindle inversion when reaches hole-bottom.

Explanation:

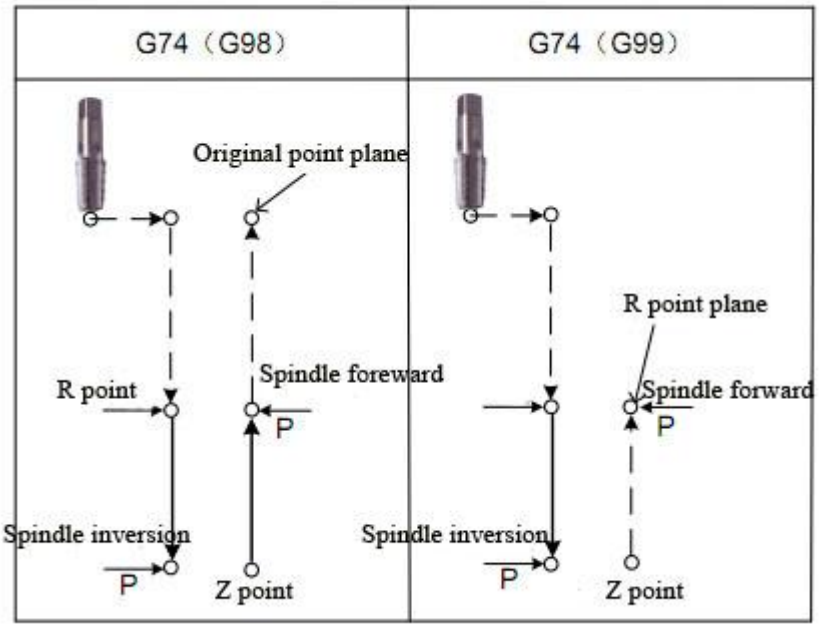
X_Y_: hole positioning data;

Z_: increment program shows the distance from R point to hole-bottom; absolute programming shows the absolute coordinate value of hole-bottom;

R_: increment programming shows the distance from original point plane to R point plane; absolute programming means the R point absolute coordinate value;

P_: pause time;

F_: cutting feed speed;



Picture 3-4-12-1

Spindle executes tapping in anticlockwise direction, and rotates in opposite direction in order to return spindle axis at hole-bottom and generates thread during it.

Feeding rate is omitted during tapping. Machine tool wouldn't stop when speeding paused until finish returning motions.

Assistant function M code makes the spindle rotate before specify G74. Spindle axis rotates anticlockwise if there are no commands, system adjust for clockwise rotation according to current spindle instruction rotation in R plane.

When G84 and M instruction in the same program segment, execute M code at the same time with the first hole-positioning, then system processes the next drilling motion.

Only execute M code to the first hole when specify repeat times K.

P is modal instruction, minimum value is set by data parameter P336, maximum value is set by parameter P337. It runs in the minimum value when P is smaller than P336, and runs in the max value when the set P is greater than P337. If specify in the program segment which not execute hole-drilling processing, P can't be stored as modal data.

When specify tool length offset G43, G44 or G49 in fixed cycle, add or cancel offset at the same time when positioning to R point.

In each feeding mode, the relationships of screw lead, feeding speed and spindle speed:

Feeding speed F = tapping pitch * spindle speed S

For examples: tapping M12*15 screw hole in workpiece, parameter as below:

S500 = 500 r/min; F=1.5×500=750mm/min;

When it is multi-head screw, multiply by head quantity can be F value.

For example:

M4 S100; spindle axis starts to rotate

G90 G99 G74 X300 Y-250 Z-150 R-120 P300 F120; Positioning, drill the first holes and then return to R point.

Y-550; positioning, drill the 2nd holes and then return to R point.

Y-750;positioning, drill the 3rd holes and then return to R point.

X1000; positioning, drill the 4th holes and then returns to R point.

Y-550;positioning, drill the 5th holes and then returns to R point.

G98 Y-750; positioning, drill the 6th holes and then returns to original point plane.

G80;

G28 G91 X0 Y0 Z0; return to reference point

M5; spindle stop rotation

M30;

Cancel: G code (G00 to G03) of 01 group, G60 is modal G code (bit parameter NO: 52#2 sets to be 1) and G74 can be specified in the same program segment, or G74 will be canceled.

Tool offset: tool radius offset will be omitted during fixed cycle positioning.

3.4.13 Precision boring cycle G76

Instruction format: G76 X_Y_Z_Q_R_P_F_K_

Function:

Precision boring cycle is suit for hole;

Spindle stop rotating when reaches hole-bottom, cutting tool leaves workpiece and be processd the surface, then turn back;

Aviod the trace when knife back that affect the surface smooth finish and tool damage.

Explanation:

X_Y_: hole positioning data;

Z_: increment program shows the distance from R point to hole-bottom; absolute programming shows the absolute coordinate value of hole-bottom;

R_: increment programming shows the distance from original point plane to R point plane; absolute programming

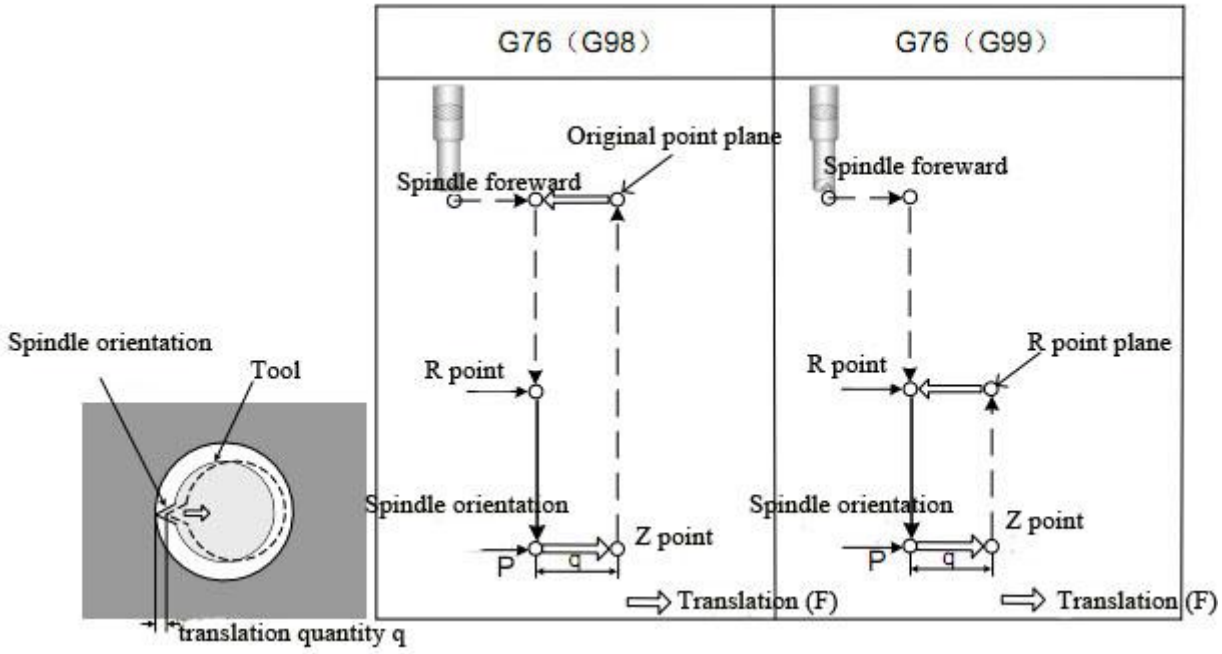
means the R point absolute coordinate value;

Q_: offset value of hole-bottom;

P_: pause time;

F_: cutting feed speed;

K_: times of precision boring.



Picture 3-4-13-1

When tool reaches hole-bottom, spindle stop in the fixed rotation position, tool moves retract in the opposite direction of tool nose. It protects the processing plane and achieves precision boring processing. Parameter Q specified the retract distance. Specify retract direction by bit parameter NO: 44#4 and NO: 44#5, Q must be positive value. The symbol wouldn't work even it is negative value. The offset value of Q in hole-bottom is the modal value which stores in fixed cycle, and it must be specified carelly, because it is also the cutting depth of G73 and G83.

Rotate spindle by assist function before specify G76.

When G76 and M instruction in the same program segment, execute M code at the same time with the first hole-positioning, then system processes the next drilling motion.

Only execute M code to the first hole when specify repeat times K.

When specify tool length offset G43, G44 or G49 in fixed cycle, add or cancel offset at the same time when positioning to R point.

Axis switch: fixed cycle must be canceled before change drilling axis.

Boring processing: it wouldn't execute boring processing in the program segment which don't include X, Y, Z and R axis or other axis.

For example:

M3 S500; spindle axis starts to rotate

G90 G99 G76 X300 Y-250; Positioning, drill the first holes and then return to R point.

Z-150 R-100 Q5; orientation in hole-bottom and then move 5mm;

P1000 F120; stop 1 second in hole-bottom;

Y-550; positioning, drill the 2nd holes and then return to R point.

Y-750;positioning, drill the 3rd holes and then return to R point.

X1000; positioning, drill the 4th holes and then returns to R point.

Y-550;positioning, drill the 5th holes and then returns to R point.

G98 Y-750; positioning, drill the 6th holes and then returns to original point plane.

G80G28 G91 X0 Y0 Z0; return to reference point

M5; spindle stop rotation

Cancel: G code (G00 to G03) of 01 group, G60 is modal G code (bit parameter NO: 52#2 sets to be 1) and G76 can be specified in the same program segment, or G76 will be canceled.

Tool offset: tool radius offset will be omitted during fixed cycle positioning.

3.4.14 Boring cycle G85

Instruction format:G85 X_Y_Z_R_F_K_

Function:the cycle uses for hole boring

Explanation:

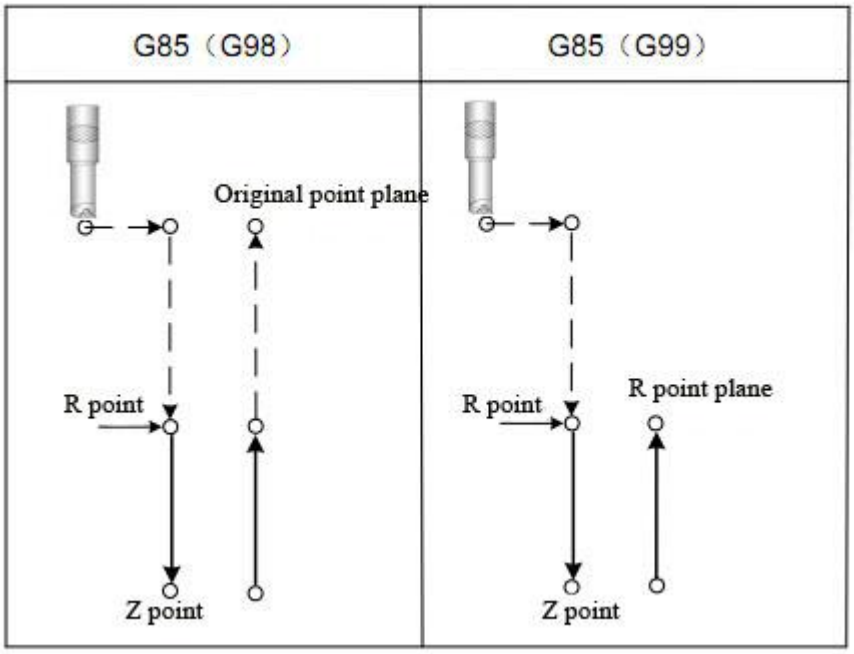
X_Y_: hole positioning data;

Z_: increment program shows the distance from R point to hole-bottom; absolute programming shows the absolute coordinate value of hole-bottom;

R_: increment programming shows the distance from original point plane to R point plane; absolute programming means the R point absolute coordinate value;

F_: cutting feed speed;

K_: times of precision boring.



Picture 3-4-14-1

Move to R point rapidly after along X, Y axis positioning, then execute hole-boring from R point to Z point, execute cutting feed when reaches hole-bottom and then return to R point.

Rotate spindle by assist function before specify G85.

When G85 and M instruction in the same program segment, execute M code at the same time with the first hole-positioning, then system processes the next drilling motion.

Only execute M code to the first hole when specify repeat times K.

When specify tool length offset G43, G44 or G49 in fixed cycle, add or cancel offset at the same time when positioning to R point.

Axis switch: fixed cycle must be canceled before change drilling axis.

Boring processing: it wouldn't execute boring processing in the program segment which don't include X, Y, Z and R axis or other axis.

For example:

M3 S100; spindle axis starts to rotate

G90 G99 G85 X300 Y-250 Z-150 R-120 F120; Positioning, drill the first holes and then return to R point.

Y-550; positioning, drill the 2nd holes and then return to R point.

Y-750;positioning, drill the 3rd holes and then return to R point.

X1000; positioning, drill the 4th holes and then returns to R point.

Y-550;positioning, drill the 5th holes and then returns to R point.

G98 Y-750; positioning, drill the 6th holes and then returns to original point plane.

G80;

G28 G91 X0 Y0 Z0; return to reference point

M5; spindle stop rotation

M30;

Cancel: G code (G00 to G03) of 01 group, G60 is modal G code (bit parameter NO: 52#2 sets to be 1) and G85 can be specified in the same program segment, or G85 will be canceled.

Tool offset: tool radius offset will be omitted during fixed cycle positioning.

3.4.15 Hole boring cycle G86

Instruction format:G86 X_Y_Z_R_F_K_

Function:the cycle uses for hole boring processing cycle

Explanation:

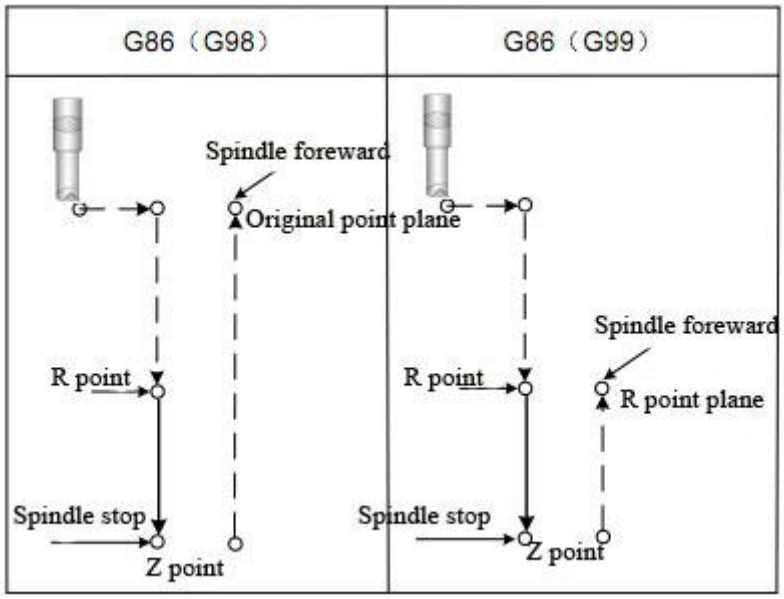
X_Y_: hole positioning data;

Z_: increment program shows the distance from R point to hole-bottom; absolute programming shows the absolute coordinate value of hole-bottom;

R_: increment programming shows the distance from original point plane to R point plane; absolute programming means the R point absolute coordinate value;

F_: cutting feed speed;

K_: times of precision boring.



Picture 3-4-15-1

Move to R point rapidly after along X, Y axis positioning, then execute hole-boring from R point to Z point, execute cutting feed when reaches hole-bottom and then return.

Rotate spindle by assist function before specify G86.

When G86 and M instruction in the same program segment, execute M code at the same time with the first hole-positioning, then system processes the next drilling motion.

Only execute M code to the first hole when specify repeat times K.

When specify tool length offset G43, G44 or G49 in fixed cycle, add or cancel offset at the same time when positioning to R point.

Axis switch: fixed cycle must be canceled before change drilling axis.

Boring processing: it wouldn't execute boring processing in the program segment which don't include X, Y, Z and R axis or other axis.

For example:

M3 S2000;spindle axis starts to rotate

G90 G99 G86 X300 Y-250 Z-150 R-100 F120;Positioning, drill the first holes and then returnto R point.

Y-550; positioning, drill the 2nd holes and then return to R point.

Y-750;positioning, drill the 3rd holes and then return to R point.

X1000; positioning, drill the 4th holes and then returns to R point.

Y-550;positioning, drill the 5th holes and then returns to R point.

G98 Y-750; positioning, drill the 6th holes and then returns to original point plane.

G80;

G28 G91 X0 Y0 Z0; return to reference point

M5; spindle stop rotation

M30;

Cancel: G code (G00 to G03) of 01 group, G60 is modal G code (bit parameter NO: 52#2 sets to be 1) and G86 can be specified in the same program segment, or G86 will be canceled.

Tool offset: tool radius offset will be omitted during fixed cycle positioning.

3.4.16 Hole boring cycle, back hole boring cycle G87

Instruction format:G87 X_Y_Z_R_F_K_

Function:the cycle uses for precision hole-boring

Explanation:

X_Y_: hole positioning data;

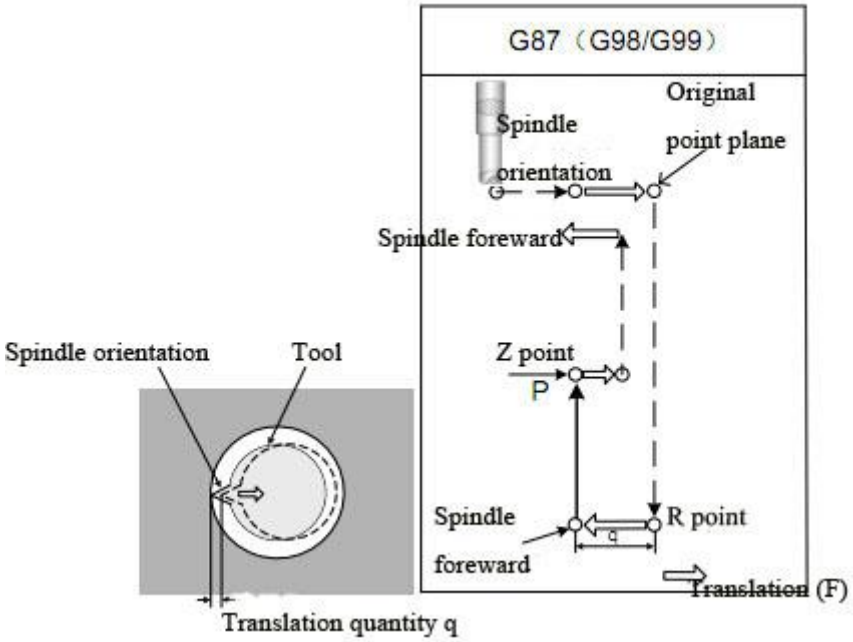
Z_: increment program shows the distance from R point to hole-bottom; absolute programming shows the absolute coordinate value of hole-bottom;

R_: increment programming shows the distance from original point plane to R point plane; absolute programming means the R point absolute coordinate value;

Q_: offset value of hole-bottom;

F_: cutting feed speed;

K_: feeding speed of precision boring.



Picture 3-4-16-1

When tool reaches hole-bottom, spindle stop in the fixed rotation position, tool moves retract in the opposite direction of tool nose. It protects the processing plane and achieves precision boring processing. Parameter Q specified the retract distance. Specify retract direction by bit parameter NO: 44#4 and NO: 44#5, Q must be positive value. The symbol wouldn't work even it is negative value. The offset value of Q in hole-bottom is the modal value which stores in fixed cycle, and it must be specified carelly, because it is also the cutting depth of G73 and G83.

Along X, Y axis positioning, stop tool after spindle axis orientation, and moves in the opposite direction of tool nose. And moves in feeding speed at hole-bottom R point, then tool moves in tool nose direction and spindle rotation foreward.Hole boring foreward until Z point, after Z point spindle axis orientation again, spindle axis stop on the rotation position and tool retracts at the opposite direction of tool nose, then tool returns to original plane. Tool offsets spindle rotates foreward in tool nose direction and execute next program segment.

Parameter Q specifies the distance of tool retract. Specify retract direction by bit parameter NO: 44#4 and NO: 44#5, Q must be positive value. Although use a negative number, symbol doesn't work.Q in the bottom of the hole deviation is saved in a fixed cycle modal value must be careful, because it is used for G73 and G83 cutting depth.

Use secondary function M code rotates the spindle before pointed G87.

G87 command and M command the same program period specified, in the first hole positioning movement of M code execution at the same time. Then, the system is processing the next borehole.

When pointed repeat times K, only execute M code to the first hole, others no need to execute M code.

When in a fixed loop, specify length bias G43 prop, G44, or G49, while positioning to the "R" plus offset values or cancelled.

Axis switch must cancel fixed loop before switching boring axis.

Bore process: not including X, Y, Z, R or other auxiliary axis programs, not execute bore process

Tips:

When in bore cycle programming, Z and R value should be pointed, usually, Z position will in the up of R position. Or system will alarm.

For example:

M3 S500 Spindle start rotate

G90 G99 G87 X300Y-250 Z-120 R-150 Q5 P1000 F120;

(Positioning, Bore 1 hole, in the initial position orientation and then offset by 5 mm in Z stop 1 second)

Y-550; Positioning, Bore 2 hole, then back to R

Y-750; Positioning, Bore 3 hole, then back to R

X1000; Positioning, Bore 4 hole, then back to R

Y-550; Positioning, Bore 5 hole, then back to R

G98 Y-750; Positioning, Bore 6 hole, then back to R

G80 G28 G91 X0 Y0 Z0; Back to reference point

M5; Spindle axis stop rotate

Cancel: cannot point 01 group G code (G00 to G003) in the same program segment; G60 is modal G code, Position parameter NO: 52#2 set to 1) and G87, or G87 will be cancelled.

Tool Offset: In the process of fixed loop positioning tool radius offset is ignored

3.4.17 Boring Loop G88

Command format: G88 X_Y_Z_R_P_F_

Function: This loop use for boring

Description:

X_Y_: Hole Location position;

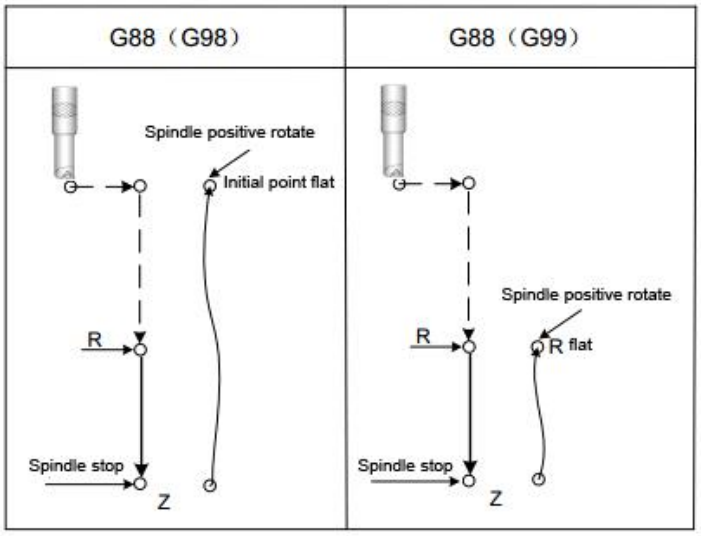
Z_: Incremental programming pointed the distance from R to the bottom of the hole; absolute program indicate the

hole bottom's absolute coordinates.

R_: Incremental programming indicates the distance from initial point to R; absolute program indicate R absolute coordinates.

P_: Pause time;

F_: Cutting feed rate.



Picture 3-4-17-1

Along X and Y axis positioning, move fast to R point, then execute boring hole from R to Z point, after boring hole finish, execute paused, then spindle stop. Tool from the bottom of Z point manual back to R point (G99) or initial point (G98) finished, then start spindle positive rotate.

Before pointed G88 code, please use M code to rotate spindle.

G88 command and M command in the same program segment, in the first hole positioning operation at the same time execute M code, then, system process the next hole boring.

When specified repeat times K, only execute M code for the first hole, and not execute M code to other holes.

P is modal command, min. value is setting by data parameter P336, max value is setting by data parameter P337. When P less than P336 parameter setting value, then work as min value, if greater than P337 parameter setting value, then work as max value. If pointed in the unexecuted boring process program, P cannot save as modal data.

In the fixed loop, specified tool length offset G43 G44 or G49, when positioning R point, add offset or cancel offset.

Axis Switch: Cancel fixed loop before switch boring axis.

Bore process: not including X, Y, Z, R or other auxiliary axis programs, not execute bore process

Example:

M3 S2000 Spindle start rotate

G90 G99 G88 X300 Y-250 Z-150 R-100 P1000 F120; Positioning, bore 1 hole, then back to R.

Y-550; Positioning, bore 2 hole, then back to R.

Y-750; Positioning, bore 3 hole, then back to R.

X1000; Positioning, bore 4 hole, then back to R

G98 Y-750; Positioning, bore 5 hole, then back to R.

G80 G28 G91 X0 Y0 Z0; Back to reference point.

M5; Spindle stop rotate.

Cancel: cannot point 01 group G code (G00 to G003) in the same program segment; G60 is modal G code, Position parameter NO: 52#2 set to 1) and G88, or G88 will be cancelled.

Tool Offset: In the process of fixed loop positioning tool radius offset is ignored

3.4.18 Hole Loop G89

Command format: G88 X_Y_Z_R_P_F_

Function: This loop use for boring

Description:

X_Y_: Hole Location Data;

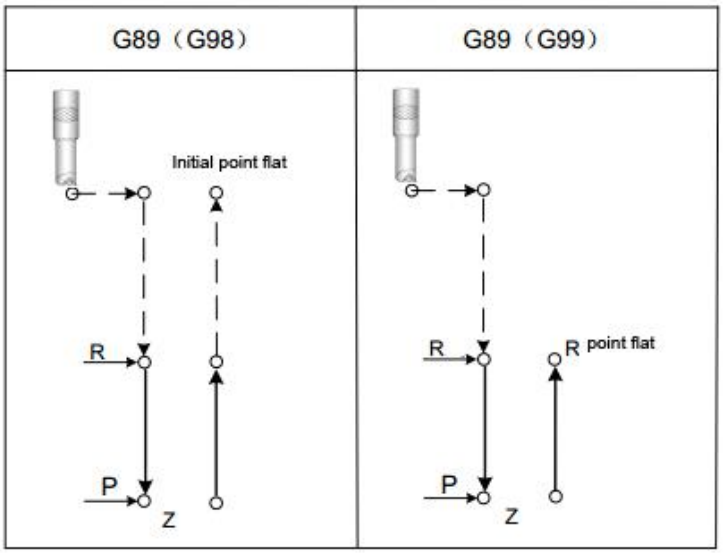
Z_:Incremental programming pointed the distance from R to the bottom of the hole; absolute program indicate the hole bottom's absolute coordinates.

R_: Incremental programming indicates the distance from initial point flat to R; absolute program indicate R absolute coordinates.

P_: Pause time;

F_: Cutting feed rate.

K_: Repeat count



Picture 3-4-18-1

This loop is almost the same as G85, the difference is the loop will paused in the bottom of hole.

Before pointed G89 code, use secondary function M code rotate spindle.

G89 command and M command in the same program segment, in the first hole positioning operation at the same time execute M code, then, system process the next boring hole action.

When pointed repeat times K, only execute M code for the first hole, and not execute M code to other holes.

P is modal command, min. value is setting by data parameter P336, max value is setting by data parameter P337. When P less than P336 parameter setting value, then work as min value, if greater than P337 parameter setting value, then work as max value. If pointed in the unexecuted boring process program, P cannot save as modal data.

In the fixed loop, specified tool length offset G43 G44 or G49, when positioning R point, add offset or cancel offset.

Axis Switch: Cancel fixed loop before switch boring hole axis.

Bore process: not including X, Y, Z, R or other auxiliary axis programs, not execute bore process

Example:

M3 S100 Spindle start rotate

G90 G99 G89 X300 Y-250 Z-150 R-100 P1000 F120;

Positioning, bore 1 hole, then back to R, stop 1 seconds in the bottom of hole

Y-550; Positioning, bore 2 hole, then back to R.

Y-750; Positioning, bore 3 hole, then back to R.

X1000; Positioning, bore 4 hole, then back to R

Y-550; Positioning, bore 5 hole, then back to R.

G98 Y-750; Positioning, bore 6 hole, then back to initial flat

G80 G28 G91 X0 Y0 Z0; Back to reference point.

M5; Spindle stop rotate.

M30;

Cancel: cannot point 01 group G code (G00 to G003) in the same program segment; G60 is modal G code, Position parameter NO: 52#2 set to 1) and G88, or G88 will be cancelled.

Tool Offset: In the process of fixed loop positioning tool radius offset is ignored

3.4.19 Left-handed rigid tapping G74

Command Format: G74 X_Y_R_P_F_K_

Function: Spindle motor in the rigid way of work is a servo motor, the directive can be realized in the left-hand tell high-precision tapping

Description:

X_Y_: Hole Location Data;

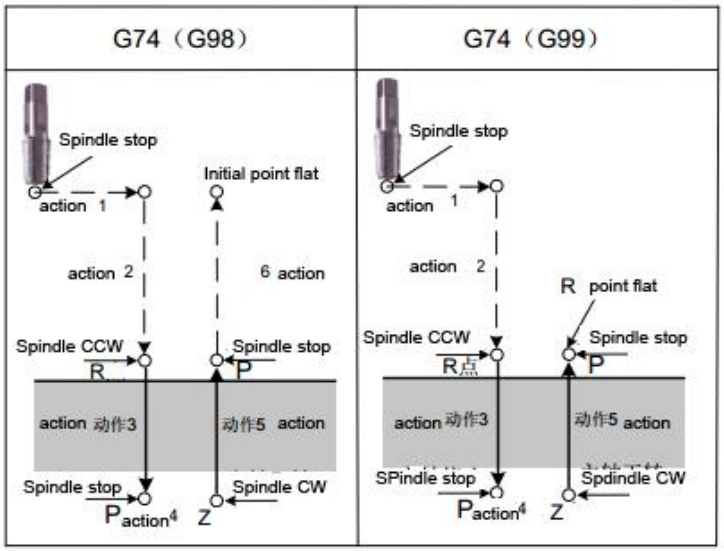
Z_:Incremental programming pointed the distance from R to the bottom of the hole; absolute program indicate the hole bottom's absolute coordinates.

R_: Incremental programming indicates the distance from initial point flat to R; absolute program indicate R absolute coordinates.

P_: Pause time;

F_: Cutting feed rate.

K_: Repeat count



Picture 3-4-19-1

Along X and Y axis positioning, Z axis move quickly to R point, execute G74 spindle start CCW, from R point to Z point to execute tapping, after tapping finished, spindle stop and execute paused, then spindle in the CCW direction rotate tool back to R point, spindle stop, then execute move quickly to initial position. When tapping is executing, feed rate and spindle rate are 100%.

Rigid Mode:

Use any method below can specified rigid tapping:

- Before tapping command specified M29 S****
- Including tapping command program segment specified M29 S****

While G74 command and M command in the same program segment, i in the first hole positioning operation at the same time execute M code, then, system process the next boring hole action.

When pointed repeat times K, only execute M code for the first hole, and not execute M code to other holes.

P is modal command, min. parameter setting by data parameter P336, max parameter setting by data parameter P337. When P less than P336 parameter setting value, then work as min value, if greater than P337 parameter setting value, then work as max value. If pointed in the unexecuted boring process program, P cannot save as modal data.

Tool length compensation, in the fixed loop, pointed tool length compensation G43, G44 or G49. When positioning R point, add offset or cancel offset.

Axis switch: Cancel fixed loop before switch tapping axis. If switch tapping axis in rigid mode, it will alarm. If

pointed S and axis move command between M29 and G84, system will alarm. If M29 pointed in rigid loop, system will alarm.

Screw lead: Feed speed/ Spindle rotate speed

Z axis feed speed= Spindle rotate speed * screw lead

Example:

Spindle speed 1000rpm;

Screw lead 1.0mm;

Z axis feed speed= 1000*1=1000mm/min;

G00 X120 Y100; Positioning

M29 S1000; Pointed rigid mode

G74 Z-100 R-20 F1000; Rigid tapping

Limited:

F: If pointed F value over cutting feed speed's upper limit value, it will alarm.

S: If speed larger than pointed level max speed, it will alarm. Speed level decided by data parameter P244~P246.

Cancel: cannot point 01 group G code (G00 to G003) in the same program segment; G60 is modal G code, Position parameter NO:48#0 No:12#0 set to 1) and G74, or G74 will be cancelled.

Tool Offset: In the process of fixed loop positioning tool radius offset is ignored

Program restart: Program restart invalid during rigid tapping period.

3.4.20 Right-handed rigid tapping G84

Command Format: G84 X_Y_R_P_F_K_

Function: Spindle motor in the rigid way of work is a servo motor, the directive can be realized in the left-hand tell high-precision tapping. Though in the same position, many times repeat execute tapping command, the thread not disorderly clasp.

Description:

X_Y_: Hole Location Data;

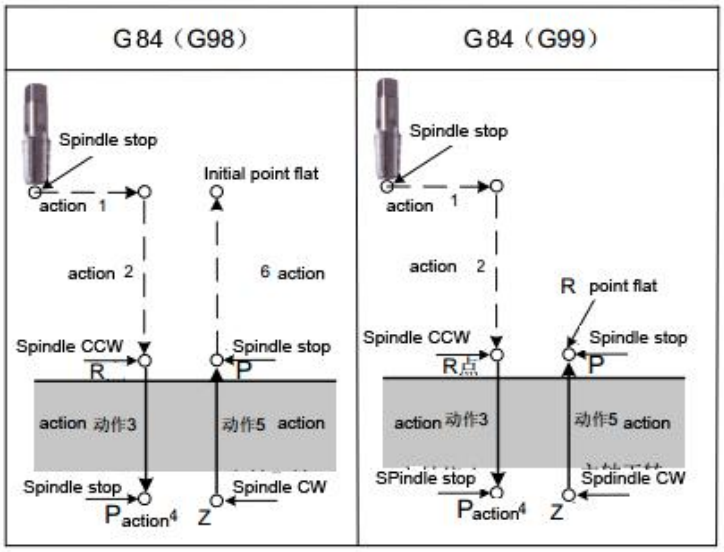
Z_:Incremental programming pointed the distance from R to the bottom of the hole; absolute program indicate the hole bottom's absolute coordinates.

R_: Incremental programming indicates the distance from initial point flat to R; absolute program indicate R absolute coordinates.

P_: Pause time;

F_: Cutting feed rate.

K_: Repeat count



Picture 3-4-20-1

Along X and Y axis positioning, Z axis move quickly to R point, execute G84 spindle start CCW, from R point to Z point to execute tapping, after tapping finished, spindle stop and execute paused, then spindle in the CCW direction rotate tool back to R point, spindle stop, then execute move quickly to initial position. When tapping is executing, feed rate and spindle rate are 100%.

Rigid Pattern:

Use any method below could pointed rigid pattern:

- Before tapping command point M29 S****
- Including tapping command program segment pointed M29 S****

While G74 command and M command in the same program segment, i in the first hole positioning operation at the same time execute M code, then, system process the next boring hole action.

When pointed repeat times K, only execute M code for the first hole, and not execute M code to other holes.

P is modal command, min. parameter setting by data parameter P336, max parameter setting by data parameter P337. When P less than P336 parameter setting value, then work as min value, if greater than P337 parameter setting value, then work as max value. If pointed in the unexecuted boring process program, P cannot save as modal data.

Tool length compensation, in the fixed loop, pointed tool length compensation G43, G44 or G49. When positioning R point, add offset or cancel offset.

Axis switch: Cancel fixed loop before switch tapping axis. If switch tapping axis in rigid mode, it will alarm. If pointed S and axis move command between M29 and G84, system will alarm. If M29 pointed in rigid loop, system will alarm.

Screw lead: Feed speed/ Spindle rotate speed

Z axis feed speed= Spindle rotate speed * screw lead

Example:

Spindle speed 1000rpm;

Screw lead 1.0mm;

Z axis feed speed= 1000*1=1000mm/min;

G00 X120 Y100; Positioning

M29 S1000; Pointed rigid mode

G74 Z-100 R-20 F1000; Rigid tapping

Limited:

F: If pointed F value over cutting feed speed's upper limit value, it will alarm.

S: If speed larger than pointed level max speed, it will alarm. Speed level decided by data parameter P244~P246.

Cancel: cannot point 01 group G code (G00 to G003) in the same program segment; G60 is modal G code, Position parameter NO:48#0 No:12#0 set to 1) and G74, or G74 will be cancelled.

Tool Offset: In the process of fixed loop positioning tool radius offset is ignored

Program restart: Program restart invalid during rigid tapping period.

3.4.21 Deep hole rigid tapping (chip) loop

Command Format: G84(or G74) X_Y_R_P_Q_F_K_

Function: In the deep hole rigid tapping, execute times feed until the bottom of hole.

Description:

X_Y_: Hole Location Data;

Z_:Incremental programming pointed the distance from R to the bottom of the hole; absolute program indicate the hole bottom's absolute coordinates.

R_: Incremental programming indicates the distance from initial point flat to R; absolute program indicate R

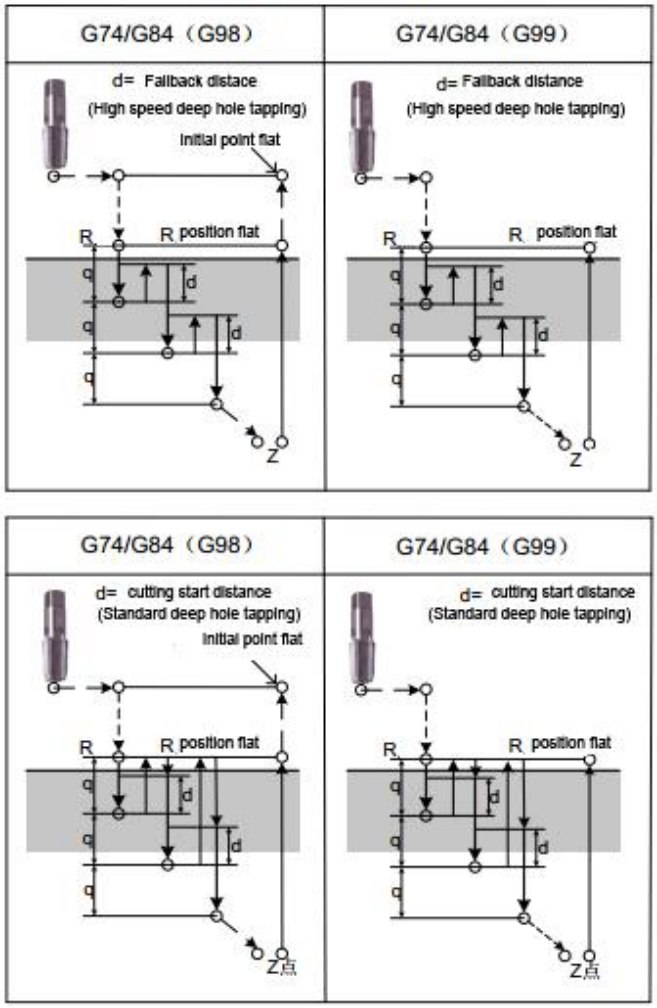
absolute coordinates.

P_: Pause time;

Q_: Depth of cut per feed cutting

F_: Cutting feed rate.

K_: Repeat count



Picture **3-4-21-1**

Deep hole rigid tapping loop have two mode: High speed deep hole tapping loop and standard deep hole tapping loop, settle by position parameter No:58#1.

When position parameter No:58#1=1is high speed hole tapping loop: Along X and Y axis position, execute move quickly to R point, from R point to the depth of cut Q (Every depth of cutting feed) execute cutting, then tool return to distance d (it is decided by data parameter P339), position parameter N0:58#6 setting rigid tapping return, feed

rate effective or not; position parameter N0:58#7 pointed return speed feed rate, position N0:58#5 setting rigid tapping cutting feed, return cutting, whether use the same time constant. When reach Z point, spindle stop, then rotate back in opposite direction.

When position N0:58#1=0, is the standard deep hole tapping loop: Along X and Y axis position, execute move quickly to R point, from R point to the depth of cut Q (Every depth of cutting feed) execute cutting, then execute return to R point, position parameter N0:58#6 setting rigid tapping return, feed rate effective or not; position parameter N0:58#7 pointed return speed feed rate, from R point to cutting destination distance d (it is decided by data parameter P339), as cutting speed F's value re-execute cutting, position N0:58#5 setting rigid tapping cutting feed, return cutting, whether use the same time constant. When reach Z point, spindle stop, then rotate back in opposite direction.

Limited:

F: If specified F value is larger than cutting feed speed's max. value, it will alarm.

S: If speed larger than specifed max.gear' speed, it will alarm. Gear speed decided by data parameter P244~P246.

Cancel: cannot specify 01 group G code (G00 to G003) and G84(or G74) in the same program segment; otherwise G84 (G74) will be cancelled.

Tool Offset: In the process of fixed loop positioning tool radius offset is ignored

Program restart: Program restart is invalid during rigid tapping period.

3.4.22 Fixed loop cancel G80

Command format: G80

Function: Cancel fixed loop

Description:

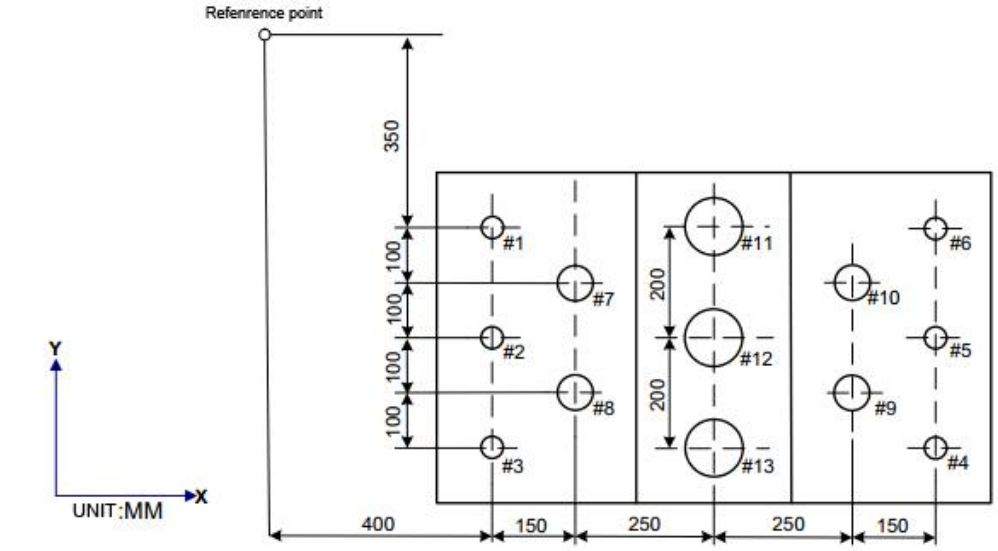
Cancel all fixed loop, execute normal operation. R and Z point also be cancelled, hole drilling, hole boring data are be cleared.

Example:

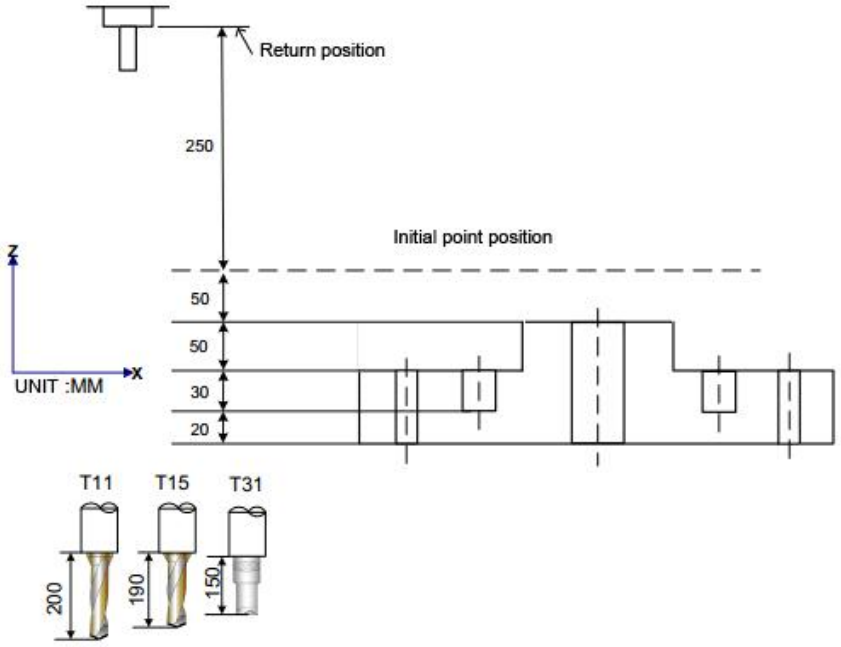
M3 S100	Spindle start rotate
G90 G99 G88 X300 Y-250 Z-150 R-120 F120;	Positoning, bore 1 hole, then back to R.
Y-550;	Positioning, bore 2 hole, then back to R.
Y-750;	Positioning, bore 3 hole, then back to R.
X1000;	Positioning, bore 4 hole, then back to R

Y-550;	Positioning, bore 5 hole, then back to R.
G98 Y-750;	Positioning, bore 6 hole, then back to G80:
G28 G91 X0 Y0 Z0;	Back to reference point cancel fixed loop
M5;	Spindle stop rotate.

Example: Below use tool length compensate, comprehension declare fixed loop's use.



1～6...Drill Φ10 hole
7～10...Drill Φ20 hole
#11～13.Bore Φ95 hole



Offset No.11's value is 200, offset No. 15's value is 190, offset No. 31's value is 150 use as offset value separately setting, program as below:

N001 G92 X0 Y0 Z0; Coordinate system setting to reference point

N002 G90 G00 Z250 T11 M6; Switch tool

N003 G43 Z0 H11; In the initial point working flat tool length compensate

N004 S300 M3; Spindle start

N005 G99 G81 X400 Y-350 Z-153 R-97 F-120; after positioning, process #1hole

N006 Y-550; after positioning, process #2 hole, return back R point flat.

N007 G98 Y-750; after positioning, process #3 hole, return back initial point flat

N008 G99 X1200; after positioning, process #4 hole, return back R point flat.

N009 Y-550; after positioning, process#5 hole, return back R point flat.

N010 G98 X1200; after positioning, process#6 hole, return back initial point flat

N011 G00 X0 Y0 M5; Return back to reference point, spindle stop.

N012 G49 Z250 T15 M6; cancel tool length compensate, switch tool.

N013 G43 Z0 H15; Initial point flat, tool length compensate.

N014 S200 M3: Spindle start.

N015 G99 G82 X550 Y-450 Z-130 R-97 P30 F70; after positioning, process #7 hole, return back R point flat

N016 G98 Y-650; after positioning, process #8 hole, return back initial point flat

N017 G99 X1050; after positioning, process #9 hole, return back R point flat.

N018 G98 Y-450; after positioning, process #10 hole, return back initial point flat.

N019 G00 X0 Y0 M5; Return back to reference point, spindle stop

N020 G49 Z250 T31 M6; cancel tool length compensate, switch tool.

N021 G43 Z0 H31; initial point flat tool length compensate

N022 S100 M3; spindle start.

N023 G85 G99 X800 Y-350 Z-153 R47 F50; after positioning process #11 hole, return back R point flat

N024 G91 Y-200; after positioning process #12 hole, return back R point flat

N025 Y-200; after positioning process #13 hole, return back R point flat

N026 G00 G90 X0 Y0 M5; Return back to reference point, spindle stop.

N027 G49 Z0; Cancel tool length compensate

N028 M30; Program stop

3.5 Tool Compensate G Code

3.5.1 Tool compensate G43, G44, G49

Function:

G43 specified positive tool compensation

G44 specified negative tool compensation

G49 cancel tool length compensation

Command format:

System support A/B two tool length offset method, use position parameter No: 3#4 setting tool lengths offset mode.

Method A:

G43 Z_H_;

G44

Method B:

G17 G43 Z_H;

G17 G44 Z_H;

G18 G43 Y_H;

G18 G44 Y_H;

G19 G43 X_H;

G19 G44 X_H;

Tool length offset method cancel: G49 or H0.

Description:

Above command's function is shift an offset value from the specified axis destination position. The D-value between programming supposed tool length value (Usually set as the first tool) and actual process use tool length value salved in advance in the offset save room, so it is no need to change program, only change tool length compensate could use different length tool process accessories.

G43, G44 pointed different offset direction, using H code pointed offset number.

• Offset direction

G43 Positive direction offset (Normally use offset method)

G44 Negative direction offset

No matter absolute command, or incremental command, in G43, in the program pointed axis command destination coordinate value plus H code pointed offset value (setting in the offset save room); G44, minus H code pointed offset value, and then use the calculated result as destination coordinate value.

G43, G44 are model G code, they are effective before meet other G code in the same group.

- Offset value's pointed

H code pointed length offset number, the offset number corresponding offset value and program Z axis move command value plus or minus, then become into new Z axis move command. According to the necessary offset number could pointed H00~H256.

Offset value setting range as below:

Format 3-5-1-1

	Mm input
Compensate H	-999.9999mm~+999.999mm

Offset number 00, it means H00 corresponding offset value is 0. H00 corresponding offset value cannot set in system.

Attention: Offset number change caused to offset value change, only use new offset value instead of old offset value, not use new offset value and old compensate value plus.

Example:

H01……Offset value 20

H02……Offset value 30

G90 G43 Z100 H01; ……..Z goes to 120

G90 G43 Z100 H02; ……..Z goes to 130

- Offset number's effective turns

Once length offset model build, the current offset number effective at once, when offset number change, new offset value instead of old offset value at once.

For example:

0XXXXX;

H01;

G43 Z10; (1) Offset number H01 effective

G44 Z20 H02; (2) Offset number H02 effective

H03; (3) Offset number H03 effective

G49; (4) Offset value cancel, H00 effective

M30:

- Cancel tool length compensate

Use G49 or H00 cancel tool length compensate. After command G49 or H00, system cancel tool length compensate at once.

