

User Manual

**3 Axis 5000Khz
Stand Alone Controller
V4.0**

Warnings and Notes as Used in this Publication

Warning

Warning notices are used in this publication to emphasize that hazardous voltages, currents, temperatures, or other conditions that could cause personal injury exist in this equipment or may be associated with its use.

In situations where inattention could cause either personal injury or damage to equipment, a Warning notice is used.

Caution

Caution notices are used where equipment might be damaged if care is not taken.

Note

Notes merely call attention to information that is especially significant to understanding and operating the equipment.

This document is based on information available at the time of its publication. While efforts have been made to be accurate, the information contained herein does not purport to cover all details or variations in hardware or software, nor to provide for every possible contingency in connection with installation, operation, or maintenance. Features may be described herein which are not present in all hardware and software systems. Shenzhen Guanhong Automation assumes no obligation of notice to holders of this document with respect to changes subsequently made.

Shenzhen Guanhong Automation makes no representation or warranty, expressed, implied, or statutory with respect to, and assumes no responsibility for the accuracy, completeness, sufficiency, or usefulness of the information contained herein. No warranties of merchantability or fitness for purpose shall apply.

SAFETY PRECAUTIONS

This section describes the safety precautions related to the use of CNC units. It is essential that these precautions be observed by users to ensure the safe operation of machines equipped with a CNC unit (all descriptions in this section assume this configuration). Note that some precautions are related only to specific functions, and thus may not be applicable to certain CNC units.

Users must also observe the safety precautions related to the machine, as described in the relevant manual supplied by the machine tool builder. Before attempting to operate the machine or create a program to control the operation of the machine, the operator must become fully familiar with the contents of this manual and relevant manual supplied by the machine tool builder.

1 Definition of Warning , Caution, and Note

This manual includes safety precautions for protecting the user and preventing damage to the machine. Precautions are classified into Warning and Caution according to their bearing on safety. Also, supplementary information is described as a Note. Read the Warning, Caution, and Note thoroughly before attempting to use the machine.

WARNING

Applied when there is a danger of the user being injured or when there is a danger of both the user being injured and the equipment being damaged if the approved procedure is not observed.

CAUTION

Applied when there is a danger of the equipment being damaged, if the approved procedure is not observed.

NOTE

The Note is used to indicate supplementary information other than Warning and Caution.

Read this manual carefully, and store it in a safe place !

2 GENERAL WARNINGS AND CAUTIONS

Warning

1. Never attempt to machine a workpiece without first checking the operation of the machine.

Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

2. Before operating the machine, thoroughly check the entered data. Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

3. Ensure that the specified feedrate is appropriate for the intended operation. Generally, for each machine, there is a maximum allowable feedrate. The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate. If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

4. When using a tool compensation function, thoroughly check the direction and amount of Compensation. Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

5. The parameters for the CNC and PMC are factory-set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change. Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

6. Immediately after switching on the power, do not touch any of the keys on the MDI panel until the position display or alarm screen appears on the CNC unit. Some of the keys on the MDI panel are dedicated to maintenance or other special operations. Pressing any of these keys may place the CNC unit in other than its normal state. Starting the machine in this state may cause it to behave unexpectedly.

7. The operator's manual and programming manual supplied with a CNC unit provide an overall description of the machine's functions, including any optional functions. Note that the optional functions will vary from one machine model to another. Therefore, some functions described in the manuals may not actually be available for a particular model. Check the specification of the machine if in doubt.

8. Some functions may have been implemented at the request of the machine–tool builder. When using such functions, refer to the manual supplied by the machine–tool builder for details of their use and any related cautions.

NOTE: Programs,parameters, and macro variables are stored in nonvolatile memory in the CNC unit. Usually, they are retained even if the power is turned off. Such data may be deleted inadvertently, however, or it may prove necessary to delete all data from nonvolatile memory as part of error recovery. To guard against the occurrence of the above, and assure quick restoration of deleted data, backup all vital data, and keep the backup copy in a safe place.

3 WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

This section covers the major safety precautions related to programming. Before attempting to perform programming, read the supplied operator's manual and programming manual carefully such that you are fully familiar with their contents.

Warning

1. Coordinate system setting

If a coordinate system is established incorrectly, the machine may behave unexpectedly as a result of the program issuing an otherwise valid move command.

Such an unexpected operation may damage the tool, the machine itself, the workpiece, or cause injury to the user.

2. Positioning by nonlinear interpolation

When performing positioning by nonlinear interpolation (positioning by nonlinear movement between the start and end points), the tool path must be carefully confirmed before performing programming. Positioning involves rapid traverse. If the tool collides with the workpiece, it may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3. Function involving a rotation axis

When programming polar coordinate interpolation or normal-direction (perpendicular) control, pay careful attention to the speed of the rotation axis. Incorrect programming may result in the rotation axis speed becoming excessively high, such that centrifugal force causes the chuck to lose its grip on the workpiece if the latter is not mounted securely. Such mishap is likely to damage the tool, the machine itself, the workpiece, or cause injury to the user.

4. Inch/metric conversion

Switching between inch and metric inputs does not convert the measurement units of data such as the workpiece origin offset, parameter, and current position. Before starting the machine, therefore, determine which measurement units are being used. Attempting to perform an operation with invalid data specified may damage the tool, the machine itself, the workpiece, or cause injury to the user.

5. Constant surface speed control

When an axis subject to constant surface speed control approaches the origin of the workpiece coordinate system, the spindle speed may become excessively high. Therefore, it is necessary to specify a maximum allowable speed. Specifying the maximum allowable speed incorrectly may damage the tool, the machine itself, the workpiece, or cause injury to the user.

6. Stroke check

After switching on the power, perform a manual reference position return as required. Stroke check is not possible before manual reference position return is performed. Note that when stroke check is disabled, an alarm is not issued even if a stroke limit is exceeded, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

7. Absolute/incremental mode

If a program created with absolute values is run in incremental mode, or vice versa, the machine may behave unexpectedly.

8. Plane selection

If an incorrect plane is specified for circular interpolation, helical interpolation, or a canned cycle, the machine may behave unexpectedly. Refer to the descriptions of the respective functions for details.

9. Compensation function

If a command based on the machine coordinate system or a reference position return command is issued in compensation function mode, compensation is temporarily canceled, resulting in the unexpected behavior of the machine. Before issuing any of the above commands, therefore, always cancel compensation function mode.

4 WARNINGS AND CAUTIONS RELATED TO HANDLING

This section presents safety precautions related to the handling of machine tools. Before attempting to operate your machine, read the supplied operator's manual and programming manual carefully, such that you are fully familiar with their contents.

Warning

1. Manual operation

When operating the machine manually, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and feedrate have been specified correctly. Incorrect operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

2. Manual reference position return

After switching on the power, perform manual reference position return as required. If the machine is operated without first performing manual reference position return, it may behave unexpectedly. Stroke check is not possible before manual reference position return is performed.

An unexpected operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3. Manual handle feed

In manual handle feed, rotating the handle with a large scale factor, such as 100, applied causes the tool and table to move rapidly. Careless handling may damage the tool and/or machine, or cause injury to the user.

4. Disabled override

If override is disabled (according to the specification in a macro variable) during threading or other tapping, the speed cannot be predicted, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

5. Origin/preset operation

Basically, never attempt an origin/preset operation when the machine is operating under the control of a program. Otherwise, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the tool, or causing injury to the user.

6. Workpiece coordinate system shift

Manual intervention, machine lock, or mirror imaging may shift the workpiece coordinate system. Before attempting to operate the machine under the control of a program, confirm the coordinate system carefully. If the machine is operated under the control of a program without making allowances for any shift in the workpiece coordinate system, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

7. Software operator's panel and menu switches

Using the software operator's panel and menu switches, in combination with the MDI panel, it is possible to specify operations not supported by the machine operator's panel, such as mode change, override value change, and jog feed commands.

Note, however, that if the MDI panel keys are operated inadvertently, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

8. Manual intervention

If manual intervention is performed during programmed operation of the machine, the tool path may vary when the machine is restarted. Before restarting the machine after manual intervention, therefore, confirm the settings of the manual absolute switches, parameters, and absolute/incremental command mode.

9. Feed hold, override, and single block

The feed hold, feedrate override, and single block functions can be disabled using custom macro system variable #3004. Be careful when operating the machine in this case.

10. Dry run

Usually, a dry run is used to confirm the operation of the machine. During a dry run, the machine operates at dry run speed, which differs from the corresponding programmed feedrate. Note that the dry run speed may sometimes be higher than the programmed feed rate.

11. Cutter and tool nose radius compensation in MDI mode

Pay careful attention to a tool path specified by a command in MDI mode, because tool nose radius compensation is not applied. When a command is entered from the MDI to interrupt in automatic operation in tool nose radius compensation mode, pay particular attention to the tool path when automatic operation is subsequently resumed. Refer to the descriptions of the corresponding functions for details.

12. Program editing

If the machine is stopped, after which the machining program is edited (modification, insertion, or deletion), the machine may behave unexpectedly if machining is resumed under the control of that program. Basically, do not modify, insert, or delete commands from a machining program while it is in use.

5 WARNINGS RELATED TO DAILY MAINTENANCE

WARNING

1. Memory backup battery replacement

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.

When replacing the batteries, be careful not to touch the high – voltage circuits (marked and fitted with an insulating cover). Touching the uncovered high – voltage circuits presents an extremely dangerous electric shock hazard.

NOTE: The CNC uses batteries to preserve the contents of its memory, because it must retain data such as programs, offsets, and parameters even while external power is not applied. If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen. When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the contents of the CNC memory will be lost. Refer to the maintenance section of the operator's manual for details of the battery replacement procedure.

2. Absolute pulse coder battery replacement

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work. When replacing the batteries, be careful not to touch the high – voltage circuits (marked and fitted with an insulating cover). Touching the uncovered high – voltage circuits presents an extremely dangerous

electric shock hazard.

NOTE: The absolute pulse coder uses batteries to preserve its absolute position. If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen. When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the absolute position data held by the pulse coder will be lost.

3. Fuse replacement

For some units, the chapter covering daily maintenance in the operator's manual or programming manual describes the fuse replacement procedure.

Before replacing a blown fuse, however, it is necessary to locate and remove the cause of the blown fuse. For this reason, only those personnel who have received approved safety and maintenance training may perform this work. When replacing a fuse with the cabinet open, be careful not to touch the high - voltage circuits (marked and fitted with an insulating cover).

Touching an uncovered high - voltage circuit presents an extremely dangerous electric shock hazard.

Special Attention:

- 1. All the functions of A axis are effective when configure with fourth axis system.***
- 2. When use this system for the first time, please read carefully all the details of each chapter so as to make it work more efficiency and safer.***
- 3. The "Run" button on the panel of system can be used when debugging (P9_D14 in Other parameter to set "Valid" or "Invalid"), must plus an external "Run" button when fitting system, otherwise it may cause accident because of the using life of this key!!! The "Start" key of panel on CNC controller are prohibited to using for long time, otherwise the consequences has nothing to do with my company.***
- 4. The "Pause" button on the panel of system can be used when debugging, P203 on User parameter are set its function to "valid" or "Invalid", must plus an external "Pause" button when user using this cnc system at real processing, otherwise it may cause accident because of the using life of this key!!! The "Pause" key of panel on CNC controller are prohibited to using for long time, otherwise the consequences has nothing to do with my company.***
- 5. Don't plug & insert connectors when system is on power.***
- 6. We will use some short name to stands for parameter number. For example: P -- Parameter , P1 stands for the No.1 parameter .Example: P9-D6,also No.9 parameter, No.7 bit from right to left side.***
- 7. There are some bit parameters on CNC system, different bit parameters have different means, some bit parameters are for inner system,don't change these bits parameter we don't explain,keep same as ex-factory set.***

CONTENTS

Chapter 1 Preface.....	1
1.1 Characteristics.....	2
1.2 Technical Specifications.....	2
1.3 System operation condition.....	3
1.4 G Code List.....	4
Chapter 2 Programming.....	6
2.1 Basic Concepts of Programming.....	6
2.1.1 Tool Movement Along Workpiece Parts Figure Interpolation.....	7
2.1.2 Feed Function.....	8
2.1.3 Part Drawing And Tool Movement.....	8
2.1.4 Coordinate System.....	9
2.1.5 Indicated Command Dimensions for Moving the Tool.....	11
2.1.6 Cutting Speed-Spindle Speed Function.....	12
2.1.7 Tool Function.....	13
2.1.8 Command For Machine Operations.....	13
2.2 Configuration of Program.....	14
2.3 Main Program & Subprogram.....	17
2.4 Program Run.....	17
2.5 Tool Figure And Tool Motion By Program.....	18
2.6 Tool Movement Range-Stroke.....	19
Chapter 3 G INSTRCUTIONS.....	20
3.1 INTRODUCTION.....	20
3.2 G Code List.....	20
3.3 Positioning (Rapid Traverse) (G00).....	24
3.4 Linear Interpolation (G01).....	25
3.5 Circular Interpolation (G02/G03).....	27
3.6 Helical Interpolation (G02/G03).....	30
3.7 3D Space Arc Interpolation G06.....	31
3.8 Dwell (G04).....	32
3.9 Skip Function(G31,G311).....	33
3.10 Reference Position.....	34
3.11 Return to Starting Point (G26,G261~G264).....	35
3.12 Save Current Position (G25).....	35
3.13 Return to Specified Position (G61,G611~G614).....	35
3.14 Coordinate System.....	36
3.14.1 Machine Coordinate System (G53).....	36
3.14.2 Workpiece Coordinate system(G54/G55/G56/G57/G58/G59).....	37
3.14.3 Local Coordinate System (G52).....	39
3.15 Plane Selection(G17/G18/G19).....	41
3.16 Absolute and Incremental Programming (G90/G91).....	41

3.17 Pole coordinate instruction(G15/G16).....	42
3.18 Inch/Metric Conversion (G20/G21).....	44
3.19 Feed Mode.....	44
3.20 Processing Mode (G60/G64).....	45
3.21 Scaling (G36/G37).....	46
3.22 Programmable mirror image (G11/G12).....	47
3.23 Rotate Coordinate Instruction (G68/G69).....	48
3.24 Tool Length Offset (G43/G44/G49).....	50
3.25 Tool Offset(G45/G46/G47/G48).....	53
3.26 Tool Radius Compensation C (G40/G41/G42).....	56
3.27 Details of Tool Compensation C.....	60
3.27.1 Inside and outside.....	60
3.27.2 Tool movement when start-up.....	60
3.28 Canned Cycle (G73-G89).....	66
3.28.1 High-speed Peck Drilling Cycle(G73).....	69
3.28.2 Left-handed Peck Rigid Tapping Cycle (G74).....	71
3.28.3 Fine Boring Cycle (G76).....	73
3.28.4 Canned Cycle Cancel (G80).....	74
3.28.5 Drilling cycle, Spot Drilling (G81).....	75
3.28.6 Drilling Cycle Counter Boring Cycle (G82).....	76
3.28.7 Peck Drilling Cycle (G83).....	77
3.28.8 Right-handed Peck Rigid Tapping Cycle (G84).....	78
3.28.9 Boring Cycle (G85).....	80
3.28.10 Boring Cycle (G86).....	81
3.28.11 Back Boring Cycle (G87).....	82
3.28.12 Boring Cycle (G89).....	84
3.28.13 Example of Canned Cycle.....	85
3.29 Block Cycle (G22,G800).....	87
3.30 Macro program instruction(G65/G66/G67).....	88
3.30.1 Non-Mode Macro Command G65.....	88
3.30.2 Mode Macro Command G66/G67.....	88
3.30.3 Macro Program Instruction.....	89
3.30.3.1 Input Instruction: WAT.....	89
3.30.3.2 Output Instruction: OUT.....	89
3.30.3.3 Assignment Instruction: =.....	89
3.30.3.4 Unconditional Jump: GOTO n.....	89
3.30.3.5 Conditional Jump.....	89
3.30.3.6 Loop Command.....	90
3.30.4 Operators' meaning.....	91
3.30.5 Arithmetic & Logic Operation.....	91
3.30.6 Local Variable.....	91
3.30.7 Global Variable.....	91
3.30.8 System Variable.....	92
3.30.9 System Parameter Variable.....	92

3.30.10 I/O variable.....	92
3.30.11 Message Hint Dialog Box.....	92
3.30.12 Build Processing Program Automatically.....	93
3.31 User-defined Macro Instruction (G110-G170).....	94
3.31.1 Bolt Hole Circle Drilling Cycle.....	94
3.31.2 Inner Circle Groove Roughing Cycle (G110/G111).....	95
3.31.3 Inner Circle Groove Fine Cycle (G112/G113).....	97
3.31.4 Outer Circle Fine Milling Cycle (G114/G115).....	99
3.31.5 Outer Circle Rough Milling Cycle (G116/G117).....	100
3.31.6 Outer Rectangle Rough Milling Cycle (G132/G133).....	102
3.31.7 Inner Rectangle Groove Roughing Cycle (G134/G135).....	105
3.31.8 Inner Rectangle Groove Fine Milling Cycle (G136/G137).....	107
3.31.9 Outer Rectangle Fine Milling Cycle (G138/G139).....	108
3.31.10 Tool Change for Linear Tool Magazine (G120).....	110
3.31.11 Tool change for Hat Tool Magazine (G121/G122).....	111
Chapter 4 M INSTRCUTIONS.....	113
4.1 M Function (Auxiliary Function).....	113
4.1.1 Program Stop(M00).....	113
4.1.2 Optional Stop (M01).....	113
4.1.4 Cycle of Program (M20).....	113
4.1.5 Account of Workpiece(M87).....	113
4.1.6 Unconditional Jump (M97).....	113
4.2 Subprogram Configuration.....	114
4.2.1 Calling of Subprogram (M98).....	114
4.2.2 End of Subprogram (M99).....	114
4.3 Standard PLC M Command List.....	115
4.3.1 M Output Command List.....	115
4.3.1.1 Spindle Control (M03/M04/M05).....	116
4.3.1.2 Spindle Gear Shifting(M41/M42/M43/M44).....	117
4.3.1.3 Coolant(M08/M09).....	117
4.3.1.4 Lubricate(M32/M33).....	118
4.3.1.5 Chuck(M10/M11).....	118
4.3.1.6 Huff(M59/M58).....	118
4.3.1.7 Condition Output of Machine Tool(M65/M67/M69).....	119
4.3.2 M Input Command List.....	119
4.4 Analog Speed of Spindle(S , SS).....	119
4.5 Comprehensive examples for programming.....	120
4.6 Usage for automatic tool setting gauge.....	121
4.7 Usage for automatic dividing center.....	122
4.8 Usage of Following Function of Axes.....	122
Chapter 5 Operation.....	123
5.1 Operational Panel.....	123
5.2 Function Menu.....	123

5.3 Editing Keyboard.....	124
5.4 Machine Control Panel.....	124
5.5 Manual Operation.....	126
5.5.1 Manual Continuous.....	126
5.5.2 Manual Increment.....	127
5.5.3 Manual pulse generator(Handwheel).....	127
5.5.4 Manual Reference Position Return.....	128
5.6 Auto Operation.....	130
5.6.1 Automatic Processing Mode.....	130
5.6.2 Processing at arbitrary program line or with arbitrary tool.....	131
5.6.2.1 Start from “nth” line(block).....	131
5.6.2.2 Start from “N**” line.....	131
5.6.3 Start Program.....	131
5.6.4 Halt Program.....	131
5.6.5 Emergency Stop.....	131
5.6.6 Alarm.....	132
5.6.7 Indicator Light Output.....	133
5.6.8 DNC function.....	133
5.6.8.1 RS232-DNC.....	133
5.6.8.2 USB-DNC.....	133
5.7 External Electrical Connection.....	133
5.7.1 Limitation.....	133
5.7.1.1 Software limitation.....	134
5.7.1.2 External Switch for limitation.....	134
5.7.1.3 Suggestion Usage.....	134
5.8 Diagnosis.....	135
5.9 Programming Operation.....	137
5.9.1 Editing.....	138
5.9.2 Copy.....	140
5.9.3 Delete.....	140
5.9.4 Rename.....	140
5.9.5 Information.....	140
5.9.6 Compile.....	140
5.9.7 Folder management.....	140
5.9.8 Execute Program.....	140
5.9.9 Communication.....	140
5.9.10 U-disk management.....	141
5.9.10.1 Function Keys of USB-disk.....	142
5.9.10.2 Management of Processing Program.....	142
5.9.10.3 Management of Parameters & Software.....	143
Chapter 6 Parameter List.....	144
6.1 User Parameter.....	145
6.2 Speed parameter.....	150
6.3 Axis parameter.....	160

6.4 Tool parameter.....	168
6.5 Other Parameter.....	169
6.6 Workpiece Coordinate Parameter.....	177
6.6.1 How to set up the workpiece coordinate system?.....	178
6.6.2 How to adjust the offset value after set well?.....	178
6.7 Password.....	179
6.8 Redeem.....	180
6.8.1 Radius Compensation.....	180
6.8.2 Length of redeem.....	181
6.8.3 Tool Sets List.....	182
6.8.4 Set quantity.....	182
6.9 Screw Compensation.....	183
Chapter 7 Installation & Connection.....	186
7.1 system installation and connection.....	186
7.2 system installation dimension.....	186
7.3 System Rear View.....	187
7.4 Interface Connection Graph.....	188
7.4.1 CN6 Communication Socket (Female/DB9).....	188
7.4.2 CN4 Turret Socket (Female/DB15).....	189
7.4.3 CN9 Spindle Encoder Socket (Female/DB9).....	190
7.4.4 CN3 IO1 Control Socket (Female/DB25).....	191
7.4.5 CN10 IO2 Socket (Female/DB25).....	192
7.4.6 CN5 XYZA Drive Socket (Male/DB25).....	193
7.4.7 CN11 MPG/Handheld Box Socket (Male/DB15).....	194
7.4.7.1 Electrical handwheel (Manual pulse generator).....	194
7.4.7.2 Using for Band Switch.....	195
7.4.7.3 External Switch for Run/Halt.....	195
7.4.7.4 Using for External Emergency Stop.....	195
7.5 SZGH-CNC-IO-12 IO Relay Board.....	196
7.6 Daily Maintenance and Repair.....	199
7.6.1 Maintain.....	199
7.6.2 Ordinary Problem.....	199
Appendix I: Wiring Diagram of CN3 Plug.....	200
Appendix II: Wiring Diagram of CN10 Plug.....	201
Appendix III: Wiring Diagram of CN4 Plug.....	202

Chapter 1 Preface

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

CNC controller is the newest universal CNC system for milling machine, drilling machine, boring machine , which is developed by our company. The control circuit is using the latest industrial high-speed ARM 11 processor, large-scale field and programmable FPGA technology, multi-layer PCB, the machine adopts the high integrated chip and surface mount components, the structure is more compact and reasonable so that make sure the reliability and stability of the system.

CNC controller can control four feeding axes, two analog spindles, 2ms interpolation in real-time control with high speed (2MHz Pulse Output Frequency) and 1um precision,which obviously improve the efficiency, precision and surface quality of processing parts. Adopt 800x600 dot matrix TFT LCD adaptive brightness LCD display, LED backlight brightness uniformity and long service life, and overcomes the display brightness by environmental temperature changes the shortcoming. Full screen English menu display, operation is simple and convenient.

This manual introduce the programming and using method of milling system.



Fig1.1 SZGH-CNC990MDb

Note: This picture of CNC controller just for reference,maybe we will make update in the future!

1.1 Characteristics

- 1) 800*600 8.4 inch real color LCD Display
- 2) Support ATC function, Macro function and PLC function
- 3) Support Linear tool magazine,umbrella tool magazine
- 4) 128MB Memory, 100Mb user store room
- 5) 2MHz Pulse Output Frequency, Default Pulse equivalent is 1um.
- 6) PLC On-line Display,Monitor & Design
- 7) High anti-jamming switch power(220VAC -> 24VDC & 5VDC)
- 8) With USB interface, for upgrade & copy programs
- 9) Fully English Display & keys,easy to operate
- 10) Analog voltage output of 0~10V in two channels, support double spindles
- 11) Adapted servo spindle can realize position,rigid tapping,threading of spindle
- 12) Basic I/Os : 40*24
- 13) Built-in screw compensation,Reverse backlash compensation
- 14) English menu, program and interface, full screen edition
- 15) Support macro variable dialog box & Running program by input points

1.2 Technical Specifications

Max Number of control axes

- Number of control axes: 4 axes (X Y Z A)
- Number of linkage axes: 4 axes
- Number of PLC control axes: 4 axes

Feeding axes function

- Minimum command unit: 0.001mm
- Position command range: +/- 99999.999
- Max speed: 30 m/min Feeding speed: 0.001-15m/min
- G00 rapid override: Total 8 levels: 0~150%, real-time adjusting
- Feeding override: Total 16 levels: 0~150%, real-time adjusting
- Spindle override: Total 16 levels: 5%~150%, real-time adjusting
- Interpolation mode: Interpolation of linear ,arc ,thread and rigid tapping
- Auto chamfer

Spindle function

- Analog voltage 0~10V output in two channels ,support two-spindle control
- Spindle encoder feedback in one channel, resolution of spindle encoder can be set
- Spindle speed: It is set by speed parameter,max spindle speed also corresponding to 10V
- Spindle override: Total 16 levels: 5%~150%, real-time adjusting
- Spindle constant surface speed control

Precision compensation

- Backlash compensation/Pitch error compensation in memory type
- Built-in Thread Compensation

PLC function

- Refresh cycle: 8ms
- PLC program can be altered on PC , download by USB interface

- I/Os : 40*24 I/Os
- Support On-line display,monitor & alter ladder

Man-machine interface

- 8.4" large screen real-color LCD , the resolution is 480 000
- Display in Chinese or English
- Display in 3D tool path
- Real-time clock
- Operation management
- Operate mode:Auto, Manual, MDI, mechanical zero return, MPG/single step.
- Operation authority of multiple-level management
- Alarm record

Edit program

- Program capacity: 128M
- Editing function: program/block/characters research ,rewriting and deleting
- Program format: ISO-840 code,support Macro command programming, programming of relative coordinate ,absolute coordinate and hybrid coordinate,fit to ISO-841 international standard.
- Calling program: Support macro program ,subprogram

Safety function

- Emergency stop
- Hardware travel limit
- Software travel limit
- Data restoring and recovering
- User-defined alarm hint

1.3 System operation condition

1) Power supplying

AC 220V(+10%/-15%), Frequency 50Hz±2%. Power: \leq 200W.

Note: it should use isolation transform to supply power first input:380V-220VAC or add noise filter.

2) Climate condition

Item	Working Conditions	Storage&Delivery Conditions
Ambient Temperature	5 °C~45 °C	0 °C~+55 °C
Ambient Humidity	40~80%	\leq 90%(40 °C)
Atmosphere Pressure	86kPa~106kPa	86kPa~106kPa
Altitude	\leq 1000m	\leq 1000m

3) operation environment :

No excessive flour dust, no acid, no alkali gas and explosive gas, no strong electromagnetic interference.

1.4 G Code List

CODE	Description	CODE	Description
G00	Rapid Positioning	G17	XY plane selection
G01	Linear Interpolation	G18	ZX plane selection
G02	Circular Interpolation CW	G19	YZ plane selection
G03	Circular Interpolation CCW	G90	Absolute value programming
G31	Jumping function	G91	Incremental value programming
G311	Jumping function	G30	Return to zero point of coordinate
G04	Dwell	G301	X-axis Return to zero point of coordinate
G15	Polar coordinate command cancel	G302	Y-axis Return to zero point of coordinate
G16	Polar coordinate command	G303	Z-axis Return to zero point of coordinate
G20	Inch input	G304	A-axis Return to zero point of coordinate
G21	Millimeter input	G26	Return to starting point of program
G37	Scaling Cancel	G261	X-axis Return to starting point of program
G36	Scaling	G262	Y-axis Return to starting point of program
G12	Programmable mirror image Cancel	G263	Z-axis Return to starting point of program
G11	Programmable mirror image	G264	A-axis Return to starting point of program
G60	Exact stop & position	G265	B-axis Return to starting point of program
G64	Continuous track processing	G25	Save current coordinate value
G65	Macro command non-mode calling	G61	Return to position of G25
G66	Macro command mode calling	G611	X-axis return to position of G25
G67	Macro program mode calling calling	G612	Y-axis return to position of G25
G68	Constant surface speed control	G613	Z-axis return to position of G25
G69	Constant surface speed control cancel	G614	A-axis return to position of G25
G73	High Speed Peck Drilling Cycle	G615	B-axis return to position of G25
G80	Canned cycle for drilling cancel	G28	Return to home of machine
G81	Dot Drilling cycle	G281	X-axis return to home of machine

G82	Boring Stage hole cycle	G282	Y-axis return to home of machine
G83	Deep hole drilling cycle	G283	Z-axis return to home of machine
G85	Boring Cycle	G284	A-axis return to home of machine
G86	Boring Cycle	G285	B-axis return to home of machine
G89	Boring Cycle	G52	Local Coordinate System Setup
G74	Reverse Tapping Cycle	G53	Machine coordinate system
G84	Tapping Cycle	G54	Workpiece coordinate system 1
G94	Feed per minute	G55	Workpiece coordinate system 2
G95	Feed per revolution	G56	Workpiece coordinate system 3
G98	Return to initial level	G57	Workpiece coordinate system 4
G99	Return to R point level	G58	Workpiece coordinate system 5
G110	CCW round groove roughing	G22	Program Cycle
G111	CW round groove roughing	G800	Program Cycle Cancel
G112	Finishing in CCW full circle	G40	Tool nose radius compensation cancel
G113	Finishing in CW full circle	G41	Tool nose radius left compensation
G114	CCW outer circle finishing	G42	Canned Tapping Cycle
G115	CW outer circle finishing	G43	Radial Cutting Cycle
G116	CCW outer circle groove roughing	G44	Program Cycle
G117	CW outer circle groove roughing	G49	Program Cycle Cancel
G132	CCW rectangle outer roughing	G45	Polar coordinate command cancel
G133	CW rectangle outer roughing	G46	Polar coordinate command
G134	CCW rectangle groove roughing	G47	Absolute value programming
G135	CW rectangle groove roughing	G48	Incremental value programming
G136	CCW rectangle groove finishing	G120	Tool changing for linear tool magazine
G137	CW rectangle groove finishing	G121	Tool changing for umbrella tool magazine
G138	CCW rectangle outer finishing	G122	Tool changing for Arm tool magazine
G139	CW rectangle outer finishing	G06	3D Space Arc Interpolation

Chapter 2 Programming

CNC milling System is highly effective automatic equipment according to programmed program to process workpiece. Programming is using the CNC system control language according to the requirement and drawing of the workpiece to describe the processing trajectory and the assistant action. Ideal system not only could promise process qualified workpiece, but also make the functions of milling reasonable application and fully use, so it is very important to programming, this chapter will introduce many kinds of instructions and usage of CNC program, please read carefully.

2.1 Basic Concepts of Programming

Item	SZGH-CNC990MDb
No. of Basic Controllable Axes	3 (X, Y, Z)
Expansion of controlled axis(total)	Up to 4(X, Y, Z, A)

Table 2-1-1

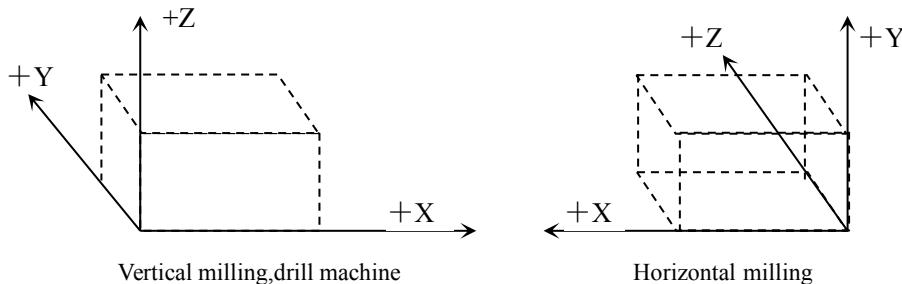
Program Segment: a complete command line consisted of instruction and data .

Program: is a congregation of program segment by machining logic structure in order to complete the machining of workpiece.

Machine Coordinate System: The establishment of coordinate is based on machine's zero point. The milling machine coordinate axis and its direction should follow to "ISO841" standard. The method as follow: Through right hand rule we can make the program coordinate, The Z axis is parallel as spindle, The X axis is horizontal, The Y axis is determined by right hand rule. The A, B, C are rotated axis or assistant axis which parallel as X,Y,Z axis. Furthermore, The coordinate axis direction is the increasing workpiece dimension direction.

When without set work coordinate, take machine coordinate as work coordinate.

Machine Coordinate & direction
sketch map



Working Coordinate System: Work piece processing uses the coordinate system is called as the work piece coordinate system, it is set by CNC. The work piece coordinate system could change to move its zero point.

Set the workpiece coordinates:

- 1.Use G54 ~ G59: Use coordinate parameters to set 6 workpiece coordinate system.
- 2.G52: Set a value behind G52 to set the workpiece coordinate system in program.

With absolute value instruction ,it must use the above method to establish the work piece coordinate system.

Local coordinate system: Set local coordinate system of workpiece coordinate system in order to programme easily when programming in workpiece coordinate system.

Absolute Programming: coordinates data of programming mode based on established absolute coordinate system. Absolute coordinate value is corresponding to homing point of coordinate system. It is settlements by “G90 X_Y_Z_A_”.

Relative Programming(increment programming): It is distance and direction of operation end point ,compared with starting point. It is settlements by “G90”.

Mode Instruction: The instruction which can remain the function in the program. It works both in this program and program in the future.

In the same operation, there may be several mode instruction, such as M03(spindle clockwise),M04(spindle counter clockwise),M05(spindle stop).They are all mode used to control spindle. The mode of same kind are categorized into one mode group. At any time it must be one of them, and there is only one of them. The original chosen mode is called mode origin. In the above mode group, M05 is such a mode origin .

Suspending Mode(destroying mode): It is instruction which can turn mode instruction into mode origin or destroy the mode. Such as M20(program ending instruction), meaning the end of operation and returning to original status.

Non-Mode instruction: It is the instruction which has no function to store, and only works in the segment of program.

2.1.1 Tool Movement Along Workpiece Parts Figure Interpolation

The tool moves along straight lines and arcs constituting the workpiece parts figure(See Fig2.1.1)

The function of moving the tool along straight lines and arcs is called the interpolation.

a) Tool movement along a straight line

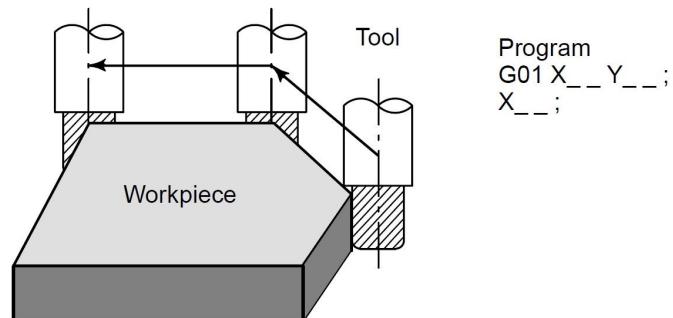


Fig2.1.1 Tool Movement along a line

b) Tool movement along an arc

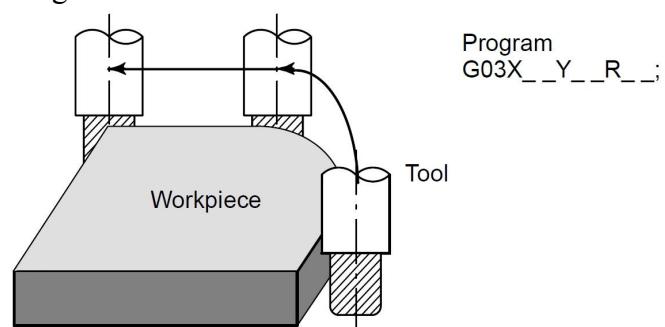


Fig2.1.2 Tool Movement along an arc

Symbols of the programmed commands G01, G02, ... are called the preparatory function and specify the type of interpolation conducted in the control unit.

(a) Movement along straight line (b) Movement along arc

G01 Y—
X—Y—;

G03X—Y—R—;

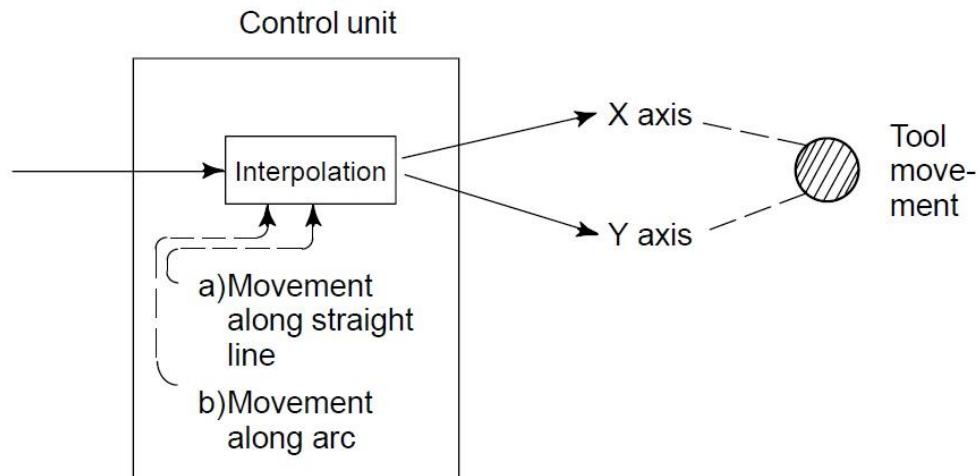
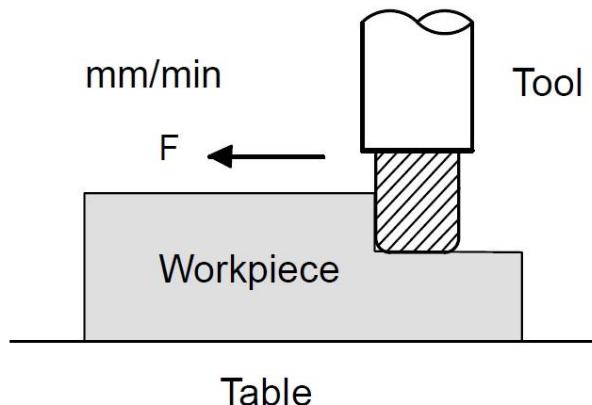


Fig2.1.3 Interpolation

Note: Some machine move tables instead of tools but this manual assumes that tools are moved against workpieces.

2.1.2 Feed Function

Movement of the table at a specified speed for cutting a workpiece is called the feed.



Table

Fig2.1.4 Feed Function

Feedrate can be specified by using actual numeric. For example, to feed the tool at a rate of 150 mm/min, specify the following in the program: F150.0

The function of deciding the feed rate is called the feed function.

2.1.3 Part Drawing And Tool Movement

A CNC machine tool is provided with a fixed position. Normally, tool change and programming of absolute zero point as described later are performed at this position. This position is called the reference position.

Reference position

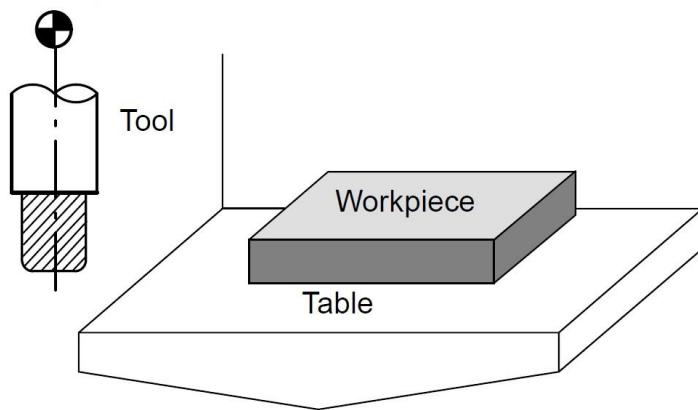


Fig2.1.5 Reference Position

The tool can be moved to the reference position in two ways:

- (1) Manual reference position return (See Chapter 5.5.4)

Reference position return is performed by manual button operation.

- (2) Automatic reference position return (See Chapter 3.10)

In general, manual reference position return is performed first after the power is turned on. In order to move the tool to the reference position for tool change thereafter, the function of automatic reference position return is used.

2.1.4 Coordinate System

Coordinate system on part drawing and coordinate system specified by CNC coordinate system.

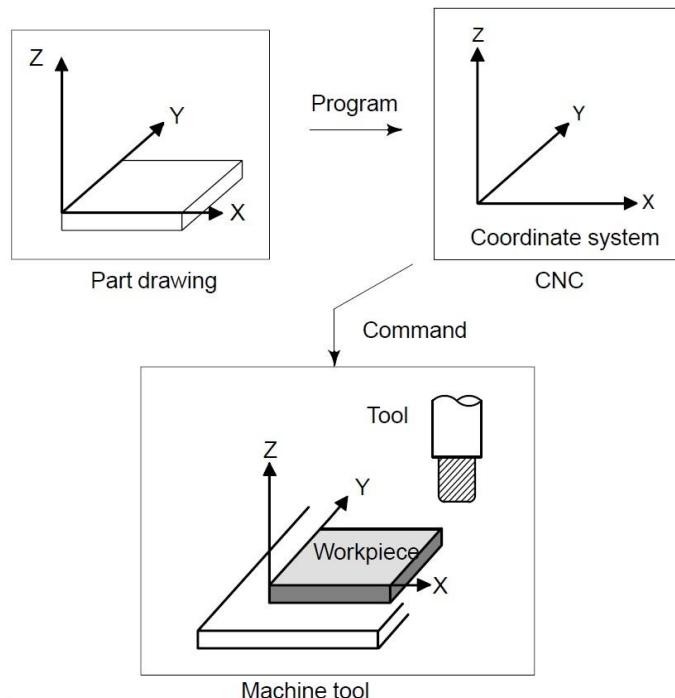


Fig2.1.6 Coordinate System

The following two coordinate systems are specified at different locations: (See chapter 3.14)

- (1) Coordinate system on part drawing

The coordinate system is written on the part drawing. As the program data, the coordinate

values on this coordinate system are used.

(2) Coordinate system specified by the CNC

The coordinate system is prepared on the actual machine tool table. This can be achieved by programming the distance from the current position of the tool to the zero point of the coordinate system to be set.

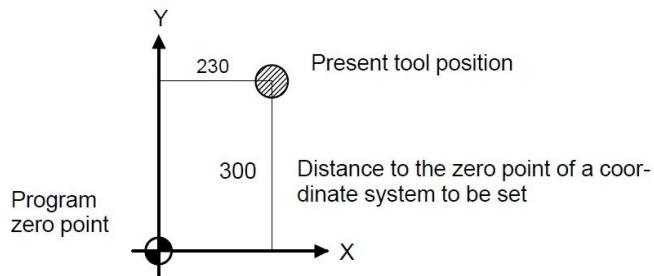


Fig2.1.7 Coordinate system specified by the CNC

The positional relation between these two coordinate system is determined when a workpiece is set on the table.

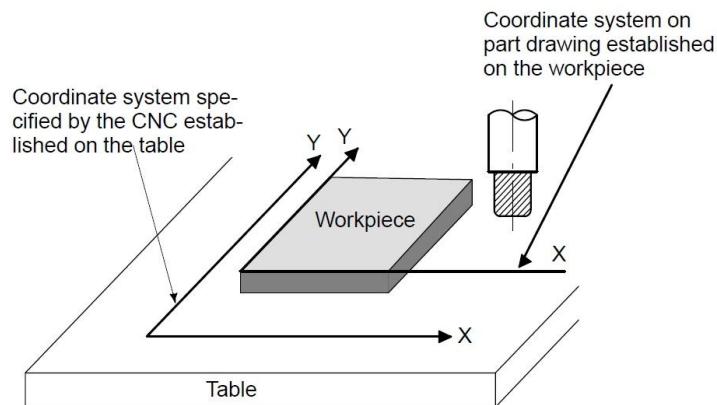


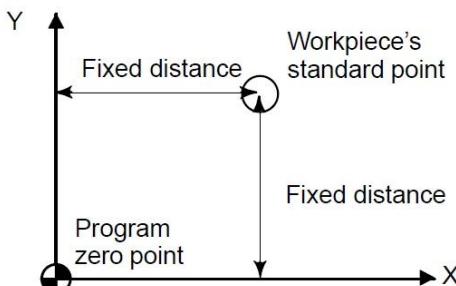
Fig2.1.8 Coordinate system specified by CNC on part drawing

The tool moves on the coordinate system specified by the CNC in accordance with the command program generated with respect to the coordinate system on the part drawing, and cuts a workpiece into a shape on the drawing.

Therefore, in order to correctly cut the workpiece as specified on the drawing, the two coordinate systems must be set at the same position.

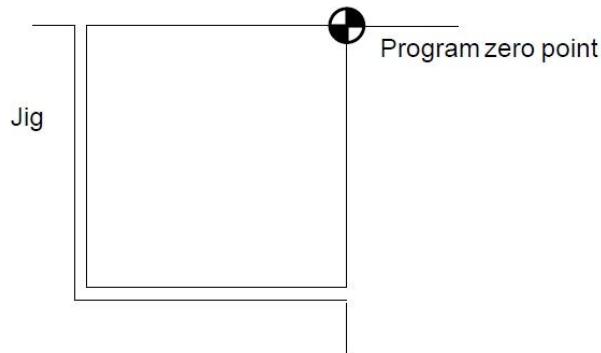
To set the two coordinate systems at the same position, simple methods shall be used according to workpiece shape, the number of machining.

(1) Using a standard plane and point of the workpiece



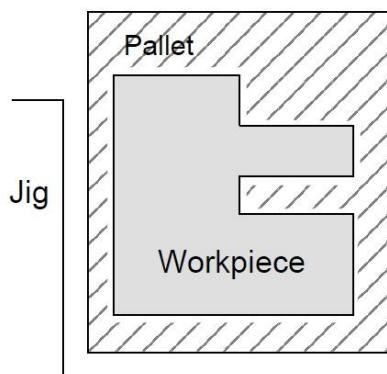
Bring the tool center to the workpiece standard point.
And set the coordinate system specified by CNC at this position

(2) Mounting a workpiece directly against the jig



Meet the tool center to the reference position. And set the coordinate system specified by CNC at this position. (Jig shall be mounted on the predetermined point from the reference position.)

(3) Mounting a workpiece on a pallet, then mounting the workpiece and pallet on the jig

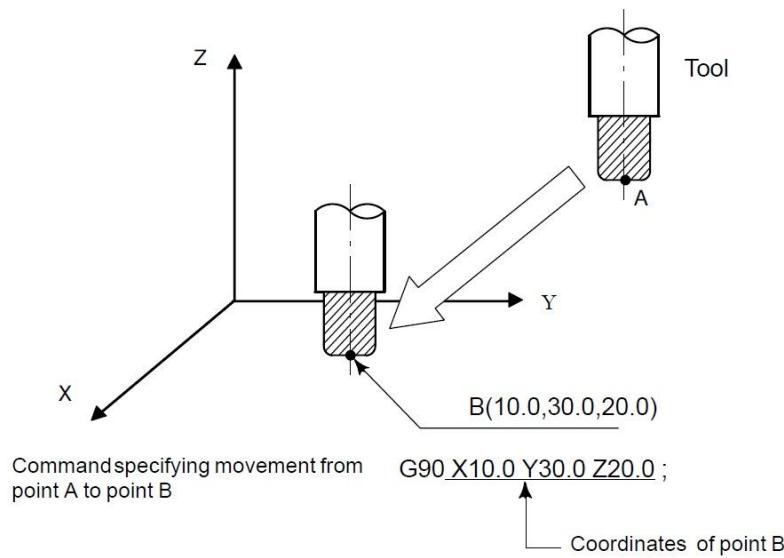


(Jig and coordinate system shall be specified by the same as (2)).

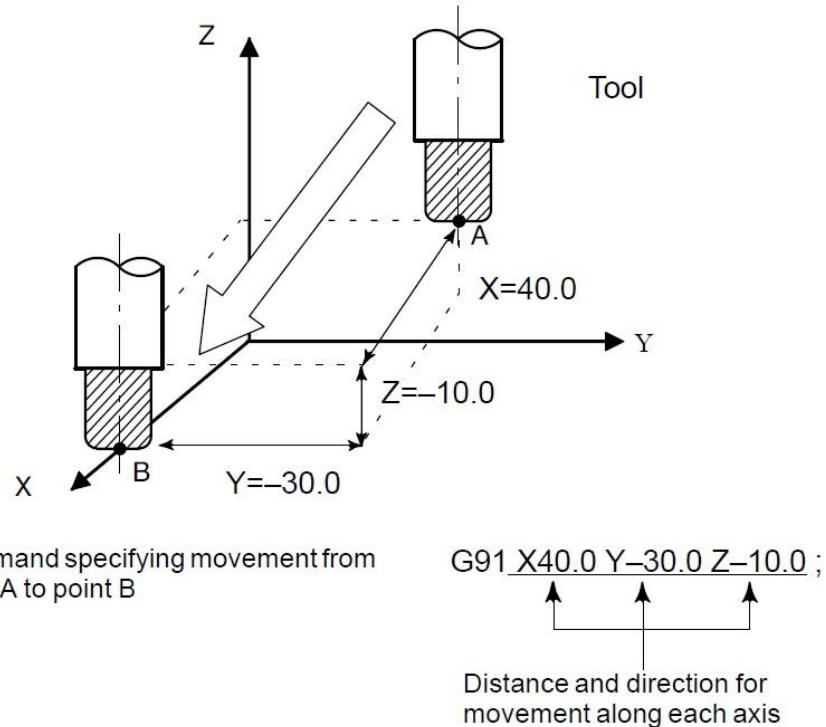
2.1.5 Indicated Command Dimensions for Moving the Tool

Command for moving the tool can be indicated by absolute command or incremental command.(See Chapter 3.16)

(a) **Absolute Command:** the tool moves to a point at “the distance from zero point of the coordinate system” that is to the position of the coordinate values.



(b) **Incremental Command:** Specify the distance from the previous tool position to the next tool position.



2.1.6 Cutting Speed-Spindle Speed Function

The speed of the tool with respect to the workpiece when the workpiece is cut is called the cutting speed.

As for the CNC, the cutting speed can be specified by the spindle speed in min^{-1} unit.

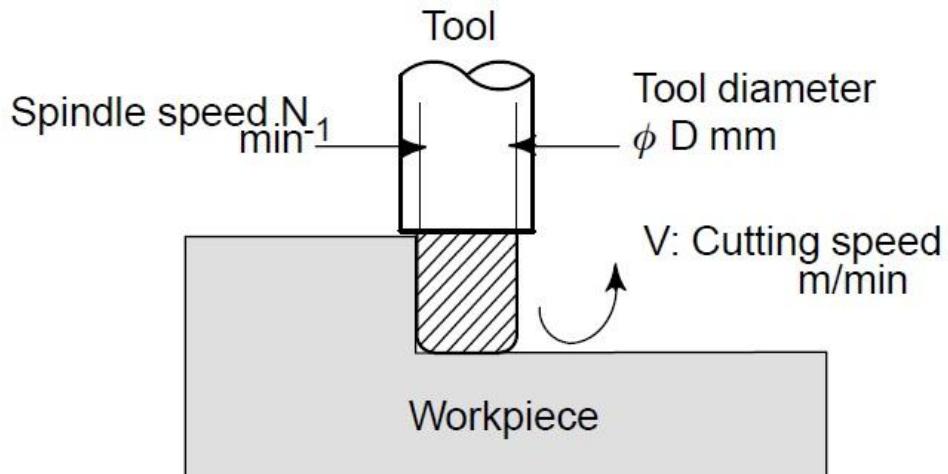


Fig2.1.9 Cutting Speed

<When a workpiece should be machined with a tool 100mm in diameter at a cutting speed of 80m/min.>

The spindle speed is approximately 250 min^{-1} , which is obtained from $N=1000v/\pi D$. Hence the following command is required:

S250;

Commands related to the spindle are called the spindle speed function (See Chapter 4.4).

2.1.7 Tool Function

When drilling, tapping, boring, milling or the like, is performed, it is necessary to select a suitable tool. When a number is assigned to each tool and the number is specified in the program, the corresponding tool is selected.

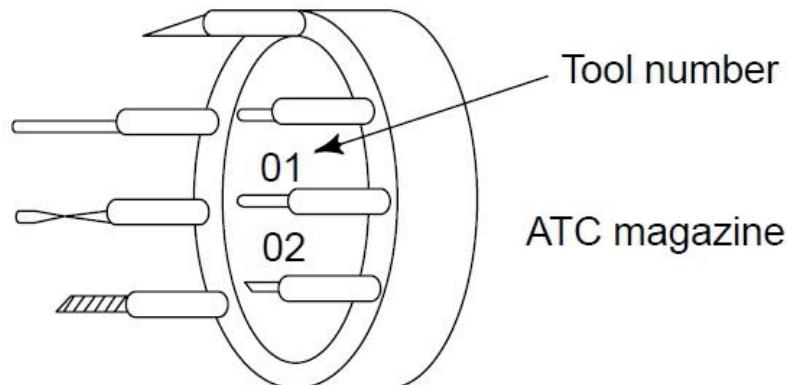


Fig2.1.10 Tool Magazine

<When No.01 is assigned to a drilling tool>

When the tool is stored at location 01 in the ATC magazine, the tool can be selected by specifying T01. This is called the tool function.

2.1.8 Command For Machine Operations

When machining is actually started, it is necessary to rotate the spindle, and feed coolant. For this purpose, On-Off operations of spindle motor and coolant valve should be controlled.

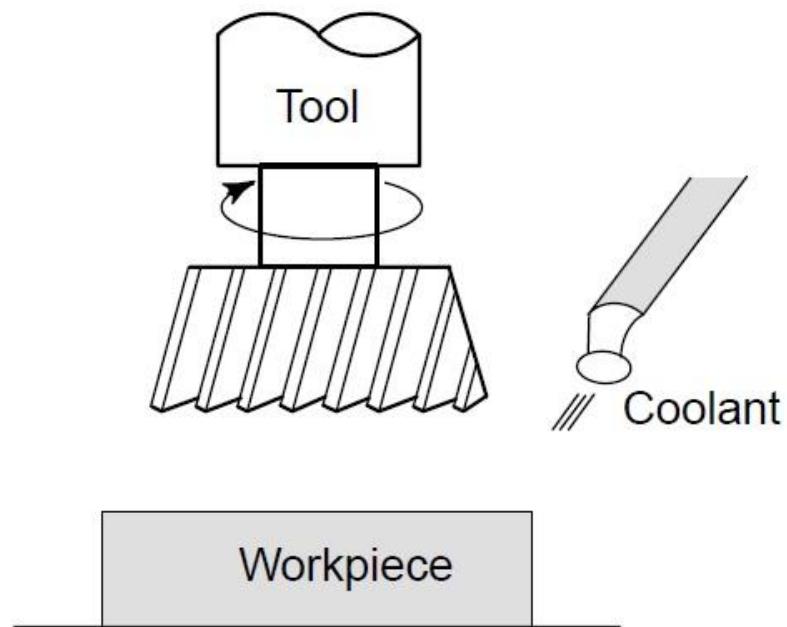


Fig2.1.11 Miscellaneous Function

The function of specifying the On-Off operations of the components of the machine is called the miscellaneous function. In general, the function is specified by an M code (See chapter 4)

For example, when M03 is specified, the spindle is rotated clockwise at the specified spindle speed.

2.2 Configuration of Program

A group of commands given to the CNC for operating the machine is called the program. User needs to compile part programs according to instruction formats of CNC system. By specifying the commands, the tool is moved along a straight line or an arc, or the spindle motor is turned on and off.

In the program, specify the commands in the sequence of actual tool movements.

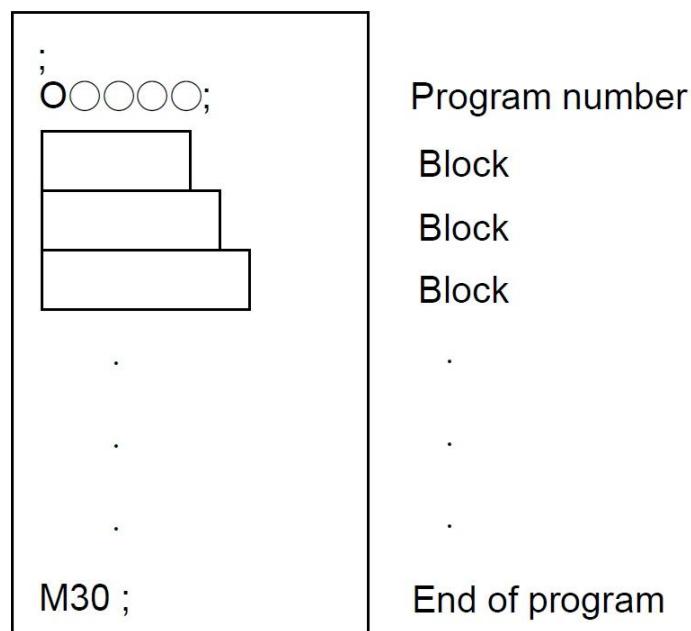


Fig2.1.12 Program Configuration

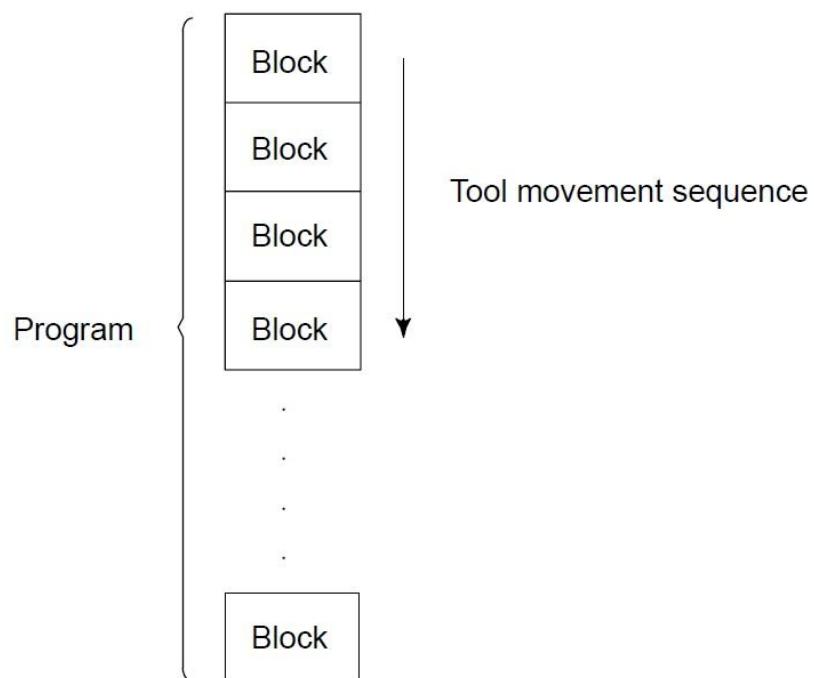


Fig2.1.13 Configuration of Program

Normally, a program number is specified after the end - of - block (;) code at the beginning of the program, and a program end code (M02 or M30) is specified at the end of the program.

See the general structure of program as follows:

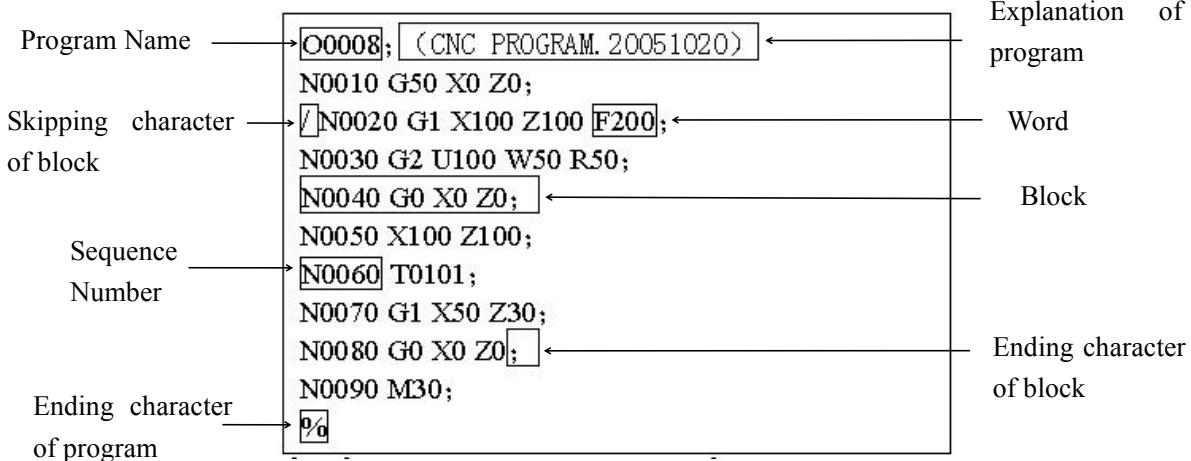


Fig2.28 General Structure of Program

Program Name: consist of alphabet & number (Eg.: O0001). There are countless programs stored in the system. To identify it, each program has only one program name (there is no the same program name).

Note: It doesn't allow exist blank on program name.

A group of commands at each step of the sequence is called the block. The program consists of a group of blocks for a series of machining. The number for discriminating each block is called the sequence number, and the number for discriminating each program is called the program number.

Block: a group of commands at each step of the sequence.

The program consists of a group of blocks for a series of machining. The number for discriminating each block is called the sequence number, and the number for discriminating each program is called the program number.

The block and the program have the following configurations.

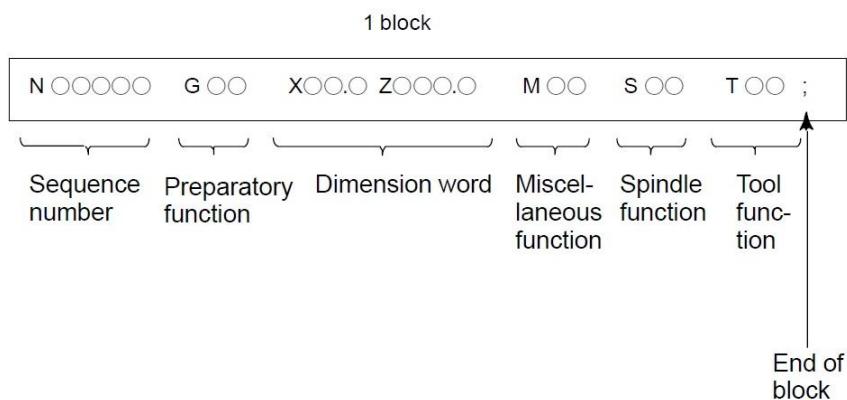


Fig2.29 Block Configuration

A block begins with a sequence number that identifies that block and ends with an end - of - block code.

This manual indicates the end - of - block code by ; (LF in the ISO code and CR in the EIA code).

The contents of the dimension word depend on the preparatory function.

In this manual, the portion of the dimension word may be represent as IP_.

There is only one for other addresses except for N, G, S, T, H, L in one block, otherwise the

system alarms. The last word in the same address is valid when there are more N, G, S, T, H, L in the same block. The last G instruction is valid when there are more G instructions which are in the same group in one block.

Word is the basic instruction unit to command CNC system to complete the control function, composed of an English letter (called instruction address) and the following number (operation instruction with/without sign). The instruction address describes the meaning of its following operation instruction and there may be different meaning in the same instruction address when the different words are combined together. Table 2-1 is Word List of SZGH-CNC990MDB system.

Functions	Address symbol	meaning	Data range
Name of file		Name of machining workpiece	0-9, A-Z
Program segment No.	N	No. of program segment	0000-9999
Preparation function	G	Content and mode of designated instruction operation	00-99
Auxiliary function	M	Auxiliary operation instruction	00-99
Tool chosen	T H D	No. /compensation of Tool	01-99
Spindle function	S	Set the speed of 1st spindle;	0-99999
	SS	Set the speed of 2nd spindle;	
Cutting speed	F	Speed per minute	0.01-15m/m
Coordinates character	X Y Z A	The coordinates value of X Y Z A axes.	+/-99999.999mm
Core coordinates	I J K	X Z axes and Z axes core coordinate increment value	+/-99999.999mm
Step length	R	Circular arc radius	0.001-99999.999mm
Delay time	P,X	Designated delay time of G04	0.001-65s
Program entrance	P	Entrance of calling program name	0000-9999
Repeat times	L	Times of cycle or subprogram calling	1-9999
Line skip	/	No executive program when “/” is in front of program line	P9_D12 in Other parameter: shield the function

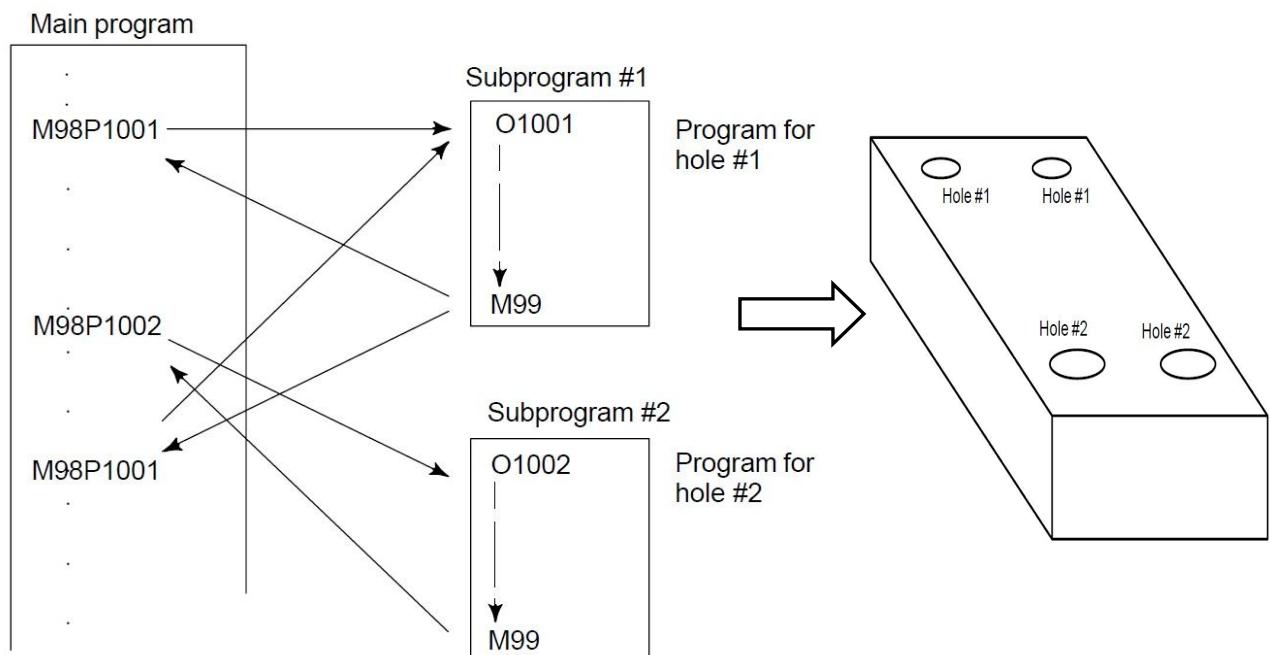
Attention 1: “-” means this data can be use.

Attention 2: In front of the numeral is 0, indicated this data only write the effective data.

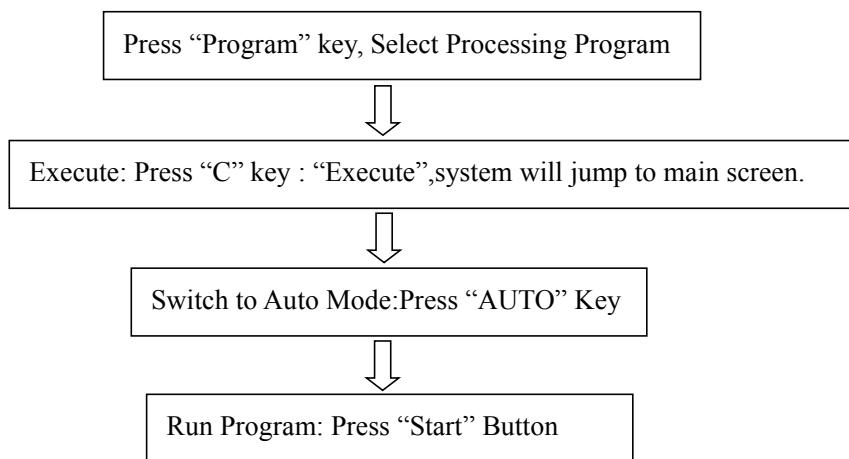
Attention 3: The digital presentation is a figure, when is two, top digit expression integer figure biggest figure, after low position expresses decimal point most imperial throne.

2.3 Main Program & Subprogram

When machining of the same pattern appears at many portions of a program, a program for the pattern is created. This is called the subprogram. On the other hand, the original program is called the main program. When a subprogram execution command appears during execution of the main program, commands of the subprogram are executed. When execution of the subprogram is finished, the sequence returns to the main program.



2.4 Program Run



Suggestion: Before running program, please compile program, and ensure program is right.

2.5 Tool Figure And Tool Motion By Program

Machining using the end of cutter--tool length compensation function.(See chapter 3.24)

Usually, several tools are used for machining one workpiece. The tools have different tool length. It is very troublesome to change the program in accordance with the tools.

Therefore, the length of each tool used should be measured in advance. By setting the difference between the length of the standard tool and the length of each tool in the CNC (data display and setting: See Chapter 3.24), machining can be performed without altering the program even when the tool is changed. This function is called tool length compensation.

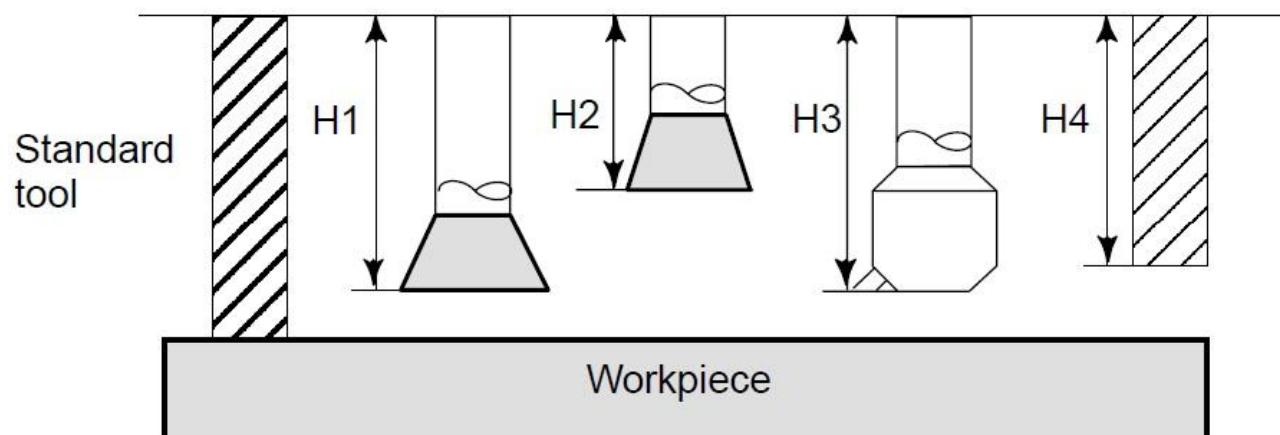


Fig2.5.1 Tool Length Compensation

Machining using the side of cutter -- Cutter Compensation Function

Because a cutter has a radius, the center of the cutter path goes around the workpiece with the cutter radius deviated.

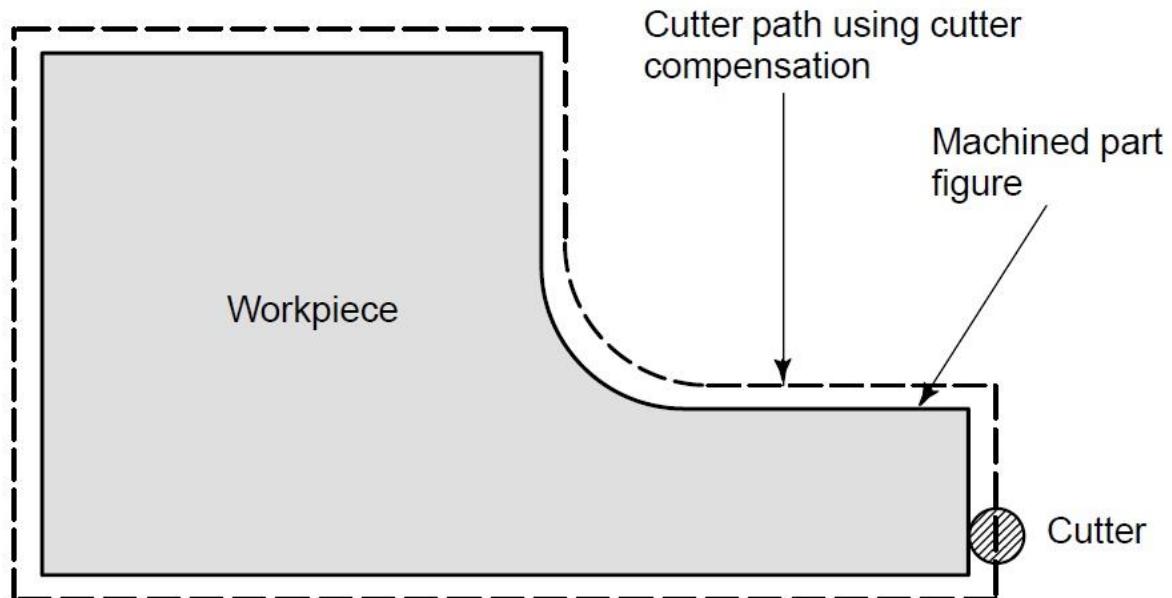
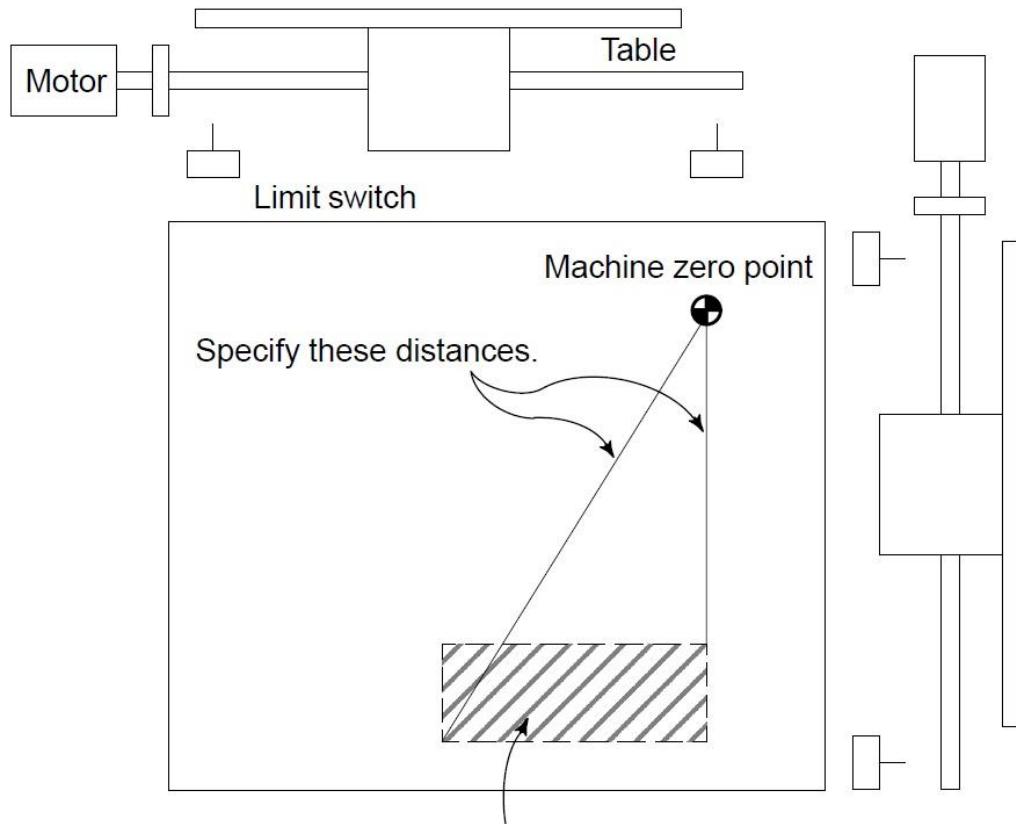


Fig2.5.2 Tool Radius Compensation

If radius of cutters are stored in the CNC(Data display and Setting: See Chapter7.8.1), the tool can be moved by cutter radius apart from the machining part figure. This function is called cutter compensation, also tool radius compensation.

2.6 Tool Movement Range-Stroke

Limit switches are installed at the ends of each axis on the machine to prevent tools from moving beyond the ends. The range in which tools can move is called the stroke.



Tools cannot enter this area. The area is specified by data in memory or a program.

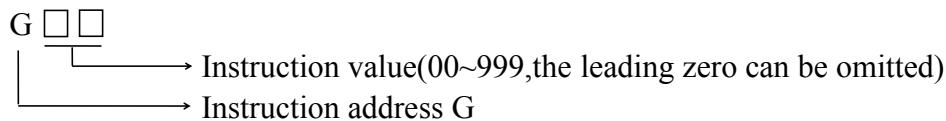
Fig2.6.1 Stroke

Besides strokes defined with limit switches, the operator can define an area which the tool cannot enter using a program or data in memory. This function is called stroke check. Please check Chapter 6.7.1.

Chapter 3 G INSTRCUTIONS

3.1 INTRODUCTION

G instruction consists of instruction address G and its following 1~3 bits instruction value, used for defining the motion mode of tool relative to the workpiece, defining the coordinates and so on. Refer to G instructions as Table 3.



A number of following address G determines the meaning of the command for the concerned block.

G codes are divided into the following two types

Type	Meaning
One-shot G code	The G code is effective only in the block in which it is specified
Modal G code	The G code is effective until another G code of the same group is specified.

Eg.: G01 and G00 are modal G codes.

G01X_;
Z_;
X_;
G00Z_; } **G01 is effective in this range**

3.2 G Code List

1. If CNC enters the clear state ,also when the power is turned on or CNC is reset, the modal G codes change as follows.

1) G codes marked with “ ” in Table 3 are enabled ,which is initial modal codes.

2) When system is cleared due to power-on or reset,which ever specified, either G20 or G21 , remains effective.

2. G codes of group 00 are single-shot G codes.

3. G codes of different groups can be specified in the same block.

If G codes of the same group are specified in the same block,the G code specified last is valid.

4. G codes of different groups can be specified in the same block.

If G codes of the same group are specified in the same block, the G code specified last is valid.

5. If a G code of group 01 is specified in a canned cycle, the canned cycle is canceled in the same way as when a G80 command is specified. G codes of group 01 are not affected by G codes for specifying a canned cycle.

