

Thapar Institute of Engineering & Technology – Patiala

Manufacturing Processes

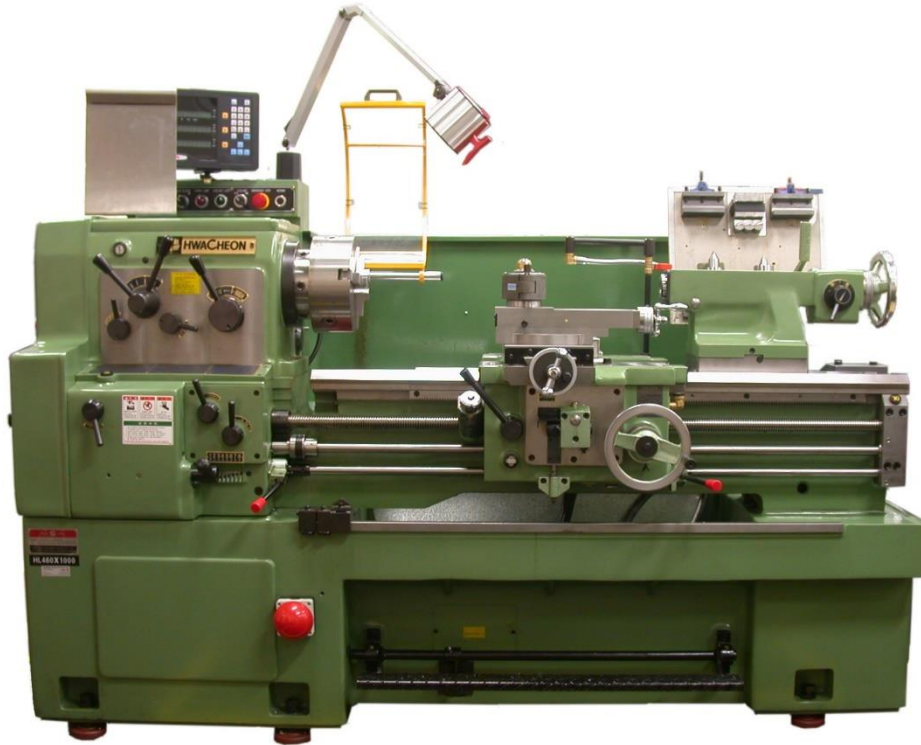
UES 102

Thapar Institute of Engineering & Technology
(Deemed to be University)
Bhadson Road, Patiala, Punjab, Pin-147004
Contact No. : +91-175-2393201
Email : info@thapar.edu

CNC Machining

ti
THAPAR INSTITUTE
OF ENGINEERING & TECHNOLOGY
(Deemed to be University)

Machine Tool Use for Machining

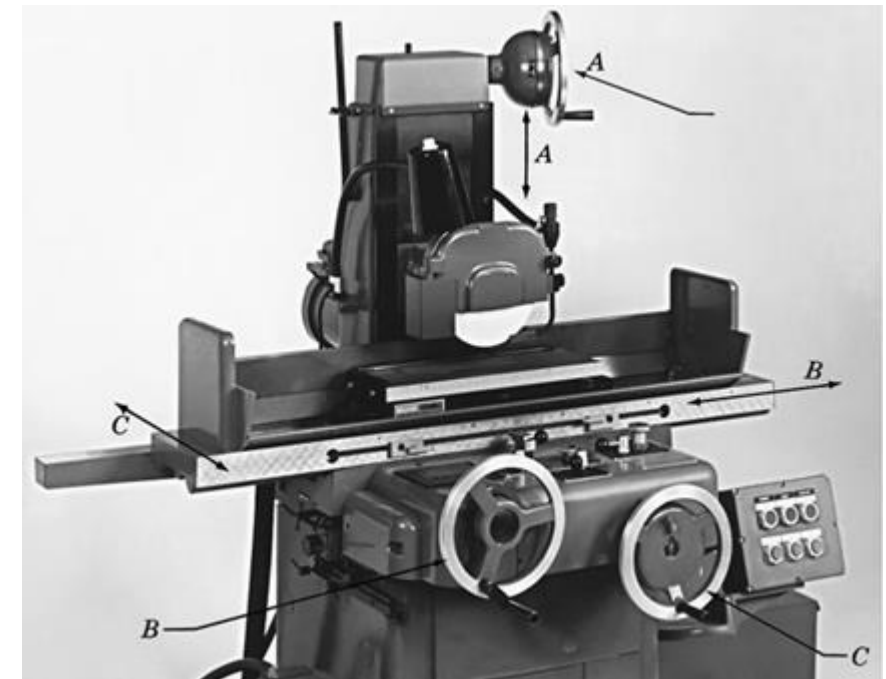
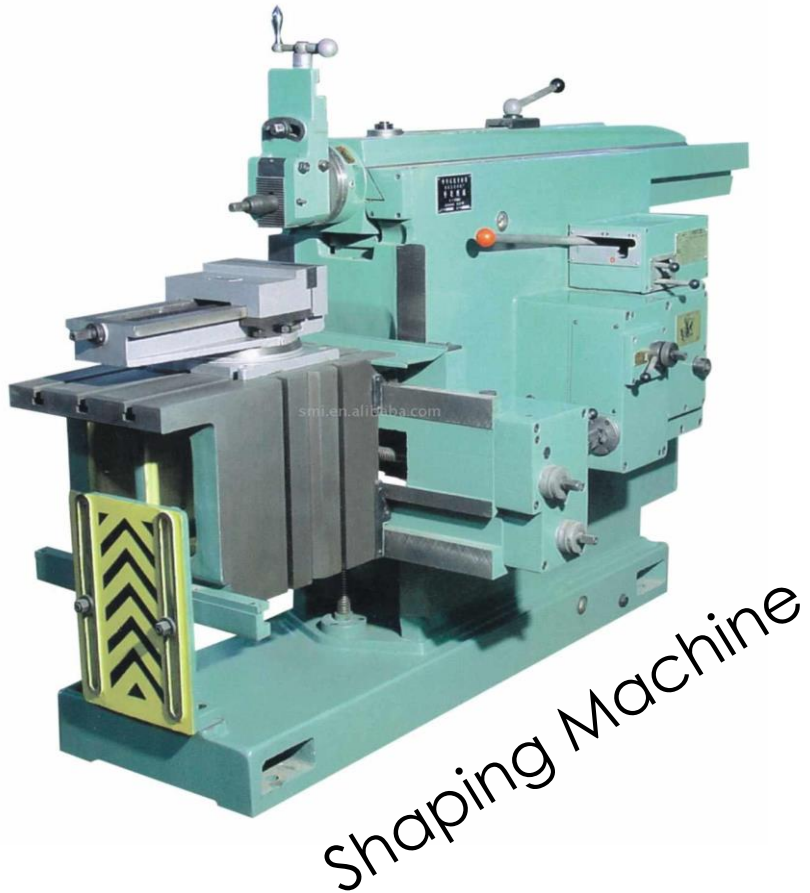


Lathe

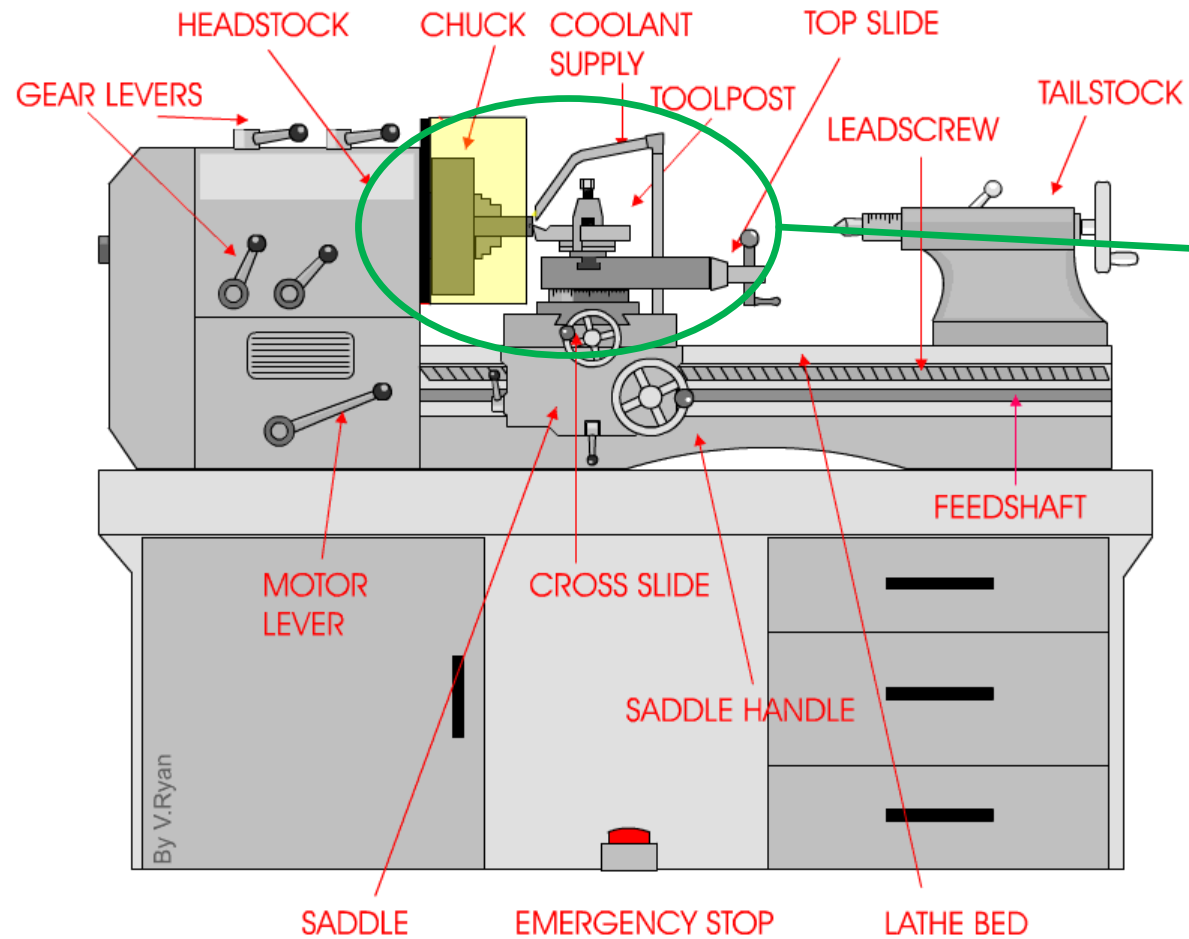


Milling Machine

Machine Tool Use for Machining

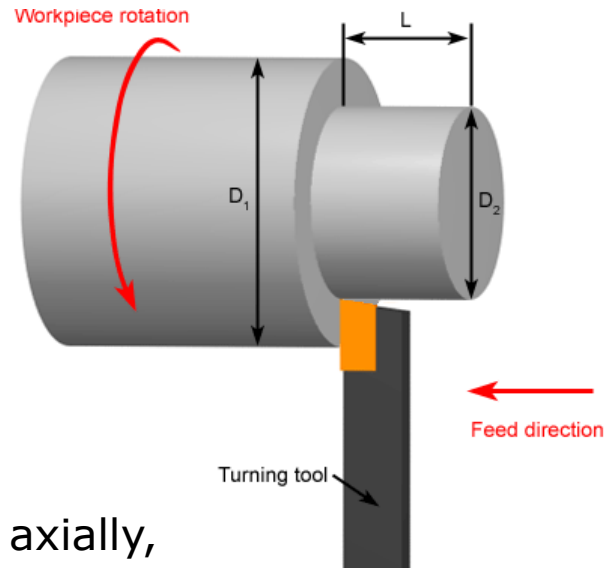
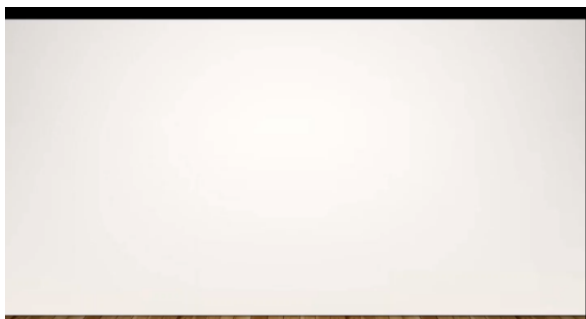


LATHE



Components of Lathe

Various Operations That Can Be Performed in Lathe - External



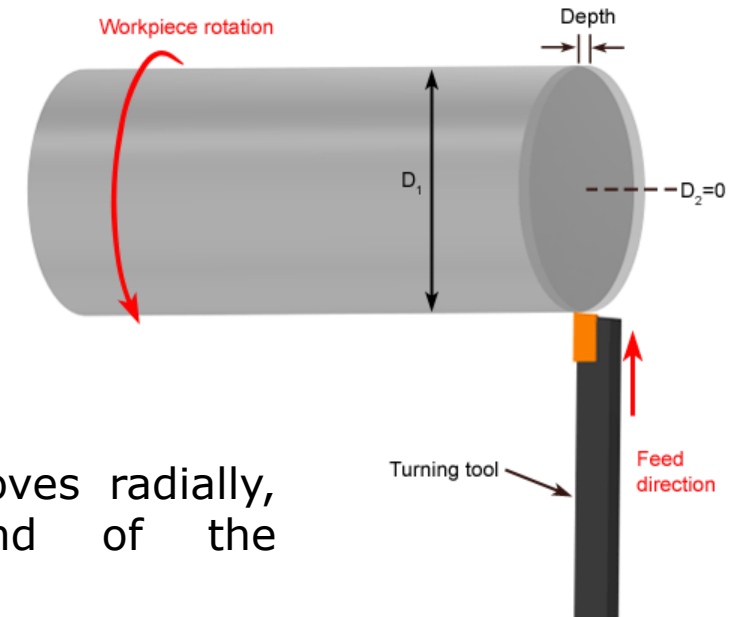
Turning

- Turning tool moves axially, along the side of the workpiece
- removing material to form steps, tapers, chamfers, and contours.

