MACHINING

Level - III

Learning Guide 3

Unit of Competence: Perform Advanced lathe CNC Operations

Module Title: Performing Advanced lathe CNC Operations

LG Code: **IND MAC3 03 0217**

Instruction Sheet	Learning Guide #1

This learning guide is developed to provide you the necessary information regarding the following **content coverage** and topics:

1. Determine job requirements
2. Prepare CNC lathe machining process / Write program
3. Perform appropriate Lathe operations

This guide will also assist you to attain the learning outcome stated in the cover page. Specifically, **upon completion of this Learning Guide, you will be able to**:

- Cutting tools, instruments and machine accessories are selected according to the requirements of the operation.
- Cutting speed and feeds rate are calculated based on work- piece and cutting tool material specifications
- written in standard CNC lathe operations, code format and in accordance with standard operating procedures.
- CNC lathe operations are performed to produce component according to drawing specifications.

Learning Instructions:

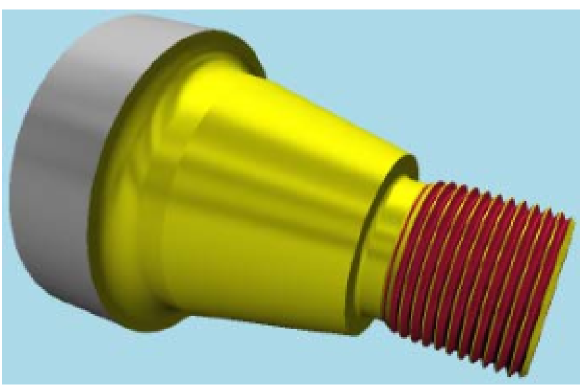
1. Read the specific objectives of this Learning Guide.
2. Follow the instructions described below 3 to 100.
3. Read the information written in the information "Sheet.
4. Accomplish the "Self-check test.
5. Do the "LAP test".

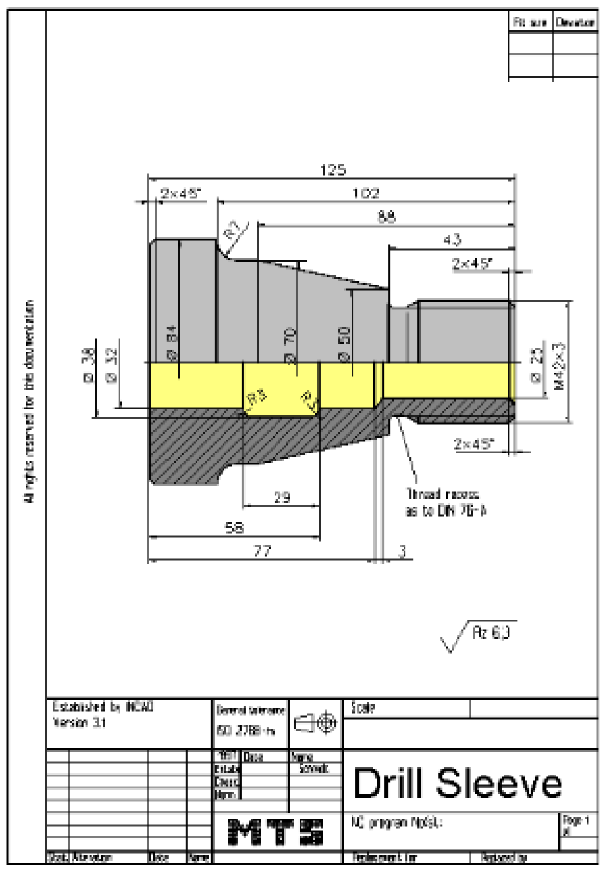
Determine job requirements

Introduction:

In CNC Milling Machining, operator or machinist must have to understand the drawing, interpret and analyze it as a reference to produce program manual script prior to manufacture the part. The following are guide before machining a part:

1. Drawings need to be interpreted to produce component as per specifications.
2. The sequence of operation should be established as well to determine what steps in producing the component according to specification.
3. Cutting tools are selected according to the requirements of the operation.
4. Cutting speed and feed rate calculated based on work- piece and cutting tool material.
5. Process / job adjustment sheets are filled up with relevant machine, tool and raw material data.



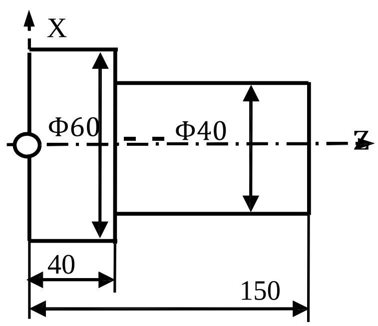


CNC Programming

To operate CNC machine tool, the first step is to understand the part drawing and produce a program manual script. The procedure for machining a part is as follows

1) Read drawing
2) Produce the program manual script
3) Input the program manual script by using the machine control panel
4) Manufacture a part

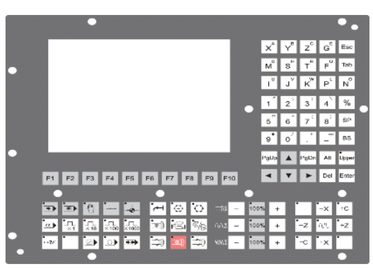
1. Read drawing



2. Produce the program manual script

N1 T0106
N2 M03 S460
N3 G00 X90Z20
N4 G00 X31Z3
N5 G01 Z-50 F100
N6 G00 X36
N7 Z3
…

3. Input the program manual script



4. Manufacture a part

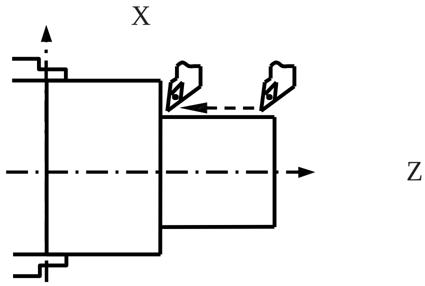


Figure 1.1 The workflow of operation of CNC machine tool

Interpolation

Interpolation refers to an operation in which the machine tool moves along the workpiece parts. There are five methods of interpolation: linear, circular, helical, parabolic, and cubic. Most CNC machine can provide linear interpolation and circular interpolation. The other three methods of interpolation (helical, parabolic, and cubic interpolation) are usually used to manufacture the complex shapes, such as aerospace parts. In this manual, linear and circular interpolation are introduced.

Linear Interpolation

There are two kinds of linear interpolation:

1) Tool movement along a straight line

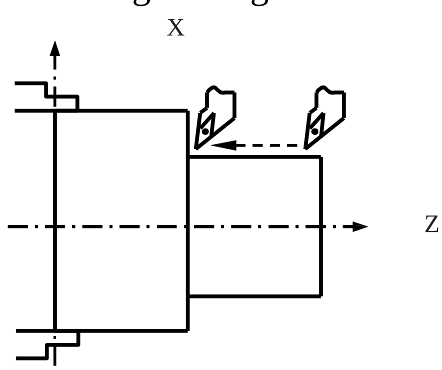


Figure 1.2 Linear Interpolation (1)

2) Tool movement along the taper line

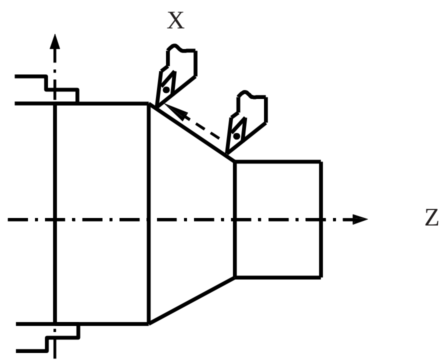


Figure 1.3 Linear Interpolation (2)

Circular Interpolation

Figure 1.4 shows a tool movement along an arc.

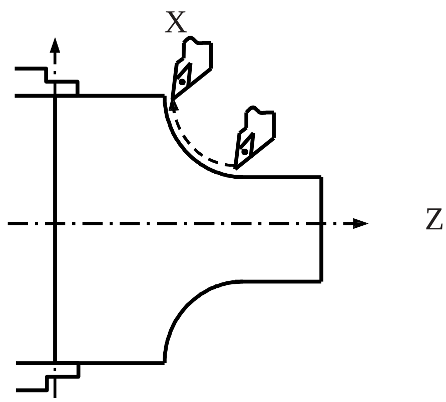


Figure 1.4 Circular Interpolation

Note:

In this manual, it is assumed that tools are moved against workpieces.

Thread Cutting

There are several kinds of threads: cylindrical, taper or face threads. To cut threads on a workpiece, the tool is moved with spindle rotation synchronously.

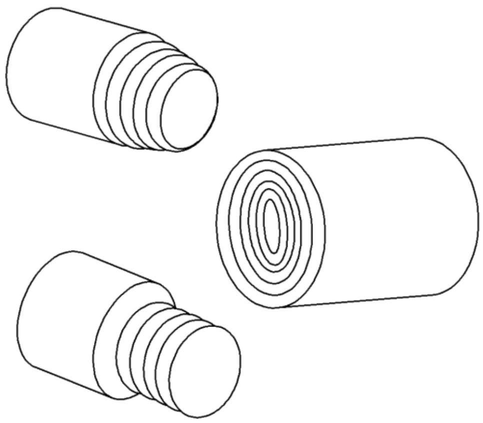


Figure 1.5 Thread Cutting

Feed Function

- Feed refers to an operation in which the tool moves at a specified speed to cut a workpiece.
- Feedrate refers to a specified speed, and numeric is used to specified the feedrate.
- Feed function refers to an operation to control the feedrate.

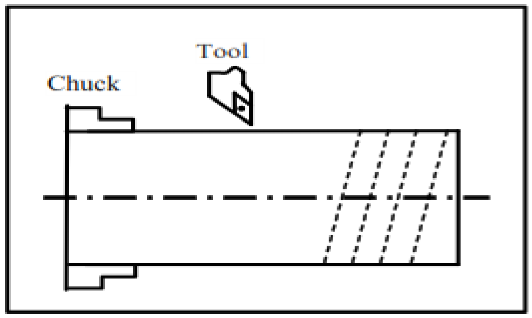


Figure 1.6 Feed Function

For example:

F2.0 //feed the tool 2mm, while the workpiece makes one turn

Coordinate System

Reference Point

Reference point is a fixed position on CNC machine tool, which is determined by cams and measuring system. Generally, it is used when the tool is required to exchange or the coordinate system is required to set.

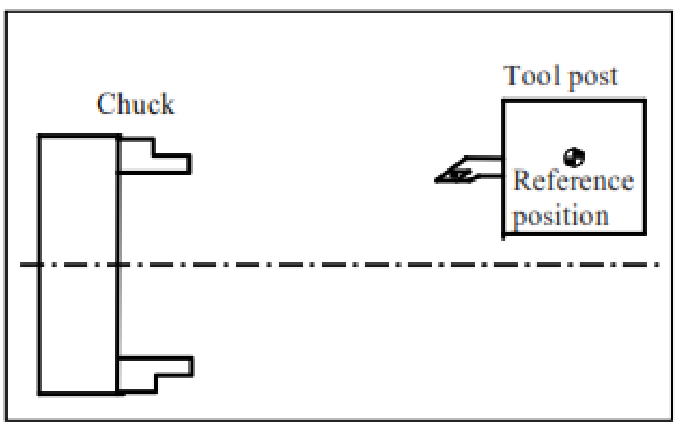


Figure 1.7 Reference Point

There are two ways to move to the reference point:

- Manual reference position return: The tool is moved to the reference point by operating the button on the machine control panel. It is only used when the machine is turned on.
- Automatic reference position return: It is used after the manual reference position return has been used. In this manual, this would be introduced.

Machine Coordinate System

The coordinate system is set on a CNC machine tool. Figure 1.8 is a machine coordinate system of turning machine, and shows the direction of axes:

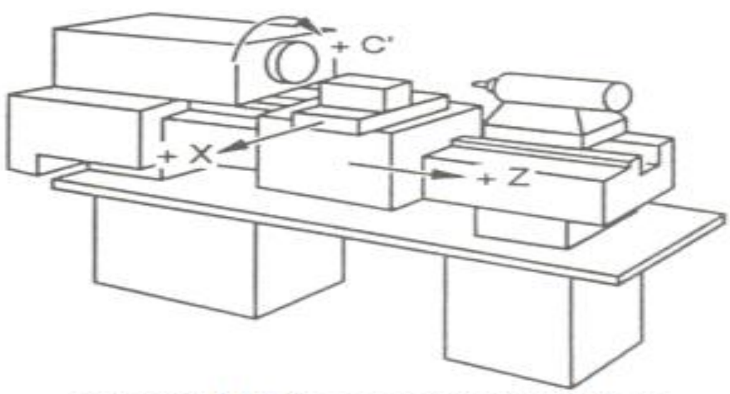


Figure 1.8 Machine Coordinate System

In general, three basic linear coordinate axes of motion are X, Y, Z. Moreover, X, Y, Z axis of rotation is named as A, B, C correspondently. Due to different types of turning machine, the axis direction can be decided by following the rule – "three finger rule" of the right hand.

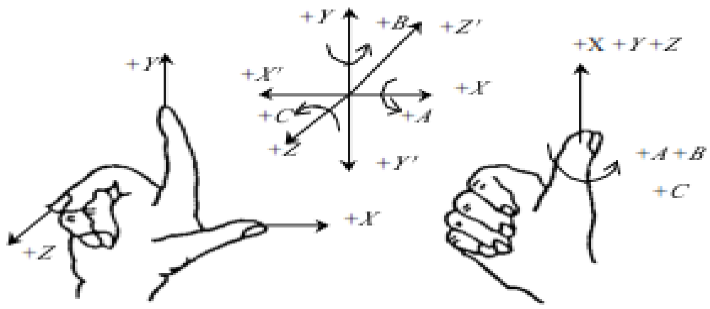


Figure 1.9 “three finger rule”

- The thumb points the X axis. X axis controls the cross motion of the cutting tool. “+X” means that the tool is away from the spindle centerline
- The index points the Y axis. Y axis is usually a virtual axis.
- The middle finger points the Z axis. Z axis controls the motion of the cutting tool. “+Z” means that the tool is away from the spindle.

Workpiece Coordinate System

The coordinate system is set on a workpiece. The data in the NC program is from the workpiece coordinate system.

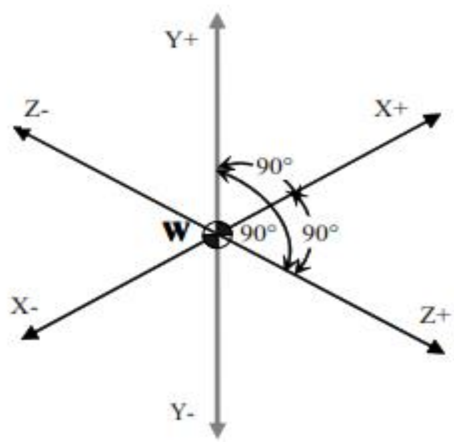


Figure 1.10 Workpiece Coordinate System

Example: Those four points can be defined on workpiece coordinate system:

P1 corresponds to X25 Z-7.5
P2 corresponds to X40 Z-15

P3 corresponds to X40 Z-25

P4 corresponds to X60 Z-35

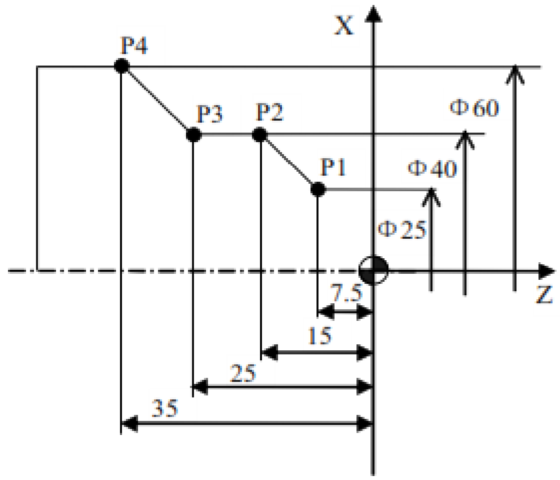


Figure 1.11 Example of defining points on workpiece coordinate system

Setting Two Coordinate Systems at the Same Position

There are two methods used to define two coordinate systems at the same position.

1) The coordinate zero point is set at chuck face

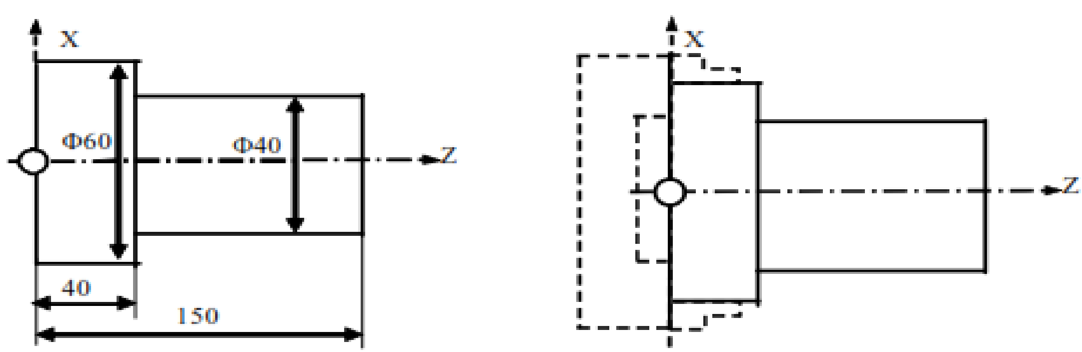


Figure 1.12 The coordinate zero point set at chuck face

2) The coordinate zero point is set at the end face of workpiece

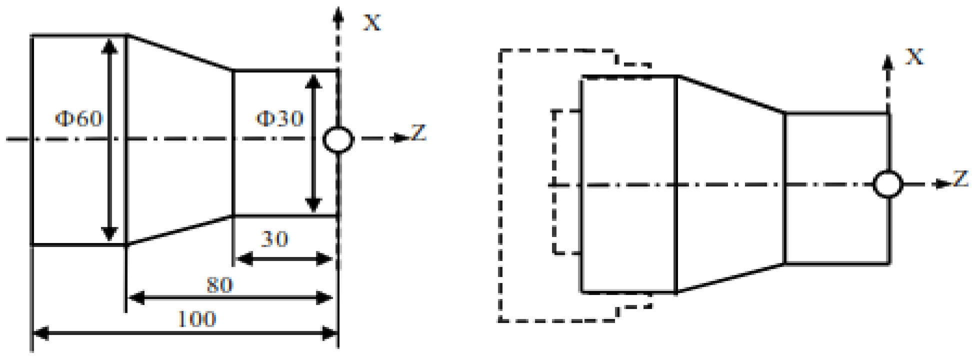


Figure 1.13 The coordinate zero point set at the end face of workpiece

Absolute Commands

The absolute dimension describes a point at "the distance from zero point of the coordinate system".

