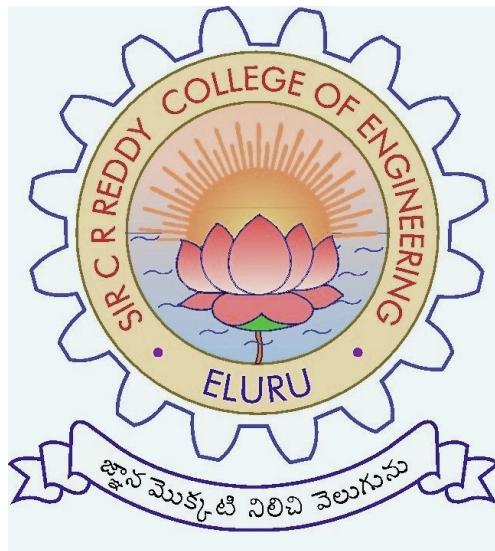


**SIR C.R.REDDY COLLEGE OF ENGINEERING
ELURU-534007**

**CAD/CAM LABORATORY
MANUAL**

IV/IV B.TECH (Mechanical): I SEMESTER



DEPARTMENT OF MECHANICAL ENGINEERING

DEPARTMENT OF MECHANICAL ENGINEERING**COMPUTER AIDED DESIGN LAB****LIST OF EXPERIMENTS**

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INTRODUCTION TO GRAPHICS PACKAGE

The engineering drawing has been and is an integral part of industry and it is a link between engineering design and manufacturing. Information is quickly communicated in the form of drawings prepared according to prescribed drafting standards.

What is Computer Aided Design?

The use of interactive graphics programs to develop 2D and 3D models, assemblies, part lists, and dimensional drawings of various components, structures or objects with the help of a computer is called Computer-Aided Design(CAD). The data generated by a CAD system can be directly utilized by a CAM (Computer Aided Manufacturing) system. Thus CAD and CAM are interrelated to each other.

Advantages of CAD

- Reduction of drafting labor.
- Direct cost savings.
- High accuracy (up to one millionth of a unit)
- Improvement in the general flow of information through the company.
- Evaluation of alternative designs.
- Use of common parts in multiple products.

Before we design and manufacture any component, the object must be presented in the form of drawings along with different views and dimensions. Further, the bill of materials etc. must also be supplied. With the advent of computers and relevant software packages, the drafting drifted from manual to computer aided drafting. The networking between the shop floor sections and design sections makes it easier to share the data and information. The modifications (in the drawings, materials or process of operation) if required, can be discussed on line and can be implemented without any time lag. An attempt is made here to introduce the graphics package in the following topics.

CAD LAB EXPERIMENTS

1. INITIATING THE GRAPHICS PACKAGE

Aim: Setting the paper size, space; setting the limits, units; use of snap and grid commands.

Requirements: Auto CAD software, PC with a min. of 256MB RAM.

Description:

LIMITS: sets and controls the drawing boundaries and grid display.

The drawing limits are two-dimensional points in the World Co-ordinate System that represent a lower-left limit and an upper-right limit. Limits cannot be imposed on the Z direction.

Turn on the drawing limits and restrict within the rectangular area. Drawing limits also determine the area of the drawing that can be displayed with or without grid dots.

Command: LIMITS

ON/ OFF/ Lower left corner < current> :

- ❑ **On:** Turns on limits checking. When the “limits checking” is on, AutoCAD rejects attempts to enter points outside the drawing limits.
- ❑ **Off:** Turns off limits checking, but maintains the current values, for the next time when the “limits checking” is turned on.

Procedure: Enter the command “LIMITS”.

Reset model space limits.

Specify lower left corner or [ON/OFF], < 0.0000, 0.0000 > : 0,0 (return)

Specify upper right corner < 120.0000, 120.0000 > : 120,120 (or any value) (return)

Command: zoom (return)

Use of Snap And Grid Commands

GRID displays a dot grid

At the command prompt, enter ‘GRID’.

Command: GRID

Grid Spacing (X) or ON / OFF / Snap / Aspect / Current > :

You can turn grid on and off with F7 key or by double clicking the GRID button in the status bar.

SNAP restricts cursor movement to specified intervals

At the command prompt, enter ‘SNAP’.

The points you enter with a pointing device can be locked into alignment by SNAP. You can rotate the SNAP grid, set differing X and Y spacing, or choose an isometric format for the snap grid.

A change in the snap grid affects only the co-ordinate of new points. Objects already in the drawing retain their existing co-ordinates.

The snap grid is invisible. Use SNAP with GRID to display a separate visible grid of dots. Set the spacing of two grids to equal or related values dots.

Command: SNAP

Snap spacing or ON / OFF / Aspect / Rotate / Style < Current > :

- Snap spacing:** Actives snap mode using the current snap grid resolution, rotation and style.
- ON:** Actives snap mode using the current snap grid resolution, rotation and style.
- OFF:** Deactivates snap mode but retains the values and modes.
- Rotate:** Sets the rotation of the snap grid with respect to the drawing and the display screen. Any angle between –90 and + 90 can be specified. A positive angle rotates the grid counterclockwise about the base point.

Base point <current > :

Rotation angle < current >:

- Style:** Selects the format of the snap grid, which is standard or Isometric. Standard displays a rectangular grid that is parallel to the XY plane of the current UCS. X and Y spacing may differ
- Spacing:** Changes overall spacing.
- Aspect:** Changes the horizontal and vertical spacing individually.

Isometric displays an isometric grid where the grid points are initially at 30 and 150-degree angles.

Vertical spacing < current >:

2. DRAWING OF PRIMITIVES

Aim: Drawing various graphics primitives.

Requirements: Auto CAD software, PC with a min. of 256MB RAM.

Description: The command “LINE” creates straight line segments. From the draw menu, select Line. At the command prompt, enter Line or L. Line command allows creating a line where the end points are specified in two-dimensional or three-dimensional coordinates.

Procedure:

Command: limits (return)

Reset model space limits:

Specify lower left corner or [ON/OFF], < 0.0000, 0.0000 > : 0,0 (return)

Specify upper right corner < 120.0000, 120.0000 > : 120,120 (or any value) (return)

Command: zoom (return)

Command: line (return)

Specify first (point P1): 50,50 (return)

Specify next point (P2): 100,100 (return)

AutoCAD prompts for points, allowing to draw continuous line segments.

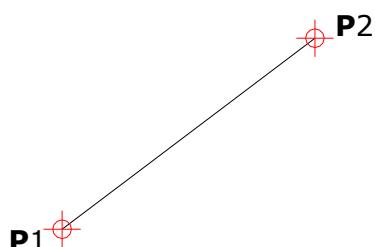
U (Undoing a line)

This allows erasing the most recent segment in a line sequence

C (closing a line sequence)

This allows closing a sequence of continuous line segments, resulting in a polygon.

You may specify the points randomly with the mouse or by any of the coordinate systems.



Lines can be drawn, using absolute mode (eg.100,100) or incremental mode (eg.@100,100) or polar mode(eg.100<90).

Circle

The command “CIRCLE” creates a circle.

From Draw menu, select Circle.

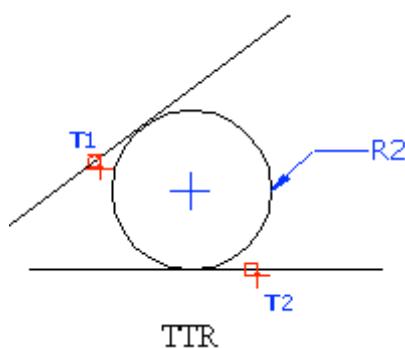
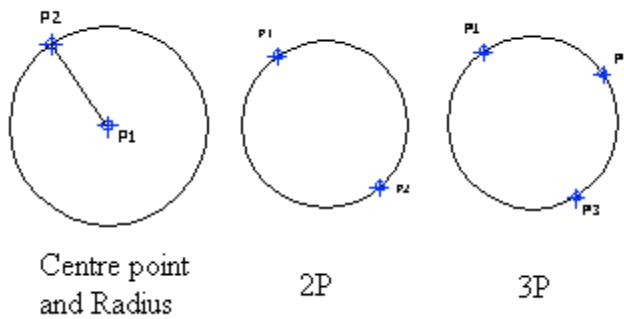
At the command prompt enter CIRCLE OR C.

Specify center point for circle: 50,50 (return)

Specify radius of circle [or diameter]: 50 (return)

The other options for drawing a circle are 2P, 3P and TTR.

With 2P option, the user defines two points. These two points are at the diameter of the circle. 3P option allows you to define 3 non-collinear points. The circle fits in these three points. TTR option allows drawing a circle by defining two tangent points and specifying a radius. AutoCAD by default asks for a centre point and radius.



ARC

ARC allows to create an arc segment

From Draw menu, select Arc

At the command prompt, enter ARC or A

Command: ARC

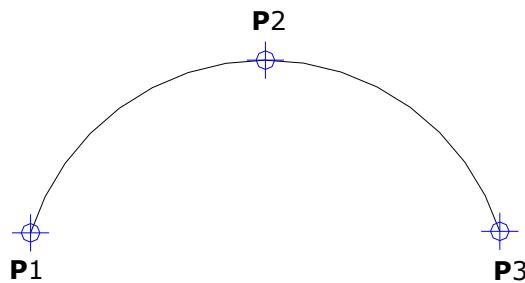
Center / < start point >: specify a point P1

Start point - This specifies the start point of an arc

Center / End/ < second point>:

Specify a point P2

Second point - This draws an arc using three specific points along the circumference of the arc. The first and the third points from the end points of the arc. The second point can be any point on the circumference of the arc.

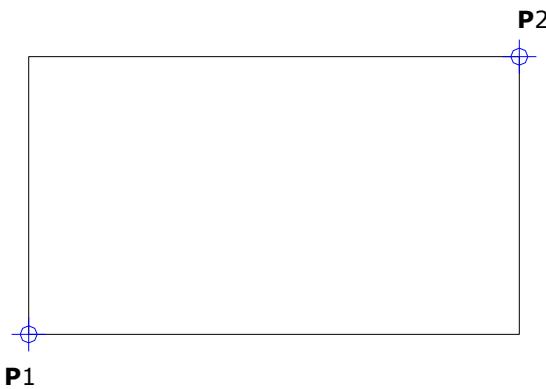


RECTANG

RECTANG draws a rectangular polyline

First corner : specify a point P1

Other corner : specify a point P2



POLYGON

POLYGON allows you to create polygons with 3 to 1024 numbers of sides.

From Draw menu, select Polygon

At the command prompt, enter POLYGON or POL

Command: POLYGON

Number of sides < current>

Edge /< centre of polygon >:

Number of sides

This specifies the number of sides the polygon must have. AutoCAD accepts any value between 3 to 1024

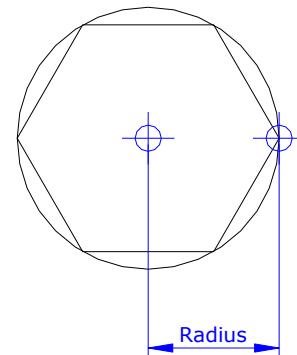
Centre of polygon

Defines the center point of the polygon. Specify a point

Inscribed in circle / Circumscribed about circle (I/C) < I > :

Inscribed in circle

This specifies the radius of a circle on which all vertices of the polygon lie.



Circumscribed about circle

This specifies the distance from the centre of polygon to the midpoints of the edges of the polygon.

