User Manual

SZGH-CNC990TDb(series)
Lathe Control System
V4.0

Shenzhen Guanhong Automation CO.,LTD

Website: www.szghauto.com

Add:QingShuiWan Building,No 7-1 Tangkeng Road, Liuyue community, Henggang Street , Longgang District, Shenzhen City,Guangdong Province, China

Post code: 518100

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 merchant-ability 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 verse, 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 . 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

CNC machine tool is an electron-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.

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

SZGH-CNC990TDb series CNC control system is economic type CNC control system for lathe machine , which is developed by Shenzhen Guanhong Automation Co.,Ltd. And we have already made updates basic original CNC990TDb CNC Controller.

Fig1.1 SZGH-CNC990TDb

1.1 Characteristics

- 1) 800*600 8.4 inch real color LCD Display
- 2) Support ATC function , Macro function and PLC function
- 3) Electric Turret & Binary code Turret & Special Turret, Max: 99 pieces of tools
- 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 Z Y(C) 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: 240 m/min Feeding speed:0.001-30m/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 chamfering

Thread

• Acceleration and deceleration function
• Common thread(follow the spindle) / Rigid thread
• Single-headed/Multi thread of straight ,taper and terminal surface in metric system/inch system,equal and variable pitch thread
• Thread retract in length ,angle and speed characteristics can be set
• Thread pitch:0.1~1000.000mm or 0.1~99 tooth/inch
• Rapid traverse: linear type or S type
• The starting speed,finishing speed and time of acceleration and deceleration are set by parameter

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
• Rigid tapping

Tool Function

•Tool length compensation
• tool nose radius compensation (C type)
• Tool wearing compensation
• Method of setting tools: Tool-setting in fixed position ,trial cutting tool -setting,,auto tool setting
• Tool offset executing mode: Rewriting coordinate mode,tool traverse mode

Precision compensation

• Backlash compensation/Pitch error compensation in memory type
• Built-in Screw Compensation

PLC function

• Refresh cycle: 8ms
• PLC program can be altered on PC or on CNC , 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 two-dimensional 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
• Calling program: Support macro program ,subprogram

Community function

• RS232: Files of part program can be transmitted
• USB: File operation and file directly processing in flash disk,support PLC programs,flash disk of software upgrade.

Safety function

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

1.3 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	G65	Macro command non-mode calling
G32	Threading Cutting	G66	Macro command mode calling
G31	Jumping function	G67	Macro program mode calling calling
G311	Jumping function	G40	Tool nose radius compensation cancel
G70	Finishing Cycle	G41	Tool nose radius left compensation
G71	Axial Roughing in Cycle	G42	Tool nose radius right compensation
G72	Radial Roughing in cycle	G26	Return to starting point of program
G73	Close Cutting Cycle	G261	X-axis Return to starting point of program
G74	Axial Grooving Cycle	G262	Y-axis Return to starting point of program
G75	Radial Grooving Cycle	G263	Z-axis Return to starting point of program
G76	Multiple Thread cutting cycle	G264	A-axis Return to starting point of program
G90	Axial Cutting cycle	G265	B-axis Return to starting point of program
G92	Thread cutting cycle	G25	Save current coordinate value
G93	Canned Tapping Cycle	G61	Return to position of G25
G94	Radial Cutting Cycle	G611	X-axis return to position of G25
G22	Program Cycle	G612	Y-axis return to position of G25
G800	Program Cycle Cancel	G613	Z-axis return to position of G25
G15	Polar coordinate command cancel	G614	A-axis return to position of G25
G16	Polar coordinate command	G615	B-axis return to position of G25
G990	Absolute value programming	G28	Return to home of machine
G991	Incremental value programming	G281	X-axis return to home of machine
G20	Inch input	G282	Y-axis return to home of machine
G21	Millimeter input	G283	Z-axis return to home of machine
G04	Dwell	G284	A-axis return to home of machine
G60	Exact stop & position	G285	B-axis return to home of machine
G64	Continuous track processing	M800	C-axis return to zero position of SP-encoder
G50	Set max speed of spindle	M881	C-axis do orientation
G52	Set local coordinate system	G53	Machine coordinate system
G184	Setup/Offset current coordinate value	G54	Workpiece coordinate system 1
G185	Set/Offset all coordinate value	G55	Workpiece coordinate system 2
G96	Constant surface speed control	G56	Workpiece coordinate system 3
G97	Constant surface speed control cancel	G57	Workpiece coordinate system 4
G98	Feeding/min	G58	Workpiece coordinate system 5
G99	Feeding/rev	G59	Workpiece coordinate system 6
G06	3D Space Arc Interpolation	Note: System also includes M codes&Other codes.	

1.4 System operation condition

1) Power supplying

AC 220V(+10%/-15%), Frequency 50Hz±2%. Power:≤ 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℃~45℃	0℃~+55℃
Ambient Humidity	40~80%	≤90%(40℃)
Atmosphere Pressure	86kPa~106kPa	86kPa~106kPa
Altitude	≤1000m	≤1000m

3) operation environment :

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

1.5 Wiring Sketch

Fig1.2 Wiring Sketch for Total CNC Lathe System

Chapter 2 Programming

CNC machine tool is an electron-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.

Software used for controlling SZGH-CNC990TDb Turning Machine CNC system is divided into system software (NC for short) and PLC software (PLC for short). NC system is used for controlling display, communication, edit, decoding, interpolation and acceleration/deceleration, and PLC system for controlling explanations, executions, inputs and outputs of ladder diagrams.

Programming is a course of workpiece contours, machining technologies, technology parameters and tool parameters being edit into part programs according to special CNC programming instructions.CNC machining is a course of CNC controlling a machine tool to complete machining of workpiece according requirements of part programs. Technology flow of CNC machining is as following:

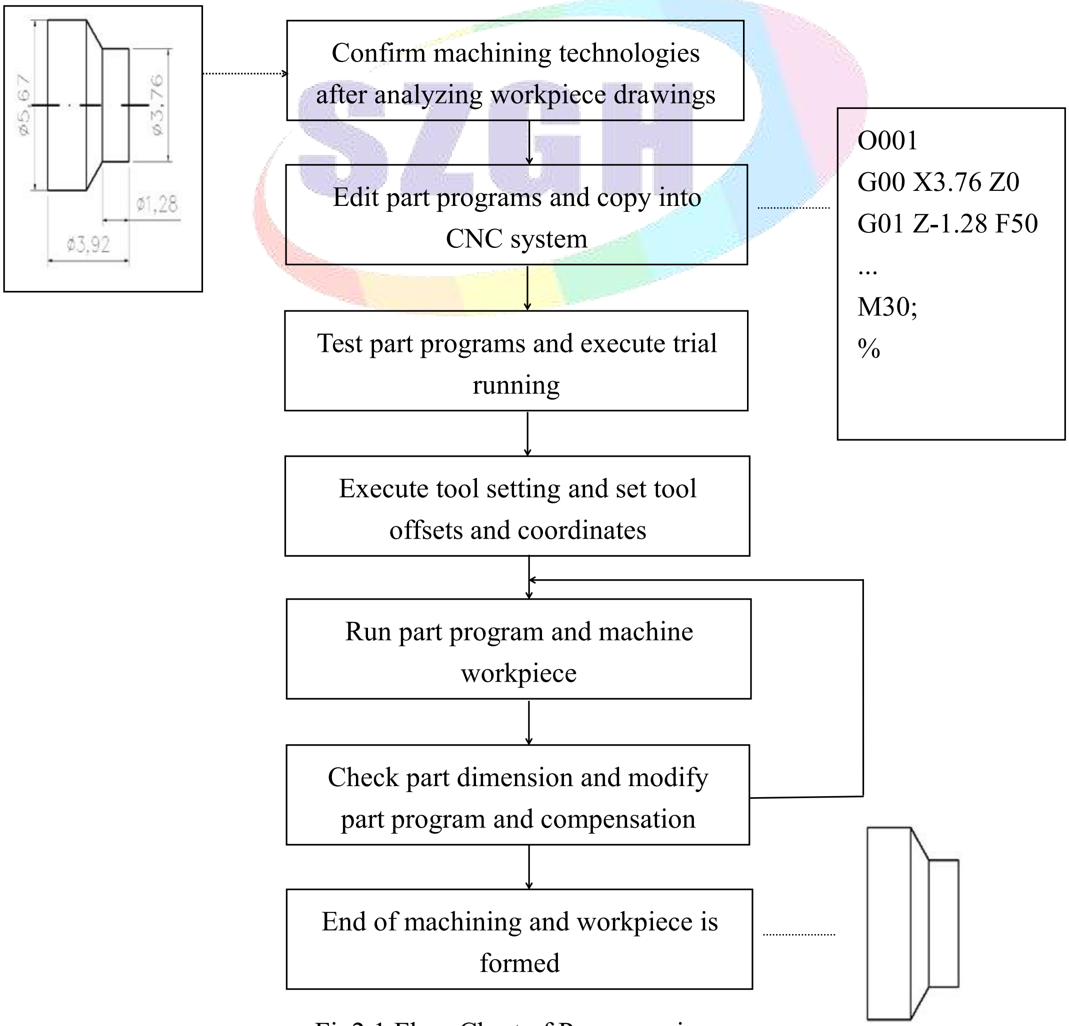


Fig2.1 Flow Chart of Programming

2.1 Basic Concept of Programming

2.1.1 Definition of Coordinate System

Sketch map of CNC turning machine is as following:

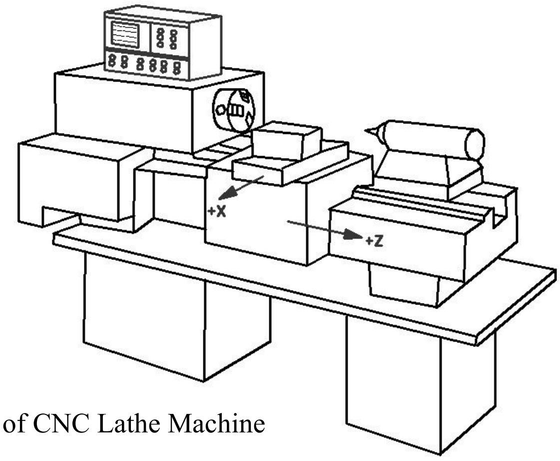


Fig2.2 Sketch Map of CNC Lathe Machine

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

CNC system is employed with a rectangular coordinate system composed of X, Z axis. X axis is perpendicular with axes of spindle and Z axis is parallel with axes of spindle; direction of approach to the workpiece is negative direction and direction are away from workpiece is positive direction.

According to their relative position between the tool-post and spindle, there are 2 kinds of turning coordinate system; one is Front tool-post coordinate system(Fig2.3) , the other is Rear tool-post coordinate system(Fig2.4).It shows exactly the opposite direction in X direction but the same direction in Z direction from figures. In the manual, following figures and examples are based on Front tool-post coordinate system. **Type of CNC Lathe Machine: P3 in Tool parameter.**

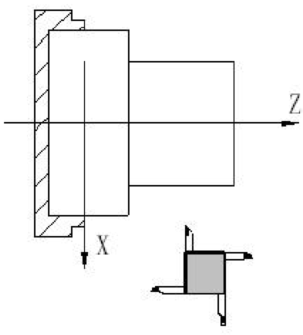


Fig2.3 Front toolpost coordinate system

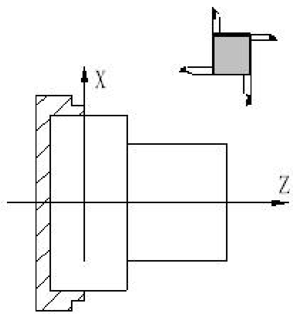


Fig2.4 Rear toolpost coordinate system

2.1.2 Machine Coordinate System and Machine Reference Point

Machine tool coordinate system is a benchmark one used for CNC counting coordinates and a fixed point on the machine tool.

Machine tool origin is named **machine reference point , machine zero** or **home**, which is specified by a reference point return switch on the machine tool. Usually, the reference point return switch is installed on max stroke in X, Z positive direction. The system considers the current coordinates of machine tool as zeroes and sets the machine tool coordinate system according to the current position as the coordinate origin after having executed the machine reference point return.

Machine Reference position is offset point based on machine zero.Offset value is set by P32(X-axis) & P33(Z-axis) in Axis parameter. If P32&P33=0, machine zero & reference position is same. Normally, tool change and programming of absolute zero point as described later are performed at this position.

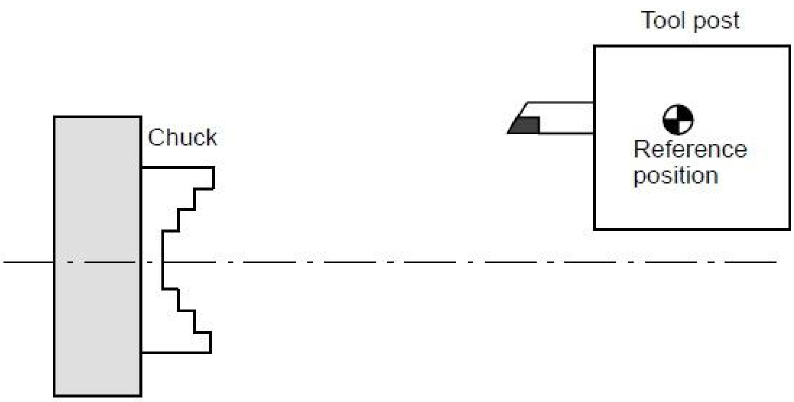


Fig2.5 Reference Position

The system considers the current coordinates of machine tool as zeroes and sets the machine tool coordinate system according to the current position as the coordinate origin after having executed return of machine reference point.

Note: Do not execute the machine reference point return without the reference point switch installed on the machine tool , otherwise movement over limitation of stroke , and broke machine.

2.1.3 Workpiece Coordinate System and Program Reference Point

Workpiece coordinate system is set to a rectangular coordinate system according to part drawings , also named **floating coordinate system**. After the workpiece is clamped on the machine tool, G50 is executed to set an absolute coordinates of tool’s current position according to the relative position of tool and workpiece, and so the workpiece system has been created. The current position of tool is named **program reference point** and the tool returns to the position after executing the program reference point return. Usually, Z axis is consistent with the axes of spindle and X axis is placed on the heading or the ending of workpiece. The workpiece will be valid until it is replaced by a new one.

The current position of workpiece coordinate system set by G50 is named the **program reference point** and the system returns to it after executing the program reference point return.

Note: Do not execute the program reference point return without using G50 to set the workpiece coordinate system after power on.

2.1.4 Coordinate System

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

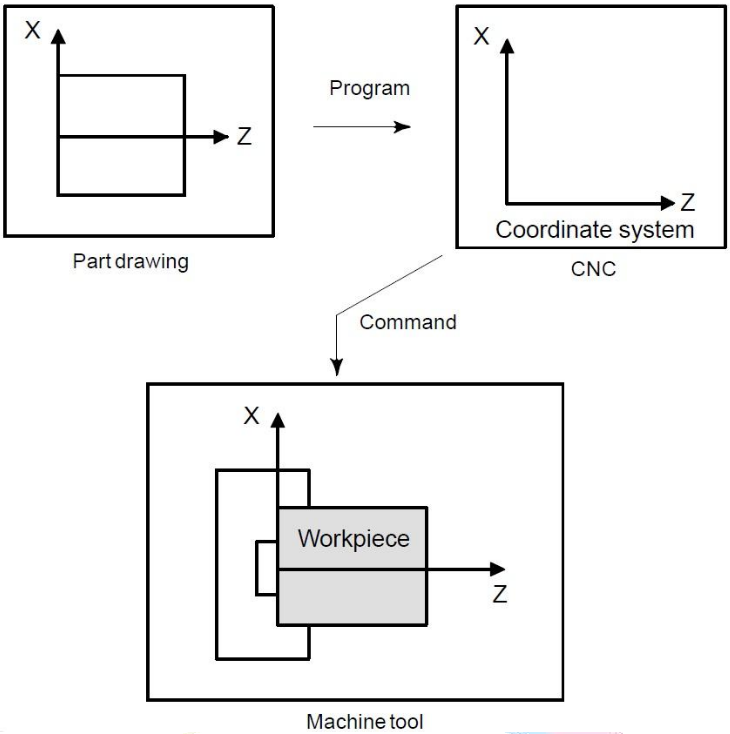


Fig2.6 Coordinate system

The following two coordinate systems are specified at different locations:

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. 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.

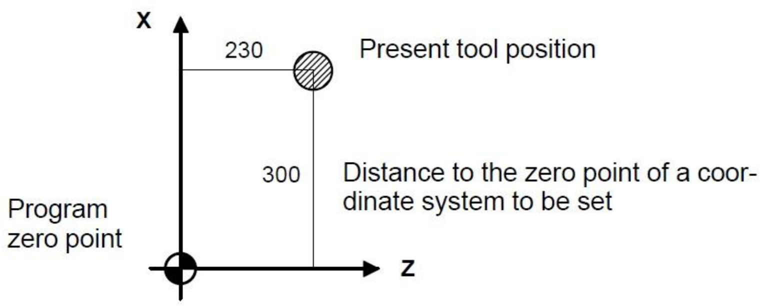


Fig2.7 Coordinate system specified by the CNC

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.

The following method is usually used to define two coordinate systems at the same location.

1. When coordinate zero point is set at chuck face

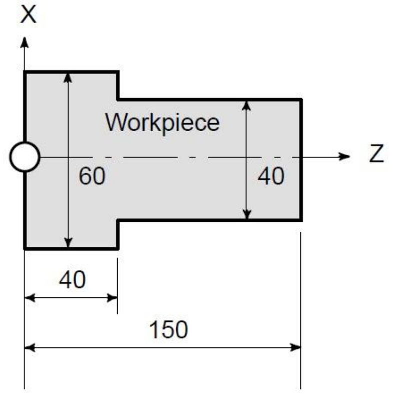


Fig2.8 Coordinates and dimensions on part drawing

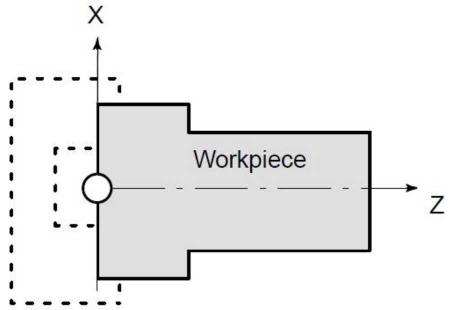


Fig2.9 Coordinate system on lathe as specified by CNC (made to coincide with the coordinate system on part drawing)

2. When coordinate zero point is set at work end face.

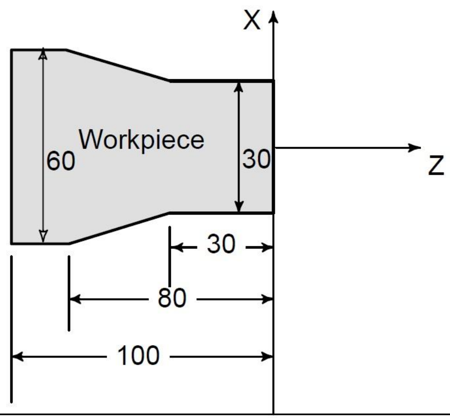


Fig2.10 Coordinates and dimensions on part drawing

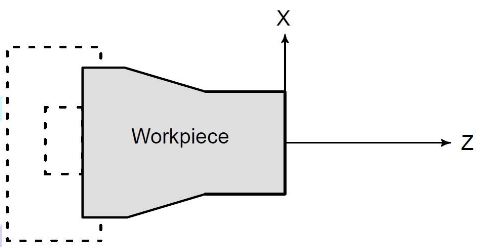


Fig2.11 Coordinates system on lathe as specified by CNC (made to coincide with the coordinate system on part drawing)

2.1.5 Interpolation

The tool moves along straight lines and arcs constituting the workpiece parts figure.

Interpolation is defined as a planar or three dimensional contour formed by path of 2 or multiple axes moving at the same time, also called **Contour control**. The controlled moving axis is called link axis when the interpolation is executed. The moving distance, direction and speed of it are controlled synchronously in the course of running to form the required complex motion path. Fixed point control is defined that the motion path in the course of running are not controlled but end point of one axis or multiple axes moving.

Linear Interpolation: Tool movement alone a straight line

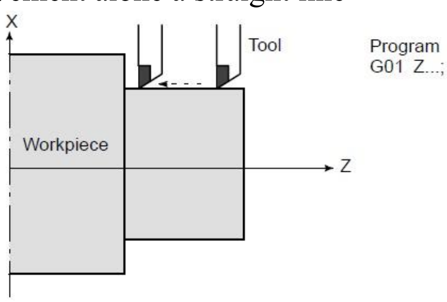


Fig2.12 Tool movement along the straight line which is parallel to Z–axis

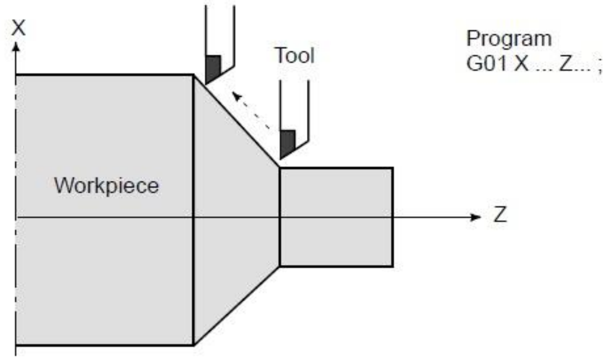


Fig2.13 Tool movement along the taper line

Arc Interpolation: Tool movement along an arc

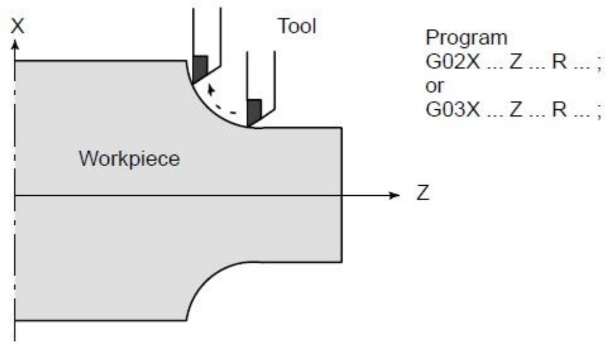


Fig2.14 Tool movement along an arc

Thread Interpolation(Thread Cutting): Threads can be cut by moving the tool in synchronization with spindle rotation. In a program, specify the thread cutting function by G32.

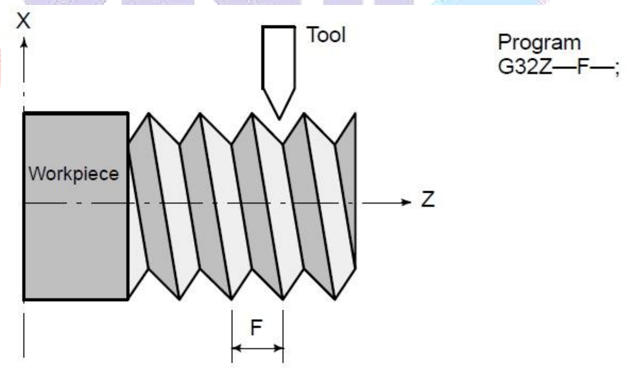


Fig2.15 Straight thread cutting

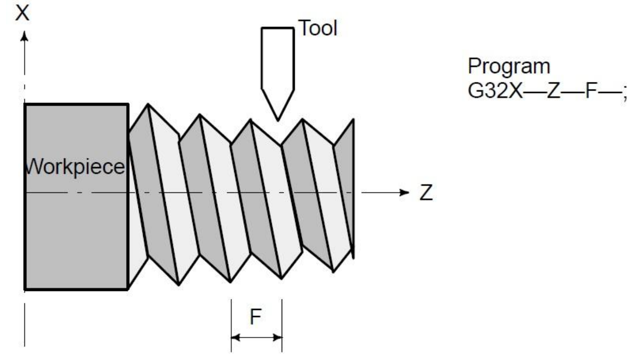


Fig2.16 Taper thread cutting

Feed: Movement of the tool at a specified speed for cutting a workpiece is called the feed.

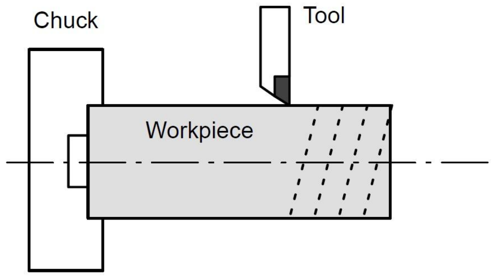


Fig2.17 Feed Function

Feed Rates can be specified by using actual numeric. Eg.: **F2.0** , which can be used to feed the tool 2 mm while the workpiece makes one turn.

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

2.1.6 Absolute Programming & Incremental Programming

Methods of command for moving the tool can be indicated by absolute or incremental designation

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.

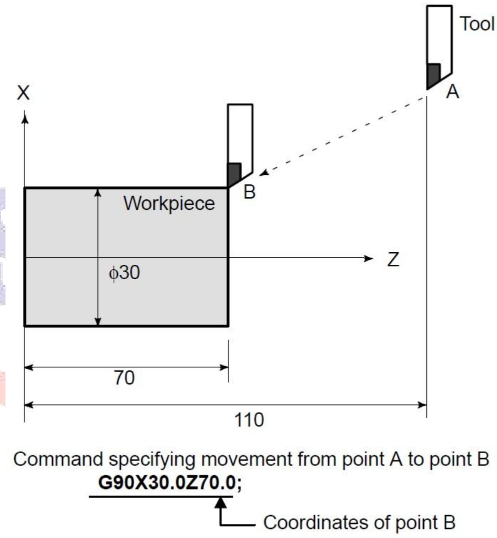


Fig2.18 Absolute Command

Incremental command: Specify the distance from previous tool position to the next tool position.

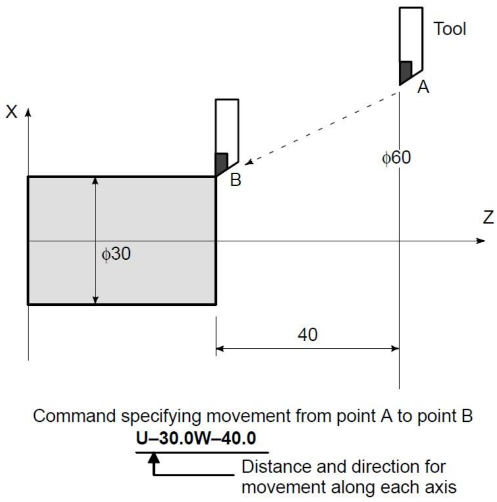


Fig2.19 Incremental command

2.1.7 Diameter Programming & Radius Programming

Since the work cross section is usually circular in CNC lathe control programming, its dimensions can be specified in two ways :

Diameter and Radius

When the diameter is specified, it is called diameter programming and when the radius is specified, it is called radius programming.

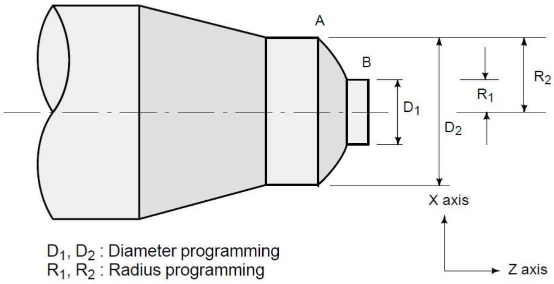
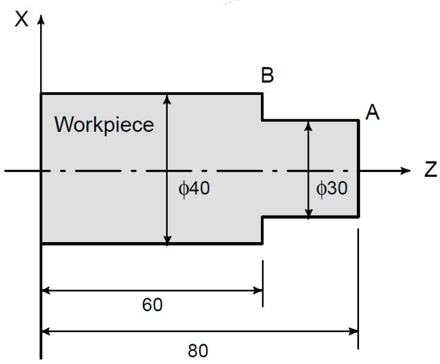


Fig 2.20 Diameter Programming & Radius Programming

Dimensions of the X axis can be set in diameter or in radius. Diameter programming or radius programming is employed independently in each machine. P16 in User parameter is set for diameter command&radius command.

1. Diameter Programming

In diameter programming, specify the diameter value indicated on the drawing as the value of the X axis.

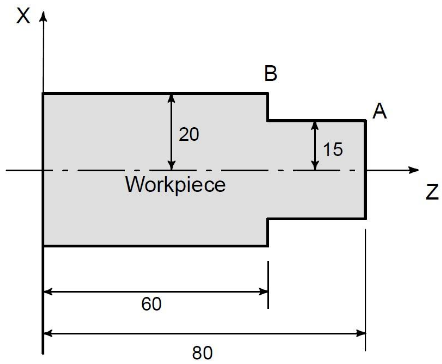


Coordinate values of points A and B
A(30.0, 80.0) , B(40.0, 60.0)

Fig2.21 Diameter Programming

2. Radius Programming

In radius programming, specify the distance from the center of the workpiece, i.e. the radius value as the value of the X axis.



Coordinate values of points A and B
A(15.0, 80.0) , B(20.0, 60.0)

Fig2.22 Radius Programming

When using diameter programming, note the conditions listed in the table in the following,

Item	Notes
X axis command	Specified with a diameter value
Incremental command	Specified with a diameter value In the above figure, specifies D2 minus D1 for tool path B to A of Fig2.14.
Coordinate system setting (G50)	Specifies a coordinate value with a diameter value
Component of tool offset value	P16 in User Parameter determines either diameter or radius value programming for X-axis
Parameters in canned cycle, such as cutting depth along X axis. (R)	Specifies a radius value
Radius designation in circular interpolation (R, I, K, and etc.)	Specifies a radius value
Feedrate along axis	Specifies change of radius/rev. or change of radius/min.
Display of axis position	Displayed as diameter value

2.1.8 Cutting Speed & Spindle Speed

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

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

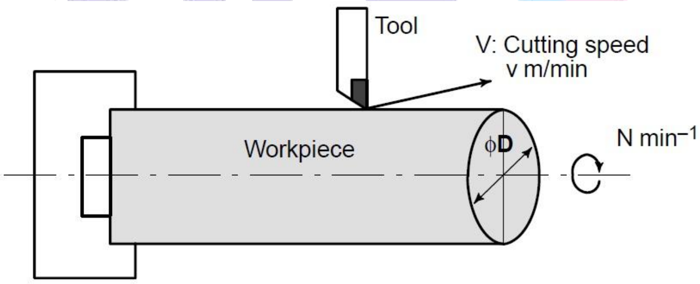


Fig2.23 Cutting Speed

Eg.: <When a workpiece 200 mm in diameter should be machined at a cutting speed of 300 m/min. >

The spindle speed is approximately 478 min^{-1}, which is obtained from $N=1000v/\pi D$. Hence the following command is required:

S478 ;

Commands related to the spindle speed are called the spindle speed function.

The cutting speed v (m/min) can also be specified directly by the speed value. Even when the workpiece diameter is changed, the CNC changes the spindle speed so that the cutting speed remains constant.

This function is called the constant surface speed control function.

2.1.9 Tool Function

Selection of tool used for various machining.

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.

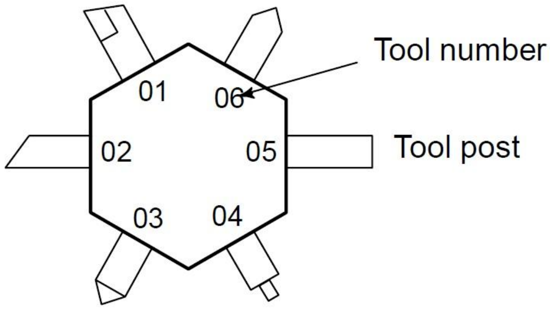


Fig2.24 Tool used for various machining

Example: <When No.01 is assigned to a roughing tool>

When the tool is stored at location 01 of the tool post, the tool can be selected by specifying **T0101**.This is called the tool function

2.1.10 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

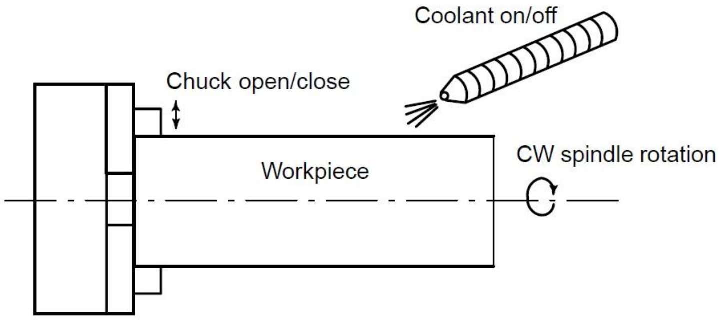


Fig2.25 Command for machine Operations

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.

For example, when M03 is specified, spindle is rotated with CW direction at specified 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 spindle motor is turned on and off.

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

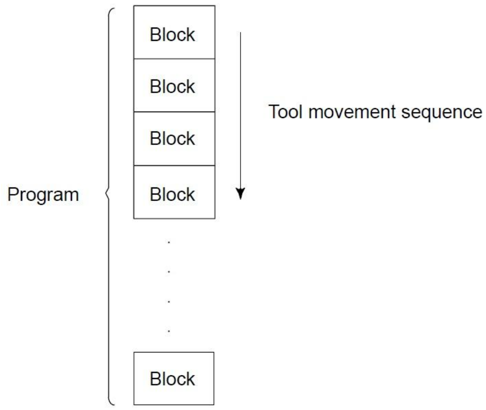


Fig2.26 Configuration of Program

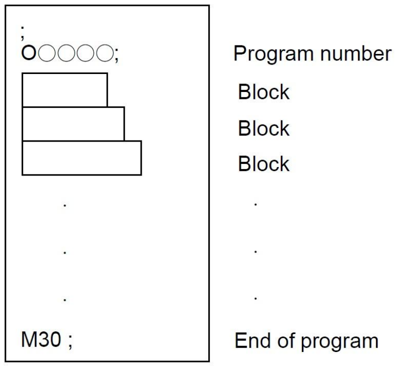


Fig2.27 Program Configuration

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:

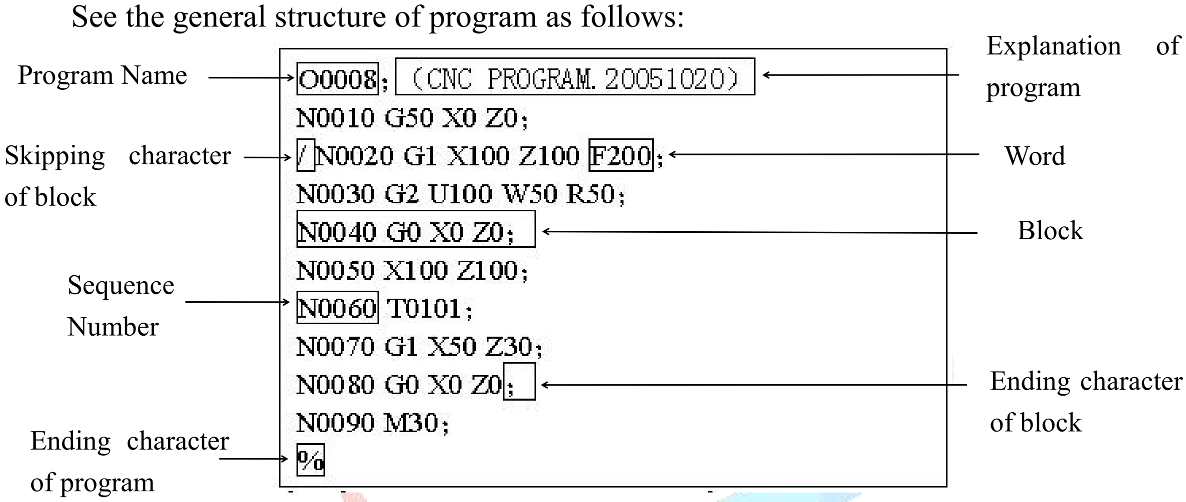


Fig2.28 General Structure of Program

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.

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-CNC990TDb system.

Table 2-1 Word List

Address	Data range	Functions
N	00000~9999	Block number
G	00~99	Preparatory function
	100~150	User-defined G macro function
M	00~99	Auxiliary function output
T	01-99	Tool function
S	0-99999(rpm)	Specify Speed of 1st Spindle
SS	0-99999(rpm)	Specify Speed of 2nd Spindle
	0.01-15000mm/min	Feedrate per minute
F	0.001-500mm/r	Feedrate per rev

	0.1~1000mm	Thread Lead in Metric
X	±99999.999mm	Coordinates in X direction
	0～9999.999(s)	Dwell time
U	±99999.999mm	Increment in X direction
	Finish allowance in X direction in G71,G72,G73	
	Cutting depth in G71	
	Moving distance of tool retraction in X direction in G73	
Z	±99999.999mm	Coordinates in Z direction
W	±99999.999mm	Increment in Z direction
	Finish allowance in Z direction in G71,G72,G73	
	Cutting depth in G72	
	Moving distance of tool retraction in Z direction in G73	
I	00-99 teeth/inch	Thread Lead in Inch
	±99999.999mm	Vector of arc center relative to starting point I in X direction
K	±99999.999mm	Vector of arc center relative to starting point in Z direction
R	0.001-99999.999mm	Arc radius
	Moving distance of cycle tool retraction in G71,G72	
	Cycle times of roughing in G73	
	Moving distance of tool retraction after Cutting in G74, G75	
	Moving distance of tool retraction after cutting to the end point in G74, G75	
	Finishing allowance in G76	
	Taper in G90, G92, G94, G96	
P	0.001-65s	Dwell time
	0000-99999	Calling subprogram number
	Circular moving distance in X direction in G74, G75	
	Thread cutting parameter in G76	
	Initial block number of finishing in the compound cycle instruction	
L	1~9999	Cycle times
	1~9999	Cycle times of calling subprogram
	1-99	Heads of multi-head thread
T	0000-9999	Tool function
M	00-99	Auxiliary function output, subprogram call, etc.
	880-889	User-defined M Macro Function
/	Program skip	

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.

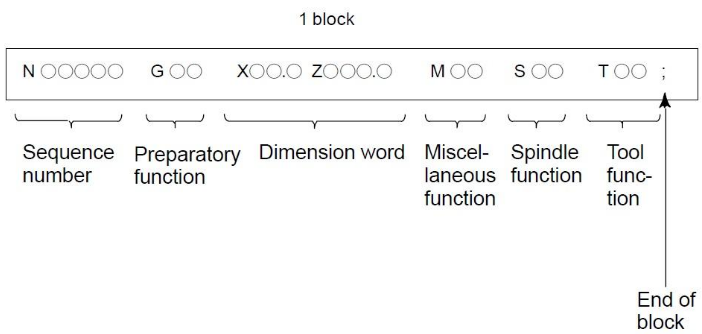


Fig2.29 Block Configuration

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).

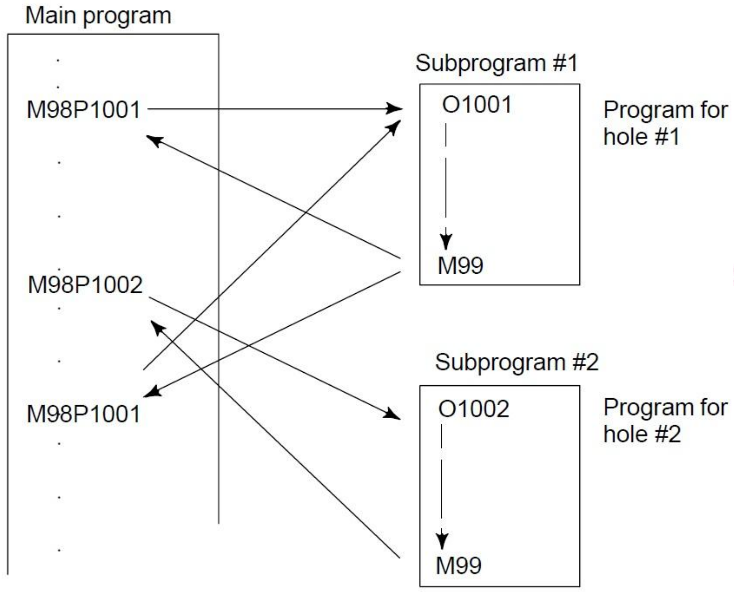
The contents of the dimension word depend on the preparatory function.

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.

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

Press “Program” key, Select one Processing Program

⇩

Execute: Press “C” key : “Execute”,system will jump to main screen.

⇩

Switch to Auto Mode:Press “AUTO” Key

⇩

Run Program: Press “Start” Button

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

Chapter 3 G INSTRCUTIONS

3.1 INTRODUCTION

G instruction consists of instruction address G and its following 1～2 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.

G □□

→ Instruction value(00~99,the leading zero can be omitted)

→ Instruction address G

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_;  }  G01 is effective in this range
   X_;  }
G00Z_;
```

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 3 G Code List

Word	Ground	Functions	Page
G00	01	Positioning(Rapid Traverse)	
G01		Linear Interpolation(Cutting feed)	
G02		Circular Interpolation CW	
G03		Circular Interpolation CCW	
G32	01	Thread Cutting	
G332		Thread Interpolation with Circular CW	
G333		Thread Interpolation with Circular CCW	
G70	01	Finishing Cycle	
G71		Axial Roughing Turning Cycle	
G72		Radial Roughing Facing Cycle	
G73		Pattern Repeating Cycle	
G74		End Face Peck Drilling	
G75		Outer diameter/Internal diameter Grooving Cycle	
G76		Multi Threading Cycle	
G90	01	Outer diameter/internal diameter cutting cycle	
G92		Thread Cutting Cycle	
G93		Canned Tapping Cycle	
G94		Endface turning cycle	
G31	01	Skip function (No alarm)	
G311		Skip function (alarm)	
G22	01	Program Block Cycle	
G800		Program Block Cycle Cancel	
G26	01	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	
G61	01	Return G25 coordinate of G25	
G611		Return the coordinate position of X-Axis in G25	
G612		Return the coordinate position of Y(C)-Axis in G25	
G613		Return the coordinate position of Z-Axis in G25	
G614		Return the coordinate position of A-Axis in G25	
G25	01	Save value of current coordinate	
G28	01	Return to reference position	
G281		X-Axis return to reference position	
G282		Y(C)-Axis Return to reference position	
G283		Z-Axis Return to reference position	
G284		A-Axis Return to reference position	
G50	01	Coordinate system setting or max. spindle speed setting	
G52		Local coordinate system setting	
G53		Machine Coordinate System	
G54	06	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	
G184	01	Setup/offset coordinate of current tool	
G185		Setup/offset coordinate of all tools	
G96	02	Constant surface speed control	
G97		Constant surface speed control cancel	
G98	02	Feeding per minute	
G99		Feeding per revolution	
G15	02	Polar coordinate interpolation cancel mode	
G16		Polar coordinate interpolation mode	
G990	02	Absolute programming	
G991		Incremental programming	
G20	02	Input in Inch	
G21		Input in mm	
G65	00	Macro calling	
G66	12	Macro modal call	
G67		Macro modal call cancel	
G04	03	Dwell	
G60	04	Exact Stop & Positioning	
G64		Continuous Path Processing	
G40	05	Tool nose radius compensation cancel	
G41		Tool nose radius compensation left	
G42		Tool nose radius compensation right	
G17	16	XpYp plane selection	
G18		ZpXp plane selection	
G19		YpZp plane selection	

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.

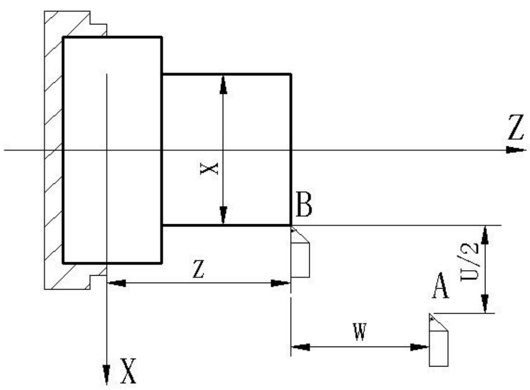


Fig3.1 Coordinate Code of G00

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.

Non-linear 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).

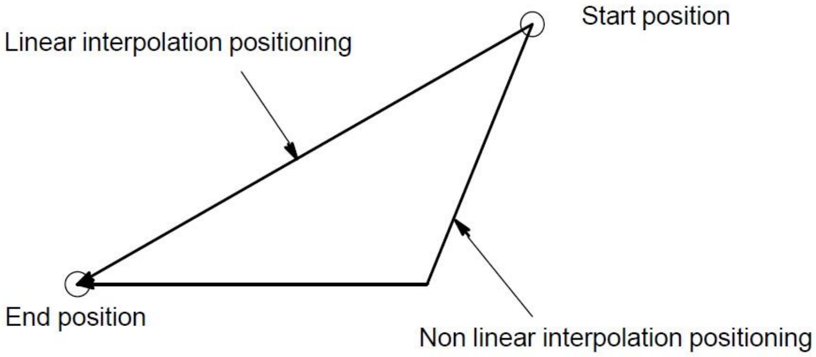


Fig3.2 Mode of Tool Path

P1 & P2 in Speed parameter is set for rapid traverse rate in the G00 command for X & Z 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.

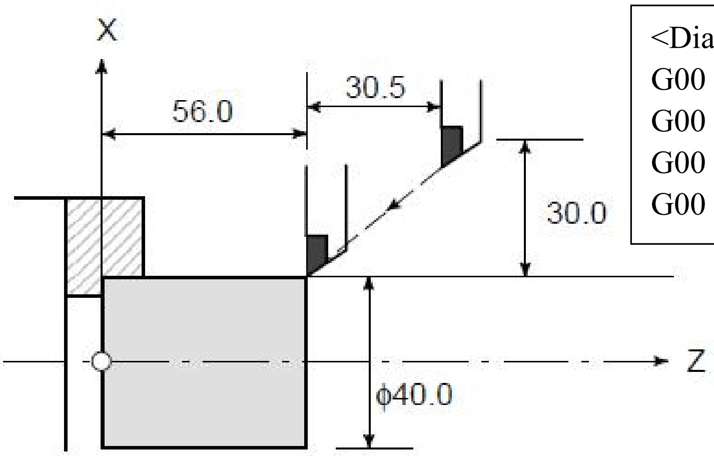
In the positioning mode actuated by G00, the tool is accelerated to a predetermined speed at the start of a block and is decelerated at the end of a block.

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.

Example:



<Diameter Programming>
G00 X40.0 Z56.0 ; (Absolute command)
G00 U-60.0 W-30.5 ; (Incremental command)
G00 U-60.0 Z56.0 ; (Compound programming)
G00 X40.0 W-30.5 ; (Compound programming)

Fig3.3 Positioning of G00

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(C)/V_ A_ F_ ;

X,Z,Y(C), 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 :

$$\mathbf{G01}\,\alpha\underline{\alpha}\,\beta\underline{\beta}\quad F\underline{f};$$

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

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

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

Example: <Diameter Programming>

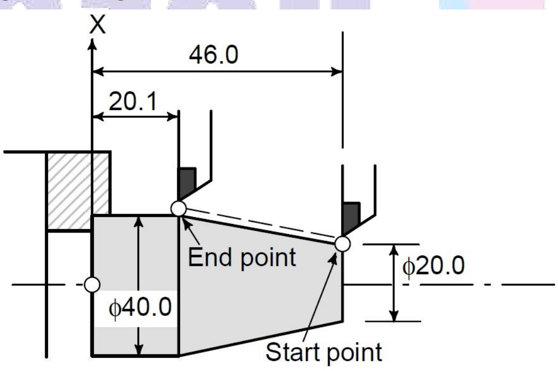


Fig3.4 Linear Interpolation of G01

G01 X40.0 Z20.1 F20 ; (Absolute command)

G01 U20.0 W-25.9 F20 ; (Incremental command)

G01 instruction can also specify movement of either X-axis or 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 with G1.

3.5 Circular Interpolation (G02/G03)

These commands will move a tool along a circular arc.

Format: Arc in the ZpXp plane (Default)

$$\mathbf{G18}\begin{Bmatrix}\mathbf{G02}\\ \mathbf{G03}\end{Bmatrix}\mathbf{Xp\_Zp\_}\begin{Bmatrix}\mathbf{I\_K\_}\\ \mathbf{R\_}\end{Bmatrix}\mathbf{F\_}$$

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 without sign (always with radius value)
F	Feedrate along the arc

Note: G18 is default set , which can be omitted.

Direction of circular interpolation: When turret in different position of lathe machine, Front tool-post system & Rear tool-post system , the direction of G02&G03 is opposite in these two system.

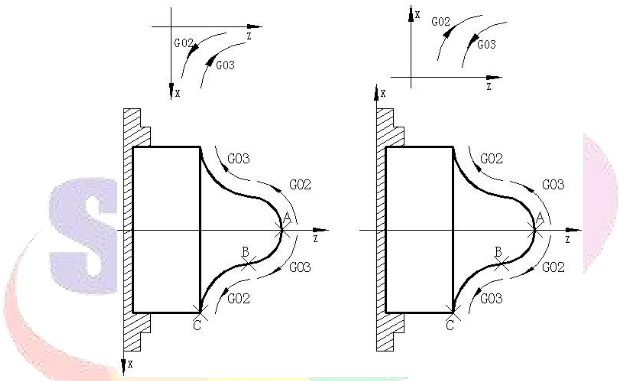


Fig3.5.1 Direction of G02&G03 at different tool-post system

"Clockwise"(G02) and "counterclockwise"(G03) on the ZpXp plane are defined when the XpYp plane is viewed in the positive－to－negative direction of the Yp axis in the Cartesian coordinate system.

Distance moved on an arc: The end point of an arc is specified by address X(U) , Z(W) or Y(V), and is expressed as an absolute or incremental value according to G990 or G991. 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 and K for the Xp and Zp axes, respectively. The numerical value following I 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 G990 and G991, as shown below.

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

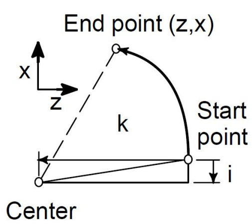


Fig3.5.2 Define & Direction of I & K

I0 and K0 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 Xp and Zp are omitted (the end point is the same as the start point) and the center is specified with I 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 and K. In this case, one arc is less than 180°, and the other is more than 180° are considered. If Xp and Zp 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.)

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

2.If I, 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 than180° when R is negative, and it is less than or equal to 180° when R is positive.

Example 1 : Command of circular interpolation X,Z

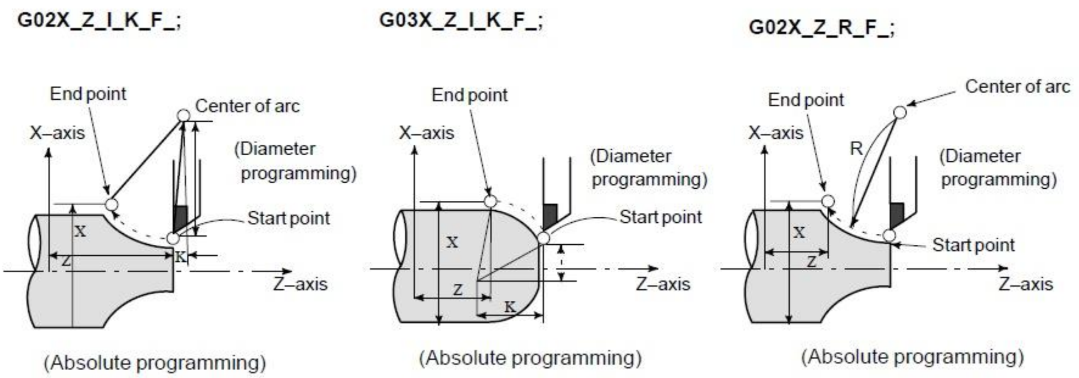
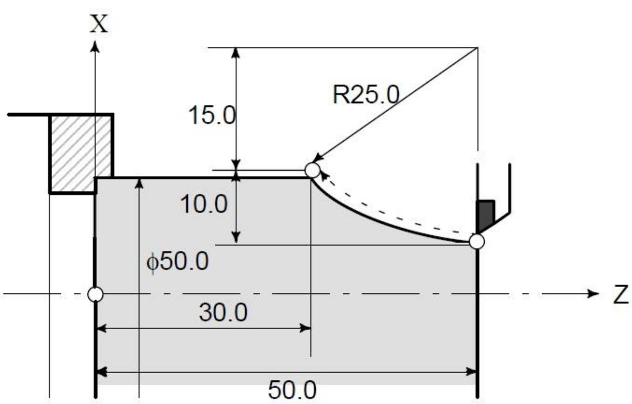


Fig3.5.3 Command of circular interpolation

Example 2: Processing same path with different command.



<Diameter Programming>

- G02 X50.0 Z30.0 I25.0 F0.3
- G02 U20.0 W-20.0 I25.0 F0.3
- G02 X50.0 Z30.0 R25.0 F0.3
- G02 U20.0 W-20.0 R25.0 F0.3

Fig3.5.4 Processing same path with different command

Example 3: Application of Circular Interpolation

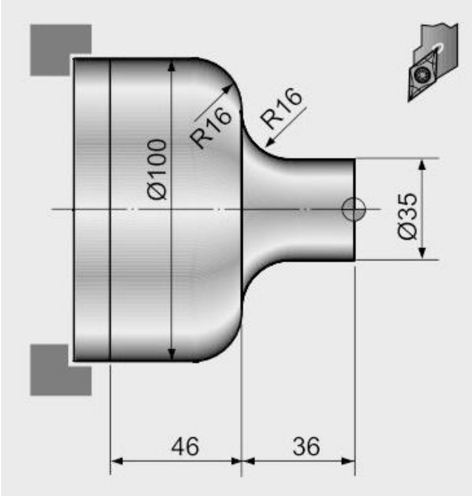


Fig3.5.5 Example 3