Example: These four point in absolute dimensions are the following:

P1 corresponds to X25 Z-7.5

P2 corresponds to X40 Z-15

P3 corresponds to X40 Z-25

P4 corresponds to X60 Z-35

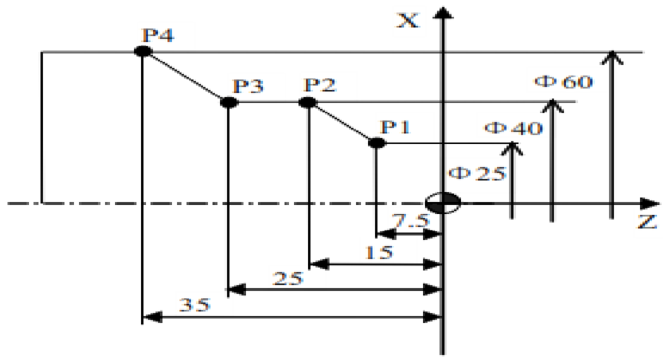


Figure 1.14 Absolute Dimension

Incremental Commands

The incremental dimension describes a distance from the previous tool position to the next tool position.

Example: These four point in incremental dimensions are the following:

P1	X25 Z-	//with reference to the
P2	X15 Z-	//with reference to P1
P3	Z-10	//with reference to P2
P4	X20 Z-	//with reference to P3

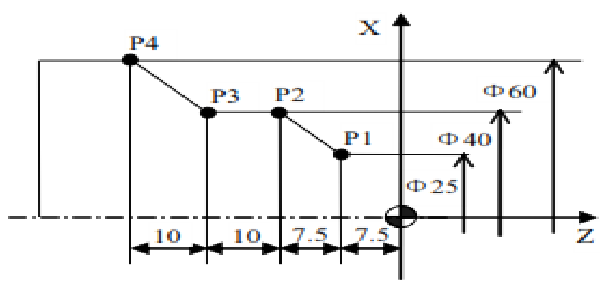


Figure 1.15 Incremental Dimension

Diameter/Radius Programming

The coordinate dimension on X axis can be set in diameter or radius. It should be noted that diameter programming or radius programming should be applied independently on each machine.

Example: Describe the points by diameter programming.

A corresponds to X30 Z80

B corresponds to X40 Z60

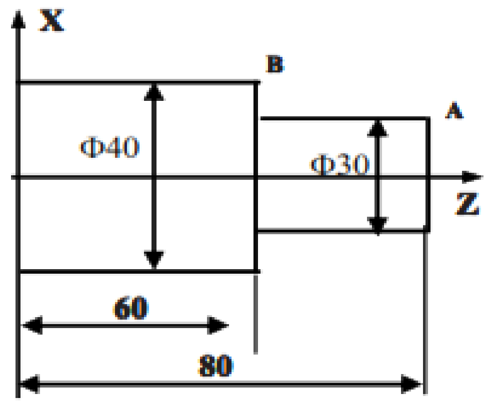


Figure 1.16 Diameter Programming

Example: Describe the points by radius programming.

A corresponds to X15 Z80

B corresponds to X20 Z60

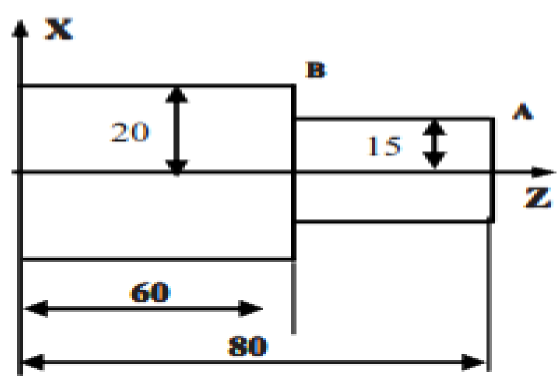


Figure 1.17 Radius Programming

Spindle Speed Function

The cutting speed (v) refers to the speed of the tool with respect to the workpiece when the workpiece is cut. The unit of the cutting speed is m/min. As for the CNC, the cutting speed can be specified by the spindle speed (N) in min^{-1}.

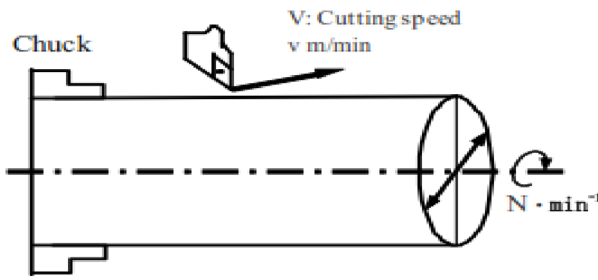


Figure 1.18 Cutting Speed and Spindle Speed

The formula to get the spindle speed is: $N = \dfrac{1000 * v}{\pi D}$

N: the spindle speed

v: cutting speed

D: diameter value of the workpiece

Example: When the diameter of workpiece is 200mm, and the cutting speed is 300m/min,

then the spindle speed: $N = \dfrac{1000 * v}{\pi D} = \dfrac{1000 * 300}{\pi * 200} \approx 478 r/m$

The constant surface speed refers to the cutting speed even when the workpiece diameter is changed, and the CNC changes the spindle speed.

Tool Function

Tool Selection

It is necessary to select a suitable tool when drilling, tapping, boring or the like is performed. As it is shown in Figure 1.19, a number is assigned to each tool. Then this number is used in the program to specify that the corresponding tool is selected.

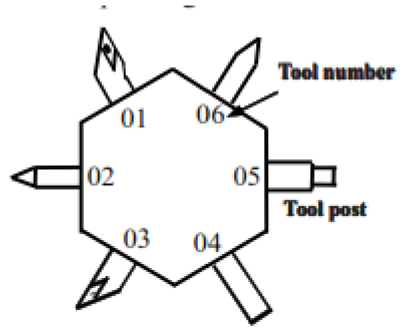


Figure 1.19 Tool Selection

Tool Offset

When writing a program, the operator just use the workpiece dimensions according to the dimensions in the part drawing. The tool nose radius center, the tool direction of the turning tool, and the tool length are not taken into account. However, when machining a workpiece, the tool path is affected by the tool geometry.

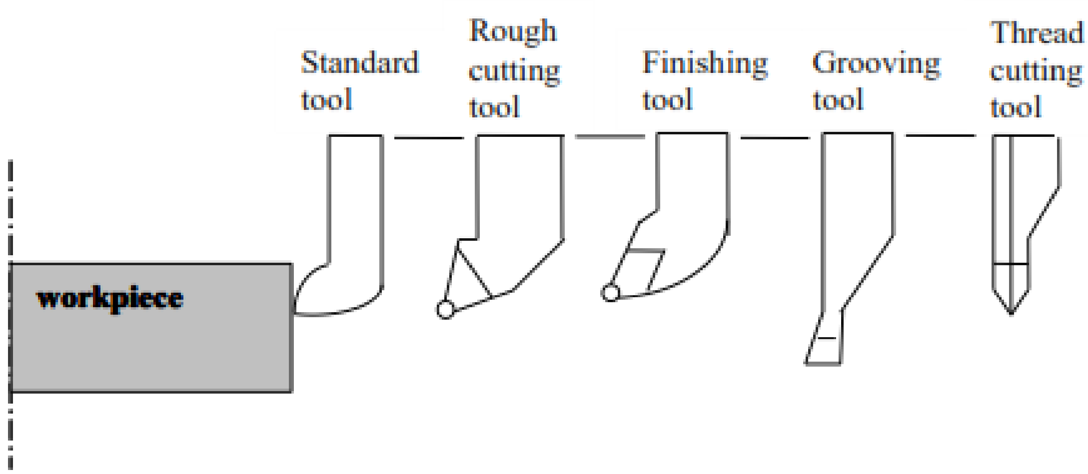


Figure 1.20 Tool Offset

Tool Length Compensation

There are two kind of ways to specify the value of tool length compensation.

- Absolute value of tool length compensation (the distance between tool tip and machine reference point)

- ➢ Incremental value of tool length compensation (the distance between tool tip and the standard tool)

As it is shown in Figure 1.21, L1 is the tool length on X axis. L2 is the tool length on Z axis. It should be noted that the tool wear values on X axis or Z axis are also contained in the tool length compensation.

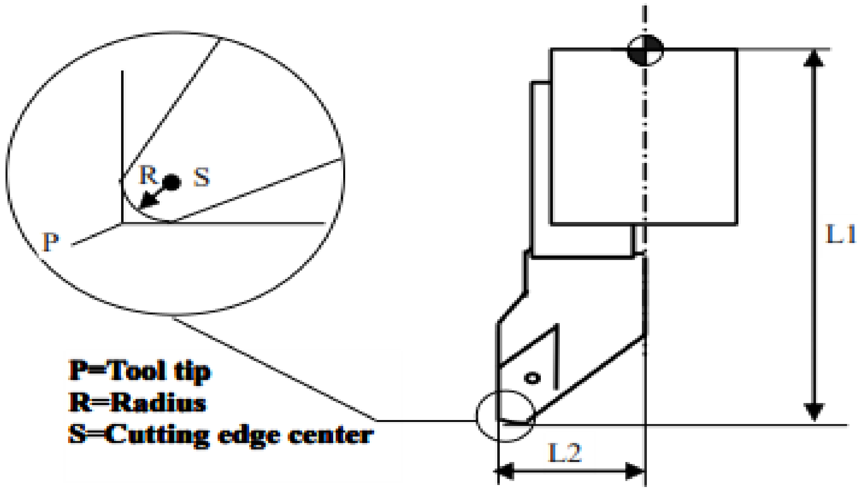


Figure 1.21 Tool Length Compensation

Tool Radius Compensation

Figure 1.22 shows the imaginary tool nose as a start position when writing a program.

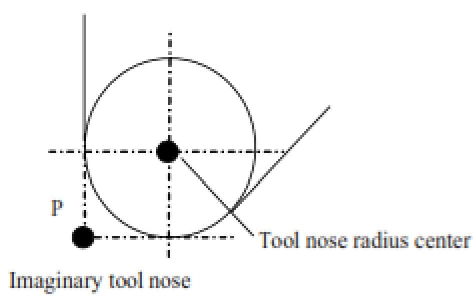


Figure 1.22 The imaginary tool nose

Miscellaneous Function

Miscellaneous function refers to the operation to control the spindle, feed, and coolant. In general, it is specified by an M code.
When a move command and M code are specified in the same block, there are two ways to execute these commands:

1) Pre-M function
   M command is executed before the completion of move command
2) Post-M function
   M command is executed after the completion of move command.

The sequence of the execution depends on the specification of the machine tool builder.

Program Configuration

Structure of an NC Program

As it is shown in Figure 1.25, an NC program consists of a sequence of NC **blocks**. Each block is one of machining steps. **Commands** in each block are the instruction.

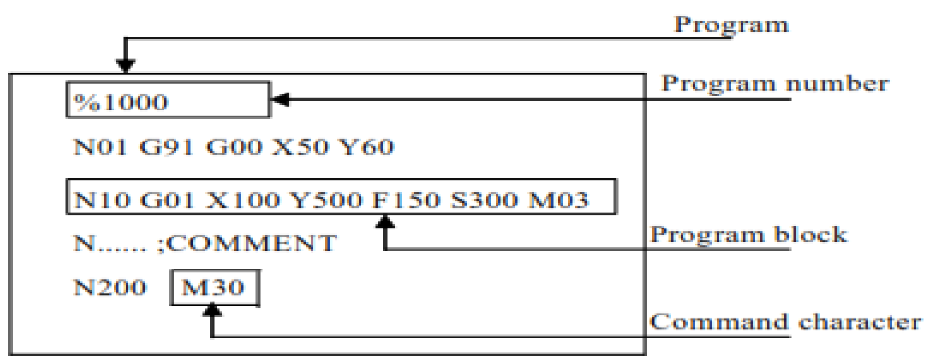


Figure 1.25 Structure of an NC Program

- Format of **program name**

The program name must be specified in the format OXXXX (X could be letters or numbers).

- Format of **program number**

The program number should be started with %XXXX or OXXXX (X could be numbers only).

- Format of **blocks**

A block starts with the program block number.

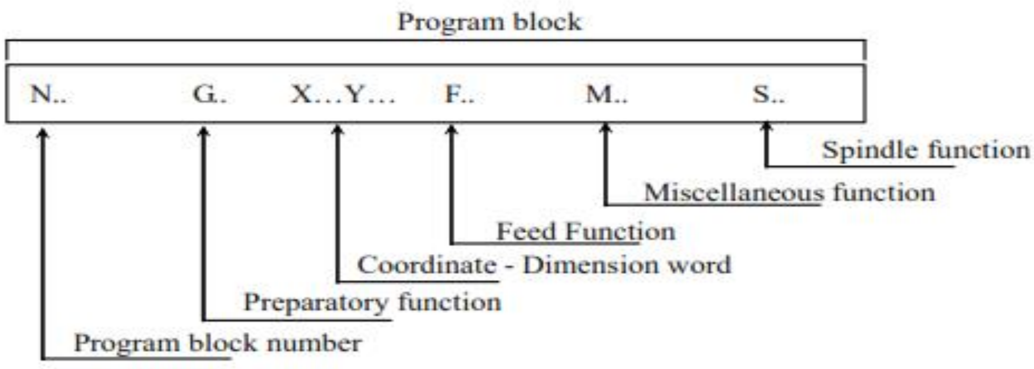


Figure 1.26 Structure of Block

- Format of **end of program**

The last block should contain M02 or M03 to indicate the end of program.

- Format of **Comments**

All information after the ";" is regarded as comments.

All information between "( )" is regarded as comments.

Main Program and Subprogram

There are two type of program: main program and subprogram. The CNC operates according to the main program. When a execution command of subprogram is at the execution line of the main program, the subprogram is called. When the execution of subprogram is finished, the system returns control to the main program.

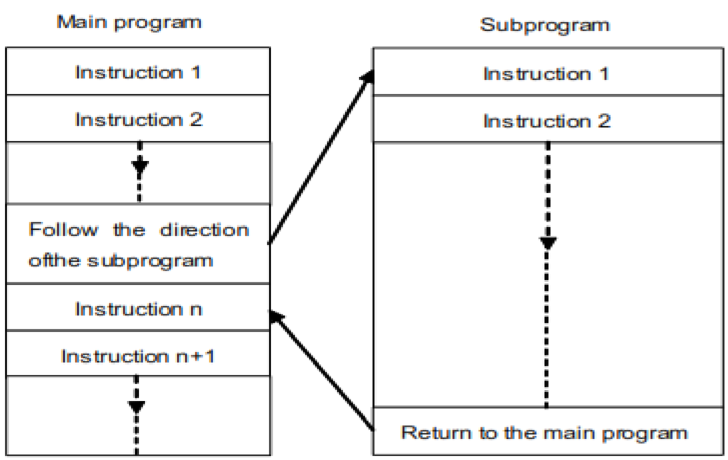


Figure 1.27 Main program and subprogram

Note:

Main program and its subprogram must be written in a same file with a different program codes.

Preparatory Function (G code)

There are two types of G code: one-shot G code, and modal G code.

Table 2 1 Type of G code

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

Example: G01 and G00 are modal G codes.

G00X_
Z_ G00 is effective in this range
X_
G01Z_

G code List

The following table is the list of G code in HNC system.

Table 2 2 G code list

G code	Group	Function
G00	01	Positioning (Rapid traverse)
◣ G01		Linear interpolation (Cutting feed)
G02		Circular interpolation CW
G03		Circular interpolation CCW
G04	00	Dwell
G20	08	Input in inch
◣ G21		Input in mm
G28	00	Reference point return
G29		Auto return from reference point
G32	01	Thread cutting with constant lead
G34		Tapping
◣ G36	17	Diameter programming
G37		Radius programming
◣ G40	09	Tool nose radius compensation cancel
G41		Tool nose radius compensation on the left
G42		Tool nose radius compensation on the right
G46	16	Setting the limit of spindle speed
◣ G50	04	Canceling the workpiece's origin movement
G51		Moving the origin of workpiece coordinate system
G53	00	Selecting a machine coordinate system
◣ G54	11	Setting a workpiece coordinate system
G55		
G56		
G57		
G58		

G71		Stock Removal in Turning
G72		Stock Removal in Facing
G73		Pattern repeating
G74		Front drilling cycle
G75	06	Side drilling cycle
G76		Multiple thread cutting cycle
G80		Internal diameter/Outer diameter cutting cycle
G81		End face turning cycle
G82		Thread cutting cycle
◣G90	13	Absolute programming
G91		Incremental programming
G92	00	Setting a coordinate system
◣G94	14	Feedrate per minute
G95		Feedrate per revolution
G96	16	Constant cutting speed
◣G97		Constant cutting speed cancel

Explanation:

1) G codes in 00 group are one-shot G code, while the other groups are modal G code.

2) ◣ means that it is default setting.

Interpolation Functions

This chapter would introduce:

1) Positioning Command (G00)
2) Linear Interpolation (G01)
3) Circular Interpolation (G02, G03)
4) Chamfering and Rounding (G01, G02, G03)
5) Thread Cutting with Constant Lead (G32)
6) Tapping (G34)

Positioning (G00)

Programming

G00 X(U)… Z(W)…

Explanation of the parameters

X, Z Coordinate value of the end point in the absolute command

U, W Coordinate value of the end point in the incremental command

Function

The tool is moved at the highest possible speed (rapid traverse). If the rapid traverse movement is required to execute simultaneously on several axes, the rapid traverse speed is decided by the axis which takes the most time. The operator can use this function to position the tool rapidly, to travel around the workpiece, or to approach the tool change position.

Example

Move tool from P1 (45, 90) to P2 (10, 20) at the rapid traverse speed.

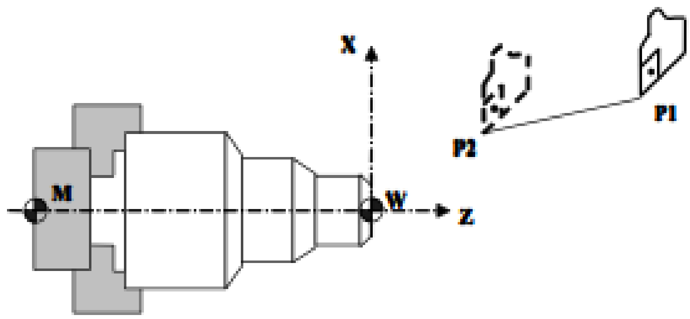


Figure 3.1 Positioning (Rapid Traverse)

Absolute programming:

G00 X10 Z20

Incremental programming:

G00 U30 W70

Linear Interpolation (G01)

Programming

G01 X(U)… Z(W)… F…

Explanation of the parameters

X, Z Coordinate value of the end point in the absolute command

U, W Coordinate value of the end point in the incremental command

F Feed rate. It is effective until a new value is specified.

Function

The tool is moved along the straight line at the specified feed rate.

Example 1

Use G01 command to rough machining and finish machining the simple cylinder part.

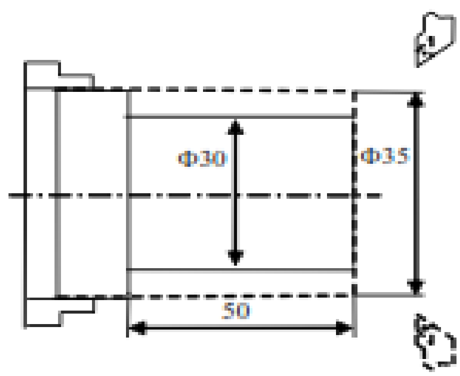


Figure 3.2 Linear Interpolation – Example 1

%3306（Absolute command）

N1 T0106
N2 M03 S460
N3 G00 X90Z20
N4 G00 X31Z3
N5 G01 Z-50 F100
N6 G00 X36
N7 Z3
N8 X30
N9 G01 Z-50 F80
N10 G00 X36
N11 X90 Z20
N12 M05
N13 M30

%3306 （Incremental command）

N1 T0101
N2 M03 S460
N3 G00 X90Z20
N4 G00 X31Z3
N5 G01 W-53 F100
N6 G00 U5
N7 W53
N8 U-6
N9 G01 Z-50 F80
N10 G00 X36
N11 X90 Z20
N12 M05
N13 M30

Example 2

Use G01 command to rough machining and finish machining simple conical part.