6. G codes are displayed for each group number.

7. When a G code not listed in the G code list is specified or a G code that corresponding function is disabled.

Table 1 Standard G Code List

Word	Ground	Functions	Page
G00	01	Positioning(Rapid Traverse)	
G01		Linear Interpolation(Cutting feed)	
G02		Circular Interpolation CW(clockwise)	
G03		Circular Interpolation CCW(counter clockwise)	
G04	00	Dwell,Exact stop	
G06		3D Space Arc Interpolation	
G11	12	Programmable mirror image	
G12		Programmable mirror image cancel	
G15	11	Polar coordinate interpolation cancel mode	
G16		Polar coordinate interpolation mode	
G17	02	XpYp plane selection	
G18		ZpXp plane selection	
G19		YpZp plane selection	
G20	06	Input in Inch	
G21		Input in mm	
G22	08	Program Block Cycle	
G800		Program Block Cycle Cancel	
G25	00	Save value of current coordinate	
G26		ALL-Axis go starting point	
G261		X-Axis go starting point	
G262		Y-Axis go starting point	
G263		Z-Axis go starting point	
G264		A-Axis go starting point	
G28		Return to reference position	
G281		X-Axis return to reference position	
G282		Y-Axis Return to reference position	
G283		Z-Axis Return to reference position	
G284		A-Axis Return to reference position	
G30		Return to zero point of coordinate	
G301		X-axis Return to zero point of coordinate	
G302		Y-axis Return to zero point of coordinate	
G303		Z-axis Return to zero point of coordinate	
G304		A-axis Return to zero point of coordinate	
G31	12	Skip function (No alarm)	
G311		Skip function (alarm)	
G36	12	Scaling	
G37		Scaling Cancel	
G40	07	Tool nose radius compensation cancel	
G41		Tool nose radius compensation left	
G42		Tool nose radius compensation right	

G43	08	Tool lengthen compensation + direction	
G44		Tool lengthen compensation - direction	
G45	00	Tool offset increase	
G46		Tool offset decrease	
G47		Tool offset double increase	
G48		Tool offset double decrease	
G49		Tool length compensation cancel	
G50		Coordinate system setting or max. spindle speed setting	
G52	00	Local coordinate system setting	
G53		Machine Coordinate System	
G54	14	Workpiece Coordinate System-1 Selection	
G55		Workpiece Coordinate System-2 Selection	
G56		Workpiece Coordinate System-3 Selection	
G57		Workpiece Coordinate System-4 Selection	
G58		Workpiece Coordinate System-5 Selection	
G59		Workpiece Coordinate System-6 Selection	
G61	04	Return G25 coordinate of G25	
G611		Return the coordinate position of X-Axis in G25	
G612		Return the coordinate position of Y-Axis in G25	
G613		Return the coordinate position of Z-Axis in G25	
G614		Return the coordinate position of A-Axis in G25	
G60	04	Exact Stop & Positioning	
G64		Continuous Path Processing	
G65	00	Macro calling	
G66	12	Macro modal call	
G67		Macro modal call cancel	
G68	13	Coordinate rotation	
G69		Coordinate rotation cancel	
G73	09	Pecking Drilling Cycle	
G74		Pattern Repeating Cycle	
G76		Fine Boring Cycle	
G80		Canned cycle cancel/external operation function cancel	
G81		Drilling Cycle, spot boring cycle or external operational function	
G82		Drilling Cycle or counter boring cycle	
G83		Peck drilling cycle	
G84		End Face Peck Drilling	
G85		Finishing Cycle	
G86		Axial Roughing Turning Cycle	
G87		Back Boring Cycle	
G89		Radial Roughing Facing Cycle	
G90	03	Absolute programming	
G91		Incremental programming	
G94	04	Feed per minute	

G95		Feed per revolution	
G96	15	Constant surface speed control	
G97		Constant surface speed control cancel	
G98	10	Return to initial point in canned cycle	
G99		Return to R point in canned cycle	

Table 2 Special G Code List

G110		CCW Inner Round groove roughing	
G111		CW Inner Round groove roughing	
G112		Finishing in CCW full circle	
G113		Finishing in CW full circle	
G114		CCW outer circle finishing	
G115		CW outer circle finishing	
G116		CCW outer circle groove roughing	
G117		CW outer circle groove roughing	
G120	09	Tool changing for linear tool magazine	
G121		Tool changing for umbrella tool magazine(A-axis)	
G122		Tool changing for umbrella tool magazine(A-axis)	
G132		CCW rectangle outer roughing	
G133		CW rectangle outer roughing	
G134		CCW rectangle groove roughing	
G135		CW rectangle groove roughing	
G136		CCW rectangle groove finishing	
G137		CW rectangle groove finishing	
G138		CCW rectangle outer finishing	
G139		CW rectangle outer finishing	

3.3 Positioning (Rapid Traverse) (G00)

G00 command moves a tool to the position in the workpiece system specified with an absolute or an incremental command at a rapid traverse rate.

In the absolute command, coordinate value of the end point is programmed.

In the incremental command the distance the tool moves is programmed.

Format: G00 X(U)_ Z(W)_ Y/C(V)_ A_ ;

Either of the following tool paths can be selected according to P9_D6 (Bit 6 of No.9 parameter) in Other parameter.

Nonlinear interpolation positioning

The tool is positioned with the rapid traverse rate for each axis separately. The tool path is normally straight.

Linear interpolation positioning

The tool path is the same as in linear interpolation (G01). The tool is positioned within the shortest possible time at a speed that is not more than the rapid traverse rate for each axis. However, the tool path is not the same as in linear interpolation (G01).

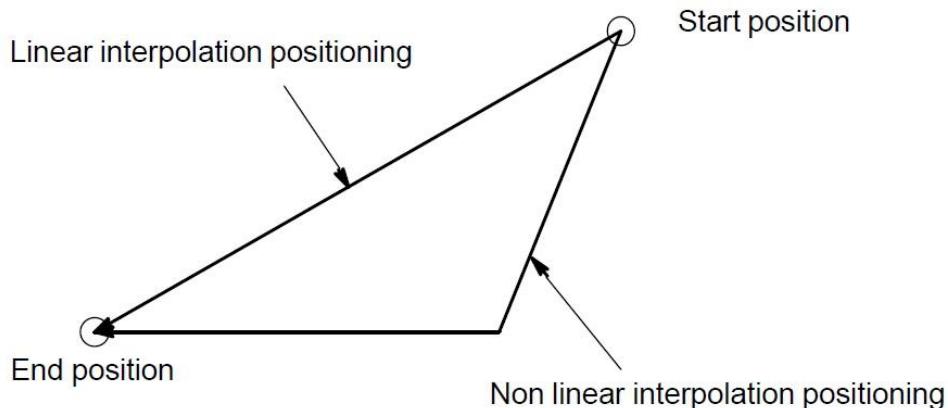


Fig3.3.1 Mode of Tool Path

P1 & P2 & P3 in Speed parameter is set for rapid traverse rate in the G00 command for each axis independently.

The speed rate of G00 can be divided into 5%~100%, total six gears, it can be selected by the key on panel.

G00 is mode instruction, when the next instruction is G00 too, it can be omitted. G00 can be written G0.

In the positioning mode actuated by G00, the tool is accelerated to a predetermined speed at the start of a block and is decelerated at the end of a block. Execution proceeds to the next block after confirming the in-position, which means that the feed motor is within the specified range.

Note: 1. When Rotary Axis positioning in absolute programming, G00 is actuated with nearest path ; when in incremental programming, G00 is actuated with arithmetic path.

2. The rapid traverse rate cannot be specified in the address F.

3. Even if linear interpolation positioning is specified, nonlinear interpolation positioning is used in the following cases. Therefore, be careful to ensure that the tool does not foul the workpiece.

- *G28 specifying positioning between the reference and intermediate positions.*
- *G53*

3.4 Linear Interpolation (G01)

A tools move along a line to the specified position at the feedrate specified in F.

Format: G01 X/U_ Z/W_ Y/V_ A_ F_ ;

X,Z,Y, A means motion axis. For an absolute command, the coordinates of an end point , and for an incremental command, the distance the tool moves.

F: Speed of tool feed(Feedrate)

The feedrate specified in F is effective until a new value is specified. It need not be specified for each block.

The feedrate commanded by the F code is measured along the tool path.

If the F code is not commanded, the feedrate is regarded as zero.

For feed-per-minute mode under 2-axis simultaneous control, the feedrate for a movement along each axis as follows :

G01 $\alpha\alpha\beta\beta\gamma\gamma\zeta\zeta$ F \underline{f} ;

$$\text{Feed rate of } \alpha \text{ axis direction : } F\alpha = \frac{\alpha}{L} \times f$$

$$\text{Feed rate of } \beta \text{ axis direction : } F\beta = \frac{\beta}{L} \times f$$

$$\text{Feed rate of } \gamma \text{ axis direction : } F\gamma = \frac{\gamma}{L} \times f$$

$$\text{Feed rate of } \zeta \text{ axis direction : } F\zeta = \frac{\zeta}{L} \times f$$

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

The feedrate of the rotary axis is command in the unit of deg/min (the unit is decimal point position)

When the straight line axis α (such as X, Y, or Z) and the rotating axis β (such as A, B, or C) are linearly interpolated, the feed rate is that in which the tangential feed rate in the α & β Cartesian coordinate system is commanded by F (mm/min).

β -axis feedrate is obtained; at first, the time required for distribution is calculated by using the above formula, then the β -axis feedrate unit is changed to deg/min.

A calculation example is as follow.

G91 G01 X20.0 A40.0 F300.0

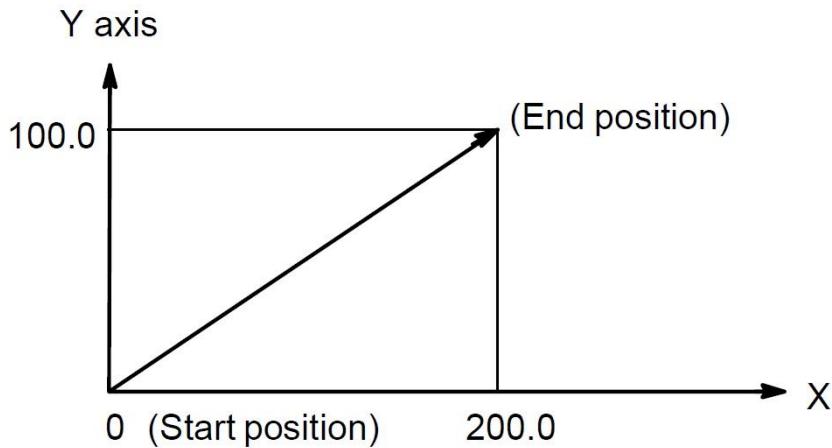
This changes the unit of the A axis from 40.0 deg to 40mm with metric input. The time required for distribution is calculated as follow:

$$\frac{\sqrt{20^2 + 40^2}}{300} \approx 0.14907(\text{min})$$

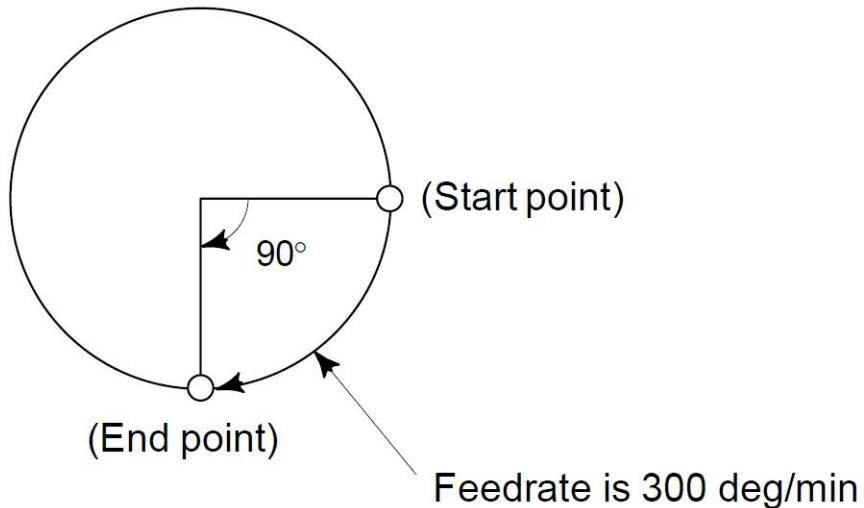
The feedrate for the C axis is $\frac{40}{0.14907} \approx 268.3 \text{ deg/min}$

In simultaneous 3 axes control, the feedrate is calculated the same way as in 2 axes control.

Example1: (G91) G01 X200.0 Y100.0 F200.0 ;



Example2: G91 G01 A-90.0 G300.0 ; Feedrate of 300deg/min



G01 instruction can also specify movement of X-axis/Y-axis/Z-axis separately.

G01 is F feed rate can be motivated by the panel to override adjusted up or down to adjust the range (0% -150%).

G01 instruction can also be directly written G1.

3.5 Circular Interpolation (G02/G03)

These commands will move a tool along a circular arc.

Format: Arc in the XpYp plane

G17 $\left\{ \begin{array}{l} \mathbf{G02} \\ \mathbf{G03} \end{array} \right\}$ **Xp_Yp_** $\left\{ \begin{array}{l} \mathbf{I_J_} \\ \mathbf{R_} \end{array} \right\}$ **F_** ;

Arc in the ZpXp plane

G18 $\left\{ \begin{array}{l} \mathbf{G02} \\ \mathbf{G03} \end{array} \right\}$ **Xp_Zp_** $\left\{ \begin{array}{l} \mathbf{I_K_} \\ \mathbf{R_} \end{array} \right\}$ **F_**

Arc in the YpZp plane

G19 $\left\{ \begin{array}{l} \mathbf{G02} \\ \mathbf{G03} \end{array} \right\}$ **Yp_Zp_** $\left\{ \begin{array}{l} \mathbf{J_K_} \\ \mathbf{R_} \end{array} \right\}$ **F_**

Code	Description
G17	Specification of arc on XpYp plane
G18	Specification of arc on ZpXp plane(Default)
G19	Specification of arc on YpZp plane
G02	Circular Interpolation Clockwise direction (CW)
G03	Circular Interpolation Counterclockwise direction (CCW)
X/Z/Y	Position of end point in workpiece coordinate
U/W/V	Distance from start point to end point
I	X axis distance from start point to center of an arc with sign(radius value)
K	Z axis distance from start point to center of an arc with sign(radius value)
J	Y axis distance from start point to center of an arc with sign(radius value)
R	Arc radius with sign (always with radius value)
F	Feedrate along the arc

Direction of circular interpolation: “Clockwise”(G02) and “Counter clockwise”(G03) on the XpYp plane(ZpXp plane or YpZp plane) are defined when the XpYp plane is viewed in the positive to negative direction of the Zp axis (Yp axis or Xp axis respectively) in the Cartesian coordinate system, See the figure below.

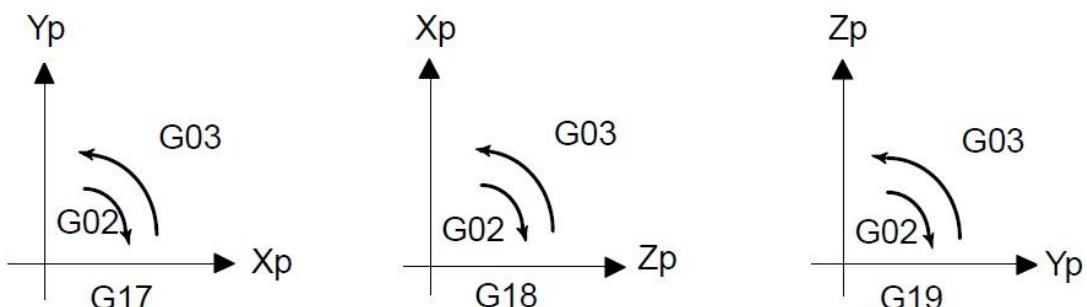


Fig3.5.1 Direction of G02&G03 at different plane

Distance moved on an arc: The end point of an arc is specified by address X_p , Y_p or Z_p , and is expressed as an absolute or incremental value according to G90 or G91. For the incremental value, the distance of the end point which is viewed from the start point of the arc is specified.

Distance from the start point to the center of arc: The arc center is specified by addresses I , J and K for the X_p , Y_p and Z_p axes, respectively. The numerical value following I , J or K , however, is a vector component in which the arc center is seen from the start point, and is always specified as an incremental value irrespective of G90 and G91, as shown below.

I , and K must be signed according to the direction.

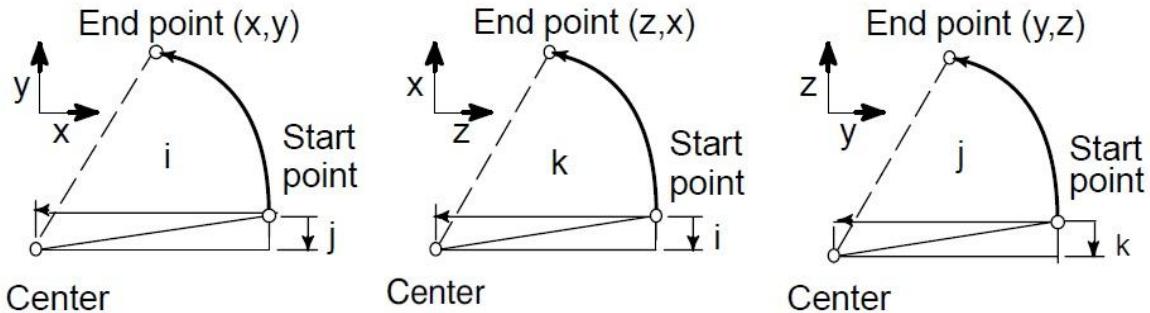


Fig3.5.2 Define & Direction of I, J & K

I_0 , J_0 and K_0 can be omitted.

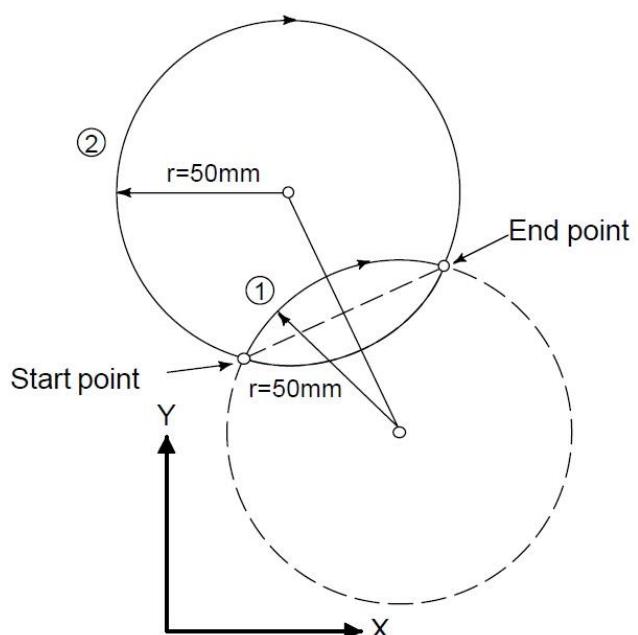
If the distance is from the end point to the center of arc , which exceeds by the value in a parameter of P41 in Speed parameter(Original value+4).

Full-circle programming: When X_p , Y_p and Z_p are omitted (the end point is the same as the start point) and the center is specified with I , J and K , a 360° arc (circle) is specified.

Arc radius: The distance between an arc and the center of a circle that contains the arc can be specified using the radius, R , of the circle instead of I , J and K . In this case, one arc is less than 180° , and the other is more than 180° are considered. When an arc exceeds 180° is commanded, the radius must be specified with a negative value. If X_p , Y_p and Z_p are all omitted, if the end point is located at the same position as the start point and when R is used, an arc of 0° is programmed.

G02R ; (The cutter does not move.)

For arc (1) [Less than 180°] G91 G02 X _p 60.0 Y _p 20.0 R50.0 F300.0;
For arc (2) [Greater than 180°] G91 G02 X _p 60.0 Y _p 20.0 R-50.0 F300.0;



Feedrate: The feedrate in circular interpolation is equal to the feed rate specified by the F code, and the feedrate along the arc(the tangential feedrate of the arc) is controlled to be the specified feedrate.

The error between the specified feedrate and the actual tool feedrate is $\pm 2\%$ or less. However, this feed rate is measured along the arc after the cutter compensation is applied.

Note: 1. When I = 0 , J = 0 or K = 0, they can be omitted; one of I, K or R must be input, otherwise the system alarms.

2. If I,J,K, and R addresses are specified simultaneously, the arc specified by address R takes precedence and the other are ignored.

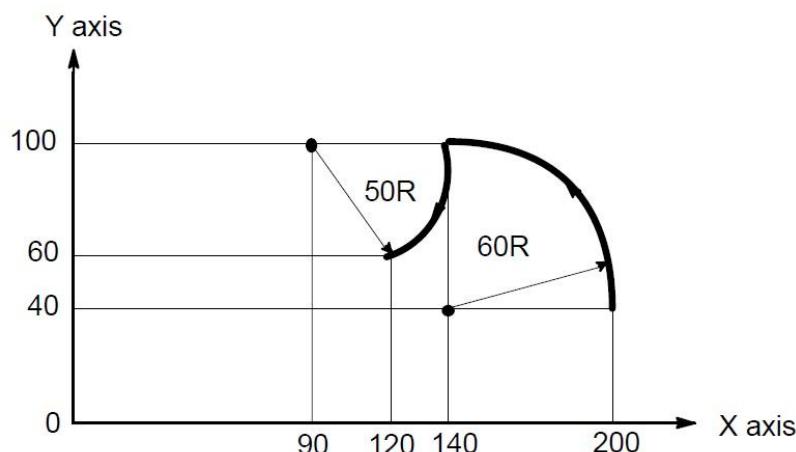
3. Processing arc workpiece usually use ball tool(arc tool) in the actual process, it must use function of tool radius compensation in programming, that's G41 G42 instruction.

4. Arc path can be more than and less than 180° when R is commanded, the arc is more than 180° when R is negative, and it is less than or equal to 180° when R is positive.

5. If an axis not comprising the specified plane is commanded, an alarm is displayed.

6. When an arc having a center angle approaching 180° is specified, the calculated center coordinates may contain an error. In such a case, specify the center of the arc with I,J, and K.

Example:



The above tool path can be programmed as follows:

(1) In absolute programming

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

(2) In incremental programming

```
G92 G03 X-60.0 Y60.0 R60.0 F300. ;
G02 X-20.0 Y-40.0 R50.0
```

Or

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

3.6 Helical Interpolation (G02/G03)

Helical interpolation which moves helical is enabled by specifying up to two other axes which move synchronously with the circular interpolation by circular commands.

Format:

Synchronously with arc of XpYp plane

G17 $\left\{ \begin{array}{l} \text{G02} \\ \text{G03} \end{array} \right\}$ $X_p _ Y_p _ \left\{ \begin{array}{l} I _ J _ \\ R _ \end{array} \right\} \alpha _ (\beta _) F _ ;$

Synchronously with arc of ZpXp plane

G18 $\left\{ \begin{array}{l} \text{G02} \\ \text{G03} \end{array} \right\}$ $X_p _ Z_p _ \left\{ \begin{array}{l} I _ K _ \\ R _ \end{array} \right\} \alpha _ (\beta _) F _ ;$

Synchronously with arc of YpZp plane

G19 $\left\{ \begin{array}{l} \text{G02} \\ \text{G03} \end{array} \right\}$ $Y_p _ Z_p _ \left\{ \begin{array}{l} J _ K _ \\ R _ \end{array} \right\} \alpha _ (\beta _) F _ ;$

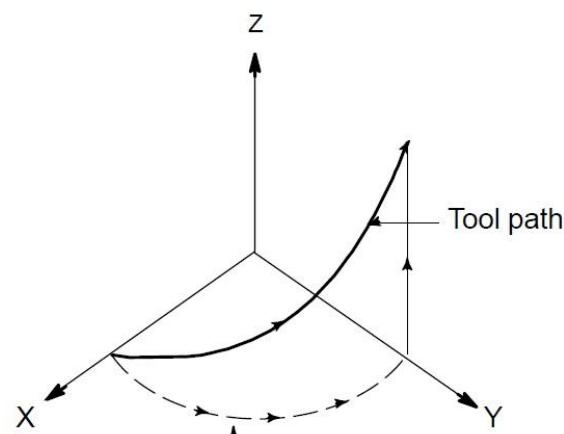
α, β : Any one axis where circular interpolation is not applied.

Up to two other axes can be specified.

The command method is to simply or secondary add a more command axis which is not circular interpolation axes. An F command specifies a feed rate along a circular arc. Therefore, the feed rate of the linear axis is as follows:

$$F * \frac{\text{Length of Linear Axis}}{\text{Length of Circular Arc}}$$

Determine the feed rate so the linear axis feed rate does not exceed any of the various limit values. Cutter compensation is applied only for a circular arc. Tool offset and tool length compensation cannot be used in a block in which a helical interpolation is commanded.



The feedrate along the circumference of two circular interpolated axes is the specified feedrate.

3.7 3D Space Arc Interpolation G06

When user don't know position of circle center & radius, but know coordinate position of 3 points on arc, now user can use G06 code to processing arc, and direction is decided by middle point between starting point & end point.

Format: G06 X_ Y_ Z_ I_ J_ K_ F_

G06: Modal command

I: Increment Coordinate Value of Middle point relative to starting point in X direction
Radius Designation, with direction;

J: Increment Coordinate Value of Middle point relative to starting point in Y direction
With direction

K: Increment Coordinate Value of Middle point relative to starting point in Z direction
With direction

F: Cutting speed

Note:

1. *Middle point is any position point except starting point & end point.*
2. *System will alarm when three points are at one line.*
3. *When I,J,K are omitted, default value is I=0, J=0, K=0. But they cannot be omitted all at same time, otherwise system will alarm.*
4. *The meanings of I,J,K are similar to I,J,K of G02/G03.*
5. *G06 cannot be used for processing total round.*
6. *Compute of G06 command is very large, it only can work smoothly on modbus system, at normal system, it would work not smoothly.*

Example:

G0 X10 Y28 Z10

G06 X30 Y98 Z10 I5 J-6 K-5 F100

 X130 Y198 Z120 I55 J-86 K-65

G0 X0 Z0

M02

3.8 Dwell (G04)

By specifying a dwell, the execution of the next block is delayed by the specified time.

Format: G04 P_ ; or G04 X_ ; or G04 U_ ;

P_ : Specify a time (decimal point not permitted) , unit: ms (millisecond)

X_ : Specify a time (decimal point permitted) , unit: s (second)

U_ : Specify a time (decimal point permitted) , unit: s (second)

Example 1:

G04 X1; delay 1s.

G04 P1000; delay 1s.

G04 U1; delay 1s.

Special application: G04 can be as an accurate stop instruction, such as processing corner kinds of workpiece, it appears over cutting sometimes, if use G04 instruction around the corner, it will clear the over cutting.

Example 2:

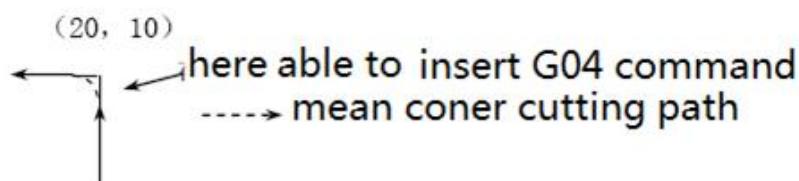


Fig3.22.1 Application of Dwell(G04)

Program:

N150 G01 X20 Z10 F100;

N160 G04 P150; (Clear the over cutting)

N170 G01 W-10;

.....

Note: Set P21 in User parameter, also by setting intervals between blocks of G01/G02/G03 to clear the over cutting.

3.9 Skip Function(G31,G311)

Linear interpolation can be commanded by specifying axial move following the G31 command, like G01. If an external skip signal is input during the execution of this command, execution of the 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.

For details of how to use this function, refer to the manual supplied by the machine tool builder.

Format: **G31 IP_ P_ ;**
G311 IP_ P_ ;

G31&G311 are One-shot G code.(It is effective only in the block in which it is specified).

IP_ : coordinate value ;

P_ : Specify jumping line number & detecting if input point is valid;

P(a)(b)(c)

a: Jumping line number specified by “ N**” ;when missed, stop running current line, and jump to next block and run;

b: 10 or 20 ; 10 means that when input point is valid, skip to specified line, when input point is invalid, don't skip, keep on running or alarm hint; 20 means that when input point is invalid, skip to specified line ; when input point is valid, don't skip , keep on running or alarm hint;

c: Specify detecting input point. address of input point, X00~X39

P 56 10 24 } when Input point X24 is valid, stop running current
a b c } line, jump to N56 block and run.

Difference between G31 & G311: When system don't detect signal of specified input point, G31 don't hint alarm & keep on running program; G311 will hint that don't input is valid & stop running, after Press “Enter”, it will go on run program ;

The coordinate values when the skip signal is turned on can be used in a custom macro because they are stored in the custom macro system variable as follows:

#5021 X axis coordinate value

#5022 Third axis (Cs axis) coordinate value

#5023 Z axis coordinate value

#5024 4th axis (A axis) coordinate value

Example: G31 X50 Z100 F100 P331022 ;if X22 is valid then go to N33(line no.).

G311 X50 Z100 F100 P2021 ;if X21 is invalid then go to next line.Valid-Alarm.

3.10 Reference Position

A CNC machine tool has a special position where, generally, the tool is exchanged or the coordinate system is set, as described later. This position is referred to as a reference position.

Reference Position Return(G28/G281/G282/G283/G284/G301/G302/G303/G304): The reference position is a fixed position on a machine tool to which the tool can easily be moved by the reference position return function. If there is machine zero point(hardware switch), it is also reference point; when user set float zero point(software switch) as home and system will take float zero point as reference point.

Tools are automatically moved to the reference position via an intermediate position along a specified axis. Or, tools are automatically moved from the reference position to a specified position via an intermediate position along a specified axis.

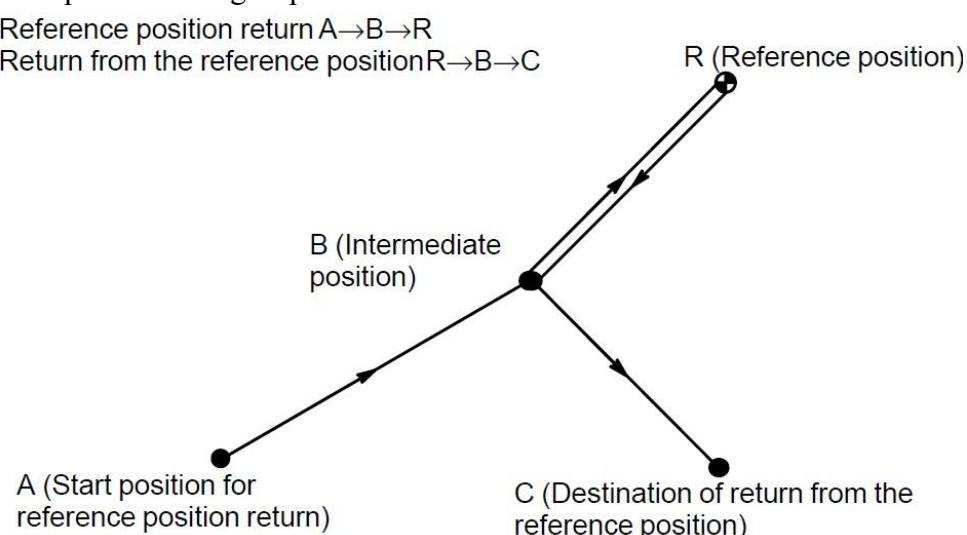


Fig3.9.1 Reference Position Return & Return from Reference Position

Format: G28 X(U)_Y(V)_Z(W)_A_ ; Reference Position Return

X(U)_Y(V)_Z(W)_A_ are intermediate position(Absolute/incremental command).

G281 ; Only X-axis return to reference position
G282 ; Only Y-axis return to reference position,
G283 ; Only Z-axis return to reference position,
G284 ; Only A-axis return to reference position,
G301 ; Only X-axis return to coordinate zero point,
G302 ; Only Y-axis return to coordinate zero point,
G303 ; Only Z-axis return to coordinate zero point,
G304 ; Only A-axis return to coordinate zero point,

Positioning to the intermediate or reference positions are performed at the rapid traverse rate of each axis. Therefore, for safety, the cutter compensation, and tool length compensation should be canceled before executing this command.

Note: 1. When use G28 code to return to reference position, if just specify intermediate position of some axes, which can return to reference position, others axes that don't be specified cannot return to reference position.

2. Before run these codes, reference position must be set well.

3. After return to reference position, system will cancel tool compensation automatically.

3.11 Return to Starting Point (G26,G261~G264)

These instructions are used for return back to the starting point of the program. Starting point is coordinate position of N0000 block. The returning speed is same to G00 speed.

Format:

G26 ; All Feeding axes return to starting point.
G261 ; X-axis return to starting point
G262 ; Y-axis return to starting point
G263 ; Z-axis return to starting point
G264 ; A-axis return to starting point

3.12 Save Current Position (G25)

G25 is used for memory current coordinate position of all axes(XZYA), save current position as specified point.

Format: G25 ; Save current coordinate

3.13 Return to Specified Position (G61,G611~G614)

These instructions are used for return to point specified by G25.

G61 ; all axes return to specified point ;
G611 ; X-axis returns to specified point ;
G612 ; Y-axis returns to specified point ;
G613 ; Z-axis returns to specified point ;
G614 ; A-axis returns to specified point ;

Note: If user don't use G25 to save current position, these instructions will return to starting point as G26.

Example: N0000 G0 X20 Z80 ;
N0001 G01 U5 W-16 F200 ;
N0002 W-100 ;
N0003 G00 U10 ;
N0004 Z80 ;
N0005 G25 ; save current position (X35,Z80)
N0006 G01 U10 W-30 ;
N0007 G0 X100 Z200 ;
N0008 G61 ; return to (X35,Z80)
N0009 M2 ; End of program

3.14 Coordinate System

By teaching the CNC a desired tool position, the tool can be moved to the position. Such a tool position is represented by coordinate in a coordinate system. Coordinates are specified using program axes.

When three programming axes, X-axis, Y-axis and Z-axis are used, coordinates are specified as follows: **X_Y_Z_**

This command is referred to as a dimension word.

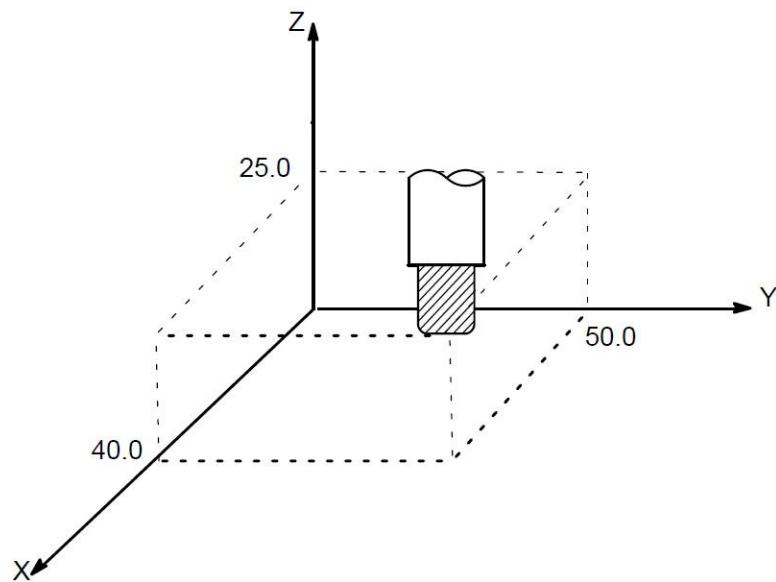


Fig3.10.1 Tool Position specified by X40.0 Y50.0 Z25.0

Coordinates are specified in one of following three coordinate systems:

- (1) Machine coordinate system
- (2) Workpiece coordinate system
- (3) Local coordinate system

The number of the axes of a coordinate system varies from one machine to another. So, in this manual, a dimension word is represented as IP_.

3.14.1 Machine Coordinate System (G53)

The point that is specific to a machine and serves as the reference of the machine is referred to as the machine zero point. A machine tool builder sets a machine zero point for each machine.

A coordinate system with a machine zero point set as its origin is referred to as a machine coordinate system.

A machine coordinate system is set by performing manual reference position return after power-on(See Chapter 5.5.4). A machine coordinate system, once set, remains unchanged.

Format: G53 IP_;

When a position has been specified as a set of machine coordinates, the tool moves to that position by means of rapid traverse. G53, used for selecting the machine coordinate system, is a one-shot G code. Any commands based on the selected machine coordinate system are thus effective only in the block containing G53. When the tool is to be moved to a machine-specific position such as a tool change position, program the movement in a machine coordinate system based on G53.

When manual reference position return is performed after power-on, a machine coordinate

system is set so that the reference position is at the coordinate values of (α, β) set by P46, P47, P48 & P49 in Axis parameter .

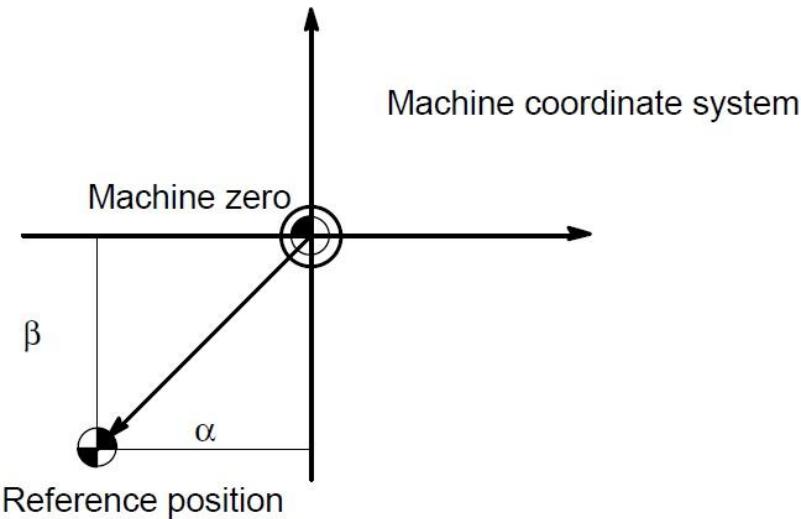


Fig3.10.2 Machine coordinate system

3.14.2 Workpiece Coordinate system(G54/G55/G56/G57/G58/G59)

A coordinate system used for machining a workpiece is referred to as a workpiece coordinate system. A workpiece coordinate system is to be set with the CNC beforehand (**setting a workpiece coordinate system**).

A machining program sets a workpiece coordinate system (**selecting a workpiece coordinate system**).

A set workpiece coordinate system can be changed by shifting its origin (**changing a workpiece coordinate system**).

A workpiece coordinate system can be set using one of three methods:

(1)Method using G50

A workpiece coordinate system is set by specifying a value after G50 in the program.

(2) Automatic setting

A workpiece coordinate system is automatically set when manual reference position return is performed .

(3) Method of using G54 to G59

Make settings on the MDI panel to preset six workpiece coordinate systems. Then, use program commands G54 to G59 to select which workpiece coordinate system to use.

When an absolute command is used, a workpiece coordinate system must be established in any of the ways described above.

The user can choose from set workpiece coordinate systems as described below.

Setting a workpiece coordinate system by G50

Format G50 IP_

A workpiece coordinate system is shifted to set a new workpiece coordinate system so that a point on the tool, such as the tool tip, is at specified coordinates(IP_). Then, the amount of coordinate system shift is added to all the workpiece zero point offset values. This means that if a coordinates system is set using G50 during tool length offset, a coordinate system in which the position before offset matches the position specified in G50 is set.

Tool compensation is canceled temporarily with G50.

Warning: When a coordinate system is set with G50 after an external workpiece zero point offset value is set, the coordinate system is not affected by the external workpiece zero point offset value. When G50 X100Z80; is specified, for example, the coordinate system having its current tool reference position at X=100 & Z=80 is set.

Example:

Example1: Setting the coordinate system by the G92 X25.2 Z23.0; command

(The tool tip is the start point for the program)

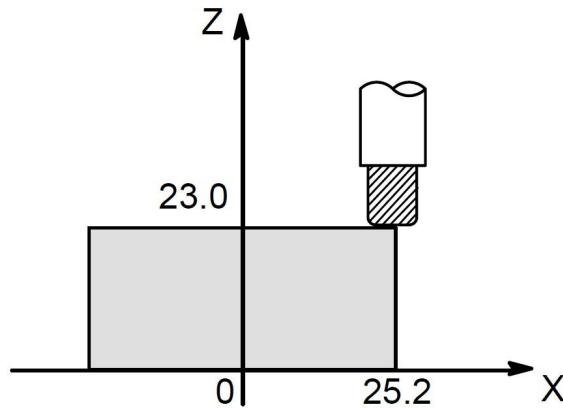


Fig3.10.3 Example1 with G50

Example2: Setting the coordinate system by the G50 X600.0 Z1200.0 ; command

(The base point on the tool holder is the start point for the program)

(If an absolute command is issued, the base point moves to the command position. In order to move the tool tip to the commanded position, the difference from the tool tip to the base point is compensated by tool length offset.)

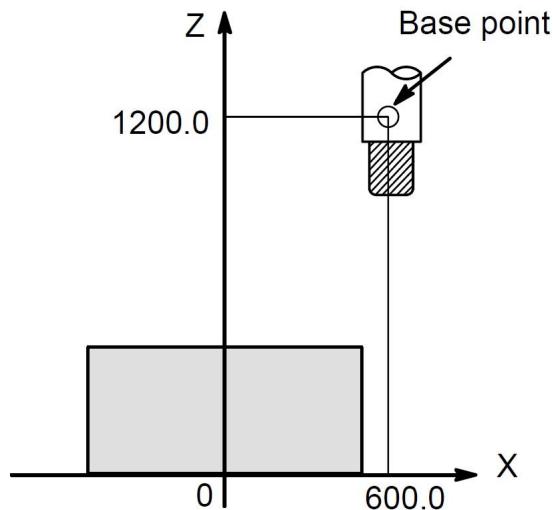


Fig3.10.4 Example2 with G50

Choosing from six workpiece coordinate systems set in the MDI: By specifying a G code from G54 to G59, one of the workpiece coordinate systems 1 to 6 can be selected.

G54 Workpiece coordinate system 1

G55 Workpiece coordinate system 2

G56 Workpiece coordinate system 3

G57 Workpiece coordinate system 4

G58 Workpiece coordinate system 5

G59 Workpiece coordinate system 6

Workpiece coordinate system 1 to 6 are established after reference position return after the power is turned on. When the power is turned on, G54 workpiece coordinate system is selected.

G90 G55 G00 X40.0 Y100.0 ;

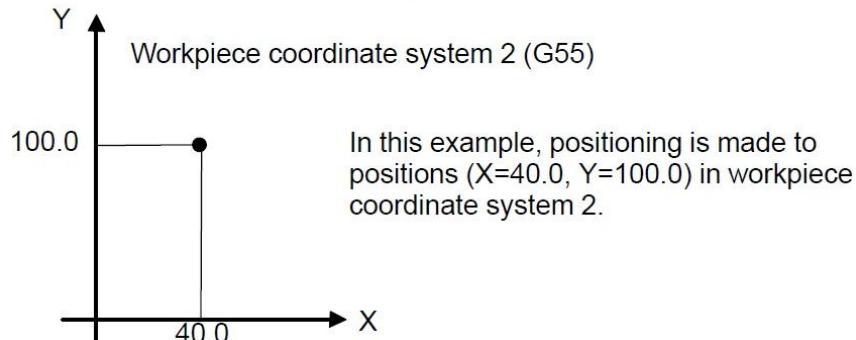


Fig3.10.5 Workpiece Coordinate system

Changing Workpiece Coordinate System: The six workpiece coordinate systems specified with G54 to G59 can be changed by changing an external workpiece zero point offset value or workpiece zero point offset value.

Three methods are available to change an external workpiece zero point offset value or workpiece zero point offset value.

- (1) Inputting from the MDI panel
- (2) Programming by G50
- (3) Using the external data input function

An external workpiece origin offset can be changed by using a signal input to the CNC, also alter coordinate system in Coordinate parameter.

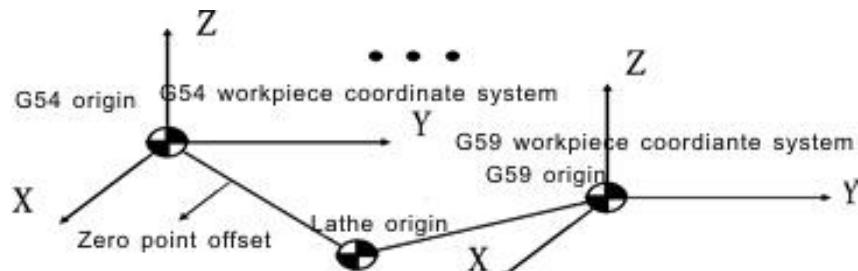


Fig3.10.6 Changing workpiece coordinate system

Example: G00 G54 X50 Y60 Z70

Move to X50 Y60 Z70 of G54 with speed of G00.

3.14.3 Local Coordinate System (G52)

When a program is created in a workpiece coordinate system, a child workpiece coordinate system may be set for easier programming. Such a child coordinate system is referred to as a local coordinate system.

Format: G52 IP_ ; Setting the local coordinate system

G52 IP 0 ; Canceling of the local coordinate system

IP_ : Origin of the local coordinate system

By specifying G52IP_ ;, a local coordinate system can be set in all the workpiece coordinate systems (G54 to G59). The origin of each local coordinate system is set at the position specified by IP_ in the workpiece coordinate system.

Once a local coordinate system is established, the coordinates in the local coordinate system are used in an axis shift command. The local coordinate system can be changed by specifying the G52 command with the zero point of a new local coordinate system in the workpiece coordinate system.

To cancel the local coordinate system and specify the coordinate value in the workpiece coordinate system, match the zero point of the local coordinate system with that of the workpiece coordinate system.

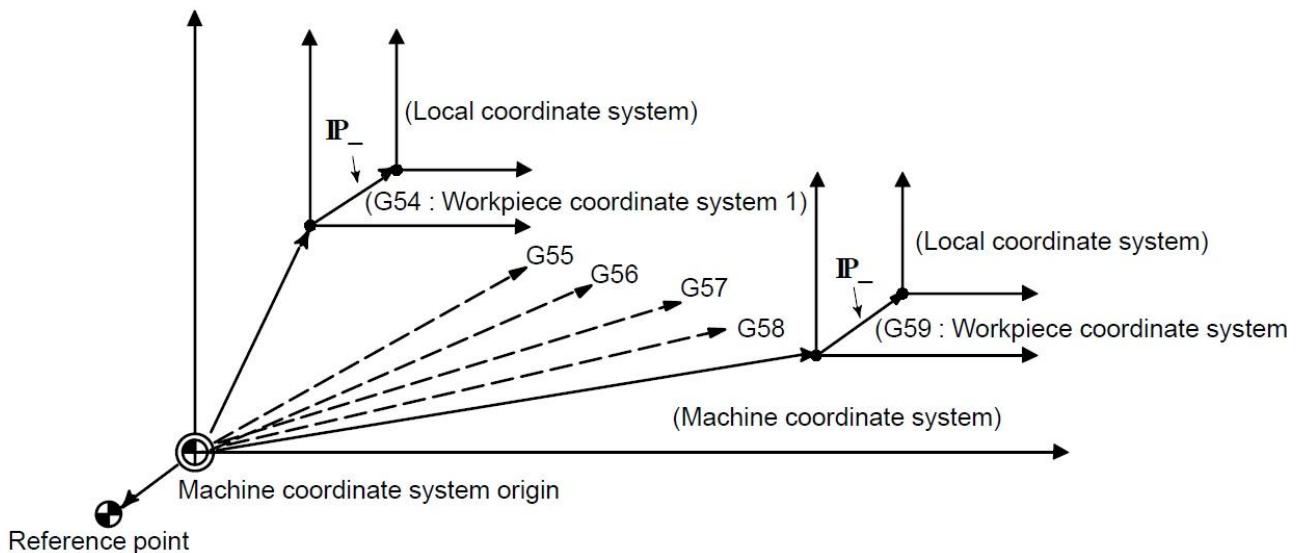


Fig3.10.7 Setting the Local Coordinate System

Note: 1. The local coordinate system setting does not change the workpiece and machine coordinate systems.

2. When G50 is used to define a work coordinate system, if coordinates are not specified for all axes of a local coordinate system, the local coordinate system remains unchanged. If coordinates are specified for any axis of a local coordinate system, the local coordinate system is canceled.

3. G52 cancels the offset temporarily in tool nose radius compensation.

4. Command a move command immediately after the G52 block in the absolute mode.

Example:

G52 X100 Y60; set local coordinate system on G54;

If tool position X100,Y100 is commanded at local coordinate system when tool is positioned at (200,160) in G54 mode, Local coordinate system (X' -Y') shift by vector A is created

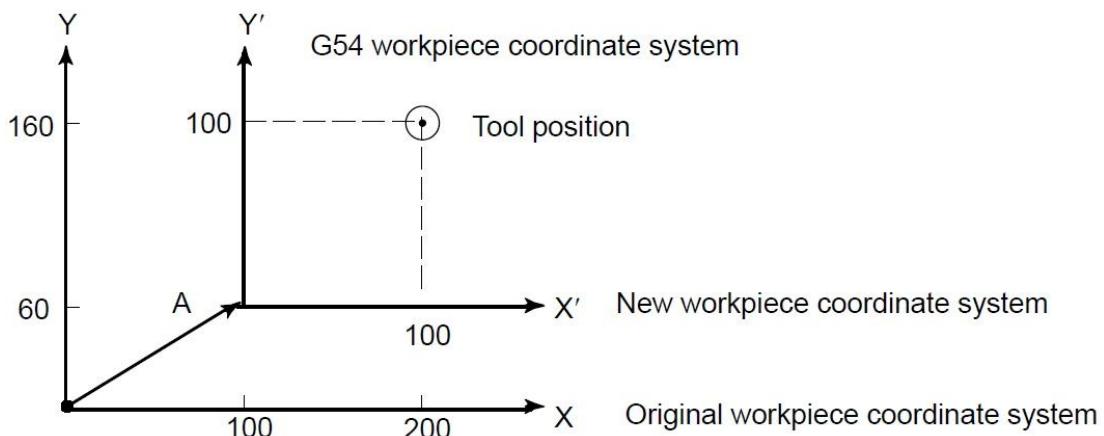


Fig3.10.6 Example of Local Coordinate System

3.15 Plane Selection(G17/G18/G19)

Select the planes for circular interpolation, cutter compensation, and drilling by G-code.

Format:	G17 (Mode,Original)	;XY Plane selection
	G18 (Mode)	;ZX Plane selection
	G19 (Mode)	;YZ Plane selection

The following table lists G-codes and the planes selected by them.

G code	Selected Plane	Xp	Yp	Zp
G17	Xp Yp Plane	X-axis or an axis parallel to it	Y-axis or an axis parallel to it	Z-axis or an axis parallel to it
G18	Zp Xp Plane			
G19	Yp Zp plane			

Used to ensure circular interpolation plane.this instruction does not produce motion.

3.16 Absolute and Incremental Programming (G90/G91)

There are two ways to command travels of the tool: absolute command and incremental command. In the absolute command, coordinate value of the end position is programmed; in the incremental command, move distance of the position itself is programmed. G90 and G91 are used to command absolute or incremental command respectively.

Format: G90 (Mode,initial) ;Absolute command

G91 (Mode) ;Incremental command

Note:Rotating axis programming,calculation is with nearest of in absolute coordinate system, calculation is with programming in incremental coordinate system.

Example:

G90 X40.0 Y70.0 ; Absolute command

G91 X-60.0 Y40.0 ; Incremental command

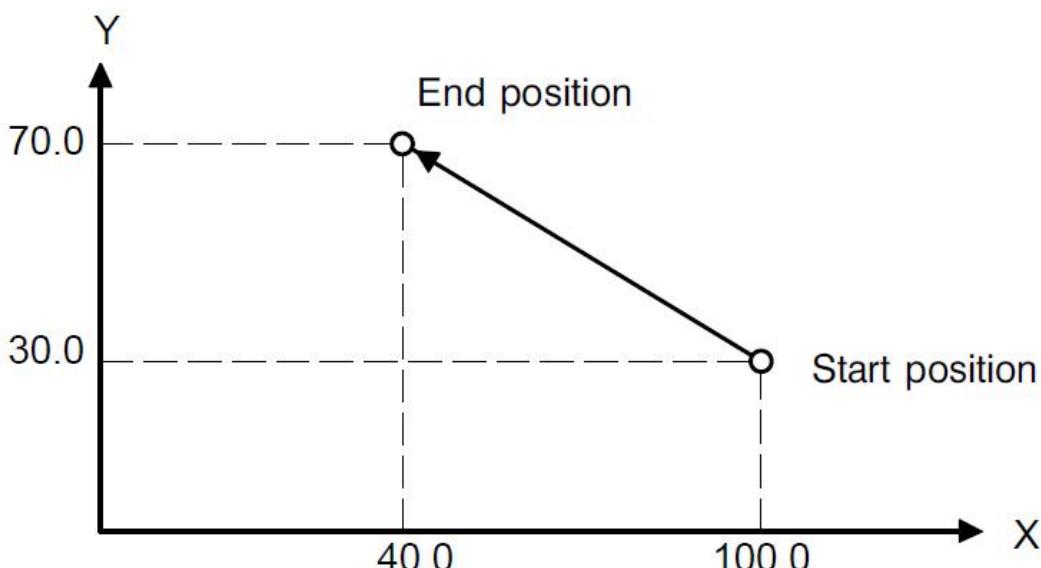


Fig3.11.1 Example of Absolute/Incremental Programming

3.17 Pole coordinate instruction(G15/G16)

The end point coordinate value can be input in polar coordinate (Radius & Angle).

The plus direction of the angle is counterclockwise of the selected plane first axis + direction , and the minus direction is clockwise.

Both radius & angle can be commanded in either absolute or incremental command(G90,G91).

Format: G□□ G○○ G16; Starting the polar coordinate command(Polar Mode)

G◇◇ IP_;
...
] Polar coordinate command

G15 Cancel Pole coordinate command;

G16 ; Polar Coordinate command

G15 ; Polar coordinate command cancel

G□□ ; Plane selection of the polar coordinate command(G17,G18 or G19)

G○○ ; G90 specifies the zero point of the work coordinate system as the origin of the polar coordinate system, from which a radius is measured. G91 specifies the current position as the origin of the polar coordinate system,from which a radius is measured.

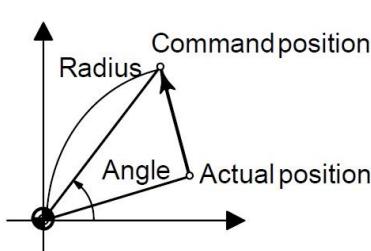
IP_ ; Specifying the addresses of axes constituting the plane selected for the polar coordinate system, and their values.

First axis: Radius of pole coordinate.

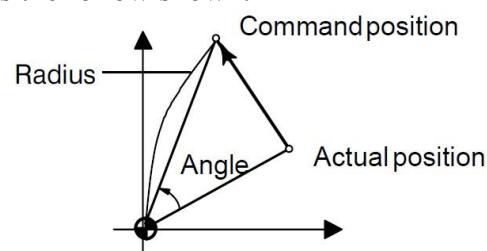
Second axis: Angle of pole coordinate.

Setting the zero point of workpiece coordinate system as origin of pole coordinate system:

Specify the radius(the distance between the zero point and the point) to be programmed with an absolute command. The zero point of the work coordinate system is set as the origin of the polar coordinate system. When a local coordinate system(G52) is used, the origin of local coordinate system becomes the center of polar coordinate. As the follow shown:

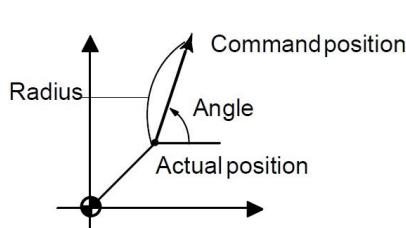


When the angle is specified with an absolute command

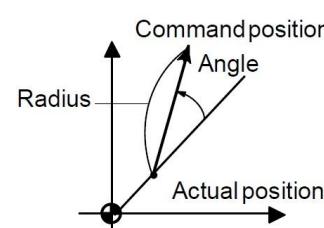


When the angle is specified with an incremental command

Setting the current position as the origin of polar coordinate system: Specify the radius (the distance between the current position and the point) to be programmed with an incremental command. The current position is set as the origin of the polar coordinate system.

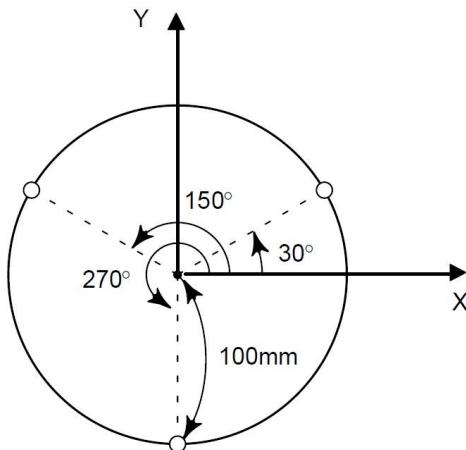


When the angle is specified with an absolute command



When the angle is specified with an incremental command

Example: Bolt hole circle



- The zero point of the work coordinate system is set as the origin of the polar coordinate system.
- The XY plane is selected.

Fig3.13.1 Example of Polar Coordinate

(a) Specifying angles and a radius with absolute commands.

N1 G17 G90 G16 X0 Y0;

Specifying the polar coordinate command and selecting the XY-plane setting the zero point of the workpiece coordinate system as the origin of the polar coordinate system.

N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0;

Specifying a distance of 100mm and an angle of 30 degrees

N3 Y150.0;

Specifying a distance of 100mm and an angle of 150 degrees

N4 Y270.0;

Specifying a distance of 100mm and an angle of 270 degrees

N5 G15 G80;

Canceling the polar coordinate command & drilling cycle.

(b) Specifying angles with incremental commands and a radius with absolute commands

N1 G17 G90 G16;

Specifying the polar coordinate command and selecting the XY plane. Setting the zero point of the workpiece coordinate system as the origin of the polar coordinate system.

N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0

Specifying a distance of 100mm and an angle of 30 degrees

N3 G91 Y120.0;

Specifying a distance of 100mm and an angle of +120 degrees

N4 Y120.0;

Specifying a distance of 100mm and an angle of +120 degrees

N5 G15 G80;

Canceling the polar coordinate command.

(c) Specifying a radius in the polar coordinate mode

In the polar coordinate mode, specify a radius for circular interpolation or helical cutting (G02, G03) with R.

Neither optional-angle chamfer nor corner rounding can be commanded in polar coordinate mode.

3.18 Inch/Metric Conversion (G20/G21)

Either inch or metric input can be selected by G code.

Format: **G20 ; Inch input**
G21 ; mm input

This G code must be specified in an independent block before setting the coordinate system at the beginning of the program. After the G code for inch/metric conversion is specified, the unit of input data is switched to the least inch or metric input increment of increment system IS - B or IS - C. The unit of data input for degrees remains unchanged.

The unit systems for the following values are changed after inch/metric conversion:

- Feedrate commanded by F code
- Positional command
- Work zero point offset value
- Tool compensation value
- Unit of scale for manual pulse generator
- Movement distance in incremental feed
- Some parameters

When the power is turned on, the G code is the same as that held before the power was turned off.

Note: 1. *G20 and G21 must not be switched during a program.*

2. *Movement from the intermediate point is the same as that for manual reference position return. The direction in which the tool moves from the intermediate point is the same as the reference position return direction.*

3. *When the least input increment and the least command increment systems are different, the maximum error is half of the least command increment. This error is not accumulated.*

4. *The inch and metric input can also be switched using setting of data setting.*

3.19 Feed Mode

Feedrate of linear interpolation(G01),circular interpolation(G02/G03),etc. are commanded with numbers after the F code.In cutting feed, the next block is executed so that the feedrate change from the previous block is minimized.

Format: Feed per minute: **G94 ; G code (group 5) for feed per minute**
F_ ; Feedrate command (mm/min or inch/min)

Feed per revolution **G95 ; G code (group 5) for feed per revolution**
F_ ; Feedrate command (mm/rev or inch/rev)

Tangential speed constant control: Cutting feed is controlled so that the tangential feedrate is always set at a specified feedrate.

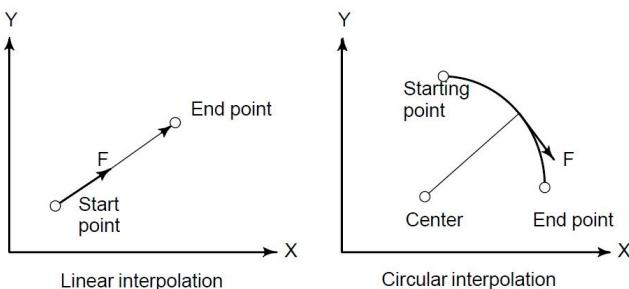


Fig3.15.1 Tangential feedrate

Feed per minute (G94): After specifying G94 (in the feed per minute mode), the amount of feed of the tool per minute is to be directly specified by setting a number after F. G94 is a modal code. Once a G94 is specified, it is valid until G95 (feed per revolution) is specified. At power-on, the feed per minute mode is set.

An override from 0% to 150% can be applied to feed per minute with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

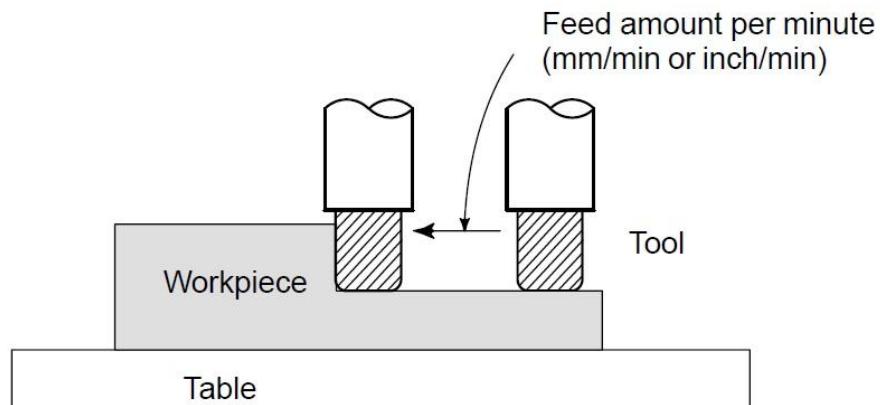


Fig3.15.2 Feed per minute

Warning: No override can be used for some commands such as for threading.

Feed per revolution (G95): After specifying G95 (in the feed per revolution mode), the amount of feed of the tool per spindle revolution is to be directly specified by setting a number after F. G95 is a modal code. Once a G95 is specified, it is valid until G94 (feed per minute) is specified.

An override from 0% to 150% can be applied to feed per minute with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

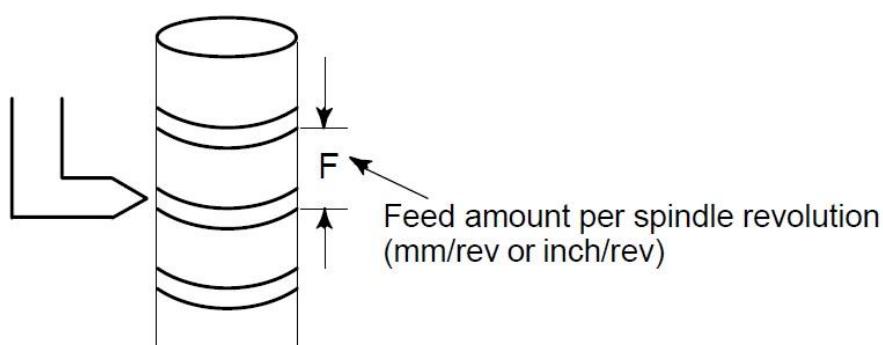


Fig3.15.3 Feed per revolution

Note: When the speed of spindle is low, feedrate fluctuation may occur. The slower the spindle rotates, the more frequently feedrate fluctuation occurs.

3.20 Processing Mode (G60/G64)

According to process requirements, user can specify the connection way between program block by G60/G64.

Format: **G60** ; Accurate Positioning Processing
G64 ; Continuous Path Processing

Both G60 and G64 are modal instructions.

3.21 Scaling (G36/G37)

A programmed figure can be magnified or reduced.(scaling)

The dimensions specified with X_{_}, Y_{_}, and Z_{_} can each be scaled up or down with the same or different rates of magnification. The magnification rate can be specified in the program.

Unless specified in the program, the magnification rate specified in the parameter is applied.

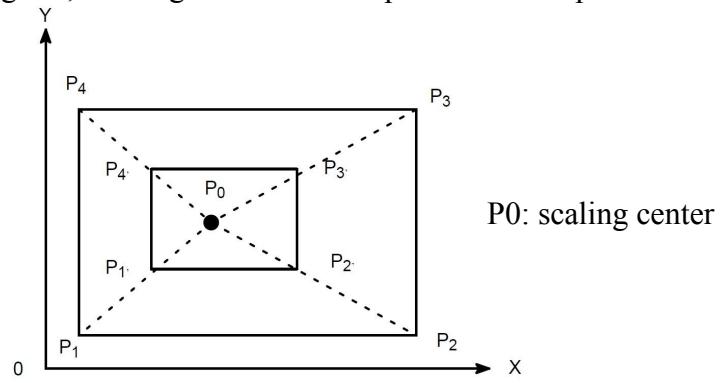


Fig3.15.1 Scaling(P₁P₂P₃P₄ -> P₁'P₂'P₃'P₄')

Format: G36 X_{_}Y_{_}Z_{_}R_{_} ;Scaling start
 ... } Scaling is effective(scaling mode)
 ... }
G37 ;Cancel

R_{_}: scaling magnification.

P₁' P₂' P₃' P₄' magnify to P₁ P₂ P₃ P₄, R=P₀P₄'/P₀P₄. When P₁ P₂ P₃ P₄ reduce to P₁' P₂' P₃' P₄', R=P₀P₄/P₀P₄'. So: R<1 when magnifying, R>1 when reducing, R=1 can be default.

Note: 1. Specify G36 in a separate block. After the figure is enlarged or reduced, specify G37 to cancel the scaling mode.

2. If someone axis didn't specify, this axis won't scaling.

3. the scaling is invalid for value of tool compensation & tool offset.

4. In scaling mode, G27, G28, G61& G30, or commands related to the coordinate system(G52-G59) must not be specified. When any of these G codes is necessary, specify it after canceling scaling mode.

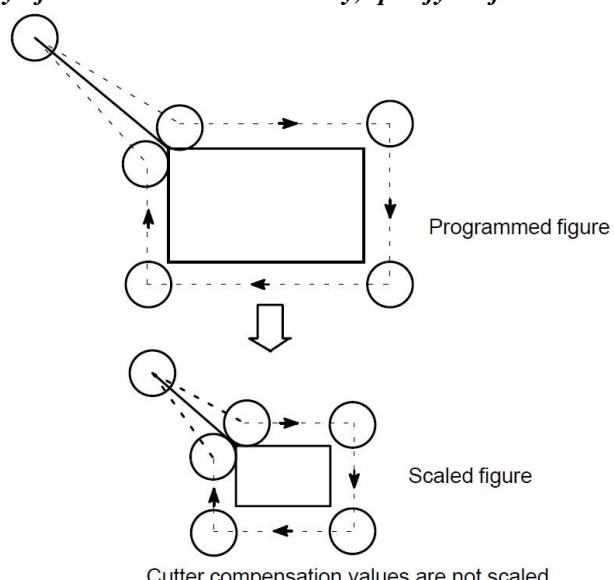


Fig3.15.2 Scaling during tool compensation

3.22 Programmable mirror image (G11/G12)

A mirror image of a programmed command can be produced with respect to a programmed axis of symmetry.(See Fig3.16.1)

Format: G11 X_ Y_ (Z_ X_) (Y_ Z_) (mode) ;according to XYZ symmetry axis

G12 (mode,original) ;Cancel Mirror.

Programmable mirror image finish processing of symmetric point in this axis by change the feeding direction of axis,which is specified by current interpolation plane G17(X_ Y_) /G18(Z_ X_) /G19(Y_ Z_). The value behind of X_ Y_ Z_ is the coordinate value of current coordinate system.

Example of mirror image program

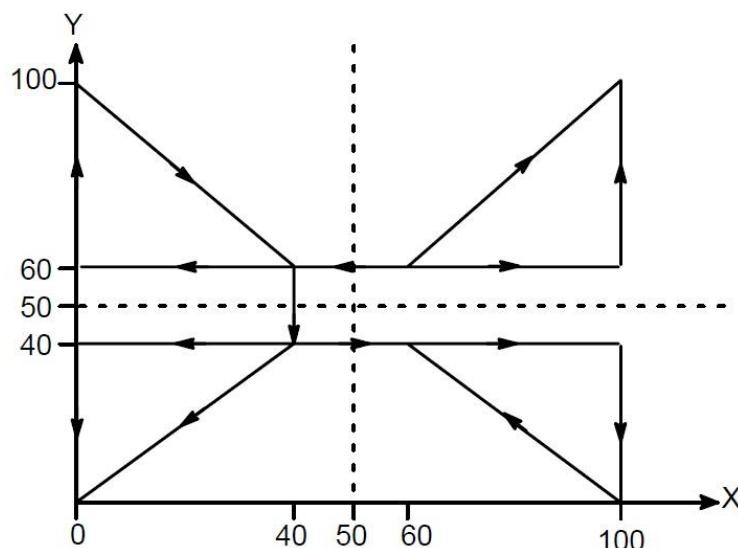


Fig3.16.1 Example of a mirror image program

Sub program: KG11

```
N10 G00 G90 X60.0 Y60.0 ;
N20 G01 X100.0 F100 ;
N30 G01 Y100.0 ;
N40 G01 X60.0 Y60.0 ;
N50 M99 ;
```

Main program

```
N10 G00 G90;
N20 M98 PKG11;
N30 G11 X50.0
N40 M98 PKG11;
N50 G11 X50.0 Y50.0
N60 M98 PKG11;
N70 G11 Y50.0
N80 M98 P%9000;
N90 G12;
N100 M12;
```

Note: before specifying the G code for reference position return(G26,G28,G61,G30) or coordinate system setting(G52-G59),cancel the mirror image mode.

3.23 Rotate Coordinate Instruction (G68/G69)

A programming shape can be rotated, by using this function it becomes possible, for example, to modify a program using a rotating command when a workpiece has been placed with some angle rotated from the programmed position on the machine. Further, when there is a pattern comprising some identical shapes in the positions rotated from a shape, the time required for programming and the length of the program can be reduced by preparing a subprogram of the shape and calling it after rotation. In short: By designating a rotation angle with G68 in the program, actual machining will be performed on the rotated coordinate.

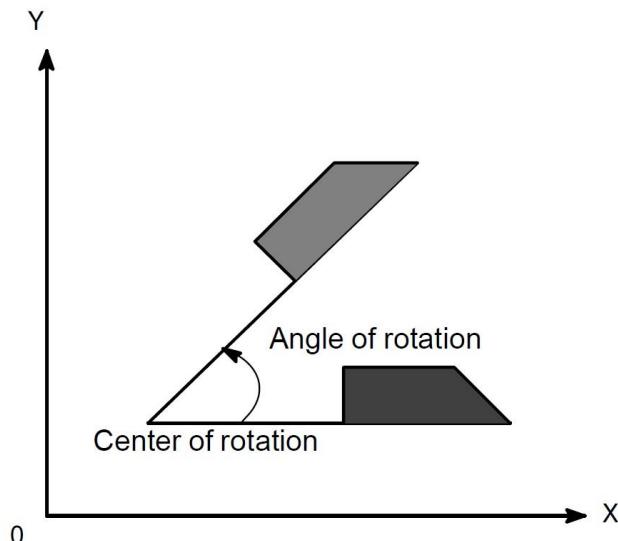


Fig3.17.1 coordinate system rotation

User can specify the Center-point (origin) and Angle-of-rotation, and whole the pattern of operation will be executed there. Anyway, if the shape of workpiece is comprised of many same graphics, the graphics unit can be compiled to subprogram, then the subprogram is on tap for rotation instruction of main program.

Format	
$\left\{ \begin{array}{l} G17 \\ G18 \\ G19 \end{array} \right\}$	G68 α β R ; Start rotation of a coordinate system.
...	Coordinate system rotation mode (The coordinate system is rotated)
G69 ;	Coordinate system rotation cancel command
Meaning of command	
G17(G18 or G19) : Select the plane in which contains the figure to be rotated.	
α β	: Absolute command for two of the X, Y and Z axes that correspond to the current plane selected by command(G17/G18/G19). The command specifies the coordinates of the center of rotation for the values specified subsequent to G68.
R	: Rotating angle, with a positive value indicates counter clockwise rotation. With negative value indicates clockwise rotation.
Least input increment: 0.001 deg; Valid data range: -360.000 ~ 360.000 .	

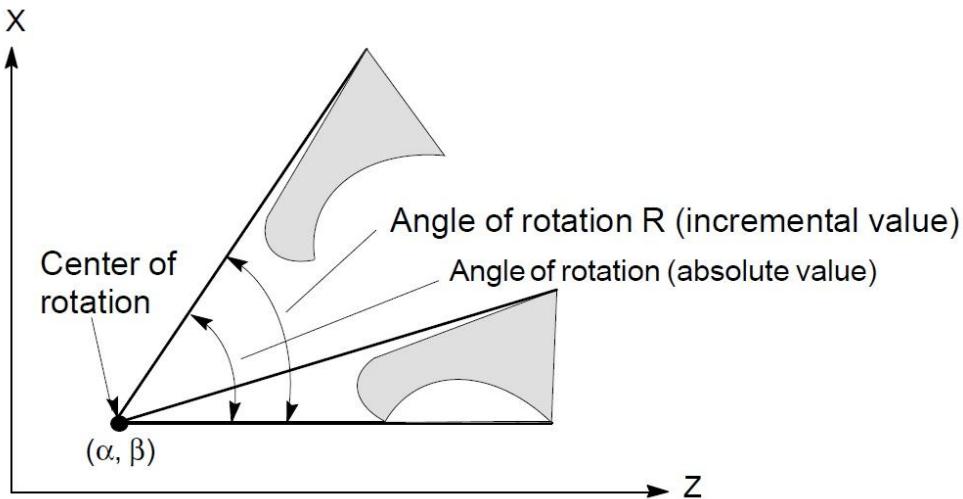


Fig3.17.2 coordinate system rotation

Note: When a decimal fraction is used to specify angular displacement (R_), the 1st digit corresponds to degree units.

G code for selecting a plane: G17/G18/G19: The G code for selecting a plane(G17 G18 or G19) can be specified before the block containing the G code for coordinate system rotation(G68). G17, G18 or G19 must not be designated in the mode of coordinate system rotation.

Incremental command in coordinate system rotation mode: The center of rotation for an incremental command programmed after G68 but before an absolute command is the tool position when G68 was programmed.(See Fig3.17.3).

Center of rotation: When α β is not programmed, the tool position when G68 was programmed is assumed as the center of rotation.

Coordinate system rotation cancel command: The G code used to cancel coordinate system rotation (G69) may be specified in a block in which another command is specified.

Tool compensation: Cutter compensation, tool length compensation, tool offset, and other compensation operations are executed after the coordinate system is rotated.

Limitations:

Commands related to reference position return and the coordinate system: In coordinate system rotation mode, G codes related to reference position return (G28,G26,G61,G30, etc.) and these for changing the coordinate system(G52-G59, etc.)must not be specified. If any of these G codes is necessary, specify it only after canceling coordinate system rotation mode.

Incremental command: The first move command after the coordinate system rotation cancel command (G69) must be specified with absolute values. If an incremental move command is specified, correct movement will not be performed.

Example1: Absolute/Incremental Position commands:

```

N1 G01 X-500 Y-500 F2000 G17;
N2 G68 X700 Y300 R60;
N3 G90 G01 X0 Y0 F2000;
(G91 X500.0 Y500.0)
N4 G91 X1000
N5 G02 Y1000 R1000
N6 G03 X-1000 I-500 J-500;

```

```

N7 G01 Y-1000
N8 G69
N9 G90 X-500 Y-500
N10 M02;

```

Tool path when the incremental command is designated in the N3 block (in parenthesis)

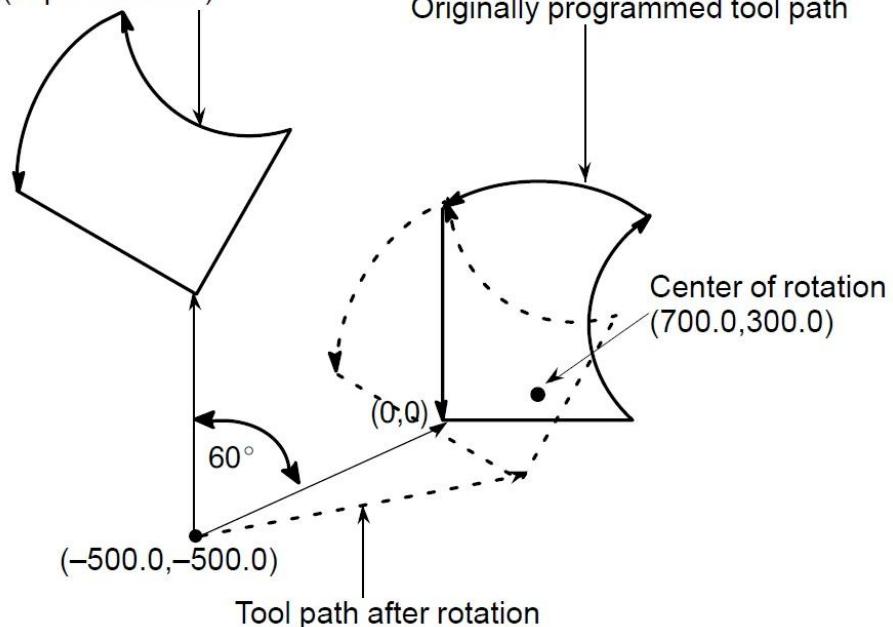
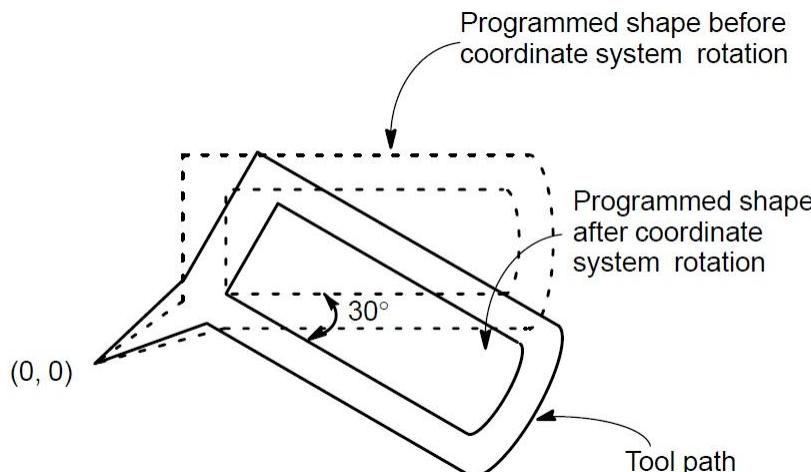


Fig3.17.3 Absolute/Incremental command during coordinate system Rotation

Example2: Cutter compensation C and coordinate rotation

It is possible to specify G68 and G69 in cutter compensation C mode. The rotation plane must coincide with the plane of cutter compensation C.



```

N1 G01 X0 Y0 G69 ;
N2 G42 G90 X1000 Y1000 F1000 D01;
N3 G66 R-30000;
N4 G91 X2000 ;
N5 G03 Y1000 R1000 J500
N6 G01 X-2000;
N7 Y-1000 ;
N8 G69 ;
N9 G90 X0 Y0 ;
N10 G42 ;
N11 M30 ;

```

Fig3.17.4 Cutter compensation C and Coordinate system rotation

3.24 Tool Length Offset (G43/G44/G49)

This function can be used for setting the difference between tool length assumed during programming and the actual tool length of the tool used into the offset memory. It is possible to compensate the difference without changing the program.

Specify the direction of offset with G43 or G44. Select a tool length offset value from the offset memory by entering the corresponding address and number(H code).