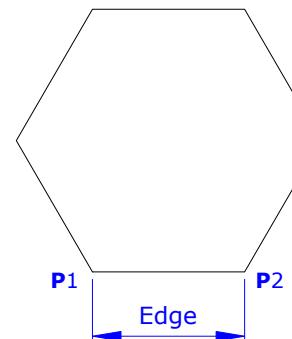
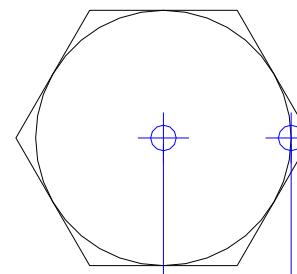
Radius of circle:

Edge

Defines a polygon by specifying the endpoints of the first edge.

First endpoint of edge

Second endpoint on edge



ELLIPSE

ELLIPSE creates an ellipse or an elliptical arc

From Draw menu, select Ellipse

At the command prompt, enter Ellipse or EI

Command: ELLIPSE

Arc/ center/ <Axis endpoint 1> :

Axis endpoint 1:

This is the first step towards defining the first axis. The two end points of the axis are specified. When the first point is specified in the above option, it prompts for the second point.

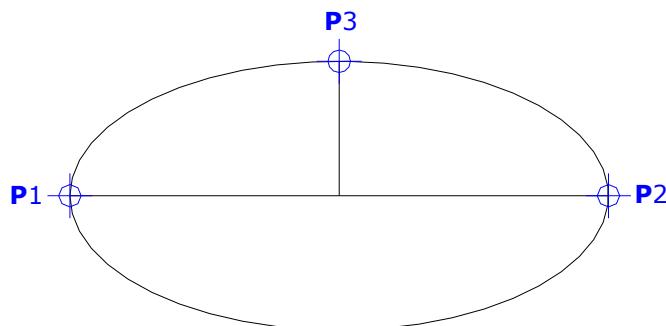
Axis endpoint 2 :

< other axis distance > / Rotation :

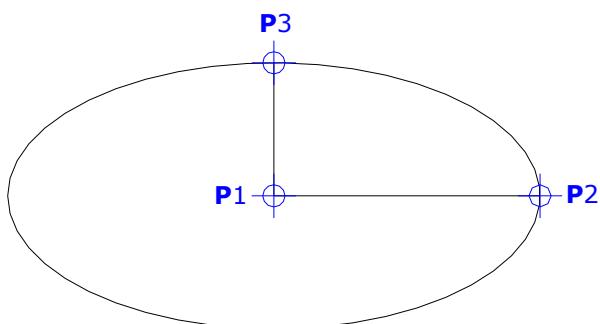
The first axis can define either the major or the minor axis of the ellipse.

Other axis distance

Defines the second axis as the distance from the centre of ellipse in a direction perpendicular to the first axis.



Ellipse by axis end points



Ellipse by centre point

3. FLANGE

Aim: Drawing a flange

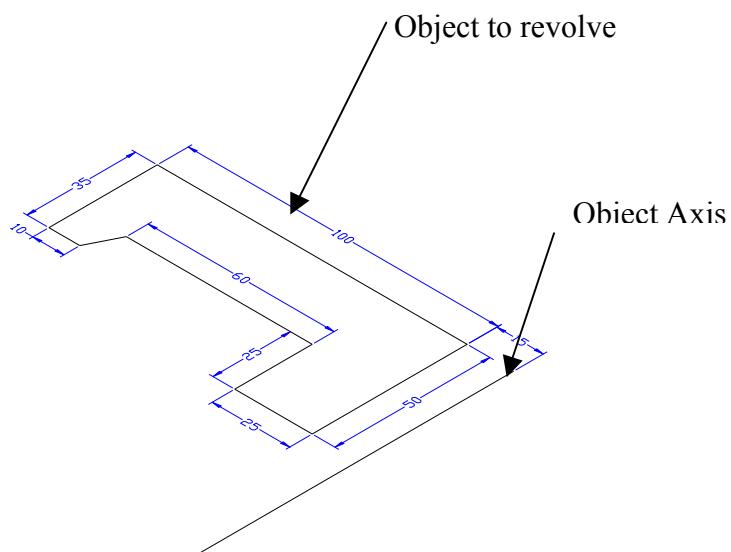
Procedure:

FIRST STEP

Draw PLINE of required cross section

PLINE creates two-dimensional poly lines

A 2D poly line is a connected sequence of a line and arc segments, and is created by AutoCAD as a single object using region command.



SECOND STEP

REVOLVE the drawn pline about required axis and a specified angle

Command: Revolve

REVOLVE creates a solid by revolving a two-dimensional object about an axis.

Select objects : Use object selection method

Axis of revolution – Object / X / Y / < Start point axis > : Specify a point or enter an option.

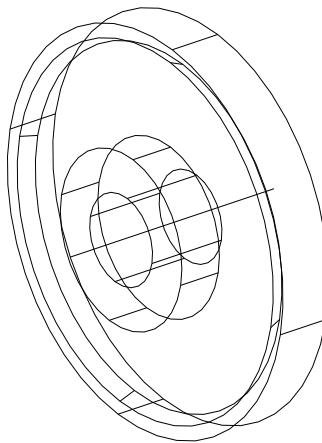
Start point of axis:

Specifies the first and second point of the axis if revolution.

Angle of revolution <full circle > : Specify an angle or press Enter

THIRD STEP

Draw a circle on the face of the flange and use the polar array command for making number of holes on the face.

**ARRAY**

ARRAY creates multiple copies of objects in a pattern.

From modify menu, select Array.

At the command prompt, enter ARRAY or AR

Select objects: select objects by any of the selection methods.

Rectangular / Polar array (R/P) < current> :

Polar array

This creates an array about a defined center point

Center point of array:

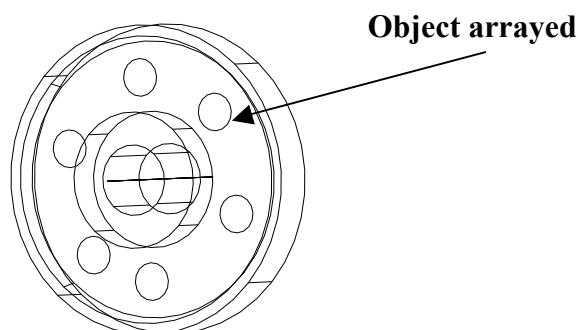
Number of items:

Angle to fill (+ = ccw, - = cw) < 360 > :

Rotate objects as they are copied ? < Y > :

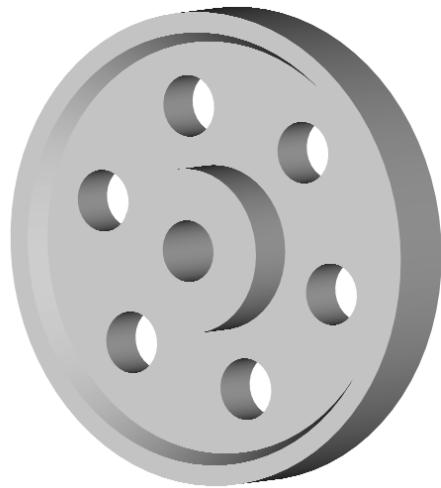
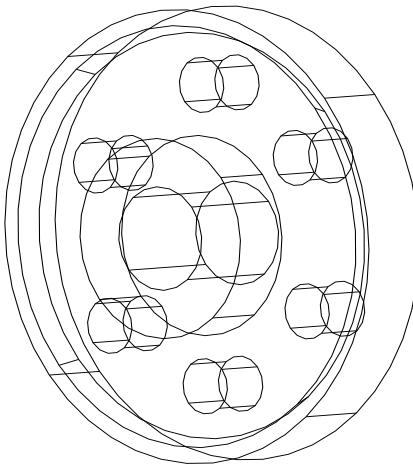
Angle between items:

Rotate objects as they are copied? < Y > :



FOURTH STEP

Extrude all the six components to the required path and distance using EXTRUDE command. Then subtract all the components from the flange. You will get a required flange as shown below.



4. BUSHING ASSEMBLY

Aim: Drawing a bushing assembly

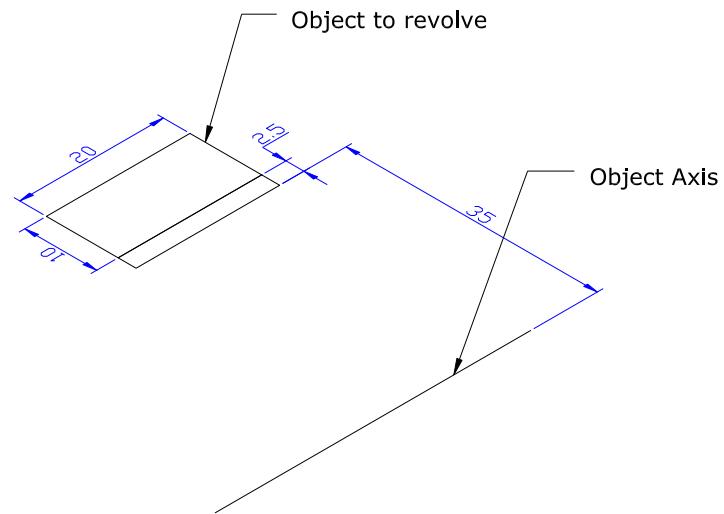
Procedure:

FIRST STEP

Draw PLINE of required cross section

PLINE creates two-dimensional polylines

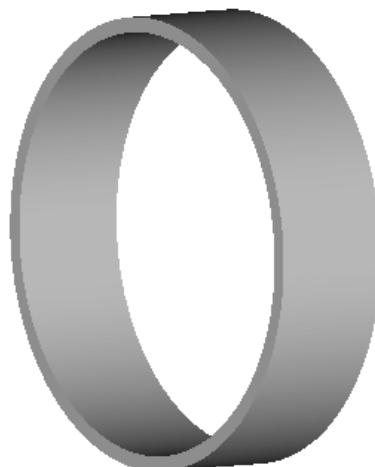
A 2D polyline is a connected sequence of a line and arc segments, and is created by AutoCAD as a single object.



SECOND STEP

REVOLVE the drawn pline about required axis and a specified angle

Command: Revolve



REVOLVE creates a solid by revolving a two-dimensional object about an axis.

Select objects: Use object selection method

Axis of revolution – Object / X / Y / < Start point axis >: Specify a point or enter an option.

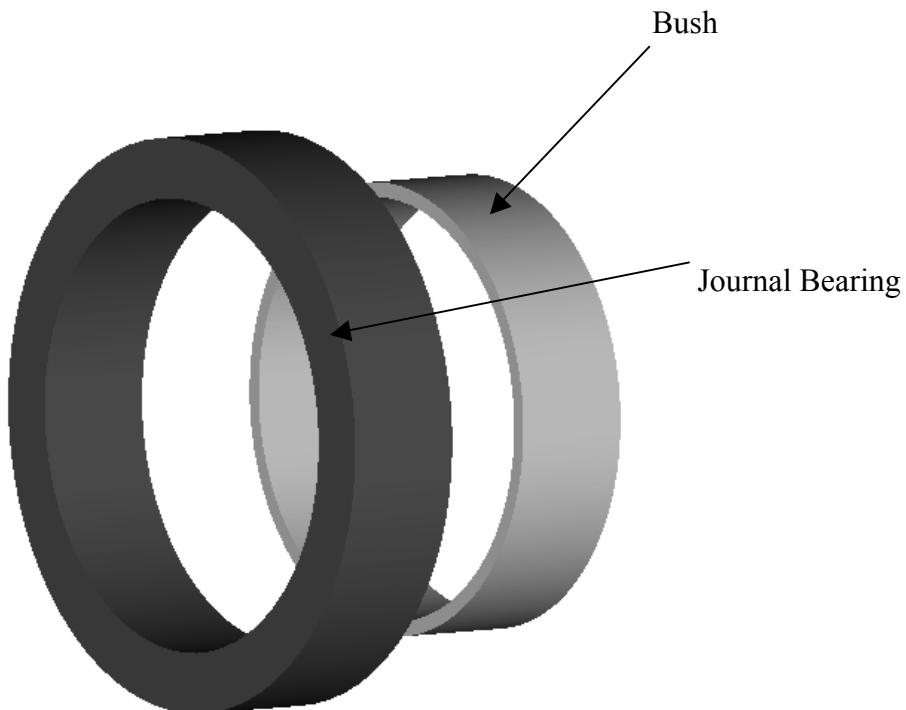
Start point of axis:

Specifies the first and second point of the axis if revolution.