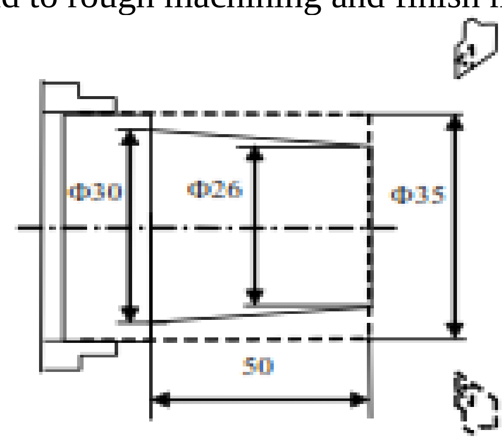


Figure 3.3 Linear Interpolation – Example 2

```
%3307
N1 T0101
N2 M03 S460
N3 G00 X100Z40
N4 G00 X26.6 Z5
N5 G01 X31 Z-50 F100
N6 G00 X36
N7 X100 Z40
N8 T0202
N9 G00 X25.6 Z5
N10 G01 X30 Z-50 F80
N11 G00 X36
N12 X100 Z40
N13 M05
N14 M30
```

Example 3

Use G01 command to rough machining and finish machining the part.

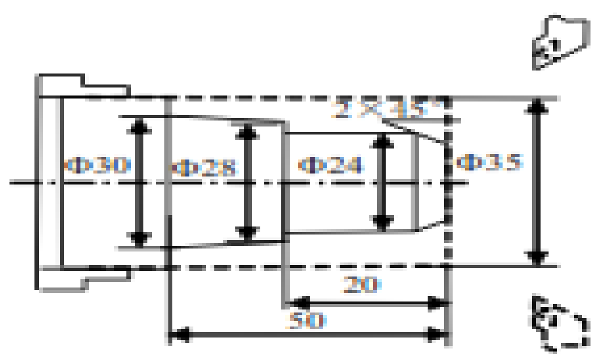


Figure 3.4 Linear Interpolation – Example 3

```
%3308
N1 T0101
N2 M03 S450
N3 G00 X100 Z40
N4 G00 X31 Z3
N5 G01 Z-50 F100
N6 G00 X36
N7 Z3
N8 X25
N9 G01 Z-20 F100
N10 G00 X36
N11 Z3
N12 X15
N13 G01 U14 W-7 F100
N14 G00 X36
N15 X100 Z40
N16 T0202
N17 G00 X100Z40
N18 G00 X14 Z3
N19 G01 X24 Z-2 F80
N20 Z-20
N21 X28
N22 X30 Z-50
N23 G00 X36
N24 X80 Z10
N24 M05
N25 M30
```

Circulation Interpolation (G02, G03)

Programming

$$\begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X(U)\_Z(W)\_ \begin{Bmatrix} I\_K\_ \\ R\_ \end{Bmatrix} F\_$$

Explanation of the parameters

G02 a circular path in clockwise direction (CW)
G03 a circular path in counterclockwise direction (CCW)

X, Z Coordinate values of the circle end point in absolute command

U, W Coordinate values of the circle end point with reference to the circle starting point in incremental command.

I, K Coordinate values of the circle center point with reference to the circle starting point in incremental command.

R Circle radius. R is valid when I, K, R are all specified in this command.

F Feed rate

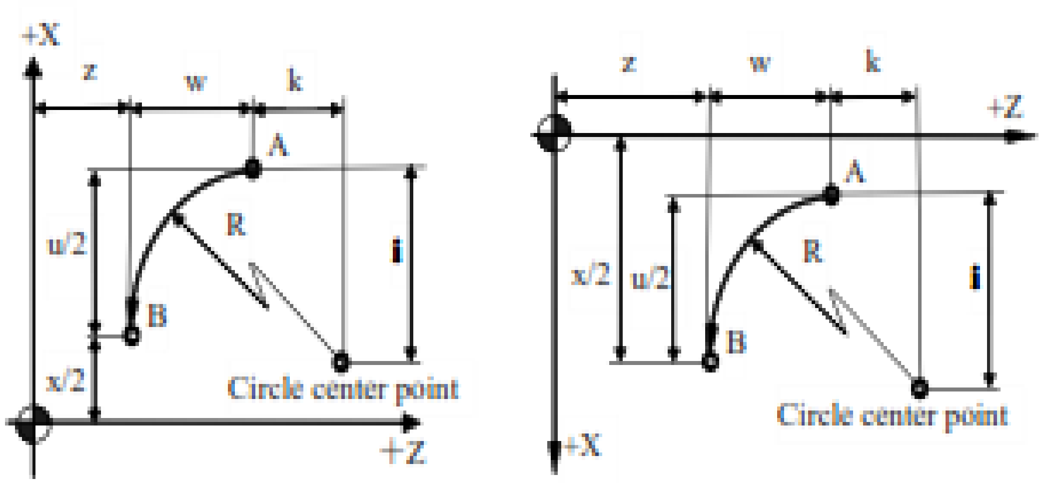


Figure 3.5 Description of G02/G03 parameter

G02 and G03 are defined when the working plane is specified. Figure 3.6 shows the direction of circular interpolation.

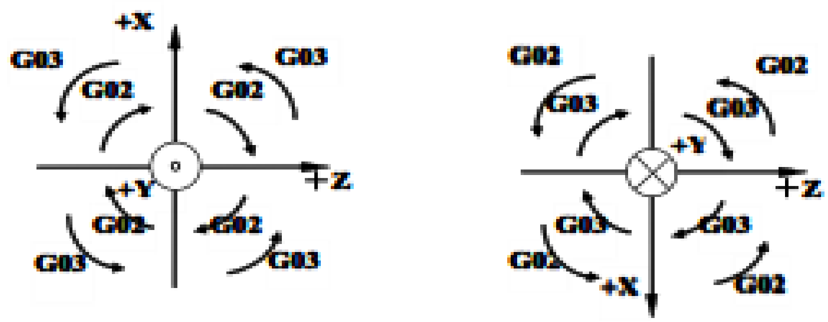


Figure 3.6 Direction of Circular Interpolation

Function

The tool is moved along a full circle or arcs.

Example 1

Use the circular interpolation command to program.

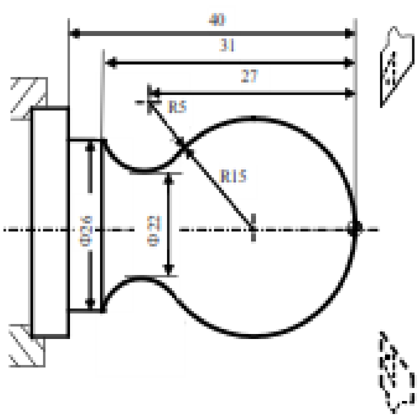


Figure 3.7 Circular Interpolation – Example 1

```
%3309
N1 T0101
N2 G00 X40 Z5
N3 M03 S400
N4 G00 X0
N5 G01 Z0 F60
N6 G03 U24 W-24 R15
N7 G02 X26 Z-31 R5
N8 G01 Z-40
N9 X40 Z5
N10 M30
```

Example 2

Use the circular interpolation command to program

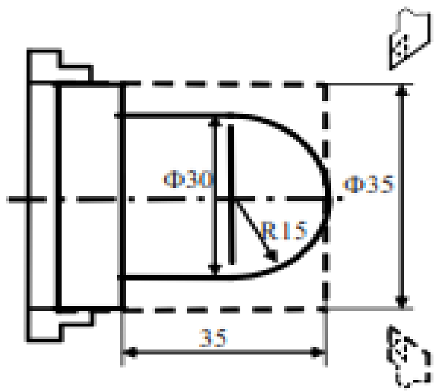


Figure 3.8 Circular Interpolation – Example 2

%3310（Absolute programming）	%3310（Incremental programming）
N1 T0101	N1 T0101
N2 M03 S460	N2 M03 S460
N3 G00 X90Z20	N3 G00 X90Z20
N4 G00 X0 Z3	N4 G00 U-90 W-17
N5 G01 Z0 F100	N5 G01 W-3 F100
N6 G03 X30 Z-15 R15	N6 G03 U30 W-15 R15
N7 G01 Z-35	N7 G01 W-20
N8 X36	N8 X36
N9 G00 X90 Z20	N9 G00 X90 Z20
N10 M05	N10 M05
N11 M30	N11 M30

Example 3

Use the circular interpolation command to program.

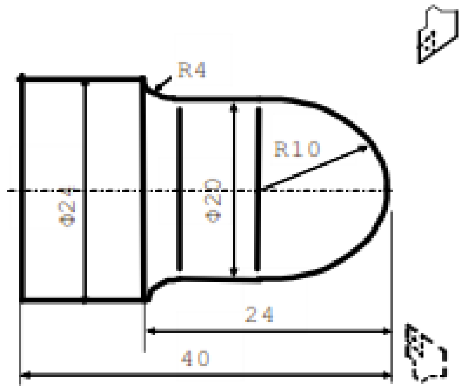


Figure 3.9 Circular Interpolation – Example 3

```
%3311
N1 T0101
N2 M03 S460
N3 G00 X100 Z40
N4 G00 X0 Z3
N5 G01 Z0 F100
N6 G03 X20 Z-10 R10
N7 G01 Z-20
N8 G02 X24 Z-24 R4
N9 G01 Z-40
N10 G00 X30
N11 X100 Z40
N12 M05
N13 M30
```

Example 4

Use the circular interpolation command to program

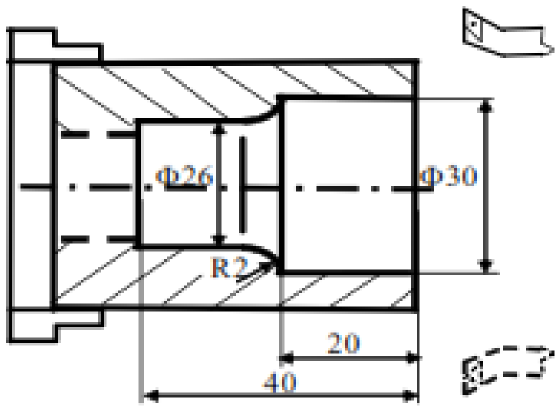


Figure 3.10 Circular Interpolation – Example 4

```
%3312
N1 T0101
N2 M03 S460
N3 G00 X80 Z10
N4 G00 X30 Z3
N5 G01 Z-20 F100
N6 G02 X26 Z-22 R2
N7 G01 Z-40
N8 G00 X24
N9 Z3
N10 X80 Z10
N11 M05
N12 M30
```

Chamfering and Rounding (G01, G02, G03)

Note: These commands can not be used in thread cutting.

Chamfering (G01)

Programming

G01 X(U)_ Z(W)_ C_

Explanation of the parameters

X, Z	Coordinate values of the intersection (point G) in absolute command
U, W	Coordinate values of the intersection (point G) in incremental command
C	Width of chamfer in original direction of movement (c)

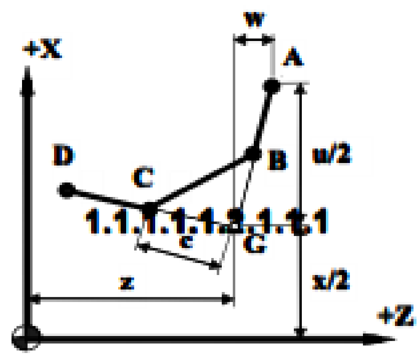


Figure 3.11 Chamfering (G01)

Function

A chamfer can be inserted between two blocks which intersect at a right angle (point A→B →C).

Note: The length of GA should be more than the length of GB

Rounding (G01)

Programming

G01 X(U)_ Z(W)_ R_

Explanation of the parameters

X, Z	Coordinate values of the intersection (point G) in absolute command
U, W	Coordinate values of the intersection (pint G) in incremental command
R	Radius of the rounding (r)

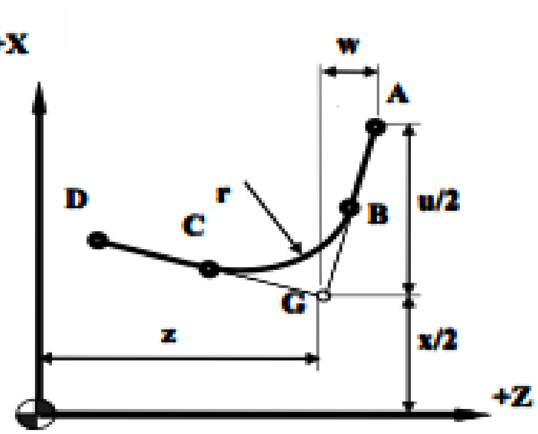


Figure 3.12 Rounding (G01)

Function

A corner can be inserted between two blocks which intersect at a right angle (point A→B→ C).

Note: The length of GA should be more than the length of GB

Example

Use the chamfering and rounding command (G01):

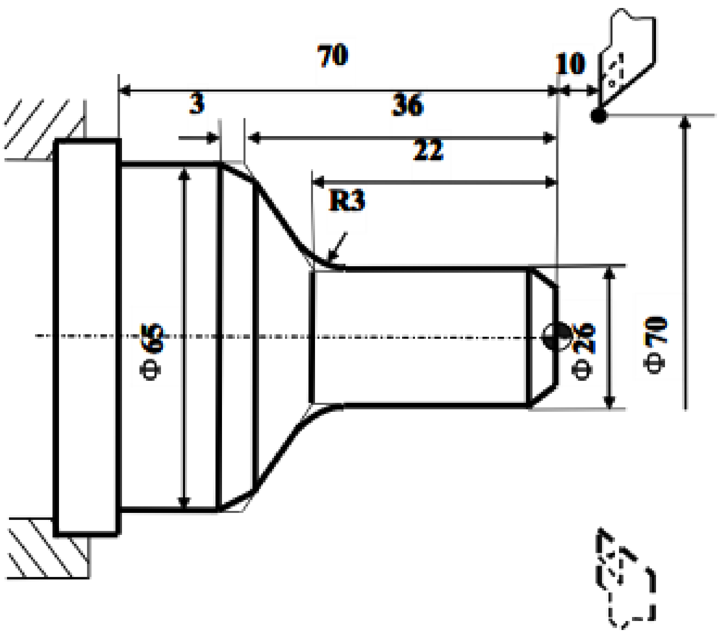


Figure 3.13 Chamfering and Rounding (G01) - Example

```
%3314
N1 M03 S460
N2 G00 U-70 W-10
N3 G01 U26 C3 F100
N4 W-22 R3
N5 U39 W-14 C3
N6 W-34
N7 G00 U5 W80
N8 M30
```

Chamfering (G02, G03)

Programming

$$\begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X(U)\_\ Z(W)\_\ R\ \_\ \ RL = \_$$

Explanation of the parameters

X, Z Coordinate values of the intersection (point G) in absolute command
U, W Coordinate values of the intersection (point G) with reference to the circle starting point (point A) in incremental command
R Circle Radius (r)
RL= Width of chamfer in original direction of movement (RL)

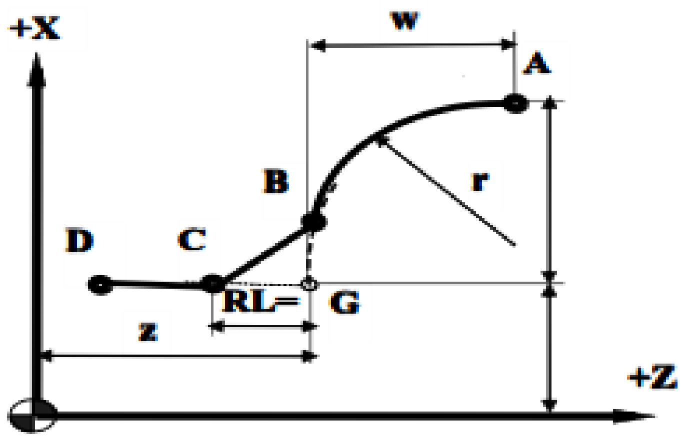


Figure 3.14 Chamfering (G02/G03)

Function

A chamfer can be inserted between two blocks which intersect at a right angle (point A→B →C).

Note: RL must be capitalized letters.

Rounding (G02, G03)

Programming

$$\begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X(U)\_\ Z(W)\_\ R\ \_\ \ RC = \_$$

Explanation of the parameters

X, Z Coordinate values of the intersection (point G) in absolute command
U, W Coordinate values of the intersection (point G) with reference to the circle starting point (point A) in incremental command
R Circle radius (r)
RC Radius of rounding (rc)

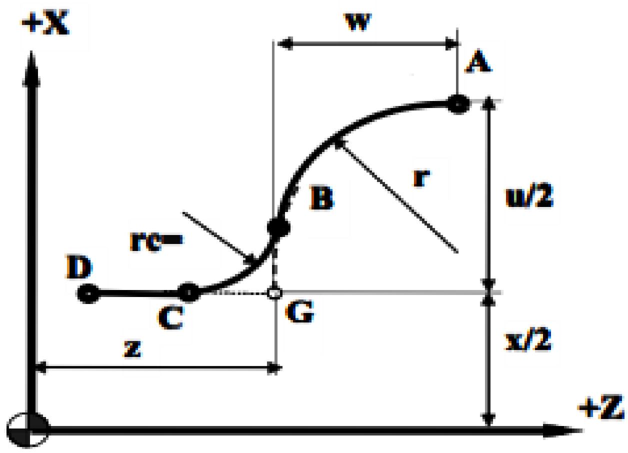


Figure 3.15 Rounding (G02/G03)

Function

A corner can be inserted between two blocks which intersect at a right angle (point A→B→ C).

Note: RC must be capitalized letters.