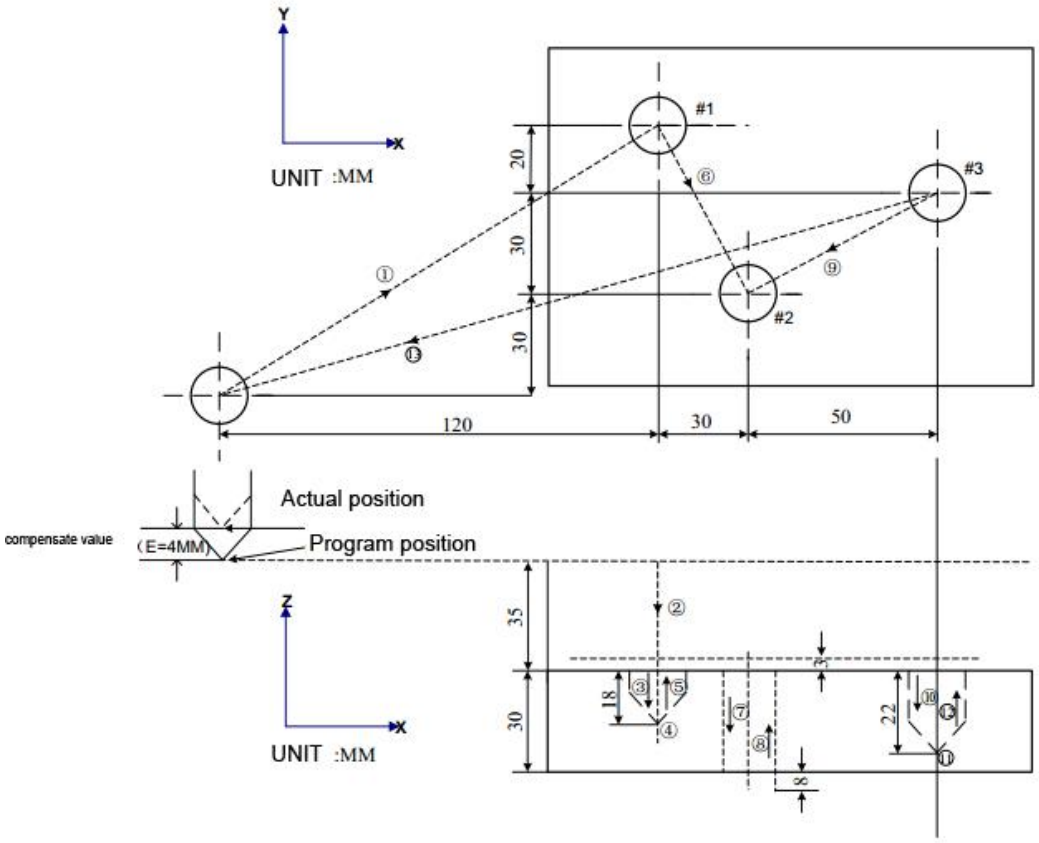
Attention:

- **Tool length offset B mode is along two or two more axis execute, use G49 cancel all axes' offset value, use H00 only cancel the offset value of perpendicular to the axis in the flat .**
- **Suggest add Z axis move command for length compensates build and cancel, or it will use the current point build and cancel length compensate. So, use G49 please also confirm Z axis in the safety height, to avoid tool knock or work piece damage.**
- Tool length offset mode G53 G28 or G30 command

Tool length offset mode pointed G53 G28 or G30 command, tool length offset axis's value move to the pointed position cancel (G53 move to command position cancel, G28, G30 move to reference point cancel), but mode code show no change to G49, and except tool length offset value axis, others axes no cancel, all axes move to the pointed

position cancel length offset value; when G28 or G30 with G49 in a same segment, all axes move to reference point cancel length offset value. Cancel tool length offset value will be recovered in the next program segment.

- Tool length compensate examples:
- Tool length compensate (Process#1,#2, #3 hole)
- H01=Offset value -4



Picture 3-5-1-1

```
N1 G91 G00 X120 Y80 ;    (1)
N2 G43 Z-32 H01 ;        (2)
N3 G01 Z-21 F200 ;       (3)
N4 G04 P2000 ;           (4)
N5 G00 Z21 ;             (5)
N6 X30 Y-50 ;            (6)
N7 G01 Z-41 F200 ;       (7)
```

```
N8 G00 Z41 ;          (8)
N9 X50 Y30 ;          (9)
N10 G01 Z-25 F100 ;   (10)
N11 G04 P2000 ;       (11)
N12 G00 Z57 H00 ;     (12)
N13 X-200 Y-60 ;      (13)
N14 M30 ;
```

3.5.2 Tool Radius Compensate G40/G41/G42

Command format:

G41 D_X_Y_;
G42 D_X_Y_;
G40 X_Y_;

Function:

G41 pointed tool move direction's left compensate.

G42 pointed tool move direction's right compensate

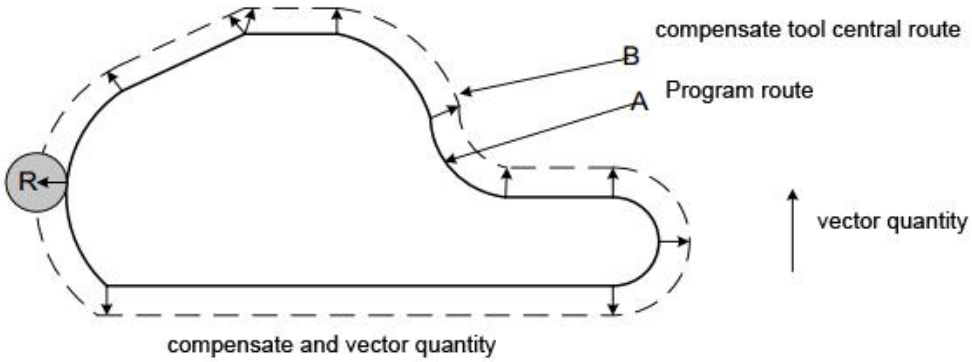
G40 cancel tool radius compensates.

Description:

- Tool radius compensate function

Picture as below use radius R's tool cutting work piece A, tool central route is picture B, route B distance A is R.

Tool offset work piece A radius's distance is compensate.



Picture 3-5-2-1

Program people use tool radius compensate mode program process programming, during process, test tool diameter and saved into CNC's storage, tool route become into compensate route B.

- Compensate value (D value)

D code pointed radius offset number, and its offset number corresponding offset value with move command value in the program plus or minus, come into new move command. According to the necessaries could pointed D00~D256.

Using LCD/MDI panel, could setting offset number corresponding offset value in advance in offset storage.

Compensate value's setting range as below:

Format 3-5-2-1

	Mm input
Compensate D	-999.9999mm~+999.999mm

Attention: D00's compensate value is 0 in system; user cannot setting or modify it.

Compensate flat's change must work after cancel compensate mode. No cancel compensate mode and change compensate flat, system will alarm.

- Flat choose and vector quantity

Compensate calculate is execute during the G17 G18 G19 choose flat. The flat called compensate flat. For example, when choose XY flat, in the program use (X.Y) execute compensate calculate and vector quantity calculate. It is no effect to the coordinate which outside compensate flat.

In the three axis control at the same time, only the tool path of the projection flat in compensation.

Compensate flat's change must after cancel compensate mode.

Format 3-5-2-2

G code	Compensate flat
G17	X-Y flat
G18	Z-X flat
G19	Y-Z flat

- G40 G41 and G42

Using G40 G42 and G42 command tool radius compensate vector' cancel and execute. They are with G00 G01 G02 G03 command combination, definition a mode confirm compensate vector's value and direction.

Format 3-5-2-3

G code	Function
G40	Tool radius compensate cancel
G41	Tool radius left compensate
G42	Tool radius right compensate

- Tool radius compensate mode G53 G28 G30 command

Tool radius compensate mode pointed G53 G28 or G30 command, tool radius offset axis's value move to the

pointed position cancel (G53 move to command position cancel, G28, G30 move to reference point cancel), but mode code show no change to G40, and except tool length offset value axis, others axes no cancel, when G53 with G40 in a same segment, all axes move to the pointed position cancel radius offset compensate; when G28 or G30 with G40 in a same segment, all axes move to reference point cancel length offset value. Cancel tool radius offset value will be recovered in the next program segment.

Attention: In the compensate mode, position parameter N0:19#4 choose pointed G28 G30 command move to the middle of the point, compensate whether cancel in pause.

Cancel Tool Radius Compensate (G40)

In G00, G02 status, use below command, G40 X_Y_;

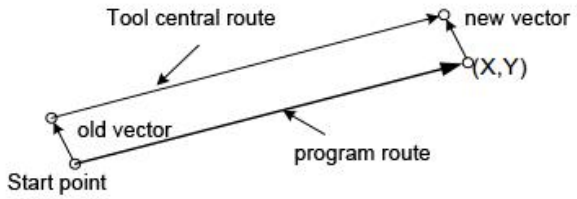
From start point's old vector is forward to destination work in linear motion. In G00, every axes are forward to destination in quickly motion. Use the command, let system from tool compensate status into cancel tool compensate status.

If only G40, without command X_Y_, tool no motion.

Tool Radius Compensate Left (G41)

- G00 G01

G41 X_Y_D_; command in the destination of program segment, to a new vector which is perpendicular to the (X,Y) direction, tool move from start point (old vector)'s cutting-edge to new vector's cutting edge.



Picture 3-5-2-2

Old vector value is 0, use command, let system from tool compensate status into cancel tool compensate status. At the time, D code pointed offset value.

- G02 G03,

  G41……;

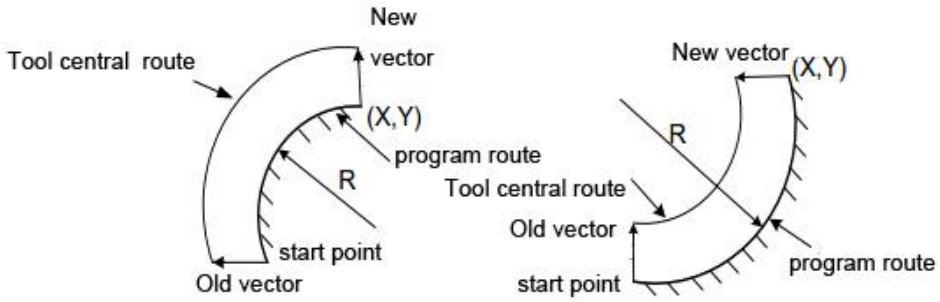
  ……

  ……

G02/G03 X_Y_R_;

According above program could make new vector, it is located in the center of circular arc and the end of a wire, see the arc forward direction, pointing to the left (or right), tool central from the arc's old vector cutting-edge to new vector cutting edge move along the arc. But the premise is the old vector already make out.

Offset vector is from start point or destination point directing to cycle arc centre or deviate from the center.



Picture 3-5-2-3

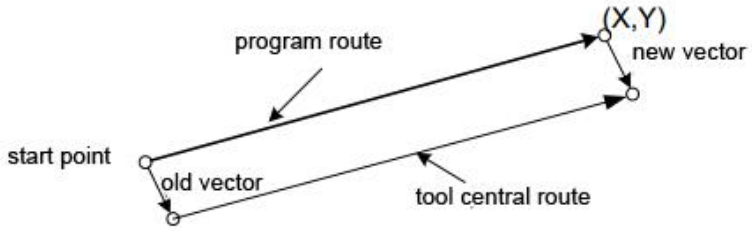
Tool Radius Compensate Right (G42)

G42 is contrary to G41, along tool forward direction, tool is offset on the work piece's right. That is to say G42's vector direction is contrary to G41's. Except the vector direction, offset method is the same as G41.

1) G00、G01

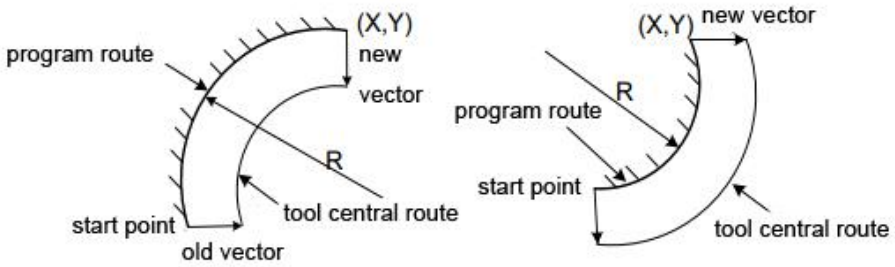
G42 X_ Y_ D_ ;

G42 X_ Y_ ;



Picture 3-5-2-4

2) G02、G03



Picture 3-5-2-5

- Attentions about Offset
- Offset number's pointed

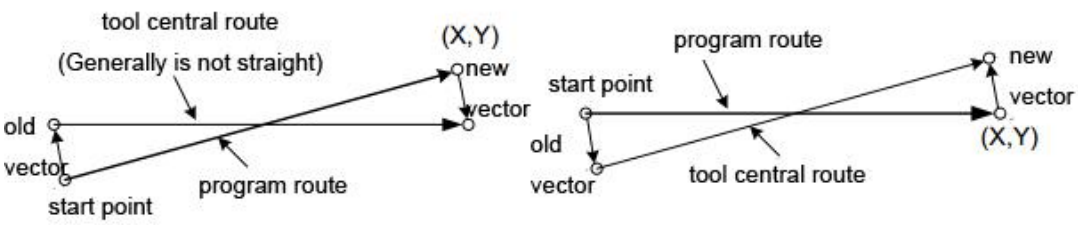
G41 G42 G40 is mode command, offset number use D code pointed. It is could point anywhere before offset cancel status change into tool radius compensate. G41 G42 commands after must with move command, or it will alarm.

- About cancel offset status change into tool radius compensate status

The move command from cancel offset status change into tool radius compensate status must be positioning (G00) or linear interpolation (G01), cannot use arc interpolation (G02 G03).

- Tool radius compensate left and right change

Offset direction from left change to right, or from right change to left, usually pass offset cancel status. But positioning (G00) or linear interpolation (G01) could no pass offset cancel status, change directly. In this situation, tool route as below picture show:

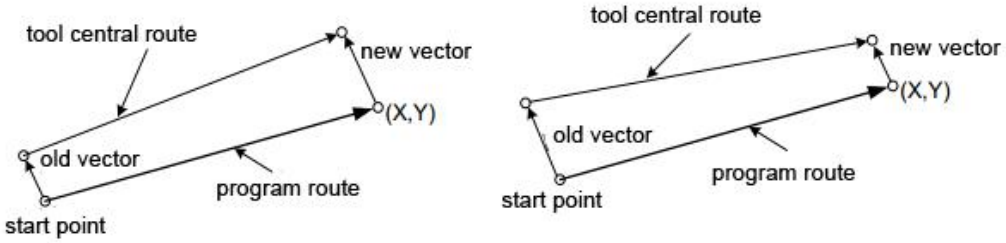


Picture **3-5-2-6**

```
G1G41 D_X_ Y_ ;                G42 D_X_ Y_ ;
......                         ......
G1G42 D_X_ Y_ ;                G41 D_X_ Y_ ;
```

- Offset value's change

Offset value's change, usually is in offset cancel status, in switch tool, for positioning (G00) and linear interpolation, it is also could change in offset status, details please reference as below:



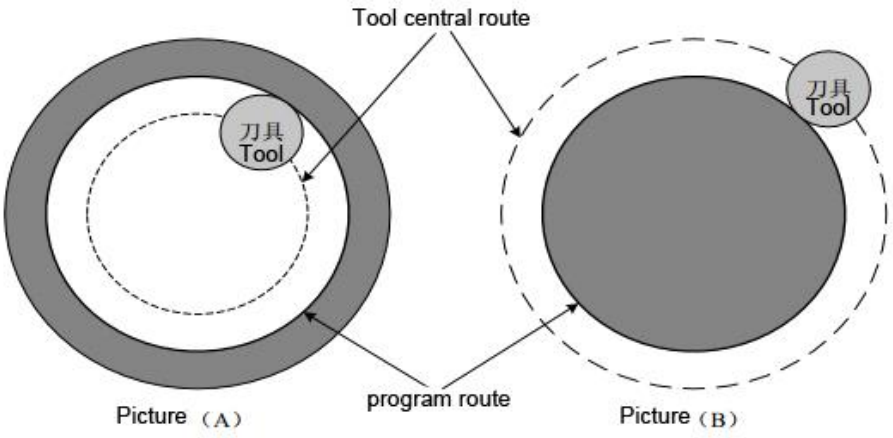
Picture **3-5-2-7** (Offset value's change)

• Offset value's positive negative and tool central route

If set offset value in negative, then process work piece is equal to change G41 G42 in program, so, along the outside of work piece cutting change into along inside of work piece cutting, original along inside of work piece change into outside of work piece process.

As below picture, suppose offset value is positive:

When tool route program as (A) picture show, if set offset value in negative, then tool route work as (B) picture show; The same, when tool route program as (B) picture show, if set offset value in negative, then tool route work as (A) picture show.



Picture 3-5-2-8

General graphics with sharp corners are common (graphics with sharp arc interpolation). But offset value set in negative, cannot process work piece inside circle. Cutting a certain Angle, the Angle of inside there insert appropriate radius of circular arc, smooth transition, to cutting.

Left compensate or right compensate is to see compensate direction on the left side or right side of tool work piece, G41 or G42 let system into compensate mode, G40 let system cancel compensate mode.

Compensate program example as below:

Program segment (1) named start, in this segment G41 command let compensate cancel mode change into compensate mode. In the end of this segment, tool centre use tool radius, which is perpendicular to the next program route (from P1 to P2) direction compensate. Tool compensate value use D07 point, it means compensate number set 7, G41 means tool route left compensate.

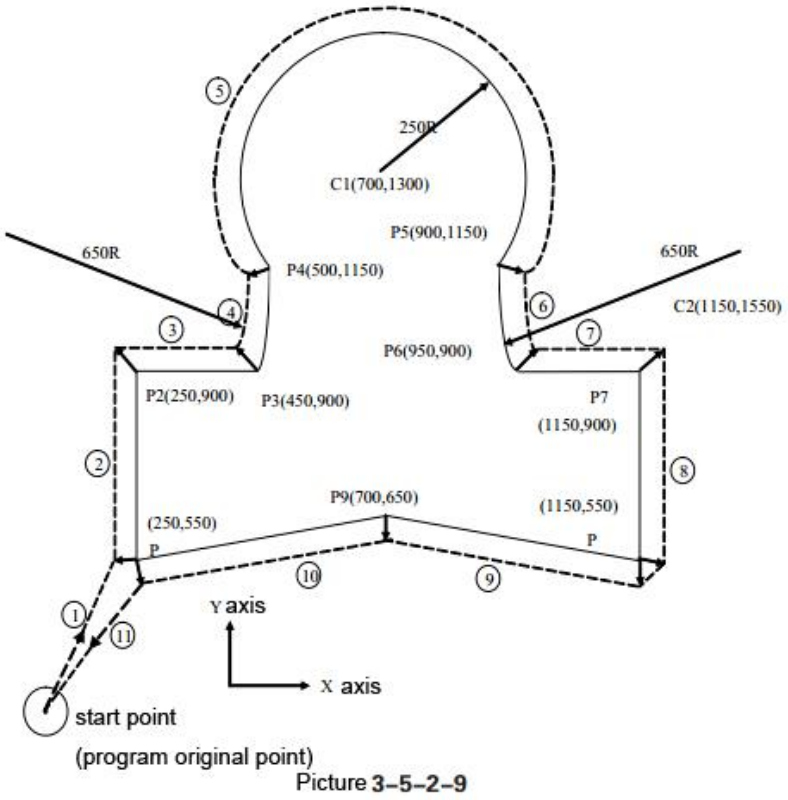
Compensate start, when work piece shape into P1-P2……P9-P10-P11, tool route compensate execute automatically.

Tool route compensate program example as below:

```
G92 X0 Y0 Z0 ;
(1) N1 G90 G17 G0 G41 D7 X250 Y550 ;
(2) N2 G1 Y900 F150 ;
(3) N3 X450 ;
(4) N4 G3 X500 Y1150 R650 ;
```

(compensate value must use compensate number setting in advance)

```
(5) N5 G2 X900 R-250 ;
(6) N6 G3 X950 Y900 R650 ;
(7) N7 G1 X1150 ;
(8) N8 Y550 ;
(9) N9 X700 Y650 ;
(10) N10 X250 Y550 ;
(11) N11 G0 G40 X0 Y0 ;
```

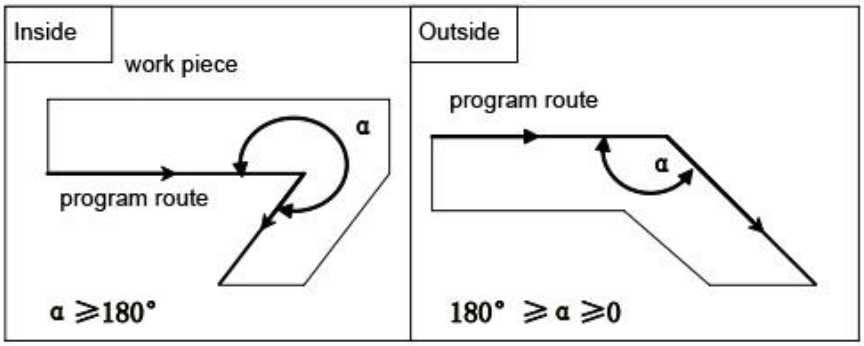


Picture 3-5-2-9

3.5.3 Tool Radius Compensate Details Description

Concept:

Inside and outside: when two segment command build tool route's angle over 180 degree, the route named inside, when the angle between 0 degree and 180 degree named outside



picture 3-5-3-1

Meaning of Symbol

Use below symbols in the after pictures

--S show the position single program execute 1 time.

--SS show the position single program execute 2 time.

--SSS show the position single program execute 3 time.

--L show the tool moving along a straight line.

--C show the tool moving along a arc.

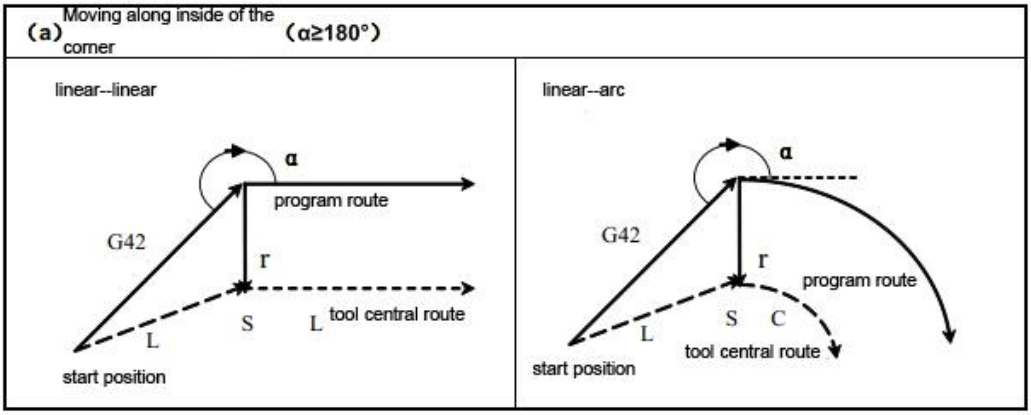
--r show tool radius compensates value.

--Intersection point is two programming track after they r offset the position of the fellowship with one another

--0 show tool central

- **Tool Movement When Tool Start Work**

From offset cancel mode change into offset mode, tool movement as below show (tool start work)

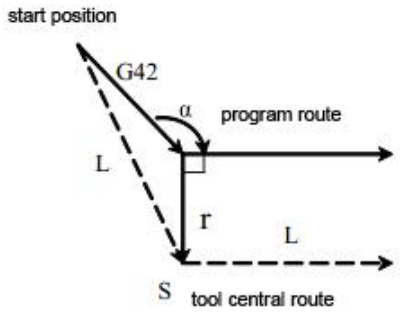


(b) moving along outside of the obtuse angle corner **(180°>α≥90°)**

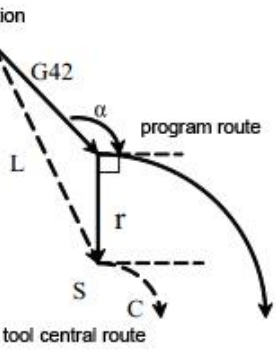
Tool route have A and B mode action when compensate start or cancel, chosed by position parameter N0:19#3.

A

linear--linear

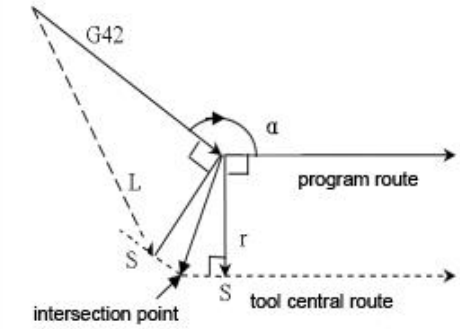


linear--arc



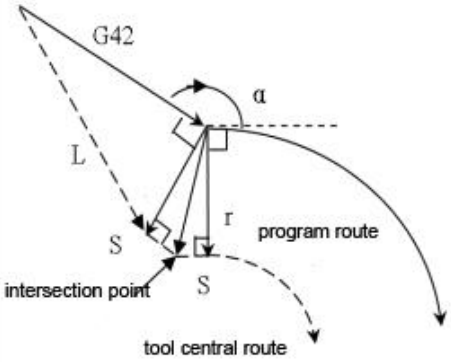
B

linear--linear



Remark:Intersection point is continuous two compensation path intersecting location procedures section

linear--arc

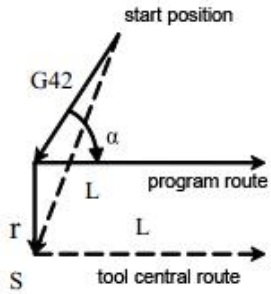


(C) moving along outside of the acute angle **(α<90°)**

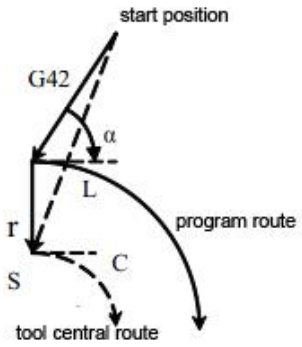
Tool route have A and B mode action when compensate start or cancel, chosed by position parameter N0:19#3

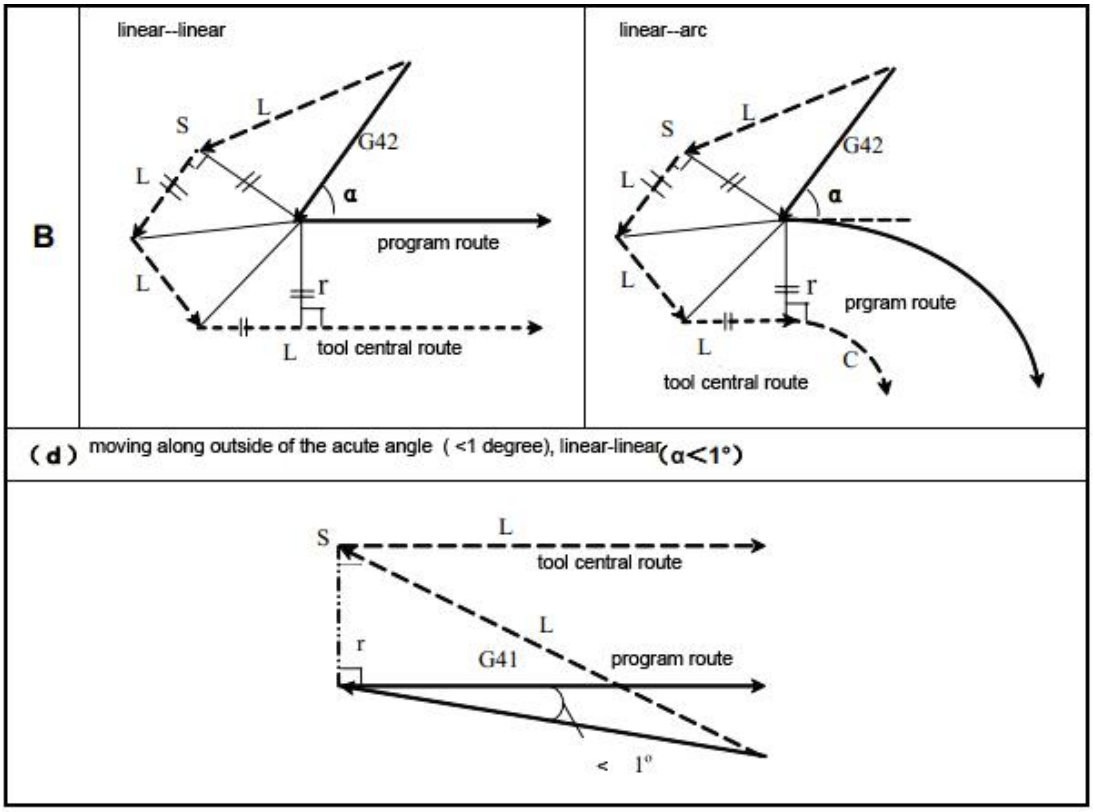
A

linear--linear



linear--arc

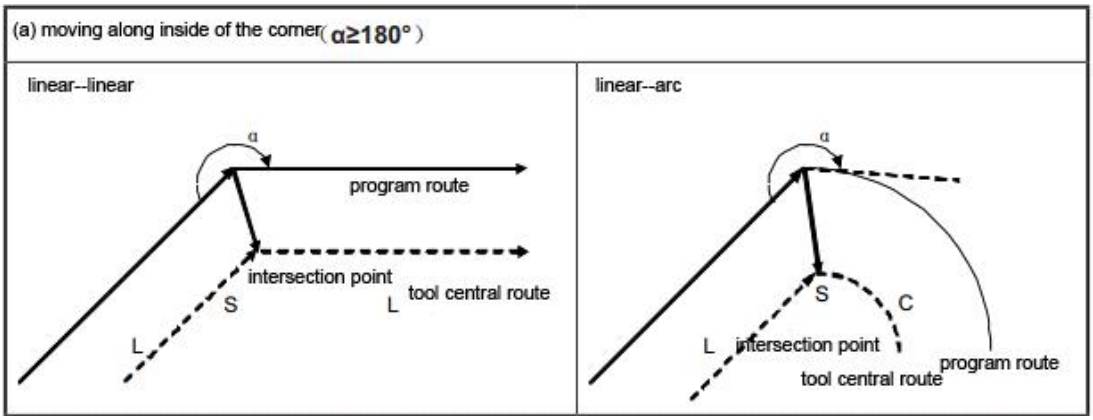


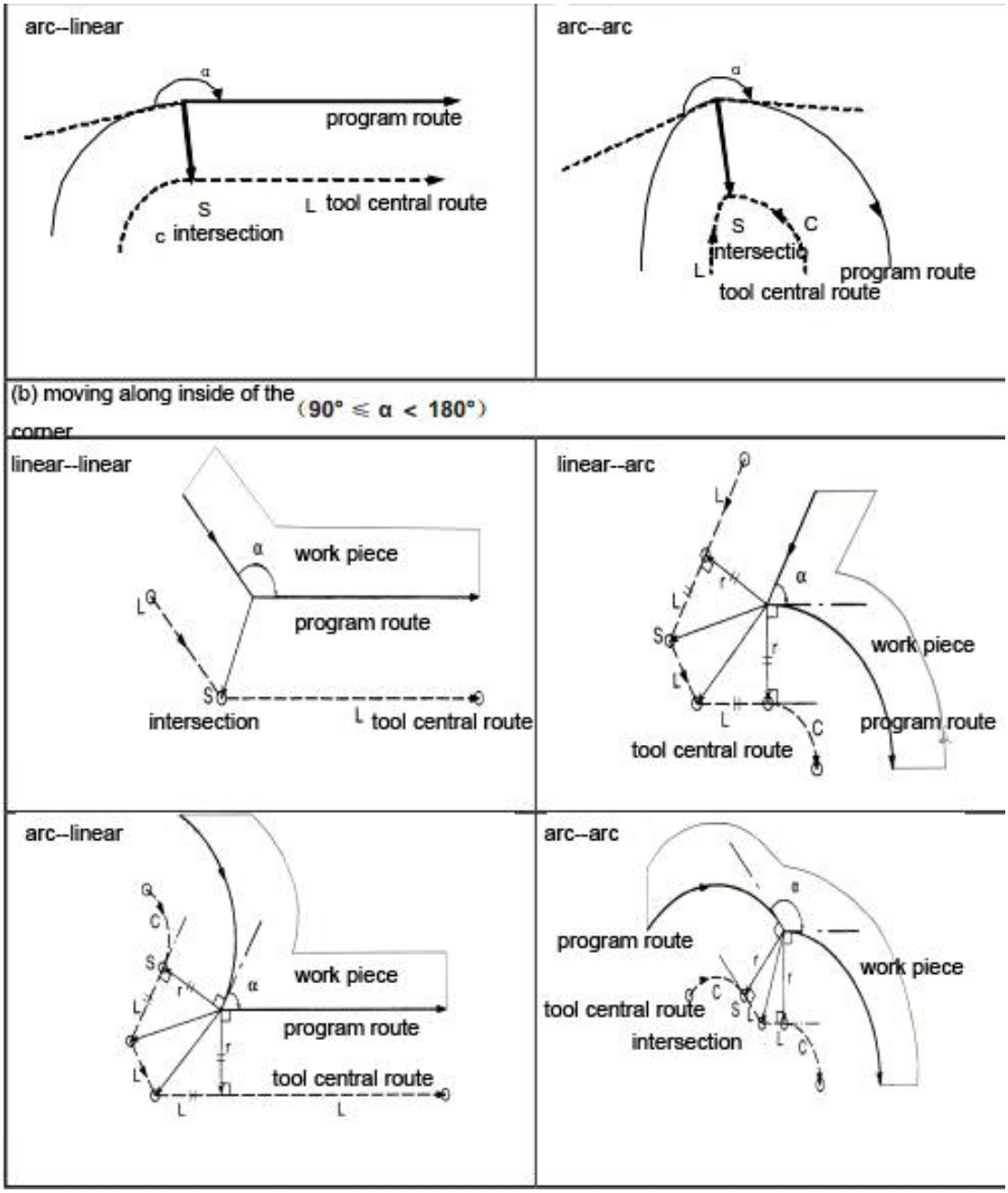


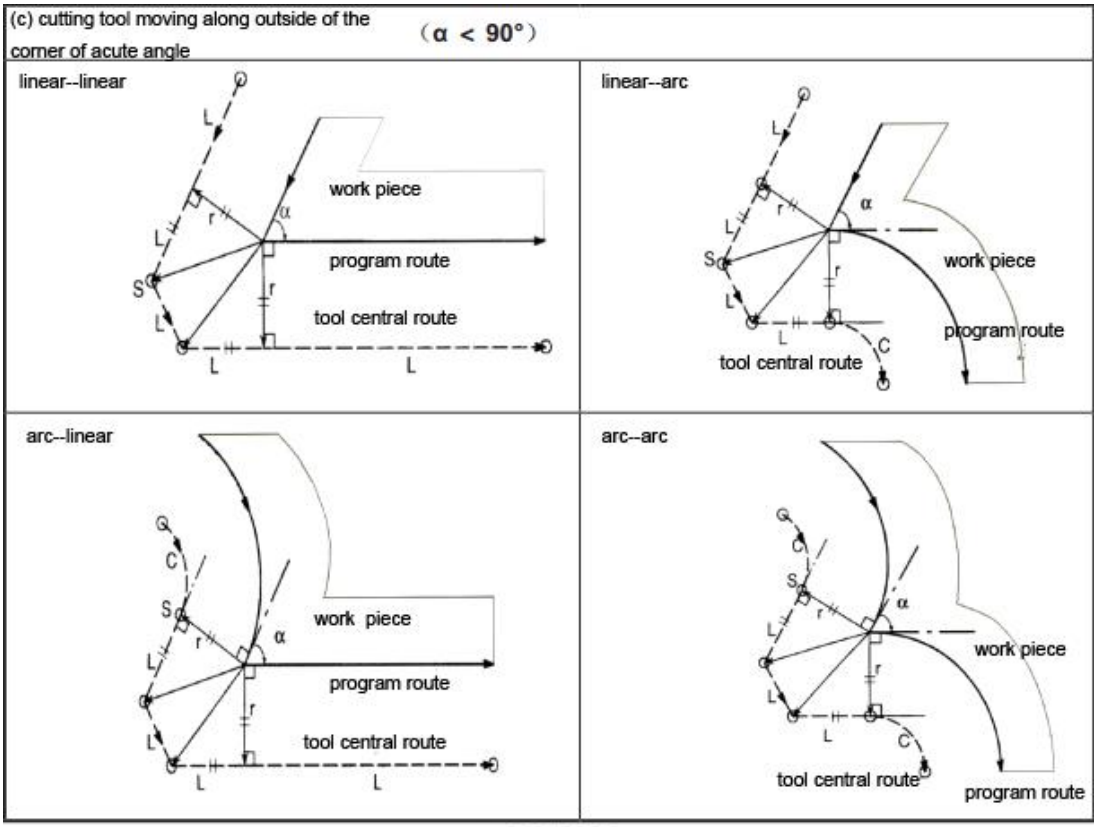
Picture 3-5-3-2

- **Tool Movement in Offset Mode**

In compensation mode cannot change compensation palne, or it will alarm and tool stop at the same time. In offset mode, tool movement as below picture show:



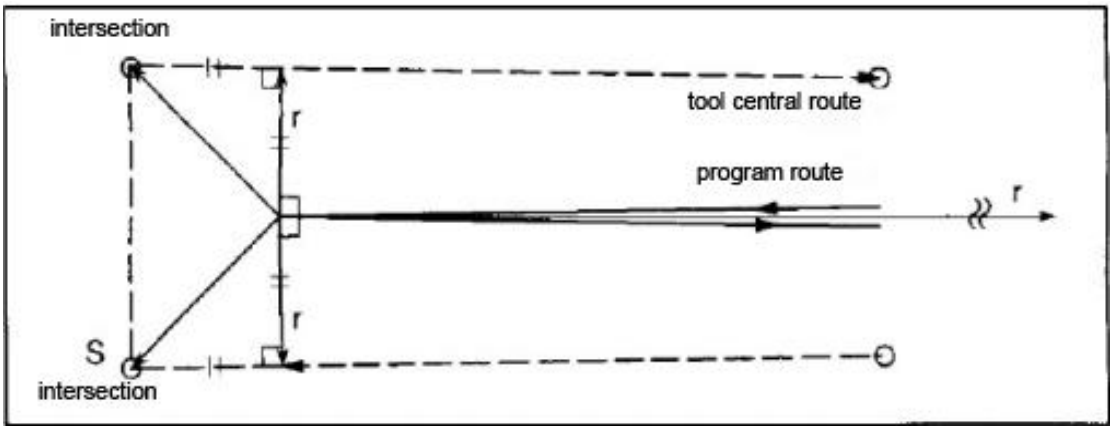

arc--linear
α
program route
S
L tool central route
C intersection
arc--arc
α
S
C
intersection
L
program route
tool central route
(b) moving along inside of the corner (90° ≤ α < 180°)
linear--linear
α
work piece
program route
L
S
intersection
L tool central route
linear--arc
L
α
r
S
work piece
program route
tool central route
C
arc--linear
C
S
r
α
work piece
program route
tool central route
L
arc--arc
program route
work piece
tool central route
intersection
C
S
L



Picture 3-5-3-3

Moving around a super-long vector tool movement (a< 1°)

Linear—Linear

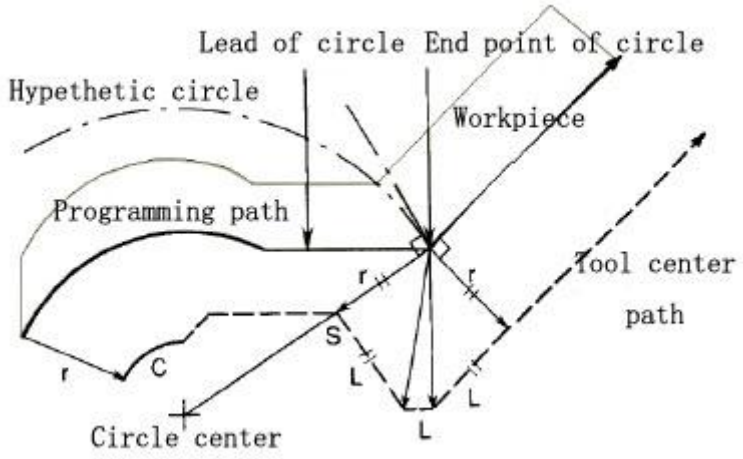


Arc-Linear, Linear-Arc and Arc-Arc, handle as the same method.

- **Special Situation**

The end point of the circular arc not on the arc

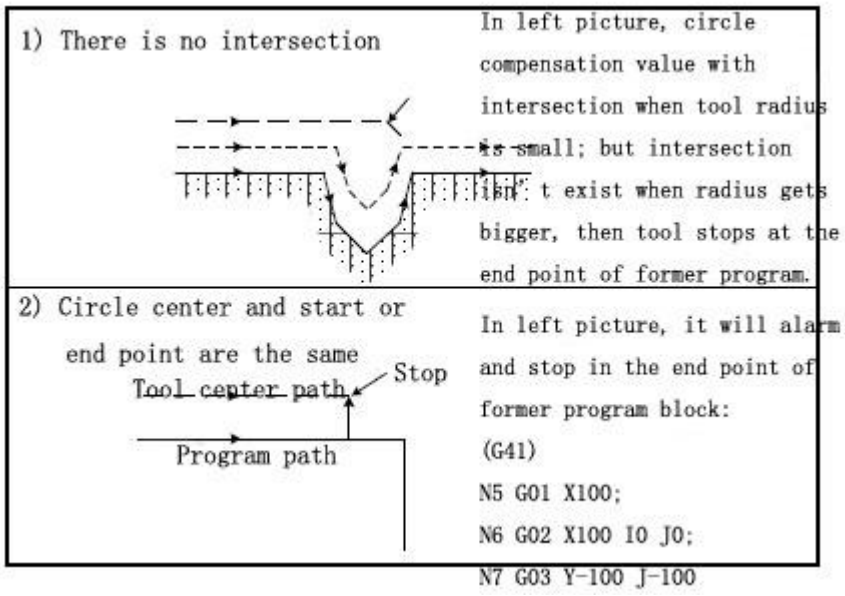
If one linear end point in the arc program wrong as the arc end point (as below picture show), then system think tool radius compensation's program route is one supposed round's centre is the original arc's centre, and through the pointed end point. According to the supposed system build vector and realize compensate ……



The above narrative applies to items moves between two circular trajectory.

No intersection point

If the tool radius compensation is small enough to compensation after the formation of the two circular arc cutting tool center trajectory intersection in point (P). If the tool radius compensation pointed over-large value, intersection P possible does not come out ……



Picture 3-5-3-4

4.Tool Move in Offset Cancel Mode

In compensate mode, when as a condition to satisfy any of the following procedures section executes, system into compensation cancel mode, the program of action is called compensation.

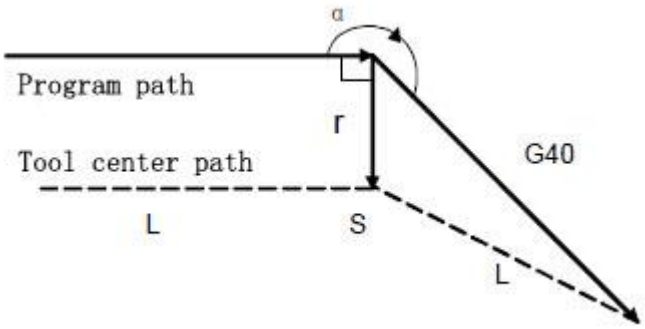
a) Command G40

b) Tool radius compensate number is 0.

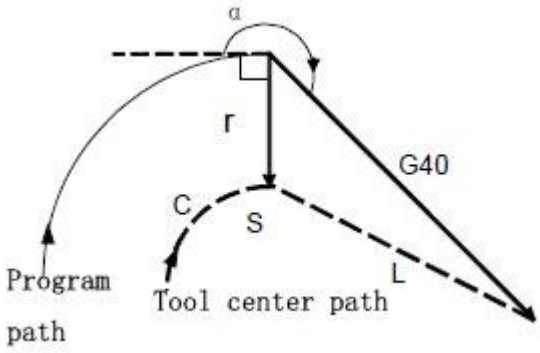
During execute compensate cancel, cannot use arc command (G03 and G02) cancel. If command arc alarm and tool stop.

(a) Move along with the inside of angle (a≤180º)

Straight line - Straight line



Circle – Straight line

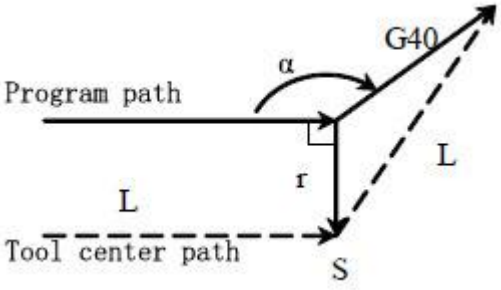


(b) Move along with the inside of angle (90 º≤a≤180º)

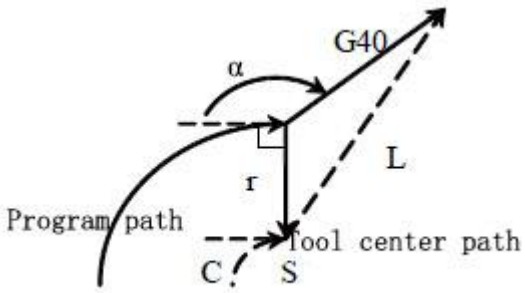
There are two tool path A type and B type in compensation start and cancel motion, chosen by parameter No:19#3

Straight line - Straight line

A

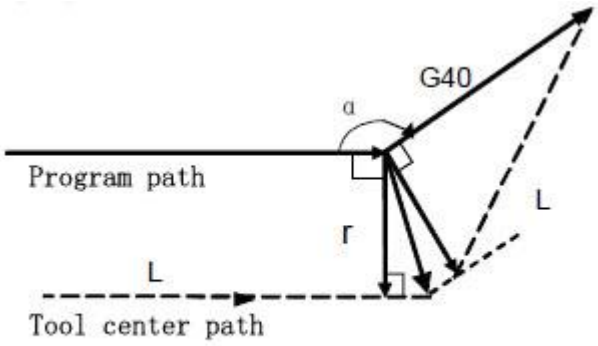


Circle – Straight line

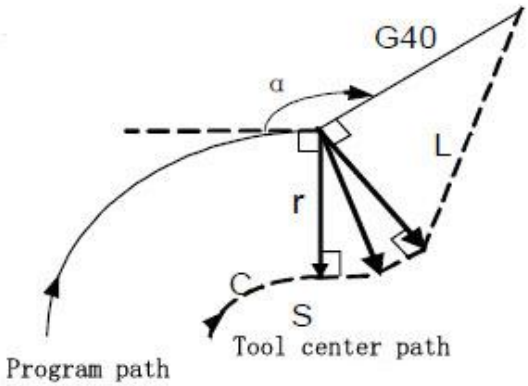


Straight line - Straight line

B



Circle – Straight line

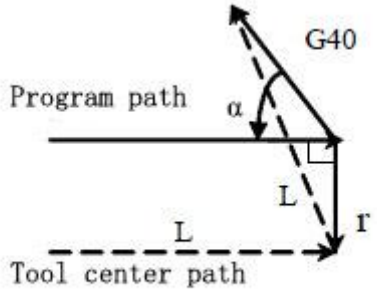


(c) Move along with the outside of acute angle (a<90 º)

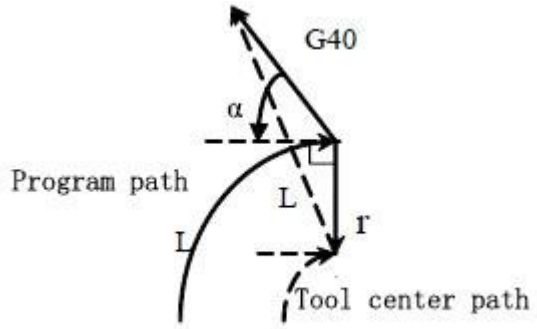
There are two tool path A type and B type in compensation start and cancel motion, chosen by parameter No:19#3

A

Straight line - Straight line

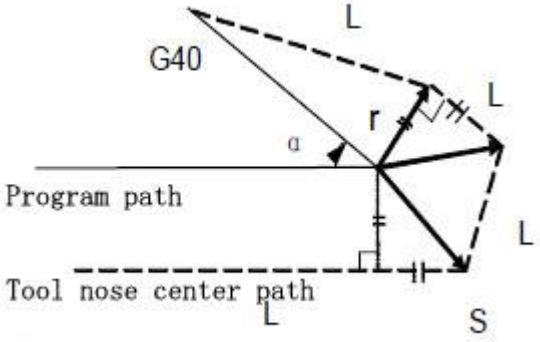


Circle – Straight line

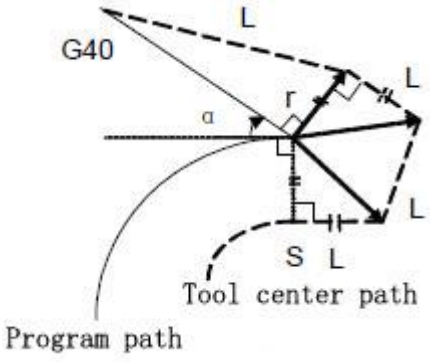


B

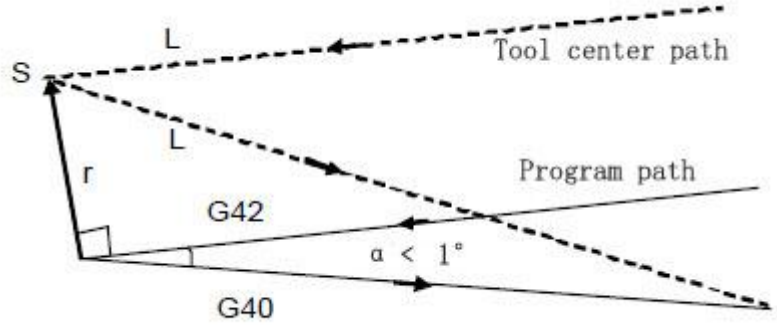
Straight line - Straight line



Circle – Straight line



(d) Move along with the outside of acute angle which is smaller than 1 º, Straight line - Straight line (a<1 º)



5.Change Compensate Direction in Compensate Mode

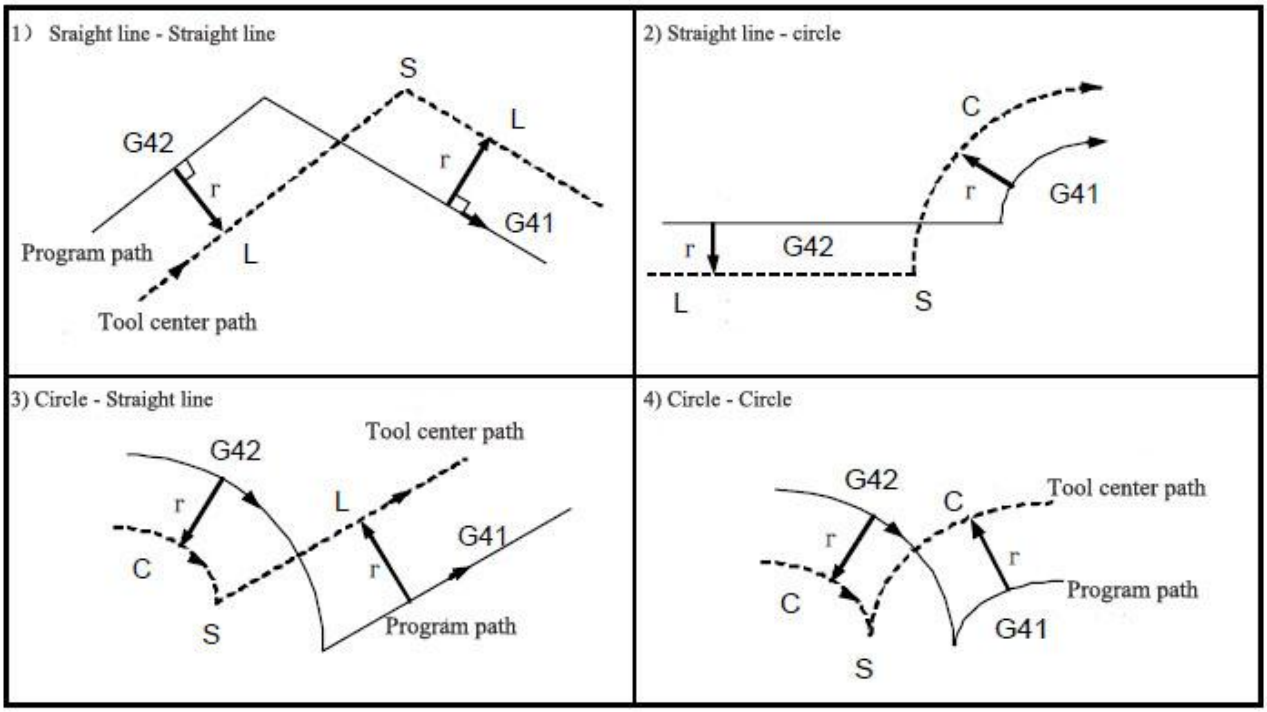
Tool radius compensate G code (G41 and G42) decide compensate direction, compensate vector symbol as below:

Format 3-5-3-1

Compensation symbol	+	-
G code		
G41	Left side compensation	Right side compensation

G42	Right side compensation	Left side compensation

In special situation, in compensate mode could change compensate direction. But program can't be changed after start up beginning program block and the latter program block. Compensate direction change, have no concept of inside and outside. Below compensate vector supposed as positive.