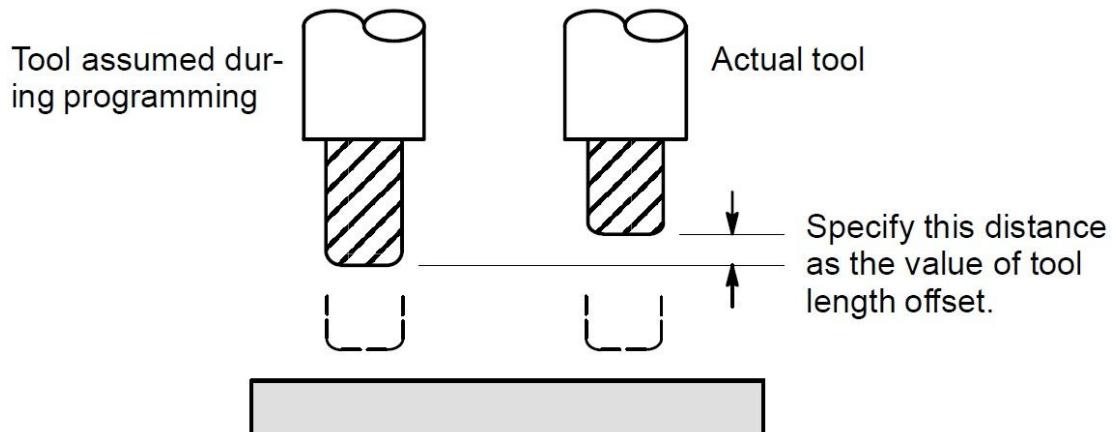


Fig3.18.1 Tool Length Offset

The following two methods of tool length offset can be used, depending on the axis along which tool length offset can be made.

- ✓ **Tool length offset A:** Compensates for the difference in tool length along the Z-axis.
- ✓ **Tool length offset B:** Compensates for the difference in tool length along the X-axis, Y-axis , or Z-axis.

Tool length offset A	G43 H_ G44 H_	Offset along the Z-axis
Tool length offset 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_ ;	G43: Positive Offset G44: Negative Offset G17: XY Plane selection, offset on Z-axis G18: ZX Plane selection, offset on Y-axis G19: YZ Plane selection, offset on X-axis
Tool length offset cancel	G49; or H0 ;	Cancel Length Offset

Selection of tool length offset: Select tool length offset A, B, or C, by setting P47 on Speed parameter. When value of P47 +0, default set, tool length offset A; When +1, tool length offset B;

At tool length offset B, length offset can be shift to another axis when the axes are specified in two or more blocks.

The tool length offset value assigned to the number(offset number) specified in the H code is selected from offset memory and added to or subtracted from the moving command in the program.

When G43 is specified, the tool length offset value(store in offset memory) specified with the H code is added to the coordinates of the end position specified by a command in the program.

When G44 is specified, the same value is subtracted from the coordinate of the end position. The resulting coordinates indicate the end position after compensation, regardless of whether the absolute or incremental mode is selected.

G43 and G44 are modal G codes. They are valid until another G code belonging to the same group is used.

Warning: When the tool length offset value is changed due to a change of offset number, the offset value changes to the new tool length offset value, the new tool length offset value is not added to the old tool length offset value.

To cancel tool length offset, specify G49 or H0. After G49 or H0 is specified, the system immediately cancels the offset mode.

Note: The tool length offset value corresponding to offset No.0, that is, H0 always means 0. It is impossible to set any other tool length offset value to H0.

Example1: H1: tool length offset value is 20.0 (Tool Length Offset A)

H2: tool length offset value is 30.0

G90 G43 Z100.0 H1; Z will move to 120.0

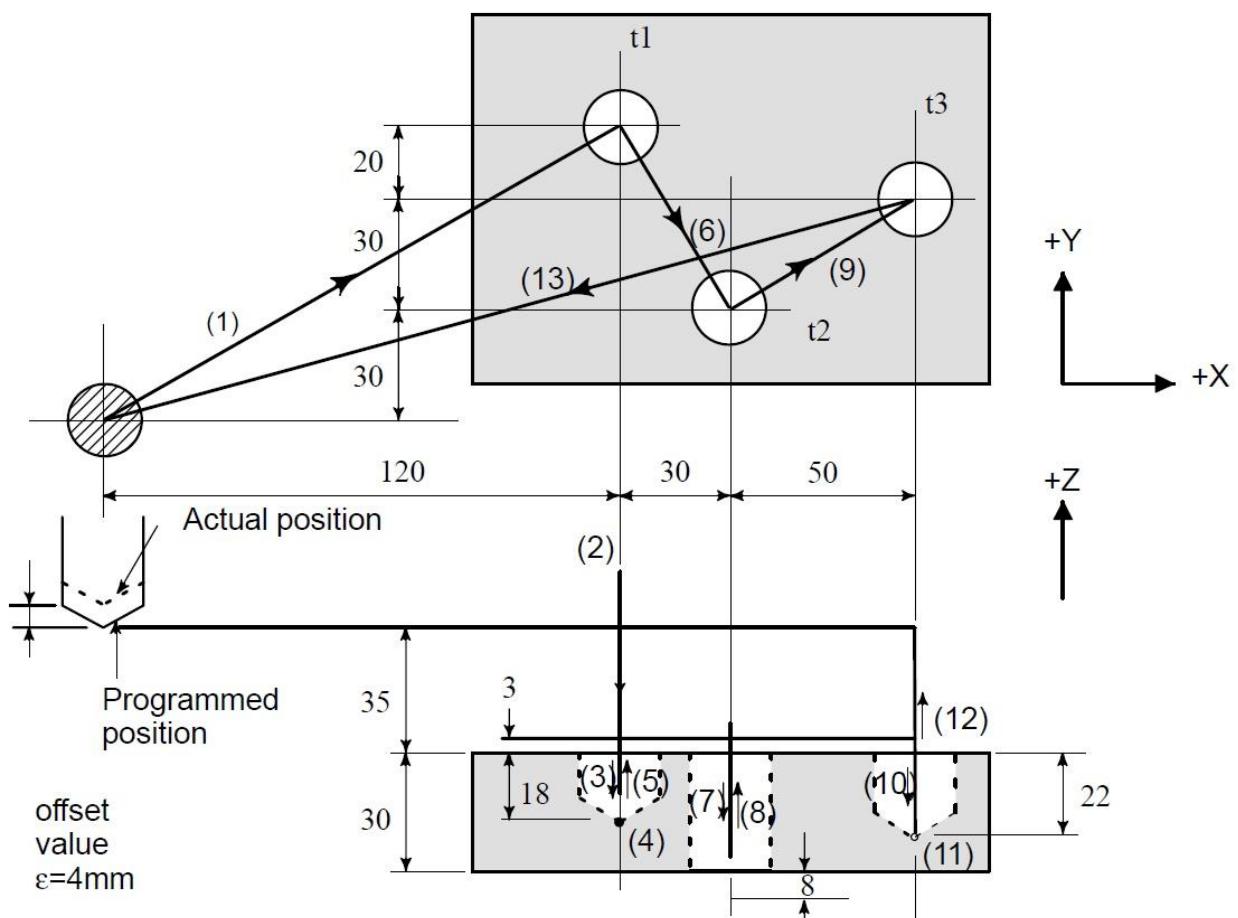
G90 G43 Z100.0 H2; Z will move to 130.0

Example2: Offset in X and Y axes.(Tool Length Offset B)

G19 G43 H_ ; Offset in X-axis

G18 G43 H_ ; Offset in Y-axis

Example3: H1=-4.0(Tool length offset value)



N1 G91 G00 X120.0 Y80.0;	N5 G0 Z21.0 ;	N9 X50.0 Y30.0 ;	N13 X-200.0 Y-60.0 ;
N2 G43 Z-32.0 H1;	N6 X30.0 Y-50.0 ;	N10 G01 Z-25.0 ;	N14 M2 ;
N3 G01 Z-21.0 F1000 ;	N7 G01 Z-41.0 ;	N11 G04 P2000 ;	
N4 G04 P2000 ;	N8 G00 Z41.0 ;	N12 G00 Z57.0 H0 ;	

3.25 Tool Offset(G45/G46/G47/G48)

The programmed travel distance of the tool can be increased or decreased by a specified tool offset value. The tool offset function can also be applied to an additional axis.

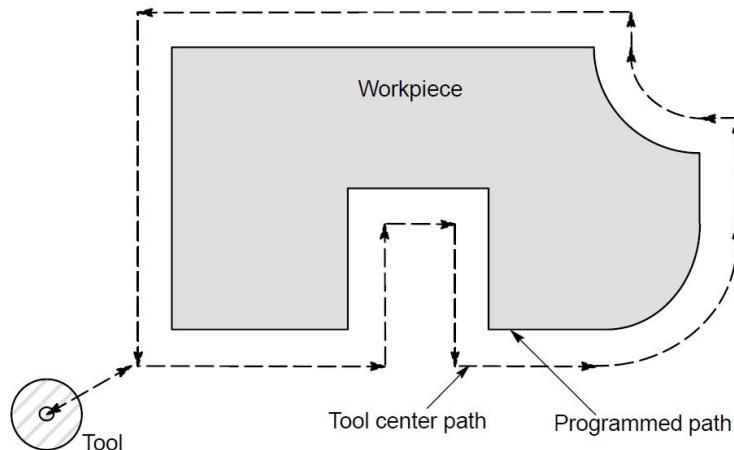


Fig3.19.1 Tool offset

Format:	G45 IP_D_;	Increase the travel distance by the tool offset value
	G46 IP_D_;	Decrease the travel distance by the tool offset value
	G47 IP_D_;	Increase the travel distance by twice tool offset value
	G48 IP_D_;	Decrease the travel distance by twice tool offset value

G45-G48: One-shot G code for increasing or decreasing the travel distance.

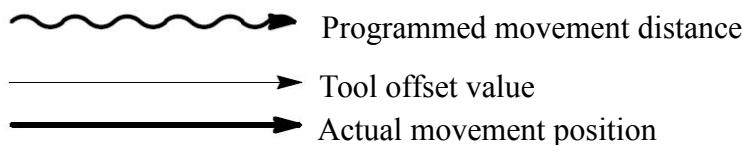
IP: Command for moving the tool

D: Code for specifying the tool offset value

As shown in Table3.19.2, the travel distance of the tool is increased or decreased by the specified tool offset value.

Table3.19.2 Increase and decrease of the tool travel distance

G code	When a positive tool offset value is specified	When a negative tool offset value is specified
G45	 Start position End position	 Start position End position
G46	 Start position End position	 Start position End position
G47	 Start position End position	 Start position End position
G48	 Start position End position	 Start position End position



In the absolute mode, the travel distance is increased or decreased as the tool is moved from the end position of the previous block to the position specified by the block containing G45 to G48.

If a move command with a travel distance of zero is specified in the incremental command (G91) mode, the tool is moved by the distance corresponding to the specified tool offset value.

If a move command with a travel distance of zero is specified in the absolute command (G90) mode, the tool is not moved.

Once selected by D code, the tool offset value remains unchanged until another tool offset value is selected. D0 always indicates a tool offset value of zero.

Warning: 1. When G45-G48 is specified to n axes (n=1-4) simultaneously in a motion block, offset is applied to all n axes. When the cutter is offset only for cutter radius or diameter in taper cutting, over-cutting or undercutting occurs. Therefore, use cutter compensation (G40/G41/G42) shown in chapter 3.26.

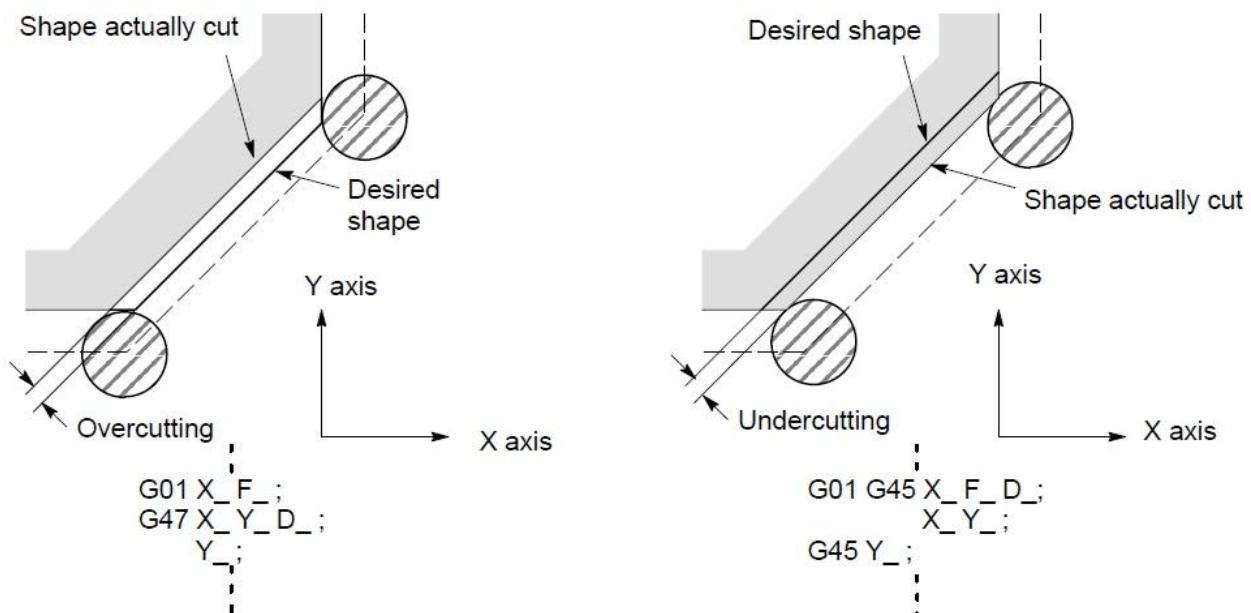
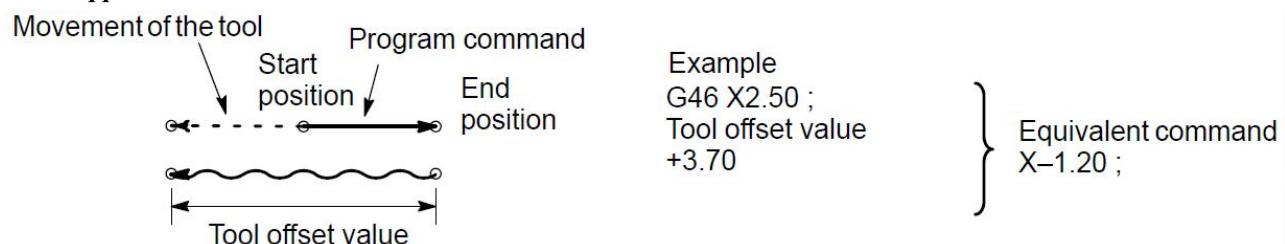


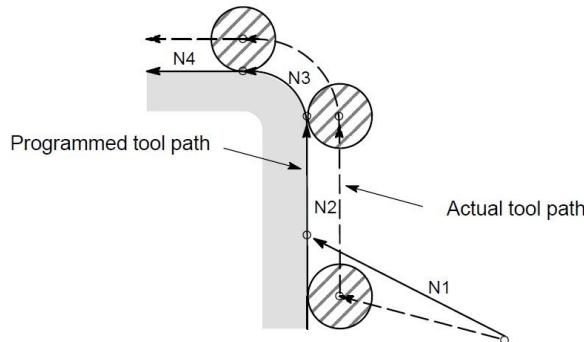
Fig3.19.3 Tool offset for over-cutting & undercutting

2. G45-G48 (tool offset) must not be used in the G41 or G42 (cutter compensation) mode.

Note: 1. When the specified direction is reversed by decrease as shown in the figure below, the tool moves in the opposite direction.



2. Tool offset can be applied to circular interpolation (G02, G03) with the G45 to G48 commands only for 1/4 and 3/4 circles using addresses I, J and K by the parameter setting, providing that the coordinate rotation be not specified at the same time. This function is provided for compatibility with the conventional CNC tape without any cutter compensation. The function should not be used when a new CNC program is prepared.



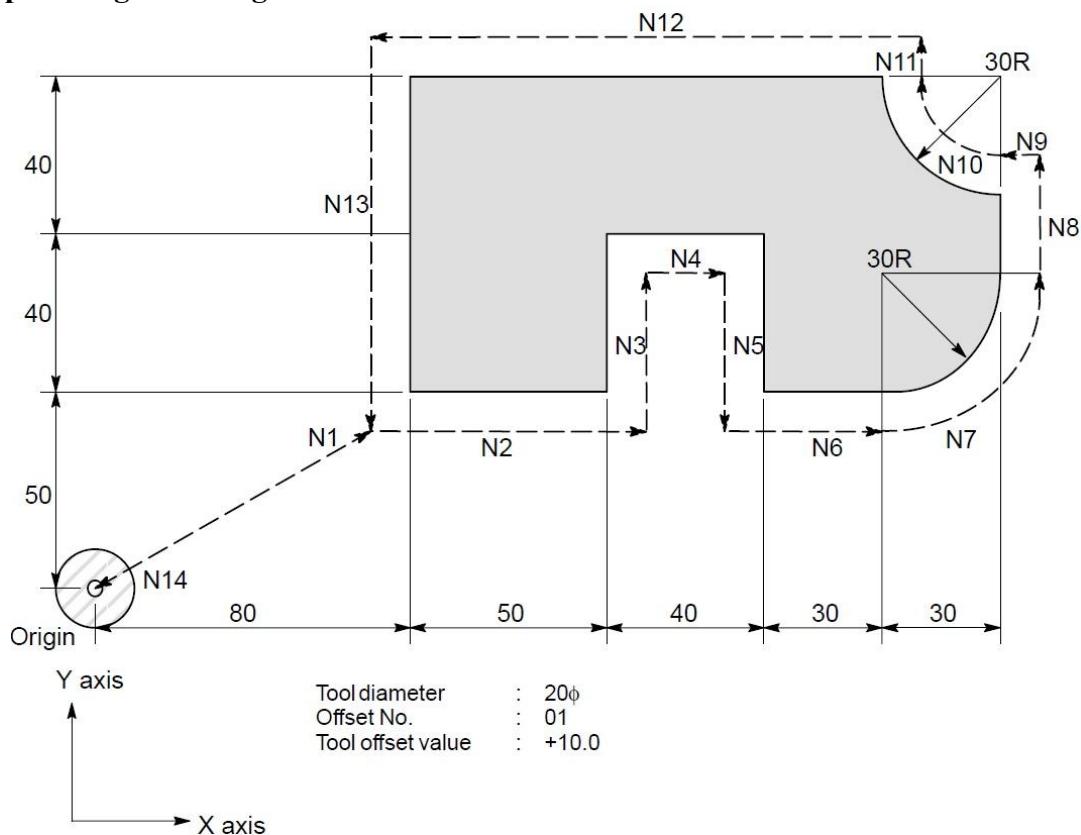
Program

```
N1 G46 G00 X_ Y_
N2 G45 G01 Y_ F_
N3 G45 G03 X_ Y_ I_ ;
N4 G01 X_
```

Fig3.19.4 Tool offset for circular interpolation

3.G45-G48 are ignored in canned cycle mode. Perform tool offset by specifying G45 to G48 before entering canned cycle mode and cancel the offset after releasing the canned cycle mode.

Example: Program using tool offset



Program

N1 G91 G46 G00 X80.0 Y50.0 D01	N2 G47 G01 X50.0 F120.0	N3 Y40.0
N4 G48 X40.0	N5 Y-40.0;	N6 G45 X30.0
N7 G45 G03 X30.0 Y30.0 J30.0	N8 G45 G01 Y20.0	N9 G46 X0
N10 G46 G02 X-30.0 Y30.0 J30.0	N11 G45 G01 Y0 ;	N12 G47 X-120.0
N13 G47 Y-80.0	N14 G46 G00 X-80.0 Y-50.0	

N9: Decrease toward the positive direction for movement amount “0”. The tool moves in the -X direction by the offset value. N11:Decrease toward the positive direction for movement amount “0”. The tool moves in the +Y direction by the offset value

3.26 Tool Radius Compensation C (G40/G41/G42)

When the tool is moved, the tool path can be shifted by radius of the tool.

To make an offset as large as the radius of the tool, CNC firstly establish offset vector with a length equal to the radius of the tool(start-up). The offset vector is perpendicular to tool path. The tail of vector is on the workpiece side and the head position to the center of the tool.

If a linear interpolation or circular interpolation command is specified after start-up, the tool path can be shifted by the length of the offset vector during machining.

To return the tool to the start position at end of machining, cancel the tool compensation mode.

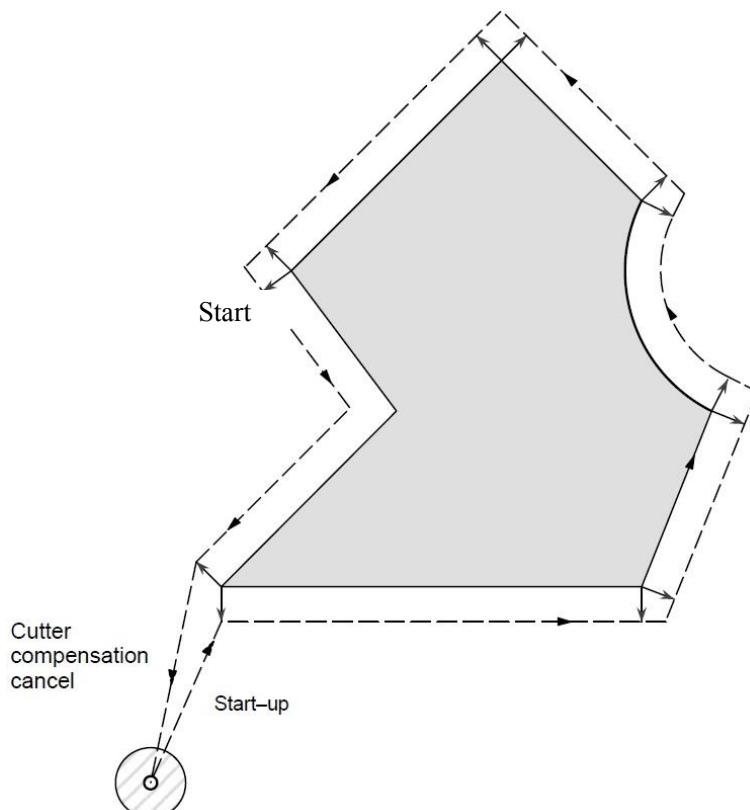


Fig3.20.1 Outline of tool compensation C

Format: **G41 IP_ D_ ; Tool compensation left**
G42 IP_ D_ ; Tool compensation right
G40 ; Cancel compensation.

IP_ : command for axis movement

D_ : code for specifying as the cutter compensation value.(D1-D99)

At the beginning when power is applied the control is in the cancel mode. In the cancel mode, the vector is always 0, and the tool center path coincides with the programmed path.

Start Up: When a tool compensation command(G41 or G42, nonzero dimension words in the offset plane, and D code other than D0) is specified in the offset cancel mode, the CNC enters the offset mode. Moving the tool with this command is called start-up.

Specify positioning(G00) or linear interpolation (G01) for start-up. If circular interpolation (G02, G03) is specified, CNC system will hint alarm.

When processing the start-up block and subsequent blocks, the CNC pre-read two blocks.

Offset mode: In the offset mode, compensation is accomplished by positioning (G00), linear interpolation (G01), or circular interpolation (G02, G03). If two or more blocks that do not move the tool (miscellaneous function, dwell, etc.) are processed in the offset mode, the tool will make either an excessive or insufficient cut. If the offset plane is switched in the offset mode, CNC system will alarm and the tool is stopped.

Offset mode cancel: In the offset mode, when a block which satisfies any one of the following conditions is executed, the CNC enters the offset cancel mode, and the action of this block is called the offset cancel.

1. G40 has been commanded.

2. 0 has been commanded as the offset number for tool compensation.

When performing offset cancel, circular arc commands (G02 and G03) are not available. If a circular arc is commanded, CNC system will alarm & stops tool.

In the offset cancel, the control executes the instructions in that block and the block in the cutter compensation buffer. In the meantime, in the case of a single block mode, after reading one block, the control executes it and stops. By pushing the cycle start button once more, one block is executed without reading the next block. Then the control is in the cancel mode, and normally, the block to be executed next will be stored in the buffer register and the next block is not read into the buffer for tool compensation.

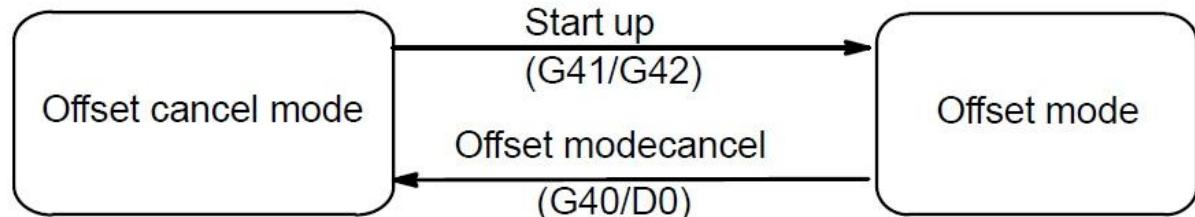


Fig3.20.2 Changing the offset mode

In general, the cutter compensation value shall be changed in the cancel mode, when changing tools. If the tool compensation value is changed in offset mode, the vector at the end point of the block is calculated for the new cutter compensation value.

Calculated from the cutter compensation value in the block N6

Calculated from the cutter compensation value in the block N7

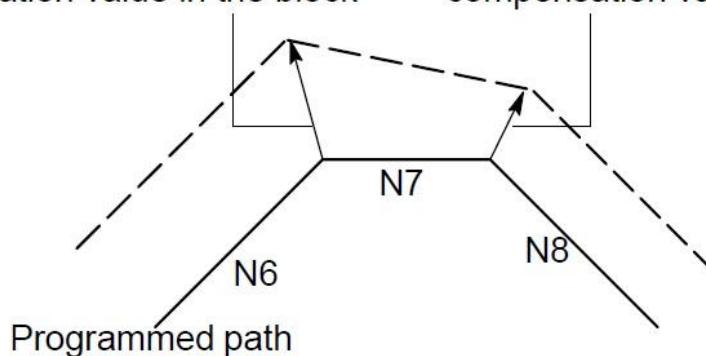


Fig3.20.3 Changing the cutter compensation value

If the offset amount is negative(-), distribution is made for a figure in which G41's and G42's are all replaced with each other on the program. Consequently, if the tool center is passing around the outside of the workpiece, it will pass around the inside, and vice versa.

The figure below shows one example. Generally, the offset amount is programmed to be

positive(+). When a tool path is programmed as in (1),, if the offset amount is made negative (-), the tool center moves as in (2), and vice versa. Consequently, the same tape permits cutting both male and female shapes, and any gap between them can be adjusted by the selection of the offset amount. Applicable if start-up and cancel is A type.

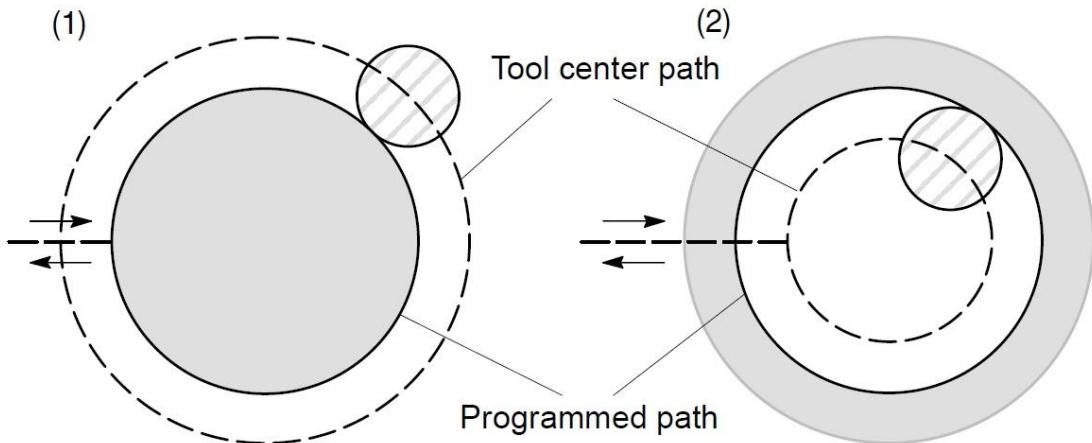


Fig3.20.4 Tool center paths when positive and negative tool compensation values are specified.

Note: 1. The tool compensation value corresponding to offset No.0, that is, D0 always means 0. It is impossible to set D0 to any other offset amount.

2. **Tool compensation C can be specified by D code.**

The offset vector is the two dimensional vector that is equal to the tool compensation value assigned by D code. It is calculated inside the control unit, and its direction is updated in accordance with the progress of the tool in each block. The offset vector is deleted by reset.

Specify a tool compensation value with a number assigned to it. The number consists of 01 to 99 after address D (D code). The D code is valid until another D code is specified. The D code is used to specify the tool offset value as well as the cutter compensation value.

Offset calculation is carried out in the plane determined by G17, G18 and G19, (G codes for plane selection). This plane is called the offset plane. Compensation is not executed for the coordinate of a position which is not in the specified plane. The programmed values are used as they are. In simultaneous 3 axes control, the tool path projected on the offset plane is compensated.

The offset plane is changed during the offset cancel mode. If it is performed during the offset mode, CNC system will alarm and stop moving.

Example:

G54 X0 Y0 Z0 ;Specifies a workpiece coordinates system.
 The tool is positioned at the start position (X0, Y0, Z0).

N1 G90 G17 G00 G41 D03 X250.0 Y550.0; Starts tool compensation(start-up). Tool is shifted to the left of the programmed path by the distance specified in D03. In other words the tool path is shifted by the radius of the tool (offset mode)Because D03 is set to 15 beforehand(the radius of the tool is 15mm).

N2 G01 Y900.0 F150 ;Specify machining from P1 to P2.

N3 X450.0 ;Specify machining from P2 to P3.

N4 G03 X500.0 Y1150.0 R650.0 ;Specifies machining from P3 to P4.

N5 G02 X900.0 R-250.0 ;Specifies machining from P4 to P5.

N6 G03 X950.0 Y900.0 R650.0 ;Specifies machining from P5 to P6.

N7 G01 X1150.0 ;.....Specifies machining from P6 to P7.

N8 Y550.0 ;.....Specifies machining from P7 to P8.

N9 X700.0 Y650.0.....Specifies machining from P8 to P9.

N10 X250.0 Y550.0.....Specifies machining from P9 to P1.

N11 G00 G40 X0 Y0.....Cancel the offset mode.

The tool is returned to the start position (X0, Y0, Z0).

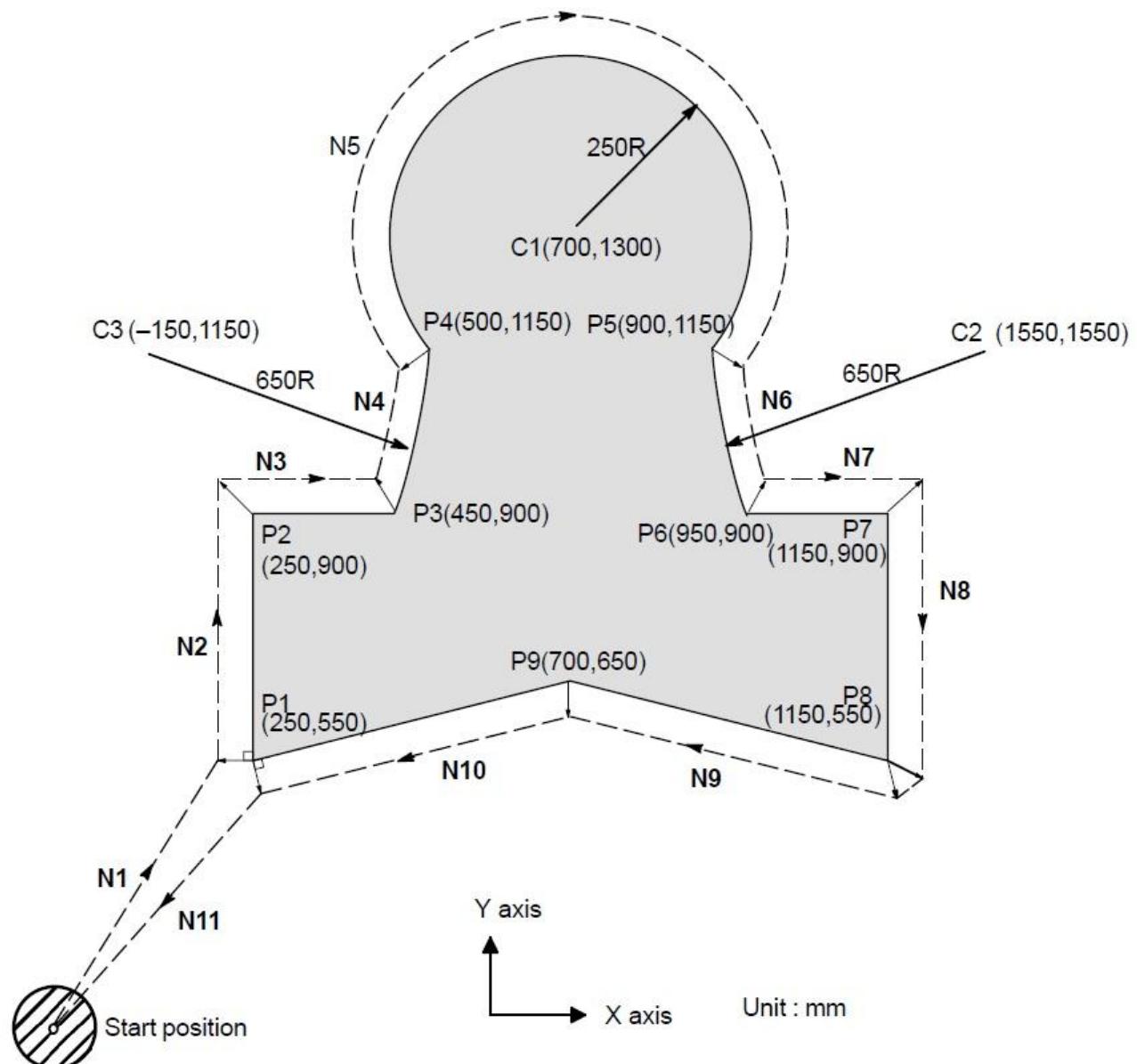


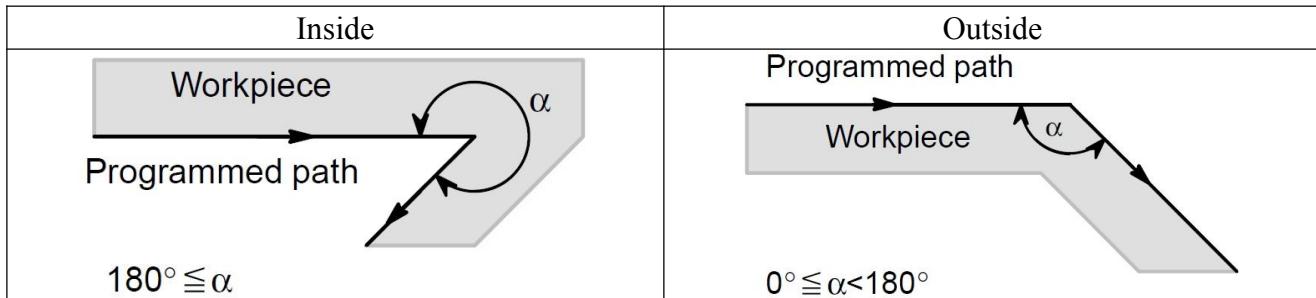
Fig3.20.5 Example of Tool Compensation C

3.27 Details of Tool Compensation C

This section provides a detailed explanation of the movement of the tool for tool compensation C outlined in Section 3.20.

3.27.1 Inside and outside

When an angle of intersection created by tool paths specified with move commands for two blocks is over 180° , it is referred to as "Inside". When the angle is between 0° and 180° , it is referred to as "Outside".



The following symbols are used in subsequent figures:

S: a position at which a single block is executed once. L: the tool moves along a straight line.
 SS: a position at which a single block is executed twice. O: the center of the tool.
 C: the tool moves along a arc. r: the tool compensation value.

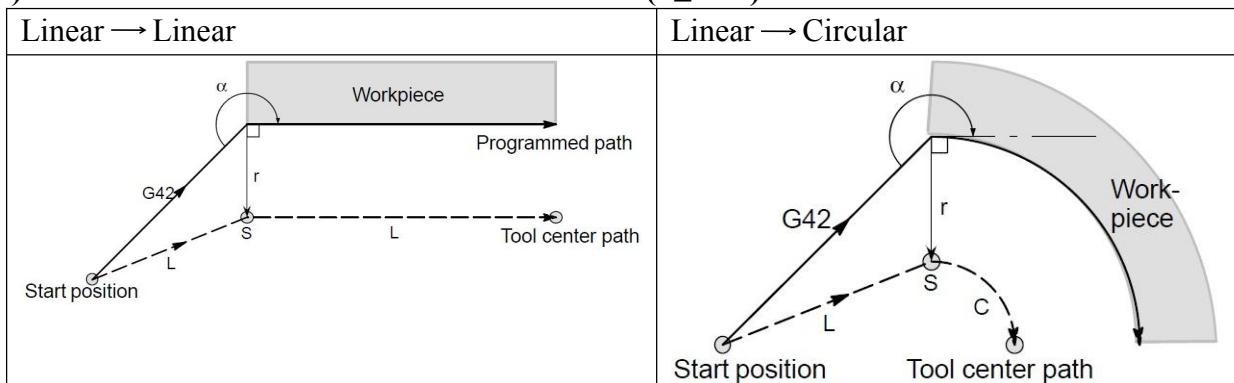
An intersection is a position at which the programmed paths of two blocks intersect with each other after they are shifted by r.

3.27.2 Tool movement when start-up

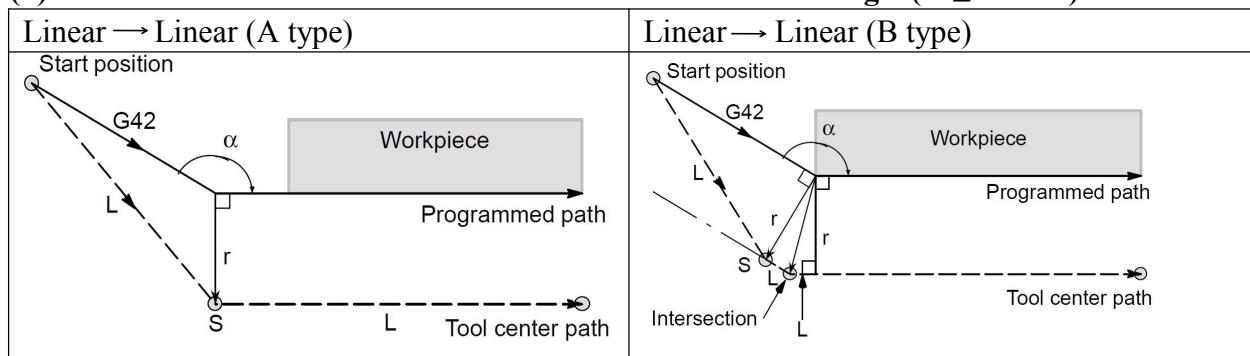
When the offset cancel mode is changed to offset mode, the tool moves as illustrated below:

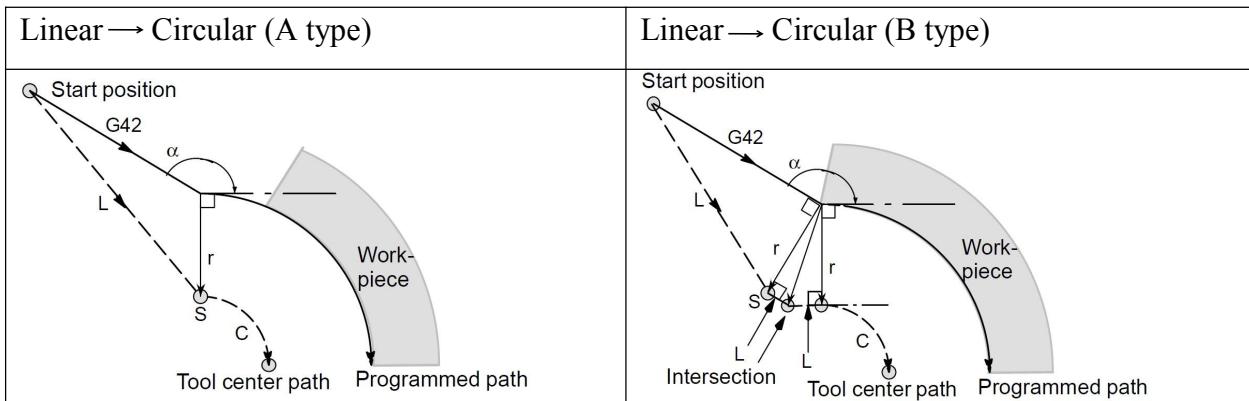
Tool path in start-up has two types A and B, and they are selected by P1 on Tool parameter.

(1) Tool movement around an inside of a corner ($\alpha \geq 180^\circ$)

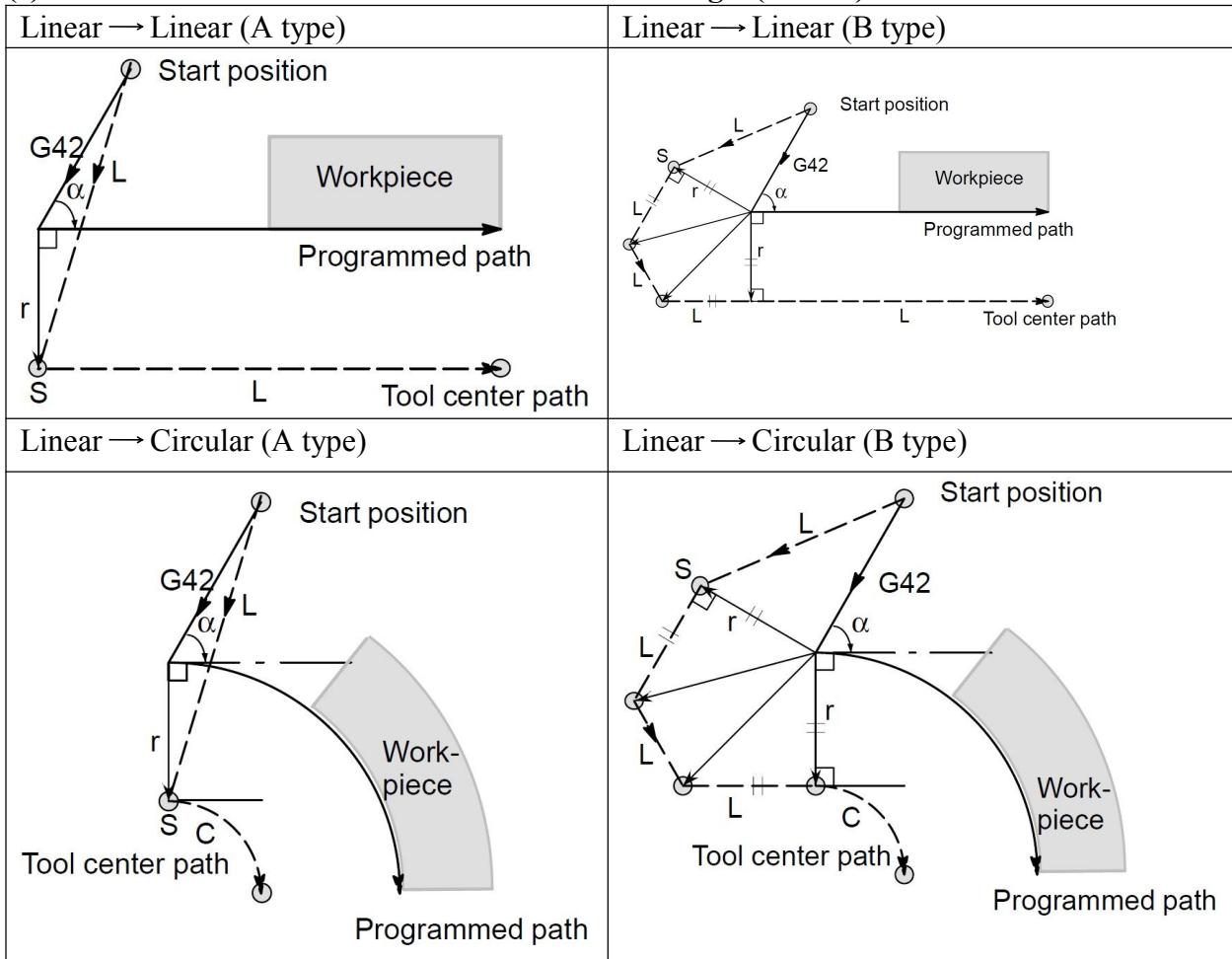


(b) Tool movement around the outside of a corner at obtuse angle ($90^\circ \leq \alpha < 180^\circ$)





(c) Tool movement around the outside of an acute angle ($\alpha < 90^\circ$)



(d) Tool movement around outside linear to linear at an acute angle less than 1 degree ($\alpha < 1^\circ$)

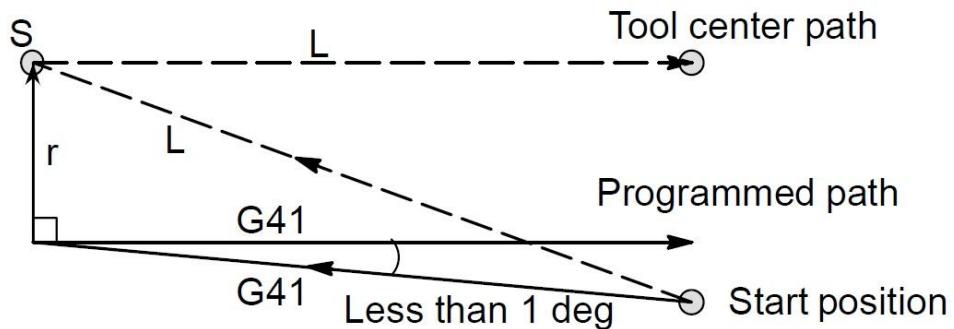


Fig3.21.1 Tool Movement at angle less than 1°

d) A block without tool movement specified at start-up: If the command is specified at start-up, the offset vector isn't created. for the definition of blocks that don't move the tool, see chapter 3.20.

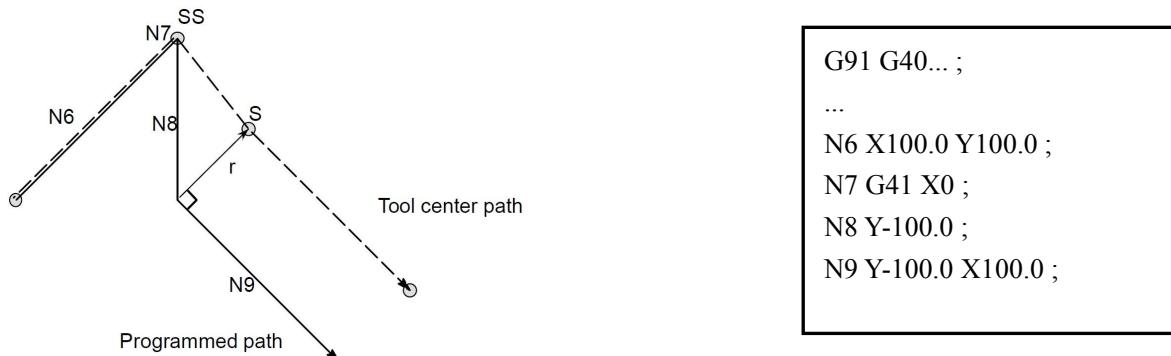
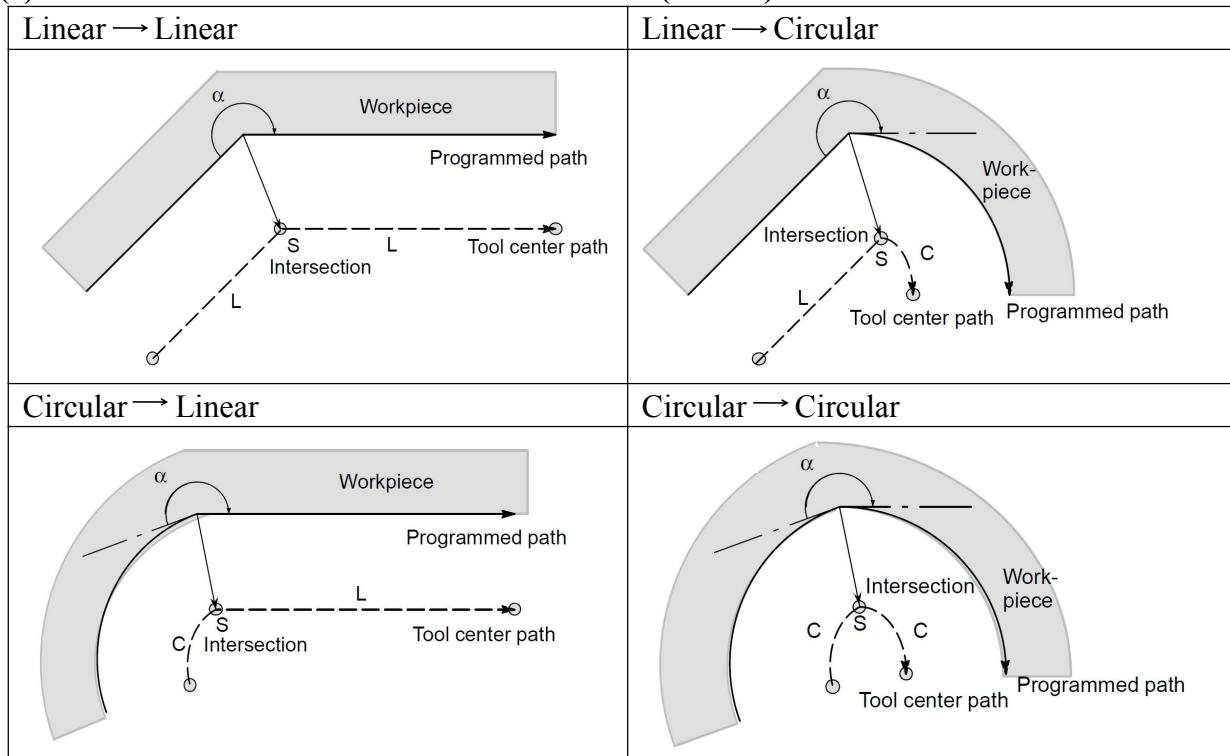


Fig3.21.2 A block without tool movement

3) Tool Movement in Offset Mode

In the offset mode, the tool moves as illustrated below:

(a) Tool movement around the inside of a corner ($180^\circ \leq \alpha$)



(b) Tool movement around the inside ($\alpha < 1^\circ$) with an abnormally long vector, linear→linear.

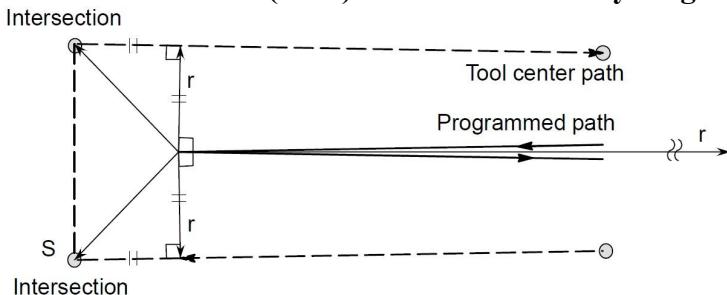
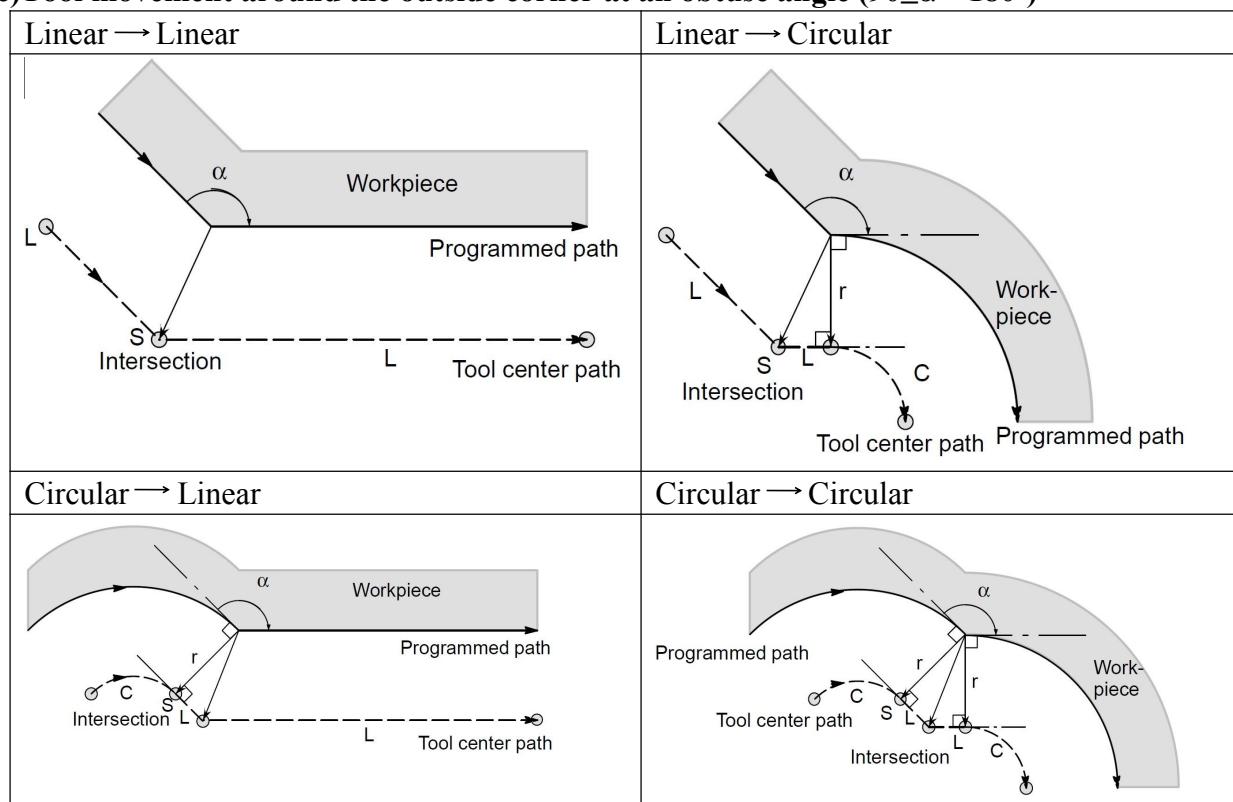


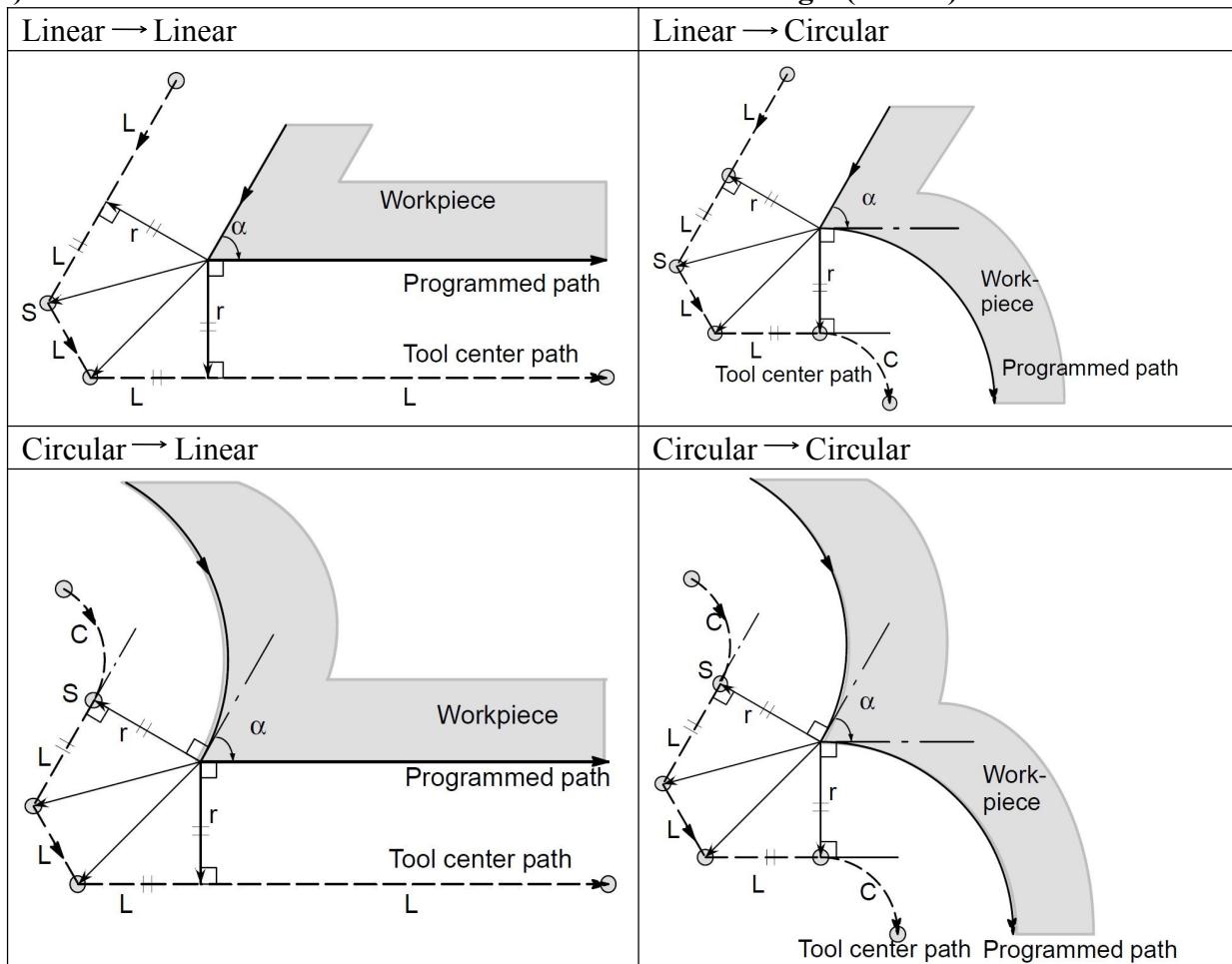
Fig3.21.3 Tool Movement at angle less than 1°

Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

(c) Tool movement around the outside corner at an obtuse angle ($90^\circ \leq \alpha < 180^\circ$)



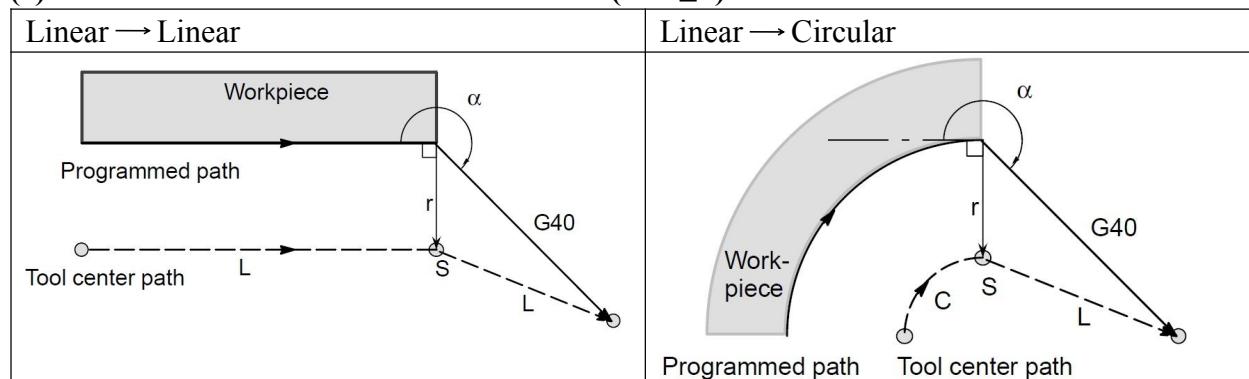
(d) Tool movement around the outside corner at an acute angle ($\alpha < 90^\circ$)



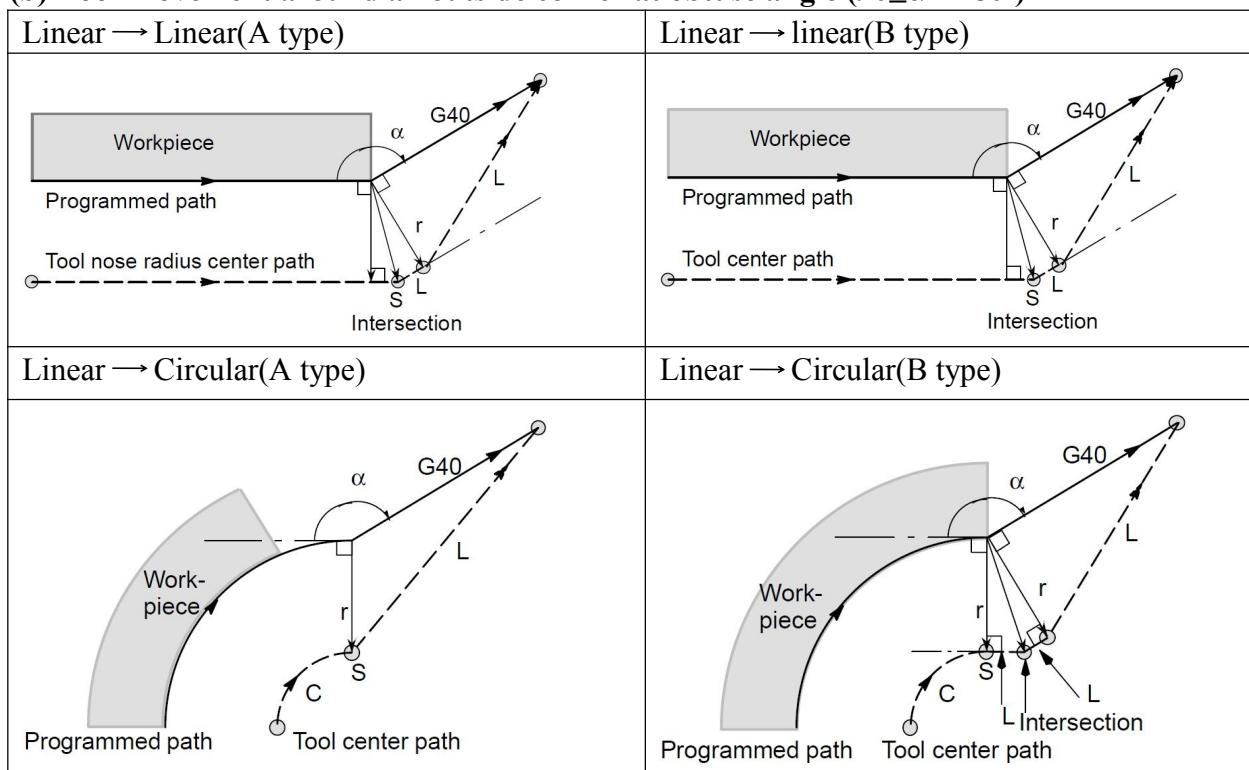
4) Tool Movement in Offset Mode Cancel

Tool path has two types, A and B; and they are selected by P2 in Tool parameter.

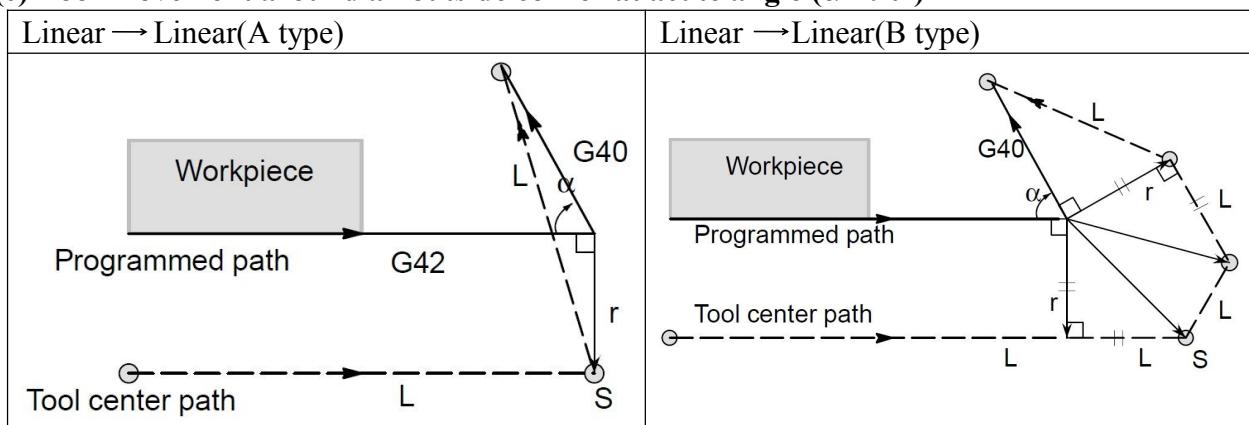
(a) Tool movement around an inside corner ($180^\circ \leq \alpha$)

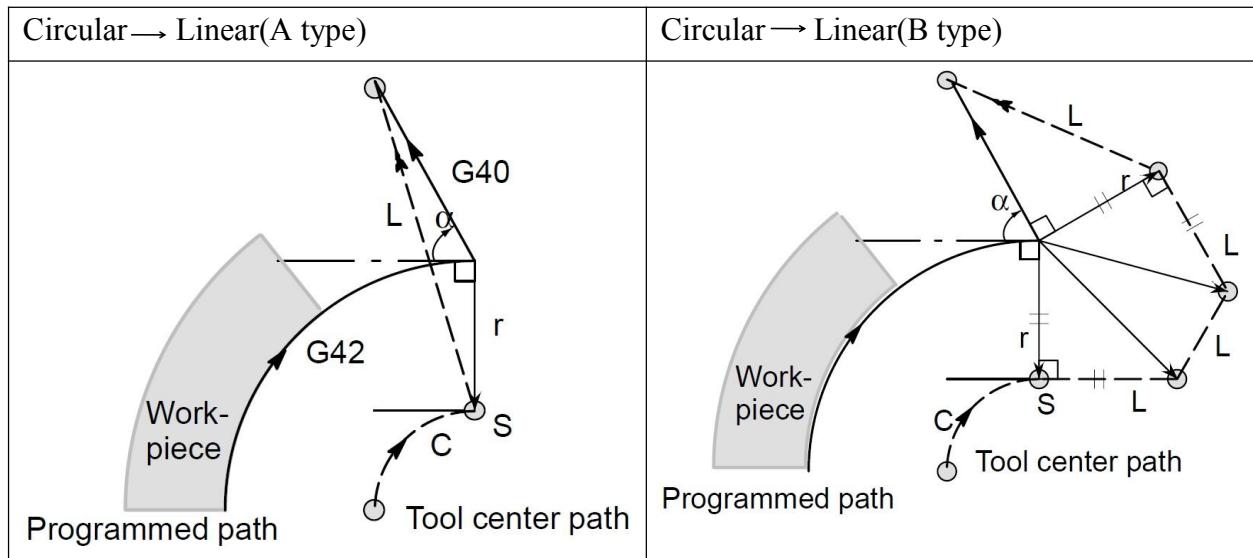


(b) Tool Movement around an outside corner at obtuse angle ($90^\circ \leq \alpha < 180^\circ$)



(c) Tool Movement around an outside corner at acute angle ($\alpha < 90^\circ$)





(d) Tool movement around outside linear to linear at an acute angle less than 1 degree($\alpha < 1^\circ$)

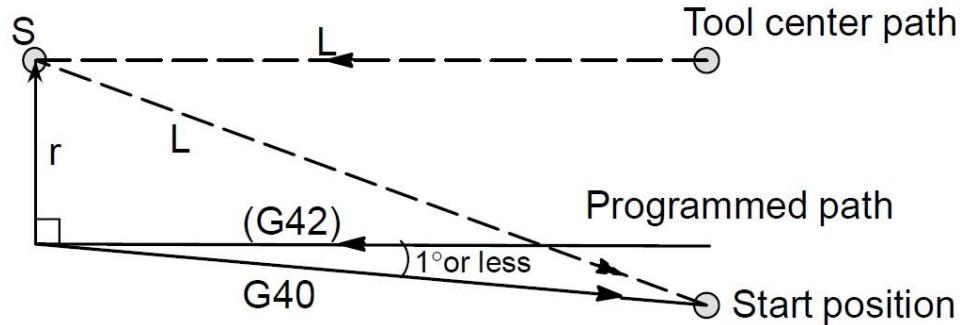


Fig3.21.4 Tool Movement at angle less than 1

(e) A block without tool movement specified together with offset cancel

When a block without tool movement is commanded together with an offset cancel, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in the earlier block, the vector is canceled in the next move command.

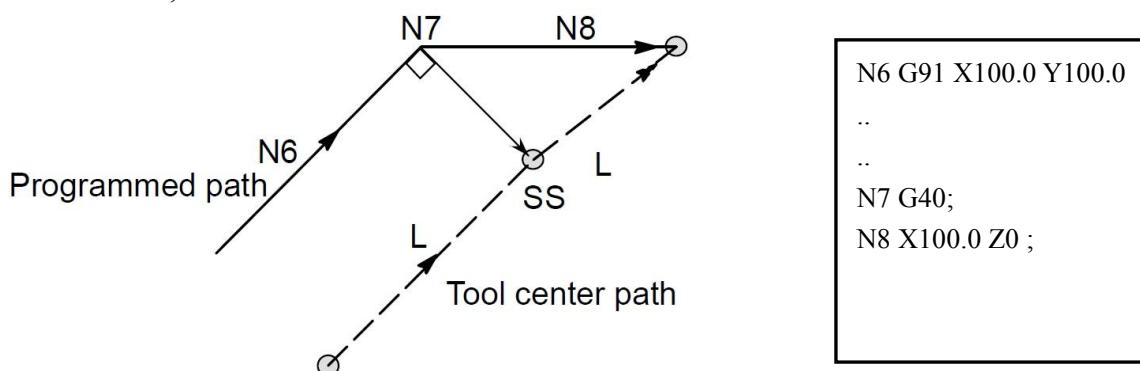


Fig3.21.5 A block without tool movement

3.28 Canned Cycle (G73-G89)

Canned cycles make it easier for the programmer to create programs. With a canned cycle, a frequently-used machining operation can be specified in a single block with a G function; without canned cycles, normally more than one block is required. In addition, the use of canned cycles can shorten the program to save memory.

Table3.22 Canned Cycles

G code	Feed mode (-Z direction)	Operation at the bottom of hole	Retraction (+Z direction)	Application
G73	Intermittent feed	-	Rapid traverse	High-speed peck drilling cycle
G74	Feed	Dwell → SP CW	Feed	Left-hand tapping cycle
G76	Feed	Oriented SP Stop	Rapid traverse	Fine boring Cycle
G80	-	-	-	Cancel
G81	Feed	-	Rapid traverse	Drilling cycle, spot drilling cycle
G82	Feed	Dwell	Rapid traverse	Drilling cycle, spot drilling cycle
G83	Intermittent feed	-	Rapid traverse	Peck drilling cycle
G84	Feed	Dwell → SP CCW	Feed	Tapping cycle
G85	Feed	-	Feed	Boring cycle
G86	Feed	Spindle Stop	Rapid traverse	Boring cycle
G87	Feed	Spindle CW	Rapid traverse	Back boring cycle
G89	Feed	Dwell	Feed	Boring cycle

A canned cycle consists of a sequence of six operations(Fig3.22.1)

- Operation 1 Position of X and Y axis(including also another axis)
- Operation 2 Rapid traverse up to point R level
- Operation 3 Hole Machining
- Operation 4 Operation at the bottom of a hole
- Operation 5 Retraction to point R level
- Operation 6 Rapid traverse up to the initial point

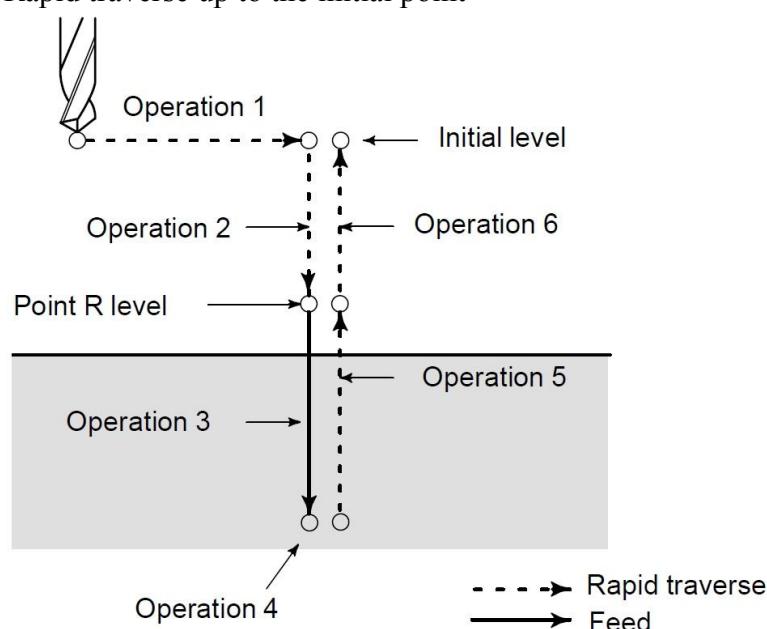


Fig3.22.1 Canned cycle operation sequence

The positioning plane is determined by plane selection code G17,G18 or G19. The positioning axis is an axis other than the drilling axis.P47 in Speed parameter + 2 is set for this function.

Although canned cycles include tapping and boring cycles as well as drilling cycles, in this chapter, only the term drilling will be used to refer to operations implemented with canned cycles.

The drilling axis is a basic axis(X,Y, or Z) not used to define the positioning plane, or any axis parallel to that basic axis.The axis(basic axis or parallel axis) is used as the drilling axis is determined according to the axis address for the drilling axis specified in the same block as G codes G73 to G89.If no axis address is specified for the drilling axis, the basic axis is assumed to be the drilling axis.

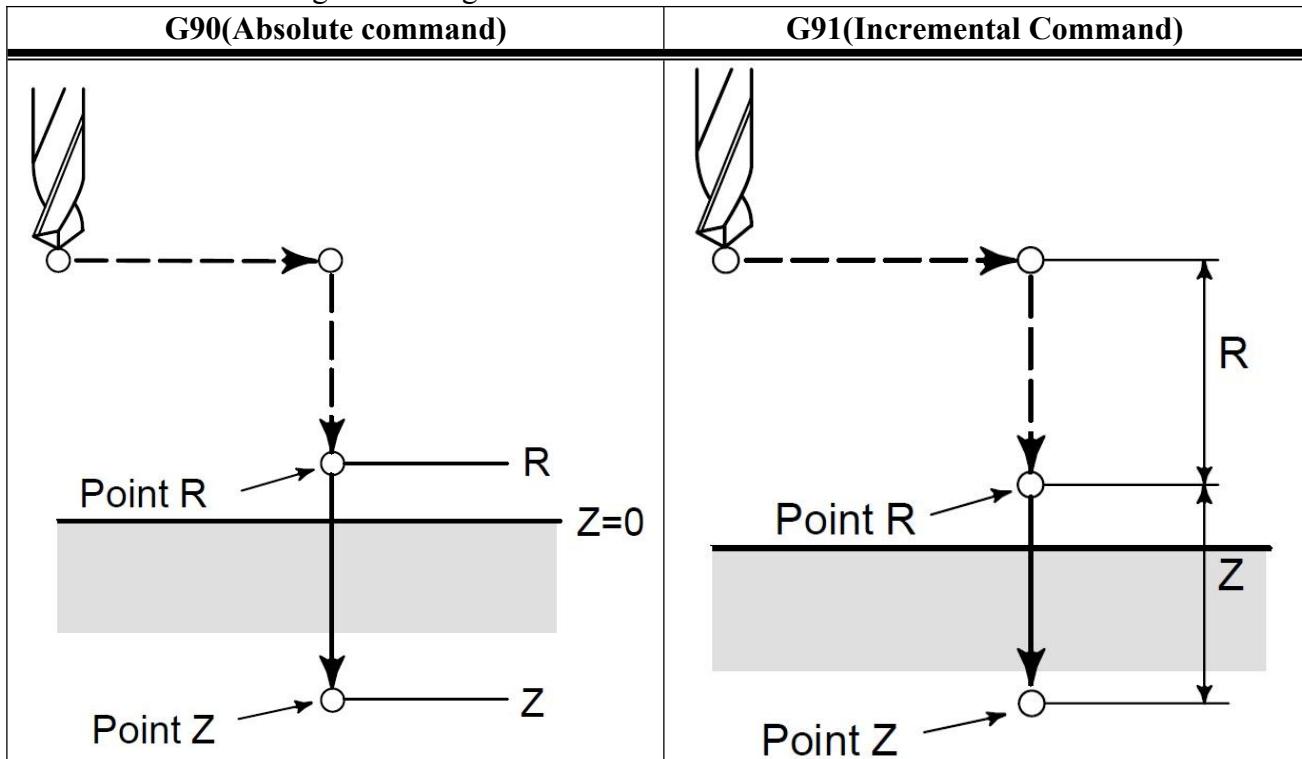
G code	Positioning Plane	Drilling axis
G17	Xp-Yp plane	Zp
G18	Zp-Xp plane	Yp
G19	Yp-Zp plane	Xp

Xp: X axis or an axis parallel to the X axis

Yp: Y axis or an axis parallel to the Y axis

ZP: Z axis or an axis parallel to the Z axis

Travel distance along the drilling axis varies for G90 and G91 as follows:



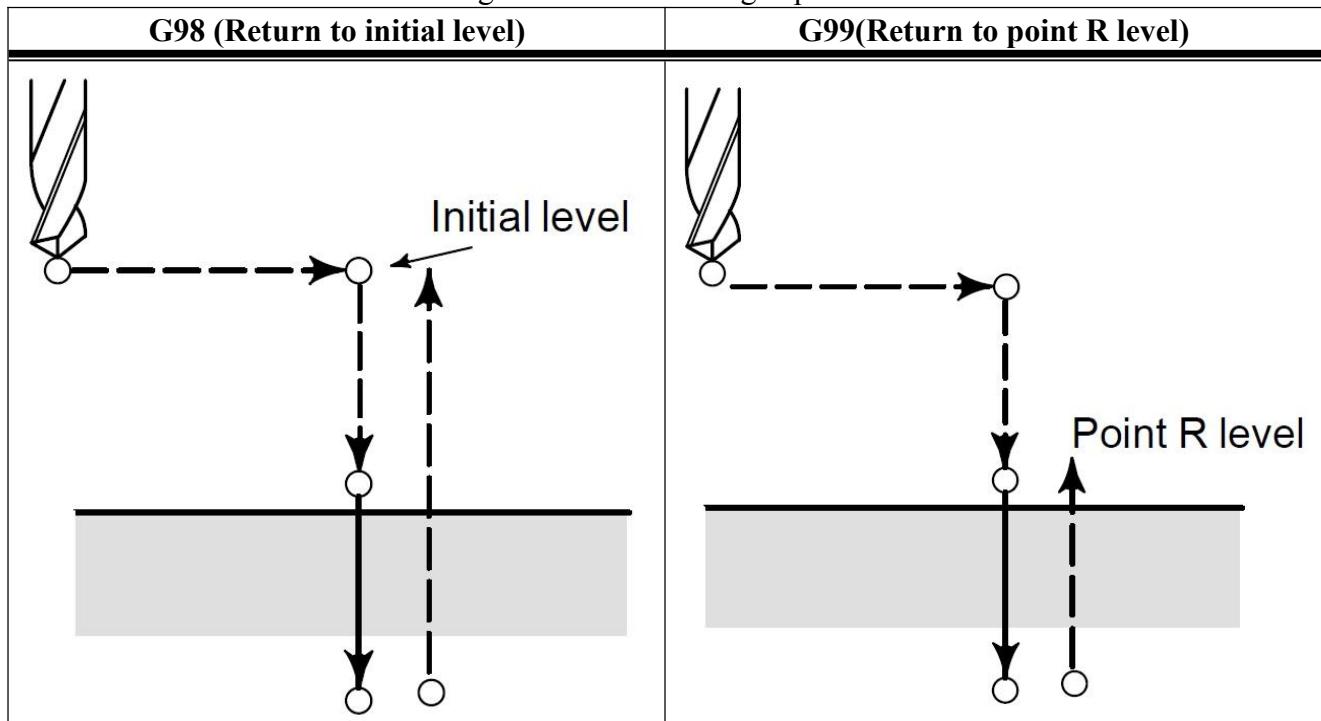
Drilling Mode: G73, G74 and G81 to G89 are modal G codes and remain in effect until canceled. When in effect, the current state is the drilling mode. Once drilling data is specified in the drilling mode, the data is retained until modified or canceled.