```
<Program>
N20 G50 S2000 T0300
G96 S200 M03
G42 G00 X35.0 Z5.0 T0303 M08
G01 Z-20.0 F0.2
G02 X67.0 Z-36.0 R16.0
G01 X68.0
G03 X100.0 Z-52.0 R16.0
G01 Z-82.0
G40 G00 X200.0 Z200.0 M09 T03.00
M30
```

SZGH-CNC990TD CNC system support many kinds of thread cutting, which includes straight thread,tapered thread,scroll thread,thread cutting with variable lead, Continuous thread cutting, multiple-thread cutting, metric/inch single.Length and angle of thread run-out can be changed,multiple-thread is machined by single side to protect tool and improve smooth finish of its surface.

When machine tool needs to do thread cutting, spindle must fix encoder.When transmission of spindle and encoder is not 1:1, we need to set transmission ratio by set P412&P413 in Axis parameter.

P10 in Axis parameter is set for pulses of per rev of spindle(Resolution*Poles).

P412 in Axis parameter is set for teeth of spindle motor.

P413 in Axis parameter is set for teeth of spindle encoder.

X or Z axis traverses to start machine after the system receives spindle signal per rev in thread cutting, and so one thread is machined by multiple roughing, finishing without changing spindle speed.

There is a big error in the thread pitch because there are the acceleration and the deceleration at the starting and ending of thread cutting in X, Z direction, and so there is length of thread lead-in and distance of tool retraction at the actual starting and ending of thread cutting.

The traverse speed of tool in X, Z direction is defined by spindle speed instead of cutting feedrate override in thread cutting when the pitch is defined. The spindle override control is valid in thread cutting. When the spindle speed is changed, there is error in pitch caused by acceleration/deceleration in X, Z direction, and so the spindle speed cannot be changed.

Note:

1. When spindle encoder can't feedback real position of spindle chuck(also not 1:1), teeth of spindle motor is more than teeth of spindle encoder , it must match with key-sets of our company ;

2. In the process of thread cutting, Feeding speed and override is invalid.

3. In the process of thread cutting, spindle will not stop whatever you operate, if the user operate suspend,the system will suspend after processed this segment.

4. The spindle speed must remain constant from rough cutting through finish cutting. If not, incorrect thread lead will occur.

3.6 Thread Cutting (G32)

Tapered screws and scroll threads in addition to equal lead straight threads can be cut by using a G32 command.

The spindle speed is read from the position coder on the spindle in real time and converted to a cutting feedrate for feed－per minute mode, which is used to move the tool.

3.6.1 Constant Lead Threading

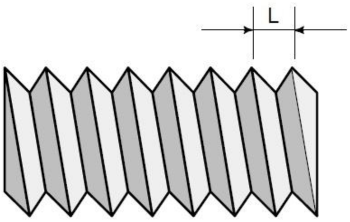


Fig3.6.1 Straight thread

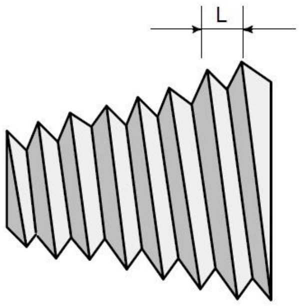


Fig3.6.2 Tapered Screw

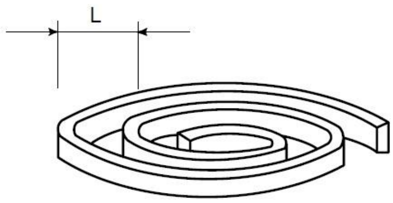


Fig3.6.3 Scroll Thread

Straight thread: only input the direction and length of Z-axis;

Tapper thread: must input the direction and length of X-axis and Z-axis;

Scroll thread(head face thread): only input the direction and length of X-axis;

Format: G32 Z(W)_X(U)_ F(I)_ SP(Q)_

G32 is the spiral interpolation machining instruction.it is modal.

X(U)_ , Z(W)_ is end point in absolutely/correspond coordinate system.

F_: metric lead(pitch) of long axis, is moving distance of long axis when the spindle rotates one rev: 0.1-1000mm; Max Lead=Lines (Resolution) of Spindle encoder /10 mm. After F is executed, it is valid until F with specified pitch is executed again.

I_: teeth per inch,is ones per inch(25.4 mm) in long axis, and also is circles of spindle rotation when the long axis traverses one inch(25.4 mm):0.1~99 tooth/inch. After I is executed, it is valid until I with specified pitch is executed again.

SP_: Initial angle(offset angle)between spindle rotation one rev and starting point of thread cutting: 0~360000(unit: 0.001 degree). SP is non-modal parameter, must be defined every time, otherwise it is 0°.

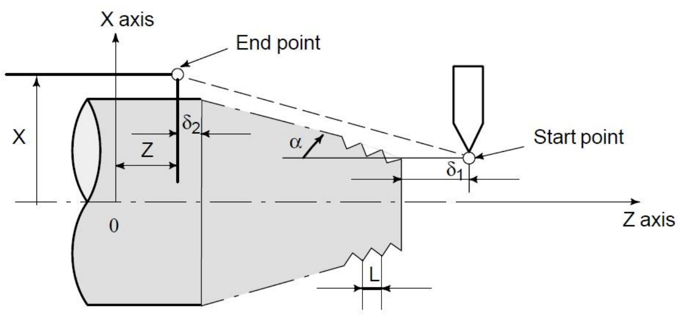
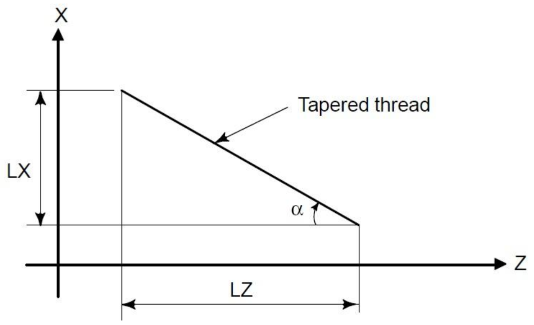


Fig3.6.4 Example of Thread Cutting

In general, thread cutting is repeated along the same tool path in rough cutting through finish cutting for a screw. Since thread cutting starts when the position coder mounted on the spindle outputs a 1–turn signal, threading is started at a fixed point and the tool path on the workpiece is unchanged for repeated thread cutting.



α ≥ 45º , lead is LZ , Z-axis is long axis
α ≤ 45º , lead is LX , X-axis is long axis

Fig3.6.5 LZ & LX of a tapered thread

In general, the lag of the servo system, etc. will produce somewhat incorrect leads at the starting and ending points of a thread cut. To compensate for this, a threading length somewhat longer than required should be specified.

Example1: Straight Thread Cutting

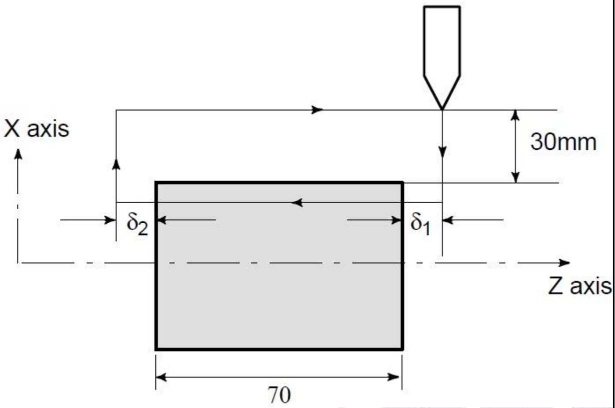


Fig3.6.6 Example of Straight Thread Cutting

Requirement: (Metric input, Diameter programming)
Thread Lead:4mm , δ_1=3mm ;δ_2=3mm ; Depth of cut:1mm(cut twice)

```
G00 U-62.0 ;
G32 W-74.5 F4.0 ;
G00 U62.0 ;
W74.5 ;
U-64.0 ;
(For the second cut,cut 1mm)
G32 W-74.5 ;
G00 U64.0 ;
W74.5 ;
```

Example2: Tapered Thread Cutting

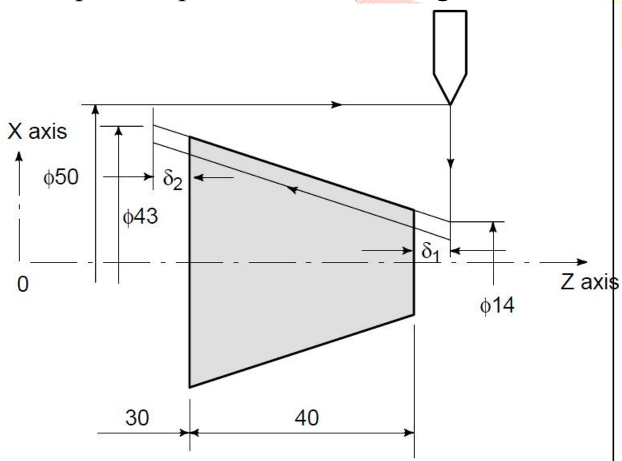


Fig3.6.7 Example of Tapered Thread Cutting

Requirement: (Metric input, Diameter programming)
Thread Lead:3.5mm in the direction of Z-axis, δ_1=2mm ;δ_2=1mm; Depth Cut(X):1mm (cut twice)

```
G00 X12.0 Z72.0 ;
G32 X-41.0 Z29.0 F3.5 ;
G00 X50.0 ;
Z72.0 ;
X10.0 ;
(For the second cut,cut 1mm)
G32 X39.0 Z29.0 ;
G00 X50.0 ;
Z72.0 ;
```

3.6.2 Continuous Thread Cutting

This function for continuous thread cutting is such that fractional pulses output to a joint between move blocks are overlapped with the next move for pulse processing and output (block overlap) .

Therefore, discontinuous machining sections caused by the interruption of move during continuously block machining are eliminated, thus making it possible to continuously direct the block for thread cutting instructions.

Since the system is controlled in such a manner that the synchronous with the spindle does not deviate in the joint between blocks wherever possible, it is possible to performed special thread cutting operation in which the lead and shape change midway.

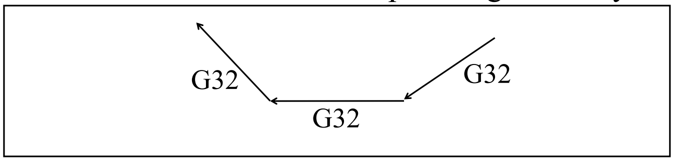


Fig3.6.8 Continuous Thread Cutting

Even when the same section is repeated for thread cutting while changing the depth of cut, this system allows a correct machining without impairing the threads.

3.6.3 Thread Cutting With Variable Lead

Variable-lead thread cutting is finished by continuous input G32 command, input a thread length of each program, lead of thread(F) is different. At the second cycle , CNC system will not detect the encoder synchronization signal.

The start angle (Q) increment is 0.001 degrees. Note that no decimal point can be specified. Example: For a shift angle of 180 degrees, specify SP180000.

SP180.000 cannot be specified, because it contains a decimal point.

3.7 Circular Thread Cutting(G332/G333)

Format: G332 Z(W)_ X(U)_ R_ F(I)_ SP_;

G332: Clockwise circular thread cutting

Format: G333 Z(W)_ X(U)_ R_ F(I)_ SP_

G333: Counter-Clockwise circular thread cutting

Z(W)_X(U)_: end point of thread cutting

R: radius of circular(negative number means degree over 180º)

F(I)_: lead(pitch) of thread ;

SP_ : start angle The start angle is not a continuous–state (modal) value. It must be specified each time it is used. If a value is not specified, 0 is assumed.

Using method refer to G02, G03, G32 instructions.

3.8 Canned Cycle(G90,G92,G93,G94)

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

There are three canned cycles : Outer diameter/internal diameter cutting canned cycle (G90), Thread cutting canned cycle (G92), Canned tapping cycle (G93) , and End face turning canned cycle (G94).

Note: 1. Explanatory diagrams in this chapter uses diameter programming in X axis.

2. In radius programming, changes U/2 with U and X/2 with X.

3.8.1 Outer Diameter/Internal Diameter Cutting Cycle (G90)

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

a) Straight Cutting Cycle : Format: G90 X(U)_ Z(W)_ F_ ;

X: absolute coordinates of cutting end point in X direction

U: different value of absolute coordinates between end point and starting point of cutting in X direction

Z: different value of absolute coordinates between end point and starting point of cutting in Z direction

W: different value of absolute coordinates between end point and starting point of cutting in Z direction

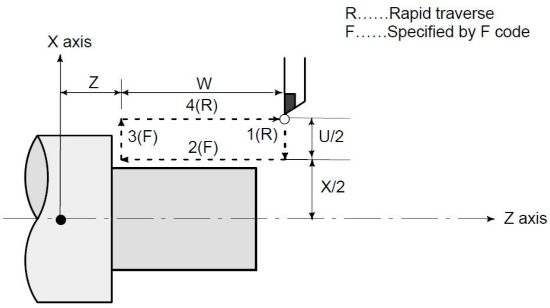


Fig3.8.1 Straight Cutting Cycle

In incremental programming, the sign of the numbers following address U and W depends on the direction of paths 1 and 2. In the Fig3.8.1, the signs of U and W are negative.

In single block mode, operations 1, 2, 3 and 4 are performed by pressing the cycle start button once.

b) Taper Cutting Cycle: Format: G90 X(U)_ Z(W)_ R_ F_ ;

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

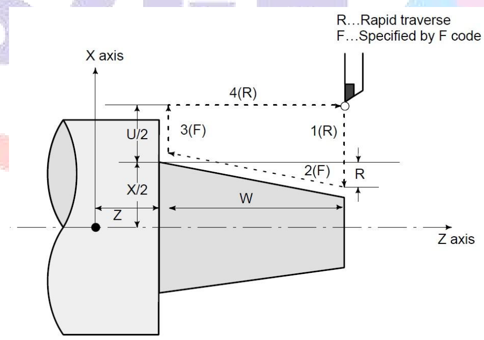
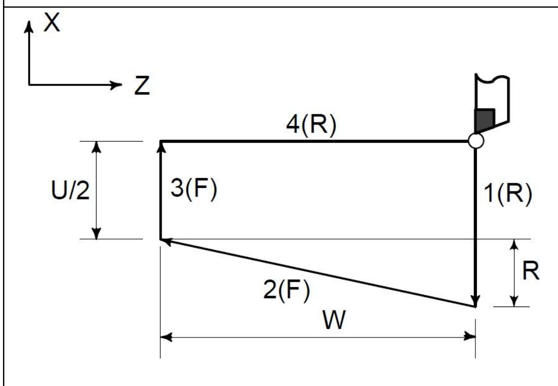
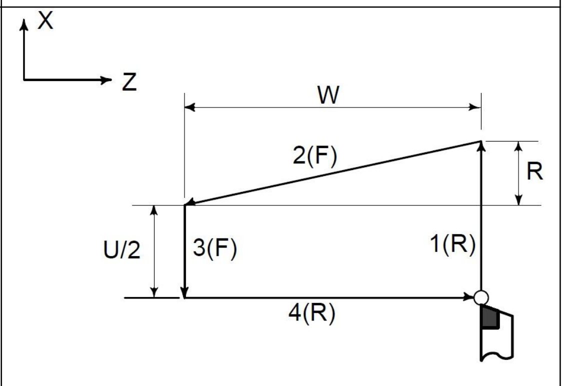
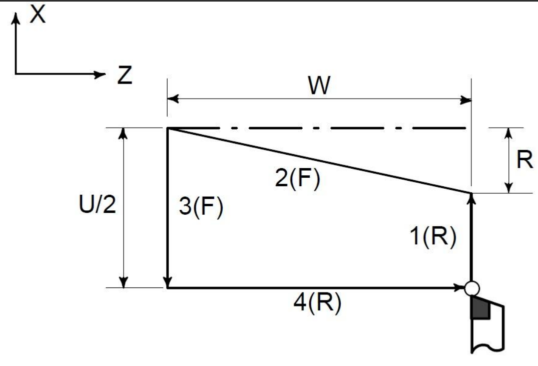
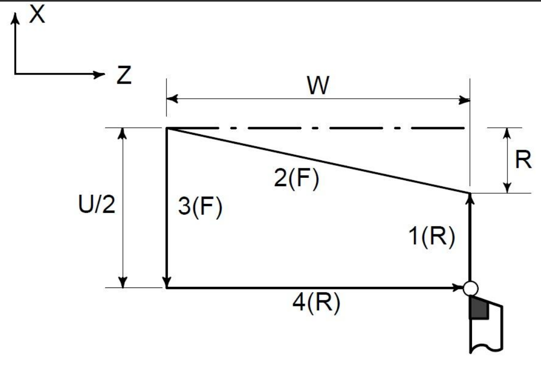


Fig3.8.2 Taper Cutting Cycle

c) Signs of numbers specified in the taper cutting cycle

In incremental programming, the relationship between the signs of the numbers following address U, W, and R, and the tool paths are as follows:

1. U<0 , W<0 , R<0	**2. U>0,W<0, R>0**
	

3. U<0, W<0, R>0 **at |R| ≤ |U/2|**	**4. U>0, W<0, R<0** **at |R| ≤ |U/2|**
	

d) Cycle Process & examples

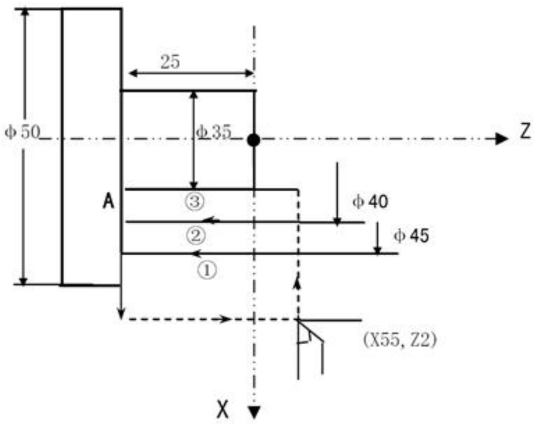
① The tool rapidly traverses from starting point to cutting starting point in X direction;

② Cutting feed(linear interpolation) from the cutting starting point to cutting end point;

③ Retract the tool at feedrate in X direction (opposite direction to the above-mentioned ①), and return to the position which the absolute coordinates and the starting point are the same;

④ The tool rapidly traverses to return to the starting point and the cycle is completed.

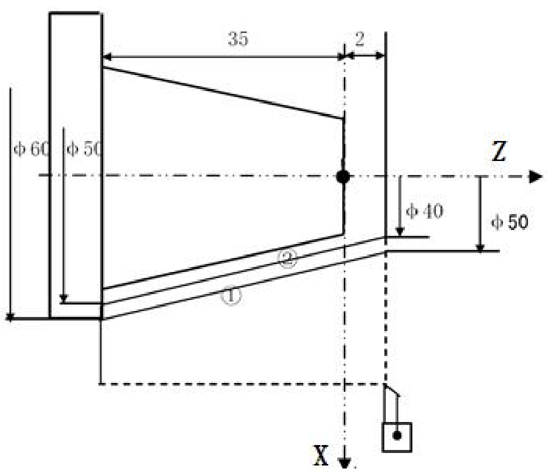
Example1: Use G90 to Process Cylinder Surface




```
N10 T0101 ;
N20 G00 X55 Z4 M03 ;
N30 G01 Z2 F100 M08 ;
N40 G90 X45 Z-25 ;
N50 X40 ;
N60 X35 ;
N70 G00 X100 Z100 ;
N80 T0100 M09 ;
N90 M05 ;
N100 M30 ;
```

Fig3.8.3 Usage of G90

Example: Use G90 to Process Taper Surface




```
N10 M03 S1000 ;
N20 T0101 ;
N30 G00 X65 Z5 ;
N50 G96 S120 ;
N60 G99 G01 Z2 F1 M08 ;
N70 G90 X60 Z-35 R-5 F0.2 ;
N80 X50 ;
N90 G00 G98 X100 Z100 M09 ;
N100 G97 S1000 T0100 ;
N110 M05 ;
N120 M30 ;
```

Fig3.8.4 Usage of G90

3.8.2 Thread Cutting Cycle (G92)

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

(1) Straight Thread Cutting: G92 X(U)_ Z(W)_ F/I_ ;

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

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

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

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

F=0.001~1000 mm, metric thread pitch. After F value is executed, it is reserved and can be omitted; I=0.1~99 toots/inch, metric thread teeth per inch.

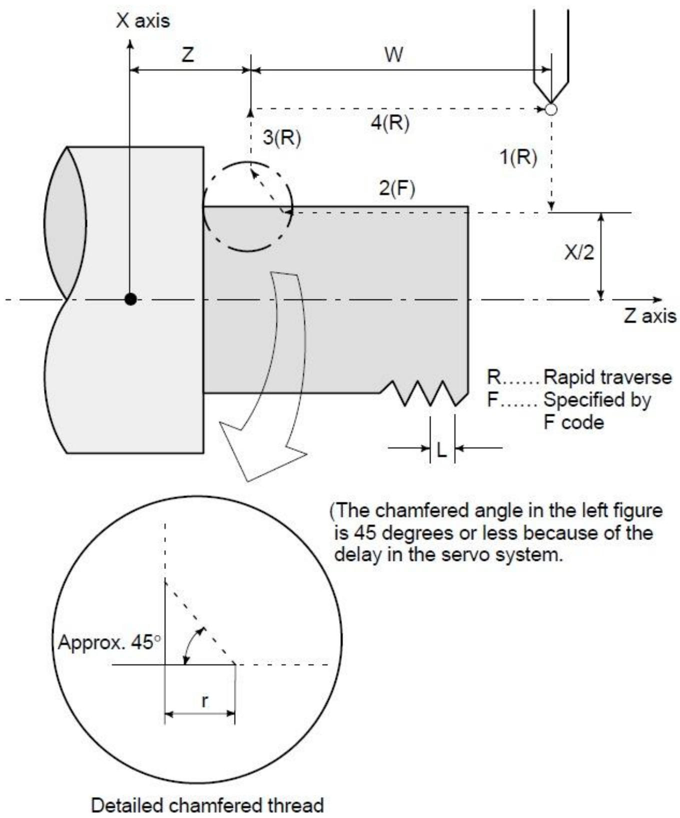


Fig3.8.5 Straight Thread Cutting

In incremental programming, the sign of numbers following addresses U and W depends on the direction of paths 1 and 2. That is , if the direction of path 1 is the negative along the X axis, the value of U is negative. The range of thread leads, limitation of spindle speed, etc. are the same as in G32 (thread cutting).

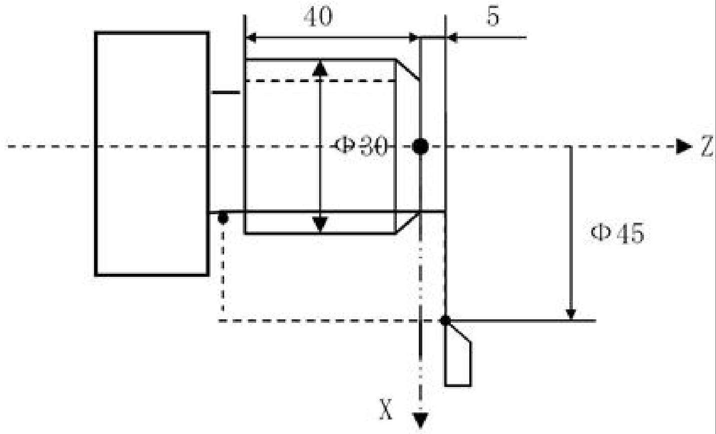


Fig3.8.6 Example of Straight Thread Cutting

```
Program:
N10 M03 S1000;
N20 T0101;
N30 G00 X45 Z5;
N40 G92 X29.2 Z-45 F1.5;
N50 X28.6;
N60 X28.2;
N70 X28.04;
N80 G00 X100 Z50;
N90 T0100 M05;
N100 M30:
```

(2)Taper Thread Cutting : G92 X(U)_ Z(W)_ R_ F/I_ ;

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

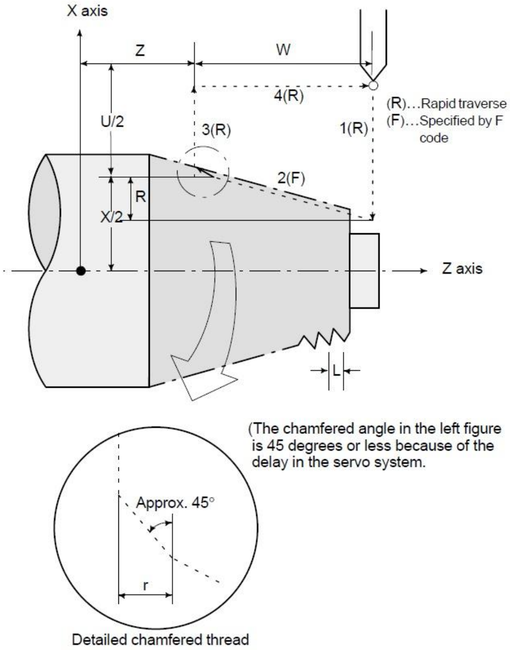


Fig3.8.7 Taper Thread Cutting

Example: Process of taper screw of inner hole, pitch is 11 tooth/inch,(coning is 1:32)

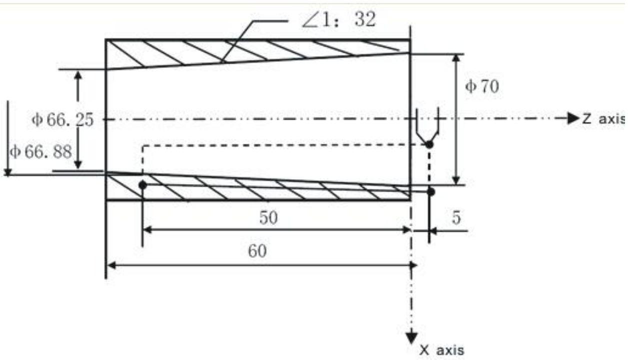


Fig3.8.8 Example of Taper Thread Cutting

```
Program:
N10 M03 S1000;
N20 T0101;
N30 G00 X55 Z10;
N40 G01 X60 Z5 F100;
N50 G90 X66.25 Z-60 R1.875;
N60 G92 X66.88 Z-50 R1.4 I11;
N70 X66.9 I11;
N80 X67 I11;
N90 X67.4 I11;
N100 X67.6 I11;
N110 X67.8 I11;
N120 G00 X100 Z50;
N130 T0100 M05;
N140 M30;
```

Note:

1.When processing inch thread, pitch I is non-mode,just be effective in one sentence, so every segments should plus I in thread cycle.

2.Speed of processing thread ,which should be less than 3000mm/min, is Pitch (F) multi Speed of spindle (S).

3.The retract speed of X axis ,which should be less than 5000mm/min, is F*S*P24 (Speed parameter)*0.1 . Eg.:When processing F2, S1200,this value of P24 parameter should be less than 20.

(3) Deceleration or acceleration control in thread cutting cycle:

At the end of thread, because of the index of deceleration control,cause the distance of pitch is in-homogeneous,the higher speed of spindle the longer of in-homogeneous pitch. To reduce the error,should reduce the index of deceleration or acceleration time, but it will cause the motor stuck if match the step motor. In order to solve this problem:

- could choose Z axis according to linear acceleration or deceleration speed constant;
- could choose the X axis with the rapid speed G00 to back tail.

The relevant parameter is as follows (see the chapter of parameter):

In Speed parameter

P22: the acceleration or deceleration constant of Z axis in thread processing

P23: the acceleration or deceleration constant of Y axis in thread processing

P24: The backing tail speed rate of servo motor in thread cycle

P25: The starting speed of servo motor in thread cycle

P26: The maximum backing tail speed of servo motor in thread cycle

(4) Multi Thread Cutting : G92 X_ Z_ F_ L_ [or SP];

L_: Multi threads: 1~100 and it is modal parameter. (the system defaults it is single thread when L is omitted); repeat times of G92 is L.

SP_: Initial angle(offset angle)between spindle rotation one rev and starting point of thread cutting: 0~360000(unit: 0.001 degree). SP is non-modal parameter, must be defined every time, otherwise it is 0°.

Note: Cannot use SP to specify when processing multi threads.

Such as: L03, 3 threads, continuous executing 3 times G92. First time,processing at once when spindle rotate one rev; Second time,after offset of 120 degrees ,begin cutting thread; Third time, after 240º offset, begin cutting thread,

Example:

G92 X50.Z-100 F5 L5 ; at X50, process 5 threads.

X48.5 ; at X48.5, processing 5 threads.

X45 ; at X45, processing 5 threads.

G00 X100 Z100 ;

......

(5) Back Tail of Thread : G92 X_ Z_ F/I_ P_;

P: volume of backing tail: the default value of P could be set by P20 in User parameter (Default when powering on).

Set unit: P1 means 0.1 pitch ; P10 means 1 pitch .

Scope: 1--225, when the set value beyond to the range is invalid.

(6) Back tail at any angle

When cutting thread without backing fuller, the system must have the function of automatic backing tail in thread processing to produce a qualified section of thread. Including the program format of backing tail in thread:

G92 X_ Z_ F_ J_ K_P_;

● J, K set the ratio of back tail X, Z. When J2 K1, X is twice faster than Z.

● P: back tail volume. Setting: 0.1 pitch. Set range: 1~255 (beyond to this range is invalid). The default value can be set by No.20 parameter in process parameter (Default when powering on).

● J, K, P are mode value.

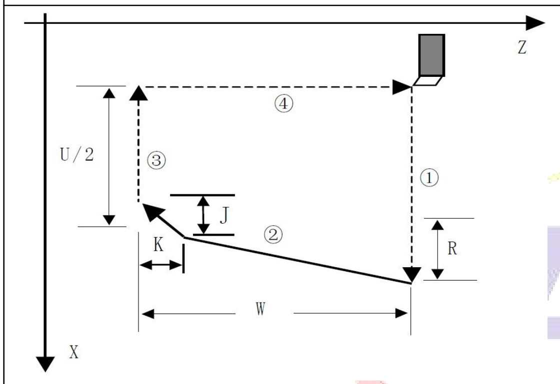
● When executing J0 or K0 in G92, cancel any angle specify, fixed 45 degrees.The default value is 45 degrees when powering on.

● When J K are set to be negative number, or beyond to 65535,it's invalid setting. The range: 1~65535.

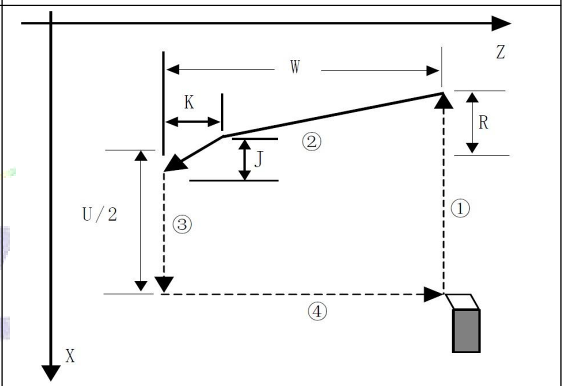
(7) Path of G92 Instruction

Relative position between thread cutting end point and starting point with U, W, R and tool path and thread run-out direction with different U, W, R signs.

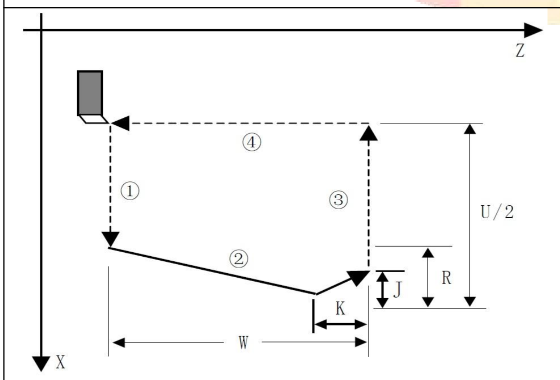
1) U>0 , W>0 , R>0



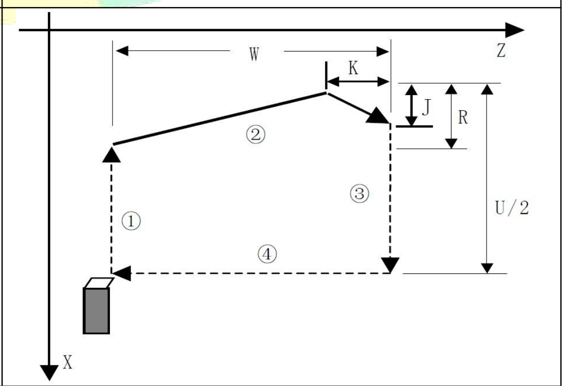
2) U<0 , W<0 , R<0



3) U>0 , W>0 , R<0 & |R| ≤ |U/2|



4) U<0 , W>0 , R>0 & |R| ≤ |U/2|



3.8.3 Canned Tapping Cycle (G93)

Tool path is from starting point to end point and then from end point to starting point. The tool traverses one pitch when the spindle rotates one rev, the pitch is consistent with pitch of tool and there is spiral grooving in internal hole of workpiece and the internal machining can be completed one time.

Canned Tapping Cycle in Z direction Format: G93 Z(W)_ F/I_ ;

Z(W): starting point and end point in Z direction are the same one not to execute the thread cutting when Z or W is not input;

G93 is modal instruction.

F: metric thread pitch

I: teeth per inch thread

Tapping has two kinds of method:

① Tracking the spindle encoder: In Axis parameter: P411=0

Note: spindle must fix with spindle encoder(CN9 plug is connected to SP-encoder).

② Interpolation between spindle servo & Z axis: In Axis Parameter: P405=0, P410=92, P411=4.

Cycle process:

1) Tool infeed in Z negative direction ;
2) Stop spindle (output M05 signal) after the tool reaches the specified end point in Z direction in programming ;
3) Test spindle after completely stopping ;
4) Spindle rotation with reverse direction automatically ;
5) The tool retracts to starting point in Z direction ;
6) Stop spindle (output M05 signal) ;
7) Spindle recover rotation as before G93.

Example: Tapping M10 * 1.5

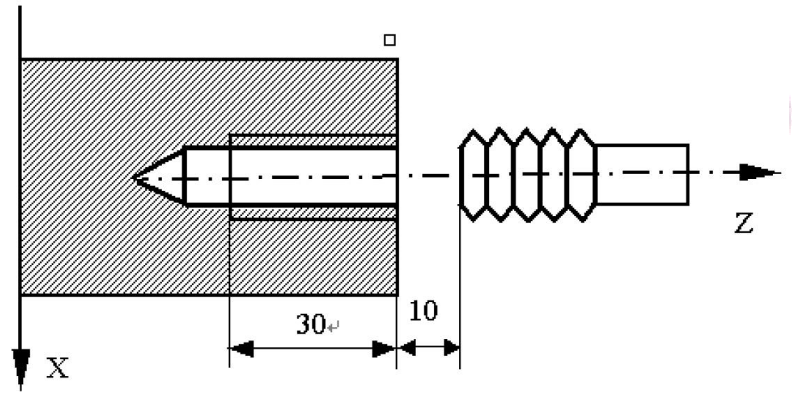


Fig3.8.9 Example of Canned Tapping Cycle

Program: O0011

```
G00 Z90 X0 M03 ;
G93 Z50 F1.5 ;
M03
G00 X60 Z100
M30
```

Example2:

```
G93 Z-100 F5      ; tapping cycle to Z-100;
Z-101             ;  tapping cycle to Z-101;
G00 X50           ;  G00
```

Solution of Canned Tapping Cycle in X direction:

1) P41 in Speed parameter : "compensation mode of arc reverse backlash " , set to 2 , canned tapping cycle in X direction ;

2) Add G19 into this segment when tapping in X direction ; Add G17 into this segment when tapping in Z axis ; Add G18 into next segment after finish tapping.

Example:

G93 G19 X-100 F2 ; Canned Tapping Cycle in X direction
G93 G17 Z-100 F2 ; Canned Tapping Cycle in Z direction
G18 G0 X30 ; Cancel Canned Tapping Cycle

Note:

1. ***If execute G93 after Z moving in positive direction,due to opposite direction, system will make reverse backlash compensation firstly. We should set P13, reverse backlash parameter in Axis parameter . If configured with stepper motor & stuck , we could set the smaller speed value of reverse backlash compensation , also P41-1 & P41-2. Or input the instruction that let Z axis moves with negative direction before executing G93.***
2. ***The parameters of spindle breaking time will affect the start rotating time after stop. Please pay attention to setting these parameters***
3. ***Z-axis must move in negative direction when tapping.***
4. ***Must start spindle rotating before executing G93.***
5. ***The breaking time of spindle should be short.***
6. ***The rotating speed of spindle should be not too high.***
7. ***For specifying inch thread when specifying I is the same as G32 & G92.***
8. ***When choosing the acceleration and deceleration control mode, if the spindle speed change, there is some delay when making the thread change. So choose the non-speed up or down if require the accuracy. However, configured with stepper motor, the speed of spindle cannot be too high, otherwise it will cause the stuck.***

3.8.4 End Face Turning Cycle G94

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

a) Face Cutting Cycle : G94 X(U)_ Z(W)_ F_ ;

X: absolute coordinates of end point of cutting in X direction Unit:mm;

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

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

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

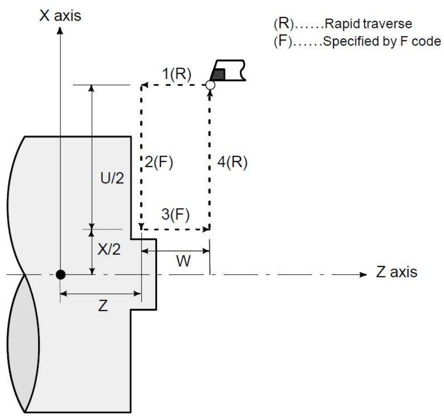


Fig3.8.10 End face loop cutting

In incremental programming, the sign of numbers following addresses U and W depends on the direction of paths 1 and 2. That is, if the direction of the path is in the negative direction of the Z axis, the value of W is negative.

In single block mode, operations 1, 2, 3, and 4 are performed by pressing the cycle start button once.

Example:

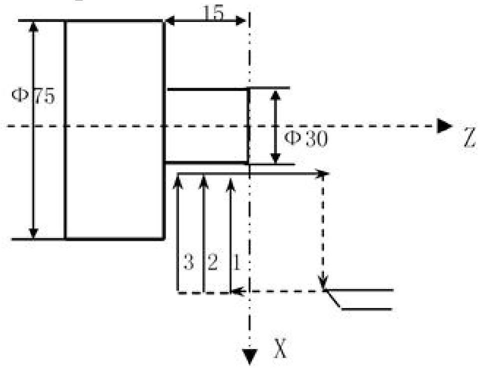


Fig3.8.11 Usage of G94

Program:
N10 M03 S1000;
N20 T0101;
N30 G00 X85 Z10 M08;
N40 G01 Z5 F200;
N50 G94 X30 Z-5 F100;
N60 Z－10;
N70 Z－15;
N80 G00 X100 Z60 M09;
N90 T0100 M05;
N100 M30;

b)Taper Face Cutting Cycle : G94 X(U)_ Z(W)_ R_ F_;

R_: different value(R value) of absolute coordinates from end point to starting point of cutting in X direction. When the sign of R is not the same that of U, R.

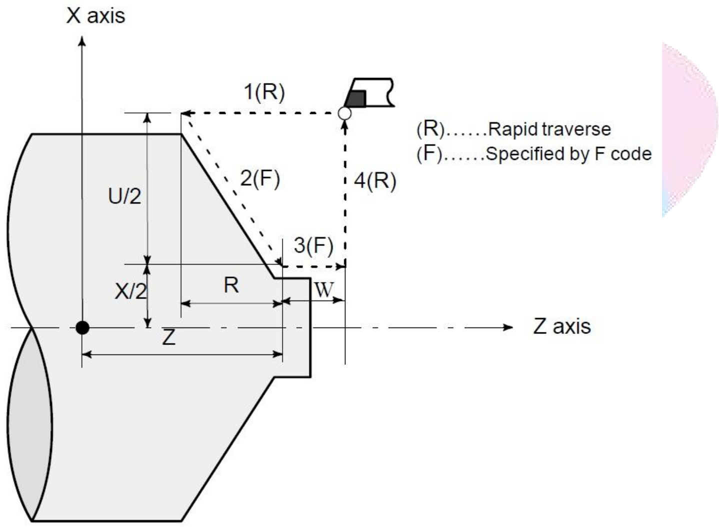
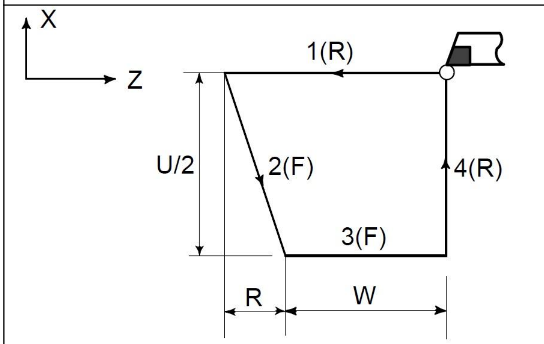
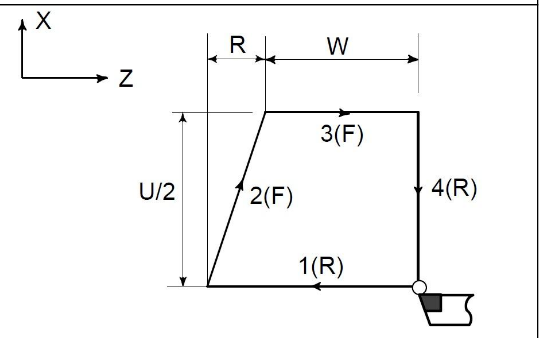
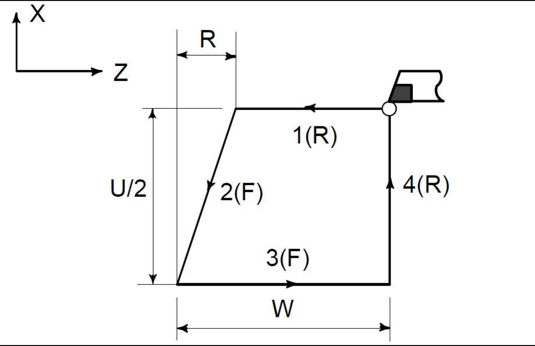
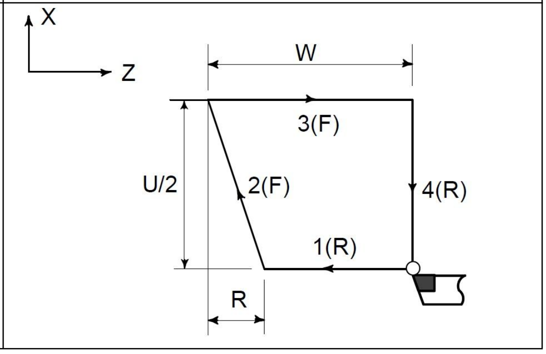


Fig3.8.12 Taper Face Cutting Cycle

c) Signs of numbers specified in the taper cutting cycle

In incremental programming, the relationship between the signs of the numbers following address U, W, and R, and the tool paths are as follows:

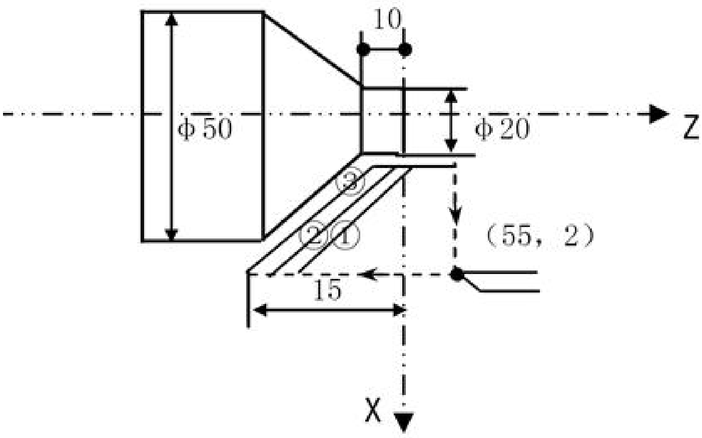
1. U<0 , W<0 , R<0	2. U>0 , W<0 , R
	

<table>
<tr><td>3. U<0 , W<0 , R>0 at |R| ≤ |W|</td><td>4. U>0 , W<0 , R<0 at |R| ≤ |W|</td></tr>
<tr><td>
</td><td>
</td></tr>
</table>

d) Process of Cycle

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

e) Example of Taper Face Cutting Cycle



Program
……
N40 G01 X55 Z2 F200;
N50 G94 X20 Z0 R-5 F100;
N60 Z-5;
N70 Z-10;
N80 G00 X Z;
……
(R in N50 program:
R-5=-15-(-10)=-5mm)

Fig3.8.13 Example of Taper Face Cutting Cycle

Note:

1. Since data values of X (U), Z (W) and R during canned cycle are modal, if X (U), Z (W), or R is not newly commanded, the previously specified data is effective , except that lead I in Inch thread processing. Thus, when the Z axis movement amount does not vary as in the example below, a canned cycle can be repeated only by specifying the movement commands for the X–axis.

However, these data are cleared, if a one–shot G code expect for G04 (dwell) or a G code in the group 01 except for G90, G92, G94 is commanded.

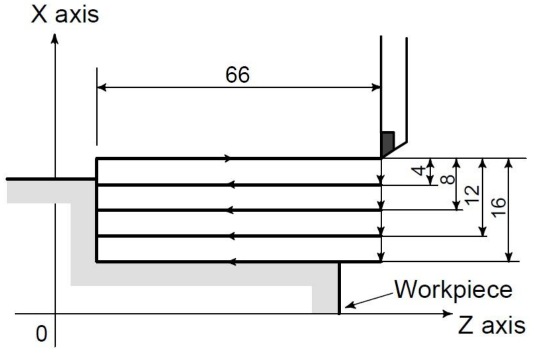
2. The following two applications can be performed.

(1) If an EOB(;) or zero movement commands are specified for the block following that specified with a canned cycle, the same canned cycle is repeated.

(2) Only use "Start" button to run program when input codes in MDI

(3) If the M, S, T function is commanded during the canned cycle mode, both the canned cycle and M, S, or T function can be performed simultaneously. If this is inconvenient, cancel the canned cycle once as in the program examples below (specify G00 or G01) and execute the M, S, or T command. After the execution of M, S, or T terminates, command the canned cycle again.

Example1:



The Cycle in the above figure is executed by the following program.

N030 G90 U-8 .0 W-66.0 F0.4 ;
N031 U-16.0 ;
N032 U-24.0 ;
N033 U-32.0 ;

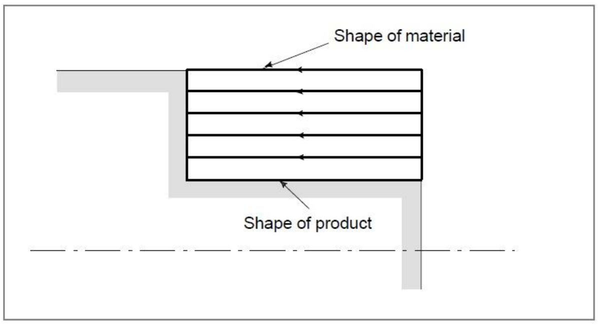
Example2:

N003 T0101 ;
...
N010 G90 X20.0 Z10.0 F0.2
N011 G00 T0202
N012 G90 X20.5 Z10.0

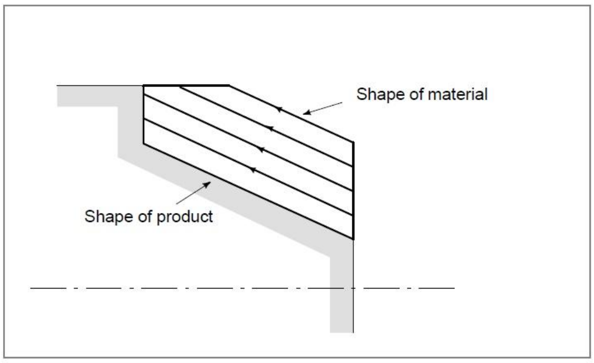
3.8.5 Usage for Canned Cycle

An appropriate canned cycle is selected according to the shape of the material and the shape of the product.

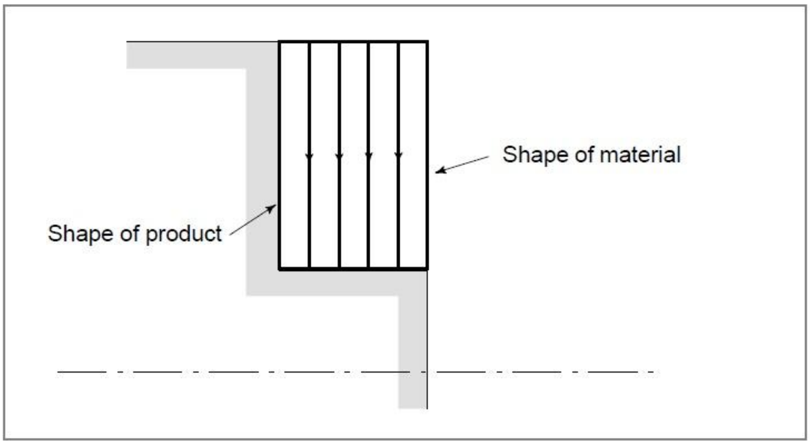
1) Straight Cutting Cycle (G90):



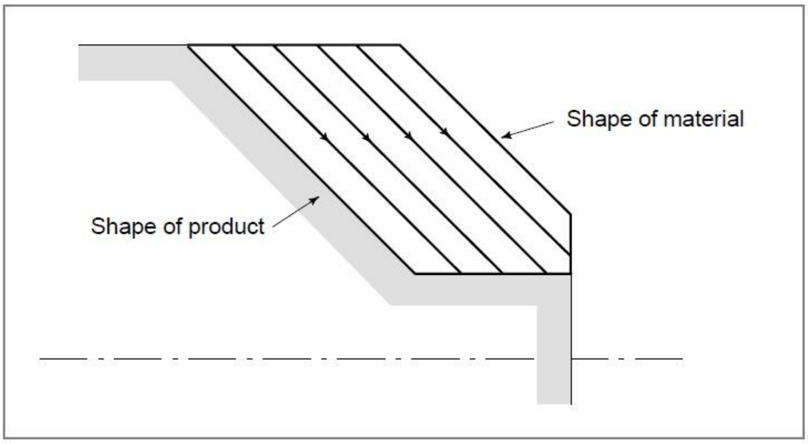
2) Taper Cutting Cycle (G90):



3) Face Cutting Cycle (G94):



4) Taper Face Cutting Cycle (G94):



3.9 Multiple Repetitive Cycle Instructions(G70~G76)

Several types of canned cycles are provided to make programming easier. For instance, the data of the finish work shape describes the tool path for rough machining. And also, a canned cycles for the thread cutting is available.

Multiple cycle instructions of the system includes: Axial roughing turning cycle G71, Radial roughing facing cycle G72, Pattern Repeating Cycle G73, Finishing cycle G70, End face peck drilling cycle G74, Outer/Internal diameter grooving cycle G75 and Multiple thread cutting cycle G76.

When the system executes these instructions, it automatically counts the cutting times and the cutting path according to the programmed path, travels of tool infeed and tool retraction, executes multiple machining cycle (tool infeed → cutting → retract tool → tool infeed), automatically completes the roughing, finishing workpiece and the starting point and the end point of instruction are the same one.

3.9.1 Axial Roughing Turning Cycle (G71)

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

Format: G71 U(△d) R(e);

G71 P(ns) Q(nf) U(△u) W(△w) F(f) S(s) T(t) ;

N(ns) ...... ;

......

F___

S___

T___

N(nf) ...... ;

The move command between A and B is specified in the block from sequence number ns to nf .

△d: Depth of cut (radius designation)

Designate without sign. The cutting direction depends on the direction AA'. This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (P1 in User parameter), and the parameter is changed by the program command.

e : Escaping amount

This designation is modal and is not changed until the other value is designated. Also this value can be specified by the parameter (P2 in User parameter), and the parameter is changed by the program command.

ns : Sequence number of the first block for the program of finishing shape.

nf : Sequence number of the last block for the program of finishing shape.

△u: Distance and direction of finishing allowance in X direction (diameter designation). Also this value can be specified by the parameter (P4 in User parameter). Input negative number when processing inner hole.

△w: Distance and direction of finishing allowance in Z direction. Also this value can be specified by the parameter (P5 in User parameter).

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

f,s,t : Any F , S, or T function contained in blocks ns to nf in the cycle is ignored, and the F, S, or T function in this G71 block is effective.

If a finished shape of A to A' to B is given by a program as in the figure below, the specified area is removed by Δd (depth of cut), with finishing allowance Δu/2 and Δw left.

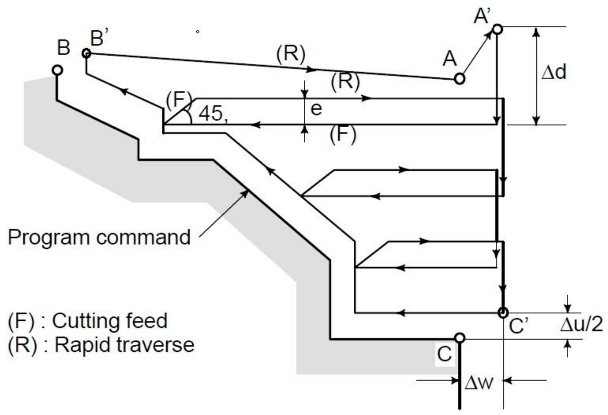


Fig3.9.1 Cutting Path in Axial Roughing Turning Cycle

Execution process:(reference as Fig3.9.1)

① Rapid traverse from A point to A' point , the travel in X direction is Δu, and the travel in Z direction is Δw;

② The travel in X direction from A' point is Δd(tool infeed), ns block is for tool infeed at rapid traverse speed with G0, is for tool infeed at feedrate F with G71, and its direction of tool infeed is that of A→ C point;

③ Cutting feeds to the roughing path in Z direction, and its direction is the same that of coordinates in Z direction C→B point;

④ The travel of tool retraction is e (45°straight line)at feedrate in X, Z direction, the directions of tool retraction is opposite to that of too infeed;

⑤ Rapid retract at rapid traverse speed in Z direction to the position which is the same that of the coordinates in Z direction of A'-C' point;

⑥ After executing the tool infeed (Δd+e)again in X direction, the end point of traversing tool is still on the middle point of straight line between A' and C'(the tool does not reach or exceed C'), and after executing the tool infeed (Δd+e)again, execute ③;after executing the tool infeed (Δd+e)again, the end point of tool traversing reaches C' point or exceeds the straight line between A' →C' point and execute the tool infeed to C' point in X direction and the execute the next step;

⑦ Cutting feed from C' to B' point along the roughing path;

⑧ Rapid traverse to B' from A point and the program jumps to the next clock following nf block after G71 cycle is ended.

Note:

1. G71 in the use of rough machining cycle,The cycle machining is performed by G71 command with P and Q specification. F, S, and T functions which are specified in the move command between points A and B are ineffective and those specified in G71 block or the previous block are effective. But F S T in program of ns→nf is effectively to fine machining , invalid in rough machining cycle.

2. The tool path between A' and B must be steadily increasing or decreasing pattern in both X and Z axis. When the tool path between A and A' is programmed by G00/G01, cutting along AA' is performed in G00/G01 mode respectively.

3. When the constant surface speed control function is enabled, G96 or G97 command specified in the move command between points A and B are ineffective, and that specified in G71 block or the previous block is effective.

4. The tool path between A and A' is specified in the block with sequence number "ns" including G00 or G01, and in this block, a move command in the Z axis cannot be specified.

5. The subprogram cannot be called from the block between sequence number "ns" and "nf".

6. The following four cutting patterns are considered in G71 instruction. All of these cutting cycles are made paralleled to Z axis and the sign of Δu and Δw are as follows:

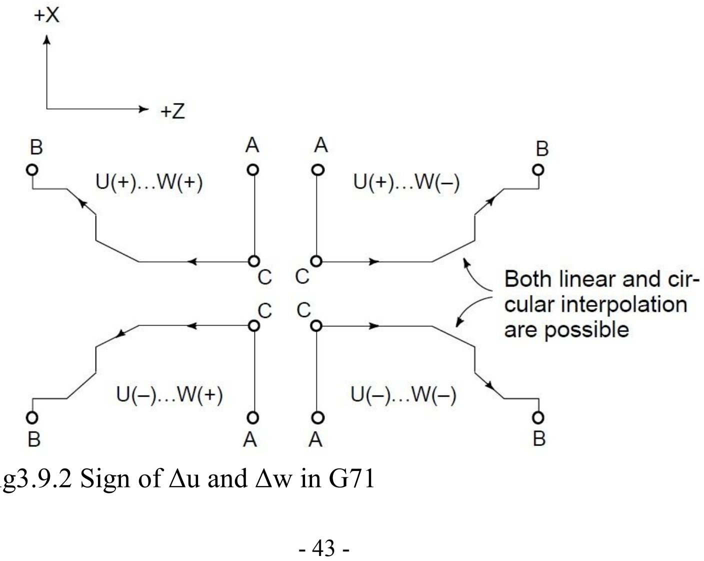


Fig3.9.2 Sign of Δu and Δw in G71

3.9.2 Radial Roughing Facing Cycle (G72)

As shown in the figure below, this cycle is the same as G71 except that cutting is made by a operation parallel to X axis.

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

Format: G72 W(△d) R(e) ;

G72 P(ns) Q(ns) U(△u) W(△w) F(f) S(s) T(t) ;

N(ns) ...... ;

......

F___

S___

T___

N(nf) ...... ;

The move command between A and B is specified in the block from sequence number ns to nf .

The meanings of △d、e、ns、nf、△u、△w、f、s、t are same as those in G71.

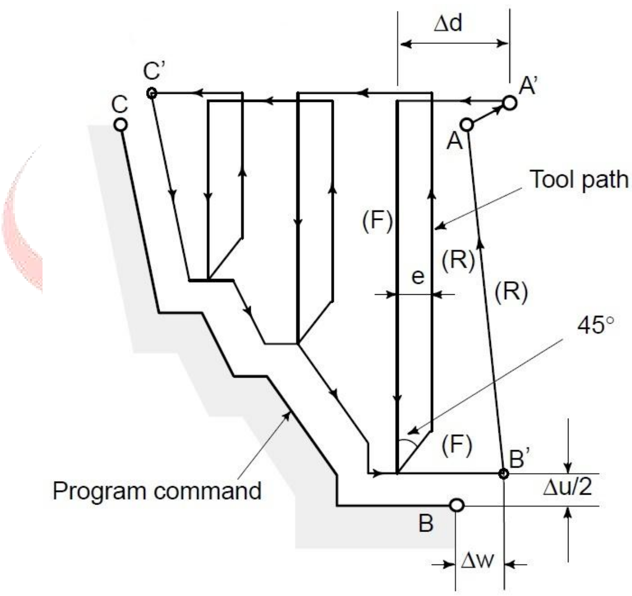


Fig3.9.3 Cutting Path in Radial Roughing Facing Cycle

Execution Process: (reference as Fig3.9.3)

① Rapid traverse from A point to A' point, the travel in X direction is Δu, and the travel in Z direction is Δw;

② The travel in Z direction from A' is Δd(tool infeed), ns block is for tool infeed at rapid traverse speed with G0, is for tool infeed at G72feedrate F in G1, and its direction of tool infeed is that of A→C point;

③ Cutting feeds to the roughing path in X direction, and its direction is the same that of coordinates in X direction C→B point;

④ The travel of tool retraction is e (45º straight line)at feedrate in X, Z direction, the directions of tool retraction is opposite to that of tool infeed ;

⑤ Rapidly retract at rapid traverse speed in X direction to the position which is the same that of the coordinates in Z direction ;

⑥ After executing the tool infeed (Δ d+e)again in Z direction, the end point of traversing tool is still on the middle point of straight line between A’ and C’(the tool does not reach or exceed C’), and after executing the tool infeed (Δ d+e)again, execute ③; after executing the tool infeed (Δ d+e)again, the end point of tool traversing reaches C’ point or exceeds the straight line between A’➞C’ point and
execute the tool infeed to C’ point in Z direction and the execute the next step;

⑦ Cutting feed from C’ to B’ point along the roughing path;

⑧ Rapidly traverse from B’ to A point and the program jumps to the next clock following nf block after G71 cycle is completed.

Use G72 to cut the shape, there are four situation. No matter what kind of is the tool parallel the X axis to cut again. △ u, △ w symbols are as follow

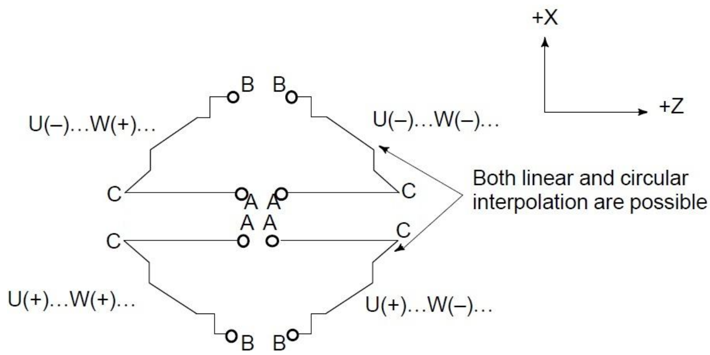


Fig3.9.4 Signs of numbers specified with u and W in G72

The tool path between A and C is specified in the block with sequence number “ns” including G00 or G01, and in this block, a move command in the X axis cannot be specified. The tool path between C and B must be steadily increasing and decreasing pattern in both X and Z axes. Whether the cutting along AC is G00 or G01 mode is determined by the command between A and C.

3.9.3 Pattern Repeating Cycle (G73)

This function permits cutting a fixed pattern repeatedly, with a pattern being displaced bit by bit. By this cutting cycle, it is possible to efficiently cut work whose rough shape has already been made by a rough machining, forging or casting method,etc.The pattern commanded in the program should be as follows:A→A’→B.

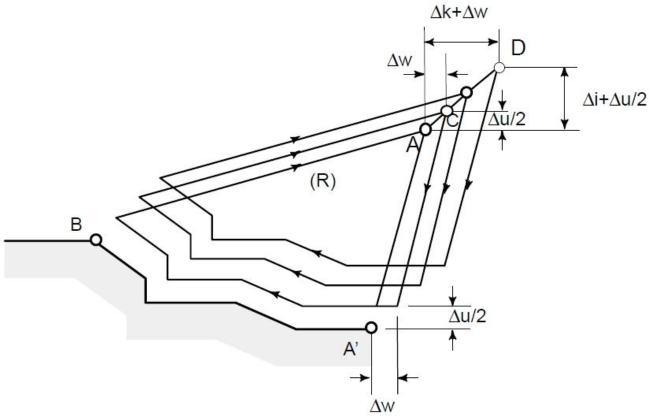


Fig3.9.5 Cutting Path in Pattern Repeating Cycle

Format: G73 U(△i) W(△k) R(d) ;

G73 P(ns) Q (nf) U(△u) W(△w) F(f) S(s) T(t) ;

N(ns) …… ;

…… ;

F_

S_

T_

N(nf) ……;

The move command between A and B is specified in the blocks from sequence number ns to nf.

△i: Distance and direction of relief in the X axis direction (Radius designation).

This designation is modal and is not changed until the other value is designated. Also this value can be specified by P7 in User parameter, and the parameter is changed by the program command.

△k: Distance and direction of relief in the Z axis direction.

This designation is modal and is not changed until the other value is designated. Also this value can be specified by P8 in User parameter, and the parameter is changed by the program command.

d: The number of division.

This value is the same as the repetitive count for rough cutting. This designation is modal and is not changed until the other value is designated. Also, this value can be specified by P6 in User parameter, and the parameter is changed by the program command.

ns : Sequence number of the first block for the program of finishing shape.

nf : Sequence number of the last block for the program of finishing shape.

u : Distance and direction of finishing allowance in X direction (diameter/radius designation)

w : Distance and direction of finishing allowance in Z direction

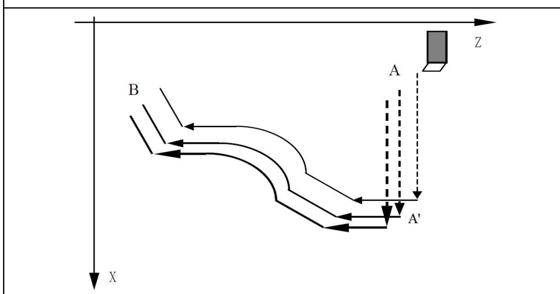
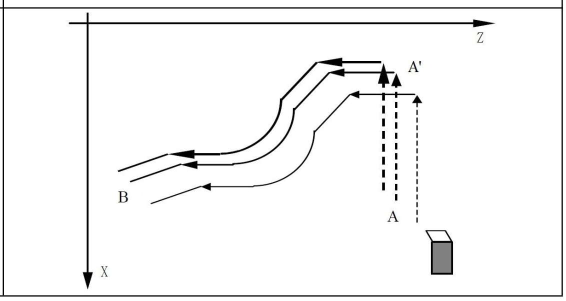
f,s,t : Any F, S, and T function contained in the blocks between sequence number “ns” and “nf” are ignored, and the F, S, and T functions in this G73 block are effective. Others is same as G71.

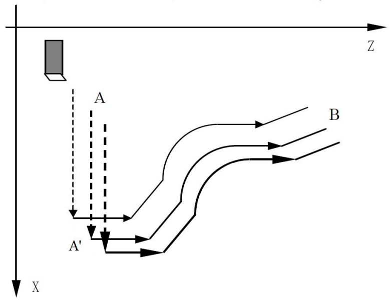
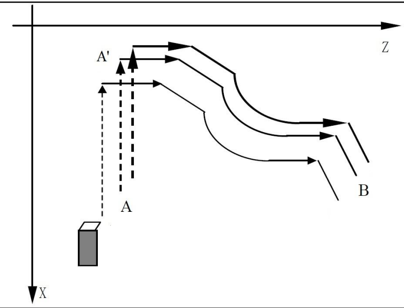
Note: 1. The cycle is according to the program which is between P and Q in G73.The tool backs to A point automatically after finish cycle.

2. Increase or decrease X or Z axis is invalid when using G73.

3. While the values Δi and Δk, or Δu and Δw are specified by address U and W respectively, the meanings of them are determined by the presence of addresses P and Q in G73 block. When P and Q are not specified in a same block, addresses U and W indicates Δi and Δk respectively. When P and Q are specified in a same block, addresses U and W indicates Δu and Δw respectively.

4. The cycle machining is performed by G73 command with P and Q specification.The four cutting patterns are considered. Take care of the sign of Δu, Δw, Δk, and Δi.When the machining cycle is terminated, the tool returns to point A.

1) Δi<0 Δk>0 , △u<0 Δw>0 ;	**2) Δi>0 Δk>0 , △u>0 Δw>0 ;**
	

3) Δi<0 Δk<0 , △u<0 Δw<0 ;	4) Δi>0 Δk<0 , △u>0 Δw<0 ;
	

3.9.4 Finishing Cycle (G70)

After rough cutting by G71, G72 or G73, the following command permits finishing.

Format: G70 P(ns) Q(nf) ;

(ns) : Sequence number of the first block for the program of finishing shape.

(nf) : Sequence number of the last block for the program of finishing shape.

Note:

1 F, S, and T functions specified in the block G71, G72, G73 are not effective but those specified between sequence numbers "ns" and "nf" are effective in G70.

2 When the cycle machining by G70 is terminated, the tool is returned to the start point and the next block is read.

3 In blocks between "ns" and "nf" referred in G70 through G73, the subprogram cannot be called.

3.9.5 Usages of G71,G72,G73 & G70

3.9.5.1 Example of G71&G70

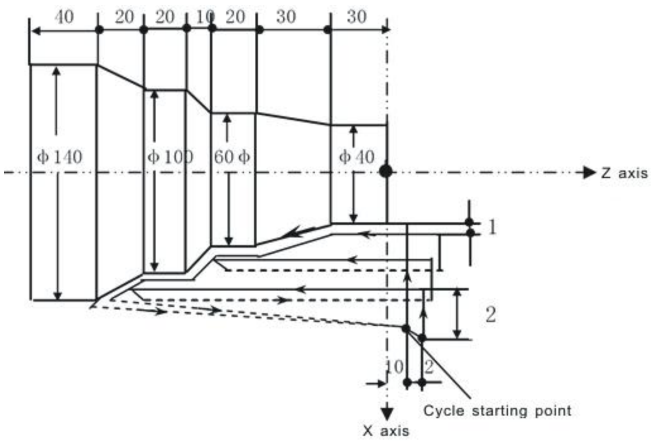


Fig3.9.6 Using of G71&G70

```
N10 M03 S1500;
N20 T0101;
N30 G00 X160 Z10;
N40 G71 U2 R1;
N50 G71 P60 Q120 U2 W1 F100 S2000
N60 G00 X40;
N70 G01 Z-30 F80;
N80 X60 W-30;
N90 W-20;
N100 X100 W-10;
N110 W-20;
N120 X140 W-20;
N130 G70 P60 Q120;
N140 G00 X200 Z50;
N150 T0100 M05;
N160 M30 ;
```

3.9.5.2 Example of G72 & G70

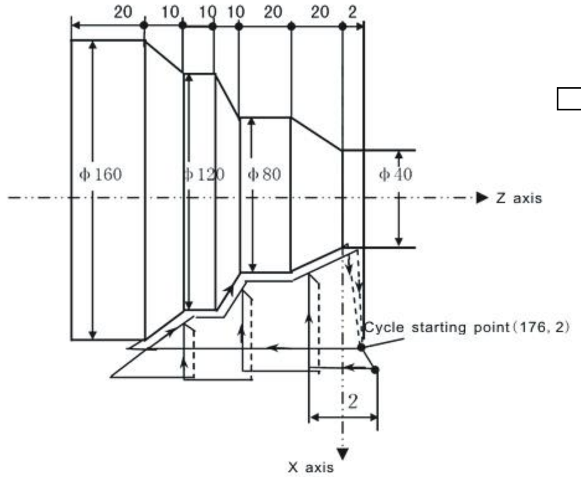


Fig3.9.7 Using of G72&G70

```
N10 M03 S2000;
N20 T0202;
N30 G00 X176 Z2;
N40 G72 W2 R1;
N50 G72 P60 Q120 U2 W1 F100 ;
N60 G00 Z-72;
N70 G01 X160 Z-70 F80;
N80 X120 W10;
N90 W10;
N100 X80 W10;
N110 W20;
N120 X36 W22.08;
N130 G70 P60 Q120;
N140 G00 X200 Z50;
N150 T0200 M05;
N160 M30;
```

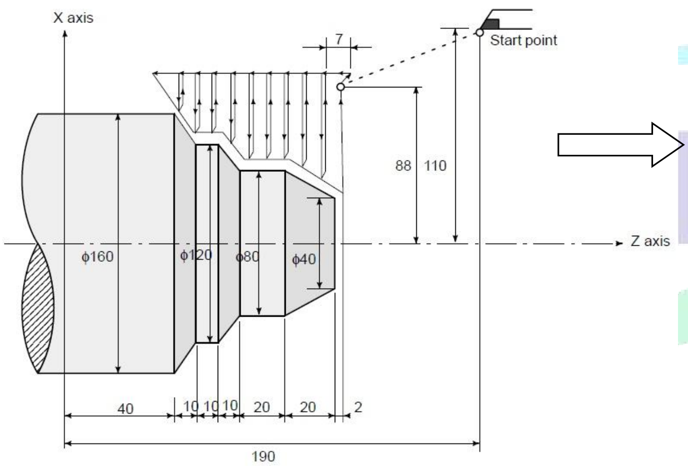


Fig3.9.8 Using of G72&G70

```
(Diameter designation, metric input)
N010 G50 X220.0 Z190.0 ;
N011 G00 X176.0 Z132.0 ;
N012 G72 W7.0 R1.0 ;
N013 G72 P014 Q019 U4.0 W2.0 F0.3
S550 ;
N014 G00 Z58.0 S700 ;
N015 G01 X120.0 W12.0 F0.15 ;
N016 W10.0 ;
N017 X80.0 W10.0 ;
N018 W20.0 ;
N019 X36.0 W22.0 ;
N020 G70 P014 Q019 ;
```

3.9.5.3 Example of G73&G70

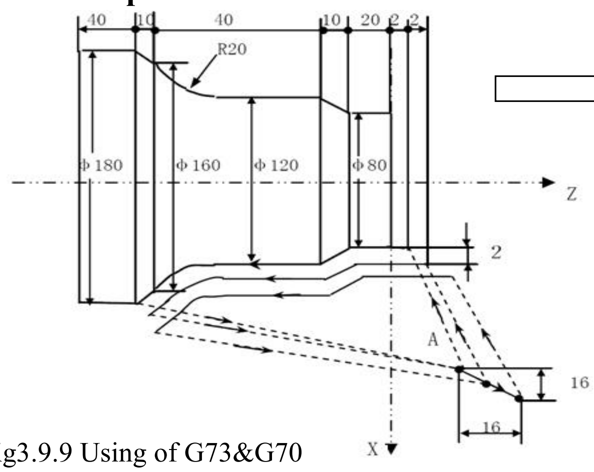
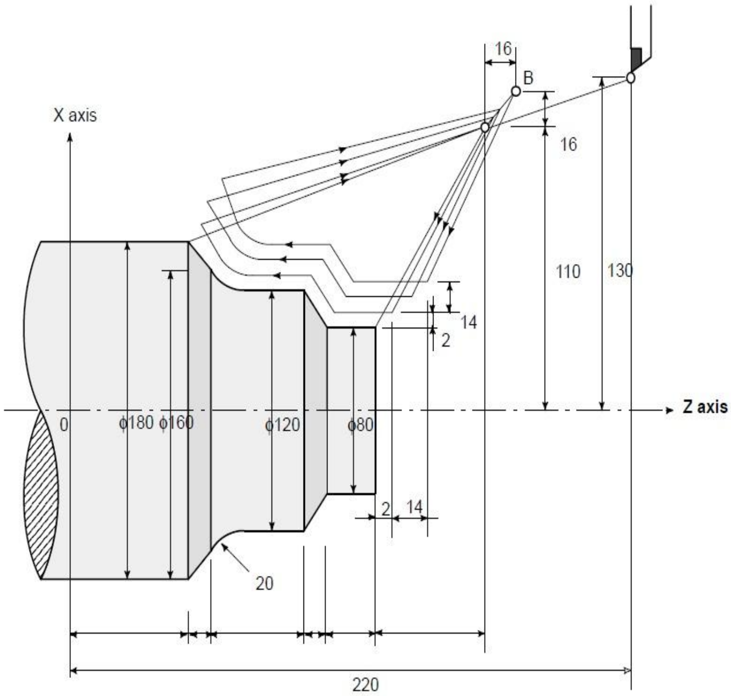


Fig3.9.9 Using of G73&G70

```
N10 M03 S3000;
N20 T0303;
N30 G00 X220 Z40;
N40 G73 U14 W14 R0.010;
N50 G73 P60 Q110 U4 W2 F100;
N60 G00 X80 Z2;
N70 G01 Z-20 F80;
N80 X120 W-10;
N90 W-20;
N100 G02 X160 W-20 R20;
N110 G01 X180 W-10;
N120 G70 P60 Q110;
N130 G00 X250 Z50;
N140 T0300 M05;
N150 M30;
```

(b)




```
(Diameter designation, metric input)
N010 G50 X260.0 Z220.0;
N011 G00 X220.0 Z160.0 ;
N012 G73 U14.0 W14.0 R3 ;
N013 G73 P014 Q019 U4.0 W2.0
F0.3 S0180 ;
N014 G00 X80.0 W–40.0 ;
N015 G01 W–20.0 F0.15 S0600 ;
N017 W–20.0 S0400 ;
N018 G02 X160.0 W–20.0 R20.0;
N019 G01 X180.0 W–10.0 S280;
N020 G70 P014 Q019 ;
```

Fig3.9.10 Using of G73 & G70

3.9.6 End Face Peck Drilling Cycle (G74)

The following program generates the cutting path shown in Fig3.9.11. Chip breaking is possible in this cycle as shown below. If X (U) and Pare omitted, operation only in the Z axis results, to be used for drilling.

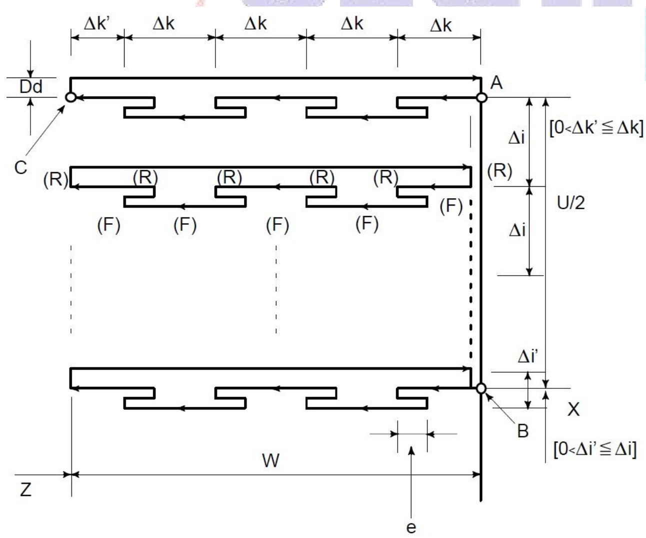


Fig3.9.11 Cutting Path in End Face Peck Drilling Cycle

Format: G74 R(e) ;

G74 X(u) P(△i) Z(w) Q(△k) R(△d) F(f) ;

e: Return amount;

This designation is modal and is not changed until the other value is designated. Also this value can be specified by P10 in User parameter, and the parameter is changed by the program command.

X : X component of point B

U : Incremental amount from A to B
Z : Z component of point C
W : Increment amount from A to C
Δi : Movement amount in X direction (without sign).
Δk : Depth of cut in Z direction (without sign),also can be set by P9 in User parameter. (Unit: um)
Δd : Relief amount of the tool at the cutting bottom. The sign of Δd is always plus(+). However, if address X (U) and Δi are omitted, the relief direction can be specified by the desired sign.
f : Feed rate

NOTE

1. While both e and d are specified by address R, the meanings of them are determined by the present of address X (U). When X(U) is specified, d is used.

2. The cycle machining is performed by G74 command with X(U) specification.

Execution process:(Fig3.9.11)

① Axial (Z axis) cutting feed △k from the starting point of G74, feed in Z negative direction when the coordinates of cutting end point is less than that of starting point in Z direction, otherwise, feed in Z positive direction;

②Axial(Z axis) rapid tool retraction e and its direction is opposite to feeding direction;

③ Cutting feed(Δk+e) again in Z direction, the end point of cutting feed is still in it between starting point An of axial cutting cycle and end point of axial tool infeed, cutting feed (Δk+e)again in Z direction and execute ②; after cutting feed (Δk+e)again in Z direction, end point of cutting feed is on Bn or isn't on it between An and Bn cutting feed to Bn in Z direction ,then execute ④;

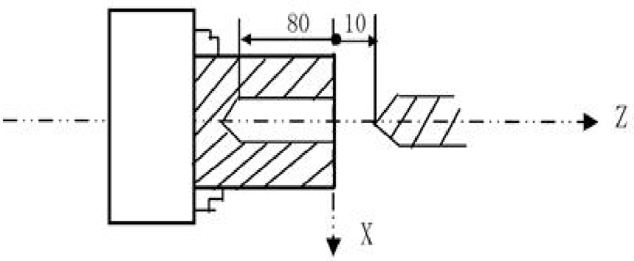
④ Radial(X axis) rapid tool retraction △d（radius value）to Cn , when the coordinates of Bf (cutting end point) is less than that of A (starting point) in X direction, retract tool in X positive, otherwise, retract tool in X negative direction;

⑤ Axial(Z axial) rapid retract tool to Dn, No. n axial cutting cycle is completed. If the current axial cutting cycle is not the last one, execute ⑥ ; if it is the previous one before the last axial cutting cycle, execute ⑦;

⑥ Radial(X axial)rapid tool infeed, and it direction is opposite to ④ retract tool. If the end point of tool infeed is still on it between A and Af (starting point of last axial cutting cycle) after tool infeed(△d+△i) (radius value) in X direction, i.e. n→An+1 and then execute ① (start the next axial cutting cycle); if the end point of tool infeed is not on it between Dn and Af after tool infeed (△d+△i) (radius value) in X direction, rapidly traverse to Af and execute ① to start the first axial cutting cycle;

⑦ Rapidly traverse to return to A in X direction, and G74 is completed.

Example:




```
N10 G00 X0 Z10;
N20 G74 R2;
N30 G74 Z-80 Q10000 F800;
N40 G00 X50 Z50;
N50 M30;
```

Fig3.9.12 Example of G74

3.9.7 Outer Diameter/Internal Diameter Drilling Cycle (G75)

The following program generates the cutting path shown in Fig3.9.13.

This is equivalent to G74 except that X is replaced by Z. Chip breaking is possible in this cycle, and grooving in X axis and peck drilling in X axis (in this case, Z, W, and Q are omitted) are possible.

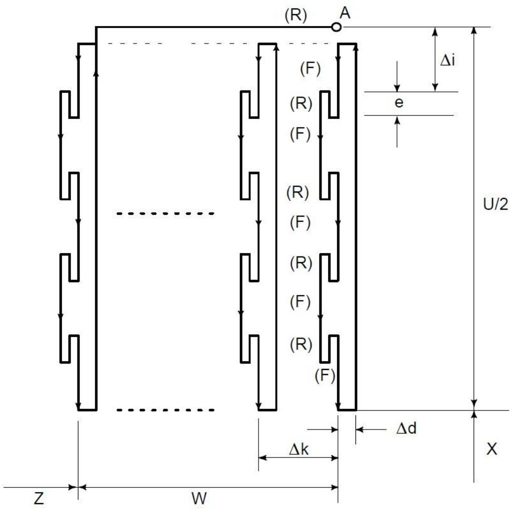


Fig3.9.13 Cutting Path in Outer/Internal Diameter Drilling Cycle

Format: G75 R(<u>e</u>) ;
G75 X(<u>U</u>) P(<u>Δi</u>) Z(<u>w</u>) Q(<u>△k</u>) R(<u>Δ d</u>) F(<u>f</u>) ;

X : X component of point B
U : Incremental amount from A to B
Z : Z component of point C
W : Increment amount from A to C

Δi : Movement amount in X direction (without sign),also can be set by P9 in User parameter.(Unit: um)

Δk : Depth of cut in Z direction (without sign),

Δd : Relief amount of the tool at the cutting bottom. The sign of Δd is always plus(+). However, if address X (U) and Δi are omitted, the relief direction can be specified by the desired sign.

f : Feed rate

Both G74 and G75 are used for grooving and drilling, and permit the tool to relief automatically. Four symmetrical patterns are considered,respectively.

Execution process:(Fig3.9.14)

① Radial (X axis) cutting feed △i from the starting point of radial cutting cycle, feed in X negative direction when the coordinates of cutting end point is less than that of starting point in X direction,otherwise, feed in X positive direction;

② Radial(X axis) rapid tool retraction e and its direction is opposite to the feed direction of ①;

③ Cutting feed(Δ k+e) again in X direction, the end point of cutting feed is still in it between starting point An of radial cutting cycle and end point of radial tool infeed, cutting feed (Δ i+e)again in X direction and execute ②; after cutting feed (Δ i+e)again in X direction, the end point of cutting feed is on Bn or is not on it between An and Bn cutting feed to Bn in X direction and then execute ④ ;

④ Axial(Z axis) rapid tool retraction △d（radius value）to Cn , when the coordinates of Bf (cutting end point) is less than that of A (starting point) in Z direction, retract tool in Z positive, otherwise, retract tool in Z negative direction;

⑤ Radial(Z axis) rapid retract tool to Dn, No. n radial cutting cycle is completed. The current radial cutting cycle is not the last one, execute ⑥ ; if it is the previous one before the last radial cutting cycle, execute ⑦;

⑥ Axial(X axis)rapid tool infeed, and it direction is opposite to ④ retract tool. If the end point of tool infeed is still on it between A and Af (starting point of last radial cutting cycle) after tool infeed (△d+△k) (radius value) in Z direction, i.e. Dn→An+1 and then execute ① (start the next radial cutting cycle); if the end point of tool infeed is not on it between Dn and Af after tool infeed (△d+△k) in Z direction, rapidly traverse to Af and execute ① to start the first radial cutting cycle;

⑦ Rapidly traverse to return to A in Z direction, and G75 is completed.

Example:

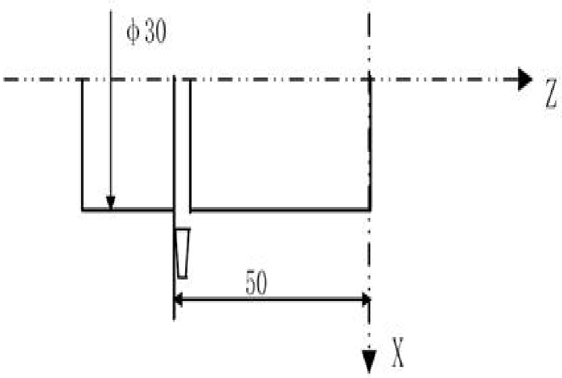


Fig3.9.14 Example of G75

```
N10 M03 S1000;
N20 T0101;
N30 G00 X35 Z-50;
N40 G75 R1;
N50 G75 X-1 P5000 F60;
N60 G00 X100 Z50 M09;
N70 M05;
N80 T0100;
N90 M30;
```

3.9.8 Multiple Thread Cutting Cycle (G76)

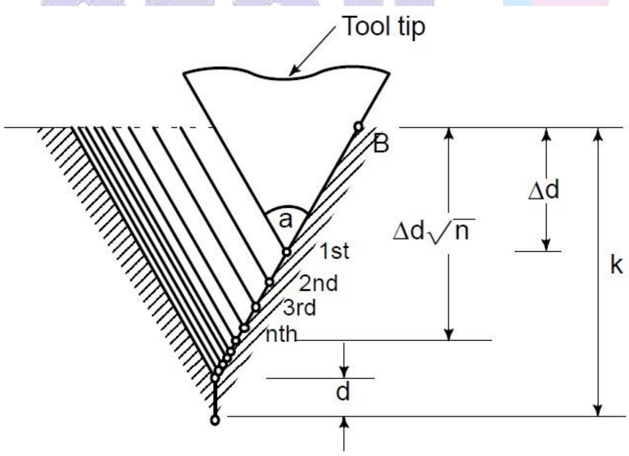


Fig3.9.15 Details of Cutting

Format: **G76 P(b)(c)(m)(r)(a) Q(Δdmin) R(d) ;**

G76 X(U)_ Z(W)_ R(i) P(k) Q(Δd) F(I)_ L(L)[or SP];

P actually consists of multiple values which control the thread behavior.

b: 0———Infeed incrementally;
1———Equidistant infeed;
2———If the fist feed is too long in digression feed,so divide into two infeed.

c: 0———right enter;
1———left enter;
2———middle enter
3———right and left enter, the first feed is middle.

m: Repetitive count in finishing (1 to 99)

This designation is modal and is not changed until the other value is designated. Also this value can be specified by P11 in User parameter, and the parameter is changed by the program command.

r: Chamfer amount.

When the thread lead is expressed by L, the value of L can be set from 0.0L to 9.9L in 0.1L increment (2–digit number from 00 to 90). This designation is modal and is not changed until the other value is designated. Also this value can be specified by P12 in User parameter, and the parameter is changed by the program command.

a: Angle of tool tip

One of six kinds of angle, 80°, 60°, 55°, 30°, 29°, and 0°, can be selected,and specified by 2–digit number. This designation is modal and is not changed until the other value is designated. Also this value can be specified by P13 in User Parameter, and the parameter is changed by the program command.

b ,c,m, r, and a are specified by address P at the same time.

Example: When b=2, c=3, m=1, r=1.2K, a=60°, specify as shown below (K is lead of thread).

$P\frac{2}{b}\frac{3}{c}\frac{01}{m}\frac{12}{r}\frac{60}{a}$, coding instruct is : P23011260

Q(Δdmin): Minimum cutting depth. (specified by the radius value)

When the cutting depth of one cycle operation ($\Delta d^*\sqrt{n} - \Delta d^*\sqrt{n-1}$) becomes smaller than this limit, the cutting depth is clamped at this value. This designation is modal and is not changed until the other value is designated. Also this value can be specified by P14 in User Parameter, and the parameter is changed by the program command. Unit: um.

R(d): Finishing allowance.

This designation is modal and is not changed until the other value is designated. Also this value can be specified by P15 in User parameter, and the parameter is changed by the program command.

X: Absolute coordinates (unit: mm) of thread end point in X direction;

U: Different value (unit: mm) of absolute coordinates between thread end point and starting point in X direction;

Z: Absolute coordinates (unit: mm) of thread end point in Z direction;

W: Different value (unit: mm) of absolute coordinates between thread end point and starting point in Z direction;

R(i): Difference of thread radius.

If i = 0, ordinary straight thread cutting can be made.

P(k): Height of thread. This value is specified by the radius value.

F: metric thread pitch. (same as G32).0.1~500.000mm.

I: Thread teeth per inch for inch thread. 0.1~99 teeth.

L: multiple thread head numbers.

SP: starting angle: 0-360°, unit is 0.001 degree. No specify means 0 degree.

Δ d: Depth of cut in 1st cut (radius value) .Or infeed times.

First cut amount (with G32 threading) in microns; or feed times.

P24 in User parameter is set for meaning of Q (Δ d) of G76. [P24 = 8 ,times of roughing infeed]. When P24 = 8, Q (Δ d) means times that needed to complete the roughing cycle, the default is 1; otherwise Q (Δ d) means that depth of cut in 1st cut. Q (Δ d) , times of infeed, also there are modes of equidistant infeed and decrements infeed.

The infeed amount and infeed times of roughing in all case are as follows:

1) b = 0, P24 ≠ 8, every infeed depth: $\nabla d\sqrt{\text{n}}$;

2) b = 0, P24 = 8, each infeed depth is: the same way as a) according to Δd calculated as: $\frac{(\text{k}-d)}{\sqrt{\nabla d}}\sqrt{n}$, infeed times is Δd;

3) b=1, P24≠8, amount of each infeed: Δd,times of roughing infeed is (k-d)/Δd;

4) b=1, P24=8,the amount of each infeed: (k-d)/Δd,roughing feed times for Δd;

The thread cutting cycle as shown in Fig3.9.16 is programmed by G76 command.

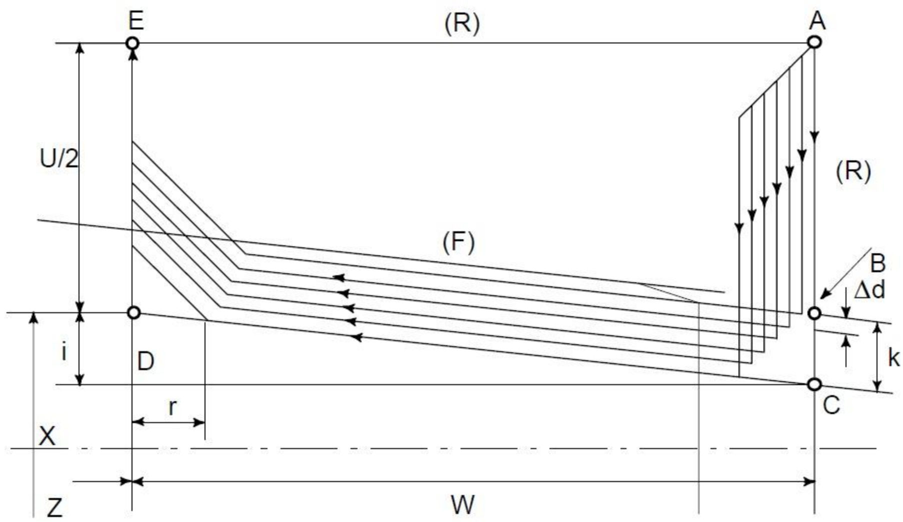


Fig3.9.16 Cutting path in multiple thread cut

Execution process:

① The tool rapidly traverses to B(1st),and the thread cutting depth is △d. The tool only traverses in X direction when a=0; the tool traverses in X and Z direction and its direction is the same that of A→D when a≠0;

② The tool cuts threads paralleling with C→D to the intersection of D→E (r≠0: thread run-out);

③ The tool rapidly traverses to E point in X direction;

④ The tool rapidly traverses to A point in Z direction and the single roughing cycle is completed;

⑤ The tool rapidly traverses again to tool infeed to B(nth) (is the roughing times), the cutting depth is the bigger value of ($\sqrt{n}\times$△d),($\sqrt{\text{n}-1}\times$△d+△dmin) and execute ② if the cutting depth is less than（k-d）; if the cutting depth is more than or equal to（k-d）, the tool infeeds（k-d）to B((n+1)th) ,and then,execute ⑥ to complete the last thread roughing;

⑥ The tool cuts threads paralleling with C→D to the intersection of D→E (r≠0: thread run-out);

⑦ The tool rapidly traverses to E point in X direction;

⑧ The tool rapidly traverses to A point in Z direction and the thread roughing cycle is completed to execute the finishing;

⑨ After the tool rapidly traverses to B(the cutting depth is k and the cutting travel is d), execute the thread finishing, at last the tool returns to A point and so the thread finishing cycle is completed;

⑩ If the finishing cycle times is less than m, execute ⑨ to perform the finishing cycle, the thread cutting depth is k and the cutting travel is 0; if the finishing cycle times is equal to m, G76 compound thread machining cycle is completed.

Thread Cutting Cycle Retract:

When feed hold is applied during threading in the multiple thread cutting cycle (G76), the tool quickly retracts in the same way as in chamfering performed at the end of the thread cutting cycle. The tool goes back to the start point of the cycle. When cycle start is triggered, the multiple thread cutting cycle resumes.

Note: 1. The meanings of data specified by address P, Q, and R determined by the presence of X(U) and X (W).

2. The cycle machining is performed by G76 command with X (U) and Z (W) specification. By using this cycle , one edge cutting is performed and the load on the tool tip is reduced.Making the cutting depth d for the first path, and dn for the nth path, cutting amount per one cycle is held constant.Four symmetrical patterns are considered corresponding to the sign of each address. The internal thread cutting is available. In the above figure, the feed rate between C and D is specified by address F, and in the other path, at rapid traverse. The sign of incremental dimensions for the above figure is as follows:

U, W : minus (determined by the direction of the tool path AC and CD.)

R(i) : minus (determined by the direction of the tool path AC.)

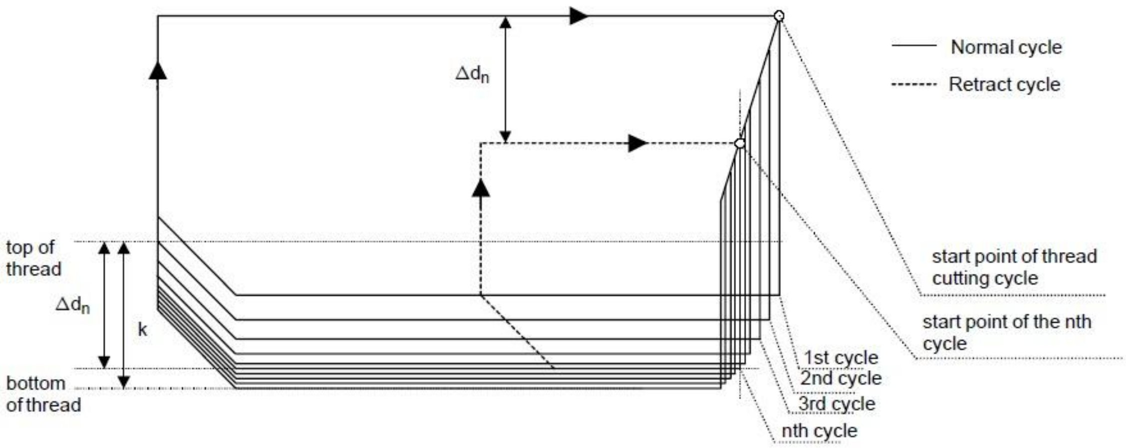
P(k) : plus (always)

Q(△d) : plus (always)

3. Notes on thread cutting are the same as those on G32 thread cutting and G92 thread cutting cycle.

4. The designation of chamfering is also effective for G92 thread cutting cycle.

5. The tool returns to the cycle start point (cutting depth dn) as soon as the feed hold status is entered during thread cutting. (dn : cutting depth in nth cut)



6. If start point of thread cutting cycle is close to a workpiece, tool may interfere with the workpiece during retract cycle because of passing along the retract cycle route described at Note 5. Therefore start point of thread cutting cycle must be at least k(height of thread) away from the top of thread.

7. pay attention to cut thread, use G32 to cut is the same as using G92.

8. specify the chamfering amount of thread, it's also effective to the G92 thread cutting circle.

9. when matching step motor, because of the acceleration or deceleration the thread in tail will be in-homogeneous. So should choose the linear acceleration or deceleration to control X axis with G00 to back tail fast.

Example: Multiple Repetitive Cycle G76

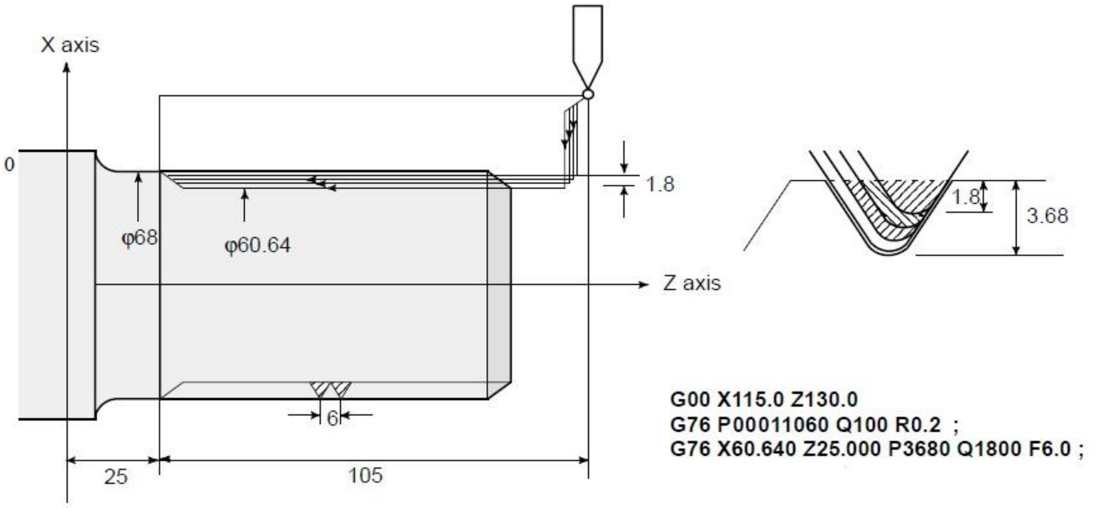


Fig3.9.16 Example of G76

3.9.9 Notes On Multiple Repetitive Cycle (G70 ~ G76)

1. In the blocks where the multiple repetitive cycle are commanded, the addresses P, Q, X, Z, U, W, and R should be specified correctly for each block.

2. In the block which is specified by address P of G71, G72 or G73, G00 or G01 group should be commanded.

3. In MDI mode, G70, G71, G72, or G73 cannot be commanded.G74, G75, and G76 can be commanded in MDI mode.

4. In the blocks in which G70, G71, G72, or G73 are commanded and between the sequence number specified by P and Q, M98 (subprogram call) and M99 (subprogram end) cannot be commanded.

5. In the blocks between the sequence number specified by P and Q, the following commands cannot be specified.

- One shot G code except for G04 (dwell)
- 01 group G code except for G00, G01, G02, and G03
- 06 group G code
- M98 / M99

6. While a multiple repetitive cycle (G70~G76) is being executed, it is possible to stop the cycle and to perform manual operation. But, when the cycle operation is restarted, the tool should be returned to the position where the cycle operation is stopped. If the cycle operation is restarted without returning to the stop position, the movement in manual operation is added to the absolute value, and the tool path is shifted by the movement amount in manual operation.

7. When G70, G71, G72, or G73 is executed, the sequence number specified by address P and Q should not be specified twice or more in the same program.

8. The blocks between the sequence number specified by P and Q on the multiple repetitive cycle must not be programmed by using “Direct Drawing Dimensions Programming” or “Chamfering/Corner R”.

9. G74, G75, and G76 also do not support the input of a decimal point for P or Q. The least input increments are used as the units in which the amount of travel and depth of cut are specified.

10. When #1 = 2500 is executed using a custom macro, 2500.000 is assigned to #1. In such a case, P#1 is equivalent to P2500.

11. Tool nose radius compensation cannot be applied to G71, G72, G73, G74, G75, or G76.

12. The multiple repetitive cycle cannot be executing during DNC operation.

13. Interruption type custom macro cannot be executed during executing the multiple repetitive cycle.

3.10 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 machine coordinate value
#5022 Third axis (Cs axis) machine coordinate value
#5023 Z axis machine coordinate value
#5024 4th axis (A axis) machine coordinate value

Example: G31 X50 Z100 F100 P**33**1022 ;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.

3.11 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

Example:

```
N0000   M03 M08
N0001   G0    X200 Z200
N0002   G01 W-100 F300
N0003   G22   L6              ; 6 times cycle
N0004   G01   U-22   F100
N0005   W－11   U6
N0006   W-30
N0007   W-10 U5
N0008   G0   U10
N0009   W51
N0010   G800                  ;loop end
N0011   G26
N0012   M30
```

3.12 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(C)-axis return to starting point
G263 ; Z-axis return to starting point
G264 ; A-axis return to starting point

3.13 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

3.14 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(C)-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
```

3.15 Return to Reference Position (G28)

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.For example, the reference position is used as a position at which tools are automatically changed.