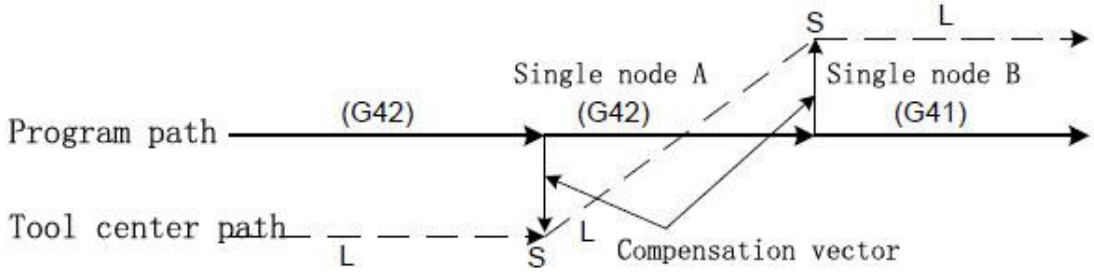


Picture 3-5-3-6

(v) If compensate execute normally, but no intersection.

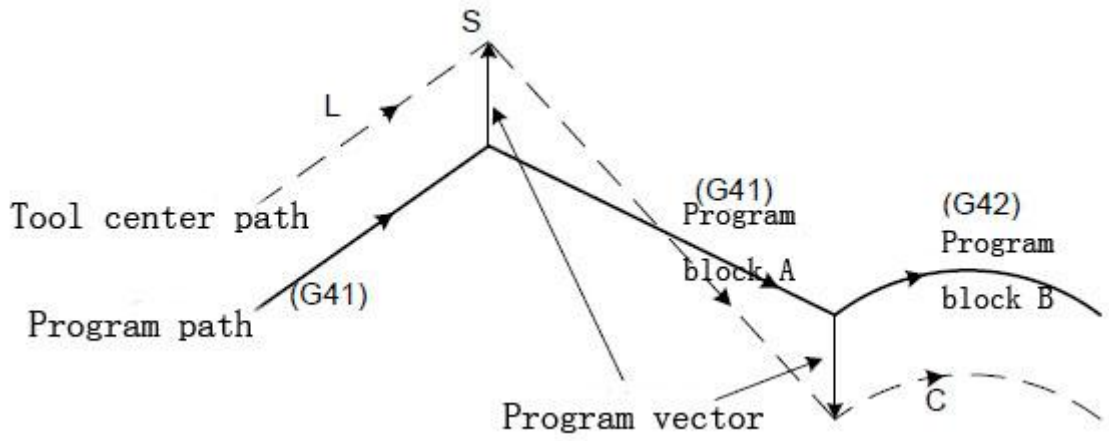
When use G41 and G42 change program segment A to segment B's offset direction, if don't need compensate route intersection, in program segment B' start point do perpendicular to the vector of the program segment B.

(1) Straight line - Straight line



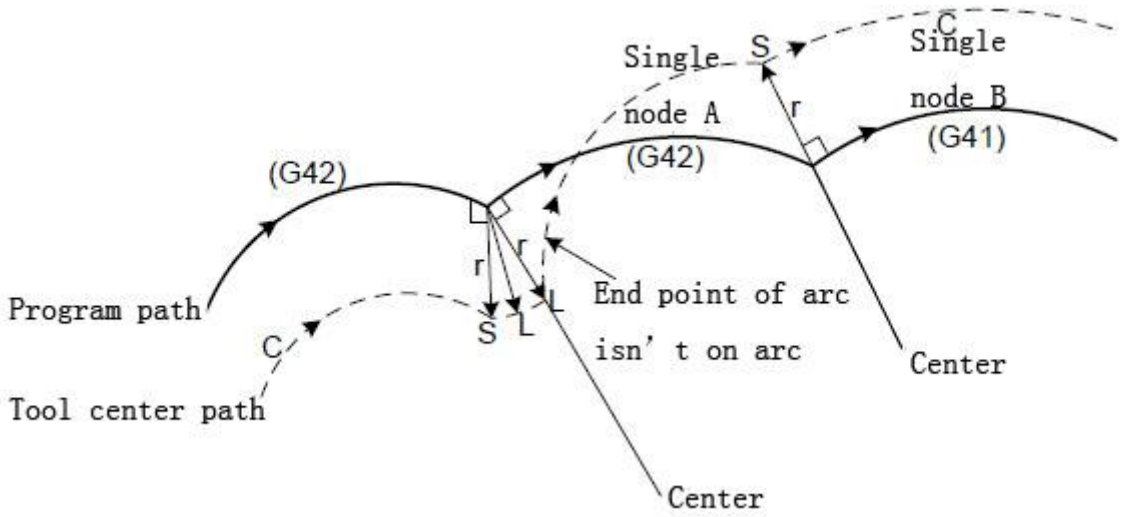
Picture 3-5-3-7

(2) Straight line – Circle arc



Picture 3-5-3-8

(3) Circle arc – Circle arc



Picture 3-5-3-9

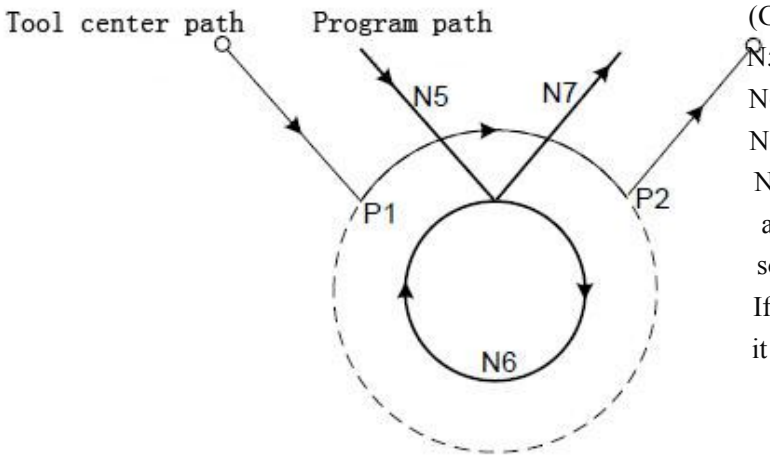
(vi)Tool radius compensate make tool central route's length more than a circumference, usually below situations will not happen. But G41 and G42 change; it will occurfollowing conditions:

Arc--arc (linear--arc)

Change tool compemsate direction system will be alarm, when tool number D0, arc command cannot cancel tool compemsate.

Linear--linear

Tool compensation direction can be changed.



(G42)
N5 G02 G91 X500 Y-700;
N6 G41 G02 J-500;
N7 G42 G01 X500 Y700;
Now, tool center path is not circle arc, it is an arc from P1 to P2. In some conditions, it may alarm.
If hope tool move by entire circle, it must be segment instructions.

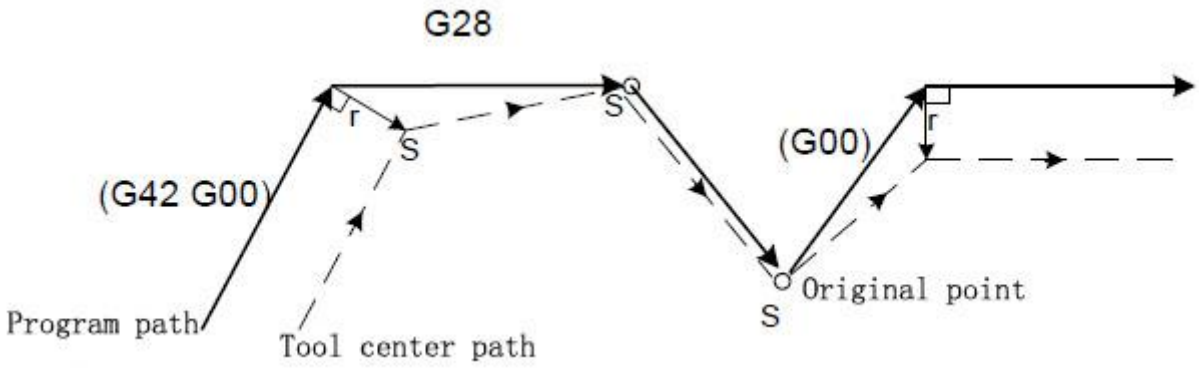
Picture 3-5-3-10

6. Compensation to Cancel Temporarily

In compensate mode, position parameter N0:19#4 choose pointed G28,G30 code command, compensate whether cancel in middle point or not. The operation's details method, please reference compemsate cancel and compensate start's detials description.

a) G28 returns reference automaticly

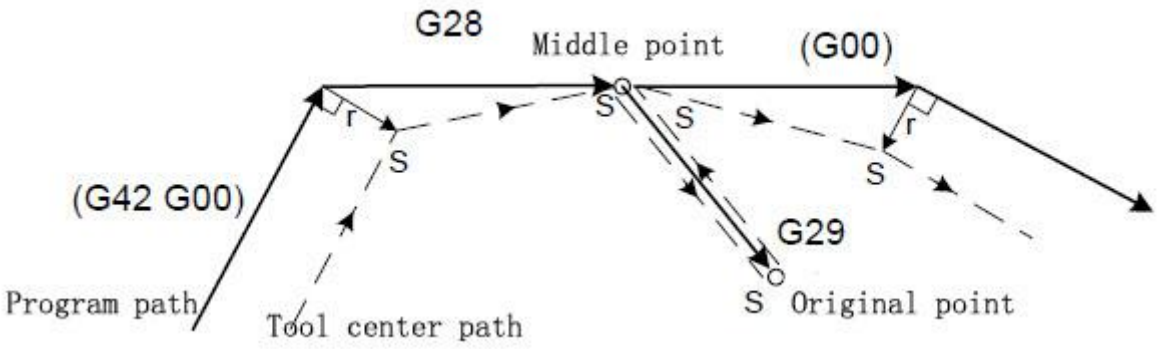
In compensate mode,if command G28， compensate will cancel in middle point.After reference point back, compemsate mode will recover automaticlly.



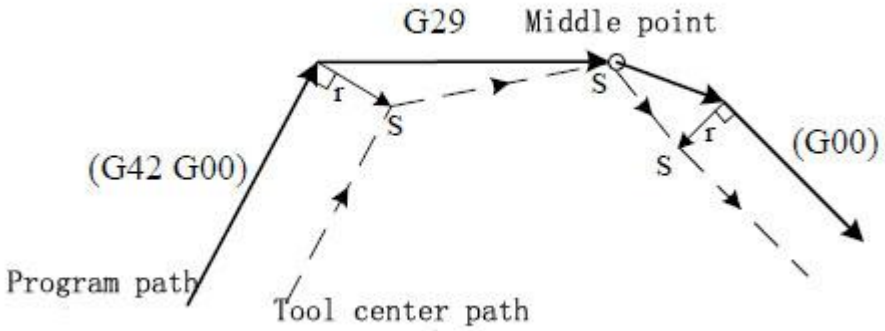
Picture 3-5-3-11

b) G29 from reference point return automaticlly.

In compensate mode, command G29, commpensate will cancel in middle point, compensate mode will recover in next program segment.



Picture3-5-3-12



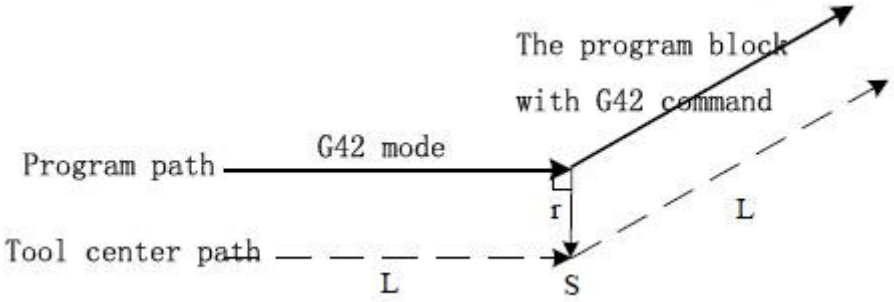
Picture3-5-3-13

7. Compensate mode's tool radius G code

In compensate mode,pointed tool radius G code (G41, G42), relative to the direction of movement can form a vector at right angles to the former procedure, has nothing to do with the processing of the inside and outside.But if in arc command pointed this G code,or it cannot get the correct arc.

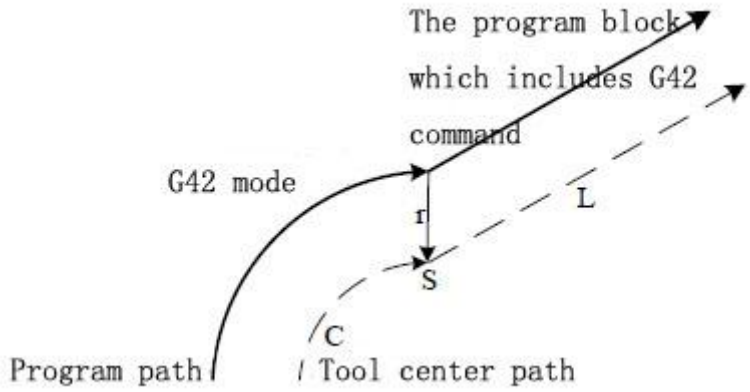
When use tool radius compensate G (G41, G42) change compensate direction, please reference (5).

Straight line - Straight line



Picture3-5-3-14

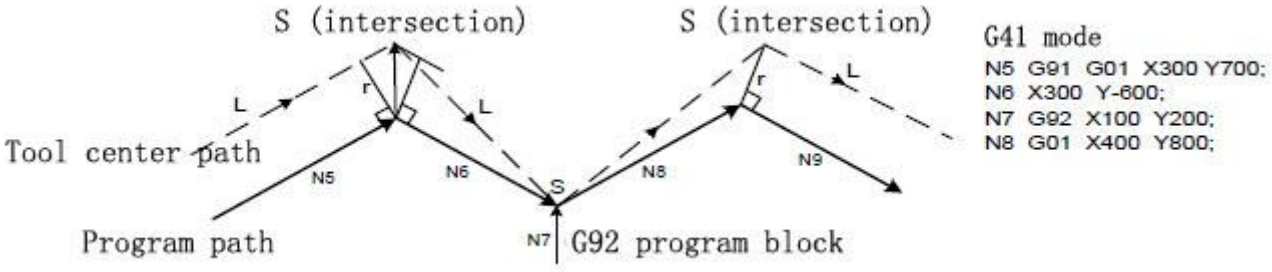
Circle arc - Straight line



Picture3-5-3-15

8. Command of Cancel Compensate Vector Temporarily

In compensate mode, if pointed G92 code(Absolute coordinate program), compensate vector will cancel temporarily.After that,compensate vector will recover automaticly.At the same time, different from compemsate cancel mode, tool move directly from intersection to compensate vector cancel's command point.When compensate mode recover, tool move back to intersection.



Picture 3-5-3-16

9. Tool No Move Program Segment

In the following program segment, tool have no movement. In these program segments,even in the tool radius compensate mode no move.

(1) M05; M code output

(2) S21; S code output

(3) G04 X10; Pause

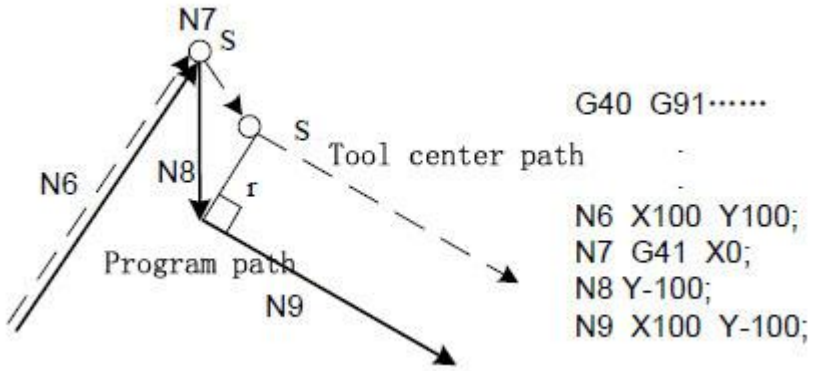
(4) (G17) Z100; Compensate flat have no move command

(5) G90; Only G code

(6) G01 G91 X0; Move vector is zero

a) Command in the beginning of compensate

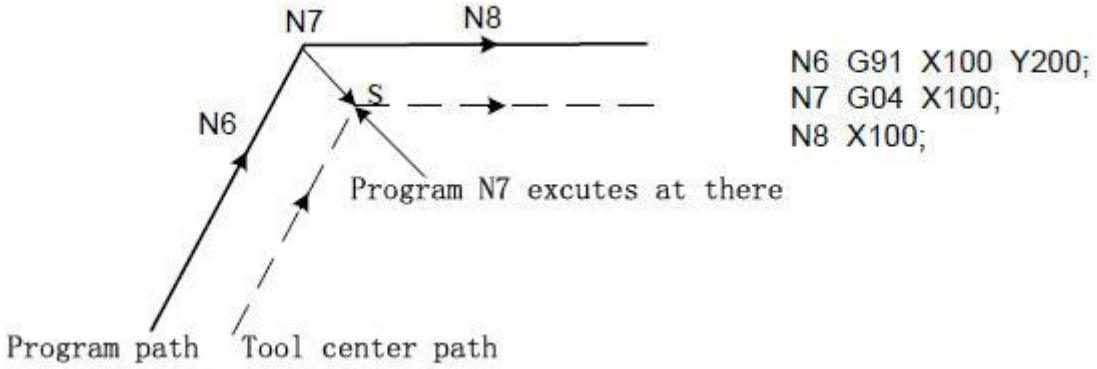
If tool start program segment have no move, system will have tool start action in next move command.



Picture 3-5-3-17

b) In Compensate Mode

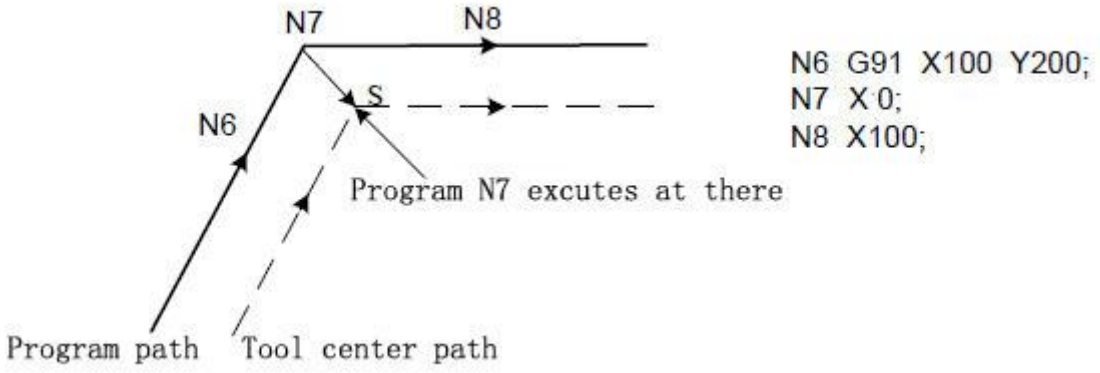
In compensate mode only command one no tool movement program segment, vector and tool central route as same as no command program segment. (Reference item (3) command mode), no tool movement program segment in single program segment stop point execute.



Picture3-5

-3-18

But, when program segment move vector is zero, even only point one program segment, tool move is the same as two or two more without tool movement's program segments.

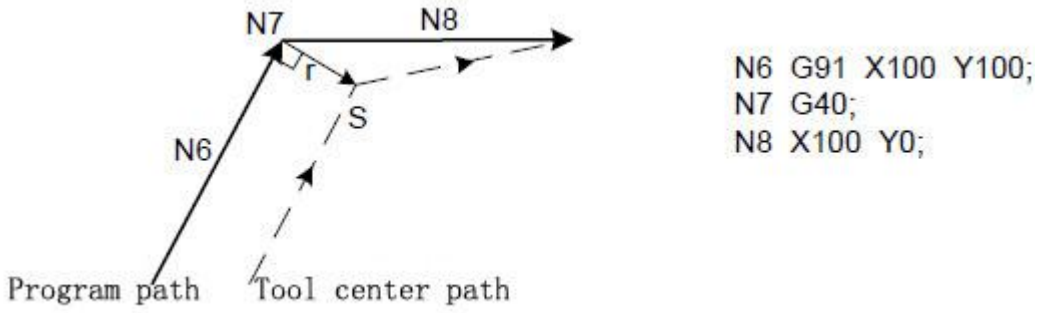


Picture3-5-3-19

Remark: Above program segment working untder G1G41 situation, G0 route is not tally with picture.

c) With Compensate Cancel Together Command

When with compensate cancel together command's program segment have no tool movement, will form length as compensate vector, direction perpendicular to the before program segment move direction's vector, this vector cancel in next move command.



Picture3-5-3-20

10. Corner Movement

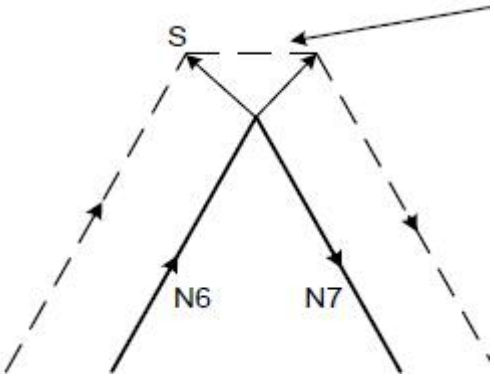
If in program segment ending, come out two more vectors, tool from one vector linear move to another vector. This move called corner movement.

If$\Delta VX \leq \Delta V$ limit and $\Delta VY \leq \Delta V$ limit, latter vectors are ignored.

If these vector are not the same, come out a movement, which is along the corner.The move belong to the latter program segment.

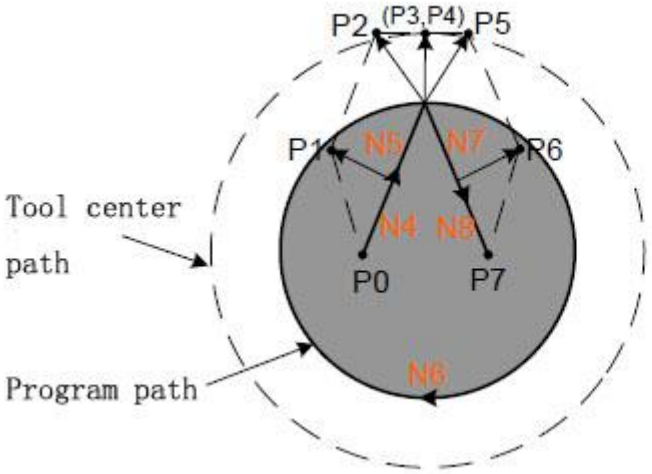
Thess movements are belong to program block N6, so feeding ratio is equal to the feeding ratio of program block N6. If program N6 is G00 mode, tool moves in rapidly feeding ratio; if program block N6 is G01, G02, G03

modes, tool moves in cutting feed ratio.



Picture3-5-3-21

But, if in the next program segment's route over half-round, no execute above function. Reasons as below:




```
N4 G41 G91 X150 Y200;
N5 X150 Y200;
N6 G02 J-600;
N7 G01 X150 Y-200;
N8 G40 X150 Y-200;
```

Picture 3-5-3-22

If vector is no ignored, tool route as below:

P0-P1-P2-P3 (arc)-P4-P5-P6-P7

But, if the distance between P2 and P3 is ignored, then P3 is also ignored. Tool route as below:

P0-P1-P2-P4-P6-P7 program segment N6's arc cutting ignore.

11. Interference Checking

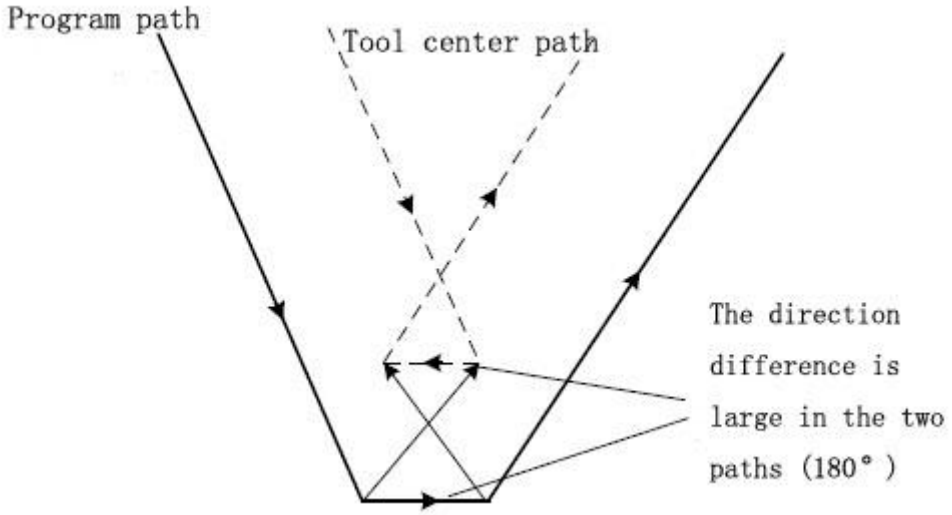
Excessive cutting tool called "interference". Interference could checking tool excessive cutting in advance.Such as syntax checking after loader interference is detected, the system will alarm prompt. Position Parameter N0:19#2 setting radius compensate whether interference checking or not.

Interference checking basic terms, tool central route

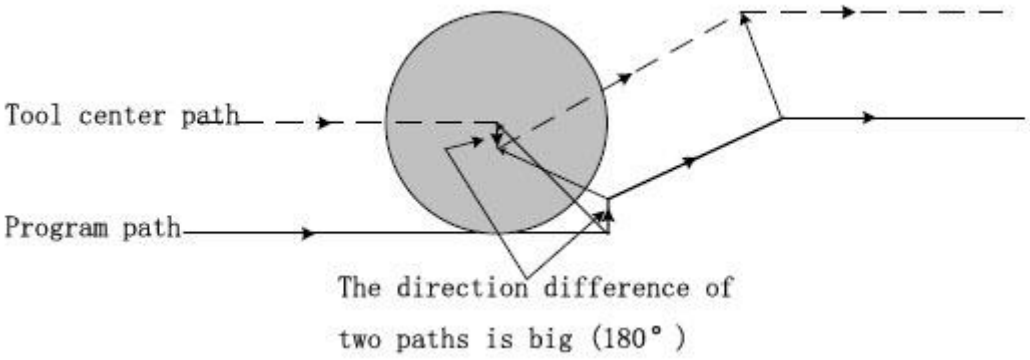
(1) Tool route direction is different from program route direction. (Path between the included Angle between 90

degrees to 270 degrees）

(2) Arc processing, except above terms, angle between the tool central route's start point and destination point have big difference to the angle between program route start point and destination. (Above 180 degrees)



Picture3-5-3-23



Picture3-5-3-24

12. Manual Operation

Tool nose radius compensate's manual operation, please reference to manual operation. If tool length compensate in the tool radius compensate execute, tool radius'compensate vector as been change.

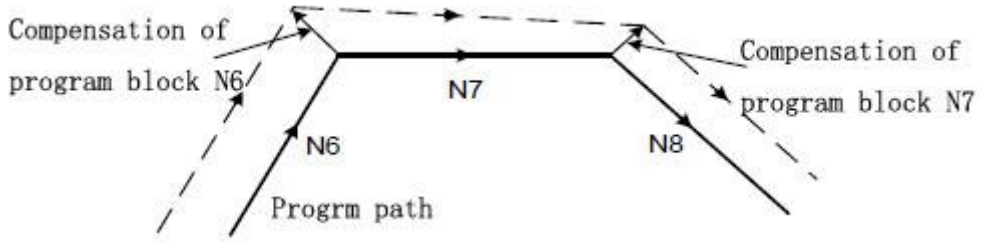
13. Compensate Attentions in Usual

a) Command compensate vector

Compensate vector use D code point compensate number. Once pointed, D code keep effective until another D code pointed, or cancel compensate.D code use for tool radius compensate point compensate vector, also use tool offset's offset value.

b) Change Compensate Vector

Usually, change tool, compensate vector must change in cancel mode. If in compensate mode change compensate vector, in program segment ending point calculate new compensate vector.

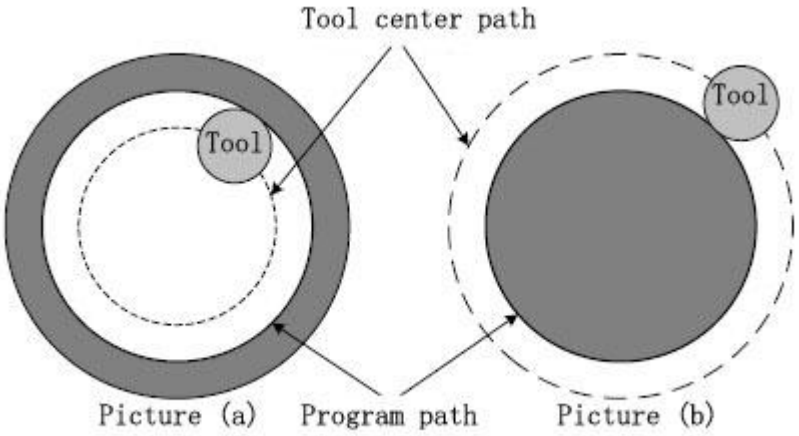


Picture3-5-3-25

c) Compensate vector's positive negitive and tool central route

If compensate vector is negitive (-), program G41 and G42 code change to each other.If tool central move along the outside of work piece, it will move along inside.

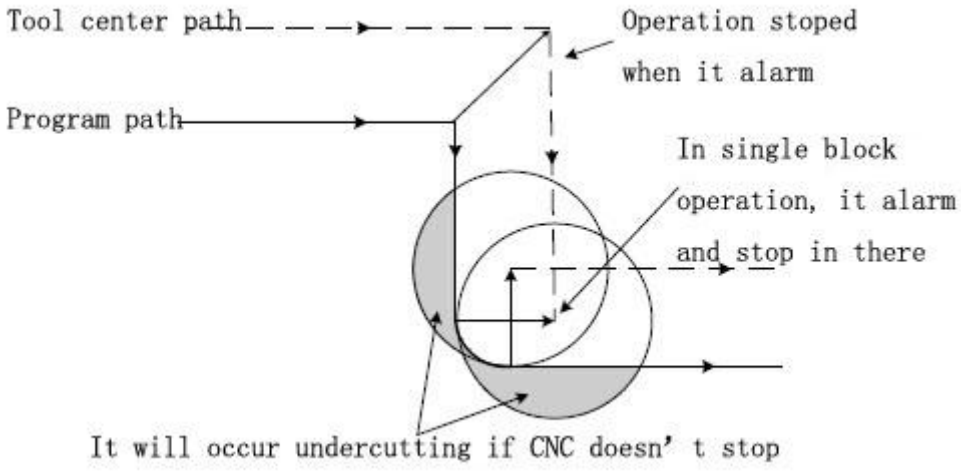
As below example show, usually, process program compensate is (+). When tool route as picture (a) programming, if compensate vector is (-), tool centre move as picture (b).



Picture3-5-3-26

d) Use tool radius compensate over-cutting

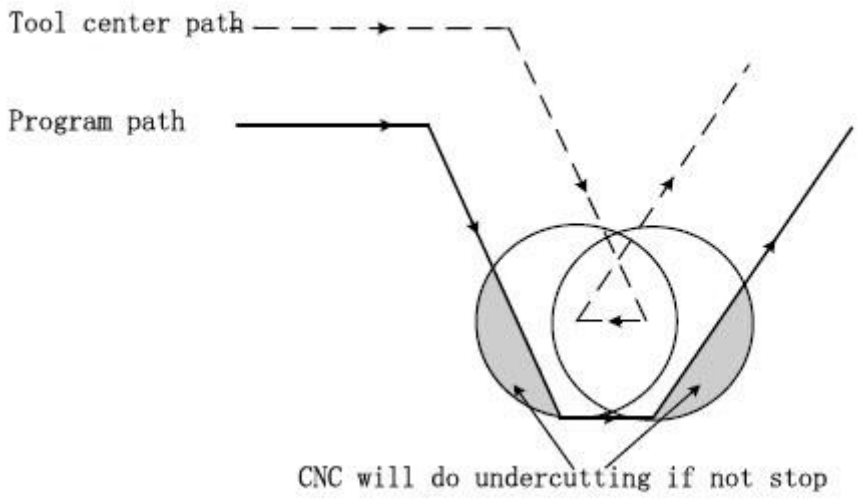
(1) Use arc, which smaller than tool radius inside processingwhen corner radius smaller than tool radius, because tool inside compensate will come out over-cutting, it will be alarm.Because tool use single segment execute mode, when stop, will come out over-cutting.



Picture3-5-3-27

(2) When using a smaller than tool radius of cutting groove machining

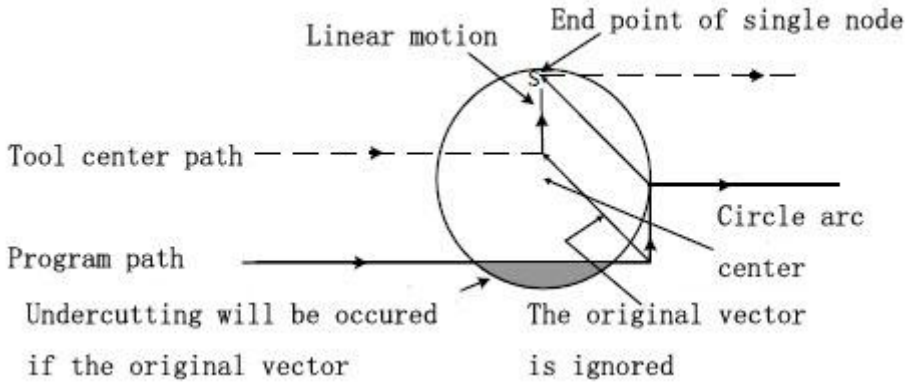
Use cutting groove which is smaller than tool radius processing, because of tool radius compensate force tool central route to program route countermovement, it will be alarm.



Picture3-5-3-28

(3) When the poor lag which smaller than tool radius processing

If in the program, there is segment difference smaller than tool radius, use arc process command this segment difference processing, normal compensate tool central route change the counter-direction of program direction. Single period of execution stops here. If not in single period of processing mode, automatic operation will continue. If the section is a straight line, Won't produce alarm. Make correct cutting, but there is no cutting part will remain.



Picture3-5-3-29

Start cutting tool radius compensation and the Z axis movement

Usually at the beginning of machining, tool radius compensation effectively, after cutting tool moving along the Z axis is apart from the workpiece. The above situation, if you want to move along the Z axis is divided into fast feed and cutting feed, please refer to the following two procedures:

If program block N3 (Z axis movement instruction)

Separate as below:

N1 G91G00X500Y500H01;

N3 Z－250;

N5 G01Z－50F1;

N6 Y100F2;

N1 G91 G0 X500 Y500 H01;

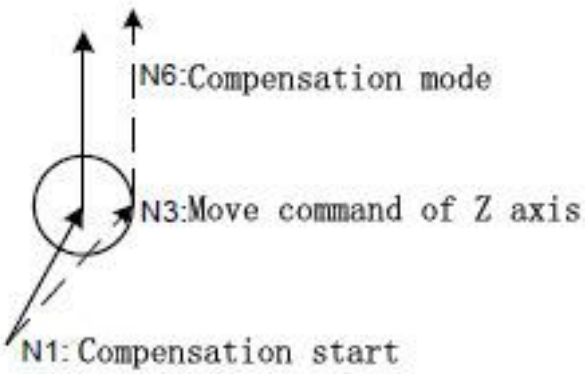
N3 G01 Z-300 F1;

N6 Y100 F2;

When performing N3 N6 into the buffer,

Also use the relationship between them,

such as the proper compensation right



Picture3-5-3-30

3.5.4 Corner offset circular arc interpolation (G39)

Instruction format: G39 or

I_ J_

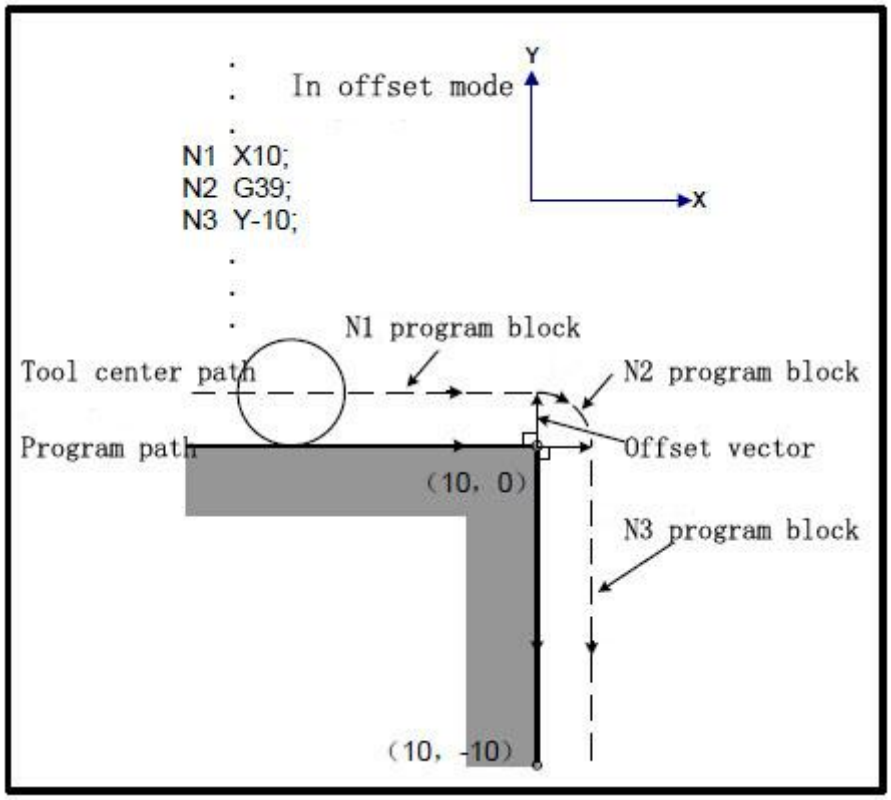
G39 I_ K_

J_ K_

Function: During the period of tool radius compensation, in a biased way instruction G39, can offset arc instruction corner compensation, corner radius compensation is equal to the offset value. By a reference NO: 19 # 7 is set in the radius compensation, the corner arc is valid.

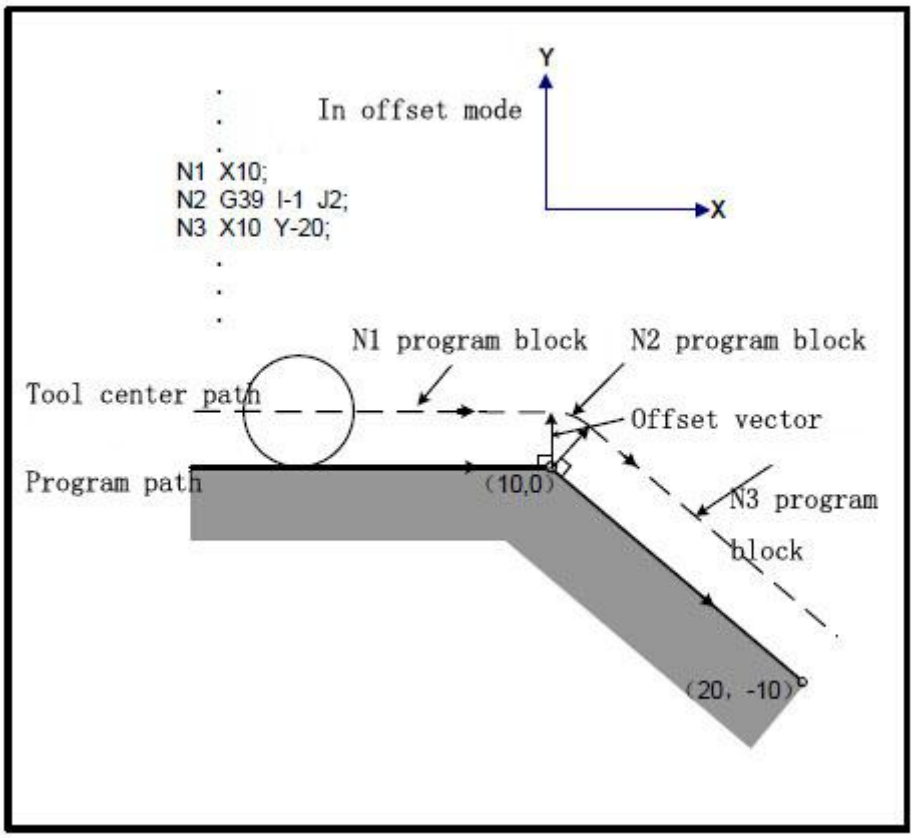
Explanation:

1. Specified G39 instruction, can perform its radius compensation value is equal to the corner of the circular arc interpolation.
2. G41 or G42 arc in front of the directive is clockwise or counterclockwise, G39 for non-modal G code.
3. With G39 command (not I, J and K) programming, the corner form the arc, so the arc at the end of the vector is perpendicular to the beginning of the next paragraph. As shown in the figure below:



Picture 3-5-4-1 G39 withno I, J and K

4. When G39 and I, J and K is instruction, the formation of circular arc at the corner, so in the arc end vector is perpendicular to the determined by the I, J and K value vector, as the following picture shows:



Picture 3-5-4-2 G39 with I, J and K

3.5.5 Cutter compensation value, compensation with input program (G10)

Instruction format:G10 L10 P_ R_; Geometric compensation value of H code

G10 L12 P_ R_; Geometric compensation value of D code

G10 L11 P_ R_; Wear compensation value of H code

G10 L13 P_ R_; Wear compensation value of D code

P: Cutter compensation

R: Cutter compensation value of the absolute value instruction

(G90) mode

Tool offset value of increment value instruction (G91) mode,

this value plus tool compensation value.

Explanation:

Cutting tool compensation value effective input range:

Geometric compensation: Metric input±999.999mm; British input±99.9999inch

Wear compensation: Metric input±999.999mm; British input±99.9999inch

Note: 1. When switch Metric system and British system, NO: 3 # 0 setting tool offset is automatic transformation;

2. The maximum wear compensation is conditioned by several P331.

3.6 Feeding G code

3.6.1 Feeding mode G64/G61/G63

Instruction format:

Orientation mode G61

Tapping mode G63

Cutting mode G64

Function:

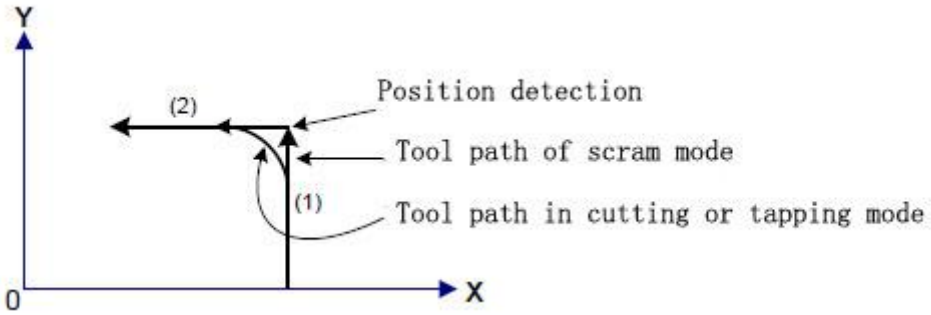
Orientation mode G61: this function is valid once specified until G62, G63 and G64 specified, Cutting tool at the end of the procedures section deceleration executive in place to check, and then execute the next program block.

Tapping mode G63: this function is valid once specified until G61, G62 and G64 specified. Cutting tools don't slow down at the end of the block and executing the next program. Feed speed ratio and feed suspended are invalid when specify G63.

Cutting mode G64: this function is valid once specified until G61, G62 and G63 specified, Cutting tools don't slow down at the end of the block and executing the next program.

Explanation:：

1. None parameter format;
2. G64 is system default feeding mode, the end point of program block wouldn’t decelerate, and execute the next block directly.
3. The purpose of arriving detection in orientation mode is check whether the servo motor is arrived at the location specified range.
4. In orientation mode, the paths of tool movement are different in cutting and tapping mode.



Picture 3-6-1-1 Tool path from program block 1 to 2

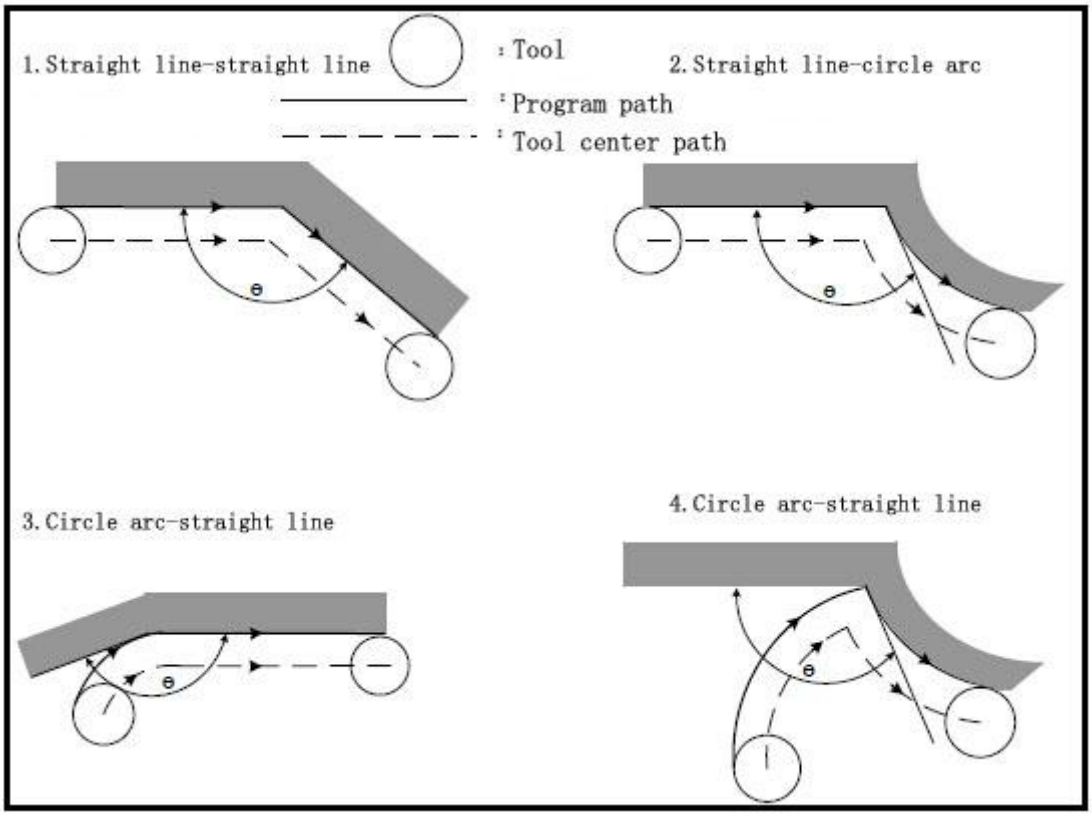
3.6.2 Automatic corner ratio (G62)

Instruction format: G62

Function: Automatic corner ration is G62, the function is valid once specified until G61, G63 and G64 specified. During the period of tool radius compensation, for cutting feed speed ratio when the cutting tool moves along the inner corner, To inhibit the cutting quantity of per unit time, then can a good surface machining precision.

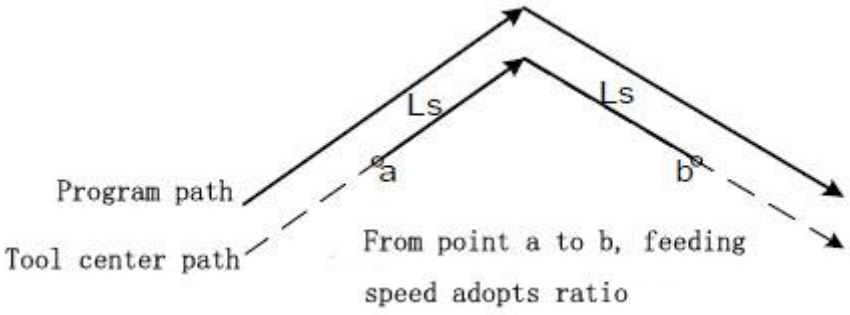
Explanation:

1. When the tool radius compensation, when Knives move in the circle and corner, Automatic deceleration to minimize tool on the negative, then can a good surface machining precision.
2. Specify G62 and tool radius compensation function and processing in the corner, On both ends of the corner of automatically fix the inlet to speed, there are four in the corner as shown in FIG. 3-6-2. In the picture: $2° \leq \theta \leq \theta p \leq 178°$. θp is set by P160.