Example

Use the chamfering and rounding command (G02/G03):

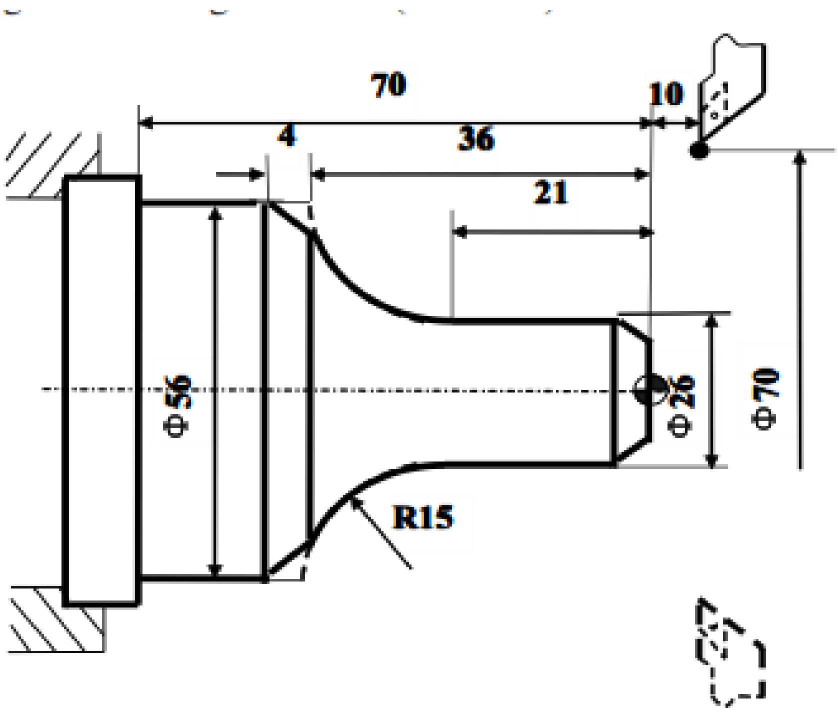


Figure 3.16 Chamfering and Rounding (G02/G03) - Example

```
%3315
N1 T0101
N2 G00 X70 Z10 M03 S460
N3 G00 X0 Z4
N4 G01 W-4 F100
N5 X26 C3
N6 Z-21
N7 G02 U30 W-15 R15 RL=4
N8 G01 Z-70
N9 G00 U10
N10 X70 Z10
N11 M30
```

Thread Cutting with Constant Lead (G32)

Programming

G32 X(U)_Z(W)_R_E__P_F_____

Explanation of the parameters

X, Z Coordinate values of end point in absolute command
U, W Coordinate values of end point with reference to the starting point in incremental command
R, E Coordinate value of retraction amount with reference to the end point in incremental command. In general, R is set as two times value of thread lead, and E is set as the thread height.
P Start point offset. It is used for multiple threads.
F Thread lead per revolution

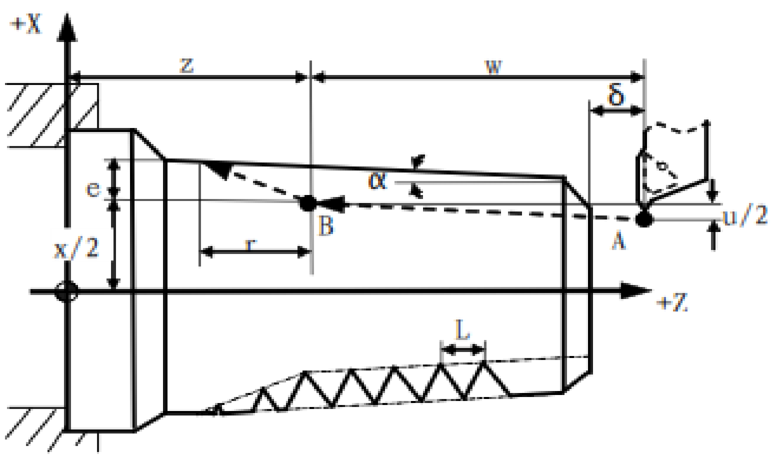


Figure 3.17 Thread Cutting with Constant Lead

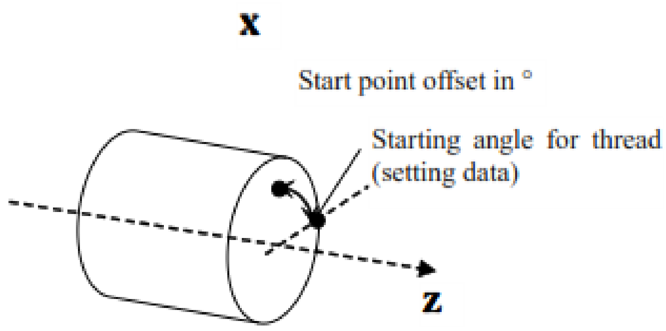


Figure 3.18 Start point Offset

Function

Cylindrical thread, taper thread and face thread can be machined with G32.

Note:

1) The spindle speed should remain constant during rough cutting and finish cutting.
2) The feed hold function is ineffective during the thread cutting. Even though the "feed hold" button is pressed, it is effective until the thread cutting is done.
3) It is not recommended to use the constant surface speed control during the thread cutting.
4) Allowant amount must be specified to avoid the error.

Example

Given that F=1.5mm, δ =1.5mm, δ' =1mm, cutting for four times and each cutting depth is separately: 0.8mm, 0.6 mm, 0.4mm, 0.16mm. It is diameter programming.

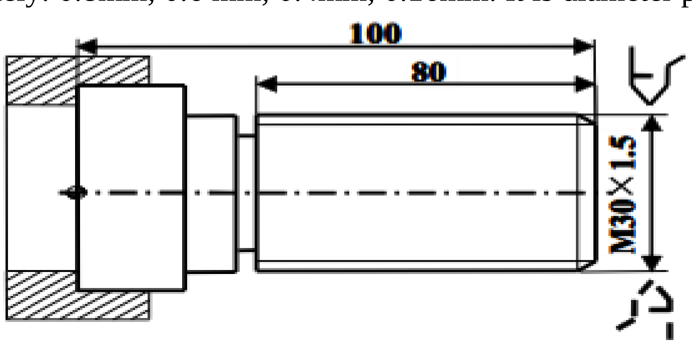


Figure 3.19 Thread Cutting – Example

```
%3316
N1 T0101
N2 G00 X50 Z120
N3 M03 S300
N4 G00 X29.2 Z101.5
N5 G32 Z19 F1.5
N6 G00 X40
N7 Z101.5
N8 X28.6
N9 G32 Z19 F1.5
N10 G00 X40
N11 Z101.5
N12 X28.2
N13 G32 Z19 F1.5
N14 G00 X40
N15 Z101.5
N16 U-11.96
N17 G32 W-82.5 F1.5
N18 G00 X40
N19 X50 Z120
N20 M05
N21 M30
```

Tapping (G34)

Programming

G34 K_ F_ P_

Explanation of the parameters

K The distance from the starting point to the bottom of the hole
F Thread lead
P Dwell time at the bottom of a hole

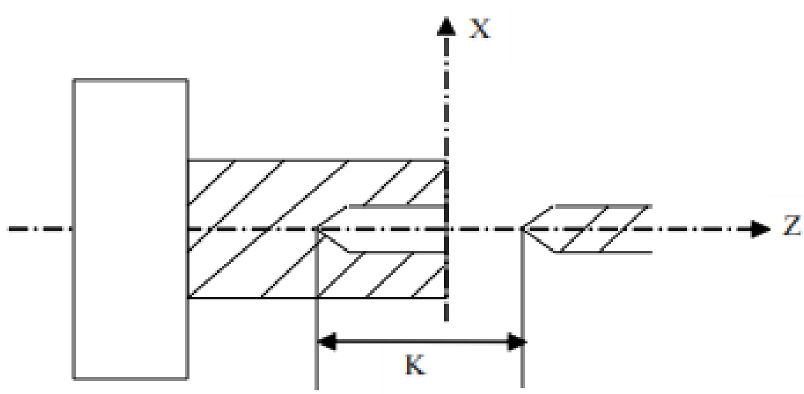


Figure 3.20 Rigid Tapping

Function

With this command, the operator can rigid tap a thread.
In general, there is overshoot of the tap at the bottom of the thread during the spindle-braking portion of the tapping cycle. It can be set by PMC parameters (Table 3-1) to eliminate the overshoot errors.

Feed Function

There are two kinds of feed functions:

1. Rapid Traverse
The tool is moved at the rapid traverse speed set in CNC.
2. Cutting Feed
The tool is moved at the programmed cutting feed rate.

Rapid Traverse (G00)

Positioning command (G00) is to move the tool at the rapid traverse speed (the highest possible speed).
This rapid traverse speed can be controlled by the machine control panel. For more detailed information, please refer to turning operation manual.

Cutting Feed (G94, G95)

Programming

G94 [F_]
G95 [F_]

Explanation of the parameters

G94 feed rate per minute.
On linear axis, the unit of feed rate is mm/min, or in/min. On rational axis, the unit of feed rate is degree/min.
G95 feed rate per revolution
The unit of feed rate is mm/rev, or in/rev.

Note:

1) G94 is the default setting
2) G95 is only used when there is spindle encoder.

Function

The feed rate can be set by G94 or G95.

Dwell (G04)

Programming

G04 P_

Explanation of the parameters

P dwell time (specified in seconds)

Function

It can be used to interrupt machining to get the smooth surface. It can be used to control the groove cutting, drilling, and turning path.

Coordinate System

This chapter would introduce:

1) Reference Position Return (G28)
2) Auto Return from Reference Position (G29)
3) Setting a Workpiece Coordinate System (G92)
4) Selecting a Machine Coordinate System (G53)
5) Selecting a Workpiece Coordinate System (G54~G59)
6) Origin of a Workpiece Coordinate System (G51, G50)
7) Absolute and Incremental Programming (G90, G91)
8) Diameter and Radius Programming (G36, G37)
9) Inch/Metric Conversion (G20, G21)

Reference Position Return (G28)

Programming
G28 X(U)_ Z(W)_

Explanation of the parameters

X, Z Coordinate values of the intermediate point in absolute command
U,W Coordinate values of the intermediate point with reference to the starting point in incremental command

Function
The tool is moved to the intermediate point rapidly, and then returned to the reference point.

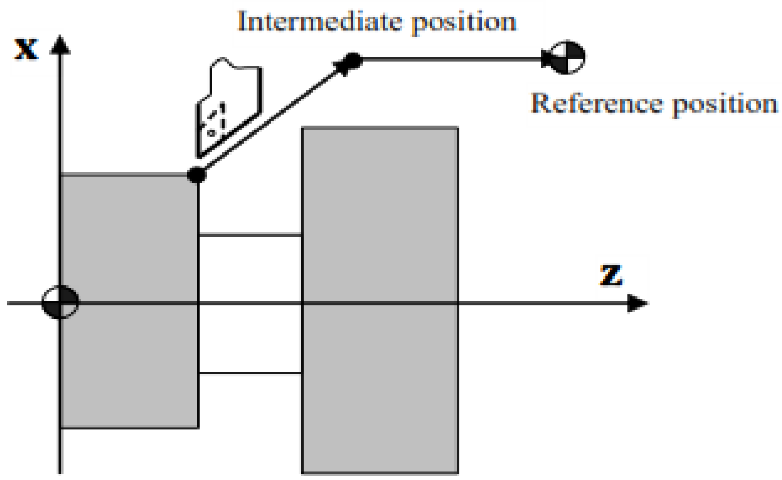


Figure 5.1 Reference Position Return

Note:

1) In general, G28 is used to change tools or cancel the mechanical error. Tool radius compensation and tool length compensation should be cancelled when G28 is executed.
2) G28 can not only make the tool move to the reference point, but also can save the intermediate position to be used in G29.
3) When the power is on and manual reference position return is not available, G28 is same as the manual reference position return. The direction of this reference position return (G28) is set by the axis parameter – reference approach direction.
4) G28 is one-shot G code.

Auto Return from Reference Position (G29)

Programming

G29 X(U)_ Z(W)_

Explanation of the parameters

X, Z Coordinate value of the end point in absolute command

U, W Coordinate value of the end point in incremental command

Function

The tool is moved rapidly from the intermediate point defined in G28 to the end point. Thus, G29 is generally used after G28 is defined.

Note:

G29 is one-shot G code.

Example
Use G28, G29 command to program the track shown in. It moves from the starting point A to the intermediate point B, and then returns to the reference point R. At last, it moves from the reference point R to the end point C through the intermediate point B.

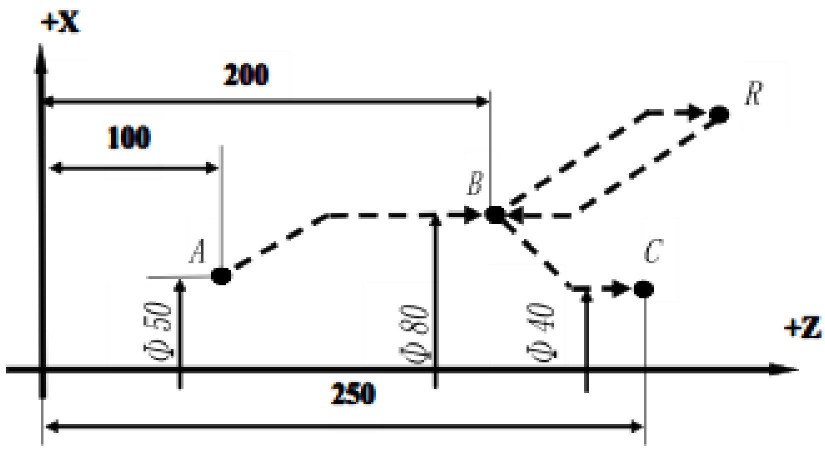


Figure 5.2 Reference Position – Example

%3317

N1 T0101

N2 G00 X50 Z100

N3 G28 X80 Z200

N4 G29 X40 Z250

N5 G00 X50Z100

N6 M30

Setting a Workpiece Coordinate System (G92)

Programming
G92 X_ Z_

Explanation of the parameters
X, Z Coordinate values of the tool position in the workpiece coordinate system.

Functions
G92 can set a workpiece coordinate system based on the current tool position (X_ Z_).

Example

Use G92 to set a workpiece coordinate system.

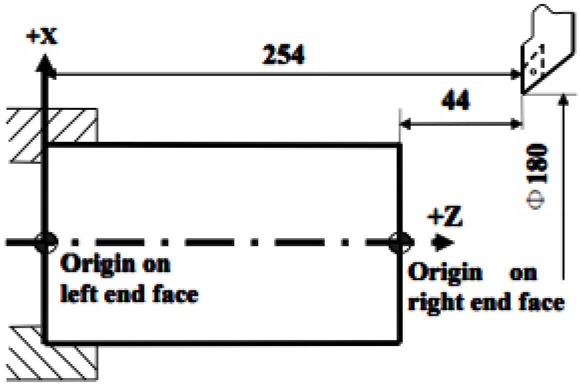


Figure 5.3 Setting a Coordinate System – Example

If the origin is set on the left end face,
G92 X180 Z254
If the origin is set on the right end face
G92 X180 Z44

Selecting a Machine Coordinate System (G53)

Programming

G53 X_Z_

Explanation of the parameters

X, Z   Absolute coordinate values of a point in the machine coordinate system.

Function

A machine coordinate system is selected, and the tool moves to the position at the rapid traverse speed.

Note:

1) Absolute values must be specified in G53. The incremental values would be ignored by G53.
2) G53 is one-shot G code.

Selecting a Workpiece Coordinate System (G54~G59)

Programming

$$\begin{Bmatrix} G54 \\ G55 \\ G56 \\ G57 \\ G58 \\ G59 \end{Bmatrix} X\_\ Z\_$$

Explanation of the parameters

X, Z Coordinate values of the point in absolute command

Function

There are six workpiece coordinate system to be selected. If one coordinate system is selected, the tool is moved to a specified point.

Note:

1) The workpiece coordinate system must be set before these commands (G54~G59) are used. The workpiece coordinate system can be set by using the MDI panel. For detailed information, please refer to the turning operation manual.
2) Reference position must be returned before these commands (G54~G59) are executed.
3) G54 is the default setting.

Example

Select one of workpiece coordinate system, and the tool path is Current point→A→B.

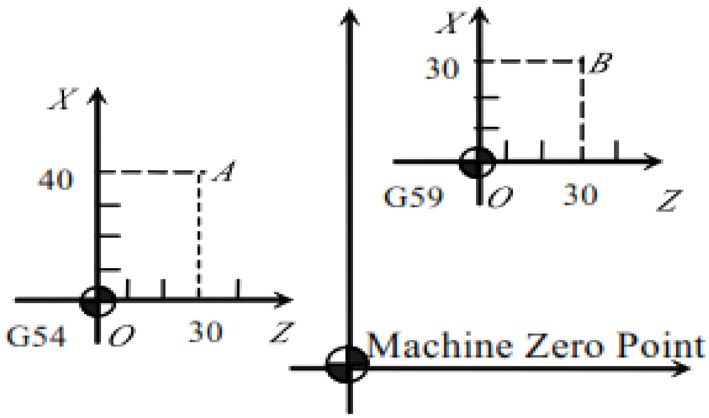


Figure 5.4 Workpiece Coordinate System – Example

```
%3303
N01 G54 G00 G90 X40 Z30
N02 G59
N03 G00 X30 Z30
N04 M30
```

Origin of a Workpiece Coordinate System (G51, G50)

Programming

G51U_ W_ G50

Explanation of the parameters

G51 can move the origin of workpiece coordinate system.

U, W Coordinate values of the position in incremental command

G50 can cancel the movement.

Function

The origin of workpiece coordinate system can be moved.

Note:

1) G51 is only effective when T command or G54~G59 is defined in the program.
2) G50 is only effective when T command or G54~G59 is defined in the program.

Example

```
%1234
G51 U30 W10
M98 P1111 L4
G50
T0101
G01 X30 Z14
M30

%1111
T0101
G01 X32 Z25
G01 X34.444 Z99.123
M99
```

Absolute and Incremental Programming (G90, G91)

Programming G90
X_ Z_ G91 U_W_
Explanation of the parameters
G90 Absolute programming
X, Z Coordinate values on X axis and Z axis in the coordinate system
G91 Incremental programming
U, W Coordinate values with reference to the previous position in the coordinate system
Function
The tool is moved to the specified position.

Example
Move the tool from point 1 to point 2 through point 3, and then return to the current point.

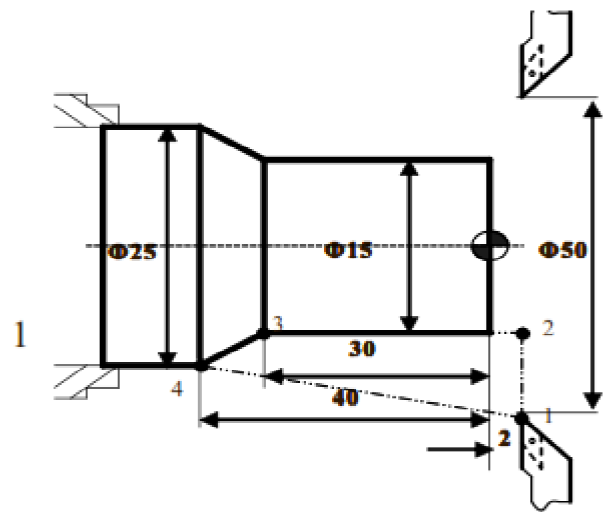


Figure 5.5 Absolute and Incremental Programming – Example

Absolute Programming	Incremental Programming	Absolute and Incremental
%0001 N 1 T0101 N 2 M03 S460 N3 G90 G00 X50 Z2 N4 G01 X15 N 5 Z-30 N 6 X25 Z-40 N 7 X50 Z2 N 8 M30	%0001 N 1 M03 S460 N 2 G91 G01 X-35 N 3 Z-32 N 4 X10 Z-10 N 5 X25 Z42 N 6 M30	%0001 N 1 T0101 N 2 M03 S460 N 3 G00 X50 Z2 N 4 G01 X15 N 5 Z-30 N 6 U10 Z-40 N 7 X50 W42 N 8 M30

Diameter and Radius Programming (G36, G37)

Programming
G36
G37

Explanation of the parameters
G36 Diameter programming
G37 Radius programming

Function
The coordinate value on X axis is specified in two ways: diameter or radius. It allows to program the dimension straight from the drawing without conversion.
Note:

1) In all the examples of this book, we always use diameter programming if the radius programming is not specified.
2) If the machine parameter is set to diameter programming, then diameter programming is the default setting. However, G36 and G37 can be used to exchange. The system shows the diameter value.
3) If the system parameter is set to radius programming, then radius programming is the default setting. However, G36 and G37 can be used to exchange. The system shows the radius value.