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.

Reference Position offset of axes to home can be set by parameters.

X-axis offset: P32 in Axis parameters , unit: 0.01mm.

Z-axis offset: P33 in Axis parameters, unit: 0.01mm.

C-axis offset: P116 in Axis parameters, unit: 0.01mm.

A-axis offset: P214 in Axis parameters, unit: 0.01mm.

User could return reference point in Manual (Press "Return" key) or in Auto (Use G28 instruction).

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

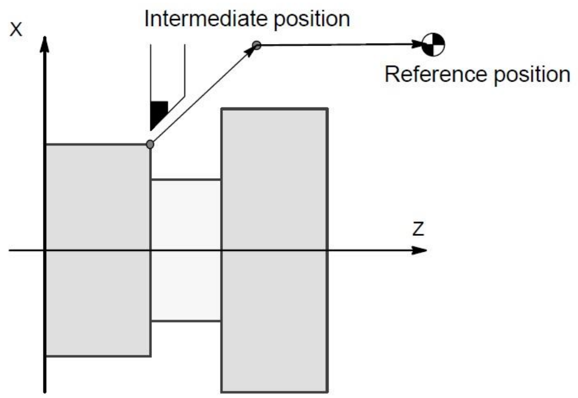


Fig3.15.1 Reference Position Return

Format: G28 IP_;

IP_:Command specifying the intermediate position of all feeding axes (Absolute/incremental command)

G281 ; Only X-axis return to reference position
G282 ; Only Y(C)-axis return to reference position,
G283 ; Only Z-axis return to reference position,
G284 ; Only A-axis return to reference position,

Note: 1. When use G282 instruction, Y(C)-axis return reference position,the system only detect Y0 (PIN2 of CN3 plug),don't detect Z0 signal of encoder.

2. When use M800, C-axis return to zero position(Z0) of encoder.please check details of M800 in M code chapter.

3. When the G28 instruction is specified when manual return to the reference position has not been performed after the power has been turned on, the movement from the intermediate point is the same as in manual return to the reference position.In this case, the tool moves in the direction for reference position return specified by P28 in Axis parameter. Therefore the specified intermediate position must be a position to which reference position return is possible.

4. When use G28 instruction to return to reference position, if just specify intermediate position of some axes, which can return to reference position, the others axes that don't be specified cannot return to reference position.

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

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

3.16 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 coordinates in a coordinate system. Coordinates are specified using program axes.

When two program axes, the X – axis and Z – axis, are used, coordinates are specified as follows:

X_Z_

This command is referred to as a dimension word.

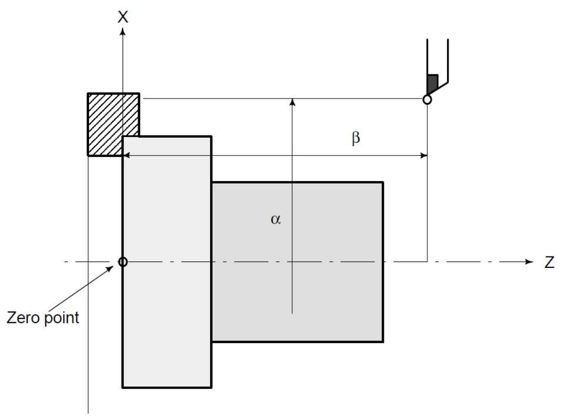


Fig3.16.1 Tool Position Specified by XαZβ

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_**.

3.16.1 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 until the power is turned off.

G53 IP_ ;

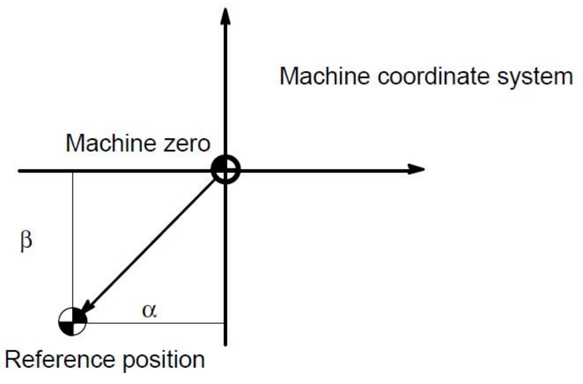
IP_ : absolute dimension word

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. The G53 command must be specified using absolute values. If incremental values are specified, the G53 command is ignored. 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.

Note: 1. When the G53 command is specified, cancel the tool nose radius compensation and tool offset.

2. Since the machine coordinate system must be set before the G53 command is specified, at least one manual reference position return or automatic reference position return by the G28 command must be performed after the power is turned on. This is not necessary when an absolute–position detector is attached.

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 P32 & P33 in Axis parameter .



3.16.2 Workpiece Coordinate System

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 NC 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 toG59 to select which workpiece coordinate system to use.

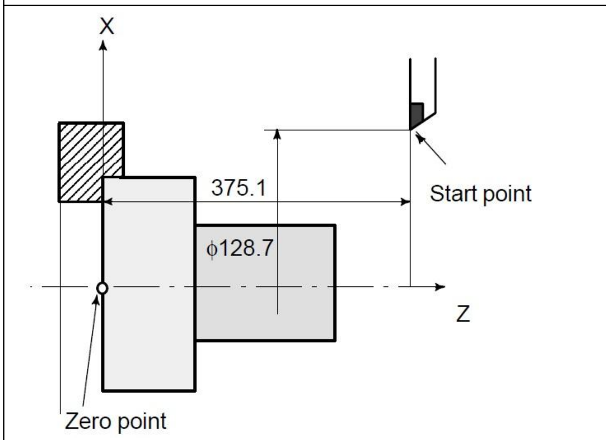
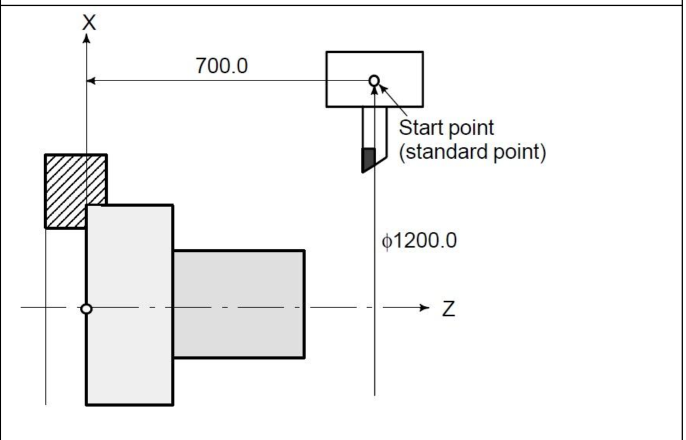
When an absolute command is used, a workpiece coordinate system must be established in any of the ways described above.

3.16.3 Setting a Workpiece Coordinate System(G50)

Format: G50 IP_ ;

A workpiece coordinate system is set so that a point on the tool, such as the tool tip, is at specified coordinates. If IP is an incremental command value, the work coordinate system is defined so that the current tool position coincides with the result of adding the specified incremental value to the coordinates of the previous tool position. If a coordinate system is set using G50 during offset, a coordinate system in which the position before offset matches the position specified in G50 is set. And after workpiece coordinate system set well, absolute coordinate position of following commands are based on this coordinate system. Setting solution of G50 workpiece coordinate system(See Chapter of coordinate system set)

Note: In the status of compensation, if use G50 to set the workpiece coordinate system, position before make tool compensation is based on G50 workpiece coordinate system. Usually cancel the tool compensation first of all before starting program. After return to reference position, system will cancel tool compensation automatically.

Example1: Setting the coordinate system by the G50X128.7Z375.1; command (Diameter designation)	Example2: Base Point Setting the coordinate system by the G50X1200.0Z700.0; command (Diameter designation)
	

3.16.4 Selecting a Workpiece Coordinate System

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

(1)G50 or automatic workpiece coordinate system setting

Once a workpiece coordinate system is selected, absolute commands work with the workpiece coordinate system.

(2)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.

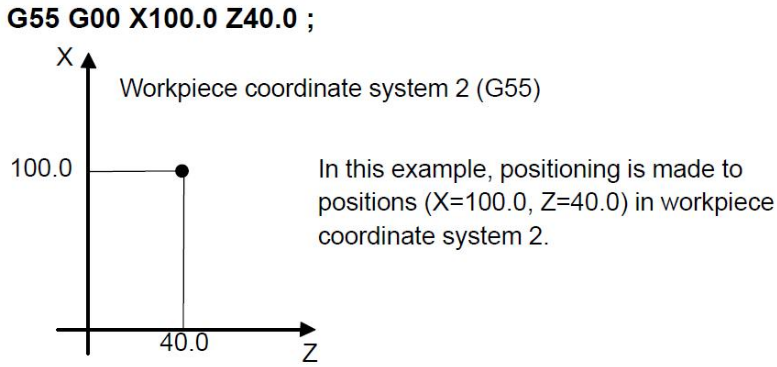


Fig3.16.2 Example of Workpiece Coordinate System

3.16.5 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 (see chapter 5.6.5)
(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 Coor parameter.

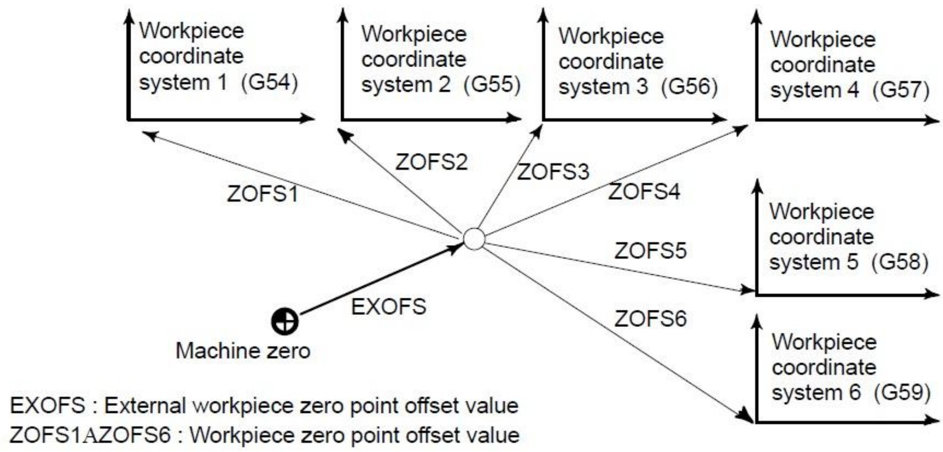


Fig3.16.3 Workpiece Zero Point Offset

Format: Changing by G50 : G50 IP_ ;

By specifying G50IP_;, a workpiece coordinate system (selected with a code from G54 to G59) is shifted to set a new workpiece coordinate system so that the current tool position matches the specified coordinates (IP_).

If IP is an incremental command value, the work coordinate system is defined so that the current tool position coincides with the result of adding the specified incremental value to the coordinates of the previous tool position. (Coordinate system shift)

Then, the amount of coordinate system shift is added to all the workpiece zero point offset values. This means that all the workpiece coordinate systems are shifted by the same amount.

Examples:

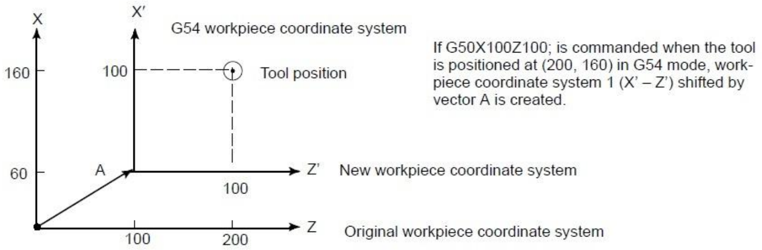


Fig3.16.4 Example1 of Workpiece coordinate System

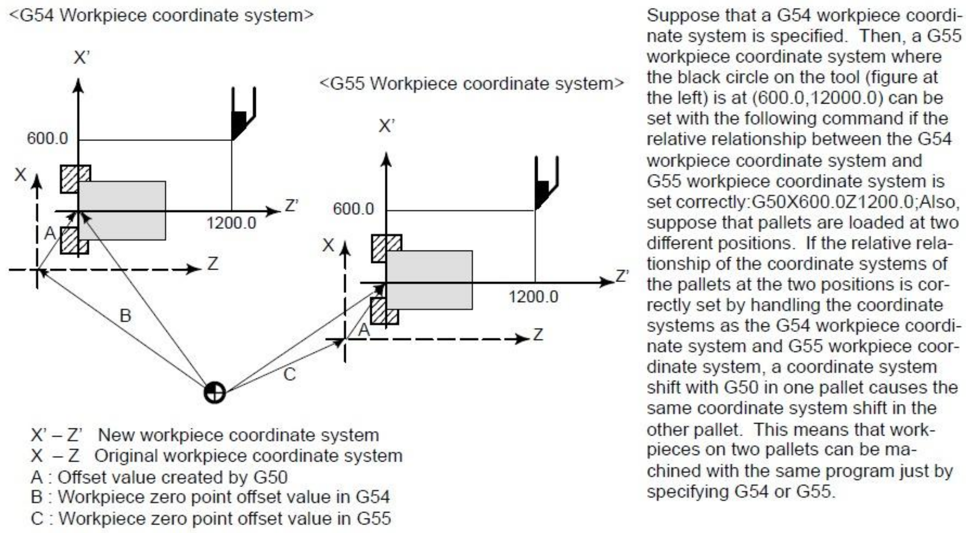


Fig3.16.5 Example2 of Workpiece Coordinate System

3.16.6 Workpiece Coordinate System Shift

When the coordinate system actually set by the G50 command or the automatic system setting deviates from the programmed work system, the set coordinate system can be shifted .

Set the desired shift amount in the work coordinate system shift memory.

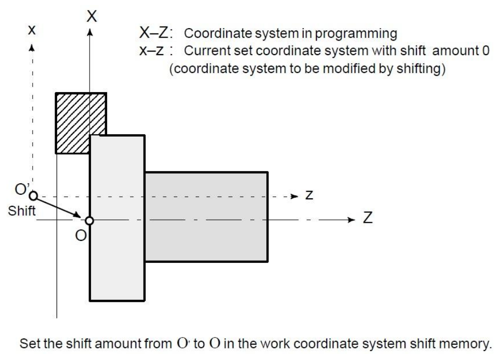


Fig3.16.6 Workpiece Coordinate System Shift

3.16.7 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.

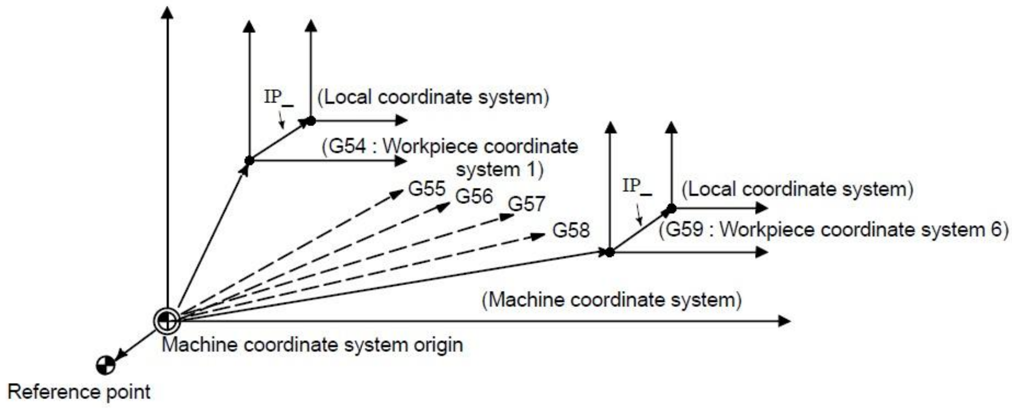


Fig3.16.7 Setting the Local Coordinate System

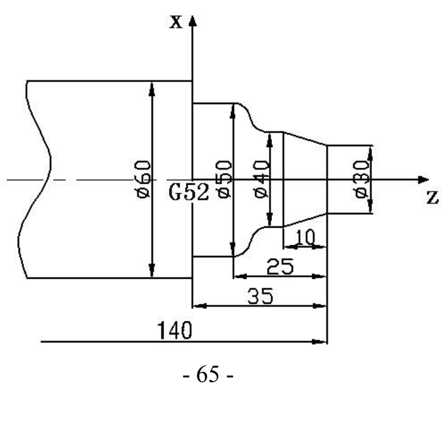
Note:1.The local coordinate system setting does not change the workpiece and machine coordinate systems.

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.

Example1:



N1 G00 X60 Z20
N2 G52 X0 Z-236
N3 T0101
N4 M03 S800 M08
N5 G01 Z35 F100
N6 X-1
N7 X70
N8 G71 U2 R1
N8 G71 P10 Q15 U0.5 W0.5 F150
N10 G01 X30
N11 X40 Z25
N12 Z20
N13 G02 X50 Z15 R5
N14 G03 X60 Z10 R5
N15 G01 Z0
N16 G00 X70
N17 G52 X0 Z0
N18 M05
N19 M30

Example2:Processing 5pieces same workpiece on one workpiece.

G22 L5
X50 Z60; workpiece
......
G52 W-50; each Z-axis relative offset 50mm
G800
G52 X0 Z0; cancel the local coordinate system
G0 X100 Z100
M02

3.17 Constant Surface Speed Control (G96/G97)

Specify the surface speed (relative speed between the tool and workpiece) following S. The spindle is rotated so that the surface speed is constant regardless of the position of the tool.

Format: G96 S<u>OOOO</u> ; Constant surface speed control command
↑ Surface Speed (m/min or feet/min)
G97 S<u>OOOO</u> ; Constant surface speed control cancel command
↑ Spindle Speed (min^{-1})
G50 S_ ; The maximum spindle speed (min^{-1}) follows S.

G96 (constant surface speed control command) is a modal G code. After a G96 command is specified, the program enters the constant surface speed control mode (G96 mode) and specified S values are assumed as a surface speed. A G96 command must specify the axis along which constant surface speed control is applied. A G97 command cancels the G96 mode.When constant surface speed control is applied, a spindle speed higher than the value specified in G50S_; (maximum spindle speed) is clamped at the maximum spindle speed. When the power is turned on, the maximum spindle speed is not yet set and the speed is not clamped. S (surface speed) commands in the G96 mode are assumed as S = 0 (the surface speed is 0) until M03 (rotating the spindle in the positive direction) or M04 (rotating the spindle in the negative direction) appears in the program.

Spindle Speed(min^{-1})

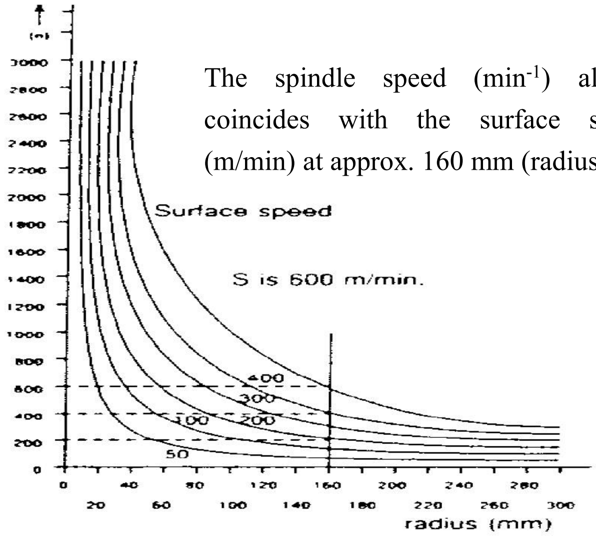


Fig3.17.1 Relation between workpiece radius, spindle speed and surface speed

To execute the constant surface speed control, it is necessary to set a workpiece coordinate system , Z axis, (axis to which the constant surface speed control applies) becomes zero.

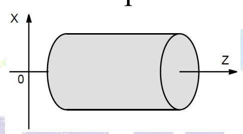
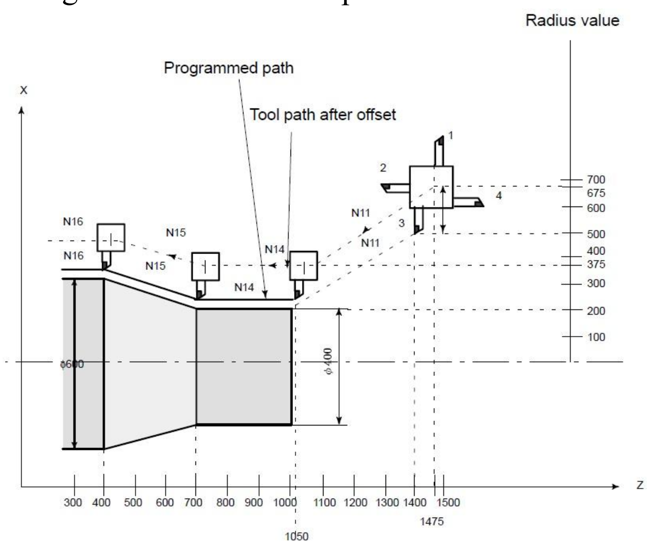


Fig3.17.2 Example of the workpiece coordinate system for constant surface speed control

The constant surface speed control is also effective during threading. Accordingly, it is recommended that the constant surface speed control be invalidated with G97 command before starting the scroll threading and taper threading, because the response problem in the servo system may not be considered when the spindle speed changes.

In a rapid traverse block specified by G00, the constant surface speed control is not made by calculating the surface speed to a transient change of the tool position, but is made by calculating the surface speed based on the position at the end point of the rapid traverse block, on the condition that cutting is not executed at rapid traverse.



Example1:

N8 G00 X1000.0Z1400.0 ;

N9 T33;

N11 X400.0Z1050.0;

N12 G50S3000 ; (Designation of max. spindle speed)

N13 G96S200 ; (Surface speed 200m/min)

N14 G01 Z 700.0F1000 ;

N15 X600.0Z 400.0;

N16 Z ⋯ ;

Fig3.17.3 Example of Constant surface speed Control for G00

The CNC calculates the spindle speed which is proportional to the specified surface speed at the position of the programmed coordinate value on the X axis. This is not the value calculated according to the X axis coordinate after offset when offset is valid. At the end point N15 in the example above, the speed at 600 diameter. (Which is not the turret center but the tool nose) is 200 m/min. If X axis coordinate value is negative, the CNC uses the absolute value.

Press "absolute coordinate" to change the spindle speed in the status of constant linear speed G96.

Note: 1. use constant linear speed, the spindle must be able to change speed automatically. (such as: Servo Spindle,Inverter Spindle).Max SP-speed is set by parameter.

2. Min SP-speed in condition of G96 can be set by P35 in Speed parameter.

3. Spindle override don't work when processing with constant surface speed.

Example2:

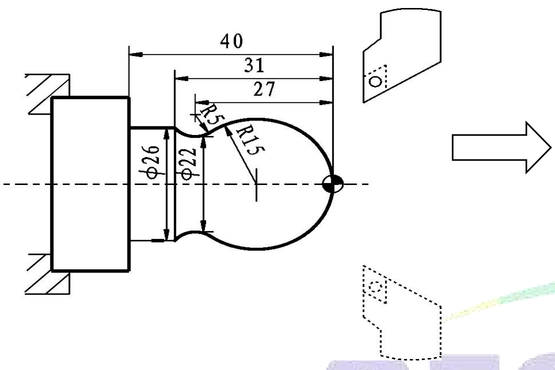


Fig3.17.4 Example2

```
N1 T0102 X40 Z5
N2 M03 S400
N3 G96 S80
N4 G00 X0
N5 G01 Z0 F60
N6 G03 U24 W-24 R15
N7 G02 X26 Z-31 R5
N8 G01 Z-40
N9 X40 Z5
N10 G97 S300
N11 M30
```

3.18 Cutting Feed (G98,G99)

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.

Two modes of specification are available:

1. Feed per minute (G98)

After F, specify the amount of feed of the tool per minute.

2. Feed per revolution (G99)

After F, specify the amount of feed of the tool per spindle revolution.

Format: G98 ; Feed per minute instruction

F_; Feedrate command (mm/min or inch/min)

G99 ; Feed per revolution instruction

F_ ; Feedrate command (mm/rev or inch/rev)

Cutting feed is controlled so that the tangential feedrate is always set at a specified feedrate.

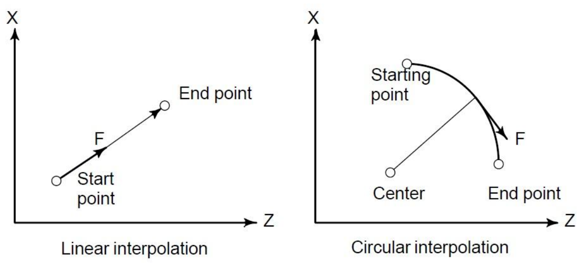


Fig3.18.1 Tangential feedrate (F)

After specifying G98 (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. G98 is a modal code. Once a G98 is specified, it is valid until G99 (feed per revolution) is specified. At power－on, the feed per revolution mode is set.

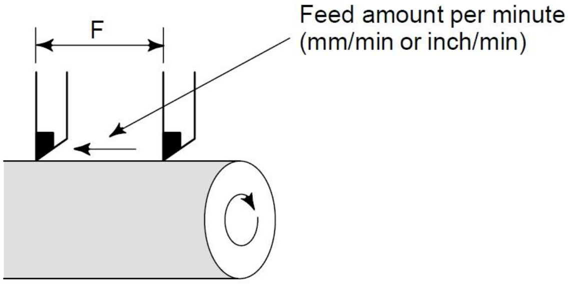


Fig3.18.2 Feed Per Minute

After specifying G99 (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. G99 is a modal code. Once a G99 is specified, it is valid until G98 (feed per minute) is specified.

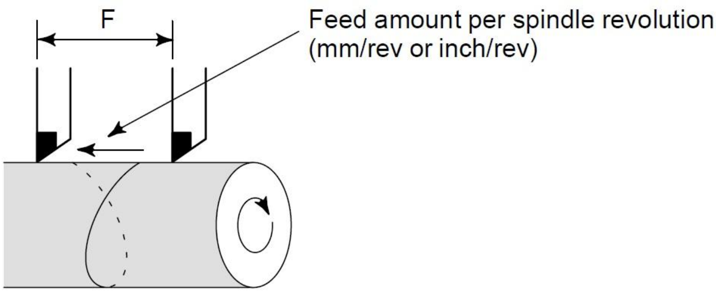


Fig3.18.2 Feed Per Revolution

Note: 1. When using G99 instruction, Spindle must configured with encoder,otherwise system will stay on.

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

3.19 Pole Coordinate Interpolation (G15/G16)

Polar coordinate interpolation is a function that exercises contour control in converting a command programmed in a Cartesian coordinate system to the movement of a linear axis (movement of a tool) and the movement of a rotary axis (rotation of a workpiece). This method is useful in cutting a front surface and grinding a cam shaft on a lathe.

Polar coordinates input directive allows radius and angle in polar coordinates, the angle of the positive Z direction is counterclockwise turned, while the negative direction is a clockwise turn. Radius with absolute command value instruction (Z), the angle with absolute command (X).

G16 IP_ ; Starts polar coordinate interpolation mode (enables polar coordinate interpolation)

...
...
...
Specify linear or circular interpolation using coordinates in a Cartesian coordinate system consisting of a linear axis and rotary axis (virtual axis).

G15 ; Polar coordinate interpolation mode is canceled (for not performing polar coordinate interpolation)

Explanations:

1. Z specify zero as the origin of the polar coordinate system from the point of measuring the radius of the workpiece coordinate system.

2. IP_: Plane selection & home of polar coordinate system

Z_ : Radius of polar coordinate system

X_ : Angle of polar coordinate system

3. Z set to zero as the origin of the polar coordinate system workpiece coordinate system:

By absolute programming instructions a specified radius (distance between the zero point and programming). Zero set workpiece coordinate system for the origin of the polar coordinate system

G16 starts the polar coordinate interpolation mode and selects a polar coordinate interpolation plane(Fig3.19.1).Polar coordinate interpolation is performed on this plane.

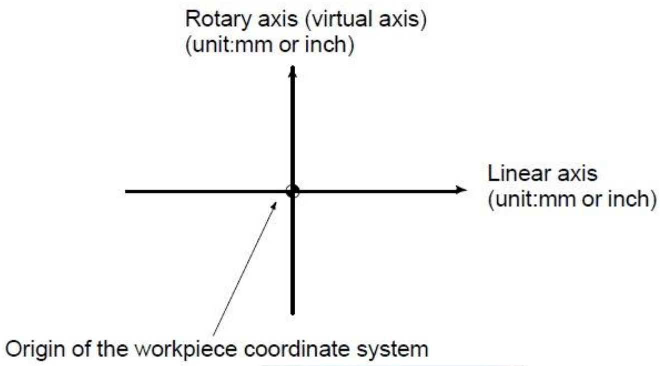


Fig3.19.1 Polar Coordinate Interpolation Plane

When the power is turned on or the system is reset, polar coordinate interpolation is canceled (G15).

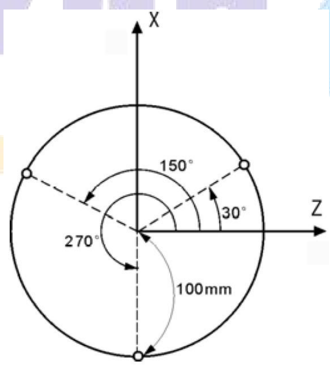


Fig3.19.2 Example of Polar coordinate System

Example: N1 G16 X0 Z0 ; setup polar coordinate

N2 G01 X30.0 Z100.0 F200.0

N3 X150.0

N4 X270.0

N5 G15 ; cancel polar coordinate system

N6 M02

3.20 Absolute and Incremental Programming (G990,G991)

There are two ways to command travels of the tool; the absolute command, and the 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.

Absolute programming or incremental programming is used depending on the command used. See following tables.

G code System	A	B or C
Command method	Address word	G990,G991

G code system A

	Absolute command	**Increment command**
X axis move command Z axis move command C axis move command	X Z C	U W V

G code system B or C

Absolute command	**G990 IP_ ;**
Incremental command	**G991 IP_ ;**

Example: Tool movement from point P to point Q (diameter programming is used for the X-axis)

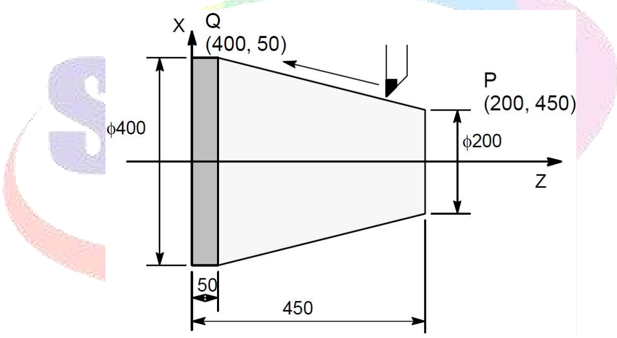


Fig3.20.1 Example of Absolute and Incremental Programming

	G code system A	G code system B or C
Absolute command	X400.0 Z50.0	G90 X400.0 Z50.0
Incremental command	U200.0 W-400.0	G91 X200.0 Z-400.0

Note: 1. Absolute and incremental commands can be used together in a block. In the above example, the following command can be specified : X400.0 W–400.0 ;

2. When both X and U or W and Z are used together in a block, the one specified later is valid.

3. Incremental commands cannot be used when names of the axes are A and B during G code system A is selected.

3.21 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**

Unit explanations for G20/G21

1.Min unit: 0.0001inch(G20/INCH); 0.001mm(G21/Metric)

2.Command unit of G20/INCH for Position command is inch.

3.Command unit of G20/INCH for Feeding Speed (F) command is inch/min.

4.Unit of G20/INCH for Tool Offset(Redeem)/G54-G59 offset value is inch.

5.Unit of G20/INCH for distance type parameter(like max travel,compensation value,...) will be to 0.1inch from mm(G21/Metric) ; & change um(G21/Metric) to 0.0001inch(G20/INCH)

6.Unit of G20/INCH for speed type parameters will be to 0.1inch/min from mm/min(Metric).

7.INCH Unit for accelerate type parameters will be 0.1inch/min/s from mm/min/s(Metric)

8.Handwheel Increment(Rate) will be 0.0001inch(*1), 0.001inch(*10), 0.01inch(*100).

9.Pulse equivalent will be changed when G20/INCH,system will adjust automatically.Electric Ratio is figured out with G21/Metric.

10.Coordinate display of G20/INCH is with 4bits data after dot.

11.Degree unit of G20/INCH is display with 0.1, also 1degree display with 0.1inch.

12.Shift mode between INCH/Metric: run G20/G21 on MDI,reboot CNC.

13. It needs to posit tool again after shift INCH/Metric mode.

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.

5. User can ask SZGH or agent/distributor for parameter display unit on CNC with INCH mode before order.

6. Suggestion that keep one processing mode(Metric/INCH) on one CNC control system.

3.22 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 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:

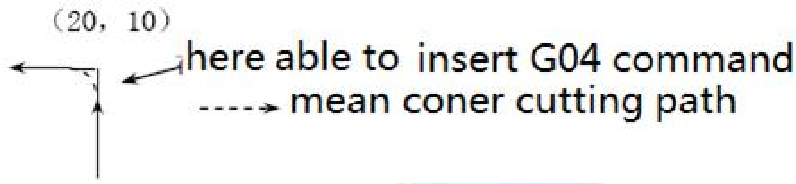


Fig3.22.1 Application of Dwell(G04)

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.

3.23 Positioning/Continuous Path Processing(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.

3.24 Workpiece Position and Move Command

In tool nose radius compensation, the position of the workpiece with respect to the tool must be specified.G40, G41, and, G42 are modal.

G code	Workpiece Position	Tool Path
G40	(Cancel)	Moving along the programmed path
G41	Right side	Moving on the left side the programmed path
G42	Left side	Moving on the right side the programmed path

The tool is offset to the opposite side of the workpiece.

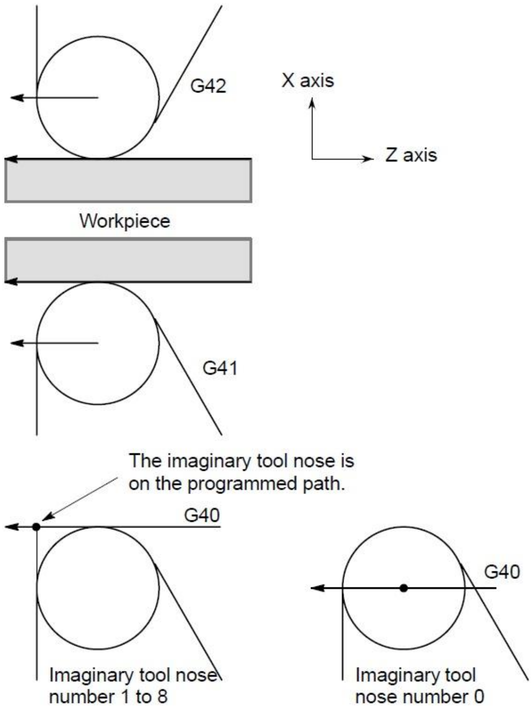


Fig2.24.1 a Workpiece Position of G41/G42

The workpiece position can be changed by setting the coordinate system as shown below.

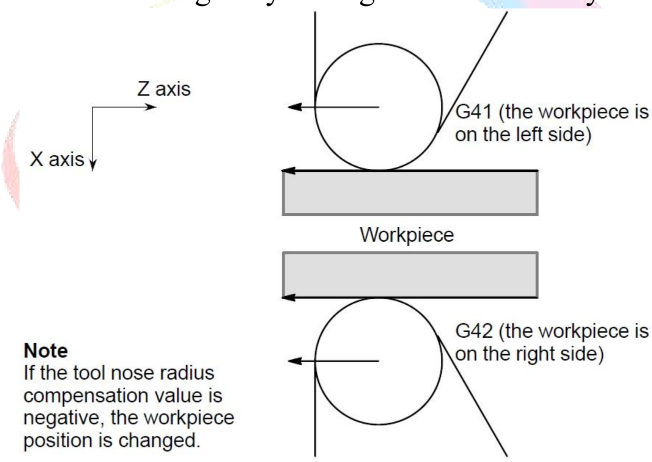


Fig3.24.2 b Workpiece Position of G41/G42

When the tool is moving, the tool nose maintains contact with the workpiece.

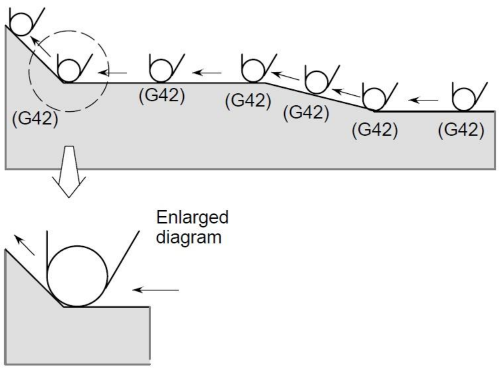


Fig3.24.3 Tool movement when the workpiece position does not change

The workpiece position against the toll changes at the corner of the programmed path as shown in the following figure.

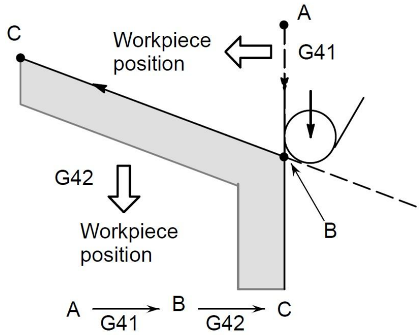


Fig3.24.4 Tool movement when the workpiece position changes

Although the workpiece does not exist on the right side of the programmed path in the above case. the existence of the workpiece is assumed in the movement from A to B. The workpiece position must not be changed in the block next to the start－up block. In the above example, if the block specifying motion from A to B were the start－up block, the tool path would not be the same as the one shown.

The block in which the mode changes to G41 or G42 from G40 is called the start－up block.

G40 _ ;

G41 _ ; (Start-up block)

Transient tool movements for offset are performed in the start–up block. In the block after the start-up block, the tool nose center is positioned Vertically to the programmed path of that block at the start position.

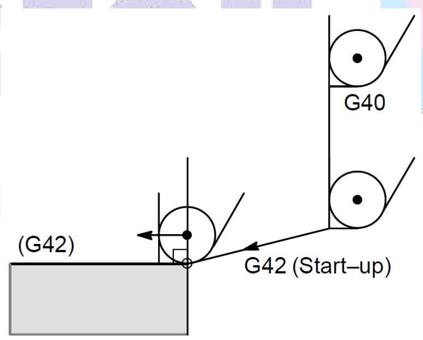


Fig3.24.5 Start-Up

The block in which the mode changes to G40 from G41 or G42 is called the offset cancel block.

G41 _ ;

G40 _ ; (Offset cancel block)

The tool nose center moves to a position vertical to the programmed path in the block before the cancel block. The tool is positioned at the end position in the offset cancel block (G40) as shown below.

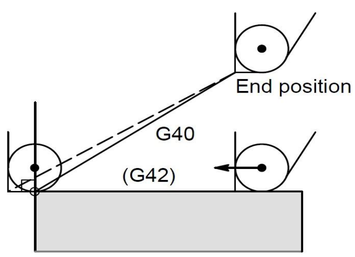


Fig3.24.6 Offset Cancel

When is specified again in G41/G42 mode , the tool nose center is positioned vertical to the programmed path of the preceding block at the end position of the preceding block.

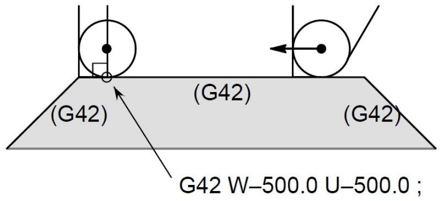


Fig3.24.7 Specification of G41/G42 in G41/G42 mode

In the block that first specifies G41/G42, the above positioning of the tool nose center is not performed.

When you wish to retract the tool in the direction specified by X(U) and Z(W) , cancel the tool nose radius compensation at the end of machining the first block in the figure below, specify the following :

G40 X(U) _ Z(W) _ I _ K _ ;

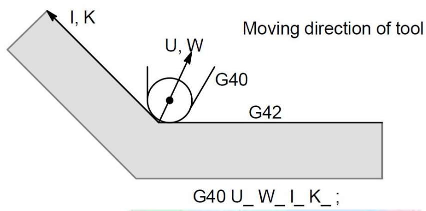


Fig3.24.8 Tool movement when the moving direction of the tool in a block

The workpiece position specified by addresses I and K is the same as that in the preceding block. If I and/or K is specified with G40 in the cancel mode, the I and/or K is ignored.

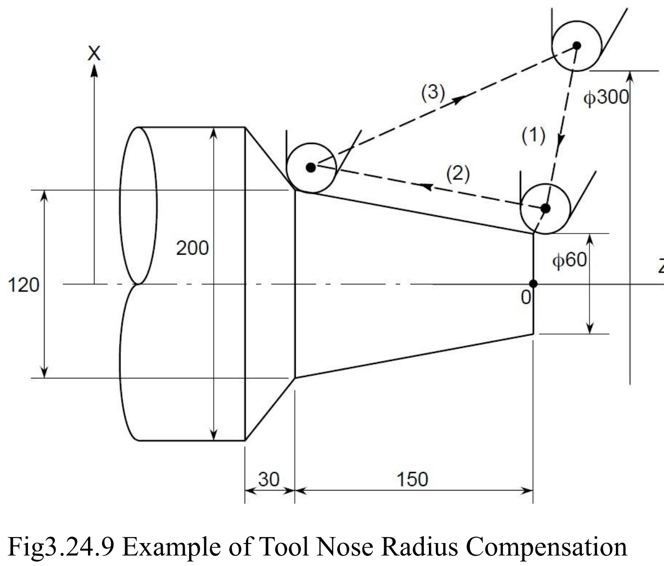
G40 X_ Z_ I_ K_;	Tool nose radius compensation
G40 G02 X_ Z_ I_ K_;	Circular interpolation

G40 G01 X_ Z_ ;

G40 G01 X_ Z_ I_ K_ ; Offset cancel mode (I and k are ineffective.)

The numeral s followed I and K should always be specified as radius values.

Example



(G40 mode)
1.G42 G00 X60.0 ;
2.G01 X120.0 W–150.0 F10 ;
3.G40 G00 X300.0 W150.0 I40.0 K–30.0 ;

Fig3.24.9 Example of Tool Nose Radius Compensation

Tool nose radius compensation with G90 (outer diameter/internal diameter cutting cycle) or G94 (end face turning cycle) is as follows, :

1. Motion for imaginary tool nose numbers

For each path in the cycle, the tool nose center path is generally parallel to the programmed path.

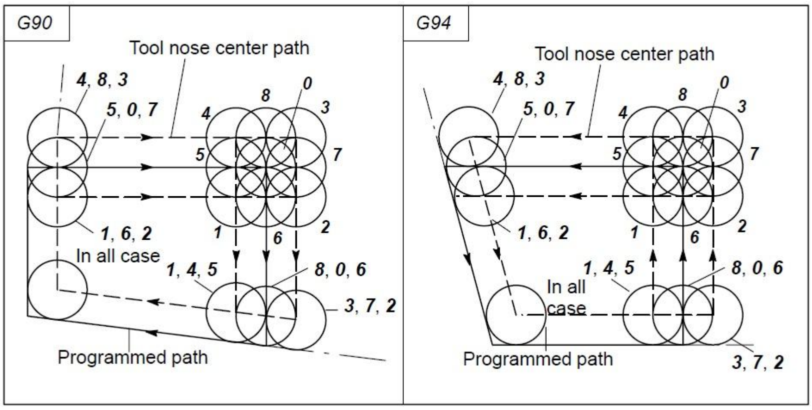


Fig3.24.10 Motion for imaginary tool nose numbers

2. Direction of the offset

The offset direction is indicated in the figure below regardless of the G41/G42 mode.

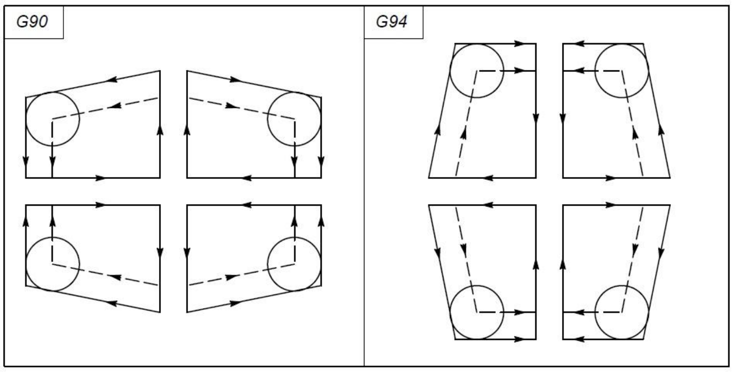
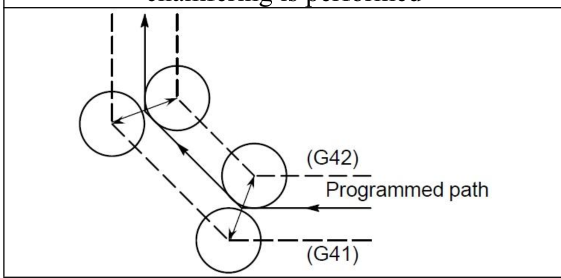
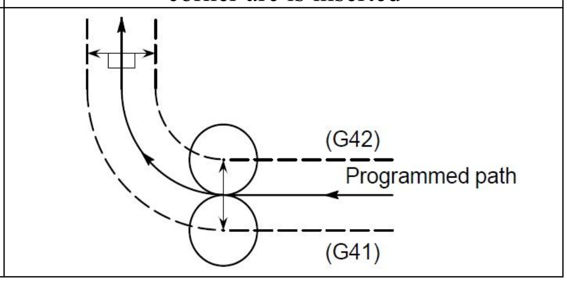


Fig3.24.11 Direction of the offset

Movement after compensation is shown below.

Tool nose radius compensation when chamfering is performed	Tool nose radius compensation when a corner arc is inserted
	

3.25 Details of Tool Nose Radius Compensation (G40/G41/G42)

The tool nose radius center offset vector is a two dimensional vector equal to the offset value specified in a T code, and the is calculated in the CNC.

Its dimension changes block by block according to tool movement. This offset vector (simply called vector herein after) is internally crated by the control unit as required for proper offsetting and to calculate a tool path with exact offset (by tool nose radius) from the programmed path.

This vector is deleted by resetting.

The vector always accompanies the tool as the tool advances.

Proper understanding of vector is essential to accurate programming.

Read the description below on how vectors are created carefully.

G40, G41 or G42 is used to delete or generate vectors.

