MACOTECH

User Manual

SZGH-CNC1000TDb(series) Lathe Control System -PROGRAMMING- V2.2

BOOK1

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. GE Fanuc Automation assumes no obligation of notice to holders of this document with respect to changes subsequently made.

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

SAFETY PRECAUTIONS

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

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

1 Definition of Warning , Caution, and Note

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

WARNING

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

CAUTION

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

NOTE

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

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

2 GENERAL WARNINGS AND CAUTIONS

Warning

1. Never attempt to machine a workpiece without first checking the operation of the machine. Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
2. Before operating the machine, thoroughly check the entered data. Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
3. Ensure that the specified feedrate is appropriate for the intended operation. Generally, for each machine, there is a maximum allowable feedrate. The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate. If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
4. When using a tool compensation function, thoroughly check the direction and amount of Compensation. Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
5. The parameters for the CNC and PMC are factory-set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change. Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
6. Immediately after switching on the power, do not touch any of the keys on the MDI panel until the position display or alarm screen appears on the CNC unit. Some of the keys on the MDI panel are dedicated to maintenance or other special operations. Pressing any of these keys may place the CNC unit in other than its normal state. Starting the machine in this state may cause it to behave unexpectedly.
7. The operator's manual and programming manual supplied with a CNC unit provide an overall description of the machine's functions, including any optional functions. Note that the optional functions will vary from one machine model to another. Therefore, some functions described in the manuals may not actually be available for a particular model. Check the specification of the machine if in doubt.
8. Some functions may have been implemented at the request of the machine–tool builder. When using such functions, refer to the manual supplied by the machine–tool builder for details of their use and any related cautions.

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

3 WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

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

Warning

1.Coordinate system setting

If a coordinate system is established incorrectly, the machine may behave unexpectedly as a result of the program issuing an otherwise valid move command.
Such an unexpected operation may damage the tool, the machine itself, the workpiece, or cause injury to the user.

2. Positioning by nonlinear interpolation

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

3. Function involving a rotation axis

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

4. Inch/metric conversion

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

5. Constant surface speed control

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

6. Stroke check

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

7. Absolute/incremental mode

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

8. Plane selection

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

9. Compensation function

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

4 WARNINGS AND CAUTIONS RELATED TO HANDLING

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

Warning

1. Manual operation

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

2. Manual reference position return

After switching on the power, perform manual reference position return as required. If the machine is operated without first performing manual reference position return, it may behave unexpectedly. Stroke check is not possible before manual reference position return is performed. An unexpected operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3. Manual handle feed

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

4. Disabled override

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

5. Origin/preset operation

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

6. Workpiece coordinate system shift

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

7. Software operator's panel and menu switches

Using the software operator' s panel and menu switches, in combination with the MDI panel, it is possible to specify operations not supported by the machine operator' s panel, such as mode change, override value change, and jog feed commands.
Note, however, that if the MDI panel keys are operated inadvertently, the machine may behave

unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

8. Manual intervention

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

9. Feed hold, override, and single block

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

10. Dry run

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

11. Cutter and tool nose radius compensation in MDI mode

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

12. Program editing

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

5 WARNINGS RELATED TO DAILY MAINTENANCE

WARNING

1. Memory backup battery replacement

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

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

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

CONTENTS

SAFETY PRECAUTIONS II
1 Definition of Warning , Caution, and Note II
2 GENERAL WARNINGS AND CAUTIONS III
3 WARNINGS AND CAUTIONS RELATED TO PROGRAMMING IV
4 WARNINGS AND CAUTIONS RELATED TO HANDLING V
5 WARNINGS RELATED TO DAILY MAINTENANCE VI
Chapter 1 Preface 1
1.1 Characteristics 2
1.2 Technical Specifications 2
1.3 G Code List 4
1.4 System operation condition 5
Chapter 2 Programming 6
2.1 Basic Concept of Programming 7
2.1.1 Definition of Coordinate System 7
2.1.2 Machine Coordinate System and Machine Reference Point 8
2.1.3 Workpiece Coordinate System and Program Reference Point 8
2.1.4 Coordinate System 9
2.1.5 Interpolation 10
2.1.6 Absolute Programming & Incremental Programming 12
2.1.7 Diameter Programming & Radius Programming 13
2.1.8 Cutting Speed & Spindle Speed 14
2.1.9 Tool Function 15
2.1.10 Command For Machine Operations 15
2.2 Configuration of Program 16
2.3 Main Program & Subprogram 19
2.4 Program Run 19
Chapter 3 G INSTRCUTIONS 20
3.1 INTRODUCTION 20
3.2 G Code List 20
3.3 Positioning (Rapid Traverse) (G00) 22
3.4 Linear Interpolation (G01) 24
3.5 Circular Interpolation (G02/G03) 24
3.6 Thread Cutting (G32) 28
3.6.1 Constant Lead Threading 28
3.6.2 Continous Thread Cutting 29
3.6.3 Thread Cutting With Variable Lead 30
3.7 Circular Thread Cutting(G332/G333) 30
3.8 Canned Cycle(G90,G92,G93,G94) 30
3.8.1 Outer Diameter/Internal Diameter Cutting Cycle (G90) 30
3.8.2 Thread Cutting Cycle (G92) 33
3.8.3 Canned Tapping Cycle (G93) 36
3.8.4 End Face Turning Cycle G94 38

3.8.5 Usage for Canned Cycle....41
3.9 Multiple Repetitive Cycle Instructions(G70~G76)....42
3.9.1 Axial Roughing Turning Cycle (G71)....42
3.9.2 Radial Roughing Facing Cycle (G72)....45
3.9.3 Pattern Repeating Cycle (G73)....46
3.9.4 Finishing Cycle (G70)....48
3.9.5 Usages of G71,G72,G73 & G70....48
3.9.5.1 Example of G71&G70....48
3.9.5.2 Example of G72 & G70....49
3.9.5.3 Example of G73&G70....49
3.9.6 End Face Peck Drilling Cycle (G74)....51
3.9.7 Outer Diameter/Internal Diameter Drilling Cycle (G75)....52
3.9.8 Multiple Thread Cutting Cycle (G76)....54
3.9.9 Notes On Multiple Repetitive Cycle (G70 ~ G76)....58
3.10 Skip Function(G31,G311)....58
3.11 Block Cycle (G22,G800)....59
3.12 Return to Starting Point (G26,G261~G264)....60
3.13 Save Current Position (G25)....60
3.14 Return to Specified Position (G61,G611~G614)....60
3.15 Return to Reference Position (G28)....61
3.16 Coordinate System....62
3.16.1 Machine Coordiante System (G53)....62
3.16.2 Workpiece Coordinate System....63
3.16.3 Setting a Workpiece Coordinate System(G50)....63
3.16.4 Selecting a Workpiece Coordinate System....64
3.16.5 Changing Workpiece Coordinate System....65
3.16.6 Workpiece Coordinate System Shift....66
3.16.7 Local Coordinate System....67
3.17 Constant Surface Speed Control (G96/G97)....68
3.18 Cutting Feed (G98,G99)....70
3.19 Pole Coordinate Interpolation (G15/G16)....71
3.20 Absolute and Incremental Programming (G990,G991)....72
3.21 Inch/Metric Conversion (G20/G21)....73
3.22 Dwell (G04)....74
3.23 Positioning/Continous Path Processing(G60/G64)....74
3.24 Workpiece Position and Move Command....75
3.25 Details of Tool Nose Radius Compensatation (G40/G41/G42)....78
3.26 Tool Nose Radius Compensation of Offset C....79
3.27 Automatical beveling (I) and smoothing(R)....84
3.28 3D Space Arc Interpolation G06....85
3.29 Macro program instruction(G65/G66/G67)....86
3.29.1 Non-Mode Macro Command G65....86
3.29.2 Mode Macro Command G66/G67....86
3.29.3 Macro Program Instruction....87

3.29.3.1 Input Instruction: WAT.......87
3.29.3.2 Output Instruction: OUT.......87
3.29.3.3 Assignment Instruction: =.......87
3.29.3.4 Unconditional Jump: GOTO n.......87
3.29.3.5 Conditional Jump.......87
3.29.3.6 Loop Command.......88
3.29.4 Operators' meaning.......89
3.29.5 Arithmetic & Logic Operation.......89
3.29.6 Local Variable.......89
3.29.7 Global Variable.......90
3.29.8 System Variable.......90
3.29.9 System Parameter Variable.......90
3.29.10 I/O variable.......90
3.29.11 Message Hint Dialog Box.......90
3.29.12 Build Processing Program Automatically.......91
3.30 Complex function for Turning & Milling.......91
3.31 Polar Coordinate Interpolation(G12.1/G13.1).......92
Chapter 4 M INSTRCUTIONS.......95
4.1 M Function (Auxiliary Function).......95
4.1.1 Program Stop(M00).......95
4.1.2 Optional Stop (M01).......95
4.1.3 End of Program (M02,M30).......95
4.1.4 Cycle of Program (M20).......95
4.1.5 Account of Workpiece(M87).......95
4.1.6 Uncondtional Jump (M97).......95
4.2 Subprogram Configuration.......95
4.2.1 Calling of Subprogram (M98).......96
4.2.2 End of Subprogram (M99).......96
4.3 Standard PLC M Command List.......97
4.3.1 M Output Command List.......97
4.3.1.1 Spindle Control (M03/M04/M05).......98
4.3.1.2 Spindle Gear Shifting(M41/M42/M43/M44).......99
4.3.1.3 Coolant(M08/M09).......99
4.3.1.4 Lubricate(M32/M33).......100
4.3.1.5 Chuck(M10/M11).......100
4.3.1.6 Tailstock(M79/M78).......100
4.3.1.7 Condition Output of Machine Tool(M65/M67/M69).......101
4.3.2 M Input Command List.......101
4.4 Analog Speed of Spindle(S , SS).......102
4.5 T Tool Function Command.......102
4.6 User-defined macro instruction(G120-G160,M880-M889).......103
4.7 Synthetic instance for programming.......104

Chapter 1 Preface

CNC machine tool is an electro-mechanical integrated product, composed of Numerical Control Systems of Machine Tools, machines, electric control components, hydraulic components, pneumatic components, 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 themain of CNC machine tools.

SZGH-CNC1000TDb series CNC control system is standard type CNC control system for lathe machine , which is developed by Shenzhen Guanhong Automation Co.,Ltd. And we have already made updates basic original CNC1000TDb CNC Controller.

Fig1.1 SZGH-CNC1000TDb

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 Pcs of tools
- 4) 128MB Memory , 100Mb user store room(Updated!)
- 5) 5MHz Pulse Output Frequency, Max speed is 300m/min(Updated!)
- 6) PLC On-line Display,Monitor & Design(Updated!)
- 7) High anti-jamming switch power(220VAC -> 24VDC & 5VDC)
- 8) Built-in plc programs, which can be edited freely.
- 9) With USB interface, for upgrade & copy programs
- 10) Display in English, which can be selected by parameter.
- 11) Analog voltage output of 0~10V in two channels, support double spindles
- 12) Adapted servo spindle can realize position,rigid tapping,threading of spindle
- 13) Basic I/Os : 56*32
- 14) Built-in screw compensation
- 15) English menu, program and interface, full screen edition
- 16) Support macro variable dialog box & Running program by input point (Updated!)

