Manufacturing Processes
UTA026

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

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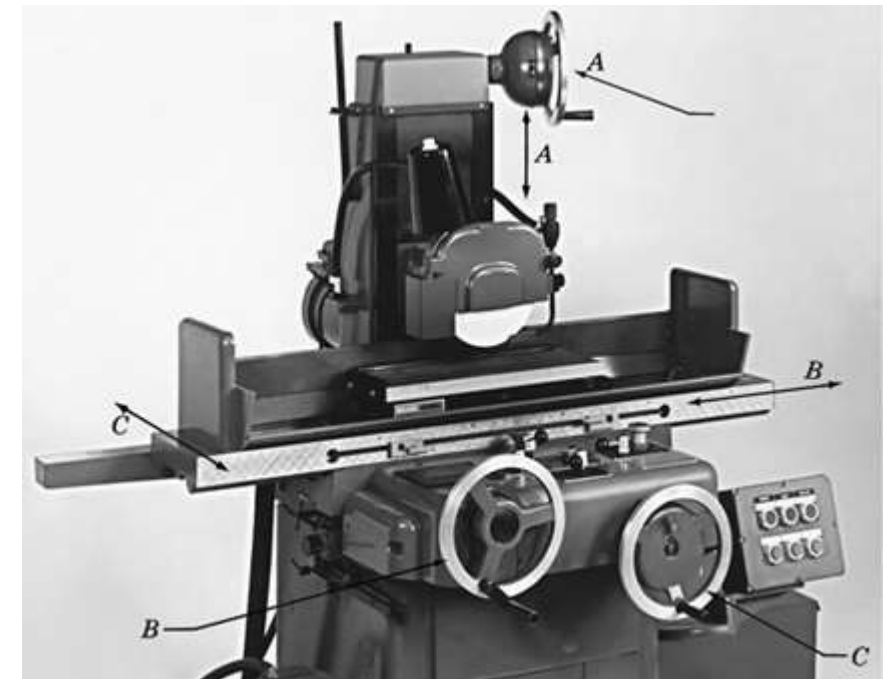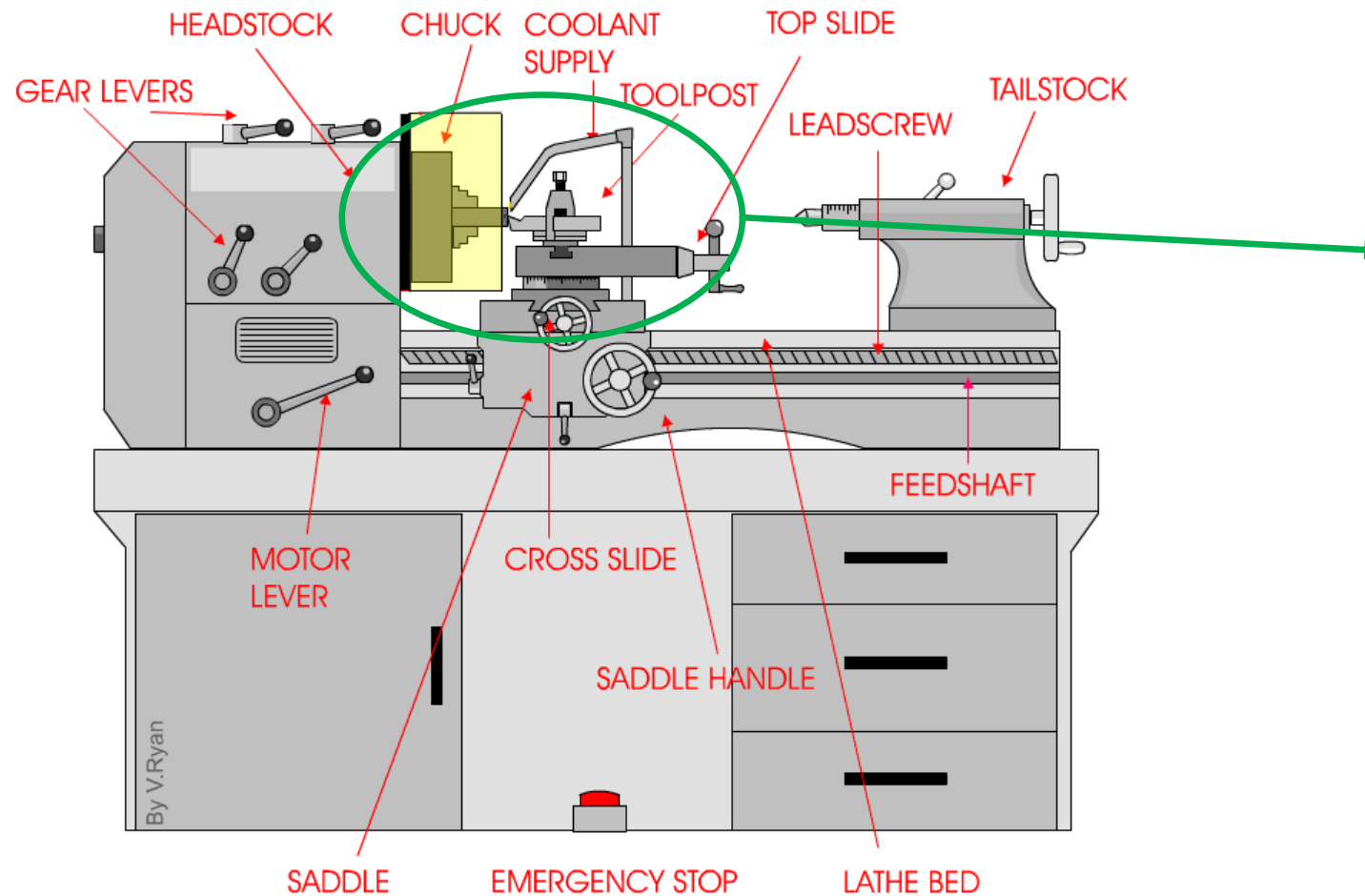(Deemed to be University)

Machine Tool Use for Machining

Lathe

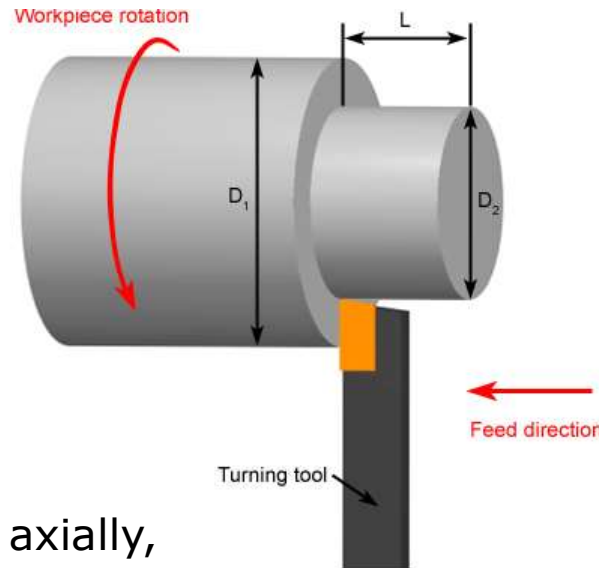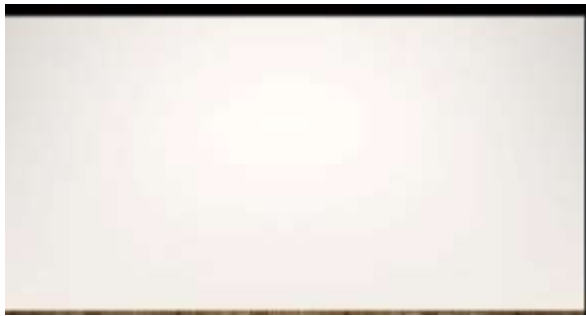Milling Machine

Machine Tool Use for Machining

Shaping Machine

Drilling Machine

Grinding Machine

LATHE

Components of Lathe

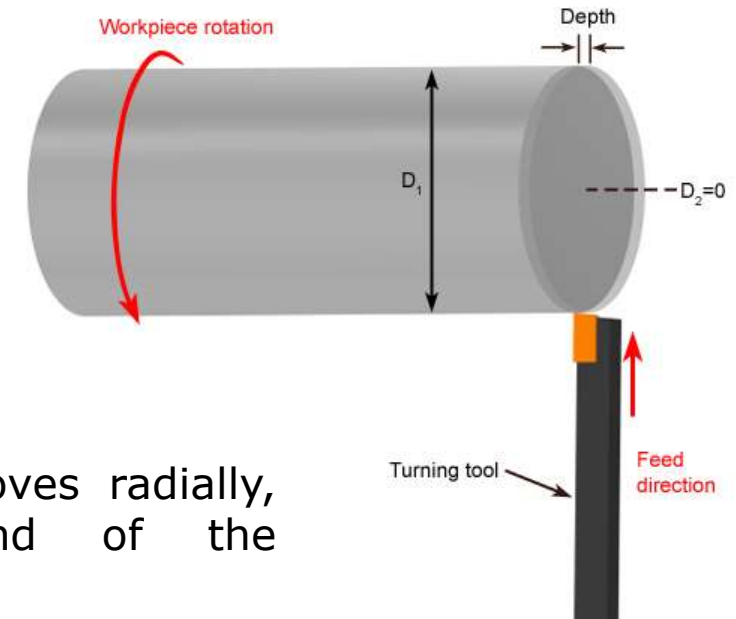# Various Operations That Can Be Performed in Lathe – External