Angle of revolution <full circle >: Specify an angle or press Enter

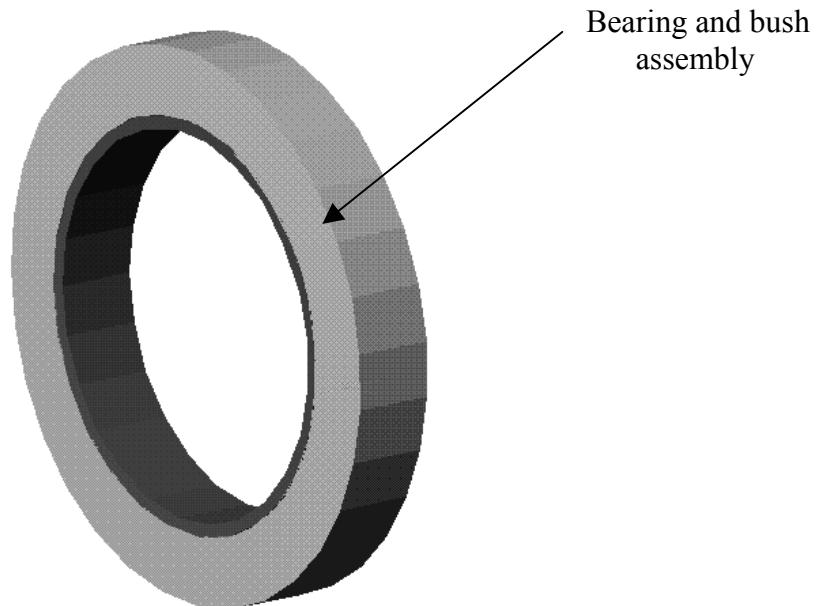
THIRD STEP

Repeat the same procedure for drawing the bearing



FOURTH STEP

Assemble the bush and bearing



5. DIMENSIONING THE DRAWING AND ADDING TEXT

Aim: To dimension the drawing and to add text.

Description:

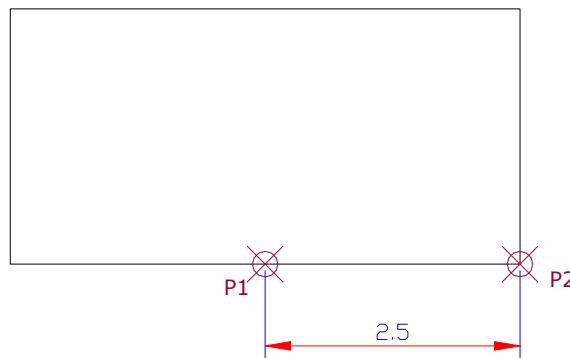
DIMLINEAR: DIMLINEAR creates linear dimensions

From Dimension menu, select Linear

At the command prompt, enter DIMLINEAR or DLI

Command: DIMLINEAR

First extension line origin or RETURN to select: Specify a point or press Enter for automatic extension lines



Point specification

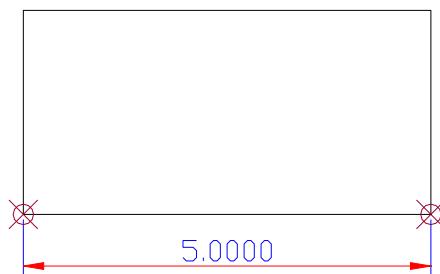
If you specify a point (P1) for the first extension line, AutoCAD prompts for the second extension line origin

Second extension line origin: specify a point (P2)

Object selection-Automatic extension lines

If you press enter to select an object, AutoCAD automatically determines the origin points of the first and second extension lines.

If you select a circle, the endpoints of its diameter are used as the origin points. When the point used to select the circle is close to the north or south quadrant point,



AutoCAD draws a horizontal dimension. When the point used to select the circle is close to the east or west quadrant point, AutoCAD draws a vertical dimension.

Dimension line location (Text/Angle/Horizontal/Vertical/Rotated):

Specify a point (P2) or enter an option.

Text: Allows you to customize the text.

Angle: Changes the angle of the dimension text.

Enter text angle: specify an angle

After you specify the angle, AutoCAD prompts

Dimension line location (Text/Angle/Horizontal/Vertical/Rotated)

Horizontal: Creates horizontal linear dimensions

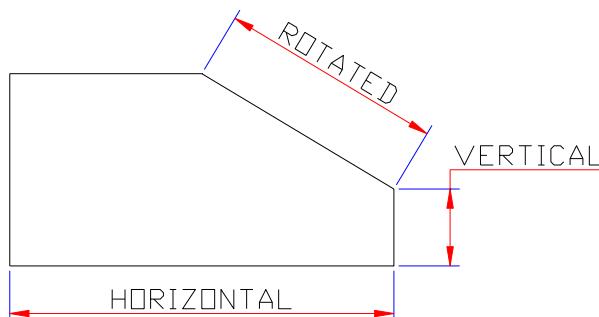
Dimension line location (Text/ Angle): Specify a point or enter an option

Vertical: Creates vertical dimensions

Dimension line location (Text/ Angle): Specify a point or enter an option

Rotate: Creates rotated linear dimensions

Dimension line angle<current>: specify an angle or press enter



DIMALIGNED

DIMALIGNED creates an aligned linear dimension

From dimension menu, select Aligned.

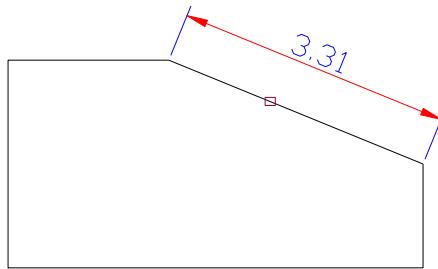
At the command prompt, enter DIMALIGNED or DAL

Command: DIMALIGNED

First extension line origin or RETURN to select: Specify a point for manual extension lines, or press enter for automatic extension lines.

Object selection for Automatic extension lines Automatically determines the origin points of the first and second extension lines.

Select object to dimension: Select an object



DIMRADIUS

DIMRADIUS Creates radial dimensions for circles and arcs

From Dimension menu, select Radius

At the command prompt, enter DIMRADIUS or DRA

Command: DIMRADIUS

Select arc or circle: select an arc or a circle

AutoCAD measures the radius and displays the text with the letter **R** in the front of it

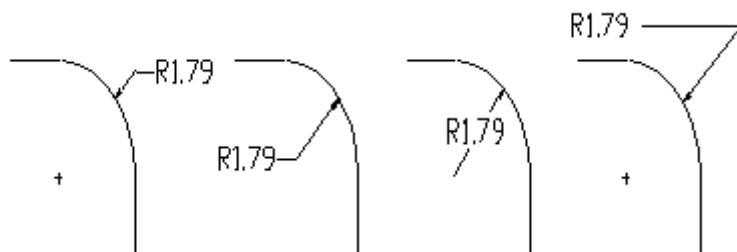
Dimension line location (Text/ Angle): specify a point or enter an option

Text

Dimension text < measured value >: Enter text or press

Angle

Enter text angle: specify an angle



Different Methods of Radius Dimensioning

DIMDIAMETER

DIMDIAMETER creates diameter dimensions for circles and arcs

From Dimension menu, select Diameter

At the command: prompt, enter DIMDIAMETER or DDI

Command: DIMDIAMETER

Select arc or circle: *select an arc or circle*

AutoCAD measures the diameter and displays the text with symbol in front of it. The next prompt is

Dimension line location (Text/ Angle): *specify a point or enter an option*

Text

Dimension text < measured value>: *Enter text or press return*

Dimension line location:

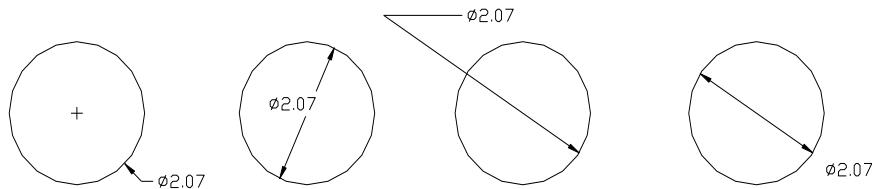
Angle

Changes the angle of the dimension text

Enter text angle: *specify an angle*

After you specify the angle, AutoCAD redisplays

Dimension line location (Text/ Angle):



Different methods of diameter dimensioning

DIMANGULAR

DIMANGULAR creates angular dimensions.

From dimension menu, select Angular

At the command prompt, enter DIMANGULAR or DAN

Command: DIMANGULAR

Select arc, circle, line or RETURN

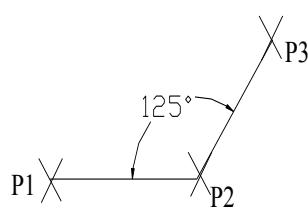
Select the appropriate object or press enter to specify three points

Three point specification

Angle vertex: Specify a point (P1)

First angle end point: specify a point (P2)

Second angle end point: *specify a point (P3)*



Arc selection

If you select an arc, AutoCAD determines the defining points for a three-point angular dimension. The center of the arc is the angle vertex. The arc endpoints become the origin points of the extension lines.

Circle selection

If you select a circle, the centre of the circle is the angle vertex. The selection point is used as the origin of the first extension line

AutoCAD draws the dimension line as an arc between the extension lines. The extension lines are drawn from the angle endpoints to the intersection of the dimension line.

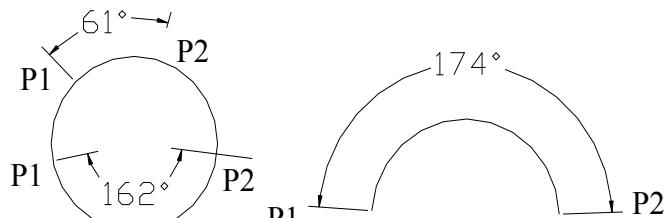
DIMBASELINE

DIMBASELINE creates a linear, angular, or ordinate dimension from the baseline of the previous or selected dimension

From the Dimension menu, select Baseline or DBA

Command: DIMBASELINE

Second extension line origin or RETURN to select: *specify a point or press to select a base dimension*



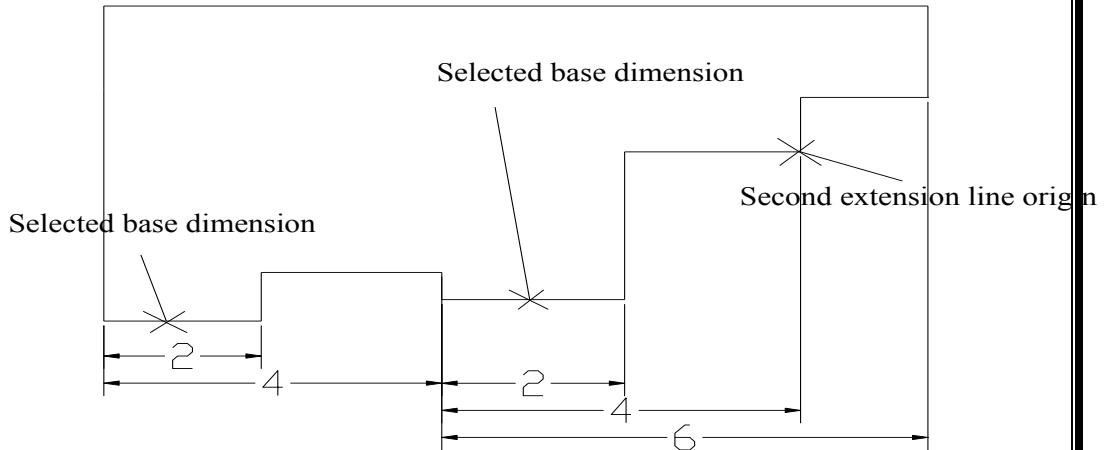
Angular dimension on arc and circle

If the previous dimension was linear, angular or ordinate, AutoCAD uses the first extension line origin of that dimension for the first extension line origin of the baseline dimension. After you specify a point, AutoCAD draws the baseline dimension and redisplays the prompt.

