

LAB MANUAL FOR Engineering Drawing and CAD Sessional



**University of Global Village
DEPARTMENT OF MECHANICAL ENGINEERING**

COURSE INFORMATION

CREDIT	1
ASSESMENT	50
COURSE CODE	ME 0715-2102

CLOs	
1. Constructing geometry using basic modify tools in AutoCAD.	CLO 1
2. Utilizing tools such as Erase, Copy, Move, Rotate, Scale, Align, Offset, Mirror, Stretch, Lengthen, Break, Trim, and Extend to create complex geometric shapes.	CLO 2
3. Generating orthographic projections of simple engineering parts.	CLO 3
4. Generating Isometric projections of simple engineering parts.	CLO 4

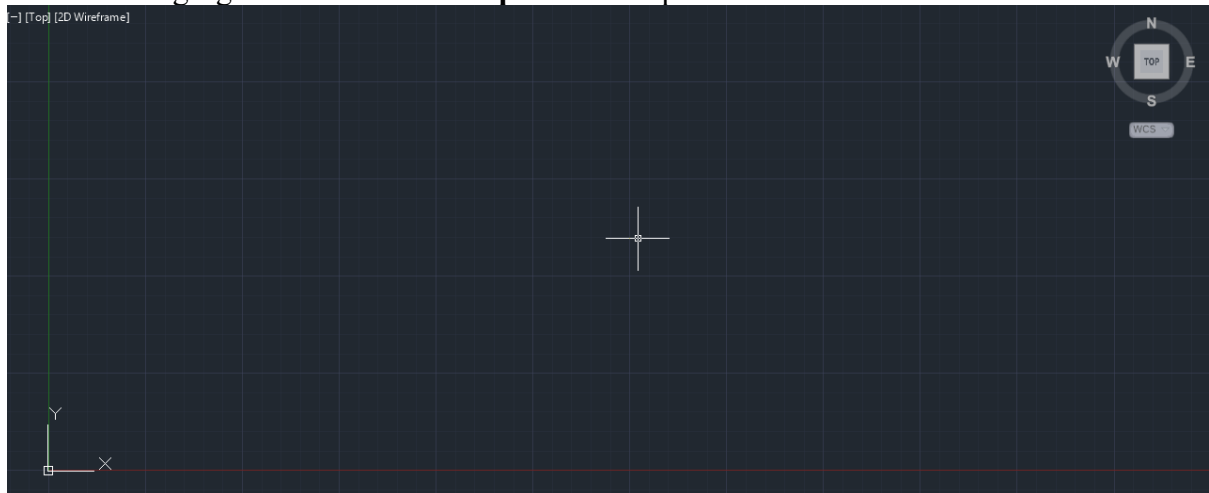
Sl. No.	Content	Week
1	Introduction to AutoCAD , unit, line	
2	Creates a 2D polyline, a single object that is composed of line and arc segments, circle.	
3	Introduction to polyline, polygon, ellipse	
4	Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill	
5	To construct a geometry using basic modify Tools like Erase, Copy, Move, Rotate, Offset, Mirror	
6	Introduction to stretch, break, trim, extend, explode.	
7	Develop geometry of planar machine parts by using Advanced Editing Commands like Fillet, Chamfer, Array	
8	Draw the orthographic views of 3D machine parts and organizing your work by dimensioning, managing with layers	
9	Exercise	

LAB SESSION 1

Introduction to AutoCAD , unit, line

Instructions:

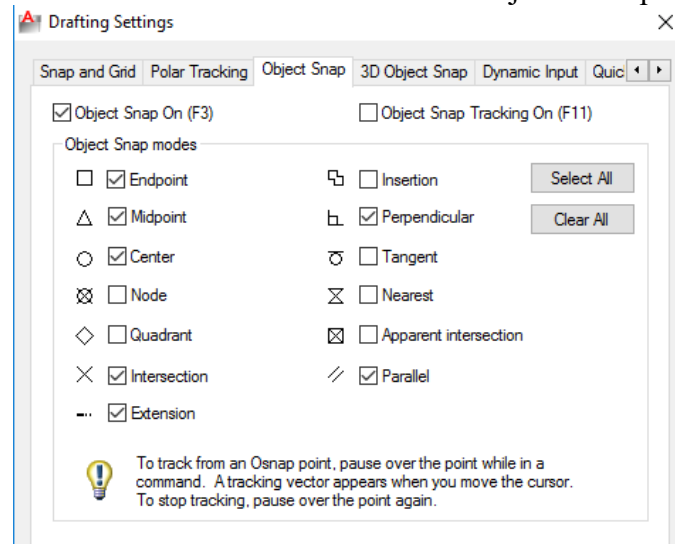
1. Open **AutoCAD** by double clicking on its icon situated somewhere on the desktop.
2. The following figure shows the **workspace**. Workspace is the area where a user can work.