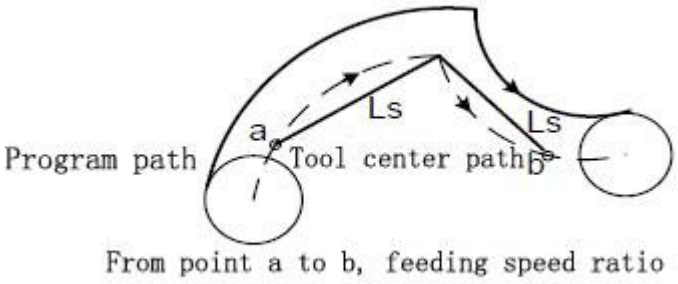
Picture 3-6-2-1

3. When the corner was identified as inside corner, before and after the corner, Perform feed speed ratio. Execution of feed ratio of the distance is Ls and Le, Ls and Le are the distance from tool center to corner. As picture 3-6-2-2 shows, Ls+Le ≤ 2mm.



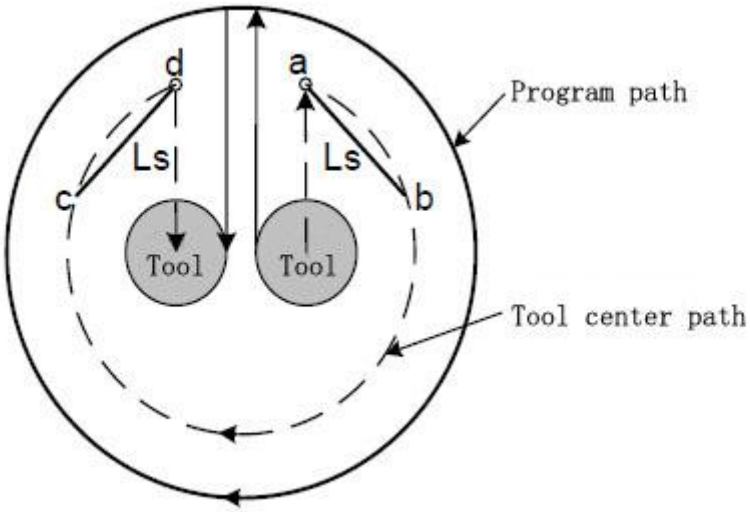
Picture 3-6-2-2 Straight line–Straight line

4. When programming track consists of two circular arcs, if the start and end points were in the same quadrant or in adjacent quadrant, feed speed Shi ratio. As picture 3-6-2-3 shows.



Picture 3-6-2-3

5. Consider a program has a line into the arc, also has a circular arc to a straight line, as shown in picture 3-6-2-4, from point a to point b and point from c to d Point of feed speed ratio.



Picture 3-6-2-4

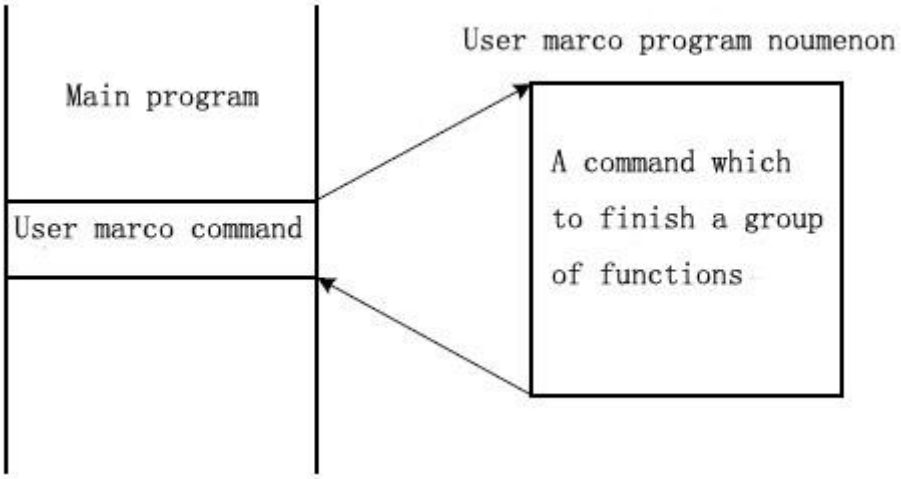
Limitation:

1. During acceleration/deceleration before interpolation, in the corner is invalid.
2. If you have any knife block or corner before including G41 or G42 block, after the inside corner ratio is invalid.
3. If the offset is zero, does not perform within the corner

3.7 Marco G code

3.7.1 User Marco program

By a set of instructions to realize some functions like a subroutine in advance in the memory, With an instruction on behalf of these functions. As long as write the representative instruction in the program to achieve those functions. This is a set of instructions called “user macro program ontology”. User macro program ontology is also sometimes referred to as "macro program, user macro instructions, also known as macro program calls.



Picture 3-7-1-1

3.7.2 Marco variable

In the user macro program, you can use the general CNC instruction, also can use variables, operation and transfer instruction.

Start in user macro program and end in M99.

```
O0066;                    Program number
G65 H01 ……;               Operation command
G90 G00 X#101 ……;         Use CNC command of variable
……
……
……
G65 H82 ……;               Transfer command
……
……
M99;          User marco program noumenon finished
```

Picture 3-7-1-2

1. The usage of variable

User macro program with variable can command parameter values of ontology. A variable's value can be made by the main program assignment or set through the LCD/MDI, or in ontology, perform user macro program is assigned to the computed value.

Multiple variables can be used, these variables with a number to distinguish.

(1) Representations of variables

Use # subsequent variable to represent the variable, format as follows:

#i(i = 1, 2, 3, 4 ……)

(For example) #5, #109, #1005

(2) Variable references

After the numerical with variable can be displacement parameter values

(For example)

F#103 When #103=15, it is the same with F15 instruction;

G#130 When #130=3, it is the same with G3.

Note: 1. Digital O and N (program number and the serial number) cannot reference variable. O#100 and N#120 can't be programmable.

2. If more than the parameter value of maximum instruction, it can't be used. M#30 exceeds the max instruction value when #30 = 120

3. Display and set a variable's value: A variable's value can be displayed on the LCD screen, Can also use MDI way to variable value.

2. The types of variables

Depending on the variable number, divided into empty variables, local variables, public variables and system variables, their use and nature is different.

(1) Empty variable #0: (This variable is always empty; no values can give the variable)

(2) Local variable #1 ~ #50: a local variable can only be used to store data in the macro program, for example: Operation result. Automatic reset when cut off the power supply and reset or the end of the program (execute M30, M02). Call Marco program, Independent variable to a local variable assignment.

(3) Public variable #100 ~ #199, #500 ~ #999:

Public variables in the main program and by the main program calls each user macro program is common. Namely a variable used in the user macro program using # # I and other macros that will be the same. So a macro program in computing results public variable # I can be used for other macro program.

Form 3-7-2-1

Variable number	Varible type	Function
#100 ~ #199	Public variable	It clear when cut off power, all reset to be "empty" when power on

#500 ~ #999		Data store in document and wouldn't lose even power off

(4) System variables: System variable is used to read and write CNC runtime changes of various kinds of data.

1) Interface input signal #1000 --- #1047 (Read PLC input signal positional)

2) Interface input signal #1100 --- #1147 (Write PMC input signal positional)

3) Tool length compensation value #1500 --- #1755 (Can be read)

4) The length of the wear compensation value #1800 --- #2055 (Can be read)

5) Tool radius compensation value #2100 --- #2355 (Can be read)

6) Wear radius compensation value #2400 --- #2655 (Can be read)

7) Knife library data table #2700 --- #2955 (Read only, cannot be written)

8) Alarm #3000

9) User tables #3500 --- #3755 (Read only, cannot be written)

10) Modal information #4000 --- #4030 (Read only, cannot be written)

11) Location information #5001 --- #5030 (Read only, cannot be written)

12) The workpiece zero offset #5201 --- #5235 (Read and write are valid)

13) Additional workpiece coordinate system #7001 --- #7250 (Read and write are valid)

3. System variables in detail

(1) Modal information

Form 3-7-2-2

Variable number	Function	Group number
#4000	G10, G11	00
#4001	G00, G01, G02, G03	01
#4002	G17, G18, G19	02
#4003	G90, G91	03
#4004	G94, G95	04
#4005	G54, G55, G56, G57, G58, G59	05
#4006	G20, G21	06
#4007	G40, G41, G42	07
#4008	G43, G44, G49	08

#4009	G73, G74, G76, G80, G81, G82, G83, G84, G85, G86, G87, G88, G89	09
#4010	G98, G99	10
#4011	G15, G16	11
#4012	G50, G51	12
#4013	G68, G69	13
#4014	G61, G62, G63, G64	14
#4015	G96, G97	15
#4016	To extend	16
#4017	To extend	17
#4018	To extend	18
#4019	To extend	19
#4020	To extend	20
#4021	To extend	21
#4022	D	22
#4023	H	23
#4024	F	24
#4025	M	25
#4026	S	26
#4027	T	27
#4028	N	28
#4029	O	29
#4030	P (Additional workpiece coordinate system chosen at now)	30

Note: 1. P code to the currently selected additional workpiece coordinate system。

2. When performing the G# 4002, in the value of the # 4002 is 17, 18, or 19

3. Modal information can't write, can only be read.

(2) Current position information

Form 3-7-2-3

Variable	Position information	Relative	Read operation	Tool compensation

number		coordinate system	when moves	value
#5001	X axis program block end position (ABSIO)	Workpiece coordinate system	Yes	Tool nose doesn't be taken in to consideration (program command position)
#5002	Y axis program block end position (ABSIO)			
#5003	Z axis program block end position (ABSIO)			
#5004	The 4th axis program block end position (ABSIO)			
#5006	X axis program block end position (ABSMT)	Machine coordinate system	No	
#5007	Y axis program block end position (ABSMT)			
#5008	Z axis program block end position (ABSMT)			
#5009	The 4th axis program block end position (ABSMT)			
#5011	X axis program block end position (ABSOT)	Workpiece coordinate system		
#5012	Y axis program block end position (ABSOT)			
#5013	Z axis program block end position (ABSOT)			
#5014	The 4th axis program block end position (ABSOT)			
#5016	X axis program block end position (ABSKP)		Yes	
#5017	Y axis program block end position (ABSKP)			

#5018	Z axis program block end position (ABSKP)			
#5019	The 4th axis program block end position (ABSKP)			
#5021	X axis tool length compensation value			
#5022	Y axis tool length compensation value		No	
#5023	Z axis tool length compensation value			
#5024	The 4th axis tool length compensation value			
#5026	X axis servo position compensation			
#5027	Y axis servo position compensation			
#5028	Z axis servo position compensation			
#5029	The 4th axis servo position compensation			

Note: 1. ABSIO: in workpiece coordinate system, the previous program block end point coordinates value;

2. ABSMT: in machine coordinate system, current machine coordinate position;

3. ABSOT: in workpiece coordinate system, current coordinate position;

4. ABSKP: in workpiece coordinate system, the skip information valid position in G31 program block.

(3) The workpiece zero offset and additional zero offset

Form 3-7-2-4

Variable number	Function
#5201	The 1st axis external workpiece zero point offset value
……	……
#5204	The 4th axis external workpiece zero point offset value
#5206	The 1st axis G54 workpiece zero point offset value
……	……
#5209	The 4th axis G54 workpiece zero point offset value

#5211	The 1st axis G55 workpiece zero point offset value
……	……
#5214	The 4th axis G55 workpiece zero point offset value
#5216	The 1st axis G56 workpiece zero point offset value
……	……
#5219	The 4th axis G56 workpiece zero point offset value
#5221	The 1st axis G57 workpiece zero point offset value
……	……
#5224	The 4th axis G57 workpiece zero point offset value
#5226	The 1st axis G58 workpiece zero point offset value
……	……
#5229	The 4th axis G58 workpiece zero point offset value
#5231	The 1st axis G59 workpiece zero point offset value
……	……
#5234	The 4th axis G59 workpiece zero point offset value
#7001	The 1st axis G54 P1 workpiece zero point offset value
……	……
#7004	The 4th axis G54 P1 workpiece zero point offset value
#7006	The 1st axis G54 P2 workpiece zero point offset value
……	……
#7009	The 4th axis G54 P2 workpiece zero point offset value
#7246	The 1st axis G54 P50 workpiece zero point offset value
……	……
#7249	The 4th axis G54 P50 workpiece zero point offset value

4. Local variable

Address a corresponding relationship with the local variables:

Form 3-7-2-5

Independent variable address	Local variable No.	Independent variable address	Local variable No.
A	#1	Q	#17

B	#2	R	#18
C	#3	S	#19
I	#4	T	#20
J	#5	U	#21
K	#6	V	#22
D	#7	W	#23
E	#8	X	#24
F	#9	Y	#25
M	#13	Z	#26

Note: 1. Assignment in English letters after add value, the other 20 English letters can give independent variable assignment except G, L, O, N, H and P, assigned to once with each letter, from A-B-C-D… to X - Y - Z, assign a value not in alphabetical order, no assignment of address can be omitted.

2. Before using any of the independent variables must specify G65.

5. Matters needing attention about user macro program ontology

(1) The method that input by key

Press # after parameter G, X, Y, Z, R, I, J, K, F, H, M, S, T, P and Q, # will be input in.

(2) In MDI state, Instructions operation and movement instruction are valid.

(3) H, P, Q and R of operation, movement instruction before or after G65 are regards as parameter of G65 instruction.

H02 G65 P#100 Q#101 R#102; Correct

N100 G65 H01 P#100 Q10; Correct

(4) The range of variable value: -9999.9999 ~ 9999.9999

(5) The operation result of variable value can be decimals, the precision is 0.0001. Except H11 (or operation), H12 (and operation), H13 (nor operation), H23 (complementation)

For example:

#100 = 35, #101 = 10, #102 = 5

#110 = #100÷#101 (= 3.5)

#111 = #110×#102 (= 17.5)

#120 = #100×#102 (= 175)

#121 = #120÷#101 (= 17.5)

(6) Operation, the transfer instruction execution time, Due to the different conditions, general average may be

considered for the 10 milliseconds.

3.7.3 User Marco program call

When execute instruction G65, in order to address the P specified user macro program is invoked, data by arguments passed to the user macro program in the body.

Instruction format as below:

G65 P □□□□□ L □□□□ <specify independent variables>;

□□□□□ L □□□□: The called macro program of the ontology;

<Specify independent variables>: number of calling;

After G65, address P specified by user macro program application number, Marco program call numbers specified by L, transfer data with the independent variable to the macro program.

When request repeat, specify the repeat time from 1 to 9999, default time is 1 when omit L.

Using arguments specified, its value assigned to the corresponding local variables

Note:

1. It alarm (PS, 078) when can't retrieve address specified subroutine number P;

2. 90000~00000 sub-program are system reserved program, when user call this sub-program, but the cursor is still at M98 instruction block, program interface still display main-program content. (Sub-program content can be displayed by amend bit parameter No: 29#5);

3. Marco program can't be called in DNC mode.

3.7.4 Operation and transfer instruction

1. Normal format:

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

m: 01 ~ 99 is operation instructions or transfer function.

#i: Variable name which stores in operation results.

#j: Variable 1 of running calculates. It can be constant. Constant express directly, without #.

#k: Variable 2 of running calculates. It can also be constant.

Meaning: #i = #j ○ #k (Calculation symbol, specified by Hm)

(For example)

P#100 Q#101 R#102……#100 = #101 ○ #102;

P#100 Q#101 R15 ……#100 = #101 ○ 15;

P#100 Q-100 R#102……#100 = -100 ○ #10.

H code which specified by G65, there is no any effects to offset value.

G code	H code	Function	Definition
G65	H01	Assignment	#i = #j
G65	H02	Plus	#i = #j + #k
G65	H03	Minus	#i = #j - #k
G65	H04	Multiple	#i = #j * #k
G65	H05	Divide	#i = #j ÷ #k
G65	H11	Logic add (OR)	#i = #j or #k
G65	H12	Logic multiple (AND)	#i = #j and #k
G65	H13	XOR	#i = #j XOR #k
G65	H21	Square root	#i = √ #j
G65	H22	Absolute value	#i = |#j|
G65	H23	MOD	#i = #j-trunc (#j÷#k)×#k
G65	H26	Composite multiplication or divide	#i = (#i×#j) ÷ #k
G65	H27	Composite square root	#j2#i =√ +#k2
G65	H31	Sine	#i = #j * SIN(#k)
G65	H32	Cosine	#i = #j * COS(#k)
G65	H33	Tangent	#i = #j * TAN(#k)
G65	H34	Arc tangent	#i = ATAN (#j/#k)
G65	H80	Non-conditional jump	Turn to N
G65	H81	Conditional jump 1	IF #j < #k, GOTO N
G65	H82	Conditional jump 2	IF #j < #k, GOTO N
G65	H83	Conditional jump 3	IF #j < #k, GOTO N
G65	H84	Conditional jump 4	IF #j < #k, GOTO N
G65	H85	Conditional jump 5	IF #j < #k, GOTO N
G65	H86	Conditional jump 6	IF #j < #k, GOTO N

Picture 3-7-4-1

2. Operational orders:

1) Variable assignment: #I = #J

G65 H01 P#I Q#J;

(For example) G65 H01 P#101 Q105; (#101 = 1005)

G65 H01 P#101 Q#110; (#101 = #110)

G65 H01 P#101 Q-#102; (#101 = -#102)

2) Additive operation: #I = #J + #K

G65 H02 P#I Q#J R#K;

(For example) G65 H02 P#101 Q#102 R15; (#101 = #102 + 15)

3) Subtraction operation: #I = #J – K

G65 H03 P#I Q#J R#K;

(For example) G65 H03 P#101 Q#102 R#103; (#101 = #102 – 103)

4) Multiply operation: #I=#J×#K

G65 H04 P#I Q#J R#K;

(For example) G65 H04 P#101 Q#102 R#103; (#101 = #102×#103)

5) Division operation:#I=#J÷#K

G65 H05 P#I Q#J R#K;

(For example) G65 H05 P#101 Q#102 R#103; (#101 = #102÷#103)

6) Logic add (OR):#I=#J.OR. # K

G65 H11 P#I Q#J R#K;

(For example) G65 H11 P#101 Q#102 R#103: (#101 = #102.OR. #103)

7) Logic multiplication (AND):#I=#J. AND. #K

G65 H12 P#I Q#J R#K;

(For example) G65 H12 P# 101 Q#102 R#103; (#101 = #102.AND.#103)

8) XOR: #I=#J. XOR. # K

G65 H13 P#I Q#J R#K;

(For example) G65 H13 P#101 Q#102 R#103; (#101 = #102. XOR. #103)

9) Square root: #I = √ #j

G65 H21 P#I Q#J;

10) Absolute value: #I= |#J|

G65 H22 P#I Q#J;

(For example) G65 H22 P#101 Q#102; (#201 = |#102|)

11) MOD: #I=#J-TRUNC (#J/#K) × #K, TRUNC: leave the decimal part

G65 H23 P#I Q#J R#K;

(For example) G65 H23 P#101 Q#102 R#103; (#101 = #102-TRUNC (#102/#103) × #103)

12) Composite multiplication or divide: #I= (#I×#J) ÷#K

G65 H26 P#I Q#J R# k;

(For example) G65 H26 P#101 Q#102 R#103; (#101 = (#101×#102) ÷ #103)

13) Composite square root: #I= √ #j2+# k2

G65 H27 P#I Q#J R#k;

(For example) G65 H27 P#101 Q#102 R#103; (#101 = √ #1022+#1032)

14) Sine: #I = #J•SIN (#K) (Unit: degree)

G65 H31 P#I Q#J R#K;

(For example) G65 H31 P#101 Q#102 R#103; (#101 = #102•SIN (#103))

15) Cosine: #I=#J•COS (#K) (Unit: degree)

G65 H32 P#I Q#J R#k;

(For example) G65 H32 P#101 Q#102 R#103; (#101 =#102•COS (#103))

16) Tangent: #I=#J•TAM (#K) (Unit: degree)

G65 H33 P#I Q#J R#K;

(For example) G65 H33 P#101 Q#102 R#103; (#101 = #102•TAM (#103))

17) Arc Tangent: #I=ATAN (#J/#K) (Unit: degree)

G65 H34 P#I Q#J R#k;

(For example) G65 H34 P#101 Q#102 R#103; (#101 =ATAN (#102/#103))

Note: 1. Unit of angle variable is degree.

2. In each operation, when Q and R didn't be specified, their values are zero.

3. Trunc: truncating operation, give up decimal part.

3. Transfer order

1) Unconditional transfer

G65 H80 Pn; n: sequence number

(For example) G65 H80 P120; (Turn to N120 program block)

2) Conditional transfer1#J.EQ. #K (=)

G65 H81 Pn Q#J R#K; n: Sequence number

(For example) G65 H81 P1000 Q#101 R#102;

When #101= #102, turn to N1000 program block, when #101 ≠ #102, program execute in sequence.

3) Conditional transfer 2#J.NE. #K (≠)

G65 H82 Pn Q#J R#K; n: Sequence number

(For example) G65 H82 P1000 Q#101 R#102;

When #101≠ #102, turn to N1000 program block, when #101 = #102, program execute in sequence.

4) Conditional transfer 3#J.GT. #K (>)

G65 H83 Pn Q#J R#K; n: Sequence number

(For example) G65 H83 P1000 Q#101 R#102;

When #101> #102, turn to N1000 program block, when #101 ≤ #102, program execute in sequence.

5) Conditional transfer 4#J.LT. #K (<)

G65 H84 Pn Q#J R#K; n: Sequence number

(For example) G65 H84 P1000 Q#101 R#102；

When #101< #102, turn to N1000 program block, when #101 ≥#102, program execute in sequence.

6) Conditional transfer 5#J.GE. #K (≥)

G65 H85 Pn Q#J R#K; n: Sequence number

(For example) G65 H85 P1000 Q#101 R#102;

When #101≥#102, turn to N1000 program block, when #101<#102, program execute in sequence.

7) Conditional transfer 6 #J. LE. #K (≤)

G65 H86 Pn Q#J R#K; n: Sequence number

(For example) G65 H86 P1000 Q#101 R#102;

When #101≤#102, turn to N1000 program block, when #101>#102, program execute in sequence.

Note: Variable can be used to specify sequence number. For example: G65 H81 P#100 Q#101 R#102; when meet the conditional, program moves to the sequence number of program block which specified by #10

4. Logical AND, logical OR and logical NOT orders

For example:

G65 H01 P#101 Q3;

G65 H01 P#102 Q5;

G65 H11 P#100 Q#101 Q#102;

5 express as binary system is 101, 3 express as 011, calculate result is #100=7;

G65 H12 P#100 Q#101 Q#102;

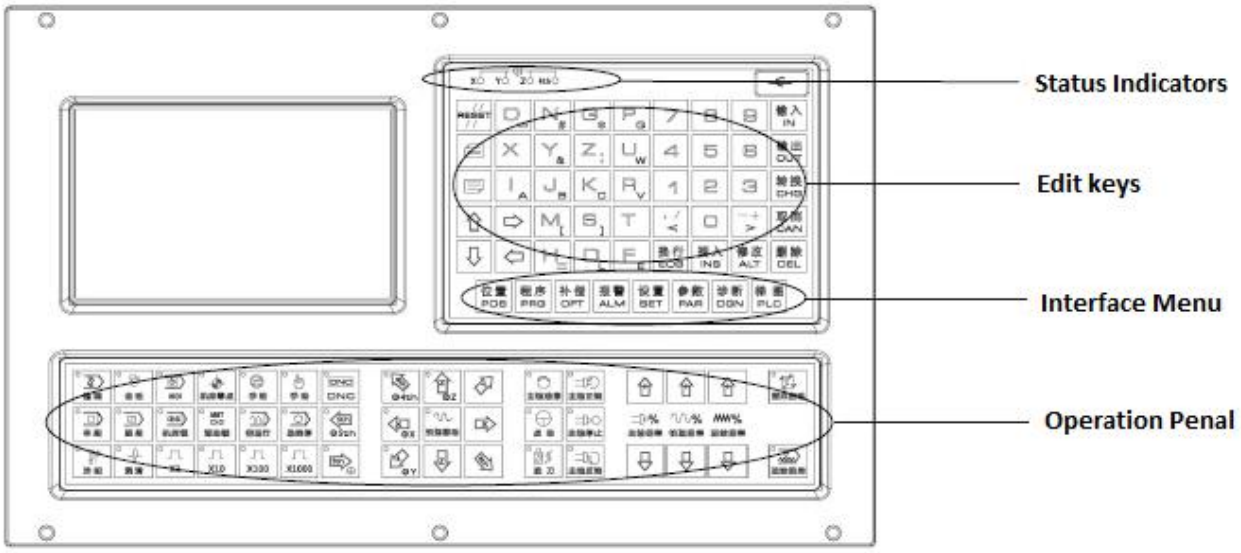
5 express as binary system is 101, 3 express as 011, calculate result is #100=1;

Volume 2 Operation explanation

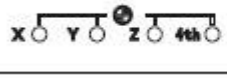
Chapter 1 Operation mode and display interface

1.1 Panel divide

ECN*000M series adopts integrated operation panel, panel divides as below:



1.1.1 State instruction

	Axis back to zero finished indicator light

1.1.2 Display menu

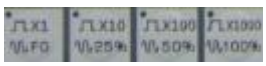
Menu key	Remark
位置 POS	Enter the position interface.Position interfaces with relative coordinates, the absolute coordinates, coordinates, cheng prison four surface, etc
程序 PRG	Enter the program interface.Program interface program content, status, directory, file directory page.
补偿 OFT	Compensation interface, macro variables into the interface, repeatedly key in the transformation between the three interfaces.Compensation interface can display tool offset wear;Macro interface can display CNC macro variables
报警 ALM	Access to the alarm interface, alarm log key repeatedly in the transformation between the two interfaces.Alarm interface has CNC, PLC alarm two pages;Alarm log can display alarm and eliminate the alarm history
设置 SET	Enter setup interface, graphical interface, key can be repeatedly in the transformation between the two interfaces. Settings interface have switch Settings, G54 - G59, data manipulation, permissions Settings, time Settings;Graphical interface display of feed shaft movement trajectory

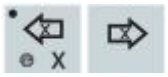
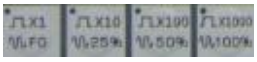
参数 PAR	Enter the state parameters, data, and the parameters of the screw filling interface.Keys can be repeatedly transformation between surface from all walks of life
诊断 DGN	The state of CNC diagnostic interface, PLC, PLC data, machine soft panels, version information interface.Keys can be repeatedly transformation between surface from all walks of life.CNC diagnostic interface, PLC, PLC according to CNC internal signal state, PLC the status information of each address, data.Machine tool machine tool can be soft panel soft keyboard operation;Version information display interface of CNC software, hardware and PLC version number
梯图 PLC	Information into the ladder figure interface, PLC, PLC ladder diagram and the parameters of the PLC, PLC diagnosis interface, key can be repeatedly transformation between surface from all walks of life.

1.1.3 Machine panel

Key of ECN*000M series machine panel are definite by PLC program (ladder diagram), please reference machine manufacturer's manual for each key's meaning.

Machine panel key' meaning which definite by ECN*000M series system standard PLC program as below:

Key	Name	Function explanation	Operation mode when function is valid
	Feed hold key	Program, MDI code paused	Automatic mode, input mode, DNC mode
	Cycle start key	Program, MDI code start	Automatic mode, input mode, DNC mode
	Feed ratio key	Adjustment of feed speed	Automatic mode, input mode, edit mode, machine back to zero, MPG mode, single step mode, manual mode, DNC mode
	Rapid ratio key	Adjustment of rapid move speed	Automatic mode, input mode machine back to zero, manual mode, DNC mode
	Spindle ratio key	Adjustment of spindle speed (spindle rotate speed analog control mode is valid)	Automatic mode, input mode, edit mode, machine back to zero, MPG mode, single step mode, manual mode, DNC mode
	Manual tool	Manual tool change	Automatic mode, MPG mode, single step

	change key		mode, manual mode
	Inching on-off key	Spindle inching state on/off	Automatic mode, MPG mode, single step mode, manual mode
	Lubrication on-off key	Machine tool lubrication on/off	Automatic mode, input mode, edit mode, machine back to zero, MPG mode, single step mode, manual mode, DNC mode
	Cooling liquid on-off key	Cooling liquid on/off	Automatic mode, input mode, edit mode, machine back to zero, MPG mode, single step mode, manual mode, DNC mode
	X axis feed key	Manual, single step operate mode, each axis positive/negative movement	Machine back to zero, single step mode, manual mode
	Y axis feed key		
	Z axis feed key		
	4th axis feed key		
	MPG/single step increment chosen and rapid ratio chosen key	MPG movement per club 1/10/100/1000*Min single each step 1/10/100/1000*Min rapid ration Fo, 25%, F50%, F100%	Automatic mode, input mode, machine back to zero, MPG mode, single step mode, manual mode, DNC mode
	Optional stop	M01 operation is paused when opitional stop is valid	Automatic mode, input mode, DNC mode
	Single-block switch	Program single-block running/continuous running state switch, single	Automatic mode, input mode, DNC mode
	Program option skip switch	If the program header which has"/" skipped	Automatic mode, input mode, DNC mode
	Machine tool	Machine lock indicator light	Automatic mode, input mode, machine

	lock switch	is no when machine is locked, feed axis output is invalid.	back to zero, MPG mode, single step mode, manual mode, DNC mode
	Assist function lock switch	Lock indicator light is on Assis function locked	Automatic mode, input mode, DNC mode
	Dry running switch	Dry running indicator light is on when it is valid, processing program/MDI code	Automatic mode, input mode, DNC mode
	Edit mode choose key	Enter edit operation mode	Automatic mode, input mode, machine back to zero, MPG mode, single step mode, manual mode, DNC mode
	Automatic mode choose key	Enter auto operation mode	Automatic mode, edit mode, machine back to zero, MPG mode, single step mode, manual mode, DNC mode
	Input mode choose key	Enter input (MDI) operation mode	Automatic mode, edit mode, machine back to zero, MPG mode, single step mode, manual mode, DNC mode
	Machine return to zero mode choose key	Enter machine back to zero operation mode	Automatic mode, input mode, edit mode, machine back to zero, MPG mode, single step mode, DNC mode
	Single step/MPG mode choose key	Enter single step or MPG operation mode (Parameter choose one of them)	Automatic mode, input mode, edit mode, machine back to zero, manual mode, DNC mode
	Manual mode choose key	Enter manual operation mode	Automatic mode, input mode, edit mode, machine back to zero, MPG mode, single step mode, DNC mode
	DNC mode choose key	Enter DNC operation mode	Automatic mode, input mode, edit mode, machine back to zero, MPG mode, single

			step mode, DNC mode

1.2 Operation mode summarize

ECN*000M series has edit, input, homing, single step/MPG, manual and DNC 7 kinds of operation modes.

- Edit operation mode

  In edit operation mode, it can do generate, delete, and amend etc. operations in edit operation modes.

- Automatic operation mode

  In automatic operation mode, run program automatically.

- Input operation mode

  In input operation mode, the parameters of input and the input and execute code。

- Machine back to zero operation mode

  In machine back to zero operation mode, feed shaft back to the machine zero point can be performed, respectively.

- single step/MPG operation mode

  In single step/MPG feeding mode, CNC moves according to the selected mobile increment.

- Manual operation mode

  In manual operation mode, Manual feed, manual can be rapid, feed ratio adjustment, quick ratio and spindle stop switch, lubrication, coolant fluid switch, spindle point move, manual operation pine dao/clip, etc.

- DNC operation mode

  DNC operation mode, DNC runs program.

1.3 Display interface

ECN*000M series has position interface, program interface etc. 9 interfaces, each interface has multiple display interfaces. Each interface and operation mode is independent.

1.3.1 Position interface

Press 位置 POS key enter position interface, position interface has absolute coordinate, relative coordinate, comprehensive coordinate and monitor pages. They can be read by  or 

Absolute coordinate display page

The X, Y, Z coordinate values are Cutting tool in the absolute position of the workpiece coordinate system, CNC electric on X, Y, Z coordinates to maintain, workpiece coordinate system by G92, G54 - G59 specified.



Note: In editing, automatic, entry, display speed "programming"; in the machine back to zero, program back to zero, manual mode display speed "manual". Under the handwheel way according to "hand wheel incremental"; in a single step mode shows "step increment"

Actual speed: In the actual processing, feed rate after operation of actual processing speed

Feeding rate: With the ratio of feed rate switch option;

G function code: Segment that is currently being executed in the G code of values;

The quantity of processing: When the program execution of the ((or M99) of the main program, the processing number 1;

Machining time: Timing starts when the automatic operation after start-up, the unit of time in order for the hours, minutes and seconds;

Quick ratio: Show the current rapid rate;

Spindle override: When the parameters of the NO. 001 Bit4 bit set to 0, according to the main shaft ratio;

S0000: The spindle rotational speed of main shaft encoder feedback, You must install the main shaft encoder to show the spindle speed；

The current tool: Knife number designated by the T code in the program.

Tool Offset: Tool length compensation H00 current processing procedure; D00 radius compensation current processing program。

The quantity of processing and cutting off electricity memory, reset method is as follows:

Quantity of processing reset: hold the 取消 CAN key, and N # key.

Cutting time reset: hold the 取消 CAN key, and M [key.

2) Relative coordinates to display the page

Shows the relative coordinate values for the current position relative to the point of reference for the relative coordinates, CNC on electric relative coordinates. Relative coordinates can be reset at any time. Relative coordinates after reset, the current point for relative reference point.

Relative coordinates reset method:

Under the relative coordinate display page hold the key X until the page X flashing, press 取消 CAN key, X coordinate values;

Under the relative coordinate display page hold the key Y & until the page Y flashing, press 取消 CAN key, X coordinate values;

Under the relative coordinate display page flashing, hold the key Z : until the page Z key, press 取消 CAN key, Z coordinates reset.

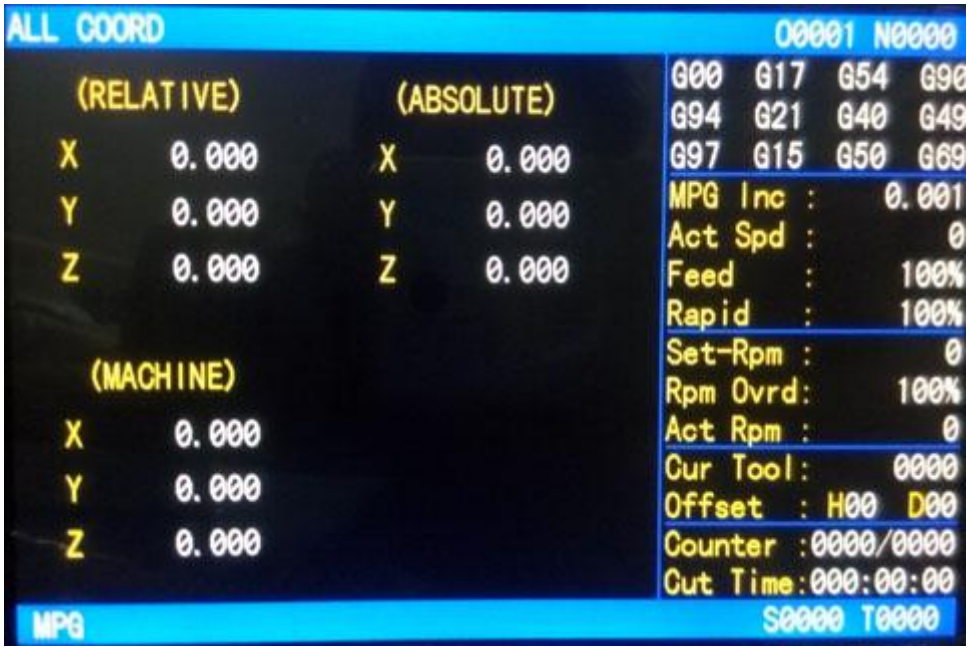
Note: if the 4th, 5th are effective, its reset method.

3) Comprehensive coordinate display page

In the comprehensive position page, also showed relative coordinate, absolute coordinates, machine tool coordinate, the residual amount of mobile (you moves only in automatically and record the mode shows).

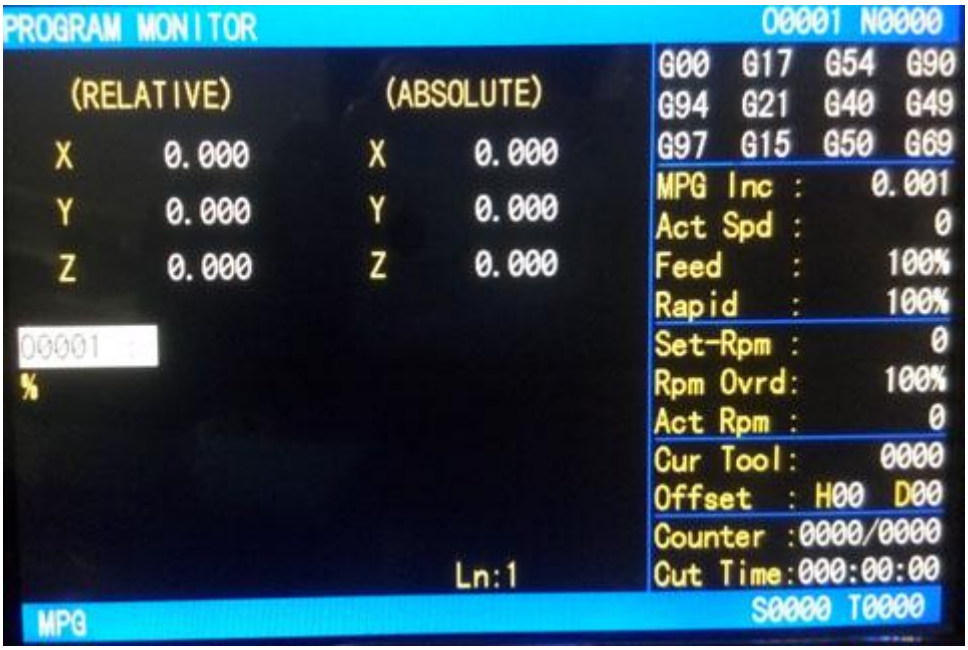
The indications of the machine tool coordinate for the coordinates of the current position in machine coordinate system, machine tool coordinate system was established through back to the machine zero point. Motion values are the difference value ofprogram blockor MDI code'sdifference value of target location and current location.

Display the page as follows:



4) Monitor display page

In process monitor display page, at the same time show the current position of the absolute coordinates, relative coordinates (if the state parameter No. 25 Bit0 bit is set to 1, shows the current position of the absolute coordinates and mobile) and the program of six section, in the program is run, according to the procedures section dynamic refresh, the cursor is located in the segment that is currently running.



Note: At the location interface, press 转换 CHG the key the sidebar at the bottom right corner to switch between cutting time and the system time.

1.3.2 Programming interface

Key 程序 PRG into the program interface, the interface program content, program state, directory, file directory four pages, through repeated 程序 PRG key switch in each page.

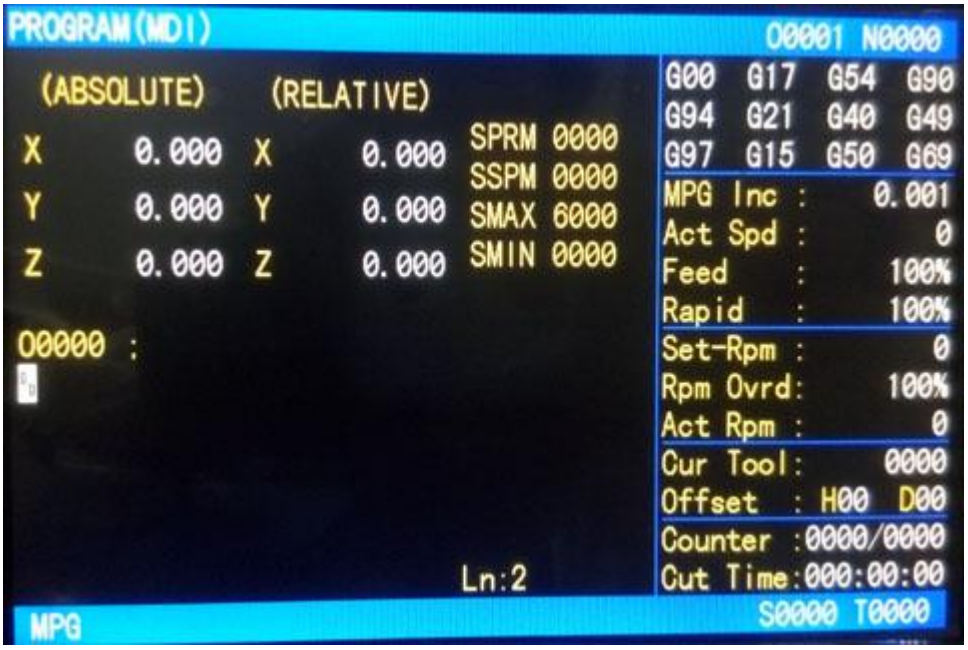
1) Program content pages

In the program content page, display program content, including the current procedures section. In the edit mode key operation, key or content forward and backward look at the program.



2) Application status page

In the program content page, key 程序 PRG will enter the application status page



3) The directory page

The application status page, keys 程序 PRG will enter the directory page. Under the interface, lists all the processing procedures, for the party Users find want to select the program and system at the right shows the current page program before 12 line segments。

Display the contents of the program directory page:

(a) Existing application number:

Displayed in the CNC has been deposited in the number of programs (including subroutine)

(b) The remaining application number:

Display can also deposit in the CNC program

(c) Has been in storage:

According to CNC has been deposited in the part program takes up the amount of storage（KB）

(d) The remaining amount:

According to the residual capacity of CNC program storage parts of (KB)

(e) Program directory:

According to the size of the part program name showed in turn into parts the application number of the program

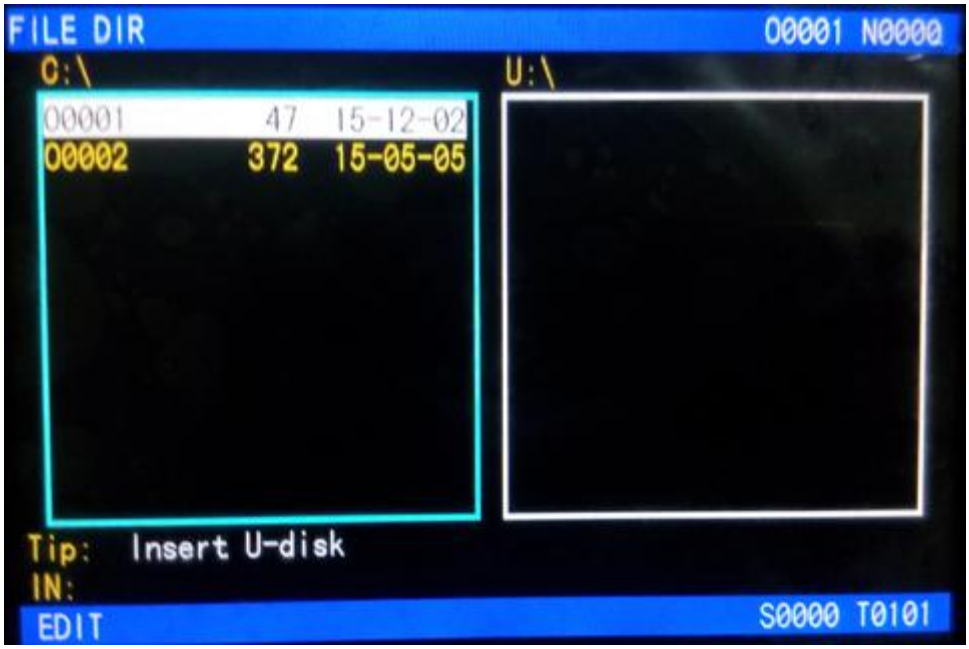
(f) Program size:



The size of the display of CNC program storage space

4) File directory page

In the directory page, key 程序 PRG will enter the file directory page. Page shows the following:



1.3.3 Tool offset wear, macro variable interface

Key 补偿 OFT is a composite key, press a key 补偿 OFT to enter from other display page knife repairing interface, then key 补偿 OFT macro variables into the interface。

1）Tool offset wear interface

Tool offset wear interface with a total of 4 pages, a total of 32 biases, wear number (No. 001 to No. 032) for the use of the user, through the key or according to each page, as shown in figure display page：



2）Public variable interface

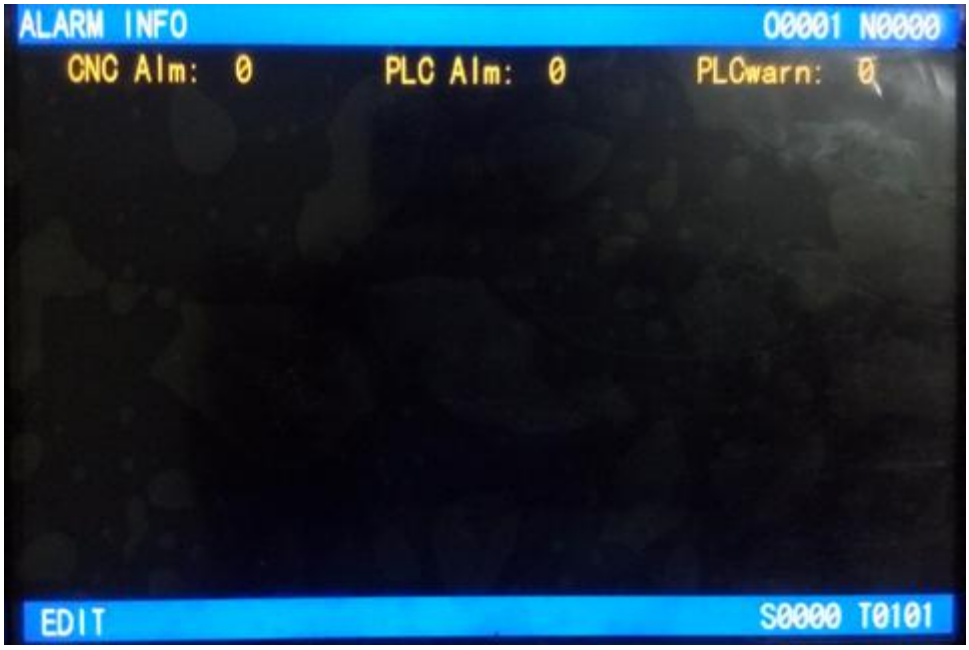
Macro variables interface have 20 pages, but through the key or display page, macro variable page shows 600 (No. 100 to No. 199 and No. 500 to No. 999) macro variables, macro variable values can be set directly by macro code to specify or keyboard.



1.3.4 Alarm interface

1) Alarm:

Key 报警 ALM into the alarm interface, through the key or to see all alarm display, page shows as below:



Note: the removal of the alarm: when the alarm has been cancelled, the key to eliminate alarm content.

(No. 100 alarm must at the same time key and a key 取消 CAN to remove).

2) Alarm Logging:

Key 取消 CAN again log into the interface of alarm record. Through the key or to view a total of 200 alarm log information;

Order: the latest alarm log information in the front of the first page, and so forth. When the alarm log every more than 200, a history log information is cleared after the most.



Note: the alarm log must be removed manually: under the level 2 password press 取消 CAN + ↵, **to eliminate all of the log information:**

1.3.5 Interface change

Key 设置 SET is a composite key, press key 设置 SET ,controller display setting interface, press it again entering the graphical interface, If press the key 设置 SET once again and again,it will change between setting and graphical interface.

1) Setting interface

Setting interface has five pages, can switch by page down/up key.

Switch Settings: display parameters, program, and program's S/N off and on.

Parameters switch : ON: you can modify the parameters;OFF: forbidden to modify parameters.

Program switch: ON: you can edit program; OFF: editing program is prohibited.

Program's S/N: ON: when editing program to automatically generate program segment number;

OFF: block number won't be generated automatically, should manually input.



Operation: data in this page for CNC data (status parameters, data parameters,screw compensation parameters, tool offset, etc.) backup and restore.

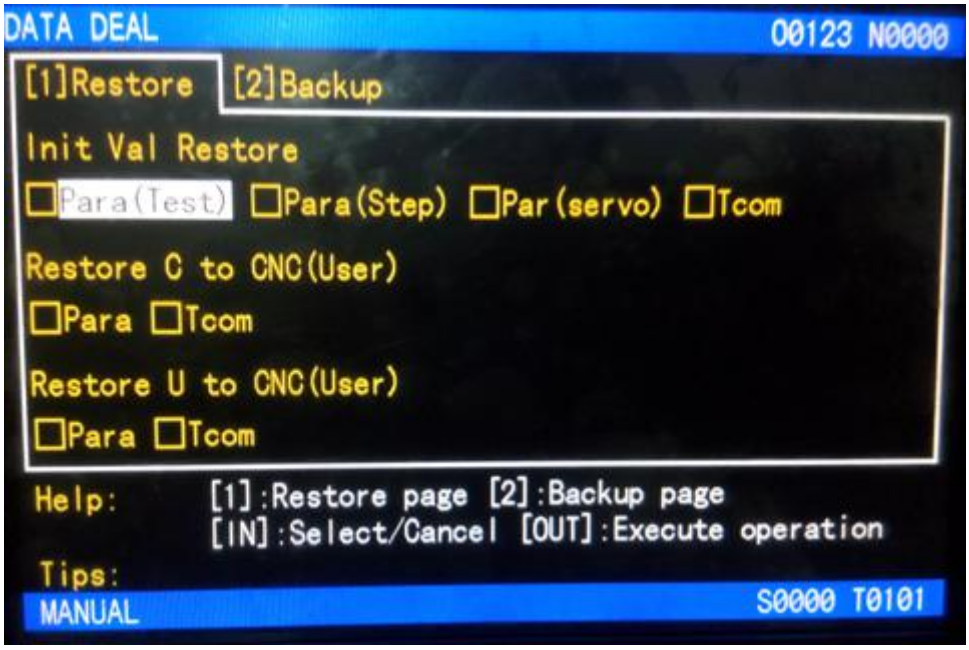
The initialization operation: can make parameters, tool offsets and screw compensation parameters to default values.

C disk data reduction to the CNC: the backup data files in the system disk restore to the system.

U disk data reduction to CNC: the backup data in the U disk restore to the system.

CNC data backup to disk C: system current parameters, tool offsets, screw compensation parameters and ladder diagram backup to system.

CNC data backup to usb: system current parameters, tool offsets, screw compensation parameters and ladder diagram backup to U disk.



Permissions Settings: display, set user operation level.