Second extension line origin

If the previous dimension was linear, angular or ordinate, if you press enter at the second extension line origin prompt, AutoCAD prompts you to select linear, angular or ordinate dimension to use as the basis for the baseline dimension.

Select base dimension: *Select a linear, angular or ordinate dimension*



Linear baseline dimensioning

DIMCONTINUE

DIMCONTINUE continues a linear, angular or ordinate dimension from the second extension line of the previous or a selected dimension.

From Dimension menu, select Continue

At the command: prompt enter DIMCONTINUE or DCO

Command: DIMCONTINUE

If the previous dimension was linear, angular or ordinate, AutoCAD uses the origin of that dimension's second extension line for the origin of the next dimension's first extension line.

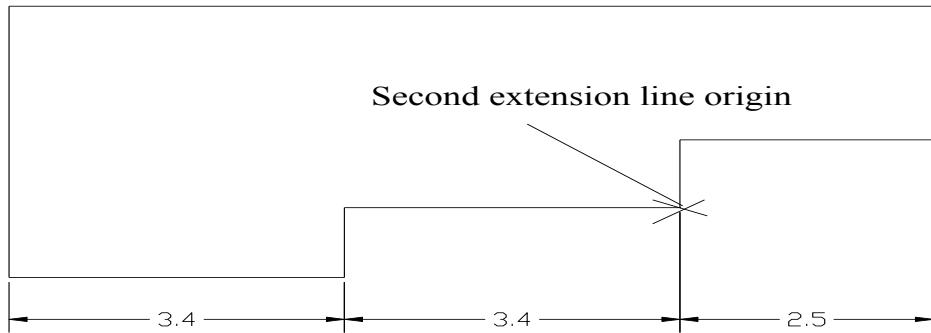
Select extension line origin or RETURN to select:

Specify a point or press enter to select a continued dimension

After you specify a point, AutoCAD draws the next dimension. AutoCAD redisplays the prompt

Select extension line origin or RETURN to select:

To end the command, press enter



Linear continuous dimensioning

DIMORDINATE

DIMORDINATE creates ordinate point dimensions

From Dimension menu, select Ordinate.

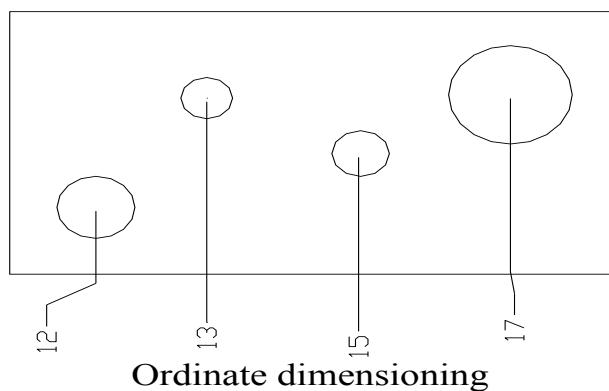
At the Command prompt, enter DIMORDINATE or DOR

Ordinate dimensions display the X or Y coordinate of a feature along with a simple leader line. These dimensions are also known as datum dimensions. AutoCAD uses the current UCS to determine the measurement.

Command: DIMORDINATE

Select feature: specify a point or snap to an object

Leader endpoint (X datum /Y datum /Text): Specify a point or enter an option



TEXT OBJECT

Description:

You can use TEXT to enter several lines of text that you can rotate, justify, and resize. As you type at the Enter Text prompt, the text you are typing is displayed on the screen. Each line of text is a separate object. To end a line and begin another, press ENTER after entering characters at the Enter Text prompt. To end TEXT, press ENTER without entering any characters at the Enter Text prompt.

By applying a style to the text, you can use a variety of character patterns or fonts that you can stretch, compress, make oblique, mirror, or align in a vertical column.

Procedure:

Draw menu: Text Single Line Text

Command line: text

Current text style: *current*

Current text height: *current*

Specify start point of text or [Justify/Style]: Specify a point or enter an option

If TEXT was the last command entered, pressing ENTER at the Specify Start Point of Text prompt skips the prompts for height and rotation angle and immediately displays the Enter Text prompt. The text is placed directly beneath the previous line of text. The point specified at the prompt is also stored as the insertion point of the text.

START POINT

Specifies a start point for the text object.

Specify height <current>: Specify a point (1), enter a value, or press ENTER

The Specify Height prompt is displayed only if the current text style does not have a fixed height.

Specify rotation angle of text <*current*>: Specify an angle or press ENTER

Enter text: Enter text and press ENTER to exit the command

JUSTIFY

Controls justification of the text.

Enter an option [Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]:

You can also enter any of these options at the Specify Start Point of Text prompt.

Align

Specifies both text height and text orientation by designating the endpoints of the baseline.

Specify first endpoint of text baseline: Specify a point (1)

Specify second endpoint of text baseline: Specify a point (2)

Enter text: Enter text and press ENTER to exit the command

The size of the characters adjusts in proportion to their height. The longer the text string, the shorter the characters.

Fit

Specifies the text that fits within an area and at an orientation defined with two points and a height. Available for horizontally oriented text only.

Specify first endpoint of text baseline: Specify a point (1)

Specify second endpoint of text baseline: Specify a point (2)

Specify height <current>:

Enter text: Enter text and press ENTER to exit the command

The height is the distance in drawing units that the uppercase letters extend from the baseline. Designated text height is the distance between the start point and a point you specify. The longer the text string, the narrower the characters. The height of the characters remains constant.

Center

Aligns text from the horizontal center of the baseline, which you specify with a point.

Specify center point of text: Specify a point (1)

Specify height <current>:

Specify rotation angle of text <current>:

Enter text: Enter text and press ENTER to exit the command

The rotation angle specifies the orientation of the text baseline with respect to the center point. You can designate the angle by specifying a point. The text baseline runs from the start point toward the specified point. If you specify a point to the left of the center point, AutoCAD draws the text upside down.

Middle

Aligns text at the horizontal center of the baseline and the vertical center of the height you specify. Middle-aligned text does not rest on the baseline.

Specify middle point of text: Specify a point (1)

Specify height <current>:

Specify rotation angle of text <current>:

Enter text: Enter text and press ENTER to exit the command

The Middle option differs from the MC option in that it uses the midpoint of all text, including descenders. The MC option uses the midpoint of the height of uppercase letters.

Right

Right-justifies the text at the baseline, which you specify with a point.

Specify right endpoint of text baseline: Specify a point (1)

Specify height <current>:

Specify rotation angle of text <current>:

Enter text: Enter text and press ENTER to exit the command

TL (Top Left)

Left-justifies text at a point specified for the top of the text. Available for horizontally oriented text only.

Specify top-left point of text: Specify a point (1)

Specify height <current>:

Specify rotation angle of text <current>:

Enter text: Enter text and press ENTER to exit the command

TC (Top Center)

Centers text at a point specified for the top of the text. Available for horizontally oriented text only.

Specify top-center point of text: Specify a point (1)

Specify height <current>:

Specify rotation angle of text <current>:

Enter text: Enter text and press ENTER to exit the command

TR (Top Right)

Right-justifies text at a point specified for the top of the text. Available for horizontally oriented text only.

Specify top-right point of text: Specify a point (1)

Specify height <current>:

Specify rotation angle of text <current>:

Enter text: Enter text and press ENTER to exit the command

ML (Middle Left)

Left-justifies text at a point specified for the middle of the text. Available for horizontally oriented text only.

Specify middle-left point of text: Specify a point (1)

Specify height <current>:

Specify rotation angle of text <current>:

Enter text: Enter text and press ENTER to exit the command

MC (Middle Center)

Centers the text both horizontally and vertically at the middle of the text. Available for horizontally oriented text only.

Specify middle-center point of text: Specify a point (1)

Specify height of text <current>:

Specify rotation angle of text <current>:

Enter text: Enter text and press ENTER to exit the command

The MC option differs from the Middle option in that it uses the midpoint of the height of uppercase letters. The Middle option uses the midpoint of all text, including descenders.

MR (Middle Right)

Right-justifies text at a point specified for the middle of the text. Available for horizontally oriented text only.

Specify middle-right point of text: Specify a point (1)

Specify height <current>:

Specify rotation angle of text <current>:

Enter text: Enter text and press ENTER to exit the command

BL (Bottom Left)

Left-justifies text at a point specified for the baseline. Available for horizontally oriented text only.

Specify bottom-left point of text: Specify a point (1)

Specify height <current>:

Specify rotation angle of text <current>:

Enter text: Enter text and press ENTER to exit the command

BC (Bottom Center)

Centers text at a point specified for the baseline. Available for horizontally oriented text only.

Specify bottom-center point of text: Specify a point (1)

Specify height <current>:

Specify rotation angle of text <current>:

Enter text: Enter text and press ENTER to exit the command

BR (Bottom Right)

Right-justifies text at a point specified for the baseline. Available for horizontally oriented text only.

Specify bottom-right point of text: Specify a point (1)

Specify height <current>:

Specify rotation angle of text <current>:

Enter text: Enter text and press ENTER to exit the command

6. SETTING THE LAYERS AND APPLICATION OF LAYERS

Aim: To set the layers and to learn and manage the layers.

Description:

LAYER: The command “LAYER” manages layers. It makes a layer current, adds new layers, deletes layers and renames layers. One can assign properties to layers, turn layers *on and off*, freeze and thaw layers globally or by view port, lock and unlock layers, set plot styles for layers, and turn plotting on and off for layers. Also, one can filter the layer names displayed in the Layer Properties Manager, and can save and restore layer states and properties settings.

By creating layers, one can associate similar types of objects by assigning them to the same layer. For example, construction lines, text, dimensions, and title blocks can be put on separate layers and can then be controlled to check

- Whether the objects on a layer are visible in any view ports
- Whether and how the objects are plotted
- What color is assigned to all the objects on a layer
- What default line type and line weight is assigned to all objects on a layer
- Whether objects on a layer can be modified

When a new drawing is begun, AutoCAD creates a special layer named 0. By default, layer 0 is assigned colour number 7 (white or black depending upon your background colour), the CONTINUOUS line type, a line weight of Default (the default setting is .01 inch or .25 mm), and the NORMAL plot style. Layer 0 cannot be deleted or renamed.

Procedure:

From Format menu, select Layer

At the command prompt, enter LAYER or LA

The Layer & Line type properties dialog box appears

Command: LAYER

Various icons and buttons in the dialog box indicate various statuses of layers and layer management tools.

Named Layer Filters

Determines which layers to display in the list of layers. You can filter layers based on whether they're xref-dependent, or whether they contain objects. You can also filter layers based on name, visibility, color, line type, line weight, plot style name, whether they are plotted, or whether they are frozen in the current view port or in new view ports.