Example

Use Diameter programming and Radius programming for the same path

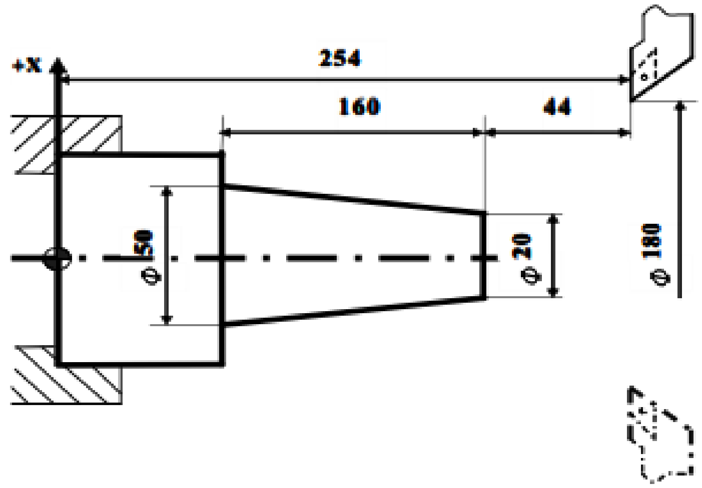


Figure 5.6 Diameter and Radius Programming – Example

Diameter Programming	Radius Programming	Compound Programming
%3304	%3314	%3314
N1 G92 X180 Z254	N1 G37 M03 S460	N1 T0101
N2 M03 S460	N2 G54 G00 X90 Z254	N2 M03 S460
N3 G01 X20 W-44	N3 G01 X10 W-44	N3 G37G00 X90 Z254
N4 U30 Z50	N4 U15 Z50	N4 G01 X10 W-44
N5 G00 X180 Z254	N5 G00 X90 Z254	N5 G36 U30 Z50
N6 M30	N6 M30	N6 G00 X180 Z254
		N7 M30

Inch/Metric Conversion (G20, G21)

Programming
G20
G21

Explanation of the parameters
G20: Inch input
G21: Metric input
The units of linear axis and circular axis are shown in the following table

Table 5 1. Unit of Linear axis and Circular axis

	Linear axis	Circular axis
Inch system (G20)	Inch	Degree
Metric system (G21)	Mm	Degree

Function

Depending on the part drawing, the workpiece geometries can be programmed in metric measures or inches.

Spindle Speed Function

Spindle function controls the spindle speed (S), the unit of spindle speed is r/min. Spindle speed is the cutting speed when it is at the constant speed, the unit of speed is m/min.
S is modal G code command; it is only available when the spindle is adjustable. Spindle speed programmed by S code can be adjusted by overrides on the machine control panel.

This chapter would introduce

1) Limit of spindle speed (G46)
2) Constant surface cutting control (G96, G97).

1 Limit of Spindle Speed (G46)

Programming

G46 X_ P_

Explanation of the parameters

X The minimum speed of the spindle when using constant surface speed（r/min

P The maximum speed of the spindle when using constant surface speed（r/min

Function
G46 command can set the minimum of spindle speed, and the maximum of spindle speed.

Note:
It can only used with G96 (constant surface speed control command).

2 Constant Surface Speed Control (G96, G97)

Programming
G96 S G97 S

Explanation of the parameters

G96	activate the constant surface speed
S	surface speed (m/min)
G97	deactivate the constant surface speed
S	spindle speed (r/min)

Function
G96 and G97 commands are to control the constant surface speed.

Note:
1) The spindle speed must be controlled automatically when the constant surface cutting command is executed.
2) The maximum of spindle speed can be set by the axis parameter.

Example

Use the constant surface control command

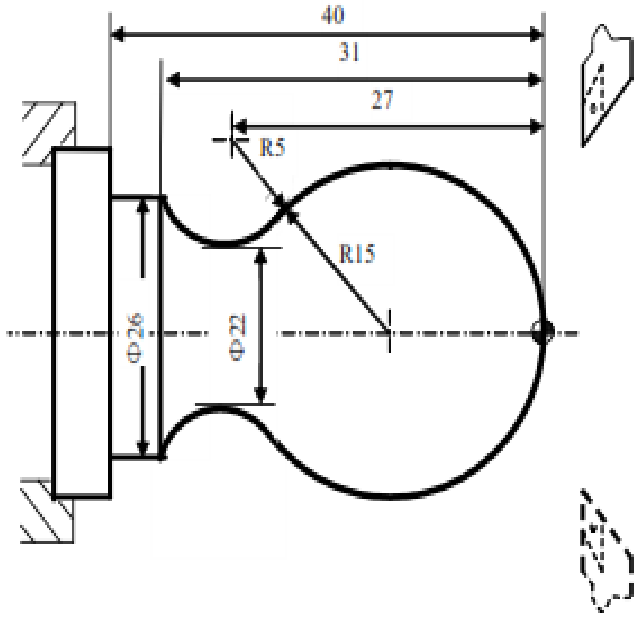


Figure 6.1 Constant Surface Control – Example

```
%3318
N1 T0101
N2 G00 X40 Z5
N3 M03 S460
N4 G96 S80
N5 G46 X400 P900
N5 G00 X0
N6 G01 Z0 F60
N7 G03 U24 W-24 R15
N8 G02 X26 Z-31 R5
N9 G01 Z-40
N10 X40 Z5
N11 G97 S300
N12 M30
```

Tool Function

This chapter would introduce:

1) Too selection and Tool offset (T code)
2) Tool radius compensation (G40, G41, G42)

1 Tool Selection and Tool Offset (T code)

Programming

T XX XX

Explanation of the parameters

XX Tool number (two digits). The number of tool depends on manufacture's configuration.

XX Tool offset number (two digits). It corresponds to the specific compensation value.

Functions

To select the desired tool, T command makes the turret turn, selects a cutter, and calls the compensation value.

Note:

1) T command is only effective when it is used with tool move command, such as G00
2) When T command and tool move command are in the same program block, T command is executed at first.
3) The same tool can have different compensation values. For example, T0101, T0102, T0103 are possible.
4) Different tool can have same compensation values. For example, T0101, T0201, and T0301 are possible.

Example

```
%0012
N01 T0101
N02 M03 S460
N03 G00 X45 Z0
N04 G01 X10 F100
N05 G00 X80 Z30
N06 T0202
N07 G00 X40 Z5
N08 G01 Z-20 F100
N09 G00 X80 Z30
N10 M30
```

2 Tool Radius Compensation (G40, G41, G42)

Programming

$$\begin{Bmatrix} G40 \\ G41 \\ G42 \end{Bmatrix} \begin{Bmatrix} G00 \\ G01 \end{Bmatrix} X\_\ Z\_$$

Explanation of the parameters

G40 Deactivate tool radius compensation

G41 Activate tool radius compensation, tool operates in machining operation to the left of the contour.

G42 Activate tool radius compensation, tool operates in machining operation to the right of the contour.

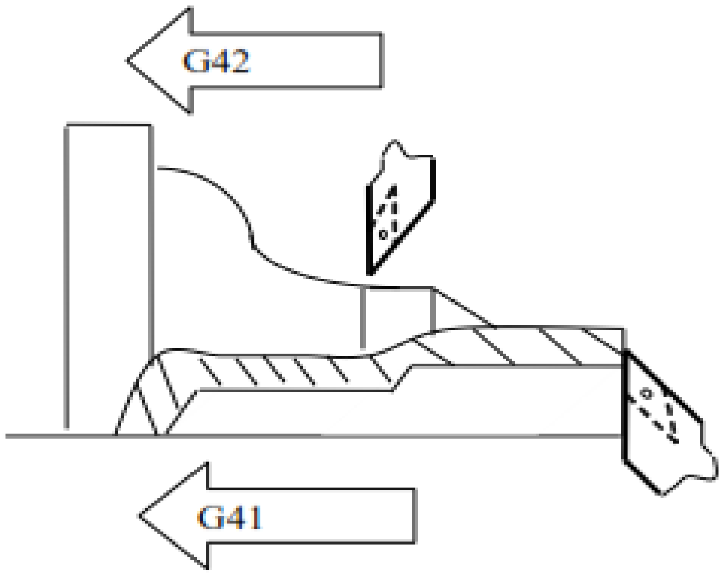


Figure 7.1 Tool Radius Compensation

X, Z Coordinate values of the end point. It is the point where the tool radius compensation is activated or deactivated.

Function

These commands can control the tool radius compensation to get the equidistant tool paths for different tools.

Note:

1) G40, G41, and G42 must be used with G00 or G01.
2) The tool radius compensation value is assigned in T code.

Example

Use the tool radius compensation, and program for the part shown in Figure 7.2

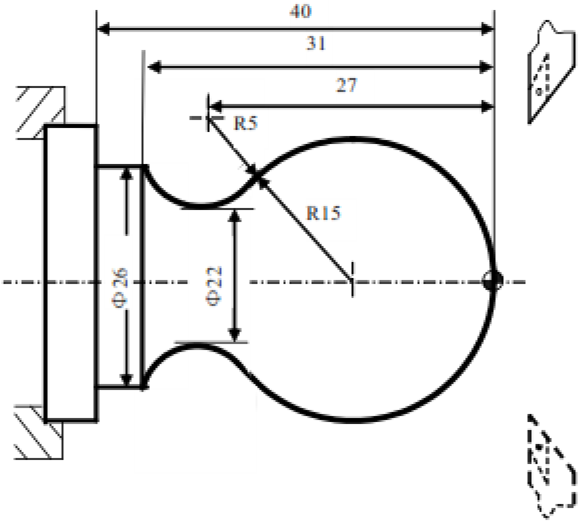


Figure 7.2 Tool Radius Compensation

```
%3323
N1 T0101
N2 M03 S400
N3 G00 X40 Z5
N4 G00 X0
N5 G01 G42 Z0 F60
N6 G03 U24 W-24 R15
N7 G02 X26 Z-31 R5
N8 G01 Z-40
N9 G00 X30
N10 G40 X40 Z5
N11 M30
```

Miscellaneous Function

As it is mentioned in Chapter 1.8, there are two ways of execution when a move command and M code are specified in the same block.

1) Pre-M function
   M command is executed before the completion of move command.
2) Post-M function
   M command is executed after the completion of move command

There are two types of M code: one-shot M code, and modal M code.

Table 8 1 Type of M code

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

M code List

The following is a list of M command.

Table 8 2 M code List

CNC M-function	Type of Mode	Function	Pre/Post-M function
M00	One-shot	Program stop	Post-M function
M01	One-shot	Optional stop	Post-M function
M02	One-shot	End of program	Post-M function
M30	One-shot	End of program with return to the beginning of program	Post-M function
M98	One-shot	Calling of subprogram	Post-M function
M99	One-shot	End of subprogram	Post-M function
PLC M-function	**Type of Mode**	**Function**	**Pre/Post-M function**
M03	Modal	Spindle forward rotation	Pre-M function
M04	Modal	Spindle reverse rotation	Pre-M function
M05	Modal	◣Spindle stop	Post-M function
M07	Modal	Number1 Coolant on	Pre-M function
M08	Modal	Number2 Coolant on	Pre-M function
M09	Modal	◣Coolant off	Post-M function

◣: default setting

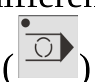
CNC M-Function

Program Stop (M00)

M00 is one-shot M function, and it is post-M function

The program can be stopped, so that the operator could measure the tool and the part, adjust part and change speed manually, and so on.
When the program is stopped, the spindle is stopped and the coolant is off. All of the current modal information remains unchanged. Resuming program could be executed by pushing “Cycle Run” button on the machine control panel.

Optional Stop (M01)

M01 is one-shot M function, and it is post-M function
Similarly to M00, M01 can also stop the program. All of the modal information is maintained. The difference between M00 and M01 is that the operator must press M01 button () on the machine control panel. Otherwise, the program would not be stopped even if there is M01 code in the program.

End of Program (M02)

M02 is one-shot M function, and it is post-M function

When M02 is executed, spindle, feed and coolant are all stopped. It is usually at the end of the last program block. To restart the program, press “Cycle Run” button on the operational panel.

End of Program with return to the beginning of program (M30)

M30 is one-shot M function, and it is post-M function

Similarly to M02, M30 can also stop the program. The difference is that M30 returns control to the beginning of program. To restart the program, press “Cycle Run” button on the operational panel.

Subprogram Control (M98, M99)

- End of Subprogram (M99)

M99 indicates the end of subprogram and returns control to the main program. It is one-shot

M function, and it is post-M function.

- Calling a Subprogram(M98)

  M98 P_ L_

P program number of the subprogram
L repeated times of subprogram

M98 is used to call a subprogram. It is one-shot M function. Moreover, it is post-M function.

Example

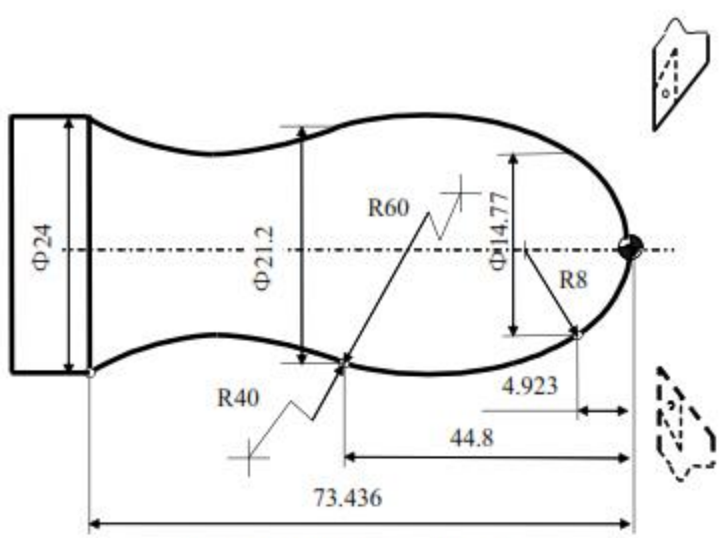


Figure 8.1 Subprogram Control - Example

```
%3111
N1 G92 X32 Z1
N2 G00 Z0 M03 S46
N3 M98 P0003 L5
N4 G36 G00 X32 Z1
N5 M05
N6 M30
%0003
N1 G37 G01 U-12 F100
N2 G03 U7.385 W-4.923 R8
N3 U3.215 W-39.877 R60
N4 G02 U1.4 W-28.636 R40
N5 G00 U4
N6 W73.436
N7 G01 U-5 F100
N8 M99
```

PLC M Function

Spindle Control (M03, M04, M05)

M03 starts spindle to rotate CW at the set speed set in the program.
M04 starts spindle to rotate CCW at the set speed in the program.
M05 stops spindle.
M03, M04 are modal M code, and they are pre-M function. M05 is modal M code, and it is post-M function. M05 is the default setting.

Coolant Control (M07, M08, M09)

M07, M08 can turn on the coolant.
M09 can turn off the coolant.
M07 and M08 are modal M code, and they are pre-M function. M09 is one-shot M code, and it is post-M function. Moreover, M09 is the default setting.

Functions to Simplify Programming

This chapter would introduce:

1) Canned Cycle

   Internal diameter/ Outer diameter cutting cycle

   (G80) End face turning cycle (G81)

   Thread cutting cycle (G82)

   End face peck drilling cycle (G74)

   Outer diameter grooving cycle (G75)

2) Multiple Repetitive Cycle

   Stock Removal in Turning (G71)

Stock Removal in Facing (G72)

Pattern Repeating (G73)

Multiple Thread Cutting Cycle (G76)

1 Canned Cycles

To simplify programming, the canned cycle command can execute the specific operation using one G code, instead of several separated G commands in the program.

Internal Diameter/Outer Diameter Cutting Cycle (G80)

- **Straight Cutting Cycle**

Programming
G80 X(U)_ Z(W)_ F_

Explanation of the parameters
X, Z Coordinate values of end point (point C) in absolute command
U, W Coordinate values of end point (point C) with reference to the initial point (point A) in incremental command
F Feed rate

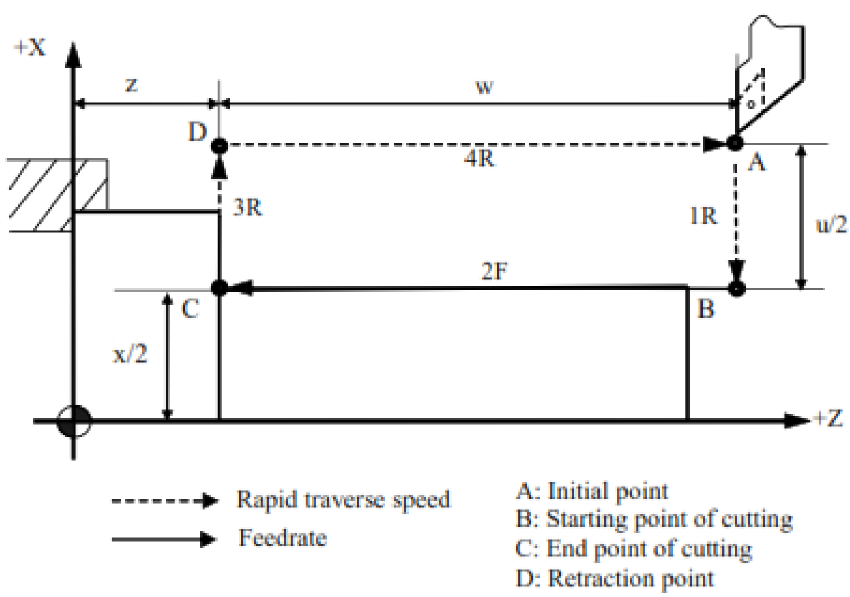


Figure 9.1 Straight cutting cycle (G80)

Function

This command can implement the straight cutting. The machining path is A→B→C→D→A.

- **Taper Cutting Cycle**

Programming

G80 X(U)_ Z(W)_ I_ F_

Explanation of the parameters

X, Z Coordinate values of end point (point C) in absolute command

U, W Coordinate values of end point (point C) with reference to the initial point (point A) in incremental command

I The radius difference between starting point B and end point C. It is negative, if the radius of point B is less than the radius of point C. Otherwise, it is positive.