1.2 Technical Specifications

Max Number of control axes

• Number of control axes: 5 axes (X Z Y(C) A B)

• Number of linkage axes: 5 axes

• Number of PLC control axes: 5 axes

Feeding axes function

• Minimum command unit: 0.001mm

• Position command range: +/- 99999.999

• Max speed: 300m/minFeeding speed:0.001-15m/min

• G00 rapid override: Total 8 levels: 0~150%,real-time adjusting

• Feeding override: Total 16 levels: 0~150%,real-time adjusting

• Spindle override: Total 16 levels: 5%~150%,real-time adjusting

• Interpolation mode: Interpolation of linear ,arc ,thread and rigid tapping

• Auto chamfering

Thread

• Acceleration and deceleration funciton

• Common thread(follow the spindle) / Rigid thread

• Single-headed/Multi thread of straight ,taper and therminal suface in metric system/inch system,wqual and variable pitch thread

• Thread retractin 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 speed control for two spindles

• 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 coressponding to 10V
• Spindle override: Total 16 levels: 5%~150%,real-time adjusting
• Spindle constant surface speed control
• Ragid 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 exexuting mode: Rewriting coordinate mode,tool traverse mode

Precision compensation

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

PLC function

• Refresh cycle: 8ms
• PLC program can be altered on PC , download by USB interface
• I/Os : 56*32 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 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 operationand 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 curret 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	Continous track processing	M800	C-axis return to zero position of SP-encoder
G50	Set max speed of spindle	M801	C-axis ready to stop
G52	Set local coordinate system	G53	Machine coordinate system
G184	Setup/Offset curret coordinte 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 must use isolation transform to supply power first input:380V

2) Climate condition

Item	Working Conditions	Storage&Delivery Conditions
Ambient Temperature	0℃~45℃	-40℃~+70℃
Ambient Humidity	≤95%(no freezing)	≤95%(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 electro-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-CNC1000TDb 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:

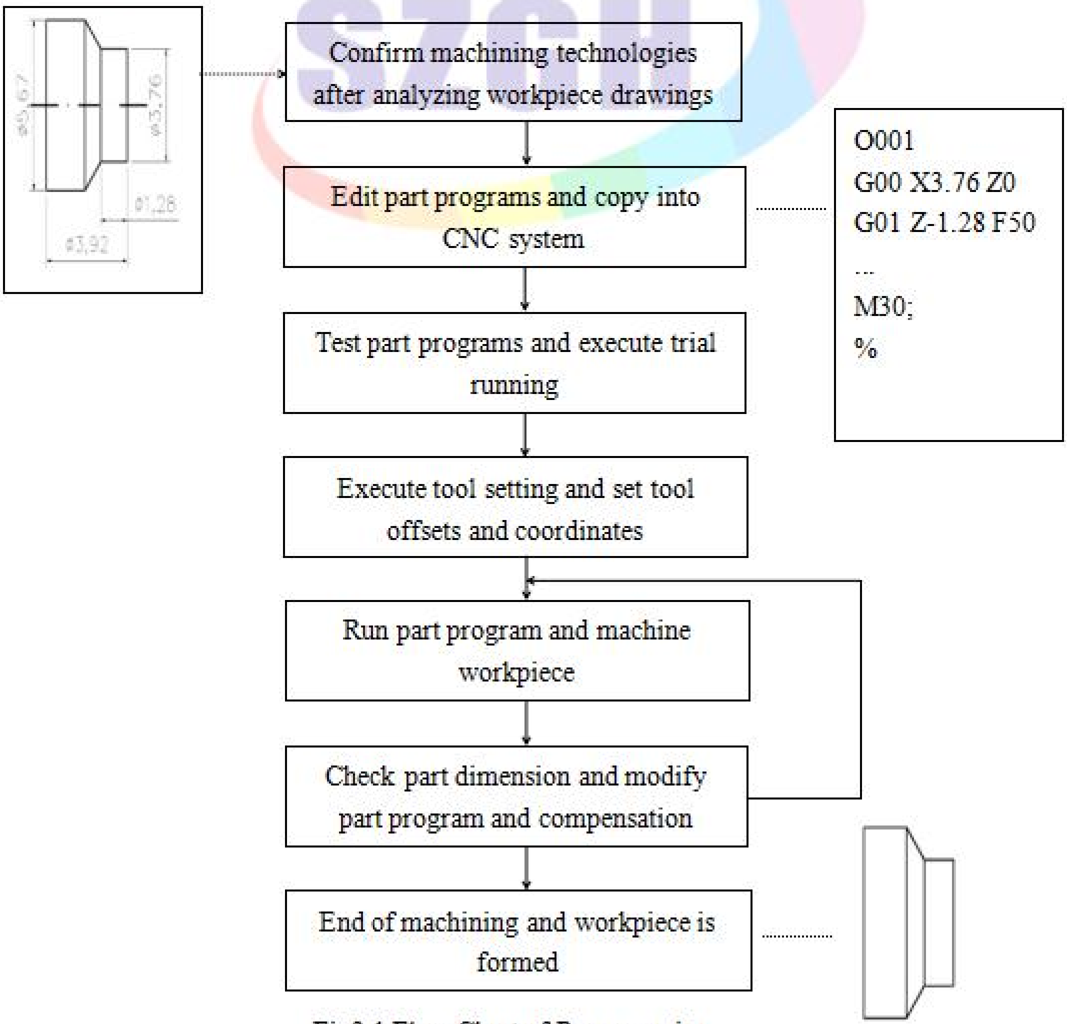


Fig2.1 Flow Chart of Programming

2.1 Basic Concept of Programming

2.1.1 Definition of Coordinate System

Sketch map of CNC turning machine is as following:

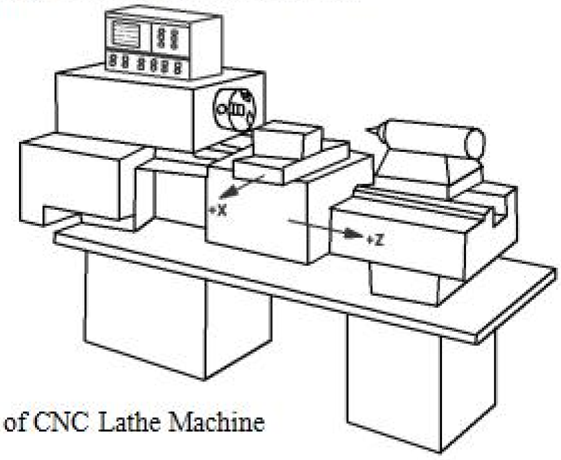
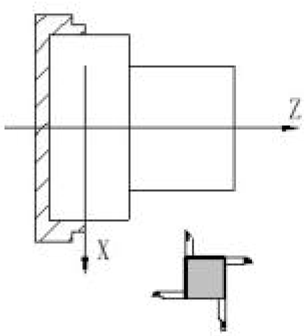


Fig2.2 Sketch Map of CNC Lathe Machine

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 toolpost and spindle, there are 2 kinds of turning coordinate system; one is Front toolpost coordinate sytem(Fig2.3) , the other is Rear toolppost 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 toolpost coordinate sytem. **Type of CNC Lathe Machine: P3 in Tool parameter.**



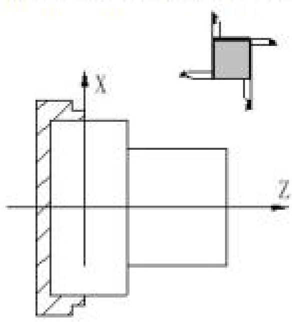


Fig2.3 Front toolpost coordinate system Fig2.4 Rear toolpost coordinate system

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.

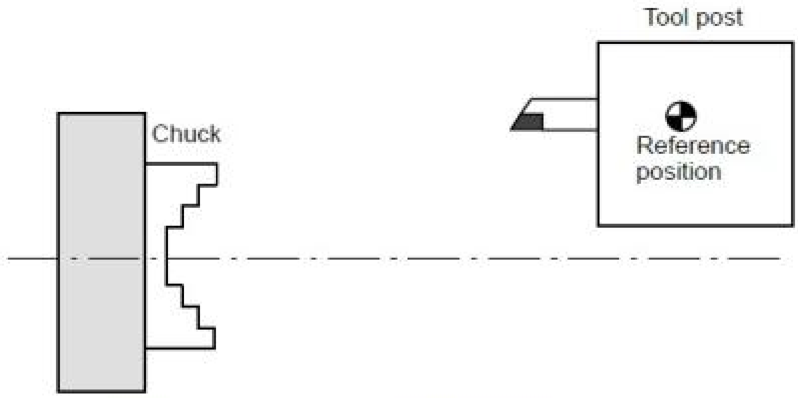


Fig2.5 Reference Position

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.

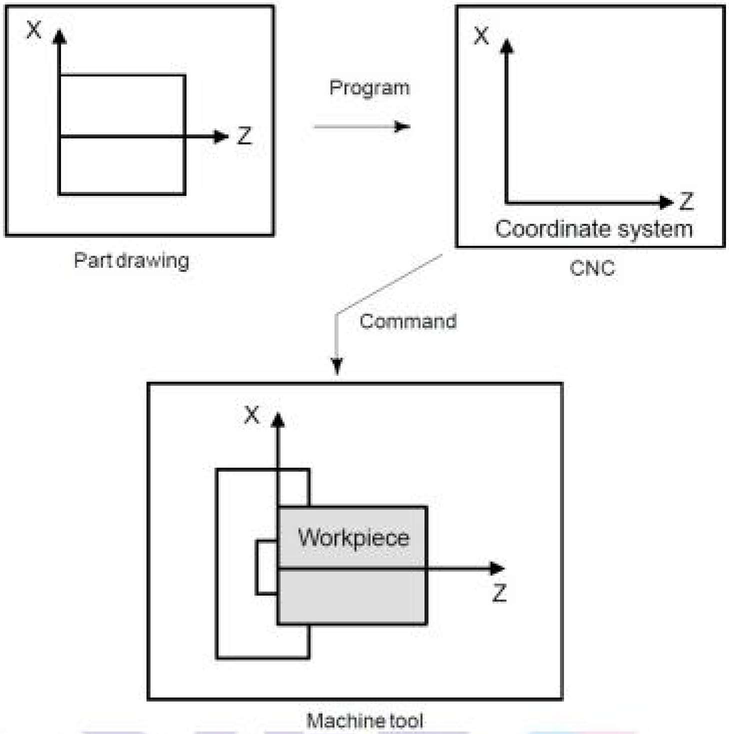


Fig2.6 Coordinate system

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.

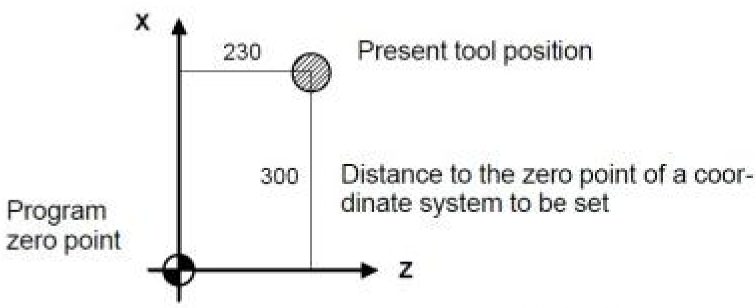


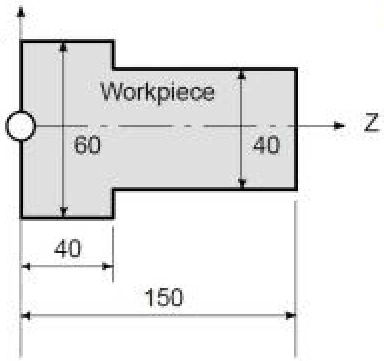
Fig2.7 Coordinate system specified by the CNC

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.