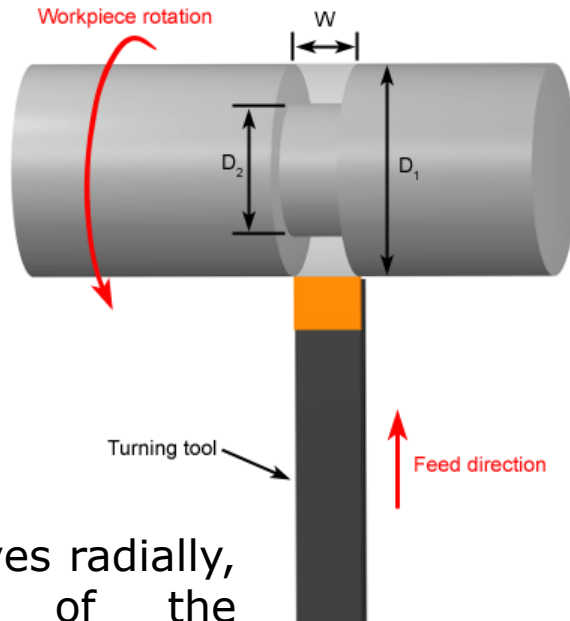


Facing

- Turning tool moves radially, along the end of the workpiece
- removing a thin layer of material to provide a smooth flat surface

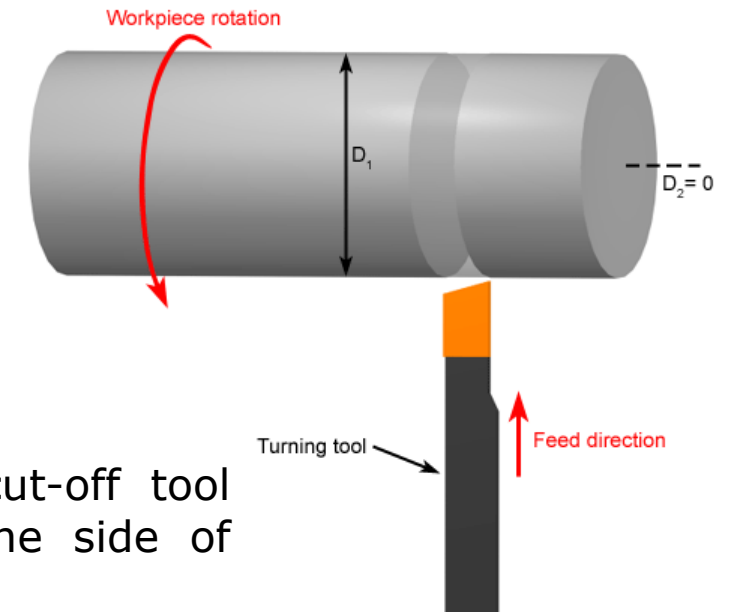


Various Operations That Can Be Performed in Lathe - External



Grooving

- Grooving tool moves radially, into the side of the workpiece,
- cutting a groove equal in width to the cutting tool.



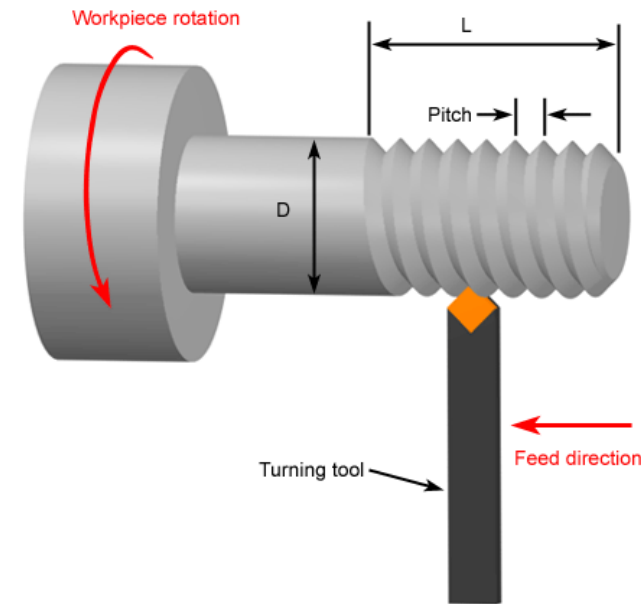
Cut-off (parting) -

- Similar to grooving, cut-off tool moves radially, into the side of the workpiece,
- continues until the centre or inner diameter of the workpiece is reached.

Various Operations That Can Be Performed in Lathe - External

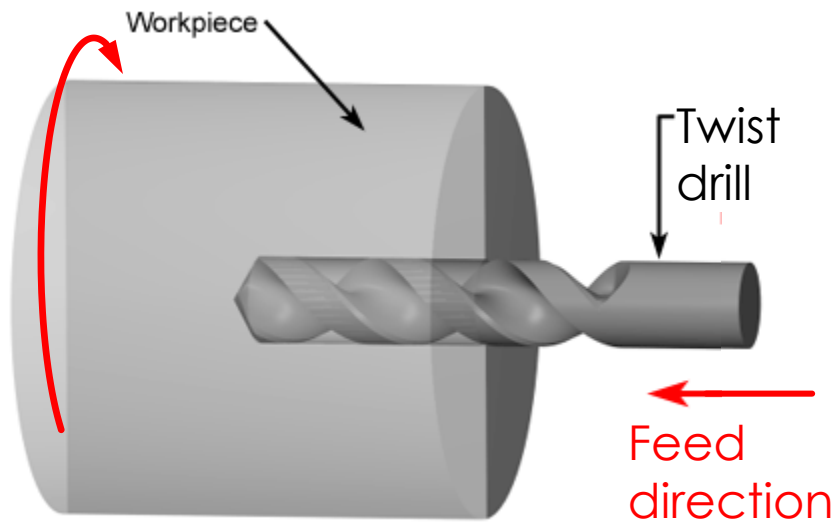
Thread cutting

- Threading tool, typically with a 60° pointed nose, moves axially, along the side of the workpiece,
- cutting threads into the outer surface.



Thread cutting

Various Operations That Can Be Performed in Lathe - Internal

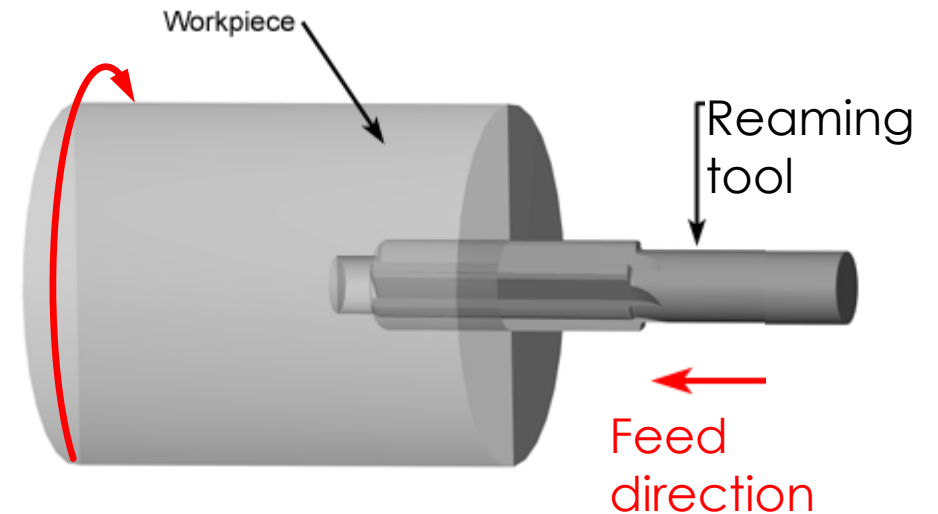


Drilling

- A drill enters the workpiece axially through the end,
- Cuts a hole with a diameter equal to that of the tool.

Reaming

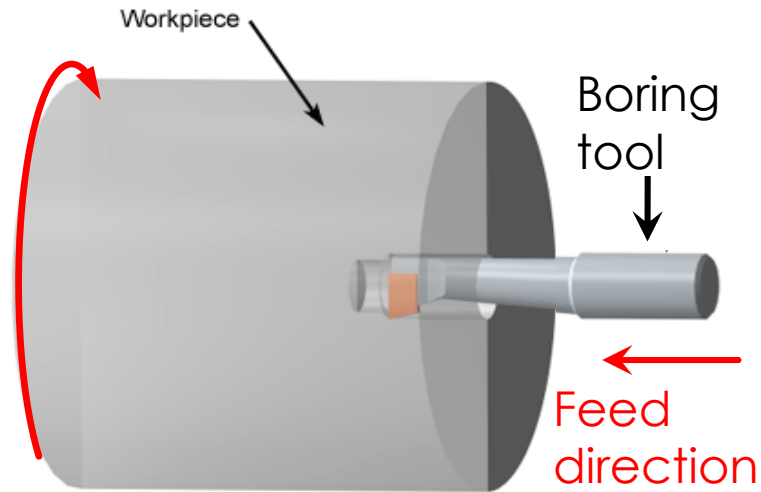
- A reamer enters the workpiece axially through the end
- Enlarges an existing hole to the diameter of the tool.
- Performed after drilling to obtain both a more accurate diameter and a smoother internal finish



Various Operations That Can Be Performed in Lathe - Internal

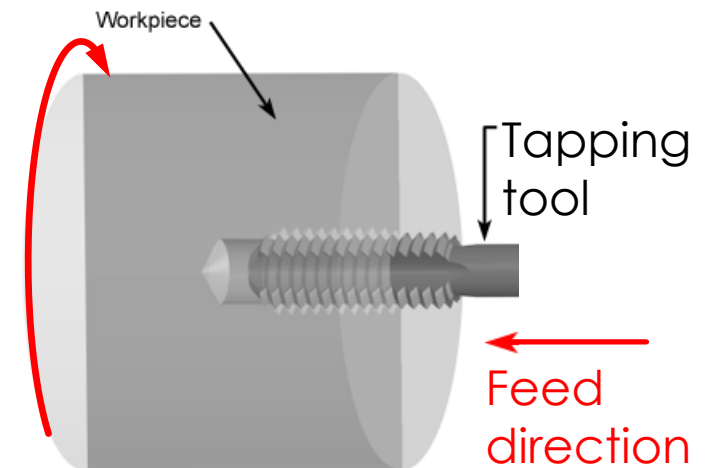
Boring

- A boring tool enters the workpiece axially,
- Cuts along an internal surface to form different features, such as steps, tapers, chamfers, and contours.
- Enlarge the existing hole.



Tapping

- A tap enters the workpiece axially through the end
- Cuts internal threads into an existing hole.
- The existing hole is typically drilled by the required tap drill size that will accommodate the desired tap.



Computer Numerical Control (CNC) Machines

Made with **KINEMASTER**

- ▶ High Quality
- ▶ High Accuracy & Precision
- ▶ High Production rate
- ▶ Manufacturing of Complex components

STAR INFOTECH

CNC Lathe - G Code

- **G-code** is the most widely used as numerical control (NC) programming language
- Generally it is telling the computerized machine tools **what type of action to perform** or **how to make something**
- Such as:
 - ✓ Where to move the cutting tool
 - ✓ How fast to move the cutting tool
 - ✓ Which path the cutting tool will move
- Within a **machine tool**, a **cutting tool** is moved according to the instructions of **G-code** through a **toolpath and cuts away** material to leave only the finished workpiece.