F Feed rate

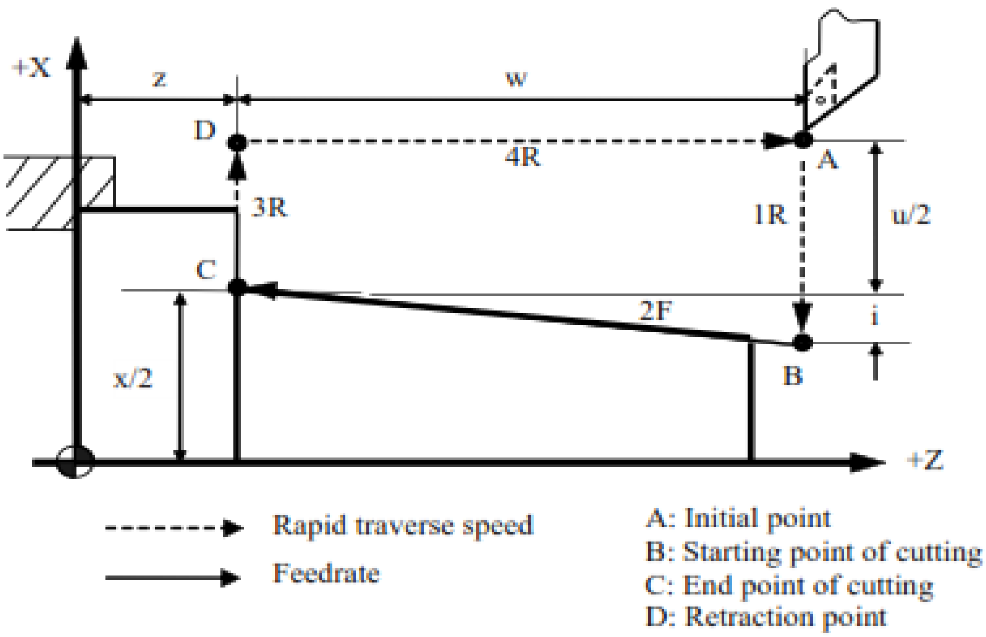


Figure 9.2 Taper Cutting Cycle (G80)

Function

This command can implement the taper cutting. The machining path is A→B→C→D→A.

Example

Use G80 command to machine the cylindrical part in two steps – rough machining and finish machining.

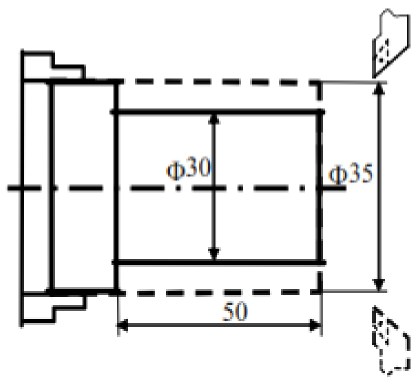


Figure 9.3 Internal Diameter/Outer Diameter Cutting Cycle – Example 1

```
%3320
N1 T0101
N2 M03 S460
N3 G00 X90Z20
N4 X40 Z3
N5 G80 X31 Z-50 F100
N6 G80 X30 Z-50 F80
N7 G00X90 Z20
N8 M30
```

Example

Use G80 command to machine the tapered part in two steps – rough machining and finish machining.

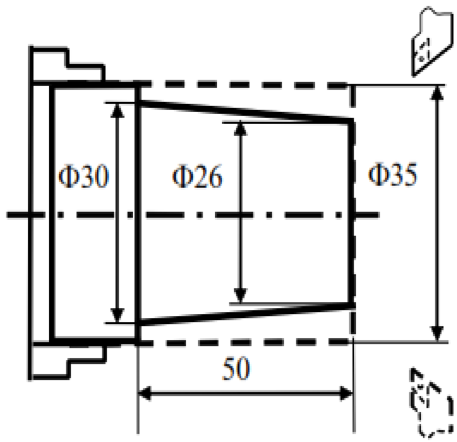


Figure 9.4 Internal Diameter/Outer Diameter Cutting Cycle – Example 2

```
%3321
N1 T0101
N2 G00 X100Z40 M03 S460
N3 G00 X40 Z5
N4 G80 X31 Z-50 I-2.2 F100
N5 G00 X100 Z40
N6 T0202
N7 G00 X40 Z5
N8 G80 X30 Z-50 I-2.2 F80
N9 G00 X100 Z40
N10 M05
N11 M30
```

Example

Use G80 command to machine the tapered part in two steps – rough machining and finish machining.

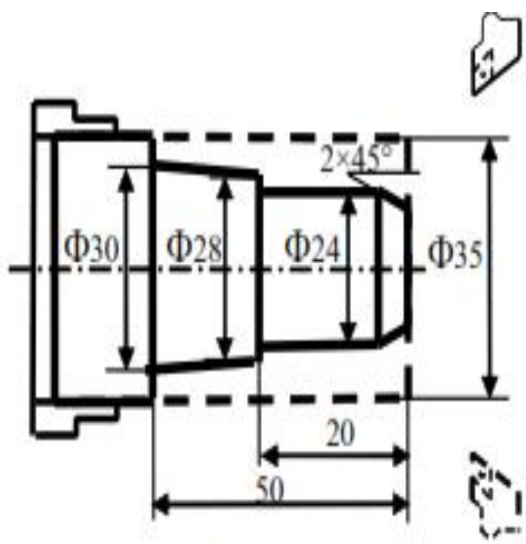


Figure 9.5 Internal Diameter/Outer Diameter Cutting Cycle – Example 3

```
%3322
N1 T0101
N2 M03 S460
N3 G00 X100 Z40
N4 X40 Z3
N5 G80 X31 Z-50 F100
N6 G80 X25 Z-20
N7 G80 X29 Z-4 I-7 F100
N8 G00 X100 Z40
N9 T0202
N10 G00 X100 Z40
N11 G00 X14 Z3
N12 G01 X24 Z-2 F80
N13 Z-20
N14 X28
N15 X30 Z-50
N16 G00 X36
N17 X80 Z10
N18 M05
N19 M30
```

End Face Turning Cycle (G81)

- **Face Cutting Cycle**

Programming

G81 X(U)_ Z(W)_ F_

Explanation of the parameters

X, Z Coordinate values of end point (point C) in absolute command
U, W Coordinate values of end point (point C) with reference to the initial point (point A) in incremental command
F Feed rate

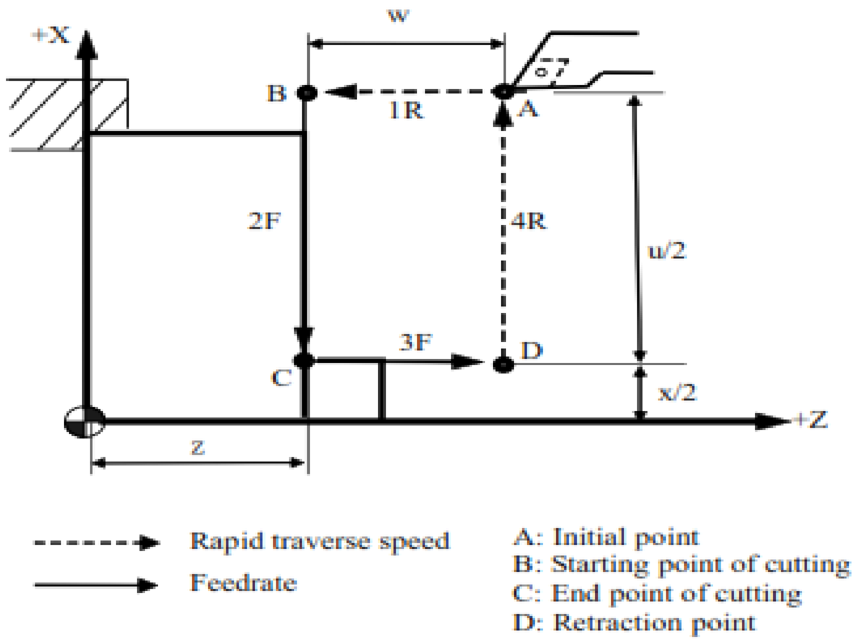


Figure 9.6 Face Cutting Cycle (G81)

Function

This command can implement the end face cutting. The machining path is A→B→C→D→A.

- **Taper Face Cutting Cycle**

Programming

G81 X(U)_ Z(W)_ K_ F_

Explanation of the parameters

X, Z Coordinate values of end point (point C) in absolute command

U, W Coordinate values of end point (point C) with reference to the initial point (point A) in incremental command

K The distance on Z axis of the starting point (point B) with reference to the end point (point C). It is negative, if the value of point C on Z axis is more than point B’s. It is positive, if the value of point C on Z axis is less than point B’s.

F Feed rate

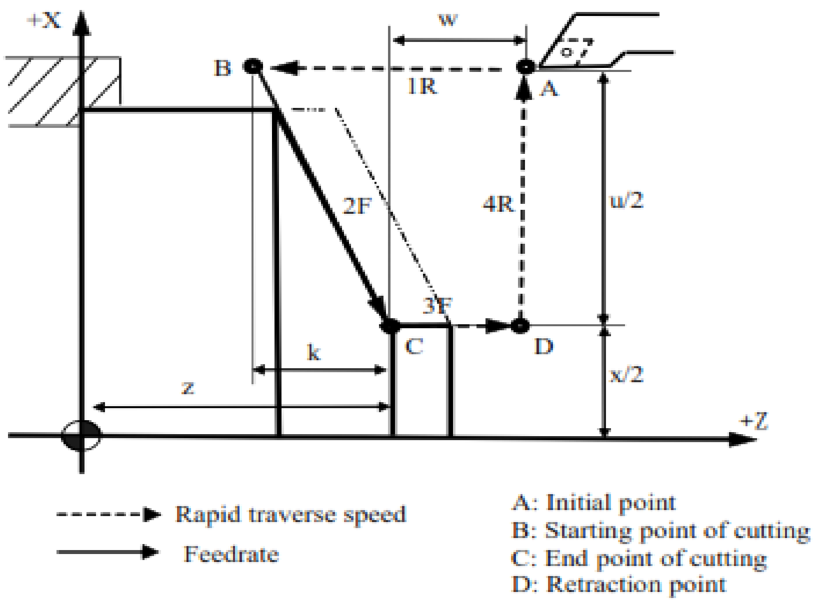


Figure 9.7 Taper Face Cutting Cycle (G81)

Function

This command can implement the taper face cutting. The machining path is A→B→C→D→A.

Example

Use G81 to program. The dashed line stands for the roughcast.

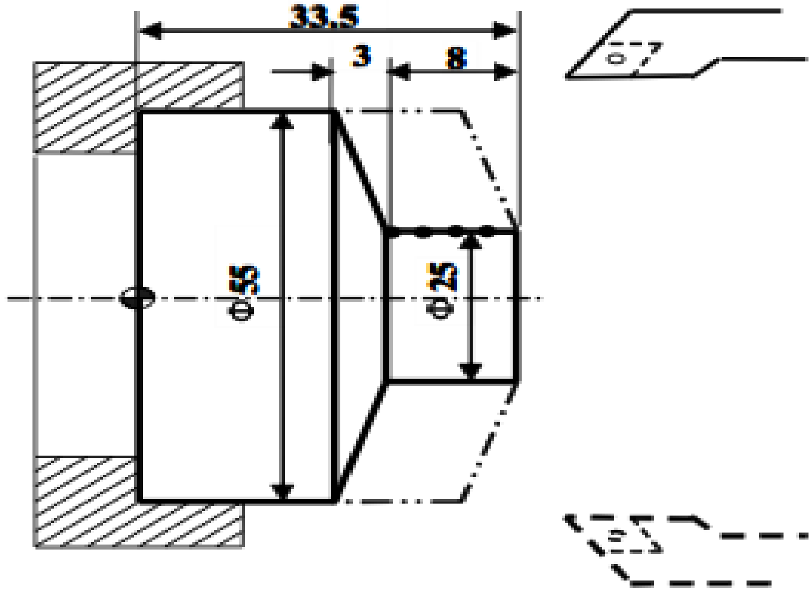


Figure 9.8 End Face Turning Cycle (G81)

```
%3323
N1 T0101
N2 G00 X60 Z45
N3 M03 S460
N4 G81 X25 Z31.5 K-3.5 F100
N5 X25 Z29.5 K-3.5
N6 X25 Z27.5 K-3.5
N7 X25 Z25.5 K-3.5
N8 M05
N9 M30
```

Thread Cutting Cycle (G82)

- **Cylindrical Thread Cutting Cycle**

Programming

G82 X(U)_ Z(W)_ R_ E_ C_ P_ F(J)_

Explanation of the parameters

X, Z Coordinate values of end point (point C) in absolute command

U, W Coordinate values of end point (point C) with reference to the initial point (point A) in incremental command

R, E Coordinate value of retraction amount with reference to the end point (point C) in incremental command.

C The number of thread head. It is single thread when C is 0 or 1

P Start point offset. It is used for multiple threads.

F Thread lead per revolution

J Thread lead in inch measurement

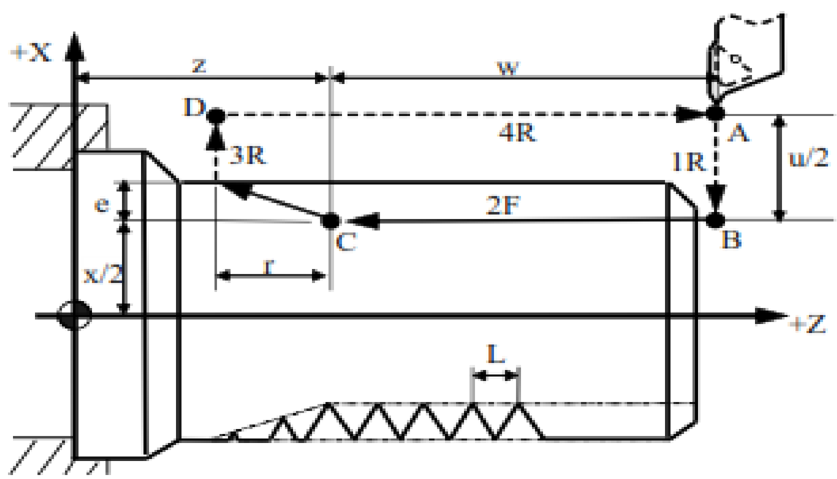


Figure 9.9 Cylindrical Thread Cutting Cycle (G82)

Function

This command can implement the cylindrical thread cutting. The machining path is A→B→C→D→A. Moreover, this command is same as G32 (Thread cutting with constant lead).

- **Taper Thread Cutting Cycle**

Programming

G82 X(U)_ Z(W)_ I_ R_ E_ C_ P_ F(J)_

Explanation of the parameters

X, Z Coordinate values of end point (point C) in absolute command
U, W Coordinate values of end point (point C) with reference to the initial point (point A) in incremental command

I The radius difference between starting point B and end point C. It is negative, if the radius of point B is less than the radius of point C. Otherwise, it is positive.
R, E Coordinate value of retraction amount with reference to the end point (point C) in incremental command.
C The number of thread head. It is single thread when C is 0 or 1.
P Start point offset. It is used for multiple threads.
F Thread lead per revolution
J Thread lead in inch measurement

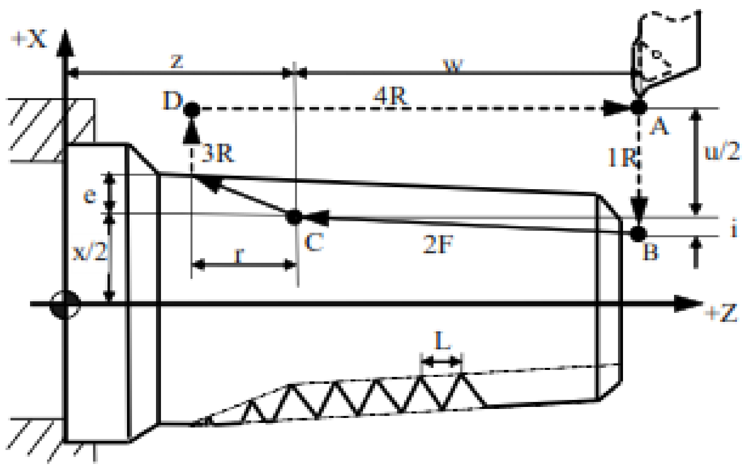


Figure 9.10 Taper Thread Cutting Cycle (G82)

Function

This command can implement the taper thread cutting. The machining path is A→B→C→D→A.

Example

Use G82 command to program. The screw's pitch is 1.5, and the number of thread head is 2.

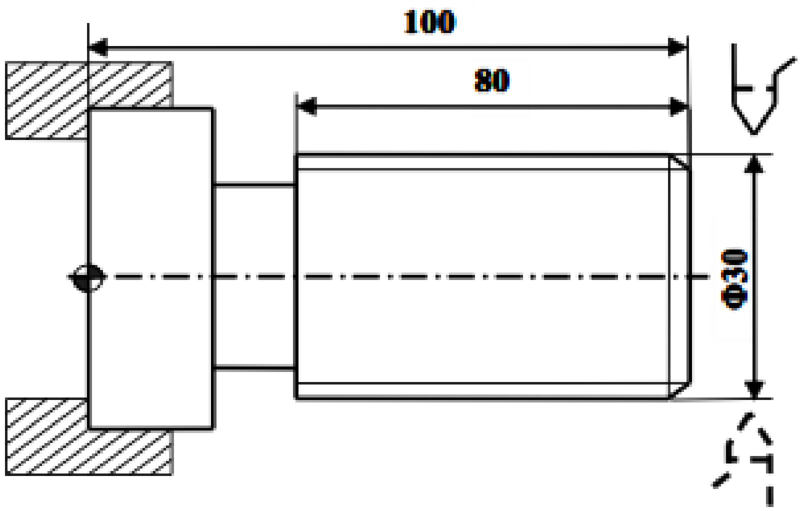


Figure 9.11 Thread Cutting Cycle - Example

```
%3324
N1 G54 G00 X35 Z104
N2 M03 S300
N3 G82 X29.2 Z18.5 C2 P180 F3
N4 X28.6 Z18.5 C2 P180 F3
N5 X28.2 Z18.5 C2 P180 F3
N6 X28.04 Z18.5 C2 P180 F3
N7 M30
```

End Face Peck Drilling Cycle (G74)

Programming

G74 Z(W)_ R(e) Q(△K) F_

Explanation of the parameters

Z Coordinate value on Z axis of the end point in absolute command
W Coordinate value on Z axis of the end point with reference to the starting point in incremental command
R Retraction amount(e) for each feed. It must be absolute value.
Q Depth of drilling(△K) for each feed. It must be absolute value.
F Feed rate

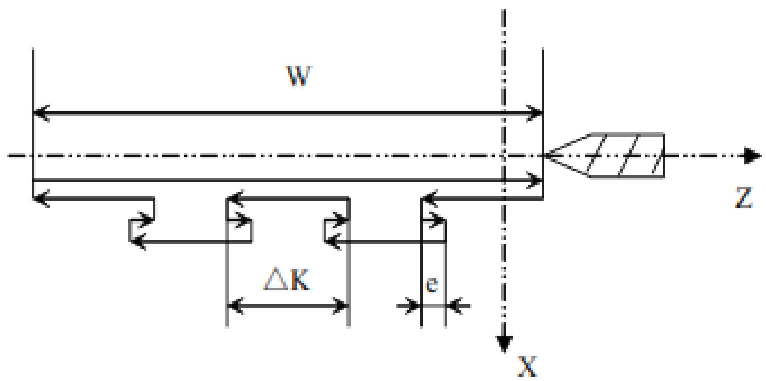


Figure 9.12 End Face Peck Drilling Cycle (G74)

Function

This command can drill a hole on end face.

Example

Use G74 to drill a hole on a workpiece.