Th d to define two coordinate systems at the same location.

1 t at chuck face



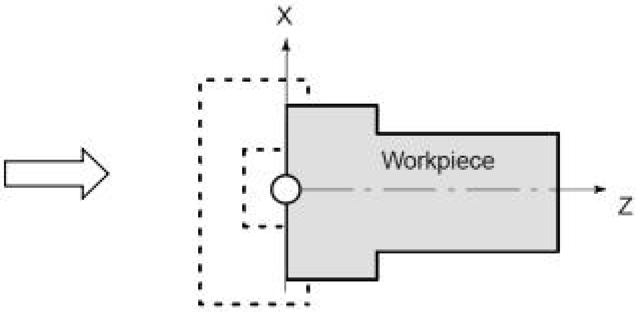
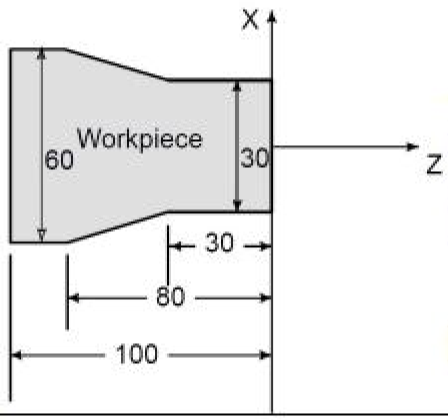


Fig2.8 Coordinates and dimensions on part drawing

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.



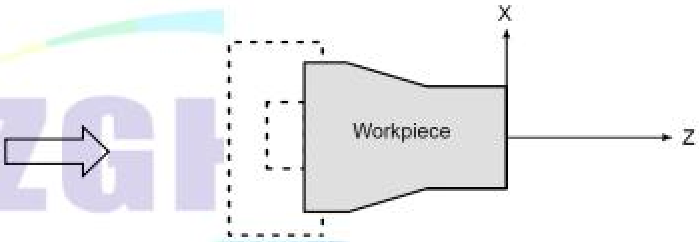


Fig2.10 Coordinates and dimensions on part drawing

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

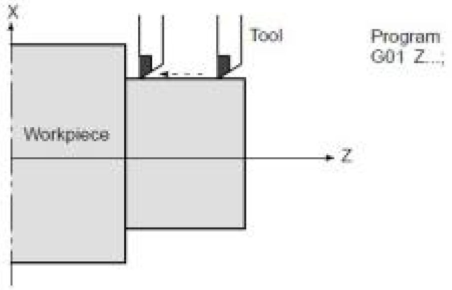


Fig2.12 Tool movement along the straight line which is parallel to Z–axis

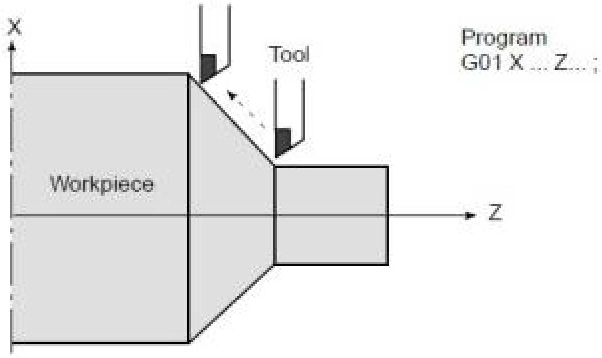


Fig2.13 Tool movement along the taper line

Arc Interpolation: Tool movement along an arc

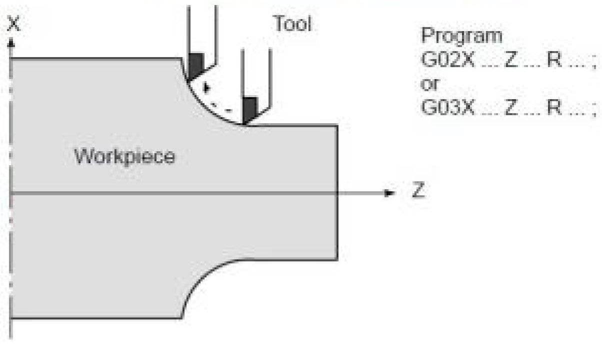


Fig2.14 Tool movement along an arc

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.

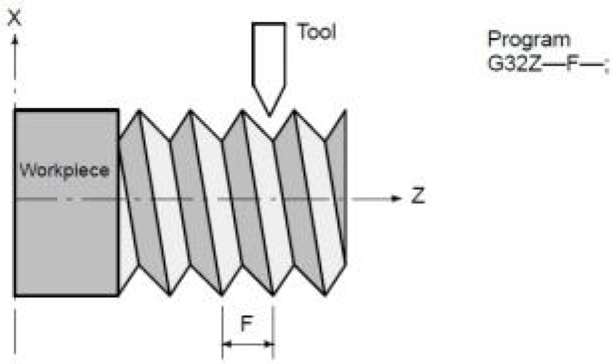


Fig2.15 Straight thread cutting

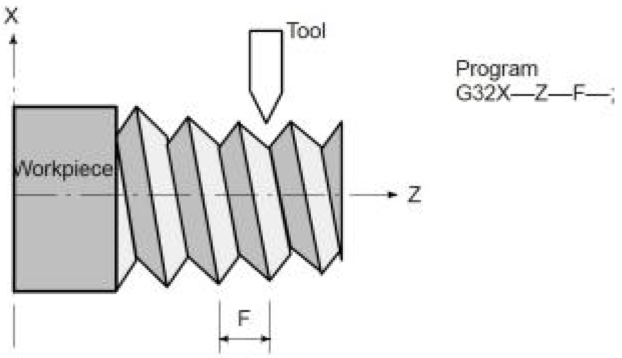


Fig2.16 Taper thread cutting

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

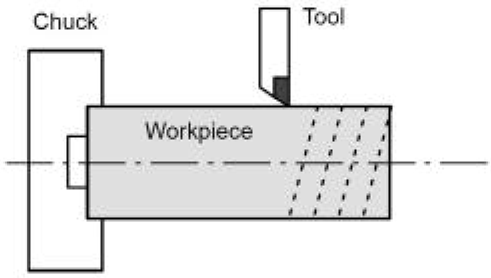


Fig2.17 Feed Function

Feed Rates can be specified by using actual numerics. 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.

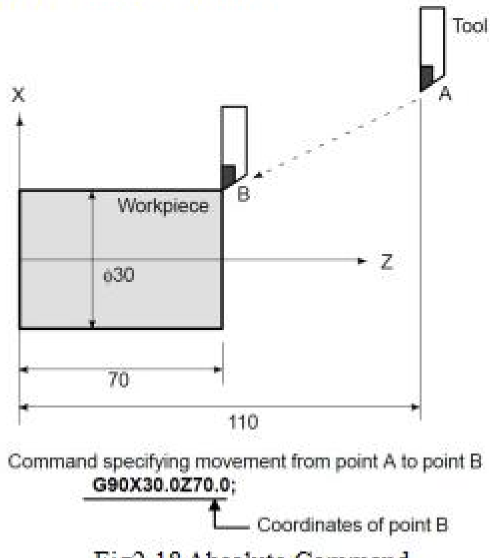


Fig2.18 Absolute Command

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

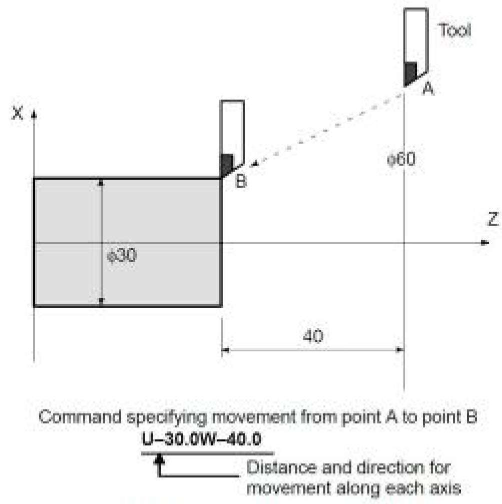


Fig2.19 Incremental command

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.

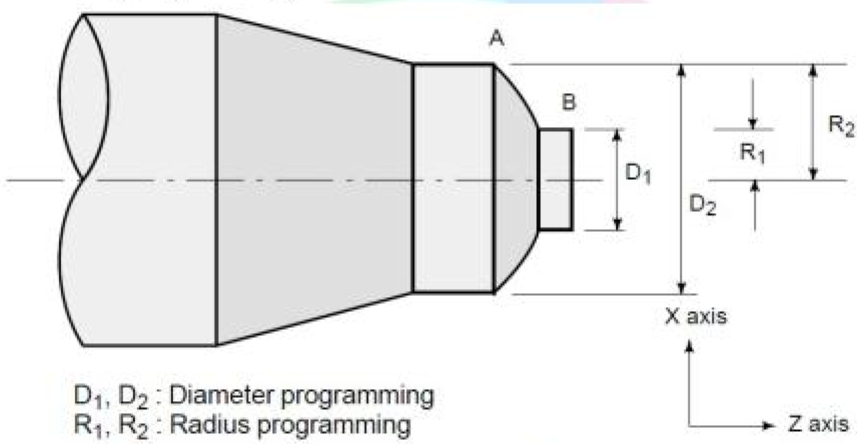
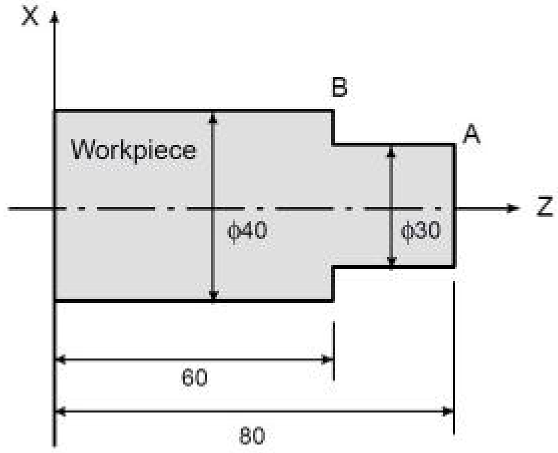


Fig 2.20 Diameter Programming & Radius Programming

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.

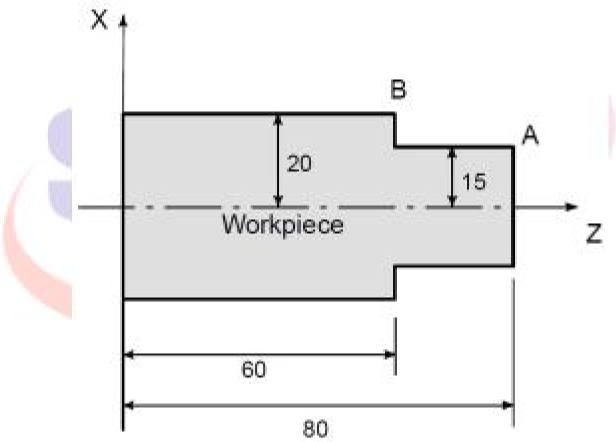


Coordinate values of points A and B
A(30.0, 80.0), B(40.0, 60.0)

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
Parameters in canned cycle, such as cutting depth along X axis. (R)	Specifies a radius value
Radius designation in circular interpolation	Specifies a radius value

(R, I, K, and etc.)	
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.