Specify all necessary drilling data at the beginning of canned cycles; when canned cycles are being performed, specify data modifications only.

Return point level G98/G99: When the tool reaches the bottom of a hole, the tool may be returned to point R or to the initial level. These operations are specified with G98 and G99. The following illustrates how the tool moves when G98 or G99 is specified. Generally, G99 is used for

the first drilling operation and G98 is used for the last drilling operation.

The initial level doesn't change even when drilling is performed in the G99 mode.



Repeat: To repeat drilling for equally-spaced holes, specify the number of repeats in L_.

K is effect only within the block where it is specified. The max value of L is 9999. Default=1.

To cancel a canned cycle, use G80 or a group 01 G code.

Group 01 G codes

G00: Positioning (Rapid traverse)

G01: Linear interpolation

G02: Circular interpolation or helical interpolation (CW)

G03: Counter-Circular interpolation or helical interpolation (CCW)

Symbol in figures: Subsequent sections explain the individual canned cycles. Figures in these explanations use the following symbols:

- Positioning (rapid traverse G00)
- Cutting feed (linear interpolation G01)
- Manual feed
- OSS (The spindle stops at a fixed rotation position)
- Shift (rapid traverse G00)
- P Dwell

3.28.1 High-speed Peck Drilling Cycle(G73)

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

Format: **G73 X_Y_Z_R_Q_F_L_ ;**

X_Y_: hole position data

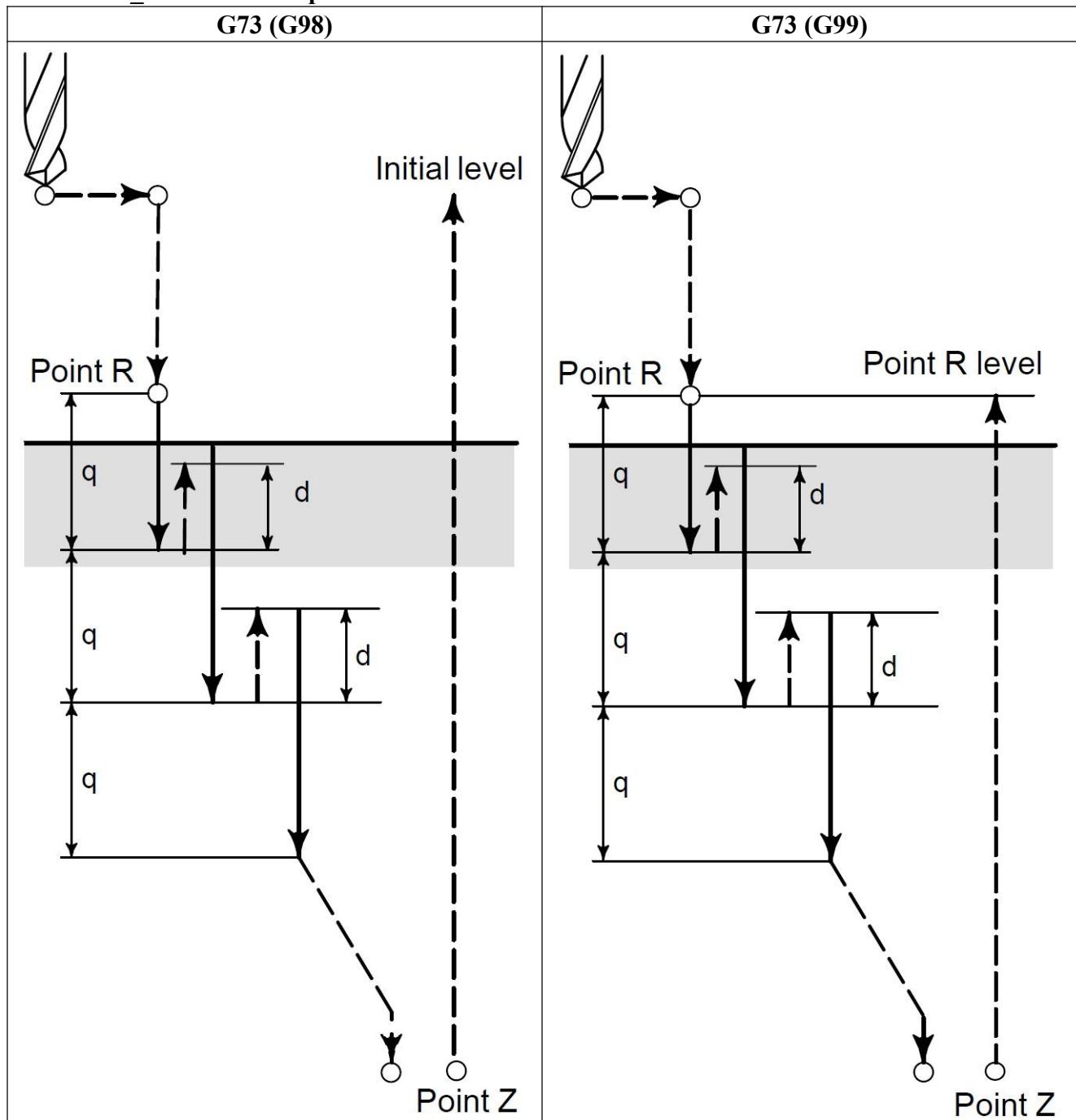
Z_: The distance from point R to bottom of the hole

R_: The distance from the initial level to point R level

Q_: Depth every time

F_: Cutting feedrate

L_: Number of repeats



The high-speed peck drilling cycle performs intermittent feeding along the Z-axis. When this cycle is used, chips can be removed from the hole easily, and a smaller value can be set for retraction. This allows drilling to be performed effectively.

P1 in User parameter sets the value of clearance(d).

The tool is retracted in rapid traverse.

Before specifying G73, rotate the spindle using a miscellaneous function (M code). When the G73 code and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When a tool length offset(G43 G44 or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Note: 1. In a block that doesn't contain R,Q , drilling isn't performed.Specify R and Q in blocs that perform drilling. If they are specified in a block that doesn't perform drilling, they cannot be stored as modal data.

2. Do not specify a G code of the 01 group(G00-G03) and G73 in a single block.Otherwise, G73 will be canceled.

3. In the canned cycle mode, tool offsets are ignored.

4. Before the drilling axis can be changed, the canned cycle must be canceled.

Example:

M3 S2000 ;	Cause the spindle to start rotating
G90 G99 G73 X300. Y-250. Z-150. R-100. Q15. F120. ;	Position, Drill 1st hole, then return to point R.
Y-550. ;	Position, Drill 2nd hole, then return to point R.
Y-750. ;	Position, Drill 3rd hole, then return to point R.
X1000. ;	Position, Drill 4th hole, then return to point R.
Y-550. ;	Position, Drill 5th hole, then return to point R.
G98 Y-750. ;	Position, Drill 6th hole, then return to the initial level.
G80 G28 G91 X0 Y0 Z0;	Return to the reference position.
M05 ;	Cause the spindle to stop rotating

3.28.2 Left-handed Peck Rigid Tapping Cycle (G74)

This cycle performs left-handed tapping. In the left-handed tapping cycle, when the bottom of the hole has been reached, the spindle rotates clockwise.

Tapping a deep hole in rigid tapping mode may be difficult due to chips sticking to the tool or increased cutting resistance. In such cases, the peck rigid tapping cycle is useful.

This cycle may be performed in standard mode or rigid tapping mode.

In standard mode, the spindle is rotated and stopped along with a movement along the tapping axis using miscellaneous functions M03 (Rotating the spindle clockwise), M04 (rotating the spindle counterclockwise), and M05 (Stopping the spindle) to perform tapping following with SP-Encoder.

In Rigid tapping mode, tapping is performed by controlling the spindle motor as if it were a servo motor and by interpolation between the tapping axis and spindle. When tapping is performed in rigid mode, the spindle rotates one turn every time a certain feed(thread lead) which takes place along the tapping axis. This operation doe not vary even during acceleration or deceleration.

Format: G74 X_ Y_ Z_ R_ P_ K_ Q_ L_ S_ ;

X_ Y_ : Hole position data

Z_ : The distance from point R to the bottom of the hole

R_ : The distance from initial level to point R level

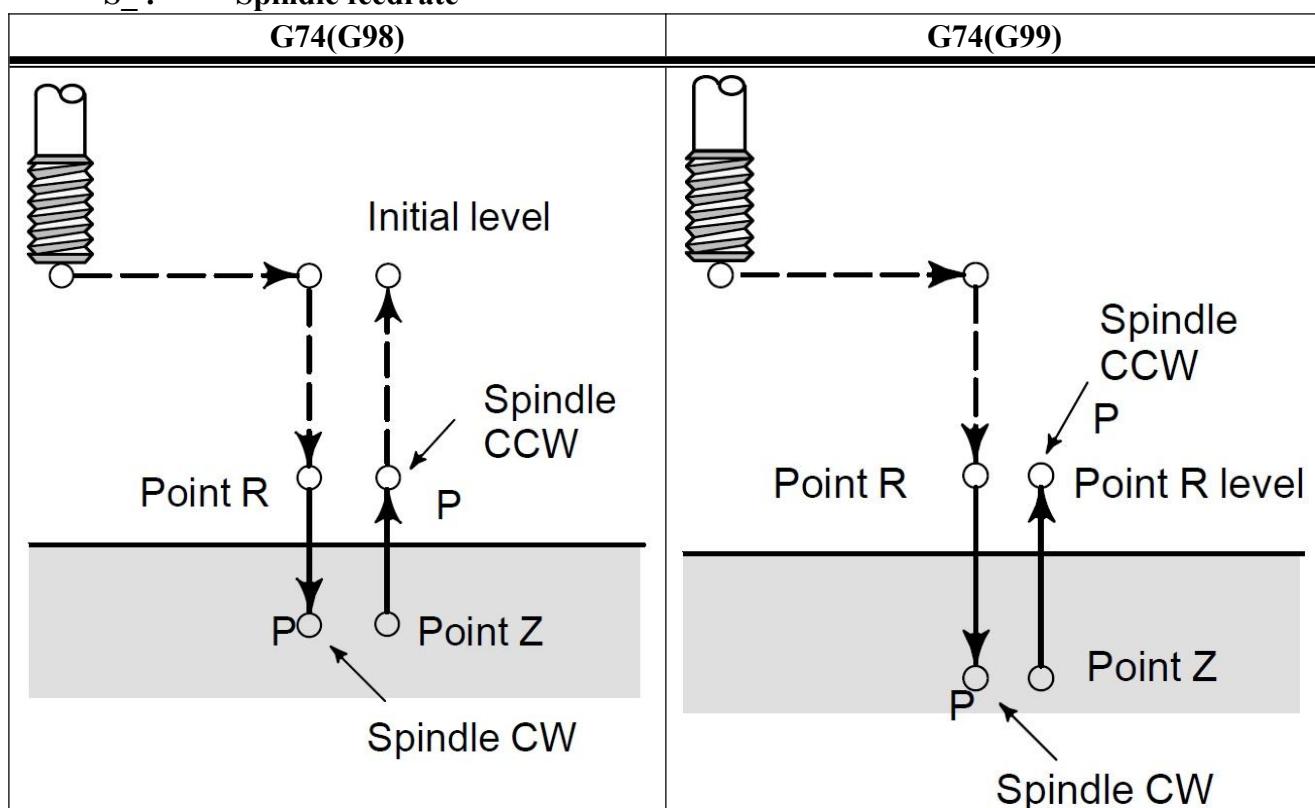
P_ : Dwell time unit:s

K_ : Pitch

Q_ : Depth of cut for each cutting feed

L_ : Number of repeats (if required)

S_ : Spindle feedrate



Tapping is performed by turning the spindle counterclockwise. When the bottom of the hole

has been reached, the spindle is rotated clockwise for retraction. This creates a reverse thread.

Feedrate overrides are ignored during left-handed tapping. A feed hold doesn't stop the machine until the return operation is completed.

Before specifying G74, use a miscellaneous function(M code) to rotate the spindle counterclockwise.

When the G74 command and an M code are specified in the same block, the M code is executed at the time of the first position operation. The system then proceeds to the next drilling operation.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Note: 1. Before the drilling axis can be changed, the canned cycle must be canceled.

2. In a block that does not contain R, K, drilling is not performed.

3. Specify P in blocs that perform drilling. If they are specified in a block that doesn't perform drilling, they cannot be stored as modal data.

4. Do not specify a G code of the 01 group(G00-G03) and G73 in a single block. Otherwise, G73 will be canceled.

5. In the canned cycle mode, tool offsets are ignored.

6. During rigid tapping, Spindle-feedrate override and Feed-feedrate override don't work.

Parameters Sets for Rigid Tapping:

1, Standard tapping, following Spindle_Encoder mode, P411=2 in Axis parameter, CNC system must be configured with SP-encoder

Special Note: When the transmission ratio between Spindle and Encoder is not 1:1, it must be configured with SZGH Transfer Board & modify P412 & P413 in Axis parameter;

412, Number of spindle teeth

413, Number of encoder teeth

2, Rigid tapping, interpolation between Spindle & Drilling axis, P405=5, P410=94, P411=3 in Axis parameter..

M75: Shift control mode of spindle to Position Control Mode(Pulse+Direction). After rigid tapping, M74, cancel position control mode of spindle.

Rigid mode eliminates the need to use a floating tap required in the standard tapping mode, thus allowing faster and more precise tapping.

Example:

N1 M4 S100 ; Cause the spindle to start rotating CCW.

N2 G90 G99 G74 X300. Y-250. Z-150. R-100. K5 ;

Position, tapping hole 1, then return to point R.

N3 Y-550. ; Position, tapping hole 2, then return to point R.

N4 Y-750. ; Position, tapping hole 3, then return to point R.

N5 X1000. ; Position, tapping hole 4, then return to point R.

N6 Y-750. ; Position, tapping hole 5, then return to point R.

N7 G98 Y-550. K5; Position, tapping hole 6, then return to the initial level

N8 G80 G28 G91 X0 Y0 Z0; Return to the reference position.

N9 M05; Cause the spindle to stop rotating.

3.28.3 Fine Boring Cycle (G76)

The fine boring cycle bores a hole precisely. When the bottom of the hole has been reached, the spindle stops, and the tool is moved always from the machined surface of the workpiece and retracted with the direction that set by P3 in User parameter.

Format: **G76 X_ Y_ Z_ R_ Q_ P_ F_ L_ ;**

X_ Y_ : Hole position data.

Z_ : The distance from point R to the bottom of the hole.

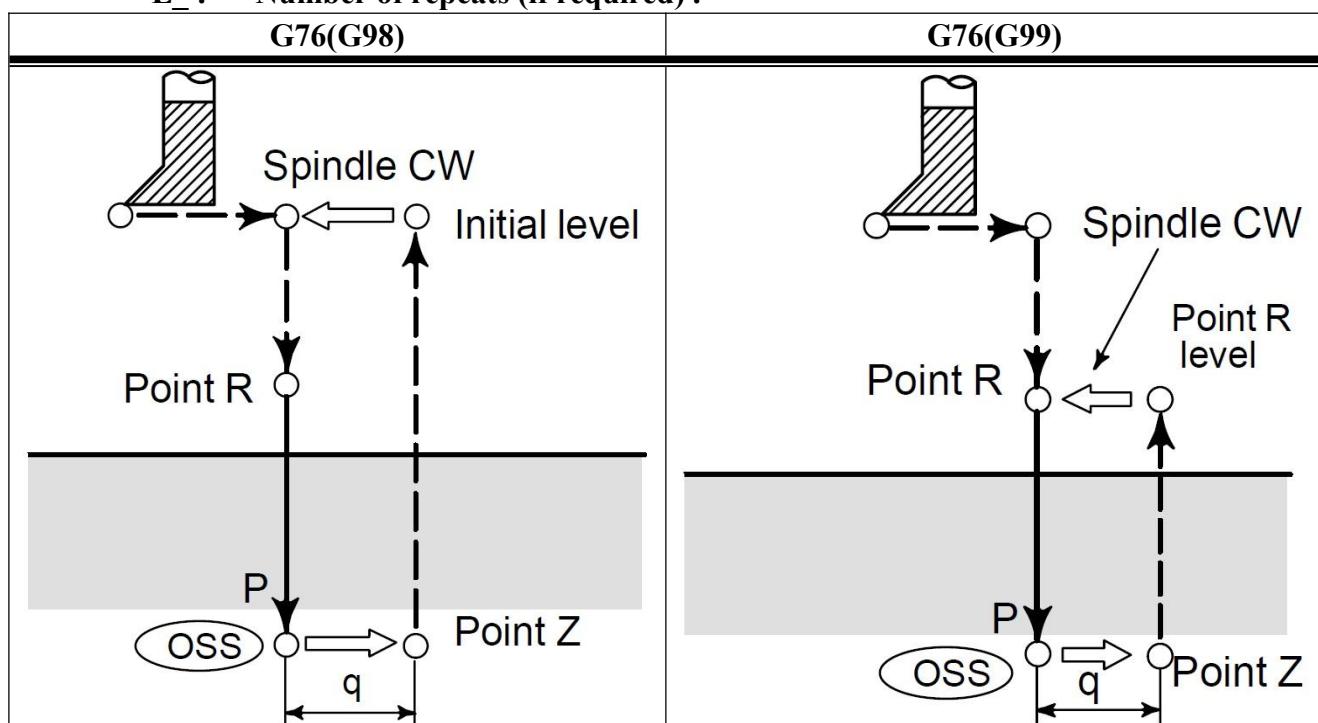
R_ : The distance from the initial level to point R level.

Q_ : Shift amount at the bottom of a hole.

P_ : Dwell time at the bottom of a hole. Unit:s.

F_ : Cutting feedrate.

L_ : Number of repeats (if required) .



Warning: Q(shift at the bottom of a hole) is a modal value retained within canned cycles. It must be specified carefully because it is also used as the depth of cut for G73 and G83.

Direction of shift after oriented spindle stop is set by P3 in User parameter.

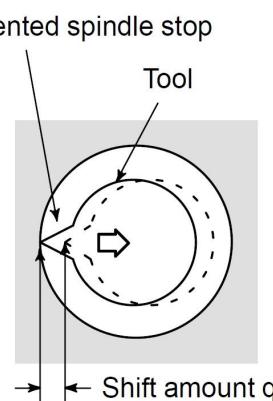


Fig3.22.3 Shift amount & direction

When the bottom of the hole has been reached, the spindle is stopped at the fixed rotation position, and the tool is moved in the direction opposite to the tool tip and retracted. This ensures that the machined surface is not damaged and enable precise and efficient boring to be performed.

Before specifying G76, use a miscellaneous function (M code) to rotate the spindle. When The G76 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next operation.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R..

Note: 1. Before the drilling axis can be changed, the canned cycle must be canceled.

2. In a block that does not contain R, Q, boring is not performed.

3. Be sure to specify a positive value in Q. If Q is specified with a negative value, the sign is ignored.

Set the direction of shift is set by P3 of User parameter. Specify P & Q in a block that perform as boring. If they are specified in a block that doesn't perform boring, they cannot be stored as modal data.

4. Do not specify a G code of the 01 group(G00-G03) and G76 in a single block. Otherwise, G76 will be canceled and alarm.

5. In the canned cycle mode, tool offsets are ignored.

Warning: The spindle system must support orientation function when use G76. SZGH CNC system output M61(Pin19_CN10 plug) for orientation, M22(Pin5_CN10 plug) detects orientation end.

Example:

N10 M3 S500 ; Cause the spindle to start rotating.

N20 G90 G99 G75 X300 Y-250 Z-150 R-100 Q5 P1 F120;

Position, bore hole 1, then return to point R. Orient at the bottom of the hole, then shift by 5mm. Stop at the bottom of the hole for 1s.

N30 Y-350 ; Position, drill hole 2, then return to point R.

N40 Y-550 ; Position, drill hole 2, then return to point R.

N50 X1000 ; Position, drill hole 2, then return to point R.

N60 Y-550 ; Position, drill hole 2, then return to point R.

N70 G98 Y-750 ; Position, drill hole 2, then return to the initial level.

N80 G80 G28 G91 X0 Y0 Z0 ; Return to the reference position

N90 M5 ; Cause the spindle to stop rotating.

3.28.4 Canned Cycle Cancel (G80)

G80 cancels canned cycle.

Format: **G80** ;

All canned cycle are canceled to perform normal operation. Point R and point Z are cleared. This means that R=0 and Z=0 in incremental mode. Other drilling data is also canceled(cleared).

3.28.5 Drilling cycle, Spot Drilling (G81)

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

Format: **G81 X_ Y_ Z_ R_ F_ L_ ;**

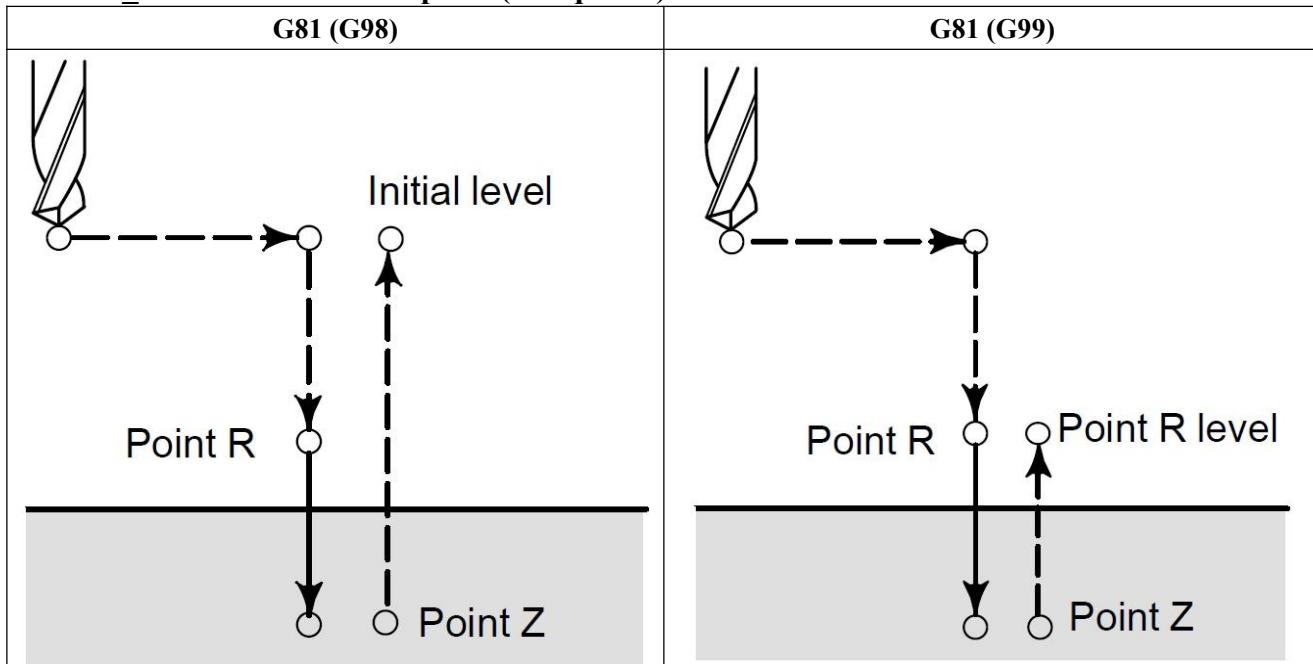
X_ Y_ : Hole position data

Z_ : The distance from point R to bottom of the hole

R_ : The distance from initial level to point R level

F_ : Cutting feedrate

L_ : Number of repeats (if required)



After positioning along the X axis and Y axis, rapid traverse is performed to point R. Drilling is performed from point R to point Z. Then the tool is retracted in rapid traverse.

Before specifying G81, use miscellaneous function (M code) to rotate the spindle. When the G81 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When a tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset is applied at the time of position to point R.

Note: In the canned cycle mode, tool offsets are ignored.

Example:

M3 S2000 ; Cause the spindle to start rotating

G90 G99 G81 X300 Y-250 Z-150 R-100 F120 ; Position, drill hole 1, then return to point R.

Y-550 ; Position, drill hole 2, then return to point R.

Y-750 ; Position, drill hole 3, then return to point R.

X1000 ; Position, drill hole 4, then return to point R.

G98 Y-750 ; Position, drill hole 5, then return to the initial level.

G80 G28 G91 X0 Y0 Z0 ; Return to the reference position

M5 ; Cause the spindle to stop rotating

3.28.6 Drilling Cycle Counter Boring Cycle (G82)

This cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole. At the bottom, a dwell is performed, then the tool is retracted in rapid traverse. This cycle is used to drill holes more accurately with respect to depth.

Format: **G82 X_Y_Z_R_P_F_L ;**

X_Y : Hole position data

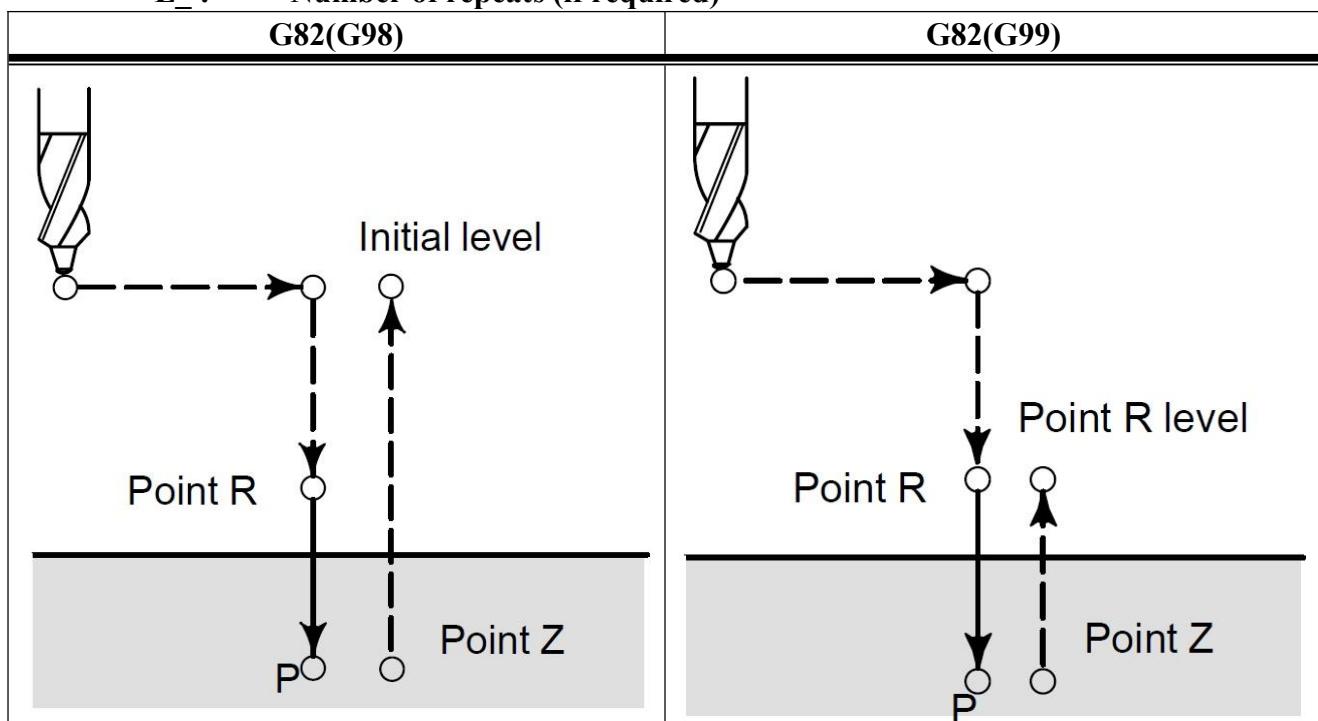
Z : The distance from R point to bottom of the hole

R : The distance from initial level to point R level

P : Dwell time at the bottom of a hole. Unit: s.

F : Cutting feed rate

L : Number of repeats (if required)



After positioning along the X and Y axis, rapid traverse is performed to point R. Then drilling is performed from point R to point Z. When the bottom of the hole has been reached, a dwell is performed. Then the tool is retracted in rapid traverse.

Before specifying G82, use miscellaneous function (M code) to rotate the spindle. When the G82 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When a tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset is applied at the time of position to point R. In the canned cycle mode, tool offsets are ignored.

Example: M3 S2000 ; Cause the spindle to start rotating

G90 G99 G82 X300. Y-250. Z-150. R-100. P1 F120. ;

Position, drill hole 1, and dwell for 1s at the bottom of the hole, then return to point R.

Y-550. ; Position, drill hole 2, then return to point R.

G98 Y-750. ; Position, drilling hole 3, then return to initial level.

G0 X0 Y0 Z0 ; Return to the zero point.

M05 ; Cause the spindle to stop rotating.

3.28.7 Peck Drilling Cycle (G83)

This cycle performs peck drilling. It performs intermittent cutting feed to the bottom of a hole while removing shavings from the hole.

Format: **G83 X₁ Y₁ Z₁ R₁ Q₁ F₁ L₁ ;**

X₁ Y₁ : Hole position data

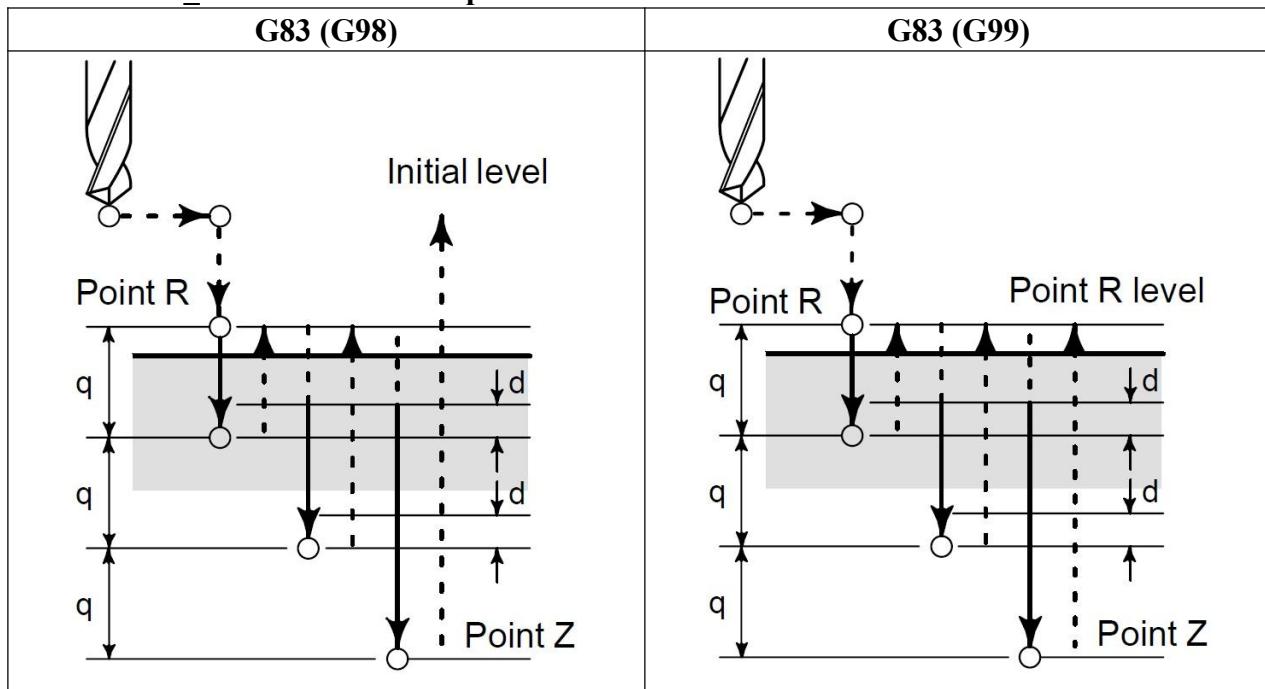
Z₁ : The distance from point R to the bottom of the hole

R₁ : The distance from initial level to point R level

Q₁ : Depth of cut for each cutting feed

F₁ : Cutting speed

L₁ : Number of repeats



Q represents the depth of each cutting feed. It must be specified in an incremental value. In the second and subsequent cutting feed, rapid traverse is performed up to a d point just before where the last drilling ended, and cutting feed is performed again. d is set by P2 in User parameter.

Be sure to specify a positive value in Q. Negative values are ignored.

Before specifying G83, use miscellaneous function (M code) to rotate the spindle. When the G83 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When a tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset is applied at the time of position to point R. In the canned cycle mode, tool offsets are ignored.

Example: M3 S2000 ; Cause the spindle to start rotating.

G90 G99 G83 X300. Y-250. Z-150. R-100. Q15. F120. ;

Position, drill hole 1, then return to point R.

Y-550.; Position, drill hole 1, then return to point R.

G98 Y-750.; Position, drill hole 1, then return to the initial level.

G80; Cancel canned cycle

M05; Cause the spindle to stop rotating.

3.28.8 Right-handed Peck Rigid Tapping Cycle (G84)

Tapping a deep hole in rigid tapping mode may be difficult due to chips sticking to the tool or increased cutting resistance. In such cases, the peck rigid tapping cycle is useful.

This cycle may be performed in standard mode or rigid tapping mode.

In standard mode, the spindle is rotated and stopped along with a movement along the tapping axis using miscellaneous functions M03 (Rotating the spindle clockwise), M04 (rotating the spindle counterclockwise), and M05 (Stopping the spindle) to perform tapping following with SP-Encoder.

In Rigid tapping mode, tapping is performed by controlling the spindle motor as if it were a servo motor and by interpolation between the tapping axis and spindle. When tapping is performed in rigid mode, the spindle rotates one turn every time a certain feed(thread lead) which takes place along the tapping axis. This operation doe not vary even during acceleration or deceleration.

In this cycle, cutting is performed several times until the bottom of the hole is reached.

Format: **G84 X_ Y_ Z_ R_ P_ Q_ K_ L_ S_ ;**

X_ Y_ : Hole position data

Z_ : The distance from point R to the bottom of the hole

R_ : The distance from the initial level to point R level

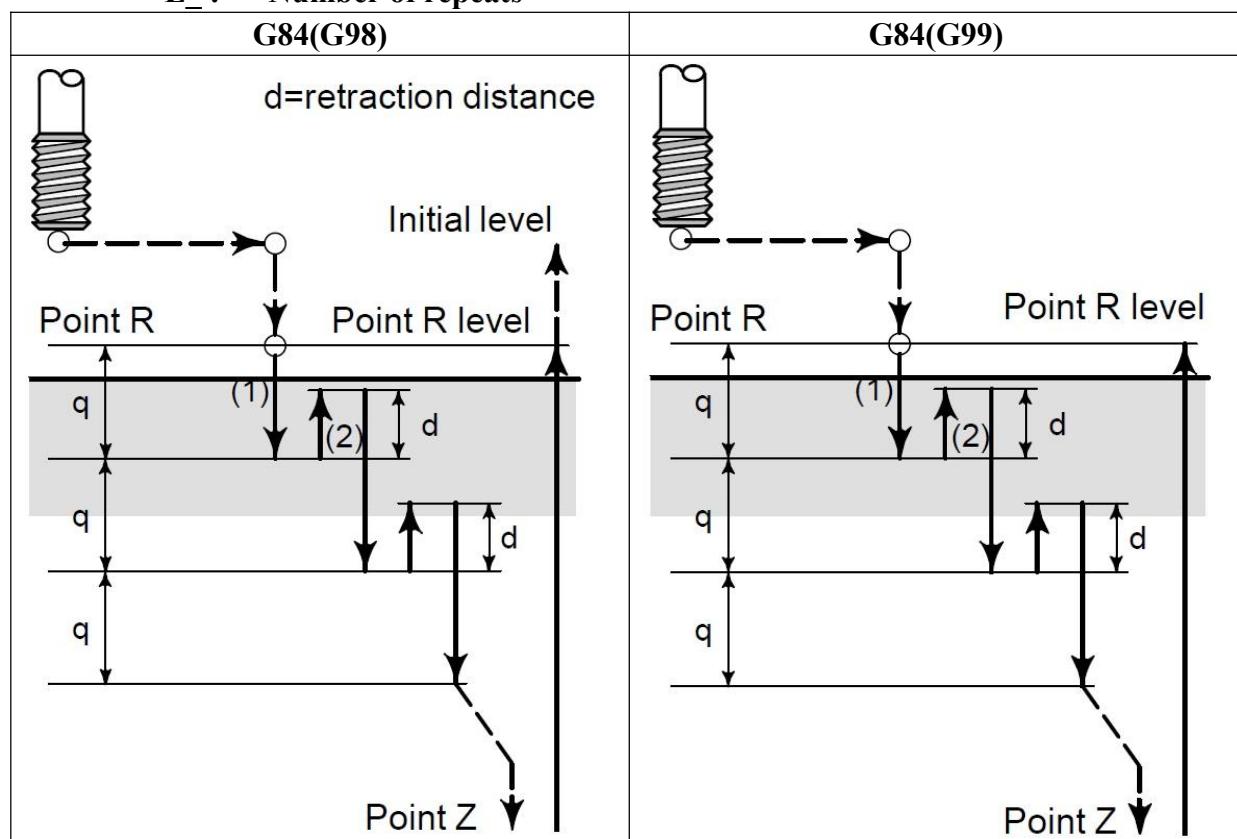
P_ : Dwell time at the bottom of the hole and at point R. Unit s.

Q_ : Depth of cut for each cutting feed

K_ : Pitch of Screw

S_ : Speed of Spindle

L_ : Number of repeats



After positioning analog X and Y axis,rapid traverse is performed to point R. From point R,

cutting is performed with depth Q (depth of each cutting feed), then the tool is retracted by distance d. When point Z has been reached, after dwell time P, the spindle is stopped, then rotated in the counter-clockwise direction(CCW) for retraction.

Set the retraction distance, d, P7 in User parameter

Set the retraction mode, P6 in User parameter. 0: Retract with d in rapid traverse, 1: Retract to point R level.

During rigid tapping, Speed of feeding axes is set by SP_Speed(S) & threading pitch .

Note: 1. Do not specify a G code of the 01 group(G00-G03) and G84 in a single block. Otherwise, G84 will be canceled and alarm.

2. In a block that does not contain K , rigid tapping is not performed and cannot pass compile.

3. In the canned cycle mode, tool offsets are ignored.

4. Before the drilling axis can be changed, the canned cycle must be canceled. If the drilling axis is changed in rigid mode, CNC system will alarm.

5. During rigid tapping, Spindle-feedrate override and Feed-feedrate override don't work.

Parameters Sets for Rigid Tapping:

1, Standard tapping, following Spindle_Encoder mode, P411=2 in Axis parameter, CNC system must be configured with SP-encoder

Special Note: When the transmission ratio between Spindle and Encoder is not 1:1, it must be configured with SZGH Transfer Board & modify P412 & P413 in Axis parameter;

412, Number of spindle teeth

413, Number of encoder teeth

2, Rigid tapping, interpolation between Spindle & Drilling axis, P405=5, P410=94, P411=3 in Axis parameter..

M75: Shift control mode of spindle to Position Control Mode(Pulse+Direction). After rigid tapping, M74, cancel position control mode of spindle.

Rigid mode eliminates the need to use a floating tap required in the standard tapping mode, thus allowing faster and more precise tapping.

Example:

M3 S100 ; Cause the spindle to start rotating

G90 G99 G84 X300 Y-250 Z-150 R-120 P300 K5 S100 ;

Position, tapping hole 1, then return to point R

Y-550. K5; Position, tapping hole 2, then return to point R

G98 Y-750. K5; Position, tapping hole 1, then return to the initial level

G80 ; Cancel rigid tapping cycle

M30; End of program

3.28.9 Boring Cycle (G85)

This cycle is used to bore a hole.

Format: **G85 X_ Y_ Z_ R_ F_ L_ ;**

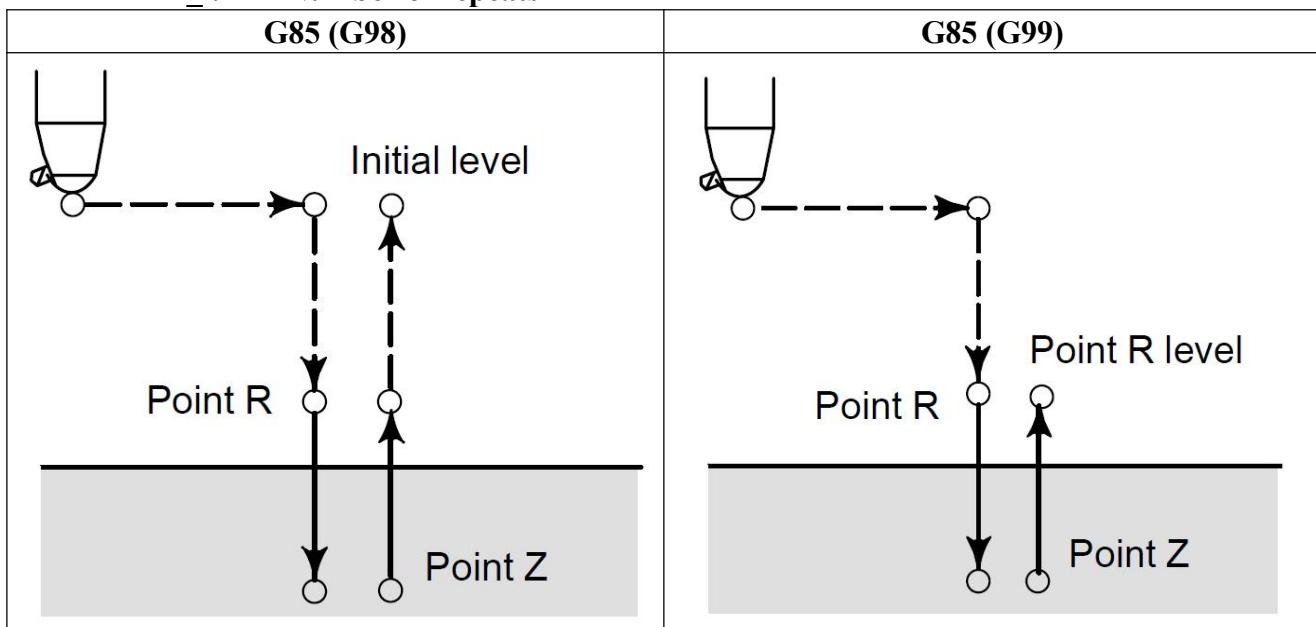
X_ Y_ : Hole position data

Z_ : The distance from point R to the bottom of the hole

R_ : The distance from the initial level to point R level

F_ : Cutting feedrate

L_ : Number of repeats



After positioning along the X and Y axis, rapid traverse is performed to point R. Drilling is performed from point R to point Z. When point Z has been reached, cutting feed is performed to return to point R.

Before specifying G85, use miscellaneous function (M code) to rotate the spindle. When the G85 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next boring operation.

When a tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset is applied at the time of position to point R. In the canned cycle mode, tool offsets are ignored.

Note: 1. Before the drilling axis can be changed, the canned cycle must be canceled.

2. Do not specify a G code of the 01 group(G00-G03) and G85 in a single block. Otherwise, G85 will be canceled and alarm.

Example: M3 S100 ; Cause the spindle to start rotating

G90 G99 G85 X300. Y-250. Z-150. R-120. F120.

Position, drill hole 1, then return to point R

Y-550 ; Position, drill hole 2, then return to point R

Y-750; Position, drill hole 3, then return to point R

X1000 ; Position, drill hole 4, then return to point R

G98 Y-750 ; Position, drill hole 5, then return to initial level

G80 ; Cancel canned cycle

M05 ; Cause the spindle to stop rotating.

3.28.10 Boring Cycle (G86)

This cycle is used to bore a hole.

Format: **G86 X_ Y_ Z_ R_ F_ L_ ;**

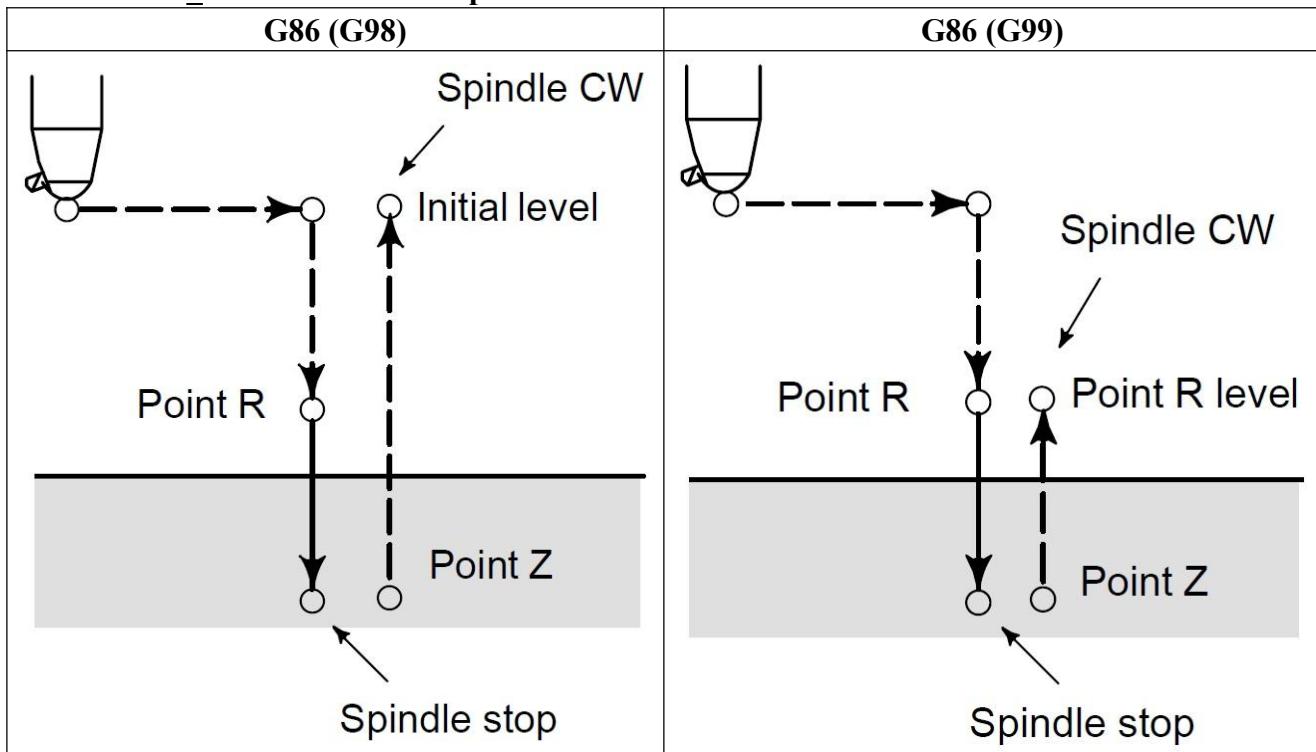
X_ Y_ : Hole position data

Z_ : The distance from point R to the bottom of the hole

R_ : The distance from the initial level to point R level

F_ : Cutting feedrate

L_ : Number of repeats



After positioning along the X and Y axis, rapid traverse is performed to point R. Drilling is performed from point R to point Z. When the spindle is stopped at the bottom of the hole, the tool is retracted in rapid traverse.

Before specifying G86, use miscellaneous function (M code) to rotate the spindle. When the G86 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next boring operation.

When a tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset is applied at the time of position to point R. In the canned cycle mode, tool offsets are ignored.

Note: 1. Before the drilling axis can be changed, the canned cycle must be canceled.

2. Do not specify a G code of the 01 group(G00-G03) and G86 in a single block. Otherwise, G86 will be canceled and alarm.

Example: M3 S2000 ; Cause the spindle to start rotating

G90 G99 G85 X300. Y-250. Z-150. R-120. F120. Position, drill hole 1, then return to point R
Y-550 ; Position, drill hole 2, then return to point R

G98 Y-750 ; Position, drill hole 3, then return to initial level

G80 ; Cancel canned cycle

M05 ; Cause the spindle to stop rotating.

3.28.11 Back Boring Cycle (G87)

This cycle performs accurate boring. When the bottom of the hole has been reached, the spindle stops, and the tool is moved always from the machined surface of the workpiece and retracted with the direction that set by P4 in User parameter.

Format: **G87 X_ Y_ Z_ R_ Q_ P_ F_ L_ ;**

X_ Y_ : Hole position data

Z_ : The distance from point R to the bottom of the hole

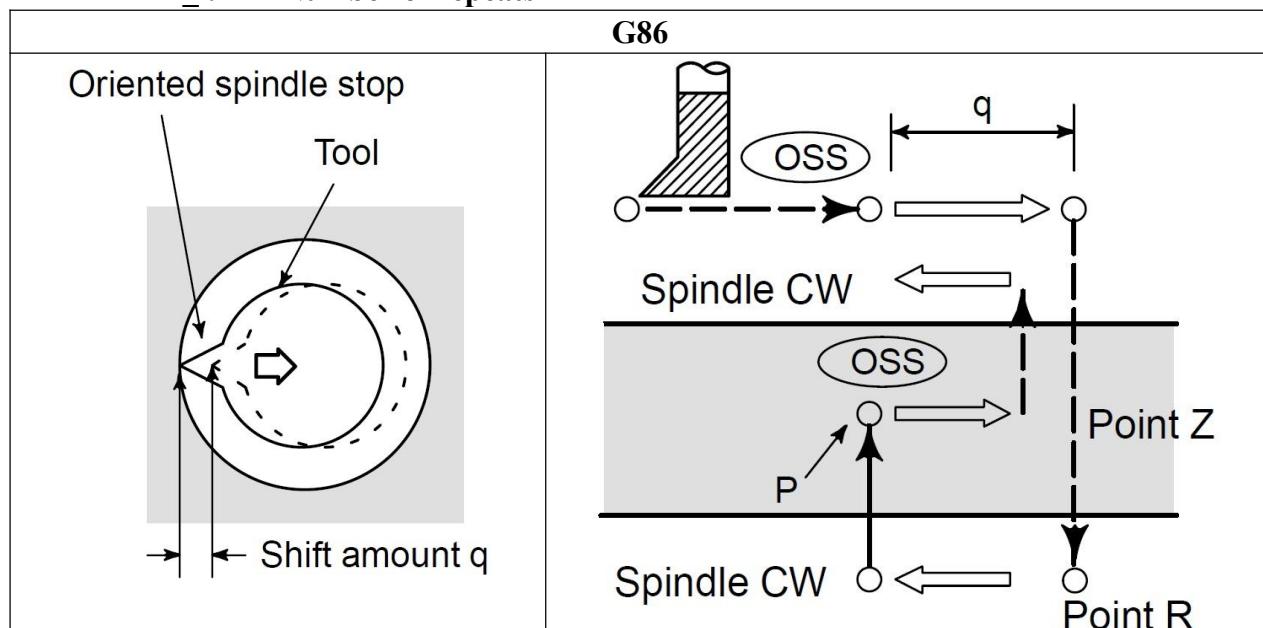
R_ : The distance from the initial level to point R(the bottom of the hole) level

Q_ : Tool shift amount

P_ : Dwell time

F_ : Cutting feedrate

L_ : Number of repeats



Warning: *Q(shift at the bottom of a hole) is a modal value retained within canned cycles. It must be specified carefully because it is also used as the depth of cut for G73 and G83.*

After positioning along the X and Y axis, the spindle is stopped at the fixed rotation position (Spindle orientation). The tool is moved in the direction (this direction is set by P4 & P5 in User parameter), opposite to the tool tip, positioning (rapid traverse) is performed to the bottom of the hole (point R). Then the tool is shifted in the direction of the tool tip and the spindle is rotated clockwise. Boring is performed in the positive direction along the Z-axis until point Z is reached. At point Z, the spindle is stopped at the fixed rotation position again (Spindle orientation), the tool is shifted in the direction opposite to the tool tip, then the tool is returned to the initial level. Then the tool is shifted in the direction of the tool tip and the spindle is rotated clockwise to proceed to the next block operation.

Before specifying G87, use miscellaneous function (M code) to rotate the spindle. When the G87 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next boring operation.

When a tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset is applied at the time of position to point R.

Note:

1. Before the drilling axis can be changed, the canned cycle must be canceled.
2. In a block that does not contain R, Q, boring is not performed.
3. Be sure to specify a positive value in Q. If Q is specified with a negative value, the sign is ignored.

Set the direction of shift is set by P4 & P5 of User parameter. Specify P & Q in a block that perform as boring. If they are specified in a block that doesn't perform boring, they cannot be stored as modal data.

4. Do not specify a G code of the 01 group(G00-G03) and G87 in a single block. Otherwise, G87 will be canceled and alarm.

5. In the canned cycle mode, tool offsets are ignored.

Warning: The spindle system must support orientation function when use G87. SZGH CNC system output M61(Pin19_CN10 plug) for orientation, M22(Pin5_CN10 plug) detects orientation end.

Example:

N10 M3 S500 ; Cause the spindle to start rotating.

N20 G90 G87 X300 Y-250 Z-150 R-100 Q5 P1 F120;

Position, bore hole 1. Orient at the initial level, then shift by 5mm. Stop at point Z for 1s.

N30 Y-350 ; Position, drill hole 2.

N40 Y-550 ; Position, drill hole 3.

N50 X1000 ; Position, drill hole 4.

N60 Y-550 ; Position, drill hole 5.

N70 G98 Y-750 ; Position, drill hole 6.

N80 G80 G28 G91 X0 Y0 Z0 ; Return to the reference position

N90 M5 ; Cause the spindle to stop rotating.

3.28.12 Boring Cycle (G89)

This cycle is used to bore a hole.

Format: **G89 X_ Y_ Z_ R_ P_ F_ L_ ;**

X_ Y_ : Hole position data

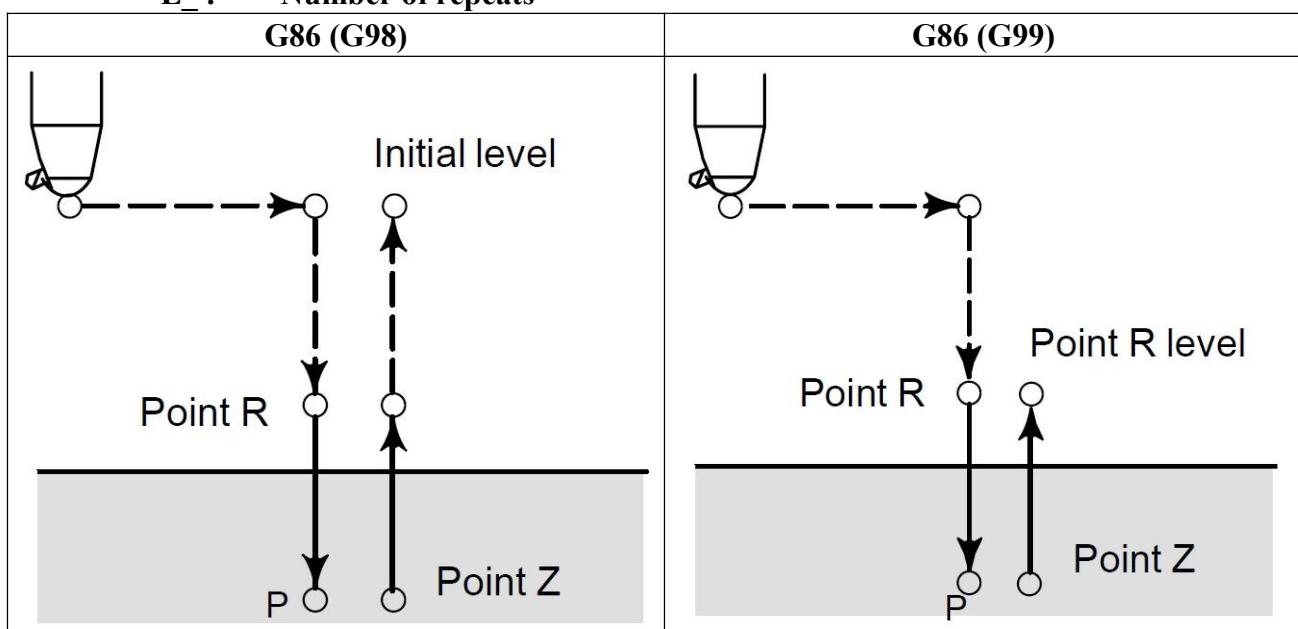
Z_ : The distance from point R to the bottom of the hole

R_ : The distance from the initial level to point R level

P_ : Dwell time at the bottom of a hole

F_ : Cutting feedrate

L_ : Number of repeats



This cycle is almost same to G85. The difference is that this cycle performs a dwell at the bottom of the hole.

Before specifying G89, use miscellaneous function (M code) to rotate the spindle. When the G89 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next boring operation.

When a tool length offset (G43, G44 or G49) is specified in the canned cycle, the offset is applied at the time of position to point R. In the canned cycle mode, tool offsets are ignored.

Note: 1. Before the drilling axis can be changed, the canned cycle must be canceled.

2. Do not specify a G code of the 01 group(G00-G03) and G86 in a single block. Otherwise, G86 will be canceled and alarm.

Example: M3 S100 ; Cause the spindle to start rotating

G90 G99 G89 X300. Y-250. Z-150. R-120. P1 F120. ;

Position, drill hole 1, then return to point R

Then stop at the bottom of the hole for 1s.

Y-550 ;

Position, drill hole 2, then return to point R

X1000 ;

Position, drill hole 3, then return to point R.

G98 Y-750 ;

Position, drill hole 5, then return to initial level.

G80 G28 G91 X0 Y0 Z0 ; Return to the reference position

M05 ;

Cause the spindle to stop rotating.

3.28.13 Example of Canned Cycle

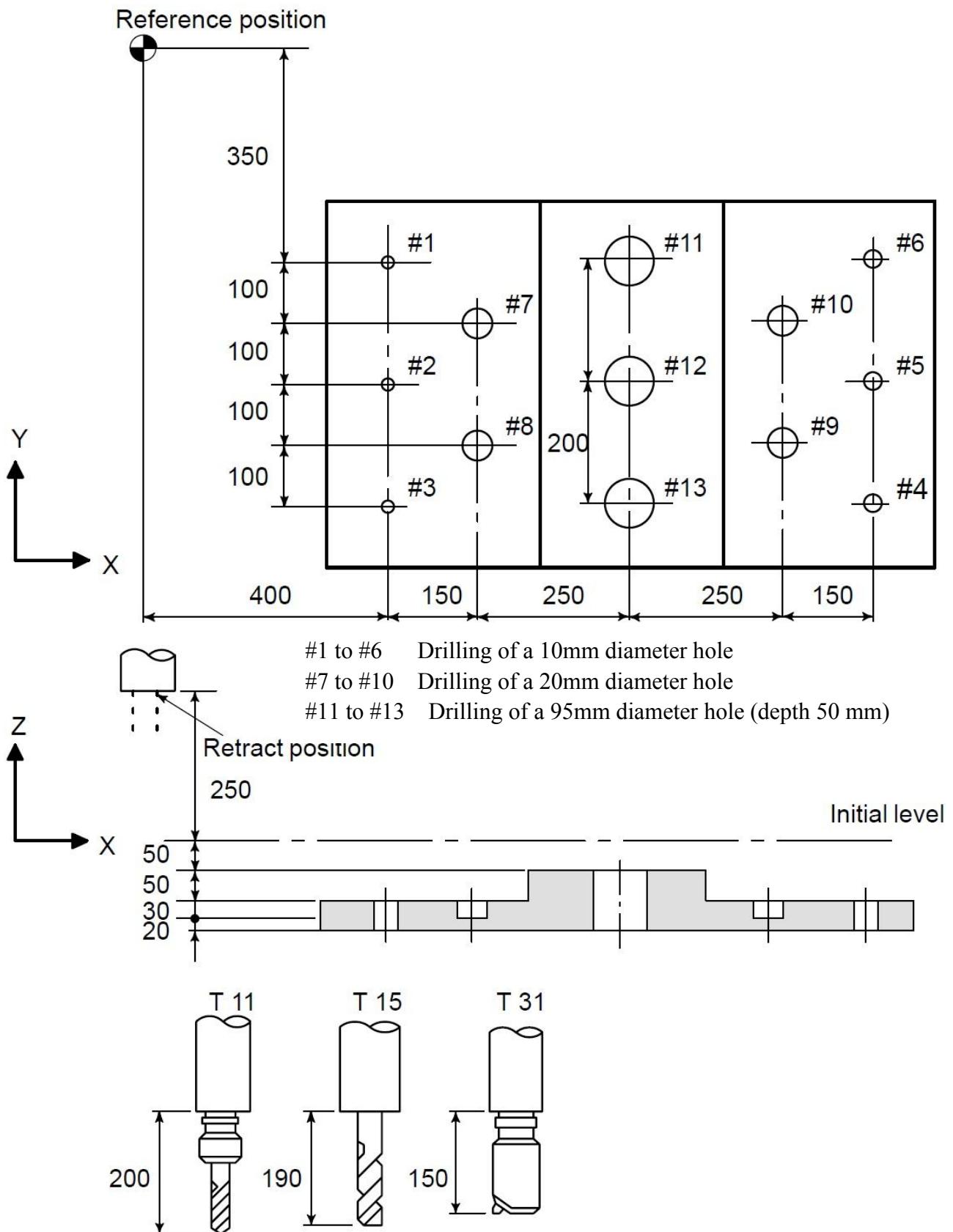


Fig3.22.13 Example of Canned Cycle

Parameter Set: In Redeem, Length offset value: +200.0 is set in No.11 ; +190 is set in No.15 ; and +150 is set in No.31.

Program Example:

N1	G54 X0 Y0 Z0 ;	Coordinate setting at reference position
N2	G90 G00 Z250.0 T11;	Tool change
N3	G43 Z0 H11 ;	Initial level, tool length offset
N4	S30 M3 ;	Start spindle CW
N5	G99 G81 X400.0 Y-350.0 Z-153.0 R-97.0 F120 ;	Positioning, then drilling hole #1
N6	Y-550.0 ;	Positioning, then drilling hole #2, return to point R level
N7	G98 Y-750.0 ;	Positioning, then drilling hole #3, return to initial level
N8	G99 X1200.0 ;	Positioning, then drilling hole #4, return to point R level
N9	Y-550.0 ;	Positioning, then drilling hole #5, return to point R level
N10	G98 Y-350.0 ;	Positioning, then drilling hole #6, return to initial level
N11	G00 X0 Y0 M5 ;	Positioning, then drilling hole #7, return to point R level
N12	G49 Z250.0 T15 ;	Tool length offset cancel, tool change
N13	G43 Z0 H15 ;	Initial level, tool length offset
N14	S20 M3 ;	Start spindle CW
N15	G99 G82 X550.0 Y-450.0 Z-130.0 R-97.0 P1 F50 ;	Positioning, then drilling hole #7, return to point R level
N16	G98 Y-650.0 ;	Positioning, then drilling hole #8, return to initial level
N17	G99 X1050.0 ;	Positioning, then drilling hole #9, return to point R level
N18	G98 Y-450.0 ;	Positioning, then drilling hole #10, return to initial level
N19	G00 X0 Y0 M5 ;	Return to reference position & Stop spindle
N20	G49 Z250.0 T31 ;	Tool length offset cancel, tool change
N21	G43 Z0 H31 ;	Initial level, tool length offset
N22	S10 M3 ;	Start spindle CW
N23	G85 G99 X800.0 Y-350.0 Z-153.0 R47.0 F150 ;	Positioning, then drilling hole #11, return to point R level
N24	G91 Y-200.0 ;	Positioning, then drilling hole #12, return to point R level
N25	Y-200.0 ;	Positioning, then drilling hole #13, return to point R level
N26	G90 G28 X0 Y0 M5 ;	Return to reference position & Stop spindle
N27	G49 Z0 G80 ;	Tool length offset cancel
N28	M02 ;	Program Stop

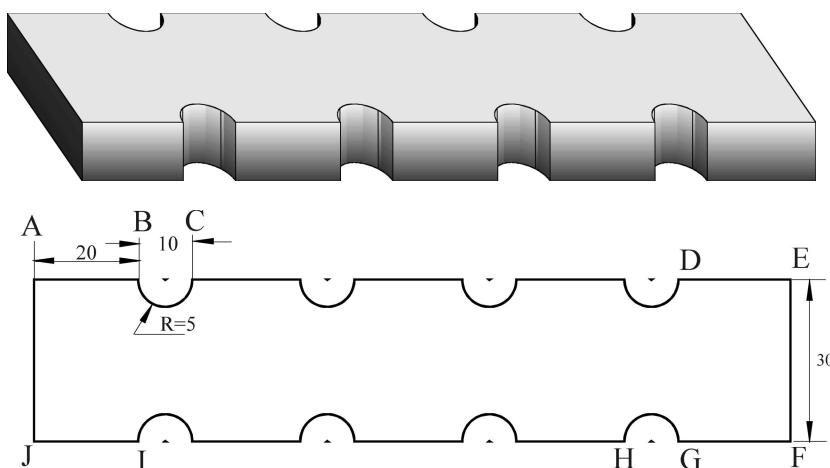
3.29 Block Cycle (G22,G800)

G22 is a program loop instruction, G800 is the end of the cycle instruction. Both must be paired for parts machining process requires repeated occasions. L is the number of cycles, ranging from 1-99999. Cycle instructions can be nested.

Format:

G22 L_
.
.
.
} **Block Cycle**
G800 ;End

For example



Program as follows:

```
N0000 G17 G90 X0 Y0 F250 M03      ;  
N0001 G91 G01 Z-10  
N0010 G22 L4                      ;  
N0020 G01 X20                      ;  
N0030 G03 X10 I5 J0 Y0          ;  
N0040 G800                      ;  
N0050 G01 X20          ;D-E line  
N0060 Y-30          ;E-F line  
N0070 G11 X140 Y-30      ;set mirror  
N0080 G22 L4          ;circulation begin  
N0090 G01 X20          ;F-G line  
N0100 G03 X10 I5          ;G-H arc  
N0110 G800          ;circulation end  
N0120 G01 X20          ;I-J line  
N0130 G01 Y-30          ;cancel mirror  
N0140 G12          ;J-A line  
N0150 G26          ;go back to program begin point  
N0160 M02          ;over
```

3.30 Macro program instruction(G65/G66/G67)

3.30.1 Non-Mode Macro Command G65

Format: G65 P_L_A_B_C_

Non-mode macro command G65 only work at current line , which is different to mode macro command(G66),which always work until macro cancel command(G67)

P_ : Specify name of macro program, E.g: P6000 , name of specified macro program is 6000 .

L_ : Set times of call macro program

<A_B_C_... ...> : Argument , which is used for transfer data to macro variable(***) ,

Transferring table is as following

Argument	Variable	Argument	Variable	Argument	Variable
A	#0	I	#7	T	#14
B	#1	J	#8	U	#15
C	#2	K	#9	V	#16
D	#3	M	#10	W	#17
E	#4	Q	#11	K	#18
F	#5	R	#12	Y	#19
H	#6	S	#13	Z	#20

Warning:

1. Macro variables #100-#155¾-#201 was occupied by system, user cannot use.

2. User cannot use G70,G71,G72,G73,G92,G76 etc loop command on Macro program.

Note: the address G, L, N, Q, P can't be used as user-defined variables.

Example:

Main program:9000	Macro program:8000
G00 X0 Z0	N1 #2=#0+#1
G65 P8000 L1 A5 B6	N2 IF (#2 EQ 10) GOTO 4
G0 X0 Z0	N3 GOO X#2
M30	N4 G00 Z#1
	N5 M99 ;Return

3.30.2 Mode Macro Command G66/G67

G66 is mode macro command , G67 is cancel mode macro command

Format: G66 P_L_A_B_C_

G67

G66 Mode macro command,which always call macro program until macro cancel command(G67)

P_ : Specify name of macro program, E.g: P7000 , name of specified macro program is 7000 .

L_ : Set times of call macro program

<A_B_C_... ...> : Argument , which is used for transfer data to macro variable(***) , the transferring table is same as above table.

Example:

Main Program : 4000

G00 X0 Z0

G66 P6000 L2 A5 B6

A8 B1

```
A9 B10
G67
M30
Macro Program: 6000
N1 #2=#0+#1
N2 IF (#2 EQ 10) GOTO 4
N3 G00 X#2
N4 G00 Z#1
N5 M99 ; Return
```

3.30.3 Macro Program Instruction

3.30.3.1 Input Instruction: WAT

Waiting for the input port X valid or invalid instruction

Format: WAT+ (-) X

Attention: "+" to means the input is effective;

"-" means the input is invalid;

"X" means the input port X00-X55; see the I/O diagnosis;

3.30.3.2 Output Instruction: OUT

Set the output port Y is valid or invalid instruction

Format: OUT +(-)Y

Attention: "+" means the output is effective;

"-" means the output is invalid;

"Y" means the output port Y00-Y31; see the I/O diagnosis;

3.30.3.3 Assignment Instruction: =

Explanation: used for assignment of a variable

Eg.: #251=890.34 #450=#123

And also it could be mathematical expression , Example: #440=#234+#470

3.30.3.4 Unconditional Jump: GOTO n

“GOTO n” is the command that for jump to the program line that is specified by sequence number (N**) unconditionally. n is the sequence number.

E.g.: GOTO 5 ; // Jump to N5 program line.

Note: when specified program line , n , is beyond sequence number of N1-N99999, CNC system will hint error.

n , program line, could be macro variable (#**)

E.g.: GOTO #100

3.30.3.5 Conditional Jump

1) IF (Conditional express) GOTO n

If condition is met, execute GOTO n ,jump to N** program line; if the condition isn't met, execute the next segment.

Example: N1 IF(#200 EQ 1) GOTO 20

N10 G00 X0

N20 G00 Z0

Explanation: If #200 is equal to 1, system will execute GOTO 20 , jump to N20 , and execute “G00 Z0”, if #200 isn't equal to 1, system don't execute operation of “GOTO 20” ,and will execute next segments , “G00 X0”,and then execute “G00 Z0”.

2) IF (Conditional express) THEN <A Expression>

<B operational segment>

If condition is met, system execute A expression , and then execute B operational segment ; if condition is not meet, execute the next segment , B operation.

Example: #101=0

```
N1 IF(#100 EQ 1) THEN #101=1
N2 IF(#101 EQ 1) GOTO 4
N3 G00 X100
N4 G00 Z100
```

Explanation: If #100 is equal to 1, system will execute “#100=1”, and then judge #101 is equal to 1 ,jump to N4 & “execute G00 Z100” ; if #100 isn’t equal to 1, system will judge #101 also isn’t equal to 1 directly , and execute “G00 X100” & “G00 Z100”.

NOTE: 1.<A expression> normally is assignment statement.

2. <A expression> after THEN must exist, otherwise system will hint grammatical errors.

Prolongation:

3) IF(conditional express)

<A operational command>

ELSE

<B operational command>

ENDIF

4) IF(conditional express)

<A operational command>

ELIF

<B operational command>

ENDIF

3.30.3.6 Loop Command

Format: (Conditions Initialization)

WHILE (conditional expression) DO n

<A operational segments>

[Alter condition of loop]

END n

<B operational segments>

When conditions are met during WHILE cycle command, execute the operational segments between DO n and END n . Otherwise,when condition isn't met, jump to the program line after END n ,also execute B operational segments.

We can nest for loops by placing one loop within another.

Note: 1. There must have operational codes that are for change condition at operational segments ,which is between Do n & END n. Otherwise system will enter endless loop.

2. Nesting of macro program loop statements of SZGH CNC system is 3 pieces of loops at most . Also n only could be 1 , 2 , 3 .

3.n of “DO n” & “END n” must keep same.

Example: #100=2 #150=5 #200=25

```
WHILE (#100 LT 3) DO 1
```

```

G00 X100
WHILE (#150 EQ 5) DO 2
G00 Y100
WHILE (#200 GE 20) DO 3
G00 Z100
#200=#200-2
END 3
#150=#150-1
END 2
#100=#100-1
END 1

```

3.30.4 Operators' meaning

Operator	Sign	Ex.	Operator	Sign	Ex.	Operator	Sign	Ex.
EQ	=	equal	GT	>	greater	LT	<	Less
NE	≠	unequal	GE	≥	G&E	LE	≤	L&E

3.30.5 Arithmetic & Logic Operation

Table:

Function	Format	Attention
Definition	#i = #j	
Addition	#i = #j + #k ;	
Subtraction	#i = #j - #k ;	
Multiplication	#i = #j * #k ;	
Division	#i = #j / #k ;	
Sin	#i = SIN(#j) ;	
Asin	#i = ASIN(#j);	
Cos	#i = COS(#j) ;	
Acos	#i = ACOS(#j);	90.5 degrees means 90 degrees & 30 points
Tan	#i = TAN(#j);	
Atan	#i = ATAN(#j);	
Square root	#i = SQRT(#j);	
Absolute value	#i = ABS(#j) ;	
Rounding off	#i= ROUND(#j);	
Round down	#i = FIX(#j);	
Round up	#i = FUP(#j);	
Natural logarithm	#i = LN(#j);	
Exponential function	#i = EXP(#j);	
Or	#i = #j OR #k ;	
Exclusive or	#i = #j XOR #k ;	Executing with binary system
And	#i = #j AND #k ;	

3.30.6 Local Variable

#0--#20 : local variables only can be used to store data in macro program, such as a result of operation, when power is off, the local variables are initialized to the empty. The argument assignment to the local variable when calling the macro program.

3.30.7 Global Variable

#21--#999 : Their meanings are the same in different macro program.

When power is off, variables of #21--#100 are initialized to zero, variables of #101--#999 data are saved and won't lost even if the power is off.

#900~#999: can be used for CMP code(compare) on PLC ladder.

3.30.8 System Variable

#1000-- : the system variables are used to change various data when reading the running CNC. For example, the current position and the compensation of tool.

Special Attention: macro variables #100~#155, #161~#165, and #190--#204 have been used in system, and users can not use these variables.

#161~#165 are used for record&display XYZAB machine coordinate value of G31/G311 codes.

#201~#204 are used on G176 (threading cycle code)

User can use global variable: #31~#100 & #205~#999, the others are used already on CNC system.

3.30.9 System Parameter Variable

#1001--#1099 : Value of X-axis length compensation for T1--T99(Unit: um)

#1101--#1199 : Value of D1 radius compensation for T1--T99(Unit: um)

#1201--#1299 : Value of Y-axis length compensation for T1--T99(Unit: um)

#1301--#1399 : Value of D2 radius compensation for T1--T99(Unit: um)

#1401--#1499 : Value of Z-axis length compensation for T1--T99(Unit: um)

#1501--#1599 : Value of D3 radius compensation for T1--T99(Unit: um)

#1601--#1699 : Value of A-axis length compensation for T1--T99(Unit: um)

#1701--#1799 : Value of D4 radius compensation for T1--T99(Unit: um)

3.30.10 I/O variable

#1800: X00-X07 (D0-D7) ; input resistor

#1801: X08-X15 (D0-D7) ; input resistor

#1802: X16-X23 (D0-D7) ; input resistor

#1802: X16-X23 (D0-D7) ; input resistor

#1803: X24-X31 (D0-D70 ; input resistor

#1804: X32-X39 (D0-D7) ; input resistor

#1808: Y00-Y15 (D0-D15) ; output resistor

#1809: Y16-Y31 (D0-D15) ; output resistor

Warning:

1. Macro variables #100~#155¾--#201 was occupied by system, user cannot use.

2. User cannot use G70,G71,G72,G73,G92,G76 etc loop command on Macro program.

Note: the address G, L, N, Q, P can't be used as user-defined variables.

3.30.11 Message Hint Dialog Box

Format: MSG(hint words) or MSG[hint words] ;

Hint words is that user want to hint message on CNC system.

Note: 1. This code can be used on normal NC programs.

2. After hint message, CNC system will pause program automatically.

Format: STAF(hint words) or STAF[hint words];

Hints words is that user want to hint message on CNC system. And CNC system don't pause program automatically.

3.30.12 Build Processing Program Automatically

3.30.12.1 New/Open a program

Format: FILEON(Program) or FILEON[Program]

Example: FILEON(AABBCC) or FILEON[AABBCC]

It means that new or open a program “AABBCC”

3.30.12.2 Close program

Format: FILECE

It means that close current opening program, if without this code, system will close current program after program is finished.

3.30.12.3 Write codes into program

Format: FILEWD(Blocks) or FILEWD[Blocks]

Example: FILEWD(G54G0X0Z0) or FILEWD[G54G0X0Z0]

It means that write a blocks of “G54G0X0Z0” into current opening program.

3.30.12.4 Write current absolute coordinate into program

Format: FILEWC

It means that write current absolute coordinate value into program.

Example:

G0X0Z0
FILEON[AABBCC]
FILEWD [G54G0X0Z0]
G1X45Z89
FILEWC
G1X99Z76
FILEWC
FILECE

After finished this program, system will new a program of “AABBCC” under directory of program, its blocks is :
G54G0X0Z0
X45Z89
X99Z76

3.30.12.5 Save all global variables with program format

FILEMS(ABC) or FILEMS(ABC)

Save #21~#999 to program of “ABC”

If user want to read these variables, it just needs to execute this “ABC” program, or take this program as subprogram.

3.30.12.6 Application Example

G0 X100 Z2
FILEON(SCANNER01)
G22 L10
G31 Z-10 P1001 F100
FILEWC
G0 Z2
U-1
G800

This is example program for scanning,which will generate program: SCANNER01

3.31 User-defined Macro Instruction (G110-G170)

Every user-defined G code is corresponding to a macro program ProgramGxxx, user cannot programme the macro program in NC system, must edit the macro code in the computer, and then copy into the system.

3.31.1 Bolt Hole Circle Drilling Cycle

Defines the G152 function: the arc model porous drilling cycle. (must copy the macro program ProgramG152 into system).