Turning

➢ Turning tool moves axially, along the side of the workpiece

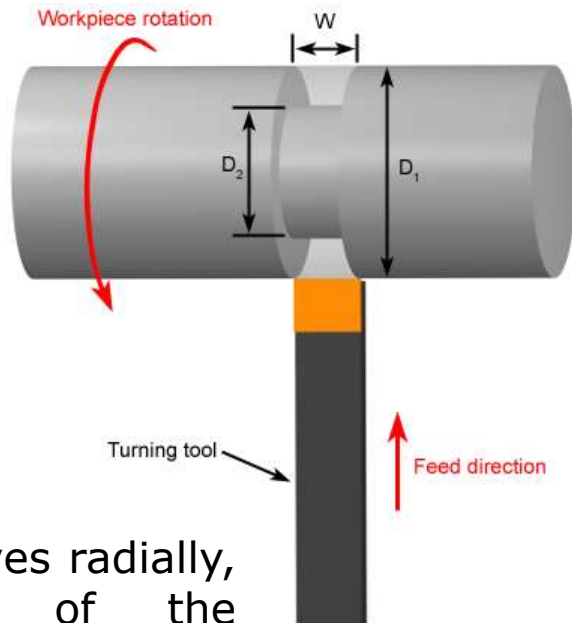➢ removing material to form steps, tapers, chamfers, and contours.

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

Workpiece rotation
W
D_2
D_1
Turning tool
Feed direction

Workpiece rotation
D_1
$D_2 = 0$
Turning tool
Feed direction

Grooving

➢ Grooving tool moves radially, into the side of the workpiece,

➢ cutting a groove equal in width to the cutting tool.
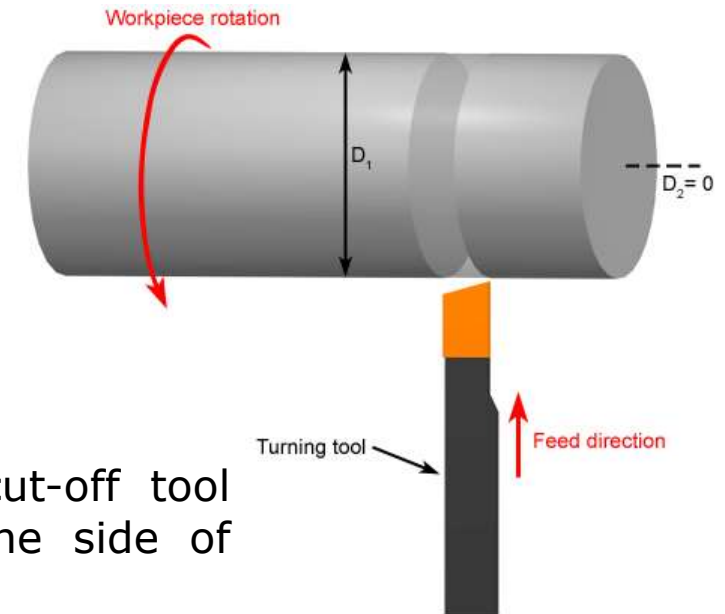
Cut-off (parting) –

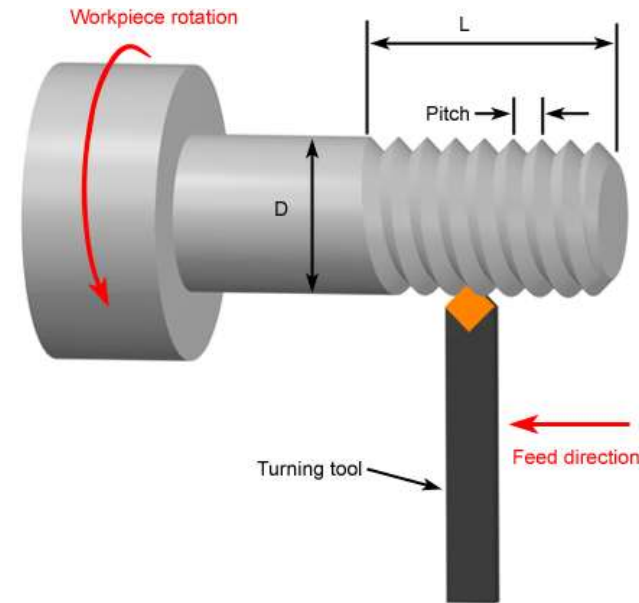➢ Similar to grooving, cut-off tool moves radially, into the side of the workpiece,

➢ continues until the centre or inner diameter of the workpiece is reached.

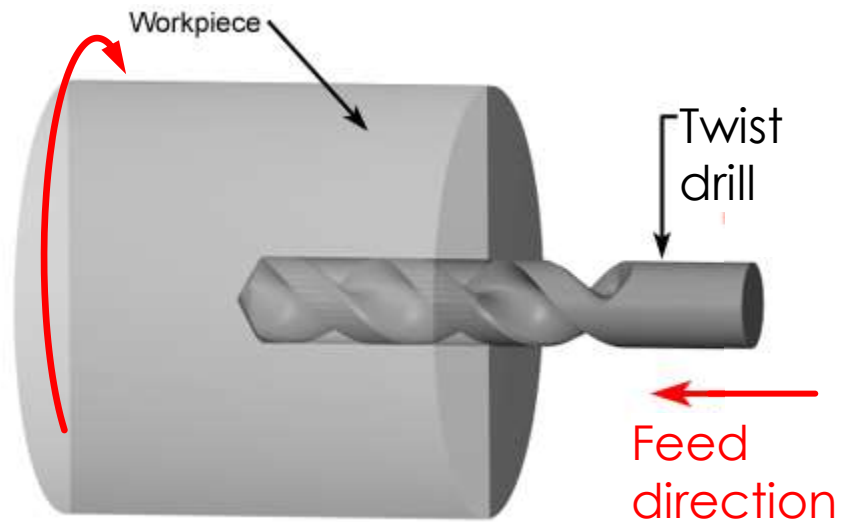# Various Operations That Can Be Performed in Lathe – External

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

Workpiece

Twist drill

Feed direction

Drilling

➤ A drill enters the workpiece axially through the end,

➤ Cuts a hole with a diameter equal to that of the tool.

Workpiece

Reaming tool

Feed direction
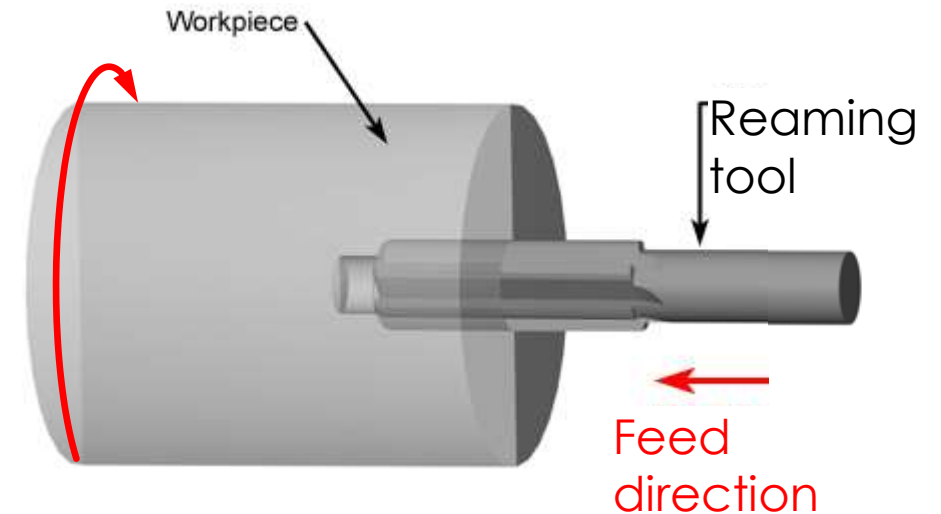
Reaming

➤ 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

Workpiece

Boring tool

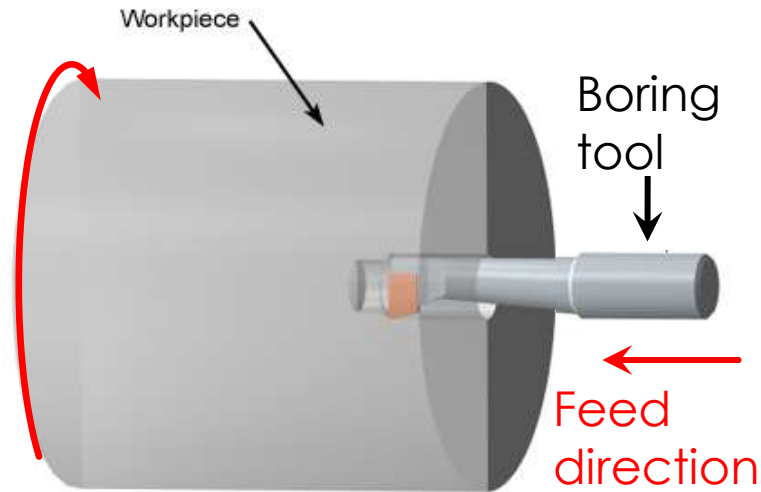Feed direction

Workpiece

Tapping tool

Feed direction

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.
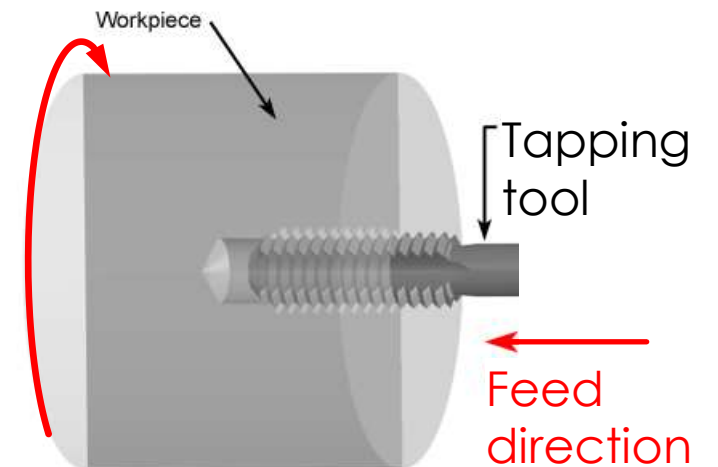
Computer Numerical Control (CNC) Machines