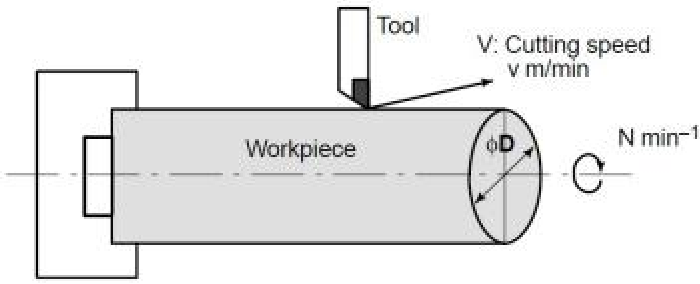


Fig2.23 Cutting Speed

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.

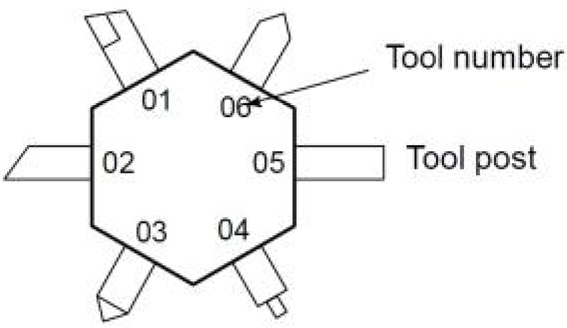


Fig2.24 Tool used for various machining

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

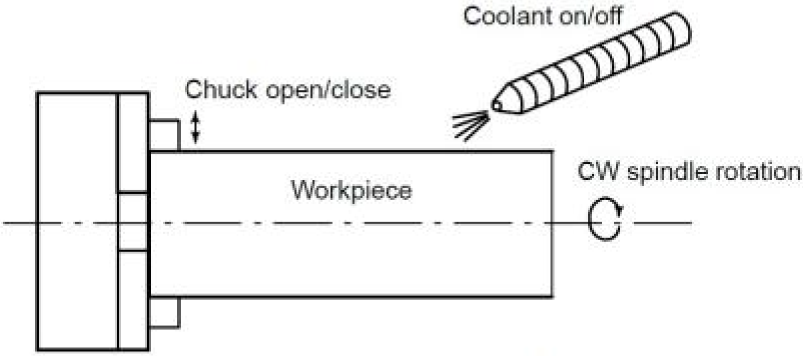


Fig2.25 Command for machine Operations

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, the spindle is rotated clockwise at the specified spindle speed.

2.2 Configuration of Program

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

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

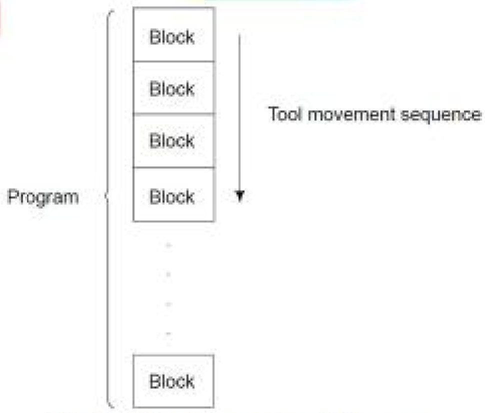


Fig2.26 Configuration of Program

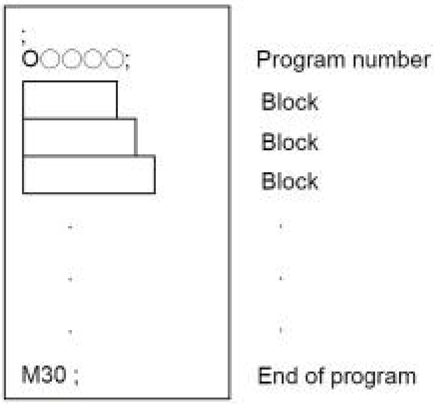


Fig2.27 Program Configuration

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:

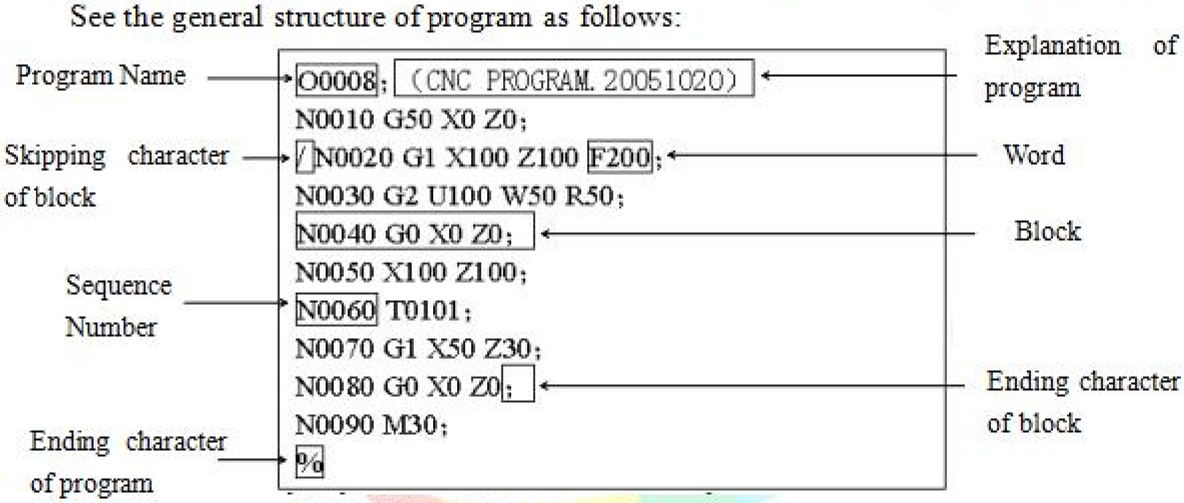


Fig2.28 General Structure of Program

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
F	0.01-15000mm/min	Feedrate per minute
	0.001-500mm/r	Feedrate per rev
	0.1~1000mm	Thread Lead in Metric
X	±99999.999mm	Coordinates in X directtion
	0~9999.999(s)	Dwell time
U	±99999.999mm	Increment in X directtion
	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 directtion
W	±99999.999mm	Increment in Z directtion
	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, program executed flow, subprogram call
	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.

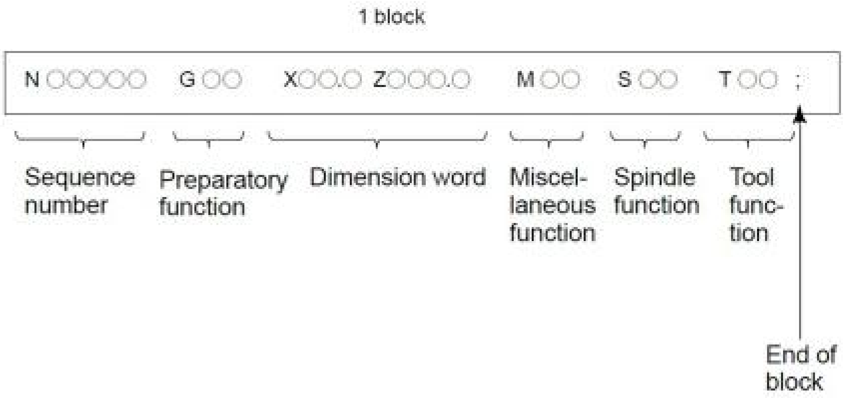


Fig2.29 Block Configuration

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).

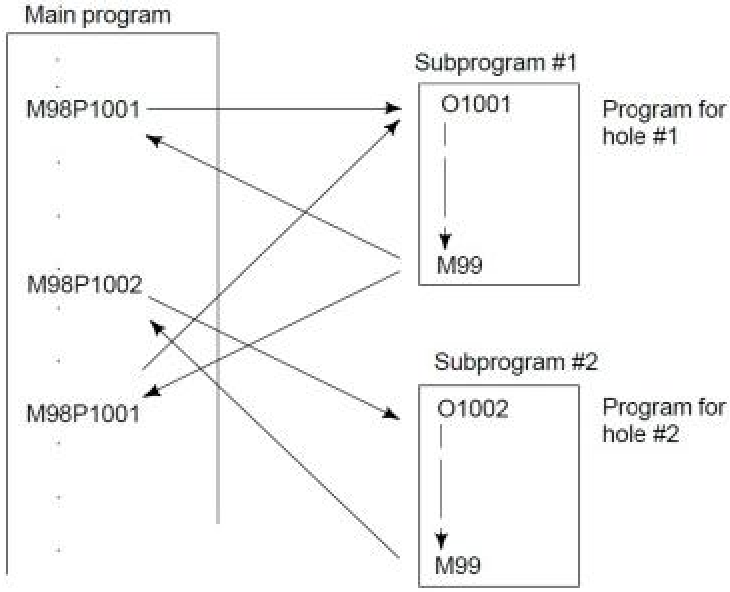
The contents of the dimension word depend on the preparatory function.

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 Processing Program

⇩

Excute: Press "F7" key : "Excut",system will jump to mn 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 comand 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 unti another G code of the same group is specified.

Eg.: G01 and G00 are modal G codes.

G01X_;
Z_;
X_; } **G01 is effective in this range**
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	00	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	00	Skip function (No alarm)	
G311		Skip function (alarm)	
G22	08	Program Block Cycle	
G800		Program Block Cycle Cancel	
G26	03	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	04	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	00	Save value of current coordinate	
G28	00	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	00	Coordinate system setting or max. spindle speed setting	
G52		Local coordinate system setting	
G53		Machine Coordinate System	
G54	14	Workpiece Coordinate System-1 Selection	
G55		Workpiece Coordinate System-2 Selection	
G56		Workpiece Coordinate System-3 Selection	
G57		Workpiece Coordinate System-4 Selection	
G58		Workpiece Coordinate System-5 Selection	
G59		Workpiece Coordinate System-6 Selection	
G184	00	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	05	Feeding per minute	
G99		Feeding per revolution	

G15	21	Polar coordinate interpolation cancel mode	
G16		Polar coordinate interpolation mode	
G990	12	Absolute programming	
G991		Incremental programming	
G20	06	Input in Inch	
G21		Input in mm	
G65	00	Macro calling	
G66	12	Macro modal call	
G67		Macro modal call cancel	
G04	00	Dwell	
G60	04	Exact Stop & Positioning	
G64		Continous Path Processing	
G40	07	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.

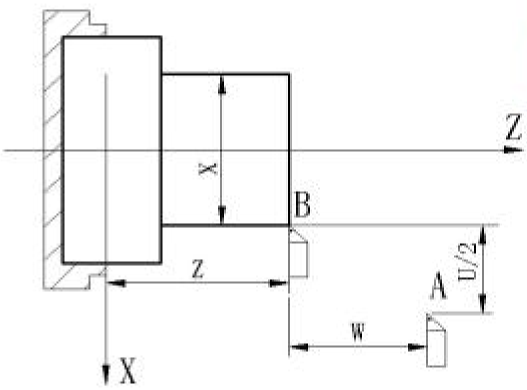


Fig3.1 Coordinate Code of G00

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.