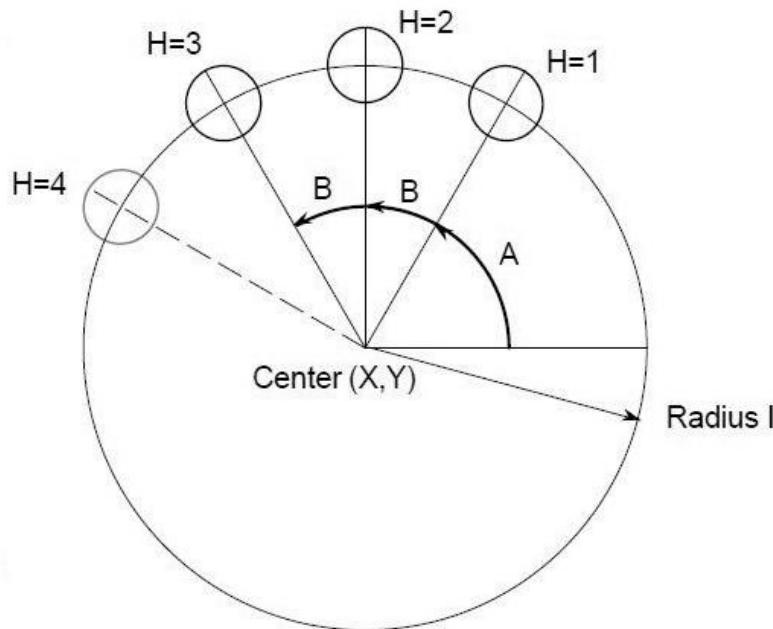


Fig3.31.1 Bolt Hole Circle Drilling Cycle

Format: G152 X_ Y_ Z_ R_ I_ A_ B_ H_ F_ ;

- X_ : The X coordinate of the center of the circle. [#18]
- Y_ : The Y coordinate of the center of the circle. [#19]
- Z_ : Hole depth [#20]
- R_ : Coordinate of an approaching point [#12]
- F_ : Cutting feedrate [#5]
- I_ : Radius of the circle [#7]
- A_ : Drilling Start angle, also angle of the first hole [#0]
- B_ : Incremental angle (Clockwise when negative value) [#1]
- H_ : Number of holes [#6]

Macro program ProgramG152 as follows:

```
#80=#0
#81=#1
#82=#2
#85=#5
#86=#6
#87=#7
#92=#12
#98=#18
```

```

#99=#19
#100=#20
#30=#4003
#31=#4014
G90
IF[#30 EQ 90] GOTO 1
G53
#98=#5001+#98
#99=#5002+#99
N1 WHILE[#86 GT 0] DO 1
#35=#98+#87*COS[#80]
#36=#99+#87*SIN[#80]
G81X#35Y#36Z#100R#92F#85
#80=#80+#81
#86=#86-1
END 1
G#30 G#31 G80
M99

```

Example: Bolt hole circle drilling cycle G152 , to drill 5 holes at intervals of 45 degrees after a start angle of 0 degrees. On the circumference of a circle with radius 40. The absolute center of the circle is (100, 50)

Program:

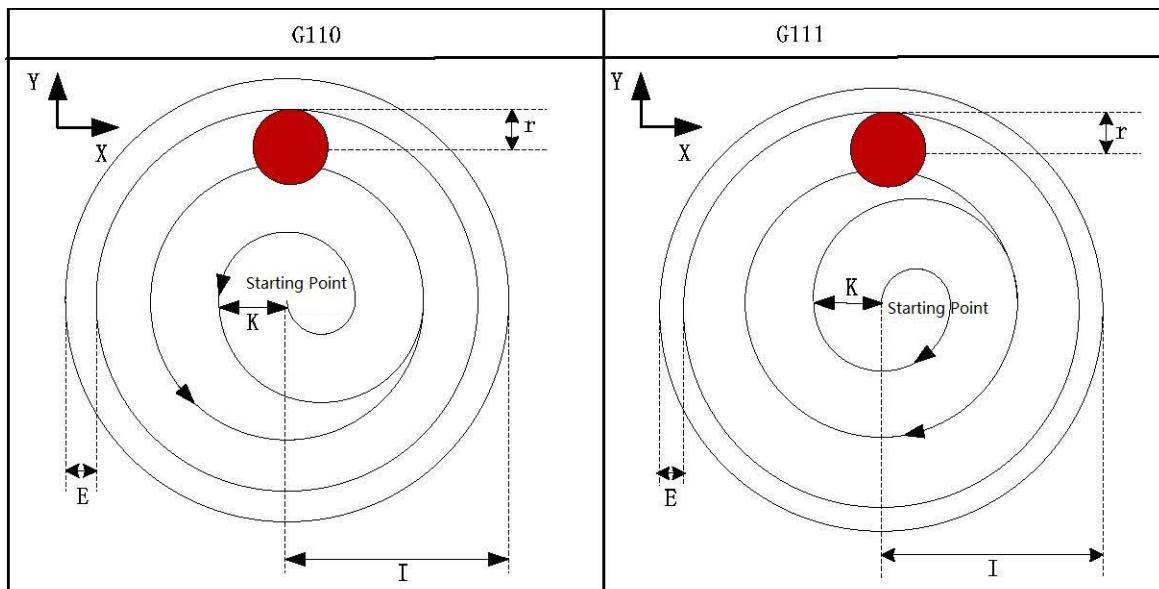
```

G90 G92 X0 Y0 Z4
G152 X100 Y50 R10 Z-20.0 F20 I40.0 A0 B45.0 H5
M30

```

3.31.2 Inner Circle Groove Roughing Cycle (G110/G111)

This cycle starts from arc center, arc interpolation with spiral mode, until milling round groove. G110 is CCW Inner circle groove roughing cycle, G111 is CW Inner circle groove roughing cycle.



Format: G110/G111 X_Y_Z_R_I_W_Q_K_V_E_D_F_

X_Y_: The starting point in XY plane

Z_: The distance from point R to the bottom of the hole

R_: The distance from the initial level to point R level

I_: Radius of Groove Round

W_: Depth of 1st cutting feed, distance from point R level

Q_: Depth of each cutting feed

K_: Incremental Width

V_: Height between unprocessed surface and cutter

E_: Allowance for fine groove

D_: Tool radius offset number

F_: Cutting Feedrate

Note: 1. Radius of groove round, I, should be greater than radius of current tool.

2. Depth of 1st cutting feed, W, should be greater than 0. If depth of 1st cutting feed exceeds the groove bottom, then the machining is performed at the groove bottom;

3. Allowance for fine milling isn't less than 0, otherwise absolute value of negative number.

4. Range of Tool radius offset number , D, is 0~32. default is 0.

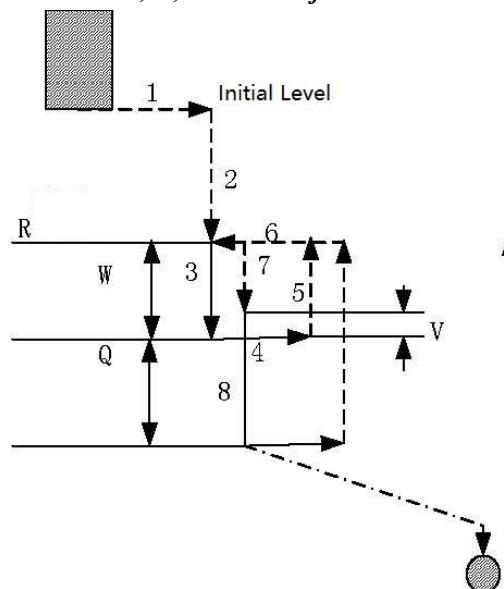


Fig3.31.2 Path of Inner Circle Groove Roughing Cycle

Process of Cycle:

- (1) Rapid positioning to the position in XY plane
- (2) Rapid down to point R level
- (3) Cut a depth (W) downward at the cutting speed by helical mode, then feed to circle center,
- (4) Mill the circle surface with a radius (I) hectically outward from the center by an increment (K) each time;
- (5) Return to point R level along Z axis;
- (6) Axis X and Y rapidly position to the starting point;
- (7) Rapid traverse to V , the distance to the unprocessed end surface;
- (8) Cut a depth(Q+V) downward along Z axis;
- (9) Repeats the operation (4)~(8) until the total depth of circle surface is finished;
- (10) Return to point R level (G99);or return to initial level(G98).

Example: Rough milling an inner circle groove using the canned cycle code G110, as shown in the figure below: (D1=5).

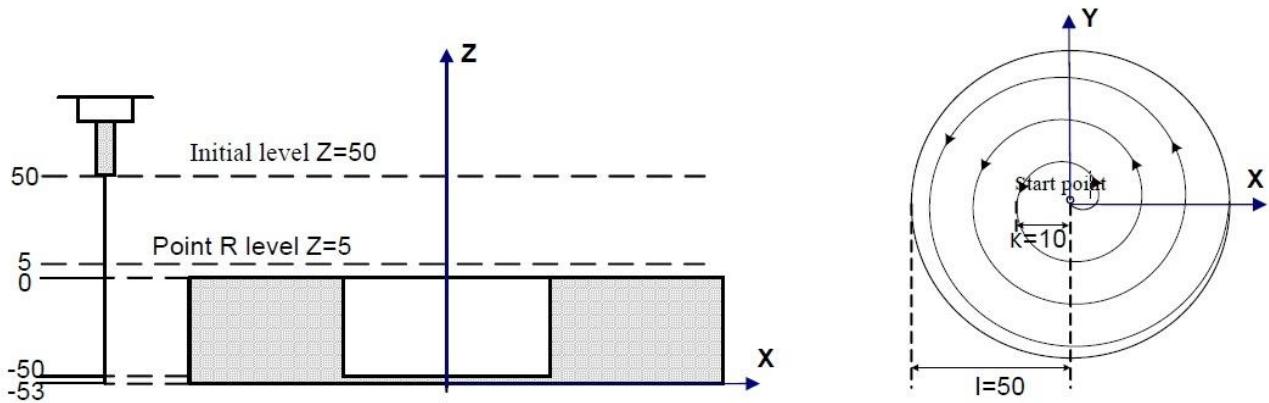


Fig3.31.3 Example of G110

G90 G00 X50 Y50 Z50 ; G00 Rapid traverse

G99 G110 X25 Y25 Z-50 R5 I50 W20 Q10 K10 E0 V10 F800 D1 ; Groove rough milling

G80 X50 Y50 Z50 ; Canned cycle cancel and return from point R level

M30 ;

3.31.3 Inner Circle Groove Fine Cycle (G112/G113)

The tool fine mills a full circle within a circle by the specified radius I and the specified direction, and it returns after finishing the fine milling.

G112: CCW Inner Circle Groove Fine Cycle ; C113: CW Inner Circle Groove Fine Cycle .

Format: G112/G113 X_ Y_ Z_ R_ I_ J_ D_ F_

X_Y_ : The starting point in XY plane

Z_ : The distance from point R to the bottom of the hole

R_ : The distance from the initial level to point R level

I_ : Radius of Fine Milling Circle

J_ : Distance from fine milling start point to circle center

D_ : Tool radius offset number

F_ : Cutting Feedrate

Note: 1. Radius of fine milling circle, I, Its absolute value is used if it is negative.

2. Distance from fine milling starting point to circle center, J, Its absolute value is used if it is negative.

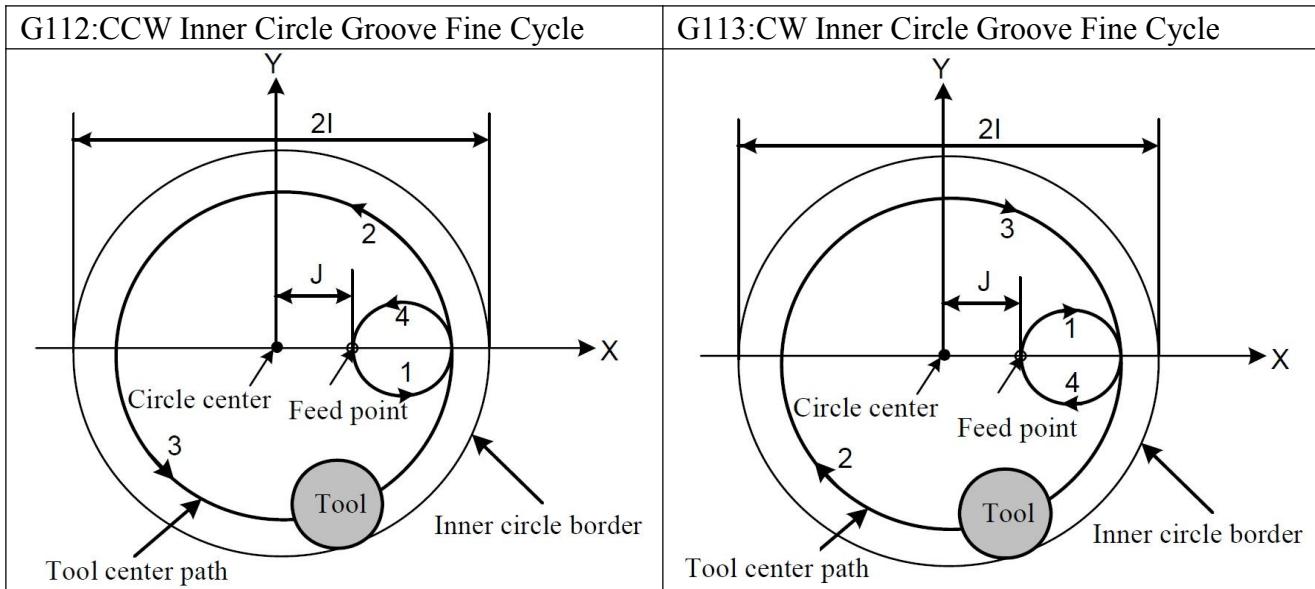
3. Allowance for fine milling isn't less than 0, otherwise absolute value of negative number.

4. Range of Tool radius offset number , D, is 0~32. default is 0.

Process of Cycle:

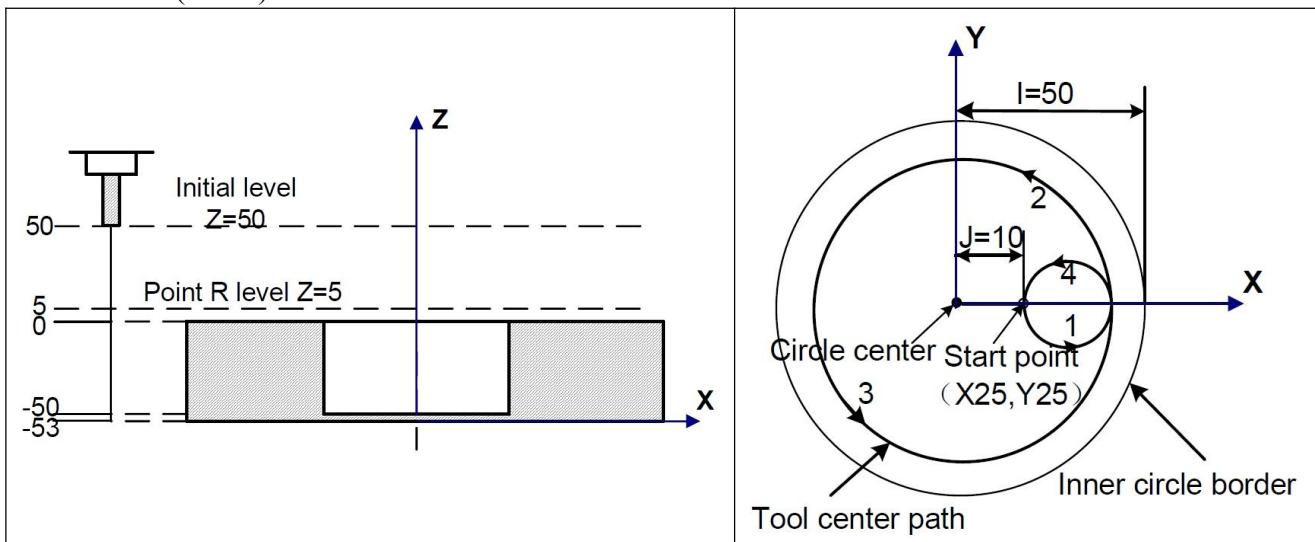
- 1) Rapid positioning to the position in XY plane;
- 2) Rapid down to point R level ;
- 3) Feed to the machining start point at the bottom of hole ;
- 4) Arc Interpolation by the transition arc 1 from the starting point;
- 5) Circle Interpolation for the whole circle by inner arc path of finish-milling ;
- 6) Circle Interpolation by transition arc 4 and return to the starting point ;
- 7) Return to the initial level or point R level according to G98/G99.

Command Path:



Note: Q, P, L is invalid when using G112/G113, but value of Q & P will be remained as canned cycle.

Example: Fine milling a circular groove that has been rough milled as follows by canned cycle code G112. (D1=5).



G90 G00 X50 Y50 Z50 ; Rapid Positioning

G99 G24 X25 Y25 Z-50 R5 I50 J10 D1 F800 ; Fine milling inner circle with CCW direction

G80 X50 Y50 Z50 ; Cancel canned cancel and return to point R level

M30 ; End of program

Note: 1. Q, P, L is invalid when using G112/G113, but value of Q & P will be remained as canned cycle.

2. When use G112/G113 is used, G codes in 01 group cannot be used, otherwise G114/G115 is replaced by other codes in group 1.

3. *Tool radius compensation: in the fixed cycle command, the tool radius compensation is ignored, the system calls the tool radius compensation specified by the program during the tool infeed.*

3.31.4 Outer Circle Fine Milling Cycle (G114/G115)

Format: G114/G115 X_ Y_ Z_ R_ I_ J_ D_ F_ ;

X_Y_ : The starting point in XY plane

Z_ : The distance from point R to the bottom of the hole

R_ : The distance from the initial level to point R level

L_ : Radius of Fine Milling Circle

J_ : Distance from fine milling start point to circle center

D_ : Tool radius offset number

F_ : Cutting Feedrate

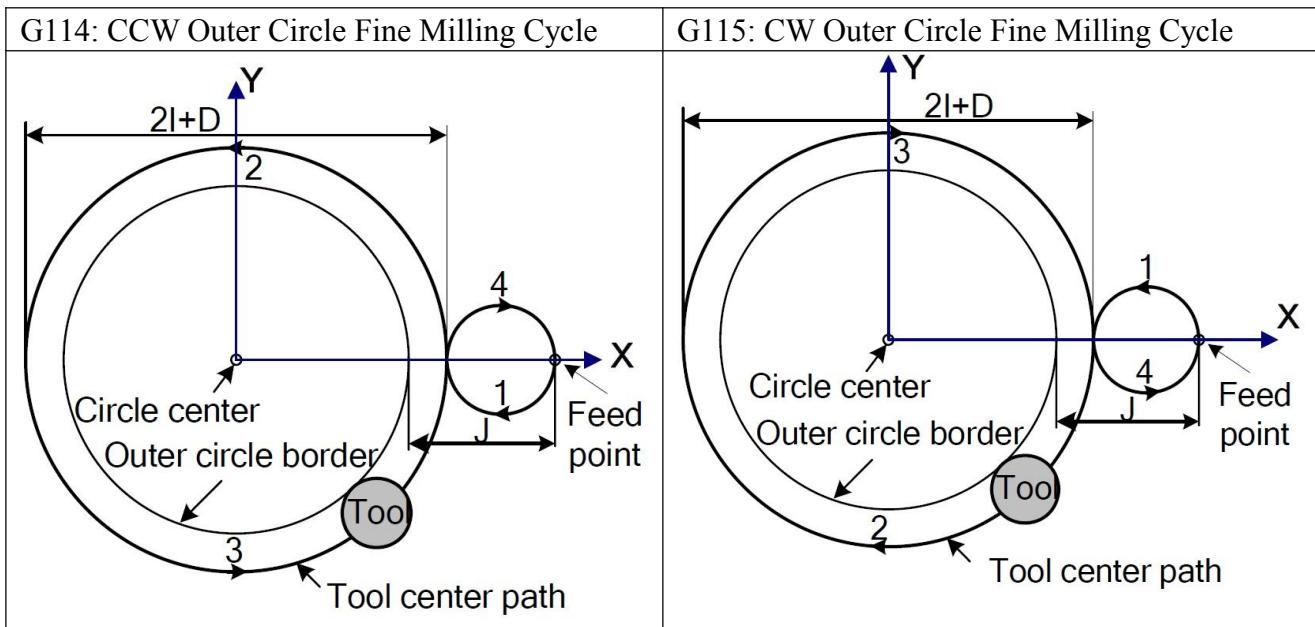
Note: 1. Radius of fine milling circle, I, Its absolute value is used if it is negative.

2. Distance from fine milling starting point to circle center, J, Its absolute value is used if it is negative.

3. Allowance for fine milling isn't less than 0, otherwise absolute value of negative number.

4. Range of Tool radius offset number , D, is 0~32. default is 0.

Command Path:



Command Path:

- (1) Rapid positioning to a location within XY plane ;
- (2) Rapid down to R level ;
- (3) Feed to the bottom of hole
- (4) Circular interpolation by the transition arc 1 from the starting point ;
- (5) Circular interpolation for the whole circle by the path of arc 2 and arc 3 ;
- (6) Circular interpolation by transition arc 4 and return to the starting point ;
- (7) Return to the initial level(G98) or point R level (G99) .

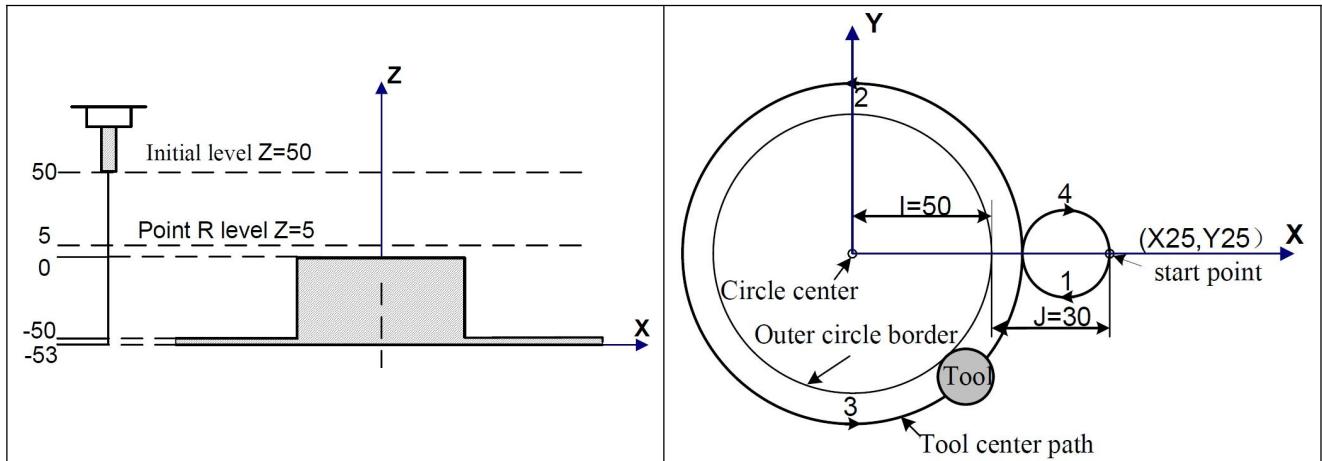
In outer circle fine milling cycle, the interpolation directions of the transition arc and fine milling arc are different. The interpolation direction in the code means the one of the fine milling.

Note: 1. Q, P, L is invalid when using G114/G115, but value of Q & P will be remained as canned cycle.

2. When use G114/G115 is used, G codes in 01 group cannot be used, otherwise G114/G115 is replaced by other codes in group 1.

3. Tool radius compensation: in the fixed cycle command, the tool radius compensation is ignored, the system calls the tool radius compensation specified by the program during the tool infeed.

Example: Fine milling a circular groove that has been rough milled as follows by the canned cycle code G114.



G90 G00 X50 Y50 Z50 ; Rapid positioning

G99 G114 X25 Y25 Z-50 R5 I50 J30 D1 F800 ; Outer circle fine milling

G80 X50 Y50 Z50 ; Cancel canned cycle and return from point R level

M30 ; End of program

3.31.5 Outer Circle Rough Milling Cycle (G116/G117)

This cycle starts from the starting point, cutter make circle interpolation with specified radius & direction , until milling a pad.

G116: CCW Outer Circle Rough Milling cycle, G111: CW Outer Circle Rough Milling Cycle.

Format: G116/G117 X_ Y_ Z_ R_ I_ J_ W_ Q_ K_ C_ E_ D_ F_ ;

X_Y_ : The starting point in XY plane

Z_ : The distance from point R to the bottom of the hole

R_ : The distance from the initial level to point R level

I_ : Radius of rough milling outer circle

J_ : Radius of workpiece

W_ : Depth of 1st cutting feed, distance from point R level

Q_ : Depth of each cutting feed in Z-axis direction

K_ : Incremental Width in XY plane

C_ : First cutting feed in X-axis direction

V_ : Height between unprocessed surface and cutter

E_ : Allowance for outer circle rough milling

D_ : Tool radius offset number

F_ : Cutting Feedrate

Note: 1. Radius of rough milling circle, I, Its absolute value is used if it is negative.

2. Radius of unprocessed workpiece, J, Its absolute value is used if it is negative.

3. Allowance for outer circle rough milling isn't less than 0, otherwise absolute value of negative number.

4. Depth of 1st cutting feed, W, should be greater than 0. If depth of 1st cutting feed exceeds the groove bottom, then the machining is performed at the groove bottom;

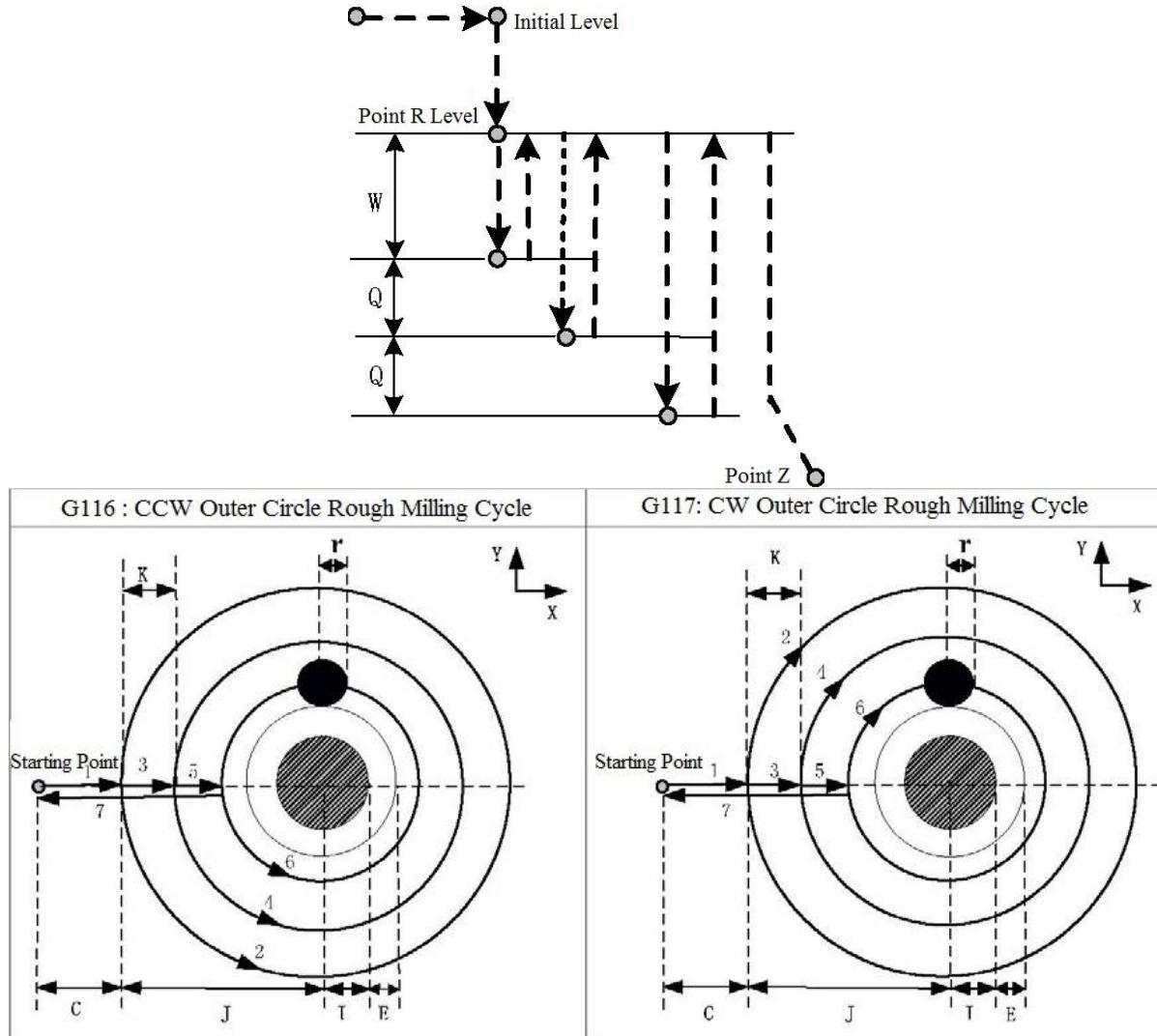
5. Depth of each cutting in Z-axis direction, Q, its absolute value is used if it is negative.

6. Incremental width in XY plane should be less than diameter of cutter, greater than 0. Its absolute value is used if it is negative. If without specify K, default K= radius of cutter.

7. Value of first cutting feed in X-axis direction, C, should not be less than radius of cutter+2.0, When positive number feed in positive direction of X-axis;when negative number, feed in negative direction of X-axis.

8. Range of Tool radius offset number , D, is 0~32. default is 0.

Command Path:



Command Process:

- (1) Rapid positioning to a location within XY plane ;
- (2) Rapid down to point R level , and then rapid down to point W level;
- (3) First cutting feed(c) in X-axis direction, Line 1 is feed with linear interpolation ;
- (4) Whole Circle interpolation with arc 2 ;
- (5) Cutting feed with increment width (K) from starting point to center until radius of round workpiece is (I+E);
- (6) Cutter rapid return to point R level in Z-axis and then rapid traverse to starting point ;
- (7) Cutter rapid down to depth of unprocessed end surface & each cutting feed (W+nQ);
- (8) Loop run (4)~(7) until finish outer circle rough milling surface with depth of Z point;
- (9) Return to point R level ;
- (10) Return to starting point in XY plane.

In outer circle fine milling cycle, the interpolation directions of the transition arc and fine

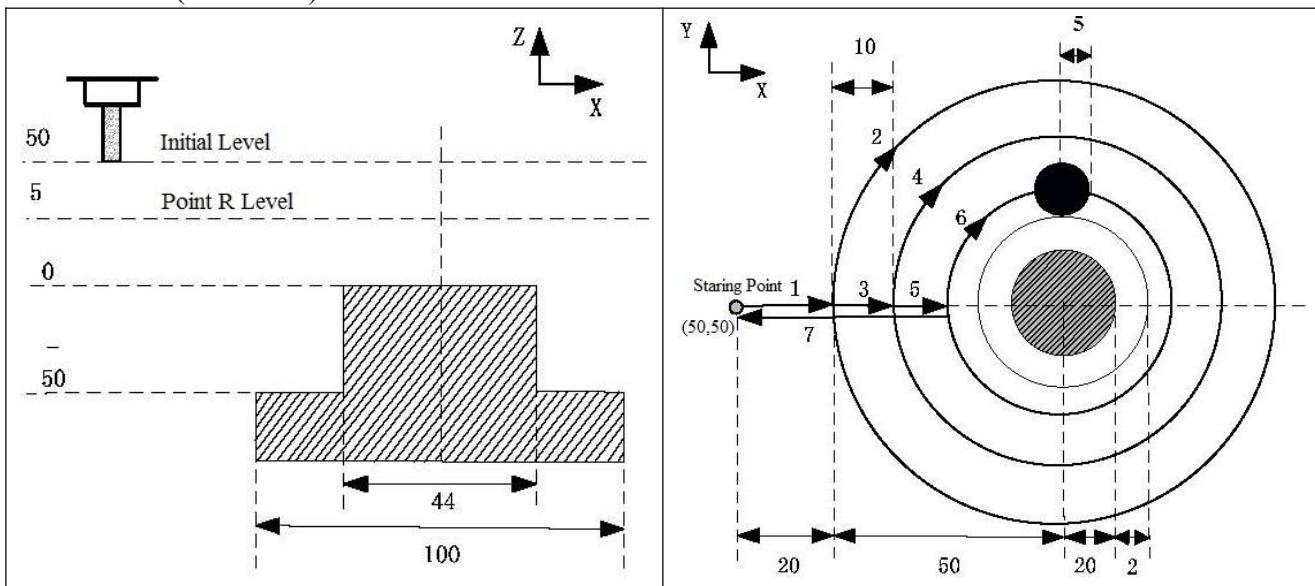
milling arc are different. The interpolation direction in the code means the one of the fine milling.

Note: 1. P, L is invalid when using G116/G117, but value of P will be remained as canned cycle.

2. When use G116/G117 is used, G codes in 01 group cannot be used, otherwise G116/G117 is replaced by other codes in group 1.

3. Tool radius compensation: in the fixed cycle command, the tool radius compensation is ignored, the system calls the tool radius compensation specified by the program during the tool infeed.

Example: Fine milling a circular pad that has been rough milled as follows by the canned cycle code G117. (D1=5mm)



G90 G00 X0 Y0 Z50 ; Rapid positioning

G99 G117 X50 Y50 Z-50 R5 I20 J50 W20 Q10 K10 C20 E2 D1 F800 ; Outer circle roughing

G80 X50 Y50 Z50 ; Cancel canned cycle and return from point R level

M30 ; End of program

3.31.6 Outer Rectangle Rough Milling Cycle (G132/G133)

This cycle starts from the starting point, cutter make circle interpolation with specified parameters , until milling a rectangle pad.

G132:CCW Outer Rectangle Rough Milling cycle,G133:CW Outer Rectangle Rough Milling cycle.

Format: G132/G133 X_ Y_ Z_ R_ I_ J_ A_ B_ W_ Q_ K_ C_ E_ D_ F_ ;

X_Y_ : The starting point in XY plane

Z_ : The distance from point R to the bottom of the hole

R_ : The distance from the initial level to point R level

I_ : Width of rough milling outer rectangle in X-axis (>0)

J_ : Width of rough milling outer rectangle in Y-axis (>0)

A_ : Width of unprocessed rectangle workpiece in X-axis (>0)

B_ : Width of unprocessed rectangle workpiece in Y-axis (>0)

W_ : Depth of 1st cutting feed, distance from point R level (>0)

Q_ : Depth of each cutting feed in Z-axis direction

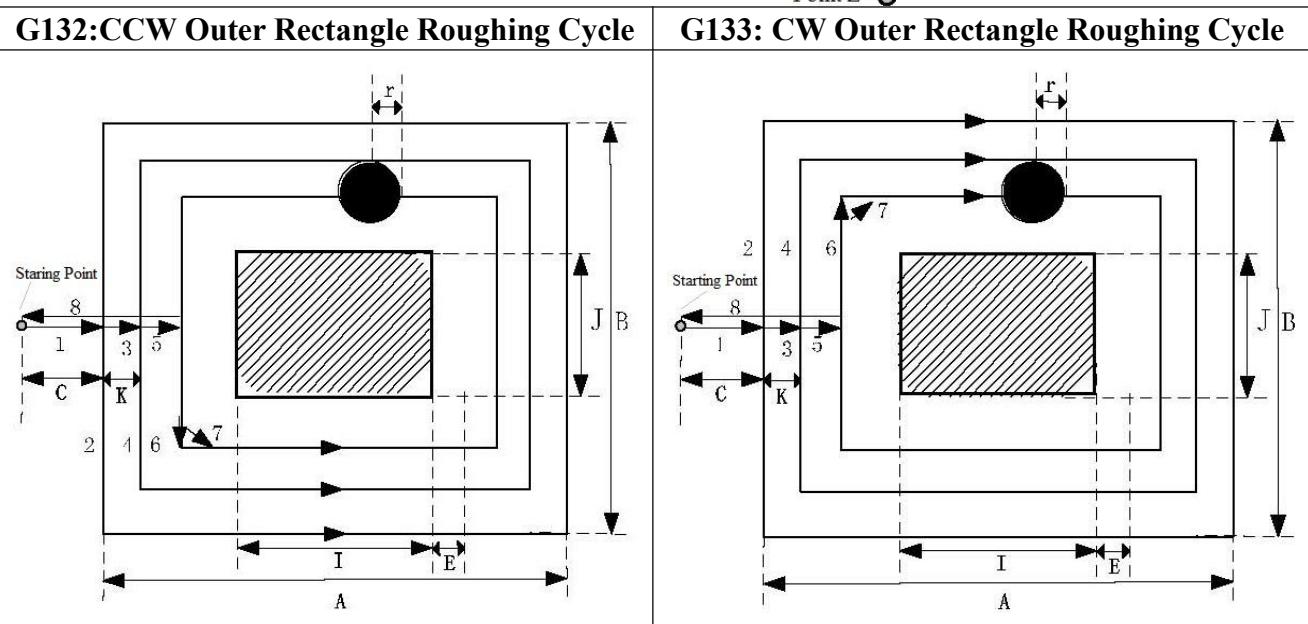
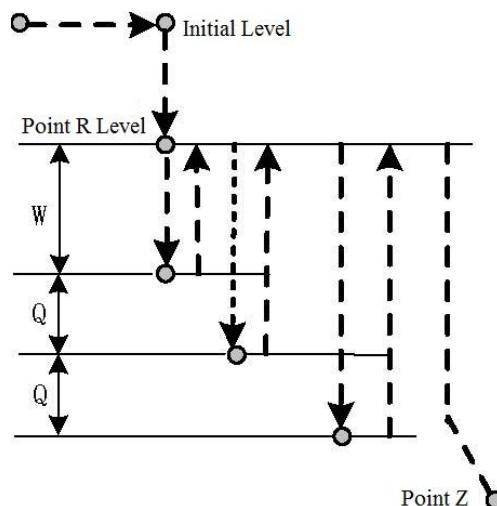
K_ : Incremental Width in XY plane (>0)

C_ : First cutting feed in X-axis direction

V_ : Height between unprocessed surface and cutter
E_ : Allowance for outer rectangle rough milling
D_ : Tool radius offset number
F_ : Cutting Feedrate

Note: 1. Width of rough milling outer rectangle, I & J , Its absolute value is used if it is negative.
 2. Width of unprocessed workpiece, A & B , Its absolute value is used if it is negative.
 3. Allowance for outer rectangle rough, E , isn't less than 0, otherwise absolute value of negative number.
 4. Depth of 1st cutting feed, W , should be greater than 0. If depth of 1st cutting feed exceeds the groove bottom, then the machining is performed at the groove bottom;
 5. Depth of each cutting in Z-axis direction, Q , its absolute value is used if it is negative.
 6. Incremental width in XY plane should be less than diameter of cutter, K , is greater than 0. Its absolute value is used if it is negative. If without specify K , default $K = \text{radius of cutter}$.
 7. Value of first cutting feed in X-axis direction, C , should not be less than radius of cutter+2.0, When positive number, feed in positive direction of X-axis; when negative number, feed in negative direction of X-axis.
 8. Range of Tool radius offset number , D , is 0~32. default is 0.

Command Path:



Command Process:

- (1) Rapid positioning to a location within XY plane ;
- (2) Rapid down to point R level , and then rapid down to point W level at first time;
- (3) First cutting feed(C) in X-axis direction, Line 1 is feed with linear interpolation ;
- (4) Whole linear interpolation with line 2 ;
- (5) Cutting feed with increment width (K) from starting point to center until Length of rectangle pad is (I+2E), Width of rectangle pad is (J+2E);
- (6) Milling corner of rectangle
- (7) Cutter rapid return to point R level in Z-axis and then rapid traverse to starting point ;
- (8) Cutter rapid down to depth of unprocessed end surface & each cutting feed (W+nQ);
- (9) Loop run (4)~(7) until finish outer circle rough milling surface with depth of Z point;
- (10) Return to point R level ;
- (11) Return to starting point in XY plane.

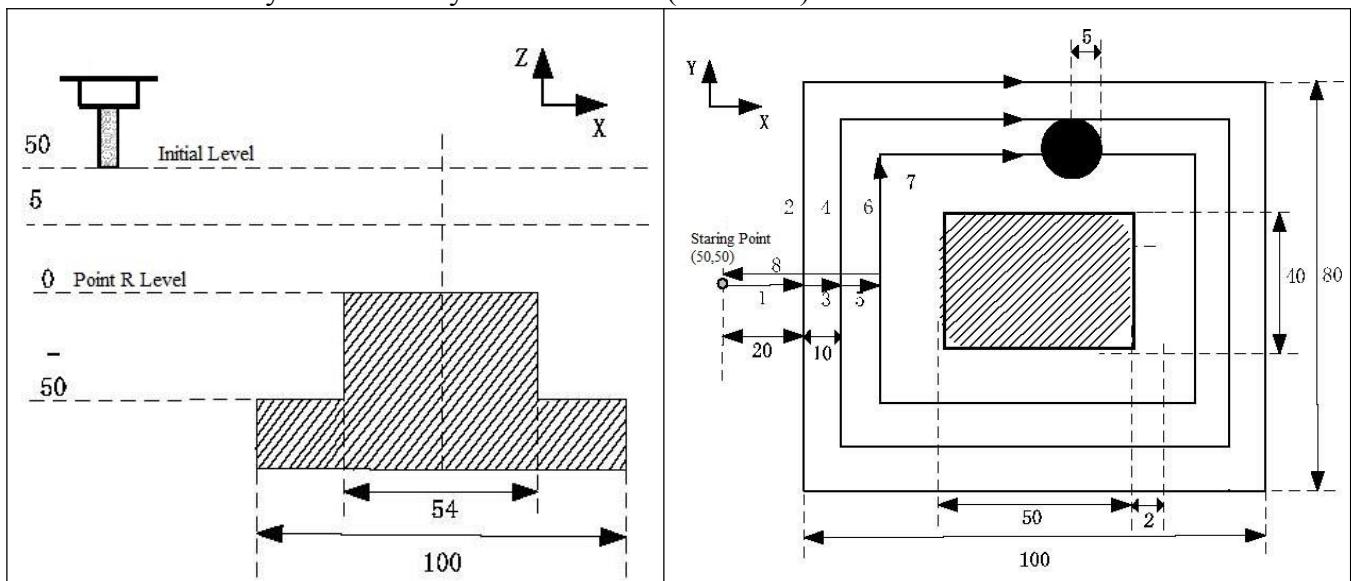
In outer circle fine milling cycle, the interpolation directions of the transition arc and fine milling arc are different. The interpolation direction in the code means the one of the fine milling.

Note: 1. P, L is invalid when using G116/G117, but value of P will be remained as canned cycle.

2. When use G116/G117 is used, G codes in 01 group cannot be used, otherwise G116/G117 is replaced by other codes in group 1.

3. Tool radius compensation: in the fixed cycle command, the tool radius compensation is ignored, the system calls the tool radius compensation specified by the program during the tool infeed.

Example: Fine milling a rectangle pad(Length=50, Width=40, Height=50) that has been rough milled as follows by the canned cycle code G133. (D1=5mm)



G90 G00 X0 Y0 Z50 ; Rapid positioning

G99 G133 X50 Y50 Z-50 R5 I50 J40 A100 B80 W20 Q10 K10 C20 E2 D1 F800 ;

Outer rectangle roughing

G80 X50 Y50 Z50 ; Cancel canned cycle and return from point R level

M30 ; End of program

3.31.7 Inner Rectangle Groove Roughing Cycle (G134/G135)

This cycle starts from rectangle center, linear interpolation cycle until milling rectangle groove.

Format: G134/G135 X Y Z R I J K W Q V E D F

X_Y: The starting point in XY plane

Z : The distance from point R to the bottom of the hole

R : The distance from the initial level to point R level

I_— : Width of rough milling inner rectangle in X-axis (>0)

J : Width of rough milling inner rectangle in Y-axis (>0)

K : Incremental Width in XY Plane

W : Depth of 1st cutting feed, distance from point R level

Q : Depth of each cutting feed

V : Height between unprocessed surface and cutter (≥ 0)

V_{\perp} : Height between unprocessed surface and cutter (> 0)
 E : Allowance for fine grooves

E_: Allowance for line groove
D_: Tool radius offset number

D₁: Tool Fadius offset number
E₁: Cutting Feedrate

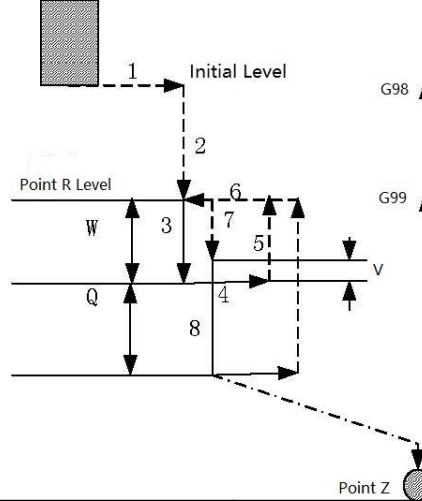
F_z: Cutting Feedrate

Note: 1. Length & Width of groove rectangle, I & J, should be greater than radius of current tool.

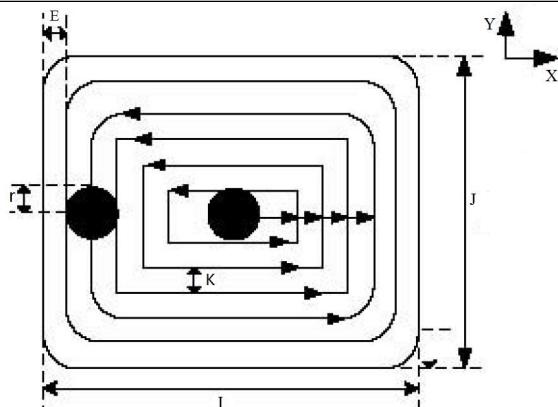
2. Depth of 1st cutting feed, W , should be greater than 0. If depth of 1st cutting feed exceeds the groove bottom, then the machining is performed at the groove bottom;

3. Allowance for rough milling isn't less than 0, otherwise absolute value of negative number.

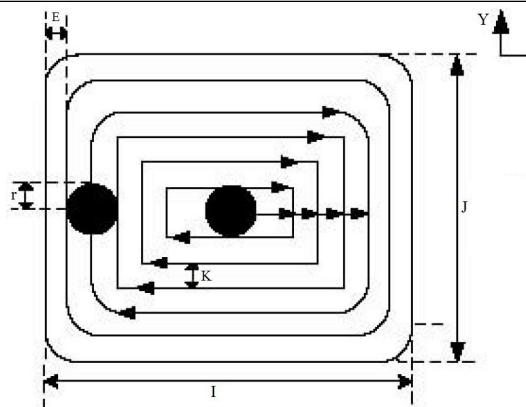
4. Range of Tool radius offset number , D , is 0~32. default is 0.



G134: CCW Inner Rectangle Groove Rough cycle



G135: CW Inner Rectangle Groove Rough Cycle



Process of Cycle:

- (1) Rapid positioning to the position in XY plane
- (2) Rapid down to point R level
- (3) Cut a depth (W) downward at the cutting speed by helical mode,
- (4) Mill the rectangle surface hectically, outward from the center, with a incremental width (K) each time;
- (5) Cutter return to point R level along Z axis;
- (6) Axis X and Y rapidly position to the center of rectangle;
- (7) Rapid traverse to V , the distance to the unprocessed end surface;
- (8) Cut a depth(Q+V) downward along Z axis;
- (9) Repeats the operation (4)~(8) until the total depth of rectangle surface is finished;
- (10) Return to point R level (G99);or return to initial level(G98).

Note: 1. P, L is invalid when using G134/G135, but value of P will be remained as canned cycle.

2. When use G134/G135 is used, G codes in 01 group cannot be used, otherwise G134/G135 is replaced by other codes in group 1.

3. Tool radius compensation: in the fixed cycle command, the tool radius compensation is ignored, the system calls the tool radius compensation specified by the program during the tool infeed.

Example: Rough milling an inner rectangle groove using the canned cycle code G134, as shown in the figure below: (D1=5, Length of rectangle =70 , Width of rectangle =50mm).

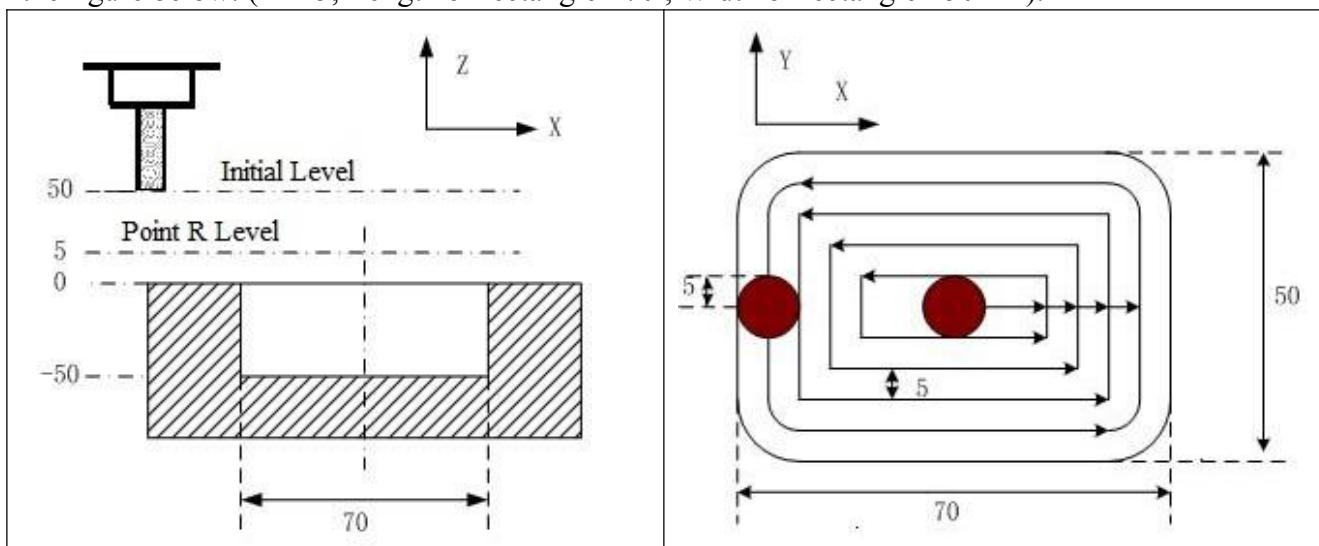


Fig3.31.7 Example of G134

G90 G00 X50 Y50 Z50 ; G00 Rapid traverse

G99 G134 X25 Y25 Z-50 R5 I70 J50 W20 Q10 K5 E0 V10 F800 D1 ; Groove rough milling

G80 X50 Y50 Z50 ; Canned cycle cancel and return from point R level

M30 ;

3.31.8 Inner Rectangle Groove Fine Milling Cycle (G136/G137)

The tool fine milling within a rectangle by the specified width & direction, and it returns after finishing the fine milling.

G136:CCW Inner Rectangle Groove Fine Milling Cycle ; C137:CW Inner Rectangle Groove Fine Milling Cycle .

Format: G136/G137 X_ Y_ Z_ R_ I_ J_ D_ K_ U_ F_ ;

X_Y_: The starting point in XY plane

Z_: The distance from point R to the bottom of the hole

R_: The distance from the initial level to point R level

I_: Width of rectangle in X-axis

J_: Width of rectangle in Y-axis

K_: Distance from fine milling start point to rectangle side in X-axis

U_: corner arc radius

D_: Tool radius offset number

F_: Cutting Feedrate

Note: 1. Width of rectangle in X-axis, I, Range is -99999999~99999999 (*min unit). Its absolute value is used if it is negative.

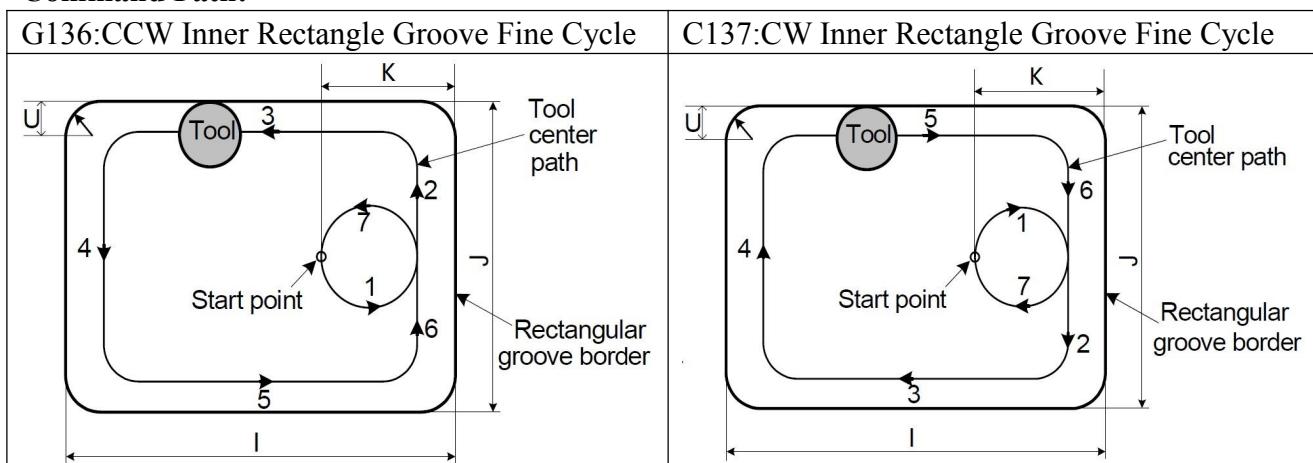
2. Width of rectangle in Y-axis, J, Range is -99999999~99999999 (*min unit). Its absolute value is used if it is negative. Value of J must be not less than diameter , otherwise alarm.

3. Distance from fine milling starting point to rectangle side in X-axis,K, is greater than radius of tool. Its absolute value is used if it is negative.

4. No corner transition if corner arc radius, U, is omitted. System will alarm when U is omitted or 0 & tool radius is less than 0.

5. Range of Tool radius offset number , D, is 0~32. default is 0.

Command Path:

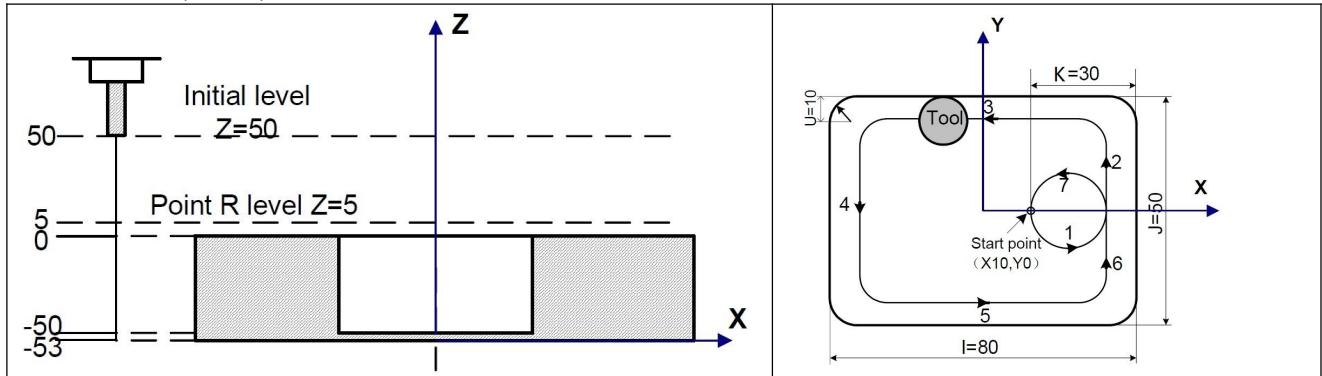


Process of Cycle:

- 1) Rapid positioning to the position in XY plane;
- 2) Rapid down to point R level ;
- 3) Cutting Feed to the machining start point at the bottom of hole ;
- 4) Perform arc interpolation by the transition arc 1 from the starting point;
- 5) Perform linear & circle interpolation by the path 2-3-4-5-6 ;
- 6) Perform circle interpolation by transition arc 7 and return to the starting point ;
- 7) Return to the initial level or point R level according to G98/G99.

Note: *Q, P, L* is invalid when using G112/G113, but value of *Q* & *P* will be remained as canned cycle.

Example: Fine milling a rectangle groove that has been rough milled as follows by canned cycle code G136. (D1=5).



G90 G00 X50 Y50 Z50 ; Rapid Positioning

G99 G24 X10 Y10 Z-50 R5 I80 J50 K30 U10 D1 F800 ; CCW fine milling rectangle

G80 X50 Y50 Z50 ; Cancel canned cancel and return to point R level

M30 ; End of program

Note: 1. When use G112/G113 is used, G codes in 01 group cannot be used, otherwise G114/G115 is replaced by other codes in group 1.

2. **Tool radius compensation:** in the fixed cycle command, the tool radius compensation is ignored, the system calls the tool radius compensation specified by the program during the tool infeed.

3.31.9 Outer Rectangle Fine Milling Cycle (G138/G139)

The tool performs fine milling outside the rectangle by the specified width & direction, and it returns after finishing the fine milling.

G138: CCW Outer Rectangle Fine Milling Cycle ; C139: CW Outer Rectangle Fine Milling Cycle .

X_Y_: The starting point in XY plane

Z_: The distance from point R to the bottom of the hole

R_: The distance from the initial level to point R level

I_: Width of rectangle in X-axis

J_: Width of rectangle in Y-axis

K_: Distance from fine milling start point to rectangle side in X-axis

U_: corner arc radius

D_: Tool radius offset number

F_: Cutting Feedrate

Note: 1. Width of rectangle in X-axis, I, Range is -99999999~99999999 (*min unit). Its absolute value is used if it is negative.

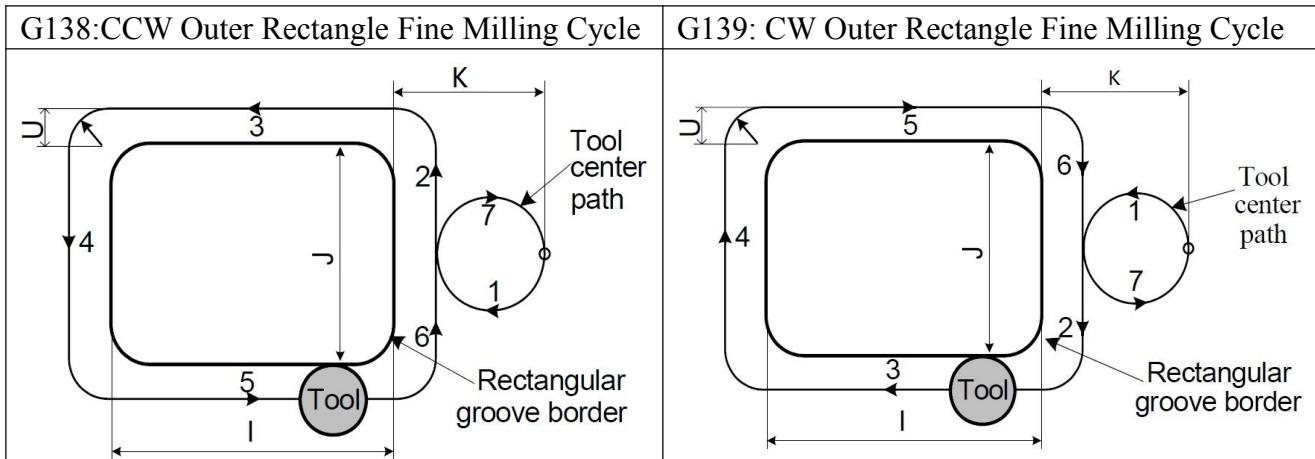
2. Width of rectangle in Y-axis, J, Range is -99999999~99999999 (*min unit). Its absolute value is used if it is negative.

3. Distance from fine milling starting point to rectangle side in X-axis, K, is greater than radius of tool. Its absolute value is used if it is negative.

4. No corner transition if corner arc radius, U, is omitted.

5. Range of Tool radius offset number, D, is 0~32. default is 0.

Command Path:



Command Path:

- (1) Rapid positioning to a location within XY plane ;
- (2) Rapid down to R level ;
- (3) Cutting Feed to the bottom of hole
- (4) Performs circular interpolation by the transition arc 1 from the starting point ;
- (5) Performs linear & circular interpolation by the path 2-3-4-5-6 ;
- (6) Performs circular interpolation by transition arc 7 and return to the starting point ;
- (7) Return to the initial level(G98) or point R level (G99) .

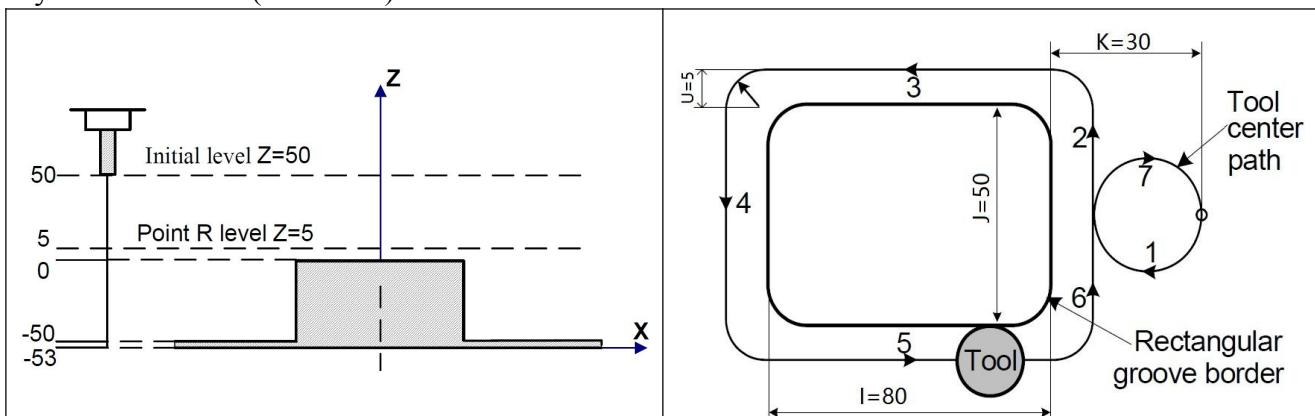
In outer rectangle fine milling cycle, the interpolation directions of the transition arc and fine milling arc are different. The interpolation direction in the code means the one of the fine milling.

Note: 1. Q, P, L is invalid when using G114/G115, but value of Q & P will be remained as canned cycle.

2. When use G114/G115 is used, G codes in 01 group cannot be used, otherwise G114/G115 is replaced by other codes in group 1.

3. Tool radius compensation: in the fixed cycle command, the tool radius compensation is ignored, the system calls the tool radius compensation specified by the program during the tool infeed.

Example: Fine milling a outer rectangle pad that has been rough milled as follows by the canned cycle code G138. (D1=5mm)



G90 G00 X50 Y50 Z50 ; Rapid positioning

G99 G138 X25 Y25 Z-50 R5 I80 J50 K30 U5 D1 F800 ; Outer rectangle fine milling

G80 X50 Y50 Z50 ; Cancel canned cycle and return from point R level

M30 ;

End of program

3.31.10 Tool Change for Linear Tool Magazine (G120)

This code is used to performs tool change for linear tool magazine in steps.

Format: G120 T_ ;

T_ : Number of exchanging tool.

a) Parameter Set: Before run this tool changing code, it needs to set related parameters well firstly of all. It needs to set P900=4 in Other parameter, and then reboot SZGH CNC system. Press “Enter” & “PgDn” key to enter dialog box.

Linear Tool Magazine 1/2		
1. Current Tool Number:	<input type="text" value="0"/>	
2. Quanity of Tools:	<input type="text" value="0"/>	
3. Position Speed of Z axis:	<input type="text" value="0"/>	
4. Position Speed_X&Y Axis:	<input type="text" value="0"/>	
5. Safe position of Z axis:	<input type="text" value="0.000"/>	
6. Position of Put Tool(Z):	<input type="text" value="0.000"/>	
7. Position of Take Tool(Z):	<input type="text" value="0.000"/>	
8. Time of Huff(M59):	<input type="text" value="0"/>	
9. If Spindle Orientation:	<input type="text" value="0"/>	
Linear Tool Magazine 2/2		
Tool_1 Coor X:	<input type="text" value="0.000"/>	Y: <input type="text" value="0.000"/>
Tool_2 Coor X:	<input type="text" value="0.000"/>	Y: <input type="text" value="0.000"/>
Tool_3 Coor X:	<input type="text" value="0.000"/>	Y: <input type="text" value="0.000"/>
Tool_4 Coor X:	<input type="text" value="0.000"/>	Y: <input type="text" value="0.000"/>
Tool_5 Coor X:	<input type="text" value="0.000"/>	Y: <input type="text" value="0.000"/>
Tool_6 Coor X:	<input type="text" value="0.000"/>	Y: <input type="text" value="0.000"/>
Tool_7 Coor X:	<input type="text" value="0.000"/>	Y: <input type="text" value="0.000"/>
Tool_8 Coor X:	<input type="text" value="0.000"/>	Y: <input type="text" value="0.000"/>
Tool_9 Coor X:	<input type="text" value="0.000"/>	Y: <input type="text" value="0.000"/>
Tool_10 Coor X:	<input type="text" value="0.000"/>	Y: <input type="text" value="0.000"/>
<input type="button" value="Enter"/> <input type="button" value="ESC"/> www.szghauto.com		
ShenZhen Guanhong Automation Co.,Ltd.		

Note: This coordinate value is position at machine coordinate system(G53).

In Other parameter: P13=1 ; set interlock between chuck and rotation of spindle

P20=0 or 1 ; set mode of chuck loose/tighten tool

In Axis parameter: P40_D1=1 ; set for detect position of chuck loose/tighten tool

b) IO ports

Signal	PIN	Function
M10	PIN21_CN3 Plug	Output for tighten tool
M71	PIN9_CN10 plug	Output for loose tool (when P20=1 in Other parameter)
M14	PIN24_CN10 plug	Detect position of tighten tool
M12	PIN11_CN10 plug	Detect position of loose tool
M61	PIN19_CN10 plug	Output for Spindle orientation
M22	PIN5_CN10 plug	Detect end of spindle orientation
M59	PIN6_CN10 plug	Output for huff/blower, to clear dust of tools

c) Steps of tool change

1) Z-axis go home(zero point of G53)	2) Spindle Orientation(it can be omitted)
3) Z-axis go to safe position(prepare to put tool)	4) XY-axis move to position of put tool
5) Z-axis down to position of put tool	6) Chuck loose tool & Huff
7) Z-axis up to safe position (prepare to take tool)	8) XY-axis move to position of take tool
9) Z-axis down to position of take tool	10) Chuck tighten tool
10) Z-axis go to safe position	Tool change is finished.

Note: 1. all coordinate value are absolute position at machine coordinate system

2. #301-#400 are taken up at control program for linear tool magazine, cannot be used to the other.

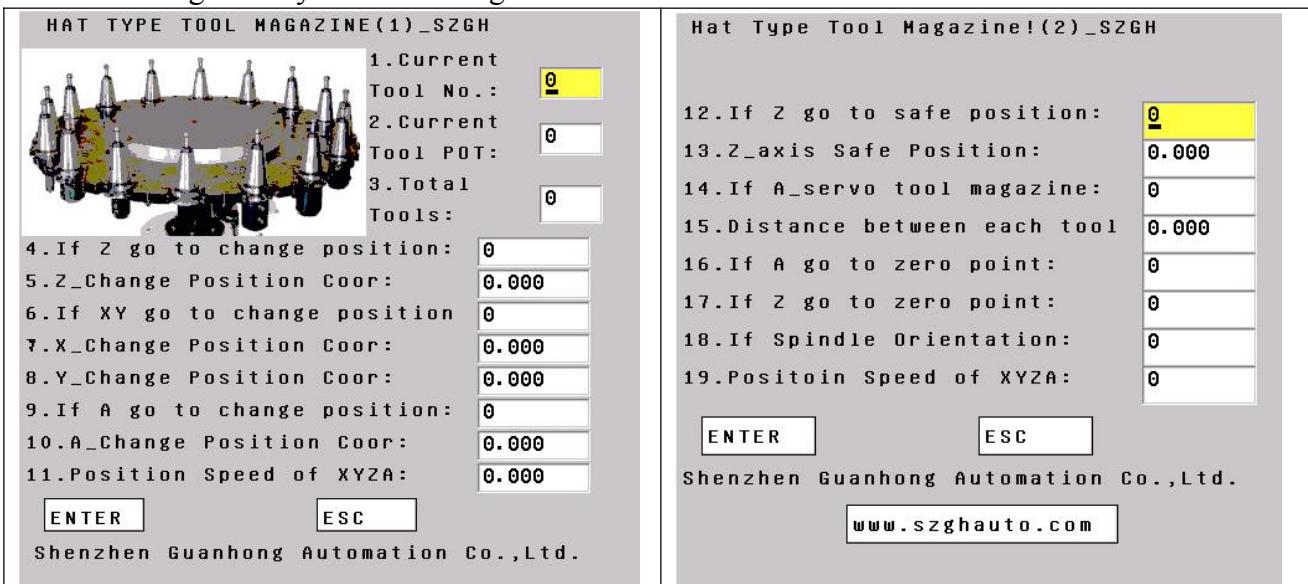
3.31.11 Tool change for Hat Tool Magazine (G121/G122)

This code is used to performs tool change for hat type tool magazine in steps & tool selection is set by ac motor(G121) or by servo motor(G122),which is controlled by A-axis.

Format: G121/G122 T_ ;

T_ : Number of exchanging tool.

a) Parameter Set: Before run this tool changing code, it needs to set related parameters well firstly of all. It needs to set P900=4 in Other parameter, and then reboot SZGH CNC system. Press “Enter” & “PgDn” key to enter dialog box.



Note: This coordinate value is position at machine coordinate system(G53).

1.Current Tool No.: It sets current tool number in spindle, if without tool in spindle, set to 0.

2.Current Tool POT: It sets current tool seat in tool magazine.

3.Total Tools: It sets total numbers, Range: don't over 20 tools.

Suggestion: When tools are over 20, please use our SZGH-CNC1000MDcb series CNC system.

4.If Z go to change position: It sets whether Z-axis go to tool change position, 1:Yes, 0:No.

5.Z_Change Position Coor: It sets Machine Coordinate value of tool change position in Z-axis.

6.If XY got to change position: It sets whether XY-axis go to tool change position, 1:Yes, 0:No.

7.X_Change Position Coor: It sets Machine Coordinate value of tool change position in X-axis.

8.Y_Change Position Coor: It sets Machine Coordinate value of tool change position in Y-axis.

9.If A go to change position: It sets whether A-axis go to tool change position, 1:Yes, 0:No.

10.A_Change Position Coor: It sets Machine Coordinate value of tool change position in A-axis.

11.Position Speed of XYZ: It sets the speed of XYZ that move to tool change position.

12.If Z go to safe position: It sets whether Z-axis go to safe position, 1:Yes, 0:No.

13.Z-axis Safe Position: It sets value of safe height in Z-axis(increment).

14.If A_Servo tool magazine: It sets if rotation magazine is controlled by A-axis, 1:Yes, 0:No

15.Distance between each tool: It sets distance between each tool.

16.If A go to zero point: It sets if A-axis go to zero point, 1: Yes, 0: No .

17.If Z go to zero point: It sets if Z-axis go to zero point , 1: Yes, 0: No .

18. If spindle Orientation: It sets if spindle do orientation, 1: Yes, 0: No .

19.Rotating Speed of A_Axis: It sets speed of rotating for magazine.

In Other parameter: P13=1 ; set interlock between chuck and rotation of spindle

P20= 0 or 1 ; set mode of chuck loose/tighten tool

In Axis parameter: P40_D1=1 ; set for detect position of chuck loose/tighten tool

b) IO ports

Signal	PIN	Function
X00	PIN1_CN4 plug	Detection tool magazine at UP position
X01	PIN2_CN4 plug	Detection tool magazine at Down position
X02	PIN3_CN4 plug	Detection of Tool seat(POT) in position for tool change
X03	PIN4_CN4 plug	Detection of SP-Orientation Finished
X04	PIN5_CN4 plug	Detection of position of safe height in Z-axis
X05	PIN6_CN4 plug	Detection of position of forwarder of tool magazine
X06	PIN7_CN4 plug	Detection of position of backward of tool magazine
X07	PIN8_CN4 plug	Counting of tools
+T	PIN12_CN4 plug	CW Rotation of Tool Magazine
-T	PIN13_CN4 plug	CCW Rotation of Tool Magazine
M10	PIN21_CN3 plug	Output for tighten tool
M71	PIN9_CN10 plug	Output for loose tool (when P20=1 in Other parameter)
M14	PIN24_CN10 plug	Detect position of tighten tool
M12	PIN11_CN10 plug	Detect position of loose tool
M61	PIN19_CN10 plug	Output for Spindle orientation
M22	PIN5_CN10 plug	Detect end of spindle orientation
M59	PIN6_CN10 plug	Output for huff/blower, to clear dust of tools

c) Steps of tool change

1) If P17=1, Z-axis go home(zero point of G53)
2) If P4=1, Z-axis go to tool change position, also coordinate value of P5
3) If P18=1, Spindle Orientation(it can be omitted when P18=0), output M61, detect M22
4) If P6=1, XY-axis move to tool change position, also coordinate value of P7 & P8
5) If P9=1, A-axis move to tool change position, also coordinate value of P9
6) Detect position of tool POT(X02), Magazine Forward(M73), detect position of forward(X05)
7) Spindle Chuck Loose tool(M11), detect position of loose tool(M12)
8_1) If P12=1, Z-axis lift with safe height(value of P19), detect position of safe height(X04)
8_2) If P12=0, Magazine down to safe position(M79), detect position of down magazine(X01)
9) Magazine rotate & select tool(Y18/Y19), detect counts of tools(T08)
10) Detect position of tool POT after selecting tool(X02),
11_1) If P12=1, Z-axis down with safe height(value of P19), detect position of tool change (-X04)
11_2) If P12=0, Magazine up to safe position(M59), detect position of up magazine(X00)
12) Spindle Chuck Tighten tool(M10), detect position of tighten tool(M12)
13) Magazine back to original position(M63), detect position of magazine_home(X06)
14) If P17=1, Z-axis go home(zero point of G53) & Tool change is finished.

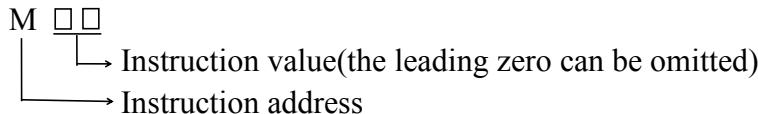
Warning: 1. all coordinate value are absolute position at machine coordinate system

2. #301-#400 are taken up at control program for linear tool magazine, cannot be used to the other.
3. Tool length compensation will be canceled after run exchanging codes G120/G121/G122.
4. System will enter Machine Coordinate System automatically after running G120/G121/G122

Chapter 4 M INSTRCUTIONS

4.1 M Function (Auxiliary Function)

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



When address M followed by a number is specified, a code signal and strobe signal are transmitted. These signals are used for turning on/off the power to the machine.

In general, only one M code is valid in a block but up to three M codes can be specified in a block (although some machines may not allow that). The correspondence between M codes and functions is up to the machine tool builder.

All M codes are processed in the machine except for M97 M98, M99,M codes for calling a subprogram , and M codes for calling a custom macro. Refer to the appropriate manual issued by the machine tool builder.

The following M codes have special meanings.

M00, M01, M02, M30, M97, M98, M99 must not be specified together with another M code.

Some M codes other than M00, M01, M02, M30, M97, M98 and M99 cannot be specified together with other M codes; each of those M codes must be specified in a single block.

4.1.1 Program Stop(M00)

Automatic operation is stopped after a block containing M00 is executed.

When the program is stopped, all existing modal information remains unchanged. The automatic operation can be restarted by actuating the cycle operation. This differs with the machine tool builder.

4.1.2 Optional Stop (M01)

Similarly to M00, automatic operation is stopped after a block containing M01 is executed. This code is only effective and program stop when input point M22(PIN5 of CN10 Plug) is valid

4.1.3 End of Program (M02,M30)

This indicates the end of the main program.Automatic operation is stopped and the CNC unit is reset.

4.1.4 Cycle of Program (M20)

Run program cycle , cycle time is set by P18 in User parameter.

4.1.5 Account of Workpiece(M87)

Number of workpieces will add one automatically as P10=0 in Other parameter.

4.1.6 Unconditional Jump (M97)

M97 P_ , jump to specified block number that specified by P.P4 said entrance line number with 4 field numbers specified program transfer main program.

Example: M97 P0120 , when executive this code, CNC will jump to “N0120” block and run.

4.2 Subprogram Configuration

There are two program types, main program and subprogram. Normally, the CNC operates according to the main program. However, when a command calling a subprogram is encountered in the main program, control is passed to the subprogram. When a command specifying a return to the main program is encountered in a subprogram, control is returned to the main program.

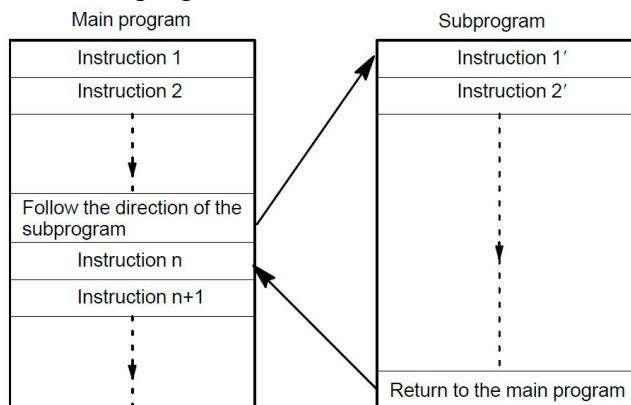


Fig4.2.1 Main program and subprogram

If a program contains a fixed sequence or frequently repeated pattern, such a sequence or pattern can be stored as a subprogram in memory to simplify the program.

A subprogram can be called from the main program.

A called subprogram can also call another subprogram.

4.2.1 Calling of Subprogram (M98)

This code is used to call a subprogram. The code and strobe signals are not sent.

M98 P_ L_ ;

P_ : specify address & name of subprogram. Eg.: Psub/1390; sub is a folder.

Subprogram can be hidden files that don't display in program district, First character of these program must be a "HIDEFILE".

Example: "HIDEFILE01", the subprogram in the program area is not displayed, user can use these type commands to call subprogram.

M98 PHIDEFILE01

or **M98 P*01**

If user want to call subprogram in USB-disk, Format is " P[_ " or " P] _ ".

Example: **M98 P[A1234** ; Call A1234 subprogram in USB-disk;

L_ : number of times the subprogram is called repeatedly. When no repetition data is specified, the subprogram is called just once.

M98 instruction can be omitted. Format: **PP_**.

Example: **PP|FFDE** ; call "FFDE" subprogram in USB-disk;

Note: 1. There must has a blank before "L_ " in this system;

2. Subprogram must be an independent program.

4.2.2 End of Subprogram (M99)

This code indicates the end of a subprogram. M99 execution returns control to the main program. No code or strobe signal is sent.

1) M99 in main program is same to M02;

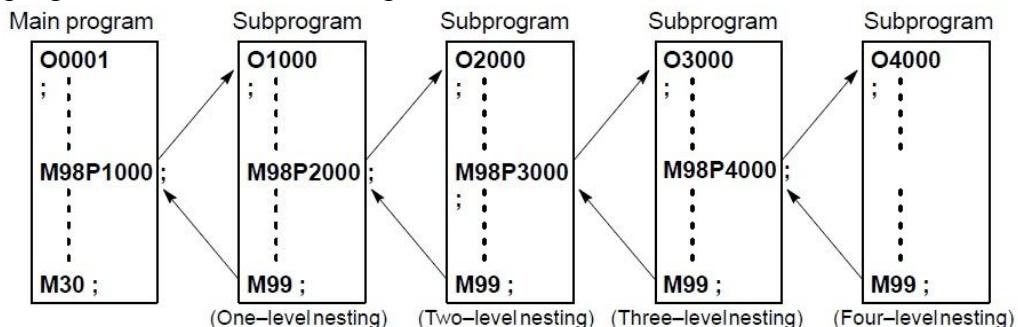
2) M99 with P in main program is same to M97 ;

3) M99 in subprogram return to next block of M98;

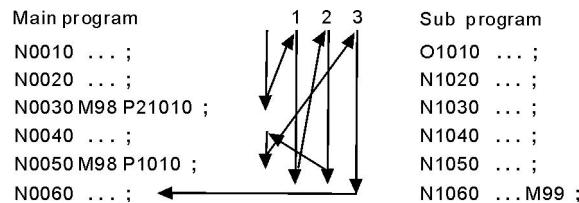
4) M99 with P in the subroutine return to specified block in main program.

When the main program calls a subprogram, it is regarded as a one-level subprogram call.

Thus, subprogram calls can be nested up to four levels as shown below.