These codes are used together with G00, G02, or G32 to specify a mode for tool motion (Offsetting).

G code	Function	Workpiece Position
G40	Tool nose radius compensation cancel	Neither
G41	Left offset along tool path	Left
G42	Right offset along tool path	Right

G41 and G42 specify an off mode, while G40 specifies cancellation of the offset.

The system enters the cancel mode immediately after the power is turned on, when the RESET button on the MDI is pushed or a program is forced to end by executing M02 or M30. (the system may not enter the cancel mode depending on the machine tool.) In the cancel mode, the vector is set to zero, and the path of the center of tool nose coincides with the programmed, path. A program must end in cancel mode. If it ends in the offset mode, the tool cannot be positioned at the end point, and the tool stops at a location the vector length away from the end point.

Note: 1. G40/G41/G42 are modal codes, can cancel each other.

2. G41/G42 without parameters, the compensation number (on behalf of the tool tip radius compensation corresponding values) specified by the T code. Its tip arc offset number and tool offset number corresponding compensation.Establish and canceled .

3. Tool nose radius compensation with G00 or G01 instruction only, not the G02 or G03.Nose radius compensation tool compensation interface "radius compensation", the definition of the turning radius; imaginary tool nose number defines the direction of the tip.

The imaginary tool nose tip number defines the positional relationship between the tool and cutter tip arc center point, which is from 0-9 ten directions, as shown above.

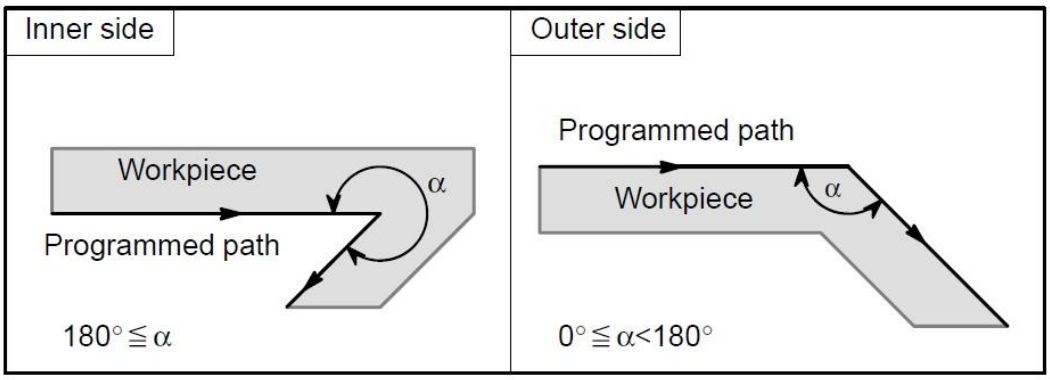
Note: For the definition of the edge position code "Posit tip" .

3.26 Tool Nose Radius Compensation of Offset C

C means the system calculates the tool trajectory of radius compensation according to the last program line and the next program line.

1) Inner side and outer side

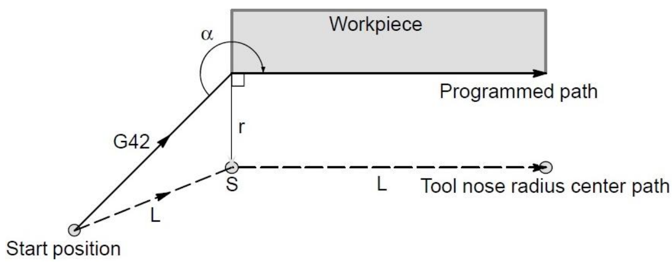
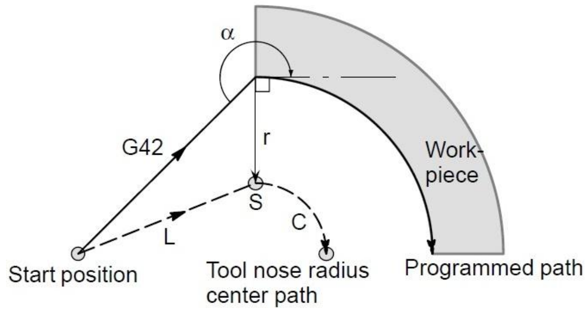
When an angle of intersection created by tool paths specified with move commands for two blocks is over 180º, it is referred to as “inner side.” When the angle is between 0º and 180º , it is referred to as “outer side.”



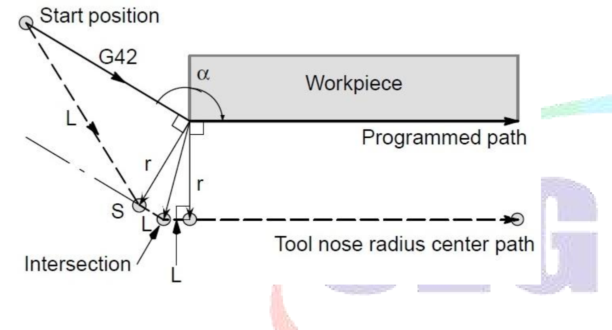
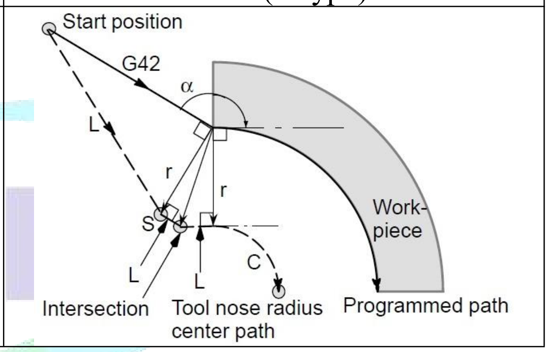
2) Tool Movement in Start-up

When the offset cancel mode is changed to offset mode, the tool moves as illustrated below (start-up):

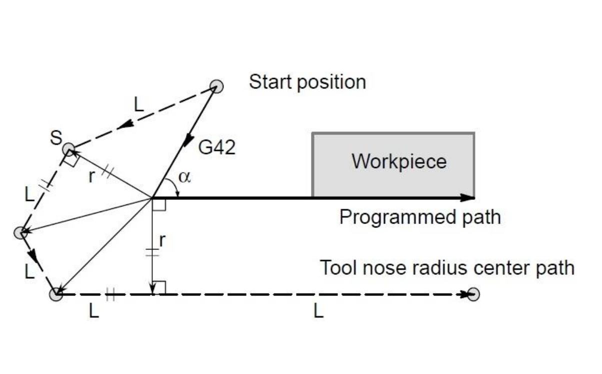
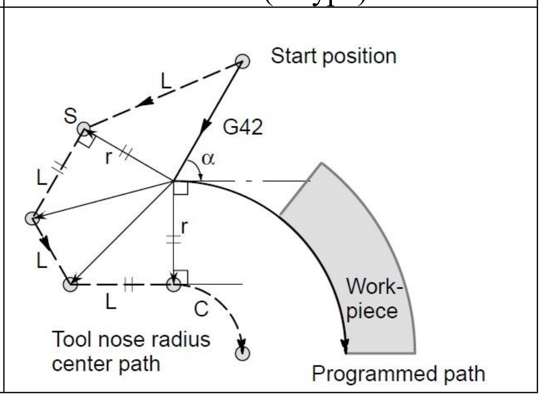
(a) Tool movement around an inner side of a corner (α≥180º)

Linear ⟶ Linear	Linear ⟶ Circular
	

(b) Tool movement around the outside of a corner at obtuse angle (90≤α＜180)

Linear ⟶ Linear (B type)	Linear ⟶ Circular (B type)
	

(c)Tool movement around the outside of an acute angle (α＜90º)

Linear ⟶ Linear (B type)	Linear ⟶ Circular (B type)
	

d)Tool movement around outside linear→linear at an acute angle less than 1 degree (α<1º)

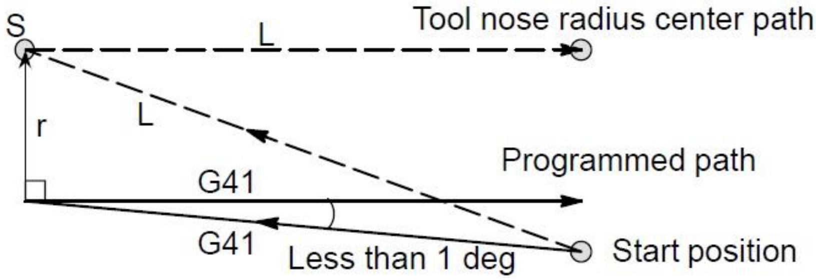


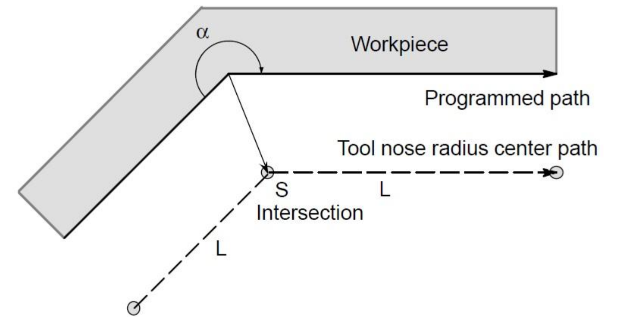
Fig3.26.1 Tool Movement at angle less than 1º

3) Tool Movement in Offset Mode

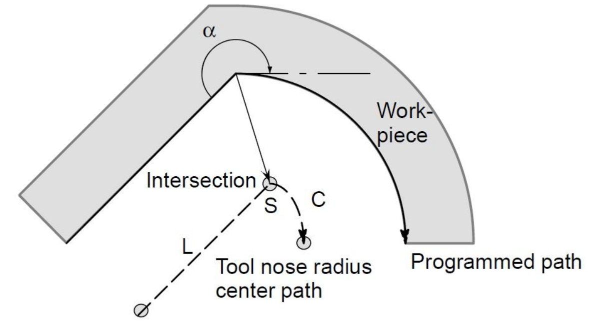
In the offset mode, the tool moves as illustrated below:

(a)Tool movement around the inside of a corner (180º≤α)

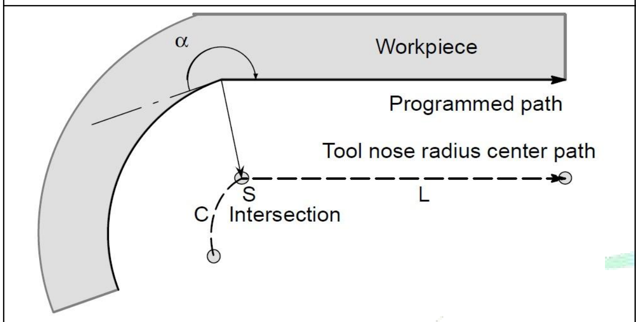
Linear ⟶ Linear



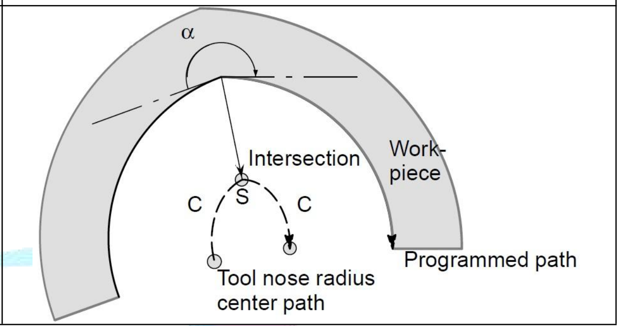
Linear ⟶ Circular



Circular ⟶ Linear



Circular ⟶ Circular



(b)Tool movement around the inside (α<1º) with an abnormally long vector, linear → linear

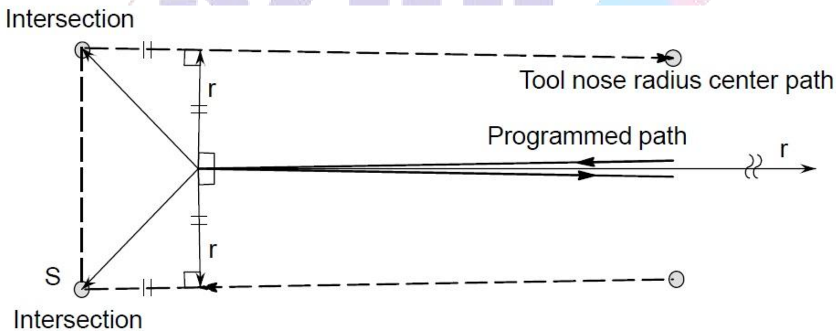
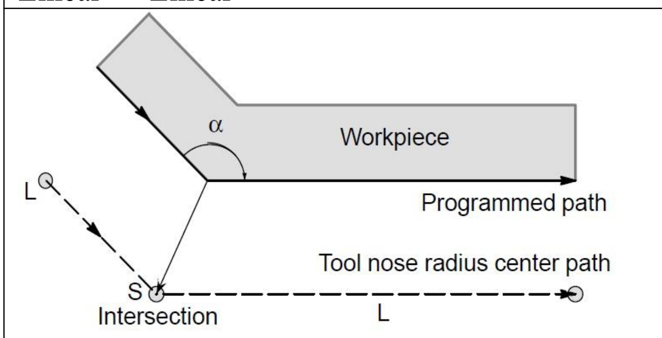


Fig3.26.2 Tool Movement at angle less than 1º

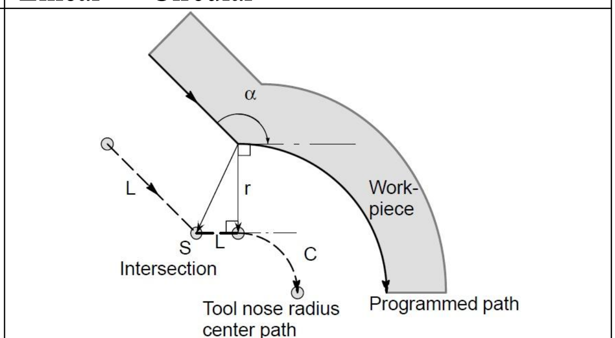
Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

(c)Tool movement around the outside corner at an obtuse angle (90≤α＜180º)

Linear ⟶ Linear

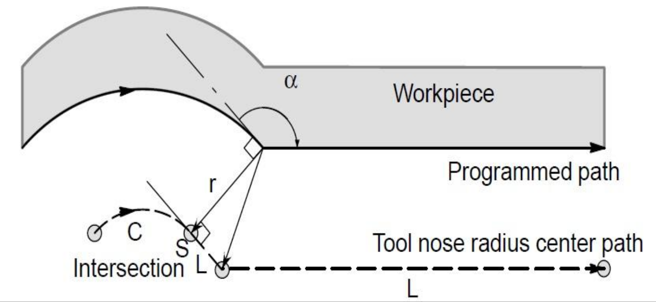


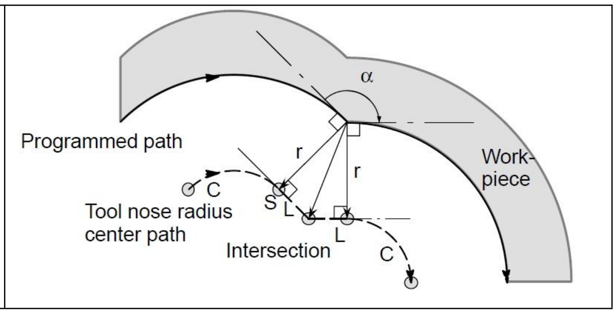
Linear ⟶ Circular



Circular ⟶ Linear

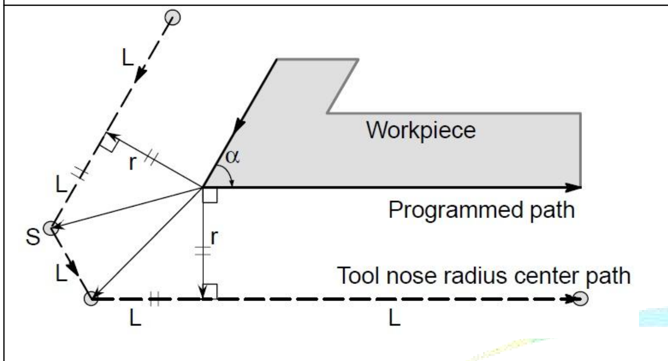
Circular ⟶ Circular



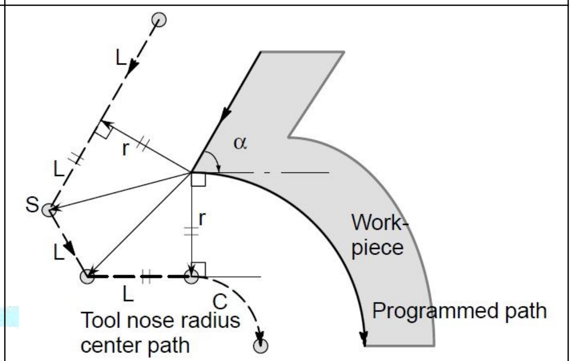


(d)Tool movement around the outside corner at an acute angle (α＜90º)

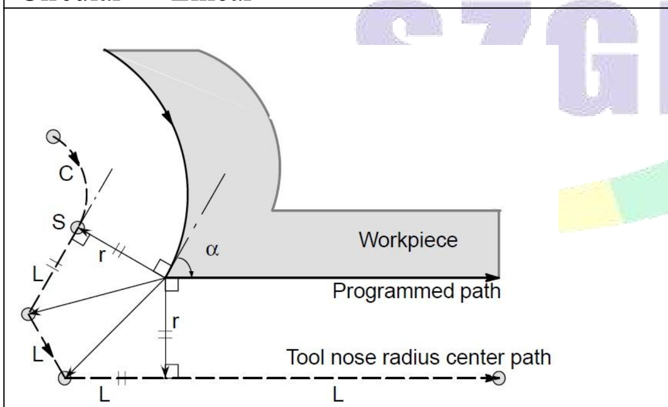
Linear → Linear



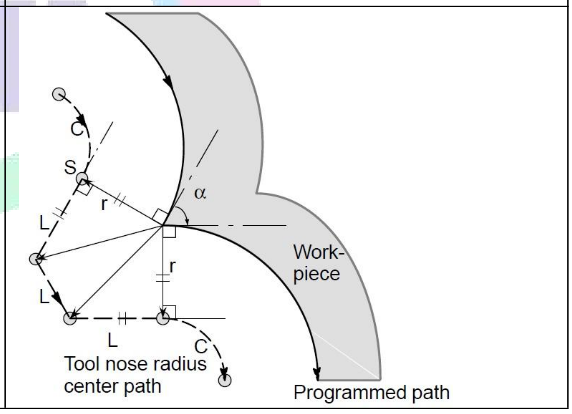
Linear → Circular



Circular → Linear



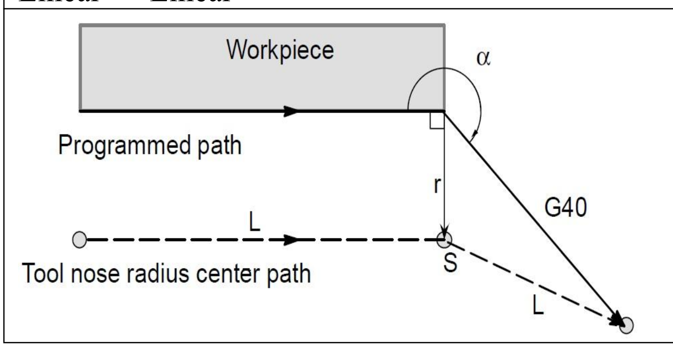
Circular → Circular



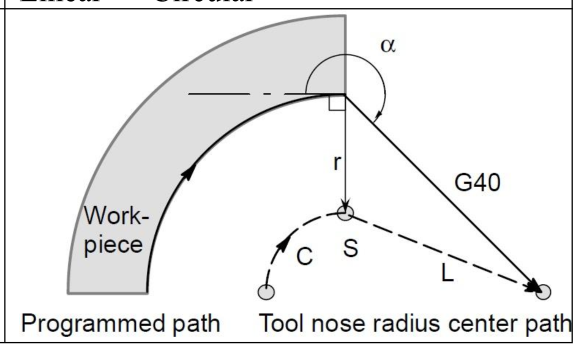
4) Tool Movement in Offset Mode Cancel

(a) Tool movement around an inside corner (180º≤α)

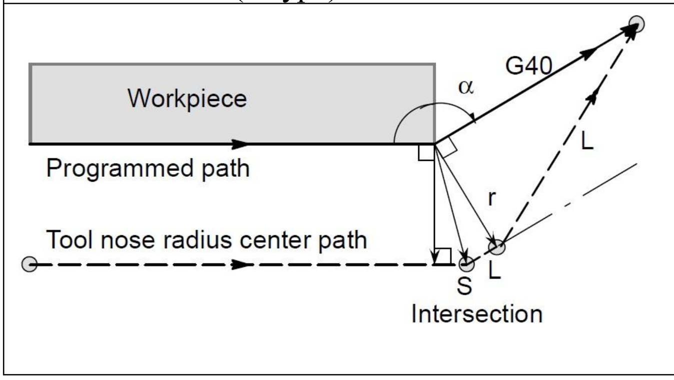
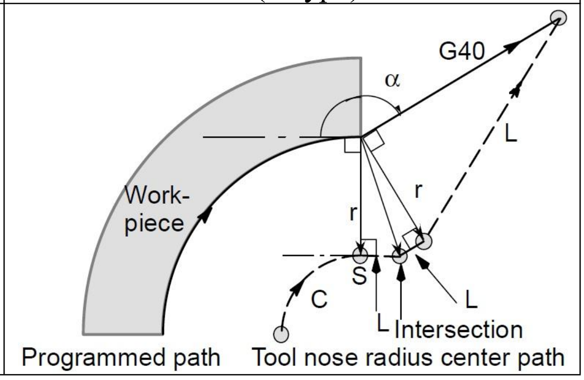
Linear → Linear



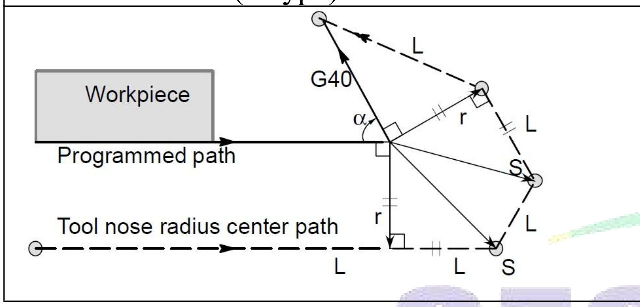
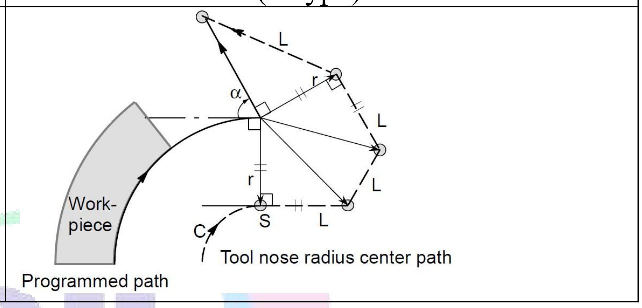
Linear → Circular



(b) Tool Movement around an outside corner at obtuse angle (90≤α＜180º)

Linear ⟶ Linear(B type)	Linear ⟶ Circular(B type)
	

(c) Tool Movement around an outside corner at acute angle (α＜90º)

Linear ⟶ Linear(B type)	Circular ⟶ Linear(B type)
	

(d) Tool movement around outside linear→linear at an acute angle less than 1 degree(α＜1º)

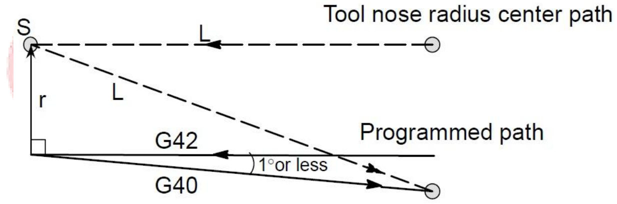


Fig3.26.3 Tool Movement at angle less than 1

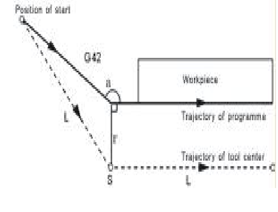
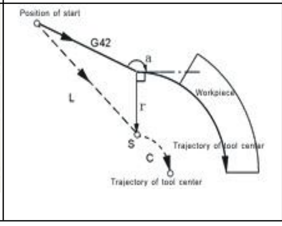
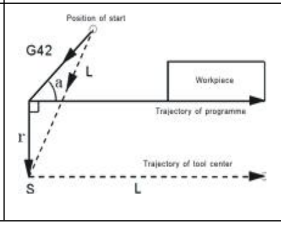
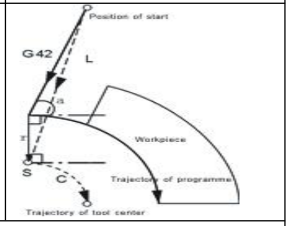
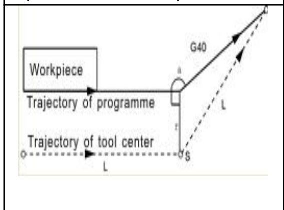
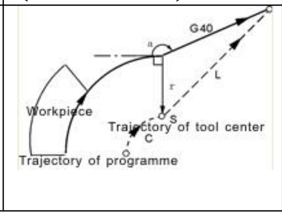
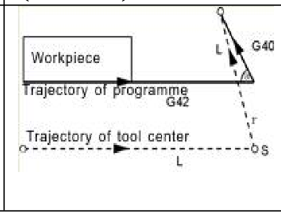
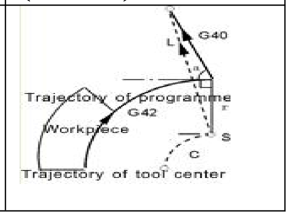
5) Change in the offset direction in the offset mode

The offset direction is decided by G codes (G41 and G42) for tool nose radius and the sign of tool nose radius compensation value as follows.

G code \ Sign of offset value	+	-
G41	Left side offset	Right side offset
G42	Right side offset	Left side offset

The offset direction can be changed in the offset mode. If the offset direction is changed in a block, a vector is generated at the intersection of the tool nose radius center path of that block and the tool nose radius center path of a preceding block. However, the change is not available in the start-up block and the block following it.

6) A Type of Tool Nose Compensation

<table>
<tr><td colspan="4">Tool Movement in Start-up</td></tr>
<tr><td>(90≤α＜180)</td><td>(90≤α＜180)</td><td>(α＜90º)</td><td>(α＜90º)</td></tr>
<tr><td>
</td><td>
</td><td>
</td><td>
</td></tr>
<tr><td colspan="4">Tool Movement in Offset Mode Cancel</td></tr>
<tr><td>(90≤α＜180)</td><td>(90≤α＜180)</td><td>(α＜90º)</td><td>(α＜90º)</td></tr>
<tr><td>
</td><td>
</td><td>
</td><td>
</td></tr>
</table>

P41 in Speed is set for A type , B type and other type for Tool Nose Radius Compensation of Offset C.

3.27 Automatic beveling (I) and smoothing(R)

Format:

Automatic Beveling

G01(G00) X(U)_ I_ ;
G01(G00) Z(W)_ } **Automatic Beveling**

G01(G00) Z(W)_ I_;
G01(G00) X(U)_ } **Automatic Beveling**

Automatic Smoothing

G01(G00) X(U)_ R_ ;
G01(G00) Z(W)_ } **Automatic Smoothing**

G01(G00) Z(W)_ R_;
G01(G00) X(U)_ } **Automatic Smoothing**

Note:

1.The address of I and R are specified with radius model. The running distance of this line and the next line must be greater than the length of beveling or radius of smoothing, otherwise the system will decrease the length of beveling or radius of smoothing to minimal running distance of this line and the next line automatically.

2. The two adjacent lines must be 90 degrees.

For example:

```
0 G54 G0 X-50 Y-50 Z20
N1 M03 S500
N2 G01 G42 D01 X0 Y0 F200
N3 G01 Z-5
N4 X100 I4                      ;Beveling4x4
N5 Y40 R6                       ;SmoothingR6
N6 X47 R5                       ;SmoothingR5
N7 Y70 I3                       ;Beveling3x3
N8 X15
N9 X0 Y40
N10 Y0
N11 G0 X-50 Y-50 G40
N12 Z50
N13 M30
```

3.28 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
```

3.29 Macro program instruction(G65/G66/G67)

3.29.1 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(#**) , Transforming 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-#155&#190-#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 G65 P8000 L1 A5 B6 G0 X0 Z0 M30	N1 #2=#0+#1 N2 IF （#2 EQ 10）GOTO 4 N3 GOO X#2 N4 G00 Z#1 N5 M99 ;Return

3.29.2 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

3.29.3 Macro Program Instruction

3.29.3.1 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;

3.29.3.2 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;

3.29.3.3 Assignment Instruction: =

Explanation: used for assignment of a variable
Eg.: #251=890.34 #450=#123
And also it could be mathematical expression , eg.: #440=#234+#470

3.29.3.4 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

3.29.3.5 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

3.29.3.6 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 is not 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
```

3.29.4 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

3.29.5 Arithmetic & Logic Operation

Table:

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

3.29.6 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.

3.29.7 Global Variable

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

When power is off, the variable #21--#100 is initialized to zero, the variable #101--#600 data is saved not to loss even if the power is off.

3.29.8 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 G76 (threading cycle code)

User can use global variable: #31~#100 & #205~#999,the others are used already on CNC system.

3.29.9 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(C)-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)

3.29.10 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
#1805: X40-X47 (D0-D7) ; input resistor
#1806: X60-X67 (D0-D7) ; input resistor
#1807: X74-X81 (D0-D7) ; Alarm of driver/Spindle
#1808: Y00-Y15 (D0-D15) ; output resistor
#1809: Y16-Y31 (D0-D15) ; output resistor
#1810: Y32-Y47 (D0-D15) ; output resistor

Warning:

1. Macro variables #100-#155&#190-#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.

3.29.11 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.

3.29.12 Build Processing Program Automatically

3.29.12.1 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”

3.29.12.2 Close program

Format: FILECE

It means that close current opening program, if without this code, system will close current program after program is finished.

3.29.12.3 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.

3.29.12.4 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 Complex function for Turning & Milling

CNC lathe system can finish milling processing, solutions is as follow:

1.Parameter sets: Name of 3rd axis should be set to “Y” axis, which is set by P101 in Axis parameter; programming mode is radius programming.

2.Related commands: The Others are same except following commands:

Code	Same functions in milling system	Code	Same functions in milling system
G990	G90 in milling system	G981	G81 in milling system
G991	G91 in milling system	G982	G82 in milling system
G994	G94 in milling system	G983	G83 in milling system
G995	G95 in milling system	G984	G84 in milling system
G998	G98 in milling system	G985	G85 in milling system
G999	G99 in milling system	G986	G86 in milling system
G973	G73 in milling system	G987	G87 in milling system
G974	G74 in milling system	G989	G89 in milling system
G976	G76 in milling system		

Note: CNC lathe system have these functions when CNC system is configured with 3rd axis.

3.31 Polar Coordinate Interpolation(G12.1/G13.1)

Polar coordinate interpolation is a function that exercises contour control in converting a command programmed in a Cartesian coordinate system to the movement of a linear axis (movement of a tool) and the movement of a rotary axis (rotation of a workpiece). This method is useful in cutting a front surface and grinding a cam shaft on a lathe.

Format:

G12.1 ; Starts polar coordinate interpolation mode (enables polar coordinate interpolation)

Specify linear or circular interpolation using coordinates in a Cartesian coordinate system consisting of a linear axis and rotary axis (virtual axis).

G13.1 ; Polar coordinate interpolation mode is canceled (for not performing polar coordinate interpolation)

Note: Specify G12.1 and G13.1 in Separate Blocks.

G12.1 starts the polar coordinate interpolation mode and selects a polar coordinate interpolation plane, G17, (Fig3.20). Polar coordinate interpolation is performed on this plane.

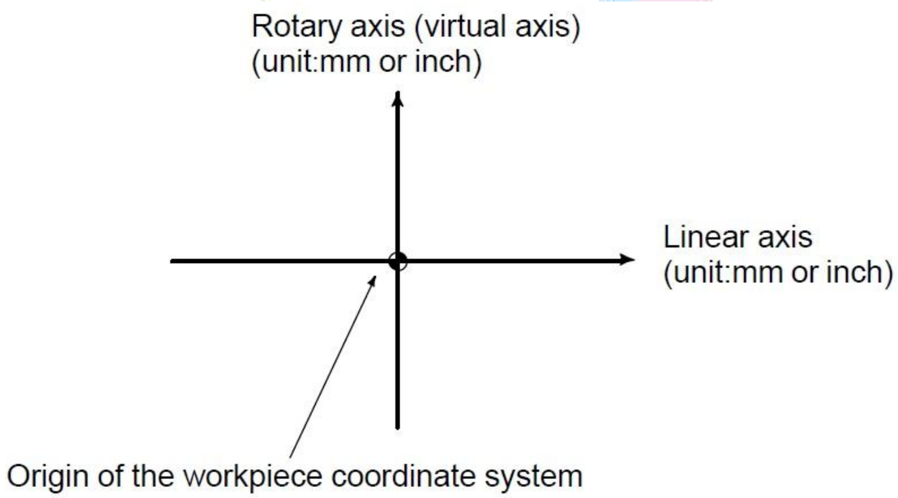


Fig3.20 Polar coordinate interpolation plane(G17)

Performed Steps of Polar coordinate interpolation: Polar coordinate interpolation program, which is based on X axis (Linear axis) & C axis(Rotary axis)

The linear and rotation axes for polar coordinate interpolation must be set in parameters (P102) beforehand.

Note: 1.When the power is turned on or the system is reset, polar coordinate interpolation is canceled (G13.1). G12.1 & G13.1 are mode codes.

2.G codes which can be specified in the polar coordinate interpolation mode

G01: linear interpolation

G02,G03 : Circular interpolation

G04: Dwell

G40,G41,G42: Tool nose radius compensation

(Polar coordinate interpolation is applied to the path after cutter compensation.)

3. When G12.1,start polar coordinate interpolation, system will shift to G17 plane automatically; When running G13.1, cancel polar coordinate interpolation, system will return to G18 plane automatically.

4. Programming mode: X-axis: Diameter/Radius programming; C-axis: Radius programming. Even when diameter programming is used for the linear axis (X-axis), radius programming is applied to the rotary axis (C-axis). Programming unit is mm, displaying unit is degree.

5. In the polar coordinate interpolation mode, program commands are specified with Cartesian coordinates on the polar coordinate interpolation plane. The axis address for the rotation axis is used as the axis address for the second axis (virtual axis) in the plane. Whether a diameter or radius is specified for the first axis in the plane is the same as for the rotation axis regardless of the specification for the first axis in the plane.

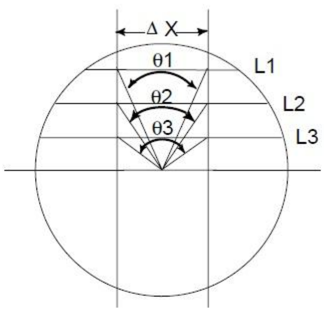
6. The virtual axis is at coordinate 0 immediately after G12.1 is specified. Polar interpolation is started assuming the angle of 0 for the position of the tool when G12.1 is specified.

7. Specify the feedrate as a speed (relative speed between the workpiece and tool) tangential to the polar coordinate interpolation plane (Cartesian coordinate system) using F.

8. The addresses for specifying the radius of an arc for circular interpolation (G02 or G03) in the polar coordinate interpolation plane depend on the first axis in the plane (linear axis). I and J in the Xp－Yp plane when the linear axis is the X－axis or an axis parallel to the X－axis.The radius of an arc can be specified also with an R command.

9.Polar coordinate interpolation converts the tool movement for a figure programmed in a Cartesian coordinate system to the tool movement in the rotation axis (C-axis) and the linear axis (X-axis). When the tool moves closer to the center of the workpiece, the C-axis component of the feedrate becomes larger and may exceed the maximum cutting feedrate for the C-axis (set by P109 in Axis parameter), causing an alarm (see the figure below). To prevent the C-axis component from exceeding the maximum cutting feedrate for the C-axis, reduce the feedrate specified with address F or create a program so that the tool (center of the tool when tool nose radius compensation is applied) does not move close to the center of the workpiece.

WARNING



Consider lines L1, L2, and L3. ΔX is the distance the tool moves per time unit at the feedrate specified with address F in the Cartesian coordinate system. As the tool moves from L1 to L2 to L3, the angle at which the tool moves per time unit corresponding to ΔX in the Cartesian coordinate system increases from θ1 to θ2 to θ3.

In other words, the C–axis component of the feedrate becomes larger as the tool moves closer to the center of the workpiece. The C component of the feedrate may exceed the maximum cutting feedrate for the C–axis because the tool movement in the Cartesian coordinate system has been converted to the tool movement for the C–axis and the X–axis.

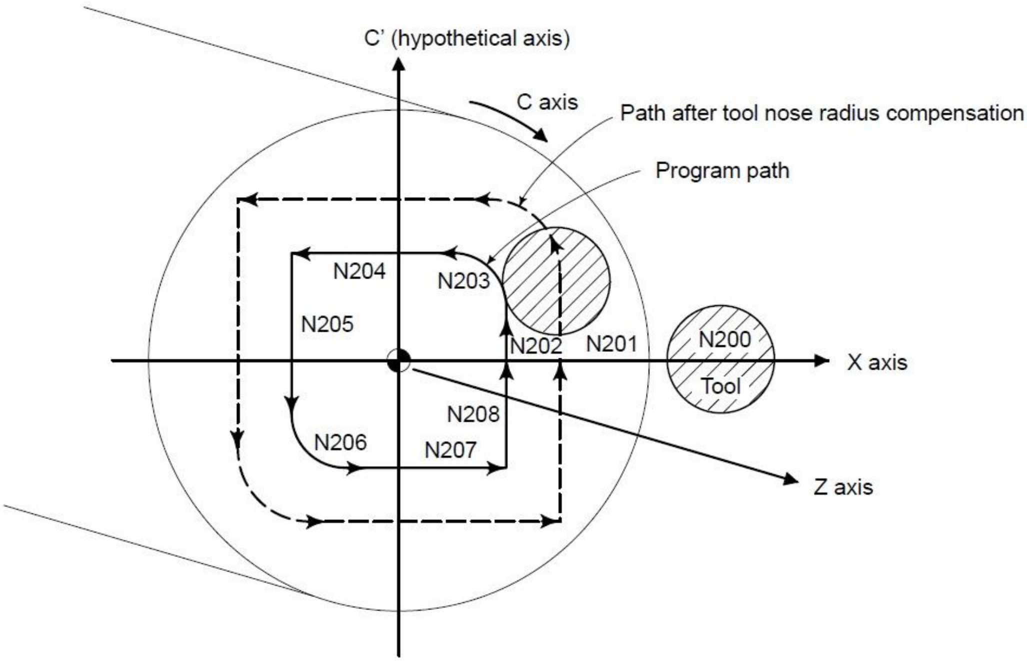
L : Distance (in mm) between the tool center and workpiece center when the tool center is the nearest to the workpiece center

R :Maximum cutting feedrate (deg/min) of the C axis

Then, a speed specifiable with address F in polar coordinate interpolation can be given by the formula below. Specify a speed allowed by the formula. The formula provides a theoretical value; in practice, a value slightly smaller than a theoretical value may need to be used due to a calculation error.

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

Example: Polar Coordinate Interpolation Program Based on X Axis (Linear Axis) and C Axis (Rotary Axis)



X axis is by diameter programming, C axis is by radius programming

N10 T0202
...
N100 G00 X150 C0 Z0 ; Positioning to start position
N200 G12.1 ; Start of polar coordinate interpolation
N201 G42 G01 X40.0 F20 ;
N202 C10 ;
N203 G03 X-20.0 C20.0 R10.0;
N204 G01 X-40.0;
N205 C-10.0;
N206 G03 X-40 C-20 R20 ;
N207 G01 X40 ;
N208 G03 X80 C-20 R20 ;
N209 G01 C0 ;
N210 X150.0 ;
N300 G13.1; Cancellation of polar coordinate interpolation
N400 Z100.0 C0 ;
...
N500 M30 ;

(N201–N210:) Geometry program (Program based on Cartesian coordinate on X-C' plane)

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.

M □□
→ Instruction value(the leading zero can be omitted)
→ Instruction address

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 workpiece 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.

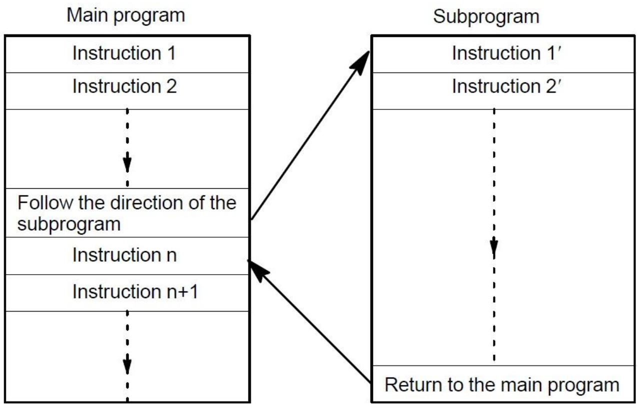


Fig4.2.1 Main program and subprogram

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;

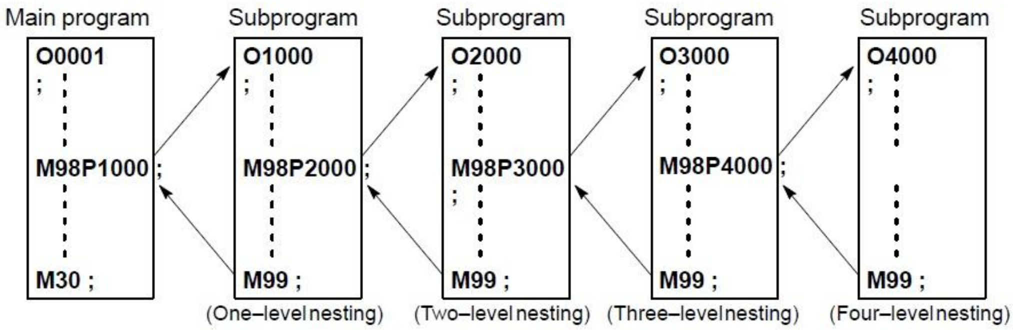
2) M99 with P in main program is same to M97 ;

3) M99 in subprogram return to next block of M98;

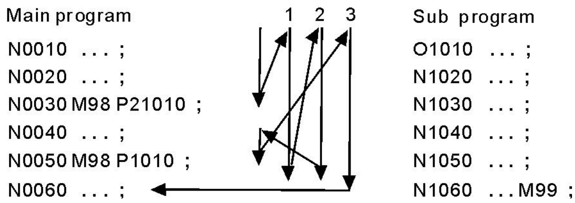
4) M99 with P in the subroutine return to specified block in main program.

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	
Tailstock	M79	Tailstock Forward	Functions interlocked and states reserved
	M78	Tailstock Backward	
Lubricate	M32	Lubrication ON	Functions interlocked and states reserved
	M33	Lubrication OFF	
Huff	M59	Huff ON	Functions interlocked and states reserved
	M58	Huff OFF	
1	M61	User-define Output 1 ON	Functions interlocked and states reserved
	M60	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 switch 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.

M71/M70: When P20 in Other parameter set to 1, M10 output M71.

M73/M72: When P21 in Other parameter set to 1, M78 output M73.

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	2nd Spindle on CW	***Note: Standard plc ladder is without these output for 2nd spindle,if need,we need to alter plc ladder.***
M204	2nd Spindle on CCW	
M205	2nd Spindle Stop	

Wiring Diagram

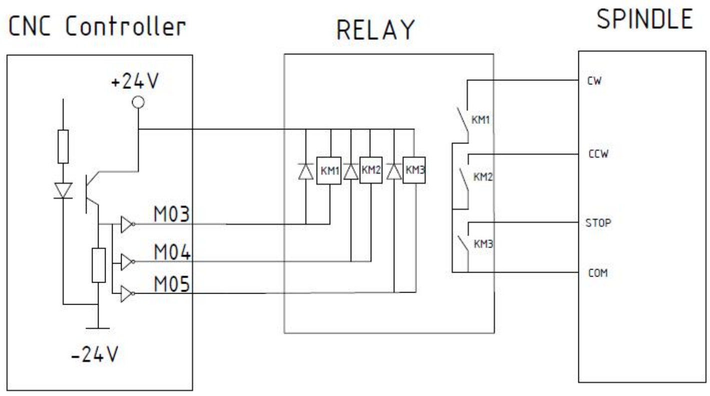


Fig4.3.1 Wiring Diagram for Spindle

According to this wiring diagram, it will consist 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 others inductance load, must 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, 1means 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.

P36: To set the max speed of spindle, also the speed of corresponding 10V.

Attention: when spindle system is with gears,this is the speed of first gear.

P37: To set the max speed of spindle(second gear), that's the turning speed of corresponding 10V instruction voltage.Unit: rpm.

P38: To set the max speed of spindle (Third gear), that's the turning speed of corresponding 10V instruction voltage. Unit: rpm.

P39: To set the max speed of spindle (Fourth gear), that's the turning speed of corresponding 10V instruction voltage. Unit: rpm.

P40: To set the highest speed of second spindle, also the speed of corresponding 10V. Unit: r/min

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.

P36 in Speed parameter is set for max speed of 1st class spindle;

P37 in Speed parameter is set for max speed of 2nd class spindle;

P38 in Speed parameter is set for max speed of 3rd class spindle;

P39 in Speed parameter is set for max speed of 4th class 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.

P15: Detect position of clamping/unclamping of chuck,0:No detect, 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*.

4.3.1.6 Tailstock(M79/M78)

M79: Tailstock forward

M78: Tailstock backward

Remark	PIN	Function
M79	PIN22_CN3 Plug	Tailstock Forward/Backward
M73	PIN22_CN10 Plug	Control chuck unclamping(Spare)
M18	PIN11_CN10 Plug	Detect position of forward_Tailstock(spare)
M28	PIN24_CN10 Plug	Detect position of backward_Tailstock(spare)
M14	PIN24_CN10 Plug	Input point for switch to control Tailstock(spare)

Parameters set for tailstock,in Other parameter:

P16: Detect Position of Forward/Backward of Tailstock; 0: no detect,1:Yes;

M18: input point for detecting position of forward of tailstock;

M28: input point for detecting position of backward of tailstock.

P21: Type of controlling signal for tailstock,0: one output controlling signal for tailstock; 1: two output controlling signals for tailstock.

M79:output point for controlling forward of tailstock , controlling code is M79;

M73:output point for controlling backward of tailstock,controlling code is M78.

P23: External Switch(Foot switch) for tailstock, reciprocating mode, price one time, tailstock forward; press twice time, tailstock backward. 0: without switch ; 1: with external switch for control tailstock; input signal is M14.

P25: Holding time of output M79/M73 of tailstock, unit: s. 0: mode type*.

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.

4.3.1.7 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	These codes can be used for conditional wait or conditional jump.
	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	
8	M18xx	Check xx input point is valid	Special application for non-defined input points
	M28xx	Check xx input point is invalid	

Note: All M input commands, valid level of all inputs 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.

c) Special Detection

M18xx & M28xx are used for detect input points by X address directly, Example: M1810: detect if X10 is valid

4.3.3 M Special Command List

No.	Code	Function
2	M320	Clear workpiece coordinate value of all axes
	M317	Clear X_workpiece coordinate
	M318	Clear Y_workpiece coordinate
	M319	Clear Z_workpiece coordinate
	M315	Clear A_workpiece coordinate
	M316	Clear B_workpiece coordinate
	M312	Clear C_workpiece coordinate
	M313	Clear Xs_workpiece coordinate
	M314	Clear Ys_workpiece coordinate
3	M420	Clear machine coordinate value of all axes
	M417	Clear X_machine coordinate
	M418	Clear Y_machine coordinate
	M419	Clear Z_machine coordinate
	M415	Clear A_machine coordinate
	M416	Clear B_machine coordinate
	M412	Clear C_machine coordinate
	M413	Clear Xs_machine coordinate
	M414	Clear Ys_machine coordinate

M880~M889 are corresponding to ProgramUser0~ProgramUser9 macro program.

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 correspond 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 T Tool Function Command

Format	Function
Tab	a: Exchanging Tool Number b: Tool compensate number

Note: 1. a=0 ; means that don't exchange tool;

2. b=0; means that don't make tool compensation , display machine coordinate;

3.Range: a= 0~99 ; b=0~99.

Eg1: T0204 ; change to No.2 tool , and make No.4 tool compensation.

T0300 ; change to No.3 tool , and don't make tool compensation.

T0004 ; don't exchange tool , only make No.4 tool compensation.

When machine tool is with electrical turret, each station can fix several tools,adopt different tool compensation to exchange tools.

Eg2: 4 station turret, there are 2 tools in 3rd station (use No.3 & No.5 tool compensation)

```
N0000   T0101    ; change to No.1 tool, and make no.1 tool compensation
N0001   G0   X30 Z500
N0002   T0303         ; change to 1st tool in 3rd station
N0003   G00   X50
N0004   T0505         ; change to 2nd tool in 3rd station
N0005   M02
```

4.6 User-defined macro instruction(G120-G160,M880-M889)

Every user-defined G code is corresponding to a macro program ProgramGxxx, the M code is corresponding to a macro program of ProgramUser0 --ProgramUser9, the user cannot programme the macro program in NC system, must edit the macro code in the computer, and then copy into the system.

For example, defines the G152 function: the arc model porous drilling cycle. (must copy the macro program ProgramG152 into system).

Format:G152 Xx Yy Zz Rr Ii Aa Bb Hh Ff;

X: The X coordinate with absolute value or incremental value of center to specify.

Y: The Y coordinate with absolute value or incremental value of center to specify.