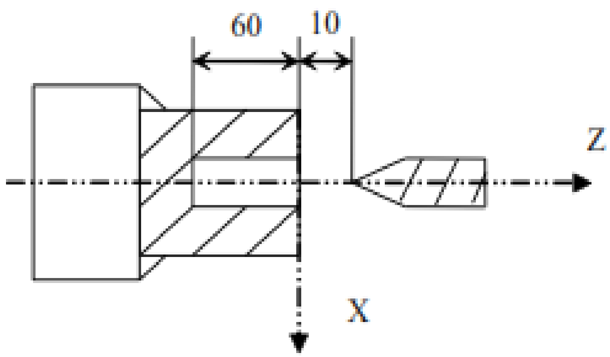


Figure 9.13 End Face Peck Drilling Cycle – Example

```
%1234
T0101
M03S500
G01 X0 Z10
G74 Z-60R1Q5F1000
M30
```

Outer Diameter Grooving Cycle (G75)

Programming

G75 X(U)_ R(e) Q(△K) F_

Explanation of the parameters

X Coordinate value on X axis of the end point in absolute command

U Coordinate value on X axis of the end point with reference to the starting point in incremental command

R Retraction amount(e) for each feed. It must be absolute value.

Q Depth of grooving(△K) for each feed. It must be absolute value.

F Feed rate

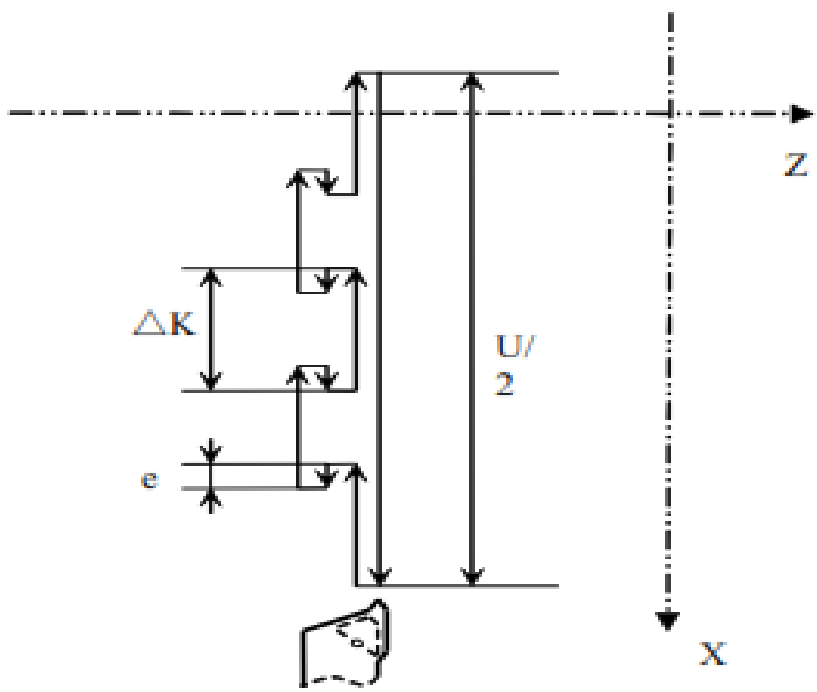


Figure 9.14 Outer Diameter Grooving Cycle (G75)

Function

This command can be used for grooving.

Example

Use G75 to groove a hole on a workpiece.

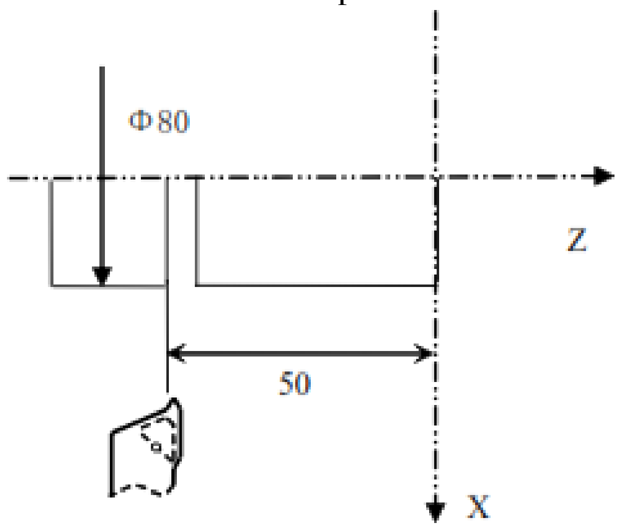


Figure 9.15 Outer Diameter Grooving Cycle - Example

```
%1234
T0101
M03S500
G01 X50 Z50
G75 X10R1Q5F1000
M30
```

Multiple Repetitive Cycle

Multiple repetitive cycle command can only use one command to finish the rough machining and the finish machining.

Stock Removal in Turning (G71)

- **Stock Removal in Turning without Groove**

Programming

G71 U(△d) R(r) P(ns) Q(nf) X(△x) Z(△z) F(f) S(s) T(t)

Explanation of the parameters

U(△d) the cutting depth (radius designation). The cutting direction depends on the direction of AA'.

R(r) Retraction amount

P(ns) Sequence number of the first block for the finishing program.
Q(nf) Sequence number of the last block for the finishing program.
X(△x) Distance and direction of finishing allowance on X axis

Z(△z) Distance and direction of finishing allowance on Z axis

F(f), S(s), T(t) F, S, T function are only effective for the rough machining, i.e, it is not effective in the finishing program – between P(ns) and Q(nf).

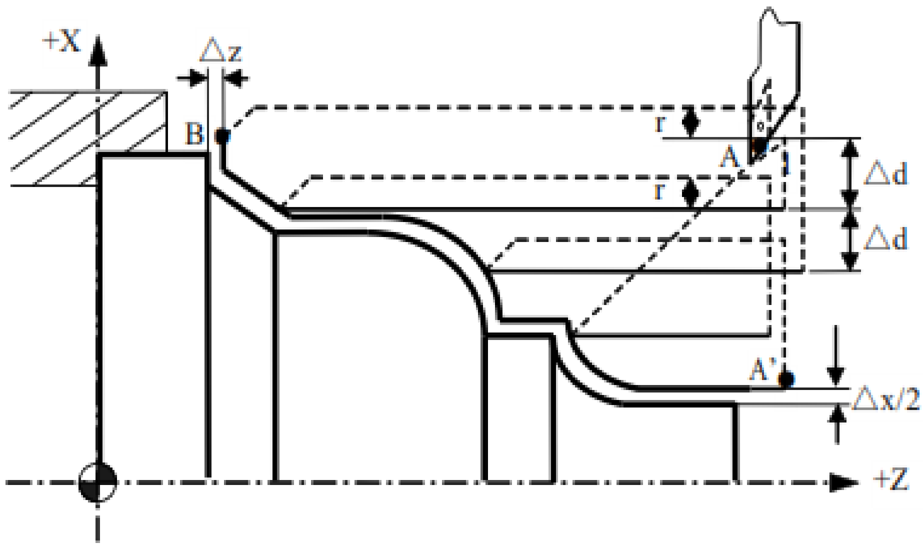


Figure 9.16 Stock Removal in Turning without Groove (G71)

Function

This command can do a stock removal in facing without groove. The machining path is A→A'→B

Note

1) G00 or G01 must be used in the finishing program – between P(ns) and Q(nf).
Otherwise, there is an alarm message.
2) G71 can not be used in MDI mode.
3) G98 and G99 can not used in the finishing program – between P(ns) and Q(nf).
4) The direction of △x and △z is shown in the following figure.

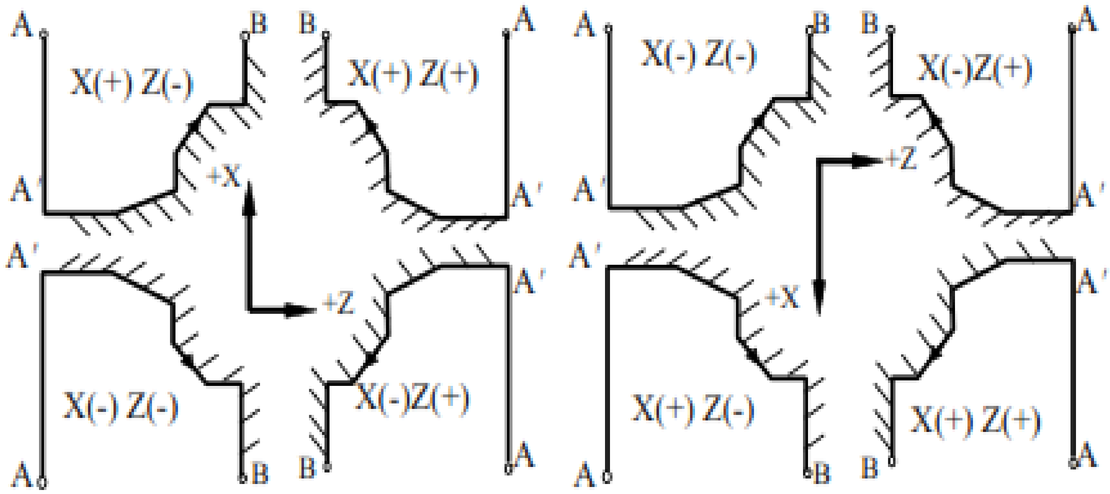


Figure 9.17 Direction of the finishing allowance in G71

Example

Example 1

The initial point A is (46, 3). The depth of cut is 1.5mm (radius designation). The retraction amount is 1mm. The finishing allowance in the X direction is 0.6mm, and the finishing allowance in the Z direction is 0.1mm. The dashed line stands for the original part.

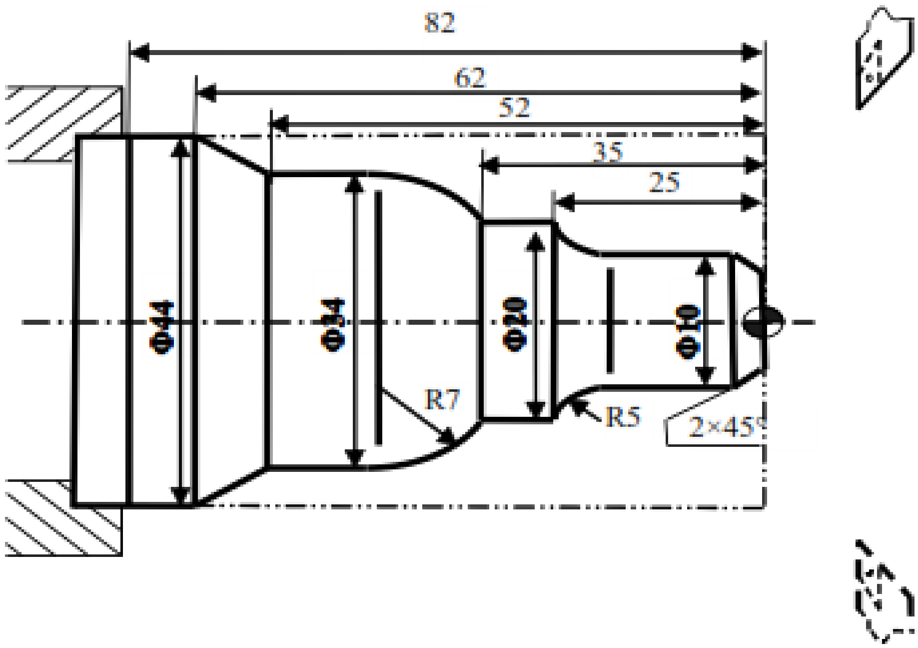


Figure 9.18 Outer Diameter Removal without Groove – Example

```
%3325
T0101
N1 G00 X80 Z80
N2 M03 S400
N3 G01 X46 Z3 F100
N4 G71U1.5R1P5Q13X0.6 Z0.1
N5 G00 X0
N6 G01 X10 Z-2
N7 Z-20
N8 G02 U10 W-5 R5
N9 G01 W-10
N10 G03 U14 W-7 R7
N11 G01 Z-52
N12 U10 W-10
N13 W-20
N14 X50
N15 G00 X80 Z80
N16 M05
N17 M30
```

Example 2

The initial point A is (6, 3). The depth of cut is 1.5mm (radius designation). The retraction amount is 1mm. The finishing allowance in the X direction is 0.6mm, and the finishing allowance in the Z direction is 0.1mm. The dashed line stands for the original part.

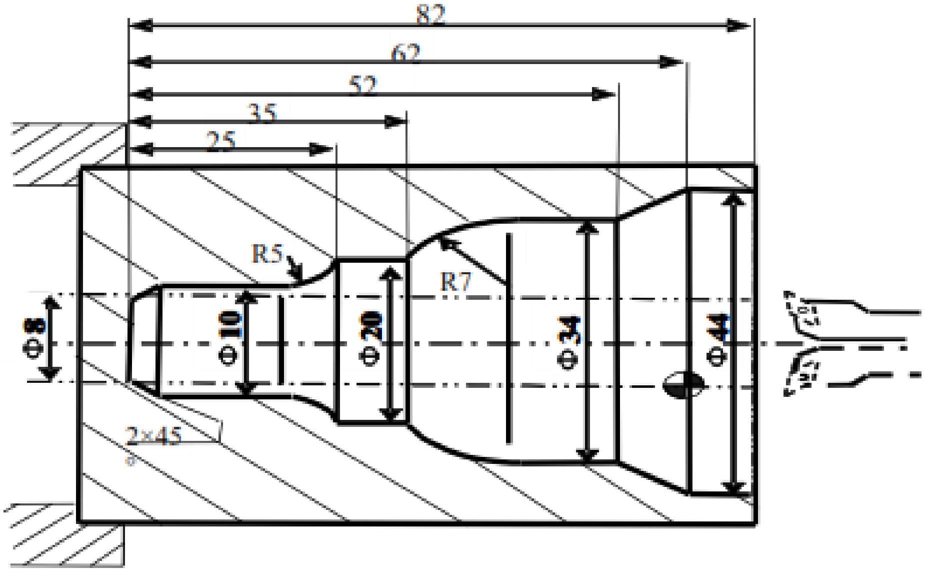


Figure 9.19 Internal Diameter Removal without Groove – Example

```
%3326
N1 T0101
N2 G00 X80 Z80
N3 M03 S400
N4 X6 Z5
G71U1R1P8Q16X-0.6Z0.1 F100
N5 G00 X80 Z80
N6 T0202
N7 G00 G41X6 Z5
N8 G00 X44
N9 G01 Z-20 F80
N10 U-10 W-10
N11 W-10
N12 G03 U-14 W-7 R7
N13 G01 W-10
N14 G02 U-10 W-5 R5
N15 G01 Z-80
N16 U-4 W-2
N17 G40 X4
N18 G00 Z80
N19 X80
N20 M30
```

- **Stock Removal in Turning with Groove**

Programming

G71 U(△d) R(r) P(ns) Q(nf) E(e) F(f) S(s) T(t)

Explanation of the parameters

U(△d) the cutting depth (radius designation). The cutting direction depends on the direction of AA'.

R(r) Retraction amount

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

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

E(e) Distance and direction of finishing allowance on X axis. It is positive when it is outer diameter cutting. It is negative when it is internal diameter cutting.

F(f), S(s), T(t) F, S, T function are only effective for the rough machining, i.e, it is not effective in the finishing program – between P(ns) and Q(nf).

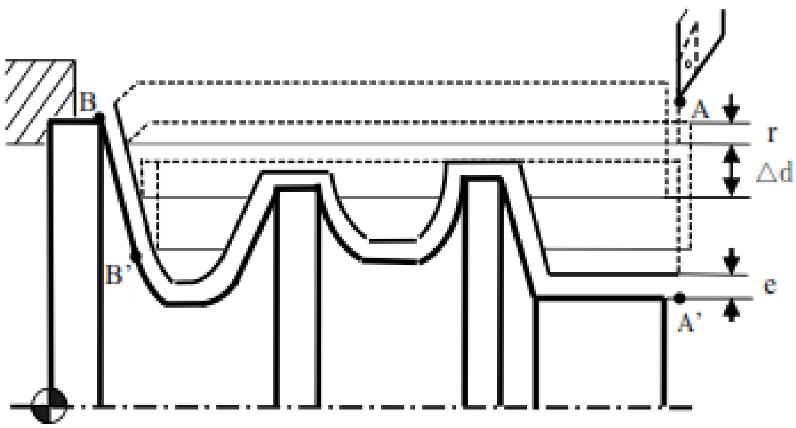


Figure 9.20 Stock Removal in Turning with Groove (G71)

Function

This command can do a stock removal in facing with groove. The machining path A→A'→B'→B.

Example

Use G71 to program.

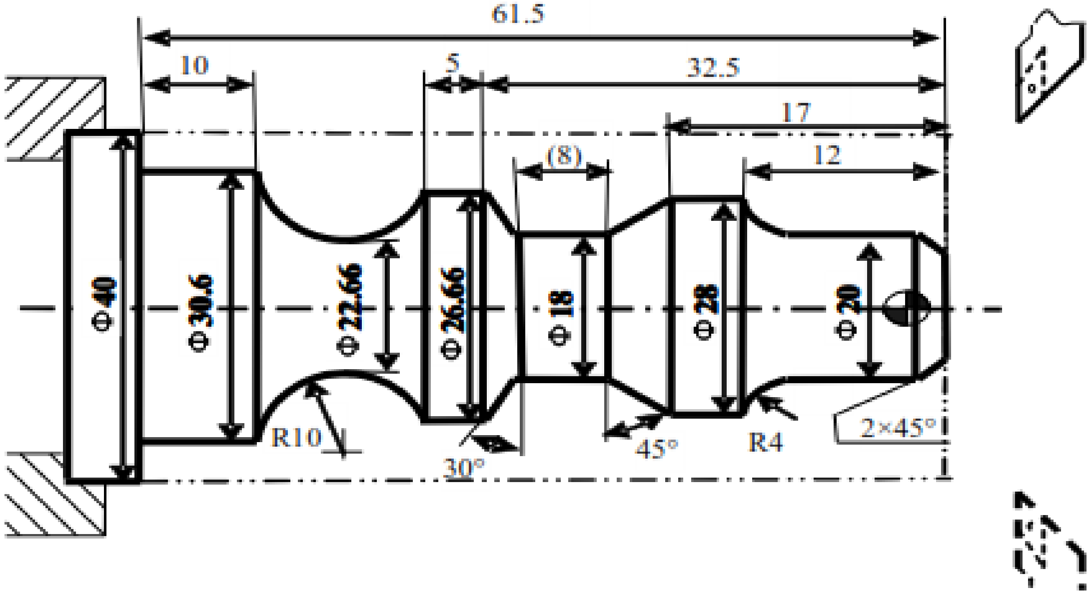


Figure 9.21 Stock Removal in Turning with Groove - Example

```
%3327
N1 T0101
N2 G00 X80 Z100
M03 S400
N3 G00 X42 Z3
N4G71U1R1P8Q19E0.3F100
N5 G00 X80 Z100
N6 T0202
N7 G00 G42 X42 Z3
N8 G00 X10
N9 G01 X20 Z-2 F80
N10 Z-8
N11 G02 X28 Z-12 R4
N12 G01 Z-17
N13 U-10 W-5
N14 W-8
N15 U8.66 W-2.5
N16 Z-37.5
N17 G02 X30.66 W-14 R10
N18 G01 W-10
N19 X40
N20 G00 G40 X80 Z100
N21 M30
```

Stock Removal in Facing (G72)

Programming

G72 W(Δd) R(r) P(ns) Q(nf) X(Δx) Z(Δz) F(f) S(s) T(t)

Explanation of the parameters

W(△d) the cutting depth (radius designation). The cutting direction depends on the direction of AA'.

R(r) Retraction amount

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

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

X(△x) Distance and direction of finishing allowance on X axis

Z(△z) Distance and direction of finishing allowance on Z axis

F(f), S(s), T(t) F, S, T function are only effective for the rough machining, i.e, it is not effective in the finishing program – between P(ns) and Q(nf).