The workspace is nothing but an **XY Cartesian Plane**. X-Axis is the horizontal axis while Y-axis is the vertical axis. Any location on this plane may accessed using XY coordinates of that location.

3. The following figure shows the **command history** (Top) and **command input line** (bottom) where the user may see the previous commands and enter various news commands through a keyboard. This command line can be used to access various tools, settings, preferences etc.
4. Before starting any work certain settings should be checked and modified if necessary. The settings include the Coordinate Entry Settings, Units to be used, Grid size, Object Snaps etc.
 - a. **Units:** User can select from a variety of units to be used in his/her engineering drawing which he/she is going to produce using AutoCAD. In the Format menu, click Units to open 'Drawing Units' window. In the sections of Length and Angle, user can select the type of units and precision, while from the section of the Insertion Scale appropriate units may be selected.
 - b. **Object Snap:** Object snap options are very useful when working on drawing. These options are in fact aids for the user so that he/she can access certain points instantly. For example if the user want to access the end-point of a certain line and want the new line to start from this point, then he/she has to know exact coordinates of the

point otherwise it won't be possible for him/her to select exactly the required point. Object snap here helps the user by highlighting and attaching the cursor with the end-point if 'end-point snap' is turned on. User may turn on and off various snaps available from the tab of Object Snap in the drafting settings.



Along with the option of turning snaps on/off a symbol is also presented that will be shown when the cursor has been snapped to a particular object.

5. Once aforementioned settings have been understood, the user can now start drawing. This lab manual will include the introduction and use of the most basic commands that are used frequently when working with AutoCAD. Various drawing tools are available in the **'Draw' Toolbar** ↓




DRAW TOOLBAR

Line:

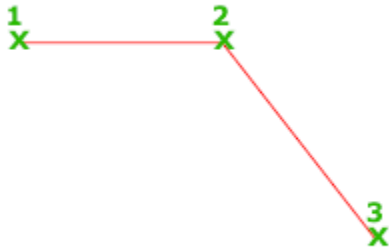
The most basic tool is the Line Tool. User can activate this tool by clicking on its icon in the 'Draw' Toolbar, or by selecting it from the 'Draw' menu or by entering alphabet 'l' (ell) in the command line and pressing spacebar. Create a series of contiguous line segments. Each segment is a line object that can be edited separately.

Access Methods

Tool Set: Drafting tab > Draw panel > Line. 

Menu: Draw > Line.

With LINE, you can create a series of contiguous line segments. Each segment is a line object that can be edited separately.



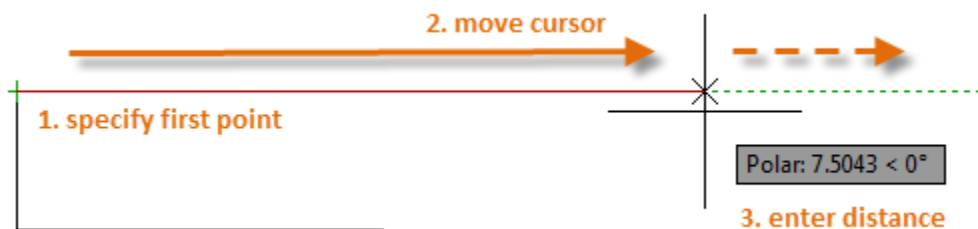
The following prompts are displayed.

Specify first point

Sets the starting point for the line. Click a point location. With object snaps or grid snap turned on, the points will be placed precisely. You can also enter coordinates. If instead, you press Enter at the prompt, a new line starts from the endpoint of the most recently created line, polyline, or arc. If the most recently created object is an arc, its endpoint defines the starting point of the line. The line is tangent to the arc.