Z: Hole depth R: Approaching fast to the point coordinate

F: Cutting feed speed I: Radius

A: The angle of the first hole B: Incremental angle specify(CW when negative)

Macro program ProgramG152 as follows:

```
#80=#0
#81=#1
#82=#2
#83=#3
```

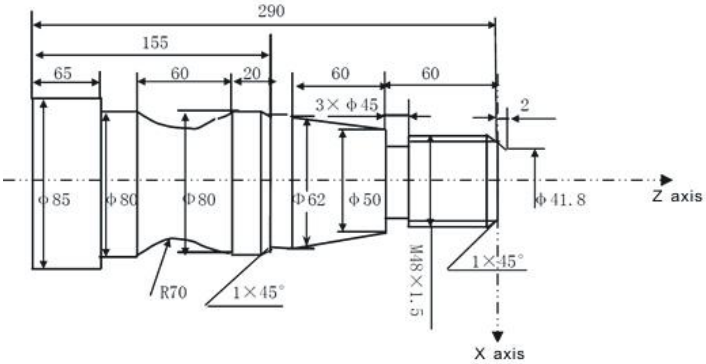
```
#84=#4
#85=#5
#86=#6
#87=#7
#88=#8
#89=#9
#90=#10
#91=#11
#92=#12
#93=#13
#94=#14
#95=#15
#96=#16
#97=#17
#98=#18
#99=#19
#100=#20
#30=#4003
#31=#4014
G90
IF[#30 EQ 90] GOTO 1
G53
#98=#5001+#98
#99=#5002+#99
N1 WHILE[#86 GT 0] DO 1
#35=#98+#87*COS[#80]
#36=#99+#87*SIN[#80]
G81X#35Y#36Z#100R#92F#85
#80=#80+#81
#86=#86-1
END 1
G#30 G#31 G80
M99
```

4.7 Synthetic instance for programming

In the actual programming, must according to the drawing and processing requirement to select the correct install folder mode and suitable tool, and combined with the actual working performance of lathe to select the right cutting allowance, for example:

Example 1：The tool is:

T01 cylindrical cutting tool; T02 cutting groove, tool width 3m; T03 thread tool with 60 degree angle



Program as follows:

N10 M03 S1000; Start spindle

N20 T0101; Choose the first tool and execute the first redeem

N30 G00 X41.8 Z2 M08; Move fast to the cutting point, cutting fluid is on

N40 G01 X48 Z-1 F100; Chamber

N50 Z-60; Fine machining for thread

N60 X50; Tool is backing

N70 X62 W－60; Fine machining in cone

N80 W－15; Fine machining in Φ62mm ex-circle

N90 X78; Tool is backing

N100 X80 W-1; Chamber

N110 W-19; Fine machining in Φ80mm ex-circle

N120 G02 X80 W-60 R70; Fine machining in arc（I63.25 K－30）

N130 G01 Z-225; Fine machining in Φ80mm ex-circle

N140 X85; Tool is backing

N150 Z-290; Fine machining in Φ85mm ex-circle

N160 X90 M09; Tool is backing, cutting fluid is off

N170 G00 X150 Z50; Move fast to the point of changing tool

N180 T0202; Change tool and set the No.2 redeem

N190 M03 S800; Change speed of spindle

N200 G00 X51 Z-60 M08; Move fast to the processing point, use the left point of tool to redeem

N210 G01 X45 F90; Cutting Φ45mm groove

N220 G00 X51; Tool is backing

N230 X150 Z50 M09; Return to the point of backing tool, cutting fluid is off

N240 T0303; Change tool and set the redeem

N250 M03 S1500; Change speed of spindle

N260 G00 X62 Z6 M08; Move fast to the processing point, cutting fluid is on

N270 G92 X47.54 Z－58 F1.5; Cutting thread is cycle

N280 X46.94;

N290 X46.54;

N300 X46.38;

N310 G00 X150 Z50 M09; Return to the point of start cutting, cutting fluid is off

N320 T0300; Cancel redeem

N330 M05; Stop spindle

N340 M30; Program is over

Chapter 5 Operation

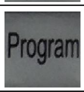
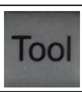
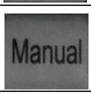
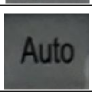
When using CNC Lathe system, just master the parameter of system, edit program, manual operation, auto operation. Then you can operate the system easily. There are some details to instruct hereinafter.

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.

Fig1.1 SZGH-CNC990TDb CNC Lathe Controller

5.2 Function Menu

Menu Keys	Comment
	Enter the interfaces of status parameter. Data parameter, and screw compensation parameter interface (interfaces can be switched by repeated press)
	Enter the program interface.
	Enter the redeem interface.
	Posit tool, set position of current tool in machine coordinate system.
	Enter controlling condition of handwheel
	Enter controlling condition of manual
	Enter controlling condition of auto
	Enter the interfaces of diagnosis
	Setup current workpiece coordinate when G54~G59. select G54~G59 on MDI.

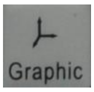
Del	Delete key	***Note: Clear number of workpiece***
Esc	Exit key	
Enter	Enter key	

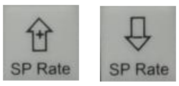
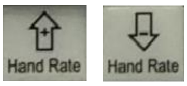
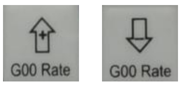
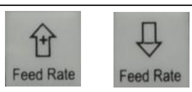
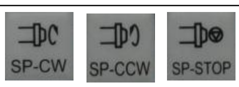
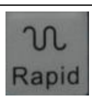
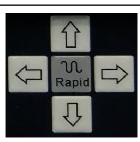
5.3 Editing Keyboard

Keys	Name	Description	
RESET	Reset key	CNC reset, stop of the feeding and moving, etc.	
X Z G M U W S T I K F E L R Y P N J V Q A B C D	Address Key	Address input, Double-address key, switch between addresses & symbol by repeated press	
- / .	Symbol key	Symbol Input	
7 8 9 4 5 6 1 2 3 0	Digital Key	Digit input	
↑ ↓ ← →	Cursor Move Keys	Move the cursor in different directions.	
PgUp PgDn	Page up/down key	Page up/down on display	***Note: Exchange of coordinate***

5.4 Machine Control Panel

Key	Designation	Explanation	Remarks and explanation
Coolant	Coolant switch	Coolant ON/OFF	Control code: M08/M09 Output Point: PIN8 of CN3
Chuck	Chuck switch	Tighten/Loosen Tool of Spindle	Control code: M10/M11 Output point:PIN21 of CN3
Thumbstall	Tailstock switch	Tailstock Forward/Backward	Control code: M79/M78 Output Point: PIN22 of CN3

	Single block key	To enter single block mode	
	Per Step mode key	To enter single step mode	Switching cycle of feeding mode between "manual continuous" to "manual increment"
	Graphic mode key	To enter graphic mode	Shift on Auto status
	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 explanation
	Spindle Rate keys	Up/Down rate of SP_Speed	Range is form 5% to 150%,16 gears totally
	Handwheel Rate 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 Axes Feeding rate	range is from 0% to 150%,16 gears totally
	Spindle control keys	CW/CCW/Stop of Spindle	Corresponding to M03/M04/M05 codes
	Rotation of SP in JOG	Holding press, CW rotation of spindle; don't press, stop rotation of spindle	
	Exchange Tool key	Manually change to next one tool	
	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 4th 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		

	Interface Switch Left side: Normal Mode Middle: F-Halt , stop feeding Right side: FS-Halt, stop feeding & rotation of spindle

Note: 1.“Manual speed set”: Press “1” “2” “3” “4” “5” “6” “7” “8” “9” keys to set feed override “F30” “F60” “F120” “F250” “F500” “F1000” “F1500” “F2000” “F2500” “F3000”.

2.“F” key: Taking mm/min as the unit to set the manual feed speed,which is also set by P7 on Speed parameter. Range:0~30000mm/min.

3.“S” key: Set the speed of the first spindle, which is set by P8 on Speed parameter. Range: 0~99999, the max value of spindle depends on the No.36 parameter in speed parameter.

4.“I” key: Modify the increment per step in manual increment mode(single step mode).

5. "T" key: Choosing current tool number & tool offset.

6. Machine coordinate clear: Press “E” key in parameter screen and then input machine password and press “Enter” to clear machine coordinate value to 0.

7. Return to zero point of G53: Press “Q” key,and then press Enter key.

8. Automatically divide center/find center of circle: Shift to G54~G59, press “K”key. Do as prompt,CNC will figure out value to offsets of G54~G59.

The system adjusts one-level menu operation, intuitive, convenient, shortcut, prompt comprehensive information. Powering on system is to enter the main screen.

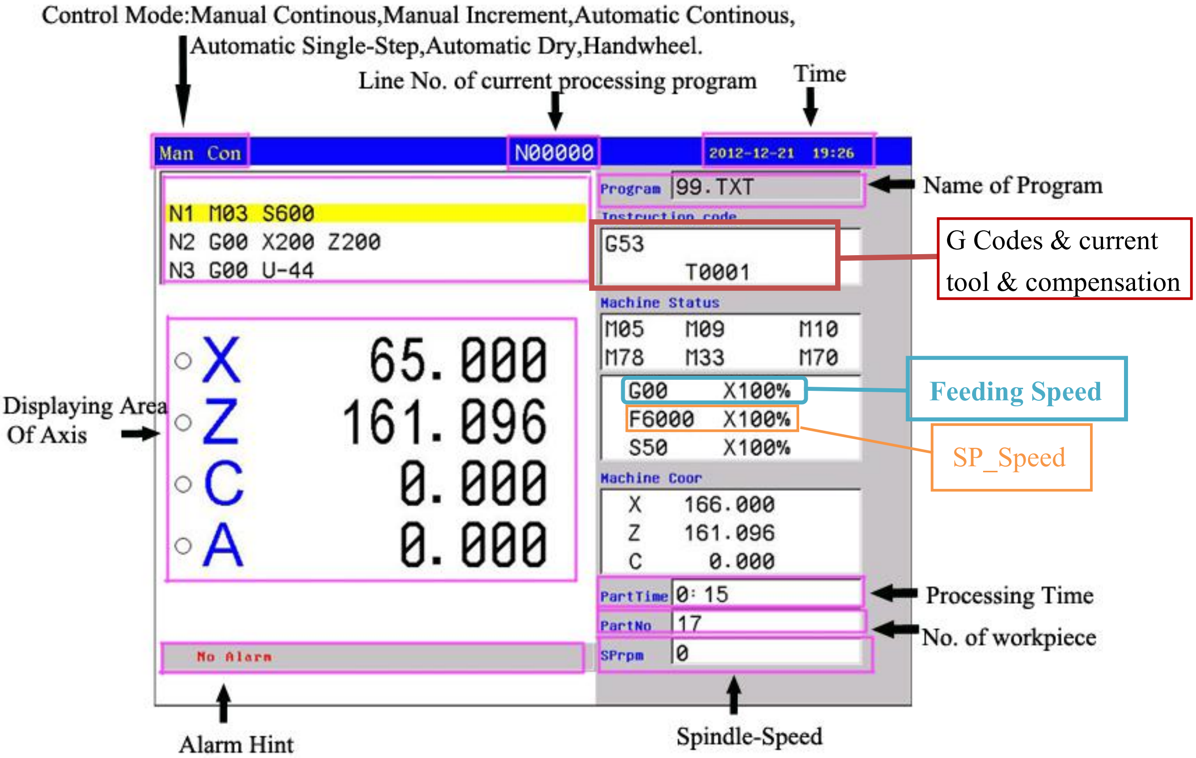


Fig5.2 Displayer

Press “Program” key enter program management area.it could edit,alter, diagnosis, delete,and copy etc.

Press “Parameter” key enter parameter management screen for refer, alter parameters.

In manual condition,the system could process workpiece.

5.5 Manual Operation

5.5.1 Manual Continuous

Continuous operation is basis 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 “↑ ↓ ← →” 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 handhold box; the other is handwheel in the operational panel.

Handwheel in handhold 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.

P17 in Speed parameter is set for acceleration / deceleration time constant.

The maximum speed is controlled by P20(X axis) P21(Z axis) in Speed parameter.

Note: 1. SZGH-CNC990TDb(c) series only support handwheel in handhold box(CN11 plug).

Fig5.3 handhold 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.

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,

P23: Bit parameter, D3:X ; D4:C(Y) ; D5:Z ; D6:A ; 1 means float zero point;
P24: Distance between reference position & current position in X direction;
P25: Distance between reference position & current position in Z direction;
P119:Distance between reference position & current position in C(Y) direction;
P217: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: P24/P25/P119/P217

Note: If current position is home of machine coordinate system, no needs to set offset of P24/P25/P119/P217 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,

P23: Bit parameter, D3:X ; D4:C(Y) ; D5:Z ; D6:A ; 0 means switch for homing;
P26: Grade of Homing,0: Just hint; 1:No use; 8: Compulsion; 9: Super Compulsion;
P27: Mode of Homing, 0 & 2: Detect Z0 signal; 1 & the others, no detect ;
P28: Bit Parameter, Direction & Sequence of Homing;
P29: Bit parameter, Type of switch for homing ;
P30: Range of detecting Z0 signal in X-axis ;
P31: Range of detecting Z0 signal in Z-axis ;
P115: Range of detecting Z0 signal in Y(C)-axis ;
P213: Range of detecting Z0 signal in A-axis ;
P32: Offset after homing in X-axis
P33: Offset after homing in Z-axis
P116: Offset after homing in Y(C)-axis
P214: Offset after homing in A-axis

In Speed Parameter,

P30: X-axis homing speed
P32: Z-axis homing speed
P31: Speed during detecting Z0 signal of X-axis

P33: Speed during detecting Z0 signal of Z-axis
In Axis parameter,
P112: C-axis homing speed
P113: Y-axis homing speed
P211: A-axis homing speed
P114: Speed during detecting Z0 signal of Y-axis
P212: Speed during detecting Z0 signal of A-axis

Note: Details about parameters set,please check part of Parameters List(see chapter 6)

B. Input points for homing in the table

Input Point	PIN	Function
X0	PIN3 of CN3 Plug	X axis homing
Z0	PIN17 of CN3 Plug	Z axis homing
M36/Y0	PIN2 of CN3 Plug	Y(C) axis homing
M34/A0	PIN4 of CN3 Plug	A axis homing

Note: 1.when system hasn't Y(C) & A axis, these two input points as function of M36&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; Z; Y(C) ; 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 3.15).

Special Attention: Every time to power up the system must back to zero point to make sure the accuracy of lathe process. The system power off unusually or in an accident, it must back to zero point, otherwise could cause trouble.

5.5.5 Alignment Tool(Posit Tool)

When machine tool use several tools to processing , length of each tool is different, so user need to ensure their offset value between tools, also posit tool, which actually is move tool to one position of workpiece, measure coordinate value of this position, and input to CNC system. System will calculates the offset and save to the related offset value of tools to register automatically.

a) Two solutions for posit tool:

A Solution (suggestion)

① Chuck clamp workpiece, set SP_Speed and Feed speed well,run spindle ;
② Select one tool that needs to setup, Example: T0202 ;
③ Turning a cylinder/bore on workpiece in Manual Continuous Mode ;
④ Exit Z axis(can't move X-axis), stop spindle ;
⑤ Measure the diameter of workpiece(cylinder/bore);
⑥ Press "Tool" key in Function Menu, input "X" & "Enter" , then import the above value of measurement into dialog box, press "Enter" to confirm ;
⑦ Cut endface of workpiece , after done, don't move Z-axis ,Stop spindle ;

⑧ Measure distance between endface of workpiece and home of Z-axis(Z0) ;

⑨ Press “Tool” in Function Menu, input “Z” & “Enter” , then import the above value of measurement into dialog box, press “Enter” to confirm ;

⑩ Posit/Setup of No.2 Tool(T02) is already done; Repeat ① – ⑨ to setup /posit/calibrate other tools.

B Solution

1) Chuck clamp workpiece, set SP_Speed and Feed speed well,run spindle ;
2) Select one tool that needs to setup, Example: T0202 ;
3) Turning a cylinder/bore on workpiece in Manual Continuous Mode ;
4) Press “Tool” in Function Menu, CNC system will hint a dialog box;
5) Exit both X axis and Z axis , & stop spindle ;
6) Measure the diameter of workpiece (cylinder or bore) ;
7) Press “X” and input above measuring value to dialog box, press “Enter” to confirm ;
8) Cut endface of workpiece with same steps above, stop feeding ;
9) Press “Tool” in Function Menu, CNC system will hint a dialog box;
10) Exit both X axis and Z axis , and stop spindle.
11) Measure distance between endface of workpiece and home of Z-axis(Z0) ;
12) Press “Z” and import the above value of measurement into dialog box, press “Enter” to confirm.
13) Posit/Setup of No.2 Tool(T02) is already done;Repeat (1) — (12) to setup /posit/calibrate other tools.

b) Difference between two methods:

A Solution	B Solution
Must ensure don’t exit posit tool axis	Posit tool axis can exit
Tool touch workpiece during exit	Tool don’t touch workpiece
Use for posit tool in Z direction	Use for posit tool in X direction

In the above processes, CNC system will save difference value between input measuring value and machine coordinate value to related tool compensation (Redeem) automatically.

If there is error as posit tool, user can adopt the redeem of tool compensation that can remove the error.

c) Special Application

When user needs to use a group of tools to processing several pieces, that needs using working coordinate system to let tools offset totally.

Method of operation is as follows:

(1) Select anyone tool; Example:T0303
(2) Select corresponding coordinates in MDI: Input G54~G59, press “Start” key;
(3) Turning a cylinder/bore on workpiece in Manual Continuous Mode ;
(4) Exit Z axis (cannot move X axis), stop spindle ;
(5) Measure the diameter of workpiece(cylinder or bore).
(6) Press “Setup”key, input “X” and “Enter”, import the measuring value, &press “Enter” key.
(7) Cut endface of workpiece with same steps above ,stop feeding & spindle ;
(8) Measure distance between endface of workpiece and home of Z-axis(Z0) ;
(9) Press “Setup”key, input “Z” and import the measuring value, & “Enter”.

CNC system will save difference between import value and workpiece coordinate value to the corresponding parameters automatically, posit tool in workpiece coordinate also set done well, and the others also set well. Tool setup is okay that user add workpiece coordinate codes (G54~G59) in first line to select the related workpiece coordinate system.

Note: 1.Every tool are independent each other, which has its own coordinate system, so user can can posit any tool at anytime , and also only affect current tool when crashing during processing. Only posit current tool well when crashing or loss of step, all is okay. Method: In the status of G53, operate method of operation at c) special application.

2. Before posit tool,please ensure that CNC system can ATC normally.

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 "↑ ↓" 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.

Switch display of coordinate: Press "PgUp"/ "PgDn" keys to switch the display which correspond to "Relative" "Absolute" "All".

Relative Coordinate	Absolute Coordinate	All Coordinate
U 0.000 W 0.000	X 0.000 Z 0.000	Relative: U 0.000, W 0.000 Machine: X 0.000, Z 0.000 Absolute: X 0.000, Z 0.000 Distance to go: X 0.000, Z 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.

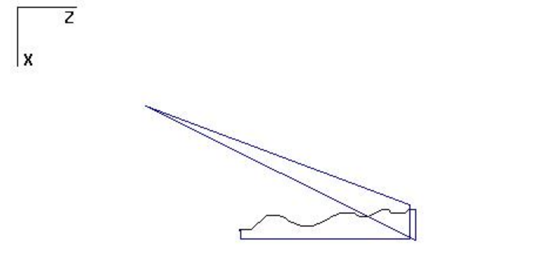
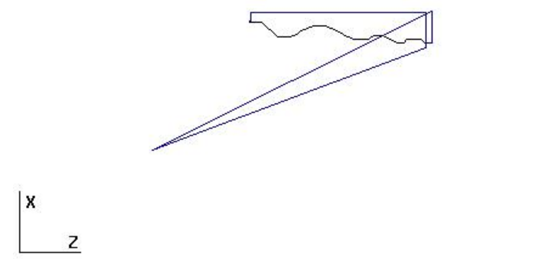
Table of Status Display about Manual/Auto

Manual Continuous	Manual Increment	Handwheel
Man Con	Man Inc 1.000	Man PulsX *100
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. There are two kinds of figure,also two types of lathe machine, one is Front Turret lathe machine ,the other is rear turret lathe machine, P3 in Tool parameter is set for type of lathe machine.

Front Tool-post Coordinate System	Rear Tool-post Coordinate System
	

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 popup 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 P5 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.

1.6.2.2 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 P5 in Speed parameter, then run the program normally.

5.6.2.3 Start from "nth" tool

The system has a function to start program from someone tool("nth" tool). At automatic process condition, press "G" and input number of tool , press "Enter" to confirm. Press "Run" to process program at the tool you import.

Note: 1.only input number of tool, not need to input number of compensation; Eg.: T0304, just import "03" ;

2. Firstly of all, system will move the starting point of "nth" block with speed which is set by P5 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.

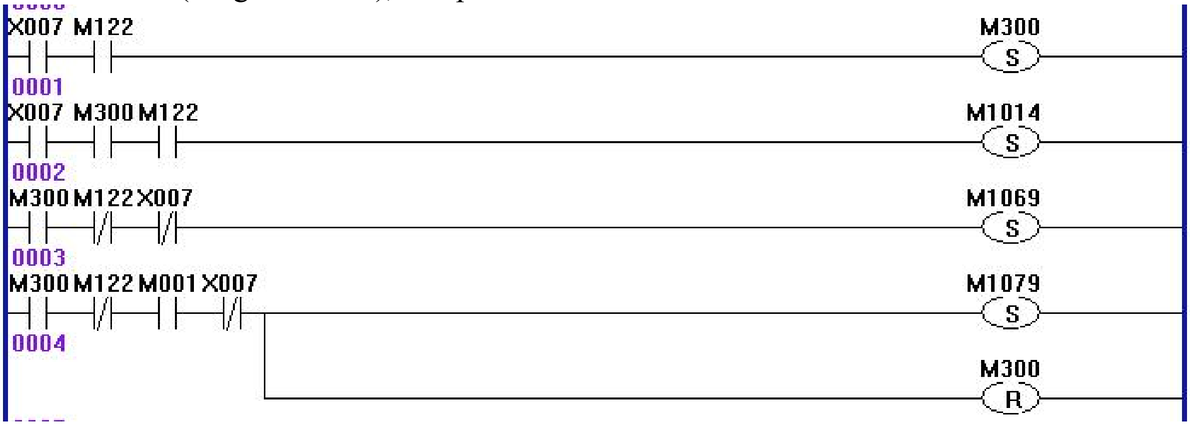
(3) Interface switch turn to the middle or right side.

(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.

(6) Use input point to activate inner macro program during processing. For example: use X7 to activate M889(ProgramUser9), add plc blocks into CNC:



M300: Middle auxiliary relay,don’t use M300 for other usages during PLC.

M122: Valid when program is on running status

M1014: Reset current program

M1069: Manual status

M01: It is valid when CNC is on Manual status

M1070~M1079: corresponding to ProgramUser0~ProgramUser9(M880~M889).

5.6.5 MDI mode

MDI,also Manual Data Input. In Manual or Auto/Coordinate status, press “M” key to enter MDI mode, also input one segment of commands, press “ESC” key to quit & exit MDI, press “Start” key/Run to execute current commands on MDI.

5.6.6 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.

5.6.7 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)
Z-axis driver has happened hardware limit	+L(PIN16 _CN3 Plug)
X-axis,C-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, the lathe will stop all actions and the screen of system shows "Emergency STOP". Wait for releasing the button.

5.6.8 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.

5.6.9 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/rear of CNC990TDb series controller.

5.6.9.1 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.

5.6.9.2 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 IOs are already used on CNC system.

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

P11: 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 Z-Negative Direction

P6: Max Range in Z-Positive Direction

P117: Max Range in C(Y)-Negative Direction

P118: Max Range in C(Y)-Positive Direction

P215: Max Range in A-Negative Direction

P216: Max Range in A-Positive Direction

Note: 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

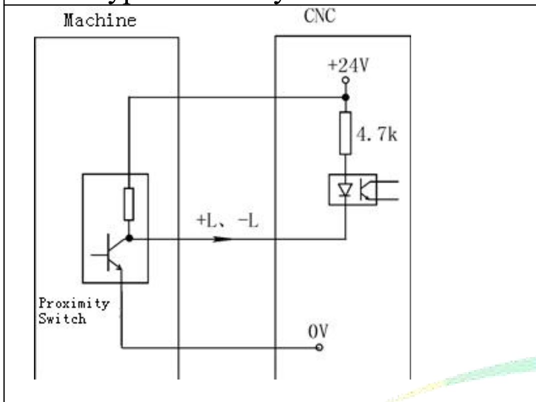
In Axis parameter:

P21, Type of switch in Positive direction [0: NO type, 1: NC Type]

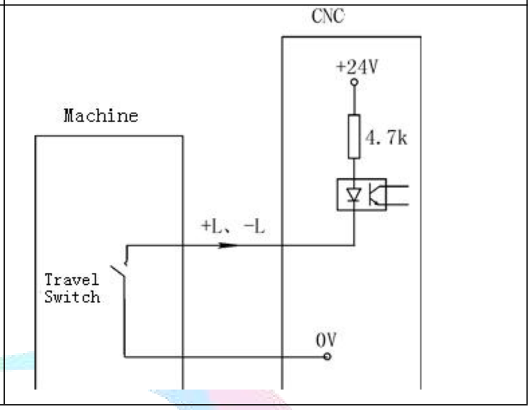
P22, Type of switch in Negative direction [0: NO type, 1: NC Type]

Wiring of Limitation

NPN Type Proximity Switch



General Switch



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.

2.1 Limitation in negative or positive direction of X axis use one input port , -L ;

2.2 Limitation in negative or positive direction of Z axis use one input port , +L ;

In Axis parameter:

P21, Type of Switch in X direction [0: NO type, 1: NC Type]

P22, Type of Switch in Z direction [0: NO type, 1: NC Type]

5.8 Diagnosis

Press “Diagnosis” key to enter the diagnosis interface.

Press “J” & “PgDn/PgUp”or “↓ ↑” to check the status of inputs and outputs.

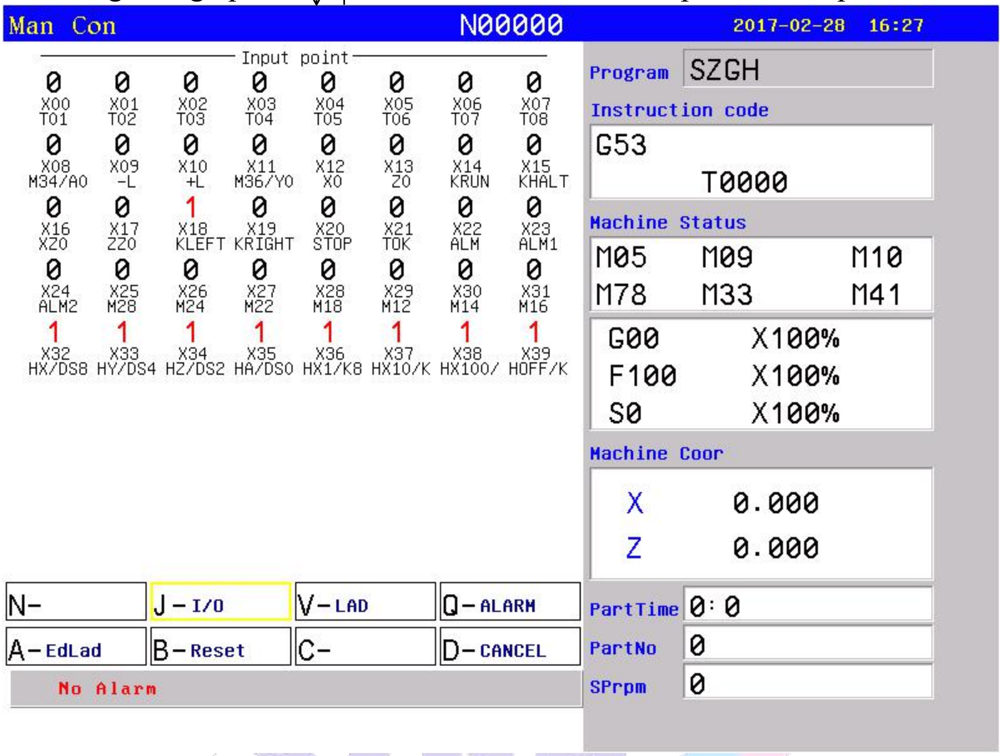


Fig5.8.1 System Diagnosis Interface(Input signal)

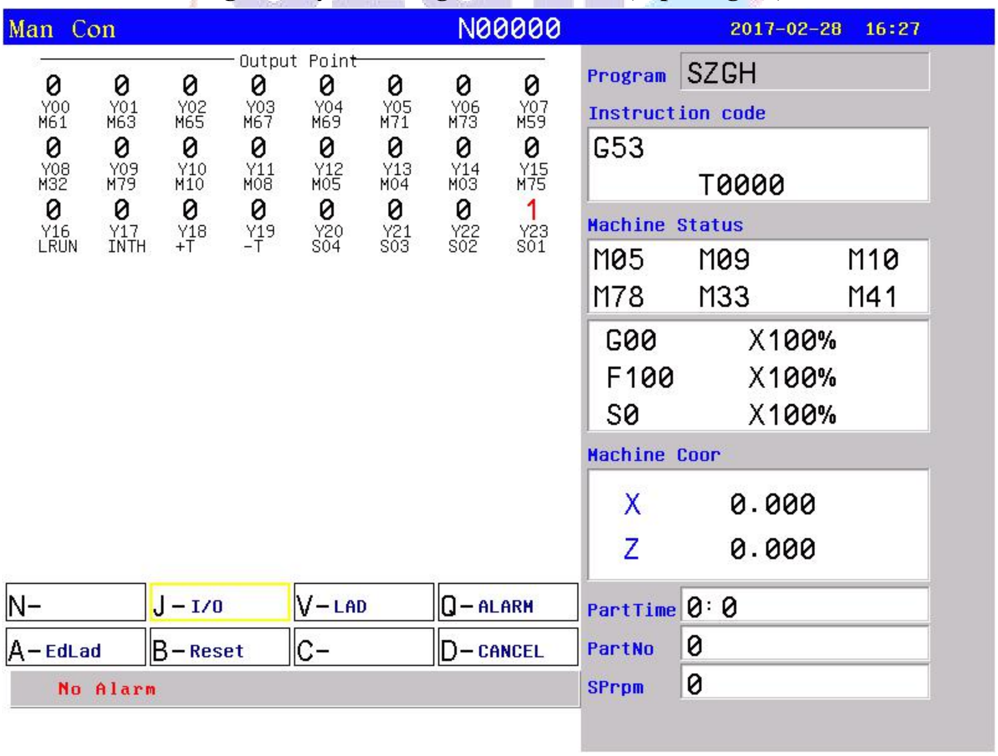


Fig5.8.2 System Diagnosis Interface(output signal)

In diagnosis interface of I/O , “0” means invalid status, “1” means valid status.

Press “V” key diagnosis screen to enter interface of check condition of PLC.

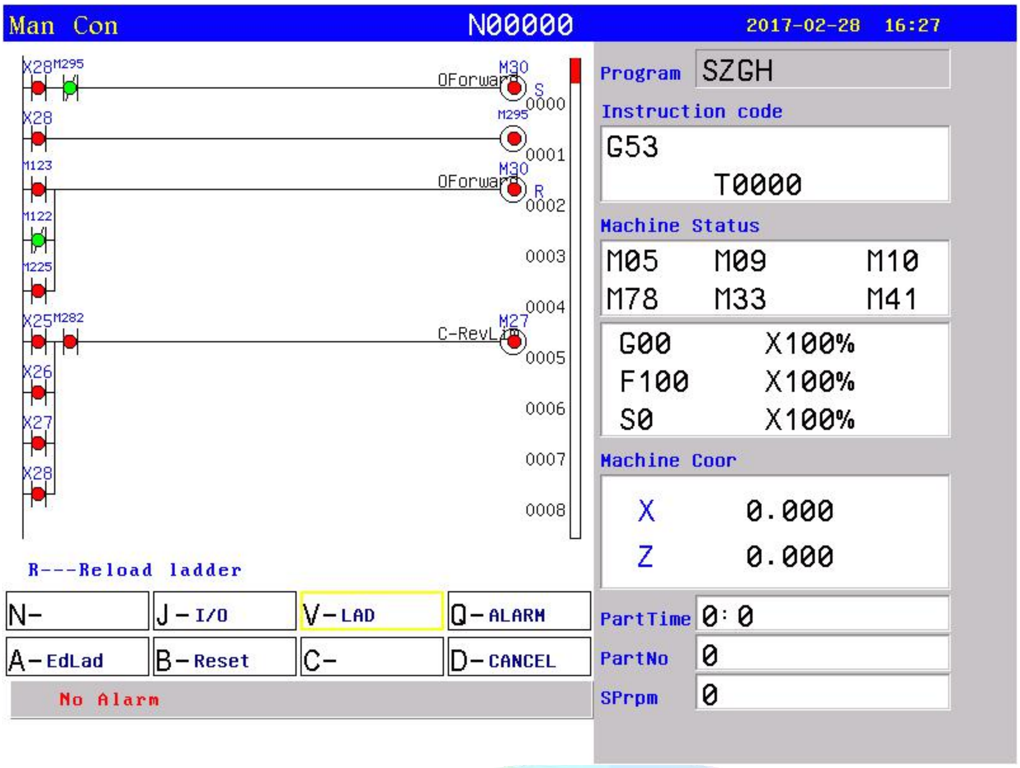


Fig5.8.3 Condition of Inner Register & I/Os

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.

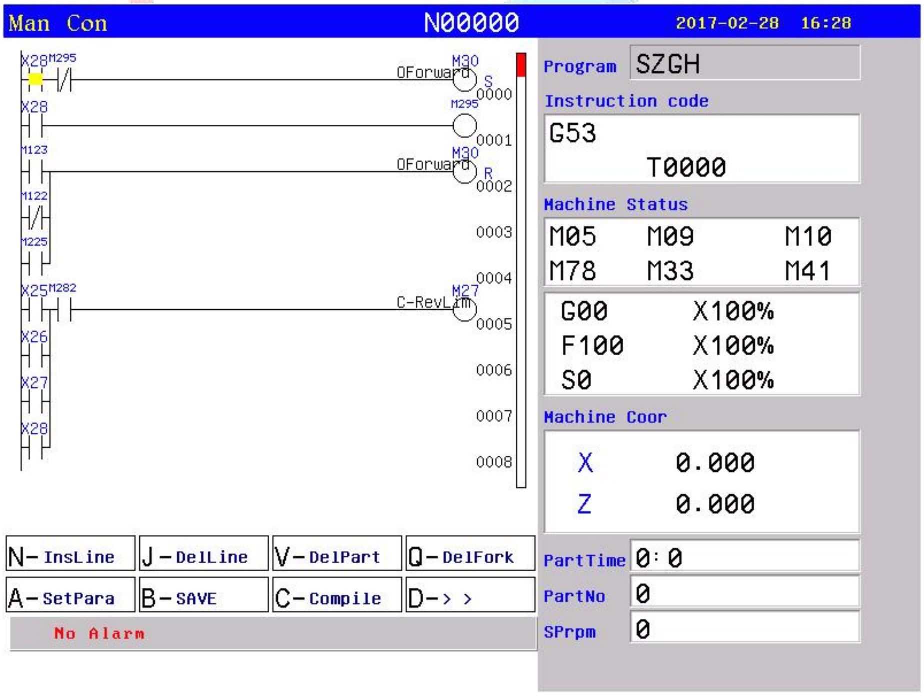


Fig5.8.4 Editing Screen of Inner Ladder

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 P1 in Password parameter set to Disable, and then user can check & edit inner ladder.

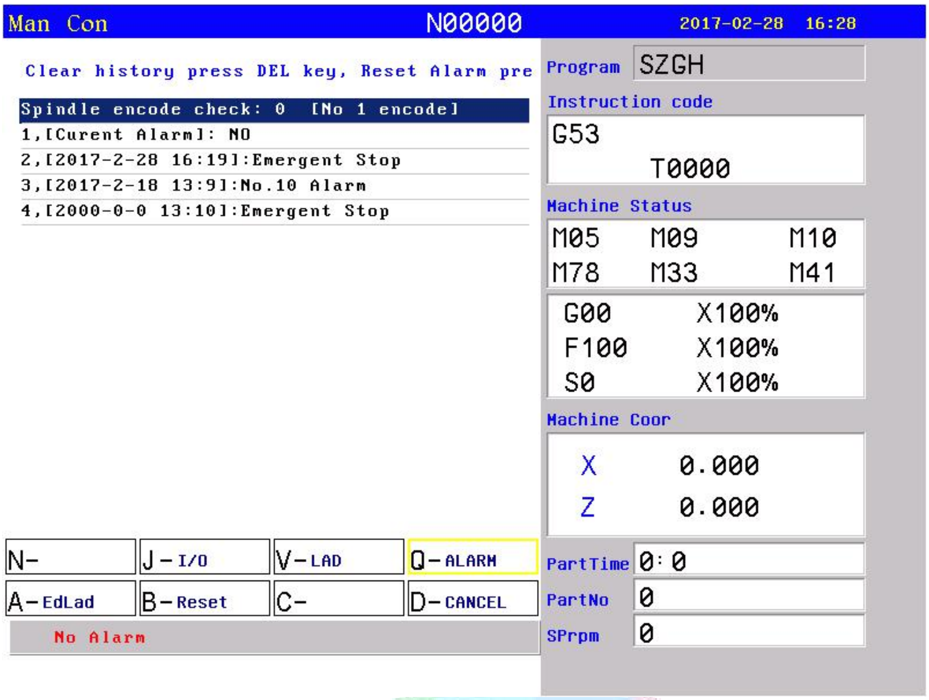


Fig5.8.5 System Diagnosis Interface(Alarm messages)

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.

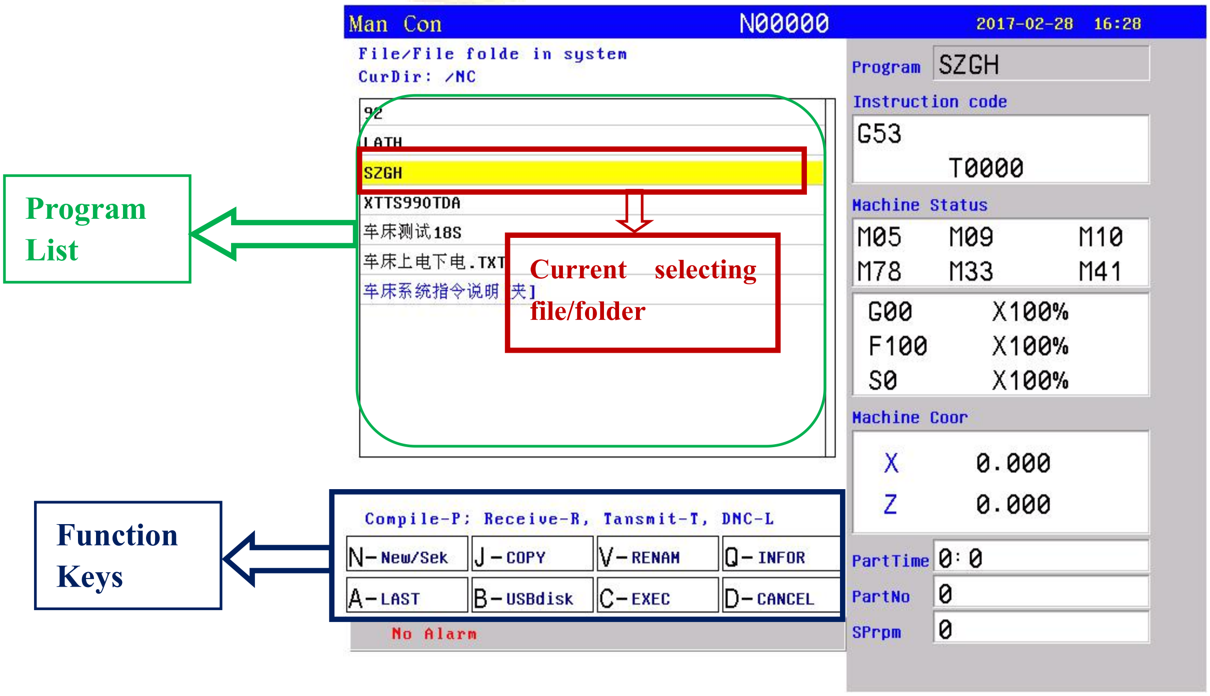


Fig5.9.1 Interface of Program

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 ↓‘↑, ” 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
J-COPY	Press “J”key to Copy selecting program(System ↔USB-disk)
V-RENAM	Press “V” 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-R	Receive file from PC with RS232-DNC
Transmit-T	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 popup 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.

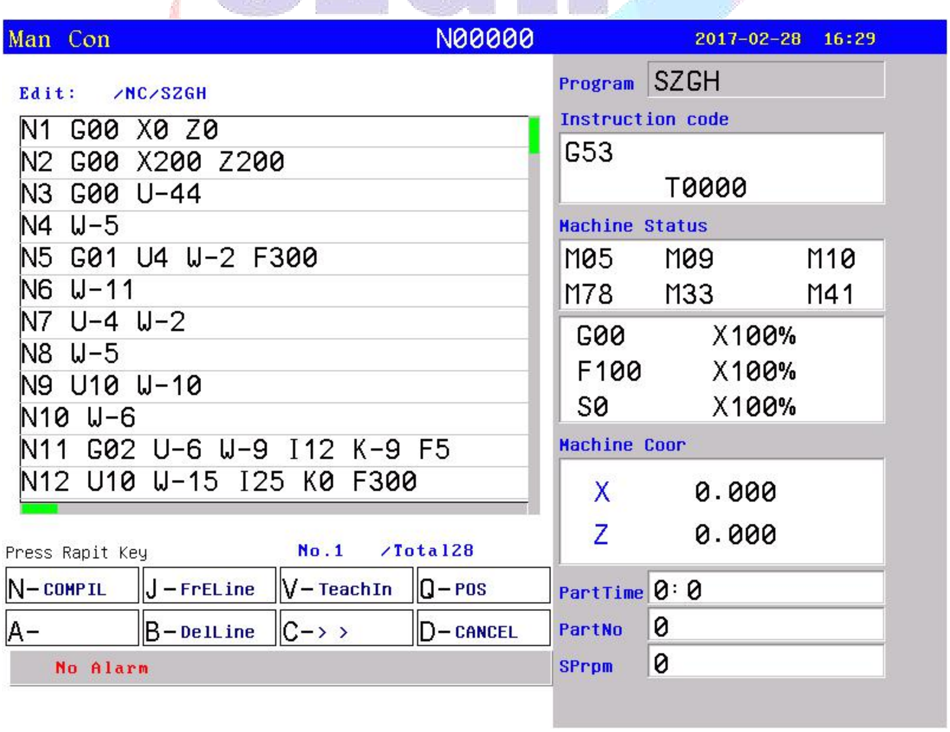


Fig5.9.2 (1)Editing interface of Program

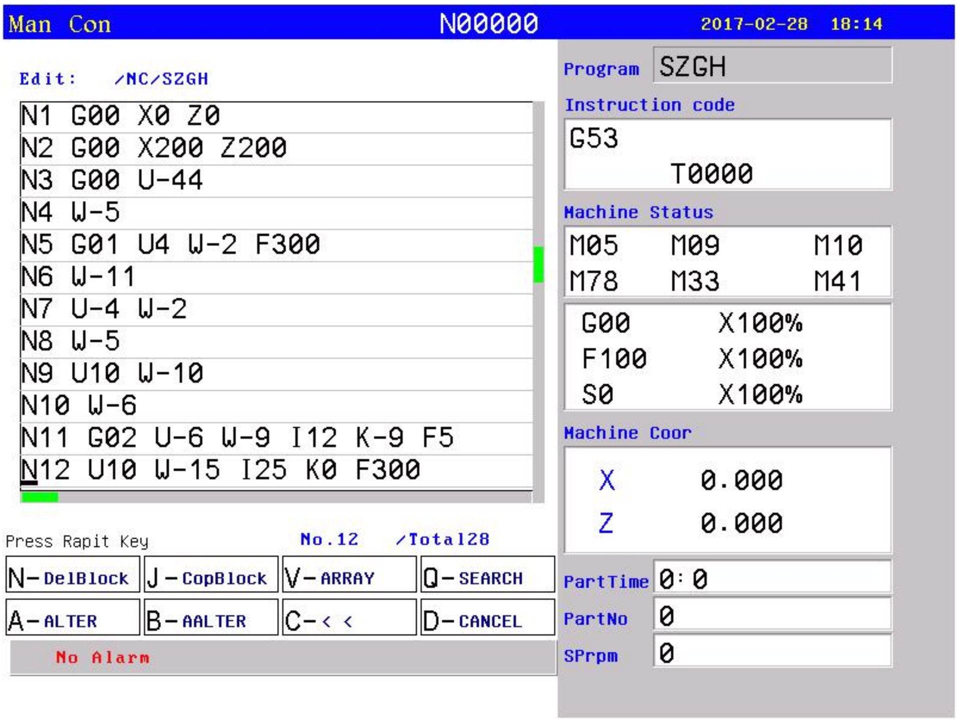


Fig5.9.3 (2)Editing interface of Program

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 + J	Cursor jump to first line or last line
Rapid + V	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 + J	Copy specified blocks from input begin line to input last line
Rapid + V	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 "J" 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 "V" 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.

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 "T" to deliver, press "Reset" to interrupt delivery.

Reception

Receive the selected program in another system or PC (Must be text file form). Press "R" 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.

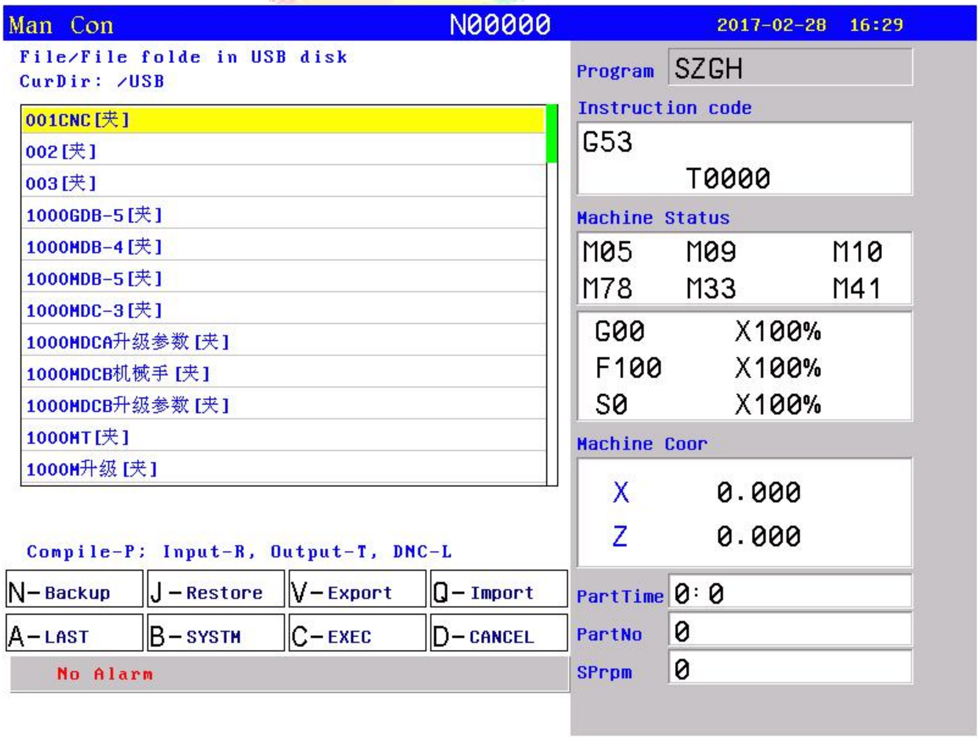


Fig5.9.4 Main Interface of USB-disk

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
J-Restore	Press “J”key to restore files at current directory of U-disk into system
V-Export	Press “V” 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-R	Press “R” key to copy all files of U-disk to system
Output-T	Press “T” 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 “R” 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 “T” 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 Transfer DXF file to G code Program

User can press “-” key to transfer dxf file to G code program after select dxf file on NC system directory.This G code program will be name with “.CNC”.

File extension of DXF files , that are imported into NC system must be with “.DXF” or “.dxf”.

CNC system will generate codes in the start and in the end of program according to “HEADDXF.txt” & “ENDDFX.txt”,(or “headdxf.txt” & “enddxf.txt” files),which are exist current NC directory when CNC transfer dxf file to G code program.

Please take P400~P401 on User parameter as reference when use this function.

Note: “headdxf.txt” & “enddxf.txt” must be saved at current directory as dxf file.

5.9.10.4 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.

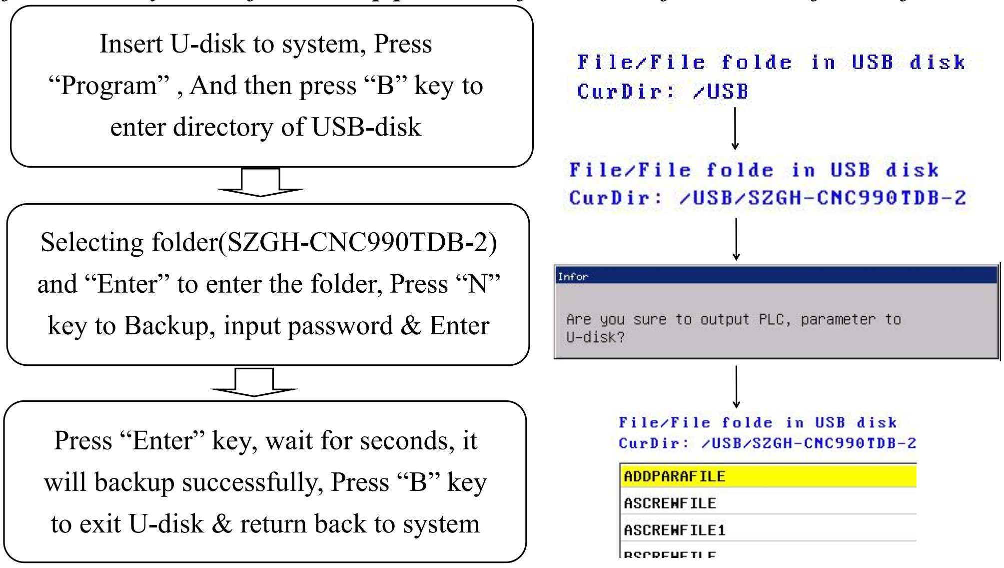


Fig5.9.5 Steps of Backup PLC & Parameter to U-disk

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.