[...] Button: Displays the Named layer filters dialog box

Invert Filter: Displays layers based on the opposites of the criteria in a named layer filter. Layers that fit the inverse criteria are displayed in the layer name list.

New

Creates a new layer. After you choose New, the list displays a layer named LAYER1. You can edit this layer immediately. To create multiple layers more quickly, you can select a layer name for editing and enter multiple layer names separated by commas. If you create a new layer, the new layer inherits the properties of the currently selected layer in the layer list (Color, On/Off state, and so on). To create layers with default settings, make sure that there are no selected layers in the list or that you select a layer with default settings before beginning layer creation.

Current

Sets the selected layer as the current layer

Delete

Deletes selected layers from the drawing file definition. You can delete only unreferenced layers. Referenced layers include layers 0 and DEFPOINTS, layers containing objects, the current layer, and xref-dependent layers. Layers that don't contain objects, are not current, and are not xref-dependent can be deleted.

List of Layers: Displays layers and their properties. To modify a property, click its icon. To quickly select all the layers, use the right-click and shortcut menu.

Names: Displays the names of the layers. You can select a name, and then click and enter a new name.

On/Off: Turns layers on and off. When a layer is on, it is visible and available for plotting. When a layer is off, it is invisible and not plotted, even if Plot is on.

Freeze/ Thaw selected layers in all view ports.

You can freeze layers to speed up ZOOM, PAN, and many other operations; improve object selection performance; and reduce regeneration time for complex drawings. AutoCAD does not display, plot, hide, render, or regenerate objects on frozen layers. When you thaw a frozen layer, AutoCAD regenerates and displays the objects on that layer. You can freeze layers in all view ports, in the current layout view port, or in new layout view ports as they are created.

Lock/Unlock: Locks and unlocks the layers. You cannot edit objects on a locked layer. Locking a layer is useful if you want to view information on a layer for reference but do not want to edit objects on that layer.

Color: Changes the color associated with the selected layers. Clicking the color name displays the select color dialog box.

Linetype: Changes the line type associated with the selected layers. Clicking any line type name displays the select Line type color dialog box.

Lineweight: Changes the line weight associated with the selected layers. Clicking any line weight name displays the select Line weight color dialog box.

Plot Style: Changes the plot style associated with the selected layers. If you are working with color-dependent plot styles you cannot change the plot style associated with a layer. Clicking any plot style displays the select plot style dialog box.

7. ISOMETRIC AND ORTHOGRAPHIC PROJECTIONS

Aim: To create isometric and orthographic projections of a drawing.

Procedure:

3DFACE

3DFACE creates a three dimensional face

From Draw menu, select Surfaces

At the command prompt, enter 3DFACE or 3F

First point: Specify a point (P1)

Second point: Specify a point (P2)

Third point: Specify a point (P3)

Fourth point: Specify a point (P4) or press Enter

AutoCAD repeats the Third point and fourth point prompts until you press Enter.

Specify points 5 and 6 at these repeating prompts. When you have finished entering points, press Enter.

Description:

HIDE

HIDE regenerates a three-dimensional model with hidden lines suppressed.

From View menu, select Hide

Procedure: At the command prompt, enter HIDE or HI

Description:

VPORTS: VPORTS divides the graphics area into multiple tiled view ports

Command: VPORTS

Procedure for orthographic projection:

Save /Restore/ Delete/ Join /Single/? / 2/ 3/ 4/: Enter an option or press Enter 3.

This option divides the current view port into **three** view ports.

Horizontal/ Vertical/ Above/ Below /Left / < Right>:

Enter an option or press Enter

The horizontal and vertical options split the area into thirds. The other options create one large view port of half the available area and two smaller ones. The Above, Below, Left, and Right options specify where the larger view port is placed.

The option “**2**” divides the current view port in half.

Horizontal< Vertical>: Enter h or press Enter **4**.

This option divides the current view port into four view ports of equal size.

Single: Returns the drawing to a single view port view, using the view from the active view port.

Join

Combines two adjacent view ports into one larger view port. The resulting view port inherits the view of the dominant view port.

Select dominant view port < current>: Press Enter or select a View port

Select view port to join:

Save

Saves the current view port configuration using a specified name.

? / Name for new view port configuration: Enter a name or ?

Entering? Lists the saved view port configurations.

Restore

Restores a previously used view port configuration.

? / Name of view port configuration to restore: Enter a name or ?

Delete

? / Name of view port configuration to delete: Enter a name or ?

REVSURF

REVSURF creates a revolved surface about a selected axis.

From Draw menu, select Surfaces > Revolved Surfaces

At the command: prompt, enter REVSURF

REVSURF constructs a polygon mesh approximating a surface of revolution by rotating a path or profile about a selected axis.

Command: REVSURF

Select path curve: Select a line, circle, or 2D or 3D polyline

Select axis of revolution: Select a line or open 2D or 3D polyline

The path curve is swept about the selected axis to define a surface.

Start angle < 0 >: Enter a value or press Enter

Included angle (+ = ccw, - = cw) < Full circle> : Enter a value or press Enter

3D

3D creates three-dimensional polygon mesh objects

From Draw menu, select Surfaces > 3D surfaces

At the command prompt, enter 3D.

The three-dimensional polygon mesh objects 3D constructs look like wire frame objects but have surfaces. Thus, 3D objects can be shaded or rendered to appear as solid objects.

Command: 3D

Selecting 3D objects from the menu displays the 3D dialog box. . Entering 3D on the command line displays the following prompt:

Box / Cone / Dish/ Mesh/ Pyramid / Sphere / Torus / Wedge: Enter an option

Box : Creates a 3D box polygon mesh.

Corner of Box: Specify a point

Length: Specify a distance

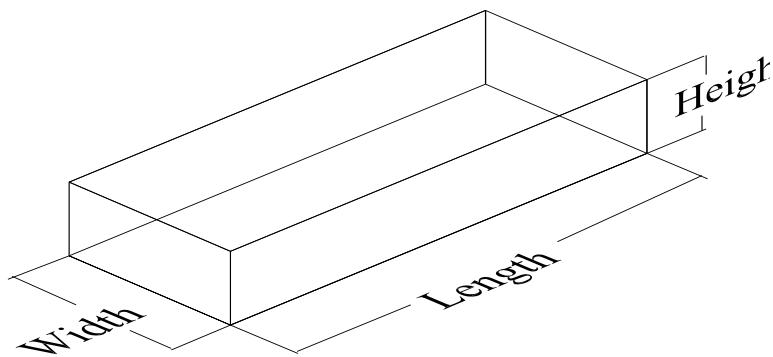
Cube / <Width >: Specify a distance or Enter C

Width: Defines the width of the box

Height: specify a distance

Rotation angle about Z axis: Specify an angle

The base point for the rotation is the first corner of the box.



Cube

Creates a cube using the length for the width and height of the box

Rotation angle about Z axis: Specify an angle

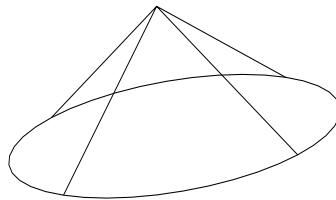
The base point for the rotation is the first corner of the box

Cone

Creates a cone –shaped polygon mesh.

Base center point: Specify a point

Diameter < radius > of base: Specify a distance or Enter D



Sphere

Command: Sphere

Centre of sphere <0,0,0>: Specify a point or press Enter

Diameter / <Radius > of sphere: Specify a distance or enter d

Torus

Torus creates a donut shaped solid

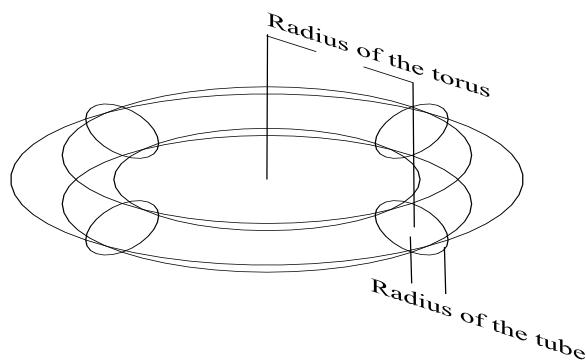
At the command prompt, enter TORUS or TOR

A torus is defined by two radius values, one for the tube and the other from the centre of the torus to the centre of the tube.

Command: TORUS

Center of torus <0,0,0 >: Specify a point or press Enter

Diameter / <Radius > of Torus: Specify a distance or enter d



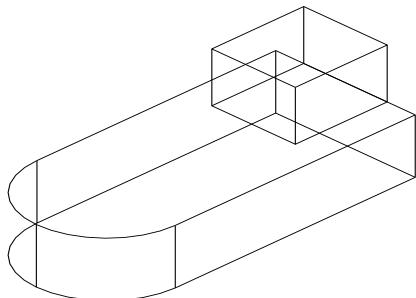
UNION

UNION creates a composite region or solid by addition

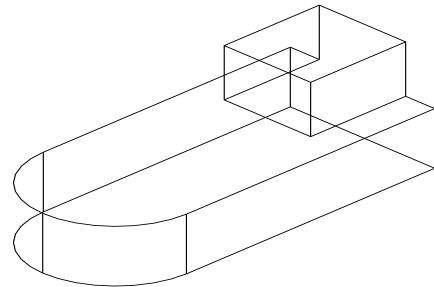
At the command prompt, enter UNION or UNI

Command: UNION

Select objects: Use an object selection method



Solid Objects selected



Solid Objects United

SUBTRACT

SUBTRACT creates a composite region or solid by subtraction

At the command: Prompt, enter SUBTRACT or SU

Command: SUBTRACT

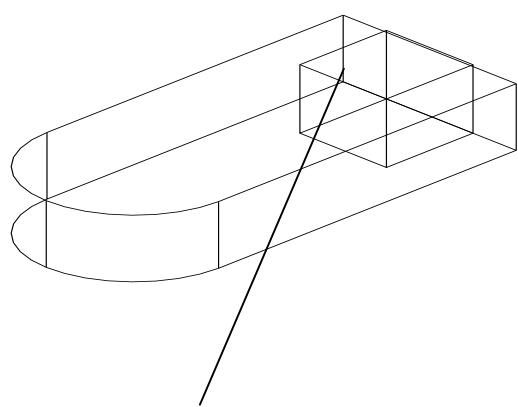
Select objects: Use an object selection method

Select objects: Press Enter

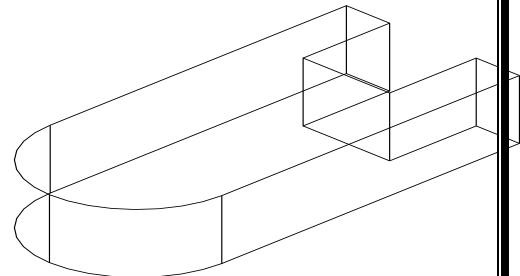
Select solids and regions to subtract

Select object: Use an object selection method.