Nonlinear interpolation positioning

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

Linear interpolation positioning

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

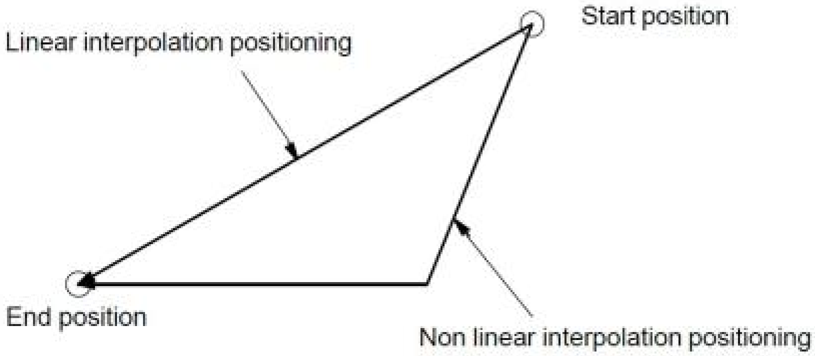


Fig3.2 Mode of Tool Path

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

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

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

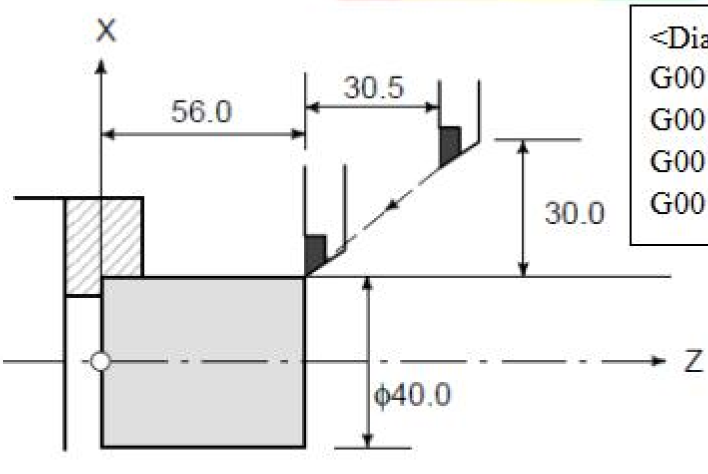
In the positioning mode actuated by G00, the tool is accelerated to a predetermined speed at the start of a block and is decelerated at the end of a block.

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

G01$\alpha\alpha\beta\beta$ ***Ff*;**

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>

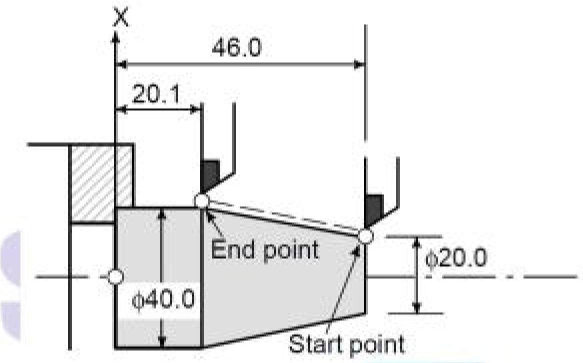


Fig3.4 Linear Interpolation of G01

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

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

G01 instruction can also specify movemtn 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 G1.

3.5 Circular Interpolation (G02/G03)

These commands will move a tool along a circular arc.

Format: Arc in the ZpXp plane (Default)

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

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 toolpost system & Rear toolpost system , the direction of G02&G03 is opposite in these two system.

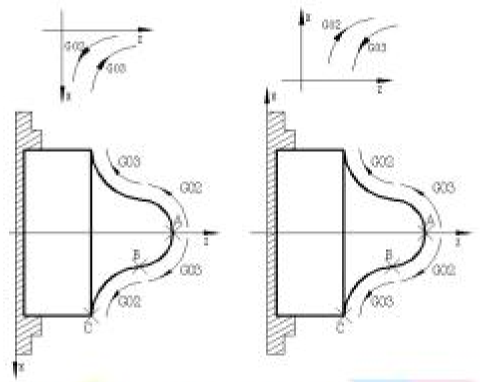


Fig3.5.1 Direction of G02&G03 at different toolpost system

"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.

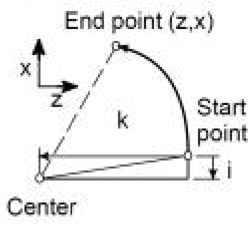


Fig3.5.2 Define & Direction of I & K

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

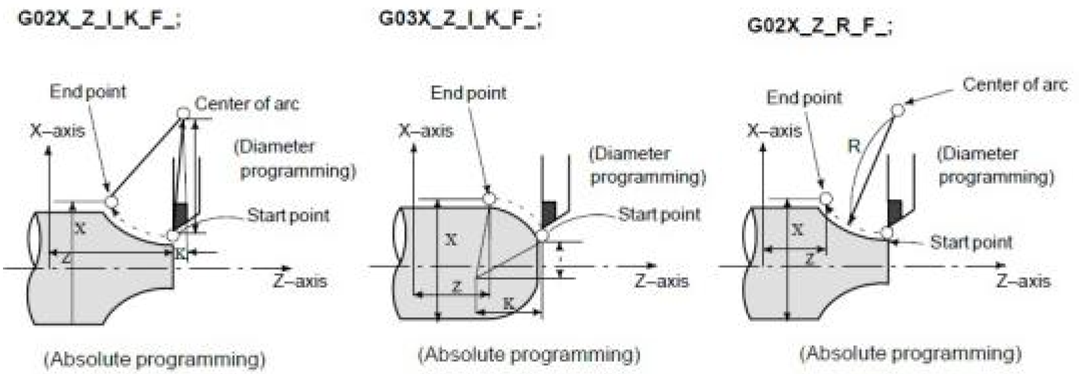
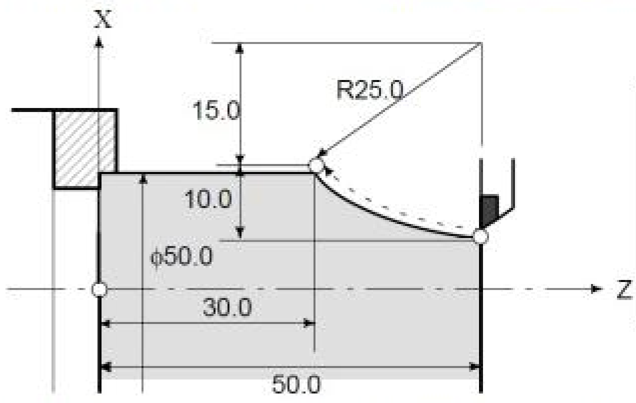


Fig3.5.3 Command of circular interpolation

Example 2: Processing same path with different command.

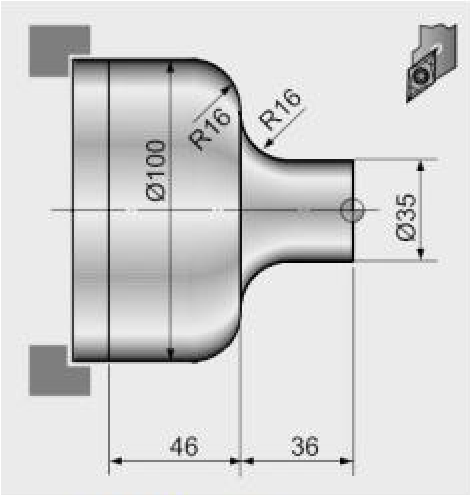


<Diameter Programming>

- 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




```
<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
```

Fig3.5.5 Example 3

SZGH-CNC1000TD CNC system support many kinds of thread cutting, which includes straight thread,tapered thread,scroll thread,thread cutting with variable lead, Continous 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 Axia 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 teeth of spindle motor is more than teeth of spindle encoder , it must match with keysets 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 processd 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

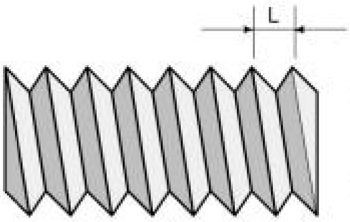


Fig3.6.1 Straight thread

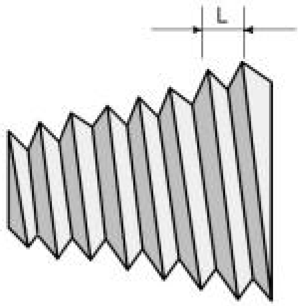


Fig3.6.2 Tapered Screw

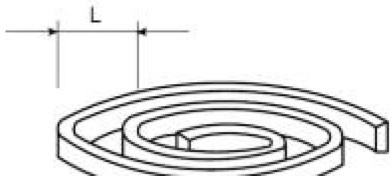


Fig3.6.3 Scroll Thread

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 /50 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°.

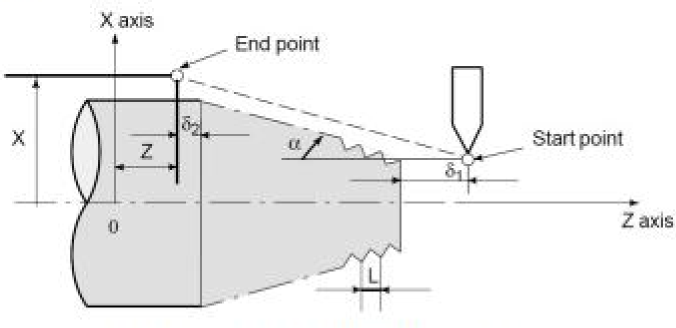
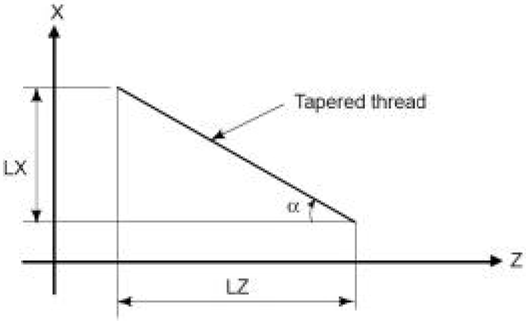


Fig3.6.4 Example of Thread Cutting

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 ≥ 45°, lead is LZ, Z-axis is long axis
α ≤ 45°, 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

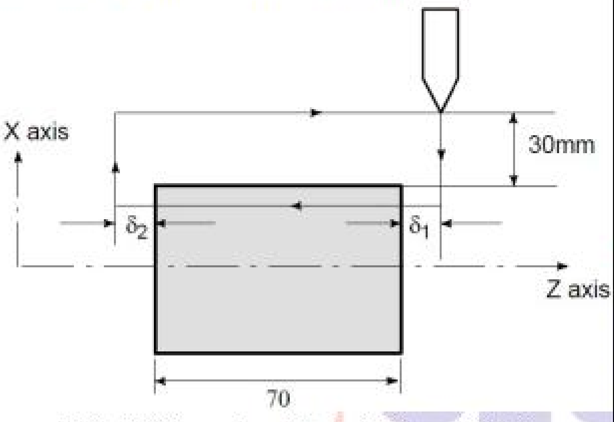


Fig3.6.6 Example of Straight Thread Cutting

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

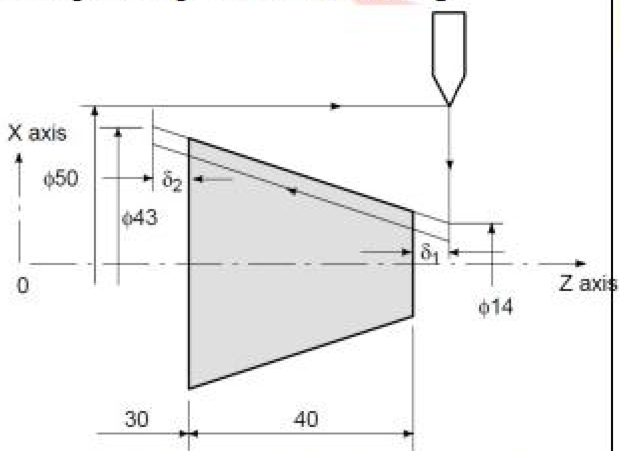


Fig3.6.7 Example of Tapered Thread Cutting

Requirement: (Metric input, Diameter programming)
Thread Lead:3.5mm in the direction of Zaxis,
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 Continous 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 synchronism 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.