CNC Lathe - G Code

- G00 – Rapid transverse (or, Rapid movement)
- G01 – Linear motion with feed
- G02 – Tool movement in clock wise direction
- G03 – Tool movement in anti-clock wise direction
- G04 – Dwell time (or, waiting time)
- G17 – XY plane
- G20 – Inches mode
- G21 – Metric mode (in mm)
- G28 – Go to machine home position in incremental mode
- G98 – Feed in mm/min
- G99 – Feed in rev/min
- U – Incremental mode in X- axis
- W – Incremental mode in Z- axis
- X – Absolute mode in X- axis
- Z – Absolute mode in Z- axis

CNC Lathe - M Code

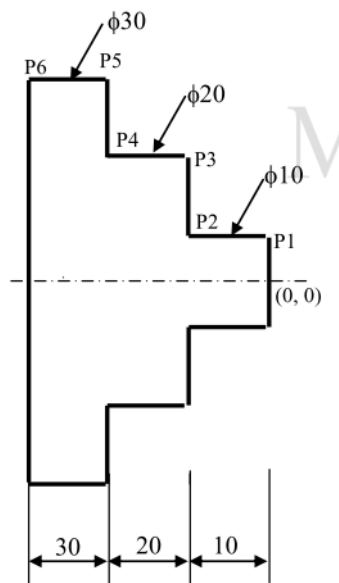
- **M-code** is used for Machine function / Auxiliary function
- Such as:
 - ✓ Rotation of spindle
 - ✓ Tool change
 - ✓ Program on or off

CNC Lathe - M Code

- ▶ M00 – Programme stop
- ▶ M02 – Program end. halts program execution. To execute the program once again, the system must reset.
- ▶ M03 – Spindle rotation clockwise
- ▶ M04 – Spindle rotation anti-clockwise
- ▶ M05 – Spindle stop
- ▶ M06 – Tool change
- ▶ M08 – Coolant on
- ▶ M09 – Coolant off
- ▶ M30 – Program Stop and Rewind. This command is used to stop the spindle, turns the coolant off, terminates and reset the CNC program
- ▶ M98 – Sub program calling
- ▶ M99 – Sub program end

Absolute System

- Absolute dimension system always refers to a **fixed reference point** in the drawing.
- This point has the function of a coordinate zero point.
- Define by **X and Z**



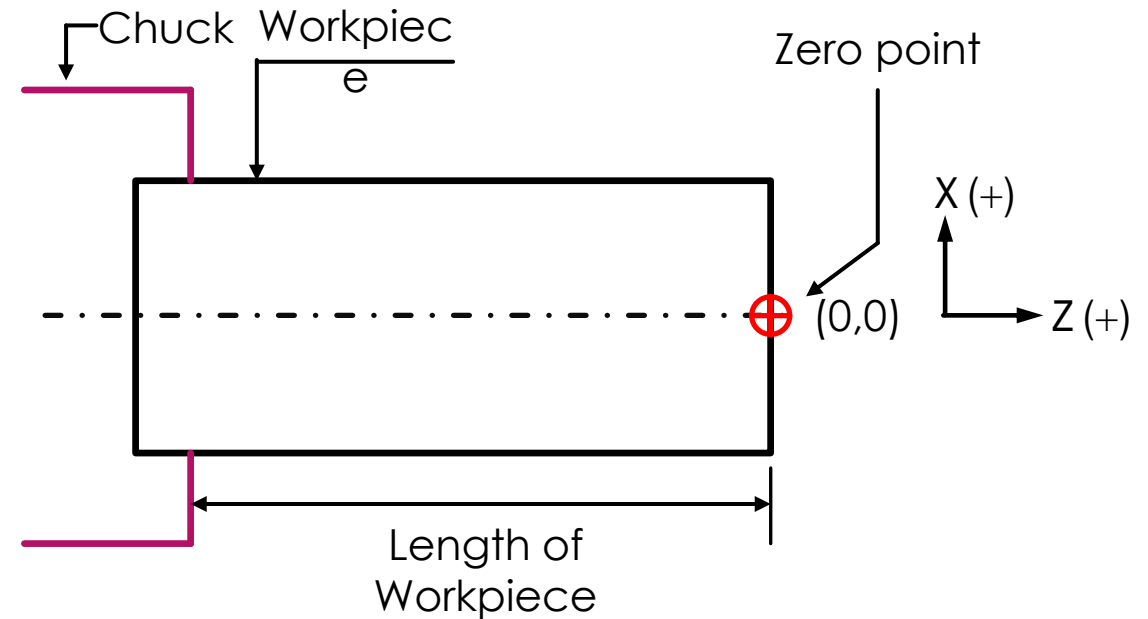
Incremental System

- Every measurement is considered from **previously dimensioned position**.
- Incremental dimensions are distance between adjacent points.
- Defined by **U and W**

ABSOLUTE DIMENSIONING			INCREMENTAL DIMENSIONING		
POINTS	X	Z	POINTS	U	W
P1	10	0	P1	10	0
P2	10	-10	P2	0	-10
P3	20	-10	P3	10	0
P4	20	-30	P4	0	-20
P5	30	-30	P5	10	0
P6	30	-60	P6	0	-30

Workpiece zero points or Program zero point

- On CNC machines, tool traverses are controlled by coordinating systems.
- Their accurate position within the machine tool is established by "Zero Points".
- The position of the workpiece zero point can be freely chosen by the programmer within the workpiece
- The workpiece zero point should be placed along the spindle axis (centre line), in line with the finished contour.



G00 - Rapid transverse (or, Rapid movement)

Syntax:

G00 X_ Z_

X= Co-ordinate in X-axis

Z= Co-ordinate in Z-axis

Example:

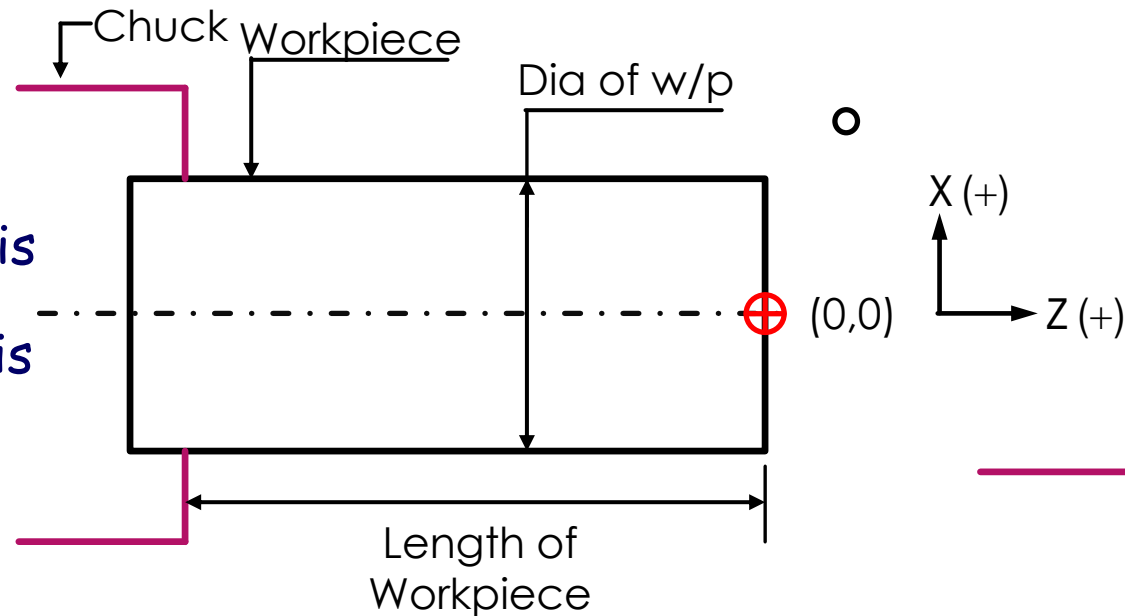
G21 G98

G28 U0 W0 / *Go tool to home position*

M06 T0101 / *Call tool no. 1*

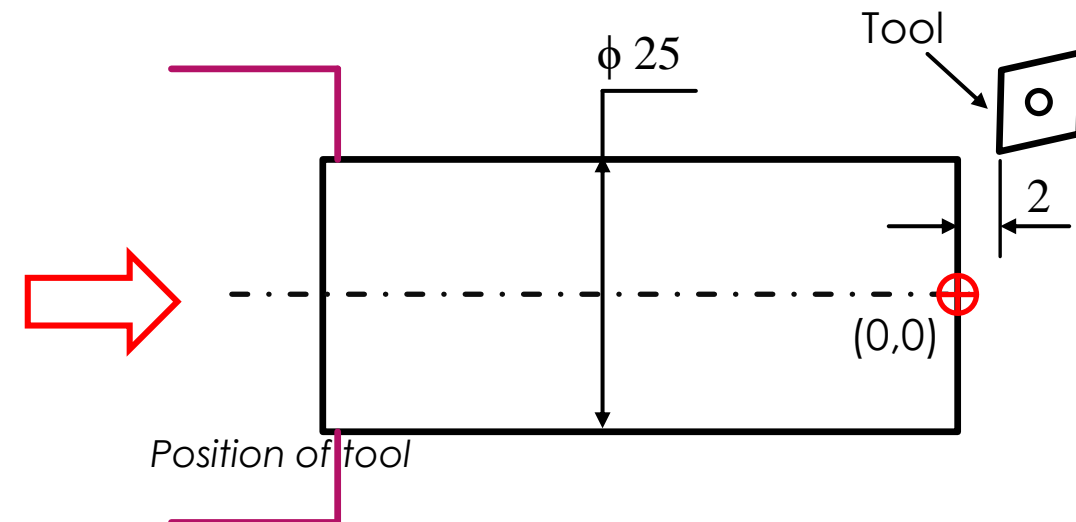
M03 S2500 / *Rotate tool at a speed of 2500 rpm*

G00 X25 Z2 / *The position of tool*



M00 - Programme stop
M02 - Program end. halts program execution. To execute the program once again, the system must reset.
M03 - Spindle rotation clockwise
M04 - Spindle rotation anti-clockwise
M05 - Spindle stop
M06 - Tool change
M08 - Coolant on
M09 - Coolant off
M30 - Program Stop and Rewind. This command is used to stop the spindle, turns the coolant off, terminates and reset the CNC program
M98 - Sub program calling
M99 - Sub program end

- G00 - Rapid transverse (or, Rapid movement)
- G01 - Linear motion with feed
- G02 - Tool movement in clock wise direction
- G03 - Tool movement in anti-clock wise direction
- G04 - Dwell time (or, waiting time)
- G17 - XY plane
- G20 - Inches mode
- G21 - Metric mode (in mm)
- G28 - Go to machine home position in incremental mode
- G98 - Feed in mm/min
- G99 - Feed in rev/min
- U - Incremental mode in X- axis
- W - Incremental mode in Z- axis
- X - Absolute mode in X- axis
- Z - Absolute mode in Z- axis



G01 - Linear Motion with Feed

Syntax:

G01 X_ Z_ F_

X= Co-ordinate in X-axis

Z= Co-ordinate in Z-axis

F= Feed in mm/min

Tool will come to the position at a feed rate

Example:

G21 G98

G28 U0 W0 / Go tool to home position

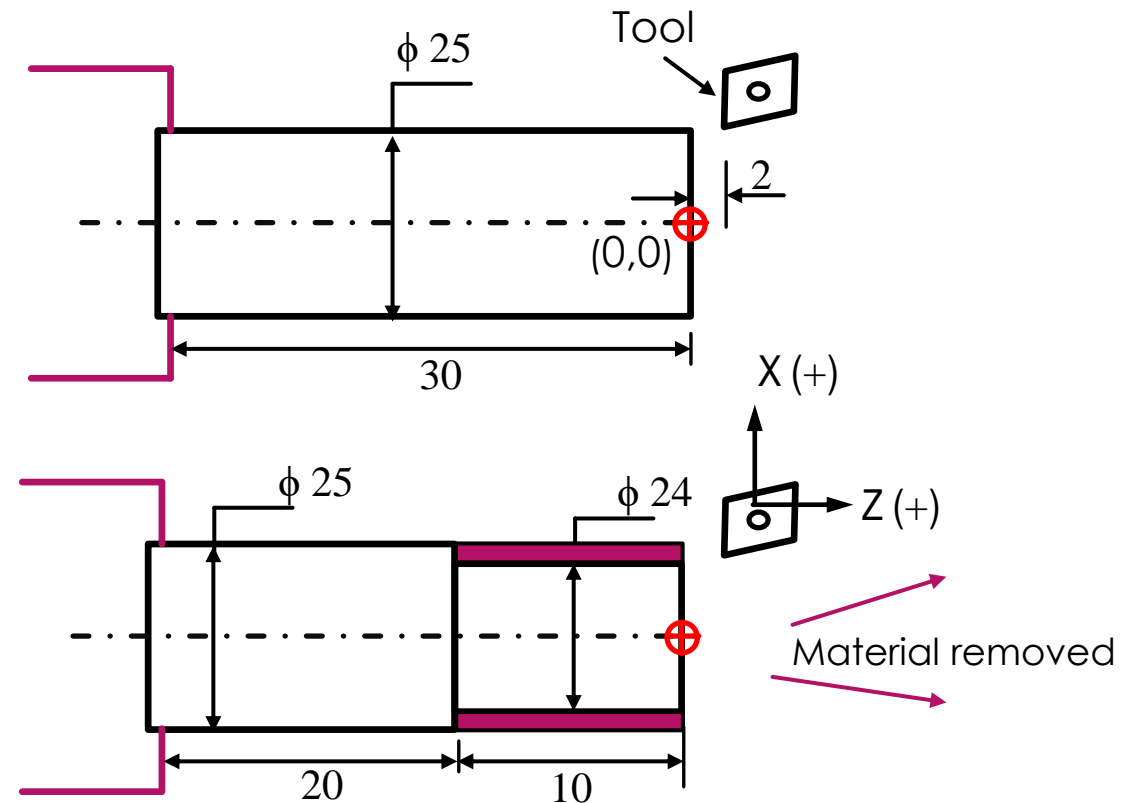
M06 T0101 / Call tool no. 1

M03 S2500 / Rotate tool at a speed of 2500 rpm

G00 X25 Z2 / First position of tool

G01 X24 F60

G01 Z -10 F60 / Final position of tool after removing material



G02 - Tool movement in clock wise direction

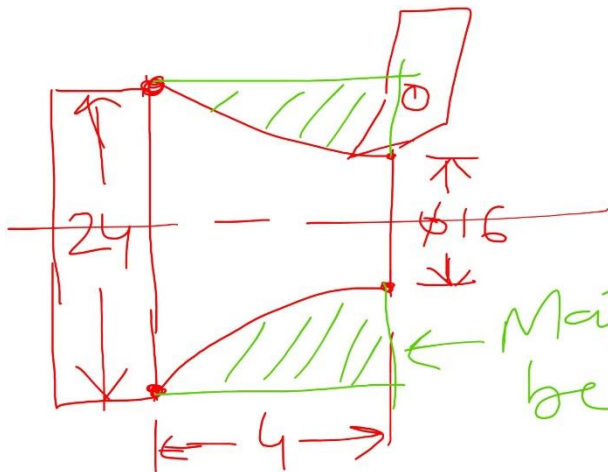
Syntax:

G02 X_ Z_ R_ F_

X= Co-ordinate in X-axis

Z= Co-ordinate in Z-axis

R= Radius of curve



Example:

G21 G98

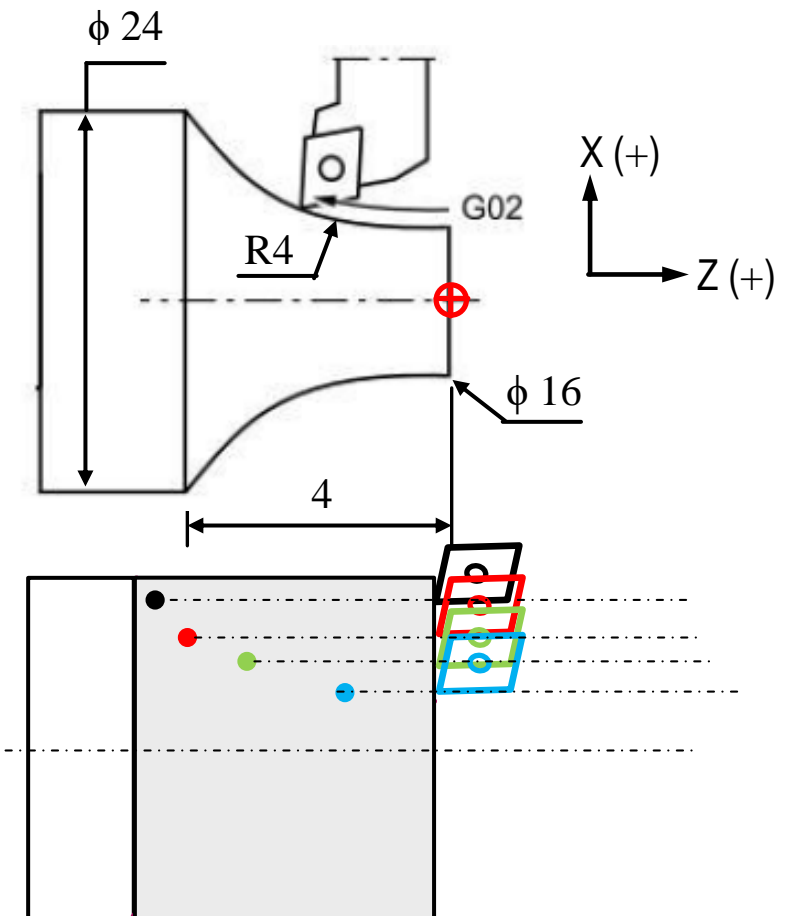
G28 U0 W0 / Go to tool home position

M06 T0101 / Call tool no. 1

M03 S2500 / Rotate tool at a speed of 2500 rpm

G00 X16 Z0 / First position of tool

G02 X24 Z -4 R4 F60 / *Final position of tool*



G03 - Tool movement in anti-clock wise direction

Syntax:

G03 X_ Z_ R_ F_

X= Co-ordinate in X-axis

Z= Co-ordinate in Z-axis

R= Radius of curve

Example:

G21 G98

G28 U0 W0 / Go to tool home position

M06 T0101 / Call tool no. 1

M03 S2500 / Rotate tool at a speed of 2500 rpm

G00 X8 Z0 / First position of tool

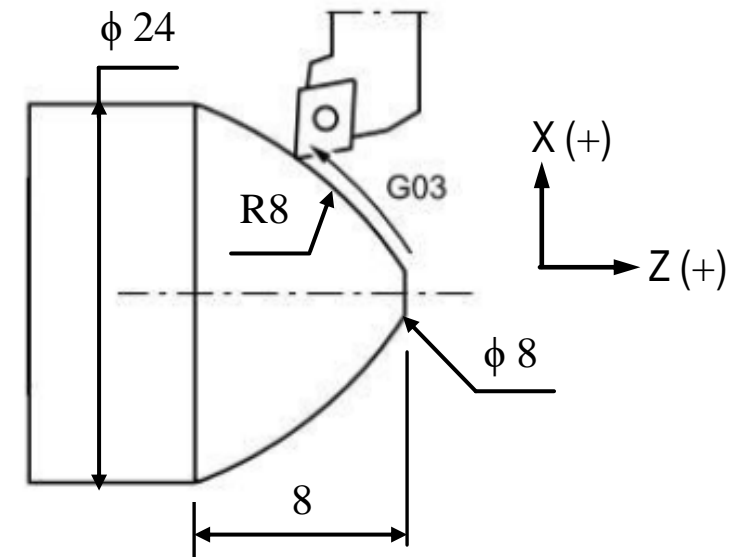
G03 X24 Z -8 R8 F60 / *Final position of tool*

G00 X 24 Z2

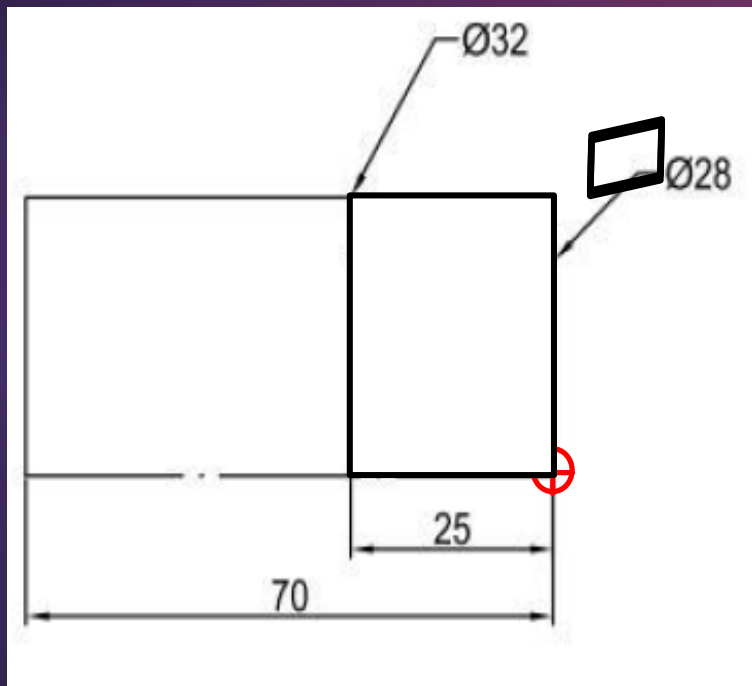
G28 U0 W0 / *Makes the tool to go to home position*

M05 / *Spindle stop*

M30 / *Program reset*



Practice



G21 G98 ----- Initial Settings

G28 U0 W0 ----- Going to home position

M06 T1 ----- Tool Change Position No. 01

M03 S1500 ----- Spindle clockwise with 1500 RPM

G00 X32 Z5 ----- Tool Moving to Tool Entry Point of X32 Z5 at Rapid Traverse

G01 X31 F80 ----- Giving First depth of cut of 0.5 mm at a feed rate of 80 mm / min

G01 Z-25----- Moving the tool towards Z-25 mm

G01 X32 ----- Retract the tool in X axis

G00 Z5 ----- Moving the tool to Z5 position

G01 X30 F80 ----- Giving Second depth of cut of 0.5 mm at a feed rate of 80 mm / min

G01 Z-25----- Moving the tool towards Z-25 mm

G01 X32 ----- Retract the tool in X axis

G00 Z5 ----- Moving the tool to Z5 position

G01 X29 F80 ----- Giving Third depth of cut of 0.5 mm at a feed rate of 80 mm / min

G01 Z-25----- Moving the tool towards Z-25 mm

G01 X32 ----- Retract the tool in X axis

G00 Z5 ----- Moving the tool to Z5 position

G01 X28 F80 ----- Giving Fourth depth of cut of 0.5 mm at a feed rate of 80 mm / min

G01 Z-25----- Moving the tool towards Z-25 mm

G01 X32 ----- Retract the tool in X axis

G00 Z5 ----- Moving the tool to Z5 position

G28 U0 W0 ----- Going to home position

M05 ----- Stop the spindle

M30 -----Program stop and rewind

G94 - Facing Cycle

Syntax:

G94 X_ Z_ F_

X= Diameter to which movement is being made

Z= Co-ordinate in Z-axis

F= Feed

Example:

G21 G98

G28 U0 W0

M06 T0101

M03 S2500

G00 X26 Z2

G94 X0 Z -0.5 F60 /

Final position of tool

Z-1

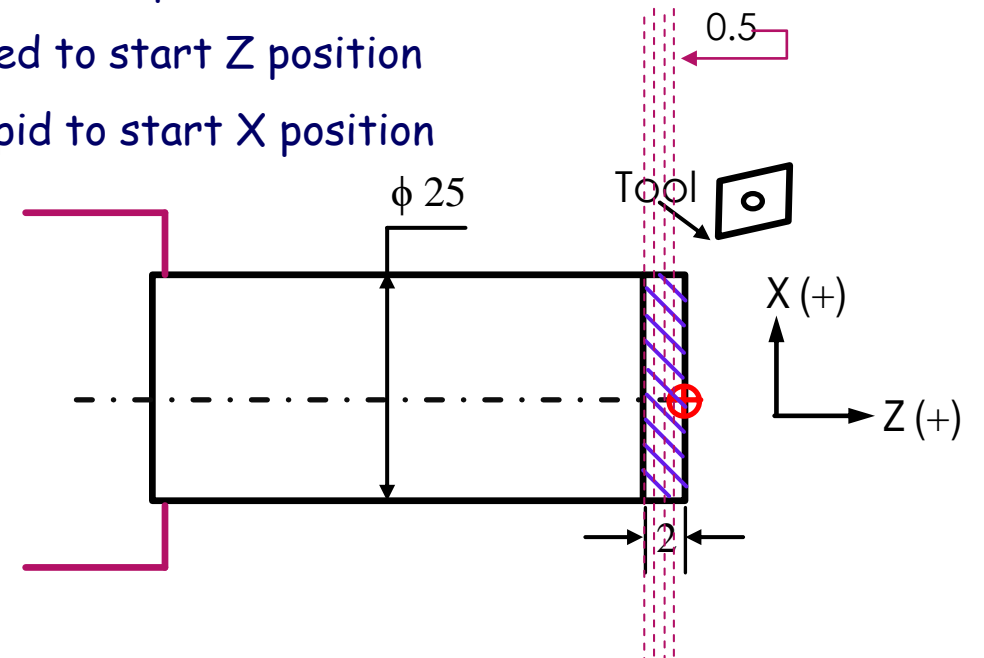
Z-1.5

Z-2

G28 U0 W0

➤ This cycle is used for stock removal in parallel tool path

1. Rapid to Z position
2. Feed to X position
3. Feed to start Z position
4. Rapid to start X position



G90 - Straight Turning Cycle

Syntax:

G90 X_ Z_ F_

X= Diameter to which movement is being made

Z= Co-ordinate in Z-axis

F= Feed rate

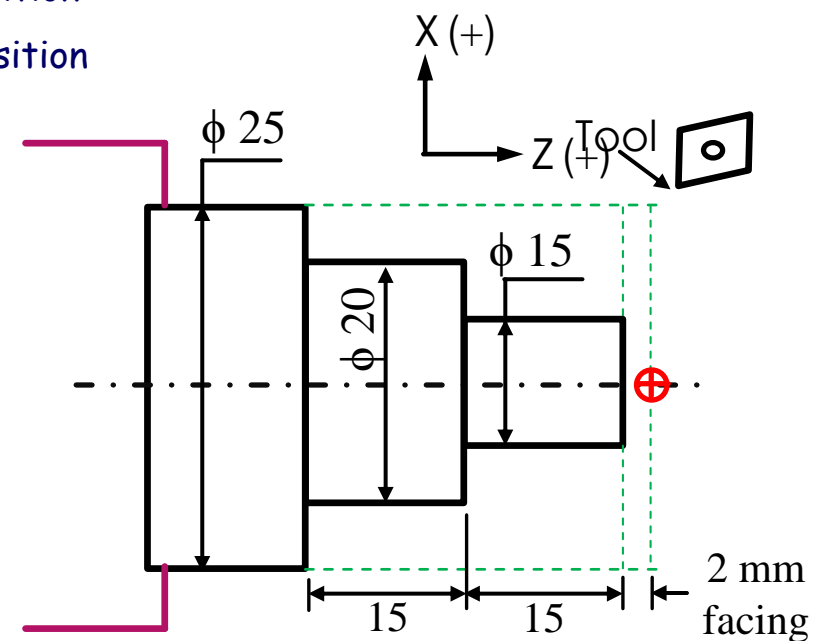
Example:

G21 G98
G28 U0 W0
M06 T0101
M03 S2500
G00 X26 Z2
G94 X0 Z -0.5 F60
Z-1
Z-1.5
Z-2
G00 X25 Z0

G90 X24 Z -32 F60
X23
X22
X21
X20
G90 X19 Z -17 F60
X18
X17
X16
X15
G28 U0 W0
M05
M30

➤ This cycle is used for stock removal in parallel tool path.

1. Rapid to x position
2. Feed to Z position
3. Feed to start X position
4. Rapid to start Z position



G71 - Multiple Turning Cycle

Syntax:

G71 U_ R_

G71 P_ Q_ U_ W_ F_

U= depth of each cut (**First one**)

R= Tool retract

P= Start block of the profile

Q= Finishing block of the profile

U= Finishing allowance in X axis (**Second one**)

W= Finishing allowance in Z axis

F= Feed rate

➤ This cycle is used when major direction of cut along the "Z" axis

G70 –Finishing Cycle

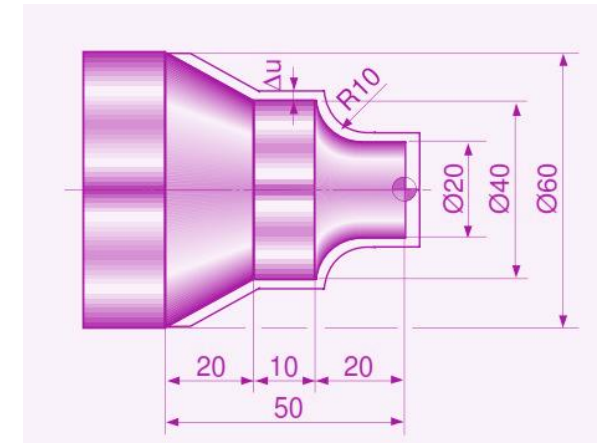
Syntax:

G70 P_ Q_ F_

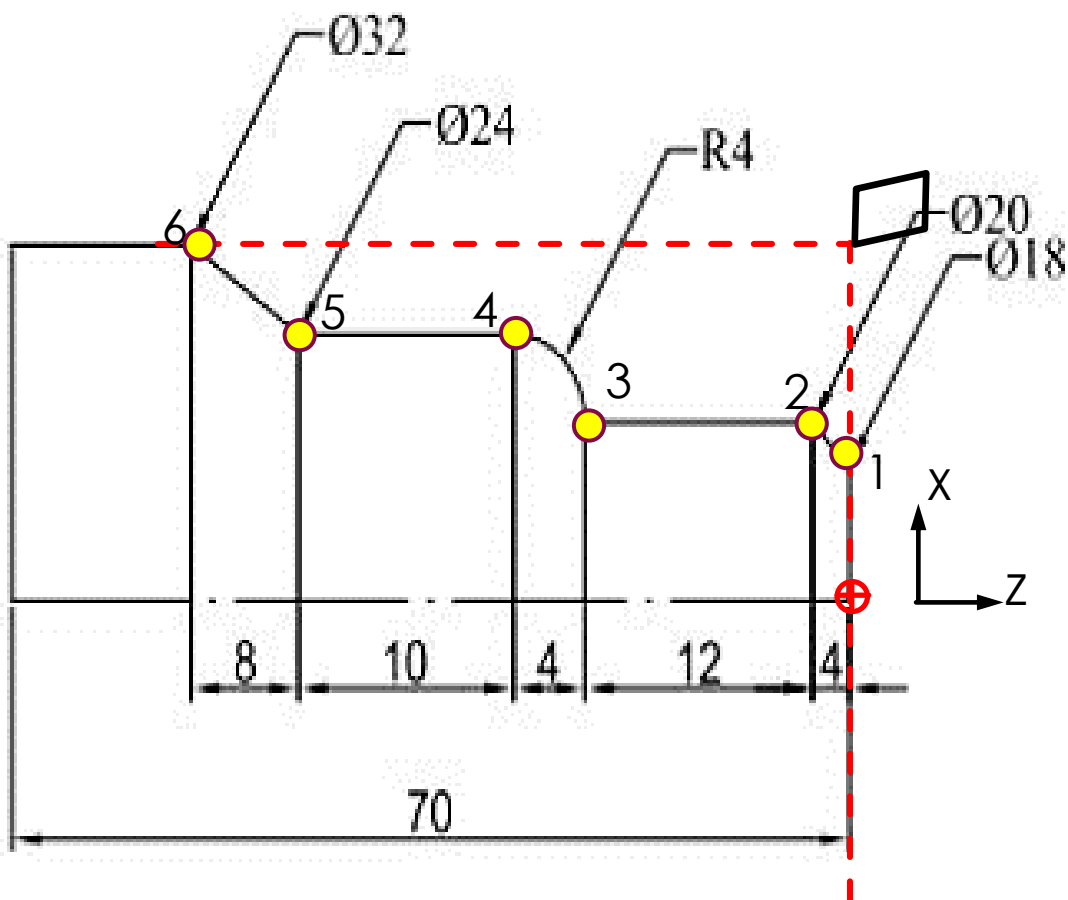
P= Start block of the profile

Q= Finishing block of the profile

F= Feed rate



G71 - Multiple Turning Cycle



```

G21 G98
G28 U0 W0
M06 T0101
M03 S1500 /Spindle clockwise with 1500 RPM
G00 X32 Z5 /Tool Moving to Tool Entry Point of X32 Z5 at Rapid Traverse
G71 U0.5 R1 /Calling G71 Cycle and defining Cycle Parameters
G71 P1 Q2 U0.1 W0.1 F100 /Defining Cycle Parameters
N1 G01 X18 F100
G01 Z0
G01 X20 Z-4
G01 Z-16
G03 X24 Z-20 R4
G01 Z-30
N2 G01 X32 Z-38
G70 P1 Q2 F80 /Finishing cycle
G28 U0 W0
M05
M30
  
```

Syntax:

G71 U_R_

G71 P_Q_U_W_F_

U= depth of each cut (First one)

R= Tool retract

P= Start block of the profile

Q= Finishing block of the profile

U= Finishing allowance in X axis (Second one)

W= Finishing allowance in Z axis

F= f
Syntax:

G70 P_Q_F_

P= Start block of the profile

Q= Finishing block of the profile

F= Feed rate

G74 - Multiple Drilling Cycle

- This cycle is designed for deep hole drilling
- 1. The drill enters the w/p into a predetermined amount
- 2. Then backing off another set amount to remove the chips

G74 R_

G74 X_ Z_ Q_ F

R - Return Amount, mm

X - Always Zero, mm

Z - Drilling Depth, mm

Q - Depth of Cut in Z axis (in Micron)

F - Feed Rate, mm /min.

Example:

G21 G98

G28 U0 W0

M06 T0202 / Call tool no. 2

M03 S1000 / Rotate tool at a speed of 1000 rpm

G00 X0 Z5 / Tool Moving to Tool Entry Point of X0 Z5 at Rapid Traverse

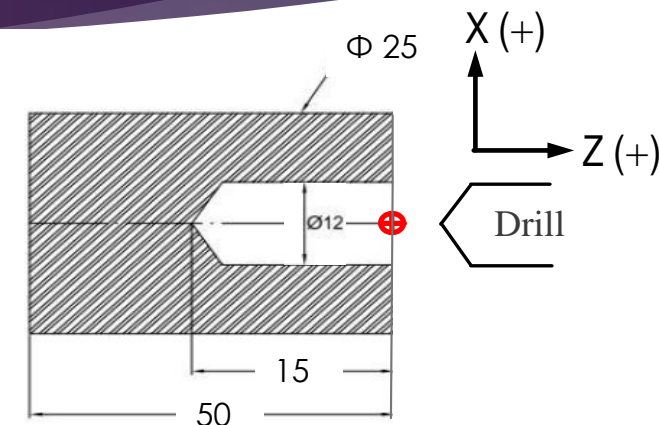
G74 R1 / Calling G74 Cycle and defining parameters

G74 X0 Z-15 Q500 F100 / make drill upto 12 mm with 500 µm doc

G28 U0 W0

M05

M30



G76 - Multiple Threading Cycle

G76 P(m) (r) (a) Q_ R_

G76 X_ Z_ P_ Q_ F_

m= no of passes for finishing operation

r= tool relief angle

a= thread angle, degree

Q= minimum cutting depth (μm in software)
(mm in machine)

R= finishing allowance (in mm)

X= core diameter (in mm)

Z= Thread length, mm

Q= depth of cut for first pass (μm in software/ mm in m/c)

P=thread height (μm in software/ mm in machine)

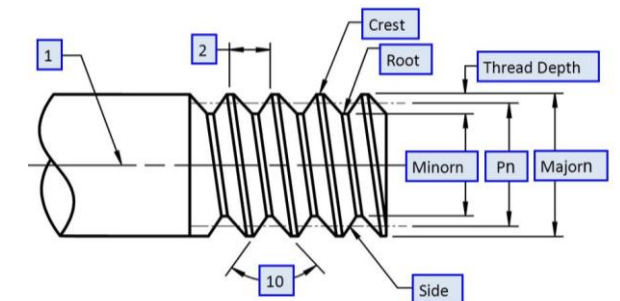
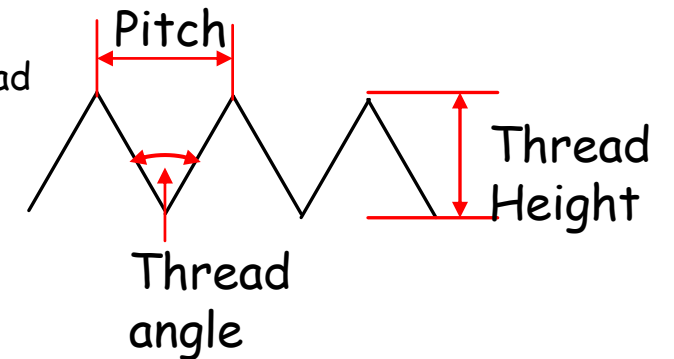
F= Pitch