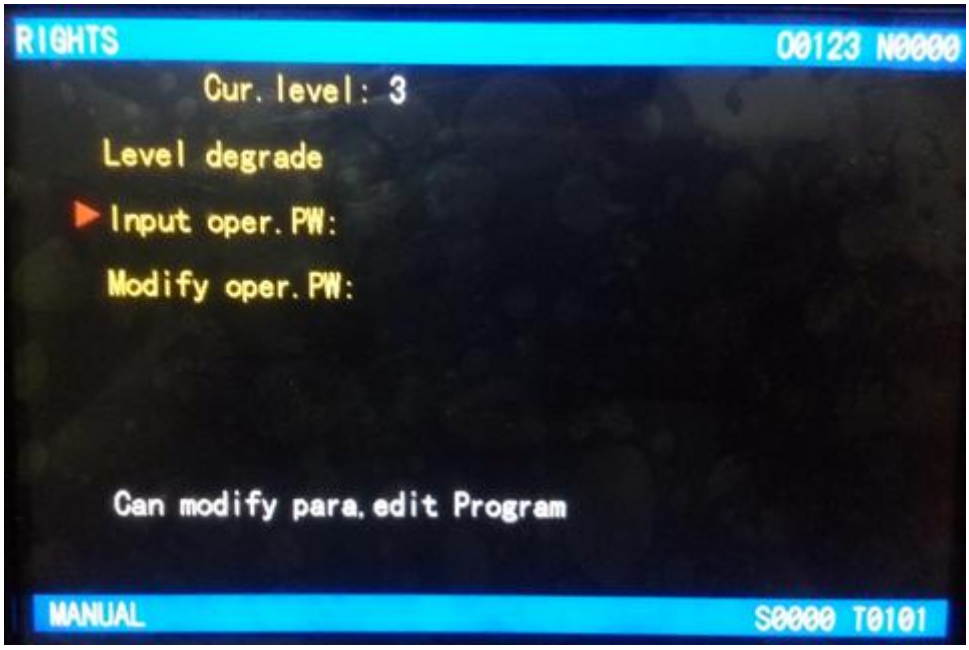
ECN*000M series password level is divided into four grade, from high to low were machine manufacturer level (level 2), the equipment management level (level 3), the engineer level (level 4), the operator level (level 5).

Machine manufacturers: allowed to modify the status parameters, data parameters, and screw compensation parameters,tool offsets data, edit program (including macro program), edit the PLC ladder diagram, download upload ladder diagram;

Equipment management level: the initial password is 12345, allowed to modify the status parameters, data parameters,tool offsets parameters,and editing program;

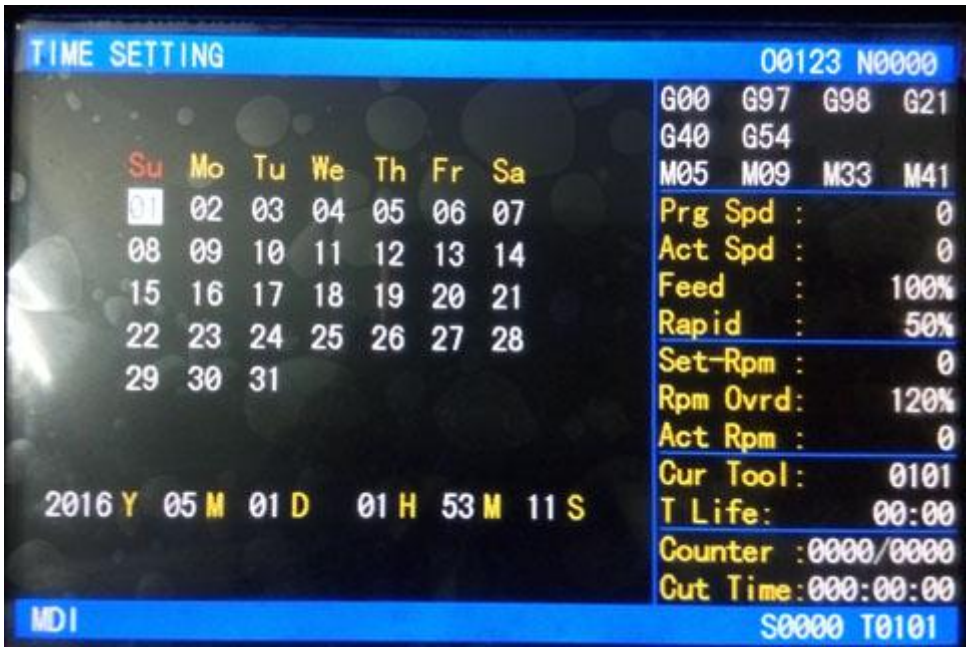
Engineer lelvel: the initial password is 1234 .can modify the tool offsets (tool setting), macro variables, program edit,but can not modify the status parameters, data parameters and screw compensation parameters.

Operator level: no password .can carry on the operate controller,tool setting, choose workpieces program, but can not edit and modify the status parameters, data parameters, and screw parameters.



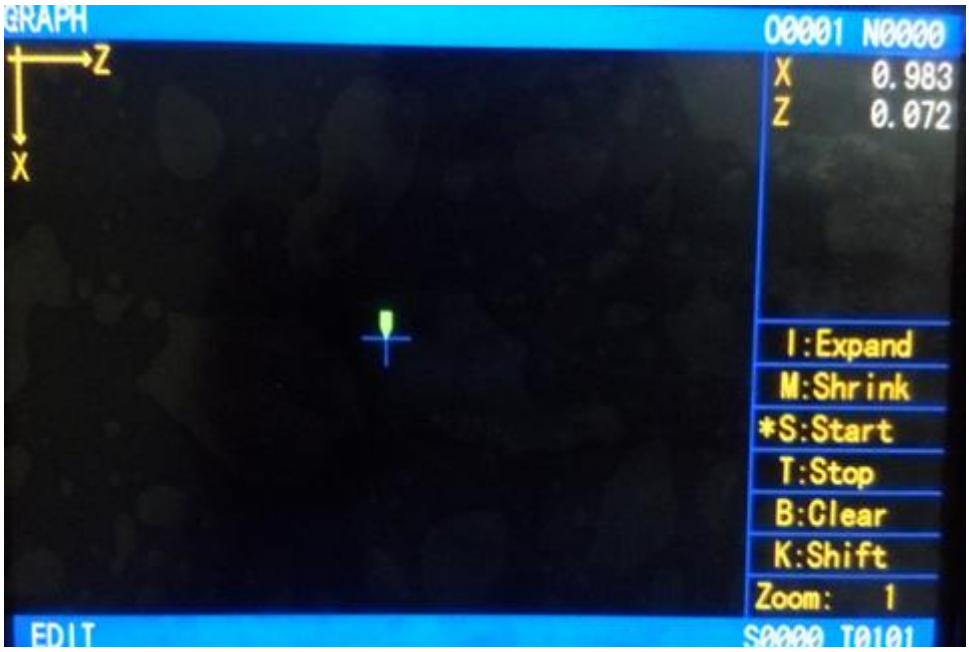
Time date: display the current time and date.

Can press key 转换 CHG into the modified model, and the key ⇨ ⇦ switch needed to modify year/month/day/minutes/seconds, press ⇩ ⇧, its value is modified, if you want to exit the modify mode, press the key 转换 CHG again.



2) Graphical User Interface

In the graphical interface, graphics can be enlarged, shrink, clearing operations.



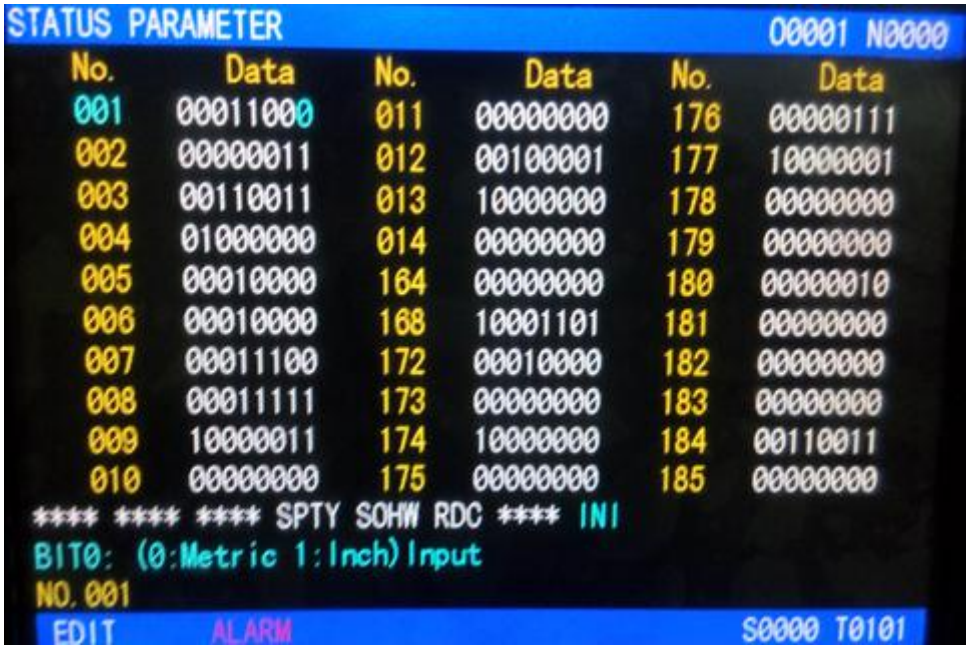
1.3.6 State parameters, data, and the parameters of the screw filling

Key 参数 PAR for a reusable key, click this key to enter repeatedly state parameters, data, and the screw filling parameters such as several interface surface。

1) State parameter interface

Key 参数 PAR to enter interface state parameters, the state parameter, a total of 60 points, according to two pages can be introduced into each page by key or view or modify the relevant parameters, specific as follows:

From status parameter page you can see,in the bottom of the page there are two line according to parameters of line, the first line shows the current parameters all abbreviations; the second line shows the current bit parameter's meaning , can use key S L to change the display parameters.



2) The data parameter interface

Press 参数 PAR key many times can enter (in status parameter interface can press) data parameter interface, then switch each page by key or view or modify the relevant parameters, specific as follows:

The bottom of the page has a line, shows the current parameter's meaning.



2) Screw parameter interface

Press 参数 PAR once again and again enter pitch error compensation interface, thread pitch error

compensation, there are a 256 compensation points display in 11 pages press or key can switch page:



1.3.7 CNC diagnosis, machine tools, PLC signal soft panel, help information, version information interface

Key 诊断 DGN is a composite key; click this key once again and again can enter the CNC diagnostic interface, PLC signal interface, and machine soft panel, help information and version information interface.

1) CNC Diagnostic interface

The I/O status of CNC controller and machine, Transmission of signals between CNC and PLC, The PLC internal data and CNC internal status can be displayed through diagnosis. Press Key 诊断 DGN into CNC diagnosis page displays, CNC diagnostic page displays have a keyboard, Status diagnosis and auxiliary function parameters, etc. By pressing key or to check.

In CNC diagnosis display page, in the bottom of the page there are two lines for detailed content, the second line shows the current bit signal's meaning, can display the diagnosis of a key or a key to change; The first line shows the current cursor in the diagnosis of all the English

abbreviations.



2) PLC Semaphore Interface

Display X0000 ~ X0063, Y0000 ~ Y0047, F0000 ~ F063, G0000 ~ G063 address status in the PLC signal interface page, etc. Repeated key 诊断 DGN into PLC interface. Press Key or to view the PLC signal status of each address.

Status page in PLC, details the bottom of the page has two lines, the second line shows the current bit meaning in Chinese, can display the key or a key to change the address bits. The first line shows the current cursor location number all the abbreviations.



3) Machine soft panel

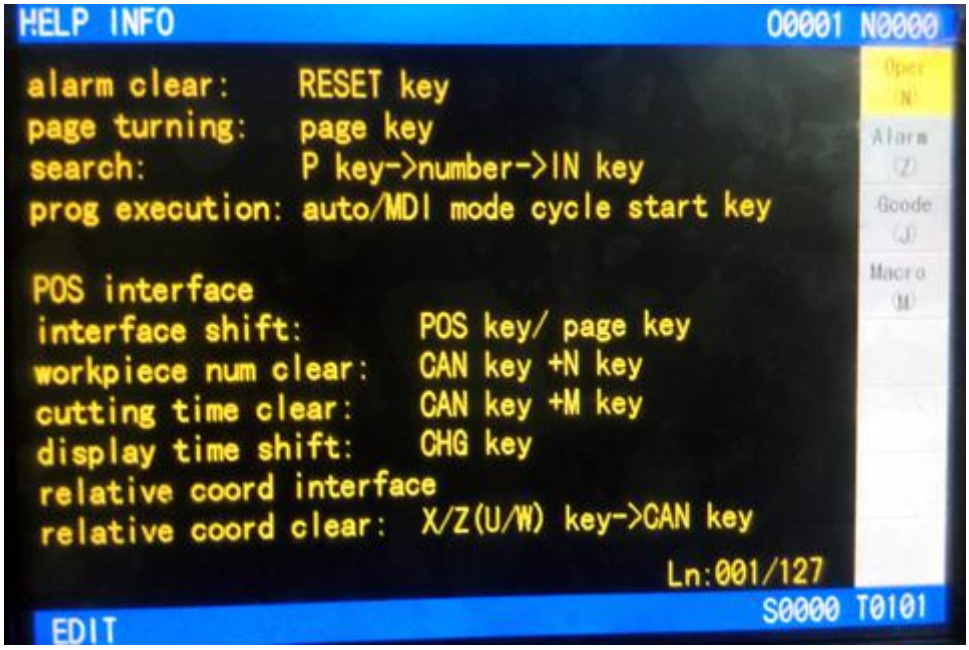
Soft panel key repeatedly into the machine tool, this page can be soft keyboard control of machine tools, machine tool soft panel page shows as below:

Note: press transformation, can switch between the columns



4) Help information

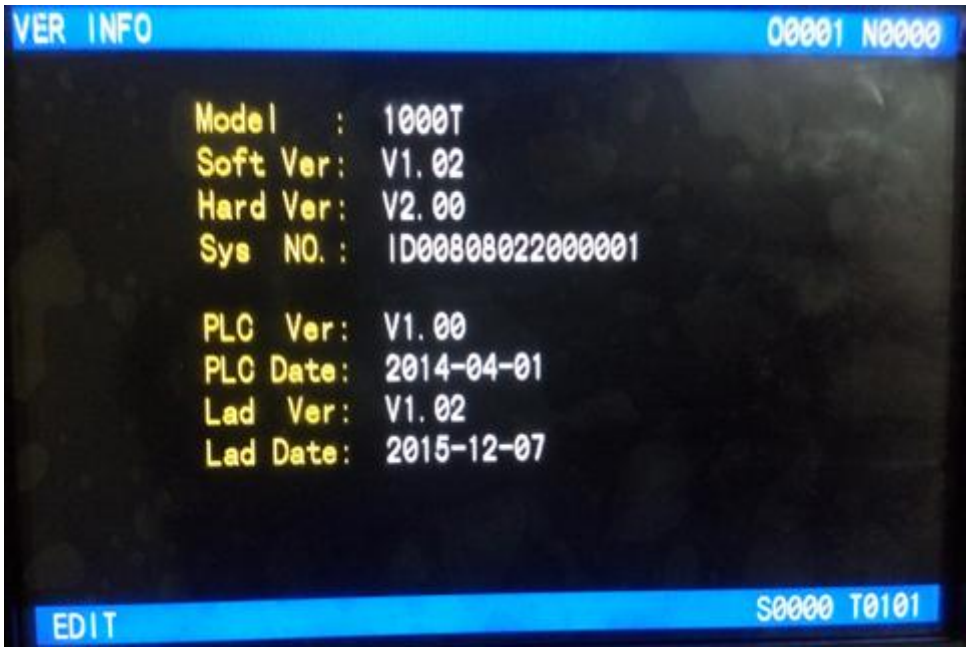
Repeated key 诊断 DGN help information into the interface. Can view the help page CNC current operating table clock, alarm table, G, macro instruction, such as information, displays the page as shown.



5) Version information

Key repeatedly enter version information interface. In the current version information page shows CNC software and hardware, system code, PLC version information, such as display the page as

shown.



Chapter 2 Start-up, shutdown and safety protection

2.1 Starting up

Beafore ECN*000M series controller is power on, please make sure:

1. The machine is normal.

2. Power supply voltage meets the requirements.

3. Wiring correctly and firmly.

After ECN*000M series controller self-checking, initialization.it will display below interface (relative coordinates).



2.2 Power off

Before controller power off, should be confirmed:

1. CNC feeding is stop status;

2. Auxiliary functions are off(like spindle ,coolant etc)

3. Power off controller power supply, then power off the machine power supply.

Note: the information about the operation to cut off the machine power supply please see illustrations of the machine tool manufacturers.

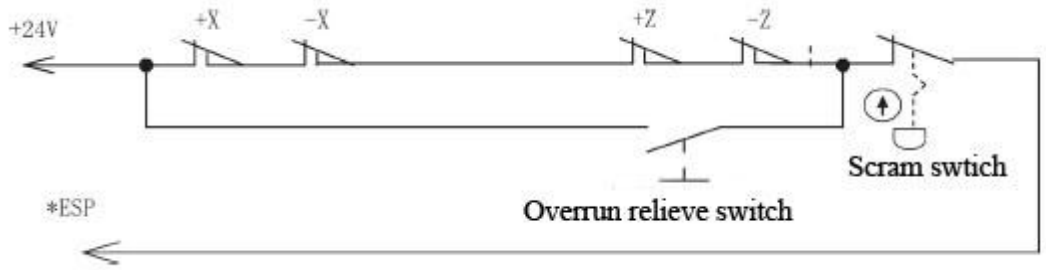
2.3 Distance protection

In order to avoid X, Y, Z axes travel over range, must use overtravel protection.

2.3.1 Hardware limit protection

In the X, Y, Z axis positive and negative maximum position installs limit switch and wiring as below

diagram, parameters No. 60 Bit2 (EALM) and Bit3 (LALM) must be set to 0. Then if overtravel, limit switch is action, axis moving is stop and controller is alarm.



When hardware overtravel, controller will alarm. The method to cancel alarm is: press overrun relieve switch key and do not loosen, switch to the alarm information page, check the alarm information, press reset key to clear alarm, then move axis in reverse direction (if positive overtravel,please move in negative direction; if negative overtravel, please move in positive direction)

2.3.2 Software limit switch

Software limit range are set by data parameter No. 010 ~ No. 019, the coordinate refer to mechanical coordinate . Bit1 status parameter No. 022 (LZR) can set whether software limit is avild before homing.

If software limit alarm, The method to cancel alarm is: switch to the alarm information page, check the alarm information, press reset key to clear alarm, then move axis in reverse direction (if positive overtravel,please move in negative direction; if negative overtravel, please move in positive direction)

2.4 Emergency operation

When machine is runing, if accident is coming, we must immediately stop machine work. Below described what you can do for our CNC controller in emergency status, But for machine, please check machine manuafacturer's user manual.

2.4.1 Reset

If controller output, axis moving are abnormal, press key 

, controller will in reset mode

1,stop all axis movement

2. M, S function output invalid (can set parameter to decide whether press 

to stop spindle

lubrication, cooling and other signals, PLC ladder diagram definition);

3. Auto running is stop, modal function and status will keep.

2.4.2 Scram

When press emergency stop key in running or dangerous condition (external stop signal is valid), CNC controller turn to stop status, movement is stop immediately, spindle rotation, such as cooling output are all closed. Release emergency stop key and clear alarm, CNC turn to reset status. Circuit connection method as shown in section 2.3.1 of this chapter。

Note 1: make sure the fault has been clean;

Note 2: press the emergency stop key before power off and power on can reduce equipment of electric shock;

Note 3: Please homming after emergency stop realse. in order to ensure the correctness of the coordinates (if machine has not installed home switch, then please do not do homing);

Note 4: only the parameters of NO. 60 Bit2 (EALM) is set to 0, external scram is available.

2.4.3 Feed hold

Can press key 

to make machine pause. but when the thread cutting, tapping,pause is not available.

2.4.4 Power off

Machine in the process of running or in dangerous or emergency conditions can be immediately cut off the power supply, in case of accident. But must pay attention to, to cut off the power supply after CNC display coordinates and the actual position may have larger deviation, must set tool again.

Chapter 3 manual manipulation

ECN*000M series controller panel key function is defined by PLC program (ladder diagram), the functional significance of each key-press please refer to the machine manufacturer's instructions.

In this chapter the following function associated with the operation panel key is for ECN*000M series controller standard of describing the PLC program, please note!

Key 

into the manual operation, manual operation can be carried out under manual feed, spindle control, ratio adjustment, tool change, etc。

3.1 Axis movment

In manual operation mode, can make the two axis manually feeding, manually fast moving.

3.1.1 Manual feeding

Hold down the feeding axis and the direction to choose key  or 

axis direction key can make the negative or positive the X axis feeding, Loosen the stop key axis moving is stop; Hold down 

or  Z axis direction key can make the Z axis to the negative or positive feed, release key axis movement is stop; Hold 

or 
Y direction key can make the Y axial negative or positive feed, loosen the key axis movement to stop; Hold 


4th axis direction key can make the 4 th axial negative or positive feed, loosen the stop key axis movement.

When manual feeding, press the key 

, the status indicates the indicator manually to fast moving state.

3.1.2 Manually move quickly

Press the key 

until the fast moving indicator of status indication area, press the 

or  key can make the negative or the X axis is fast moving, loosen the key axis movement to stop; Press the 

or  can make the Z axis to the negative or positive fast moving, loosen the key when the axial movement to stop; Hold 

or  Y direction key can make the Y axial negative or positive feed, loosen the key axis movement to stop; Hold 

 4th axis direction key can make the 4 th axial negative or positive feed, loosen the stop key axis movement. Quick ratio real-time adjustable effective.

When manually fast moving, press the key 

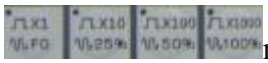
, that light is extinguished, fast moving, with manual feeding speed.

Note1: After power on, if haven't return to reference point, and when the fast moving switch is open (fast moving indicator is on), fast moving speed is manual feed speed or fast moving speed is set by status parameter NO. 012 (ISOT) option Bit0;

Note 2: In edit/the handwheel mode, key  **is invalid.**

3.1.3 Speed adjusting

In manual feed mode, can press key  to modify the manual feed ratio, there are total of 16 lelvels.

When in manual fast moving, can press  or modify the manual fast moving ratio, quick ratio have Fo, 25%, 50%, 100% four block. (Fo speed by data parameter No. 085 set)

3.2 Other manual

3.2.1 Turn anticlockwise, turn clockwise, stop control

: Manual operation mode, press this key, the spindle turn counterclockwise;

: Manual operation mode, press this key, the spindle stop;

: Manual operation mode, press this key, the spindle turn clockwise.

3.2.2 Spindle jog

: The spindle is in dynamic status.

Functional description: Press the key  on the panel can entry jog mode, the spindle should be in stop status when use spindle function.

Spindle in jog status, press keys , spindle anticlockwise jog; press Key , clockwise jog. the speed set by data parameter No. 208.

When spindle jog rotating, press key , can stop the spindle rotation at point dynamic rotation stops not output spindle brake signals.

K10.4 is set to 1, spindle point move effectively in any way. Automatic or entry under shaft in a rotating at state, to run the program will be closed at this time the spindle rotation and close the point move function.

Parameter setting:

PLC parameter K10.4 1/0: spindle point/manual in any manner, the handwheel, back to the zero modes is effective.

The data parameter No. 208: the rotation of the spindle point move

3.2.3 Coolant control



: Any operation mode, press this key, cooling fluid to switch between the switch.

Parameter setting: PLC parameter K10.1 1/0: reset spindle lubrication and cooling output keep/closed

3.2.4 Lubrication control

Functional description:

1. The automatic lubricating:

   DT13 =0: nonautomatic lubrication.

   When DT13 = 0, to flip lubrication output, press 

the machine tool operation panel key, lubrication output, repeated press lubrication output to cancel. When performing M32, lubrication output, then execute M33, lubrication output to cancel.

   When DT13 > 1, for lubricating timing output, press the machine tool operation panel key 

, lubrication output, after the parameters of the data of 112 times, lubrication output to cancel;

   Implement the M32, lubrication output, after data parameters DT13 set time, lubrication output to cancel. If DT13 set yet, at this time to perform M33, lubrication output to cancel.

2. Self-lubrication:

   DT13>0: Automatic lubrication can be installed DT53 DT13 and lubrication lubrication time interval time

   ECN*000M series after power on lubrication DT13 set time to start, and then stop the output, after DT53 setting time, repeat output lubrication, circulation in turn. Automatic lubrication, M32, M33 code, machine tool operation panel key 

is also effective.

Parameter setting:

PLC parameter: K10.1 1/0: reset spindle lubrication cooling output keep/closed

PLC parameter: K16.2 1/0: automatic lubrication lubrication is valid boot yes/no output

PLC data: DT53: automatic lubrication time interval (ms)

PLC data: DT13: automatic lubrication output time (ms)

DT05: M code execution duration (ms)

DT13: Lubrication opening time (0-60000 - ms) (0: lubrication unlimited)

3.2.5 Tool changing



: Manual operation mod, press this key, the spindle tool to loosen/clamping.

3.2.6 Axis ratio of callback

Manual operation mode, when chooses analog voltage output control spindle speed, repairable adjustable spindle speed.

Key , adjust the spindle rate change spindle speed, can realize the spindle ratio 8 magnitude real-time adjustment of 50% ~ 120%.

Chapter 4 The handwheel/a single step operation

In the handwheel/single-step operation way, machine tool according to the incremental value selected by the mobile.

ECN*000M series machine controller panel key function is defined by PLC program (ladder diagram), the functional significance of each key-press please refer to the machine manufacturer's instructions.

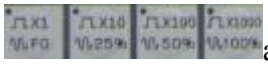
In this chapter the following function associated with the operation panel key is for ECN*000M series controller standard of describing the PLC program, please note!

4.1 Single step feed

Set the system parameter No. 001 of Bit3 to 0, key 

into a single step.

4.1.1 The choice of incremental

Key , select the mobile increment, mobile increment will display in the page. When PLC state parameters K016 BIT7 (SINC) to 1, step length value is invalid; When the BIT7 is 0, are effective.

4.1.2 Mobile direction choice

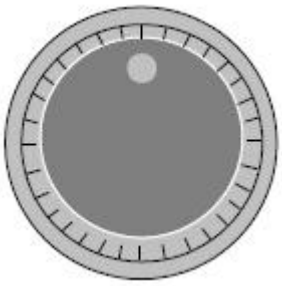
Press once or key, can make X axial negative or positive according to the single step increment feed at a time; Press once or c key, can make the Z axial negative or positive the single-step increment feed at a time. Press once  or Ykey, can make the Y axial negative or positive according to the single step increment

4.2 The handwheel (hand pulser) feed

Set the system parameter No. 001 of Bit3 to 1, key 

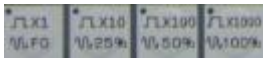
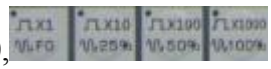
to enter the handwheel operation mode.

The handwheel shape as shown in the figure below:



The handwheel appearance figure

4.2.1 The choice of incremental

Key , select the mobile increment, mobile increment will be displayed on the page. When the PLC parameter K016 BIT7 (SINC) to 1,  step length value is invalid; When the BIT7 is 0,  are effective.

4.2.2 Axis and the direction of choice

Under the handwheel operation mode, press choose the corresponding

shaft.

The handwheel feed direction is determined by the hand wheel rotation direction. In general, the handwheel clockwise for positive feed, counterclockwise for negative feed。If sometimes the handwheel clockwise to negative feed, counterclockwise for positive feed, can exchange the handwheel end A and B signals. Also can choose by parameters of № 013 BIT0 handwheel rotates the direction of feed.

4.2.3 Other Actions

1. Turn counterclockwise, turn clockwise, stop control



: The handwheel/single-step mode, press this key, the spindle turn counterclockwise;



: The handwheel/single-step mode, press this key, the spindle stops；



: The handwheel/single-step mode, press this key, the spindle turn clockwise。

2. Spindle point move



: The spindle is in dynamic state.

At the spindle dynamic state, keys 

, anticlockwise point move; Key 

, clockwise turning point. Spindle speed at the data parameter No. 208 set. See this first 3.2.2.

3. Coolant control

See this 3.2.3

4. Lubrication control

This paper first 3.2.4

5. Manual tool change



: The handwheel/single-step mode, press this key, the cutting tool to loosen/clamping.

6. Axis ratio of callback

The handwheel/single-step mode, when choose analog voltage output control spindle speed, repairable adjustable spindle speed.

Key 

, adjust the spindle rate change spindle speed, can realize the spindle ratio 8 magnitude

real-time adjustment of 50% ~ 120%

4.2.4 Important notes

1. Handwheel calibration machine movement in relation to the see the table below:

	Each scale of mobile on the handwheel			
Handwheel incremental	0.001	0.01	0.1	1
Coordinates specified values	0.001mm	0.01mm	0.1mm	1mm

(Minimum input increment of 0.001 mm, for example)

Note1: The handwheel increment and system metric system input on the current status and the minimum input increment;
Note 2: Hand wheel rotation speed shall not be higher than 5 r/s, if more than 5 r/s, may lead to scale value and mobile.

Chapter 5 Input operations

Under the entry mode of operation, parameter setting, the code word can be input as well as the implementation of the code word.

ECN*000M series controller machine panel key function is defined by PLC program (ladder diagram), the functional significance of each key-press please refer to the machine manufacturer's instructions.

In this chapter the following function associated with the operation panel key is for ECN*000M series controller standard of describing the PLC program, please note!

5.1 The input code word

Select entry operation mode, enterprogram status page, enter program block G00 X50 Z100, operation steps are as follows:

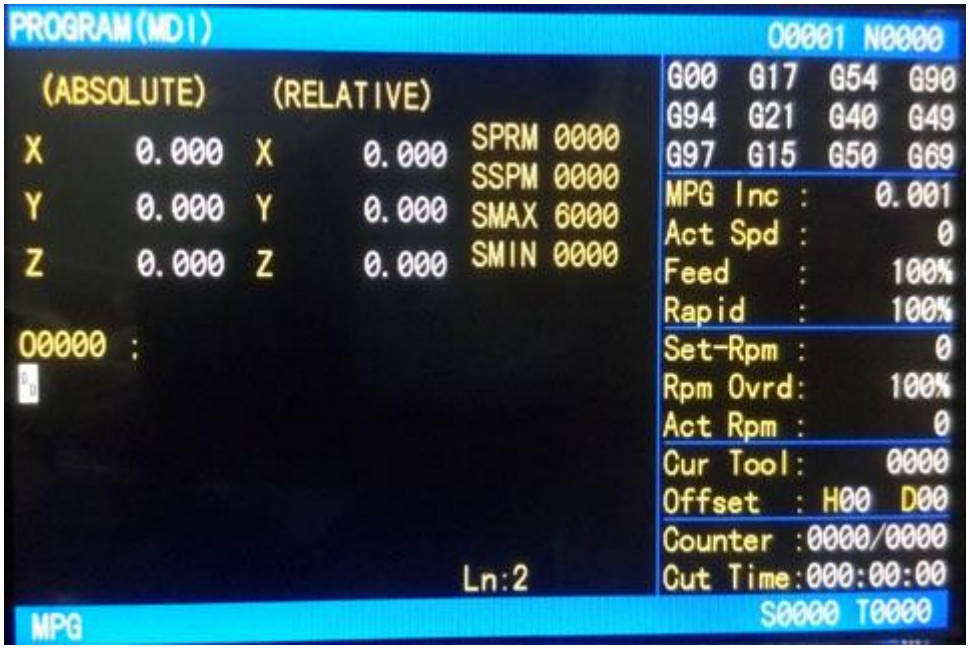
1. Key 

to enter into operation mode;

2. Key 

(key  or  key again when necessary, or more key 

) into the application status page:



3. In turn, type the address key G*, the number key 0 0;

4. In turn, type the address key Z:, the number key 1 0 0;

5. In turn, type the address key X, the number key 5 0;

Execute after the operation page displays the following (entry 4 section of the program, according to section 6 of program);

5.2 The implementation of the code word

After input code word, press the 输入 IN key,

Key RESET to perform input program block. During operation can key FEED HOLD, a key RESET and urgent stop key to stop running the procedures section.

Note: subroutine calling code (M98 P; etc.) under the MDI execution is invalid.

5.3 Preferences

Under the entry way, parameters into the interface can be modified for parameter values, detailed in this chapter 10.

5.4 modifications of data

Under the application status page, the input prior to execution of the procedures section, if the field is in the process of the input is wrong, a key 取消 CAN to cancel the show state and to modify the program

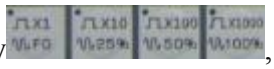
can also be key  to remove all content, input the correct procedures section again

5.5 Other Actions

1. Repairable adjustable spindle ratio the handwheel/single-step mode, when choosing analog voltage output control spindle speed, repairable adjustable spindle speed.

   Key , adjust the spindle rate change spindle speed, can realize the spindle ratio 8 magnitude real-time adjustment of 50% ~ 120%.

2. Repairable rapid rate

   Key , the rapid movement of feed speed, can realize fast movement speed for 4 real-time adjustments.

3. Repairable inlets to ratio

   Operating mode of the input, keys , inlet to the change of feed speed ratio, which can realize the actual speed for F code specifies the feed speed of 0 ~ 150% of total 16 level of real-time control.

Chapter 6 Program editing and management

In edit mode operation, can create, select, modify, copy, delete the program, also can realize CNC and CNC, the CNC two-way communication with PC.

To prevent program by accident, modify, delete, ECN*000M series controller set the program switch. Before editing program, it is necessary to open the program switch, switch Settings in this section 10.1.1.

For the convenience of management, level ECN*000M series controller provides 3 user permissions Settings. Must have the operation level of 4 class above levels (4, 3, etc.) to open the program switches, editor of the procedures involved. See section 10.3 each operation level allows operation.

6.1 The establishment of the program

6.1.1 The generation of program segment number

Program, can be programmed and not into the program segment number, is according to the procedures section into the order of execution (call)

When the switch Settings page switch in a state of "automatic sequence number," CNC segment that is not automatically generated, but can manually when programming program into the segment number.

When switch Settings page "automatic number" switch in the open state, CNC automatic generation

procedures section, edit key to automatically generate the next program of segment number, block number by the incremental value of № 042 CNC data parameter Settings (serial number automatically setting detailed in this section 10.1.1)



6.1.2 Program content input

1. Key EDIT to enter edit operation way; Key 程序 PRG program into the interface.



2. Type the address key O, digital key 1 , 2 , 3 in turn (O 0123 program, for example)

3. The key 

, set up the new program.

4. According to the good parts program input them one by one, each input one character at a time, an immediate display on the screen input character (composite key processing is repeated by the composite key, implement alternate input), after the completion of the need to switch to the other works will be stored program.

5. According to the method of step 4 can complete program other period of the input.

Note: 1.In the edit mode, only input the complete instruction word input, separate letters or Numbers, the system does not support.

2.Found that when the input program input instruction word wrong, may cancel input command keys 

one by one.

6.1.3 The movement of the cursor

1) Key 

into the editing operation mode, 

select program content display page

2) Key , the cursor move up a line; If the current column number of the cursor is greater than the line of the total number of columns and key , the cursor to the last paragraph section of the tail;

3) Key , the cursor down one line, if the current column number of the cursor is greater than the next line the total number of columns and key , the cursor at the end of the next line;

4) Key , the cursor moves to the right column; if the cursor at the end of the line, the cursor is moved to the next program first;

5) Key , the cursor moves left a column; if the cursor at the beginning, the cursor to the last paragraph section of the tail;

6) Key , page up;

7) Key , page down;

8) Display page in edit operation methods, procedures, key 

and the cursor back to the program start.

6.1.4 Words, the line number of retrieval

Word retrieval: starting from the current cursor position, up or down to find the specified characters

A method to determine the steps below:

1) Press 

select edit operation mode;

2) Key 

, display program content pages;

3) Enter the words to retrieve.

4) Key  (according to locate character with the current cursor position relationship of the characters to determine key  or a key ), display the page as below:



5) Press the up/down key again, to find the next position of words.

Note 1: if not for that input to retrieve the word will disappear.

Note 2: the macro edit mode cannot be retrieve and scan of the characters.

Note 3: in character retrieval, not retrieve the called subroutines in the characters, the characters in the subroutine to retrieve a subroutine.

The retrieval: the cursor a rapid positioning to the program specified line.

A method to determine the steps below:

1) Select edit operation mode, the program content display page;

2) Key 

(address field data for empty);

3) Input program line number;

4) Key 输入 IN, the cursor will jump to input the line number of the program。

6.1.5 Word of insert

Operating procedures are as follows:

1) Select edit operation mode, the program content display page;

2) Input to insert the words, press the key 插入 INS, the system will insert the input to the left of the cursor.

6.1.6 The deletion of the word

Operating procedures are as follows:

1) Select edit operation mode, the program content display page;

2) Put the cursor to the location of the need to delete, key 删除 DEL, the system will delete the contents of a cursor, if continuous key 删除 DEL, will remove the cursor to the right of the program content continuously.

6.1.7 Word modify

Operating procedures are as follows:

1) Select edit operation mode, the program content display page;

2) Move the cursor to the place where need to modify and input to modify the word, and then key 修改 ALT, system to replace the contents of the cursor to the content of the input.

6.1.8 The remove of the procedures section

Operating procedures are as follows:

1) Select edit operation mode, the program content display page;

2) Move the cursor to the first (column 1) delete the line of the procedures section, key N#, key 删除 DEL again to delete cursor.

6.1.9 To delete multiple procedures section

Operating procedures are as follows:

1) Select edit operation mode, the program content display page;

2) Move the cursor moves to remove the beginning of a line of the procedures section;

3) Input to remove part of the last the serial number of the procedures section;

4) Key 删除 DEL, the cursor can be delete procedures section between address with the markers.

6.1.10block delete

Operating procedures are as follows:

1) Select edit operation mode, the program content display page;

2) Delete the cursor moves to block the first code word;

3) Input to remove block finally a code word;

4) Key 删除 DEL, the cursor can be address block between deleted with the markers.

Note 1: if the deleted successfully, the input code word will disappear, or delete failed. If the specified character downs how much the same, is the distance from the current cursor on a recent by default.

6.1.11 The copy of the procedures section

Operating procedures are as follows:

1) Select edit operation mode, the program content display page;

2) Move the cursor to the copy of the beginning of a line of the procedures section;

3) Press the key N #, key 转换 CHG again, copy the cursor procedures section.

6.1.12 More than the copy of the procedures section

Operating procedures are as follows:

1) Select edit operation mode, the program content display page;

2) The cursor to the first want to copy the program code words;

3) The input serial number to copy program period of the last paragraph;

4) Procedures section between key 转换 CHG, the cursor to the input character copy complete.

6.1.13 Block replication

Operating procedures are as follows:

1) Select edit operation mode, the program content display page;

2) The cursor moved to copy the first code block;

3) Input to replicate the end of the block a code word;

4) Key 转换 CHG, the cursor to the input character between block replication is complete.

6.1.14 Block of the paste

Operating procedures are as follows:

1) Select edit operation mode, the program content display page;

2) Move the cursor to the program location of paste;

3) Key 输出 OUT, to copy the last program content is inserted into the cursor before, to complete the paste operation.

6.2 The deletion of the program

6.2.1 To delete a single program

Steps are as follows:

1) Select edit operation mode, enter the program display the page;

2) Type the address in turn key O ,the number key 0 0 0 1 (O0001 program, for example)

3) Key 删除 DEL, O 0001 program is deleted.

6.2.2 Delete all program

Steps are as follows:

1) Select edit operation mode, enter the program display the page;

2) Type the address key O in turn, the symbol key −+ > , the number key 9 9 9 9

3) Key 删除 DEL all programare deleted.

6.3 The selection of the program

When CNC has multiple programs, you can through the following three methods selection process.

6.3.1 Retrieval method

1) Select edit or automatic operation mode;

2) The key 程序 PRG, and enter the program content display screen;

3) Press the address O, type the number program;

4) According to the press ⇩ or 换行 EOB or in an automated way ⇩, on the display screen displays the retrieved process, if the program does not exist, CNC in alarm.

Note: step 4, the edit operation mode, if the program does not exist, keys 换行 EOB, CNC will create a new program.

6.3.2 Scanning method

1) Select edit or automatic operation mode;

2) Key 程序 PRG, display screen and enter the program;

3) Press the address O␣;

4) Press ⇩ or ⇧ to display next or on a program;

5) Repeat steps 3, 4, one at a show in the program.

6.3.3 The cursor recognition method

1) Select automatic operating mode (must be in the running state;

2) Display the page key 程序 PRG, enter the program directory;



3) Press ⇩ or ⇧ cursor move up/down, or press 📄 or 📄 。 up/down turn pages, or press ✕, I A he cursor to the first/last program in the directory, move the cursor to be selected on the program name (the cursor at the same time, the program content changes);

4) Key 换行 EOB or a key 输入 IN

6.4 Program name

1) Select edit operation mode, enter the program display the page content;

2) Press the address O␣, type the new program name;

3) key 修改 ALT.

6.5 Copy of program

Save the current program：

1) Select edit operation mode, enter the program display the page content;

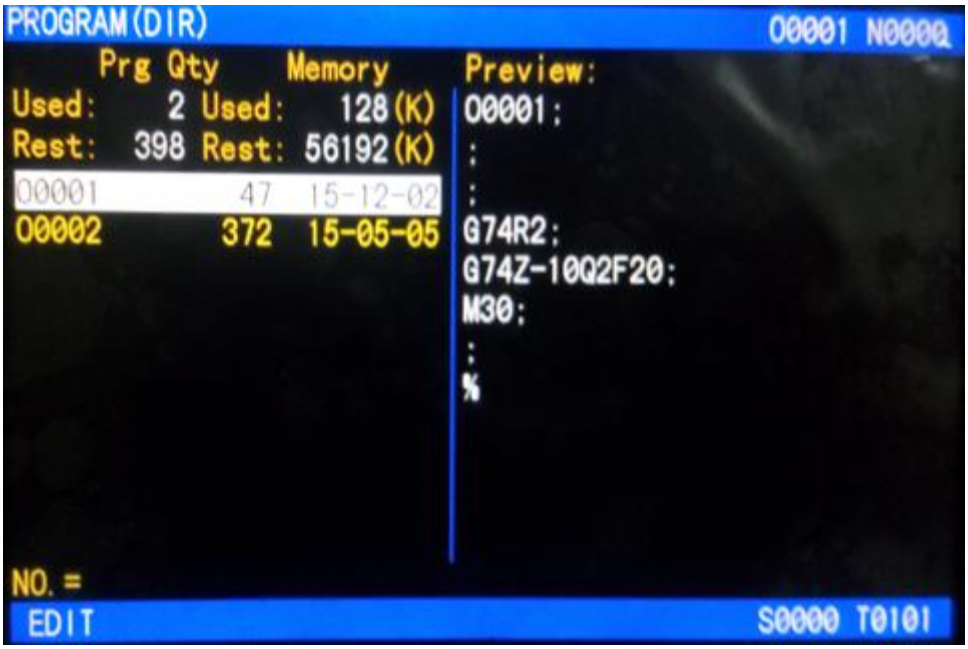
2) Address key O␣, type XinCheng serial number;

3) Key 转换 CHG.

6.6 Program management

6.6.1 Program directory

Key 程序 PRG to enter the show program directory page. In this page, in the form of table of contents shows CNC existing program name, one page only shows up to 10 program name, when the number of more than 10 existing applications, can press up/down page key displays another page program directory.



1)Open the program

Open the specified procedures: O + number + enter (or 换行 EOB) or serial number + enter (or 换行 EOB).

In edit mode, if the input sequence number does not exist will create program.

2) The deletion of the program:

1. Edit mode According to 删除 DEL remove the cursor specified procedure.

2. Edit mode O + serial number+ 删除 DEL Or the serial number+ 删除 DEL.

6.6.2 The number of stored procedures

This system the number of stored procedures for a maximum of 400, the show CNC can the current program number has been stored and the rest of the parts.

6.6.3 Memory capacity

The application of this system storage capacity (total capacity of 56 m), this shows the current has been occupied storage capacity and the remaining storage capacity.

CHAPTER 7 AUTO OPERATIONS

The key function of this ECN*000M series machine panel are defined by ladder; please refer to the manual of machine builder for each key function significance.

Please note that the following function introduction is described based on the ECN*000M series standard PLC program!

7.1 Auto run

7.1.1 Selection of the program to be run

1. Search method

1) Select the Edit or Auto mode;

2) Press 程序 PRG key to enter the PRG CONTENT page;

3) Press the address key  and key in the program No.

4) Press  or  key, the program retrieved will be shown on the screen, if the program

2. Scanning method

1) Select the Edit or Auto mode

2) Press  key to enter the PRG display page

3) Press the address key 

4) Press the  or  key to display the next or previous program;

5) Repeat the step 3, 4 above to display the saved program one by one.

3. Cursor method

a) Select the Auto mode (must in non-run state)

b) Press  key to enter the PRG LIST page;

c) Press  key to move the cursor to the name of theprogram to be selected;

d) Press  key.

7.1.2 Program Start

1. Press key to select the Auto mode

2. Press key to start the program, and the program execution begins

Note: Since the program execution begins from the block where the cursor locates, before pressing the key, make a check whether the cursor is located at the block to be executed.

7.1.3 Stop of the Auto Run

● Stop by command (M00)

1. M001

The block containing M00 is executed, the auto run is stopped. So the modal function and state are all reserved. Press the key 

or the external Run key, the program execution continues.

2. M002

Press the key 

, OPT instruction light is on and the function works. Execute the program segment which includes M01 and stop auto run, modal function and state will be stored. Program continues to execute after press the panel key 

or external run key.

● **Stop by a relevant key**

1. In Auto run, by pressing key 

or external dwell key, the machine remains the following state:

(1) The machine feed decelerate to stop;

(2) The modal function and state are saved;

(3) Program continues run after press key 

.

2. Press reset key 

(1) All axes movement is stopped.

(2) M, S function output is invalid (the automatic cut-off of signals such as spindle CCW/CW, lubrication, cooling by pressing 

key can be set by the parameters)

(3) Modal function and state is held on after the auto run.

3. If the external emergency key (external emergency signal valid) is pressed under the dangerous or emergent situation during the machine running, the CNC system enters into emergency state, and the machine moving is stopped immediately, all the output (such as spindle rotation, coolant) are cut off. If the Emergency key is released, the alarm is cancelled and CNC system enters into reset mode.

4. When the Auto mode is switched to the Machine zero, MPG/Step, the current block “dwells” immediately; when the Auto mode is switched to the Edit, MDI mode, the “dwell”is not displayed

till the current block is executed.

Note 1: Ensure that the fault has been resolved before cancelling the emergency alarm.

Note 2: The electric shock to the device may be decreased by pressing the Emergency key before power on and off.

Note 3: The Machine zero return operation should be performed again after the emergency alarm is cancelled to ensure the the coordinate correctness (but this operation is unallowed if there is no machine zero in the machine)

Note 4: Only the BIT2 (EALM) of the bit parameter No.60 is set to 0, could the external emergency stop be valid.

7.1.4 Auto run from an arbitrary block

Press 

key to enter the Edit mode, press key 

to enter the Program interface, or press  or  key to select the PRG CONTENT page:

1. Move the cursor to the block to be executed (for example, move the cursor to the 2th line head if it executes from the 2th line);

2. If the mode (G, M, T, F command) of the current block where the cursor locates is defaulted and inconsistent with the running mode of this block, the corresponding modal function should be executed to continue the next step.

3. Press 

key to enter the Auto mode, then press 

key to start the program.

7.1.5 Adjustment of the feedrate override, rapid override

In Auto mode, the running speed can be altered by adjusting the feedrate override, rapid override with no need to change the settings of the program and parameter.

● Adjustment of the feedrate override

Press the  key in 

, it can realize 16-level real time feedrate adjustment.

Press the  key each time, the feedrate override ascends a gear level till 150%

Press the  key each time, the feedrate override decends a gear level till 0;

Note 1: The actual feedrate value is specified by F in program feedrate override adjustment;

Note 2: Actual federate= value specified by F× feedrate override

● Adjustment of rapid override

It can realize the 4-level real time rapid override FO. 25%. 50%. 100% adjustment by pressing the  or 

Note 1: The max. rapid traverse speeds of each axis are set by bit parameter No.090 ~ No.094 respectively;

Actual rapid traverse rate = value set by parameter×rapid override

Note 2: When the rapid override is F0, the rapid traverse rate is set by bit parameter No.085.

7.1.6 Spindle override adjustment

While the spindle speed is controlled by the analog voltage output in Auto mode, it can be adjusted by spindle override.

Press the  key to adjust the spindle override for the spindle speed, it canrealize 8-level real-time override adjustment between 50%~120%.

7.2 DNC Running

This CNC system has a DNC function, by the connection of the DNC communication software with this system, the high speed, high capacity program can be performed in this system.

In Auto mode, press t 

key on tool panel, it enters the DNC mode. Then press the 

key to start the program DNC machining under the condition that the PC is get ready

Please refer to the DNC communication software for details.

7.3 Running State

7.3.1 Single Block Execution

When the program is to be executed for the 1st time, to avoid the programming errors, it may select Single block mode to execute the program.

In Auto mode, the methods for turning on single are as follows.

Press the 

key to make the single block indicator in State area to  light up, it means that the single block function has been selected.

In Single block mode, when the current block execution is finished, the CNC system stops;if next block is to be executed,it needs to press the 

key.

Note 1: At the mid point, the single block stops in G28 commands

Note 2: Excute call sub-program (M98), sub-program calls return code (99) single program segment is invalid. But in M98 and M99 program segment, single block stop is valid except N, O, and P.

7.3.2 Dry Running

Before the program is to be executed, in order to avoid the programming errors, it may select the Dry run mode to check the program. In Auto mode, the methods for turning on the dry running switch are as follows.

Press 

key to make the dry run indicator in State area to light up, it means that the dry run function is selected.

The speed specified by the program is invalid in Dry run, and actural feedrate is set by the DATA parameter No.82.

7.3.3 Machine Lock

In Auto mode, the ways to make machine lock function valid are as follows.

Press the 

key to make the machine lock indicator  in State area to light up, it means that i t has enterd the machine lock state.

While in the machine lock mode:

1. The machine carriage doesn't move, the "MACHINE"in the INTEGRATED POS page of the POSITION interface doesn't vary too. The RELATIVE POS and ABSOLUTE POS, DIST TO GO are refreshed normally.

2. M, S, T commands can be executed normally.

7.3.4 MST lock

In Auto mode, the ways to make MST lock function valid are as follows.

Press the 

key to make the MST lock indicato 

in State area to light up, it means that it has entered the MST lock state. And the carriage move is not performed by M, S, and TCommands.

Note: When the MST lock is valid, it has no effect on the execution of M00, M30, M98,M99.

7.3.5 Block Skip

If a block in program is not needed to be executed and not to be deleted, this block skip function can be used. When the block is headed with"/"and Block skip function is valid, this block is skipped without execution in Auto mode

In Auto mode, the way to make block skip function valid is as follows.

Press the key 

to make the block skip indicator 
in State area to light up, it means that the block skip function is valid.