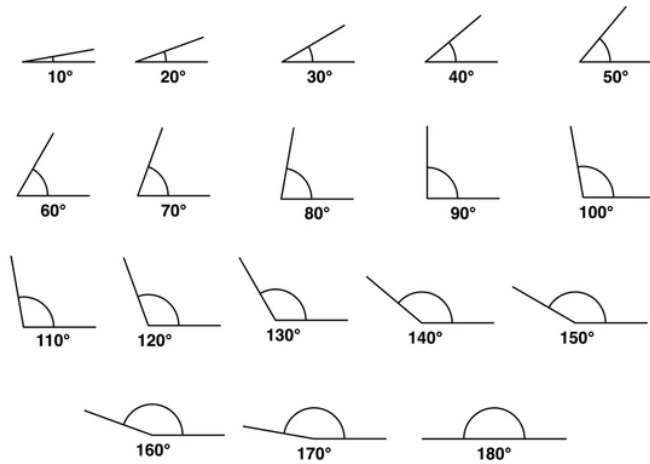
Specify next point

Specifies the endpoint of the line segment. You can also use polar and object snap tracking together with direct distance entry.



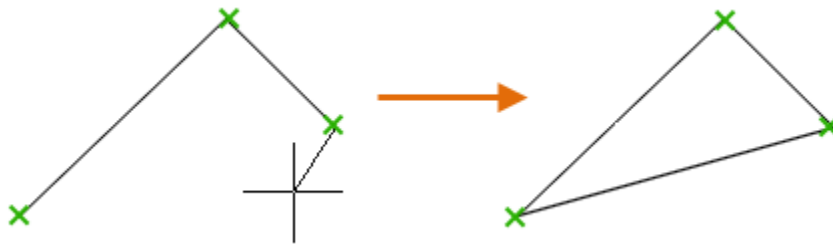
Specify angel

Specifies the angle of the line segment. Press Tab to get to the angular measurement



Close

Connects the first and last segments.



Undo


Removes the most recent segment of a line sequence.

LAB SESSION 2

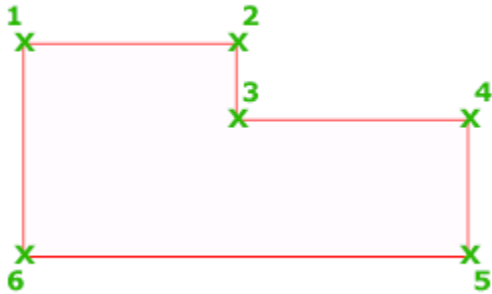
Polyline:

Creates a 2D polyline, a single object that is composed of line and arc segments, circle.

Access Methods

Tool Set: Drafting tab > Draw panel > Polyline. 

Menu: Draw > Polyline.



The following prompts are displayed.

Specify start point

Sets the starting point for the polyline.

- A temporary plus-shaped marker displays at the first point.
- Pressing Enter starts a new polyline from the last endpoint specified in creating a polyline, line, or arc.

Specify next point

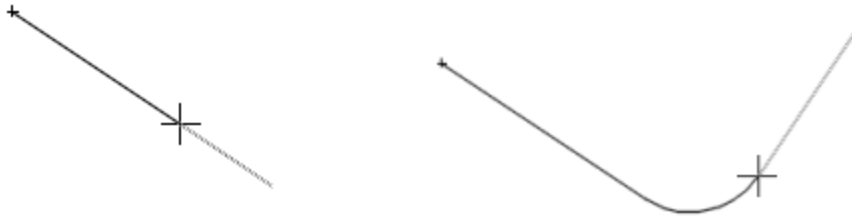
- If you specify a second point, you create straight segments.
- If you enter **a** (for Arc), you create arc segments.

Arc

Begins creating arc segments tangent to the previous segment.

Length

Creates a segment of a specified length at the same angle as the previous segment. If the previous segment is an arc, the new line segment is tangent to that arc segment.