D= Major Diameter

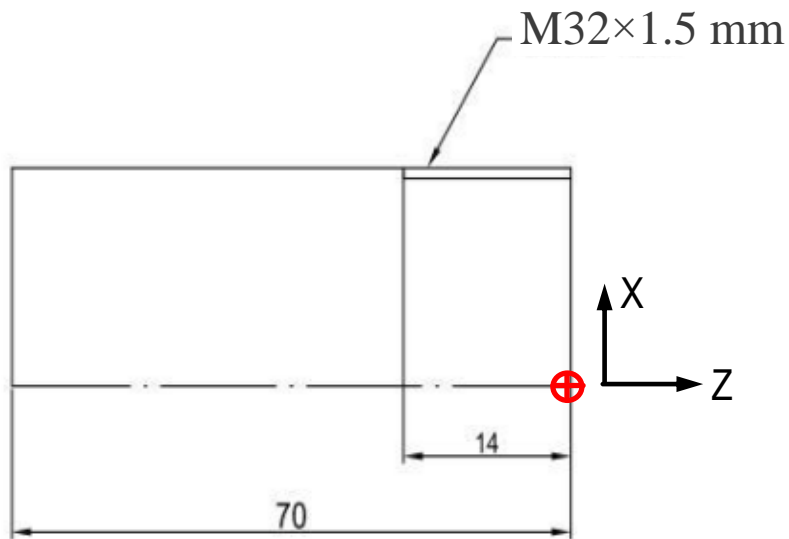
P= Thread Height

Thread Height, P = $0.613 \times \text{Pitch of the Thread}$

Core dia= $D - 2(P)$



G76 - Multiple Threading Cycle



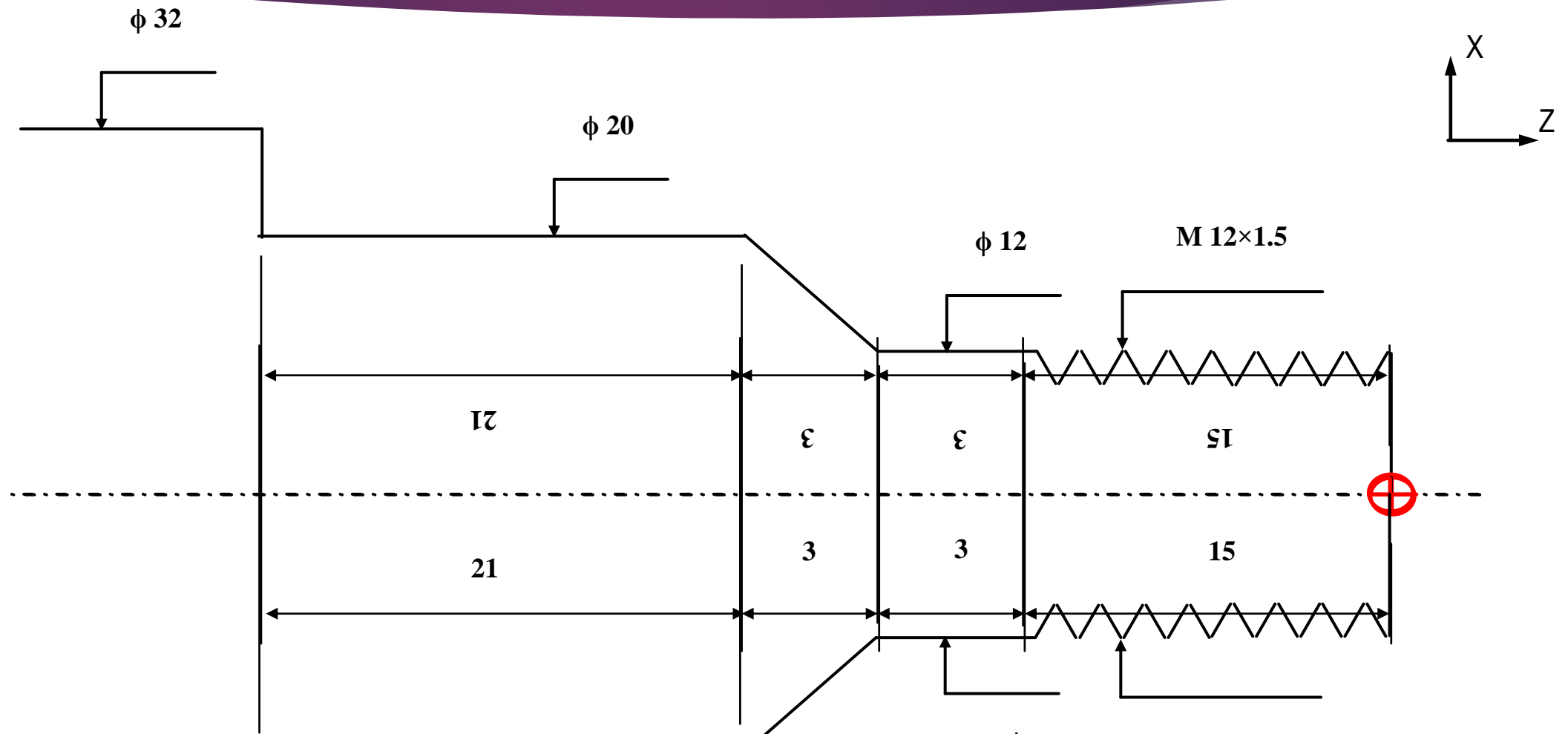
Thread Height, $P = 0.613 \times \text{Pitch of the Thread}$
 $= 0.613 \times 1.5$
 $P = 0.919 \text{ mm} = 919 \mu\text{m}$

Core diameter = Major dia - 2 (P)
 $= 32 - 2 (0.919)$
 $= 32 - 1.818$
 $= 30.162 \text{ mm}$

G21 G98 ----- Initial Settings
 G28 U0 W0 ----- Going to home position
 M06 T0101 ----- Tool Change Position No. 01
 M03 S1500 ----- Spindle clockwise with 1500 RPM
 G00 X32.5 Z5
G76 P040060 Q50 R0.04 ----- Calling G76 cycle
G76 X30.162 Z-14 P919 Q100 F1.5
 G28 U0 W0
 M05
 M30

Practice

29



O0001

G21 G98

G28 U0 W0

M06 T0101

M03 S1500 / *Spindle clockwise with 1500 RPM*

G00 X32 Z2 / *Tool Moving to Tool Entry Point of X32 Z5 at Rapid Traverse*

G71 U0.5 R1 / *Calling G71 Cycle and defining Cycle Parameters*

G71 P1 Q2 U0.1 W0.1 F100 / *Defining Cycle Parameters*

N1 G01 X12 Z0 F100

G01 X12 Z-18

G01 X20 Z-21

G01 X20 Z-42

N2 G01 X 32 Z-42

G70 P1 Q2 F100 / *Finishing cycle*

G28 U0 W0

M06 T0303 / *Calling threading tool*

M03 S500 / *Spindle clockwise rotation*

G00 X13 Z0

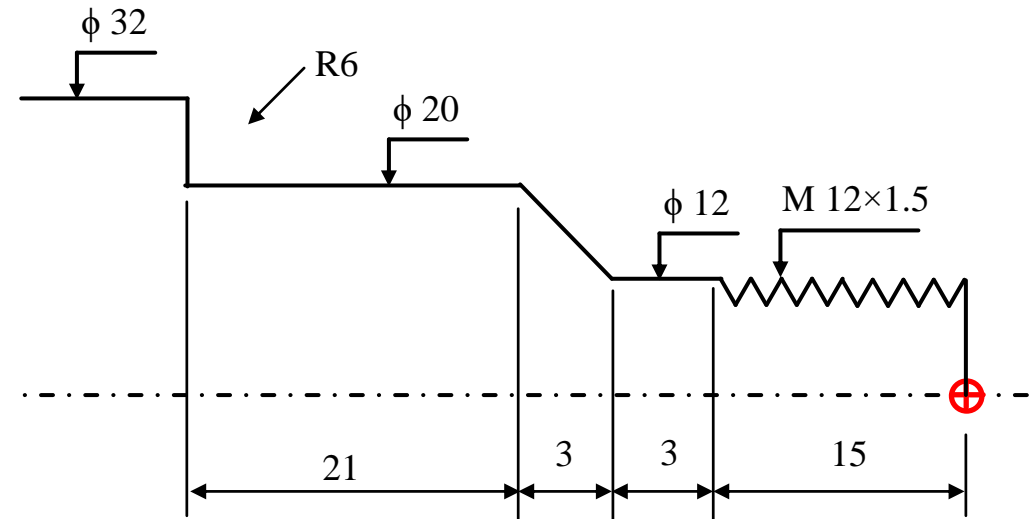
G76 P040060 Q50 R0.4 / *Call & execute threading cycle*

G76 X10.162 Z-15 Q100 P919 F1.5

G28 U0 W0 / *Tool home position*

M05

M30

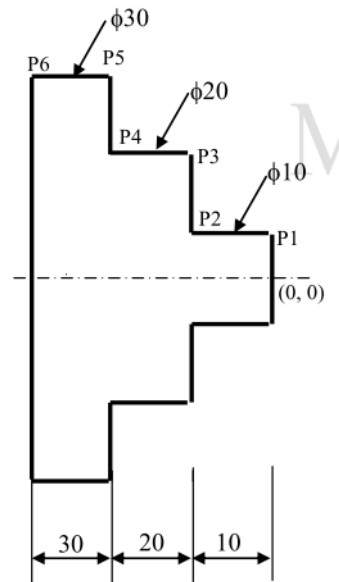


Different Measurement System

31

Absolute System

- Absolute dimension system always refers to a **fixed reference point** in the drawing.
- This point has the function of a coordinate zero point.
- Define by **X and Z**

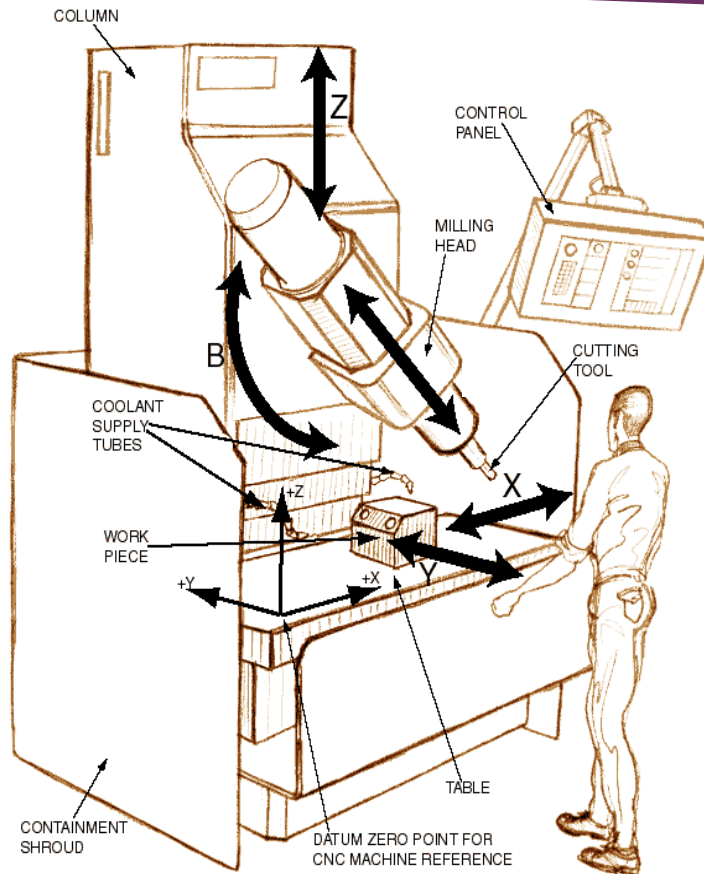


Incremental System

- Every measurement is considered from **previously dimensioned position**.
- Incremental dimensions are distance between adjacent points.
- Defined by **U and W**

ABSOLUTE DIMENSIONING			INCREMENTAL DIMENSIONING		
POINTS	X	Z	POINTS	U	W
P1					
P2					
P3					
P4					
P5					
P6					

Degree of Freedom in Milling

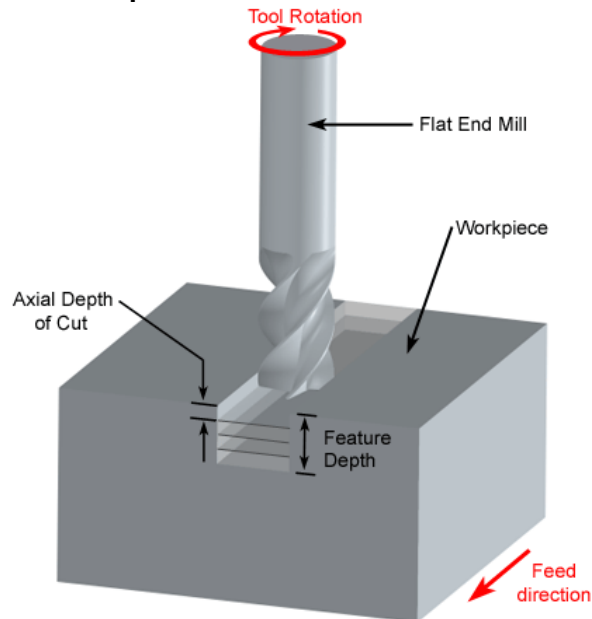


End milling tool

Various Operations That Can Be Performed in Milling

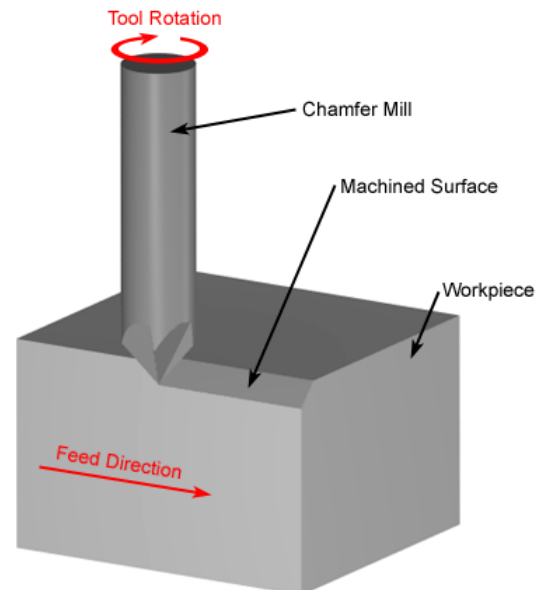
End Milling

- A end mill makes axial cuts across the workpiece
- Machine a feature, such as a profile, slot, pocket, or even a complex surface contour



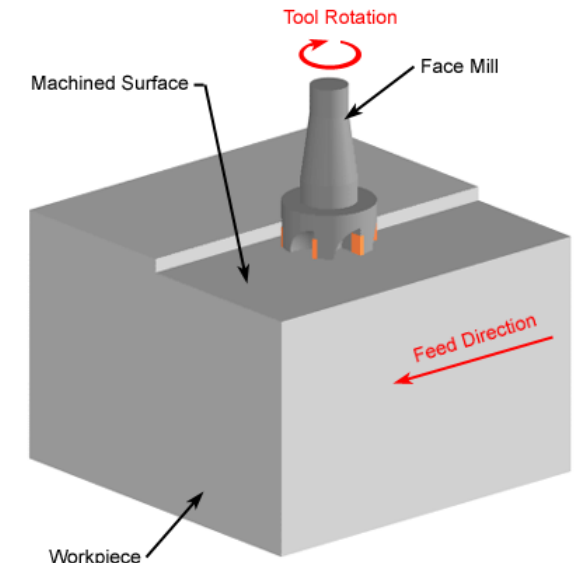
Chamfer Milling

- A chamfer end mill makes a cut along an edge of the workpiece
- Create an angled surface, known as a chamfer.