Select object: press Enter



Solid object Selected



Solid object subtracted

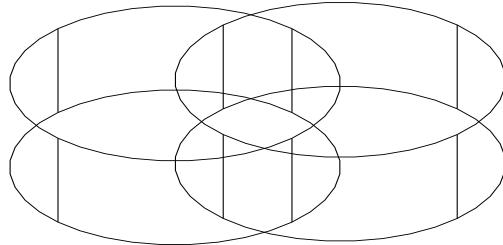
INTERSECT

INTERSECT creates composite solids or regions from the intersection of two or more solids or regions

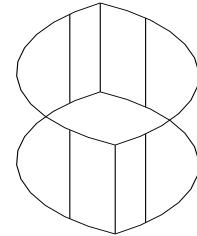
At the command prompt, enter INTERSECT or INT

Command: INTERSECT

Select objects: Use an object selection method



Two Cylinders



Intersected part

Procedure for drawing Isometric Views:

1. Change the cursor position to an Isometric plane.
2. From the tool menu, choose drafting settings.
3. In the drafting setting dialogue box, put ON the Snap & Grid tab under snap type and style, and select Isometric snap.
4. Choose O.K.
5. Cursor is changed to an Isometric plane.
6. We can cycle through the three Isometric planes by pressing F5.
7. AutoCAD through the Iso-plane top, Iso-plane right and Iso-plane left.

8. VIEWING IN THREE DIMENSIONS

Aim: To view the drawings in three dimensions.

Description: VPOINT

VPOINT sets the viewing direction for a three dimensional visualization of the drawing.

Procedure:

From menu, select 3D Viewpoint

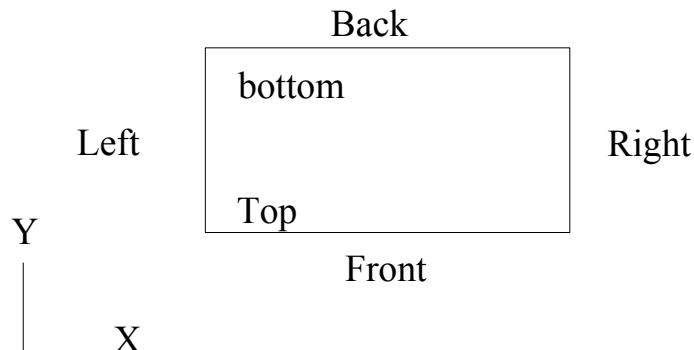
At the command prompt, enter VPOINT.

Command: VPOINT

Rotate / < View point > < Current>: Enter R, specify a point , or press to display a compass and axis tripod.

Viewpoint

AutoCAD follows some convention regarding the sides of a model. As shown in the figure assuming you are standing on top of the model facing along the +ve Y direction, the following directions are defined.



+ ve X	: Right side
- ve X	: left side
+ ve Y	: Back side
- ve Y	: Front side
+ ve Z	: Top side
-ve Z	: Bottom side

According to this rule, if you want to see the front side of the model, you need to walk along – ve Y-axis. Based on this rule, you can derive the VPOINT.

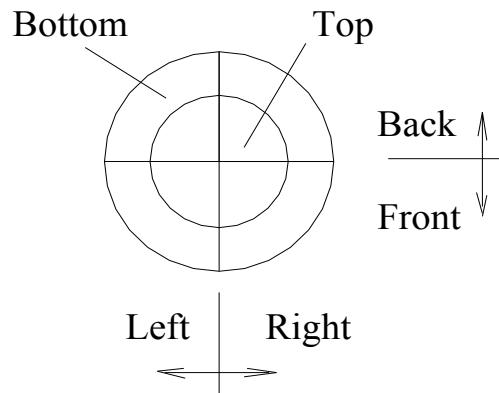
V Point Value	View
0, 0, 1	Top View
0, 0, -1	Bottom View
0, -1, 0	Front View
0, 1, 0	Back View
1,0, 0,	Right View
-1, 0, 0	Left View
1,-1, -1	Right, front ,Bottom View
-1,-1, -1	Left, front, Bottom View
1,1, -1	Right, Back, Bottom View
-1,1, -1	Left, Back, Bottom View
1,-1, 1	Right, Front, Top View
-1,-1, 1	Left, Front, Top View
1,1, 1	Right, Back, Top View
-1,1, 1	Left, Back, Top View

Compass & Tripod

Pressing enter displays a compass and axis tripod, which you use to define a viewing direction in the view port .

To select a viewing direction, move the pointing device to a location on the globe and press the pick button.

With reference to the compass, the resulted view depends on the where you select the view point on the compass.



- Any where to the right of the vertical line gives Right side view.
- Any where to the left of the vertical line gives left side view
- Any where above the horizontal line gives Backside view
- Any where below the horizontal line gives Front side view
- Any where inside the small circle gives Top view
- Any where in between small and large circles gives Bottom view

9. REMOVAL OF HIDDEN LINES - SHADING AND RENDERING

Aim: To remove the hidden lines. To apply shading on the model and also to render the model.

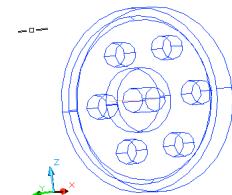
Description:

2D Wireframe

Displays the objects using lines and curves to represent the boundaries. Raster and OLE objects, linetypes, and linewidths are visible. Even if the value for the COMPASS system variable is set to 1, it does not appear in the 2D Wireframe view.

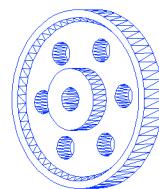
3D Wireframe

Displays the objects using lines and curves to represent the boundaries. Displays a shaded 3D UCS icon. Raster and OLE objects, linetypes, and linewidths are not visible. You can set the COMPASS system variable to 1 to view the compass. Material colors that you have applied to the objects are shown



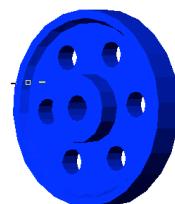
Hidden

Displays the objects using 3D wireframe representation and hides lines representing back faces.



Flat Shaded

Shades the objects between the polygon faces. The objects appear flatter and less smooth than Gouraud-shaded objects. Materials that you have applied to the objects show when the objects are flat shaded.

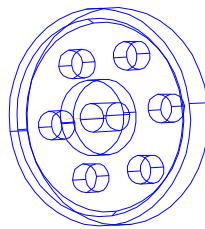


Controls the display of object shading in the current view port .

View menu: Shade

Command line: shademode

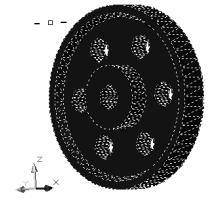
Enter option [2D wireframe/3D wireframe/Hidden/Flat/Gouraud/fLat+edges/
Gouraud +edges] <current>: Enter an option

Gouraud Shaded

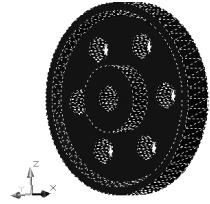
Shades the objects and smooths the edges between polygon faces. This gives the objects a smooth, realistic appearance. Materials that you have applied to the objects show when the objects are Gouraud shaded.

Flat Shaded, Edges On

Combines the Flat Shaded and Wireframe options. The objects are flat shaded with the wireframe showing through.

Gouraud Shaded, Edges On

Combines the Gouraud Shaded and Wireframe options. The objects are Gouraud shaded with the wireframe showing through.

**Procedure for Removal of hidden lines:**

1. On the view menu choose “hide” or enter the command “hide” or “hi”.
2. The lines remain hidden until you perform an action that regenerates the drawing and rediscusses a hidden line wire frame view.

Procedure for Shading:

1. Enter the command “Shade” or from the view menu choose “shade”
2. Choose one of the following option, .e. 2D wire frame/ D wireframe/ Hidden/ flat shaded/ Gouraud shaded/ flat shaded edges on/ Gouraud shaded edges on/.
3. Select the required option to shade the object.

RENDERING

View menu: Render Render

DEPARTMENT OF MECHANICAL ENGINEERING

Command line: render

Defines the scene, procedure, options, destination, sampling, and other settings for rendering.

Rendering Type

Lists Render, Photo Real, Photo Raytrace.

Scene to Render

Lists scenes, including the current view that you can select for rendering.

Rendering Procedure

Controls how RENDER behaves by default.

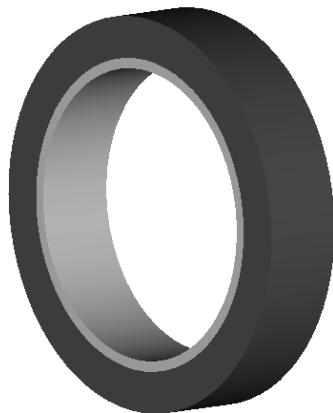
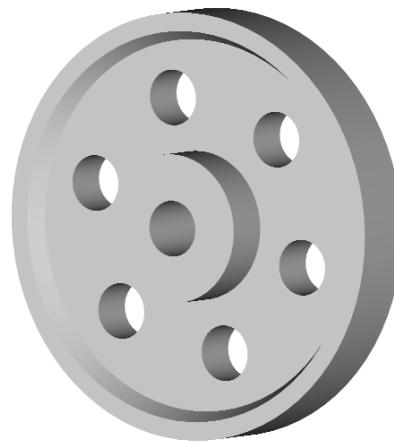
Query for Selections

Displays a prompt to select objects to render.

Crop Window

Creates a render area at render time. When you choose Render, AutoCAD prompts you to select an area on the screen before rendering proceeds. This option is available only when View port is selected under Destination.

Skip Render Dialog



Procedure for Rendering:

1. To the solid object apply the material using RMAT command.
2. To see that the particular applied material, we have to use “render”.
3. Enter the command render or RR.
4. Select the Current view.
5. Choose O.K.
6. Rendering command can also be applied for back ground and images of the object.

CAD VIVA QUESTIONS

1. What are the uses of Grid and Snap commands?
2. Write different coordinate systems and write the syntax for those coordinate systems?
3. Write diff. draw commands used for tutorial -1?
4. Write about ‘p line’ command and write the modifying command which is used to modify ‘p line’.
5. How many modes are there to draw an arc? name them.
6. Write diff modes to draw a circle.
7. Write the use of array command and name diff. types of array commands.
8. Write the difference between extend and trim commands.
9. How many times REDO command works at a time?
10. What is the use of B HATCH command?
11. Write the command which is used to dimension and object.
12. What is the difference between TEXT and M TEXT command?
13. What is the use of layers?
14. Write different layer properties and layer states**** and explain them.
15. Write the procedure to highlight a layer.
16. How to create a layer?
17. What is the use of Freeze?
18. How to create an isometric grid and oblique?
19. What is the diff. between Orthographic, Isometric and Projection?
20. What keys are used to select the isoplanes cursor?
21. Write different 3D geometric models used in Auto CAD?
22. What is the use of Extrude command?
23. What is the use of Revolve command?
24. Write diff. modifying commands used in solid modelling.
25. What do you mean by V Port? What is the use of it?
26. Write the difference between sweep and Extension command.
27. What is meant by B-rep?
28. What are the different display controls used?
29. What are diff. Graphic standards?
30. What is the use of limits command?
31. Write diff. editing commands?
32. What are the diff. selection methods?

33. What is the diff. between copy and mirror commands?
34. Explain about ‘explode’ command?
35. Explain about ‘block’ command. What are the uses and how to create it?
36. Explain about ‘W block’ command?
37. What is the one of DDedit command?
38. Write about X ref. Command. What is the diff. between insert and X ref. commands?
39. Write about Purge command.
40. Write about Donut command.
41. What do you mean by Aperture and pick box?
42. What is the diff. between Annotation and Attributes?
43. Write about Grip, ddgrip, grip color, grip hot.
44. What are the diff. extensions used for a file?
45. Write differences between copy and copy clip commands.