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

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

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 with in the workpiece

- The workpiece zero point should be placed along the spindle axis (centre line), in line with the finished contour.

Chuck   Workpiece

Zero point

$X(+)$

$(0,0)$ $Z(+)$

Length of Workpiece
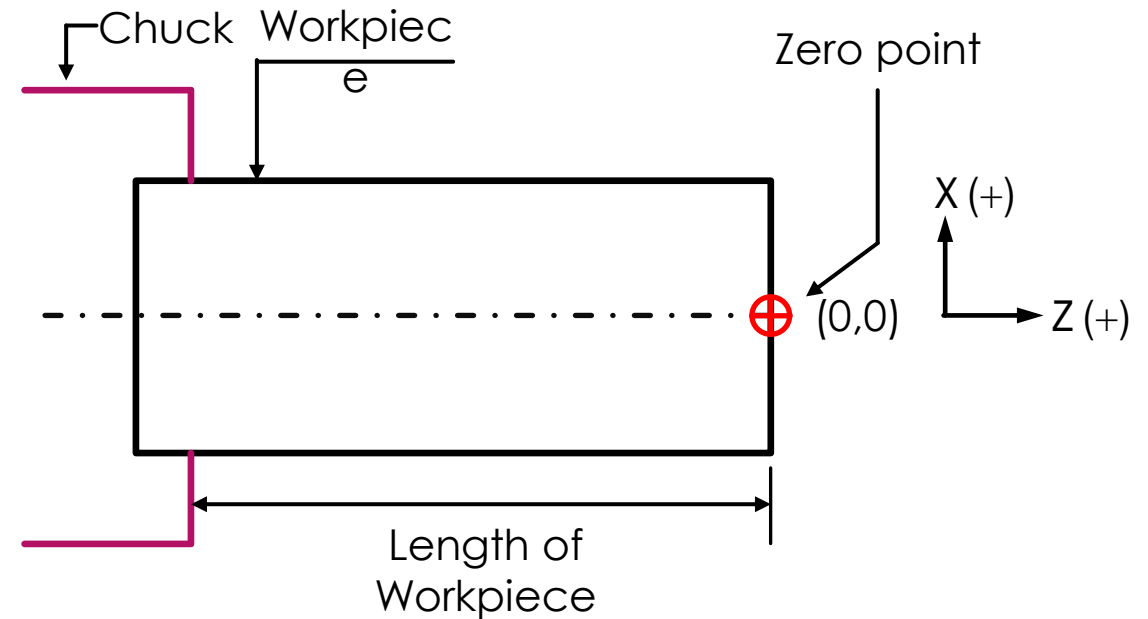
G00 – Rapid transverse (or, Rapid movement)

Tool

Syntax:

G00 X_ Z_

X= Co-ordinate in X-axis

Z= Co-ordinate in Z-axis

Chuck Workpiece

Dia of w/p

X (+)

Z (+)

(0,0)

- 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
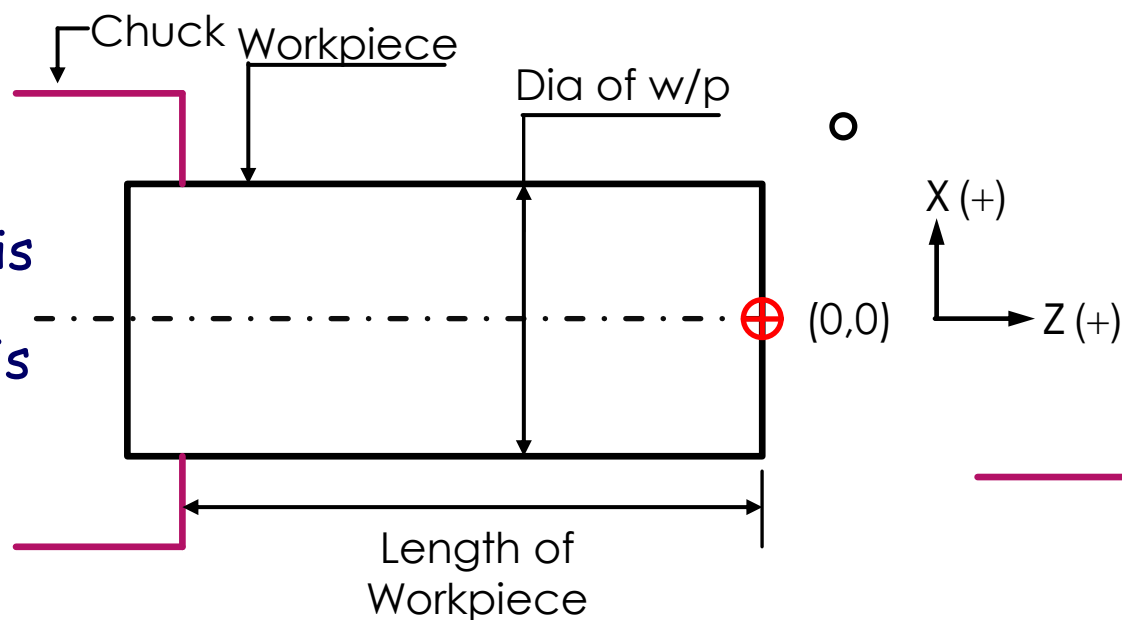
Length of Workpiece

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

φ 25

Tool

2

(0,0)

Position of tool

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:

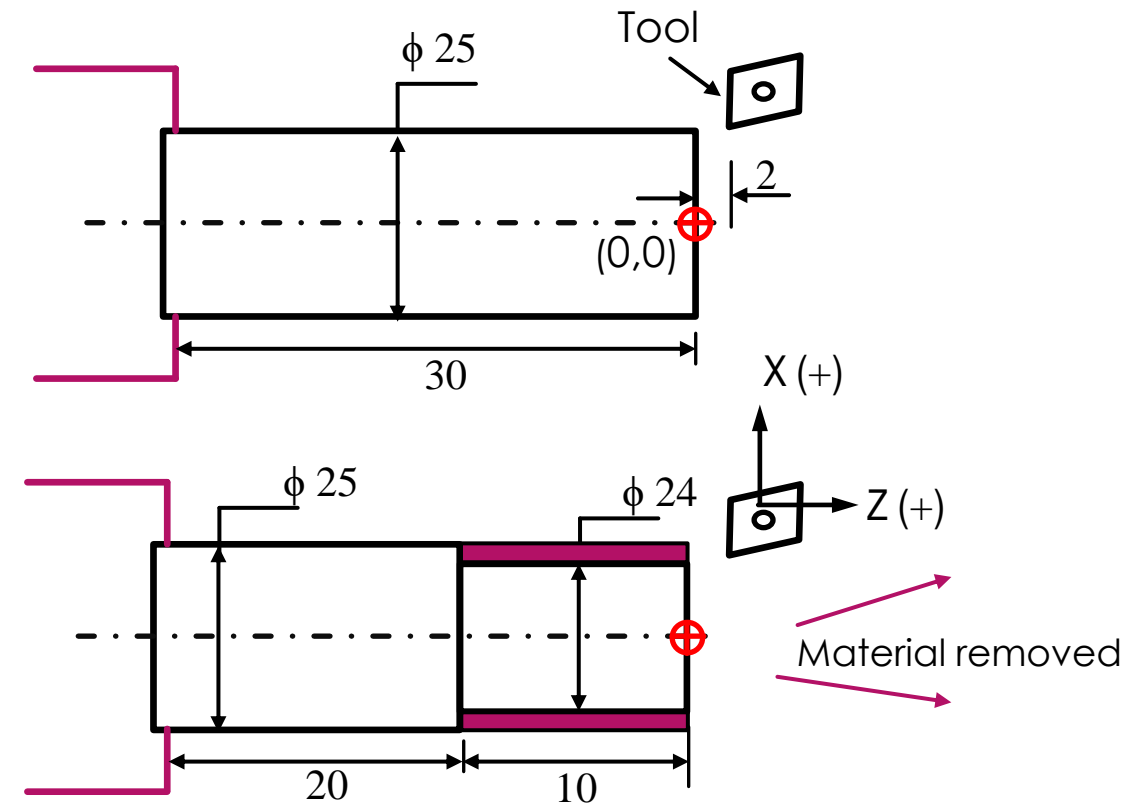G21 G98

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*

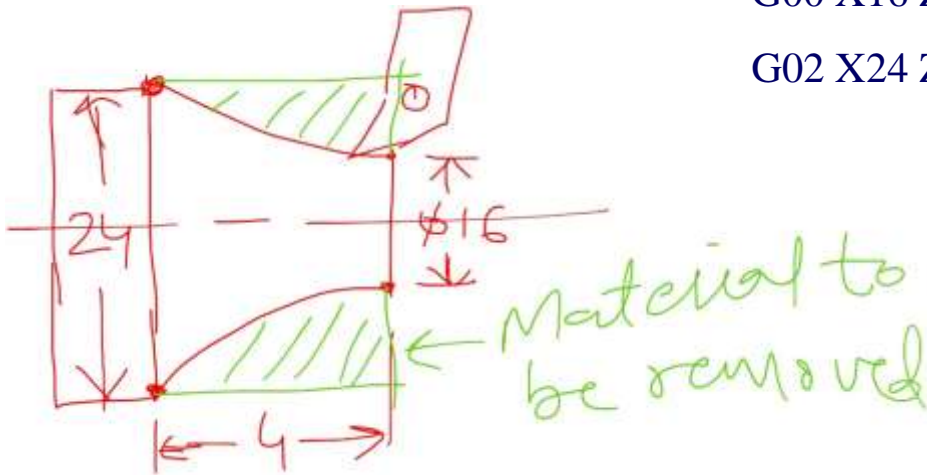# G02 – Tool movement in clock wise direction

Syntax:

G02 X_ Z_ R_ F_

X= Co-ordinate in X-axis

Z= Co-ordinate in Z-axis

R= Radius of curve
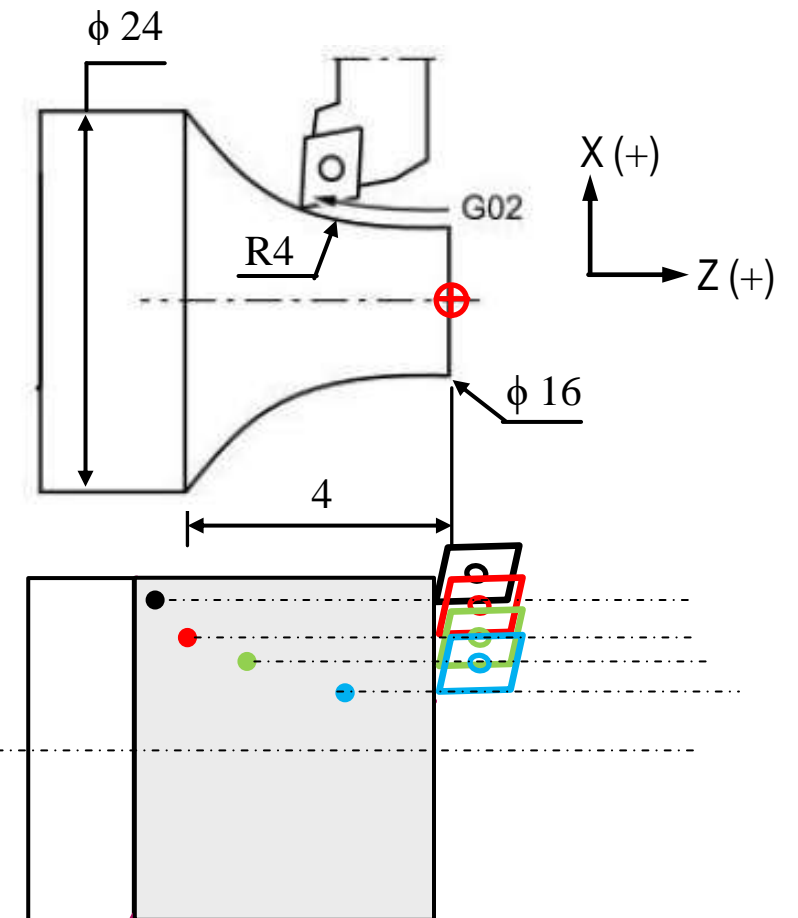
Example:

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

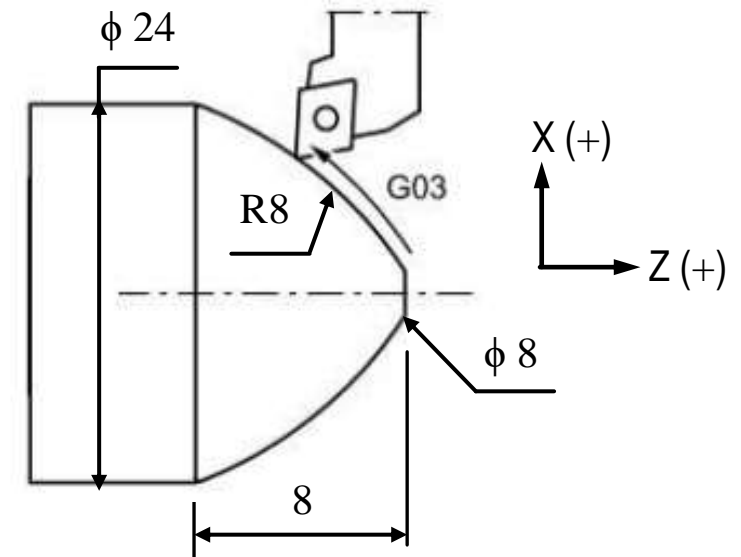G00 X8 Z0 / First position of tool

G03 X24 Z -8 R8 F60 / *Final position of tool*
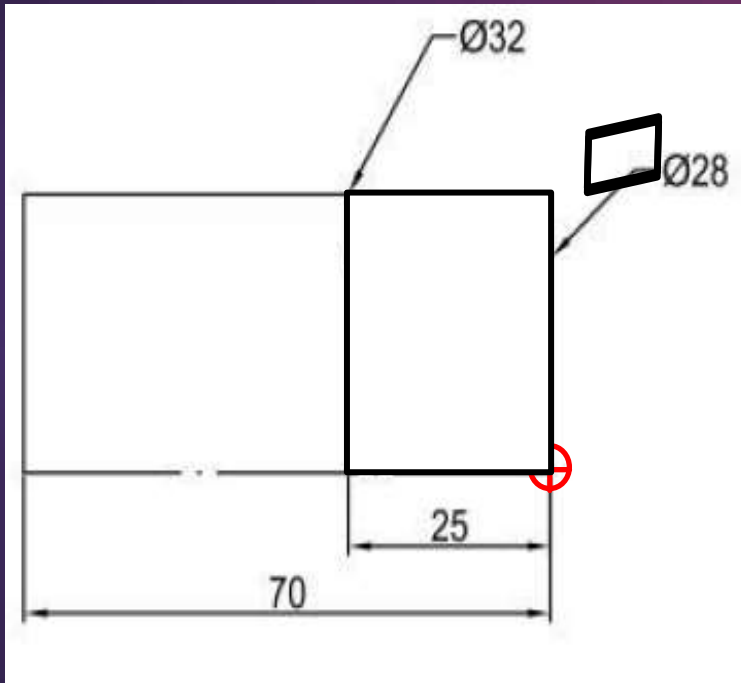
G00 X 24 Z2