Note: While the block skip function is invalid, the blocks headed with "/" signs are executed normally in Auto mode.

7.4 Other Operations

1. Press key, Coolant On/Off switchin automatic operation mode;

2.press any key of 

to achieve the operation modes switches;

3. Press key 

to achieve CNC reset;

4. Automatic lubrication function (please refer the Chapter 3 in this Volumn)

CHAPTER 8 MACHINE ZERO RETURN OPERATION

8.1 Machine Zero

8.1.1 Machine zero point

The machine coordinate system is a basic coordinate system for CNC coordinate calculation. It is an inherent coordinate system of the machine. The origin of the machine coordinate system is called machinezero (or mechanical reference point). It is defined by the zero return switches fixed on the machine. Usually the switch is fixed on the positive maximumstrokes of X, and Z axes.

8.1.2 Machine Zero Ruturn Steps

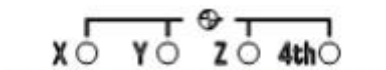
1. Press 

key, it enters the Machine zero modes, the bottom line of the screen page show“REF”, the figure is as follows:

2. Press 

key to select the machine zero of X, Y or Z axis.

3. The machine moves along the machine zero direction, and returns to the machine zero via the deceleration signal, zero signal detection. And the axis stops with the machine zero finish indicators lighting up.



Machine zero finish indicators

Note1: If the machine zero is not fixed on the machine, machine zero operation isunallowed;

Note2: The machine zero finish indicators go out as following circumstance:

1)Remove from the zero;

2) CNC outage;

Note3: After the machine zero operation, the cancellation of the tool length offset for the CNC;

Note4: See details in the 3rd part INSTALLATION AND CONNECTION for the parameters concerning with the machine zero;

Note 5: When machine zero return, the coordinate system will be reseted.

8.2 Other operations of Machine zero

1. Press key 

, spindle anticlockwise;
2. Press key 

, spindle axis stop;
3. Press key 

, spindle axis clockwise;
4. Press key 

, cooling on/off switch;
5. 

Lubrication control (please refer the third chapter of this volumn)
6. Press key 

manual loose/catch tool;
7. Callback of Spindle ratio;
8. Callback of rapid ratio;
9. Callback of feed ratio.

CHAPTER 9 DATA SETTING, BACKUP AND RESTORE

9.1 Data setting

9.1.1 Switch setting

In SWITCH SETTING page, the ON-OFF state of PARM SWT (parameter switch), PROG SWT (program switch), and AUTO SEG (auto sequence No.) can be displayed and set, the figure is as follows:



1. Press RESET key to enter the Setting interface, then press orkey to enter SWITCH SETTING page

2. Press ⇩ or ⇧ key to move the cursor to the item to be set

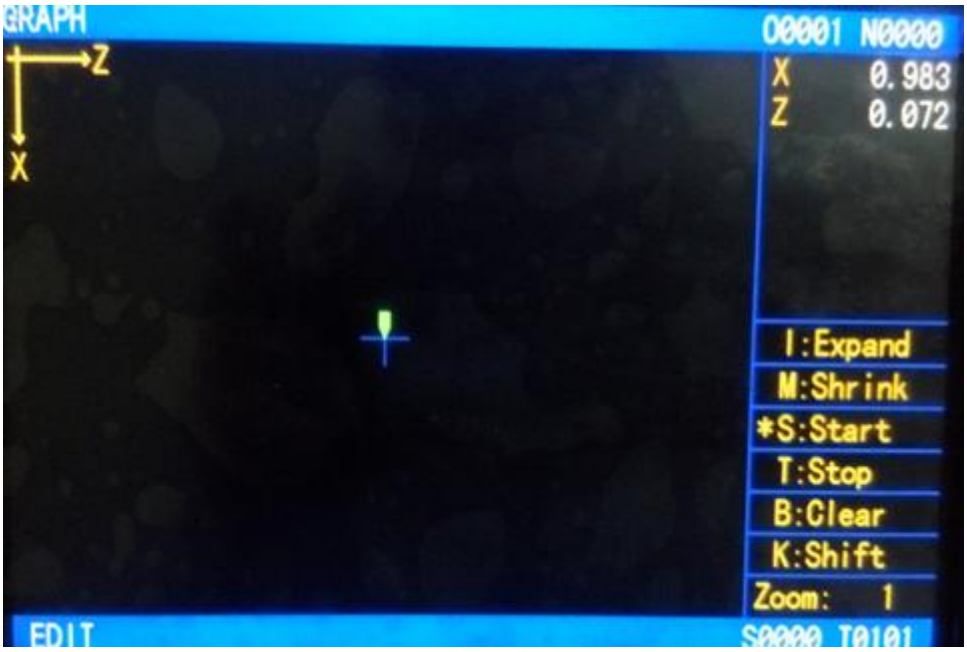
3. Press U W and R V key to shift the ON-OFF state, press U W key, “*”moves to the left to set the switch for OFF, press R V key, “*”moves to the right to set the switch for ON.

Only the PARM SWT is set to ON, could the parameter be altered; so only PROG SWT is set to ON, could programing; only auto-sequece switch opened, program edit.

Note: When parameter switch is shifted from “off”to“on”for the first time, CNC alarm occurs. Press 取消 CAN first and press RESET later then the alarm can be eliminated. For security, set parameter switch to “off” after parameter alteration is finished.

9.1.2 Graphic setting

Press RESET key twice to enter trajectory page.



The meaning of graphic parameter:

Coordinate system Settings: according to difference of before and after the knife holder coordinate system and so on, ECN*000M series controller 6 kinds of graphics display

A: Graphics trajectory of amplification, narrow

In a graphical display page, but by editing, key I A M [on the keyboard of real-time graphics trajectory amplification, narrow

B:Graphics trajectory according to start and stop and remove

In a graphical track display page, press a key S, start drawing; Press a key T, stop drawing; Press a key J, remove the current graphics trajectory

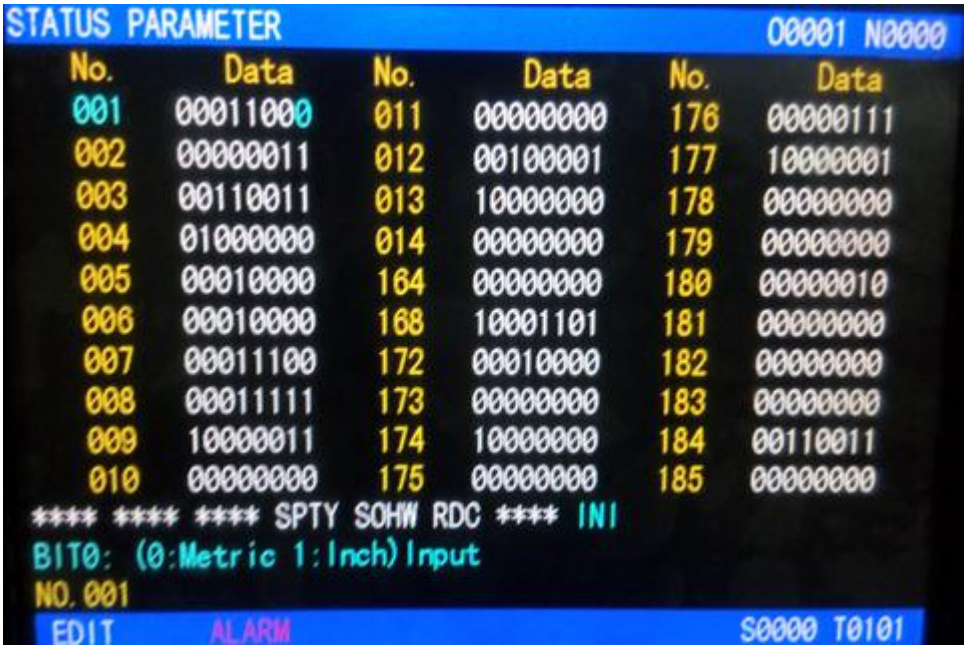
C: Graphics trajectory according to move

In a graphical track display page, mobile graphics trajectory can be realized by press the direction key.

9.1.3 Parameter setting

By the parameter setting, the characteristics of the drive unit and machine can be adjusted. See Appendix 1 for their significance

Press 参数 PAR key to enter the Parameter interface, then press or key to switch the parameter page, the figure is as picture show:



A. Alteration of the state parameter

1. Byte alteration

1) Turn on the parameter switch

2) Enter the MDI mode

3) Move the cursor to the parameter No. to be set

Method 1: Press or **key to enter the page containing the parameter to be set, press** or **key to move the cursor to the No. of the parameter to be set;**

Method 2: Press address key P, key in parameter No, then press 输入 IN key.

4) Key in the new parameter value;

5) Press 输入 IN key, the parameter value is entered and displayed;

6) For security, the PARM SWT needs to be set to OFF after all parameters setting is finished

2. Bit alteration:

1) Turn on the parameter switch;

2) Enter the MDI mode;

3) Move the cursor to the No. of the parameter to be set;

Method 1: Press 输入 IN or 输出 OUT key to enter the page of the parameter to be set, press ⇩ or ⇧ key to move the cursor to the No. of the parameter to be set

Method 2: Press address key P key in parameter No., then press 输入 IN key.

4) Press 转换 CHG key to skip to a bit of the parameter, and the bit is backlighted. Press ⇨ or ⇦ key to move the cursor to the bit to be altered, then key in 0 or 1 according to request;

5) After all parameters setting is finished, the PARM SWT needs to be set for OFF for security.

Note: After entering a bit of the parameter, press 转换 CHG key and it may skip out of the bit and into the parameter switch.

B: Alteration of the data parameter, pitch compensation

1. Data parameter alteration

1) Turn on the parameter switch;

2) Enter the MDI mode;

3) Move the cursor to the No. of the parameter to be set;

4) Key in the new parameter value;

5) Press 输入 IN key, the value is entered and displayed;

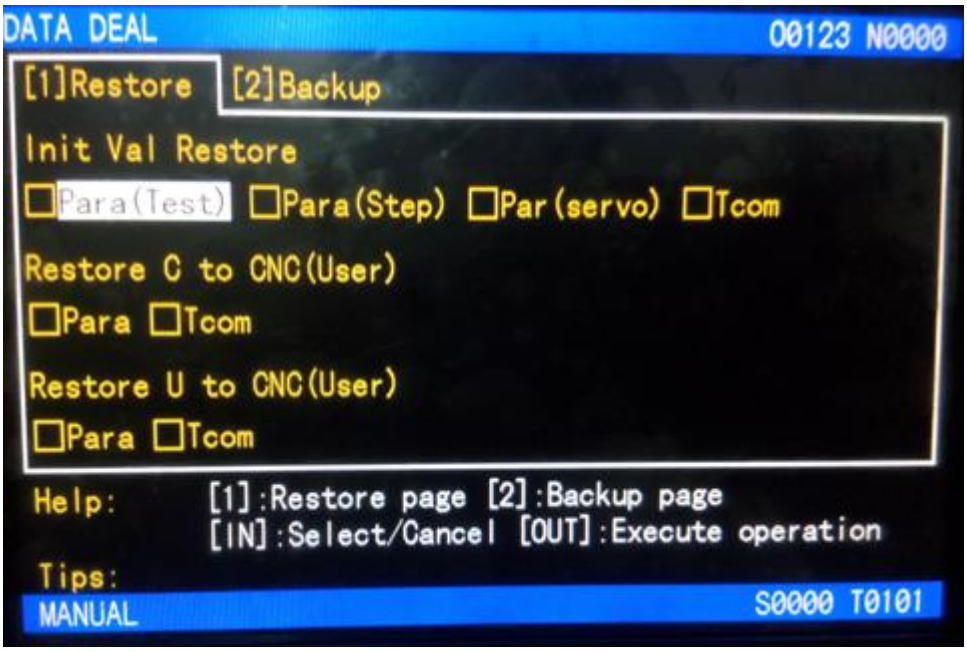
6) After all parameters settings arefinished; the PARM SWT needs to be set to OFF for security.

Explain: pitch data must be alterated in the two levels operation under the permission.

9.2 Restore and backup of data

The user data (such as state parameter, data parameter and screw data)of MST-81 seriescan be backup (stored) and

recovery (read), it doesn't affect the parts program which stored in the CNC. Data operation page display as below:



1. Turn on the parameter switch;

2. Press MDI press to input operaton mode, press 设置 SET (press key or when it is necessary) enter data operation page;

3. Press 2 enter data backup page, press 1 enter data recovery page;

4. Move cursor to the operation options and press 输入 IN to enter data recovery page;

5. Press excute all of the selected operation.

Note 1: Please don't cut off the power when backup or restore data, and it'd better not to do other operation until it finished.

Note 2: Both the 3 level and above password level users can backup and restore state parameter, data parameter and screw parameter.

9.3 The pemission setting and alteration

To prevent the processing programs, CNC parameters from malignant alteration, this ECN*000M series provides an authority setting function that is graded for 4 levels. By decending sequence, they are machine builder (2nd) level, equipment management (3rd) level, technician (4th) level, machining operation (5th) level.

The 2nd level: Modification of the CNC bit parameter, data parameter, pitch data, tool offset data, part program edit, PLC ladder transmission etc. are allowed;

The 3rd level: initial password 12345, the CNC bit parameter, data parameter; tool offset data, part program edit operations are allowed;

The 4th level: initial password 1234, tool offset data (for tool setting), macro variables, part program edit operations are allowed; but the CNC bit parameter, data parameter, pitch data operations are unallowed.

The 5th level: no password. Only the machine panel operation is allowed, and the operations of part program edit and selection, the alteration operations of CNC bit parameter, data parameterpitch data, tool offset data are unallowed;



After entering the authority setting page, the cursor locates at the “INPUT PASSWORD:”line. It may press the ⇩ ⇧ key to move the cursor to the corresponding item.

a) Press ⇧ key once, the cursor shifts a line upward. If the current cursor locates at the “SET LOWER LEVEL”line (1st line), press ⇧ key, the cursor shifts to the “UPDATE PASS:”line (end line);

b) Press ⇩ key once, the cursor shifts a line upward. If the current cursor locates at the end line, by pressing ⇩ key once, the cursor moves to the 1st line.

9.3.1 Entry of the operation level

1. After entering the PASSWORD SETTING page, move the cursor to the “INPUT PASSWORD” line;

2. Key in the password (an “*”sign added each time inputting a character);

3. Press 输入 IN key to finish the inputting, and it will enter the corresponding password level;

Note: The length of this ECN*000M series system password corresponds to the operation level, which can't be added or decreased by user at will;

Operation level	Length of password data	Original password
3rd	5bits	12345
4th	4 bits	1234
5th	none	none

Example: The current CNC level is the 4th level, as the following page shows. The 3rd level password of CNC is 12345; please alter the current level to the 3rd level.

Move the cursor to the "INPUT PASSWORD:"line, key in 12345, then press the 输入 IN key, the CNC prompts "Modify parameter and edit program", "Password passed", and the current level is the 3rd level. The page is as follows:

Note: When current operation authority is lower than or equal to the 3rd level (3rd, 4th, 5th level), the password levelis not changed if repower the CNC system. If previous level is higher than the 3rd level (0, 1st, or 2nd level), it defaults the 3rd level.

9.3.2 Alteration of the Password

Steps for password alteration:

1. After entering the PASSWORD SETTING page, enter the password by the methods in Section10.3.2;

2. Move the cursor to the"ALTER PASSWORD:"line;

3. Key in the new password, and press 输入 IN key

4. The CNC system prompts "PLEASE INPUT USER PASSWORD AGAIN", the page is as follows:

5. After reinputting the password, press 输入 IN key, if the two passwords input are identical, CNC prompts "PASSWORD UPDATED". So the password alteration is successful.

6. If the two passwords input are not identical, CNC prompts "PASSWORD CHECKOUT ERROR.".

9.3.3Lower Level Set

The demotion of the operation level is used to enter a lower level from a higher level, the steps are as follows:

1. After entering the PASSWORD SETTING page, key in the password by the method in Section 10.3.2

2. Move the cursor to the"SET LOWER LEVEL"line, if the current CNC operation is the 3rd level, the page is as

follows:

3. Press 输入 IN key, the CNC prompts “CURRENT LEVEL TO 4, OK?” the page is as follows:

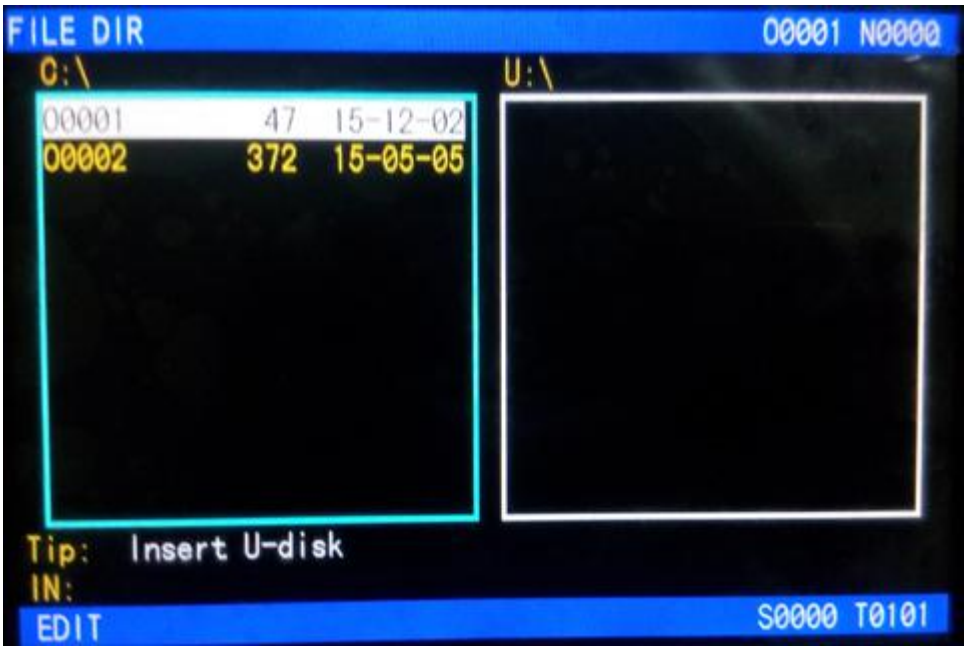
4. Press 输入 IN key again, if the demotion is successful, the page is as follows:

Note: If the current level is the 5th level, the demotion operation is unallowed.

9.4 Operation of U-disk

9.4.1 File content page

Press 程序 PRG key enter program page under un-edit state, press 程序 PRG key to enter [documentation content] interface, insert U disk as following picture:



It displayed CNC disk contents information at left of page and display USB disk contents information at right of page. It only displays “.CNC”, “.NC” and “.txt” under U disk root directory.

CNC disk will switch to USB disk by press 转换 CHG cursor, press ⇩ ⇧ can move cursor.

9.4.2 File copy

Move the cursor to the file which needs to copy, press 输出 OUT to copy it under input mode.

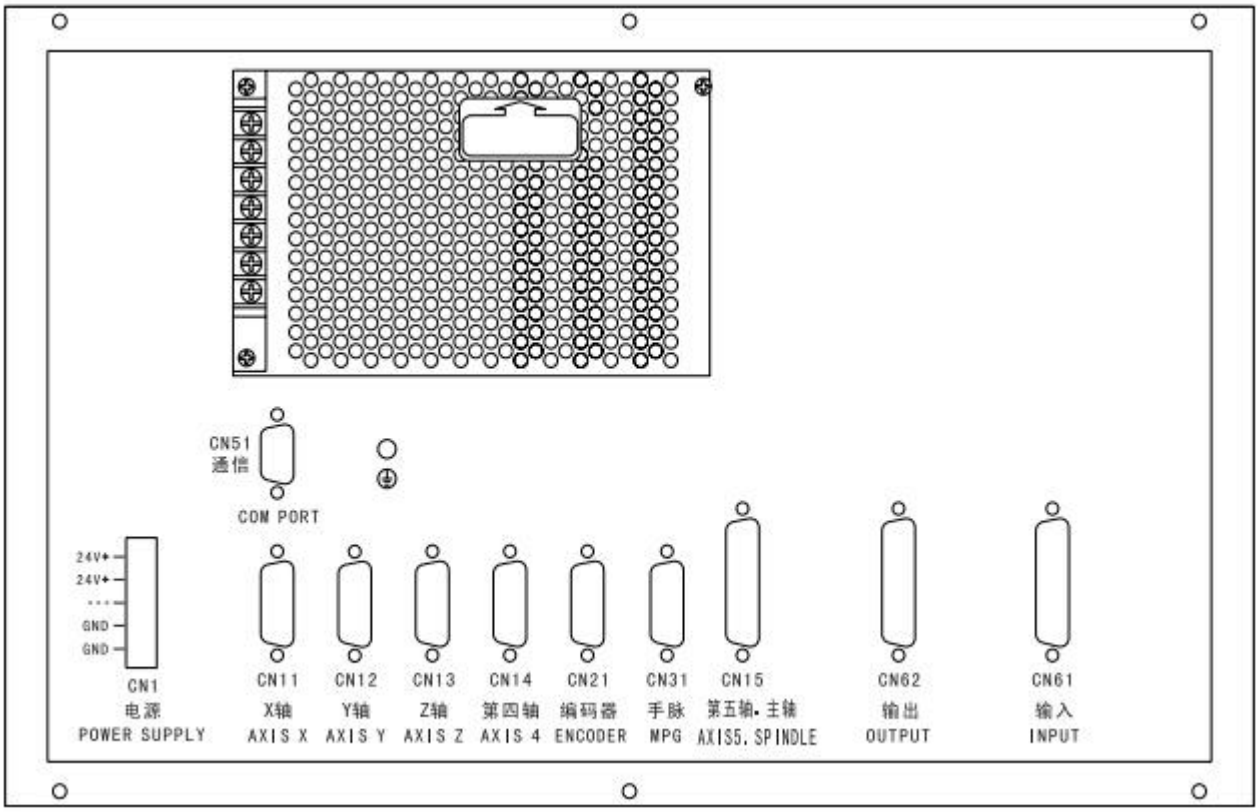
Volume 3

Installation and connection

CHAPTER 1 INSTALLATION LAYOUT

1.1 ECN*000M seriesConnection

1.1.1 ECN*000M series Back Cover Interface Layout



Picture 1-1 ECN*000M series back over interface layout

1.1.2 Interface Explanation

◎ Power box:+24V, GND power supply;

◎Filter (optional): Input AC 220C power supply, PE connects ground, output connect L, N of power supply box;

◎ CN1: Power supply port;

◎ CN11: X axis, 15-core D hole socket, for connecting X axis drive unit;

◎ CN12: Y axis, 15-core Dholesocket, for connecting Y axis drive unit;

◎ CN13: Z axis, 15-core Dholesocket, for connecting Z axis drive unit;

◎CN14: 4th axis，15-core D holesoket, for connecting 4th axis drive unit;

◎ CN15: Spindle axis, 25-core Dhole socket, for connecting spindle drive unit;

◎ CN21: Coder, 15-core Dholesocket, for connecting Encoderd;

◎ CN31: Handlewheel, 26-core Dholesocket, for connecting Encoderd;

◎ CN51: Communication, 9-core Dholesocket, for connecting pc RS232 interface;

◎ CN61:input, 44-core D hole socket, for connecting machine tool input;

◎ CN62: ouput，44-core D hole socket, for connecting machine tool output;

1.2 ECN*000M series Installation

1.2.1 External Dimensions

Please refer the appendix 1 and 2.

1.2.2 Installation Conditions of the Cabinet

◎The dust, cooling liquid and organic resolution should be effectively prevented from entering the cabinet;

◎The designed distance between the CNC back cover and the cabinet should be not less than 20cm, the inside and outside temperature difference of the cabinet should be no les than 10℃ temperature rises when the cabinet inside temperature rises;

◎ Fans should be fixed in the cabinet to ventilate it;

◎The panel should be installed in a place where the coolant can't splash;

◎The external electrical interference should be taken into consideration in cabinet design toprevent it from transferring to CNC system.

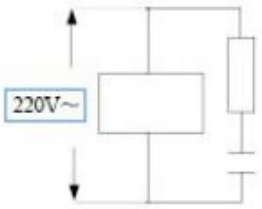
1.2.3 Protection Methods against Interference

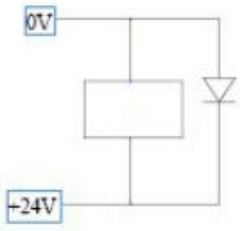
In order to ensure the CNC stable working, the anti-interference technology such as space electromagnetic radiation shielding, impact current absorbing, power mixed wave filtering are employed in CNC design.And the following measures are necessary during CNC connection:

1. Make CNC far from the interference devices (inverter, AC contactor, static generator, high-pressure generator and powered sectional devices etc.);

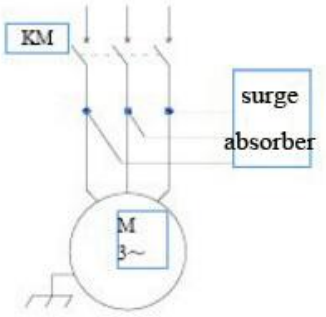
2. To supply the CNC via an isolation transformer, the machine with the CNC should be grounded; the CNC and drive unit should be connected with independent grounding wires at the grounding point;

3. To supress interference: connect parallel RC circuit at both ends of AC coil (Picture 1-3), RC circuit should approach to inductive loading as close as possible; reversely connect parallel freewheeling diode at both ends of DC coils (Picture 1-4); connect parallel surge absorber at the ends of AC motor coil (Picture 1-5);





Picture 1-3 **Picture 1-4**



Picture 1-5

4. To employ with twisted shield cable or shield cable for the leadout cable of CNC, the cable shield tier is grounded by single end at CNC side, signal cable should be as short as possible;

5. In order to decrease the mutual interference between CNC cables or CNC cables with strong-power cables, the wiring should comply with the following principles:

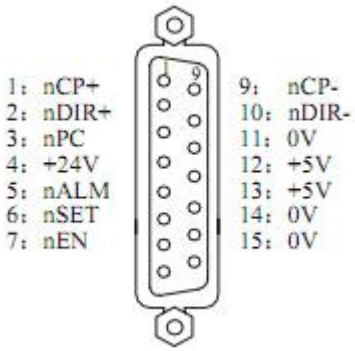
Group	Cable type	Wiring requirement
A	AC power line	Tie up A group cables with a clearance at least 10cmfrom that of B, C groups, or shield A group cables fromelectromagnetism
	AC coil	
	AC contactor	
B	DC coil (24VDC)	Tie up B and A group cables separately orshield B group cables; and the further B group cablesare from that of C group, the better it is
	DC relay (24VDC)	
	Cables between CNC andstrong-power cabinet	
	Cables between CNC and machine	
C	Cables between CNCand servo drive unit	Tie up C and A group cables separately, orshield C groupcables; and the cable distance betweenC group and B group is at least 10cm withtwisted pair cable applied.
	Position feedback cable	
	Position encoder cable	
	MPG cable	

	Other cables for shield	

CHAPTER 2 EFINITION&CONNECTION OF INTERFACESIGNALS

2.1 Connection to Drive Unit

2.1.1 Drive Interface Definition

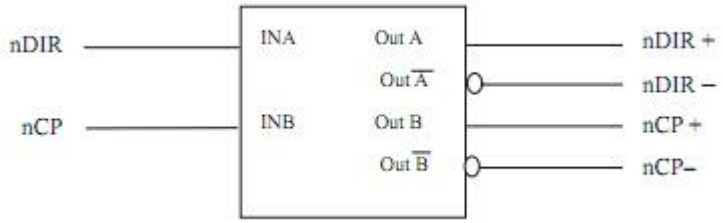


Signal	Explanation
CPn+, CPn-	Command pulse signal
DIRn+, DIRn-	Command direction sigal
PCn	Zero signal
ALMn	Drive unit alarm signal
ENn	Axis enable signal
SETn	Pusle disable signal

Picture 2-1 CN11, CN12, CN13, CN14 (15 core D hole socket)

2.1.2 Command Pulse and Direction Signals

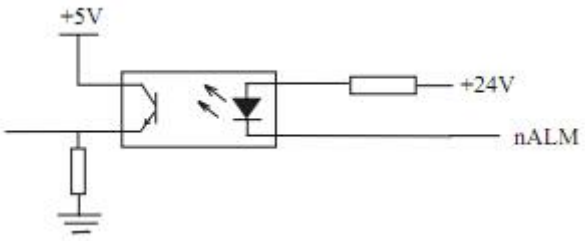
nCP+, nCP- are command pulse signals, nDIR+, nDIR- are command direction signals. These two group signals are both difference output (AM26LS31), the interior circuit for them is shown in Picture 2-2.



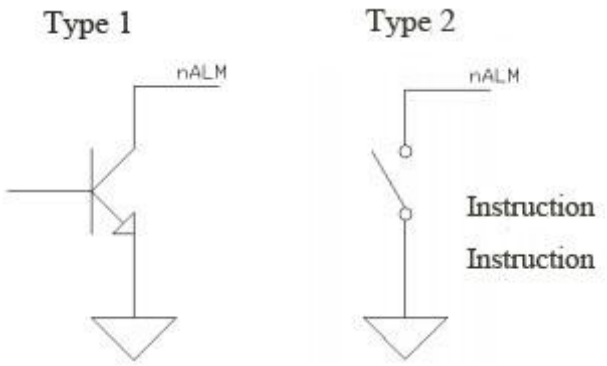
Picture 2-2 Interior circuit of command pulse and direction signals

2.1.3 Drive Unit Alarm Signal

The low or high level of the drive unit alarm level is set by the CNC bit parameter No.009 BIT0 ~ BIT4 , whose interior circuit is shown in Picture 2-3:



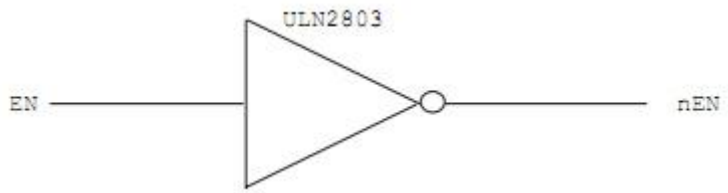
This input circuit requires that the drive unit transmits signal by the following types in Picture 2-4:



Picture 2-4 Signal types of drive unit

2.1.4 Axis Enable Signal ENn

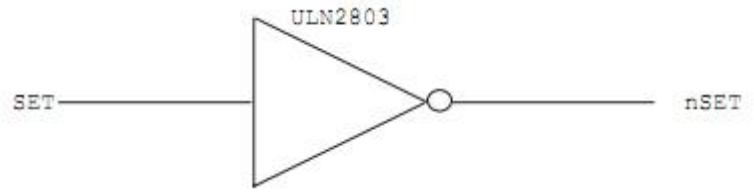
nEN signal output is valid as CNC works normally (nEN signal to 0V); when the drive unit alarm or emergency alarm occurs, CNC cuts off nEN signal output (nEN signal to0V off). The interior interface circuit is shown in Picture 2-5:



Picture 2-5 interior interface circuit for axis enable signal

2.1.5 Pulse Disable Signal SET

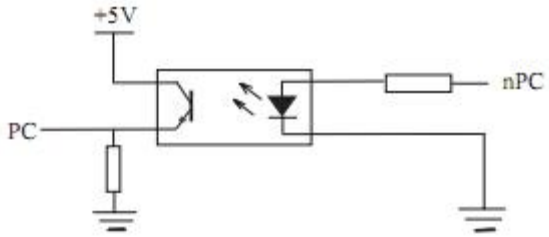
nSET signal is used to control servo input disable which can enhance the anti-disturbance capability between CNC and drive unit. This signal is at low level if there is pulse output from CNC, high resistance if not. The interior interface circuit of it is shown in Picture 2-6:



Picture 2-6 Interior interface circuit for pulse disable signal

2.1.6Zero Signal nPC

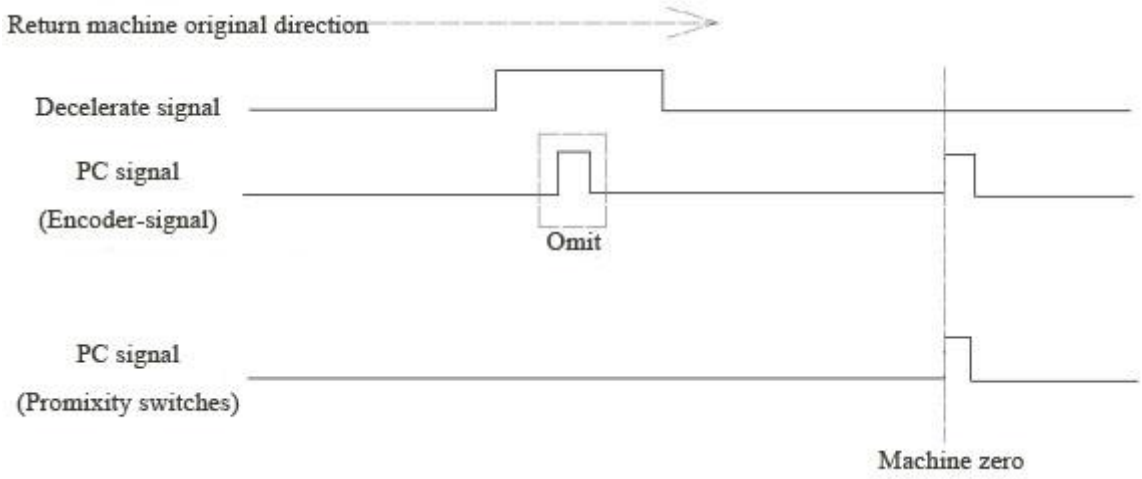
The one-rotation or approach switch signal is taken as zero signals for machine zero return. Its interior connection circuit is shown in Picture2-7.



Picture 2-7 Zero signal circuit

Note: nPC signal uses +24V level.

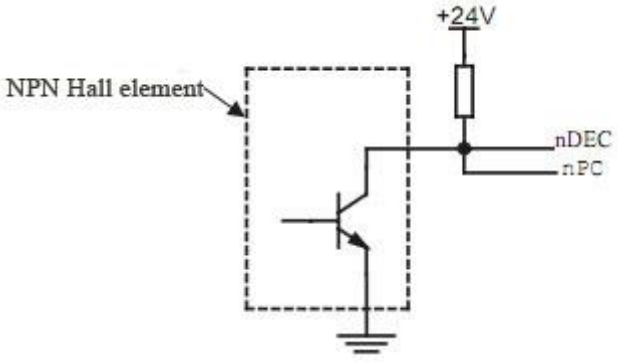
a) The wave shape that supplied by users should be like aspicture 2-8:



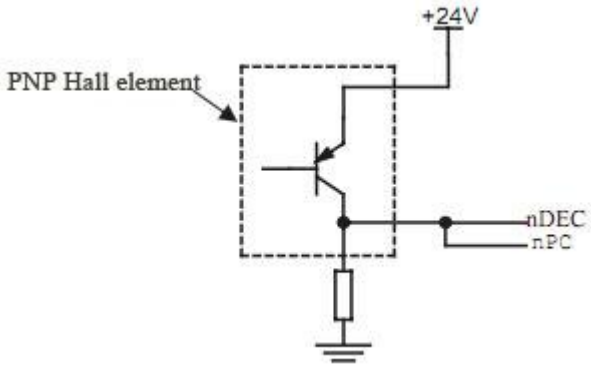
Picture 2-8 PC signal wave shape figure

Note: when machine tool backs to zero, CNC judges reference point though detect PC signal's jump after decelerate switch triped out, both rise edge and falling edge are valid.

b) The connection for NPN Hall elements taken as both deceleration signal and zero signals is shown as below:



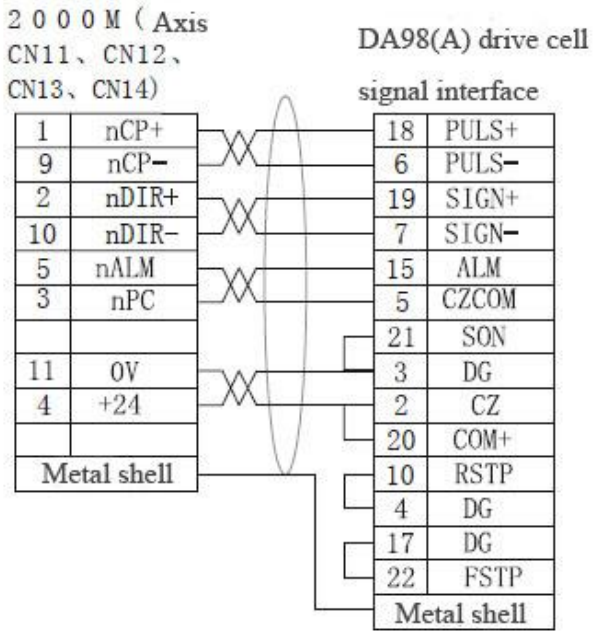
c) The connection for PNP Hall elements taken as both deceleration signal and zero signals is shown as below:



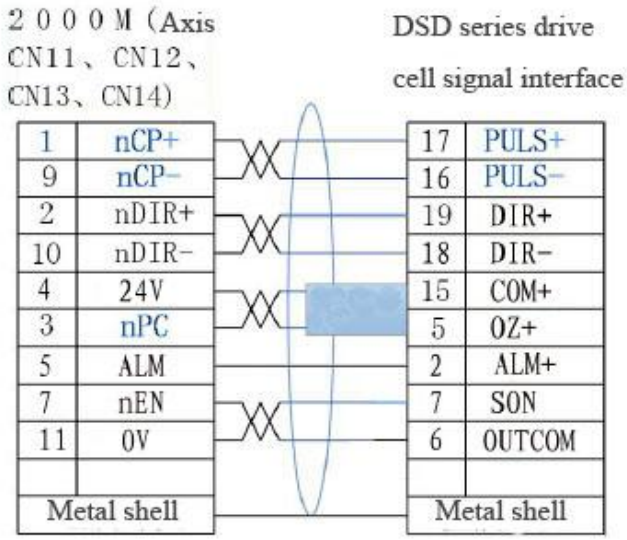
2.1.7 Connection to Drive Unit

The connection of ECN*000M series to drive unit is shown in picture 2-10:

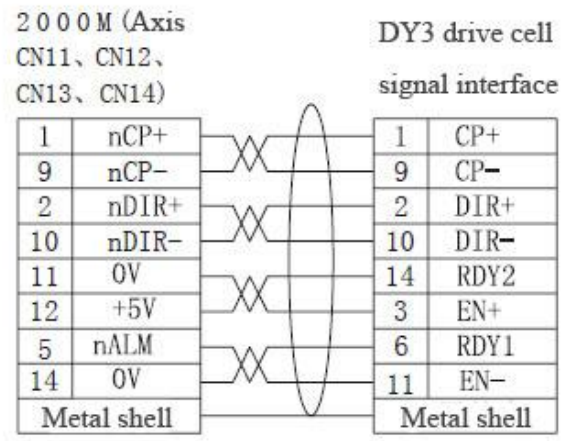
The connection of ECN*000M series andDA98(A) drive cell



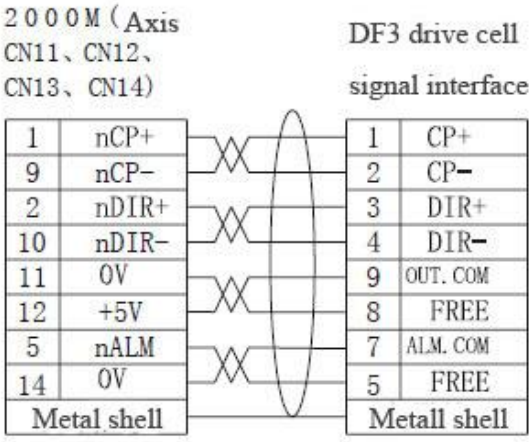
The connection of ECN*000M seriesand DSD drive cell



The connection ofECN*000M series andDY3 (A) drive cell

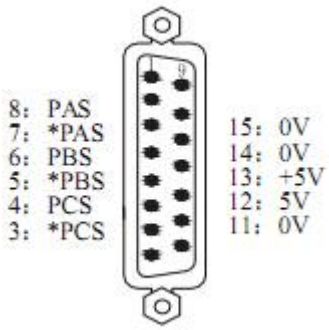


The connection ofMST-81 seriesand DF3drive cell



2.2 Connection of spindle axis encoder

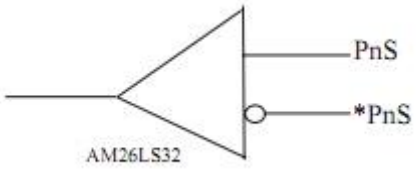
2.2.1 Spindle axis encoderinterface definition



Name	Explanation
***PAS/PAS**	**Encoder A phase pulse**
***PBS/PBS**	**Encoder B phase pulse**
***PCS/PCS**	**Encoder C phase pulse**

*PCS/PCS, *PBS/PBS, *PAS/PAS are differential input signal of C phase, B phase and A phase, adopts 26LS32 receives; *PAS/PAS, *PBS/PBS are orthogonality wave which differs 90, max sign frequency<1MHz; used encoder wire numbers are setting by parameter (range 100~5000)

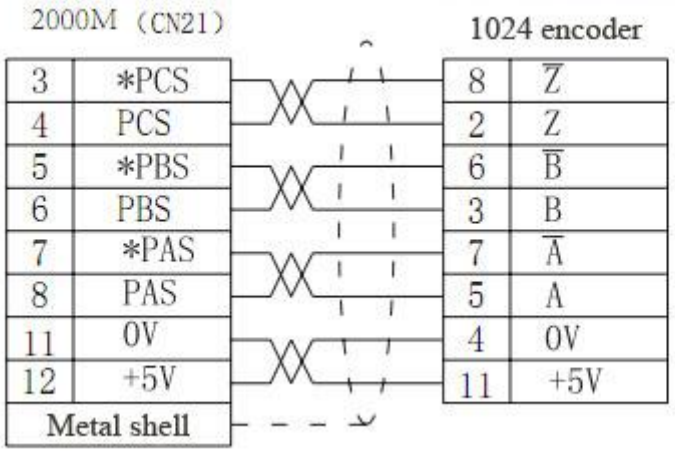
Internal connection circuit as picture 2-13 (n=A, B, C in picture)



Pciture 2-13 Encoder signal circuit

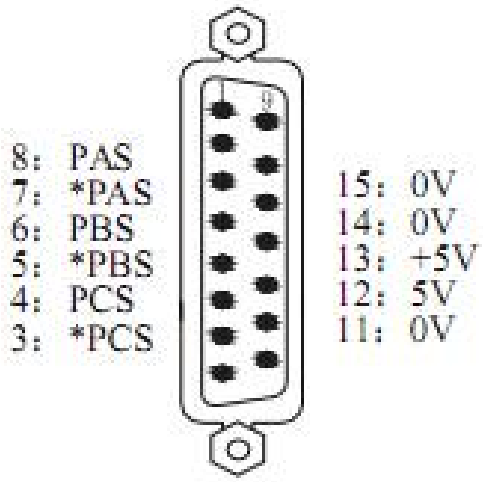
2.2.3 Connection of spindle encoder port

The connection of ECN*000M series and spindle encoder as picture 2-14, it adopts twisted-pair. (Take an example of ZLF-12-102.4BM-C05D)



2.3 Connection to Handwheel

2.3.1 Handwheel Interface Definition

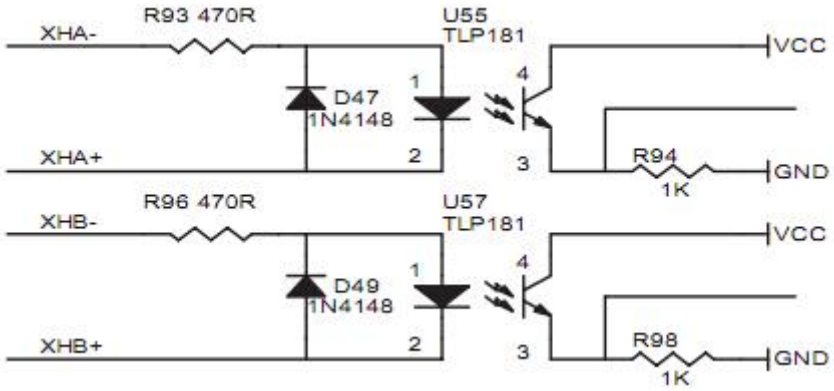


Signal	Explanation
HA+, HA-	Handwheel A phase signal
HB+, HB-	Handwheel B phase signal
X5.0	MPG X axis selection
X5.1	MPG Y axis selection
X5.2	MPG Z axis selection
X5.3	The 4th axis selection
X5.4	The 5th axis selection
X5.5	Increment *1
X5.6	Increment *10
X5.7	Increment *100
+24V	DC power supply
VCC, GND	

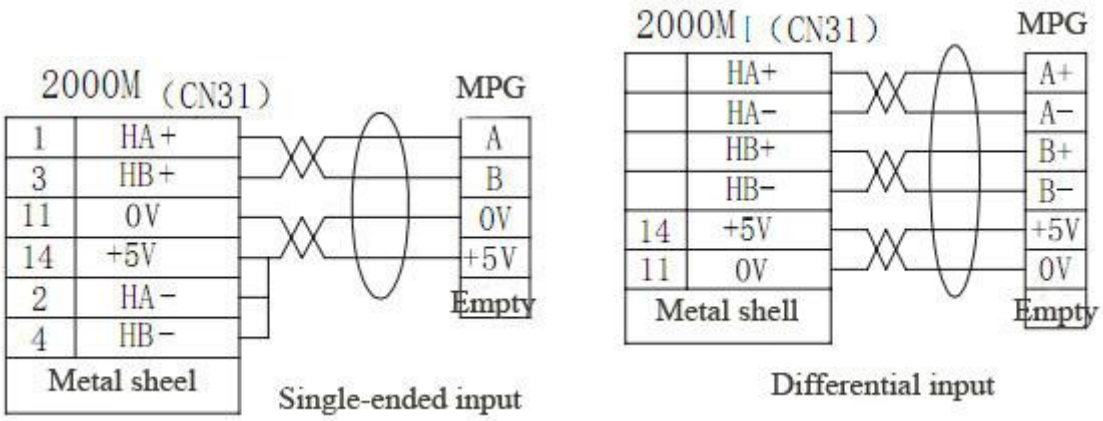
Picture 2-15 CN31 encoder port (26 cores DB type socket)

2.3.2 Signal explanation

HA. HB areA phase and B phase input signals respecitively. Internal connection circuit as picture 2-16 shows:



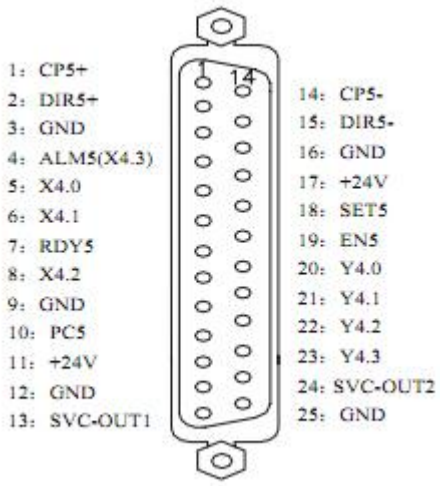
The connection of ECN*000M series and MPG as picture 2-17 shows:



Picture 2-17 the connection of ECN*000M series and MPG

2.4 Connection to Spindle Encoder

2.4.1 Spindle Encoder Interface Definition



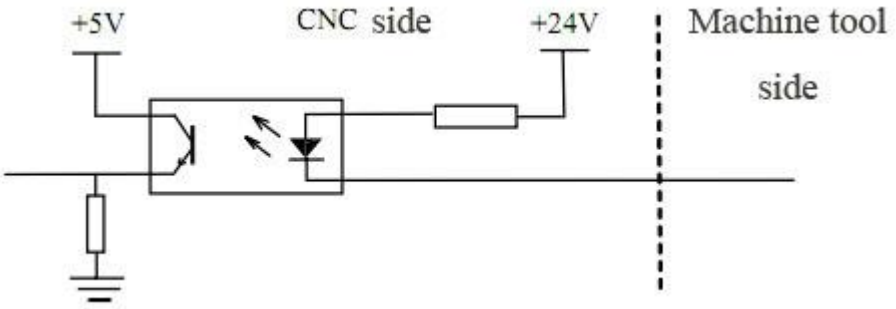
CP5+, CP5-	Spindle pulse signal
DIR5+, DIR5-	Spindle direction signal
ALM5	Spindle alarm signal
RDY5	Spindle already signal
PC5	Spindle zero signal
SVC-OUT1	Analog voltage output 1
SVC-OUT2	Analog voltage output 2
SET5	Spindle seting signal
EN5	Spindle enable signal
X4.0~X4.3	PLC address, only the low level is vaild
Y4.0~4.3	PLC address

Picture 2-18 CN15 spindle port (25 core DB hole)

Note 1: PC5 and 0V break over is valid, it is different from other feeding axis (breakover of CN11~CN14 axis port's PC and +24 is valid)

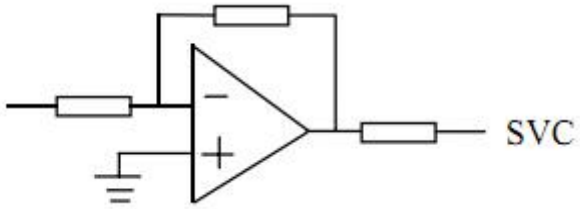
Note 2: X4.0 ~X4.3 is breakover with 0V and is different from other input signal (the breakover of other input signal is valid)

Note 3: The signal internal circuit of PC5, X4.0 ~ X4.3 as below:



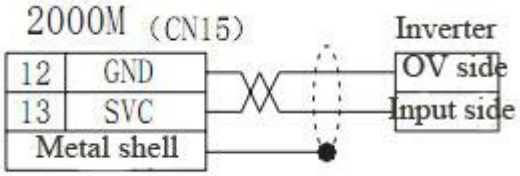
2.4.2 Connection of gerrnal inverter

Analog spindle port SVC port can output 0 ~ 10V voltage, internal circuit of signal as picture 2-20:



Picture 2-20 SVC signal circuit

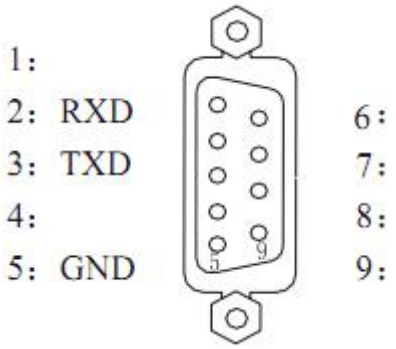
The connection of ECN*000M series and inverter as below picture 2-21:



Picture 2-21 the connection of ECN*000M series and inverter

2.5 Connection of ECN*000M series to PC serial port

2.5.1 Communication Interface Definition

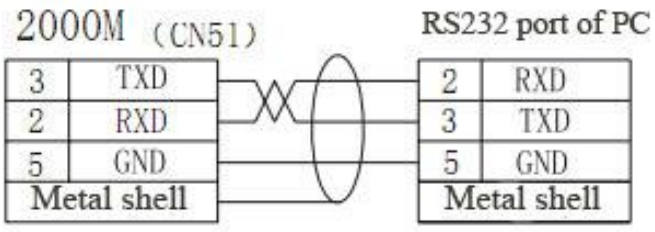


Signal	Explanation
RXD	Receive data
TXD	Send data
GND	Signal grand

Picture 2-22 CN51 communication port (9 holes)

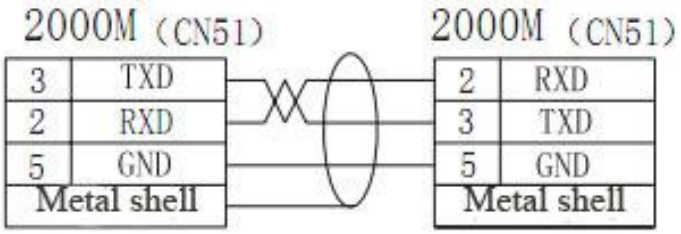
2.5.2 Communication Interface Connection

The communication between ECN*000M series and PC can be done via RS232 interface (ECN*000M series communication software needed);the connection of them is shown in picture 2-25:



Picture 2-25 connection of ECN*000M series to PC

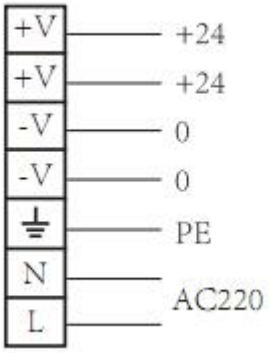
The communication of a ECN*000M series to another ECN*000M series can be made via their CN51 interfaces, and the connection of them is shown in picture 2-26:



Picture 2-26 connection of ECN*000M series and ECN*000M series

2.7 Connection of Power Interface

Power box has already connected with CN1 port before delivery; users only need to connect an external 220V AC power.