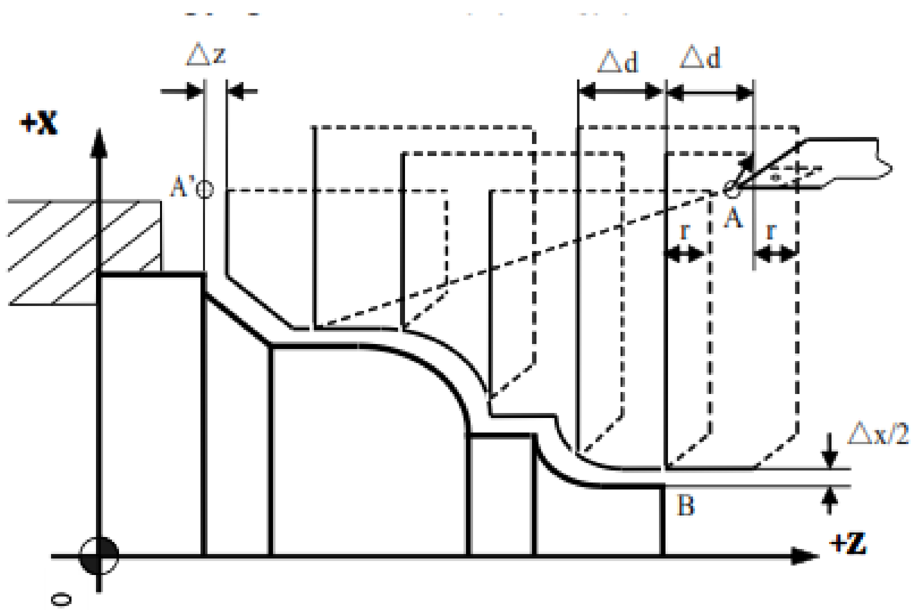


Figure 9.22 Stock Removal in Facing (G72)

Function

This command can do a stock removal in facing. The machining path is A→A'→B

Note

1) G00 or G01 must be used in the finishing program – between P(ns) and Q(nf).
Otherwise, there is an alarm message.
2) G72 can not be used in MDI mode.
3) G98 and G99 can not used in the finishing program – between P(ns) and Q(nf).
4) The direction of $\triangle$x and $\triangle$z is shown in the following figure.

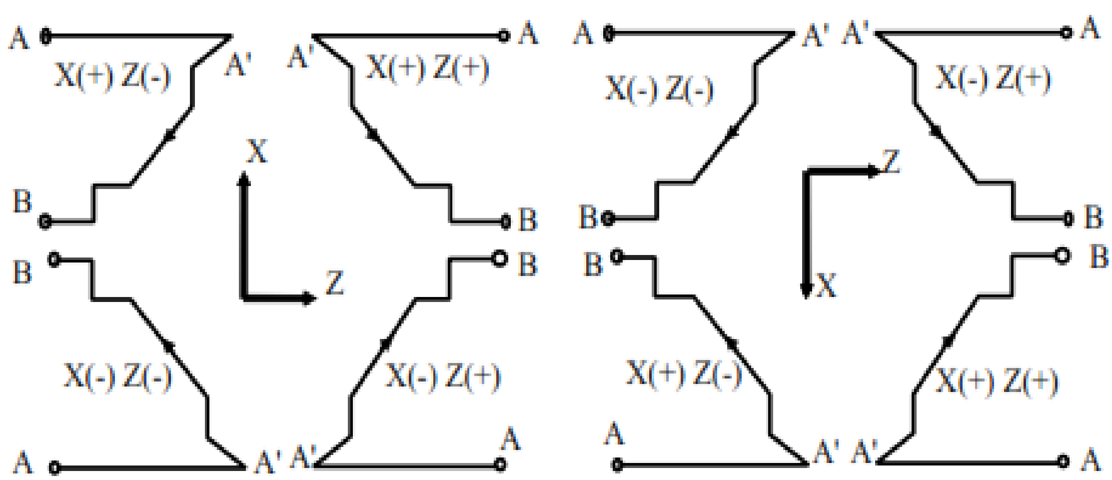


Figure 9.23 Direction of the finishing allowance in G72

Example 1

Use G72 to program. The initial point A is (80, 1). The depth of cutting is 1.2mm. The retraction amount is 1mm. The finishing allowance in the X direction is 0.2mm, and the finishing allowance in the Z direction is 0.5mm. The dashed line stands for the original part.

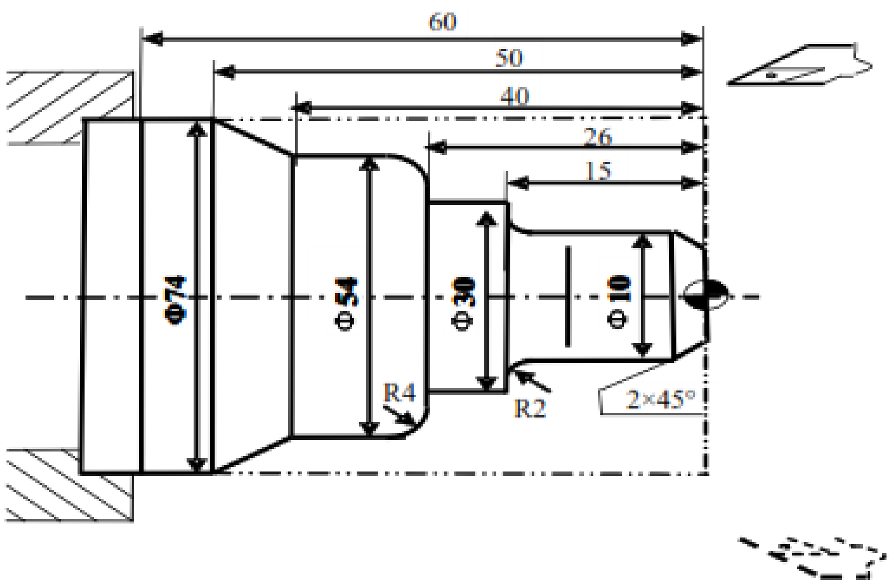


Figure 9.24 Outer Diameter Removal in Facing - Example

```
%3328
N1 T0101
N2 G00 X100 Z80
N3 M03 S400
N4 X80 Z1
N5 G72W1.2R1P8Q17X0.2Z0.5F100
N6 G00 X100 Z80
N7 G42 X80 Z1
N8 G00 Z-53
N9 G01 X54 Z-40 F80
N10 Z-30
N11 G02 U-8 W4 R4
N12 G01 X30
N13 Z-15
N14 U-16
N15 G03 U-4 W2 R2
N16 G01 Z-2
N17 U-6 W3
N18 G00 X50
N19 G40 X100 Z80
N20 M30
```

Example 2

Use G72 to program. The initial point A is (80, 1). The depth of cutting is 1.2mm. The retraction amount is 1mm. The finishing allowance in the X direction is 0.2mm, and the finishing allowance in the Z direction is 0.5mm. The dashed line stands for the original part.

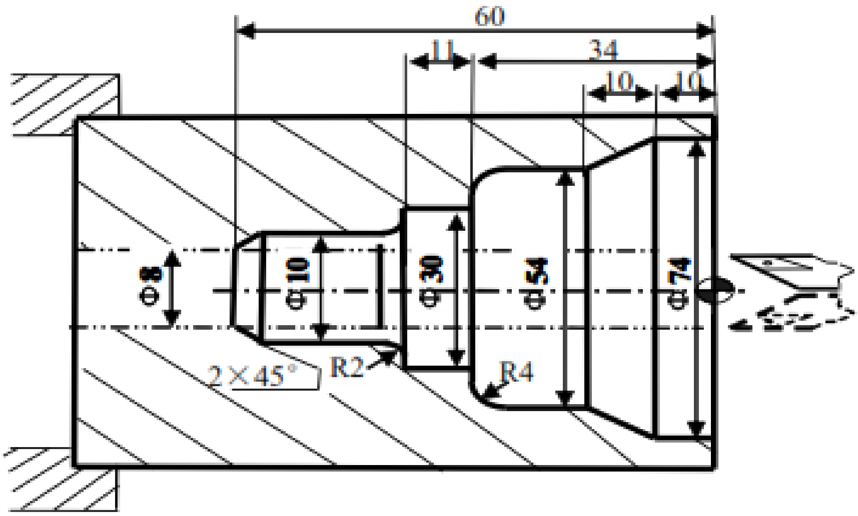


Figure 9.25 Internal Diameter Removal in Facing - Example

```
%3329
N1 T0101
N2 G00 X100 Z80
N3 M03 S400
N4 G00 X6 Z3
N5 G72W1.2R1P5Q15X-0.2Z0.5F100
N6 G00 Z-61
N7 G01 U6 W3 F80
N8 W10
N9 G03 U4 W2 R2
N10 G01 X30
N11 Z-34
N12 X46
N13 G02 U8 W4 R4
N14 G01 Z-20
N15 U20 W10
N16 Z3
N17 G00 X100 Z80
N18 M30
```

Pattern Repeating (G73)

Programming

G73 U(ΔI) W(ΔK) R(r) P(ns) Q(nf) X(Δx) Z(Δz) F(f) S(s) T(t)

Explanation of the parameters

U(△I) distance and direction of total roughing allowance in the X direction (radius designation).

W(△K) distance and direction of total roughing allowance in the X direction (radius designation)

R(r) Repeated times of cutting

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

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

X(△x) Distance and direction of finishing allowance on X axis

Z(△z) Distance and direction of finishing allowance on Z axis

F(f), S(s), T(t) F, S, T function are only effective for the rough machining, i.e, it is not effective in the finishing program – between P(ns) and Q(nf).

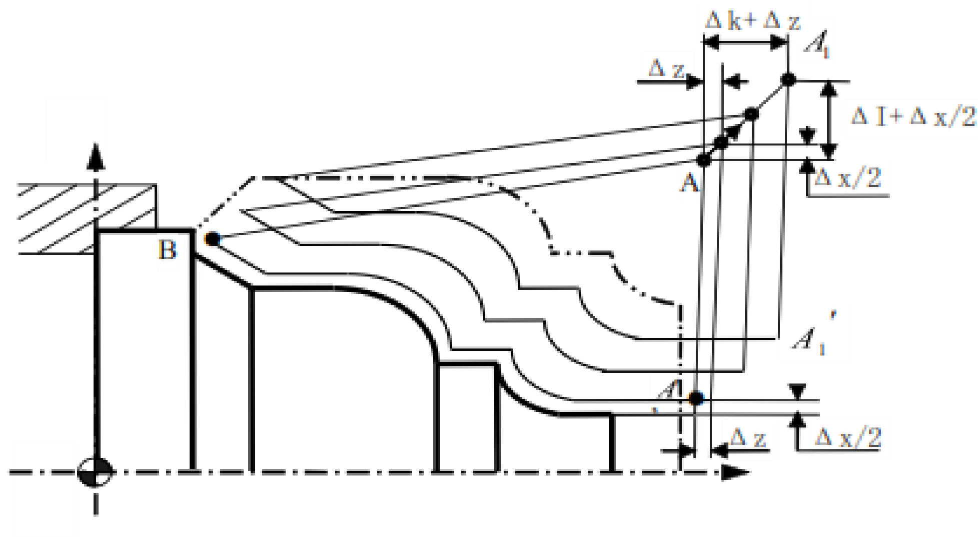


Figure 9.26 Pattern Repeating (G73)

Function

G73 command can cut a workpiece at a fixed pattern repeatedly.
The machining path is A→A'→B.

Note

1) G00 or G01 must be used in the finishing program – between P(ns) and Q(nf).
Otherwise, there is an alarm message.
2) G73 can not be used in MDI mode.
3) G98 and G99 can not used in the finishing program – between P(ns) and Q(nf).
4) The depth for each cutting on X axis = $\triangle I/r$

The depth for each cutting on Z axis = $\triangle K/r$

5) The direction of $\triangle I$ and $\triangle K$, and the direction of $\triangle x$ and $\triangle z$ should be noted.

Example

Use G73 to program. The initial point A is (60, 5). The total roughing allowance on X and Z axis are 3mm, 0.9mm, respectively. The times of rough cutting is 3. The finishing allowance on X and Z axis are 0.6mm, 0.1mm respectively. The dash-dot-line is the part's blank.

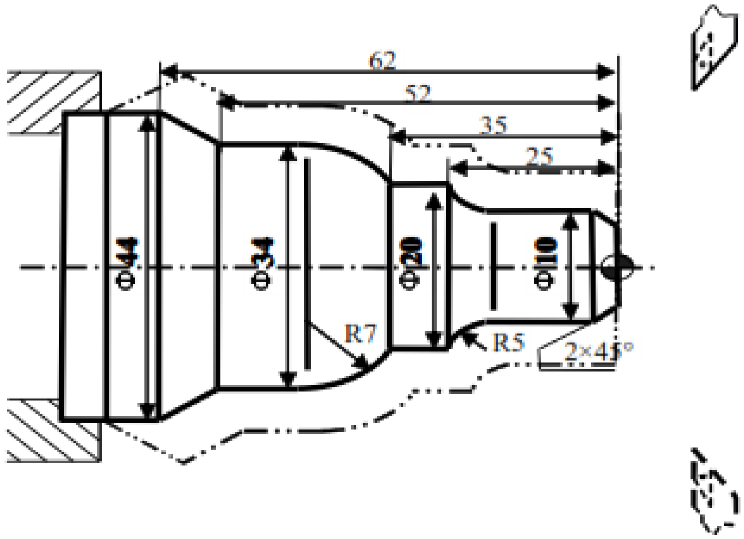


Figure 9.27 Pattern Repeating - Example

```
%3330
N1 T0101
N2 G00 X80 Z80
N3 M03 S400
N4 G00 X60 Z5
N5 G73U3W0.9R3P5Q13X0.6Z0.1F120
N6 G00 X0 Z3
N7 G01 U10 Z-2 F80
N8 Z-20
N9 G02 U10 W-5 R5
N10 G01 Z-35
N11 G03 U14 W-7 R7
N12 G01 Z-52
N13 U10 W-10
N14 U10
N15 G00 X80 Z80
N16 M30
```

Multiple Thread Cutting Cycle (G76)

Programming

G76 C(c) R(r) E(e) A(a) X(U) Z(W) I(i) K(k) U(d) V(Δdmin) Q(Δd) P(p) F(L)

Explanation of the parameters

C(c) Repetitive count in finishing (1~99)
R(r) Retraction amount on Z axis (00~99)
E(e) Retraction amount on X axis (00~99)
A(a) Angle of tool tip (two-digit number). It could be 80°, 60°, 55°, 30°, 29°, or 0°.
X, Z Coordinate value of end point (point C) in absolute command.
U, W Coordinate value of end point (point C) with reference to the initial point (point A) in incremental command

I(i) Difference of thread radius. If i=0, it is straight thread cutting.

K(k) Height of thread. This value is specified by the radius value on X axis.
U(d) The finishing allowance (radius designation).
V(Δdmin) The minimum cutting depth (radius designation). The cutting depth

isΔdmin when the cutting depth $(\Delta d\sqrt{n} - \Delta d\sqrt{n-1})$ is less thanΔdmin.

Q(Δd) Depth of cutting at the first cut (radius designation)

P(p) Start point offset.
F(L) Thread lead

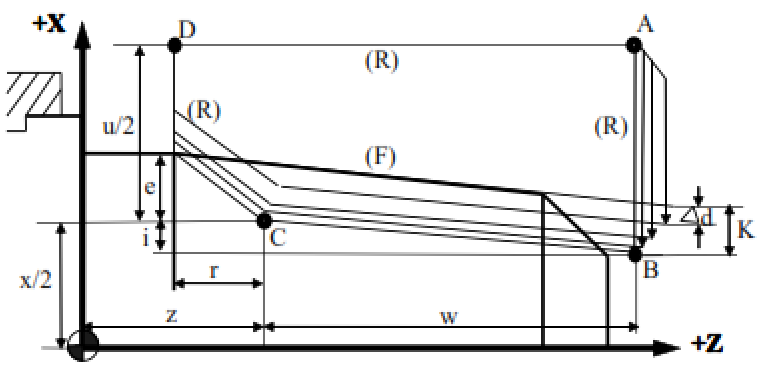


Figure 9.28 Multiple Thread Cutting Cycle (G76)

Function

G76 command can do the multiple thread cutting. The machining path is A→B→C→D.

Note

1) The signs of U and W is defined by the direction of AC and CD respectively.

2) The cutting depth in 1st cut is Δd, the cutting depth in nth cut is $\Delta d\sqrt{n}$. The bite of each cycle is $\Delta d\ (\sqrt{n}-\sqrt{n-1})$.

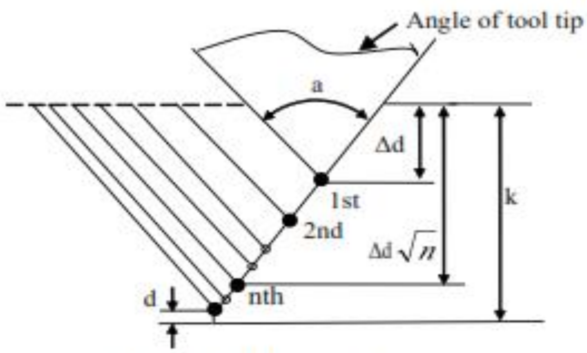


Figure 9.29 The depth of cutting

3) The cutting speed of BC path is specified by feed rate. And the other paths (AB, CD, DA) are specified by rapid traverse speed.

Example

Use G76 to program. The thread is ZM60×2. Sizes in bracket is from standards. (tan1.79=0.03125)

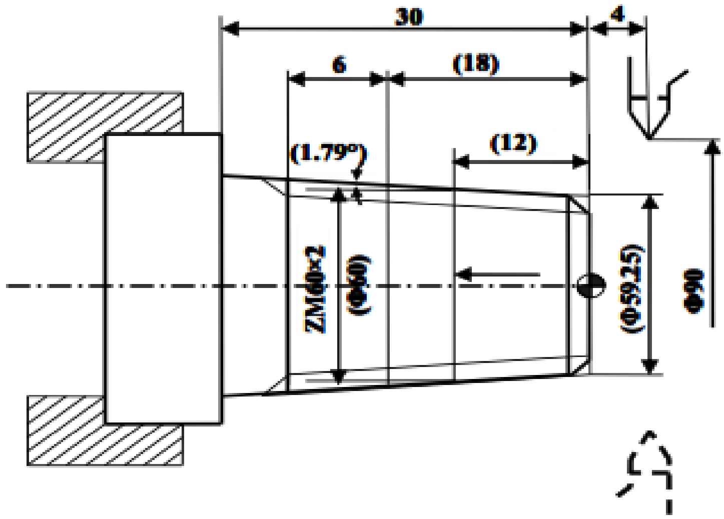


Figure 9.30 Multiple Thread Cutting Cycle - Example

```
%3331
N1 T0101
N2 G00 X100 Z100
N3 M03 S400
N4 G00 X90 Z4
N5 G80 X61.125 Z-30 I-1.063 F80
N6 G00 X100 Z100 M05
N7 T0202
N8 M03 S300
N9 G00 X90 Z4
N10 G76C2R-3E1.3A60X58.15Z-24I-0.875K1.299U0.1V0.1Q0.9F2
N11 G00 X100 Z100
N12 M05
N13 M30
```