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

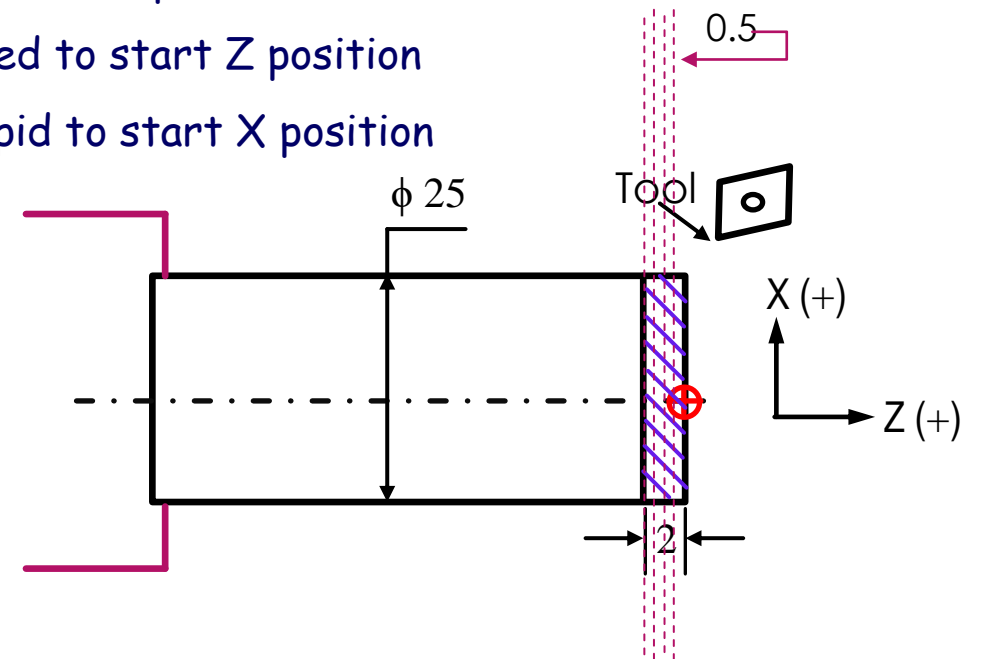G94 X0 Z -0.5 F60 /

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

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

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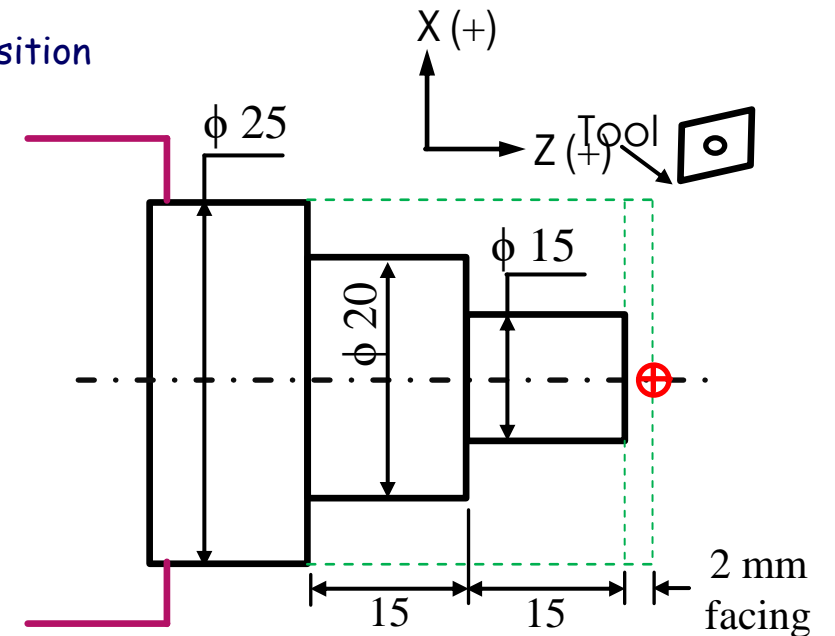3. Feed to start X position
4. Rapid to start Z position

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

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
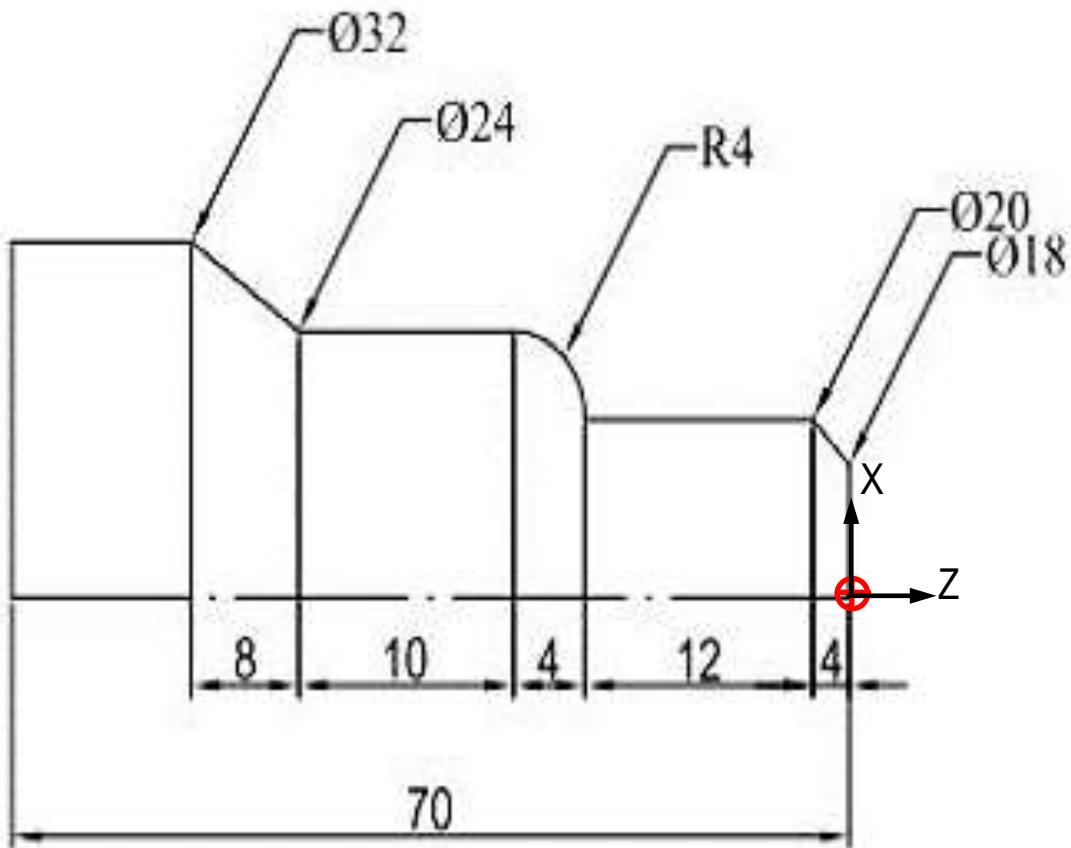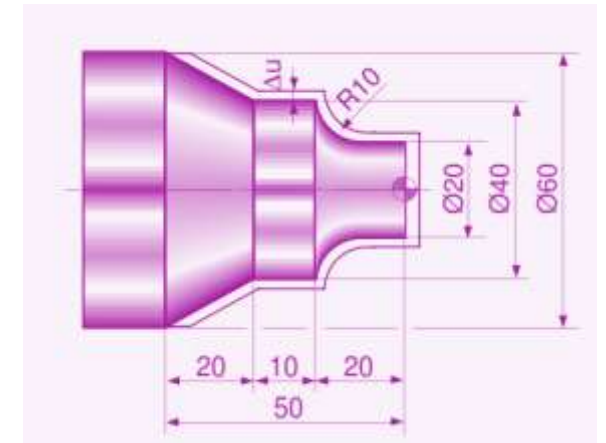
G70 –Finishing Cycle