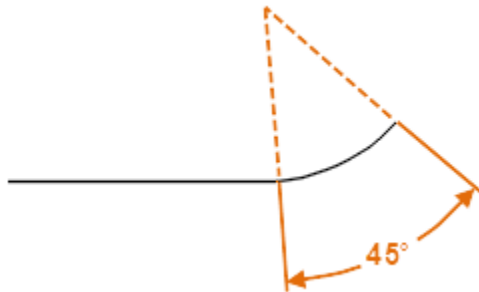
Arc-Only Prompts

Endpoint of arc

Completes an arc segment. The arc segment is tangent to the previous segment of the polyline.

Angle

Specifies the included angle of the arc segment from the start point.

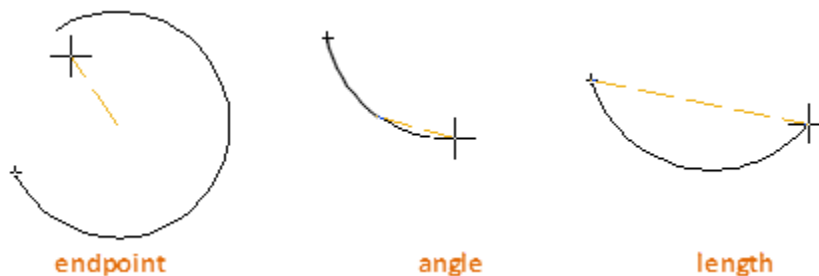


Entering a positive number creates counterclockwise arc segments. Entering a negative number creates clockwise arc segments.

Center

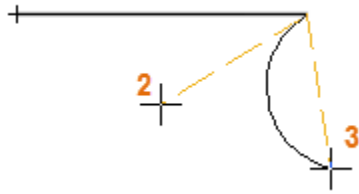
Specifies an arc segment based on its center point.

Note: For the Center option of the PLINE command, enter **ce**; for the Center object snap, enter **cen** or **center**.



Direction

Specifies the tangent for the arc segment.



- **(2) Tangent direction from the start point of the arc.** Specifies a point that establishes a tangency of the curve to the start point. The arc curves away from the vector between the start point and the tangent point.
- **(3) Endpoint of the arc.** Specifies the endpoint of the arc segment.

Tip: Press Ctrl to draw in a clockwise direction.

CIRCLE

This command is used to draw a Circle.

Access Methods

Type CIRCLE and press enter or C and press enter at “COMMAND:” prompt

Toolset: Drafting tab > Draw panel > Circle drop-down .



Menu: Draw > Circle.

The following prompts are displayed.

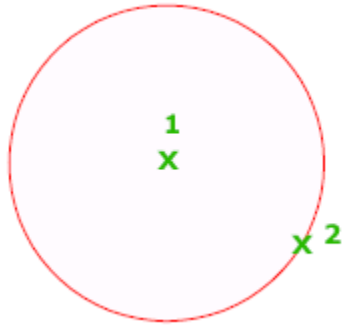
Center Point

Creates a circle based on a center point and a radius or diameter value.

Radius

Enter a value, or specify a point.

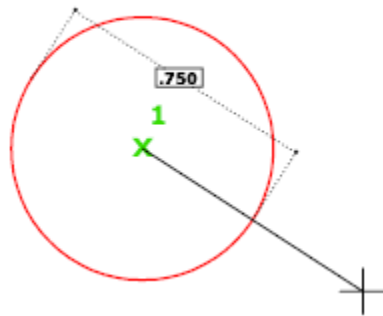
For example:



Diameter

Enter a value, or specify a second point.

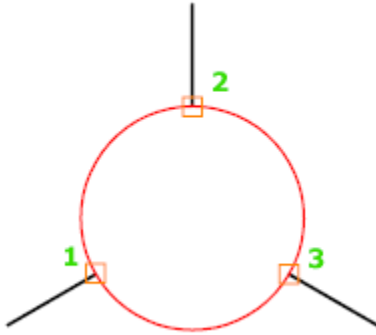
For example:



3P (Three Points) 

Creates a circle based on three points on the circumference.

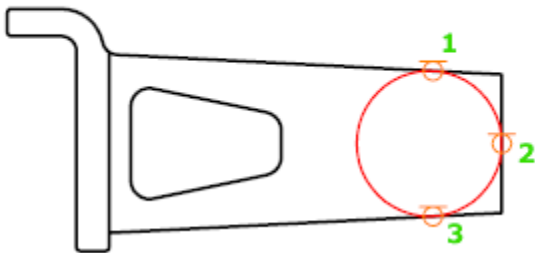
For example:



Tan, Tan, Tan 

Creates a circle tangent to three objects.