Eg.: Execution sequence of subprograms called from a main program. A subprogram can call another subprogram in the same way as a main program calls a subprogram.



A single call command can repeatedly call a subprogram up to 9999 times.

4.3 Standard PLC M Command List

4.3.1 M Output Command List

Function	Code	Introduction	Statement
Spindle	M03	Spindle on CW	Functions interlocked and states reserved
	M04	Spindle on CCW	
	M05	Spindle Stop	
Coolant	M08	Coolant ON	Functions interlocked and states reserved
	M09	Coolant OFF	
Chuck	M10	Chuck Clamping	Functions interlocked and states reserved
	M11	Chuck Unclamping	
Lubricate	M32	Lubrication ON	Functions interlocked and states reserved
	M33	Lubrication OFF	
Huff	M59	Huff ON	Functions interlocked and states reserved
	M58	Huff OFF	
Orientation	M61	Spindle Orientation	Functions interlocked and states reserved
	M60		
1	M79	User-define Output 1 ON	Functions interlocked and states reserved
	M78	User-define Output 1 Off	
2	M63	User-define Output 2 ON	Functions interlocked and states reserved
	M62	User-define Output 2 Off	
3	M65	User-define Output 3 ON	Functions interlocked and states reserved
	M64	User-define Output 3 Off	

4	M67	User-define Output 4 ON	Functions interlocked and states reserved
	M66	User-define Output 4 Off	
5	M69	User-define Output 5 ON	Functions interlocked and states reserved
	M68	User-define Output 5 Off	
6	M71	User-define Output 6 ON	Functions interlocked and states reserved
	M70	User-define Output 6 Off	
7	M73	User-define Output 7 ON	Functions interlocked and states reserved
	M72	User-define Output 7 Off	
8	M75	User-define Output 8 ON	Functions interlocked and states reserved
	M74	User-define Output 8 Off	

M75/M74: When CNC system is configured with C axis, which is used for switching the control mode (Position control Mode & Analog Speed Mode) of servo spindle. When M75 is valid, which is set servo spindle to position control mode, when M03/M04 is valid, turn off M75.

Note: All M output commands, output 0V effective level.

4.3.1.1 Spindle Control (M03/M04/M05)

M03 is for control CW of spindle, M04 is for control CCW of spindle, M05 is for stop spindle.

Input Point

M03	PIN19_CN3 Plug	CW of Spindle
M04	PIN7_CN3 Plug	CCW of Spindle
M05	PIN20_CN3 Plug	STOP of Spindle
M203	2 nd Spindle on CW	<i>Note: Standard plc ladder is without these output for 2nd spindle, if need, we need to alter plc ladder.</i>
M204	2 nd Spindle on CCW	
M205	2 nd Spindle Stop	

Wiring Diagram

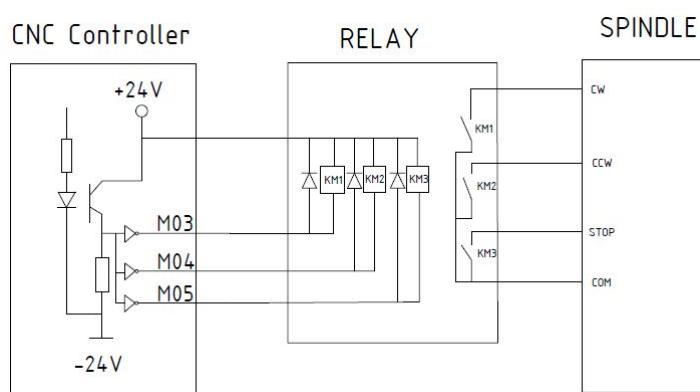


Fig4.3.1 Wiring Diagram for Spindle

According to this wiring diagram, it will consist of a control circuit with +24V when system output M03/M04/M05 signal, coil of relay works and NO type switches will be ON, and control related function of spindle.

Note: 1. Effective level of all output points is 0V.

2. When the relays and other inductance load, must be connected with the reverse diode to absorb the reverse current so as not to damage the system, if use the electromagnetic contactor, then plus resistive and capacitance spark circuit.

Parameter Set

System output dual analog for spindle for control speed of two spindles, 2nd analog output 1st analog, which set by D11_P9 in Other parameter.

In Axis parameter

P7 : Set the braking time of spindle, also the hold time of output M05, Unit: 10ms. The time less , the braking faster.

P8 : Set the braking signal is long signal 1 or short signal 0.

P9: To set system whether checking spindle feedback signal of spindle position, also the feedback signal is spindle encoder signal. To set the parameter value 1 means check; 0 means not to check.

P10: To set feedback pulse numbers of spindle encoder turn a round, the value: Line number of encoder * 4.

P11: Whether turn on the spindle or not when shifting [1 means on, 0 means off]

P51: The speed of motor when spindle shifting (unit: 1/100rpm)

P52: The direction when spindle shifting (0 means positive, 1 means negative)

P53: The stopping time when spindle shifting (unit: 10ms)

P54: Turning time of low speed when spindle shifting (unit: 10ms)

P55: Stopping delay time of spindle (Unit: 10ms)

In Speed parameter:

P8: To set the speed of spindle at manual condition. Unit: rpm.

P42: To set the max speed of spindle with 1st gear(M41), also the speed of corresponding 10V.

Attention: when spindle system is with gears, this is the speed of first gear.

P43: To set max speed of spindle(second gear/M42), Unit: rpm.

P44: To set max speed of spindle (Third gear/M43), Unit: rpm.

P45: To set the max speed of spindle (Fourth gear/M44), Unit: rpm.

P46: To set max speed of spindle on 2nd analog output , also speed of corresponding 10V.

In Other parameter:

P13: To set whether spindle and chuck is interlocking or not: 0 means they are separately; 1 means the spindle only start turning when chuck on. The thumbstall can't be use when the spindle is turning.

Setting parameter is related with the configuration of lathe and user's service condition, but consider for safe, suggest setting 1, also interlocking.

4.3.1.2 Spindle Gear Shifting(M41/M42/M43/M44)

Remark	PIN	Function	Command
S01	PIN10_CN3 Plug	Output 1st gear of spindle	M41
S02	PIN23_CN3 Plug	Output 2nd gear of spindle	M42
S03	PIN11_CN3 Plug	Output 3rd gear of spindle	M43
S04	PIN24_CN3 Plug	Output 4th gear of spindle	M44

M41/M42/M43/M44 output S01/S02/S03/S04 for shifting gear of spindle , and adjust analog voltage to adjust speed of spindle.

Note: Functions interlocked and states reserved

4.3.1.3 Coolant(M08/M09)

M08: Turn on coolant

M09: Turn off coolant

Remark	PIN	Function
M08	PIN8_CN3 Plug	Turn On/Off coolant

4.3.1.4 Lubricate(M32/M33)

M32: Turn on lubrication

M33: Turn off lubrication

Remark	PIN	Function
M32	PIN9_CN3 Plug	Turn On/Off Lubrication

In Other parameter,

P4 controls the function of lubricate automatically.

P6 is set the interval time of lubrication (Unit: s);

P5 set the time of lubrication ,also holding time of output M32(Unit: 10ms).

4.3.1.5 Chuck(M10/M11)

M10/M11 are for control clamping/unclamping of chuck.

Remark	PIN	Function
M10	PIN21_CN3 Plug	Control Chuck clamping
M71	PIN9_CN10 Plug	Control chuck unclamping(Spare)
M12	PIN11_CN10 Plug	Detect position of clamping_chuck(spare)
M14	PIN24_CN10 Plug	Detect position of unclamping_chuck(Loose)(spare)
M16	PIN12_CN10 Plug	Input point for switch to control chuck(spare)

Chuck of this system control is related with parameter as follows:

In Other parameter:

P2: Type of chuck, (Inner: Chuck to center when M10; Outer: Chuck opening outward when M10). 1 means outer, 0 means inner.

P13: Interlock between Chuck & Rotation_Spindle.0: No interlock, 1: yes.

M12: input point for detecting position of clamping of chuck;

M14: input point for detecting position of unclamping of chuck.

P20: Type of controlling signal for chuck,0: one output controlling signal for chuck; 1: two output controlling signals for chuck.

M10:output point for controlling clamping of chuck , controlling code is M10;

M71:output point for controlling unclamping of chuck,controlling code is M11.

P22: External Switch(Foot switch) for chuck, reciprocating mode, price one time clamping chuck; press twice time, unclamping chuck. 0: without switch ; 1: with external switch for control chuck; input signal is M16.

P24: Holding time of output M10/M71 of chuck, unit: s. 0: mode type*.

In Axis parameter:

P40_D1: Detect position of clamping/unclamping of chuck,0:No detect, 1: yes.

Note: 1. When user-defined signals M71/M70 , M73/M72 are used for output signal of spindle chuck and Tailstock, it can't be used for other functions.

2. M12,M14,M1,M18,M28 are multi-function codes, only one function when using.

3. *mode type means that once valid,always valid until canceling/resetting code.

4.3.1.6 Huff(M59/M58)

M59: Turn on huff M58: Turn off huff

Remark	PIN	Function
M59	PIN6_CN10 Plug	Turn on/off huff, also blower

4.3.1.7 Condition Output of Machine Tool(M65/M67/M69)

M65/M67/M69 be used for output condition of machine tool, set by parameters

Remark	PIN	Function
M65	PIN20_CN3 Plug	Output Alarm condition of machine tool
M67	PIN8_CN3 Plug	Output Running condition of machine tool
M69	PIN21_CN3 Plug	Output Pausing condition of machine tool

Parameter sets for condition output of machine tool. In Other parameter:

P28: M65/M69 are as outputs for run/halt condition of machine tool, 0: no, 1: yes.

P29: M67 is as output for alarm condition of machine tool, 0: no, 1: yes.

Note: 1. When M65, M67, M69 are used for output condition of machine tool, and then they can't be used for other functions.

2. Valid level of all output points is 0V. When fixed with inductance load, must connected with the reverse diode to protect inner circuit of CNC system.

4.3.2 M Input Command List

No.	Code	Function Introduction	Statement
1	M12	Check M12 is valid	
	M13	Check M12 is invalid	
2	M14	Check M14 is valid	
	M15	Check M14 is invalid	
3	M16	Check M16 is valid	
	M17	Check M16 is invalid	
4	M18	Check M18 is valid	
	M19	Check M18 is invalid	
5	M28	Check M28 is valid	
	M29	Check M28 is invalid	
6	M22	Check M22 is valid	
	M23	Check M22 is invalid	
7	M24	Check M24 is valid	
	M25	Check M24 is invalid	

Note: All M input commands, input effective level is 0V.

There are two kinds of special application about M input commands, as following,

a) Conditional Wait

Example: M12

When Input point M12 is valid, program goes on run following blocks, if M12 is invalid, system will wait if M12 is valid.

b) Conditional Jump

Example: M14 P0120

When the program running to this block and the system detecting if M14 input signal is valid. When M14 is valid, program will jump to 120th line of program (also N0120 block), otherwise, system will execute the next block.

4.4 Analog Speed of Spindle(S , SS)

SZGH CNC system support dual analog outputs for SP_speed.

Speed of 1st spindle is set by “S**”; Speed of 2nd spindle is set by “SS**”.

P36 in Speed parameter is set for max speed of 1st spindle; P40 in Speed parameter is set for max speed of 2nd spindle.

Note: D11_P9=1 in Other parameter is set for output 1st analog voltage to dual analog output(+10V of CN3&CN10) at same time, without function of “SS”.**

There are two gears control ways for 1st spindle as following,

(1) Four gears spindle speed electrical control, output four bits code of step speed change, M41-M44 control instructions corresponding to output S01-S04 code, with fixed speed. P50/P51/P52/P53/P54 in Axis parameter are set for mode of shifting.

(2) Four gears+Variable speed, M41-M44 instruction control, correspond the output S01-S04 code. P42/P43/P44/P45 in Speed parameter are set for maximum speed of corresponding gear, P50/P51/P52/P53/P54 in Axis parameter are set for mode of shifting.

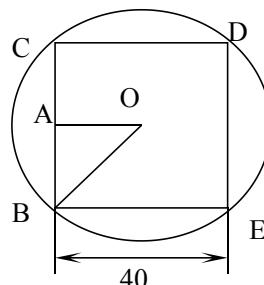
Variable speed,range is 0-99999, output 0-10V variable-frequency voltage.

Note: Output 10V is corresponding to max speed of spindle.

4.5 Comprehensive examples for programming

In the actual programming, must according to the drawings and processing requirements to select the install method and suitable tool correctly, combined with the actual working performance of milling to select the correct cutting allowance, for example:

Example 1: Cut square and cut circle



From the center to begin, the center coordinate is G54 X0 Y0 Z50

The tool radius in the D01 of the T01 parameter

N0 G54 G00 X0 Y0 Z50	Starting point for processing
N1 M03 S1000	Turn on spindle
N2 G00 Y-40 X-40 G90	Move outside of square
N3 Z10	
N4 G01 Z-10 F200	Start cutting from Z axis
N5 G01 G41 T01 D01 X-20 Y-20 F400	Move to B point
N6 Y20	Tool cuts BC line at the left of the workpiece
N7 X20	Closed angle transition in C point, cut CD line
N8 Y-20	Closed angle transition in D point, cut DE line
N9 X-25	Closed angle transition in E point,cut CB line is for smooth, move 5mm more
N10 G00 X-40 Y-40 G40	Move outside of circle
N11 G01 G41 X-20 Y-20 F500	Lengthen T01 value along A-B
N12 Z-20 F100	Start cutting from Z axis
N13 G91 G02 I20 J20 Y0 X0	Circle of contact
N14 G00 Z50	End cutting

N15 G40 G00 X0 Y0	Cancel tool compensation
N16 M05	Turn off spindle
N17 M02	Program is over

4.6 Usage for automatic tool setting gauge

1. Note for parameter:

Define macro variables of the automatic tool setting gauge function are as follows (corresponding to the other parameters P380 - P390):

P380: X_Machine coordinate value at initial position(ATS);(Unit:mm)
 P381: Y_Machine coordinate value at initial position(ATS);(Unit:mm)
 P382: Z_Machine coordinate value at initial position(ATS);(Unit:mm)
 P383: Speed in Negative direction(ATS);(mm/min)
 P384: Speed in positive speed(ATS);(mm/min)
 P385: Z_Workpiece coordinate value at surface of workpiece after ATS;(mm)
 P386: Positioning Speed (ATS);(mm/min)
 P387: Mode of Automatic tool setting (1: fixed point, 0: floating point).
 P388: Z_minimal machine coordinate value (mm);
 P389: The gap value of Z axis [The height which is the gauge surface relative to the workpiece surface(mm)];
 P390: Total number of tests for automatic tool set (1--5)

Fixed point gauge means putting the gauge in a fixed position, every-time the X Y Z axis are automatic running to the fixed point first in tool setting; But the floating point gauge search the tool setting gauge signal along negative of the Z axis.

The input point X25 is default to be the checking point of automatic tool setting gauge to input.
 2.The instruction: M880 (corresponding to ProgramUser0) automatic tool setting instruction; M882 (corresponding to ProgramUser2), M883 (corresponding to ProgramUser3) set the gap of Z axis.

3. Automatic tool setting steps:

- a)Set the P380--P388 parameter in other parameter;
- b)Set the P389 parameter in other parameter to set the gap of Z axis: this operation needs to be set only once.
 - A.Run M882 instruction in MDI to set the gap of Z axis;
 - B.Manual run Z axis to move the tool nose to the workpiece surface;
 - C.Run M883 instruction in MDI to automatic set the gap of Z axis P389 parameter in other parameter;
 - c) MDI choose the workpiece coordinate system G54/G59;
 - d) Automatic tool setting: MDI running the M880 instruction, automatic set the Z axis offset of the current workpiece coordinate system.

4.7 Usage for automatic dividing center

1. The X axis is divided center: M884(Corresponding to ProgramUser4)
 - 1) Choose the current coordinate system such as G54;
 - 2) Manually moving the X to the negative terminal of workpiece; MDI running the M884 instruction;
 - 3) Manually moving the Y to the positive terminal of workpiece; MDI running the M885 instruction, automatically divide the center of Y axis and set the middle point of workpiece as current coordinate system of Y axis origin, that's automatically setting the current coordinate system, such as the coordinate offset value of Y axis in G54.
2. The Y axis is divided center (Corresponding to ProgramUser5): M885
 - 1) Choose the coordinate system such as G54;
 - 2) Manually moving the Y to the negative terminal of workpiece; MDI running the M885 instruction;
 - 3) Manually moving the Y to the positive terminal of workpiece; MDI running the M885 instruction, automatically divide the center of Y axis and set the center point of workpiece as the current coordinate system of Y axis origin, that's automatically setting the current coordinate system, such as the offset value of Y axis in G54.

4.8 Usage of Following Function of Axes

Solution 1)

Coordinate doesn't following with control axis, it can don't display 4th axis when using.

On Axis Parameter,

P414=7 ; 4th axis is following with X-axis , synchronous to X-axis on Manual/Auto.

P414=8 ; 4th axis is following with Y-axis , synchronous to Y-axis on Manual/Auto.

P414=9 ; 4th axis is following with Z-axis , synchronous to Z-axis on Manual/Auto.

P414=17 ; C axis is following with X-axis , synchronous to X-axis on Manual/Auto.

P414=18 ; C axis is following with Y-axis , synchronous to Y-axis on Manual/Auto.

P414=19 ; C axis is following with Z-axis , synchronous to Z-axis on Manual/Auto.

P414=27 ; B axis is following with X-axis , synchronous to X-axis on Manual/Auto.

P414=28 ; B axis is following with Y-axis , synchronous to Y-axis on Manual/Auto.

P414=29 ; B axis is following with Z-axis , synchronous to Z-axis on Manual/Auto.

Solution 2)

Using command to specify one axis following with control axis on Auto status, don't follow on Manual status, coordinate also following with control axis.

M123 : Activate Y-axis following with motion of X-axis on Auto status;

M124 : Deactivate Y-axis following with motion of X-axis on Auto status

M125 : Activate A-axis following with motion of X-axis on Auto status;

M126 : Deactivate A-axis following with motion of X-axis on Auto status;

Chapter 5 Operation

When using CNC Milling system, just master the parameter of system, edit program, manual operation, auto operation. Then you can operate the system easily. There are some details to instruct hereinafter.

5.1 Operational Panel

This system panel is total controller ,which includes 8.4 inches LCD display area , function menu , editing keyboard area & machine control panel.

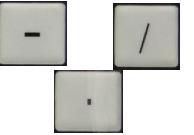
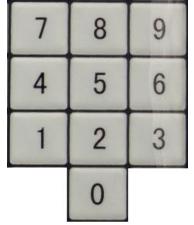


Fig5.1 SZGH-CNC990TDb CNC Milling Controller

5.2 Function Menu

Menu Keys	Comment
Parameter	Enter the interfaces of status parameter, data parameter , diagnosis and screw compensation parameter interface (interfaces can be switched by repeated press)
Program	Enter the program interface.
Redeem	Enter the redeem interface.
Diagnosis	Enter the interfaces of diagnosis
Handwheel	Enter controlling condition of handwheel
Manual	Enter controlling condition of manual
Auto	Enter controlling condition of auto
Setup	Setup current workpiece coordinate
Del	Delete key
Esc	Exit key
Enter	Enter key

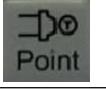
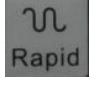
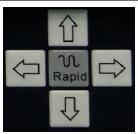
5.3 Editing Keyboard

Keys	Name	Description
	Reset key	CNC reset, stop of the feeding and moving, etc.
	Address Key & Symbol key	Address input, Double-address key, switch between addresses & symbol by repeated press
	Symbol key	Symbol Input
	Digital Key	Digit input
	Cursor Move Keys	Move the cursor in different directions.
	Shift Page	Page up/down on display <i>Note: Exchange of coordinate</i>

5.4 Machine Control Panel

Key	Designation	Explanation	Remarks and operation explanation
	Coolant	Coolant switch	Control code: M08/M09 Output Point: PIN8 of CN3
	Lubricate	Lubricate switch	Control code: M32/M33 Output point: PIN9 of CN3
	Chuck	Chuck switch	Control code: M10/M11 Output point: PIN21 of CN3
	Huff	Huff switch	Control code: M59/M58 Output point: PIN6 of CN10
	Single	Single block key	To enter single block mode

	Per Step mode key	To enter single step mode	Switching cycle from “manual continuous” to “manual increment”
	Graphic mode key	To enter graphic mode	
	Simulate switch/dry run switch		
	Pause key	Halt for program	
	Cycle start key	Press this key and the system mode automatically runs	Auto mode, MDI mode, DNC mode
	Return Home of Machine key	To enter mode of return home of machine	

Key	Designation	Explanation	Remarks and operation explanation
 	Spindle override keys	Up/Down rate of SP_Speed	Range is form 5% to 150%,16 gears totally
 	Handwheel override keys	Up/Down rate of handwheel	Rate: *100 / *10 / *1
 	Rate of G00 speed	Up/Down rate of G00_Speed	range is from 5% to 100%,16 gears totally
 	Rate of feeding speed	Up/Down rate of Feeding Speed	range is from 0% to 150%,16 gears totally
  	Spindle control keys	CW/CCW/Sto p of Spindle	Corresponding to M03/M04/M05 codes
	Rotation of SP in JOG	Point CW rotation of spindle	
	Rapid mode	Holding Rapid key+ Manual Feeding Key, for feeding with rapid speed manually. When P38=8 in Other parameter, the key is set to switch of Rapid/Normal.	
	Manual Feeding Key	For positive/negative movement of X, Y, Z and Nth axes in MANUAL mode	
	Emergency stop Driver and motor stop immediately, turns off the spindle, coolant, waits for the rise of button, and initializes values		
	Cycle Start Button, for run processing program on Auto, or run MDI		

Note: 1. "Manual speed controlled": Press "1" "2" "3" "4" "5" "6" "7" "8" "9" to set feed override "F30" "F60" "F120" "F250" "F500" "F1000" "F1500" "F2000" "F2500" "F3000".

2. "Incremental coordinate": Press "Setup" to set or set to 0 after enter "relative" coordinate mode.

3. Machine coordinate clear: Press "E" in parameter and then input password & press "Enter".

4. "F": Taking mm/min as the unit to set the manual feed speed, the input range is from 1 to 30000mm/min. And the input method according to data input method in parameter.

5. "S": Set the speed of the first spindle. The range is from 0 to 99999, the max depends on the No.36 parameter in speed parameter.

6. "I": Modify the increment per step in manual increment mode

7. "T": Choosing the position cutting tool.

The system adjusts one-level menu operation, intuitive, convenient, shortcut, prompt comprehensive information. Powering on system is to enter the main screen.

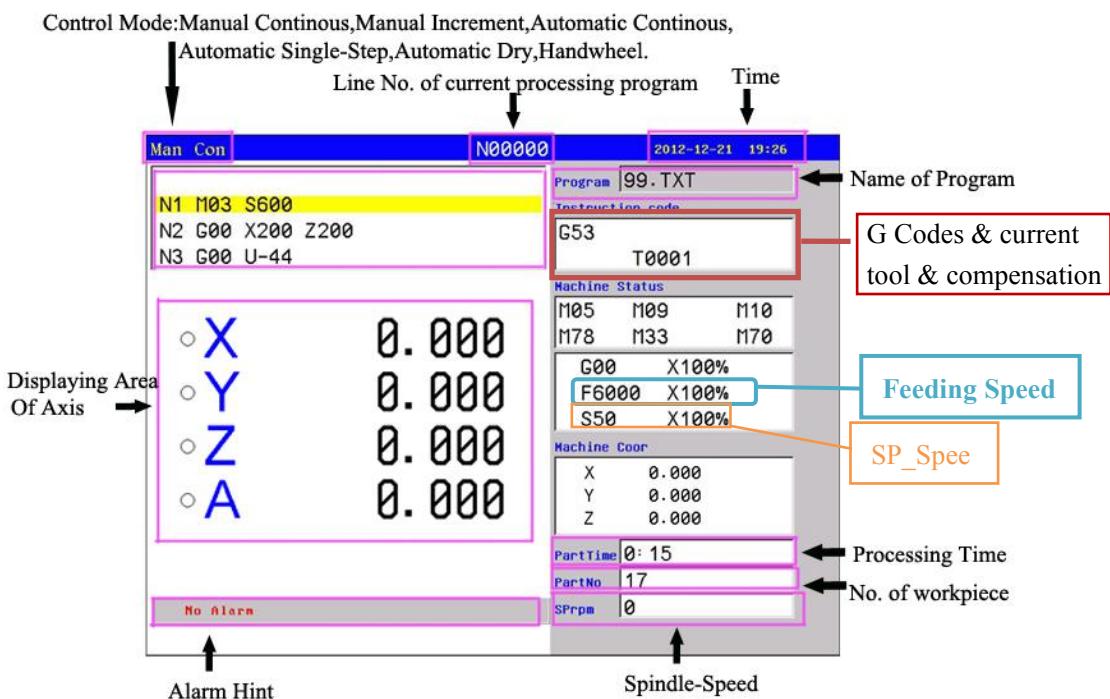


Fig5.2 Displayer

Press "Program" key enter program management area. It could edit, alter, diagnosis, delete, and copy etc.

Press "Parameter" key enter parameter management area. It could refer, alter to every parameters.

In manual condition, the system could process workpiece.

5.5 Manual Operation

5.5.1 Manual Continuous

Continuous operation is based on the time of pressing. Press to feed, up to stop feeding. Making sure the axis and using "Manual Feeding Key" to feed, feeding speed displays on the interface (F) multi Rate.

When continuous starting, press “Rapid” to switch the speed set by P1,P2 in Speed parameter, also G00 speed. If set the speed higher than the speed in parameter, the feed speed will be P1,P2 in Speed parameter times rapid override.

In order to facilitate the user single axis cutting in the manual function, setting the manual speed in manual status. Press “F” and input the speed.

When the hard limit point beyond positive and negative feed running axis two direction at, stop the feed and prompt to feed reverse direction.(the same as hereinafter)

The manual maximum speed is limited by P3 in Speed parameter, when setting the speed is higher than the value of parameter, then will be P3 in Speed parameter.

When P38=8 in other parameter, “Rapid” is change into a switch, press once to turn on (no more to always press), press again to turn off.

5.5.2 Manual Increment

This operation is to set the value of increment as the basis, press “ $\uparrow \downarrow \leftarrow \rightarrow$ ” once to run a value of increment. It will prompts “I=0010.000” in manual increment represent for the value of increment is 10mm, press “I” to revise and Enter.

The speed is the speed on display(F) times the rate.

5.5.3 Manual pulse generator(Handwheel)

There are two types of handwheel, one is handwheel in handheld box; the other is handwheel in the operational panel.

Handwheel in Handheld Box: Press “Handwheel” key to enter handwheel mode. User can operate the axes selection & feeding override of handwheel(*1/*10/*100).

Handwheel is mainly used for “Tool”, also posit tool.

Speed of handwheel pulse generator should be lower than 200r/min(100 pulses per cycle).

Parameters set for handwheel

P1 in Other parameter is set for position of handwheel.

In Speed parameter:

P23 is set for acceleration / deceleration time constant.

P24 is set for acceleration / deceleration time constant when simulate.

P25 is set max speed of X-axis with handwheel.

P26 is set max speed of Y-axis with handwheel.

P27 is set max speed of Z-axis with handwheel.

P28 is set max speed of A-axis with handwheel.

Note: 1. SZGH-CNC990MDb(c) series only support handwheel in handheld box.

2. Handwheel is no effect in auto-coordinates diagram machining, it only works in mode of coordinates.

3. When system is configured with stepper system, feeding speed shouldn't be too fast.



Fig5.3 Handheld Box(Manual Pulse Generator)

5.5.4 Manual Reference Position Return

The CNC machine tool has a position used to determine the machine position. This position is called the reference position, where the tool is replaced or the coordinate are set. Ordinarily, after the power is turned on or alarm/release emergency stop, the tool is moved to the reference position.

Manual reference position return is to move the tool to the reference position using switches and push-buttons located on the operator's panel.

In axis parameter, P38 is set for the grade of homing; there are 4 grades for homing. 0: Just hint; 1:No use; 8: Compulsion; 9: Super Compulsion;

Suggestion: *In order to ensure processing accuracy, it needs homing after reboot.*

There are two ways that return to reference position manually, one is that floating zero point, the other is that switch for homing, details as follows:

(1) Floating Zero Point Set

When user don't fix switches for Homing , user can use floating zero point as home, also reference position, also original point of machine coordinate system.

Parameters set: **In Axis parameter**,

P33: Bit parameter, D3:X ; D4:C(Y) ; D5:Z ; D6:A ; 1 means float zero point;

P34: Distance between reference position & current position in X direction;

P35: Distance between reference position & current position in Y direction;

P36: Distance between reference position & current position in Z direction;

P37: Distance between reference position & current position in A direction;

Steps of setting floating zero point(home):

1. Enabled function of float zero point of all axes(XZCA): P23= 01111001;

2. Moving each axes to designated position in order to set floating point easily.

3. Set distance between reference & designated position: P34/P35/P36/P37

Note: *If current position is home of machine coordinate system, no needs to set offset of P34/P35/P36/P37 for each axis.*

(2) Switch For Homing Set

User can fixed switches for homing,some parameters & wiring needs to done.

A. Parameters set: **in Axis Parameter**,

P33: Bit parameter, D3:X ; D4:Y ; D5:Z ; D6:A ; 0 means switch for homing;

P38: Grade of Homing,0: Just hint; 1:No use; 8: Compulsion; 9: Super Compulsion;

P39: Mode of Homing, 0 & 2: Detect Z0 signal; 1 & the others, no detect ;

P40: Bit Parameter, Direction & Sequence of Homing;

P41: Bit parameter, Type of switch for homing ;

P42: Range of detecting Z0 signal in X-axis ;

P43: Range of detecting Z0 signal in Y-axis ;

P44: Range of detecting Z0 signal in Z-axis ;

P45: Range of detecting Z0 signal in A-axis ;

P46: Offset after homing in X-axis

P47: Offset after homing in Y-axis

P48: Offset after homing in Z-axis

P49: Offset after homing in A-axis

In Speed Parameter,

- P34: X-axis homing speed
- P36: Y-axis homing speed
- P38: Z-axis homing speed
- P40: A-axis homing speed
- P35: Speed during detecting Z0 signal of X-axis
- P37: Speed during detecting Z0 signal of Y-axis
- P39: Speed during detecting Z0 signal of Z-axis
- P41: Speed during detecting Z0 signal of A-axis

Note: Details about parameters set, please check part of Parameters List(see chapter 2)

B. Input points for homing in the table

Input Point	PIN	Function
X0	PIN3 of CN3 Plug	X axis homing
Y0	PIN2 of CN3 Plug	Y axis homing
Z0	PIN17 of CN3 Plug	Z axis homing
M34/A0	PIN4 of CN3 Plug	A axis homing

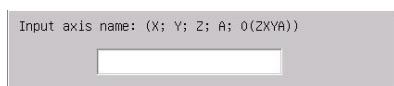
Note: 1. when system hasn't A axis, the input point is as function of M34.

2. Valid level of all input points is 0V, also common port is 0V or NPN type .

3. Wiring for homing, please see Appendix I Wiring Diagram of CN3 Plug.

C. Operation of Return Reference Position

Press “Return” in Manual mode, system will hint “Input axis name:(X; Y; Z ; A; 0(ZXYA))”, user can select one axis for homing alone, and also input “0” & Enter key , all axes go homing sequentially.



Note: 1. If user want to stop during homing, press “Emergency Stop” or “Reset” to stop.

2. After homing successfully, circle before coordinate will change to green, otherwise homing is failure.

The tool can be moved to the reference position also with program commands.

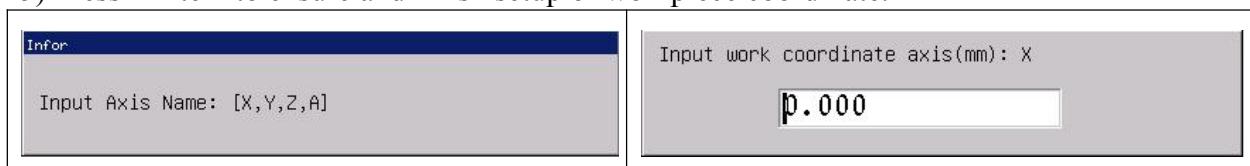
This operation is called automatic reference position return(See Chapter 3.10).

Special Attention: Every time to power up the system must back to zero point to make sure the accuracy of machine processing. The system power off unusually or in an accident, it must return to reference point ,also homing, otherwise could cause trouble.

5.5.5 Setup Workpiece Coordinate System

Steps of setup workpiece coordinate system

- 1) Press “M_{MDI}” key to enter MDI mode on Manual Condition
- 2) Input “G54 ~ G59” to select workpiece coordinate system
- 3) Move feeding axes that need to posit tool to specified position
- 4) Press “Setup” key , and select related axis and input current coordinate position
- 5) Press “Enter” to ensure and finish setup of workpiece coordinate.



5.6 Auto Operation

Auto refers to processing the editing program of workpiece. This system can start at arbitrary point, and also can start at arbitrary line or with arbitrary tool. Starting arbitrary line or with arbitrary tool must use absolute coordinate to edit the program. Press “Auto” to enter Auto mode in Manual mode. User can't move coordinate manually in Auto mode.

Select running program: Press “Program” key to enter program interface, press “ $\uparrow \downarrow$ ” to move cursor to a program which is going to run, press “C” key to select the program as processing program and switch to main screen automatically.(See Chapter 2. 4)

Switch display of coordinate: Press “PgUp”/ “PgDn” to switch the display which correspond to “Relative” “Absolute” “All”. & “Graphic”.

Relative Coordinate	Absolute coordinate	All coordinate	Graphic
• U 0.000	○ X 0.000	Relative U 0.000 V 0.000 W 0.000 A 0.000	Machine X 0.000 Y 0.000 Z 0.000 A 0.000
• V 0.000	○ Y 0.000	Work X 0.000 Y 0.000 Z 0.000 A 0.000	Distance to go X 0.000 Y 0.000 Z 0.000 A 0.000
• W 0.000	○ Z 0.000		
• A 0.000	○ A 0.000		

5.6.1 Automatic Processing Mode

“Single/continuous”: Press “Single” key to switch cycle.

“Continuous”: The program continue to execute every program segment(program line) to end or the instruction of stop to stop.

“Single ”: The program just execute one program line and end, wait another operation or press “Run” again to execute one next program line.

“Simulate”: The program is speedy simulate, the axis of coordinate can't move.

Table of Status Display about Manual/Auto

Manual Continuous	Manual Increment	Handwheel	Feed Hold
Man Con	Man Inc 1.000	Man PulsX *100	Man Con Keep
Auto Continuous	Auto Single	Simulate	
AutoCon	AutoStepStop	Imi Con Stop	

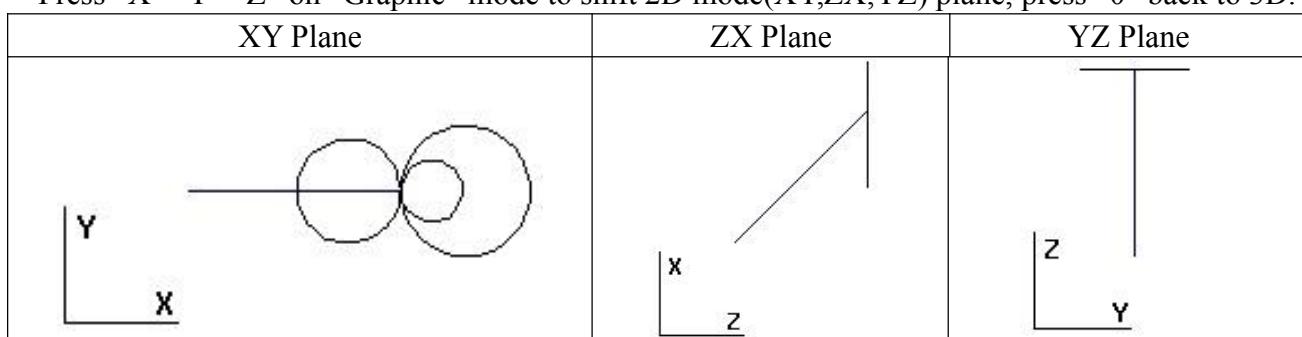
“Coordinate/Graphic”: Press “Graphic” in Auto mode to switch cycle.

“automatically coordinate”: The axis of coordinate will display with value.

“automatically Graphic”: The axis of coordinate will display with a figure.

“Feed Keep”, also feed hold, Press “Manual” twice on “AutoCon Run” condition or Press “Manual” on “AutoCon Stop” after running.

Press “X” “Y” “Z” on “Graphic” mode to shift 2D mode(XY,ZX,YZ) plane, press “0” back to 3D.



5.6.2 Processing at arbitrary program line or with arbitrary tool

5.6.2.1 Start from “nth” line(block)

At the condition of automatic processing, press “—” to pop-up a dialog box, import a number of line, press “Enter” to confirm, system will start program from this line, and display at processing program. No.1 line of program is 1th line, input number is nth line, as one block is one line.

Note: 1. The line is the actual line in the program, not the “N” stand for the line.

2. Firstly of all, system will move the starting point of “nth” block with speed which is set by P7 in Speed parameter, then run the program normally.

3. If user don’t input line number, CNC system will jump to the line that program stop last time, to facilitate user’s operations.

4. Press “Reset” to return back to the beginning of program after use “N” to search line number in coordinate screen.

5.6.2.2 Start from “N**” line

The system support that program can start from “N**” line(N** is 4 digit marked line). At automatic process condition, press “N” to popup a dialog box to import the marking line, press “Enter” to confirm. Press “Start” to run program at the “N**” line you import(mark).

*Note: 1. “N**” line is not “nth” line/block, is the “N**” stand for the line.*

2. Firstly of all, system will move the starting point of “nth” block with speed which is set by P7 in Speed parameter, then run the program normally.

5.6.3 Start Program

Start program must in the mode of “Auto”, press “Auto” key to enter mode of Auto, there are two methods to start program, as follows,

(1) Press “Start” key in the operational panel.

(2) Fix external switch to Run port(PIN18_CN3/PIN8_CN6/PIN9_CN11)

Note: PIN9_CN11 can be used as Run port, P33 in Other parameter is set for this function.

5.6.4 Halt Program

There are five methods to stop/pause program, as follows,

(1) Instructions of program: M00, M01, M02, M30, M20.

(2) Press “Single” to run a current block and stop.

(3) Interface switch turn to the middle or right side.

(4) Press “Reset” to stop all the actions of program.

(5) Fix external switch to Halt port(PIN6_CN3/PIN6_CN6/PIN10_CN11).

Note: PIN10_CN11 can be used as Halt port, P34 in Other parameter is set for this function.

5.6.5 Emergency Stop

Press “Emergency Stop” button when emergency accidents happening, the system will stop all the actions of machine tool and shows “Emergency stop” on screen.

User can fix external switch as Emergency Stop, Input point is PIN5 of CN11 plug. Type of emergency stop switch can be set by P27 in Other parameter.

After “Emergency Stop” during processing, which will affect difference between machine tool coordinate system & actual position of machine tool, in order to ensure coordinate system is same

to actual position of machine tool, it is best to do manual return reference position(homing) before processing.

5.6.6 Alarm

The screen hints alarm message when machine tool alarm, CNC system will stop processing. Only after clear alarm, and then CNC system can processing.

There are some fixed alarm, cannot be changed, as following

Alarm Hinting Message	Input Point
X-axis driver has happened hardware limit	-L(PIN15_CN3 Plug) +L(PIN16_CN3 Plug)
Y-axis driver has happened hardware limit	
Z-axis driver has happened hardware limit	
A-axis driver has happened hardware limit	
X-axis, Y-axis, Z-axis, A-axis driver is alarming	ALM(PIN12_CN5 Plug)
Spindle driver is alarming	ALM1(PIN5_CN3 Plug)
Emergency Stop	Stop(PIN5_CN11 Plug)

There are also some user-defined alarm as following

Alarm Hinting Message	Auxiliary Relay in PLC
No.0 Alarm	M80
No.1 Alarm	M81
No.2 Alarm	M82
No.3 Alarm	M83
No.4 Alarm	M84
Protect Door Is Open	M85
No.6 Alarm	M86
No.7 Alarm	M87
Loss of Lubricate Oil	M88
No.9 Alarm	M89
No.10 Alarm	M90
No.11 Alarm	M91
+5V Under-voltage	M92
+24V Under-voltage	M93
No.14 Alarm	M94
No.15 Alarm	M95

Note: these alarm hinting message & input point can be edited as user's asks on PLC.

Input point for protect door: M12, PIN11 of CN10 Plug

In Other parameter: P7: Detect switching signal of protective door, 0: no detect, 1: yes

P8: Type of switch for protective door, 0: NO type, 1: NC type.

P17: Type of alarm for servo driver, 0: NO type, 1: NC type.

P18: Type of alarm for spindle, 0: NO type, 1: NC type.

P19: Type of alarm for machine tool, 0: NO type, 1: NC type.

P26: Type of switch for emergency stop in panel, 0: NO type, 1: NC type.

P27: Type of switch for emergency stop in CN11, 0: NO type, 1: NC type.

Note: Emergency STOP: Press "Emergency STOP" button when appearing emergent accident, machine tool will stop all actions and the screen of system shows "Emergency STOP". Wait for releasing the button.

5.6.7 Indicator Light Output

Output Signal	Output Point	Parameter Set
Program Running	M69(PIN21_CN10 Plug)	P28=1 In Other parameter
Program Halt	M65(PIN20_CN10 Plug)	
Alarm	M67(PIN8_CN10 Plug)	P29=1 In Other parameter

Note: more details about indicator light output, please check Chapter 4.3.1.7.

5.6.8 DNC function

Storage room of SZGH CNC system is 128Mbit, user can adopt RS232-DNC or USB-DNC function to run the processing program that is greater than the remainder storage. RS232 port & USB port are in the front of CNC990MDb series controller.

5.6.8.1 RS232-DNC

1. Connect PC and CNC system well with the dedicated communication line , & set communication rate by P37 in Other parameter;
2. Use the dedicated communication software(SZGHCNCCS) on PC to set the related communication port and rate. Press “Transmit CNC”, select the program file to process linked, enter the status of sending program file.
3. To enter the interface of program file on CNC system, press "L" to enter the status of linked process, and program will display "RS232--DNC", press “Start” to running carry out linked process in the automatic status.
4. Turn “Interface switch” to middle or right to stop the running system in the process of linked process, press “E-Stop” or “Reset” to exit link of DNC.

Note: 1. Baud rate is related to operational environment when using serial port to send files.

2. The communication cable can't more than 10 meters length.

3. Only the dedicated communication software of this system can send program in User's computer. To set the sending speed of PC as the NC, defeat otherwise.

5.6.8.2 USB-DNC

USB-DNC is realized by U-disk, switch on U-disk and system, select program to execute in U-disk.

Press “B” to open U-disk in program interface, select corresponding program to press “C” to execute program, press “Auto” to get into automatic mode and press “Start”/ “Run” button to process the program.

Note 1. Don't unplug U-disk in the process of USB-DNC, otherwise failure.

2. Back to the system program interface from U-disk interface after finish USB-DNC.

3. After selecting the program, it is best to press “P” to compile once to make sure the program is right before executing program of USB-DNC.

5.7 External Electrical Connection

Basic I/Os of this CNC system is 40*24, some input points & output points have related functions.

5.7.1 Limitation

There are two ways to set limitation of machine tool , one is software limitation, another is fix external switch as limitation.

5.7.1.1 Software limitation

Software limitation is finished by setting working range of machine tool , also set related parameters in CNC system.

In Axis Parameter: P13: bit parameter, software-limitation of each axis is set alone.

Bit_P11	D7	D6	D5	D4	D3	D2	D1	D0
Axis			A	Z	Y	X		
Default set	0	0	0	0	0	0	0	1

1: it is invalid of software-limitation,0: valid of software-limitation.

- P3: Max Range in X-Negative Direction
- P4: Max Range in X-Positive Direction
- P5: Max Range in Y-Negative Direction
- P6: Max Range in Y-Positive Direction
- P7: Max Range in Z-Negative Direction
- P8: Max Range in Z-Positive Direction
- P9: Max Range in A-Negative Direction
- P10: Max Range in A-Positive Direction

Warning: when shift Metric/Inch, unit is changed(mm/inch), data for limitation don't change.

5.7.1.2 External Switch for limitation

Input Point of Limitation

Mark	Port	Explanation
-L	PIN15_CN3 Plug	Limitation in negative direction
+L	PIN16_CN3 Plug	Limitation in positive direction

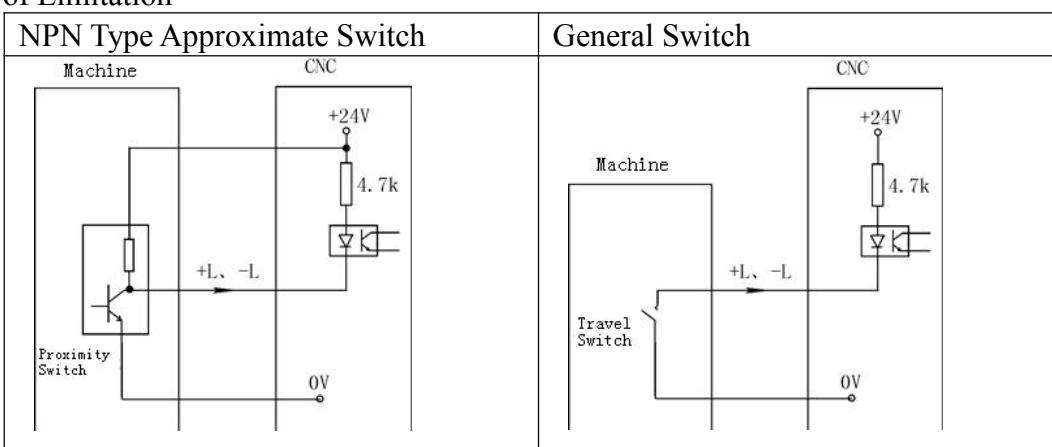
Type of Limitation Switch

In Axis parameter:

P21, Type of switch in Positive direction [0: NO type, 1: NC Type]

P22, Type of switch in Negative direction [0: NO type, 1: NC Type]

Wiring of Limitation



5.7.1.3 Suggestion Usage

Fix hardware limitation switch within the range of sets by software-limitation, P11=00000001; even if hardware switches don't work, software-limit also will work, double safe for limitation,

Note: 1. When limitation switch is NO type(normal open type) switch,use parallel connection; when limitation switch is NC type(normal close type) switch, use series connection.

2. CNC system could define X0&Z0 as limitation of X/Z axis. X0 signal as limitation and home of X axis, controlled by one switch. Z0 signal as the limitation and home of Z axis, also controlled by one switch. It needs

to restore our special PLC ladder into CNC system.

In Axis parameter:

P27, Type of Switch in positive direction, also for +L [0: NO type, 1: NC Type]

P28, Type of Switch in negative direction, also for -L [0: NO type, 1: NC Type]

5.8 Diagnosis

Press “Diagnosis” key to enter the diagnosis interface.

Press "T" & "PgDn/PgUp" or "↑ ↓" to check the status of inputs and outputs.

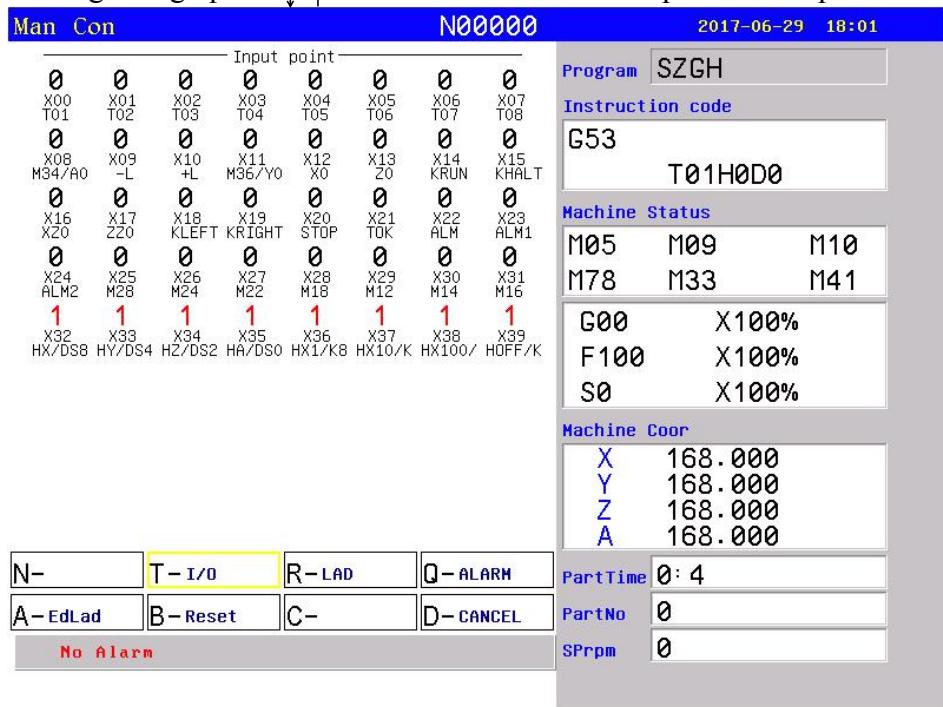


Fig5.8.1 System Diagnosis Interface(Input signal)

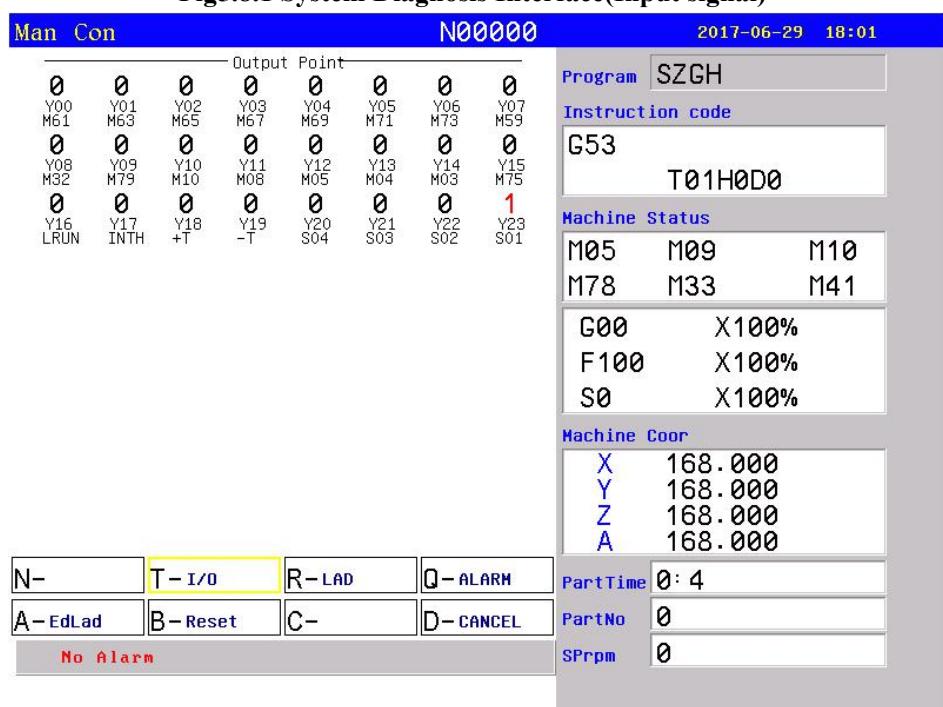


Fig5.8.2 System Diagnosis Interface(output signal)

In diagnosis interface of I/O , “0” means invalid status, “1” means valid status.
Press “R” key diagnosis screen to enter interface of check condition of PLC.

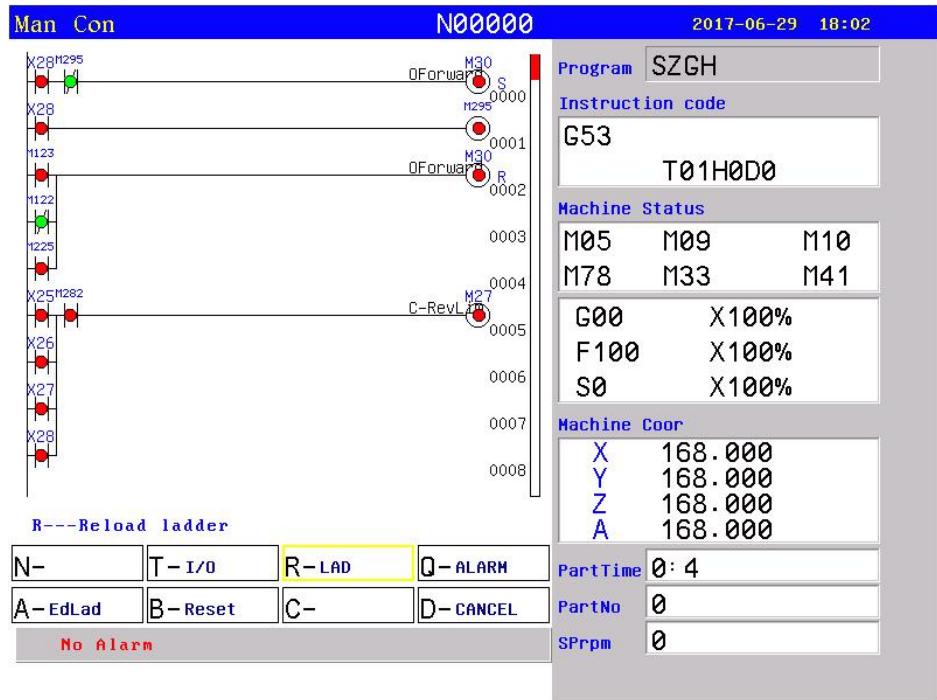


Fig5.8.3 Condition of Inner Register & I/Os

Press “PgDn”, “PgUp”, “Up arrow”, “Down Arrow” to check condition of inner registers & outputs & inputs.

“Green” means the register is valid, “Red” Means the register is invalid.

Press “A” key on diagnosis screen to enter interface of edit ladder of PLC.

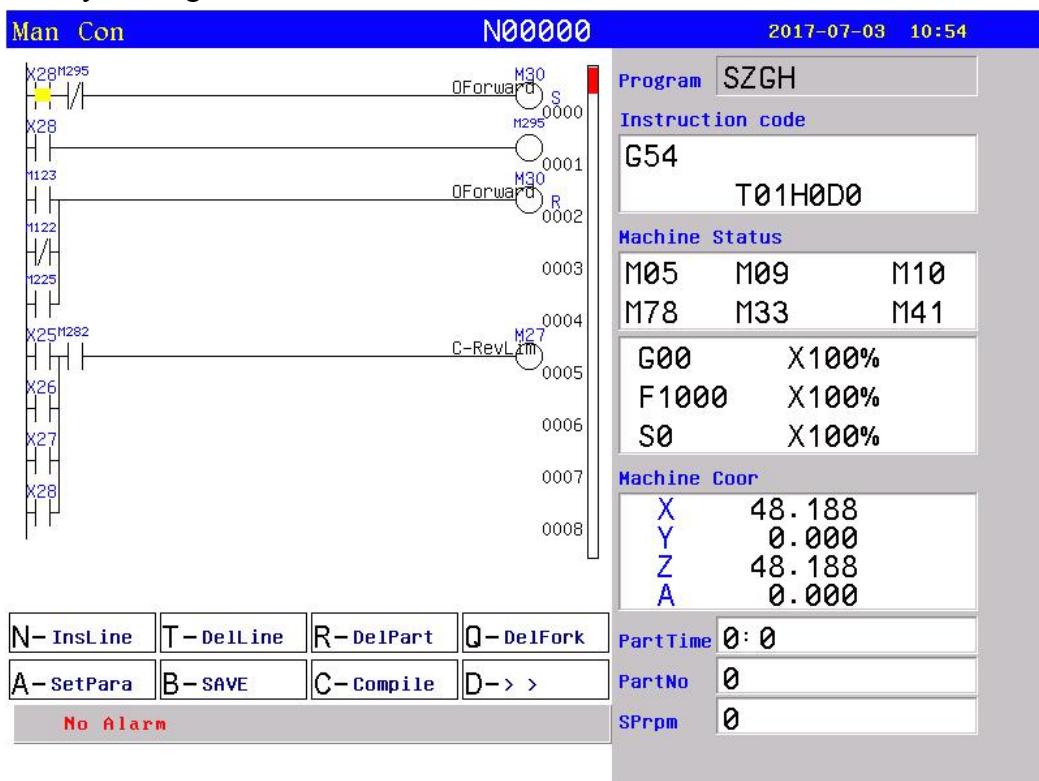


Fig5.8.4 Editing Screen of Inner Ladder

Press “S” key on these interfaces to activate search function. After finish ladder & save, it will work after reboot.

Press “R” key on condition screen of PLC, PLC will work immediately & no needs to reboot.

Note: when PI in Password parameter set to Disable, and then user can check & edit inner ladder.

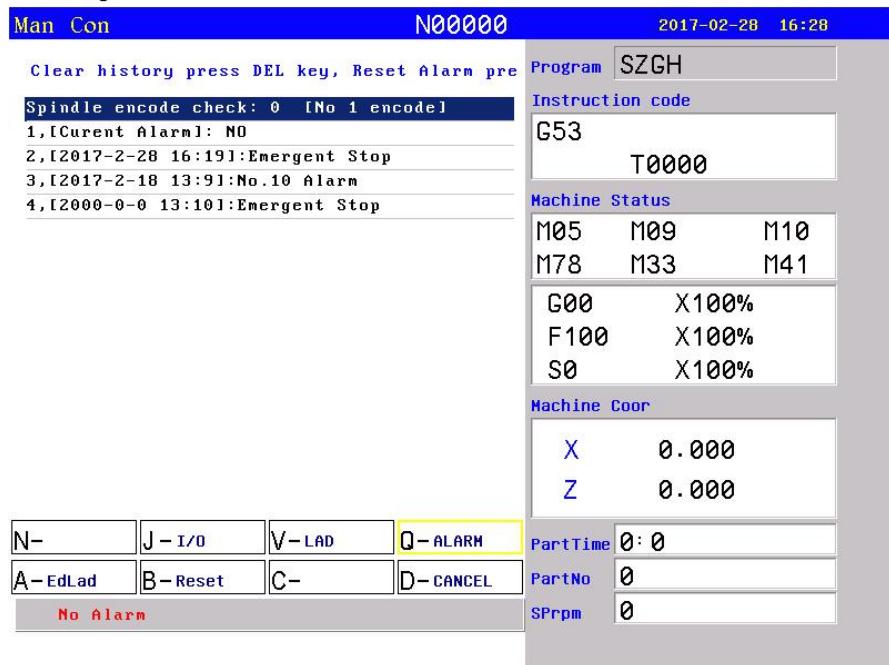


Fig5.8.5 System Diagnosis Interface(Alarm messages)

Spindle encode check:0 ,shows current resolution of SP_encoder;

1,[Current Alarm]:NO, shows current alarm condition of machine tool

Following shows history alarm messages.[Press Del key to clear alarm messages]

5.9 Programming Operation

Press “Program” to enter into status of Program as following.

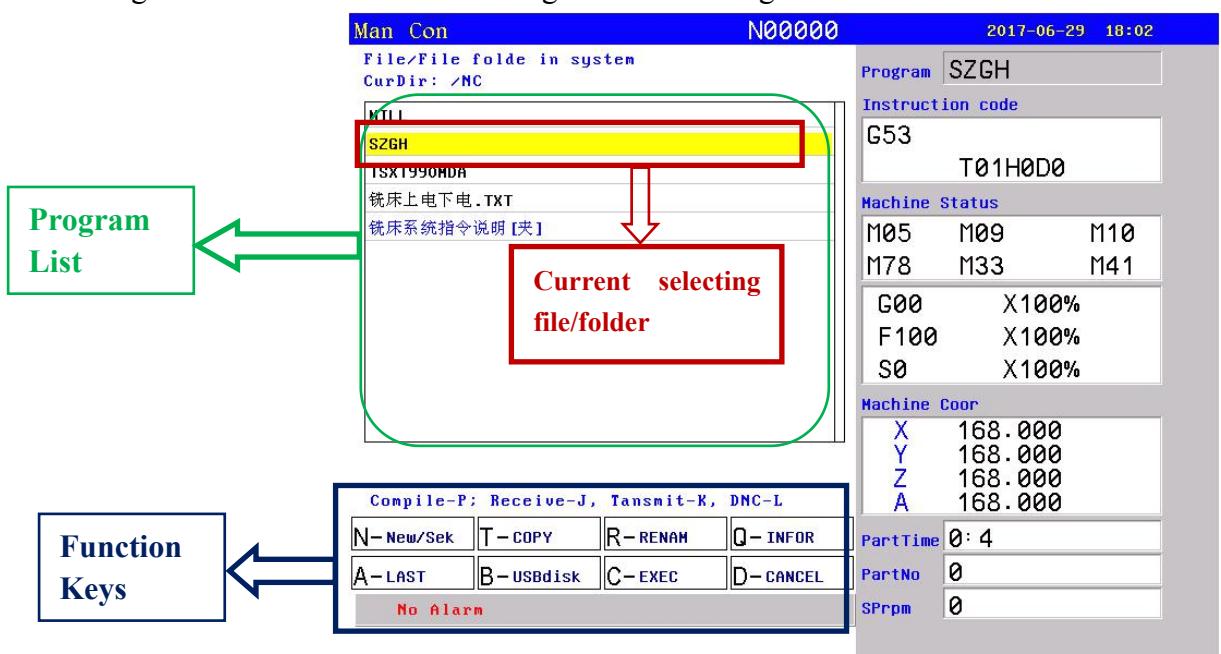


Fig5.9.1 Interface of Program

Management of program adopt mode of file/folder management, storage room of SZGH CNC system is 128Mb, there is no limitation about quantity of programs.

At program list, press “PgDn/PgUp” or \uparrow , \downarrow to select program/file and then press “Enter” to enter current program.

Name	Function
N-New/Sek	Press “N” key to New/Search a program
T-COPY	Press “T” key to Copy selecting program (System \leftrightarrow USB-disk)
R-RENAM	Press “R” key to Rename current selecting program
Q-INFOR	Press “Q” key to hints size of program & remain space of system
A-LAST	Press “A” key to return to last level
B-USBdisk	Press “B” key to open USB-disk
C-EXEC	Press “C” key to execute current program
D-CANCEL	Press “D” key to cancel or return
Compile-P	Press “P” key to compile current program
Receive-J	Receive file from PC with RS232-DNC
Transmit-K	Send file to PC with RS232-DNC
DNC-L	Open function of RS232-DNC between PC & CNC

5.9.1 Editing

Press “N” key and pop-up a dialog box to input the name of program, if the name is existing, the existing program is called up; If the name isn’t existing, the system will build a new program.

The name of program could be number, letter or mix, the length is 100 bits.

Build a new program or select a program and press “Enter” to entering the editing interface. Press “Rapid”+“C” key to shift to function interface.

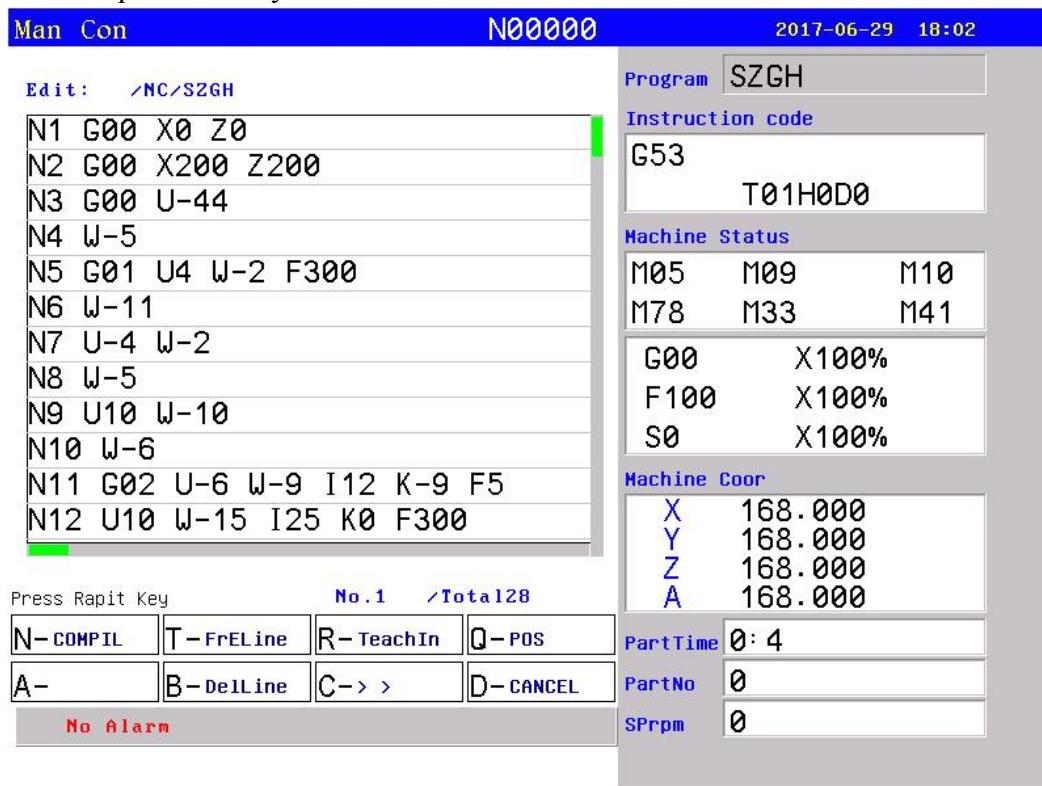


Fig5.9.2 (1)Editing interface of Program

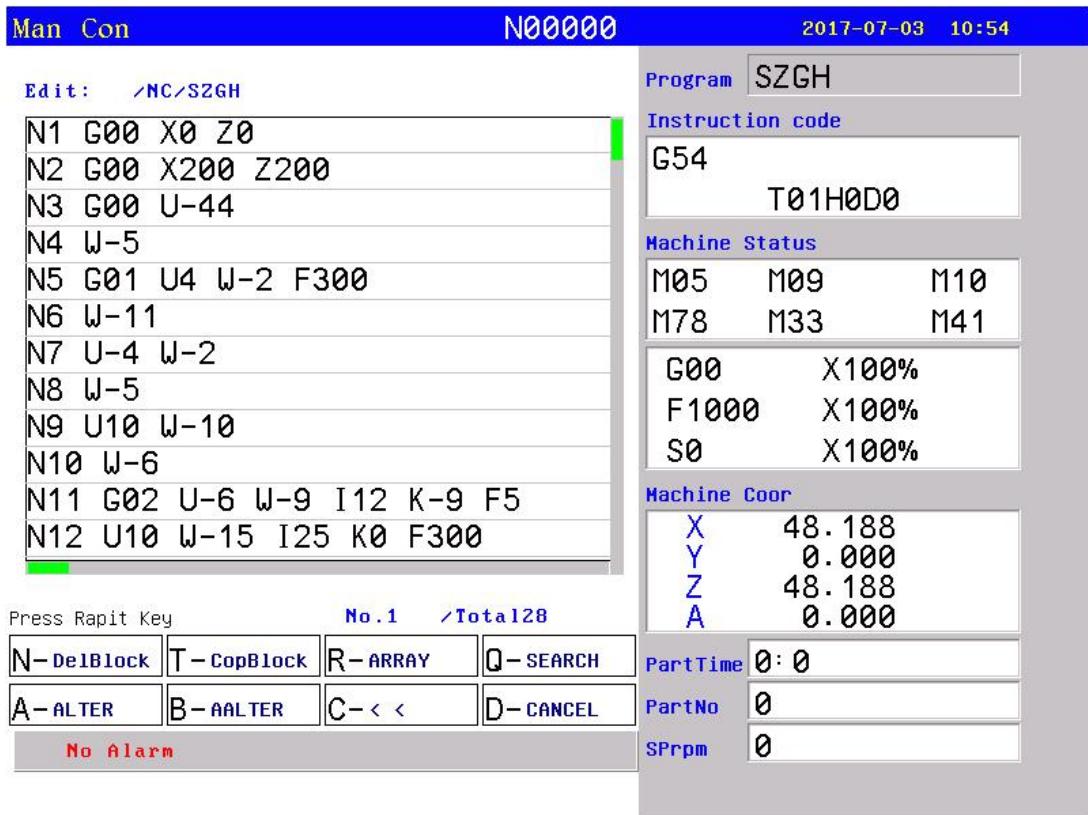


Fig5.9.3 (2)Editing interface of Program

Note: The name of all files don't allow same & blank.

The screen prompt the editing program name at the top left corner in the editing status; The left is the content, the right is the information for status, the operation in the editing status as follows:

(1) Location of current cursor:

Press “↑ ↓ ← →” to move the cursor to any position of program content

Press "PgUp" key to the last page.

Press "PgDn" key to the next page.

(2) Character Modification: Delete the character at the position of the cursor, then enter the new character.

(3) Character Insertion: Enter a new direct character at the cursor position. When the input is the letter, the letter in front of automatically generating space. If you want to enter a space, first enter a letter, and then delete this letter.

(4) Character Deletion: Press "Del" directly at the cursor position

(5) Inset a line: Press "Enter" directly, inset a line in front of the current line if the cursor is at the first line, otherwise insert a line after the current line.

(6) "Rapid" key Overlay operation:

First Function Keys	
Superposition	Function
Rapid + N	Compile current program
Rapid + T	Cursor jump to first line or last line
Rapid + R	Teaching function, enter status of Handwheel;Press X/Z/Y/C/A key to shift selecting axis, and press “Rapid”+ “Q” to read & set value of current coordinate system.

Rapid + Q	Cursor jump to the specified line
Rapid + A	Null
Rapid + B	Delete current block.
Rapid + C	Shift first function keys & second function keys
Rapid + D	Cancel
Second Function Keys	
Rapid + N	Delete specified blocks from current line to input line
Rapid + T	Copy specified blocks from input begin line to input last line
Rapid + R	Array all blocks of current program
Rapid + Q	Search specified characters from cursor line to end line
Rapid + A	Replace 1st specified characters from cursor line to end line
Rapid + B	Replace all specified characters from cursor line to end line
Rapid + C	Shift first function keys & second function keys
Rapid + D	Cancel

5.9.2 Copy

Press “↑ ↓” in main interface of Program, to select the program which need to copy and press “T” to popup a dialog box to import a new name of program, to copy which is the same content but different name , in order to modify, rename and spare.

5.9.3 Delete

Press “↑ ↓” in main interface of Program, to select program which need to delete and press “Del” key to delete the program.

Note: The operation of delete need to be careful, it can't be recovery once deleted.

5.9.4 Rename

Press “↑ ↓” in main interface of Program, to select program which need to rename and press “R” to popup a dialog box to import a new name.

5.9.5 Information

Press “↑ ↓” in main interface of Program, to select program which need to check and press “Q” to popup a dialog box to check the size of current program and remain space of the system.

5.9.6 Compile

Press “↑ ↓” in main interface of Program, to select program and press “P”, the system will check the format and grammar of program. Prompting when finding mistake automatically.

5.9.7 Folder management

User can build a folder in this system, Press “N” in main interface of Program to import a name of folder ,add “.” , and press “Enter”to build a folder and it will prompt a “[夹]” after the name.

Note: the name of folder must be different to name of other file/folder,otherwise failure.

Move the cursor to the folder and press “Enter” to open folder , user also can build a new file or folder in this folder.

Press “A” go to the last folder, also upper grade.

Move the cursor to the folder and press “Del” to delete the folder.

5.9.8 Execute Program

Press “↑ ↓” in main interface of Program to select a program and press “C” to select the processing program and switch into main interface of CNC system.

5.9.9 Communication

The system could deliver files with RS232 serial protocol .

There are two communication port for RS232 at front & rear of panel.

Remark	PIN	Function
RXD	PIN2 of CN6 Plug(rear)/Front DB9 Port	Receive Date
TXD	PIN3 of CN6 Plug(rear)/Front DB9 Port	Send Date
0V	PIN5 of CN6 Plug(rear)/Front DB9 Port	Ground

Delivery (Transmit)

Deliver the selected program in this system to another system or to PC for save. Press “↑ ↓” in main interface of Program to select the program and press “J” to deliver, press “Reset” to interrupt delivery.

Reception

Receive the selected program in another system or PC (Must be text file form). Press “K” to import a name of received program into the dialog box in main interface of Program , press “Reset” to interrupt reception.

Note: 1. Using the exclusive communication software to deliver program in User's PC.

2. The rate of deliver of both PC&CNC must be the same,otherwise failure easily.P37 in Other parameter is set the rate of CNC system.

3. The length of RS232 can't over 10 meters.

4. The number of serial port must be the same as the system setting.

5. Editing program of PC must be text file form.

5.9.10 U-disk management

To exchange files of parameter or programs with other system or PC by U-disk. It also can upgrade or back-up the software or parameter in system.

Note: : The name of folder can't have blank symbols.

Suggestion: Please prepare special USB-disk for manage files for CNC system.

Press “B” to enter the U-disk in main interface after U-disk is connected to USB port in the front of panel. Press “B” again to back interface of system.

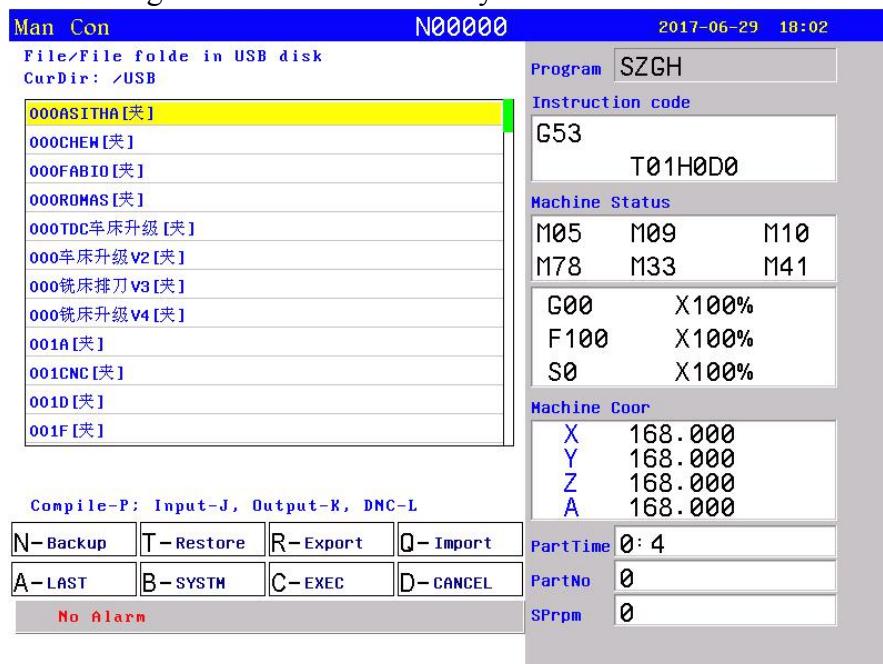


Fig5.9.4 Main Interface of USB-disk

5.9.10.1 Function Keys of USB-disk

Name	Function
N-Backup	Press “N” key to backup files of system to current directory of U-disk
T-Restore	Press “T” key to restore files at current directory of U-disk into system
R-Export	Press “R” key to export/copy file of system to U-disk
Q-Import	Press “Q” key to import/copy current file of U-disk to system
A-LAST	Press “A” key to return to last level
B-SYSTM	Press “B” key to return back to system, also exit USB-disk
C-EXEC	Press “C” key to execute current program at U-disk
D-CANCEL	Press “D” key to cancel or return
Compile-P	Press “P” key to compile current program
Input-J	Press “J” key to copy all files of U-disk to system
Output-K	Press “K” key to copy all files of system to U-disk
DNC-L	Open function of RS232-DNC between PC & CNC

5.9.10.2 Management of Processing Program

Copy the files or folder of U-disk into system

After connecting U-disk, press “B” to enter the U-disk directory in main interface of Program. Press “↑ ↓” to move cursor to select file or folder to copy and press “Q” to popup a dialog box to import name, press “Enter” to confirm. If there is the same name of program in the system, it will popup a dialog box to ask if cover the file or folder or not.

Press “J” to copy all the program in U-disk into system.

Copy the files or folder of system into U-disk

Press “↑ ↓” to move cursor to select file or folder that needs to copy to U-disk, and then press “B”, press “V” to popup a dialog box to import name in U-disk interface and press “Enter” to confirm. If there is the same name of program in the system, it will popup a dialog box to ask if cover the file/folder or not.

Press “K” to copy all the program in system to USB.

Note: 1. It must return to program directory of system, also exit U-disk by press “B” key before unplugging U-disk, otherwise the date which is copied just now will be lost.

2. The name of folder can't have blank symbol when using U-disk.

5.9.10.3 Management of Parameters & Software

User can use U-disk to deliver parameters files , system software , for upgrade and renew, back-up files and parameters of CNC system

A) Backup inner files & PLC files of system to U-disk

Note: Prepare a special empty U-disk for manage parameter files & PLC files of system better as

Parameter files is lots of about several dozens, Or setup a folder in U-disk on your computer firstly, open the folder of U-disk on system before backup parameter files & PLC files into the folder of U-disk.

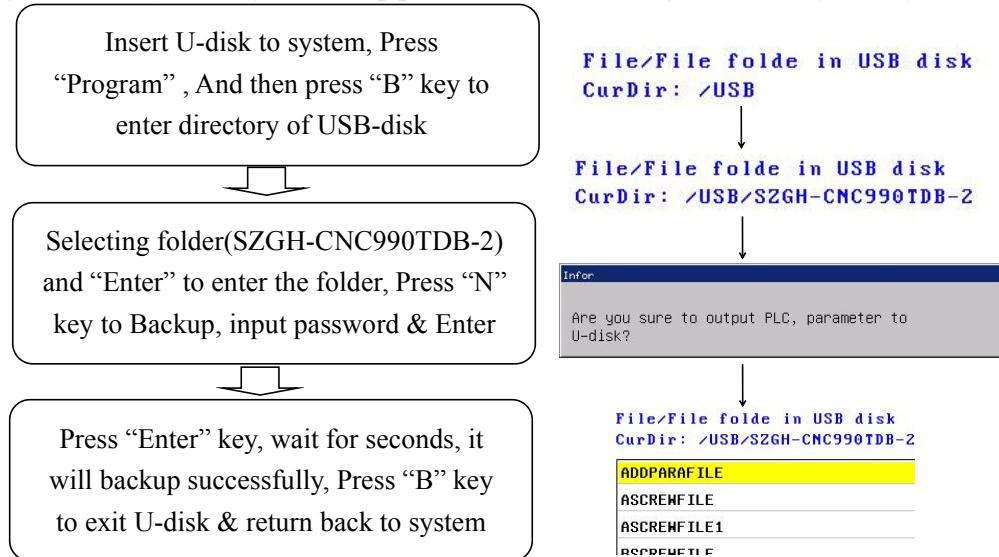


Fig5.9.5 Steps of Backup PLC & Parameter to U-disk

B) Restore parameters & PLC files into system with U-disk(Upgrade)

Note: Please put parameters & PLC for upgrade to a folder, which is better to avoid restore wrong files in U-disk into CNC system, and result to damage inner files & system.