Syntax:
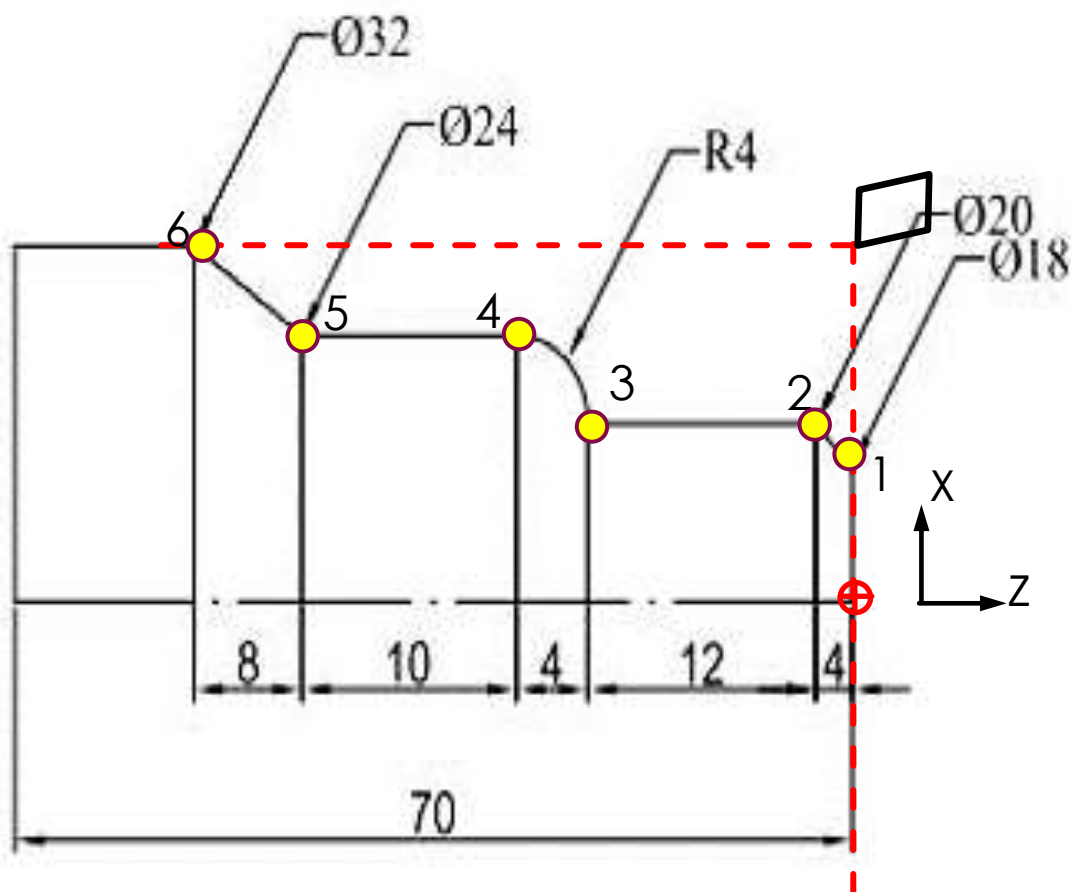
G70 P_ Q_ F_

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
```

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 a 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-12 Q500 F100 / make drill upto 12 mm with 500 μm doc

G28 U0 W0

M05

M30
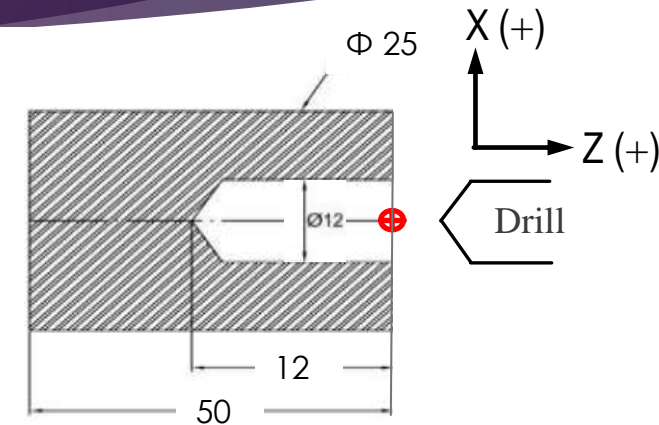
G76 – Multiple Threading Cycle

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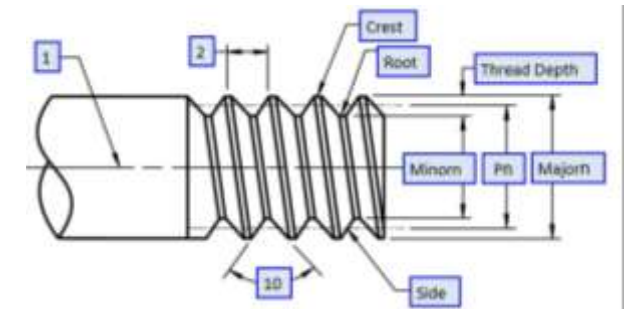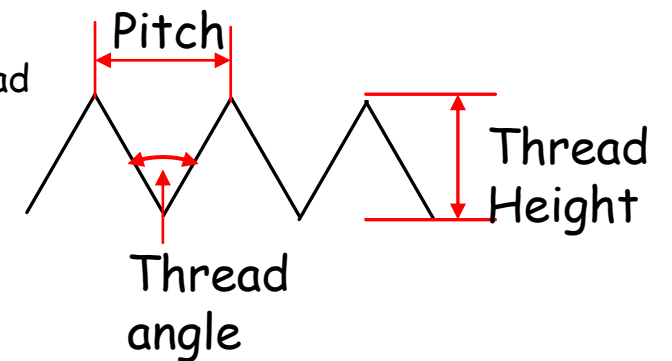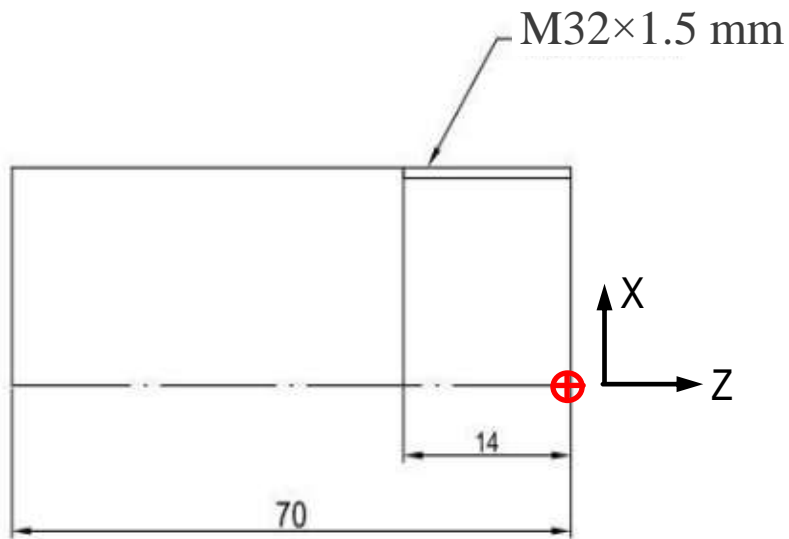P=thread height (μm in software/ mm in machine)
F= Pitch

D= Major Diameter

P= Thread Height

Thread Height, P = 0.613 x Pitch of the Thread

Core dia= D – 2 (P)

G76 – Multiple Threading Cycle

M32×1.5 mm


Thread Height, P = 0.613 x Pitch of the Thread
                 = 0.613 x 1.5
          P   = 0.919 mm =919 μm

Core diameter = Major dia – 2 (P)
          = 32 – 2 (0.919)
          = 32 – 1.818
          = 30.162 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