CAM LAB EXPERIMENTS

1. PREPARATION OF MANUAL PART PROGRAMMING FOR CNC TURNING/MILLING

MANUAL PART PROGRAMMING FOR CNC LATHE

Program in incremental mode

Tool Offset T1 X-0.3 Z 0

G92 X0 Z0

N1 G01 X-13 F80 T1

N2 Z-4

N3 X4 Z-5

N4 Z-5

N5 X-2 Z-2

N6 Z-10

N7 X10 Z-5

N8 Z-3

N9 X-2 Z-2

N10 X2 Z-2

N11 Z-6

N12 G0 X1

N13 Z44 T0

N14 M30

EXAMPLE 1: MILLING

[BILLET L 170 W 90 D 10

[EDGEMOVE X-70 Y-30

[TOOLDEF T1 L 65 D6

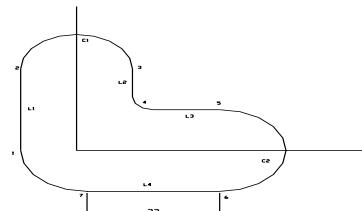
[SET DATUM X 20 Y20

N1 G00X0Y-18.4 Z5

N2 M03 S1000

N3 G01 Z-3

N4 GO1 X32

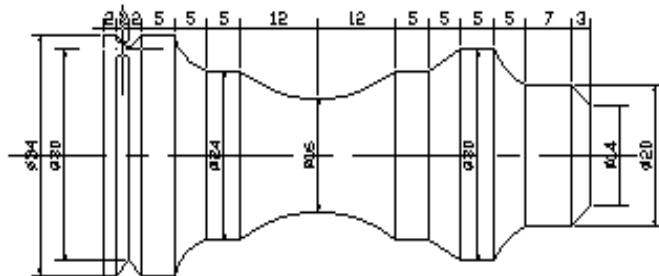


N5 G03 X32 Y18.4 R18.4
N6 G01 X18.4
N7 G01 X18.4 Y32
N8 G03 X-18.4 R18.4
N9 G01 Y0
N10 G03 X0 Y-18.4 R18.4
N11 G00 X0 Y0 Z5
N12 M05

EXAMPLE 2: MILLING

[BILLET L 170 W 90 D 10
[EDGEMOVE X-60 Y-55
[TOOLDEF T1 L 65 D6
[SET DATUM X 60 Y45
N1 G00X-30Y-43 Z5
N2 M03 S1000
N3 G01 Z-3
N4 G01 X30
N5 G03 X41.21Y-26.51 R13
N6 G01 X14.78 Y11.98
N7 G03 X-14.782 Y11.98 R18
N8 G03 X-40.25 Y-26.51
N9 G03 X-30 Y-43 R13
N10 G00 X0 Y0 Z5
N11 M05

2. PART PROGRAMMING PREPARATION THROUGH AUTOCAD



Program in incremental mode

```
Tool Offset T1 X-0.3 Z 0
G92 X0 Z0
N1 G01 X-11 F100 T1
N2 Z-1
N3 X3 Z-3
N4 Z-7
N5 G02 X5 Z-5 15 K0
N6 G01 Z-5
N7 X-3 Z-5
N8 Z-34
N9 G2 X5 Z-5 15 K0
N10 X2 Z-5
N11 X-2 Z-2
N12 X2 Z-2
N13 Z-2
N14 G0 X11
N15 Z 53
N16 G2 X0 Z-40 10 K -20
N17 G0 X0 Z 63 T0
N18 M 30
```

3. APT SYSTEM

APT GEOMETRY : Any part geometry can be built up from the basic geometric elements. A few geometric elements are defined below are.

Point definitions in APT

Syntax: <symbol> = POINT / <parameter listing>

1. POINT defined by rectangular coordinates

P1 = POINT / 50, 75, 70

P2 = POINT / 50, 25

2. POINT defined by intersection of two lines

P2 = POINT / INTOF, L1, L2

3. POINT by intersection of a line and a circle

POINT/XLARGE }
 XSMALL }
 YLARGE } NTOF, line, circle
 YSMALL

P3 = POINT / XSMALL, INTOF, L, C

4. POINT by intersection of two circles

POINT/XLARGE }
 XSMALL }
 YLARGE } NTOF, line, circle
 YSMALL

P4 = POINT / 25

C1A = CIRCLE / YSMALL, INTOF, C1, C2

Line definitions in APT

1. LINE passing through two points

LINE/ point 1, point 2

L1 = LINE / PT1, PT2

- LINE/ x1,y1, x2, y2
- L1B = LINE / 20, 30, 70, 60
- 2. LINE passing through a point and parallel to a line
LINE/point, PARREL, line
L3 = LINE / P3, PARREL, L1
- 3. LINE passing through a point and perpendicular to a line
LINE/ point, PERTO, line
L4 = LINE/P4, PERPTO, L1
- 4. LINE passing through a point and tangent to a circle
LINE / point {LEFT
RIGHT, TANTO, circle
L8 = LINE / PT, LEFT, TANTO, CR
L8A = LINE / PT, RIGHT, TANTO, CR}
- 5. LINE tangential to two circles
LINE /point LEFT {
RIGHT, TANTO, circle1, } {
RIGHT, TANTO, circle2
L9 = LINE / LEFT, TANTO, C1, RIGHT, TANTO, C2
L8A = LINE / RIGHT, TANTO, C1, RIGHT, TANTO, C2

CIRCLE DEFINITIONS IN APT

Syntax: <symbol> = CIRCLE / <parameter string>

- 1. CIRCLE defined by its center and radius
CIRCLE / x, y, radius
CIRCLE / CENTRE, point, RADIUS, radius
C1 = CIRCLE / 75, 50, CENTRE, PT1, RADIUS, 45
- 2. CIRCLE defined by its centre and tangent to a line
CIRCLE / CENTRE, point, TANTO, line
C2 = CIRCLE / CENTRE, PT2, TANTO, L1
- 3. CIRCLE defined by its centre and a point on the circumference
CIRCLE / CENTRE, point1, point2
C3 = CIRCLE / CENTRE, P3, PT3
- 4. CIRCLE passing through three points

CIRCLE / point1, point2, point3

C4 = CIRCLE / PT4, PT5, PT6

5. CIRCLE of known radius tangential to two lines

CIRCLE { XLARGE
XSMALL
YLARGE
YSMALL, line1, \$

{ XLARGE
XSMALL
YLARGE
YSMALL, line2, RADIUS, radius

C6 = CIRCLE / YLARGE, L2, YSMALL, L2, RADIUS, 15

C6A = CIRCLE / YLARGE, L1, XSMALL, L2, RADIUS, 15

6. CIRCLE of known radius and tangent to circle

CIRCLE { XLARGE
XSMALL
YLARGE
YSMALL, { IN
OUT, circle1, { IN
OUT, circle2, RADIUS, radius

C8A = CIRCLE / YLARGE, OUT, C1, IN, C2, RADIUS, 15

C81 = CIRCLE / YLARGE, IN, C3, OUT, C4, RADIUS, 50

Plane definitions in APT

Syntax: <symbol> = PLANE / <parameter string>

1. PLANE passing through three points

PLANE / point1, point2, point3

PL1 = PLANE / P1, P2, P3

MOTION COMMANDS

They are used to describe the actual machining sequence making use of geometric statements defined. The motion commands are divided into 3 groups.

1. Set up commands.
2. Point-to-point commands.
3. Continuous path motion commands.

SET UP COMMANDS:

The set up commands are required for initiating the machining operations.

Starting point command:

To establish the starting point of the cutter for the starting motion.

FROM / POINT

FROM / x, y or FROM / x, y, z

Cutter command:

To describe the cutting tool.

CUTTER / d or CUTTER / d, r

Surface commands:

The desired path of the 3-dimensional cutting tool is described by means of three interesting surfaces. These surfaces are

Part surface (PS)

Drive surface (DS)

Check surface (CS)

The part surface is the surface which is in continuous contact with the top of the tool.

Drive surface is the plane along with the cutter is in machining contact.

CS is the surface which limits the motion of the tool.]

POINT TO POINT MOTION COMMANDS :

Position in APT is achieved with two statements GOTO and GODLTA

GODLTA : incremental motion :

GODLTA / dx, dy, dz if one value is given with GODLTA,

GODLTA /dz it is implied for Z value.

GODLTA / VECTOR.

GOTO : Absolute movement command

GOTO/ point

GOTO/ x,y or GOTO/ z,y,x

When no z value is specified , it takes from ZSURF.

GOTO can also be used for position ing at PATTERN

GOTO / pattern

The type of machining is specified by POST PROCESSOR commands preceding GOTO statement.

CONTINUOUS PATH MOTION COMMANDS:

Part surface command:

Part surface is specified by PSIS

PSIS / PL (PL is a plane defined earlier)

Part surface can be established.

Parallel to X, Y plane at current Z height by AUTO PS.

NOPS (is a non-model command).

To ignore a part surface which is already in force, NOPS is used before Startup command and the original part surface is restored after the startup motion.

START UP command:

Before proceeding to start machining, the cutting tool should bring in contact with the part and drive surface within specified tolerance.

i) 3 surface startup command:

$$\text{GO} / \left\{ \begin{array}{l} \text{TO} \\ \text{ON, ds} \\ \text{PAST} \end{array} \right\} \left\{ \begin{array}{l} \text{TO} \\ \text{ON, ps,} \\ \text{PAST} \end{array} \right\} \left\{ \begin{array}{l} \text{TO} \\ \text{ON, cs} \\ \text{PAST} \end{array} \right\}$$

ii) 2 surface startup command:

$$\text{GO} / \left\{ \begin{array}{l} \text{TO} \\ \text{ON, ds} \\ \text{PAST} \end{array} \right\} \left\{ \begin{array}{l} \text{TO} \\ \text{ON, ps,} \\ \text{PAST} \end{array} \right\}$$

iii) 1 surface startup command:

$$\text{GO} / \left\{ \begin{array}{l} \text{TO} \\ \text{ON, ds} \\ \text{PAST} \end{array} \right\}$$

If no terminal modifier is specified, the default is TO. The end point of tool is always in contact with part surface, so only TO is meaningful with part surface.

To avoid ambiguity (if any) direction control commands are used.

INDIRP / point

INDIRV / vector

Tool relationship commands:

TLRGT (tool is on right side of drive surface)

TLLFT (tool is on left side of drive surface)

TLON (tool is on the drive surface)

Tool relationship should be mentioned before motion commands.

ACTION verbs:

The direction of every motion of the tool have to be specified with respective to its previous motion direction. The following six action verbs are used for this purpose.