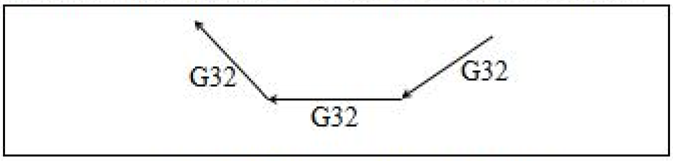


Fig3.6.8 Continuous Thread Cutting

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 finsihed by continuous input G32 command, input a thread length of each program, lead of thread(F) is different. At the second cycle , CNC sytem 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) od thread ;

SP_ : start angleThe start angle is not a continuous–state (modal) value. It must bespecified 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

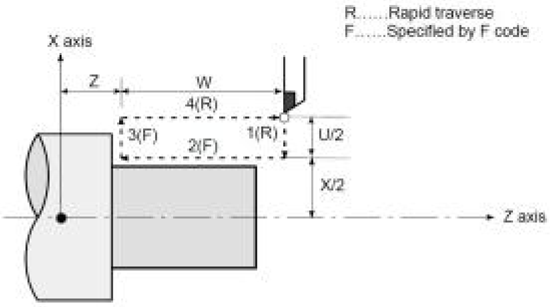


Fig3.8.1 Straight Cutting Cycle

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.

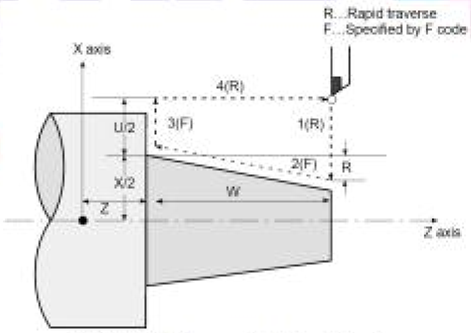
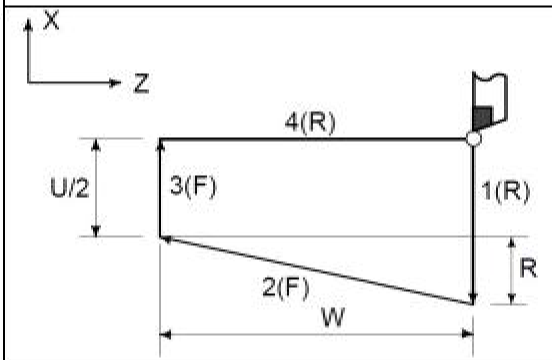
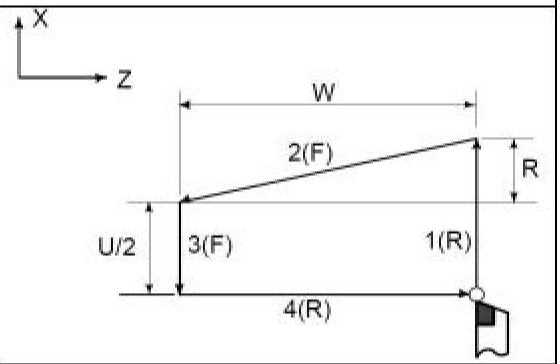
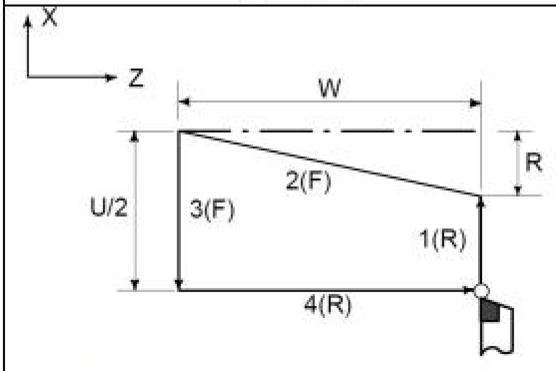
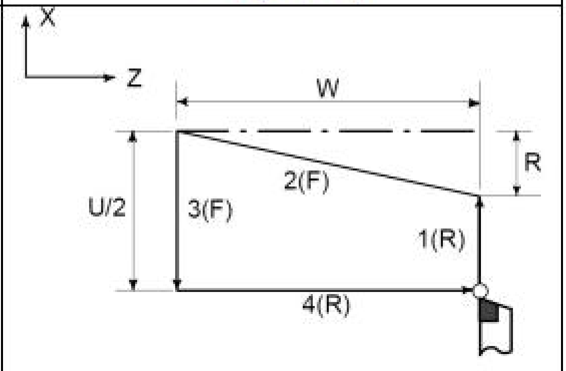


Fig3.8.2 Taper Cutting Cycle

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

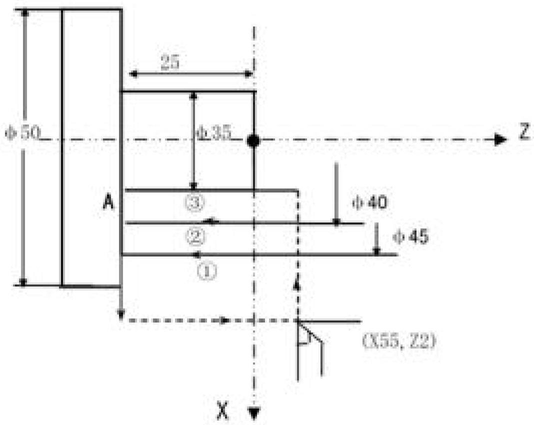
① The tool rapidly traverses from starting point to cutting starting point in X direction;

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.

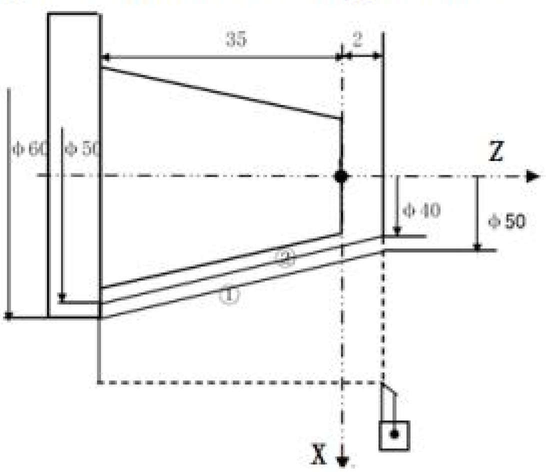
Example1: Use G90 to Process Cylinder Surface




```
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 infeeds 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.

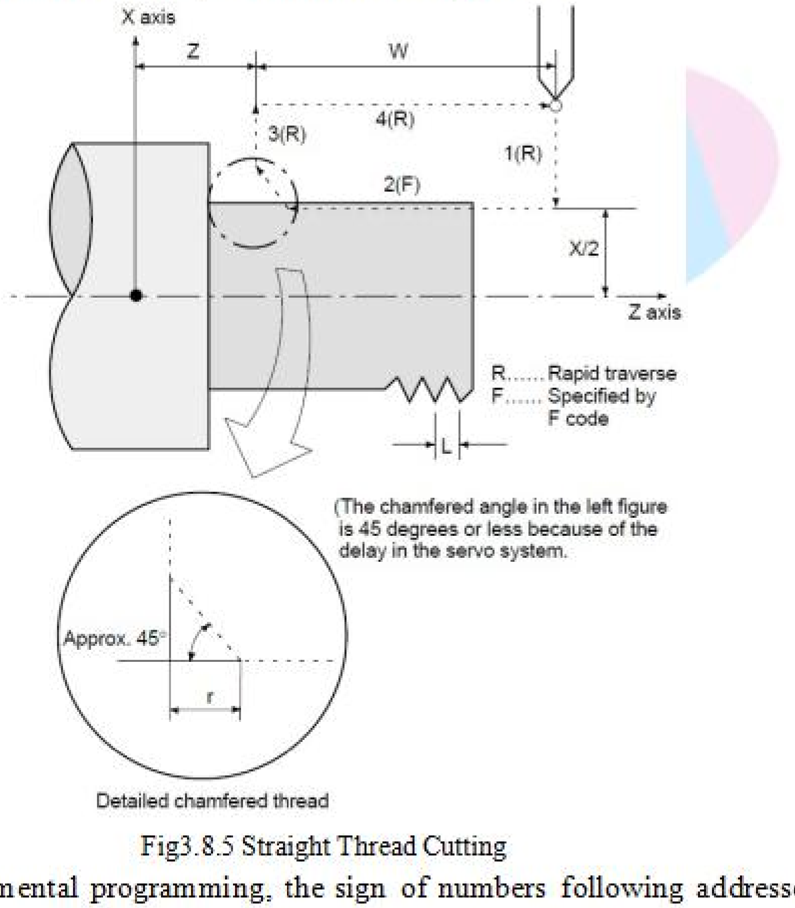


Fig3.8.5 Straight Thread Cutting

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).

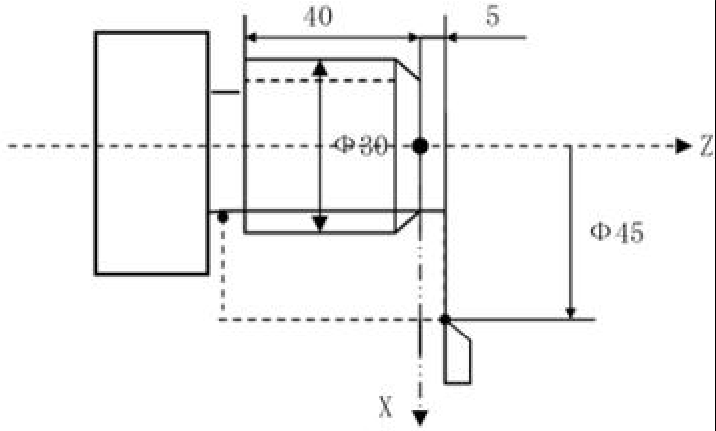


Fig3.8.6 Example of Straight Thread Cutting

```
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.