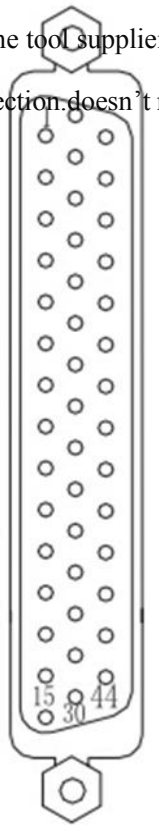


Picture 2-24 system power port CN1

2.7 I/O Interface Definitions

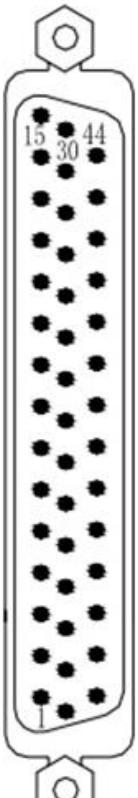
ECN*000M series milling machine tool CNC doesn't mark the fixed address I/O function meaning definied by PLC program (ladder figure), when ECN*000M series CNC assembles machine tool, I/O function decided by machine tool supplier, please refer user manual of machine tool supplier.

This section doesn't mark fixed address I/O function is descriped by ECN*000M series standard PLC program.



Subscript	Address	Function	Explanation
21 ~ 24	0V	Power interface	Power 0V side
18 ~ 20 25 ~ 28	empty	empty	empty
1	X0.0	SAGT	Protective door detection signal
2	X0.1	SP	External pause
3	X0.2	THAN	External tool clamp/unclamp
4	X0.3	DECX	X axis decelerate signal
5	X0.4		Reserve
6	X0.5	KYP	Emergency stop key
7	X0.6	LIMU	Overtravel release signal
8	X0.7	PRKY	Air pressure detect input signal
9	X1.0	TOPE	Spindle tool unclamp on position signal
10	X1.1	TCL0	Spindle tool clamp on position signal
11	X1.2	TZER	Tool changer homing key
12	X1.3	DECZ	Z axis decelerate signal
13	X1.4	ST	External cycle start signal
14	X1.5	M41I	Spindle the 1st gear on position signal
15	X1.6	M42I	Spindle the 2nd gear on position signal
16			
29	X2.0	TFRX	Tool changer forward on position signal

30	X2.1	TBAX	Tool changer backward on position signal
31	X2.2	TCUX	Tool changer tool counter signal
32	X2.3	DECY	Y axis decelerate signal
33	X2.4	DEC4	The 4th axis decelerate signal
34	X2.5	TZEX	Tool changer zero position signal
35	X2.6	TRSW	Current tool detect signal
36	X2.7	TMSW	Spindle tool detect switch signal
37	X3.0	LMIX	X axis overtravel input
38	X3.1	LMIY	Y axis overtravel input
39	X3.2	LMIZ	Z axis overtravel input
40	X3.3	TCW	Tool changer reversal key signal
41	X3.4	TCCW	Tool changer forward key signal
42	X3.5	SKIP	C31 skip signal
43	X3.6	TFRX	Tool changer forward key signal
44	X3.7	TBAX	Tool changer backward key signal



Subscript	Address	Function	Explanation
17 ~ 19 26 ~ 28	0V	Power interface	Power 0V side
20 ~ 25	+24V	Power interface	Power 24V side
1	Y0.0	COOL	Coolant output
2	Y 0.1	M32	Lubrication output
3	Y 0.2	TCLA	tool unclamp/clamp
4	Y 0.3	M03	Spindle anticlockwise rotation (forward)
5	Y 0.4	M04	Spindle clockwise rotation (reversal)
6	Y 0.5	M05	Spindle stop
7	Y 0.6	SCLP	Spindle clamp
8	Y 0.7	SPZD	Spindle brake
9	Y 1.0	S1/M41	Spindle mechanical gear output 1

10	Y 1.1	S2/M42	Spindle mechanical gear output 2
11	Y 1.2	S3/M43	Spindle mechanical gear output 3
12	Y 1.3	S4/M44	Spindle mechanical gear output 4
13	Y 1.4		Reserve
14	Y 1.5		Reserve
15	Y 1.6		Reserve
16	Y 1.7		Reserve
29	Y 2.0	TLP	Spindle tool unclamp indicator
30	Y 2.1		Reserve
31	Y 2.2	CLPY	Three - color light – yellow light
32	Y 2.3	CLPG	Three – color light – green light
33	Y 2.4	CLPR	Three – color light – red light
34	Y 2.5		Reserve
35	Y 2.6		Reserve
36	Y 2.7		Reserve
37	Y 3.0	STAO	Spindle orientation
38	Y 3.1	TCCY	Tool changer CW
39	Y 3.2	TCWY	Tool changer CCW
40	Y 3.3	TFRY	Tool changer forward
41	Y 3.4	TBAY	Tool changer bcakward
42	Y 3.5	TBAL	Tool changer backward indicator light
43	Y 3.6		Reserve
44	Y 3.7		Reserve

Note 1: some input, output ports can definit multiply functions, it is "/" in the above form;

Note 2: When output is active, the output port is 0V. If output function is inactive, the output signal is cut off by high impendance;

Note 3: If input signal and+24V breakover, the input is active. When the input signal disconnect to +24V,input is inactive.

Note 4: The +24V, COM are same as ECN*000M series power supply's terminals that have the same name;

2.7.1 Input signal

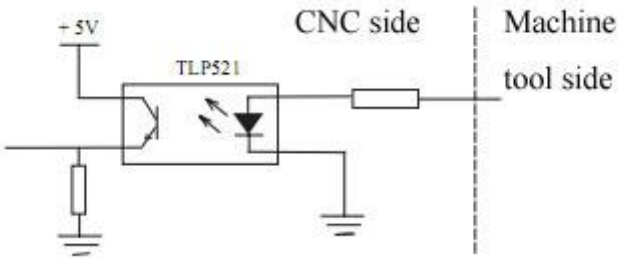
Input signal means the signal from machine to CNC, when this signal connect with +24V, the input is valid; when it disconnect with +24V, the input is invalid. The contact point of input signal at machine side should meet the following conditions:

The capacity of the contact point: DC30V, 16mA above

Leakage current between contact points in open circuit: 1mA below

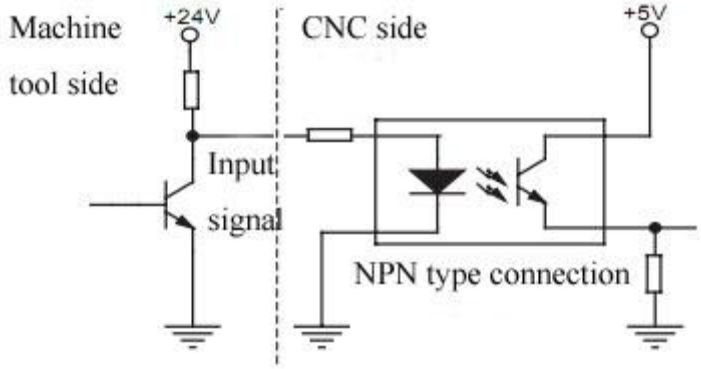
Voltage drop between contact points in closed circuit: 2V below (current 8.5mA, including cable voltage drop)

There are two external input types for input signals: one type is input by contactor whose signals are from key, stroke switch and contacts of relay at machine side, as is shown in picture 2-27:

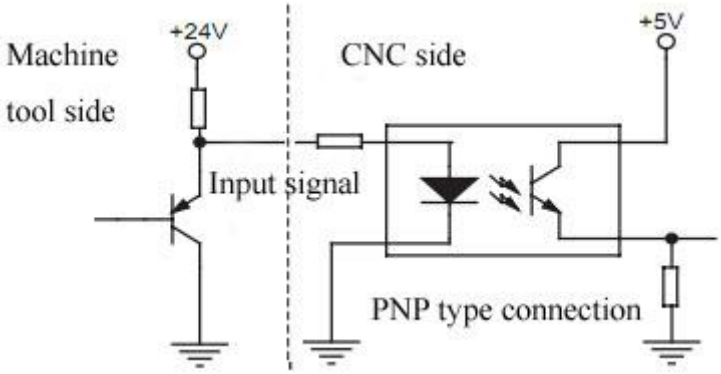


Picture 2-27

The other type is input by switch with no contactor (transistor), as is shown in Fig. 2-28A, 2-28B.



Picture 2-28A NPN type connetion

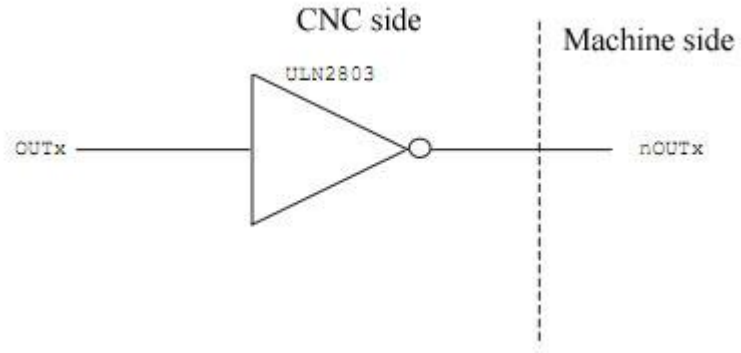


Picture 2-28B PNP type connetion

2.7.2 Output signal

The output signal is used for the machne relay and indicator, if it is on with 0V, the output function is valid; if it is off with 0V, the output function is invalid. There are total 36 digital volume outputs in

I/O interface that they all have the same structure as is shown in Fig.2-29:

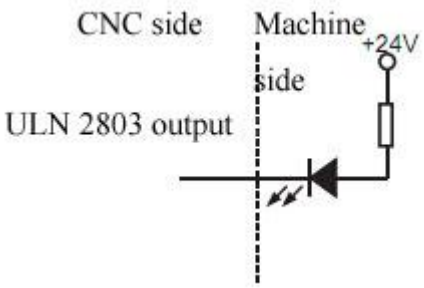


Picture 2-29 Circuit for digital volume output module

The logic signal OUTx output from the main board is sent to the input terminal of inverter (ULN2803) via a connector. And there are 2 output types for nOUTx: output with 0V, or high impedance. Its typical application is shown in follows:

- To drive OLED

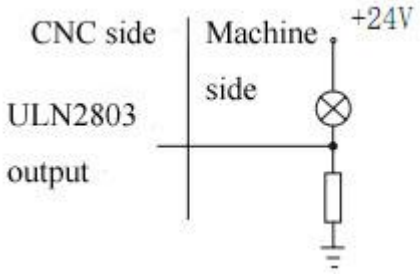
A serial resistance is needed to limit the current (usually 10mA) that goes through the OLED by sing ULN2803 output to drive LED, which is shown in picture 2-30.



Picture 2-30

- To drive filament indicator

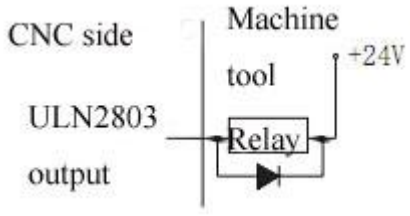
An external preheat resistance is needed to decrease the current impact at power on by using ULN2803 output to drive filament indicator, and this resistance value should be within a range that the indicator cann't light up. It is shown in picture 2-31:



Picture 2-31

- To drive inductive load (relay etc.)

To use ULN2803 output to drive an inductive load, it requires connecting a freewheeling diode near the coil to protect output circuit and deduce interference. It is shown in picture 2-32:



Picture 2-32

The meanings of output signal in I/O port definided by PLC program, output signals which defineded by standard PLC program includes S1 ~ S4 (M41 ~ M44), M3 ~ M5, M8 and M32 etc signals.

2.8 I/O function and connection

Attention!

I/O function are definied by PLC program (ladder figure), when ECN*000M series CNC assembles to CNC

machine, I/O function decided by machine manufacturer, please refer to user manual of machine manufacturer.

I/O functions descriped in this manual are standard PLC program.

2.8.1 Travel limit and scram

Relative signal

KYP: scram signal, stop alarming when disconnect with +24V;

LMIX: X axis travel limit detect input

LMIY: Y axis travel limit detect input

LMIZ: Z axis travel limit detect input

Datadignose

0	**0**	**0**		**ESP**							
Interface pin				**CN61.6**							

Signal dignose

Signal	KYP	LMIX	LMIY	LMIZ
Address dignose	X0.5	X3.0	X3.1	X3.2
interface pin	CN61.6	CN61.38	CN61.38	CN61.39

Control parameter

State parameter

0	**2**	**1**							**MESP**		

KYP=0: detect KYP scram signal

=1: don't detect KYP scram signal.

PLC state parameter

K	**1**	**0**		**LIMIT**	**LMIS**						

LMIT=1: each axis travel limit detection function is valid.

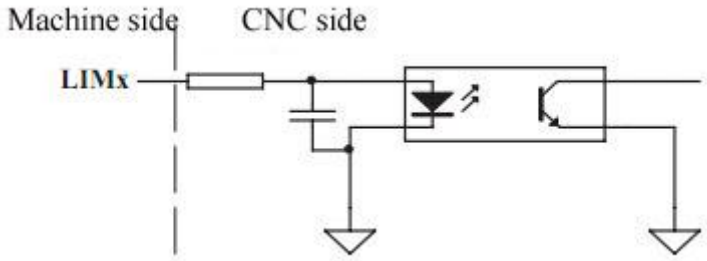
=0: each axis travel limit detection function is invalid.

LMI=1: overrun alarm when travel limit detect signal and +24V switch off.

=0: overrun alarm when limit detect signal and +24 breakover.

Signal connection

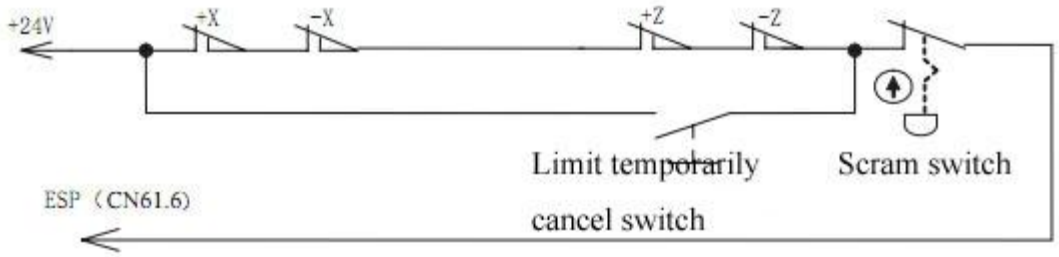
Overrun limit and scram signal (KYP) circuit as below picture 2-33:



Picture 2-33

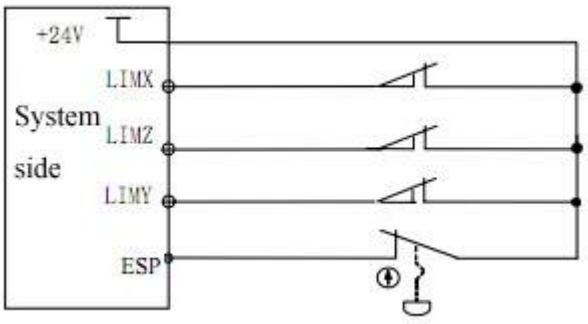
Machine external connection

① travel limit and scram connect in series, connection method as below picture2-34A:



Picture 2-34A scram, route switch connect in series

② travel limit and scram connect independent, connection method as below picture 2-34B:



Picture 2-34B scram, route switch connection in dependent

Control logic

① Travel limit and scram connect in series

CNC will appear “scram” alarm when appears overrun or press scram key, press overrun remove key if it is

overrun, press reset key cancel alarm and reversal move can relies the overrun. CNC stop pulse output when it appears scram alarm. Except above CNC processing functions, PLC program can also definit other functions when scram alarm. Standard PLC program definits functions: close M03 or M04, M08 signal output when scram alarm, and output M05 signal.

② Travel limit and scram connect independent

1.There is only one overrun touch point on each axis, it judges positive or negative overrun alarm though axis move direction.

2. It can be moved reversal when occur overrun alarm, move out of limit position and press reset to clear alarm.

Note: it must be sure that machine tool planker between positive and negative route before start overrun limit function, or the reminder alarm will be not unconformity with reality.

2.8.2 Machine zero

Relative signal

DECX: X axis deceleration signal;

DECY: Y axis deceleration signal;

DECZ: Z axis deceleration signal;

DEC4: The 4th axis deceleration signal;

DEC5: The 5th axis deceleration signal;

Data dignose

0	**0**	**0**					**DEC5**	**DEC4**	**DECZ**	**DECY**	**DECX**
Interface pins							**CN61.34**	**CN61.33**	**CN61.12**	**CN61.32**	**CN61.4**

Control parameter

K	**2**	**2**		***	***	***	**DEC5T**	**DEC4T**	**DECZ**	**DBCY**	**DECX**

DEC5T=0: 5th axis decelerates signal low level;

=1: 5th axis decelerates signal high level;

DEC4T=0: 4th axis decelerates signal low level;

=1: 4th axis decelerates signal high level;

DECZ=0: Z axis decelerates signal low level;

=1: Z axis decelerates signal high level;

DECY=0: Y axis decelerates signal low level;

=1: Y axis decelerates signal high level;

DECX=0: X axis decelerates signal low level;

=1: X axis decelerates signal high level;

0	**0**	**6**						**ZPLS**			**ZMOD**

ZMOD=1: zero mode choose front link stopper;

=0: zero modes choose after link stopper;

ZPLS=1: zero mode choice, there is a switch signal;

=0: zero mode choice, there is no switch signal;

0	**1**	**2**									**ISOT**

ISOT=1: at the time that after power on and after return machine tool zero point, manual rapidly move is vaild;

=0: at the time that after power on and after return machine tool zero point, manual rapidly move is not vaild;

0	**2**	**6**					**MZR5**	**MZR4**	**MZRY**	**MZRZ**	**MZRX**

MZRx=1: choose the zero returning direction is negative;

=0: choose the zero returning direction is positive;

Data parameter

0	**8**	**0**		**ZRNFL**

ZRNFL: low speed rate of machine zero turning

0	**7**	**0**		**ZRNFHX**

ZRNFHX:high speed machine zero turning of X axis

0	**7**	**1**		**ZRNFHY**

ZRNFHY: high speed machine zero turning of Y axis。

0	**7**	**2**		**ZRNFHZ**

ZRNFHZ: high speed machine zero turning of Z axis

0	**7**	**3**		**ZRNFH4**

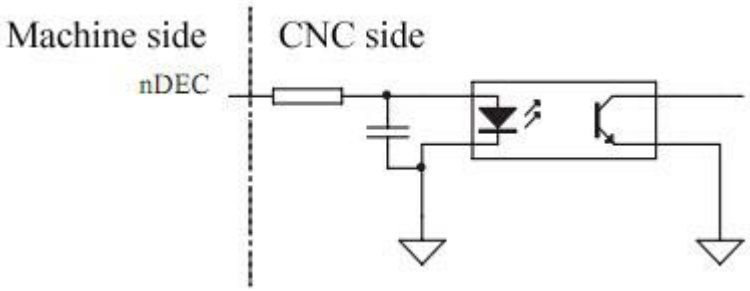
ZRNFH4: high speed machine zero turning of the 4th axis

0	**7**	**4**		**ZRNFH5**

ZRNFH5: high speed machine zero turning of the 5th axis

Signal connection

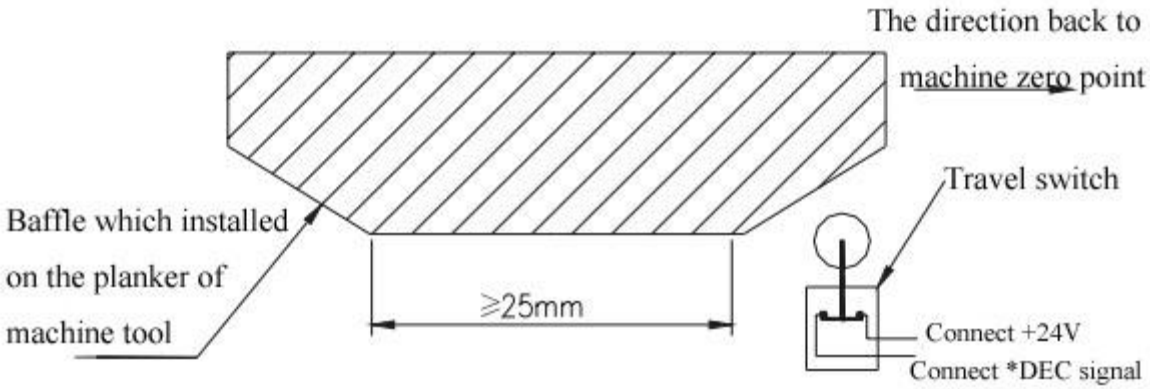
Deceleration signal internal connection circuit as below picture 2-38:



Picture 2-38

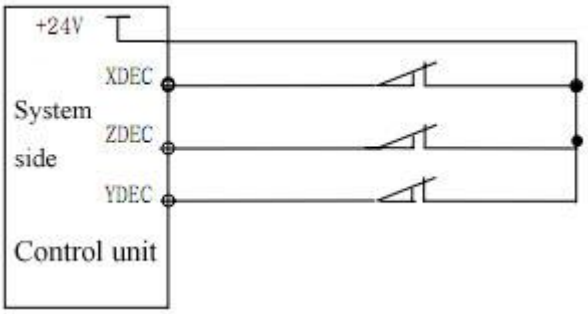
Machine zero return by regarding servo motor one-rotation signal as zero signals

① Its sketch map is shown as follows:



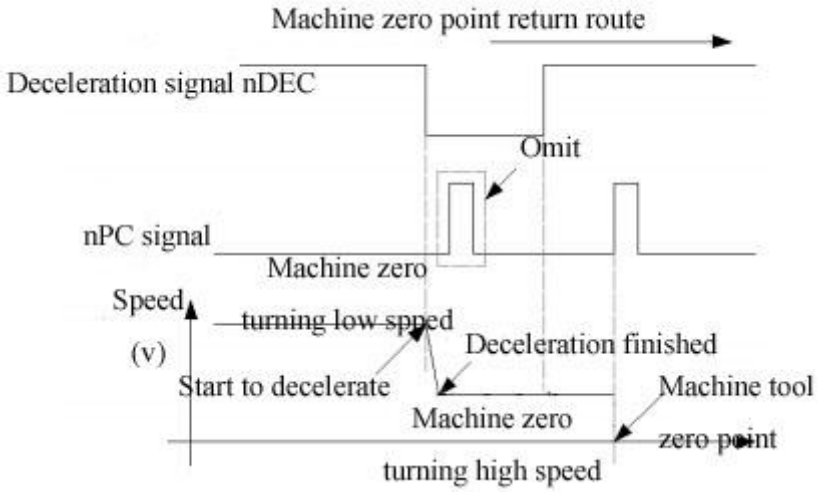
Picture 2-39

② The circuit of deceleration signal:



Picture 2-40

③ Action time sequence of machine zero return



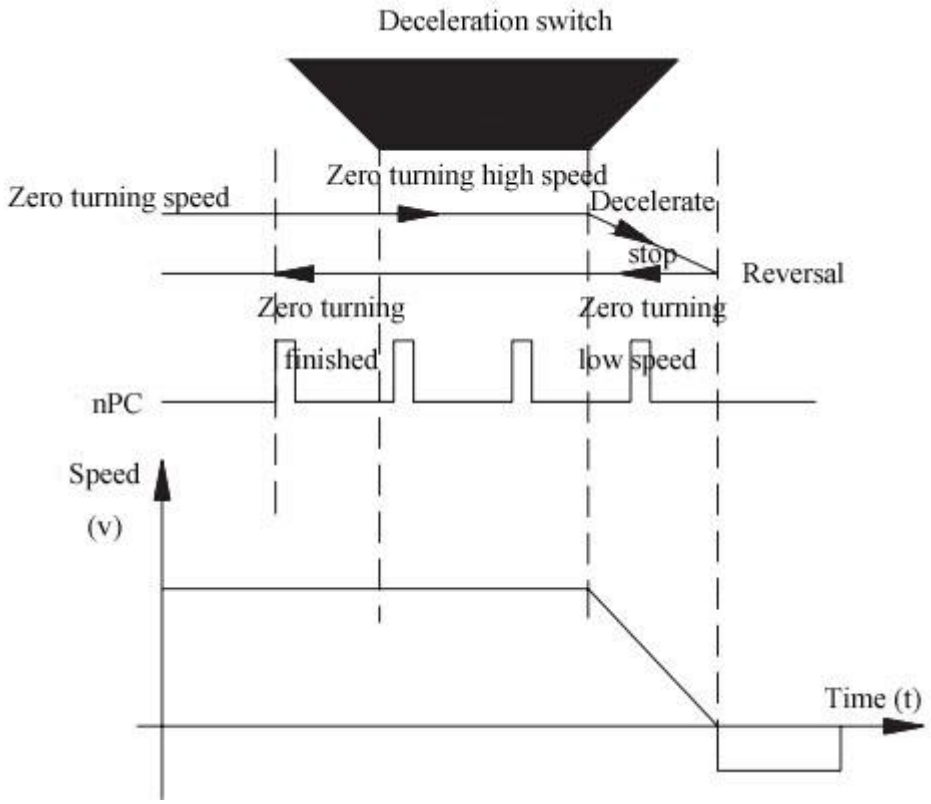
Picture 2-41-a

④ Machine zero return process

A: Select machine zero return mode, press the manual positive or negative feed key(machine zeroreturn direction is set by bit parameter No.026), the corresponding axis moves to the machine zero by a rapid speed (parameter No. 70 ~ 74). As the axis press down the deceleration switch to cut off deceleration signal, the feed slows down immediately, and it continues to run in a fixed low speed (parameter No. 80).

B: When the deceleration switch is released, the deceleration signal contact point is closed again. And CNC begins to detect the encoder one-rotation signal, if the signal level changes, the motion will be stoped.And thecorresponding zero indicator on the operator panel lights up for machine zero return completion.

When the BIT1 (ZMOD) of state parameter No.006 sets to be 1, BIT4 of K022 to be 0, choose link stopper current mode to return machine zero point, low speed signal low level is valid. By this time, zero sequence of machine zero turning as following picture:



Picture 2-41-b

A: Select machine zero return mode, press the manual positive or negative feed key(machine zeroreturn direction is set by bit parameter No.026), the corresponding axis moves to the machine zero by a rapid speed (parameter No. 70 ~ 74). As the axis press down the deceleration switch to cut off deceleration signal, the feed slows down immediately, and it continues to run in a fixed low speed .

B: When the deceleration switch is released, the deceleration signal contact point is closed again. And CNC begins to detect the encoder one-rotation signal, if the signal level changes, the motion will be stoped.And thecorresponding zero indicator on the operator panel lights up for machine zero return completion.

2.8.3 Spindle control

Relative signal (standard PLC program definition):

Signal type	Symbol	Signal port	Address	Function explanation	Remark
Input signal	SAR	CN15.6	X4.1	Spindle axis reaches signal	The signal 0V input is vaild
	SALM	CN15.4	X4.3	Spindle axis abnormal alarm input	
Output signal	M03	CN62.4	Y0.3	Spindle axis anticlockwise rotation (forward)	
	M04	CN62.5	Y0.4	Spindle axis clockwise rotation (reversal)	

	M05	CN62.6	Y0.5	Spindle stop	
	SCLP	CN62.7	Y0.6	Spindle clamp	
	SPZD	CN62.8	Y0.7	Spindle brake	
	SVF	CN62.37	Y3.0	Spindle servo break up	
Instruction format	M03			Spindle axis anticlockwise rotation (forward)	
	M04			Spindle axis clockwise rotation (reversal)	
	M05			Spindle stop	
	M20			Spindle clamp	It is valid when analog spindle
	M21			Spindle brake	

Control parameter

State parameter

K	1	0								RSJG	

RSJG=1: press RESET, cnc wouldn't close M03, M04, M08 and M32 output signal;

=0: press RESET, cnc close M03, M04, M08 and M32 output signal.

0	2	0			SAR						

Bit6 1: Dectect spindle SAR signal before cutting;

0: Doesn't dectect spindle SAR signal before cutting.

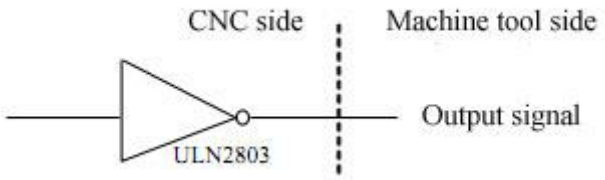
Data parameter

D 9 9

Spindle zero speed output range (r/min)

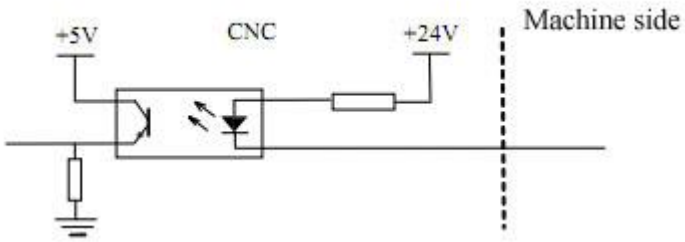
Signal connection

M03, M04, M05, SCLP, SPZD, SVF signal output circuit as below picture 2-45A shows:



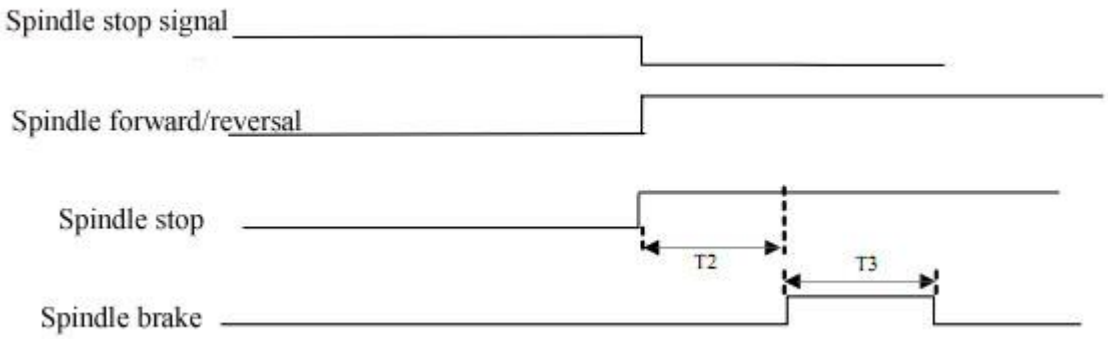
Picture 2-45A

SAR, SALM signal input circuit as below picture 2-45B shows:



Picture 2-45B

Action sequence (standard PLC program definition)



Picture 2-46

Note: T2 is the delay time from set out spindle stop signal to spindle brake signal, T3 is spindle brake hold time.

Function explantion (standard PLC program definition)

① M05 output is valid when CNC power on. When M05 output is valid, execute M03 or M04, M03 or M04 output is valid and continuous, and close M05 at the same time; M03 or M04 output is valid, execute M05, close M03 or M04 output, M05 output is valid and go on;

② M03 (M04) output is valid, execute M04 (M03) will alarm.

Note 1: when CNC scram, close M03 or M04, M08 signal output, and output M05 signal at the same time;

Note 2: when CNC reset, Bit 1 of K parameter NO.010 set if cancel the output of M03, M04:

When Bit 1 = 0, cnc reset and close the output of M03, M04;

When Bit 1 =1, cnc reset M03, M04's output state keep the same.

2.8.4 Spindle rotate speed switching value control

Relative signal (standard PLC program definition)

S01 ~ S04: Spindle rotate speed switch value control signal, S01 ~ S02 signal port which definited by standard PLC program is multiplex interface, S01 ~ S04 and M41 ~M44 share the port.

Signal diagnose

Signal	S4	S3	S2	S1
Diagnose address	Y1.3	Y1.2	Y1.1	Y1.0
Interface pins	CN62.12	CN62.11	CN62.10	CN62.09

Control parameter

State parameter

0	0	1					Analog spindle				

Bit 4 = 1: Spindle rotates speed analog voltage control;

= 0: Spindle rotates speed switch value control.

Control logic (standard PLC program definition)

S1 ~ S4 output is invalid when CNC power on. Excute one code of S01, S02, S03 and S04, corresponding S signal output is valid and hold on, and cancel the output of other S signal. Cancel output of S1 ~ S4 when execute S00 code, there is only one output of S1 ~ S4 is valid at the same time.

2.8.5 Spindle auto shift gears control

Relative signal (standard PLC program definition)

M41 ~ M44: spindle axis auto shift output signal, it can support 4 gears spindle shift automatically when choose spindle analog value control (0~10V analog voltage output)

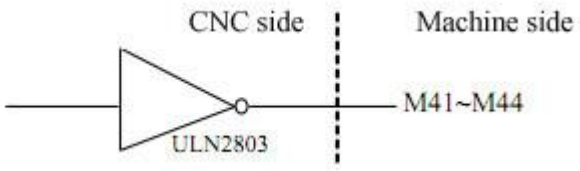
M41I, M42I: spindle axis auto shift the 1st and 2nd gears to position signal, it can support 2 gears shift position detection function,

Signal diagnose

Signal	M42I	M41I	M44	M43	M42	M41
Diagonse	X1.6	X1.5	X1.3	Y1.2	Y1.1	Y1.0
Interface pin	CN61.15	CN61.14	CN62.12	CN62.11	CN62.10	CN62.09

Signal connection

M41 ~ M44 circuit as below picture 2-47 shows:



Picture 2-47

Control parameter

State parameter

0	0	1					Analog spindle				

Bit 4 = 1: Spindle rotates speed analog control, it must be 1 when use spindle auto shift function;

= 0: Spindle rotates speed switch value control.

K	1	5						SHT	AGIM	AGIN	AGER

AGER=1: spindle auto shift function is valid;

=0: spindle auto shift function is invalid;

AGIN=1: detect shift position signal M41I and M42I when spindle auto shift to 1st and 2nd

=0: wouldn't detect shift position signal M41I and M42I when spindle auto shift to 1st and 2nd

AGIM=1: shift position signal to M41I, M42I and +24V break up is valid;

=0: shift position signal to M41I, M42I and +24V connected is valid;

SHT=1: spindle shift power down memory;

=0: spindle shift power down non-memory.

Data parameter

2	**1**	**0**		**GRMAX1**
2	**1**	**1**		**GRMAX2**
2	**1**	**2**		**GRMAX3**
2	**1**	**3**		**GRMAX4**

GRMAX1, GRMAX2, GRMAX3 and GRMAX4: they are the spindle speed which corresponding to the 1st, 2nd, 3rd and 4th when Spindle analog voltage is 10V. when spindle auto shift is valid, they are the spindle highest speed corresponding to M41, M42, M43 and M44.

2	**1**	**4**		**SFTREV**

The output voltage (0 ~ 10000, unit is mV) when spindle shift

Function description (standard PLC program definistion)

Spindle axis should be chosen under analog voltage control mode (Bit4 position of state parameter No.001 is 1), and Bit0 position of K parameter No.15 is 1, it is valid that spindle axis switches automatically; when spindle axis switches invalid, CNC will alarm when execute M41~M44. There is only on available of M41, M42, M43 and M44 at the same time.

Spindle auto switch function used for control auto switch spindle machanicam gear. When CNC execute S □□□□ code, calculate the output voltage according to current parameter (M41~44 corresponding data parameter No.210~213) which controlled by M4n, control the real rotation speed of spindle axis and S code are the same.

When power on, CNC controlled byparameter K of NO.15 Bit3 control whether to restore power in front of the main shaft gear. When the state parameters of NO. 001 of Bit4 is 0, with electricity, when the power is not memory spindle gear, default 1 gear main shaft gear, M41 ~ M44 NO output; when parameters of NO. 001 of Bit4 is 1, after power on electricity, main shaft gear memories. If you specify the gear is in line with current gear and not to shift. If you specify the gears do not agree with current gear, gear shifting, the standard of the definition of PLC shift process is as follows:

① Excute each code of M41, M42, M43, M44, output analog voltage to spindle servo or inverter according to the value (unit: mv) set by No.214;

② After delay data DT000 after (the shift time 1), shut down the original gear output signal output new shift signals at the same time;

③ When shift to 1 or 2, and Bit1 (AGIN) of № 15 K parameter is 1, the switch ④, otherwise the switch ⑤;

④ Check 1 or 2 files in the input signal M41I, M42I, if the shift in position switch ⑤;If the shift does not reach the designated position, the CNC has been waiting for shift in position signal;

⑤ Delay data DT001 (2) shift time, according to the current gear according to the data parameter NO. 210 to NO. 213 (1 ~ 4) setting output spindle analog voltage, shift to an end.

Note: CNC reset, stopping, defined as the output state of M41 ~ M44 standard PLC stays the same

2.8.6 External cycle start and feeding hold

Relative signal (standard PLC program definistion)

ST: External auto cycle start signal, it is of same function with the auto cycle start key of machine panel;

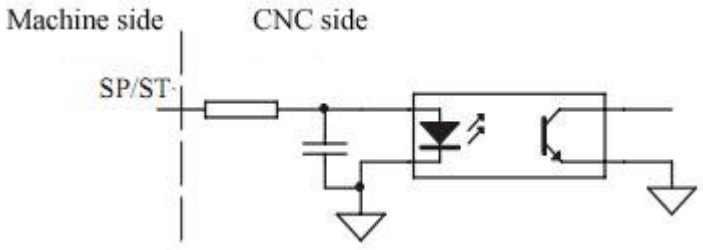
SP: External feeding hold signal, it is of same function with feeding hold key of machine panel.

Data diagnose

Signal	SP	ST
Diagnose address	X0.1	X1.4
Interface pin	CN61.2	CN61.13

Signal connection

SP/ST signal internal circuit as below picture 2-48



Picture 2-48

Control parameter

State parameter

0	2	1								MSP	MST

MST=1: External cycle start (ST) signal is invalid;

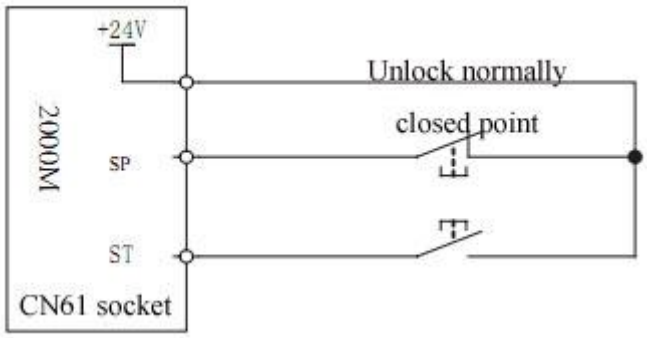
=0: External cycle start (ST) signal is valid.

MSP=1: External pause signal is invalid;

=0: External pause signal is valid. It must connect external pause switch, or CNC display “pause”

External connect circuit

SP, ST signal external connection as below picture 2-49



Picture 2-49

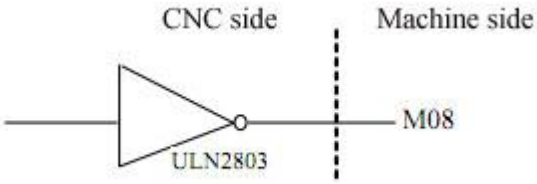
2.8.7 Cooling pump control

Relative instruction signal (standard PLC program definition

Signal type	Symbol	Signal interface	Address	Function explanation	Remark
Output signal	**M08**	**CN62.1**	**Y0.0**	**Cooling pump control output**	
Instruction format	**M08**			**Cooling liquid opend**	
	M09			**Cooling liquid closed**	

Signal connection

Internal circuit as below picture 2-50 shows:



Picture 2-50

Function description (standard PLC program definition)

M09 is valid and M08 is invalid after CNC power on. Excute M08, M08 is valid and cooling pump opened; excute M09, cancel M08 output, cooling pump closed.

Note 1: Cancel M08 output when CNC scram;

Note 2: When CNC reset, Bit 1 of CNC K parameter NO.10 sets if cancel M08 output;

Bit1=0: cancel M08 output when CNC reset;

Bit1=1: output state hold when CNC reset;

Note 3: M09 has no corresponding output signal, execute M09 cancel M08 output;

Note 4: 

key of panel can control cooling pump switch, please refer the 2nd volume "operation explanation"

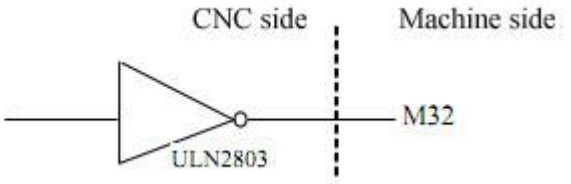
2.8.8 Lubrication control

Relative instruction signal (standard PLC program definition)

Signal type	Symbol	Signal interface	Address	Function explanation	Remark
Output signal	M32	CN62.2	Y0.1	Lubrication control output	
Instruction format	M32			Lubrication opened	
	M33			Lubrication closed	

Signal connection

Internal circuit as below picture 2-51 shows:



Control parameter

PLC state parameter

K 1 0 RSJG

RSJG=1: when press 

, CNC doesn't close M03, M04, M08 and M32 output signal.

=0: when press 

, CNC close M03, M04, M08 and M32 output signal

K	1	6							M32A		

M32A=1: when lubrication is valid, start up output lubrication

=0: when lubrication is valid, start up doesn't output lubrication

PLC data

D	T	0	5	3		

Auto lubrication interval time (0 ~ 65535ms)

D	T	0	1	3		

Auto lubrication output time (0 ~ 65535ms)

Function description

There are two lubrication functions of ECN*000M series standard PLC program definition, manual lubrication and auto lubrication set by parameter;

DT13=0: manual lubrication

>0: auto lubrication, lubrication time DT13and interval time DT53 can be set

1. Manual lubrication function is lubrication reversal output, press 

machine tool operation panel press, lubrication output cancel. When execute M32, lubrication output, lubrication output cancel through the time set by M33.

   When DT13>1, lubrication timing output, press machine operation panel 

, lubrication output cancel through the time which set by DT13; lubrication output when execute M32, lubrication output cancel through DT13 setting. If the time that set by DT13 didn't come, now execute M33 or press 

one more time, then lubrication output cancel.

2. Auto lubrication;

   When K16.2 set to be 1, the time that set by DT13 started after system power on, and then stop output, repeat output lubrication through the time that set by DT53, recycle one by one. When auto lubrication, M32, M33 and 

of machine operation panel are valid, lubrication time is the time that set by DT13.

Note 1: CNC scram, close lubrication output;

Note 2: CNC reset, Bit 1 of K parameter NO.010 set if cancels lubrication output;

When Bit1=0, CNC reset close lubrication output;

When Bit1=1, CNC reset lubrication output state hold on.

2.8.9 Protective door

Relative signal (standard PLC program definition)

SAGT: protective door detect input signal

Signal diagnose

Signal	SAGT
Diagnose address	X0.0
Interface pins	CN61.1

Control parameter

State parameter

K	1	4						SPB4	PB4		

PB4=0: protective door detection is invalid;

=1: protective door detection is valid;

SPB4 =0: when it is SAGT low level (break with +24V), the protective door is closed;

=1: when it is SAGT high level (connect with +24V), the protective door is closed;

Function description (standard PLC program definition)

① When PB4 ＝ 1, SPB4 ＝ 0, CNC can be confirmed that protective door closed when SAGT signal is broke with =24V;

② When PB4 ＝ 1, SPB4 ＝1, CNC can be confirmed that protective door closed when SAGT signal connect with =24V;

③ Protective door detection function is valid in auto mode, all the modes will give alarm that “protective door is already opened” when protective door is open and wouldn’t affect other functions;

④ In auto mode, when auto cycle restart, it will alarm when CNC detects the profective opened;

⑤ During auto operation, if CNC detects protective door opened, then axis feeding paused, close spindle axis and cooling output.

2.8.10 CNC Marco variable

Relative signal

Marco output signal: standard PLC definited 5 #1100 ～ #1105 marco output port;

Marco input signal: standard PLC definited 5 #1000 ～ #1015 marco input port;

Signal diagnose

Marco variable No.	#1105	#1104	#1103	#1102	#1101	#1100
Diagnose	Y3.7	Y3.6	Y3.5	Y3.4	Y3.3	Y3.2

Marco variable No.	#1007	#1006	#1005	#1004	#1003	#1002	#1001	#1000
Diagnose	X0.7	X0.6	X0.5	X0.4	X0.3	X0.2	X0.1	X0.0

Marco variable No.	#1015	#1014	#1013	#1012	#1011	#1010	#1009	#1008
Diagnose	X1.7	X1.6	X1.5	X1.4	X1.3	X1.2	X1.1	X1.0

Function description (standard PLC program definition)

Assignment a Marco variable to#1100 ~#1105, can change the state of UO0 ~ UO5 output signal state. It outputs 0V when assignment is "1"; it closes its output signal when assignment is "0".

The input states of interface X0.0 ~ X0.7, X1.0 ~ X1.7 can be learnt after detect the value of Marco variable #1000 ~ #1015.

2.8.11 Tricolour light

Relative signal and function definition (standard PLC program definition)

Y2.2 (CN62.31): Yellow light, it means normal state (non-running, non-alarm state)

Y2.3 (CN62.32): Green light, it means running state

Y2.4 (CN62.33): Red light, it means alarm state

2.8.12 External MPG

CN31 (MPG)	PLC address	Address symbol	Address definition function	Remark
5	X5.0	EHDX	X handlewheel	Suit for PSG-100-05E/L, ZSSY2080 type MPG
6	X5.1	EHDY	Y handlewheel	
8	X5.2	EHDZ	Z handlewheel	
9	X5.3	EMPO	Increment *1	
22	X5.4	EMP1	Increment *10	
23	X5.5	EMP2	Increment *100	

11, 12, 13	GND			
14, 15	+5V			
17, 18	+24V			

Relative parameter

State parameter

0	0	1						SOHW			

Bit3 = 0: signal step work mode;

= 1: handwheel work mode.

PLC state parameter

K	1	6		SINC							

SINC = 0: handlewheel, signal step mode *1000 gear increment is valid.

= 1: handlewheel, signal step mode *1000 gear increment is invalid.

Function description

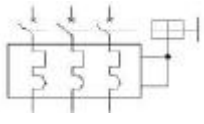
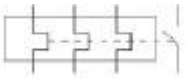
① MPG/signan step mode * 1000 increment gear choosen is invalid when SINC set to be 1, if it already chosen *1000 gear before amend the parameter, then it will change to *100mm gear automatically;

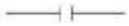
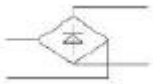
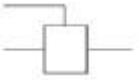
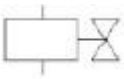
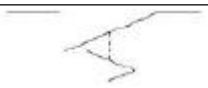
② When choose external MPG, axis choice of external MPG wouldn't clock automatically, namely when axis choice of MPG is invalid, it will be non-axis choice state;

③ When external MPG axis choice and gear choice input is valid, panel handlewheel axis choice and gear choice key is invalid; when external MPG axis choice and gear choice input is invalid, panel MPG axis choice and gear choice key is valid and clock automatically.

2.9 Electric diagram common symbols comparison

In the design of MST-81 series,ECN*000M series of DC24V power supply and working current larger electromagnetic valve used DC24V power supply must be independent, Symbols of components can be stated below:

Name	Symbol	Diagram	Name	Symbol	Diagram
Air breaker	QF		Contactor coil and assist terminal	KM	
			Hot relay and terminal	FR	

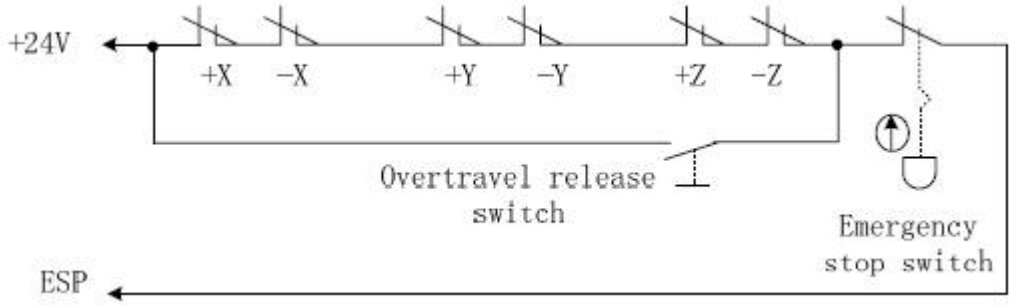
Transformer	TC		Capacitance	C	
Bridge rectifier	VC		Resistance	R	
Motor	M		Hall element		
Diode	VD		Dynamic travel switch	SQ	
Solenoid valve clue	YV		Veneer jack		
Relay clue and contact	KA		Foot switch	SA	
			Fuse	FU	

CHAPTER 3 MACHINE DEBUGGING METHODS AND STEPS

The trial run methods and steps at initial power on for this ECN*000M series are described in this chapter. The corresponding operation can be performed after the debugging by the following steps.