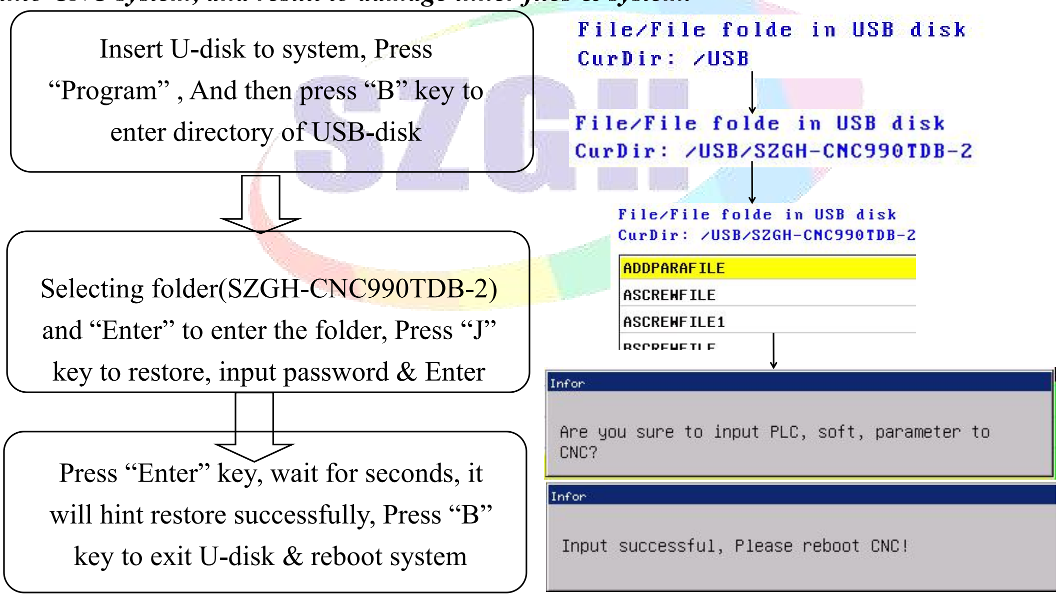


Fig5.9.6 Steps of Restore PLC & Parameter to U-disk

Chapter 6 Parameter List

At any status conditions, press “Parameter” to enter interface of parameter.

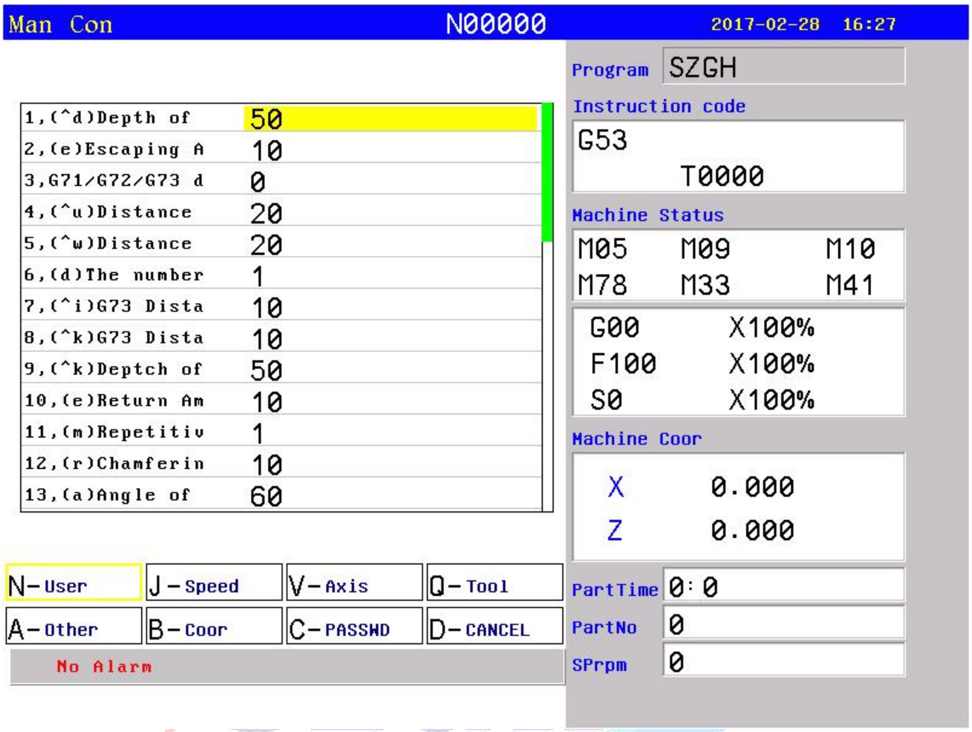


Fig6.1 Parameter List

Remark	Function
N-User	Press “N” key to enter User Parameter screen
J-Speed	Press “J” key to enter Speed Parameter screen
V-axis	Press “V” 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. Parameter list on user manual are for all axes of CNC controller, so different axes configurations are with different parameters on real CNC controller. Example: There aren’t parameters for Y-axis & A-axis on 2 Axes CNC controller(SZGH-CNC990TDb-2).

3.Short name: P2,means No.2 parameter, example: P2 in User parameter,also No.2 parameter in User parameter.

6.1 User Parameter

P	Parameter	Ex-Value
1	(^d)Depth of Cutting in G71/G72[Radius Designation] (10um)	50
2	(e)Escaping Amount[Radius Designation](10um)	10
3	Function Bit parameter of G71/G72/G73	0000000100000001
4	(^u)X_Distance&Direction of finishing allowance of G71/G72/G73 (10um)	20
5	(^w)Z_Distance&Direction of finishing allowance of G71/G72/G73 (10um)	20
6	(d)The number of division in G73	1
7	(^i)X_Distance and direction of relief of G73[Radius Designation] (10um)	10
8	(^k)Z_Distance and direction of relief of G73(10um)	10
9	(^k)Depth of Cut in Z direction of G74/G75(10um) [Without Sign]	50
10	(e)Return Amount of G74/G75(10um) [Radius Designation]	10
11	(m)Repetitive Count in Finishing of G76	1
12	(r)Chamfering amount of G76(1/10 lead)	10
13	(a)Angle of tool tip in G76(degree) [0～180°]	60
14	(^dmin)Minimum Cutting Depth of G76(10um) [Radius Designation]	20
15	(d)Finishing allowance of G76(10um)	10
16	Programming mode of X axis [1: Radius,0: Diameter]	0
17	Interlock between Running Program & SP-Rotating [1:Yes, 0:No]	0
17-1	Interlock between Running Program & SP_Chuck[1:Yes, 0:No]	0
18	Times of auto-cycle running(M20)[<0: Endless Loop]	-1
20	Length of chamfering in G92(1/10 lead)	0
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
24	Q(△d)infeed times in G76 [8:Times of rough cutting]	1
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]	1
201	Delay time before detect zero pulse when threading(ms)[>100]	1
203	Using Pause key in Panel (23103490:Yes,6326274:No)	23103490
206	G21-Metric/G20-Inch Mode(512:Metric, 1024: Inch, other: G20/G21)	512
210	Type of Graphic display area(8:manual,other:Automatic)	0
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)	0
231	Mode of "Delete" key [0:backward deletion,1:Forward delete]	0
232	Detect SP_Zero position before tapping [18:Yes,0:No]	18
233	G06 Circle teaching function[0:No, 1:Yes]	1
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)Depth of Cutting in G71/G72[Radius Designation](10um)

It is set for every default infeed in X-axis direction when W(∆d) isn't specified in G71 code, radius designation; It is also set for every default infeed in Z-axis direction when it isn't set in G72 code; unit:0.01mm

2,(e)Escaping Amount[Radius Designation](10um)

It is set for every default backward (radius) in X-axis direction when it isn't be set in G71 code;unit:0.01mm.

It is set for every default backward in Z-axis direction when it isn't be set in G72 code;unit:0.01mm.

3,Function Bit parameter of G71/G72/G73

D2: do finishing automatically for G71/G72/G73. 1:Yes ; 0: No finishing.
D3: X-axis rapid feeding for G74/G75
D4: Z-axis rapid feeding for G74/G75
D5: Automatically judge feeding/escaping direction of G71/G72/G73 commands
D6: Auto feeding point
D7: Don't return to starting point when G70
D8: Allowance is more than radius of tools when G41/G42.

4,(^u)X_Distance&Direction of finishing allowance of G71/G72/G73(10um)

It is set for every default remain of smoothing(diameter) in X-axis direction when it isn't be set in G71/G72/G73 code;unit:0.01mm.

5,(^w)Z_Distance&Direction of finishing allowance of G71/G72/G73(10um)

It is set for every default remain of smoothing(diameter) in Z-axis direction when it isn't be set in G71/G72/G73 code;unit:0.01mm.

6,(d)The number of division in G73

It is for default cycle times when it isn't be set in G73 code.

7,(^i)X_Distance and direction of relief of G73[Radius Designation](10um)

It is for default rough thickness of X axis when it isn't be set in G73 code.

8,(^k)Z_Distance and direction of relief of G73(10um)

It is for default rough thickness of Z axis when it isn't be set in G73 code.

9,(^k)Depth of Cut in Z direction of G74/G75(10um) [Without Sign]

It is for each default infeed in Z-axis direction when it isn't be set in G74;

It is for every default infeed (radius) in X-axis direction when it isn't be set in G75 code;

10,(e)Return Amount of G74/G75(10um) [Radius Designation]

It is for every default retract in Z-axis direction when it isn't be set in G74 code;

It is for every default retract (radius) in X-axis direction when it isn't be set in G75 code;unit:0.01mm.

11,(m)Repetitive Count in Finishing of G76

It is for default cycle times when it isn't be set in G76 code.(times:1-99)

12,(r)Chamfering amount of G76(1/10 lead)

It is for default length of retract chamfer when it isn't be set in G76 code.the length is 1/10 of thread lead.

13,(a)Angle of tool tip in G76(degree) [0～180°]

It is for default angle of thread tooth when it isn't be set in G76 code.unit:degree.

14,(^dmin)Minimum Cutting Depth of G76(10um) [Radius Designation]

It is for set minimal cutting depth (radius) of G76.Unit:0.1mm

15,(d)Finishing allowance of G76(10um)

It is for default remaining of finish turn when it isn't be set in G76 code.

16,Programming mode of X axis [1 mean Radius,0 mean Diameter]

There are two programming modes ,when it set as 1 that means radius programming mode,when set as 0 means that diameter programming 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 running program with running spindle;when set as 0 means running program without check spindle running.

17-1,Interlock between Running Program & SP_Chuck[1 mean Yes,0 mean No]

It is for interlock between run program and spindle chuck is tightening, when set as 1 means that running program with chuck is tightening, input point is M14, PIN24_CN10 plug;when set as 0 means that running program without check condition of chuck.

18,Times of auto-cycle running(M20)

It is for times of run M20 code in the program,negative number mean run countless times.

20,Length of chamfering in G92(1/10 lead)

It is for set default length(width) of quit and retract,the length=thread lead * 0.1.

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/deceleration. the smaller it is,the faster the Acceleration/deceleration is,but much vibration.

24,Q(△d)infeed times in G76 [8 mean times of rough cutting]

It is for set the define of Q in G76,set as “8”,it is the times of feeding in roughing.

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 X16-X23 & X26-X31.D2-D7:X26-X31; D8-D15:X16-X23.

D2 bit=1,also +4 , means X26 input is valid, execute program of "X26"/"HIDEFILEX26"
D3 bit=1,also +8, means X27 input is valid, execute program of "X27"/"HIDEFILEX27"
D4 bit=1,also +16, means X28 input is valid, execute program of "X28"/"HIDEFILEX28"
D5 bit=1,also +32, means X29 input is valid, execute program of "X29"/"HIDEFILEX29"
D6 bit=1,also +64, means X30 input is valid, execute program of "X30"/"HIDEFILEX30"
D7 bit=1,also +128, means X31 input is valid, execute program of "X31"/"HIDEFILEX31"
D8 bit=1,also +256 , means X16 input is valid, run program of "X16"/"HIDEFILEX16"
D9 bit=1,also +512, means X17 input is valid, run program of "X17"/"HIDEFILEX17"
D10 bit=1,also +1024 , means X18 input is valid, run program of "X18"/"HIDEFILEX18"
D11 bit=1,also +2048,means X19 input is valid, run program of "X19"/"HIDEFILEX19"
D12 bit=1,also +4096,means X20 input is valid, run program of "X20"/"HIDEFILEX20"
D13 bit=1,also +8192,means X21 input is valid, run program of "X21"/"HIDEFILEX21"
D14 bit=1,also +16384,means X22 input is valid, run program of "X22"/"HIDEFILEX22"
D15 bit=1,also +32768, means X23 input is valid, run program of "X23"/"HIDEFILEX23"

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.

Note: Current CNC only can transfer 2D dxf file to G code program.

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 is >=20.

Note:Modbus system can reach at 8 axes max, XYZABCXSYS display at parameter list, X:1st axis, Y:2nd axis,Z:3rd axis, A:4th axis, B:5th axis, C:6th axis, XS:7th axis,YS:8th axis.We write short name with XYZABCXSYS for feeding axes.Example: even if we can alter B-axis display to C-axis,we still use B-axis stands for 5th axis.use C-axis stands for 6th axis. Y-axis can change its display to C also.

6.2 Speed parameter

P	Parameter	EX-Value
1	G00 Speed of X-axis (mm/min)	6000.000
2	G00 Speed of Z-axis (mm/min)	6000.000
3	Manual Max Feeding Speed(mm/min)	10000.000
3-1	X_Manual Max Feeding Speed(mm/min)	0
3-2	Y_Manual Max Feeding Speed(mm/min)	0
3-3	Z_Manual Max Feeding Speed(mm/min)	0
3-4	A_Manual Max Feeding Speed(mm/min)	0
4	Auto Max Feeding Speed(mm/min)	15000.000
5	Default Speed of G01/G02/G03 (mm/min)	2000.000
6	Running Speed at Simulation Mode (mm/min)	10000.000
7	Manual Feeding Speed(mm/min)	100.00
8	Manual Spindle Speed (rpm)	0
9	Beginning Speed (mm/min)[>1]	500
10	Max Speed Increment (mm/min)[>1]	500
11	Limit G01/G02/G03 Speed of each axis [1:Yes, 0:No]	0
12	Max Speed of X_G1/G2/G3 (mm/min)	2000
13	Max Speed of Z_G1/G2/G3 (mm/min)	2000
14	X_Acceleration/Deceleration Constant [1～99999]	50000
15	Z_Acceleration/Deceleration Constant [1～99999]	50000
16	Acceleration/Deceleration Constant When Auto Run	2
17	Handwheel_Acceleration/Deceleration Constant [500--30000]	6000
18	Handwheel_Acceleration/Deceleration Constant when dry [>500]	60000
19	G00 Speed when dry with handwheel(mm/min) [>10]	1500
19-1	Start speed when dray with handwheel(mm/min)[>5]	100
20	X_Max Speed with Handwheel (mm/min)	4000
21	Z_Max Speed with Handwheel (mm/min)	4000
22	Z_Acceleration/Deceleration Constant when threading	0.000
23	X_Acceleration/Deceleration Constant when threading	50000.000
24	Ratio of Retired Speed in X-axis Servo when threading	4000
25	Beginning Speed of Retired in X-axis Step when threading	150
26	Max Retired Speed in X-axis Step when threading	6000
27	Type of Acceleration/Deceleration [0 mean line,8 mean curve]	0
28	Initial Acceleration/Deceleration Constant when curve [>=10]	100
29	Quadratic Acceleration/Deceleration Constant when curve [>=10]	500
30	Max Acceleration/Deceleration Constant when curve [>=500]	1
31	X_Homing Speed (mm/min)	2500.000
32	X_Speed for detect Z0 signal (mm/min)	250.000
33	Z_Homing Speed (mm/min)	2500.000
34	Z_Speed for detect Z0 signal (mm/min)	250.000
35	Min Speed of spindle when G96 (rpm)	1
36	Max Speed of 1st Spindle (rpm)	3000
37	Max Speed of Spindle at 2nd gear (rpm)	3000
38	Max Speed of Spindle at 3rd gear (rpm)	3000
39	Max Speed of Spindle at 4th gear (rpm)	3000
40	Max Speed of 2nd Spindle (rpm)	3000
41	Compensation Mode of Arc Backlash (0: A, 8: B)	8
42	Compensation Speed in B Mode (mm/min)	3000
42-1	Beginning Compensation Speed in B Mode (mm/min)	500
42-2	Acceleration/Deceleration Constant in B mode (mm/min/s)	60000
43	Handwheel_Stopping Speed (mm/min)[>100]	100
49	Activate Speed Processing Function[1:Yes, 0:No]	0

49-1	Stopping speed when reverse deceleration during running(mm/min)	200
50	Handwheel Stop 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	0
211	SP_Starting Speed_Tapping-Feed(rpm)	1
212	SP_Starting 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
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:Positive,1:Negative,2:near]	2
233	Control Mode of SP_homing	0
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 Z-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.

3,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.

4,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.

5,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)

6,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)

7,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.

8,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.

9,Beginning Speed of Feeding axes (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. P101~P104 are set for each axis alone.

Attention:Generally,stepper system<=100,servo system<=500.

10,Max Speed Increment (mm/min) of Feeding axes [>1]

It is max speed increment when multi-axial running track-interpolation. Also max changing value of speed. P111~P114 are set for each axis alone.

Example:when it is 300,the speed of feeding 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.

11,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.

12,Max Speed of X_G1/G2/G3 (mm/min)

It is for the Max running speed of X-axis when G1/G2/G3 interpolation.

13,Max Speed of Z_G1/G2/G3 (mm/min)

It is for the Max running speed of Z-axis when G1/G2/G3 interpolation.

14,X_Acceleration/Deceleration Constant [1～99999]

It is time constant of X-axis Acceleration/deceleration,the bigger it is ,the faster the ace/deceleration is.

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.

15,Z_Acceleration/Deceleration Constant [1～99999]

It is time constant of Z-axis Acceleration/deceleration, the bigger it is ,the faster the ace/deceleration is.

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.

16,Acceleration/Deceleration Constant When Auto Run [1-500]

It is for set constant of Acceleration/deceleration 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.

17,Handwheel_Acceleration/Deceleration Constant [500--32000]

It is for set constant of Acceleration/deceleration of Handwheel.the range is 500-32000.

18,Handwheel_Acceleration/Deceleration Constant when dry [>500]

It is for set constant of Acceleration/deceleration when handwheel start program. the range is 500-32000.when the value is less than 500,it is invalid.

19,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.

19-1,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.

20,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.

21,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.

22,Z_Acceleration/Deceleration Constant when threading

It is time constant of Z-axis ace/deceleration when do threading , Range:300~90000.The bigger it is,the faster the acceleration/deceleration.

Attention:with stepper system,it should be set smaller. When <300,it is invalid.When with servo motor ,for ensure efficiency,set 0.

23,X_Acceleration/Deceleration Constant when threading

It is time constant of Z-axis ace/deceleration when do threading, the range is from 300 to

90000. The bigger it is,the faster the acceleration/deceleration.

Attention:with stepper system,the smaller,the safer.when <300, it is invalid. When with servo motor, for ensure efficiency, set 0.

24,Ratio of Retired Speed in X-axis Servo when threading

It is ratio of X axis retried speed that configured with servo system when threading. Unit: *0.1 .

25,Beginning Speed of Retired in X-axis Step when threading

It is beginning speed of X axis retired that configured with step system when threading.

Attention: for safe,it should be less than 100mm/min.

26,Max Retired Speed in X-axis Step when threading

It is Max speed of X axis retired that configured with step system when threading.

Attention: it is related to load,the bigger the load,the smaller it is.

27,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.

28,Initial Acceleration/Deceleration Constant when curve [>=10]

It is initial acceleration/deceleration constant when P27 set curve type.Range: >=10.

29,Quadratic Acceleration/Deceleration Constant when curve [>=10]

It is quadratic constant of acceleration/deceleration when P27 set curve type. Range >=10.

30,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/deceleration constant is with line type of each axis.

31,X_Homing Speed (mm/min)

It is homing speed of X-axis.Unit:mm/min. the range is less than X_G00 speed.

32,X_Speed for detect Z0 signal (mm/min)

It is speed for check Z0 pulse signal after X-axis reach at homing switch. 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.

33,Z_Homing Speed (mm/min)

It is homing speed of Z-axis.Unit:mm/min. the range is less than Z_G00 speed.

34,Z_Speed for detect Z0 signal (mm/min)

It is speed for check Z0 pulse signal after Z-axis reach at homing switch. 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.

35,Min Speed of spindle when G96 (rpm)

It is the min speed of spindle when run G96 code , also at constant surface speed mode.

36,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).

37,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.

38,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.

39,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.

40,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.

41,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.

42,Compensation Speed in B Mode (mm/min)

It is the compensation speed in B compensation mode.unit:mm/min.

42-1,Beginning Compensation Speed in B mode (mm/min)

It is beginning compensation speed in B compensation mode.it is valid when it >10.

42-2,Acceleration/Deceleration Constant in B mode (mm/min/s)

It is acceleration/deceleration constant in B compensation mode (P41=8/12) . Range: >=10.

43,Handwheel_Stopping Speed (mm/min)[>100]

It is the speed when handwheel stop. the bigger it is ,the faster handwheel stop.

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,Stopping speed when reverse deceleration during running(mm/min)

It sets stopping speed when CNC do reverse deceleration during running.Unit:mm/min.

50,Handwheel Stop Speed(mm/min)[>100]

It sets the starting speed that 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. Set to 28 means activate all functions, 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. ***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 threading[0:No, 1:Yes]
It is set for that if wait speed of spindle is smooth when threading. 1: yes, wait speed of spindle is smooth before threading. 0: No, don’t wait.

211,SP_Starting Speed_Tapping-Feed(rpm)
It is starting speed of spindle axis for feeding when tapping.

212,SP_Starting 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.

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: Override Switch]	0
2	Switch Type for SP-Rate [0: Key, 1: Override 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 Z_Negative direction (mm)	-9999.000
6	Max Travel in Z_Positive direction (mm)	9999.000
7	SP_Braking Time (10ms)	150
8	SP_Braking is Long Signal [0: No,1: Yes]	0
9	Detect SP_Position Feedback [0: No,1: Yes]	1
10	Pulses Per Revolution of Spindle	4096
10-1	Allow Error of Resolution of SP_Encoder[>10]	0
10-2	Pulses Per Revolution of Spindle[1/1:0,Other:>99]	0
11	Soft-Limitation is valid [0:Yes, 1: No][D2X;D3C(Y);D4Z;D5A]	0000000100000001
12	X_Reverse Backlash Compensation (um)	0
13	Z_Reverse Backlash Compensation (um)	0
14	X_Direction [1:normal, 0: Reverse]	0
15	Z_Direction [1:normal, 0: Reverse]	0
16	Using Electron Gear Ratio for Feeding Axes [0:Yes, 1:No]	0
17	Numerator of X_Electron Gear	1
18	Denominator of X_Electron Gear	1
19	Numerator of Z_Electron Gear	1
20	Denominator of Z_Electron Gear	1
21	Type of Limit Switch in Positive Direction[0:NO type, 1:NC type]	0
22	Type of Limit Switch in Negative Direction[0:NO type, 1:NC type]	0
23	Type of Home [D3X;D4(C)Y;D5Z;D6A;0:Switch;1:float Zero]	01111011
24	X_Machine Coordinate of float zero point	
25	Z_Machine Coordinate of float zero point	
26	Grade of Homing[1:No,0:Prompt,8:Compulsion,9:Super compulsion]	1
27	Mode of Homing	1
28	Direction of Homing[D2:X,D3:C(Y),D4:Z,D5:A]	0000000100000001
29	Type of Switches for Homing	00000000
30	Range of Detect Z0 in X axis	45
31	Range of Detect Z0 in Z axis	60
32	Offset after homing in X axis	0
33	Offset after homing in Z axis	0
50	Spindle is rotating when shift gear[1:Yes, 0:No]	0
51	Rotating Speed of Spindle when shift gear(1/100rpm)	100
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	SP-CCW key with JOG output(M04)[8:YES]	0
66	Gear Shift Control on Spindle Axis[1:Yes, 0:No]	1
68	Time when feeding axis change direction (ms)	0
69	Mode of Manual Return Home(1:All axis, 0:Selected Axis)	0
80	X/Z axis is Rotating Axis	00000001
100	System Inner Parameter	8
101	Name of 3rd Axis [0:Y, 1:C]	1

102	Mode of Y(C) Axis [0:Rotating Axis, 1: Linear Axis]	1
104	C(Y)_Direction [1:normal, 0: Reverse]	0
105	Numerator of C(Y)_Electronic Gear	1
106	Denominator of C(Y)_Electronic Gear	1
107	C(Y)_Reverse Backlash Compensation (um)	0
108	G00 Speed of C(Y) Axis	4000
109	Max Speed of C(Y)_G1G2G3	2000
110	C(Y)_Acceleration/Deceleration Constant	50000
111	C(Y)_Max Speed with Handwheel (mm/min)	2000
112	Speed of C-axis return to Zero point of Encoder(º/min)	250.000
113	Y_Homing Speed (mm/min)	5000.000
114	Y_Speed for Detect Z0 signal (mm/min)	250.000
115	Range of detect Z0 in Y axis	80
116	Offset after homing in Y axis	0
117	Max Travel in C(Y)-Negative Direction	-9999.000
118	Max Travel in C(Y)-Positive Direction	9999.000
119	C(Y)_Machine Coordinate of float zero point	
200	System Inner Parameter	1
201	Mode of A Axis [0:Rotating Axis, 1: Linear Axis]	1
202	Base when A axis is rotating axis	1
203	A_Direction [1:normal, 0: Reverse]	1
204	Numerator of A_Electronic Gear	1
205	Denominator of A_Electronic Gear	1
206	A_Reverse Backlash Compensation (um)	0
207	G00 Speed of A Axis	4000
208	Max Speed of A_G1G2G3	2000
209	A_Acceleration/Deceleration Constant	50000
210	A_Max Speed with Handwheel (mm/min)	2000
211	Homing Speed in A Positive Direction (mm/min)	2500.000
212	Homing Speed in A Negative Direction (mm/min)	250.000
213	Range of detect Z0 in A axis	100
214	Offset after homing in A axis	0
215	Max Travel in A-Negative Direction	-9999.000
216	Max Travel in A-Positive Direction	9999.000
217	A_Machine Coordinate of float zero point	
218	Automatically Output(Y/M) for unclamp when 4^{th} Axis is running	0
219	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	92
411	Control Mode of Tapping [0:Following, 1: Interpolation]	0
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
501	Advanced distance for Z-axis with M123M125(um)[>9/<-9]	0
502	Rough allowance when advance with M123M125(um)[radius]	0

503	Max feed speed when rough processing with M123M125(mm/min)[>9]	0

Explanation about Axis Parameter:

- ***P81-P500 is the parameters for C(Y)axis, A axis (only when CNC system is configured with C(Y) & A axis, related parameters are effective), take X-axis/Z-axis as reference.***

1,Switch Type for Feed-Rate [0: Key, 1: Override Knob]

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 Override knob 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: Override Knob]

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 Override knob 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)

It is max travel in negative direction of X axis when soft-limitation, which is based on machine coordinate system.

4,Max Travel in X_Positive direction (mm)

It is max travel in positive direction of X axis when soft-limitation, which is based on machine coordinate system.

5,Max Travel in Z_Negative direction (mm)

It is max travel in negative direction of Z axis when soft-limitation, which is based on machine coordinate system.

6,Max Travel in Z_Positive direction (mm)

It is max travel in positive direction of Z axis when soft-limitation, which is based on machine coordinate system.

7,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.

8,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.

9,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.

10,Pulses Per Revolution of Spindle

It is pulses per revolution of spindle. Pulses= (Resolution of SP-encoder) * 4.

10-1,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

10-2,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.

11,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	C(Y)	X	-	-

0: means soft-limitation is valid , 1: means invalid

Example:the soft limit of X-axis is valid, P11= 00000100.

12,X_Reverse Backlash Compensation (um)

It is the value of reverse backlash compensation of X-axis, Radius designation. When direction of X-axis movement is changed ,system will make compensation with the value automatically. Unit: um

13,Z_Reverse Backlash Compensation (um)

It is the value of reverse backlash compensation of Z-axis.When direction of Z-axis movement is reversed ,system will make compensation with the value automatically. Unit: um

14,X_Direction [1:normal, 0: Reverse]

It is for set the direction of X-axis. 1: Direction of X-axis is same to direction of code; 0: Direction of X-axis is opposite to direction of code.

15,Z_Direction [1:normal, 0: Reverse]

It is for set the direction of Z-axis. 1: Direction of Z-axis is same to direction of code; 0: Direction of Z-axis is opposite to direction of code.

16,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.

17,Numerator of X_Electron Gear (1-999999)

It is Numerator of X-axis’s electron gear ratio.(X_CMR) Range: 1-999999.

18,Denominator of X_Electron Gear (1-999999)

It is Denominator of X-axis’s electron gear ratio.(X_CMD) Range: 1-999999.

19,Numerator of Z_Electron Gear (1-999999)

It is Numerator of Z-axis’s electron gear ratio. (Z_CMR) Range: 1-999999.

20,Denominator of Z_Electron Gear (1-999999)

It is Denominator of Z-axis’s electron gear ratio.(Z_CMD) 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.

$$\text{Electron Gear Ratio} = \frac{\text{Numerator}}{\text{Denominator}} = \frac{CMR}{CMD} = \frac{P}{L*1000}$$

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:

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.

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.

21,Type of Limit Switch in Positive Direction[0:NO type, 1:NC type]
It is set type of limit switch in positive direction, also type of switch that is connected to +L,PIN16_CN3 plug. 0:NO Type, 1: NC Type.

22,Type of Limit Switch in Negative Direction[0:NO type, 1:NC type]
It is set type of limit switch in negative direction, also type of switch that is connected to -L,PIN15_CN3 plug. 0:NO Type, 1: NC Type.

23,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	C(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.
D0&D1 bits are set coordinate vase of Y-axis

24,X_Machine Coordinate of float zero point
It is set the machine coordinate value of X-axis based on float zero point. The value is distance between current position of machine tool & float zero point.

25,Z_Machine Coordinate of float zero point
It is set the machine coordinate value of Z-axis based on float zero point. The value is distance between current position of machine tool & float zero point.

26,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.

27,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.

28,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	C	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.

29,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	C(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.

D6=1: Handwheel/Auto status shift automatically.

D7=1:Manual/Auto shift automatically ;when end of program,CNC will enter manual status automatically, when press Start key on manual status, CNC will enter Auto status automatically.

30,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.

31,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.

32,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.

33,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

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. 64: keep status of spindle as before gear shift.

51,Rotating Speed of Spindle when shift gear(1/100rpm)
It is rotating speed of spindle when spindle shift gear & P51=1. Unit: 1/100rpm

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,SP-CCW key with JOG output(M04)[8:YES]
It sets whether set SP-CCW key and CNC output M04 with JOG mode. 8:Yes.

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 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	-	-

100,System Inner Parameter
Inner parameter of system, cannot be altered.

101,Name of 3rd Axis [0:Y, 1:C]
It is set for name of 3rd axis, 0: set to Y , display & programming with “Y”, normally when 3rd axis is linear axis; 1: set to C, display & programming with “C”, normally when 3rd axis is rotating axis.

102,Mode of Y(C) Axis [0:Rotating Axis, 1: Linear Axis]
It is set for the mode of Y(C)-axis, 0: Rotating axis , 1: Linear axis.

104,C(Y)_Direction [1:normal, 0: Reverse]
It is for set the direction of C(Y)-axis. 1: Direction of C(Y)-axis is same to direction of code; 0: Direction of C(Y)-axis is opposite to direction of code.

105,Numerator of C(Y)_Electronic Gear
It is Numerator of C(Y)-axis’s electron gear ratio. (C_CMR) Range: 1-999999.

106,Denominator of C(Y)_Electronic Gear

It is Denominator of C(Y)-axis's electron gear ratio.(C_CMD) Range: 1-999999.

107,C(Y)_Reverse Backlash Compensation (um)

It is the value of reverse backlash compensation of C(Y)-axis.When direction of X-axis movement is changed ,system will make compensation with the value automatically. Unit: um

108,G00 Speed of C(Y) Axis (mm/min)

It is rapid speed(also speed of G00) of C(Y) axis,Max is 240000(unit:mm/min)

Attention: the value depends on machine configuration,set wrong is very easy to trouble machine tool & accident.

109,Max Speed of C(Y)_G1G2G3

It is for the Max running speed of C(Y)-axis when G1/G2/G3 interpolation.

110,C(Y)_Acceleration/Deceleration Constant

It is time constant of X-axis Acceleration/deceleration,the bigger it is ,the faster the ace/deceleration is.

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.

111,C(Y)_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.

112,Speed of C-axis return to Zero point of Encoder(°/min)

It is the speed of C-axis return to Z0 of Encoder. Unit:°/min. the range is less than the G00 speed of C-axis.

113,Y_Homing Speed (mm/min)

It is homing speed of Y-axis .Unit:mm/min. the range is less than Y_G00 speed.

114,Y_Speed for Detect Z0 signal (mm/min)

It is speed for check Z0 pulse signal after Y-axis reach at homing switch. Unit:mm/min. the range is 20-500.

Note:For ensure accuracy.the smaller it is ,the higher the accuracy is.when set well,don't change it forever.

115,Range of detect Z0 in C(Y) axis (100um)

It is range that system can detect Z0 signal of encoder in C(Y) direction.

Attention:the value must be less than the length of one rev,otherwise homing failure.

116,Offset after homing in Y axis (Unit:10um)

It is offset that Y-axis after homing. Move with G00 speed. Unit: 0.01mm

117,Max Travel in C(Y)-Negative Direction

It is max travel in negative direction of C(Y) axis when soft-limitation, which is based on machine coordinate system.

118,Max Travel in C(Y)-Positive Direction

It is max travel in positive direction of C(Y) axis when soft-limitation, which is based on machine coordinate system.

119,C(Y)_Machine Coordinate of float zero point
It is set the machine coordinate value of C(Y)-axis based on float zero point. The value is distance between current position of machine tool & float zero point.

200,System Inner Parameter
Inner parameter of system, cannot be altered.

201,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.

202,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.

203,A_Direction [1:normal, 0: Reverse]
It is for set the direction of A-axis. 1: Direction of A-axis is same to direction of code; 0: Direction of A-axis is opposite to direction of code.

204,Numerator of A_Electronic Gear
It is Numerator of A-axis's electron gear ratio. (A_CMR) Range: 1-999999.

205,Denominator of A_Electronic Gear
It is Denominator of A-axis's electron gear ratio.(A_CMD) Range: 1-999999.

206,A_Reverse Backlash Compensation (um)
It is the value of reverse backlash compensation of A-axis.When direction of A-axis movement is changed ,system will make compensation with the value automatically. Unit: um

207,G00 Speed of A Axis
It is rapid speed(also speed of G00) of A axis,Max is 240000(unit:mm/min)
Attention: the value depends on machine configuration,set wrong is very easy to trouble machine tool & accident.

208,Max Speed of A_G1G2G3
It is for the Max running speed of A-axis when G1/G2/G3 interpolation.

209,A_Acceleration/Deceleration Constant
It is time constant of A-axis acceleration/deceleration,the bigger it is ,the faster the ace/deceleration is.
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.

210,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.

211,A_Homing Speed (mm/min)
It is homing speed of A-axis .Unit:mm/min. the range is less than A_G00 speed.

212,A_Speed for Detect Z0 signal (mm/min)
It is speed for check Z0 pulse signal after A-axis reach at homing switch.

213,Range of detect Z0 in A axis
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.

214,Offset after homing in A axis
It is offset that A-axis after homing. Move with G00 speed. Unit: 0.01mm

215,Max Travel in A-Negative Direction
It is max travel in negative direction of A axis when soft-limitation, which is based on machine coordinate system.

216,Max Travel in A-Positive Direction
It is max travel in positive direction of A axis when soft-limitation, which is based on machine coordinate system.

217,A_Machine Coordinate of float zero point
It is set the machine coordinate value of A-axis based on float zero point. The value is distance between current position of machine tool & float zero point.

218,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.

219,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,o 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 threading[91 X,92 Y/C,93 Z,94 A]
It is the axis that be use for spindle when interpolation tap.

411,Control Mode of Threading Cycle [0:Following, 1: Interpolation]
[0 follow encode;4 interpolation to SP]
It is control mode of interpolation tap. 0: do threading following spindle encoder, 4: do threading cycle with interpolate axis,which is set by P410.

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.

501,Advanced distance for Z-axis with M123M125(um)[>9/<-9]
It sets advanced distance for Z-axis when use M123/M125 follow-up codes.Range:>9/<-9.

502,Rough allowance when advance with M123M125(um)[radius]
It sets rough allowance when advance with M123/M125 code, radius value,unit:um.
503,Max feed speed when rough processing with M123M125(mm/min)
It sets max feeding speed when rough processing with M123/M125 code,unit: mm/min.

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	Activate ATC Function(1:Yes, 0:No)	1
2	Tool Number of Electric Turret (+1)	5
3	Type of Lathe Machine	0
4	Max Time of Turret Positive Rotation(s)	8.000
5	Delay Time before detect after rotating(ms)	100.000
6	Delay Time after turret stop positive rotation(ms)	100.000
7	Time of Turret Negative Rotation for lock(ms)	1200.000
8	Management of Tool(0:M06,1:T code,+32768:Tool life management)	1
9	Detect Position of Lock (1: Yes, 0:No)	0
10	Mode of Setup Radius C Compensation	0
11	Mode of Cancel Radius C Compensation	0
20	Mode of Turret(1:Electric Turret; 0:Special Turret)	1
32	Filtering for Position signal or WAT Signal	1283

Explanation about Tool Parameter:

1,Active ATC function [1:Yes, 0: No]

It is for whether activate function of tool changer automatically. 0: No, lathe machine is without turret for tool magazine 1: yes, lathe machine is with turret for tool change, after tool is on position,no detect position signal of tool. 8: yes, after tool is on position, system will detect position signal of tool again after tool is on position.

Attention: 1.when the machine tool is only with linear turret ,the parameter is set to 0;

2.Set sum of tools,Press "C"key and input sum in Redeem(tool compensation) screen.

2,Tool Number of Electric Turret (+1)

It is total tool number of electric turret.The value of this parameter needs to add 1 based on actual tool number. Example:When with 4 tools on turret, set P2=5.

When lathe machine is configured with 4 tools on electric turret and 4-linear tools. On "Redeem" screen, press "C" key to set total tools, which is 8. and the parameter is 5, so T1-T4 means tools of electric turret,T5-T8 means tools of linear turret.

3,Type of Lathe Machine

It is type of lathe machine, also structure of machine tool.

0:turret in front of horizontal lathe; 1:turret behind of horizontal lathe;

8:turret in front of vertical lathe; 9:turret behind of vertical lathe.

Note: There are related introductions about type of lathe on chapter2.1.1, this parameter also setup different machine Coordinate system for machine tool.

4,Max Time of Turret Positive Rotation(s)

It is max time of turret rotates in positive direction to find position signal of tool. When system didn't find position signal of tool within the setting time, it will stop changing tool & rotation and alarm. Unit: second.

5,Delay Time before detect after rotating(ms)

It is delay time to detect position signal of tool after turret rotates in positive direction. Unit: millisecond.

6,Delay Time after turret stop positive rotation(ms)

It is delay time after turret stop rotating in positive direction, also time after reset (+T_CN4) signal , and before output (-T_CN4) signal. Unit: ms.

7,Time of Turret Negative Rotation for lock(ms)

It is the time of turret rotates in negative direction for locking turret , also the time of output (-T_CN4) signal.

Attention:The value is related to size of turret on machine tool. Motor on turret would be over-heat when the value is too big.

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.

9,Detect Position of Lock (1: Yes, 0:No)

It sets whether system detect position signal of lock. 1:Yes, detect, 0:No detect. Input point of detection is TOK, PIN9 of CN4 Plug.

10,Mode of Setup Radius C Compensation (0:A type, 1:B type)
11,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.

20,Mode of Turret(1:Electric Turret; 0:Special Turret)

It sets mode of turret. When with special turret, it needs to design “ProgramTool” & PLC for special turret.

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

+1: Rotating turret with nearest way to select tool,otherwise rotating turret with positive direction(Output +T) and select tool.

6.5 Other Parameter

P	Other Parameter	Ex-Factory
1	Type of Handwheel(0:Panel, 1:handhold)	0
2	Type of Chuck	0
3	Using Interface Switch on Panel(0: No, 88:Yes)	88
4	Lubricate Automatically (0:Yes, 1:No)	0
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	1000010001000101
10	Counting Workpieces Automatically(0:No, 1:Yes)	1
11	Increment of shift block	1
12	System Inner Parameter	9
13	Interlock between Rotation_SP & Chuck(0:No, 1:Yes)	1
13-1	Interlock between Rotation_SP & Tailstock(0:No, 1:Yes)	1
14	Coolant key is valid on Auto(0:No, 1:Yes)	1
15	Detect Position of Chuck(M10/M11)(0:No, 1:Yes)	0
16	Detect Position of Tailstock(M79/M78)(0:No, 1:Yes)	0
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
21	Control Mode of Tailstock(0:Single, 1:Double)	0
22	External Switch for Chuck(0:No, 1:Yes)	0
23	External Switch for Tailstock(0:No, 1:Yes)	0
24	Time of Chuck(s)	0.00
24-1	M10 Long Signal(0:No, 1:Yes)	0
24-2	M71 Long Signal(0:No, 1:Yes)	0
24-3	Chuck M10 or M11 boot [0:M10, 1:M11]	0
25	Time of Tailstock(s)	0.00
25-1	M79 Long Signal(0:No, 1:Yes)	0
25-2	M73 Long Signal(0:No, 1:Yes)	0
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
29-1	Alarm Output(M67) output M63 with delay time(0:No, 1:Yes)	0
29-2	Delay time before output M63	1800
30	Language(1:Chinese, 0:English)	0
31	Use Inner PMC(0:No, 1:Yes)	1
32	Use High-Speed Inner PMC(0:No, 18:Yes)	18
33	HY as Running Key(0:No, 1:Yes)	0
34	HA as Halt Key(0:No, 1:Yes)	0
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]	0
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	1
141	M78:Machine Coordinate VS Reference point 2	0.000
150	X_Reference Point_1(mm)	0.000
151	Y_Reference Point_1(mm)	0.000
152	Z_Reference Point_1(mm)	0.000
153	A_Reference Point_1(mm)	0.000
160	X_Reference Point_2(mm)	0.000
161	Y_Reference Point_2(mm)	0.000
162	Z_Reference Point_2(mm)	0.000
163	A_Reference Point_2(mm)	0.000
501	Shift Color Display of Screen(1:No, 8:Yes)	1
601	Define Parameters for Step	
602	Define Parameters for Servo	Servo
900	Display User-define Dialog Box[1:No, 4:Yes]	1
901	Homing Sequence of Axis(5bits)	1
902	Inner Parameter[2:ON]	0
903	Inner Parameter[2:SD Card,+16:5th Axis Follow,+32:4th Axis Follow]	0
904	Inner Parameter[4:File Decrypt,8:Encryption]	0
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 Other Parameter:

1,Type of Handwheel(0:Panel, 1:handhold)

It sets the type of handwheel,0:Handwheel in operational panel,1:Handwheel in handhold box.

Note: when the parameter is 1 (P1=1), CN11 is connected to handhold box; & can’t be used as SP-rate & feed-rate,only off/X/Y/Z/A/B &*1/*10/*100(also P1&P2=0 in Axis parameter if use CN11 as rate knob input)

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 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,when CNC/Spindle is in the status of M05 , chuck can clamp/unclamp;

8:fully interlock, When M05&SPRPM(spindle encoder) is 0,chuck can clamp/unclamp;

16:super interlock, Interlock by detecting rotating speed

32:Chuck & Chuck Key are invalid when CNC is running program without Pause;

56:Compulsion interlock. Chuck & Chuck Key are invalid when CNC is on Auto mode;

64:No interlock when Auto, Interlock when Manual.

Suggest set to interlock for safe.

13-1,Interlock between Rotation_SP & Tailstock (0:No, 1:Yes)

It sets interlock between rotation of spindle and tailstock(M79). The same principle as Spindle chuck, please check P13 on Other parameter.

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.

15,Detect Position of Chuck(M10/M11)(0:No, 1:Yes)

It sets if detect position of chuck. 0:No detect; 1:Yes,detect.

If P15=1, M12,PIN11_CN10 Plug, position input for Chuck(Clamp/M10); M22 , PIN5_CN10 plug, position input for Chuck(Loose/M11), after in position,then CNC do next step.

Attention: It is same pin(M12) of check of Chuck-clamp&Door-switch, only one usage is valid.If check chuck clamp if is in position,also cannot be used as check of Door-switch.

It is same pin(M22) of check of Chuck-loose& orientation end,only one usage is valid. If check chuck-loose if is in position,also cannot be used as orientation end signal input.

16,Detect Position of Tailstock(M79/M78)(0:No, 1:Yes)

It sets if detect position of tailstock. 0:No detect; 1:Yes, detect;

If P16=1, M18,PIN10_CN10 Plug, position detection input for Tailstock(Forward/M79); M28, PIN23_CN10 Plug, position detection input for Tailstock(Backward/M78). After valid, then CNC do next step.

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.

21,Control Mode of Thumbstall(0:Single, 1:Double)

It sets the control mode of tailstock, 0:Single control signal for thumbstall, also tailstock; 1:Double control signal for Thumbstall.

P21=0, one output point for thumbstall, M79:thumbstall forward, M78, also M79 is invalid: thumbstall backward;

P21=1, two output points for thumbstall, M79: output M79(PIN22_CN3 Plug) to forward thumbstall; M78: output M73(PIN22_CN10 Plug) to backward thumbstall.

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;anther 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.

24-1,M10 Long Signal(0:No, 1:Yes)

It sets the control mode of M10, 0 means short signal, holding time of output M10 can be set by parameter, 1 means long signal, M10 always valid

24-2,M71 Long Signal(0:No, 1:Yes)

It sets the control mode of M71, 0 means short signal, holding time of output M71 can be set by parameter, 1 means long signal, M71(M11) always valid.

24-3,Chuck M10 or M11 boot [0:M10, 1:M11]

It sets CNC system output M10 or M11 when booting system, 0:Output M10, 1:Output M11.

Note: If double outputs for chuck, M11,also output M71, PIN9_CN10 Plug.

25,Time of Tailstock(s)

It sets holding time of output M79/M78 for tailstock. Unit:second. 0 means M79/M78 are long signal, also always output M79/M78 is valid.

25-1,M79 Long Signal(0:NO, 1:Yes)

It sets the control mode of M79, 0 means short signal, holding time of output M79 can be set by parameter, 1 means long signal, M79 always valid

25-2,M73 Long Signal(0:NO, 1:Yes)

It sets the control mode of M73(M78), 0 means short signal, holding time of output M73 can be set by parameter, 1 means long signal, M73(M78) always valid.

Note:If double outputs, M78, also output M73, PIN22_CN10 Plug.

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.

29-1,Alarm Output(M67) output M63 with delay time(0:No, 1:Yes)

It set if output M63 with one delay time when Alarm output (M67), 0: No, 1:Yes.

29-2,Delay time before output M63

It sets delay time before output M63 after M67 alarm output.

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, 18:Yes)

It sets if use high-speed inner PMC for I/Os. 0:No,don't use PMC; 18: Yes,use High-Speed PMC. 28:Yes, use super high-speed PMC.

Warning:It is usually used for adjusting parameters. system must use High-Speed PMC when actual use,also P32=1. Otherwise system will works abnormally.

33,HY as Running Key(0:No, 1:Yes)

It sets if make HY input point, PIN9_CN11 plug as external key for RUN signal. 0:No, HY don't as RUN input signal; 1:Yes,HY as RUN signal.

Attention: Because HY signal maybe as Y(C)-axis selection signal, so when P33=1,then P1 in Axis&Other parameter only set as 0.

34,HA as Halt Key(0:No, 1:Yes)

It sets if make HA input point, PIN10_CN11 plug as external key for STOP signal. 0:No, HY don't as Halt input signal; 1:Yes,HA as Halt signal.

Attention:Because HA signal maybe as A-axis selection signal, so when P33=1,then P1 in Axis&Other parameter only set as 0.

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 displayer.

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 C(Y)_Manual Feeding is opposite;
+16	Direction of X_Manual Feeding is opposite;
+32	Direction of A_Manual Feeding is opposite;

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 1^{st} 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 2^{nd} reference point

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, 12:All]

It sets if display user-define dialog box. 1: No display; 4:Yes,display some.4108: Enter with page, +256: US0~54 are user coordinate, +8192: Shift between Coolant key & Rapid key

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 Lathe 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(C)_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(C)_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(C)_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(C)_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(C)_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(C)_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(C)_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(C)_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.

3. 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(C)_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(C)_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/C"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).

Axis Coordinate value set for all coordinate system:

1) input A###, current axis coordinate offset for all coordinate system, Example: input A12.5 on P1-2(G54-Y-axis),Y-axis of all coordinate system will offset 12.5mm.

2) Input E , clear to 0; input E###, means input absolute offset value.

3) Input EA, clear all coordinate value of current axis; input EA###, means input absolute offset value to all coordinate system for current axis.

4) On X-axis of anyone coordinate system when parameter set, input EALL, clear all coordinate coordinate value to 0.

Note: There are 54 pieces of workpiece coordinate system:G54~G59 & G54.1~G54.48.

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,curry word time: (days)

8,Version of Operational Software.

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
J-Length	Press "J" key to enter Length Compensation Interface
V-ACLEA	Press "V" 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.These parameters are used for set radius value of tool tip.

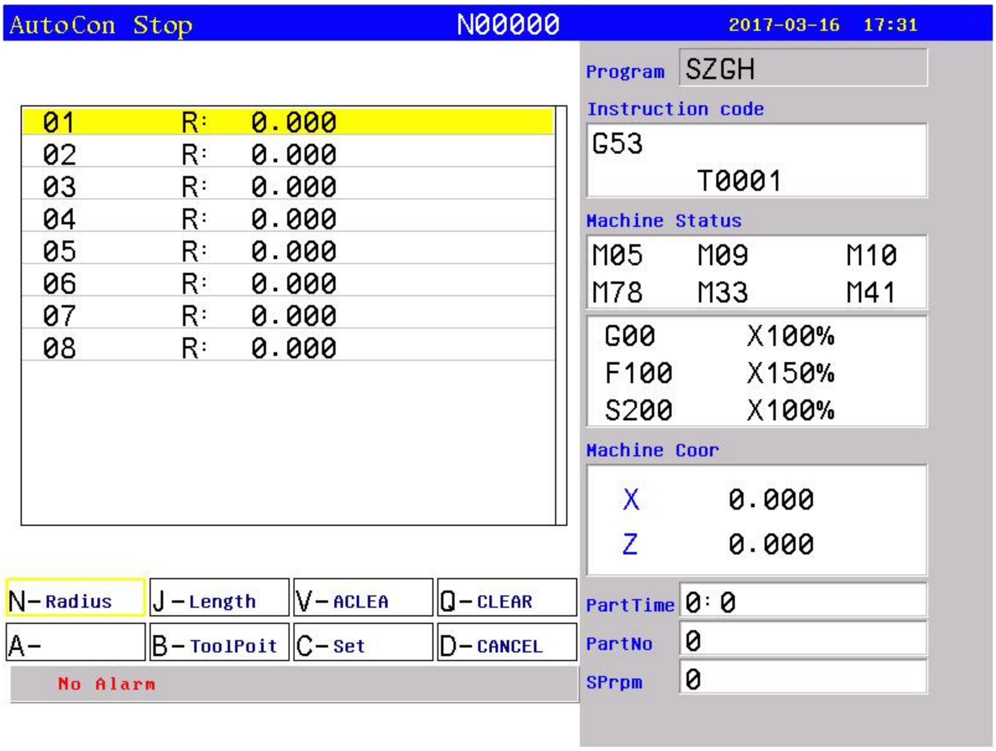


Fig6.8.1 Radius Compensation Interface

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.