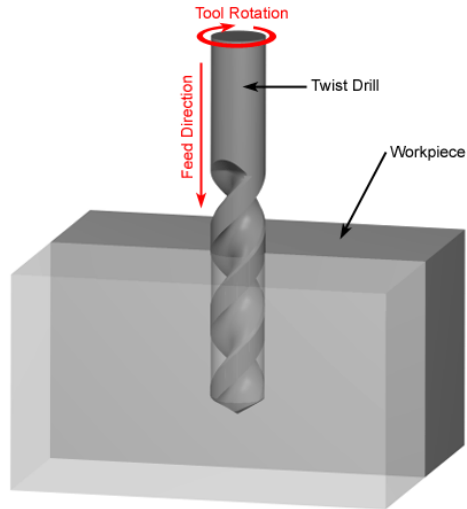


Face Milling

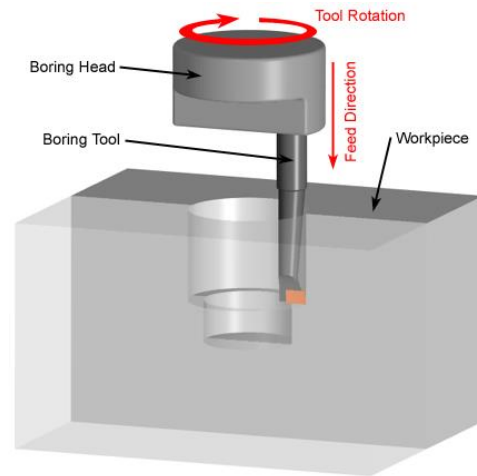
- A face mill machines a flat surface of the workpiece in order to provide a smooth finish.



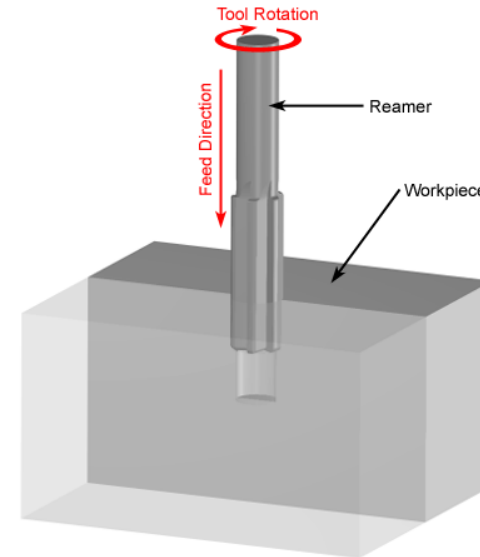
Various Operations That Can Be Performed in Milling



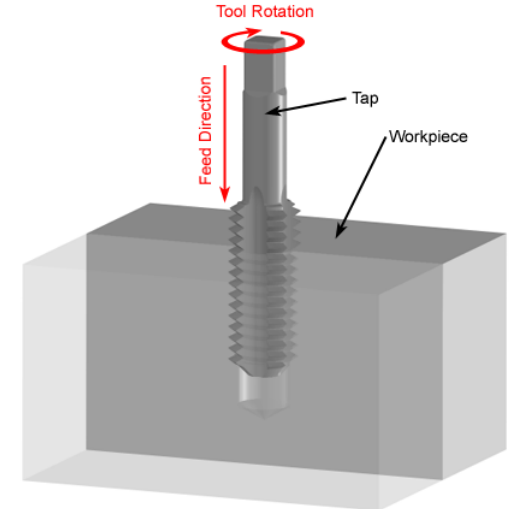
Drilling



Boring

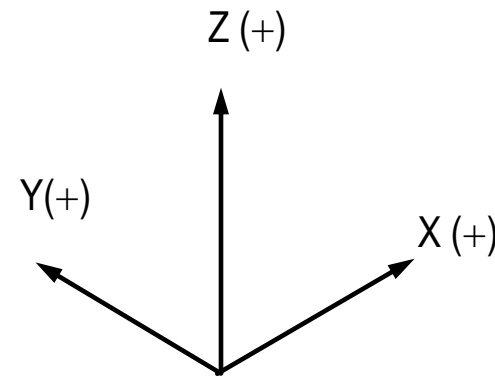
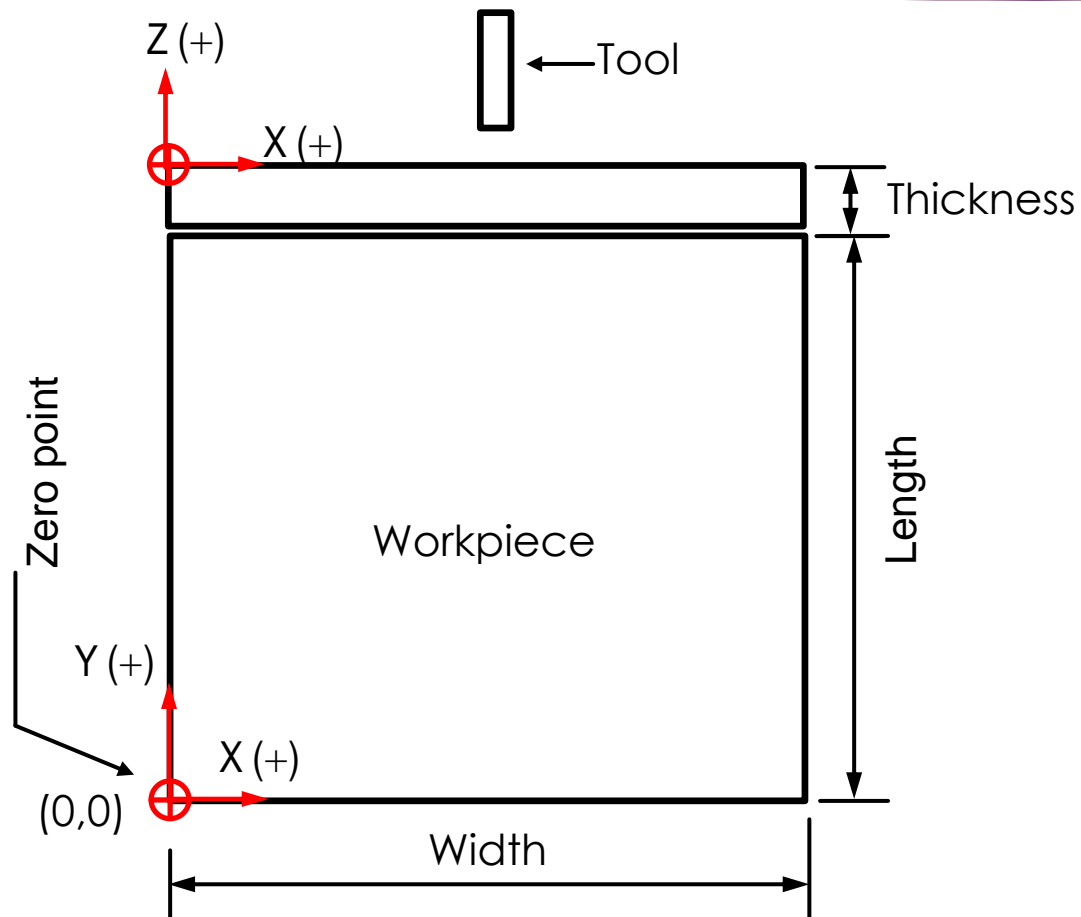


Reaming



Tapping

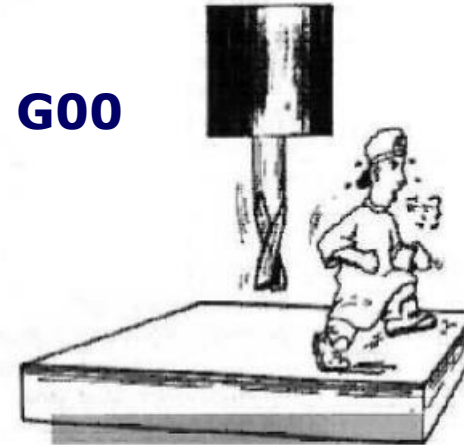
Workpiece zero points or Program zero point



CNC Milling - G Code

- **G90 – Absolute method**
- **G91 – Incremental method**
- **G94 – Feed, mm/min**
- G95 – Feed, mm/rev
- G54 to G59 – Work coordinate system
- **G43 – Height offset in downward direction**
- **H1 to H6 – Height offset for tool**
- X – Absolute mode in X- axis
- Y – Absolute mode in Y- axis
- Z - Absolute mode in Z- axis

G00



G00 X_ Y_ Z_

G02 X_ Y_ Z_ F_

G01



G01 X_ Y_ Z_ F_

G03 X_ Y_ Z_ F_

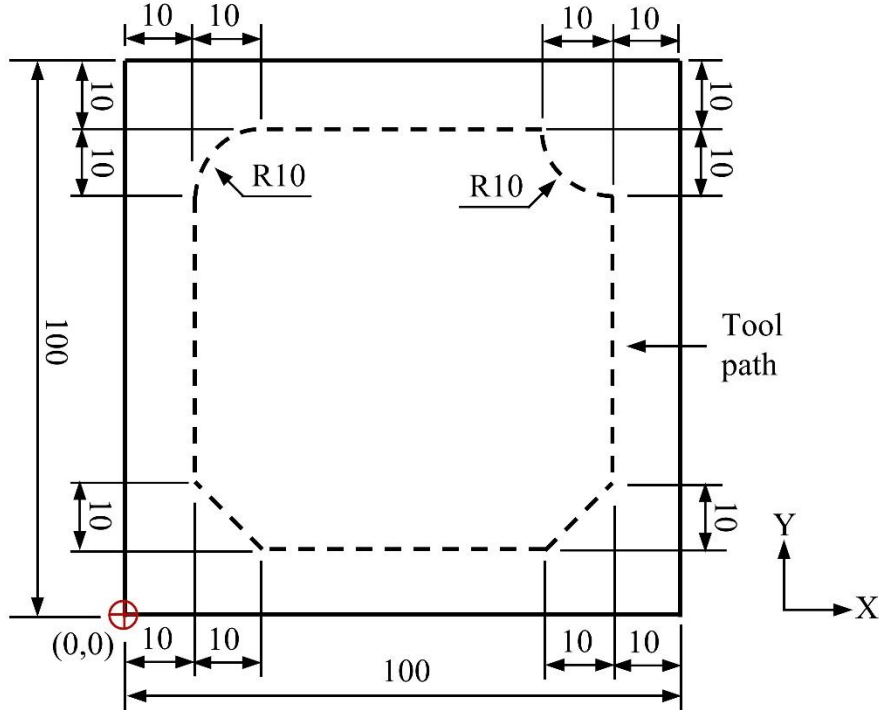
Comparison

CNC lathe

- G00 – Rapid transverse (or, Rapid movement)
- G01 – Linear motion with feed
- G02 – Tool movement in clock wise direction
- G03 – Tool movement in anti-clock wise direction
- G04 – Dwell time (or, waiting time)
- G17 – XY plane
- G20 – Inches mode
- G21 – Metric mode (in mm)
- G28 – Go to machine home position in incremental mode
- G98 – Feed in mm/min
- G99 – Feed in rev/min
- U – Incremental mode in X- axis
- W – Incremental mode in Z- axis
- X – Absolute mode in X- axis
- Z – Absolute mode in Z- axis