3.1 Emergency stop and strock limit

This ECN*000M series system has software limit function, it is suggested that te stroke limit switches are fixed in the positive or negative axes for hardware limit. The connection is shown in follows: (The chart is designed for X, Y, Z axes)



Picture 3-1

So the BIT2 (MKYP) of bit parameter No.021 should be set to 0.

And the CNC diagnostic message DGN000.7 can monitor the state of emergency stop input signal.

In Manual or MPG mode, slowly move the axes to test the validity of stroke limitswitch, correctness of alarm display, validity of overtravel release key.When the overtravel occursor Emergency Stop key is pressed,

"Emergency stop" alarm will be issued by CNC system. Thealarm can be cancelled by pressing down the Overtravel key and moving reversely.

3.2 Drive unit setting

Set BIT4～BIT0 of bit parameter No.009 according to alarm logic level of drive unit. The BIT4～BIT0 (5ALM, 4ALM, YALM, ZALM, XALM corresponding to 5th, 4th, Y, Z, X axis respectively) of bit parameter No.009 for our drive unit are all set for 1.

If the machine moving direction is not consistent with the moving command, modify the BIT4～BIT0 (DIR4, DIR5, DIRY, DIRZ, DIRX corresponding to 5th, 4th, Y, Z, X axis respectively) of bit parameter No.008.

Manual moving direction can be changed through BIT4, BIT3, BIT2, BIT1 and BIT0 bit (5VAL, 4VAL, YVAL, ZVAL, XVAL corresponding to 5t, 4th, Y, Z, X axis moving key respectively) of No. 020.

3.3 Gear Ratio Adjustment

When different thread pitch lead screw match with step motor in different step angle or servo motor in different pulse number, or connect through different ratio gear, it can make the program and real movement distance are the same through set electrical gear parameter.

Linear axis when configure step motors

$$\frac{CMR}{CMD}=\frac{360}{a\times L\times 1000}$$

CMR: command multiplier coefficient (parameter No. 000~004)

CMD: command fractional coefficient (parameter Np. 005~009)

a: step angle (degree)

L: step motor– turn to movement amount (millimeter) of corresponding machine tool

For example a=0.75 L=5

$$\frac{CMR}{CMD}=\frac{12}{125}$$

System minimum output unit is CMD/CMR=125/12(unit: 0.001mm)

Note: no matter what step angle motor it configured, the minimum programming unit of system is 0.001mm, and the minimum output unit depends on a and L, the smaller the a and L are, the higher the resolution ratio is, but the speed will go down; on the contrary, the larger the a and L are, the higher the speed is, but the resolution ratio will go down.

Linear axis configure servo motors

$$\frac{CMR}{CMD} = \frac{P}{L \times 1000}$$

L: step motor – turn to movement amount (millimeter) of corresponding machine (millinmeter)

P: motor – turn to corresponding pulse number of feedback

Rotation axis configure servo motors

$$\frac{CMR}{CMD} = \frac{P}{360000} \times \frac{ZM}{ZD}$$

P: motor – turn to corresponding pulse number of feedback

ZM: gears number of lead screw end(Passive tooth)

ZD: gears number of motor end (driving tooth)

Tapping spindle axis

$$\frac{CMR}{CMD} = \frac{P}{1000} \times \frac{ZM}{ZD}$$

P: motor – turn to corresponding pulse number of feedback

ZM: gears number of lead screw end(Passive tooth)

ZD: gears number of motor end (driving tooth)

CMR: command multiplier coefficient (parameter No. 230~232)

CMD: command fractional coefficient (parameter Np. 233~235)

3.4 Acceleration&deceleration Characteristic Adjustment

Adjust the relative CNC parameters according to the factors such as the drive unit, motor characteristics and machine load:

Data parameter №090～№094, №113～№117: X, Y, Z, 4th,and 5th axis rapid traverse rate;

Data parameter №120～№139: X,Y, Z, 4th, and 5th axis rapid traverse rate;

Data parameter №111～№112: acceleration& deceleration time constant of each axis manual feeding;

Data parameter №154: proread method, acceleration speed (mm/s/s) before cutting feed

Data parameter №102 ~ №105: MPG speed and acceleration & deceleration time constant;

Data parameter №108: step feeding max strangulation speed;

Data parameter №110: each axis manual (JOG) feeding speeds in continuous.

The larger the acceleration&deceleration time constant is, the slowertacceleration&deceleration is, the smaller the machine movement impact and the lower themachining efficiency is.

The principle for acceleration&deceleration characteristic adjustment is to properlyreduce the acceleration &

deceleration time constant and increase the acceleration&decelerationstart/end speed to improve the machining efficiency on the condition that there is no alarm, motorout-of-step and obvious machine impact. If the acceleration&deceleration time constant is set toosmall, and the start/end speed is set too large, it is easily to cause drive unit alarm, motor out-of-stepor machine vibration.

3.5 Machine zero adjust

Relative signal:

DECX: X axis deceleration signal;

DECY: Y axis deceleration signal;

DECZ: Z axis deceleration signal;

DEC4: the 4th axis axis deceleration signal;

DEC5: the 5th axis axis deceleration signal;

Diagnose data

0	**0**	**0**					**DEC5**	**DEC4**	**DECZ**	**DECY**	**DECX**
Interface pin							**CN61.34**	**CN61.33**	**CN61.12**	**CN61.32**	**CN61.4**

Control parameter

K	**2**	**2**		***	***	***	**DEC5T**	**DEC4T**	**DECZ**	**DECY**	**DECX**

DEC5T=0: the 5th axis deceleration signal low level;

=1: the 5th axis deceleration signal high level;

DEC4T=0: the 4th axis deceleration signal low level;

=1: the 4th axis deceleration signal high level;

DECY=0: Y axis deceleration signal low level;

=1: Y axis deceleration signal high level;

DECZ=0: Z axis deceleration signal low level;

=1: Z axis deceleration signal high level;

DECX=0: X axis deceleration signal low level;

=1: X axis deceleration signal high level;

0	**0**	**6**						**ZPLS**			**ZMOD**

ZMOD=1: Back to zero pattern choice pieces before;

=0: Back to zero pattern choice pieces after;

ZPLS=1: Turn back to zero mode selection, a signal;

=0: Turn back to zero mode selection, no a signal.

0	**1**	**2**									**ISOT**

ISOT=1: After electrify, back to the machine before midnight, manual fast moving is valid;

=0: After electrify, back to the machine before midnight, manual fast moving is invalid;

0	**2**	**6**					**ZMI5**	**ZMI4**	**ZMIZ**	**ZMIY**	**ZMIX**

ZMIX=1: Select the direction of shaft back to zero for negative direction back to zero;

=0: Select the direction of shaft back to zero for positive direction back to zero;

Data parameter

0	8	0		ZRNFL

ZRNFL: Back to the mechanical zero point of the low speed rate;

0	7	0		ZRNFHX

ZRNFHX: X axis and Z axis to mechanical zero point of the high speed;

0	7	1		ZRNFHY

ZRNFHY: Y axis back to mechanical zero speed at a high speed;

0	7	2		ZRNFHZ

ZRNFHZ: Z axis back to mechanical zero speed at a high speed;

0	7	3		ZRNFH4

ZRNFH4: the 4th axis back to mechanical zero speed at a high speed;

0	7	4		ZRNFH5

ZRNFH5: the 5th axis back to mechanical zero speed at a high speed;

According to the effective connection signal level, USES the way of back to zero, back to the direction of the zero adjustment related parameters:

№ 22 BIT4, BIT5, BIT6, BIT7 of K parameter:4 axis X, Y, Z, return machine zero point, slow down the signal level effectively.

State parameter BIT0 (ZMOD) of № 006 : back to the zero mode selection (0: gear block 1: after gear block before).

State parameter BIT3 of № 006: back to zero mode selection: (0: no. 1: yes) a turn signal.

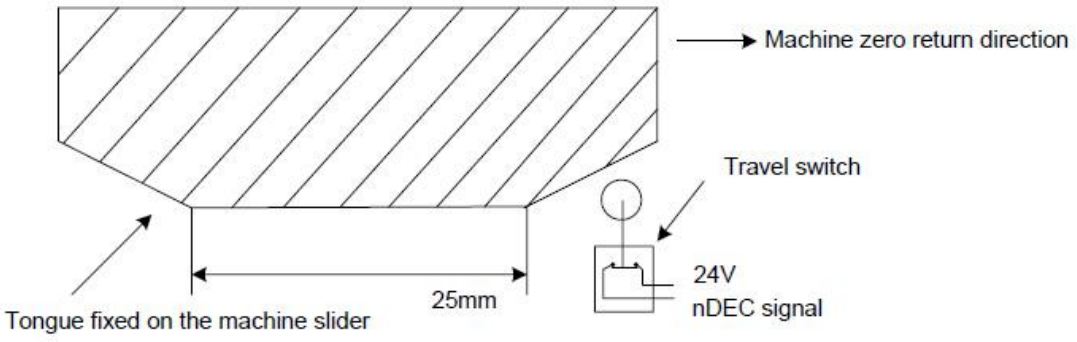
Data parameters № 080: the low speed of each axis returns to machine zero deceleration process.

Data parameters № 070 ~ № 074: the axis return machine high speed velocity of zero.

№ 26 BIT0, BIT1, BIT2 (ZMIX ZMIY ZMIZ ZMI4 ZMI5) of State parameter: Each axis direction chooses back to zero, the positive direction back to zero, or negative direction back to zero

After the confirmation of super form a switch to the executable machine tools back to zero. Usually install the machine zero point in the biggest trip, back to zero collision block effective stroke in more than 25 mm, to ensure the enough deceleration distance, can ensure the speed down, to ensure accurate back to zero. Perform the faster the speed of the machine back to zero, back to zero collision block to grow more, or you will make drag with CNC deceleration, such as machine inertia plate through back to zero speed after collision block failed to come down, there is not enough to slow down distance, affecting the precision of the back to zero. Usually there are two kinds of machine tools back to zero connection method:

1. Usually form a complete set of the ac servo motor connection: use a travel switch and servo motor a diagram of the turn signal



Picture 3-2

Using this connection, when back to the machine zero point after release of deceleration switch should be avoided at a encoder signal after the switch to release critical point position, turn a half circle of motor to turn the encoder

signal, in order to improve the precision back to zero.

Parameter Settings (recommended) as follows:

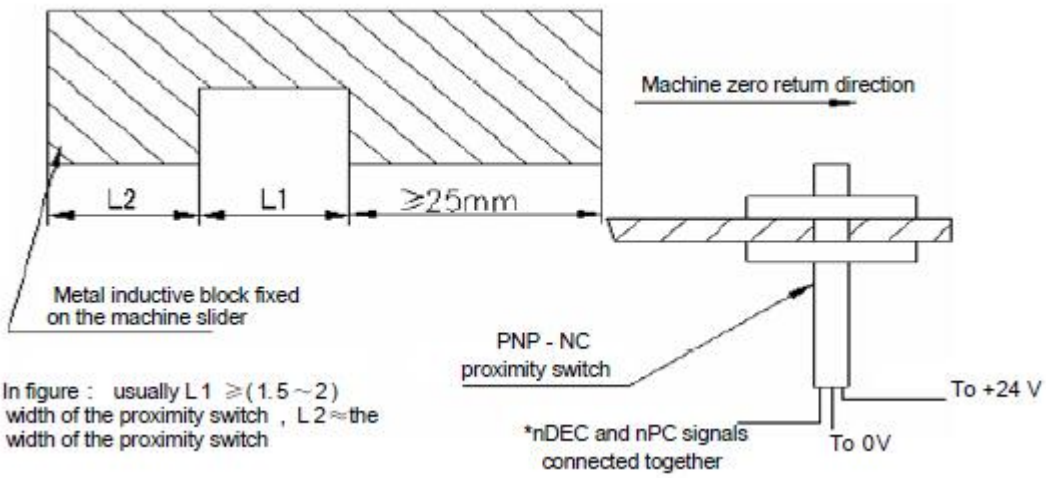
BIT0 (ZMOD) of state parameter № 006=0

BIT3(ZPLS) of state parameter № 006=1

Data parameter № 080=200

State parameter № 026 的BIT0(ZMIX), BIT1(ZMIY), BIT2(ZMIZ), BIT3(ZMI4), BIT4(ZMI5)=0

2. Usually form a complete set of stepping motor connection: use a proximity switch at the same time as the deceleration, zero signal diagram;



Form a complete set of stepping motor, parameter Settings (recommended) as follows:

BIT0 (ZMOD) of state parameter № 006=0

BIT3 (ZPLS) of state parameter № 006=0

State parameter № 026 的BIT0(ZMIX), BIT1(ZMIY), BIT2(ZMIZ), BIT3(ZMI4), BIT4(ZMI5)=0

Data parameter № 080=200

3.6 Spindle Adjustment

3.6.1 Spindle Encoder

Encoder with the linear number 100 p/r ~5000p/r is needed to be installed on the machine for threading. The linear number is set by data parameter No.109. The transmission ratio(spindle gear teeth/encoder gear teeth) between encoder and spindle is 1/255～255. The spindle gear teeth are set by CNC data parameter No.110 and the encoder gear teethare set by data parameter No.111. Synchronous belt transmission should be applied for it (no sliding transmission)

3.6.2 Switch Volume Control of Spindle Speed

When multiple speed motor control is used, motor speed control command can be defined byladder diagram as S01~S04. Relevant parameter is shown below.

Bit4=0 of parameter №001: select switching control of spindle speed.

3.6.3 Analog Voltage Control for Spindle Speed

This function can be obtained by the parameter setting of CNC. By interface outputting 0V～10V analog voltage to control inverter, the stepless shift can be obtained. And the related parameters are needed to be adjusted are:

Bit4=1 of parameter №001: for selection of spindle speed analog voltage control;

Data parameter №206: offset compensation value(mv) as spindle speed command voltage is 10V;

Data parameter №0210~№212: offset compensation value as spindle speed command voltage is 10V;

The inverter to adjust the basic parameters:

Positive &negative mode choice: determined by the terminal VF;

Frequency setting mode choice: determined by the terminal FR

If the speed by programming is not consistent with that detected by the encoder, it can be adjusted to be consistent with the actual one by adjusting the data parameter №210～№212.

Speed adjustment method: select the spindle first gear, input M03/M04 S9999 code in MDI mode to run the spindle, view the spindle speed shown on the right bottom of the screen, then reinput the displayed speed value into the parameter. The other spinle gear adjustment is identical with this.

When entering S9999 code, the voltage should be 10V, S0 is 0V. If there is an voltage error, adjust bit parameter №206 to correct the voltage offset value (corrected by manufacturer, usually not needed). When the current gear is the max.speed, if the analog voltage output by CNC is higher than 10V, set a smaller value for data parameter №100; when the S00 code is entered, if there is still slow rotation in the spindle, it means the analog voltage output by CNC is higher than 0V, so set a smaller value for data parameter №099.

If the machine is not fixed with an encoder, the spindle speed can be detected by a speed sensor, input S9999 in MDI mode to set the speed value displayed by sensor to the data parameter №210~212.

3.7 Backlash Offset

The backlash offset is input by diameter value with the unit 0.001mm, which is irrelevant to the programming by diameter or by radius. It can be measured by a dial indicator, a micrometer or a laser detector. Because the backlash offset can improve the machining precision only by accurate compensation, it is not recommended to measure it in

MPG or Step mode, but the following method is suggested:

Program editing (as Z axis)

O0001;

N10 G01 Z10 F800 G91;

N20 Z15;

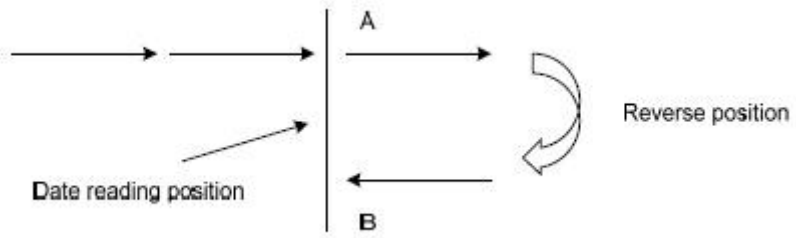
N30 Z1;

N40 Z-1;

N50 M30.

Set the backlash error offset to 0 before measuring;

Run the program by single blocks, search the measuring benchmark A after 2 positioning operations, record the current data, move 1mm in the same direction, then move 1mm reversely to point B, read the current data.



Picture 3-4 Schematic map of backlash measuring methods

Backlash error offset value =| data of point A –data of point B |; Input the calculated data to the CNC data parameter №031～№035.

Data A: dial-indicator data at point A

Data B: dial-indicator data at point B

Note 1: The backlash offset mode can be set by Bit 7 of CNC parameter No.011, and offset frequency can be set by of bit parameter №036~040;

Note 2: Check the machine backlash at regular intervals according to specific conditions to ensure machine precision

3.8 Step/MPG Adjustment

Operation panel key can be chosen to be the way for a single step or hand wheel operation, by the state parameter setting option № 001 Bit3.

Bit3=1: The handwheel operation effectively, a single step mode is invalid;

=0: Single-step operation effectively, the handwheel operation mode is invalid.

3.9 Other Adjustment

0	1	7		***	MST	MSP	MOT	MESP	***	***	***

MST =1:External Cycle Start (ST) signal invalid.

=0:External Cycle Start(ST) signal valid;

MSP =1:External Dwell (SP) signal invalid.

=0: External Stop (SP) signal valid.

CHAPTER 4 DIAGNOSIS MESSAGE

4.1 CNC Diagnosis

This diagnosis section is used to check the CNC interface signals and internal running state and it can not be modified.

4.1.1 I/O Fixed address diagnose message

0	0	0		ESP	***	***	DEC5	DEC4	DECZ	DECY	DECX
Subscript				CN61.6			CN61.34	CN61.33	CN61.12	CN61.32	CN61.4
PCL fixed address				X0.5			X2.5	X2.4	X1.3	X2.3	X0.3

DECX, DECY, DECZ, DEC4, DEC5: X, Y, Z, 4th, 5th axis machine back to zero deceleration signal

KYP: scram signal

0	0	1		***	***	***	***	***	***	***	SKIP
Subscript											CN61.42
PCL fixed address											X3.5

4.1.2 Axes Moving State and Data Diagnosis Signal of CNC

0	0	4		***	***	***	EN5	EN4	ENZ	ENY	ENX

EN5 ~ ENX: axis enable signal

0	0	5		***	***	***	SET5	SET4	SETZ	SETY	SETX

SET5 ~ SETX: axis pulse forbid signal

0	0	6		***	***	***	DRO5	DRO4	DROZ	DROY	DROX

DRO5 ~ DROX: X, Y, Z, 4th and 5th axis movement direction output

0	0	9		***	***	***	5ALM	4ALM	ZALM	YALM	XALM

5ALM ~ XALM: X, Y, Z, 4th and 5th axis alarm signal

4.1.3 Key diagnosis

DGN.016～DGN.022 are the diagnosis messages of MDI keypad key. When pressing a key in the operation panel, the corresponding bit displays “1”and“0”after releasing this key. If it displays reversely, it means there is a fault in the keypad circuit.

4.1.4 Others

1	4	5		PLC executes time (ms)
1	4	6		System runs total time (h)

4.2 PLC state

This part of diagnosis is used to detect the signal state of machine→PLC (X), PLC→machine(Y),CNC→PLC(F),PLC→CNC (G) and alarm address A, state of internal relay (R, K).

4.2.1 F signal

F000		OP	SA	STL	SPL				

OP: Automatic signal

SA: Servo ready signal

STL: Light signal cycle started

SPL: Feed suspended light signal

F001		MA		TAP	ENB	DEN		RST	AL

MA: CNC ready signal

TAP: Tapping signal

ENB: Spindle enabling signal

DEN: Distribution over the signal

RST: Reset signal

AL: Alarm signal

F002		MDRN	CUT	MSTOP	SRNMV	THRD		RPDO	

MDRN: Empty running detection signal

CUT: A cutting feed signal

MSTOP: Choose to stop testing signal

SRNMV: Program start signal

THRD: Thread cutting signal

RPDO: Rapid feed signal

F003			MEDT	MMEM	MRMT	MMDI	MJ	MH	MINC

MEDT: Editors' choice signal storage

MMEM: Run automatically select signal

MRMT: DNC operation select signal

MMDI: Manual data input selection signal

MJ: JOG Feed selection signal

MH: The handwheel feed selection signal detection

MINC: Incremental feed selection signal detection

F004			MPST	MREF	MAFL	MSBK	MABSM	MMLK	MBDT

MPST: Back to the program start signal

MREF: A manual return reference point detection signal

MAFL: Assist lock detection signal

MSBK: Single segment detection signal

MABSM: Manual absolute value detection signal

MMLK: All the locked signal axis machine tool

MBDT: Skip the optional program segment detection signal

F007						TF	SF		MF

TF: Tool function gating signal

SF: The spindle speed gating signal

MF: Assist the gating signal

F008								SCHK	

SCHK: Syntax checking for signal

F009		DM00	DM01	DM02	DM30				RCT

DM00: M00 decoded signal

DM01: M01 decoded signal2

DM01: M02decoded signal

DM30: M30 decoded signal

RCT: During the implementation of the tool change

F010		MB07	MB06	MB05	MB04	MB03	MB02	MB01	MB00

MB07: Auxiliary function code MB07

MB06: Auxiliary function code MB06

MB05: Auxiliary function code MB05

MB04: Auxiliary function code MB04

MB03: Auxiliary function code MB03

MB02: Auxiliary function code MB02

MB01: Auxiliary function code MB01

MB00: Auxiliary function code MB00

F014								DRUN	PDBG

PDBG: PLC enter debug mode

DRUN: Switching signal is prohibited

F015					EN5T	EN4T	ENZ		

EN5T: the 5th axis selection

EN4T: the 4th axis selection

ENZ: Z axis selection

F016						ZP4	ZP3	ZP2	ZP1

ZP1: X axis returns zero point finsh signals

ZP2: Y axis returns zero point finsh signals

ZP3: Z axis returns zero point finsh signals

ZP4: 4TH axis returns zero point finsh signals

F018		AR07	AR06	AR05	AR04	AR03	AR02	AR01	AR00

AR07: The spindle actual speed AR07

AR06: The spindle actual speed AR06

AR05: The spindle actual speed AR05

AR04: The spindle actual speed AR04

AR03: The spindle actual speed AR03

AR02: The spindle actual speed AR02

AR01: The spindle actual speed AR01

AR00: The spindle actual speed AR00

F019		AR15	AR14	AR13	AR12	AR11	AR10	AR09	AR08

AR15:The spindle actual speed AR15

AR14:The spindle actual speed AR14

AR13:The spindle actual speed AR13

AR12:The spindle actual speed AR12

AR11:The spindle actual speed AR11

AR10:The spindle actual speed AR10

AR09:The spindle actual speed AR09

AR08:The spindle actual speed AR08

F020								BCLP	BUCLP

BCLP: 4TH axis dividing the workbench clamping signals

BUCLP:4TH axis dividing the workbench loosing signals

F021			MST	MSP		MESP			

MST:Shielding external loop start signal

MSP: Shielding external signal

MKYP: Shielding external stop signal

F022		SB07	SB06	SB05	SB04	SB03	SB02	SB01	SB00

SB07:The spindle speed code signalSB07

SB06:The spindle speed code signalSB06

SB05:The spindle speed code signalSB05

SB04:The spindle speed code signalSB04

SB03:The spindle speed code signalSB03

SB02:The spindle speed code signalSB02

SB01:The spindle speed code signalSB01

SB00:The spindle speed code signalSB00

F026		TB07	TB06	TB05	TB04	TB03	TB02	TB01	TB00

TB07: Tool function code signal TB07

TB06: Tool function code signal TB06

TB05: Tool function code signal TB05

TB04: Tool function code signal TB04

TB03: Tool function code signal TB03

TB02: Tool function code signal TB02

TB01: Tool function code signal TB01

TB00: Tool function code signal TB00

F030		R08O	R07O	R06O	R05O	R04O	R03O	R02O	R01O

R08O: S12 bit code signal R08O

R07O: S12 bit code signal R07O

R06O: S12 bit code signal R06O

R05O: S12 bit code signal R05O

R04O: S12 bit code signal R04O

R03O: S12 bit code signal R03O

R02O: S12 bit code signal R02O

R01O: S12 bit code signal R01O

F031						R12O	R11O	R10O	R09O

R12O: S12 bit code signal R12O

R11O: S12 bit code signal R11O

R10O: S12 bit code signal R10O

R09O: S12 bit code signal R09O

F032		X1000	X100	X10	X1			RGSPM	RGSPP

X1000: step width X1000soft key

X100: step width X100 soft key

X10: step width X10 soft key

X1: step width X1 soft key

RGSPM: Rigid tapping spindle in reverse

RGSPP: Rigid tapping spindle is turned in

F033		MTAP	DTAP						RTAP

MTAP: G63 Tapping mode signal

DTAP: Rigid tapping the execution of the signal

RTAP: Rigid tapping mode signal

F034		SSTOP	SCW	Z-	Z+	Y-	Y+	X-	X+

SSTOP: The spindle stop soft key

SCW: The spindle is soft key

Z-: Z- soft key

Z+: Z+ soft key

Y-: Y- soft key

Y+: Y+ soft key

X-: X- soft key

X+: X+ soft key

F035		SCCW	MSTOP	AFLO	BDTO	SBKO	MLKO	DRNO	QFAST

SCCW: Spindle turn counterclockwise soft key

MSTOP: Select stop soft key

AFLO: Lock the soft key auxiliary function

BDTO: Jump period of soft key program

SBKO: Single paragraph soft key

MLKO: Machine tool lock soft key

DRNO: Empty running soft key

QFAST: Fast moving soft key

F036		S-	S+	FAST-	FAST+			FEED-	FEED+

S-: Spindle ratio decrease soft key

S+: Spindle ratio decrease soft key

FAST-: Quick ratio decrease soft key

FAST+: Quick ratio decrease soft key

FEED-: Feed rate minus soft key

FEED+: Feed rate minus soft key

F037					ZP5	ZP4	ZP3	ZP2	ZP1

ZP5: End signal return reference point ZP5

ZP4: End signal return reference point ZP4

ZP3: End signal return reference point ZP3

ZP2: End signal return reference point ZP2

ZP1: End signal return reference point ZP1

F038									

MV5: Axis mobile signal MV5

MV4: Axis mobile signal MV4

MV3: Axis mobile signal MV3

MV2: Axis mobile signal MV

MV1: Axis mobile signal MV1

F039					MVD5	MVD4	MVD3	MVD2	MVD1

MVD5: Axis direction signalMVD5

MVD4: Axis direction signalMVD4

MVD3: Axis direction signalMVD3

MVD2: Axis direction signalMVD2

MVD1: Axis direction signalMVD1

F040					ZRF5	ZRF4	ZRF3	ZRF2	ZRF1

ZRF5: Reference point to establish the signal ZRF5

ZRF4: Reference point to establish the signal ZRF4

ZRF3: Reference point to establish the signal ZRF3

ZRF2: Reference point to establish the signal ZRF2

ZRF1: Reference point to establish the signal ZRF1

F041					ZP15	ZP14	ZP13	ZP12	ZP11

ZP15: 5TH axis end of shaft to return to the first reference point signal

ZP14: 4TH axis end of shaft to return to the first reference point signal

ZP13: Z axisend of shaft to return to the first reference point signal

ZP12: Y axis end of shaft to return to the first reference point signal

ZP11: X axisend of shaft to return to the first reference point signal

F042					PRO5	PRO4	PRO3	PRO2	PRO1

PRO5: Return the application zero signal PRO5

PRO4: Return the application zero signal PRO4

PRO3: Return the application zero signalPRO3

PRO2: Return the application zero signalPRO2

PRO1: Return the application zero signalPRO1

F043									MSPHD

MSPHD: Spindle motion detection signal

F044					SIMSPL			FSCSL	

SIMSPL: Simulate the spindle effective

FSCSL: Cs Profile control end of the switch signal

F047		Total knife digits

F048			MST	MSP		MESP			

MST: Shielding external loop start signal

MSP: Shielding external signal

MKYP: Shielding external stop signal

F051					VAL5	VAL4	VALY	VALZ	VALX

VAL5: 5 direction of choice

VAL4: 4 direction of choice

VALY: Y direction of choice

VALZ: Z direction of choice

VALX: X direction of choice

F054		UO07	UO06	UO05	UO04	UO03	UO02	UO01	UO00

UO07: Macro output signal UO07

UO06: Macro output signalUO06

UO05: Macro output signalUO05

UO04: Macro output signalUO04

UO03: Macro output signalUO03

UO02: Macro output signalUO02

UO01: Macro output signalUO01

UO00: Macro output signalUO00

F055		UO15	UO14	UO13	UO12	UO11	UO10	UO09	UO08

UO15: Macro output signalUO15

UO14: Macro output signalUO14

UO13: Macro output signalUO13

UO12: Macro output signalUO12

UO11: Macro output signalUO11

UO10: Macro output signalUO10

UO09: Macro output signalUO09

UO08: Macro output signalUO08

F057					ZP25	ZP24	ZP23	ZP22	ZP21

ZP25: 5TH axis return to the end of the second reference point signal

ZP24: 4TH axis return to the end of the second reference point signal

ZP23: Z axis return to the end of the second reference point signal

ZP22: Y axis return to the end of the second reference point signal

ZP21: X axis return to the end of the second reference point signal

F058					ZP35	ZP34	ZP33	ZP32	ZP31

ZP35: 5TH axis return to the end of the third reference point signal

ZP34: 4THaxis return to the end of the third reference point signal

ZP33: Z axis return to the end of the third reference point signal

ZP32: Y axis return to the end of the third reference point signal

ZP31: X axis return to the end of the third reference point signal

F059					ZP45	ZP44	ZP43	ZP42	ZP41

ZP45: 5TH axis return to the end of the 4th reference point signal

ZP44: 4TH axis return to the end of the 4th reference point signal

ZP43: Z axis return to the end of the 4th reference point signal

ZP42: Y axis return to the end of the 4th reference point signal

ZP41: X axis return to the end of the 4th reference point signal

F061									ESEND

KYEND: The required number of parts to the signal

4.2.2 G Sign

G004						FIN			

FIN: Assist the end of the signal

G005		LEDT	AFL		LAXIS				

LEDT: Edit lock signal

AFL: Assist lock signal

LAXIS: The entire shaft interlocking signal

G006			SKIPP		OVC		ABSM	MSTOP	SRN

SKIPP: skip signal

OVC: Feed rate cancellation signal

ABSM: A manual signal absolute value

MSTOP: Select stop signal

SRN: Application restart signal

G007							ST		

ST: Cycle start signal

G008				SP	ESP				

SP: Feed signal

KYP: scram signal

G009							M12	M32	COOL

M12: 0/1: spindle axis tool loose/clamp signal

M32: Lubrication signal

COOL: Cooling signal

G010		JV07	JV06	JV05	JV04	JV03	JV02	JV01	JV00

JV07: Manual mobile rate signal JV07

JV06: Manual mobile rate signalJV06

JV05: Manual mobile rate signalJV05

JV04: Manual mobile rate signalJV04

JV03: Manual mobile rate signalJV03

JV02: Manual mobile rate signalJV02

JV01: Manual mobile rate signalJV01

JV00: Manual mobile rate signalJV00

G011		JV15	JV14	JV13	JV12	JV11	JV10	JV09	JV08

JV08:Manual mobile rate signalJV08

JV09:Manual mobile rate signalJV09

JV10:Manual mobile rate signalJV10

JV11:Manual mobile rate signalJV11

JV12:Manual mobile rate signalJV12

JV13:Manual mobile rate signalJV13

JV14:Manual mobile rate signalJV14

JV15:Manual mobile rate signalJV15

G012		FV07	FV06	FV05	FV04	FV03	FV02	FV01	FV00

FV07: Feed speed ratio signal FV07

FV06: Feed speed ratio signal FV06

FV05: Feed speed ratio signal FV05

FV04: Feed speed ratio signal FV04

FV03: Feed speed ratio signal FV03

FV02: Feed speed ratio signal FV02

FV01: Feed speed ratio signal FV01

FV00: Feed speed ratio signal FV00

G014		RV08	RV07	RV06	RV05	RV04	RV03	RV02	RV01

RV08: Fast feed rate signal RV08

RV07: Fast feed rate signal RV07

RV06: Fast feed rate signalRV06

RV05: Fast feed rate signalRV05

RV04: Fast feed rate signalRV04

RV03: Fast feed rate signalRV03

RV02: Fast feed rate signalRV02

RV01: Fast feed rate signalRV01

G016					SAR				

SAR: Spindle speed signal

G017						DECA	DECY	DECAZ	DECAX

DECA: 4TH axis back to zero speed signal detection

DECY: Z axis back to zero speed signal detection

DECZ: Y axis back to zero speed signal detection

DECX: X axis back to zero speed signal detection

G018						H4TH	HY	HZ	HX

H4TH: 4TH axis handwheel feed selection signal

HY: Z axis handwheel feed selection signal

HZ: Y axis handwheel feed selection signal

HX: X axis handwheel feed selection signal

G019		RT		MP2	MP1				

RT: Manual feed selection signal quickly

MP2: The handwheel ratio signal MP2

MP1: The handwheel ratio signal MP1

G021		SOV7	SOV6	SOV5	SOV4	SOV3	SOV2	SOV1	SOV0

SOV7: The spindle speed ratio signal SOV7

SOV6: The spindle speed ratio signal SOV6

SOV5: The spindle speed ratio signal SOV5

SOV4: The spindle speed ratio signal SOV4

SOV3: The spindle speed ratio signal SOV3

SOV2: The spindle speed ratio signal SOV2

SOV1: The spindle speed ratio signal SOV1

SOV0: The spindle speed ratio signal SOV0

G022		R08I	R07I	R06I	R05I	R04I	R03I	R02I	R01I

R08I: Spindle motor speed code signal R08I

R07I: Spindle motor speed code signal R07I

R06I: Spindle motor speed code signal R06I

R05I: Spindle motor speed code signal R05I

R04I: Spindle motor speed code signal R04I

R03I: Spindle motor speed code signal R03I

R02I: Spindle motor speed code signal R02I

R01I: Spindle motor speed code signal R01I

G023		SIND	SGN			R12I	R11I	R10I	R09I

SIND: Spindle motor speed code selection signal

SGN: Spindle motor speed code selection signal

R12I: Spindle motor speed code selection signalR12I

R11I: Spindle motor speed code selection signalR11I

R10I: Spindle motor speed code selection signalR10I

R09I: Spindle motor speed code selection signalR09I

G024		MRDYA							

MRDYA: Machine tool ready signal

G025				SRVB	SFRB				

SRVB: spindle axis reverse signal

SFRB: spindle axis forward signal

G026		CON							

CON: CS Profile control of switch signal

G027						+J4	+J3	+J2	+J1

+J4: Feeding axis and direction selection signal +J4

+J3: Feeding axis and direction selection signal +J3

+J2: Feeding axis and direction selection signal +J2

+J1: Feeding axis and direction selection signal +J1

G028						-J4	-J3	-J2	-J1

-J4: Feeding axis and direction selection signal-J4

-J3: Feeding axis and direction selection signal-J3

-J2: Feeding axis and direction selection signal-J2

-J1: Feeding axis and direction selection signal-J1

G030						+L4	+L3	+L2	+L1

+L4: Axis overtravel signal+L4

+L3: Axis overtravel signal+L3

+L2: Axis overtravel signal+L2

+L1: Axis overtravel signal+L1

G031						-L4	-L3	-L2	-L1

-L4: Axis overtravel signal-L4

-L3: Axis overtravel signal-L3

-L2: Axis overtravel signal-L2

-L1: Axis overtravel signal-L1

G036		BEUCL	BECLP						SPD

BEUCL: Indexing table loosen the complete signal

BECLP: Indexing table clamp the complete signal

SPD: Spindle dynamic function of the signal

G037		NT07	NT06	NT05	NT04	NT03	NT02	NT01	NT00

NT07: The current tool number NT07

NT06: The current tool number NT06

NT05: The current tool number NT05

NT04: The current tool number NT04

NT03: The current tool number NT03

NT02: The current tool number NT02

NT01: The current tool number NT01

NT00: The current tool number NT00

G043		ZRN		DNC1			MD4	MD3	MD1

ZRN: The current work mode selection 4

DNC1: DNC run the selection signal

MD4: The current work mode selection 3

MD2: The current work mode selection 2

MD1: The current work mode selection 1

G044		HDT						MLK	BDT

HDT: Manual tool change signal sequence

MLK: Machine tool lock signal (LC → CNC)

BDT: Program selected jump signal(PLC → CNC)

G046		DRN				KEY1		SBK	

DRN: Empty running signal

KEY1: Memory protection signal

SBK: Single block signals(PLC → CNC)

G048								CR2	CR1

GR2: The gear selection signal

GR1: The gear selection signal

G053		CD2	SMZ						

CDZ: Chamfering signal

SMZ: Error check signal

G054		UI07	UI06	UI05	UI04	UI03	UI02	UI01	UI00

UI07: Macro input signalUI07

UI06: Macro input signalUI06

UI05: Macro input signalUI05

UI04: Macro input signalUI04

UI03: Macro input signalUI03

UI02: Macro input signalUI02

UI01: Macro input signalUI01

UI00: Macro input signalUI00

4.2.3Anaddress (message displays request signal, standard PLC definition)

Address	Alarm number	Display
A0002.0	1216	The protective door is opened, not allowed to run automatically
A0002.1	1217	Low pressure alarm
A0002.3	1219	The spindle rotates, must not loosen the cutting tool
A0002.4	1220	The spindle rotates, tool clamping invalid alarm signal to reach the designated position
A0002.5	1221	Tool clamping in place when the signal is invalid, can not start the spindle
A0002.6	1222	The spindle tool to loosen, and may not start the spindle
A0004.0	1232	Illegal M code
A0004.1	1233	The current is not simulate the spindle, unable to perform the function of point move
A0004.2	1234	M03, M04 specified error code
A0004.4	1236	Spindle shift time is too long
A0004.5	1237	The spindle speed/position control switch time is too long

A0005.1	1241	Spindle servo abnormal alarm or frequency converter
A0007.1	1257	Door has been opened

CHAPTER 5 MEMORIZING SCREW-PITCHES ERROR COMPENSATION FUNCTION

5.1 Function Explanation

There are more or less precision errors in the screw-pitch of machine axes lead screw, it will definitely affect the parts machining precision. This ECN*000M series has the memorizing screw-pitch error.

5.2 Specifications

1. The offset is concerned with the offset origin, offset clearances, offset point, mechanical moving direction etc;

2. After performing the machine zero return, take this reference point as the offset origin, and set the offset value into the parameters according to axes compensation intervals;

3. Points to be compensated: 256 points for each axis

4. Axis to be compensated：X, Y, Z，4th, 5th axis

5. Offset range: 0～±99× for each offset point

6. Offset clearance: 1～9999999μm;

7. Offset of point N (N=0, 1, 2, 3,…255) is determined by the N, N-1 mechanical error;

8. The setting is the same as the CNC parameters input, see the explanation in the relative operation.

5.3 Parameter Setting

5.3.1 Screw-Pitch Compensation

State parameter

0	0	3				SCREW					

Bit5=1: itch error compensation function is valid;

Bit5=0: itch error compensation function is invalid.

5.3.2 Pitch error compensation the origin

Machine tools original point which corresponding to pitch error compensation in the table compensation howling pitch error compensation the origin (reference point); the origin point of pitch error compensation set by parameter № 180 ~ № 184.According to the actual demand, each axis can set any place in 0 ~ 255.

Data parameter

1	8	0		Position number of X axis pitch error compensation original point
1	8	1		Position number of Y axis pitch error compensation original point
1	8	2		Position number of Z axis pitch error compensation original point
1	8	3		Position number of the 4th axis pitch error compensation original point
1	8	4		Position number of the 5th axis pitch error compensation original point

5.3.3 Offset interval

Pitch error compensation: № 190 ~№ 194;

Input Units: metric system machine: mm, British system machine: inch;

Setting range: 1 ~ 9999.9999;

State parameter

1	9	0		Spacing distance of X axis pitch error compensation original point
1	9	1		Spacing distance of Y axis pitch error compensation original point
1	9	2		Spacing distance of Z axis pitch error compensation original point
1	9	3		Spacing distance of the 4th axis pitch error compensation original point
1	9	4		Spacing distance of the 5th axis pitch error compensation original point

5.3.4 Compensation dosage

The axis screw pitch error compensation, according to the following table parameter set, fixed amount of compensation to the radius value input, has nothing to do with programming in diameter or radius of programming, input value is mm (metric machine) or inch (british machine).

5.4 Precaution of offset value setting

①Operation privileges must be level 2 password to set and amend pitch offset parameter;

②After setting the parameters of the screw pitch error compensation, return the machine zero point can be carried

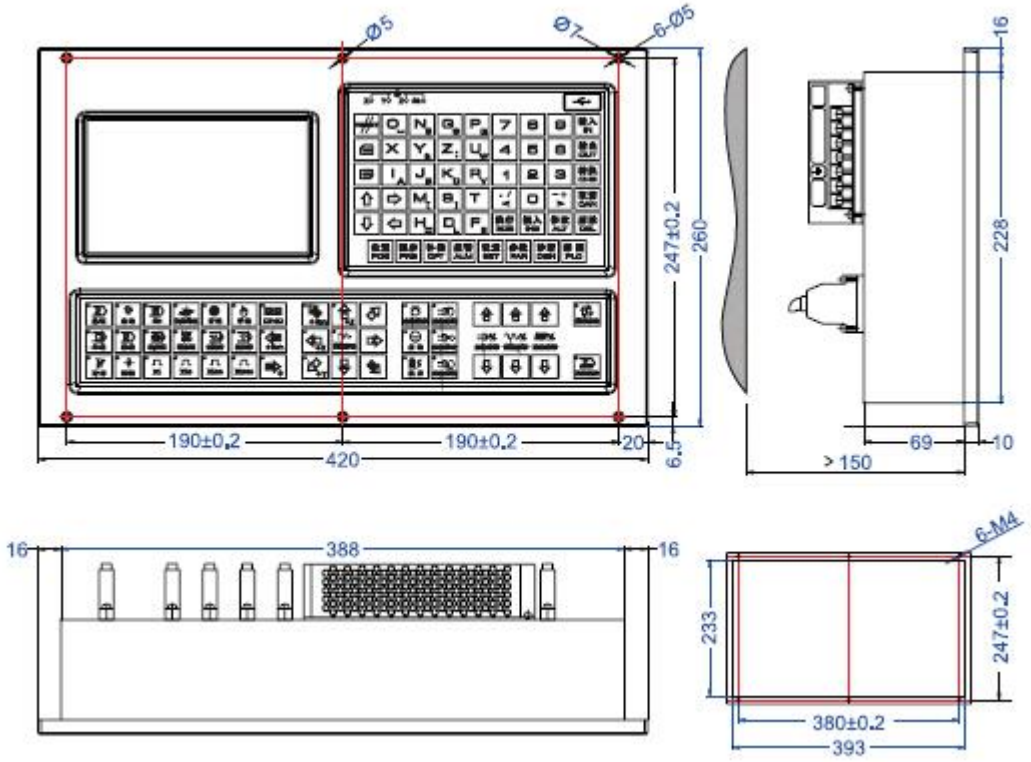
out only after proper compensation.

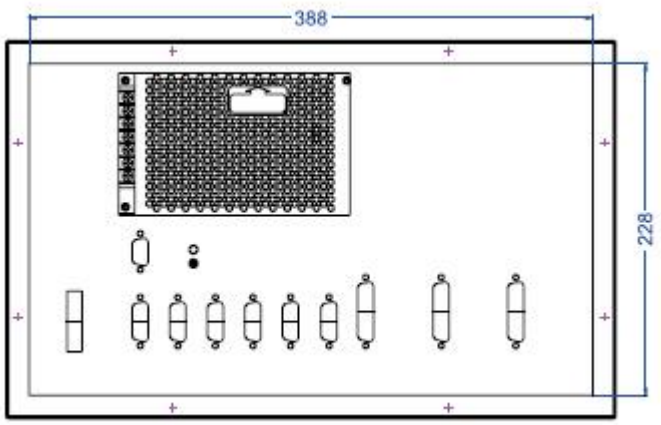
5.5 Example of the compensation parameters setting

① Data parameter№180 (screw-pitch error origin point)=33, data parameter № 185 (screw-pitch offset interval)=10.000mm, the pitch error offset point number is 33 in the following example.

Appendix

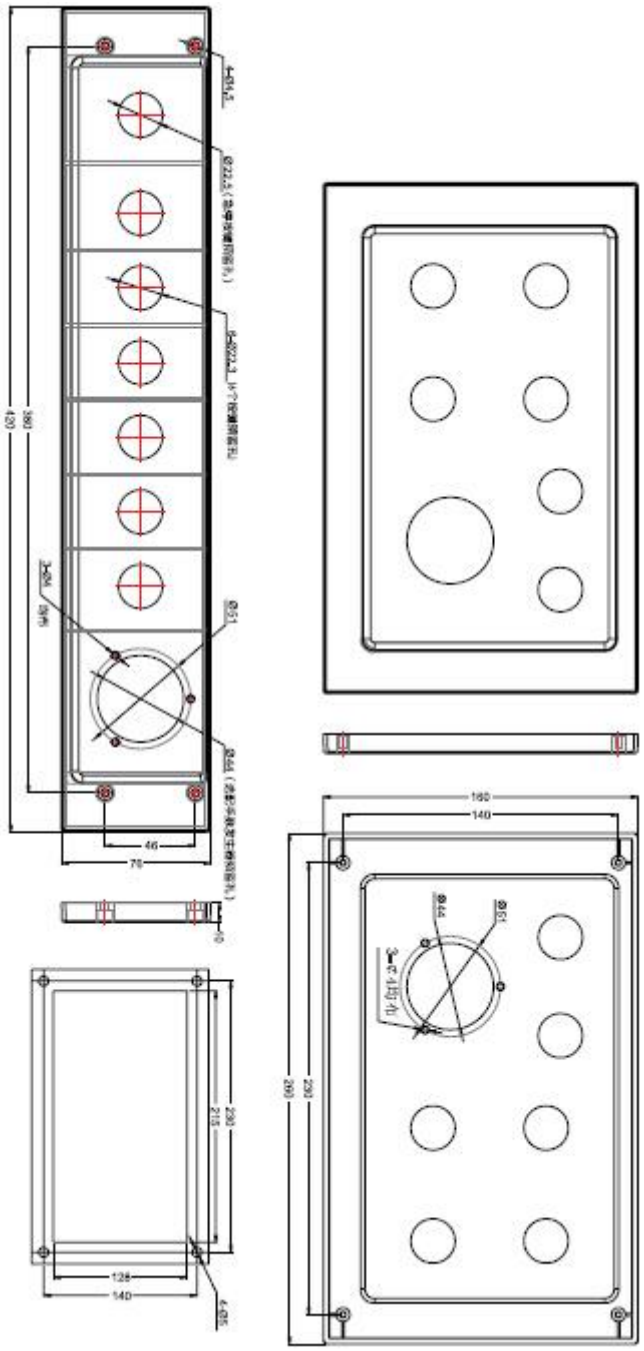
Appendix 1 Outline Dimension of ECN*000M series





ECN*000M series size

Appendix 2 Dimensions of Additional Panel



Appendix 3 Common operations list

Classfic-ation	Function	Operation	Operation mode	Display page	Keyword level	Program switch	Parameter switch	Remark
Reset	Reset X axis relative coordinate	X, 取消 CAN		Relative coordinate				1.3.1 of Volume2
	Reset Y axis relative coordinate	Y&, 取消 CAN		Relative coordinate				
	Reset Z axis relative coordinate	Z:, 取消 CAN		Relative coordinate				
	Reset processing quantity	取消 CAN + N#		Relative coordinate or absolute coorinate				
	Reset cutting time	N# + M[						
Data setting	State parameter	Parameter value, 输入 IN	Input mode	State parameter	Level 2, level 3		Open	10.1.3 of Volume 2
	Data parameter	Parameter value, 输入 IN	Input mode	Data parameter	Level 2, level 3		Open	
	X axis pitch parameter input	X, Compensation value, 输入 IN	Input mode	Pitch parameter	Level 2		Open	

	Y axis pitch parameter input	Y&, Compensation value, 输入 IN	Input mode	Pitch parameter	Level 2		Open	
	Z axis pitch parameter input	Z:, Compensation value, 输入 IN	Input mode	Pitch parameter	Level 2		Open	
	Marco variable value	Marco variable value, 输入 IN		Marco variable value	Level 2, level 3. Level4			1.1.3 of Volume 2
	Tool offset	Compensation value, 输入 IN		Tool offset	Level 2, level 3. Level4			7.4.2 of Volume 2

Classfic-ation	Function	Operation	Operation mode	Display page	Keyword level	Program switch	Parameter switch	Remark
Search	Search down from cursor	Symbol, ⇩	Edit mode	Program contect	Level 2, level 3. Level4	Open		6.1.3 of Volume 2
	Search up from cursor	Symbol, ⇧	Edit mode	Program contect				
	Search down from current place	⌴, ⇩	Edit mode or auto mode	Program contect, program catalogue or program state				6.4.1 of Volume 2
	Search up from current place	⌴, ⇧						6.4.2 of Volume 2

	Search specified program	P Q, program name, 输入 IN						
	Reach of state parameter, data parameter or pitch parameter	P Q, parameter number, 输入 IN		Program relative page				10.1.3 of Volume 2
	PLC state, PLC data search	P Q, address number, 输入 IN		PLC state and data				1.3.7 of Volume 2
Delete	Delete the symbol in cursor	删除 DEL	Edit mode	Program contect	Level 2, level 3. Level4	Open		6.1.6 of Volume 2
		取消 CAN						
	Signal program delete	转换 CHG, N #, SN, 删除 DEL						Program block includes block No. 6.1.7 of Volume 2
	Multiply program block delete	转换 CHG						6.1.8 of Volume 2
	Block delete							6.1.9 of Volume 2
	Signal program delete	O, program name, 删除 DEL						6.3.1 of Volume 2
	Entire program delete	O, −+ >, 9999, 删除 DEL						6.3.2 of Volume 2
Rename	Rename of program	O, program name, 修改 ALT	Edit mode	Program content	Level 2, level 3, level 4			
Copy	Copy of program	O, program name, 转换 CHG						

Switch setting	Open parameter swich			Switch setting	Level 2, level 3			10.1.1of Volume 2
	Open program swich				Level 2, level 3, level 4			
	Open auto SN swich							
	Close parameter swich				Level 2, level 3			
	Close program swich				Level 2, level 3, level 4			
	Close auto SN swich							

Description 1: In the "actions" column ", "said the operation between the two keys have successively order," + "means the operation between the two keys are at the same time.

Example:  ,  **Said the first key**  **, again key**  **, and then press**  +  **said press the two key at the same time.**

Explain 2: Operation mode, display page, password level, program, parameter switch and column blank said has nothing to do with the corresponding function.