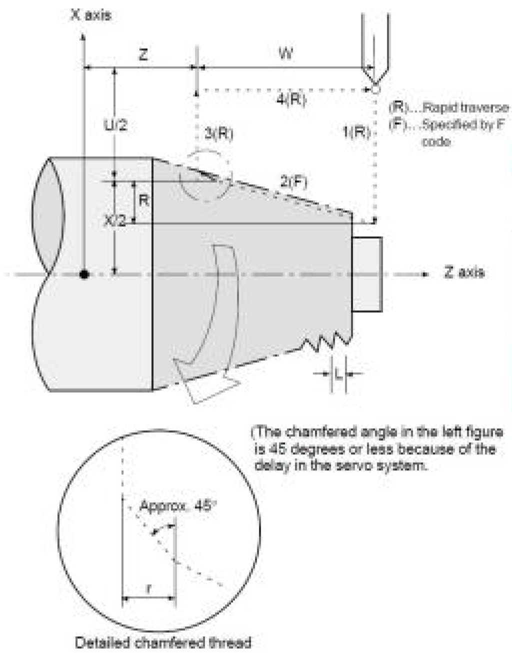


Fig3.8.7 Taper Thread Cutting

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

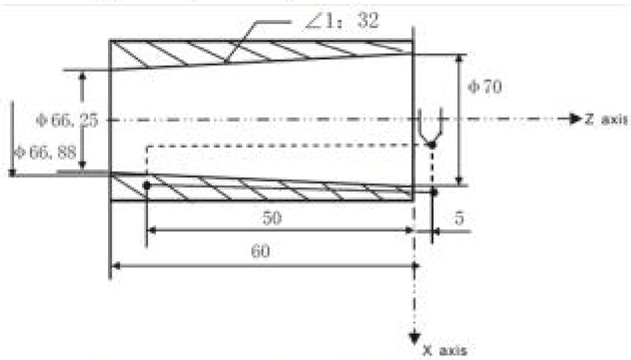


Fig3.8.8 Example of Taper Thread Cutting

```
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 inhomogeneous,the higher speed of spindle the longer of inhomogeneous 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 240° 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 automatical backing tail in thread processing to produce a qualified section of thread. Including the program format of backing tail in thread:

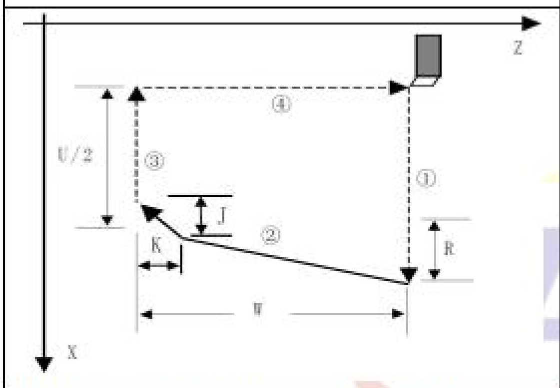
G92 X_ Z_ F_ J_ K_P_;

- 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.

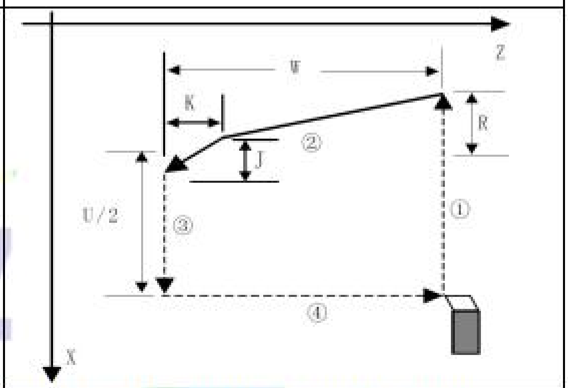
(7) Path of G92 Instruction

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.

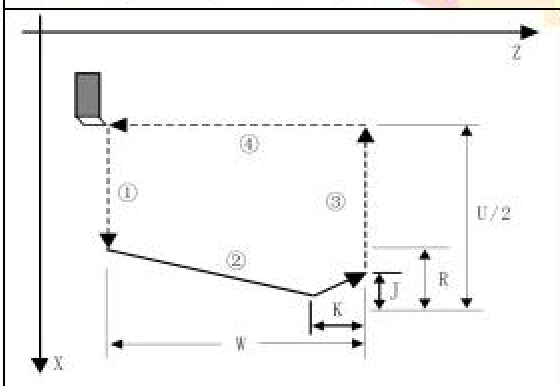
1) U>0 , W>0 , R>0



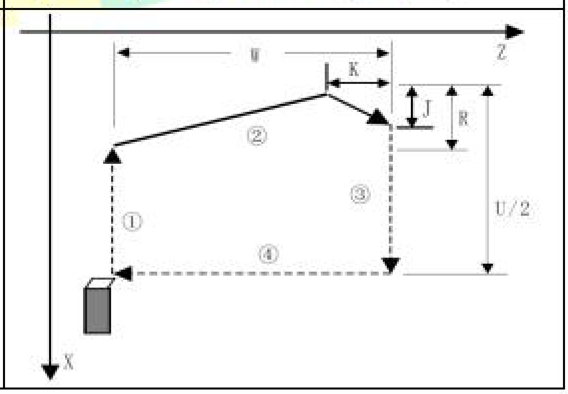
2) U<0 , W<0 , R<0



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 progamming ;
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

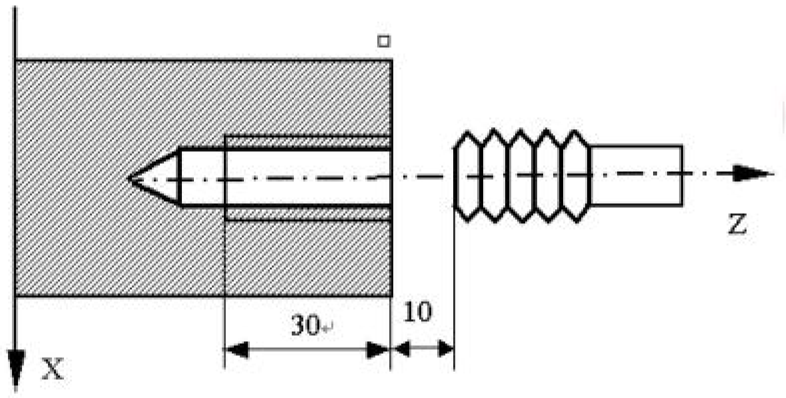


Fig3.8.9 Example of Canned Tapping Cycle

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 : "compesation 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 backlahs 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;

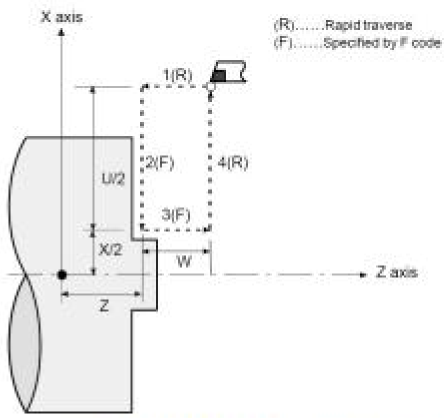


Fig3.8.10 End face loop cutting

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:

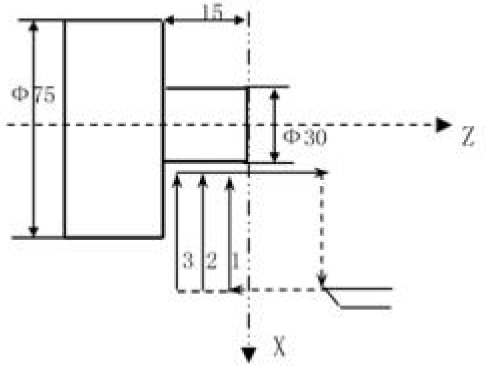


Fig3.8.11 Usage of G94

```
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.

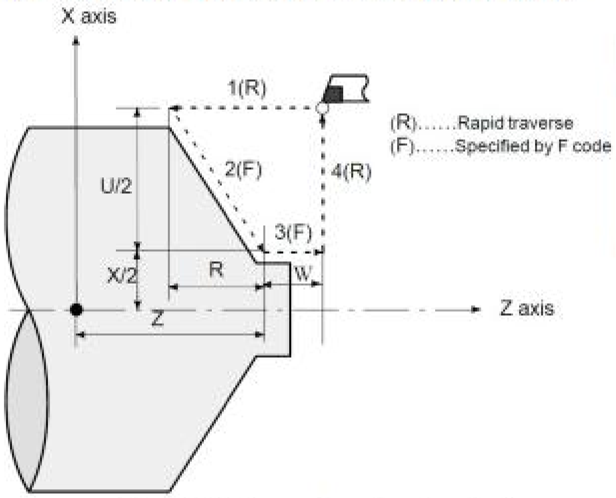
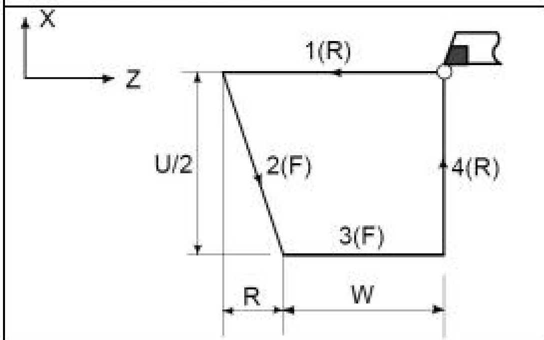
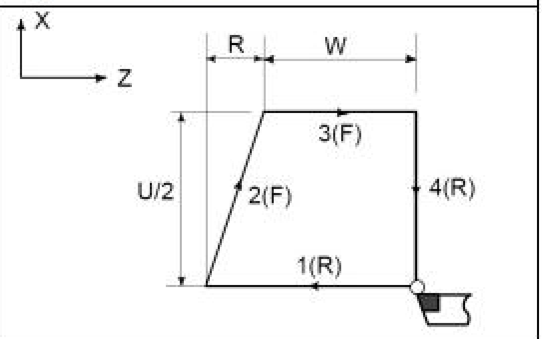
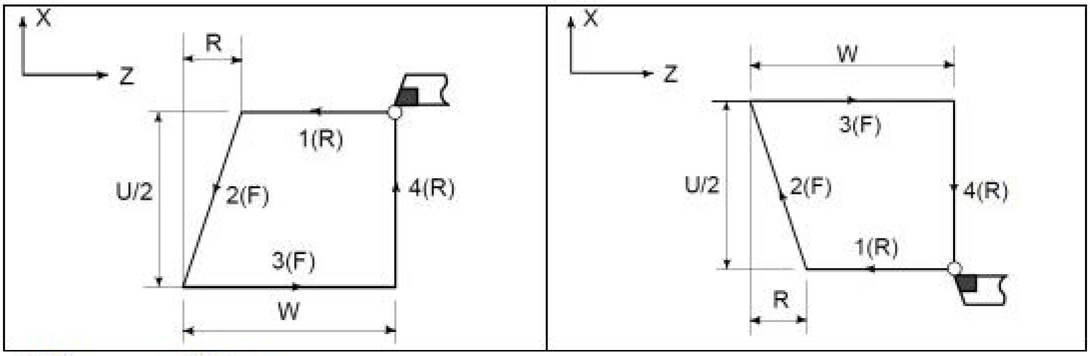


Fig3.8.12 Taper Face Cutting Cycle

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:

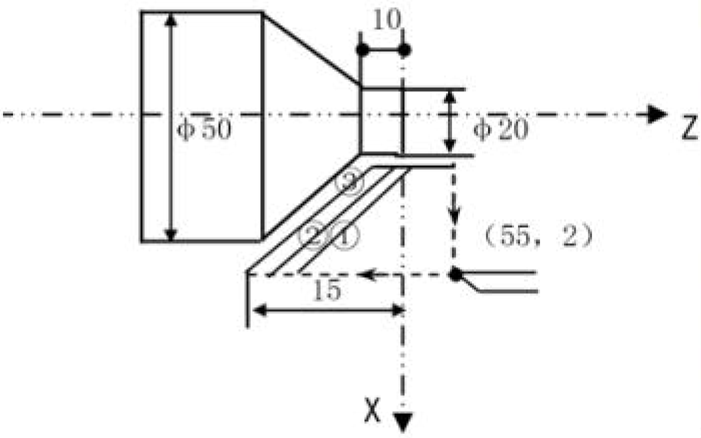
1. U<0 , W<0 , R<0	2. U>0 , W<0 , R
	
3. U<0 , W<0 , R>0 at |R| ≤ |W|	4. U>0 , W<0 , R<0 at |R| ≤ |W|



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.