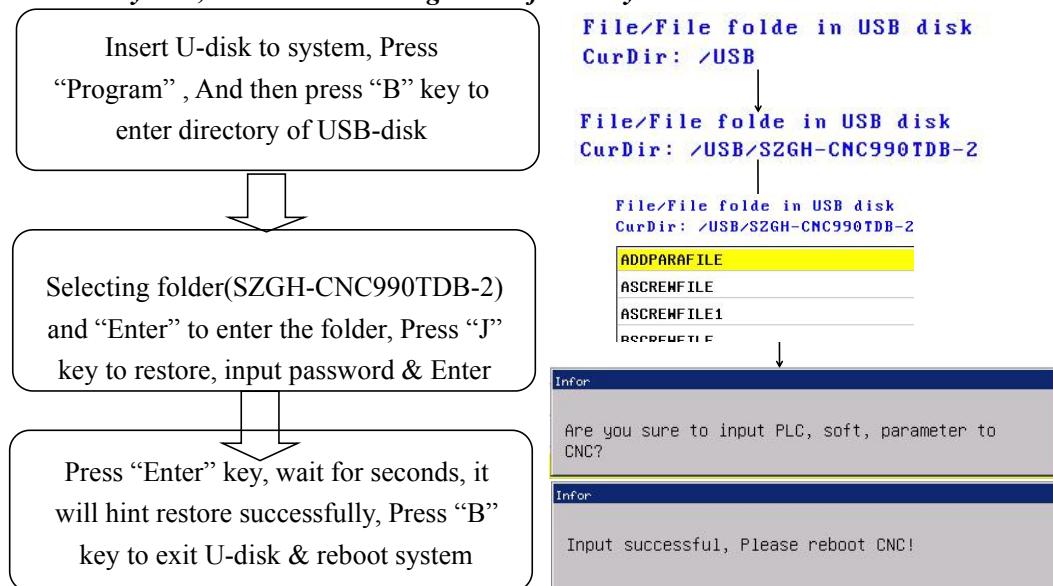


Fig5.9.6 Steps of Restore PLC & Parameter to U-disk

Chapter 6 Parameter List

At any status conditions, press “Parameter” to enter interface of parameter.

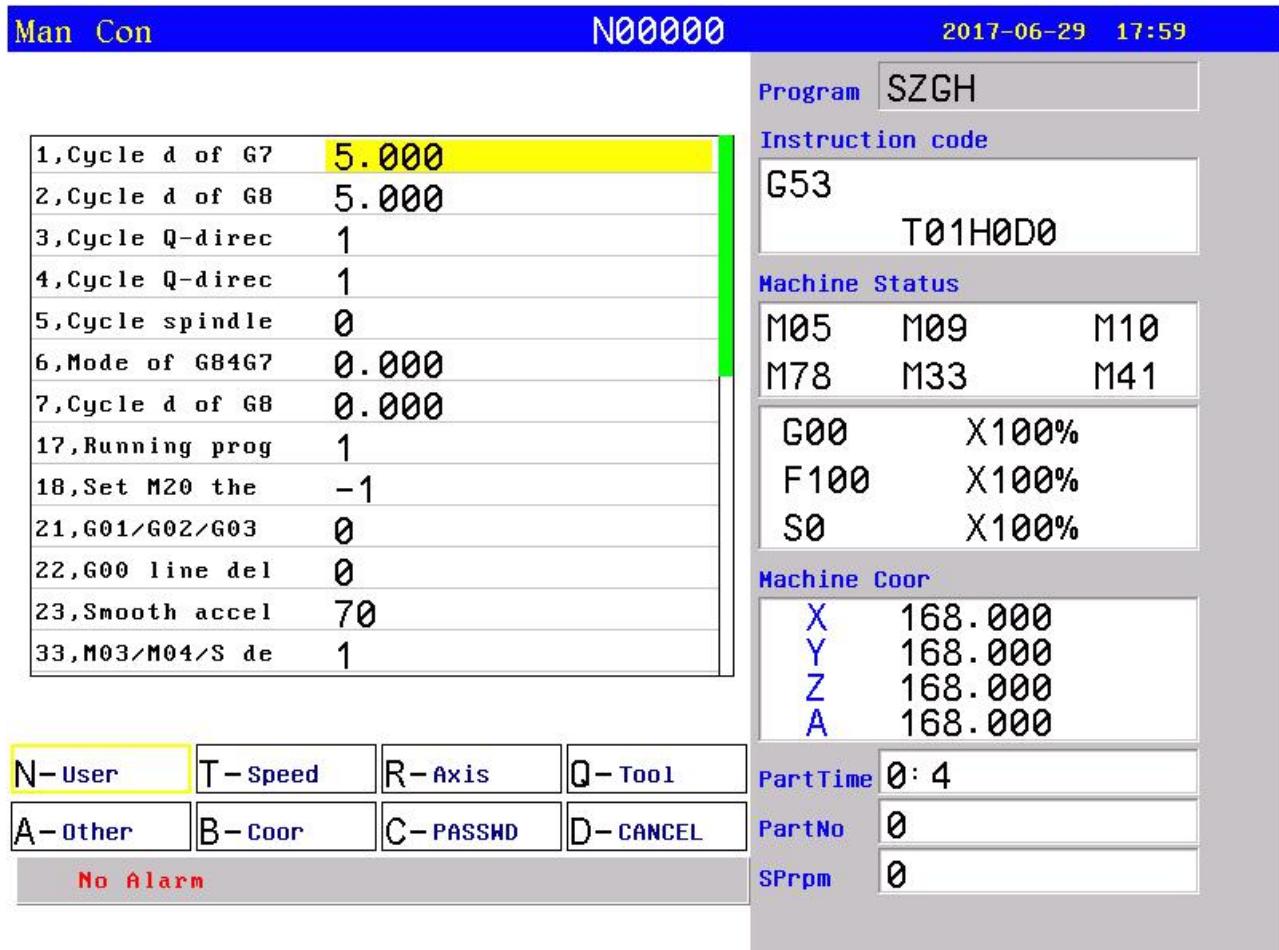


Fig6.1 Parameter List

Remark	Function
N-User	Press “N” key to enter User Parameter screen
T-Speed	Press “T” key to enter Speed Parameter screen
R-axis	Press “R” key to enter Axis Parameter screen
Q-Tool	Press “Q” key to enter Tool Parameter screen
A-Other	Press “A” key to enter Other Parameter screen
B-Coor	Press “B” key to enter Coordinate Parameter screen
C-PASSWD	Press “C” key to enter Password Parameter screen
D-CANCEL	Press “D” key to exit parameter list

After enter Parameter interface & select kind of parameter list, press “Down arrow”, “Up arrow”, “PgDn”, “PgUp” to select one parameter that need to alter, Press “Enter” enter key , popup dialog box, after alter well, press “Enter” for ensure parameter is set well.

Note: 1. Yellow cursor means current Parameter kind & Parameter Number.

2. The version of Parameter List is V21.04.

6.1 User Parameter

P	Parameter	Ex-Value
1	(d)Escaping Amount of G73 (mm)	5.000
2	(d)Escaping Amount of G83 (mm)	5.000
3	Direction of shift amount Q in G76 [G17](1:+X,2:-X,3:+Y,4:-Y)	1
4	Direction of shift amount Q in G87 [G17](1:+X,2:-X,3:+Y,4:-Y)	1
5	Stopping Angle when Spindle orientation at G76/G87 (0.1degree)	0
6	Mode of clearance in G74/G84(0:Return with d; 8: Return to R)	0
7	(d)Escaping Amount of G74/G84 (mm)	0.00
8	Pause Time Before Raise Up Tool at G73G83(ms)[Abs>5,-:G01;+:G00]	0
17	Interlock between Running Program & SP-Rotating [1:Yes, 0:No]	1
18	Times of auto-cycle running(M20)[<0: Endless Loop]	-1
21	Dwell between G01/G02/G03 blocks(ms)[>100]	0
22	Dwell between G00 blocks (ms)[>100]	0
23	Acceleration/Deceleration Constant of Handwheel [50-100]	70
33	Type of Detect SP_Speed Reached(0:M69 Relay, 8:SP_Encoder)	1
34	Allow error of SP_Speed that detected by SP_Encoder(RPM)	1
200	Waiting time to screen-saver [>=2minutes]	5
202	System Inner Parameter(cannot alter)	0
203	Inspect over-cut when tool radius offset (6326274:Yes,34818:No)	6326274
206	G21-Metric/G20-Inch Mode(512:Metric, 1024: Inch, other: G20/G21)	512
210	Type of Graphic display area(8:manual,0:Automatic)	1
211	Display X-axis Negative area (1:Yes, 0: No)	1
212	Display X-axis Positive area (1:Yes, 0: No)	1
213	Display Y-axis Negative area (1:Yes, 0: No)	1
214	Display Y-axis Positive area (1:Yes, 0: No)	1
215	Display Z-axis Negative area (1:Yes, 0: No)	1
216	Display Z-axis Positive area (1:Yes, 0: No)	1
230	Running program through input point (+4+8+16+32+64+128: X26-X31)	1
231	Mode of "Delete" key [0:backward deletion,1:Forward delete]	0
232	Detect SP_Zero position before tapping [18:Yes,0:No]	1
233	G06 Circle teaching function[0:No, 1:Yes]	0
234	Activate Program Back Function with Handwheel[+8:Yes, Other: No]	1
235	Automatically generate comments when teaching [0:Yes,1:No]	0
236	Type of Arc code when Teach-in[0:G5/G6 IJK,1:G2/G3 R]	0
237	Mode of Generate G code in Teach_In[0:Normal,1:G01]	0
238	Auto Insert one line in middle line during Teach_In[0:Yes,1:Alter]	0
307	M18xx/M28xx/WAT alarm time(ms)[>=10]	1
400	Translate DXF file to G code[1:Sequencing,4:Start point ,8:No sequencing]	0
401	DXF_X-axis coordinate of Start Point	0
402	DXF_Y-axis coordinate of Start Point	0
500	G74 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]	0

501	G81 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]	0
502	G82 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]	0
503	G83 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]	0
504	G84 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]	0
600	Internal multiplier of non-G00 instruction[>=20 valid]	105

Explanation about User Parameter(processing parameter)

1,(d)Escaping Amount of G73 (mm)

It sets every escaping amount in G73 code, unit:mm.

2,(d)Escaping Amount of G87 (mm)

It sets every escaping amount in G87 code, unit:mm.

3,Direction of offset Q in G76 [G17](1:+X,2:-X,3:+Y,4:-Y)

It sets the direction of offset Q in G76 code in XY plane(G17). 1:+X,2:-X,3:+Y,4:-Y.

4,Direction of offset Q in G87 [G17](1:+X,2:-X,3:+Y,4:-Y)

It sets the direction of offset Q in G87 code in XY plane(G17). 1:+X,2:-X,3:+Y,4:-Y.

5,Stopping Angle when Spindle orientation at G76/G87 (0.1degree)

It sets stopping angle when spindle do orientation at G76/G87 code. unit:0.1degree.

6,Mode of clearance in G74/G84(0:Return with d; 8: Return to R)

It sets mode of clearance in G74/G84;0:High-speed,return with d; 8: Normal, return to point R.

7,(d)Escaping Amount of G74/G84 (mm)

It sets every escaping amount in G74/G84 code, unit:mm.

8,Pause Time Before Raise Up Tool at G73G83(ms)[Abs>5,-:G01;+:G00]

It is set pause time when raise up tool every time on G73G83 code, Unit: ms, Range: >5 . Set to negative number, raise up tool with G01 mode, set to positive number, raise up with G00 mode.

17,Interlock between Running Program & SP-Rotating [1 mean Yes,0 mean No]

It is for interlock between run program and run spindle,when set as 1 means that running program with running spindle;when set as 0 means that running program without check spindle running.

18,Times of auto-cycle running(M20)

It is for times of run M20 code in the program,negative number mean run countless times.

21,Dwell between G01/G02/G03 blocks(ms)[>100]

It is for set delay time between G01/G02/G03,it is for solve the over-cutting in the corner.

22,Dwell between G00 blocks (ms)[>100]

It is for set delay time after run G00 ,it is effective that more than 100ms.

23,Acceleration/Deceleration Constant of Handwheel [50-100]

It is for set the constant of handwheel smoothly Acceleration/dec-eleration. the smaller it is, the faster the Acceleration/dec-eleration is, but much vibration.

33 Type of Detect SP_Speed Reached(0:M69 Relay, 8:SP_Encoder)

It sets the type of detect SP_Speed reached. 0 means that detect if M69(inner register of plc ladder) input is valid; 1 means that system detect spindle encoder for ensure spindle speed is reached.

34 Allow error of SP_Speed that detected by SP_Encoder(RPM)

It sets the allowable error of coding speed(S) & encoder feedback speed . Unit: rpm.

200,Waiting time to screen-saver [>=2minutes]

It is the time that enter protection screen when system stay in main screen and without dialog, don't enter screen protection if less than 2 minutes, press any keys to return back.

201,Delay time before detect zero pulse when threading(ms)[>100]

It is for set delay time before check Z pulse when process screw.

203,Using Pause key in Panel (23103490:Yes,6326274:No)

It is set for if using Pause key in operational panel, when set to 23103490, which means using Pause key; when set to 6326274, shield the Pause key.

206,G21-Metric/G20-Inch Mode(512:Metric, 1024: Inch, other: G20/G21)

It sets programming mode about Metric or Inch mode, 512: Programming with Metric, 1024: Programming with Inch mode, Other values: Metric/Inch mode are shifted by G20/G21

210,Type of Graphic display area(8:manual,other:Automatic)

It set the type of graphic display area, when set to 8, the graphic display area is set manually, & related parameter; when set to others, CNC system will adjust graphic display area automatically.

211,Display X-axis Negative area (1:Yes, 0: No)/212,Display X-axis Positive area (1:Yes, 0: No)

213,Display Y-axis Negative area (1:Yes, 0: No)/214,Display Y-axis Positive area (1:Yes, 0: No)

215,Display Z-axis Negative area (1:Yes, 0: No)/216,Display Z-axis Positive area (1:Yes, 0: No)

P210-P216 are set for if that CNC system display related area when type of graphic display area is manual.

230,Running program through input point (+4+8+16+32+64+128: X26-X31)

CNC system support run processing program by input points , related input points is X26-X31, Example: When P230=+4+8=12, inputs of X26 or X27 is valid, CNC system will running program of "X26"/"HIDEFILEX26" or "X27"/ "HIDEFILEX27".

231,Mode of "Delete" key [0:backward deletion,1:Forward delete]

It sets the mode of “Del” , delete key, when set to 0, press “Del” key, system will delete backward word , when set to 1, Press “Del” key, system will delete forward word.

232,Detect SP_Zero position before tapping [18:Yes,0:No]

It is set for that if system needs to detect zero position of spindle encoder when tapping.

233,G06 Circle teaching function[0:No, 1:Yes]

It sets if system use G06 circle teaching function, 0 means no use; 1 means yes, use this function.

234,Activate Program Back Function with Handwheel[+8:Yes, Other: No]

It sets if system activate program return back function that back to front processing blocks with handwheel on Auto Handwheel condition.

Note:Press Auto & Handwheel keys enter this processing condition. Press Handwheel key again exit this processing condition.

235,Automatically generate comments when teaching [0:Yes,1:No]

It sets for if generate comments automatically when user adopt teach-in function, 0: Yes, CNC will generate comments automatically when record of Teach-in.

236,Type of Arc code when Teach-in[0:G5/G6 IJK,1:G2/G3 R]

It sets type of G code format for arc when user adopt Teach-in function, 0: when arc during teach-in, use G5/G6 & IJK format; 1: G2/G3 & R format.

237,Mode of Generate G code in Teach_In[0:Normal,1:G01]

It sets mode of generate G code in Teach-in function, 0:Normal, 1: Generate G01 in each one linear movement.

238,Auto Insert one line in middle line during Teach_In[0:Yes,1:Alter]

It sets if insert one line automatically when do teach-in during middle line,0:Yes, insert one line automatically, 1:alter current middle line.

307,M18xx/M28xx/WAT alarm time(ms)[>=10]

It sets max waiting time for M18xx/M28xx, when over max time,CNC will hints alarm.unit is ms. Example set to 100 (>=10), also 1000ms, when M1810 code,detect X10 is on within 1000ms, otherwise CNC hints alarm.

400,Translate DXF file to G code[1:Sequencing,4:Start point ,8:No sequencing]

401,DXF_X-axis coordinate of Start Point

402,DXF_Y-axis coordinate of Start Point

1)When P400=1, CNC will make sequencing according to start/end point of dxf file

2)When P400=4, After make sequencing according to according to start/end point of dxf file, and compare distance with starting point(P401&P402),starting from nearest point.

3) When P400=8, CNC doesn't make sequencing automatically. When use Auto-CAD to make drawing & generate dxf file, we need to take care of sequence of drawing, as dxf files are saved with drawing sequence. So CNC generate G code files also according to this sequence.

500,G74 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]

501,G81 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]

502,G82 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]

503,G83 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]

504,G84 equal to ProgramG No.[101-170(101-150Modeless,151-170Mode)]

These parameters are for user define corresponding G codes for G74-G84. Which is for select Non-mode/mode status, and user-defined.

600, Internal multiplier of non-G00 instruction[>=20 valid]

It is set inner rate of G codes except G00. It is valid only when its value >=20.

Note: XYZABCXSYS display at parameter list, X:1st axis, Y:2nd axis, Z:3rd axis, A:4th axis. We write short name with XYZA for feeding axes.

6.2 Speed parameter

P	Parameter	EX-Value
1	G00 Speed of X-axis (mm/min)	4000.000
2	G00 Speed of Y-axis (mm/min)	4000.000
3	G00 Speed of Z-axis (mm/min)	4000.000
4	G00 Speed of A-axis (mm/min)	4000.000
5	Manual Max Feeding Speed(mm/min)	10000.000
5-1	X_Manual Max Feeding Speed(mm/min)[>=50]	0
5-2	Y_Manual Max Feeding Speed(mm/min)[>=50]	0
5-3	Z_Manual Max Feeding Speed(mm/min)[>=50]	0
5-4	A_Manual Max Feeding Speed(mm/min)[>=50]	0
6	Auto Max Feeding Speed(mm/min)	15000.000
7	Default Speed of G01/G02/G03 (mm/min)	2000.000
8	Running Speed at Simulation Mode (mm/min)	10000.000
9	Manual Feeding Speed(mm/min)	1000.00
10	Manual Spindle Speed (rpm)	0.000
11	Take-off speed(mm/min)	500
12	Maximum mutation speed(mm/min)	500
13	Limit G01/G02/G03 Speed of each axis [1:Yes, 0:No]	0
14	Max Speed of X_G1/G2/G3 (mm/min)	2000
15	Max Speed of Y_G1/G2/G3 (mm/min)	2000
16	Max Speed of Z_G1/G2/G3 (mm/min)	2000
17	Max Speed of A_G1/G2/G3 (mm/min)	2000
18	X_Acceleration/Deceleration Constant [1~99999]	50000
19	Y_Acceleration/Deceleration Constant [1~99999]	50000
20	Z_Acceleration/Deceleration Constant [1~99999]	50000
21	A_Acceleration/Deceleration Constant [1~99999]	50000
22	Acceleration/Deceleration Constant When Auto Run	2
23	Handwheel_Acceleration/Deceleration Constant [500--30000]	6000
24	Handwheel_Acceleration/Deceleration Constant when dry [>500]	60000
25	G00 Speed when dry (mm/min) [>10]	1500
25-1	Run program Handwheel G00 start speed(mm/min)[>5]	100
26	X_Max Speed with Handwheel (mm/min)	4000
27	Y_Max Speed with Handwheel (mm/min)	4000
28	Z_Max Speed with Handwheel (mm/min)	4000
29	A_Max Speed with Handwheel (mm/min)	4000
30	Type of Acceleration/Deceleration [0 mean line,8 mean curve]	0
31	Initial Acceleration/Deceleration Constant when curve [>=10]	100
32	Quadratic Acceleration/Deceleration Constant when curve [>=10]	500
33	Max Acceleration/Deceleration Constant when curve [>=500]	1
34	X_Homing Speed (mm/min)	3000.000
35	X_Speed for detect Z0 signal (mm/min)	250.000

36	Y_Homing Speed (mm/min)	3000.000
37	Y_Speed for detect Z0 signal (mm/min)	250.000
38	Z_Homing Speed (mm/min)	3000.000
39	Z_Speed for detect Z0 signal (mm/min)	250.000
40	A_Homing Speed (mm/min)	3000.000
41	A_Speed for detect Z0 signal (mm/min)	250.000
42	Max Speed of 1st Spindle (rpm)	3000
43	Max Speed of Spindle at 2nd gear (rpm)	3000
44	Max Speed of Spindle at 3rd gear (rpm)	3000
45	Max Speed of Spindle at 4th gear (rpm)	3000
46	Max Speed of 2nd Spindle (rpm)	3000
47	Compensation Mode of Arc Backlash (0: A, 8: B)	8
48	Compensation Speed in B Mode (mm/min)	3000
48-1	Beginning Compensation Speed in B Mode (mm/min)	500
48-2	Acceleration/Deceleration Constant in B mode (mm/min/s)	60000
49	Activate Speed Processing Function[1:Yes, 0:No]	1
49-1	End speed when reverse&decrease speed during running(mm/min)	200
50	Handwheel_Stopping Speed (mm/min)[>100]	100
51	SP_Speed when exit at following tapping mode(rpm)[>1]	1
52	SP_Reverse Backlash Compensation when tapping(Pulse)	1
53	Advance Retired Value before reverse rotation of following tapping(um)[10-5000]	1
54	Retired Speed when tapping (mm/min)[>=60]	1
58	Beginning Speed when hardware limitation (mm/min)	1.000
60	Activate Smooth Processing Function(+4:Manual,+8:MPG,+16:Program)	28
61	Time constant of smooth running on Manual[2-500]	50
62	Time constant of smooth running on MPG[2-500]	50
63	Time constant of smooth running on Program[2-500]	50
65	Manual enhancement smoothing processing time constant [2-50]	0
66	Handwheel enhancement smoothing processing time constant [2-50]	0
67	Program enhancement smoothing processing time constant [2-50]	0
68	Grade of smoothing process [1-9](Bigger, higher smooth)	0
101	X_Beginning Speed (mm/min)[>1]	500
102	Y_Beginning Speed (mm/min)[>1]	500
103	Z_Beginning Speed (mm/min)[>1]	500
104	A_Beginning Speed (mm/min)[>1]	500
111	X_Max Speed Increment (mm/min)[>1]	500
112	Y_Max Speed Increment (mm/min)[>1]	500
113	Z_Max Speed Increment (mm/min)[>1]	500
114	A_Max Speed Increment (mm/min)[>1]	500
200	Coherent movement is valid for G00[1:No,16:Yes]	1
210	Wait SP_Speed smooth when threading	1
211	SP_Start Speed_Tapping-Feed(rpm)	1
212	SP_Start Speed_Tapping-Extract(rpm)	1

213	Acceleration-Feed-Rigid Tapping(rpm/S)[>1]	1
214	Acceleration-Retreat-Rigid Tapping(rpm/S)[>1]	1
215	Reserve-Feed-Rigid Tapping(1/1000Rev)[>2]	1
216	Trailing-Feed-Rigid Tapping(1/1000Rev)[>2]	1
217	Reserve-Retreat-Rigid Tapping(1/1000rev)[>2]	1
218	Trailing-Retreat-Rigid Tapping(1/1000rev)[>2]	1
219	Time Constant of Smooth processing for rigid tapping[20002-20500]	1
220	Error pulses of rigid tapping(1:Error, positive: Feed-Axis; Negative: SP-Axis)	0
230	G00 Speed of SP-axis (mm/min)	1000
231	Mode for SP-axis[+4:F,+8:G90/G91,+16:Pulse]	10
232	Orientation Direction of SP-axis[0:Pos,1:Neg,2:near]	2
233	Control Mode of SP_homing	2
234	Offset Degrees when SP_Orientation(0.1Deg)[Neg:Non-Orientation]	0
235	SP_Homing Speed(0.1 rpm)	800

Explanation about Speed Parameter:

1,G00 Speed of X-axis (mm/min) 2,G00 Speed of Y-axis (mm/min)

3,G00 Speed of Z-axis (mm/min) 4,G00 Speed of A-axis (mm/min)

It is rapid speed(also speed of G00) of X/Z axis,Max is 240000(unit:mm/min)

Attention: the value depends on machine configuration, set wrong is very easy to trouble machine tool & accident.

5,Manual Max Feeding Speed(mm/min)

It is the max feeding speed in the condition of Manual , Unit:mm/min.

Attention: reference speed=G00 speed*0.5 ,in order to make sure safe.

5-1,X_Manual Max Feeding Speed(mm/min)[>=50] 5-2,Y_Manual Max Feeding Speed(mm/min)[>=50]

5-3,Z_Manual Max Feeding Speed(mm/min)[>=50] 5-4,A_Manual Max Feeding Speed(mm/min)[>=50]

6,Auto Max Feeding Speed(mm/min)

It is the max of feeding speed in Auto ,Max is 30000.Unit:mm/min. This speed could faster than G00 speed in order to each axis can reach at max speed when linkage as it only limit integrated speed.

7,Default Speed of G01/G02/G03 (mm/min)

It is the default speed of G01/G02/G03 when the speed of first interpolation code (G01/G02/G03) isn't specified in Auto-running. Max:5000 (unit:mm/min)

8,Running Speed at Simulation Mode (mm/min)

It is running speed at the mode of Simulation. (press “Simulate”,is dry run mode) Max:240000. (unit:mm/min)

9,Manual Feeding Speed(mm/min)

It is the speed of feeding axis in Manual.Range:< max feeding speed

Attention:in Manual mode, press “F” key ,can set the parameter directly.

10,Manual Spindle Speed (rpm)

It is set for speed of spindle in mode of manual. Unit:rpm.

Attention:in Manual,press “S” key ,can set the parameter directly.

11,Take-off speed(mm/min)

It is beginning speed of feeding axis when acceleration/deceleration.when it is smaller than acceleration/deceleration, accelerate/decelerate of speed start from the beginning feed speed. when it is bigger than acceleration/deceleration, the speed reached at setting speed directly. Unit: mm/min.

Attention:Generally,stepper system<=100,servo system<=500.

12,Maximum mutation speed(mm/min)

It is max speed increment when multi-axial running track-interpolation. Also max changing value of speed.

Example:when it is 300,the speed of X axis(multi-axial track-interpolation)up from F800 to F1600,800(=1600-800)>300,so the process is up from F800 to F1100, and then F1600.

13,Limit G01/G02/G03 Speed of each axis [1:Yes, 0:No]

It is set for whether limit speed of each axis when G1/G2/G3 interpolating.

14,Max Speed of X_G1/G2/G3 (mm/min) 15,Max Speed of Y_G1/G2/G3 (mm/min)

16,Max Speed of Z_G1/G2/G3 (mm/min) 17,Max Speed of A_G1/G2/G3 (mm/min)

It is for the Max running speed of each alone feeding-axis when G1/G2/G3 interpolation.

18,X_Acceleration/Deceleration Constant 19,Y_Acceleration/Deceleration Constant

20,Z_Acceleration/Deceleration Constant 21,A_Acceleration/Deceleration Constant

It is time constant of each one feeding-axis acceleration/deceleration alone,the bigger it is ,the faster the acceleration/deceleration is. Range is 1~99999

Attention:This value depends on the machine structure,the heavier the load is ,the smaller the value is. With stepper system,the value should less than 15000.

22,Acceleration/Deceleration Constant When Auto Run [1-500]

It is for set constant of Acceleration/dec-eleration in auto.the range is 1-500.It is mainly for distinguish Auto and Manual,only the difference is too much, set it is effective.

23,Handwheel_Acceleration/Deceleration Constant [500--32000]

It is for set constant of Acceleration/dec-eleration of Handwheel.the range is 500-32000.

24,Handwheel_Acceleration/Deceleration Constant when dry [>500]

It is for set constant of Acceleration/dec-eleration when handwheel start program. the range is 500-32000.when the value is less than 500,it is invalid.

25,G00 Speed when dry (mm/min) [>10]

It is the speed of G00 when when handwheel start program for simulate. it is invalid when the

value is less than 10.

25-1,Start Speed when dry with Handwheel (mm/min) [>5]

It is Starting speed of G00 when when handwheel start program for simulate. it is invalid when the value is less than 5.

26,X_Max Speed with Handwheel (mm/min)

It is for limit max speed of X-axis when use handwheel in manual.

Attention:it is valid when >100 ,otherwise invalid.And suggest don't over 4000.

27,Y_Max Speed with Handwheel (mm/min)

It is for limit max speed of Y-axis when use handwheel in manual.

Attention:it is valid when >100 ,otherwise invalid.And suggest don't over 4000.

28,Z_Max Speed with Handwheel (mm/min)

It is for limit max speed of Z-axis when use handwheel in manual.

Attention:it is valid when >100 ,otherwise invalid.And suggest don't over 4000.

29,A_Max Speed with Handwheel (mm/min)

It is for limit max speed of A-axis when use handwheel in manual.

Attention:it is valid when >100 ,otherwise invalid.And suggest don't over 4000.

30,Type of Acceleration/Deceleration [0 mean line,8 mean curve]

It sets type of acceleration/deceleration.set 0 means line type.set 8 means curve type.

Attention:In normal condition,set line type in servo system;set curve type in step system.

31,Initial Acceleration/Deceleration Constant when curve [$>=10$]

It is initial acceleration/deceleration constant when P27 set curve type.Range: $>=10$.

32,Quadratic Acceleration/Deceleration Constant when curve [$>=10$]

It is quadratic constant of acceleration/deceleration when P27 set curve type. Range $>=10$.

33,Max Acceleration/Deceleration Constant when curve [$>=500$]

It is Max acceleration/deceleration constant when P27 set curve type.

It is valid when $>=500$,otherwise the ace/dec-eleration constant is with line type of each axis.

34,X_Homing Speed (mm/min)

It is homing speed of X-axis.Unit:mm/min. the range is less than X_G00 speed.

35,X_Speed for detect Z0 signal (mm/min)

It sets for speed for off switch after X-axis reach at homing switch. & Speed for detecting Z0 phase encoder signal.Unit:mm/min. the range is 20-500.

note:it is for ensure accuracy.the smaller it is ,the higher the accuracy is.when set well,don't change it forever.

36,Y_Homing Speed (mm/min)

It is homing speed of X-axis.Unit:mm/min. the range is less than Y_G00 speed.

37,Y_Speed for detect Z0 signal (mm/min)

It sets for speed for off switch after Y-axis reach at homing switch. & Speed for detecting Z0 phase encoder signal.Unit:mm/min. the range is 20-500.

Note:it is for ensure accuracy.the smaller it is ,the higher the accuracy is.when set well,don't change it forever.

38,Z_Homing Speed (mm/min)

It is homing speed of Z-axis.Unit:mm/min. the range is less than Z_G00 speed.

39,Z_Speed for detect Z0 signal (mm/min)

It sets for speed for off switch after Z-axis reach at homing switch. & Speed for detecting Z0 phase encoder signal.Unit:mm/min. the range is 20-500.

Note:it is for ensure accuracy.the smaller it is ,the higher the accuracy is.when set well,don't change it forever.

40,A_Homing Speed (mm/min)

It is homing speed of A-axis.Unit:mm/min. the range is less than X_G00 speed.

41,A_Speed for detect Z0 signal (mm/min)

It sets for speed for off switch after A-axis reach at homing switch. & Speed for detecting Z0 phase encoder signal.Unit:mm/min. the range is 20-500.

Note:it is for ensure accuracy.the smaller it is ,the higher the accuracy is.when set well,don't change it forever.

42,Max Speed of 1st Spindle (rpm)

It is max speed of 1st spindle, also at 1st gear,M41 output for 1st gear, it is also the speed when PIN25_CN3 plug output analog voltage is 10V(Default condition).

43,Max Speed of Spindle at 2nd gear (rpm)

It is max speed of spindle at 2nd gear,M42 output for 2nd gear, it is also the speed when PIN25_CN3 plug output analog voltage is 10V at M42.

44,Max Speed of Spindle at 3rd gear (rpm)

It is max speed of spindle at 3rd gear,M43 output for 3rd gear, it is also the speed when PIN25_CN3 plug output analog voltage is 10V at M43.

45,Max Speed of Spindle at 4th gear (rpm)

It is max speed of spindle at 4th gear,M44 output for 4th gear, it is also the speed when PIN25_CN3 plug output analog voltage is 10V at M44.

46,Max Speed of 2nd Spindle (rpm)

It is the max speed of 2nd spindle,it is also the speed when PIN25_CN10 plug output analog voltage is 10V. Specified by “SS#” , unit is rpm.

47, Compensation Mode of Arc Backlash (0 mean A; 8 mean B)

It is compensation mode for arc reverse backlash.

0 means A compensation mode, which is that the bigger the reverse backlash is ,the faster the compensation speed is , in order to ensure tool don't exit pause condition. the compensation speed is less than 1000mm/min.

8 means B compensation mode , which is that the compensation speed is set by related parameters in the following.

+4: means when arc programming, IJK is the coordinate value from end point to center. In the original value of P41 plus 4(Eg.: P41=0 +4= 4) means that the IJK of G02/G03 is the coordinate value from end point to center, otherwise IJK of G02/G03 is the coordinate value from starting point to center.

48, Compensation Speed in B Mode (mm/min)

It is the compensation speed in B compensation mode.unit:mm/min.

48-1,Beginning Compensation Speed in B mode (mm/min)

It is beginning compensate speed in B compensation mode.it is valid when it >10.

48-2,Acceleration/Deceleration Constant in B mode (mm/min/s)

It is the acceleration/deceleration constant in B compensation mode (also P41=8/12) . Range: >=10.

49,Activate Speed Processing Function[1:Yes ; 0:No]

It is set for if activate speed processing function, 1 means yes, activate speed processing function, 0 means no activate the function.

49-1,End speed when reverse&decrease speed during running(mm/min)

It is set ending speed when CNC is running with reverse direction & decreasing speed, unit:mm/min.

50,Handwheel_Stopping Speed (mm/min)[>100]

It is the speed when handwheel stop. the bigger it is ,the faster handwheel stop.

51,SP_Speed when exit at following tapping mode

It is min speed before spindle reverse rotation when tapping.

52,SP_Reverse Backlash Compensation when tapping

It is reverse backlash compensation value before spindle reverse rotation when tapping. Unit: Pulse

53,Advance Retired Value before reverse rotation of following tapping

It is advance retired value before spindle reverse rotation when tapping. Unit:um. Range:10-5000

54,Retired Speed when tapping (mm/min)[>=60]

It is speed when spindle retired during tapping. Unit: mm/min

58,Beginning Speed when hardware limitation (mm/min)

It is beginning speed that motor touch hardware limitation switch. When CNC system is configured with servo, no need to set beginning speed , set to 1 normally.

60,Activate Smooth Processing Function(+4:Manual,+8:MPG,+16:Program)

It is set for if activate the function of smooth running on status of Manual/MPG/Program at processing program. +64: Low Speed; +512:Fully-new mode; Set to 28 means activate smooth processing on Manual/MPG & program processing, 1 means no.

61,Time constant of smooth running on Manual[2-500]

It is the time constant when CNC system do smooth running manually,also P60=+4. Range: 2-500.

62,Time constant of smooth running on MPG[2-500]

It is the time constant when CNC system do smooth running on status of MPG,also P60=+8. Range: 2-500.

63,Time constant of smooth running on Program[2-500]

It is the time constant when CNC system do smooth running & processing,also P60=+16. Range: 2-500.

65,Manual enhancement smoothing processing time constant [2-50]

It set time constant when CNC do smooth processing manually, Range: 2~50.

66,Handwheel enhancement smoothing processing time constant [2-50]

It set time constant when CNC do processing with handwheel, Range: 2~50.

67,Program enhancement smoothing processing time constant [2-50]

It set time constant when CNC do smooth processing automatically, Range: 2~50.

68,Grade of smoothing process [1-9](Bigger, higher smooth)

It sets grade of smooth processing, range is 1~9, When P8 are with bigger value, CNC will do processing with higher smoothly.

101,X_Beginning Speed (mm/min)[>1]

102,Y_Beginning Speed (mm/min)[>1]

103,Z_Beginning Speed (mm/min)[>1]

104,A_Beginning Speed (mm/min)[>1]

It is beginning speed of feeding axis when acceleration/deceleration.when it is smaller than acceleration/deceleration, accelerate/decelerate of speed start from the beginning feed speed. when

it is bigger than acceleration/deceleration, the speed reached at setting speed directly. Unit: mm/min.

Attention:Generally,stepper system<=100,servo system<=500.

111,X_Max Speed Increment (mm/min)[>1]

112,Y_Max Speed Increment (mm/min)[>1]

113,Z_Max Speed Increment (mm/min)[>1]

114,A_Max Speed Increment (mm/min)[>1]

It is max speed increment when multi-axial running track-interpolation. Also max changing value of speed.

Example:when it is 300,the speed of X axis(multi-axial track-interpolation)up from F800 to F1600,800(=1600-800)>300,so the process is up from F800 to F1100, and then F1600.

200,Coherent movement is valid for G00[1 is No,16 is Yes]

It is set for that if coherent movement is valid for G00. 16: yes, it is valid for G00. 1: No, it is invalid for G00.

210,Wait SP_Speed smooth when tapping[0:No, 1:Yes]

It is set for that if wait speed of spindle is smooth when tapping. 1: yes, wait speed of spindle is smooth before threading. 0: No, don't wait.

211,SP_Start Speed_Tapping-Feed(rpm)

It is starting speed of spindle axis for feeding when tapping.

212,SP_Start Speed_Tapping-Extract(rpm)

It is starting speed of spindle axis for extract when tapping

213,Acceleration-Feed-Rigid Tapping(rpm/S)[>1]

It is acceleration/deceleration for spindle feeding when rigid tapping,unit:rpm/s.

214,Acceleration-Retreat-Rigid Tapping(rpm/S)[>1]

It is acceleration/deceleration for spindle retreat when rigid tapping,unit:rpm/s.

215,Reserve-Feed-Rigid Tapping(1/1000Rev)[>2]

It is reserve for spindle feed during rigid tapping. Unit:1/1000r, Range:>2

216,Trailing-Feed-Rigid Tapping(1/1000Rev)[>2]

It is trailing for spindle feed during rigid tapping. Unit:1/1000r, Range:>2

217,Reserve-Retreat-Rigid Tapping(1/1000rev)[>2]

It is reserve for spindle retreat during rigid tapping. Unit:1/1000r, Range:>2

218,Trailing-Retreat-Rigid Tapping(1/1000rev)[>2]

It is trailing for spindle retreat during rigid tapping. Unit:1/1000r, Range:>2

219,Time Constant of Smooth processing for Rigid tapping[20002-20500]

It is time constant of smooth processing for rigid tapping, range of value is 20002-20500.

220, Error pulses of rigid tapping(1:Error, positive: Feed-Axis; Negative: SP-Axis)

It sets for display error pulses of rigid tapping, 1: means error between feedback pulses of tapping axis and feedback pulses of spindle axis(CN9), Positive value means tapping axis is advanced, negative value means spindle axis is advanced.

230,G00 Speed of SP-axis (0.1rpm)

It is G00 speed for SP-axis,which is on CN10 plug,unit:0.1rpm.

231,Mode for SP-axis[+4:F,+8:G90/G91,+16:_pulse]

It set mode for SP-axis & other special function. +4: run with F code, +8: run with G90/G91 code, +16: send pulse with display value, +8192:CNC output analog voltage per one second, +16384: run M05 & turn off analog voltage output.

232,Orientation Direction of SP-axis[0:Pos.,1:Neg.,2:Near]

It set orientation direction of SP-axis when rigid tapping with interpolate mode.0: Positive direction, 1: Negative direction, 2: nearest direction

233,Homing Control Mode of SP-Axis

It set homing control mode of SP-axis, 1: Controlled by Pulse(CN10 plug+spindle encoder), 2: Control by Spindle driver(output M61, detect M22 is valid), 16 or 32: after orientation, move SP to degree of P234, 16: Output M75: detect M22 is valid.

234,Offset Degrees when SP_Orientation(0.1Deg)[Neg:Non-Orientation]

It set offset degrees after orientation end, unit:0.1degree, if negative number, don't orientation.

235,SP_Homing Speed(0.1 rpm)

It set homing speed of Sp-axis,unit: 0.1rpm.

6.3 Axis parameter

P	Axis Parameter	Ex-Value
1	Switch Type for Feed-Rate [0: Key, 1: Band Switch]	0
2	Switch Type for SP-Rate [0: Key, 1: Band Switch]	0
3	Max Travel in X_Negative direction (mm)	-9999.000
4	Max Travel in X_Positive direction (mm)	9999.000
5	Max Travel in Y_Negative direction (mm)	-9999.000
6	Max Travel in Y_Positive direction (mm)	9999.000
7	Max Travel in Z_Negative direction (mm)	-9999.000
8	Max Travel in Z_Positive direction (mm)	9999.000
9	Max Travel in A_Negative direction (mm)	-9999.000
10	Max Travel in A_Positive direction (mm)	9999.000
11	SP_Braking Time (10ms)	150
12	SP_Braking is Long Signal [0: No,1: Yes]	0
13	Soft-Limitation is valid [0:Yes, 1: No][D2X;D3Y;D4Z;D5A]	0000000100000001
14	X_Reverse Backlash Compensation (um)	0
15	Y_Reverse Backlash Compensation (um)	0
16	Z_Reverse Backlash Compensation (um)	0
17	A_Reverse Backlash Compensation (um)	0
18	X_Direction [1:normal, 0: Reverse]	0
19	Y_Direction [1:normal, 0: Reverse]	0
20	Z_Direction [1:normal, 0: Reverse]	0
21	A_Direction [1:normal, 0: Reverse]	0
22	Using Electron Gear Ratio for Feeding Axes [0:Yes, 1:No]	0
23	Numerator of X_Electron Gear	1
24	Denominator of X_Electron Gear	1
25	Numerator of Y_Electron Gear	1
26	Denominator of Y_Electron Gear	1
27	Numerator of Z_Electron Gear	1
28	Denominator of Z_Electron Gear	1
29	Numerator of A_Electron Gear	1
30	Denominator of A_Electron Gear	1
31	Type of Limit Switch in Positive Direction[0:NO type, 1:NC type]	0
32	Type of Limit Switch in Negative Direction[0:NO type, 1:NC type]	0
33	Type of Home [D3X;D4Y;D5Z;D6A;0:Switch;1:float Zero]	01111011
34	X_Machine Coordinate of float zero point	0.00
35	Y_Machine Coordinate of float zero point	0.00
36	Z_Machine Coordinate of float zero point	0.00
37	A_Machine Coordinate of float zero point	0.00
38	Grade of Homing[1:No,0:Prompt,8:Compulsion,9:Super compulsion]	1
39	Mode of Homing	1
40	Direction of Homing[D2:X,D3:Y,D4:Z,D5:A]	0000000100000001
41	Type of Switches for Homing	00000000
42	Range of Detect Z0 in X axis	100
43	Range of Detect Z0 in Y axis	100
44	Range of Detect Z0 in Z axis	100
45	Range of Detect Z0 in A axis	100
46	Offset after homing in X axis	0
47	Offset after homing in Y axis	0
48	Offset after homing in Z axis	0

49	Offset after homing in A axis	0
50	Spindle is rotating when shift gear[1:Yes, 0:No]	0
51	Rotating Speed of Spindle when shift gear(1/100rpm)	1000
52	Rotating Direction of Spindle when shift gear[0:CW, 1:CCW]	1
53	Pause Time of Spindle when shift gear (10ms)	10
54	Braking Time of Spindle when shift gear (10ms)	10
55	Delay time between reset M03/M04 & set M05 (10ms)	0
56	Detect SP_Position Feedback [0: No, 1: Yes]	1
57	Pulses Per Revolution of Spindle	4096
57-1	Alarm Range for spindle resolution on Spindle encoder check	0
57-2	Quadruple Pulses Per Revolution of Spindle	0
58	SP_CCW is Long Signal [8: No, other: Yes]	0
66	Gear Shift Control on Spindle Axis[1:Yes, 0:No]	1
68	Delay time when feeding axes shift direction	0
69	Mode of Manual Return Home(1:All axis, 0:Selected Axis)	0
80	X/Z axis is Rotating Axis	00000001
81	Mode of Y Axis [0:Rotating Axis, 1: Linear Axis]	1
82	Workpiece coordinate when Y is rotating axis	1
83	Machine Coordinate when Y is rotating axis	1
100	System Inner Parameter	1
101	Mode of A Axis [0:Rotating Axis, 1: Linear Axis]	1
102	Base when A axis is rotating axis	-9999.000
103	Automatically Output(Y/M) for unclamp when 4 th Axis is running	0
104	Detection in position of unclamp[10000+:X,20000+M,30000+wait time]	0
323	Nearest way for rotary axis(+4:X, +8:Y, +16:Z, +32:A, +64:B)	0
404	SP_Direction when position control mode	0
405	Using Electronic Gear Ratio for Spindle [0:Yes, 1:No]	0
406	Numerator of SP_Electronic Gear Ratio in Low Gear	4096
407	Denominator of SP_Electronic Gear Ratio in Low Gear	360000
408	Numerator of SP_Electronic Gear Ratio in High Gear	4096
409	Denominator of SP_Electronic Gear Ratio in High Gear	360000
410	Coordinate Axis when spindle do tapping	94
411	Control Mode of Tapping [2:Following, 3: Interpolation]	3
412	Teeth of SP_Motor (<P413)	1
413	Teeth of SP_Encoder (>P412)	1
414	Follow-Up of A Axis[7:X, 8:Y, 9:Z]	1

Explanation about Axis Parameter:

1,Switch Type for Feed-Rate [0: Key, 1: Band Switch]

It is set switch type of Feed-Rate, Rate of Feeding axes.

0: Keys of “Feed Rate+” & “Feed Rate-” in Panel for adjust rate of feeding speed;

1: External Band switch for Feed-Rate. Port for external band switch is at CN11 plug, Pins are VDK0,VDK1,VDK2,VDK3. Total are 16 gears

2,Switch Type for SP-Rate [0: Key, 1: Band Switch]

It is set switch type of SP-Rate, Rate of SP_Speed.

0: Keys of “SP Rate+” & “SP Rate-” in Panel for adjust rate of spindle speed;

1: External Band switch for SP-Rate. Port for external band switch is at CN11 plug, Pins are VDS0,VDS1,VDS2,VDS3. Total are 16 gears

3,Max Travel in X_Negative direction (mm)

4,Max Travel in X_Positive direction (mm)

5,Max Travel in Y_Negative direction (mm)

6,Max Travel in Y_Positive direction (mm)

7,Max Travel in Z_Negative direction (mm)

8,Max Travel in Z_Positive direction (mm)

9,Max Travel in A_Negative direction (mm)

10,Max Travel in A_Positive direction (mm)

These parameters are set for max travel in negative direction/in positive direction of feeding axes when soft-limitation is valid, which is based on machine coordinate system.unit: mm.

11,SP_Braking Time (10ms)

It is the braking time of spindle,also holding time of output M05. the shorter it is,the faster the brake is. Unit: 10ms.

12,SP_Braking is Long Signal [0: No,1: Yes]

It is set for signal mode of SP_Braking. 1: Long signal, 0:short signal. It depends on braking mode of spindle system.

13,Soft-Limitation is valid [0:Yes, 1: No]

It is bit parameter, set for if soft-limitation is valid of each feeding axis.

Bit	D7	D6	D5	D4	D3	D2	D1	D0
Value	0	0	0	0	0	0	0	1
Axis	-	-	A	Z	Y	X	-	-

0: means soft-limitation is valid , 1: means invalid

Example:the soft limit of X-axis is valid, P11= 00000100.

14,X_Reverse Backlash Compensation (um) 15,Y_Reverse Backlash Compensation (um)

16,Z_Reverse Backlash Compensation (um) 17,A_Reverse Backlash Compensation (um)

These are the value of reverse backlash compensation of feeding axes. When moving direction of feeding axis is changed ,system will make compensation with the value automatically. Unit: um

18,X_Direction [1:normal, 0: Reverse] 19,Y_Direction [1:normal, 0: Reverse]

20,Z_Direction [1:normal, 0: Reverse] 21,A_Direction [1:normal, 0: Reverse]

It is for shift direction of feeding-axes. 1: Direction of feeding axis is same to direction of code; 0: Direction of feeding axis is opposite to direction of code.

22,Using Electron Gear Ratio for Feeding Axes [0:Yes, 1:No]

It is for whether using the electron gear ratio for feeding axis. 0: yes,using electron gear, 1: No, don't using electron gear.

23,Numerator of X_Electron Gear (1-999999) 24,Denominator of X_Electron Gear (1-999999)

25,Numerator of Y_Electron Gear (1-999999) 26,Denominator of Y_Electron Gear (1-999999)

27,Numerator of Z_Electron Gear (1-999999) 28,Denominator of Z_Electron Gear (1-999999)

29,Numerator of A_Electron Gear (1-999999) 30,Denominator of A_Electron Gear (1-999999)

These are Numerator & Denominator of feeding axis's electron gear ratio. Range: 1-999999.

Algorithm of P17-P20 & P105/P106 & P204/P205 parameters

Effective Range: 1-999999

Unit:non

User:Upon operating administrators

Initialization:1

Effective time:Immediately

Explain:

When lead screws with different screw pitches are configured with motors of various step angles,or with servo motors of different pulse number per round,or connections are realized through different gears,the programmed values can remain consistent with the actual moved distance by setting the parameter of the electronic gear ration of the system.

$$\text{Electron Gear Ratio} = \frac{\text{Numerator}}{\text{Denominator}} = \frac{\text{CMR}}{\text{CMD}} = \frac{P}{L * 1000}$$

CMR:Numerator of gear ratio

CMD:Denominator of gear ratio

P: pulse number per motor round

L: Moved distance per motor round(mm)

The value of CMD/CMR is the pulse equivalent,which tells the moved distance per pulse ,with its unit as 0.001mm.

Example1: The motor rotates one circle very 5000 pulses,after which the machine tool moves 5mm,then:

$$\text{CMR/CMD} = 5000 / (5 * 1000) = 1/1$$

That is to say,we can set the values as :CMR=1,CMD=1.

Here ,the pulse equivalent is 0.001mm.

Example2: The motor rotates one circle very 5000 pulses,after which the machine tool moves 10mm.

$$\text{CMR/CMD} = 5000 / (10 * 1000) = 1/2$$

That is to say,we can set the values as :CMR=1,CMD=2.

Here ,the pulse equivalent is 0.002mm.

There is one simple solution,for example,current ratio is 1:1,X0-X10(CNC move 10mm),check real distance that machine tool moves,if machine just move 5mm, set to 10:5, if machine moves 12mm, set to 10:12.also P23=10, P24=12 on Axis parameter.

31,Type of Limit Switch in Positive Direction[0:NO type, 1:NC type]

It is set type of limit switch in positive direction of all feeding axes, also type of switch that is connected to +L,PIN16_CN3 plug. 0:NO Type, 1: NC Type.

32,Type of Limit Switch in Negative Direction[0:NO type, 1:NC type]

It is set type of limit switch in negative direction of all feeding axes, also type of switch that is connected to -L,PIN15_CN3 plug. 0:NO Type, 1: NC Type.

33,Type of Home [D3X;D4(C)Y;D5Z;D6A; 0:Switch; 1:float Zero]

It is set type of home. bit parameter. Each axis set alone.

Bit	D7	D6	D5	D4	D3	D2	D1	D0
Value	0	0	0	0	0	0	0	1
Axis	-	A	Z	Y	X	-	-	-

0: Switch/Sensor for home, 1:Float Zero point for home.

Example:Only Home of X-axis is float zero point, P23=00001001.

34,X_Machine Coordinate of float zero point 35,Y_Machine Coordinate of float zero point

36,Z_Machine Coordinate of float zero point 37,A_Machine Coordinate of float zero point

It is set the machine coordinate value of feeding-axis based on float zero point. The value is distance between current position of machine tool & float zero point.

38,Grade of Homing[1:No need, 0:Prompt, 8:Compulsion, 9:Super compulsion]

It set the grade of homing for feeding axis.there is 4 grades as follow:

1 : No need. When system boots every time,no prompt and no limitation;

0 : Prompt. After system boot every time,there is a prompted box for homing , and then there aren't any limitation about homing;

8 : Compulsion. When system boots every time, there will a prompted box for homing. And then, if system don't homing successfully, it will hint "feed axis don't go home" before running program ,and don't run processing program;

9 : Super compulsion. When system boots every time, there will a prompted box for homing. And then,if system don't homing successfully,it will hints "feed axis don't go home" at each operations ,and feeding axes don't move.

39,Mode of Homing

It set mode of homing. There are 4 kinds of mode about homing.

0 : Homing after hit homing switch, move in reverse direction until homing switch is off, then detect Z0 signal of Encoder of servo motor.

1 : Homing after hit homing switch, move in reverse direction until homing switch is off.

2 : Homing after hit homing switch, move forward until homing switch is off, then detect Z0 signal of encoder of servo motor.

Other: Homing after hit homing switch,move forward until homing switch is off.

40,Direction of Homing [D2:X,D3:C(Y),D4:Z,D5:A]

It sets the direction & sequence of homing for each axis. Bit parameter.

Bit	D15	D14	D13	D12	D11	D10	D9	D8	D7	D6	D5	D4	D3	D2	D1	D0
Value	0	0	0	0	0	0	0	1	0	0	0	0	0	0	0	1
Axis	-	-	-	-	-	-	-	-	-	-	A	Z	Y	X	-	-

0:Homing in positive direction, 1:Homing in negative direction, D8: set priority of X&Z-axis go home.1 means Z-axis first,0 means X-axis first.

41,Type of Switch for Homing [D0X;D1C(Y);D2Z;D3A;1:NC ; 0:NO]

It set the type of switch for homing. Bit parameter.

Bit	D7	D6	D5	D4	D3	D2	D1	D0
Value	0	0	0	0	0	0	0	1
Axis	-	-	-	-	A	Z	Y	X

1: NC type; 0: NO type.D7=1:Manual/Auto shift automatically ;

Example:If X&Z axis are NC switch,the bit parameter is 000000101.

42,Range of Detect Z0 in X axis (unit:100um)

It is range that system can detect Z0 signal of encoder in X direction.

Attention:the value must be less than the length of one rev,otherwise homing failure.

43,Range of Detect Z0 in Y axis (unit:100um)

It is range that system can detect Z0 signal of encoder in Y direction.

Attention:the value must be less than the length of one rev,otherwise homing failure.

44,Range of Detect Z0 in Z axis (unit:100um)

It is range that system can detect Z0 signal of encoder in Z direction.

Attention:the value must be less than the length of one rev,otherwise homing failure.

45,Range of Detect Z0 in A axis (unit:100um)

It is range that system can detect Z0 signal of encoder in A direction.

Attention:the value must be less than the length of one rev,otherwise homing failure.

46,Offset after homing in X axis (unit:10um,-9999~+9999)

It is offset that X-axis after homing. Move with G00 speed. Unit: 0.01mm.

47,Offset after homing in Y axis (unit:10um,-9999~+9999)

It is offset that Y-axis after homing. Move with G00 speed. Unit: 0.01mm.

48,Offset after homing in Z axis (unit:10um,-9999~+9999)

It is offset that Z-axis after homing. Move with G00 speed. Unit: 0.01mm

49,Offset after homing in A axis (unit:10um,-9999~+9999)

It is offset that A-axis after homing. Move with G00 speed. Unit: 0.01mm.

50,Spindle is rotating when shift gear [1:Yes, 0:No]

It set if spindle is rotating when spindle shift gears.1:Yes, SP is rotating, 0: No.

51,Rotating Speed of Spindle when shift gear(1/100rpm)

It is rotating speed of spindle when spindle shift gear & P51=1.

52,Rotating Direction of Spindle when shift gear [0:CW,1:CCW]

It is rotation direction of spindle when spindle shift gear. 0: CW, output M03; 1:CCW, output M04.

53,Braking Time of Spindle when shift gear (10ms)

It is braking time of spindle when spindle shift gear. Unit: 10ms.

54,Delay time between reset M03/M04 & set M05 (10ms)

It is delay time before output M05 ,and after reset M03/M04. Unit:10ms.

55,Spindle stop time(unit:10ms)

It is the delay time between cancel M03/M04 and boot M05.unit:10ms.

56,Detect SP_Position Feedback [0: No,1: Yes]

It is for whether the system detect position feedback signal of spindle by SP_encoder. 1: detect, 0: No detect.

It is used for open that display of spindle real speed & some functions related with SP_Speed which must be on condition of transmission ratio is 1:1 between SP_encoder & spindle motor.

57,Pulses Per Revolution of Spindle

It is pulses per revolution of spindle. Pulses= (Resolution of SP-encoder) * 4.

57-1,Allow Error of Resolution of SP_Encoder[>10]

It sets resolution of spindle encoder,which is checked by CNC,also Spindle encoder check on Alarm/diagnosis screen.Range is >10

57-2,Pulses Per Revolution of Spindle[1/1:0,Other:>99]

It set pulses of spindle per one revolution(Quadruple),when teeth on encoder & on spindle axle is same(1:1), set P200-2 to 0. if not 1:1,valid range is >99,value must be integer.

58,SP_CCW is Long Signal [8: No, other: Yes]

It is set for signal mode of SP_Braking. 1: Long signal, 0:short signal. It depends on braking mode of spindle system.

66,Gear Shift Control on Spindle Axis[1:Yes, 0:No]

It sets whether set gear shift control on spindle axis.

68,Delay time when feeding axes shift direction(ms)

It sets delay time when feeding axes(XYZA) change direction, unit: ms.

80,Mode of X&Z axis

It is bit parameter, Each bit have its related function. 1: Valid, 0: Invalid.

D2:Z axis based on Workpiece coordinate system;D3:X axis based on Workpiece coordinate system; D4:Z axis based on Machine coordinate system; D5:X axis based on Machine coordinate system. D6:Z axis is rotation axis; D7:X axis is rotation axis.

Bit	D7	D6	D5	D4	D3	D2	D1	D0
Value	0	0	0	0	0	0	0	1

Function	XR	ZR	XM	ZM	XW	ZW	-	-
----------	----	----	----	----	----	----	---	---

81,Mode of Y Axis [0:Rotating Axis, 1: Linear Axis]

82,Workpiece Coordinate when Y is rotating axis[0 No;1 plan]

83,Machine Coordinate when Y is rotating axis[0 No;1 plan]

P81 is set for the mode of Y-axis, 0: Rotating axis , 1: Linear axis. When Y-axis is set as rotary axis,if based on workpiece coordinate system, set P82=1; When Y-axis is set as rotary axis,if based on machine coordinate system, set P83=1.

100,System Inner Parameter

Inner parameter of system, cannot be altered.

101,Mode of A Axis [0:Rotating Axis, 1: Linear Axis]

It is set for the mode of A-axis, 0: Rotating axis , 1: Linear axis.

102,Base when A axis is rotating axis

It is set the base of A-axis when it is rotating axis. 0:Null, +1:Based on Absolute Coordinate, +2: Based on Machine Coordinate, 3: Both.

103,Automatically Output(Y/M) for unclamp when 4th Axis is running

It is set output point(Y) or auxiliary relay(M) for unclamp automatically when 4th axis is running. Normally 4th axis is used for rotary table,which is with brake, we can define output point/auxiliary relay for control released of brake. 10000+Y output address, 20000+M auxiliary relay address. Example: we use Y27 to control & release brake of rotary table,so P103=10027.

104,Detection in position of unclamp[10000+X,20000+M,30000+wait time]

It is set input point & waiting time for detecting in position of unclamp when rotary table is with brake.10000+(X),set input point address, 20000+(M),set auxiliary relay input,30000+, set waiting time,unit is ms. Example: we use X01 as input point for detect position of unclamp,max waiting time is 1.5seconds,also P104=41501=(10001+31500)

323,Nearest way for rotary axis(+4:X, +8:Y, +16:Z, +32:A, +64:B)

It sets whether these rotary axes are moving with nearest way, +4: X-axis don't run with nearest way, +8:Y-axis, +16:Z-axis, +32:A-axis, +64:B-axis.

404,SP_Direction when position control mode

It is the direction of spindle motor,0 means reverse,1 mean normal.

405,Using Electronic Gear Ratio for Spindle [0:Yes, 1:No]

It is for whether the spindle use electron gear.

406,Numerator of SP_Electronic Gear Ratio in Low Gear (1-999999)

It is the numerator of SP-axis's electron low gear in low gear.

407,Denominator of SP_Electronic Gear Ratio in Low Gear (1-999999)

It is the denominator of SP-axis's electron low gear in low gear.

408,Numerator of SP_Electronic Gear Ratio in High Gear (1-999999)

It is the numerator of SP-axis's electron low gear in high gear.

409,Denominator of SP_Electronic Gear Ratio in High Gear (1-999999)

It is the denominator of SP-axis's electron low gear in high gear.

410,Coordinate Axis when spindle do tapping[91 X,92 Y/C,93 Z,94 A,95 B]
It is the axis that be use for spindle when interpolation tap.

411,Control Mode of Tapping [0:Following, 1: Interpolation]
[2 follow encode;3 interpolation to SP]
It is control mode of interpolation tap.

412,Teeth of SP_Motor (<P413)
It is tooth number of spindle.it <=P413.

413,Teeth of SP_Encoder (>P412)
It is tooth number of SP-encoder,it >=P412.

Attention:the tooth number of spindle must be not more than the tooth number of SP-encoder,when less,it need to install our company's adapter plate.

414,Follow-Up of A Axis[7:X, 8:Y, 9:Z]

It is set the function of follow up of A-axis. Set to 7: A-axis follow up with X-axis; Set to 8: A-axis follow up with Y-axis ; Set to 9: A-axis follow up with Z-axis. A-axis follow up on condition of both Manual & Auto.

Special Cautions:

1.Only when CNC controller is configured with related feeding axis, and there are related parameters sets for related feeding axis, such as C axis & A-axis.

2.About bit parameters, if some bits are don't specified functions for feeding axis, please keep same to ex-factory set, which should be important for inner system, otherwise it will affect normal operations of CNC system.

6.4 Tool parameter

P	Tool Parameter	Ex-Value
1	Mode of Setup Radius C Compensation	0
2	Mode of Cancel Radius C Compensation	0
8	Management of Tool(0:M06,1:T code,+32768:Tool life management)	1
32	Filtering for Position Signal or WAT Signal	1287

Explanation about Tool Parameter:

1,Mode of Setup Radius C Compensation (0:A type, 1:B type)

2,Mode of Cancel Radius C Compensation (0:A type,1:B type)

They are mode of setup/cancel radius C compensation, details at chapter3.26.

8,Management of Tool(0:M06,1:T code,+32768:Tool life management)

It sets management way for tool code,0:use M06 with T code, 1: use T code for change tool directly,+32768, activate function of tool lift management.

Press “Redeem” key again to enter tool lift management screen, press “Redeem” again,exit to Redeem screen. It can set using times or using time, and also can set current using times & using time, unit of time is second(s). When times or using time is reached, do related tool offset on next time automatically, revise increment values are set by parameters on tool life management, after revised, current using times or current using times will be cleared to 0.

32,Filtering for Position Signal or WAT Signal

It is set filtering time for position signal of electric turret or WAT signal on special turret.

Time	Position Signal	WAT signal
2ms	+256	+2048
4ms	+512	+4096
8ms	+1024	+8192

6.5 Other Parameter

P	Other Parameter	Ex-Factory
1	Type of Handwheel(0:Panel, 1:Handheld)	0
3	Using Interface Switch on Panel(0: No, 88:Yes,96: Yes when Auto)	96
4	Lubricate automatically (0:Yes, 1:No)	1
5	Time of Lubrication (10ms)	300
6	Interval of Lubricate automatically(s)	1800
7	Detection for Door Switch(0:No, 1:Yes)	0
8	Type of Door Switch(0:NO type, 1:NC type)	0
9	Special Bit Parameter	0000010001000100
10	Counting Workpieces automatically(0:No, 1:Yes)	1
11	Increment of shift block	1
12	System Inner Parameter	7
13	Interlock between Rotation_SP & Chuck(0:No, 1:Yes)	1
14	Coolant key is valid on Auto(0:No, 1:Yes)	1
17	Type of Driver Alarm(ALM)(0:NO type, 1:NC type)	0
18	Type of Spindle Alarm(ALM1)(0:NO type, 1:NC type)	0
19	Type of Machine Alarm(ALM2)(0:NO type, 1:NC type)	0
20	Control Mode of Chuck(0:Single, 1:Double)	0
22	External Switch for Chuck(0:No, 1:Yes)	0
24	Time of Chuck(s)	0.00
26	Type of Emergency Stop1(0:NO type, 1:NC type)	0
27	Type of Emergency Stop2(0:NO type, 1:NC type)	0
28	Run/Pause Output(0:No, 1:Yes)	0
29	Alarm Output(0:No, 1:Yes)	0
30	Language(1:Chinese, 0:English)	0
31	Use Inner PMC(0:No, 1:Yes)	1
32	Use High-Speed Inner PMC(0:No, 1:Yes)	1
35	Soft-Limit is Valid when no homing(0:No, 1:Yes)	1
36	Time(Year-Month-Day-Hour-Minute)	
37	Rate of RS232	6
37-1	OPC Function_Modbus_Station	0
38	Latched for Rapid Key(8:Yes)	1
39	System Inner Parameter	***
40	System Inner Parameter	-88
41	Backup Current Parameters	
42	Recovery Backup Parameters	
50	Run from middle Program ask going last line point[8:Yes,0:No]	8
51	Return Safe Point firstly when starting from break point(+4:Z,+8:X)	1
52	Machine Coordinate_Safe Point	1
120	Direction of Manual Feeding Key	1
140	M77:Machine Coordinate VS Reference point 1	0

141	M78:Machine Coordinate VS Reference point 2	1
150	X_Reference Point_1(mm)	0.00
151	Y_Reference Point_1(mm)	0.00
152	Z_Reference Point_1(mm)	0.00
153	A_Reference Point_1(mm)	0.00
160	X_Reference Point_2(mm)	0.00
161	Y_Reference Point_2(mm)	0.00
162	Z_Reference Point_2(mm)	0.00
163	A_Reference Point_2(mm)	0.00
380	X_Machine Coordinate Value at initial point(ATS)(mm)	0.00
381	Y_Machine Coordinate Value at initial point(ATS)(mm)	0.00
382	Z_Machine Coordinate Value at initial point(ATS)(mm)	0.00
383	Speed in Negative Direction(ATS)(mm/min)	0.00
384	Speed in Positive Direction(ATS)(mm/min)	0.00
385	Z_Workpiece coordinate Value at surface workpiece after ATS(mm)	0.00
386	Positioning Speed(ATS)(mm/min)	0.00
387	Mode of ATS(1:Fixed point,0: Float Point)	0.00
388	Z_Min Machine Coordinate Value(mm)	0.00
389	The Gap value of Z(mm)	0.00
390	Total detecting times when Auto tool setting(1-5)	0
501	Shift Color Display of Screen(1:No, 8:Yes)	1
601	Define Parameters for Step	
602	Define Parameters for Servo	Servo
800	System Inner parameter	
900	Display User-define Dialog Box[1:No, 4:Yes]	1
901	Homing Sequence of Axis(5bits)	1
910	High-Speed Input of M18/M22/M24/M28 for G31/G311	0
911	Using M18_Teach-in, M28_Record(0:No, 1:Yes)	0
912	“Reset”key reset Outputs(0:No, 1:Yes)	1

Note: P12&P39&P40 are System Inner Parameter, cannot be altered.

Explanation about Axis Parameter:

1, Type of Handwheel(0:Panel, 1:Handheld)

It sets the type of handwheel, 0:Handwheel in operational panel, 1:Handwheel in handheld box.

Note: when the parameter is 1 (P1=1), CN11 is connected to handheld box; & can't use rate of spindle&feeding axis, only off/X/Y/Z/A & *1/*10/*100(also P1&P2=0 in Axis parameter).

2, Type of Chuck

It set type of chuck, 0: Inside Chuck(M10: chuck clamp to center); 1:Outside Chuck(M10: chuck clamp to external).

3, Using Interface Switch on Panel(0: No, 1:Yes,)

It sets whether system use interface switch on operational panel. 0:No, don't use interface switch; 1:Yes, use interface switch.

4,Lubricate automatically (0:Yes, 1:No)

It sets whether system use lubricate automatically. 0:Yes, lubricate automatically is valid, 1:No use lubricate automatically.

Attention:Lubricate automatically according to time of running program.

5,Time of Lubrication (10ms)

It sets the time of lubricate automatically , also time of outputting M32, PIN9_CN3 Plug. Unit:0.01s.

6,Interval of Lubricate automatically(s)

It is the interval that lubricate every time,also the interval that twice M32 is valid.

7,Detection for Door Switch(0:No, 1:Yes)

It sets whether system detect the signal of protective-door. 0:No detect, 1:Yes.

Attention:1. Input point for door switch: M12, PIN11_CN10 plug.

2. After set P7=1,system can work in Manual, and stop processing in Auto.

3. Pin for detect Chuck_clamp&Door-switch are M12, only one usage is valid.

8,Type of Door Switch(0:NO type, 1:NC type)

It is type of Door-switch. 0:NO type(normal open),1:NC type(normal close).

9,Special Bit Parameter

It is bit parameter,each one bit have different functions,details as following:

Bit	D15	D14	D13	D12	D11	D10	D9	D8	D7	D6	D5	D4	D3	D2	D1	D0
Value	0	0	0	0	0	0	0	1	0	1	0	0	0	1	0	1

D0: Null; default value is 1,which cannot be altered.

D1: 1:Clear Part Number after reboot system; 0:Keep Part number.

D2: 1:Indent automatically between characters when edit; 0:No blank;

D3: Null; default value is 0,which cannot be altered.

D4: Null; default value is 0,which cannot be altered.

D5: 1:Don't stop Rotation_SP & Coolant when pressing "Reset" key;

D6: 1:Each axis run with itself speed when G00; 0: linkage movement;

D7: 1:Don't call related tool compensation when tool change manually; 0: Call related tool compensation automatically; default is 0.

D8: 1:Save status of Chuck(M10/M11) when power off;Recovery original status when booting system; 0: System output M10 automatically when booting.

D9: Select Mode of tool set & input mode of Redeem;

D10: 1:Auto Sequence for block when programming;

D11: 1:Analog of 1st spindle outputs to both +10V_CN3&CN10;

D12: 1:Shield function of "Skip" ,also "/" in the front of blocks is invalid;

D13: 1:Shield function of "Return" key on operational panel;

D14: 1:Shield function of "Start" key on operational panel;

D15: 1:Value of Redeem displays with Increment type; 0: Value of Redeem displays with absolute type;

Attention: some bits of this bit parameter cannot be altered, otherwise it maybe system will work abnormal.

10,Counting Workpieces automatically(0:No, 1:Yes)

It set whether system counting number of workpiece automatically, 0:No counting workpieces automatically; 1:Yes, counting automatically.

11,Increment of shift block

It sets the increment of block when change lines.

12,System Inner Parameter

❖ It is system inner parameter, which cannot be altered.

13,Interlock between Rotation_SP & Chuck(0:No, 1:Yes)

It sets interlock between rotation of spindle and Chuck(M10). 0:No interlock, rotation of spindle isn't related to Chuck; 1:Yes, only when chuck is clamping, and then spindle can rotate; when spindle is rotating, Chuck cannot unclamping.

14,Coolant key is valid on Auto(0:No, 1:Yes)

It sets if Press "Coolant" key is valid on Auto. 0:No, "Coolant" key doesn't work on Auto; 1:Yes, "Coolant" key also works in condition of Auto.

17,Type of Driver Alarm(ALM)(0:NO type, 1:NC type)

It sets the type of driver alarm. ALM, PIN12_CN5 plug, 0:NO type; 1:NC type.

18,Type of Spindle Alarm(ALM1)(0:NO type, 1:NC type)

It sets the type of spindle alarm. ALM1, PIN5_CN3 plug, 0:NO type; 1:NC type.

19,Type of Machine Alarm(ALM2)(0:NO type, 1:NC type)

It sets the type of machine_tool alarm. Input point is ALM2, PIN2_CN10 plug. 0:NO type; 1:NC type.

20,Control Mode of Chuck(0:Single, 1:Double)

It sets the control mode of chuck, 0:Single control signal for Chuck; 1:Double control signal for Chuck.

P20=0, one output point for Chuck, M10: clamp chuck, M11: unclamp chuck;

P20=1, two output points for Chuck, M10: output M10(PIN21_CN3 Plug) to clamp chuck; M11: output M71(PIN9_CN10 Plug) to unclamp chuck.

22,External Switch for Chuck(0:No, 1:Yes)

It sets if there is external switch for control chuck. 0:No, without switch for chuck; 1:Yes, with external switch for chuck. Input point is M16, PIN12_CN10 Plug.

Note: It is reciprocating signal. one is valid, clamp chuck; another is invalid, loose chuck.

23,External Switch for Tailstock(0:No, 1:Yes)

It sets if there is external switch for control tailstock,0:No,without switch for tailstock; 1:Yes,with switch for tailstock. Input point is M14,PIN24_CN10 Plug.

Note:Reciprocating signal.one is valid,tailstock forward;another is invalid,tailstock backward.

24,Time of Chuck(s)

It sets holding time of output M10/M11 for chuck. Unit:second. 0 means M10/M11 are long signal, also always output M10/M11 is valid.

26,Type of Emergency Stop1(0:NO type, 1:NC type)

It set thee type of switch for 1st Emergency Stop, which is at panel. 0: NO type switch; 1:NC type switch for 1st emergency stop.

27,Type of Emergency Stop2(0:NO type, 1:NC type)

It set thee type of switch for 2nd Emergency Stop, which is at panel. 0: NO type switch; 1:NC type switch for 2nd emergency stop. Input is PIN5_CN11 Plug.

28,Run/Pause Output(0:No, 1:Yes)

It sets if output the condition of Running/Pause. 0:No, don't output condition of Run/Pause; 1:Yes, output the condition of Run/Pause. And M69, PIN21_CN10 plug , output Running condition; M65,PIN20_CN10 plug, output Pause condition.

Note:These signals can be used to indicator for show condition of machine.

29,Alarm Output(0:No, 1:Yes)

It sets if output the condition of Alarm. 0:No, don't output condition of Alarm; 1:Yes, output the condition of Alarm, Output point is M67,PIN8_CN10 Plug.

Note: The signals can be used as machine-protection or show condition of machine.

30,Language(1:Chinese, 0:English)

It sets the language of system. 1: Set language to Chinese ; 0: set to English.

31,Use Inner PMC(0:No, 1:Yes)

It sets if use inner PMC function; 0:No, no use; 1:Yes, use. 32: output points of CNC & output points on PLC are valid when diagnosis; 64: output points of CNC are valid, and output points on PLC are invalid when diagnosis.

Warning:It is usually used for adjusting parameters. system must use inner IO PMC when actual use,also P31=1. Otherwise system will works abnormally.

32,Use High-Speed Inner PMC(0:No, 1:Yes)

It sets if use inner PMC function; 0:No, no use; 1:Yes, use. 32: output points of CNC & output points on PLC are valid when diagnosis; 64: output points of CNC are valid, and output points on PLC are invalid when diagnosis.

Warning:It is usually used for adjusting parameters. system must use inner IO PMC when actual use,also P31=1. Otherwise system will works abnormally.

35,Soft-Limit is Valid when no homing(0:No, 1:Yes)

It sets if soft-limit is valid when not homing. 1:Yes,valid, 0:No,invalid.

Attention:the set of this parameter is related to operation habits.

36,Time(Year-Month-Day-Hour-Minute)

It sets time and date of system. After set well,system will take this setting time as basic,according to inner timer count time and shows in display.

Example:13:33, 16th, March, 2017; set P36=2017-03-16-13-33, & Enter.

37,Rate of RS232

It sets rate of communication with RS232. Different value corresponding to different rate:[0=7200;1=9600;2=14400;3=19200;4=38400;5=57600; 6=115200].

Attention:The Rate of both CNC & PC must keep same.

37-1,OPC Function_Modbus_Station

It sets Station number of Series port with OPC modbus function, Odd parity: 10000+station number; Even parity: 20000+Station number; No parity:30000+station number; +100000:Series Port 1 as OPC.

38,Latched for Rapid Key(8:Yes)

It sets if latched for “Rapid” key on panel. 8:Yes. Reciprocating control.

41,Backup Current Parameters

It is for backup current parameters as ex-factory set. It is used for backup parameters after debugging is finished well, easy to maintain.

Attention: select this parameter,press “Enter” key twice,finish backups.

42,Recovery Backup Parameters.

It is for recovery current parameters to ex-factory set. It is normally used for recovery to ex-factory set when parameters set wrong.

Attention: after finish this operation,last parameters will be occupied.

50,Run from middle Program ask going last line point[8:Yes,0:No]

It sets whether CNC moves to end point of last block when start from middle line,8:Yes, 0:No.

51,Return Safe Point firstly when starting from break point(+4:Z,+8:X,+16:Feed Hold)

It sets whether CNC go to safe point firstly when P50=8.+4:Z-axis go to safe point; +8:X-axis; +16: feed hold.

52,Machine Coordinate_Safe Point

It sets machine coordinate value of safe point after set of P51.

120,Direction of Manual Feeding Key

It sets feeding direction of manual feeding key of each axis on panel.

Value	Function
+4	Direction of Z_Manual Feeding is opposite;
+8	Direction of Y_Manual Feeding is opposite;
+16	Direction of Z_Manual Feeding is opposite;
+32	Direction of A_Manual Feeding is opposite;

140,M77:Machine Coordinate VS Reference point 1(1001/2001/3001/4001:X/Y/Z/A)

It sets whether M77 is valid after compare axis machine coordinate value & coordinate value of 1st reference point, 1001: compare axis is X-axis, 2001:Y-axis,3001:Z-axis, 4001:A-axis. For example: P140=1001, if machine coordinate value is less than P150(X_reference point_1),X-axis value of 1st reference point , M77 is valid.

141,M78:Machine Coordinate VS Reference point 2(1002/2002/3002/4002:X/Y/Z/A)

It sets whether M78 is valid after compare axis machine coordinate value & coordinate value of 2nd reference point, 1002: compare axis is X-axis, 2002:Y-axis,3002:Z-axis, 4002:A-axis. When machine coordinate value is less than coordinate value of 2nd reference point,M78 is valid.

150,X_Reference Point_1(mm) 151,Y_Reference Point_1(mm)

152,Z_Reference Point_1(mm) 153,A_Reference Point_1(mm)

These are XYZA coordinate value for 1st reference point

160,X_Reference Point_2(mm) 161,Y_Reference Point_2(mm)

162,Z_Reference Point_2(mm) 163,A_Reference Point_2(mm)

These are XYZA coordinate value for 2nd reference point

380,X_Machine Coordinate Value at initial point(ATS)(mm)

381,Y_Machine Coordinate Value at initial point(ATS)(mm)

382,Z_Machine Coordinate Value at initial point(ATS)(mm)

383,Speed in Negative Direction(ATS)(mm/min)

384,Speed in Positive Direction(ATS)(mm/min)

385,Z_Workpiece coordinate Value at surface workpiece after ATS(mm)

386,Positioning Speed(ATS)(mm/min)

387,Mode of ATS(1:Fixed point,0: Float Point)

388,Z_Min Machine Coordinate Value(mm)

389,The Gap value of Z(mm)

390,Total detecting times when Auto tool setting(1-5)

These parameters are set for Automatic tool setter with tool setter/probe.

501,Shift Color Display of Screen(1:No, 8:Yes)

It sets if shift color display of screen, 1: No shift; 8:Yes,shift to black color.

601,Define Parameters for Step

It sets current parameters to ex-factory set for step system when machine tool is configured with stepper motor&driver.The operation is done before debugging.

602,Define Parameters for Servo

It sets current parameters to ex-factory set for servo system when machine tool is configured with servo motor&driver.The operation is done before debugging.

900,Display User-defined Dialog Box[1:No, 4:Some, 8:All]

It sets if display user-define dialog box. 1: No display; 4:Yes,display some.

901,Homing Sequence of Axis(5bits)

It sets homing sequence of each axis.Value is 5bits.D0 bit is 0. 1:X, 2:C(Y), 3:Z, 4:A. Eg.: P901=31240, Homing sequence is Z->X->Y->A.