For example:

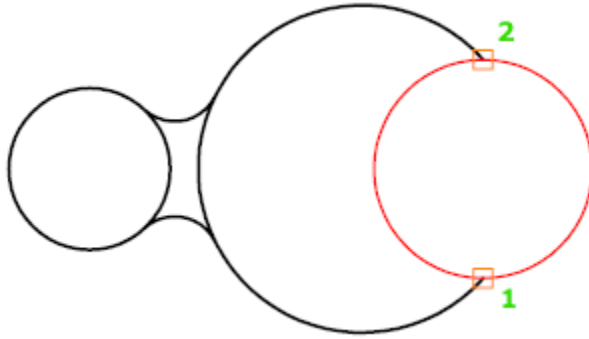


2P (Two Points)



Creates a circle based on two endpoints of the diameter.

For example:



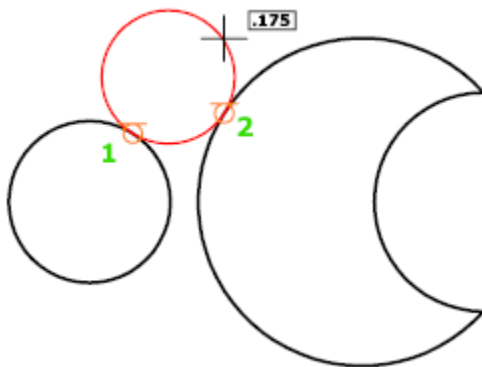
TTR (Tangent, Tangent, Radius)



Creates a circle with a specified radius and tangent to two objects.

Sometimes more than one circle matches the specified criteria. The program draws the circle of the specified radius whose tangent points are closest to the selected points.


For example:



ARC:

This tool will allow you to create arcs all the way up to 360 degrees

Access Methods

Tool Set: Drafting tab > Draw panel > Arc drop-down. 


Menu: Draw > Arc.

To create an arc, you can specify combinations of center, endpoint, start point, radius, angle, chord length, and direction values. Arcs are drawn in a counterclockwise direction by default. Hold down the Ctrl key as you drag to draw in a clockwise direction.

The following prompts are displayed.

Start point

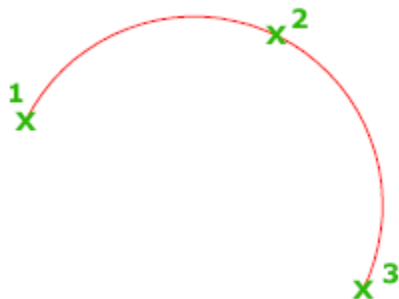
Draws an arc using three specified points on the arc's circumference. The first point is the start point (1).

Second point 

Specify the second point (2) is a point on the circumference of the arc.

End point

Specify the final point (3) on the arc.



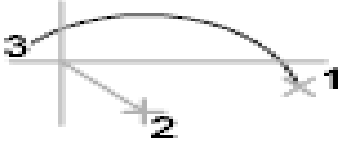
You can specify a three-point arc either clockwise or counterclockwise.

Center

Specify the second point (2) is the center of the circle of which the arc is a part.

End point

Using the center point (2), draws an arc counterclockwise from the start point (1) to an endpoint that falls on an imaginary ray drawn from the center point through the third point (3).



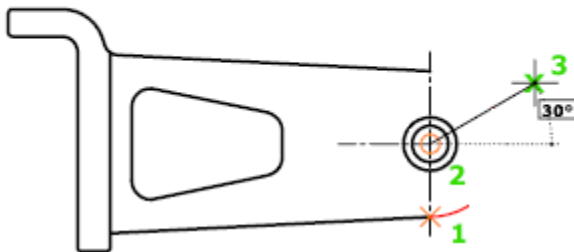
The arc does not necessarily pass through this third point, as shown in the illustration.

The distance between the start point and the center determines the radius. The endpoint is determined by a line from the center that passes through the third point.