φ 32

φ 20

φ 12

M 12×1.5

X

Z

21

3

3

15

21

3

3

15
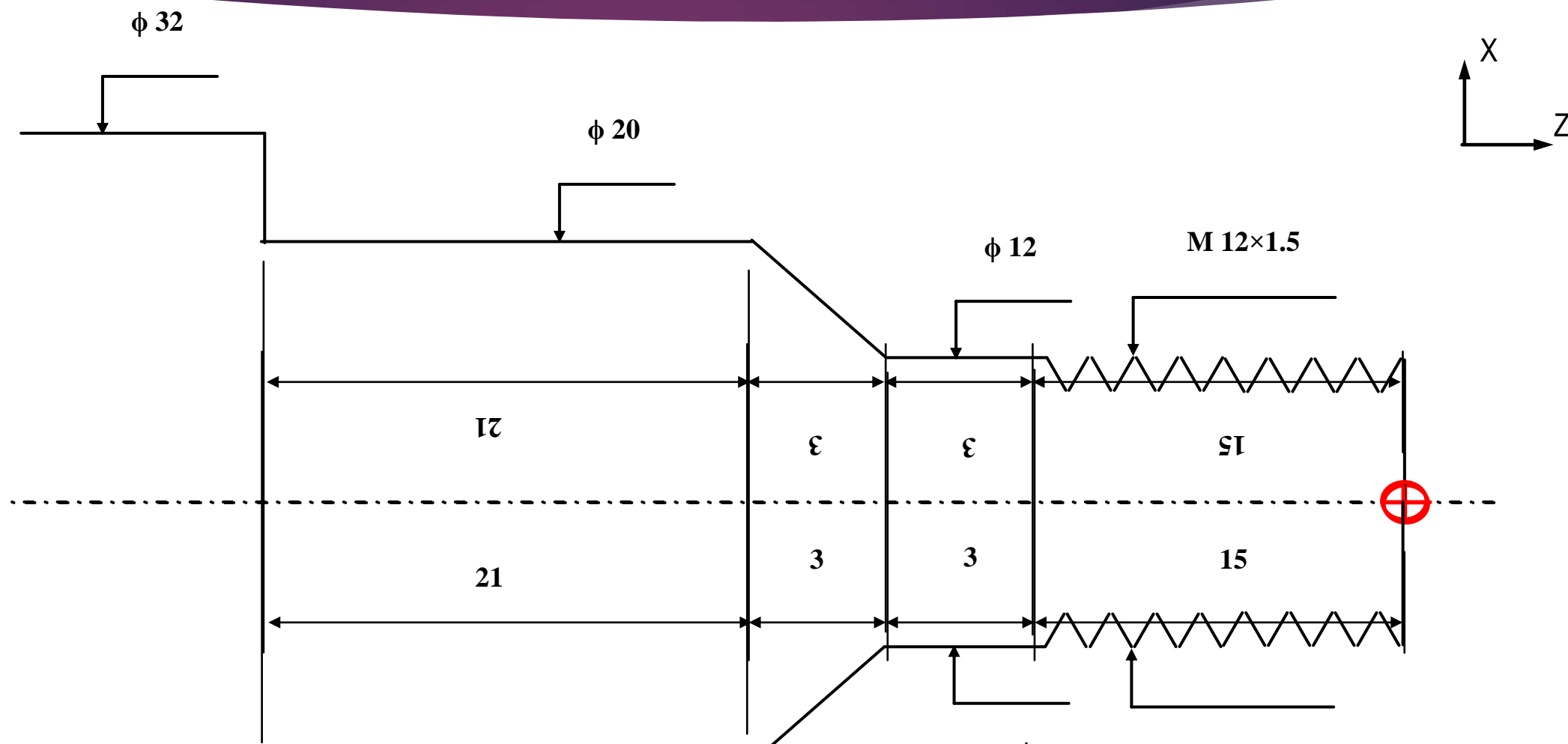
O0001
G21 G98
G28 U0 W0
M06 T0101
M03 S1500   /*Spindle clockwise with 1500 RPM*
G00 X32 Z2   /*Tool Moving to Tool Entry Point of X32 Z5 at Rapid Traverse*

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

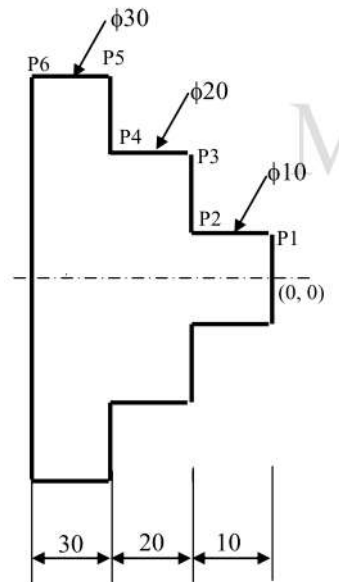φ 32

R6

φ 20

φ 12   M 12×1.5

21   3   3   15
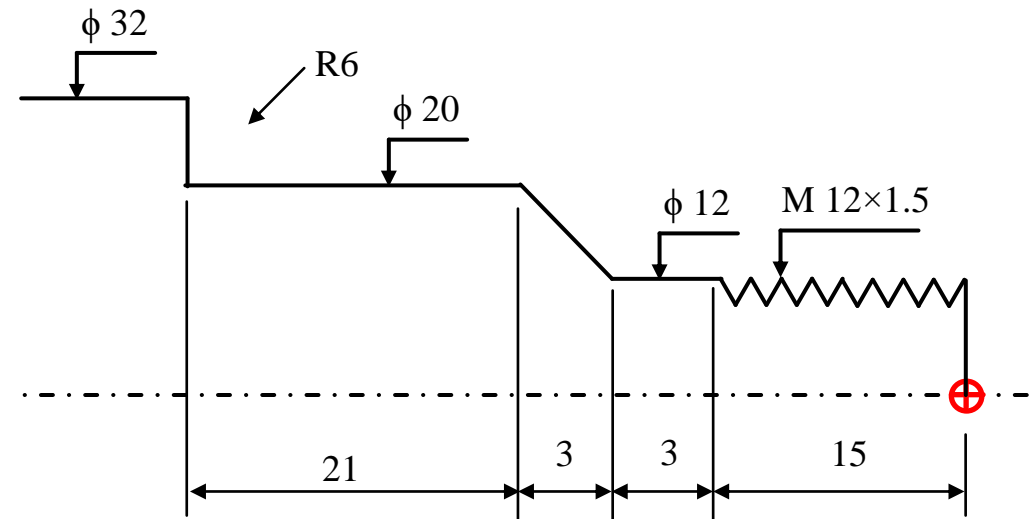
Different Measurement System

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.
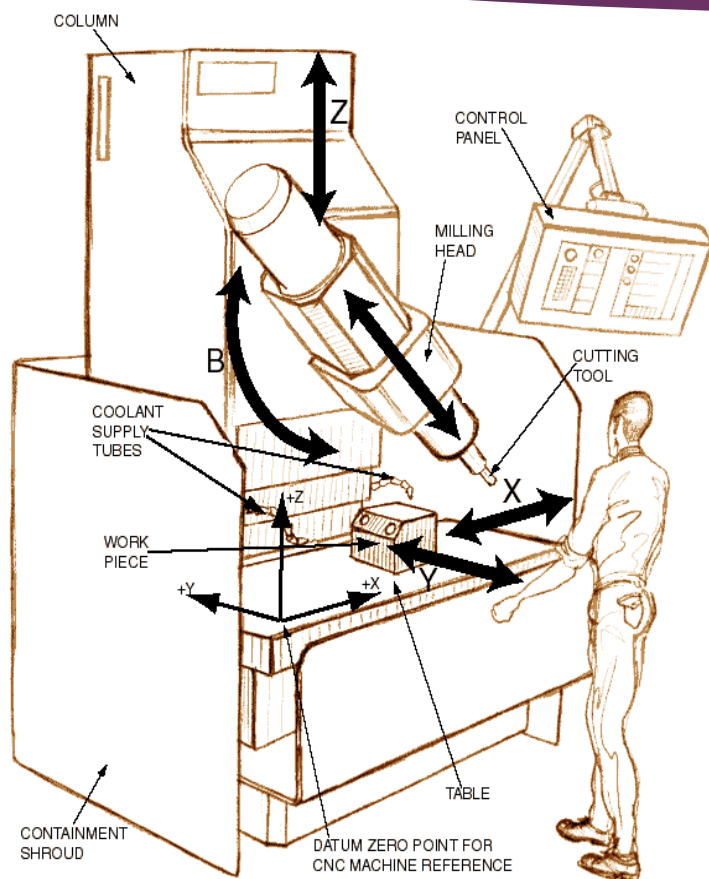
➤ Define by *X and Z*

Incremental System

➤ Every measurement is considered from **previously dimensioned position**.

➤ Incremental dimensions are distance between adjacent points.

➤ Defined by *U and W*

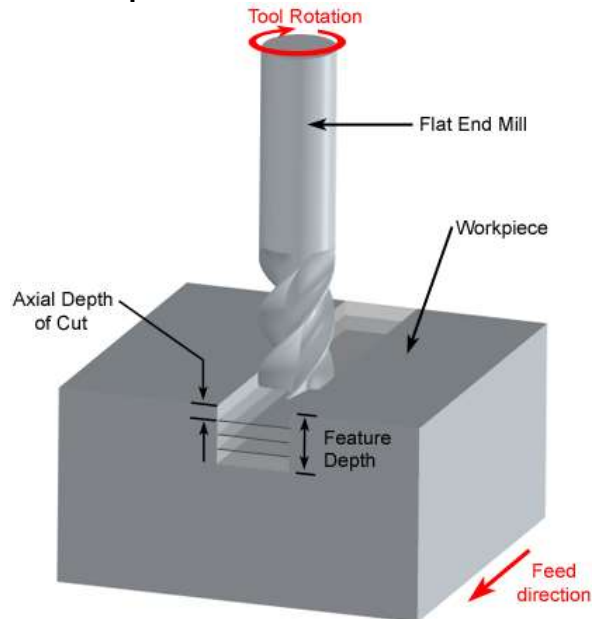| ABSOLUTE DIMENSIONING			INCREMENIAL DIMENSIONING		
POINTS	X	Z	POINTS	U	W
P1					
P2					
P3					
P4					
P5					
P6					

Degree of Freedom in Milling

End milling tool