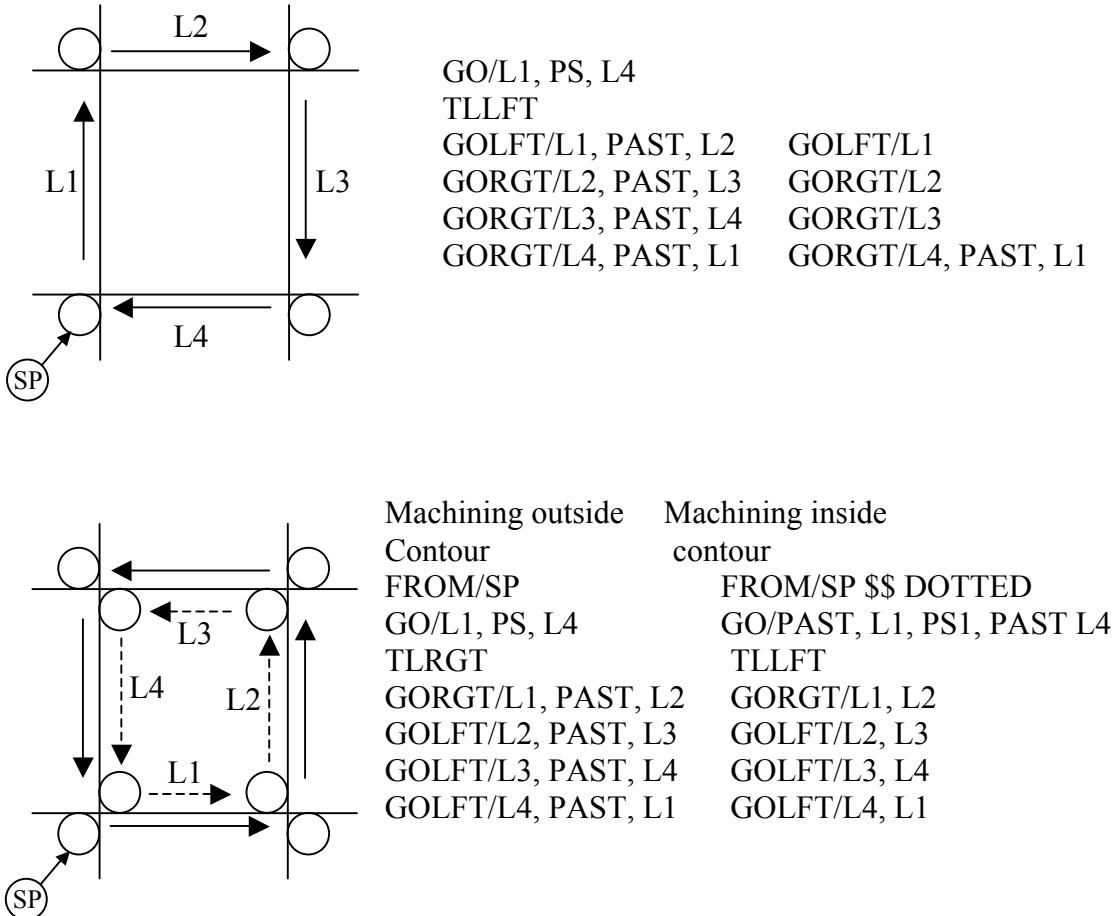
1. GO LFT / Move left along the drive surface.
2. GO RGT / Move right along the drive surface.
3. GO FWD / Move forward from a tangent position.
4. GO BACK / Move backward from tangent position.
5. GO UP / Move up along the drive surface.
6. GO DOWN / Move down along the drive surface.

To describe any motion, one of the above six action verbs are required.

Action verbs define direction w.r.t. the previous motion.

Example of simple contour:

Check surface is implied when the check surface becomes the drive surface in the subsequent command.

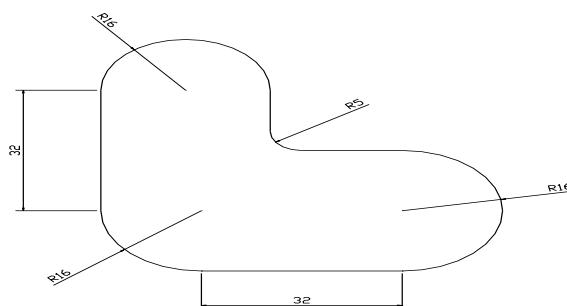


POST PROCESSOR COMMANDS

- PARTNO/ is used to indicate the name of the part program.
PARTNO/ part details.
- MACHIN/ is used to specify the machine tool.
ACHIN/ RILL, 4 to specify the 4th NC drill in the cell.
- COOLNT/ is used to turn coolant ON or OFF.
OOLNT/ MIST to turn on the air coolant.
OOLNT/ FLOOD to turn on the oil coolant.
OOLNT/ OFF to turn coolant off.
- FEDRAT/ is used to specify the feed rate of cutter motion.
PM f inch per minute
MPM f mm per minute
EDRAT/ IPR, f f inch per revolution
MPR f mm per revolution
- RAPID is used to indicate the rapid speed (similar to G00).
- SPINDL/ is used to specify the speed of the spindle.
SFM n surface feed feet per minute
SPINDL/ RPM, n, CLW n revolutions per minute
SMM CCLW n surface meters per minute
- CUTCOM/ is used for inclusion of cutter compensation.
ON
CUTCOM/ OFF, LENGTH, a a refers the register associated
RIGHT RADIUS with compensation
LEFT

CYCLE/	is used to generate canned cycles for drilling/reaming/tapping, etc.
	DRILL similar to G81 fixed cycle
	FACE IPM similar to G82 fixed cycle
CYCLE/	TAP, z, IPR, f, r similar to G84 fixed cycle
	BORE MMPR similar to G86 fixed cycle
	REAM MMPM similar to G85 fixed cycle
	z is the dept, f is the feed rate and r is the rapid plane.
CYCLE/	ON actuates the previous cycle.
	OFF suppress the cycle command (similar to G80)
LOADTL/	is used to load tool of specified number from the specified tool magazine
	position.
	LOADTL/ tool no., magpos
THREAD/	is used to activate the threading mode.
	FACE
THREAD/	TURN, INCR, a, TPI, p, DEPTH, d
	TAPER DECR MMPR
	a indicates the rate that which pitch of the thread changes. For constant pitch threads this can be omitted.
	p is the pitch in threads per inch or mm per revolution.
PRINT/	is used to print required information on the line printer.
	PRINT/ ON or PRINT/ALL
	PRINT/ OFF
CLPRINT/	is used to print the cutter location data.
	CLPRINT/ ON
	CLPRINT/ OFF
REMARK/	is used to add comments for clarity of the program.
	REMARK/ comment
FIN/	is used to indicate the finishing of the program (similar to END command).

PROGRAM EXAMPLE



PART NO/ MILLING JOB NO.1

REMARK PROGRAMMER – B.SATYANARAYANA

MACHINE / STARMILL

REMARK / GEOMETRICAL DEFINITIONS

CLPRINT / ON \$\$ prints CL data.

REMARK / GEOMETRIC STATEMENTS

SP=POINT/-4.0, -4.0
P1= POINT/-2.0
Q1=POINT/0, 0
Q2=POINT/4, 0
Q3=POINT/0,4.0,6.0
CL12=LINE/XAXIS
CL13=LINE/YAXIS
L1=LINE/PARREL, CL13, XSMALL, 2.0
L2=LINE/PARREL, CL13, XLARGE, 2.0
L3=LINE/PARREL, CL12, YLARGE, 2.0
L2=LINE/PARREL, CL12, YSMALL, 2.0
C1=CIRCLE/CENTRE, Q3, RADIUS, 2.0
C2=CIRCLE/CENTRE, Q2, RADIUS, 2.0
C3=CIRCLE/CENTRE, Q1, RADIUS, 2.0
Q11=POINT/0,0,-5.0
Q21=POINT/4,0,-5.0
Q31=POINT/0,4.06,-5.0
PS=PLANE/Q11, Q21, Q31

REMARK / MOTION STATEMENTS

FROM/SP
CUTTER/16
TOLER/0.05
SPINDL/RPM, 600, CLW
COOLNT/ON
FEDRAT/MMPM, 100
GO/TO, L1, T0, PS, TANTO, C3
AUTO PS
GODLT/-10.0
TLLFT
GOFWD/L1, TANTO, C1
GOFWD/C1, TANTO, C2
GOFWD/L2, L3
GOFWD/L3, TANTO, C2
GOFWD/C2
GOFWD/L4
GOFWD/C3, TANTO, L1
GOFWD/10.0
RAPID
GOTO/SP
COOLNT/OFF
SPINDL/OFF
CLPRINT/OFF
END
FINI

4. MACHINING OF ONE JOB ON CNC MACHINE TOOL

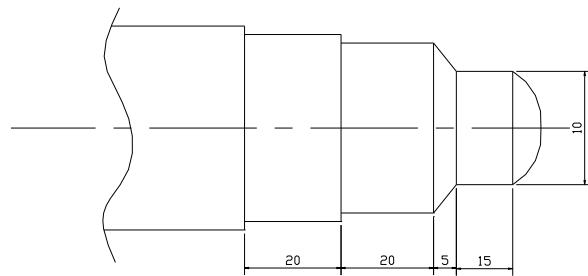


FIG NO : 1

EXAMPLE 1

[BILLET X 20 Z 80;

G28

M06 T03 ;

M03 S1500 F90 ;

G00 X20 Z1 ;

G71 U0.2 R0.5 ;

G71 P10 Q 20 U0 W0 ;

N10 G01 X0 ;

Z0;

G03 X10 Z-5 R5

EXAMPLE NO: 2

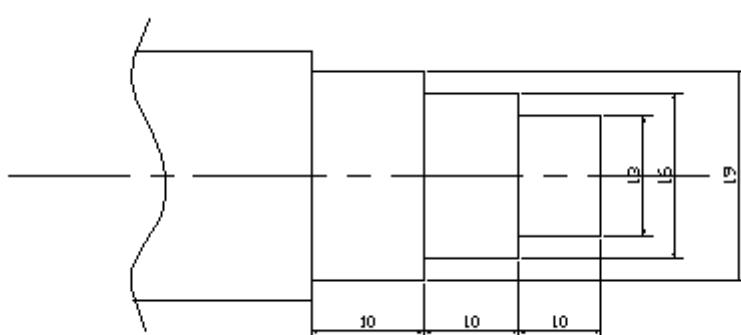


FIG NO : 2

```
[ BILLET X 20 Z 60;  
M06 T03 G28 ;  
M03 S1500 F90 ;  
G00 X21 Z1 ;  
G71 U0.2 R0.5 ;  
G71 P10 Q 20 U0.1 W0 ;  
N10 G01 X0 ;  
Z0;  
G01 X13 Z0 R0 ;  
G01 Z-10 ;  
G01 X16 Z-10 ;  
G01 Z-20;  
G01 X19 Z-20 ;  
N20 G01 Z-30;  
G28 M03 ;
```

CAM VIVA QUESTIONS

1. What is NC part programming?
2. In NC part programming “BLOCK” means?
3. In NC part programming data contains in the form of?
4. How many types NC part programming is prepared?
5. Manual part programming is prepared on?
6. Give one example to prepare NC part programming using computer language?
7. What are G, N, and M codes?
8. Explain each term given below NC program
N001 G02 X023 Y024 R5 M03
9. For circular interpolation which preparatory code is used?
10. How many number of “ dimension numbers” required for NC part programming?
11. How many digits used to represent feed word?
12. How many digits used to represent spindle speed word?
13. How many digits used to represent tool word?
14. What is magic-three code?
15. What is different between G00 and G01?
16. What is different between G90 and G91?
17. What is cutter compensation? Which codes are used for cutter compensation?
18. Explain canned cycle or fixed cycle? Which codes are used for cycle or fixed cycle?
19. Expand BLU?
20. Explain different types of tool paths used in NC part programming?
21. Explain co-ordinate system for CNC turning?

22. Explain co-ordinate system for CNC milling?
23. Expand and explain APT?
24. How many types of statements used APT programming?
25. Explain and classify motion statements in APT programming?
26. How many types point, circle and line represented in APT programming using geometrical statements?
27. Explain point-to-point motion statement and sketch?
28. Explain contour motion statement with sketch?
29. What is postprocessor in APT programming?
30. What is CNC?
31. What is DNC?
32. For plane selection which preparatory codes used?
33. Expand and explain ATC?
34. What is difference between incremental and absolute system?
35. What is difference between absolute zero and floating zero co-ordinate system?
36. Explain difference between point-to-point, linear and contour interpolation?
37. G04 represents which operation in NC part programming?
38. Explain difference between NC, CNC, and DNC?
39. Which preparatory code is used for tool nose radius compensation in turning?
40. What are different types of CNC controllers is used in now days?
41. Polar co-ordinate systems are CNC programming or not?
42. Explain High speed machining?
43. What are functions of adaptive control system in CNC machines?
44. Explain different parts in NC machines?
45. Explain different steps from design to manufacturing using NC machines?
46. What is CNC turning center?
47. What is difference between ATP and ATC?