Angle

The distance between the start point and the center determines the radius. The other end of the arc is determined by specifying an included angle that uses the center of the arc as the vertex.

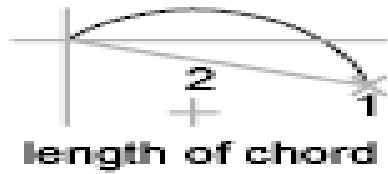
If the angle is negative, a clockwise arc is drawn.



Chord length

Draws either a minor or a major arc based on the distance of a straight line between the start point and endpoint.

If the chord length is positive, the minor arc is drawn counterclockwise from the start point. If the chord length is negative, the major arc is drawn counterclockwise.

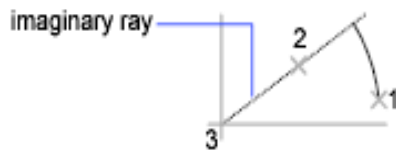


End

Specify the second point (2) is the endpoint of the arc.

Center point

Draws an arc counterclockwise from the start point (1) to an endpoint that falls on an imaginary ray drawn from the center point (3) through the second point specified (2).



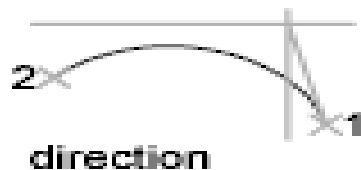
Angle

Draws an arc counterclockwise from the start point (1) to an endpoint (2), with a specified included angle. Specify the second point (2) is the endpoint of the arc. If the angle is negative, a clockwise arc is drawn.



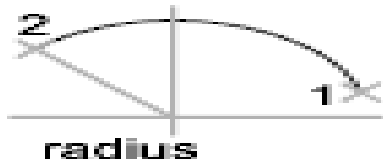
Direction

Begins the arc tangent to a specified direction. It creates any arc, major or minor, clockwise or counterclockwise, beginning with the start point (1), and ending at an endpoint (2). The direction is determined from the start point.





Draws the minor arc counterclockwise from the start point (1) to the endpoint (2). If the radius is negative, the major arc is drawn.



Center

Starts by specifying the center of the circle of which the arc is a part.

Start point

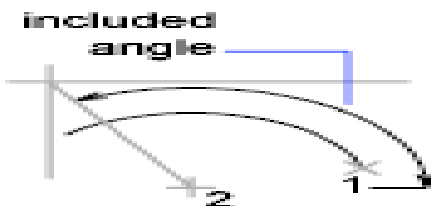
Specify start point of arc.



Draws an arc counterclockwise from the start point (2) to an endpoint that falls on an imaginary ray drawn from the center point (1) through a specified point (3).



Draws an arc counterclockwise from the start point (2) using a center point (1) with a specified included angle. If the angle is negative, a clockwise arc is drawn.



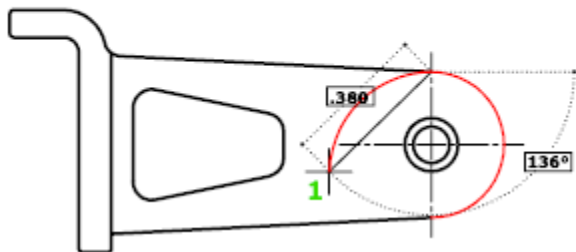
Draws either a minor or a major arc based on the distance of a straight line between the start point and endpoint.

If the chord length is positive, the minor arc is drawn counterclockwise from the start point. If the chord length is negative, the major arc is drawn counterclockwise.



Tangent to last line, arc, or polyline 

Immediately after you create a line or an arc, you can start an arc that is tangent at an endpoint by starting the ARC command and pressing ENTER at the Specify Start Point prompt. You need to specify only the endpoint of the arc.



End point of arc

Specify a point (1).

LAB SESSION 3

Introduction to polyline, polygon, ellipse

Rectangle

Creates a rectangular polyline.

Access Methods

Toolset: Drafting tab > Draw panel > Rectangle . 

Menu: Draw > Rectangle.

Creates a rectangular polyline from the specified the rectangle parameters (length, width, rotation) and type of corners (fillet, chamfer, or square).



The following prompts are displayed.

Current settings: Rotation = 0

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: *Specify a point or enter an option*

First Corner Point

Specifies a corner point of the rectangle.