910,High-Speed Input of M18/M22/M24/M28 for G31/G311(0:No, 1:Yes)

It sets if inputs of M18/M22/M24/M28 are high-speed input for G31/G311 command; 0:No, don't as input for G31/G311; 1:Yes.

911,Use M18_Teach-in, M28_Record(0:No, 1:Yes)

It set if use M18 as Teach-in function, M28 as Set function of Teach-in.

912,“Reset”key reset Outputs(0:No, 1:Yes)

It sets if “Reset”key reset output points. 0:No reset outputs; 1: Yes, reset.

6.6 Workpiece Coordinate Parameter

CNC system supports multiple coordinate system function, also 6 workpiece coordinate system(G54-G59), plus 10 workpiece coordinate system(G54.1-G54.10) and a machine coordinate system G53. A machining program can set a workpiece coordinate system can also be set up multiple workpiece coordinate system, the workpiece coordinate system can be changed to move its origin. That is the value of the parameter in the coordinates of its own coordinate origin (zero) coordinate value in the machine coordinate system.

In Milling System, normally user only need one coordinate system(G53 coordinate system), also Machine Coordinate System.

G54 to G59 can be set with 6 workpiece coordinate systems, the coordinate system settings interface can be modified 6 origin of the workpiece coordinate system coordinate value in the machine coordinate system.

P	Coordinate Parameter	Ex-Value
1-0	Current Workpiece Coordinate Set [G54-G59]	54
1-1	X_Workpiece Coordinate (G54-G59)	0.000
1-2	Y_Workpiece Coordinate (G54-G59)	0.000
1-3	Z_Workpiece Coordinate (G54-G59)	0.000
1-4	A_Workpiece Coordinate (G54-G59)	0.000
2-0	Current Workpiece coordinate Set [G54.1-G54.10]	1
2-1	X_Workpiece Coordinate (G54.1-G54.10)	0.000
2-2	Y_Workpiece Coordinate (G54.1-G54.10)	0.000
2-3	Z_Workpiece Coordinate (G54.1-G54.10)	0.000
2-4	A_Workpiece Coordinate (G54.1-G54.10)	0.000
1	X_Workpiece Coordinate of G54	0.000
2	Y_Workpiece Coordinate of G54	0.000
3	Z_Workpiece Coordinate of G54	0.000
4	A_Workpiece Coordinate of G54	0.000
6	X_Workpiece Coordinate of G55	0.000
7	Y_Workpiece Coordinate of G55	0.000
8	Z_Workpiece Coordinate of G55	0.000
9	A_Workpiece Coordinate of G55	0.000
11	X_Workpiece Coordinate of G56	0.000
12	Y_Workpiece Coordinate of G56	0.000
13	Z_Workpiece Coordinate of G56	0.000
14	A_Workpiece Coordinate of G56	0.000
16	X_Workpiece Coordinate of G57	0.000
17	Y_Workpiece Coordinate of G57	0.000
18	Z_Workpiece Coordinate of G57	0.000
19	A_Workpiece Coordinate of G57	0.000
21	X_Workpiece Coordinate of G58	0.000
22	Y_Workpiece Coordinate of G58	0.000
23	Z_Workpiece Coordinate of G58	0.000

24	A_Workpiece Coordinate of G58	0.000
26	X_Workpiece Coordinate of G59	0.000
27	Y_Workpiece Coordinate of G59	0.000
28	Z_Workpiece Coordinate of G59	0.000
29	A_Workpiece Coordinate of G59	0.000
	

Note:

1. When CNC controller is with related axes, which has related functions for feeding axes.
2. Input “E” to clear coordinate value.
3. Value Set for parameter of workpiece coordinate system is with increment type.
4. Each alone G54.1-G54.10 workpiece coordinate set just don’t display on user manual.

Explanation about Workpiece Coordinate System:

1-0, Current Workpiece Coordinate Set [G54-G59]

It is for select current workpiece coordinate from G54 to G59.

1-1 X_Workpiece Coordinate (G54-G59)

1-2 Y_Workpiece Coordinate (G54-G59)

1-3 Z_Workpiece Coordinate (G54-G59)

1-4 A_Workpiece Coordinate (G54-G59)

It sets value of related axis on workpiece coordinate system, which is set by P1-0. The value is set with Increment type.

2-0 Current Workpiece coordinate Set [G54.1-G54.10]

It is for select current workpiece coordinate from G54.1 to G54.10

2-1 X_Workpiece Coordinate (G54.1-G54.10)

2-2 Y_Workpiece Coordinate (G54.1-G54.10)

2-3 Z_Workpiece Coordinate (G54.1-G54.10)

2-4 A_Workpiece Coordinate (G54.1-G54.10)

It sets value of related axis on workpiece coordinate system, which is set by P2-0. The value is set with Increment type.

6.6.1 How to set up the workpiece coordinate system?

We set up the workpiece coordinate in the condition of Manual, the steps are following:

a). Press “MDI” key, select corresponding workpiece coordinate system (G54-G59), Example, select G55 coordinate, input G55, Press ‘Enter’, ‘Start’, selected G54 Coordinate.

b). Move machine to suitable position that easy to measure in manual, and measured the related coordinate value between this point (zero point in the workpiece) to Home of G53 coordinate system (also machine coordinate system).

c). Press “Setup”, press “X” key and ‘Enter’, ‘insert the measured value’, and ‘Enter’.

d). Press “Setup”, press “Z” key and ‘Enter’, ‘insert the measured value’, and ‘Enter’.

e). Press “Setup”, press “Y” key and ‘Enter’, ‘insert the measured value’, and ‘Enter’.

f). Press “Setup”, press “A” key and ‘Enter’, ‘insert the measured value’, and ‘Enter’.

Done well now. Enter different workpiece coordinate system, it will show the corresponding value, which also is offset value between workpiece coordinate system and machine coordinate system (G53).

6.6.2 How to adjust the offset value after set well?

If set up workpiece coordinate system well, when it needs to adjust the offset value, it could be set by enter the workpiece coordinate parameter, steps is as follow:

In the coordinate parameter screen, selected the parameter, press “Enter”, and pop up dialog, input the offset value (also Increments, example: offset 10mm in negative direction, also input -10), press “Enter”. It is okay.

Explanation: 1. when the parameter is altered well, the coordinate main screen will refresh the corresponding coordinate value soon.

2. brackets in these parameters, it means the sum, which is offset or adjust every time. It is suitable to look for the offset every time.

6.7 Password

The password is order to avoid modified accidentally and ensure the system work in normal condition. The system adopt three permissions, “CNC Factory”, “Machine Factory” and “User”.

The original condition is “CNC factory” is set, “Machine factory” and “User” isn’t set.

After set new password (set new password, it need original password), please remember the new password, and the original password wasn’t work.

Attention: the password must be 6 bit data, the data could be number and letter.

password setting include:

1, Is enable CNC Co.’s password ?

It is for inner parameter, it couldn’t be operated.

2, Is enable Machine Co.’s password ?

Display and set the parameter that is related to machine’s configuration.

3, Is enable User’s password ?

It is for whether display and set the parameter that is related to processing.

4, Modify CNC Co.’s password:

5, Modify Machine Co.’s password:

6, Modify User’s password:

7, curvy word time: (days)

Version of Operational Software.

Ex-factory Time

Note: *please must remember new password after alter password.*

6.8 Redeem

Press “Redeem” key to enter interface of redeem in any condition.

Remark	Function
N-Radius	Press “N” key to enter Radius Compensation Interface
T-Length	Press “T” key to enter Length Compensation Interface
R-ACLEA	Press “R” key to clear all compensation value.
Q-CLEAR	Press “Q” key to clear current compensation value.
A-SetTool	Press “N” key to set tool {same to Setup key on panel}
B-ToolPoit	Press “B” key to enter list of Tool Posit
C-Set	Press “C” key to set total tool number
D-CANCEL	Press “D” key to return back main interface

6.8.1 Radius Compensation

Press “N” to entering radius compensation interface on Redeem.

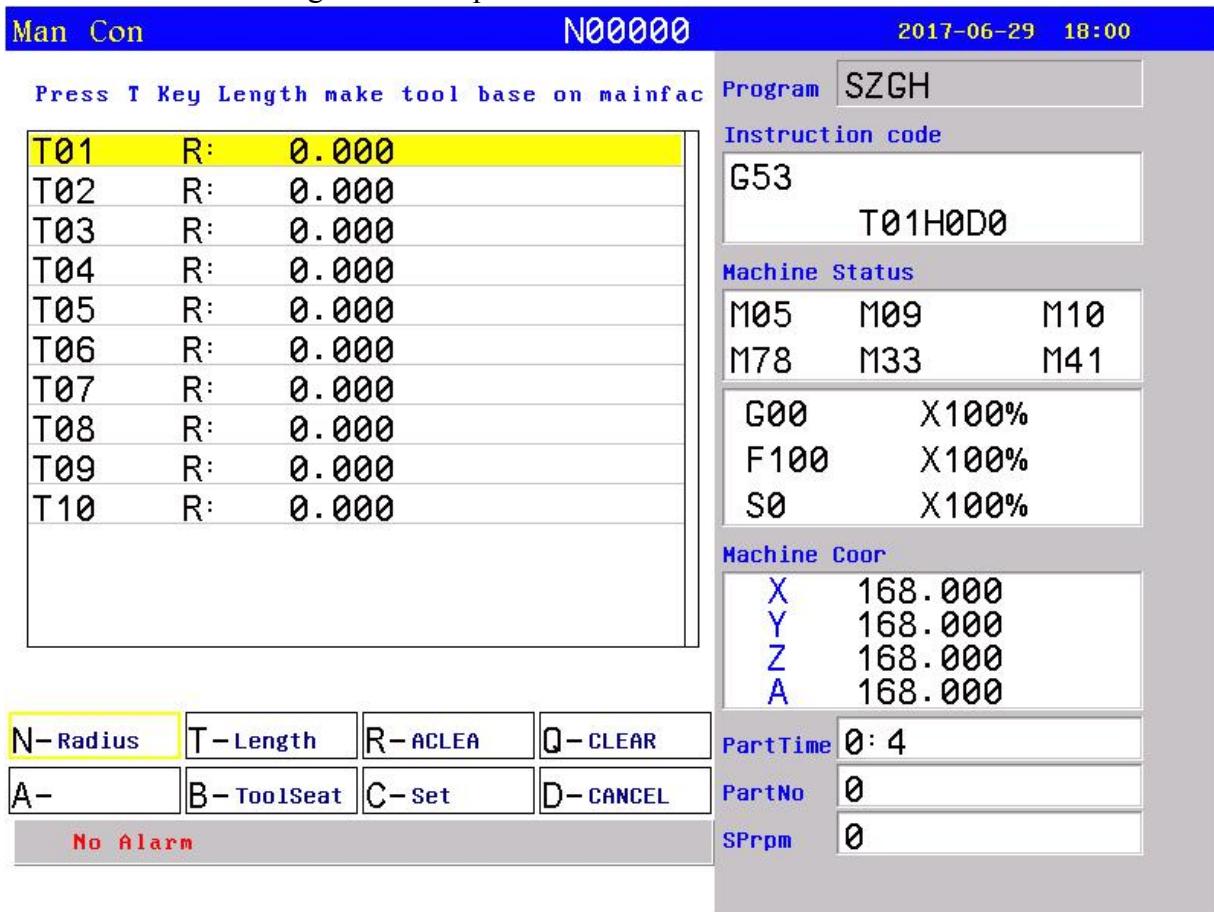


Fig6.8.1 Radius Compensation Interface

Setting Steps: Press “↑ ↓” key to move cursor to related tool and press “Enter” key to popup a dialog box “Input T# tool radius compensate R:”, input radius value of corresponding tool, press “Enter” at last.

Note: Value input is with absolute type.

6.8.2 Length of redeem

Press “T” to enter Length compensation interface on Redeem.

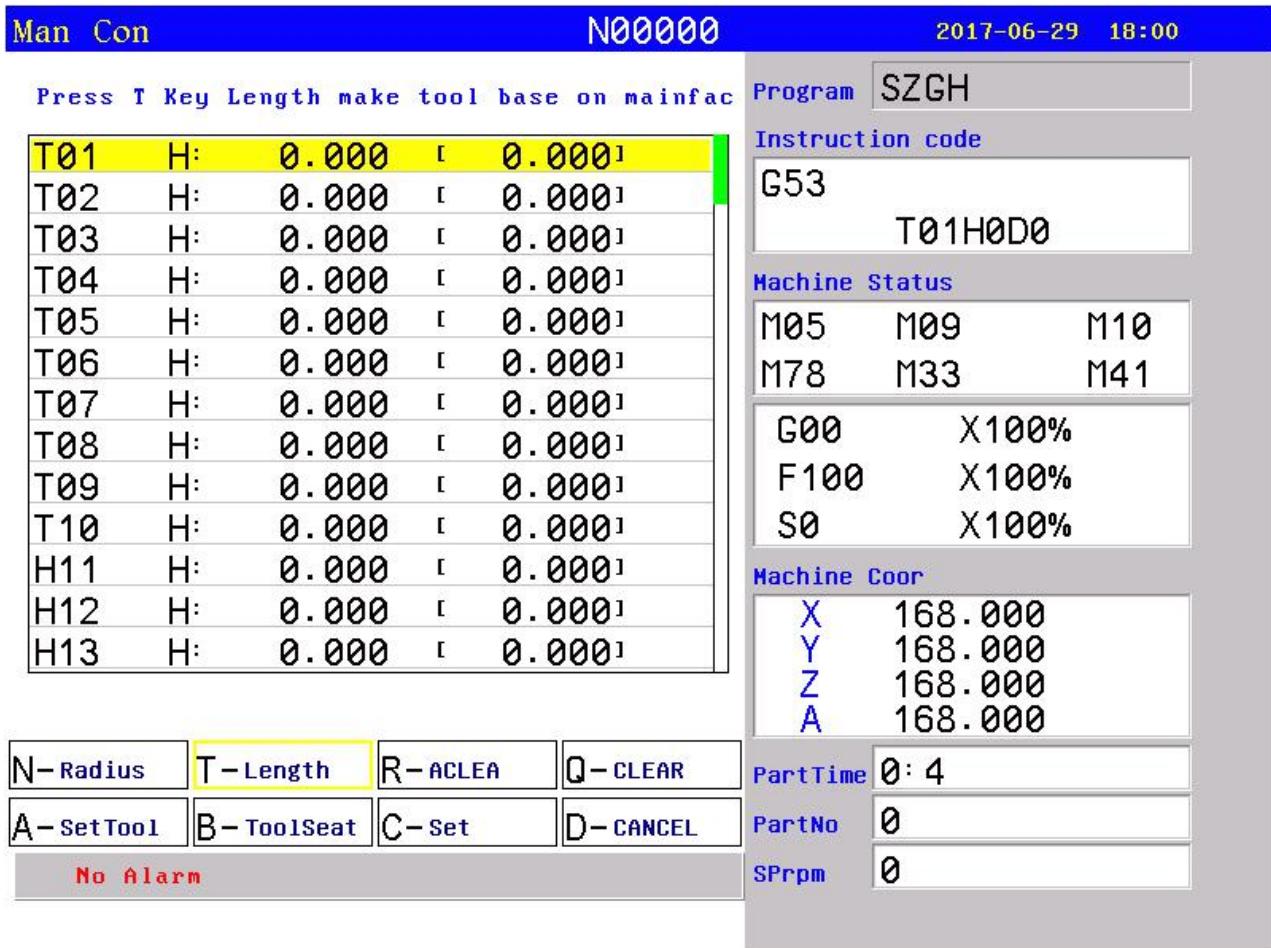


Fig6.8.2 Length Compensation Interface

Steps of modifying length compensation:

Press “↑ ↓” key to move cursor to the corresponding tool number and press “Enter” to popup a dialog box, import the modifying axis into the dialog box and import the modifying value(import 0.05 to plus 0.05, import -0.05 to reduce 0.05), press “Enter” to confirm. The system calculates current value of redeem after finishing setting.

Method of automatic Tool Set

1)Move Tool tip to tool base point, Press T key on main screen to set tool base according to prompt,input OK for confirm, and press Enter key for confirm. Normally we take surface of working table/workpiece as tool base

- 2)Press “↑ ↓” to move cursor to corresponding tool number
- 3)Press “A” to popup a dialog box, “input axis name:[X,Y,Z,A]”
- 4)Press X/Y/Z/A key, “Input Coordinate Axis(mm): X/Y/Z/A”, input coordinate value
- 4)Press “Enter” to confirm. compensation of corresponding axis is set well.

The system refresh current value of redeem after finishing setting automatically.

Method of initializing the length compensation value of tool:

Press “R” or “Q” to initialize length compensation of all or current tool.

Note: Value input is with increment type.

6.8.3 Tool Sets List

Press “B” to enter posit tool interface in redeem. The parameter is used to set type of tool sets when adopting radius compensation of tool.

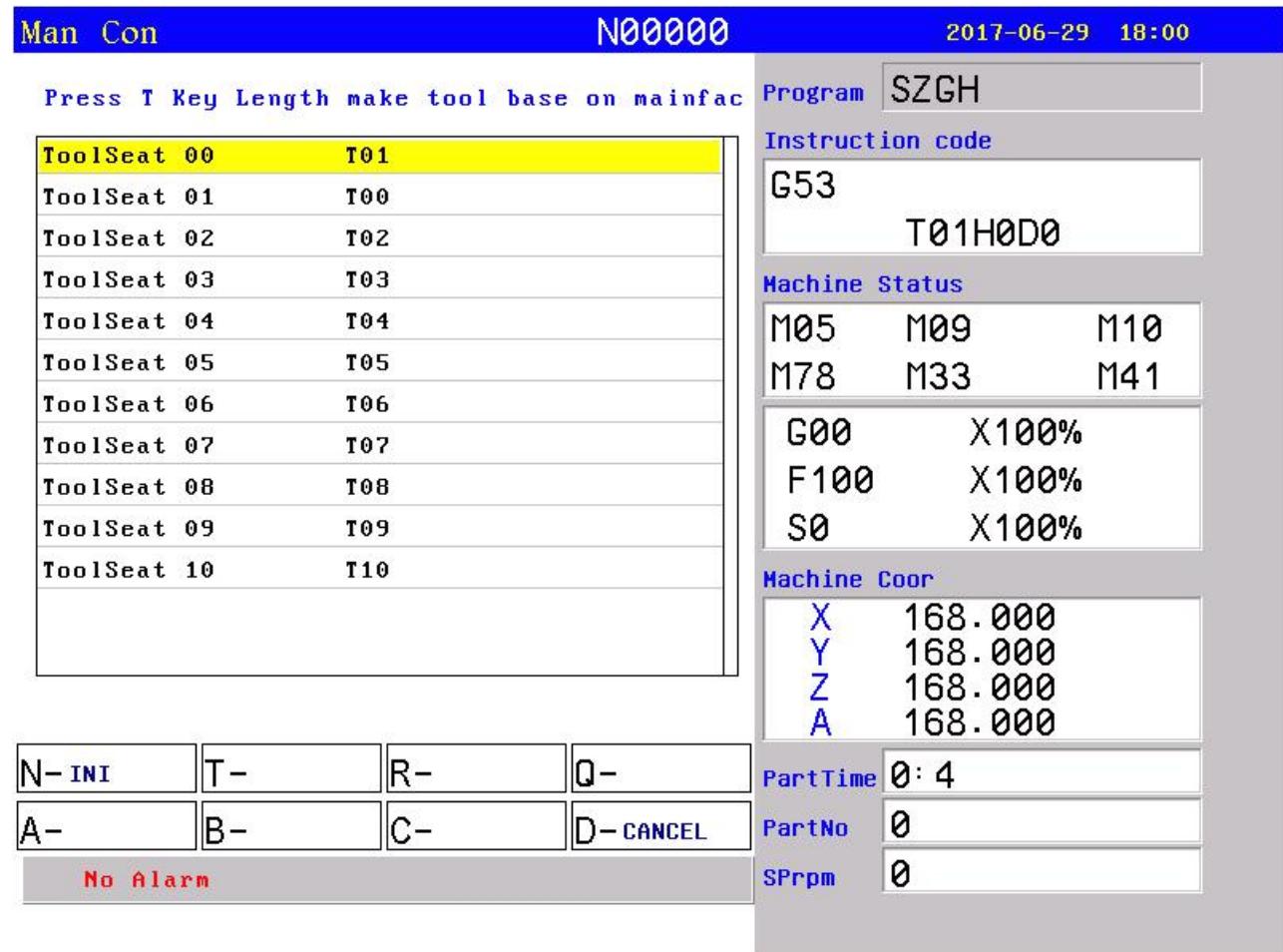


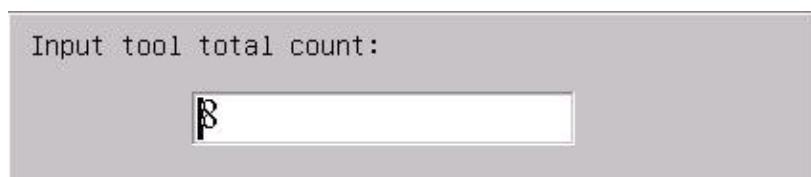
Fig6.8.3 Tool Posit Interface

Step of setting: Press “↑ ↓” to move cursor to corresponding tool number and press “Enter” to popup a dialog box, input the code of corresponding tool’s types and press “Enter” to confirm.

Press “N” key to initialize all the kinds of tool point to 0.

6.8.4 Set quantity

Press “C” key to popup a dialog box On Redeem interface to set total tools.



Including sum tools of electrical tools , linear tools and tool-post.

The CNC system supports 99 pieces of tools max.

6.9 Screw Compensation

Press “Parameter” twice to enter screw compensation interface.

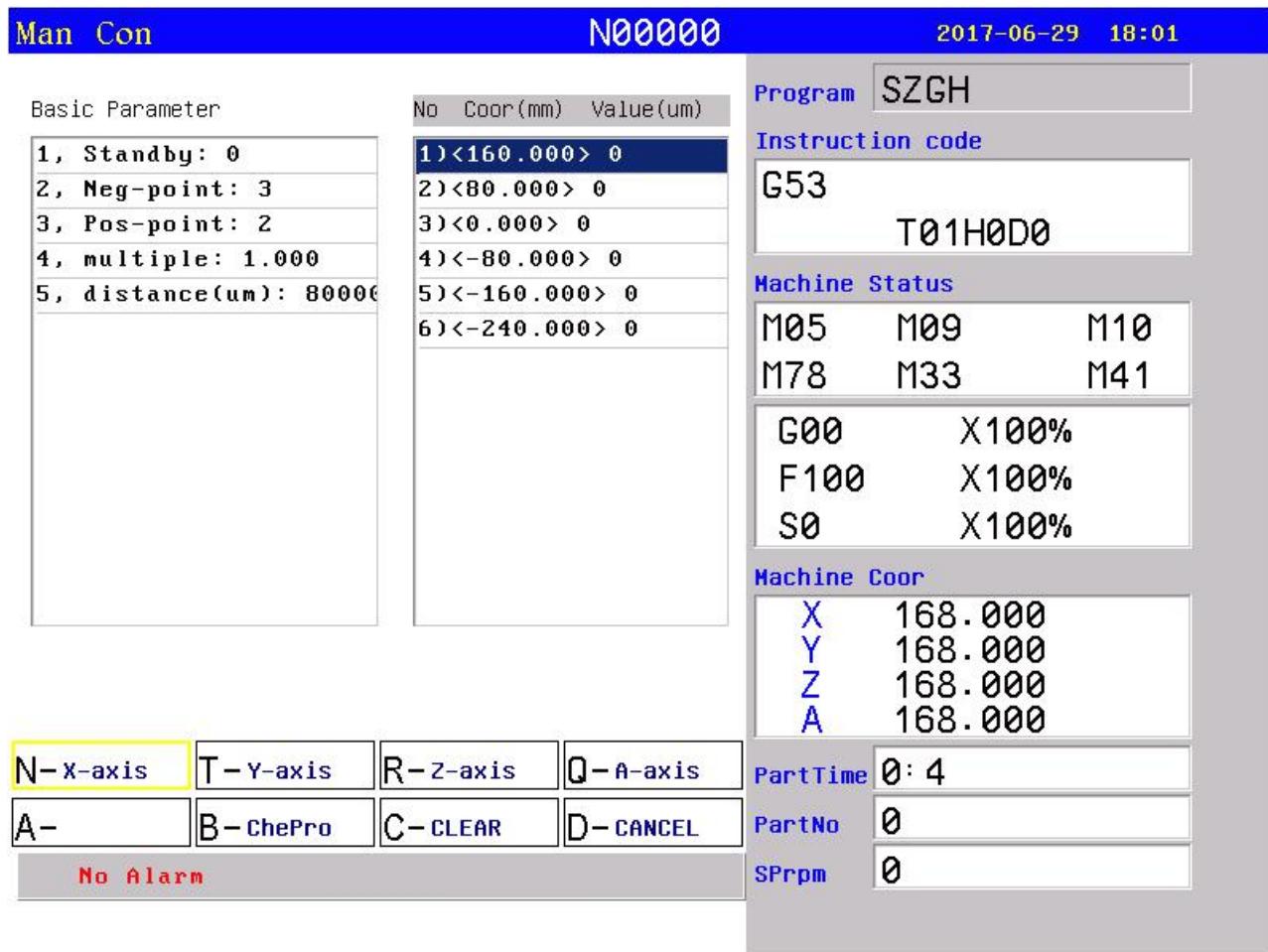


Fig6.9.1 Screw compensation interface

Screw compensation is used for automatic compensating the error of screw pitch, which due to the error of screw pitch to affect accuracy of machine. The system adopts built-in screw compensation: Take machine's home position, also datum point as the starting point when debugging, measured the error curve of screw, studied out the correctional curve according to the error curve, import the value of correctional curve into the correctional parameter and system is going to compensate according to the parameter in automatic running.

Screw compensation by the axis as the unit to set storage, set X/Y/Z/A axis separately, by pressing “N” “T” “R” “Q” to switch; Every axis of screw compensation interface has tow areas(basic parameter and set the compensation), by pressing “→ ←” to move the cursor to realize.

Storage of screw compensation curve is with each axis, set screw compensation of X Y Z A axis separately, by pressing “N” “T” “R” “Q” to switch; Every axis of screw compensation interface has two areas(basic parameter and set the compensation), which switch is through pressing “→ ←” to move the cursor .

A) Basic parameter:

Press “↑ ↓” to select current basic parameter to set in basic parameter, press “Enter” to popup a dialog box to import the error compensation of every axis and import the basic information of screw

compensation.

Basic parameter of every axis' error compensation of screw pitch includes as follows:

1. Reserve.

2. Backward checking points.

It is set for points number of compensation in negative direction.

3. Forward checking points.

It is set for points number of compensation in positive direction.

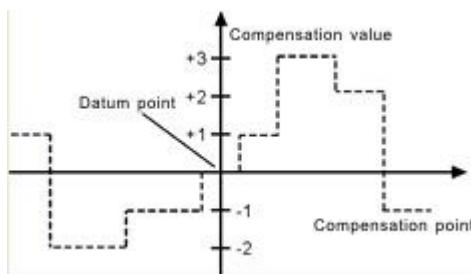
4. Multiple.

It is set for rate of compensation, also actual value=set value * multiple .

5. Distance (um).

It is set for the distance between two compensate points.

Note: Number of compensation points can be set freely, max points of each axis is 300.



B) Set compensation value (No. Coor(mm) Value(um)):

In the area of setting compensation value, it will shows the value of compensation and every axis' error compensation point of screw pitch. Press “↑ ↓ PgDn PgUp” to select current compensation point and press “Enter” to popup a dialog box to import the value of current compensation point.

Test program generation automatically

automatic generate a program of laser interferometer to check the screw compensation. Enter the screw compensation screen and set basic parameters well, press “B” key to check program” to detect program to popup a dialog box and press “Enter” to generate corresponding checking program of screw compensation.

System calculates the distance of compensation points automatically according to basic parameter. Distance is uniform, which could be set according to different axis, and user can set compensation value of each point (System requires input absolute value, relating to value of datum point).

Example1: Linear axis: when length of travel is -400mm~+800mm, distance is 50mm:

Basic parameters set as follows:

1)Backward checking points: 8

2)Forward checking points: 16

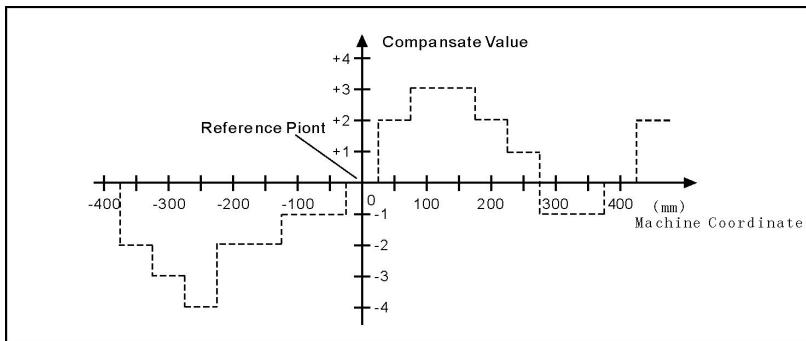
3)Multiple: 1

4)Distance(um): 50000

Corresponding compensation point and value:

No.	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	.	25
Value	+2	+1	+1	-2	0	-1	0	-1	+2	+1	0	-1	-1	-2	0	+1	+2		+1

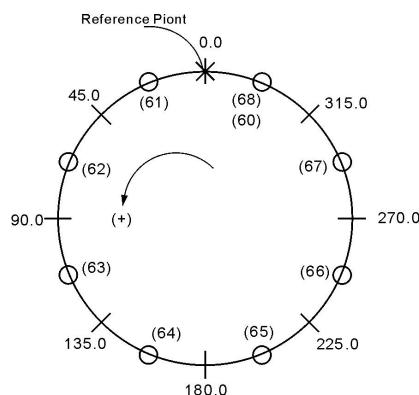
The contrasted chart of compensation points and value as follows:



Note: Zero point is reference point, don't account into checking point.

Example 2: Rotary axis: when movement per revolution is 360° , interval of points 45° , Basic parameters set as follows:

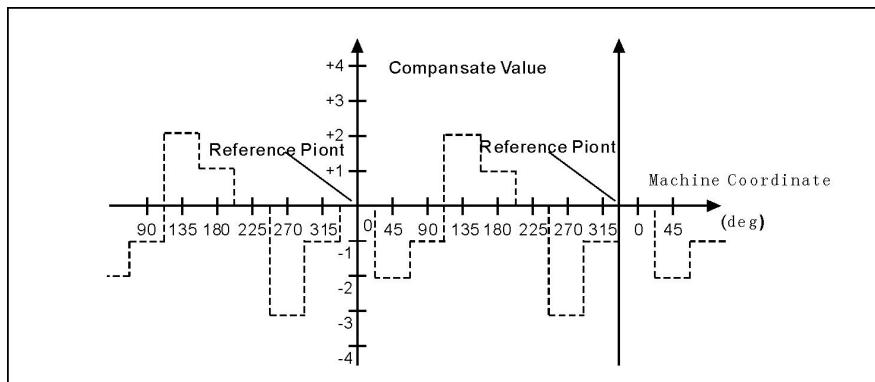
- 1) Backward checking points: 0
- 2) Forward checking points: 8
- 3) Multiple: 1
- 4) Distance(um): 45000



Output compensation value at corresponding point:

NO.	0	1	2	3	4	5	6	7	8
VALUE	+1	-2	+1	+3	-1	-1	-3	+2	+1

Compensation point and value contrast:



Note: 1. In the system, when axis is rotary axis, the coordinate value is $0 \sim 360$. 0 and 360 are at same position:

Example: When input $A0$ & $A360$, both will rotate to zero position.

2. Zero point is reference point, don't account into checking point.

Chapter 7 Installation & Connection

7.1 system installation and connection

At first, users should check whether the hardware is complete, unwound and compatible, such as: CNC system, driving power, servo motor, photoelectric encoder, electric tool carrier.

The installation of CNC system must be fastened tightly, with some spaces around to ensure the ventilation of air. Panel should be put in a place where it is not only convenient to operate and but also able to avoid hurt of heating by scrap iron.

Intense current, week current must be put separately, CNC system and driver should be possibly away from the machine intense current. In order to reduce interference, all signal cables should be kept away from AC contactor. Photoelectric encoder, limit, basic point signal are advisably not to be connected directly to CNC system through intense current box. All power cords must be earthing.

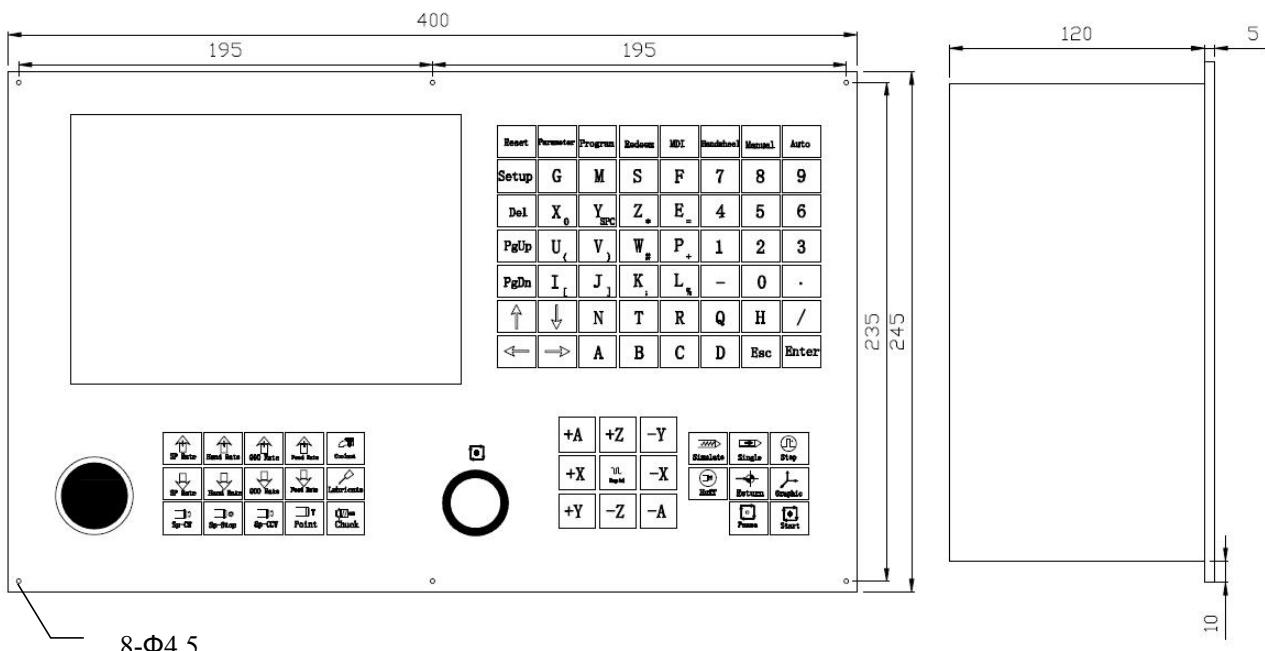
Fix all plugs with screw. Forbid to insert and extract all cables when power is on.

In installation of CNC system, panel should avoid hurting by hard and sharp materials. If the painting of other part of machine is needed, please take off CNC system to keep it clean.

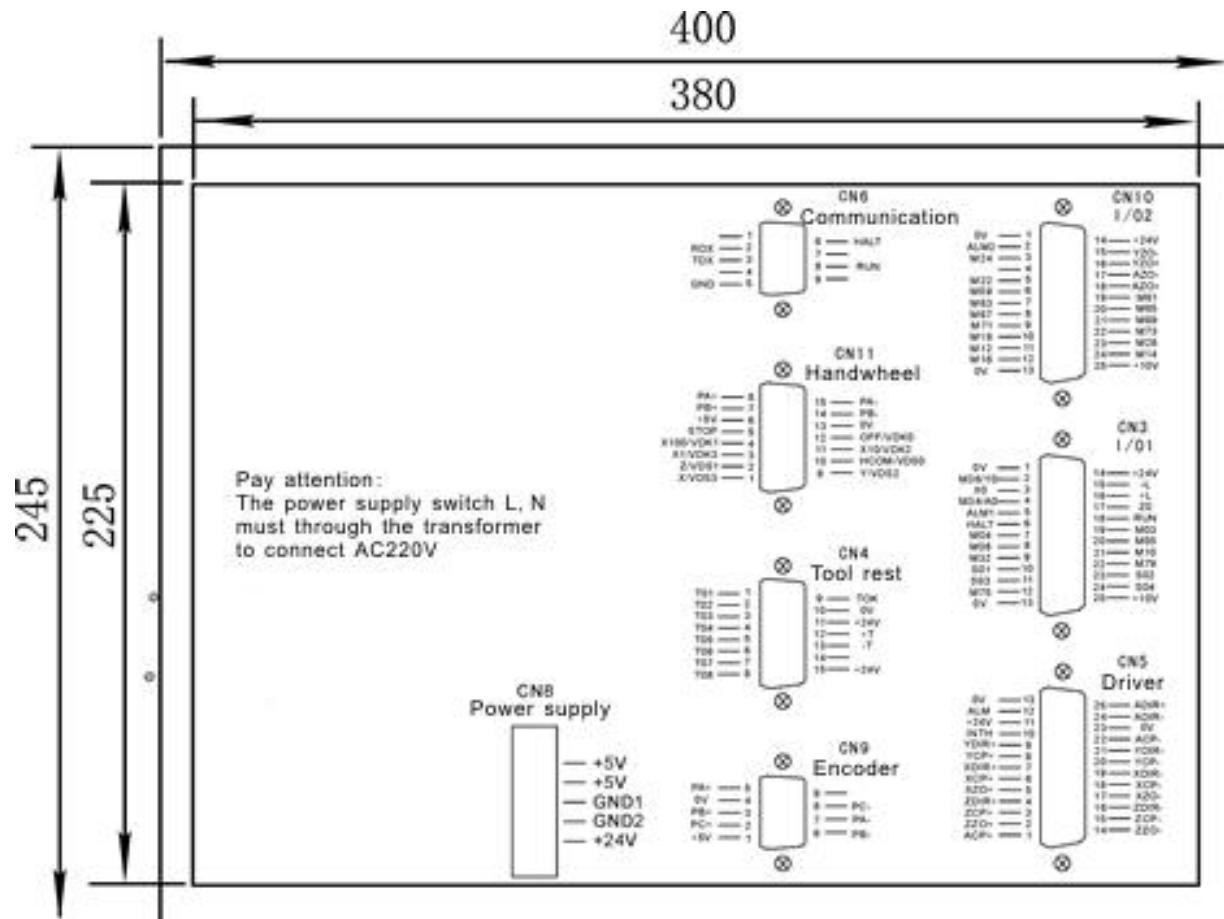
To ensure there is no strong magnet and current interference, keep away from inflammable, explosive and other danger materials.

7.2 system installation dimension

This system has two types of installation, except that the installation dimension are different, the other functions are same.



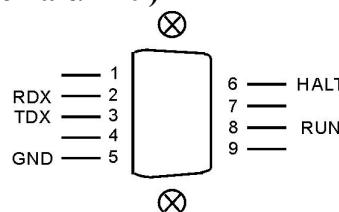
7.3 System Rear View



Attention: switching power supply L, N must be connected to AC 220V, current 0.5A through isolation transformer.

7.4 Interface Connection Graph

7.4.1 CN6 Communication Socket (Female/DB9)



CN6 communication signal with Female socket DB9

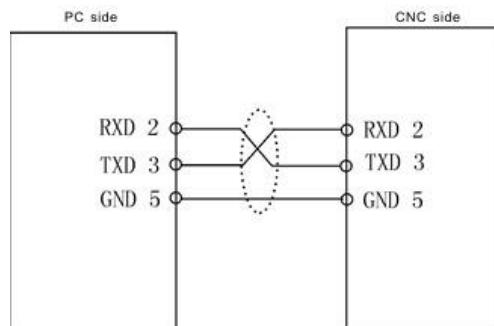
signal	pin	I/O	function	Valid
0V	5	OUT	The ground of signal	0V
RXD	2	IN	The received data signal	
TXD	3	OUT	The transmission of data signal	
RUN	8	IN	Run program	0V
HALT	6	IN	Pause program	0V

Note: 1. Connect to external PC with data communication, must be equipped with our special communication software, which is "SZGHCNCCS" software.

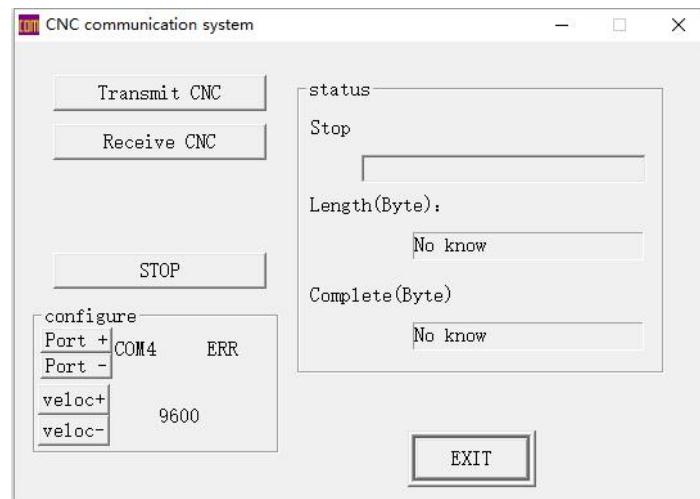
P37 in Other parameter is set rate of CNC system.

2. Communication line must adopt the shielded twisted pair cable, length shall not exceed 10m.

The signal of communication socket connect to PC:

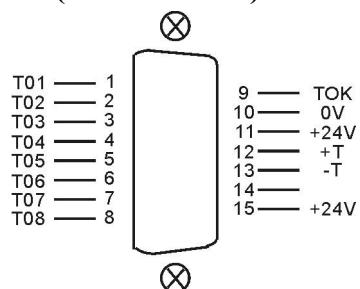


When PC programming, the files should be text files, which could be edit by Notepad or wordplay.



SZGHCNCCS software

7.4.2 CN4 Turret Socket (Female/DB15)

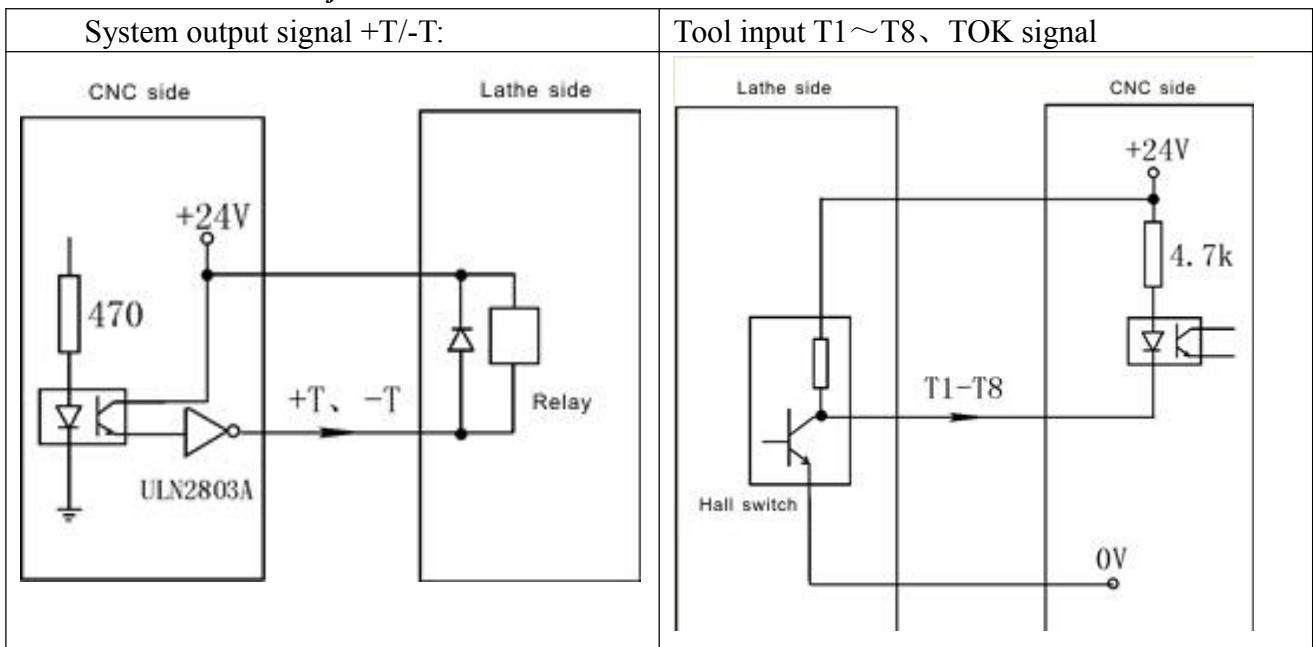


CN4 Turret signal with Female socket DB15

Signal	Pin	I/O	function	Valid
0V	10	OUT	0V	0V
+24V	11,15	OUT	+24V	+24V
+T	12	OUT	Y18 Output	0V
-T	13	OUT	Y19 Output	0V
T1	1	IN	X0 Input Signal	0V
T2	2	IN	X1 Input Signal	0V
T3	3	IN	X2 Input Signal	0V
T4	4	IN	X3 Input Signal	0V
T5	5	IN	X4 Input Signal	0V
T6	6	IN	X5 Input Signal	0V
T7	7	IN	X6 Input Signal	0V
T8	8	IN	X7 Input Signal	0V
TOK	9	IN	X21 Input signal	0V

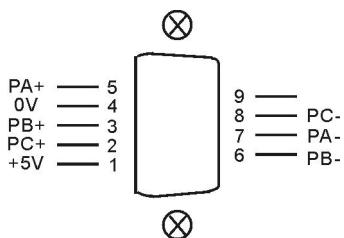
Attention: 1. All the input or output is for system, input is from exterior signal to system, the output is from system signal to external.

2. When choosing the electrical appliance plate, +T and -T control single contact middle relay, user should install two AC contactors of +T and -T.



Attention: it must plus a reverse diode in order to cancel reverse current.

7.4.3 CN9 Spindle Encoder Socket (Female/DB9)

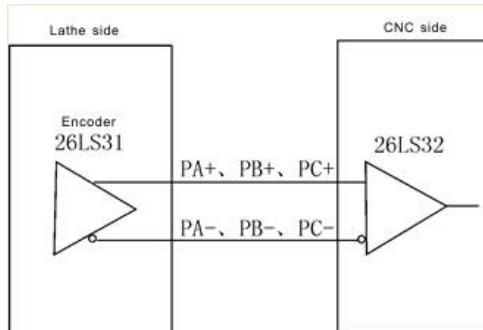


CN9 SP_Encoder signal with Female socket DB9				
Signal	Pin	I/O	Function	Valid
0V	4	OUT	0V	0V
+5V	1	OUT	+5V	+5V
PA+	5	IN	A Phase Positive signal	5V
PA-	7	IN	A Phase Negative signal	
PB+	3	IN	B Phase Positive signal	5V
PB-	6	IN	B Phase Negative signal	
PC+	2	IN	Z Phase Positive signal	5V
PC-	8	IN	Z Phase Negative signal	

Attention:

1. The output signal of encoder adopt the output way is line output, the power supply is +5V.
2. The signal line must adopt shielded twisted pair cable, the length is 20m at most.

The input signal of encoder PA PB PC:



Pay attention:

When machine is configured with inverter+ac motor and customer want to do some special processing, like G84, it needs to fix an encoder to spindle motor.

P412: number of spindle teeth

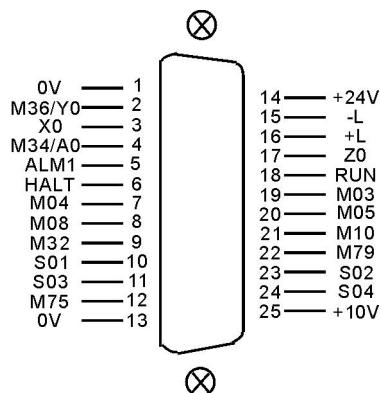
413, number of encoder teeth

When transmission ratio of spindle and encoder not as 1:1, please modify P412&P413 in Axis parameter when teeth of spindle is not more than teeth of encoder;

If teeth of spindle is more than teeth of encoder, it needs to select adapter plate of SZGH;

Note: it must be integer multiple relationship about teeth between spindle & encoder.

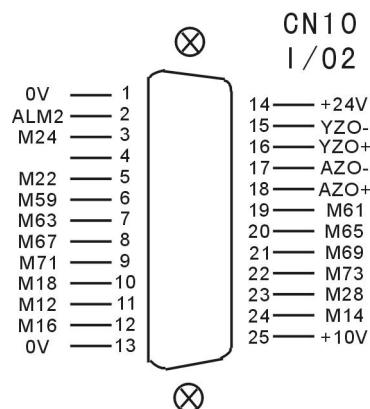
7.4.4 CN3 IO1 Control Socket (Female/DB25)



CN3 I/O1 signal with Female Socket of DB25				
signal	pin	I/O	function	Valid
0V	1	OUT	0V	0V
+24V	14	OUT	+24V	+24V
Y0	2	IN	Zero Point of Y-axis	0V
X0	3	IN	Zero Point of X-axis	0V
Z0	17	IN	Zero Point of Z-axis	0V
-L	15	IN	Positive limit	0V
+L	16	IN	Negative limit	0V
M34/A0	4	IN	M34/Zero Point of A-axis	0V
ALM1	5	IN	Alarm1 of Spindle	0V
HALT	6	IN	Pause	0V
RUN	18	IN	Run	0V
M03	19	OUT	Clockwise Rotation of Spindle	0V
M04	7	OUT	Counter clockwise Rotation of Spindle	0V
M05	20	OUT	Stop of Spindle	0V
M08	8	OUT	Coolant	0V
M10	21	OUT	Chuck	0V
M32	9	OUT	Lubrication	0V
M79	22	OUT	User-defined output8	0V
S01	10	OUT	Spindle first gear	0V
S02	23	OUT	Spindle second gear	0V
S03	11	OUT	Spindle third gear	0V
S04	24	OUT	Spindle fourth gear	0V
M75	12	OUT	User-defined output9	0V
+10V	25	OUT	Analog Output Signal of 1st spindle	0~10V
0V	13	OUT	Ground of frequency conversion	0V

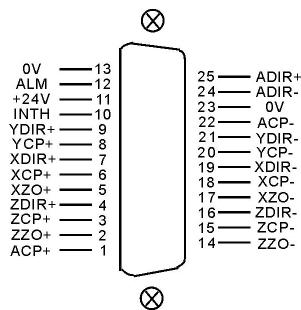
Note: when your CNC system isn't configured with Y-axis & A-axis, M36/Y0 & M34/A0 could be used as input point, controlled by M36/M34 code.

7.4.5 CN10 IO2 Socket (Female/DB25)



CN10 I/O2 signal with Female Socket of DB25				
Signal	Pin	I/O	Function	Valid
0V	1	OUT	Ground of the power supply	0V
+24V	14	OUT	24V power supply	+24V
ALM2	2	IN	Alarm2 of Machine Tool	0V
M24	3	IN	User-defined input 7	0V
M22	5	IN	M01 Quasi-stop Input	0V
M59	6	OUT	Huff Output	0V
M61	19	OUT	User-defined output1	0V
M63	7	OUT	User-defined output2	0V
M65	20	OUT	User-defined output3	0V
M67	8	OUT	User-defined output4	0V
M69	21	OUT	User-defined output5	0V
M71	9	OUT	User-defined output6	0V
M73	22	OUT	User-defined output7	0V
M18	10	IN	User-defined input1	0V
M28	23	IN	User-defined input2	0V
M12	11	IN	User-defined input3	0V
M14	24	IN	User-defined input4	0V
M16	12	IN	User-defined input5	0V
YZO+	16	IN	Positive Zero Position signal of Y-axis	5V
YZO-	15	IN	Negative Zero Position signal of Y-axis	
AZO+	18	IN	Positive Zero Position signal of A-axis	5V
AZO-	17	IN	Negative Zero Position signal of A-axis	
+10V	25	OUT	Analog Voltage of 2nd Spindle	0~10V
0V	13	OUT	Ground of frequency conversion	0V

7.4.6 CN5 XYZA Drive Socket (Male/DB25)



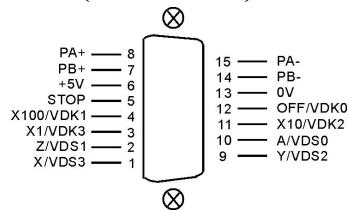
CN5 XYZA Driver with Male Socket of DB25

Signal	Pin	I/O	Function	Valid
XCP+	6	OUT	Positive Pulse signal of X-axis	5V
XCP-	18	OUT	Negative Pulse signal of X-axis	
XDIR+	7	OUT	Positive Direction signal of X-axis	5V
XDIR-	19	OUT	Negative Direction signal of X-axis	
YCP+	8	OUT	Positive Pulse signal of Y-axis	5V
YCP-	20	OUT	Negative Pulse signal of Y-axis	
YDIR+	9	OUT	Positive Pulse signal of Y-axis	5V
YDIR-	21	OUT	Negative Pulse signal of Y-axis	
XZO+	5	IN	Positive Zero position signal of X-axis	5V
XZO-	17	IN	Negative Zero position signal of X-axis	
ZCP+	3	OUT	Positive Pulse signal of Z-axis	5V
ZCP-	15	OUT	Negative Pulse signal of Z-axis	
ZDIR+	4	OUT	Positive Direction signal of Z-axis	5V
ZDIR-	16	OUT	Negative Direction signal of Z-axis	
ZZO+	2	IN	Positive Zero Position signal of Z-axis	5V
ZZO-	14	IN	Negative Zero Position signal of Z-axis	
ACP+	1	OUT	Positive Pulse signal of A-axis	5V
ACP-	22	OUT	Negative Pulse signal of A-axis	
ADIR+	25	OUT	Positive Direction signal of A-axis	5V
ADIR-	24	OUT	Negative Direction signal of A-axis	
0V	13,23	OUT	0V	0V
ALM	12	IN	Alarm signal of Servo driver	0V
+24V	11	OUT	+24V	24V
INTH	10	OUT	Reset alarm signal	0V

Note: 1. The signal line must adopt shielded twisted pair cable, the length is 20m at most.

2. Whether the alarm signal ALM is normal open or normal close is set by P17 in Other parameter.

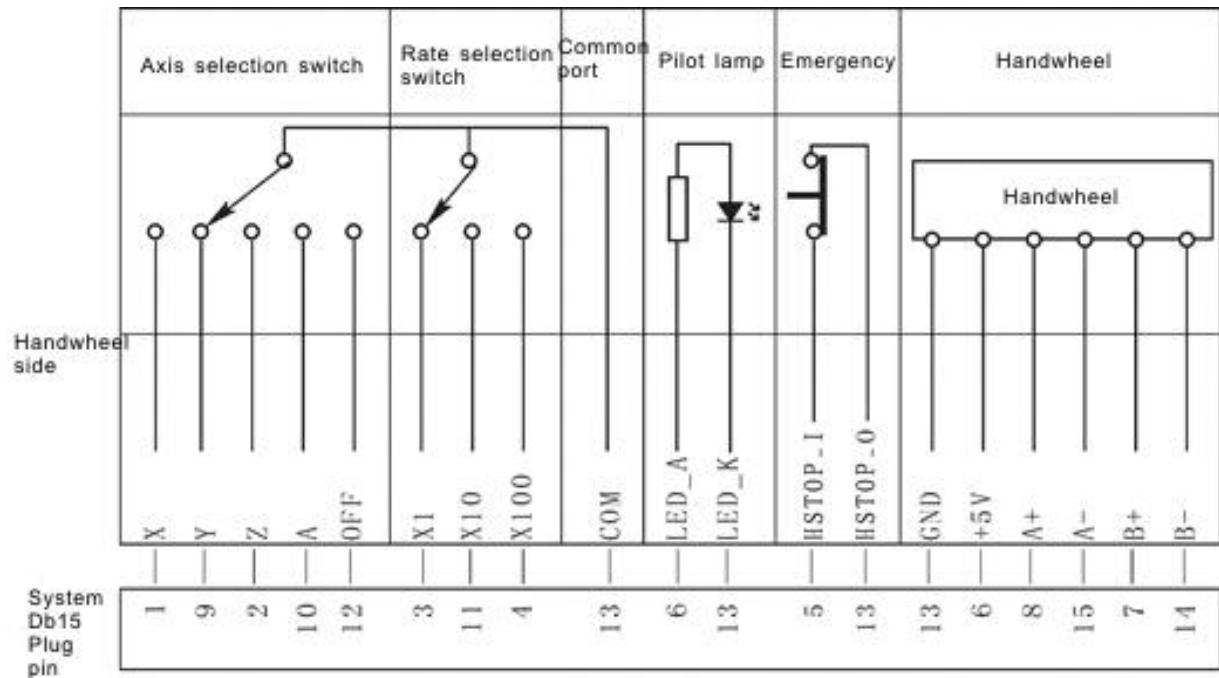
7.4.7 CN11 MPG/Handheld Box Socket (Male/DB15)



CN11 Handwheel Signal with Male Socket of DB15				
signal	pin	I/O	function	Availability
0V	13	OUT	0V	0V
+5V	6	OUT	+5V	+5V
PA+	8	IN	A signal +	5V
PA-	15	IN		
PB+	7	IN	B signal +	5V
PB-	14	IN		
STOP	5	IN	emergency stop	0V
OFF/VDK0	12	IN	Off/ feed amending 0	0V
X100/VDK1	4	IN	*100/ feed amending 1	0V
X10/VDK2	11	IN	*10/ feed amending 2	0V
X1/VDK3	3	IN	*1/ feed amending 3	0V
A/VDS0/HALT	10	IN	A/SP amending 0/halt stop	0V
Z/VDS1	2	IN	Z/SP amending 1	0V
Y/VDS2/RUN	9	IN	Y/SP amending 2/run	0V
X/VDS3	1	IN	X/SP amending 3	0V

7.4.7.1 Electrical handwheel (Manual pulse generator)

Handwheel contact diagrammatic as:

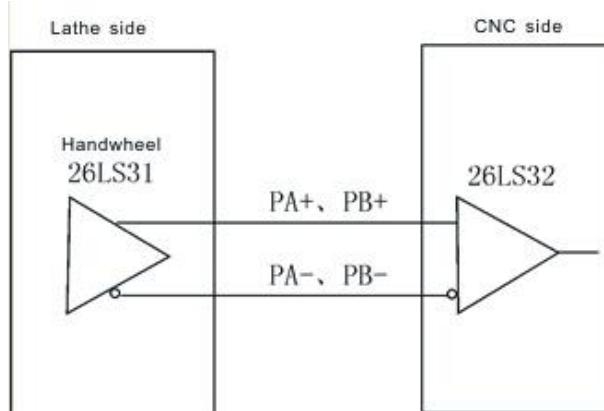


When user connect our handheld box to CN11 Plug, P1 in Other parameter needs to set 1, and cannot use band switch to adjust SP_Rate, Feed_Rate & External Run/Pause, and P1&P2 in

Axis parameter only could be “0”. X Y Z A X1 X10 X100 inputs are for axis-selection & rate, P33&P34 in Other parameter only could set to 0.

PA+ PB- PA+ PA- are corresponding input signal of handwheel pulse A B.

The input signal of handwheel:



Attention:

1. The output signal of handwheel adopts line output, the power supply is +5V.
2. Just connect PA+ PB+ if adopt voltage output.
3. Manual pulse generator needn't switch button for Enter ON/OFF handwheel, if there is a switch for Enter, it is okay that use short connection of switch.

7.4.7.2 Using for Band Switch

When P1 & P2 in Axis parameter is set to “1”, VDK0/VDK1/VDK2/VDK3 & VDS0/VDS1/VDS2/VDS3 are working, which can't as inputs for external Run/Halt button, P1 in Other parameter is 0;

VDS0(A) VDS1(Z) VDS2(Y) VDS3(X) are the input signal of adjust rate of spindle, total 16 gears. VDK0(OFF) VDK1(X100) VDK2(X10) VDK3(X1) are inputs signal of adjust Rate of Feeding speed, total 16 gears.

7.4.7.3 External Switch for Run/Halt

When P33 in Other parameter is “1”, PIN9 of CN11 plug can be as input for external Run , which running program automatically; When P34 in Other parameter is set to “1”, PIN10 of CN11 plug can be as input for external Halt, which pause processing program.

7.4.7.4 Using for External Emergency Stop

STOP signal is the input signal of external emergency button, P27 in Other parameter is set for type of switch of emergency stop button. 0: NO type, 1: NC type.

Suggestion: Configured with SZGH Handheld box(MPG), which is better to operate SZGH CNC system.

7.5 SZGH-CNC-IO-12 IO Relay Board

SZGH-CNC-IO-12 is newest version I/O Relay board with 12 pieces of relays, which is used for connecting all inputs & outputs on CN3/CN4/CN10 plugs of CNC controller to external switches & loads easily.

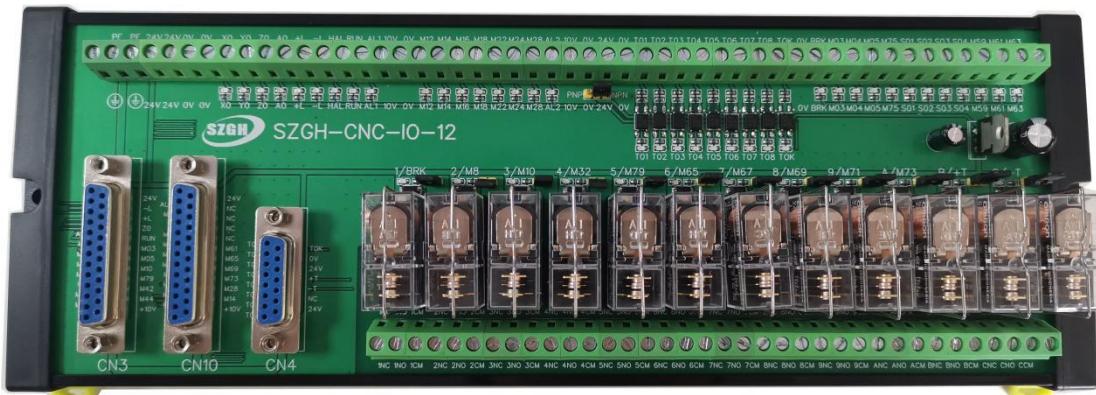


Fig1 Practical Picture of SZGH-CNC-IO-12

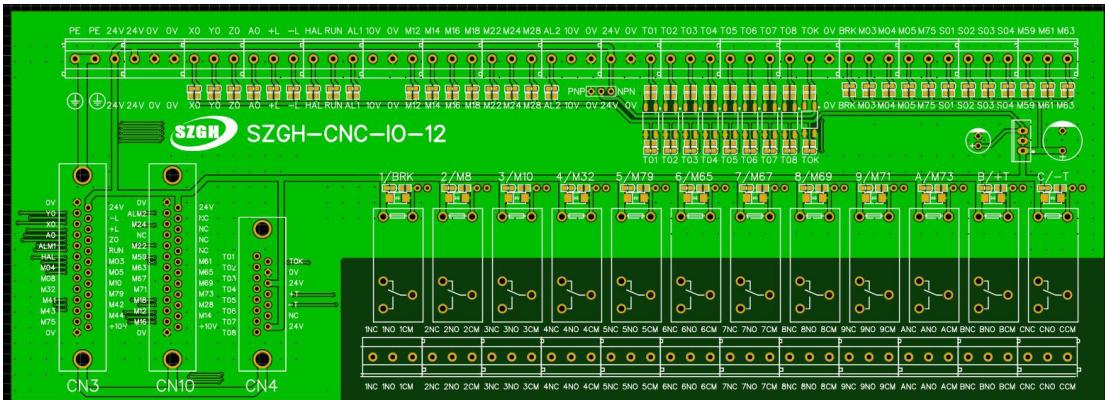


Fig2 Design Sketch of SZGH-CNC-IO-12

CN3 socket is corresponding to CN3 plug of CNC Controller one by one;

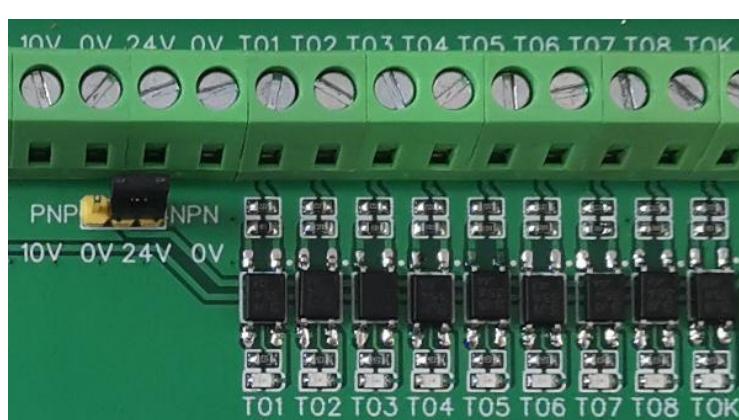
CN10 socket is corresponding to CN10 plug of CNC Controller one by one;

Note: Pin4&Pin15-Pin18 on CN10 are null.

CN4 socket is corresponding to CN4 plug of CNC Controller one by one

Note: Pin14 on CN4 plug is null.

Update Point 1:



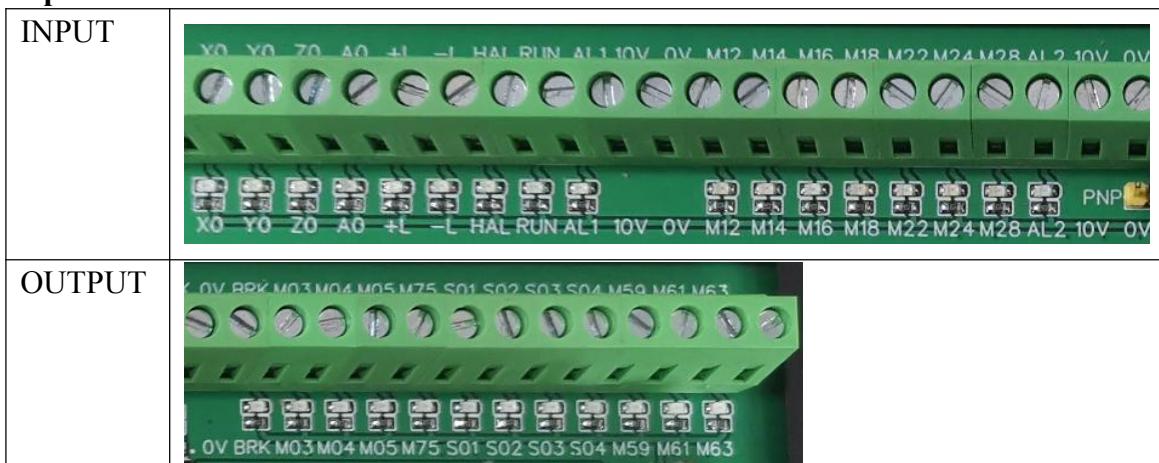
T01-T08 & TOK can support NPN & PNP input.



When short-circuit cap is at side of NPN, also COM is connected to +24V, valid level of T01-T08 & Tok inputs are 0V (also NPN), also input 0V to these inputs, which will activate valid of input points.

When short-circuit cap is at side of PNP, also COM is connected to 0V, valid level of T01-T08 & Tok inputs are +24V (also PNP), also input +24V to input these points, which will activate valid of inputs.

Update Point 2:



There are more reasonable layout, which can let user to make wiring more easily.

① Ports on upper side of SZGH-CNC-IO-12 IO relay board are includes all IOs (includes HALT/RUN/TOK/M05) of CN3 & CN4 & CN10 plugs, except output pins for relays. And there are more common ports, such as +24V, 0V, PE. And also it is more easily to expand connections of external relays.

② Strong and Weak electricity are separated, avoid wrong connections, which will damage CNC controller easily. Weak electricity are at upper side of SZGH IO relay board, Strong electricity are at output ports of relays, which is for ON/OFF loads directly.

Update Point 3:



There are 12 pieces of relays output with Omron brand relays. All outputs of relays are with NC output/NO output & COM port.

Default control signals for 12 pieces of relays as following table:

No.	1	2	3	4	5	6	7	8	9	A	B	C
Output	BRK	M8	M10	M32	M79	M65	M67	M69	M71	M73	+T	-T

1/BRK relay is for control brake on servo motor. Brake control input port is at BRK on upper side. If we don't need to control brake, we want to use this relay for M3, connect M3 to BRK, also make short circuit between BRK & M3.

There are short-circuit caps/LED indicator/reverse diode on control side of relays. Short circuit cap: we can renew define function of relays by short-circuit cap. Example: We don't need to use 3/M10 for spindle chuck, and want to use it for M4 relay control output, remove short-circuit, there are two pins, pin in the left side is control pin of relay, pin in the right side is M10 output of CNC, so we connect M4 to pin in the left side of 3/M10 relay.

We can define M65/M67/M69 outputs for Yellow/Red/Green indicators by P28 & P29 to 1 on Other parameter.

Note: Outputs ports of relays can suffer 10A, 250VAC/30VDC

Note:

1. *HAL, also HALT, Pin6_CN3 plug, AL1, also ALM1, Pin5_CN3 plug, AL2, also ALM2, Pin2_CN10 plug.*
2. *When output points is with relay, user can use them to control loads directly; when output points is output from CNC controller directly without relay, it cannot control external devices directly, otherwise it will damage CNC controller.*
3. *When power of controlling devices is over 250VAC/10A, please add contactors.*
4. *Valid level of all inputs & outputs of SZGH CNC controller is 0V.*
5. *When without Y-axis/C-axis, Y0 is input point of M36 command.*
6. *M79 is for tailstock on Lathe system, user-defined output on Milling system.*
7. *Above output ports (M3/M4/M5/M75/M41/M42/M43/M44/M59/M61/M63) are output from ports of CNC directly, without controls relays, which output drive current of these outputs are about 50mA, cannot control loads directly, otherwise it will damage these output points & CNC controller.*
8. *NC, also Normal Close ; NO, also Normal Open; CM, also Common port.*

7.6 Daily Maintenance and Repair

In order to fully use CNC system's function and promote efficiency, the most important work is correctly using system, and notice system's daily maintenance work, promote Mean Time Between Failures MTBF. Now this system's maintenance method is introduced as follows:

7.6.1 Maintain

System's using must be under the good circumstance.

Operator, programmer and repairer must be familiar with NC machining technology, and according to the requirements of the user manual correctly use, do one's best to avoid improper operation.

Everyday operator should clean the system's box and panel in case of corrupting and sundries to indemnify it.

When CNC system's using time is over three months, operator should open the system box and clean inside.

If not using system for long time, should boot the system one time every week.

7.6.2 Ordinary Problem

7.6.2.1 System can't boot

- 1) check if input power is normal.
- 2) check if power switch is turned on.
- 3) check insurance.

7.6.2.2 No display as boot

- 1) Boot again or reset.
- 2) Check if power supply's +5V, +12V, -12V, -24V are normal.
- 3) Check if transformer is bad.
- 4) Check if LCD's brightness adjustment and connection are normal.
- 5) Check if main board is normal.

7.6.2.3 System's control disorganized

- 1) Wrong operations.
- 2) Anti-jamming ability of power supply is poor.
- 3) Working environment of CNC system is too bad.

7.6.2.4 Loss of user program

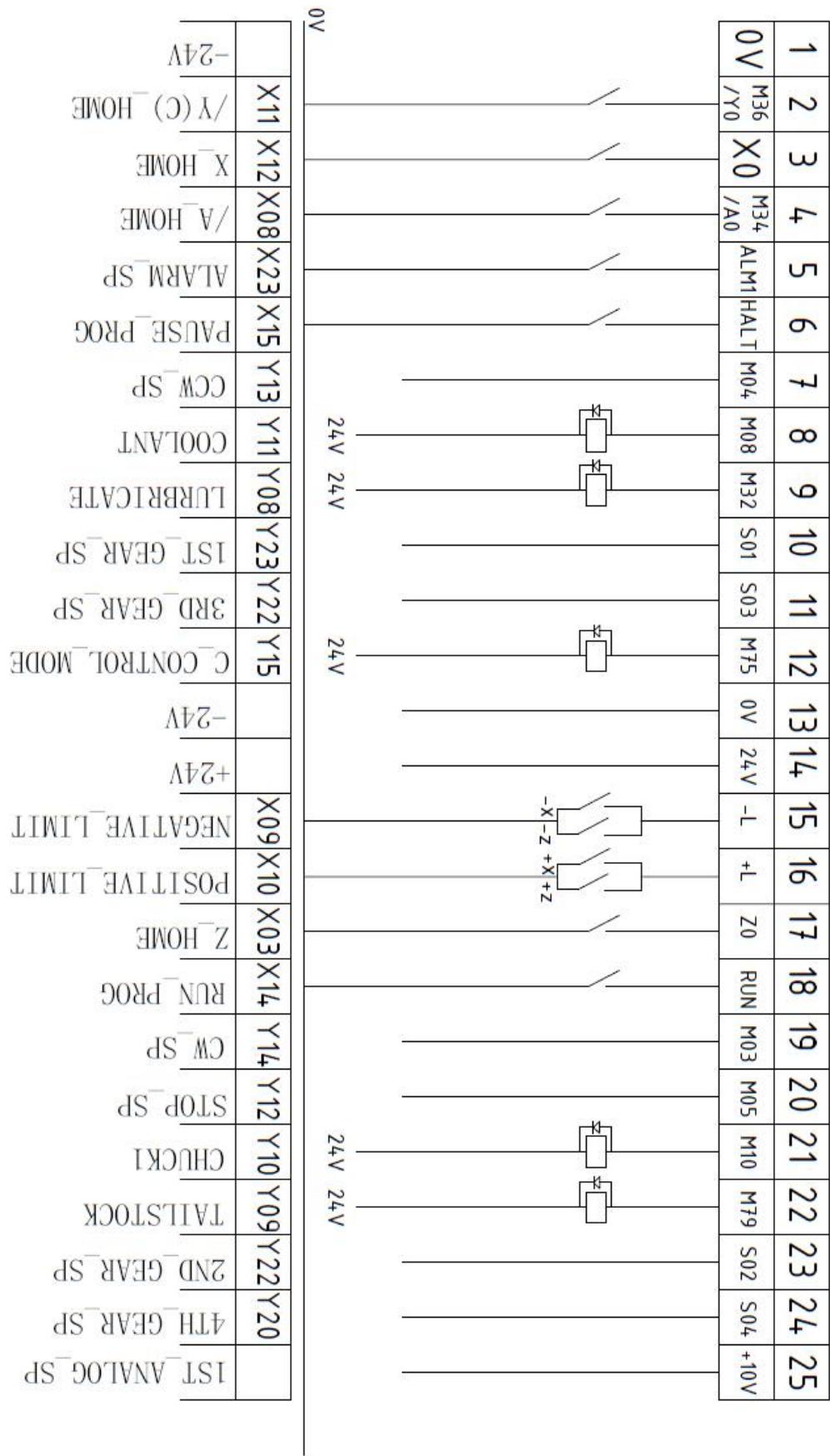
The DC battery on system main board can ensure user's program and parameter don't lose. When system isn't used for half a year or system has been used for over two years, the battery may be invalid, therefore, should exchange battery.

7.6.2.5 Machining precision is bad

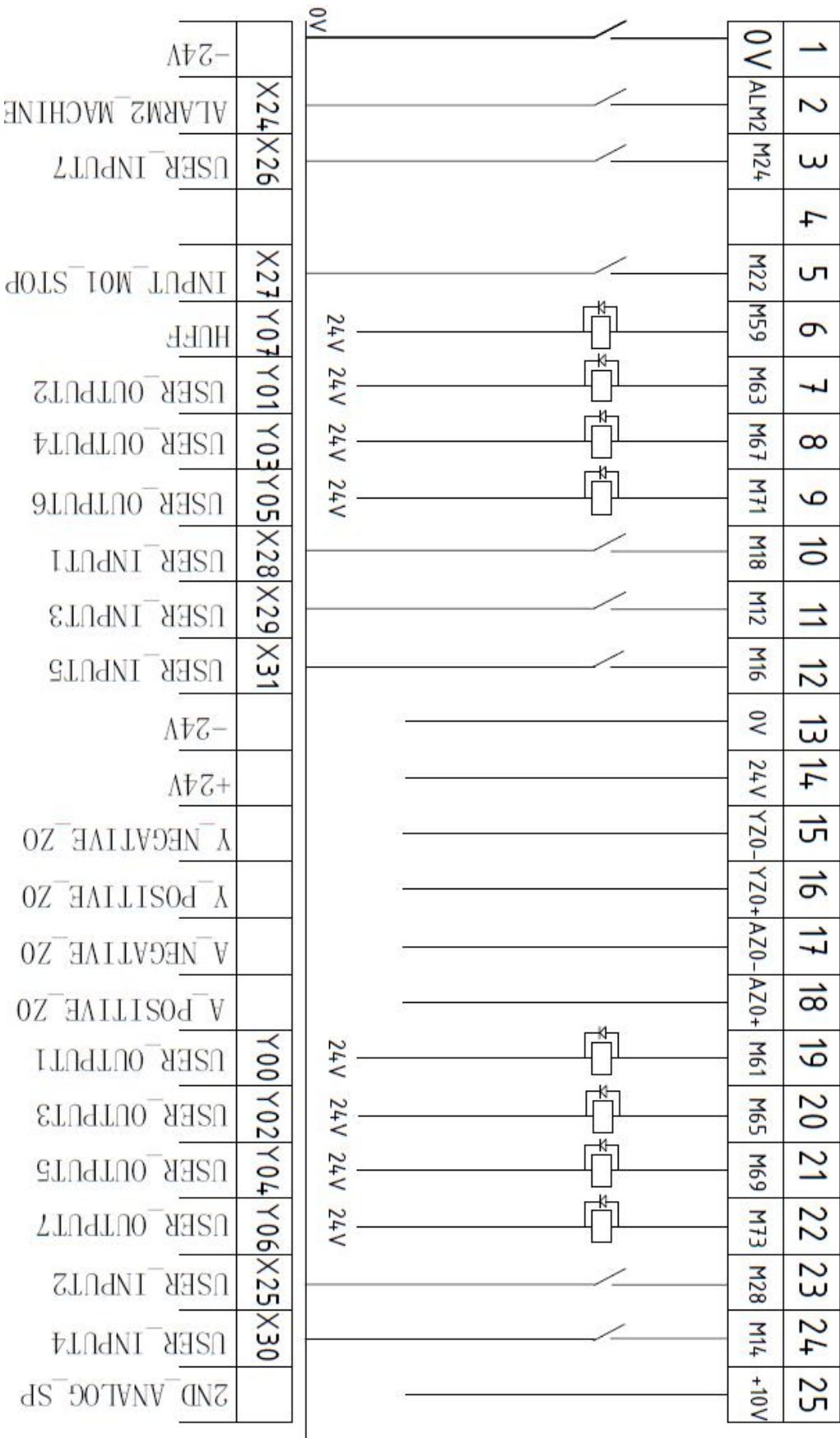
- 1) CNC Machine needs to revise backlash after using for some time.
- 2) Best to revise base point before machining in order to ensure the start point's precision.
- 3) Machining speed and cutting depth are inappropriate.
- 4) Machine connector's pin falls off.
- 5) Tool is not tightened.
- 6) Piece clamp is not good.
- 7) Tool's giving up is not equal because piece's dimension is not uniform.
- 8) Problems of machine tool

Attention: Because of many kinds of reasons this Manual book may have some mistakes. We will appreciate that you feedback to us or our agents, to provide services and technical support for every customer.

Appendix I: Wiring Diagram of CN3 Plug



Appendix II: Wiring Diagram of CN10 Plug



Appendix III: Wiring Diagram of CN4 Plug