If user want to initialize radius offset of all tools or current tool, press "V" key to clear radius offset of all tools, Press "Q" key to clear radius value of current tool.

Note: 1.Value input is with absolute type.

2.Tool offset can be called at any ease with one tool(Tab code), radius offset number is corresponding to tool offset number,also call tool offset with related tool radius offset, Example: T0102, No.2 tool offset, CNC adopt No.2 tool radius offset.

6.8.2 Length of redeem

Press “J” to enter Length compensation interface on Redeem.

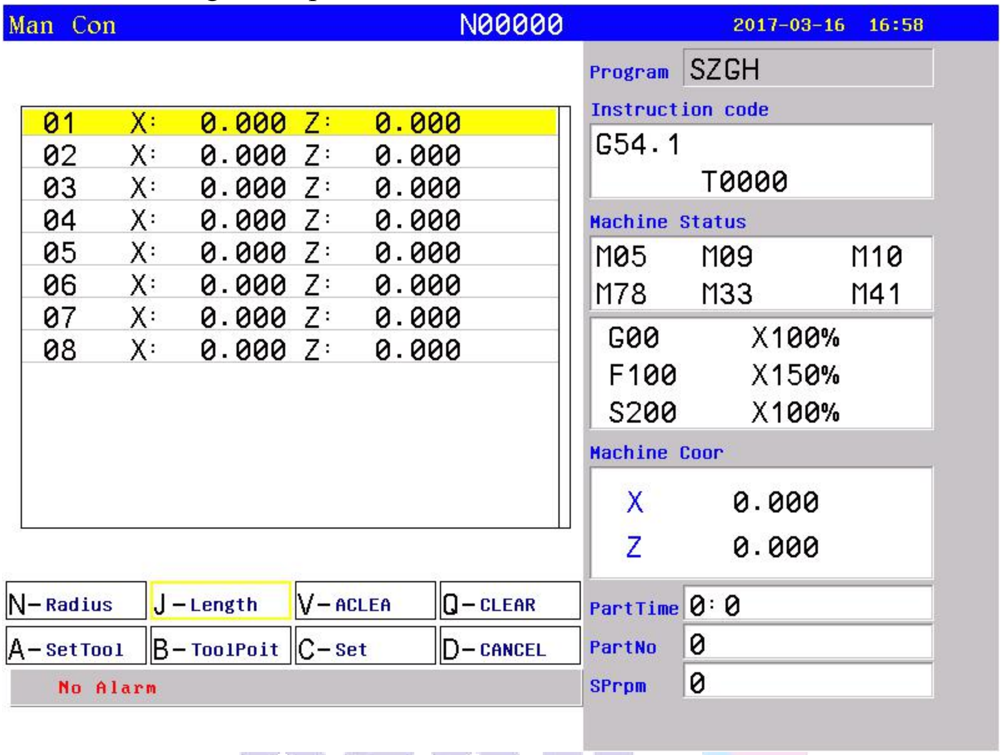


Fig6.8.2 Length Compensation Interface

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 machine tool to a position where is easy to measure coordinate of tools

2)Press “↑ ↓” to move cursor to corresponding tool number

3)Press “A” to popup a dialog box, “input axis name:[X,Z]”

4)Press X/Z key, “Input Coordinate Axis(mm): X/Z”, 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 “V” 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.

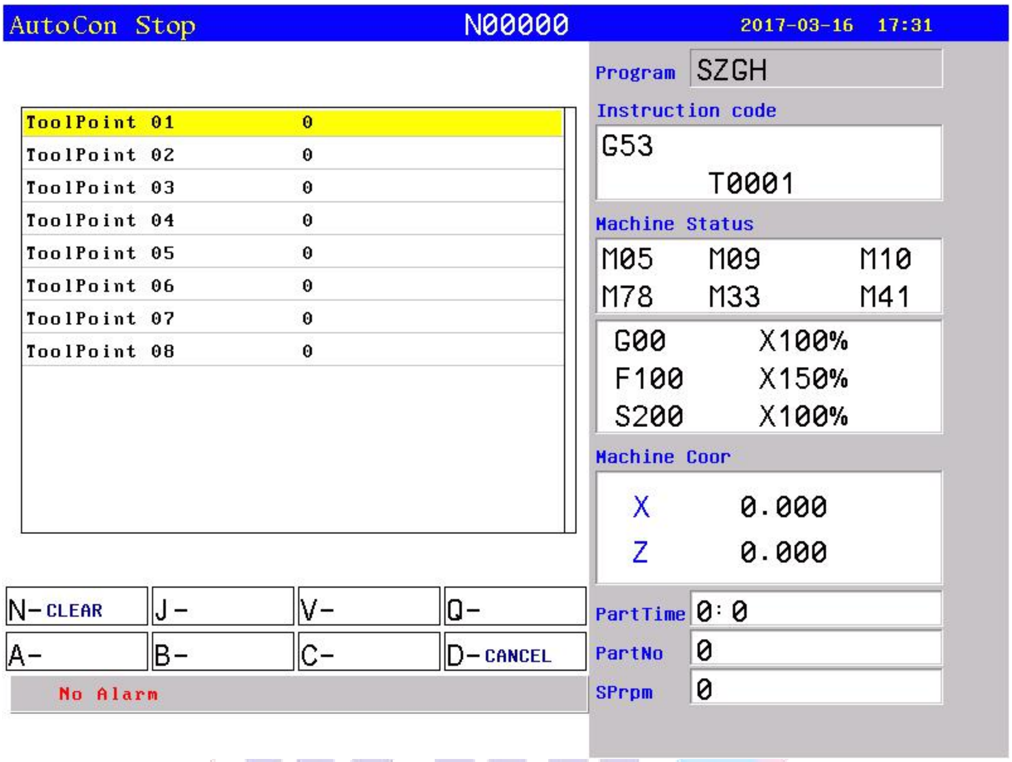


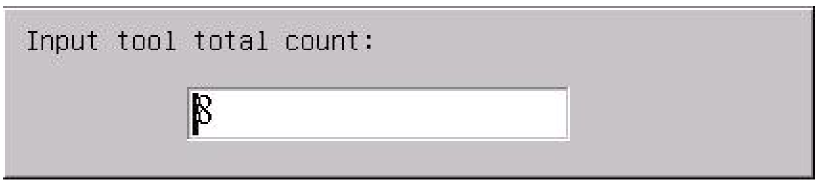
Fig6.8.3 Tool Posit Interface

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.

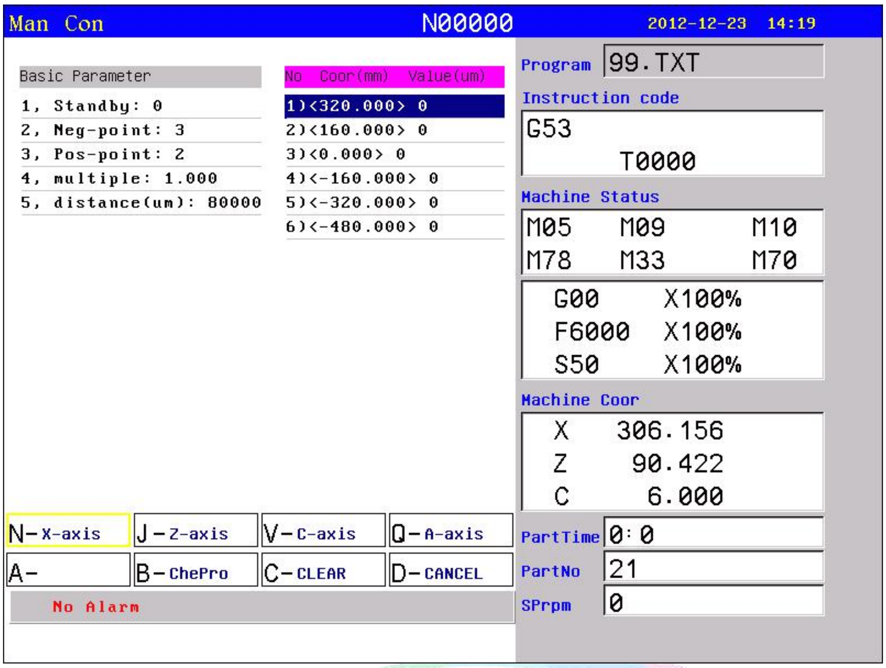


Fig6.9.1 Screw compensation interface

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/Z/C/A axis separately, by pressing “N” “J” “V” “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” “J” “V” “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.

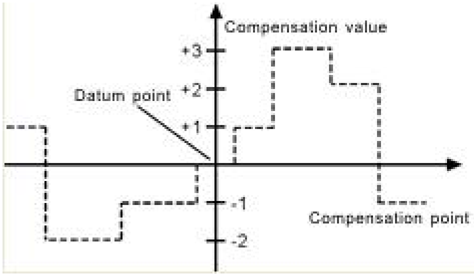
4. Multiple.

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

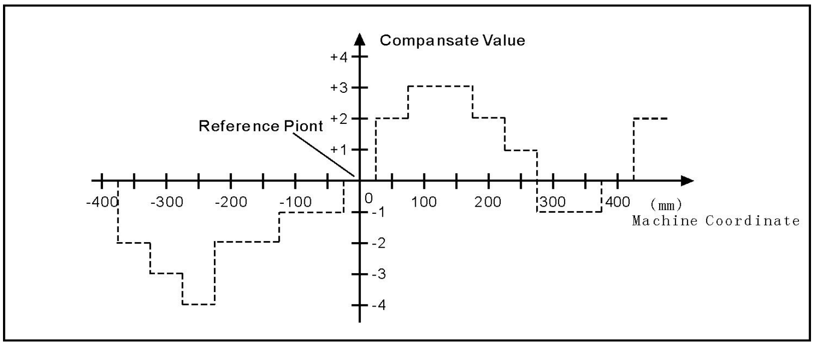
3)Multiple: 1

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

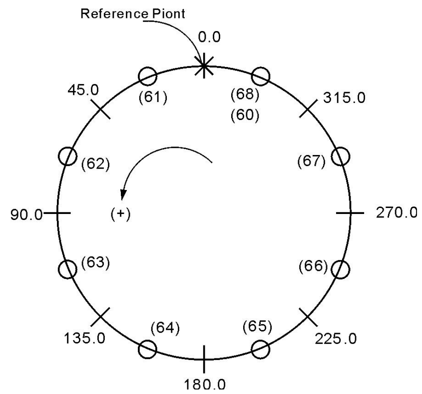
The contrasted chart of compensation points and value as follows:



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:

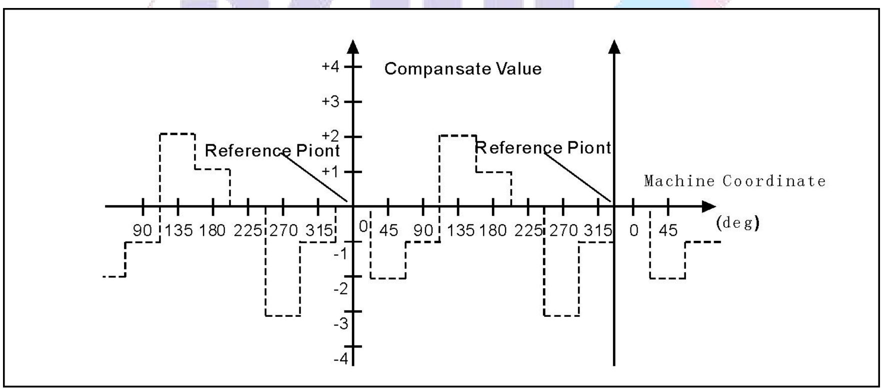
1) Backward checking points: 0
2) Forward checking points: 8
3) Multiple: 1
4) Distance(um): 45000



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 0~360. 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.

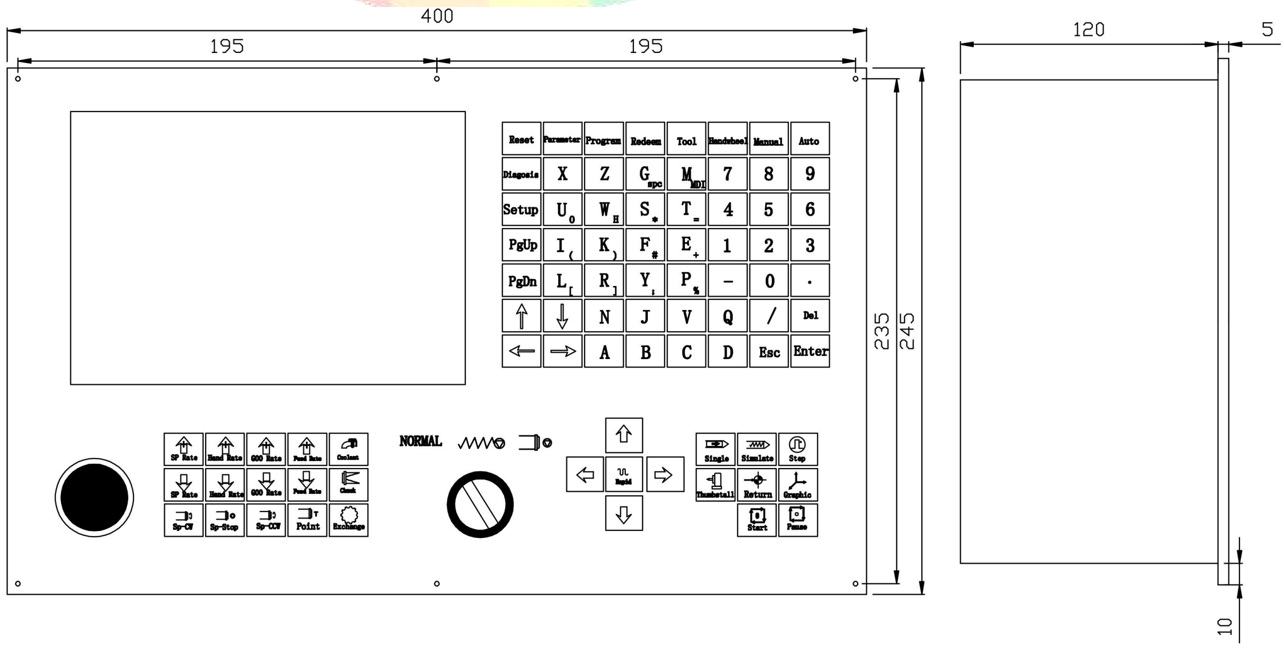
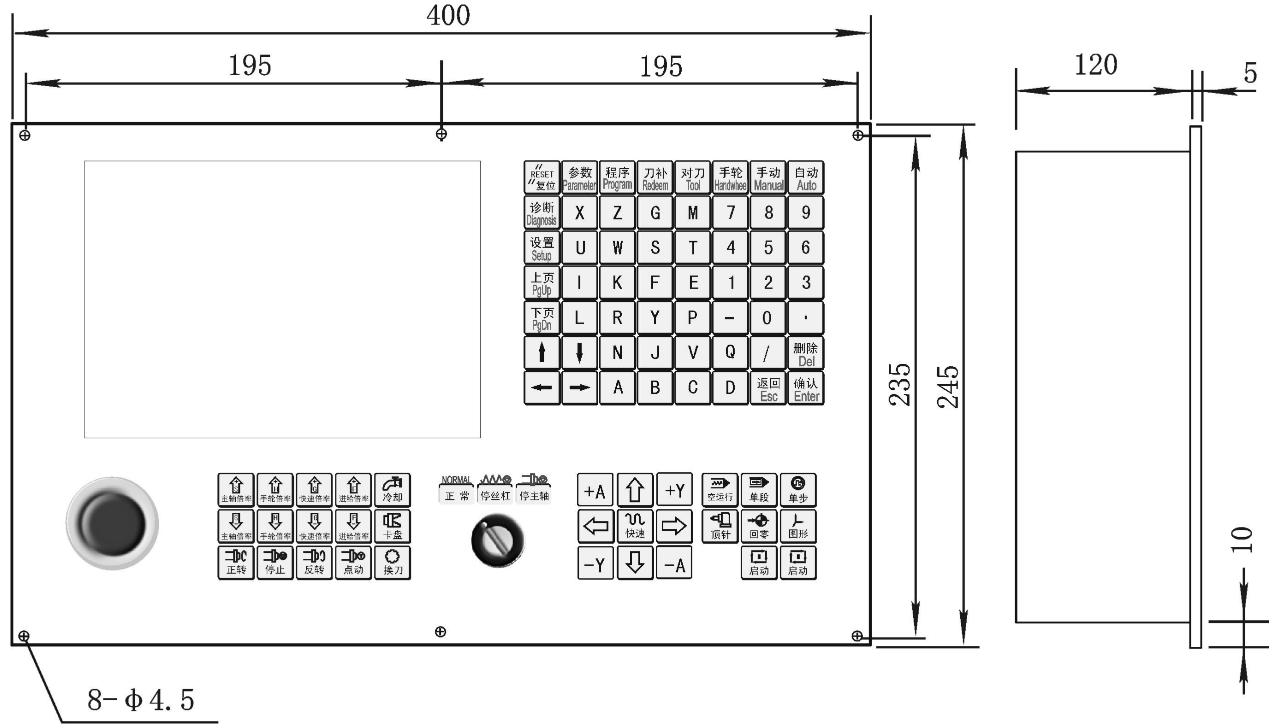
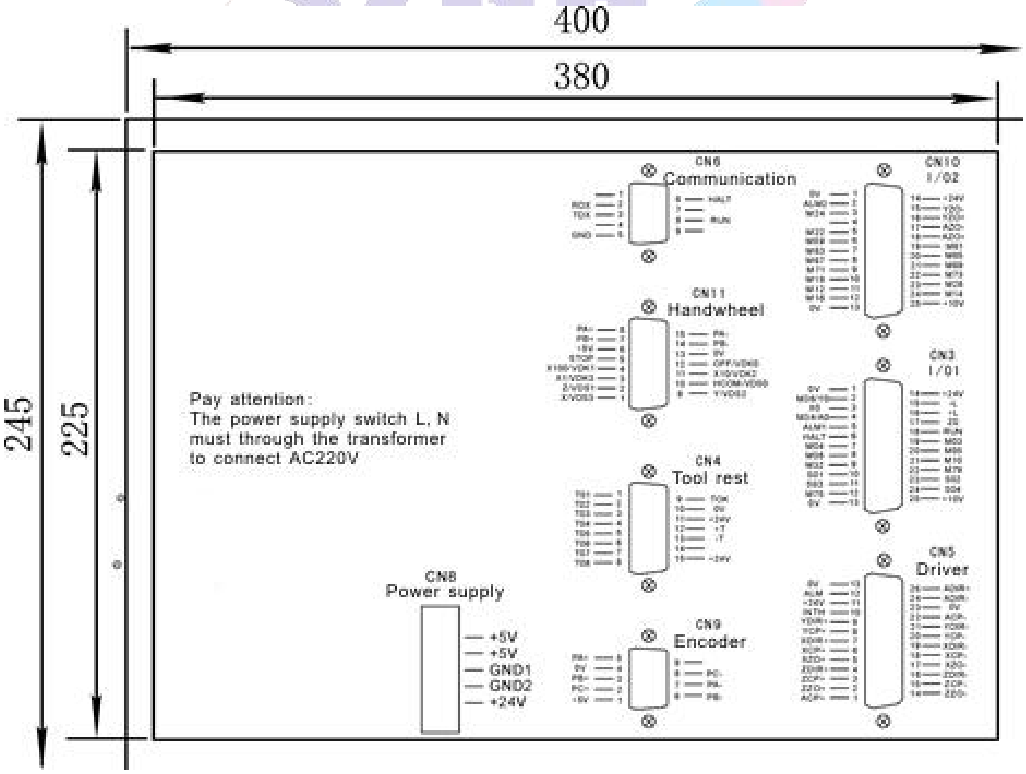


Fig.7.1 Dimension of 2 axes CNC controller



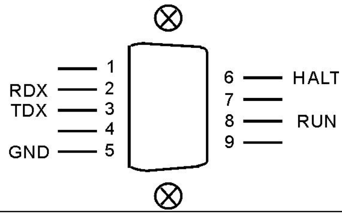
7.3 System Rear View



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)



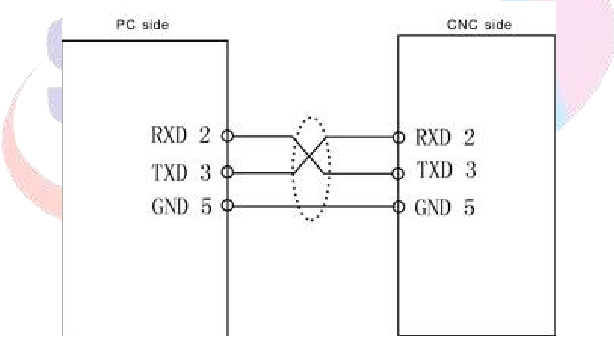
CN6 communication signal with Female socket DB9				
signal	pin	I/O	function	Valid
0V	5	OUT	The ground of signal	0V
RXD	2	IN	The received data signal	
TXD	3	OUT	The transmission of data signal	
RUN	8	IN	External Input for Run program	0V
HALT	6	IN	External Input for Pause program	0V

Note: 1. Connect to external PC with data communication, must be equipped with our special communication software, which is "SZGHCNCCS" software.

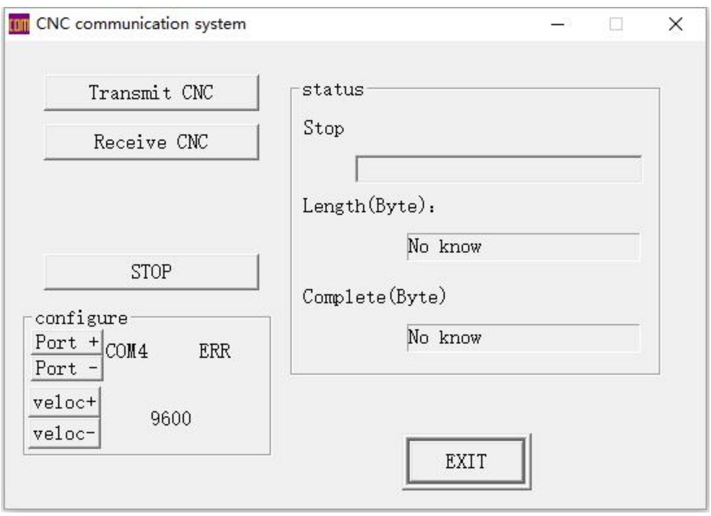
P37 in Other parameter is set rate of CNC system.

2.Communication line must adopt the shielded twisted pair cable, length shall not exceed 10m.

The signal of communication socket connect to PC:

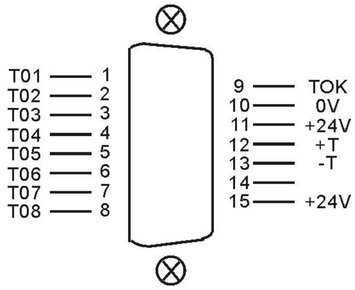


When PC programming, the files should be text files,which could be edit by Notepad or Word-pad.



SZGHCNCCS software

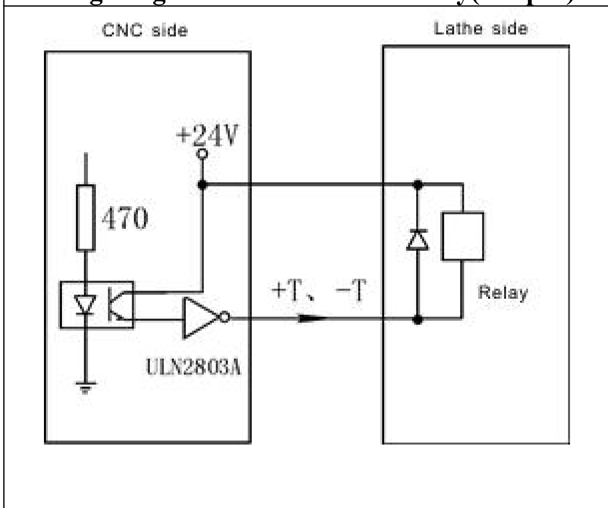
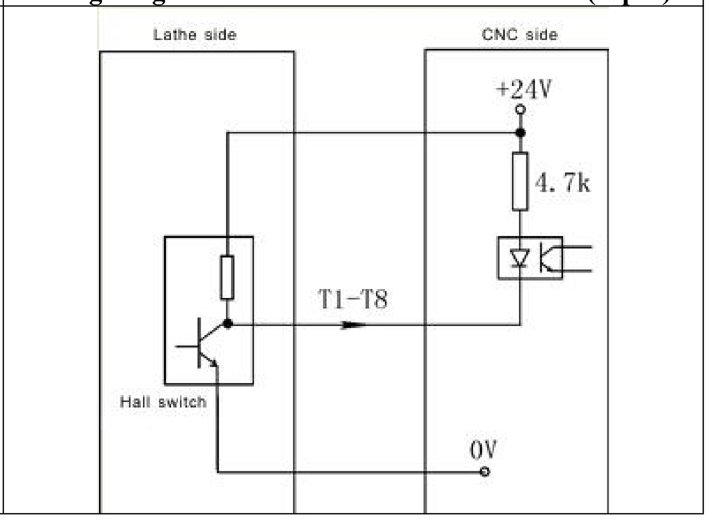
7.4.2 CN4 Turret Socket (Female/DB15)



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	Control CW Rotation of Turret	0V
-T	13	OUT	Control CW Rotation of Turret	0V
T1	1	IN	T1 Position Input Signal	0V
T2	2	IN	T2 Position Input Signal	0V
T3	3	IN	T3 Position Input Signal	0V
T4	4	IN	T4 Position Input Signal	0V
T5	5	IN	T5 Position Input Signal	0V
T6	6	IN	T6 Position Input Signal	0V
T7	7	IN	T7 Position Input Signal	0V
T8	8	IN	T8 Position Input Signal	0V
TOK	9	IN	Detect Lock signal of Turret	0V

System can control 1-99 tools. Press “C-set” to set total tools number in Redeem interface, Starting number of linear tool is set in Tool parameter. Please check Tool parameter on chapter 6.4.

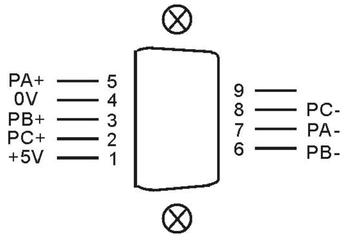
When lathe is with coding turret,we can take these as normal input points & outputs point, which is controlled by ProgramTool,and it can be edited by press “N” in Diagnosis screen.

Wiring Diagram for +T/-T with Relay(Output)	**Wiring Diagram for T01~T08 with NPN sensor (Input)**
	

Attention: 1. All the inputs & outputs are based on CNC system,Valid level of all inputs & outputs is 0V.

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.it must plus a reverse diode in order to cancel reverse current.Suggest select SZGH IO relay board.

7.4.3 CN9 Spindle Encoder Socket (Female/DB9)

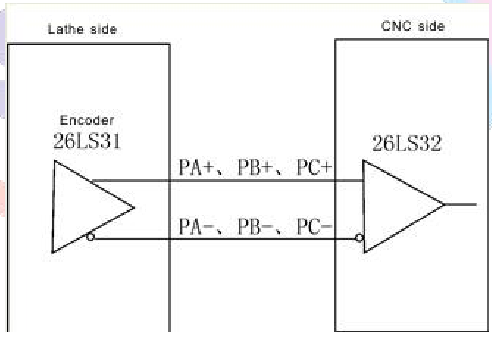


CN9 SP_Encoder signal with Female socket DB9				
Signal	Pin	I/O	Function	Valid
0V	4	OUT	0V	0V
+5V	1	OUT	+5V	+5V
PA+	5	IN	A Phase Positive signal	5V
PA-	7	IN	A Phase Negative signal	
PB+	3	IN	B Phase Positive signal	5V
PB-	6	IN	B Phase Negative signal	
PC+	2	IN	Z Phase Positive signal	5V
PC-	8	IN	Z Phase 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 threading cycle processing,like G92, it needs to fix an encoder to spindle motor.

Configuration of encoder:

P9 on Axis parameter: Whether Detecting spindle encoder

P10 on Axis parameter: Resolution of spindle encoder

P412 on Axis parameter: number of spindle teeth

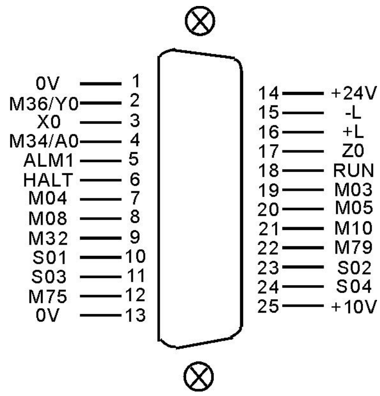
P413 on Axis parameter, number of encoder teeth

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
M36/Y0	2	IN	M36/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	Thumbstall/Tailstock	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	Shift Control mode for C-axis	0V
+10V	25	OUT	Analog Output Signal of 1st spindle	0～10V
0V	13	OUT	Ground of frequency conversion	0V

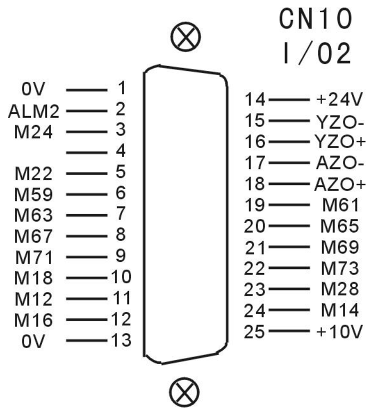
Note: 1.when your CNC system isn't configured with Y(C)-axis & A-axis,M36/Y0 & M34/A0 could be used as normal input points,controlled by M36/M34 code.

2.Valid level of inputs & outputs are 0V.

3. P36 on Speed parameter is set max speed of 1st spindle, also corresponding to +10V analog voltage of 1st spindle.

4.+24V & 0V of CN3 are from power supply directly in the rear of CNC controller.

7.4.5 CN10 IO2 Socket (Female/DB25)



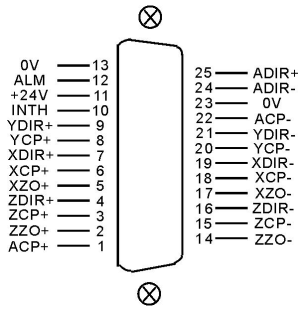
CN10 I/O2 signal with Female Socket of DB25				
Signal	Pin	I/O	Function	Valid
0V	1	OUT	Ground of the power supply	0V
+24V	14	OUT	24V power supply	+24V
ALM2	2	IN	Alarm2 of Machine Tool	0V
M24	3	IN	User-defined input 7	0V
M22	5	IN	M01 Quasi-stop Input	0V
M59	6	OUT	Huff Output	0V
M61	19	OUT	User-defined 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

Note: 1.Valid level of inputs & outputs are 0V.

2. P40 on Speed parameter is set max speed of 1st spindle, also corresponding to +10V analog voltage of 1st spindle.

3.+24V & 0V of CN10 are from power supply directly in the rear of CNC controller.

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

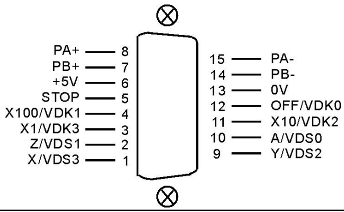
Note: 1. The signal line must adopt shielded twisted pair cable, the length is 20m at most.

2. Type of ALM signal is set by P17 in Other parameter.

3. There is only one input point, ALM, on CNC,so if NC type alarm signal,we need to make series connections; if NO type alarm signals,we need to make parallel connections.

4. When system take C axis to as rotate axis, M800 instruction is for backing to zero position of encoder, Output M75 signal to select position control mode of spindle servo, M03/M04 is to close M75 signal, spindle servo shift to speed control mode.

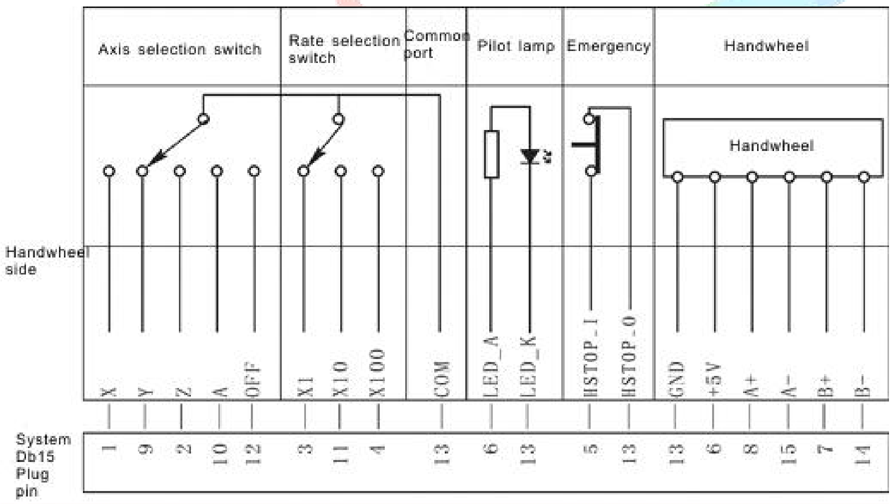
7.4.7 CN11 MPG/handhold Box Socket (Male/DB15)



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	A signal -	
PB+	7	IN	B signal +	5V
PB-	14	IN	B signal -	
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)

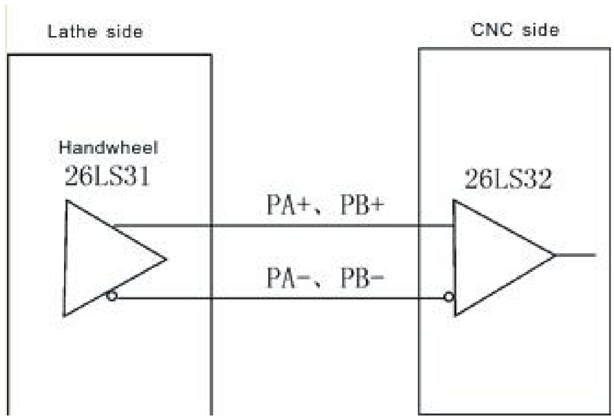
Handwheel contact diagrammatic as:



When user connect our handhold 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; P33&P34 in Other parameter only set to be 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.

7.5 SZGH-CNC-IO-12 IO Relay Board

SZGH-CNC-IO-12 is newest version I/O Relay board with 12pieces of relays, which is used for connecting all inputs & outputs on CN3/CN4/CN10 plugs of CNC controller to external switches & loads easily.

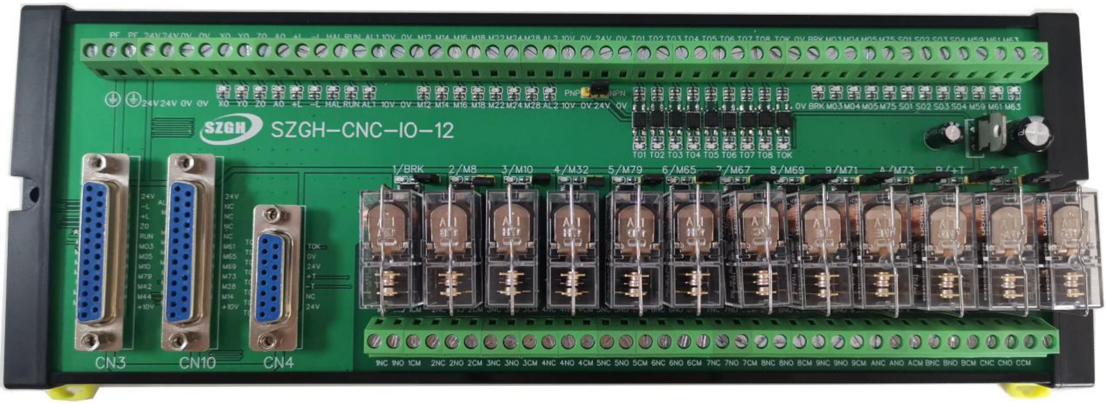


Fig1 Practical Picture of SZGH-CNC-IO-12

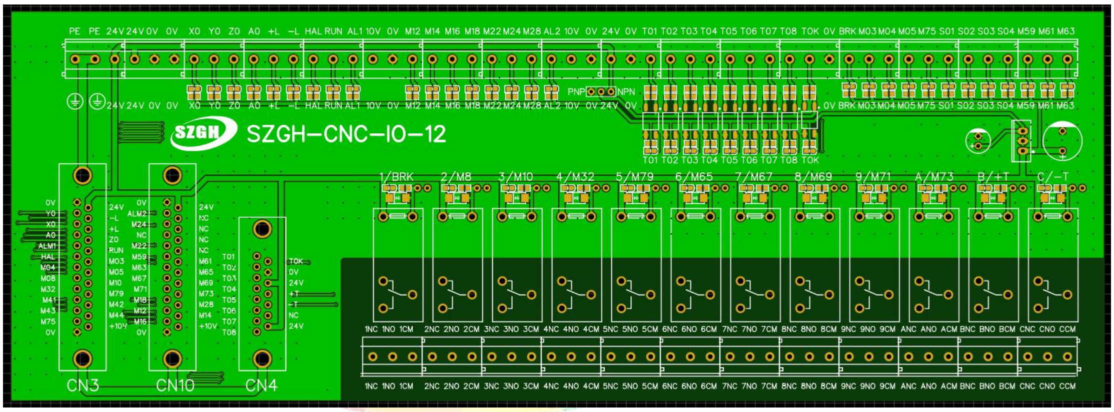


Fig2 Design Sketch of SZGH-CNC-IO-12

CN3 socket is corresponding to CN3 plug of CNC Controller one by one;

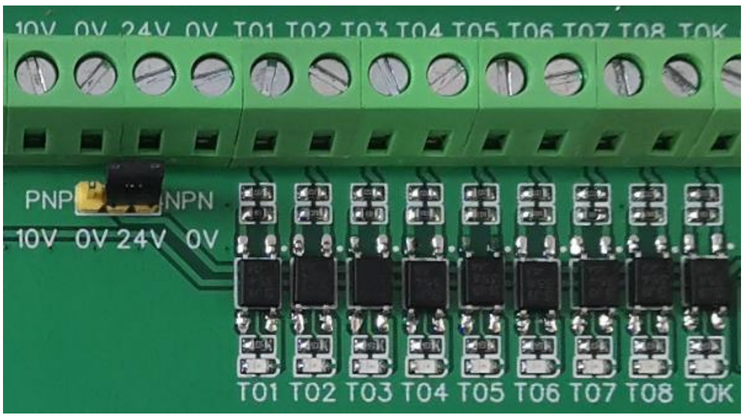
CN10 socket is corresponding to CN10 plug of CNC Controller one by one;

Note: Pin4&Pin15-Pin18 on CN10 are null.

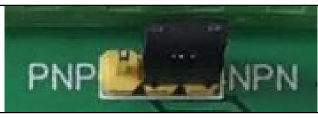
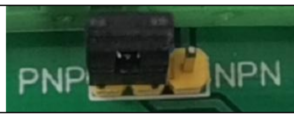
CN4 socket is corresponding to CN4 plug of CNC Controller one by one

Note:Pin14 on CN4 plug is null.

Update Point 1:



T01-T08 & TOK can support NPN & PNP input.

NPN INPUT	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:

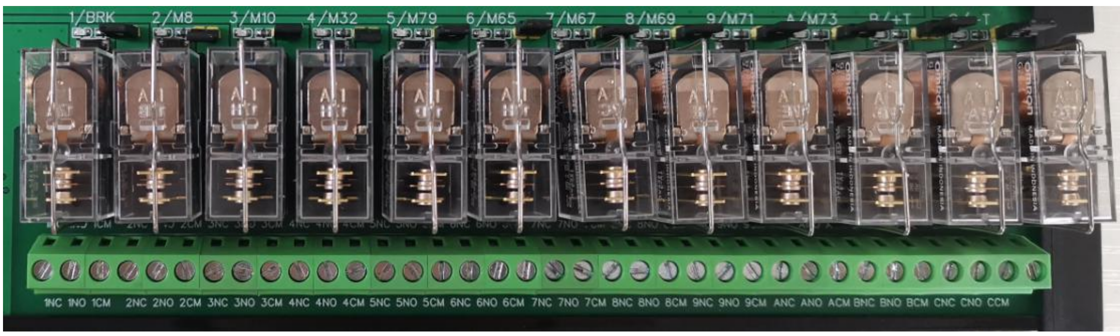
INPUT	
OUTPUT	

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 12pieces 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 plenty 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 the require of user manual correctly use，do one's best to avoid improper operation.

Everyday operator should clean the system's box and panel in case for corrupt thing and sundries to indemnify it.

When CNC system's using time is over three month,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 turn on.

3） check insurance.

7.6.2.2 No display as boot

1） Boot again or reset.
2） Check if switch power's ＋5V、＋12V、－12V、-24V are normal.
3） Check if transformer is bad.
4） Check if LCD's bright adjust and connection are normal.
5） Check if main board is normal.

7.6.2.3 System's control disorganize

1） Wrong operations.
2） Anti-jamming ability of power supply is descend.
3） Working circumstance of CNC system is too bad.

7.6.2.4 Lose of user program

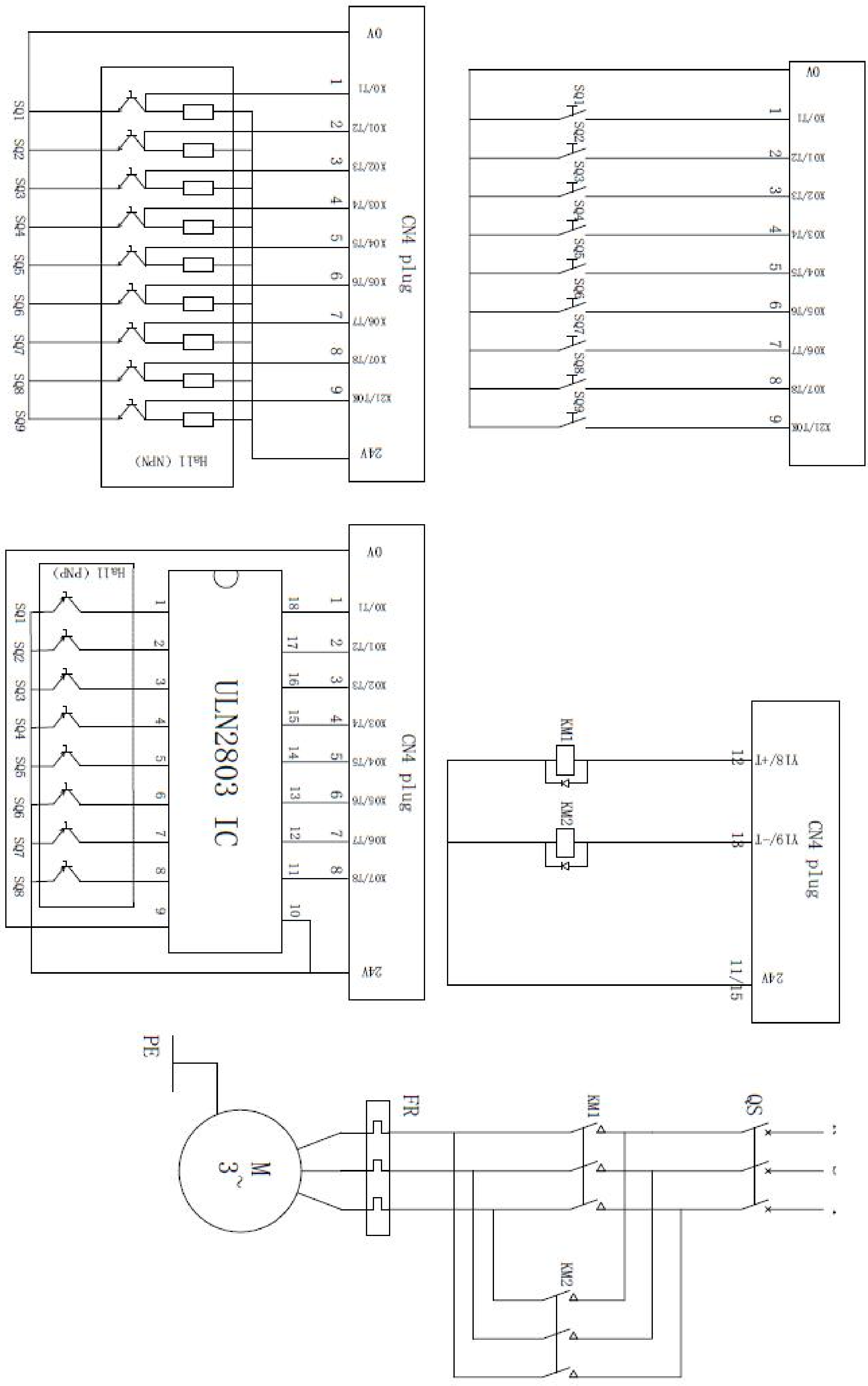
The DC battery on system main board can insure user's program and parameter don't lose.When system isn't used for half year or system has been used for over two years， the battery maybe invalidate,therefore, should exchange battery.

7.6.2.5 Machining precision is bad

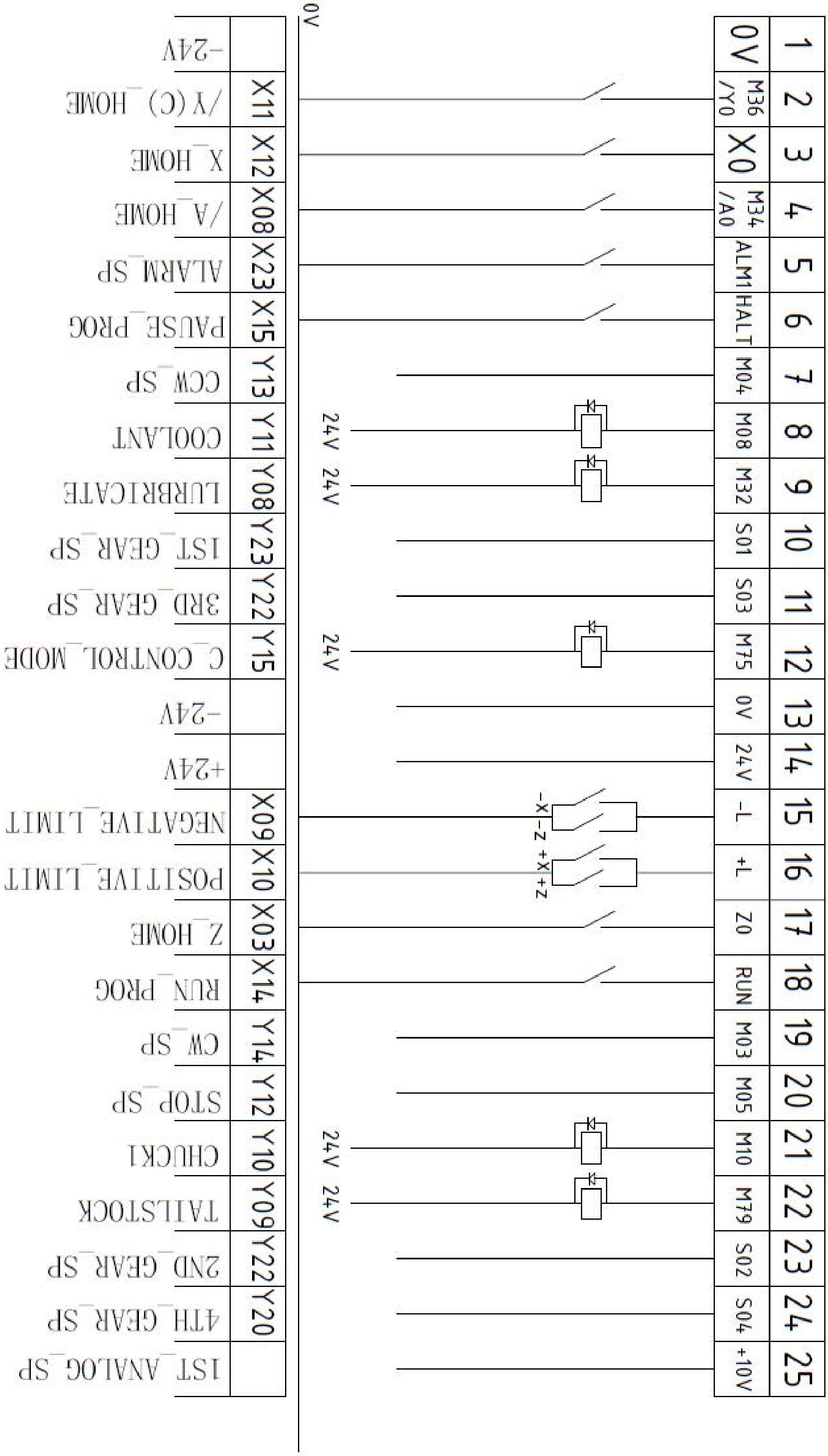
1） CNC Machine needs to revise backlash after using some time.
2） Best to revise base point before machining in order to insure the start point's precision.
3） Machining speed and cutting depth is improper.
4） Machine connector's prick melt falls off.
5） Tool isn't tightened.
6） Piece clamp isn't good.
7） Tool's giving up isn't equality because piece's dimension isn't uniformity.
8） Problems of machine Tool

Attention: Because of many kinds of reasons this manual may have some mistakes. Our company will provide the high quality service and the technical support for every customer.

Appendix I Wiring Diagram of CN4 Turret



Appendix II Wiring Diagram of CN3 Plug



Appendix III Wiring Diagram of CN10 Plug