Other Corner Point

Creates a rectangle using the specified points as diagonally opposite corners.



Area

Creates a rectangle using the area and either a length or a width. If the Chamfer or Fillet option is active, the area includes the effect of the chamfers or fillets on the corners of the rectangle.

Dimensions

Creates a rectangle using length and width values.

POLYGON

Creates an equilateral closed polyline.

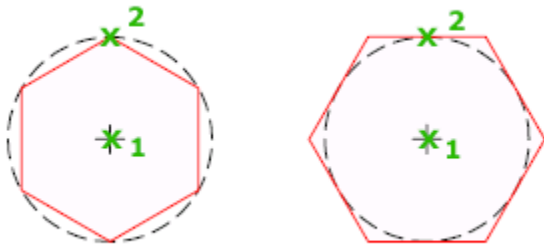
Access Methods

Tool Set: Drafting tab > Draw panel > Rectangle drop-down > Polygon.



Menu: Draw > Polygon.

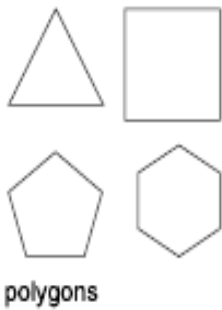
You specify the number of sides of the polygon and whether it is inscribed or circumscribed.



The following prompts are displayed.

Number of sides

Specifies the number of sides in the polygon (3-1024).

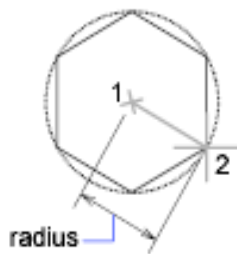


Center of polygon

Specifies the location of the center of the polygon and whether the new object is inscribed or circumscribed.

Inscribed in circle

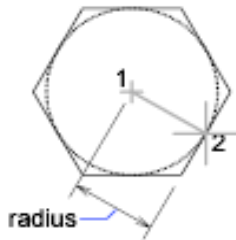
Specifies the radius of a circle on which all vertices of the polygon lie.



Specifying the radius with your pointing device determines the rotation and size of the polygon. Specifying the radius with a value draws the bottom edge of the polygon at the current snap rotation angle.

Circumscribed about circle

Specifies the distance from the center of the polygon to the midpoints of the edges of the polygon.




Specifying the radius with your pointing device determines the rotation and size of the polygon. Specifying the radius with a value draws the bottom edge of the polygon at the current snap rotation angle.

Ellipse:

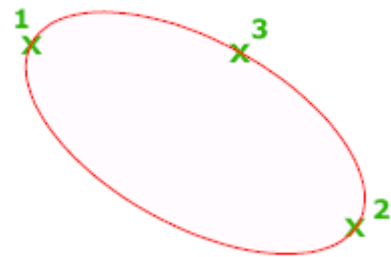
Creates an ellipse or an elliptical arc.

Access Methods

Tool Set: Drafting tab > Draw panel > Ellipse drop-down. 

Menu: Draw > Ellipse.

The first two points of the ellipse determine the location and length of the first axis. The third point determines the distance between the center of the ellipse and the end point of the second axis.



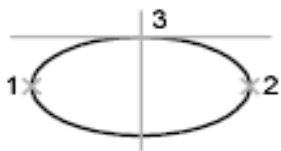
The following prompts are displayed.

Axis Endpoint 

Defines the first axis by its two endpoints. The angle of the first axis determines the angle of the ellipse. The first axis can define either the major or the minor axis of the ellipse.

Distance to Other Axis

Defines the second axis using the distance from the midpoint of the first axis to the endpoint of the second axis (3).



ellipse by axis endpoint

Rotation

Creates the ellipse by appearing to rotate a circle about the first axis.

Move the crosshairs around the center of the ellipse and click. If you enter a value, the higher the value, the greater the eccentricity of the ellipse. Entering 0 defines a circular ellipse.



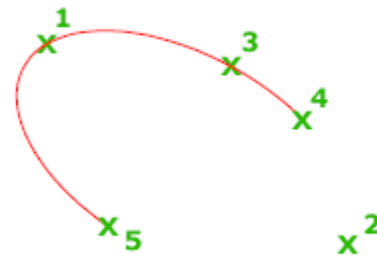
ellipse by rotation

Arc 

Creates an elliptical arc.