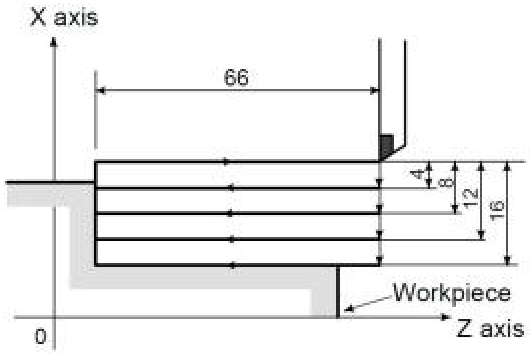
2. The following two applications can be performed.

(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 ;

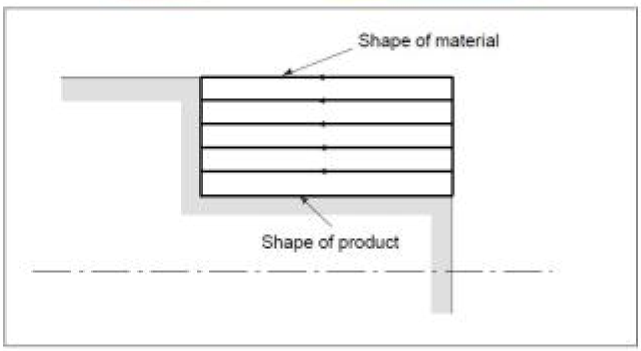
Example2:

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

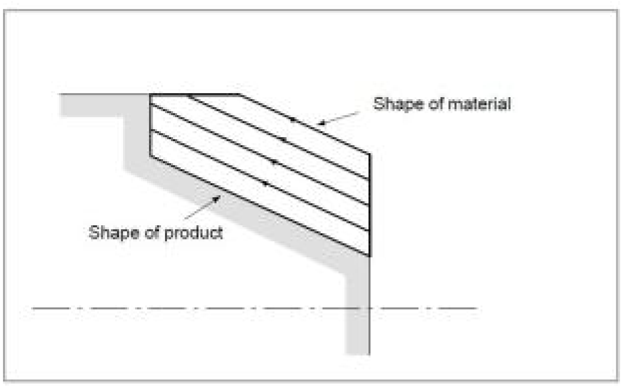
3.8.5 Usage for Canned Cycle

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

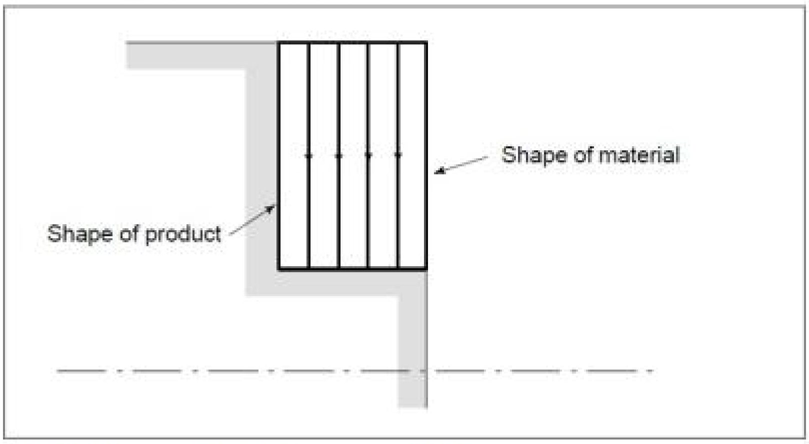
1) Straight Cutting Cycle (G90):



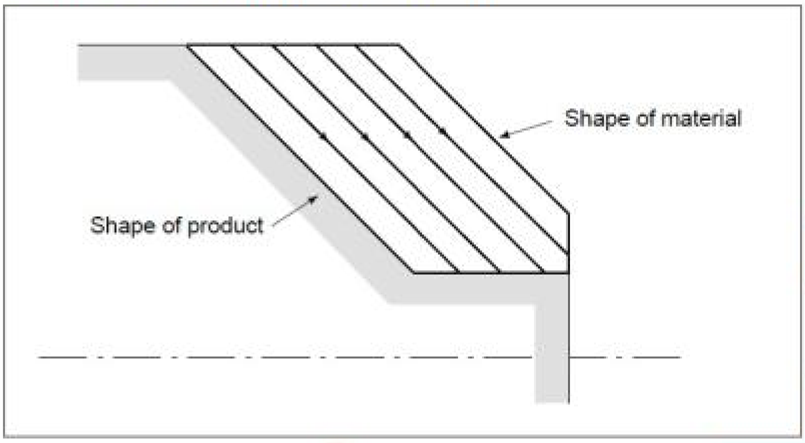
2) Taper Cutting Cycle (G90):



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 △d (depth of cut), with finishing allowance △u/2 and △w left.

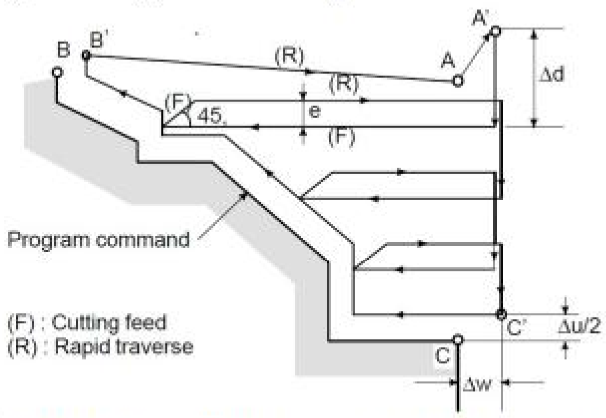


Fig3.9.1 Cutting Path in Axial Roughing Turning Cycle

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 funtion 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:

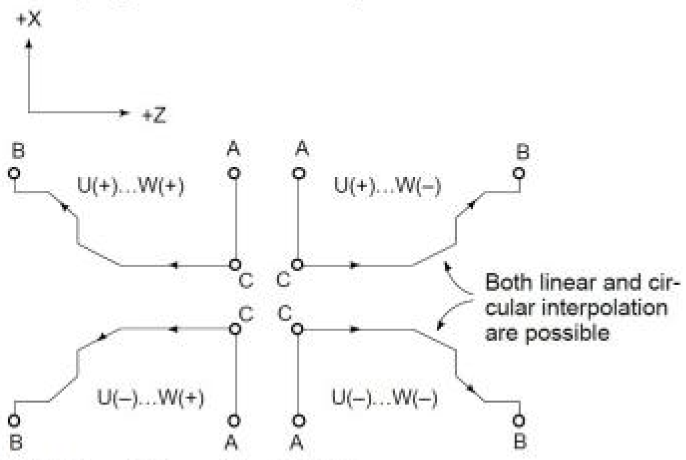


Fig3.9.2 Sign of Δu and Δw in G71

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.

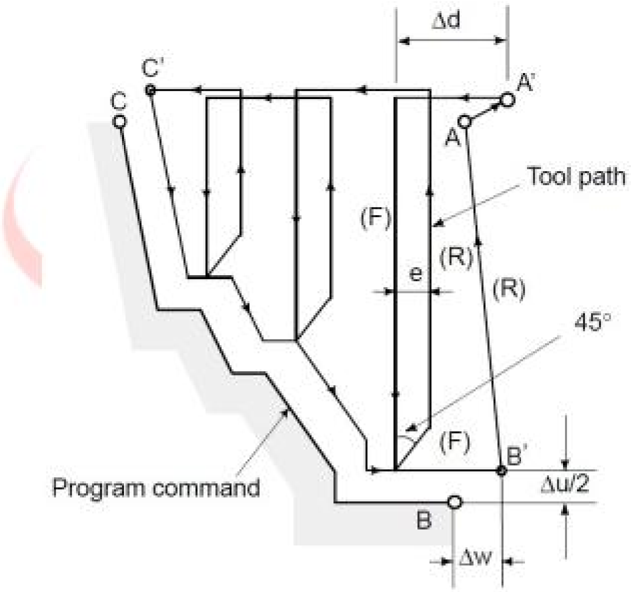


Fig3.9.3 Cutting Path in Radial Roughing Facing Cycle

Execution Process: (reference as Fig3.9.3)

① Rapid traverse from A point to A' point, the travel in X direction is △u, and the travel in Z direction is △w;

② The travel in Z direction from A' is △d(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 (45° 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

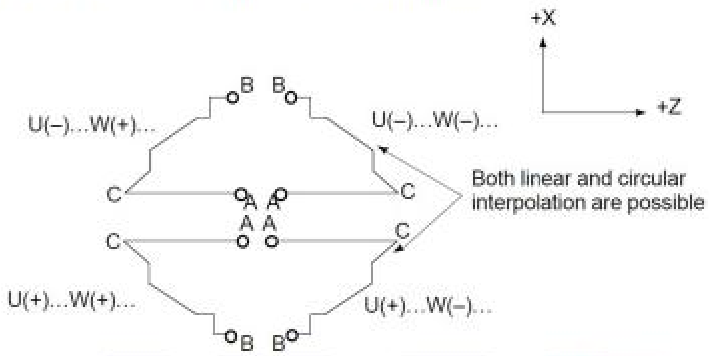


Fig3.9.4 Signs of numbers specified with u and W in G72

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.

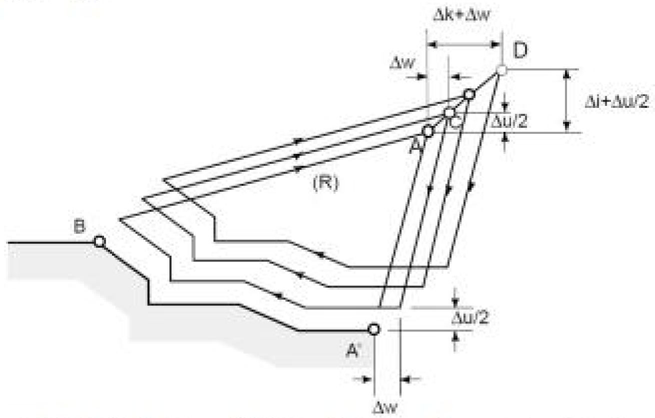


Fig3.9.5 Cutting Path in Pattern Repeating Cycle

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, addreses 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 , Δu<0 Δw>0 ;	**2) Δi>0 Δk>0 , Δu>0 Δw>0 ;**