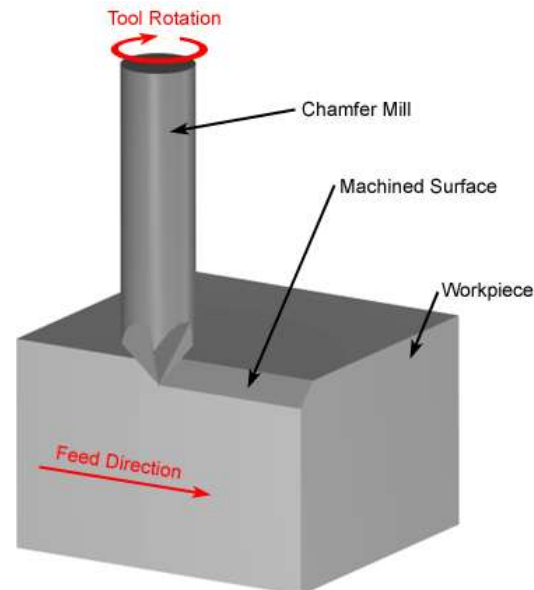# Various Operations That Can Be Performed in Milling

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
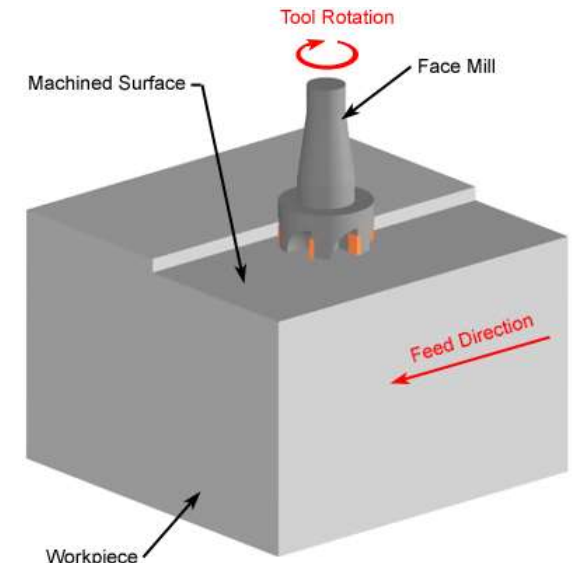

Chamfer Milling

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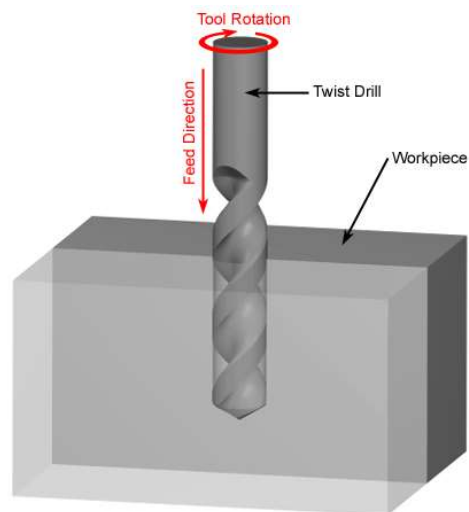
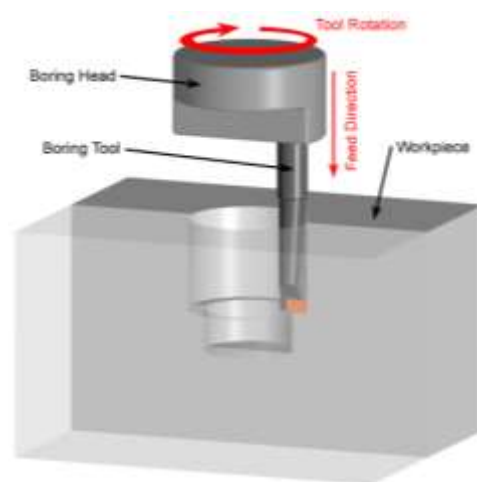➤ Create an angled surface, known as a chamfer.

Face Milling

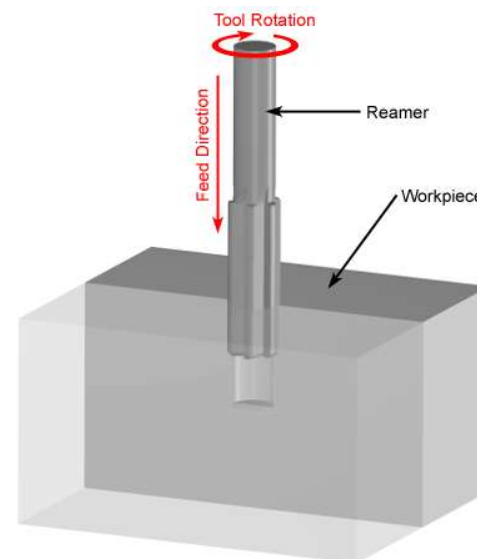➤ A face mill machines a flat surface of the workpiece in order to provide a smooth finish.

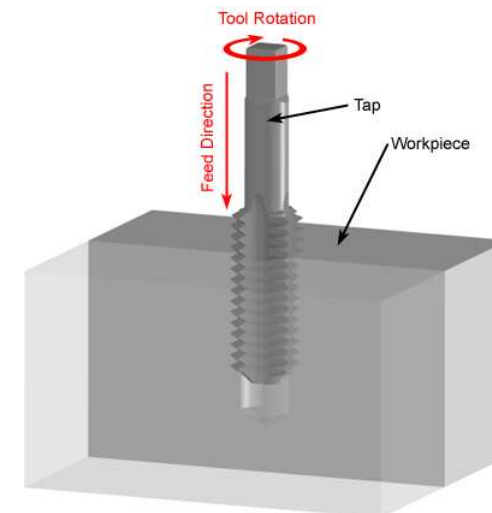# Various Operations That Can Be Performed in Milling
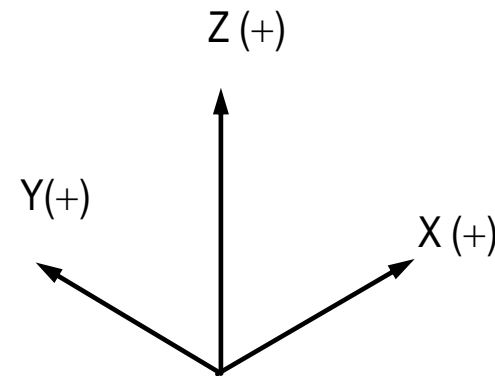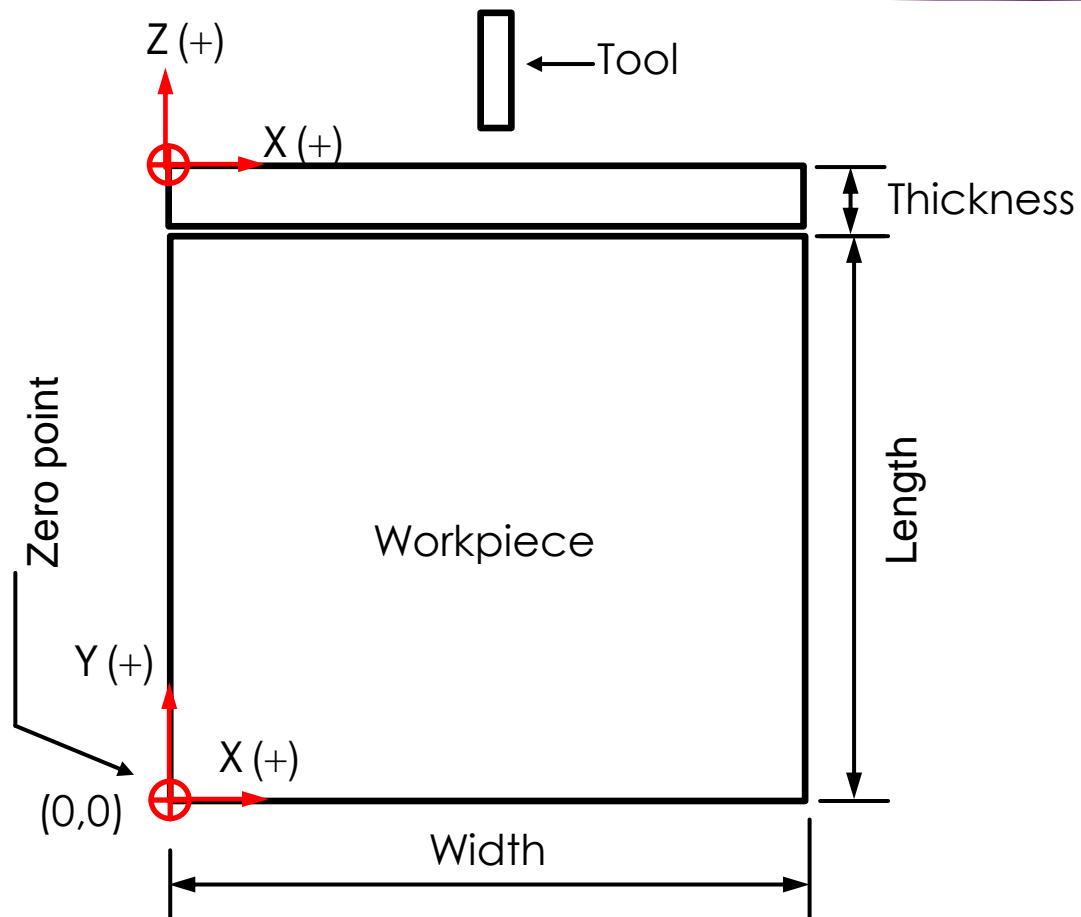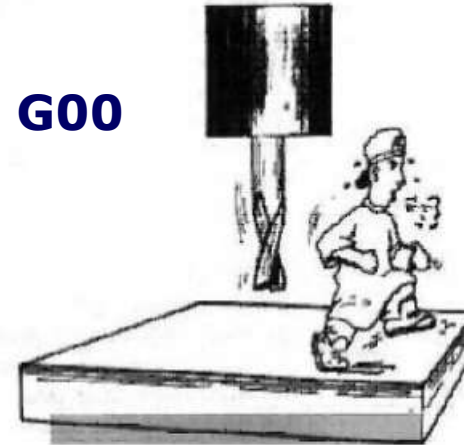

Drilling

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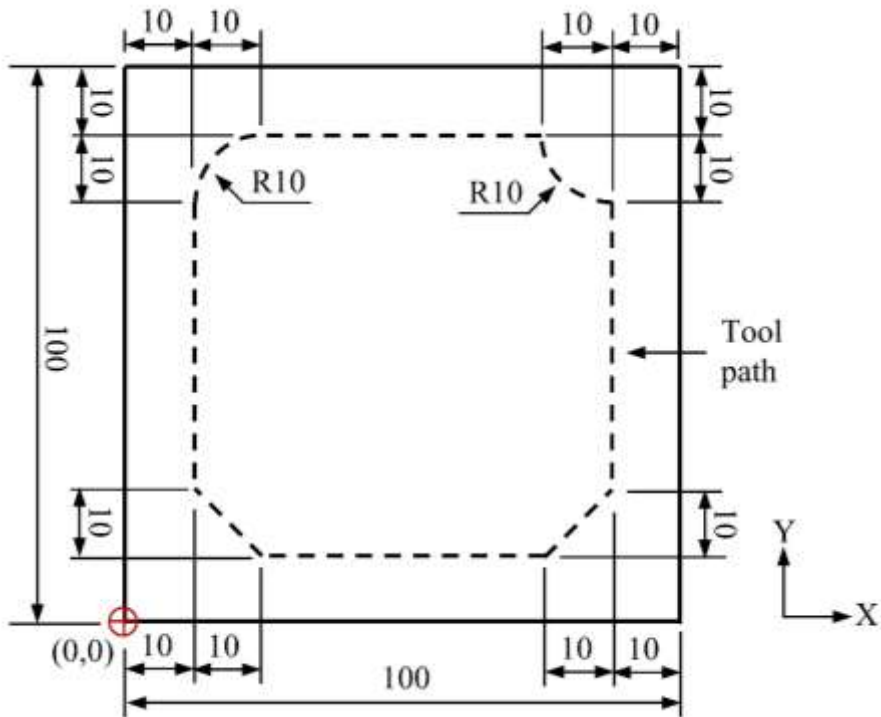
Thickness of workpiece: 12 mm, Depth of cut: 1 mm,
Feed rate: 100 mm/min, Speed: 2500, Dia of end mill: 5 mm

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*