CNC Milling

- **G90 – Absolute method**
- **G91 – Incremental method**
- **G94 – Feed, mm/min**
- **G95 – Feed, mm/rev**
- G54 to G59 – Work coordinate system
- **G43 – Height offset in downward direction**
- **H1 to H6 – Height offset for tool**
- X – Absolute mode in X- axis
- Y – Absolute mode in Y- axis
- Z – Absolute mode in Z- axis



Thickness of workpiece: 12 mm, Depth of cut: 1 mm,
Feed rate: 100 mm/min, Speed: 2500, Dia of end mill: 5 mm

G21 G94

G91 G28 X0 Y0 Z0 /tool go to home position

M06 T0101 / Call tool no. 1

M03 S2500 / tool rotates at a speed of 2500 rpm

G00 G90 G54 X0 Y0 /

G00 G43 H1 Z10

G00 X20 Y10

G01 Z-2 F100

G01 X10 Y20

G01 Y80

G02 X20 Y90 R10

G01 X80

G03 X90 Y80 R10

G01 Y20

G01 X80 Y10

G01 X20

G00 Z5

G91 G28 X0 Y0 Z0

M05

M30

Example:

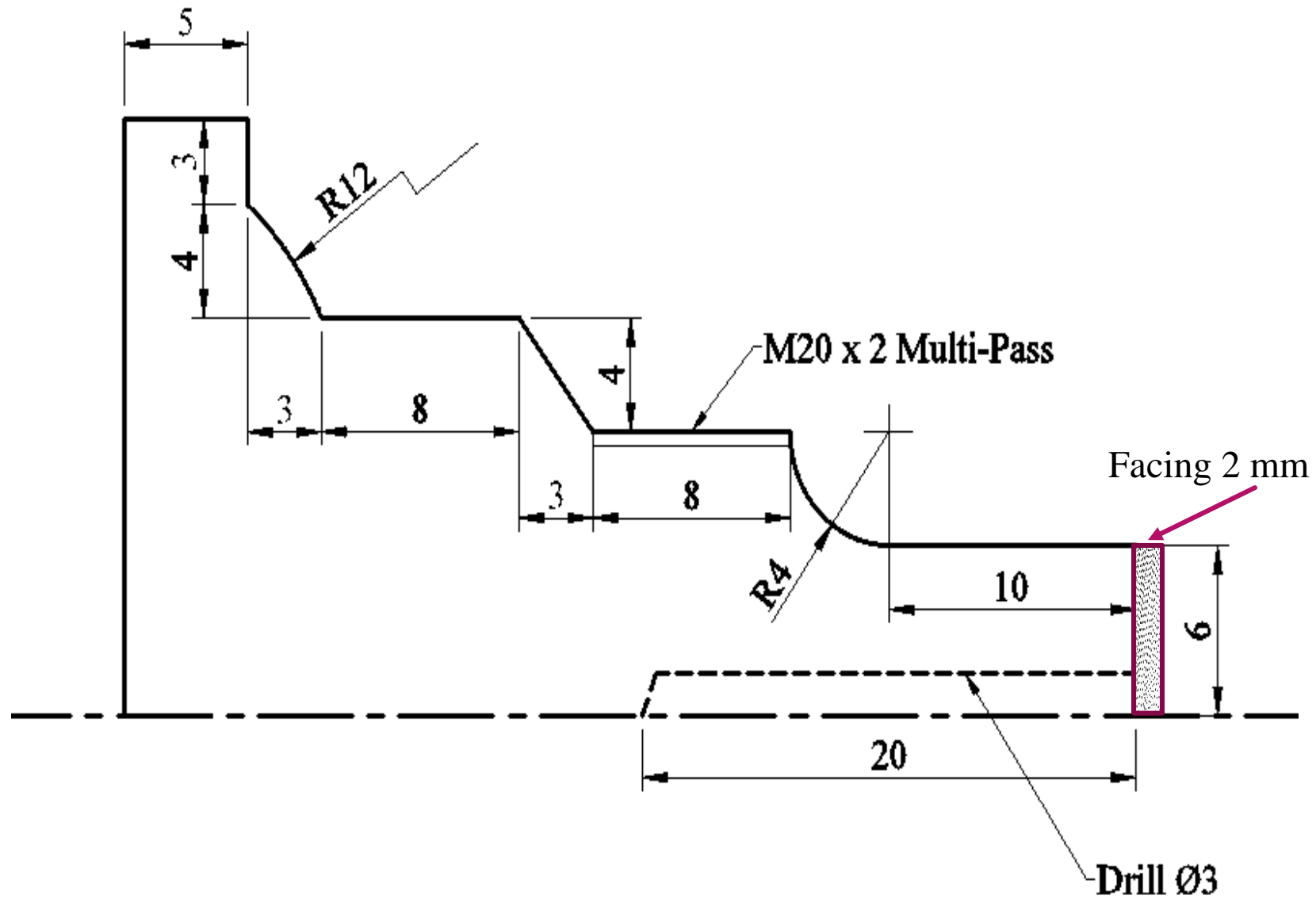
G21 G98

G28 U0 W0 / Go tool to home position

M06 T0101 / Call tool no. 1

M03 S2500 / Rotate tool at a speed of 2500 rpm

G00 X25 Z2 / The position of tool



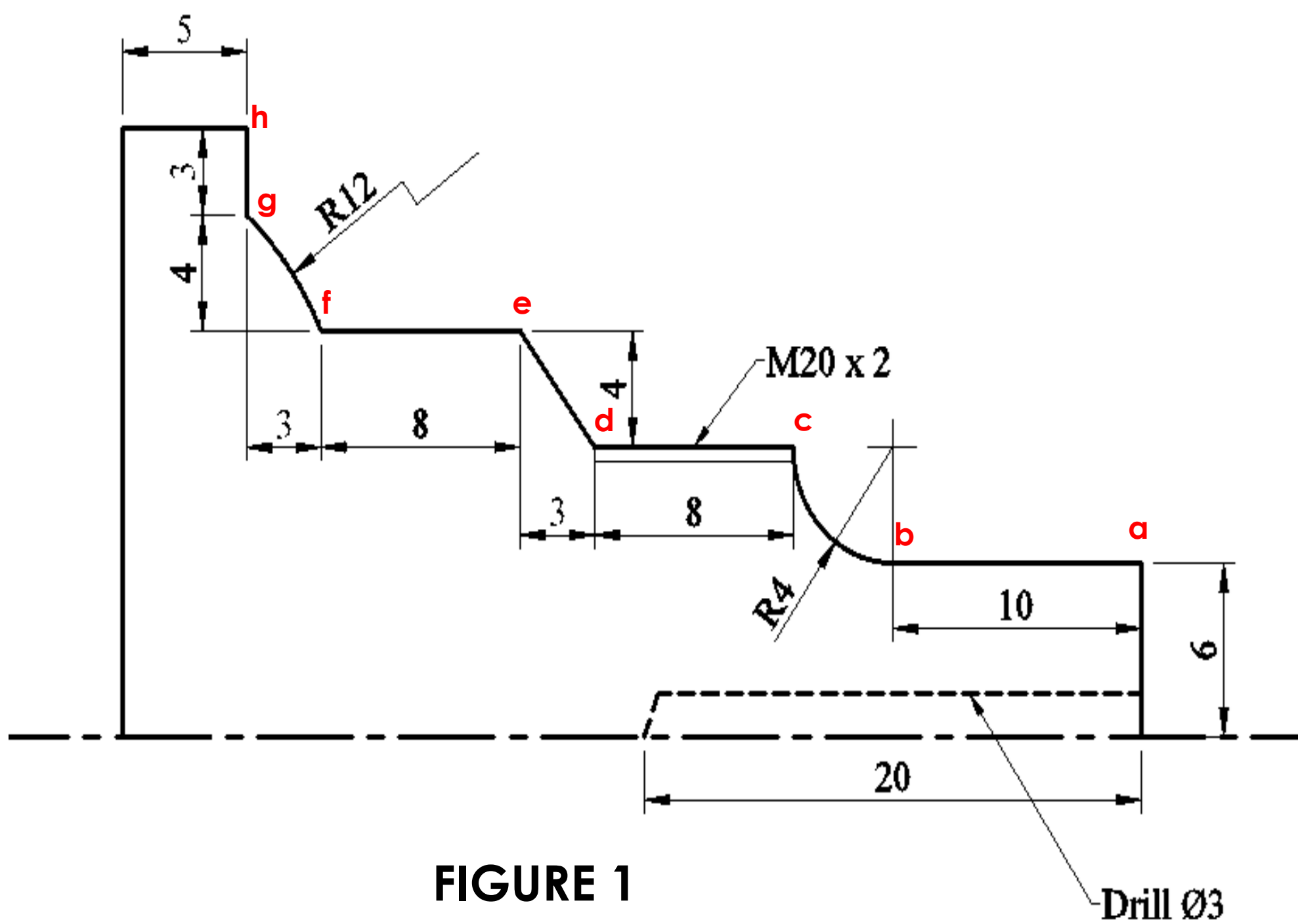
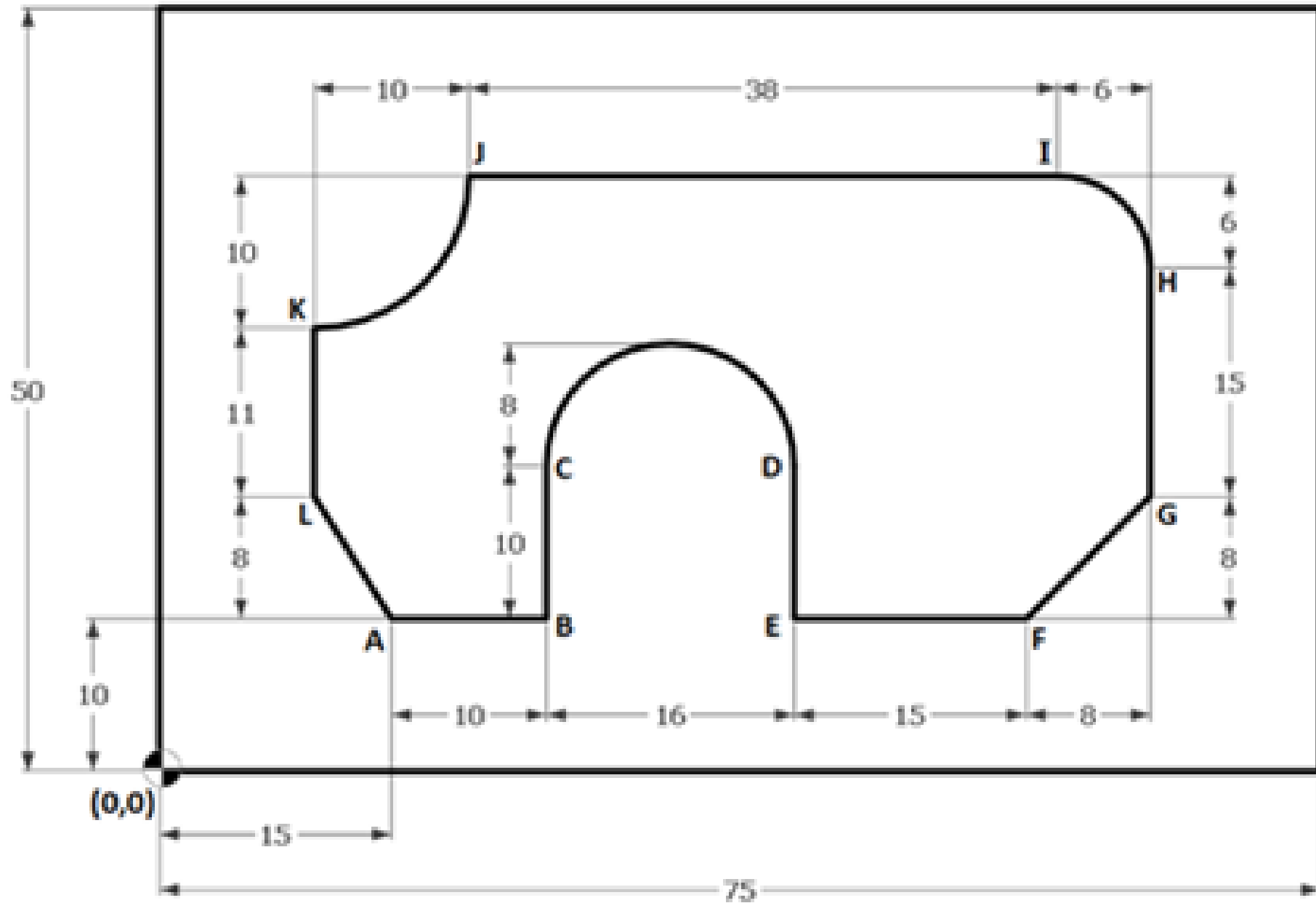


FIGURE 1

**FIGURE 2**