The angle of the first axis determines the angle of the elliptical arc. The first axis can define either the major or the minor axis depending on its size.

The first two points of the elliptical arc determine the location and length of the first axis. The third point determines the distance between the center of the elliptical arc and the endpoint of the second axis. The fourth and fifth points are the start and end angles.



Axis Endpoint

Defines the start point of the first axis.

Rotation

Defines the major to minor axis ratio of the ellipse by rotating a circle about the first axis. The higher the value from 0 through 89.4 degrees, the greater the ratio of minor to major axis. Values between 89.4 degrees and 90.6 degrees are invalid because the ellipse would otherwise appear as a straight line. Multiples of these angle values result in a mirrored effect every 90 degrees.

Start Angle

Defines the first endpoint of the elliptical arc. The Start Angle option also changes Parameter mode to Angle mode. The mode controls how the ellipse is calculated.

Parameter (specialized option)

Requires angular input, but creates the elliptical arc using the following parametric vector equation for the angle of each endpoint:

$$p(\text{angle}) = c + a * \cos(\text{angle}) + b * \sin(\text{angle})$$

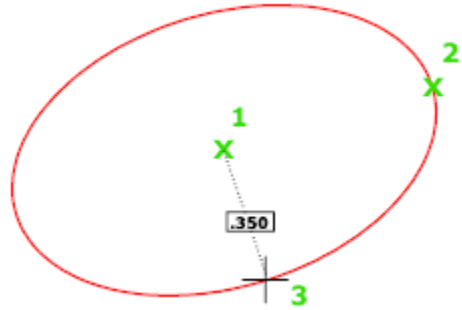
where c is the center of the ellipse and a and b are the negative lengths of its major and minor axes, respectively.

- **End Parameter:** Defines the end angle of the elliptical arc by using a parametric vector equation. The Start Parameter option toggles from Angle mode to Parameter mode. The mode controls how the ellipse is calculated.
- **Angle:** Defines the end angle of the elliptical arc. The Angle option toggles from Parameter mode to Angle mode. The mode controls how the ellipse is calculated.
- **Included Angle:** Defines an included angle beginning at the start angle.



Center

Creates an ellipse using a center point, the endpoint of the first axis, and the length of the second axis. You can specify the distances by clicking a location at the desired distance or by entering a value for the length.



Distance to Other Axis

Defines the second axis as the distance from the center of the ellipse, or midpoint of the first axis, to the point you specify.

Rotation

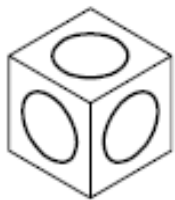
Creates the ellipse by appearing to rotate a circle about the first axis.

Move the crosshairs around the center of the ellipse and click. If you enter a value, the higher the value, the greater the eccentricity of the ellipse. Entering 0 defines a circle.

Isocircle

Creates an isometric circle in the current isometric drawing plane.

Note: The Isocircle option is available only when ISODRAFT is set to an isoplane, or the Style option of SNAP is set to Isometric.



Radius

Creates an isometric representation of a circle using a radius you specify.

Diameter

Creates an isometric representation of a circle using a diameter you specify.

LAB SESSION 4


Hatch:

Fills an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.

Access Methods



Button

 Toolbar: Drafting tool set ➤ Closed Shapes tool group ➤ Hatch

 Menu: Draw ➤ Hatch

Summary

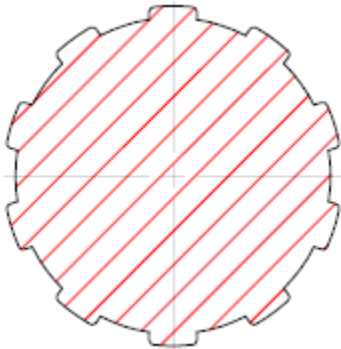
The [Hatch visor](#) or [Hatch and Gradient dialog box](#) is displayed.

If you enter **-hatch** at the Command prompt, [options are displayed](#).

Note: To prevent the creation of an enormous number of hatch lines, the maximum number of hatch lines created in a single hatch operation is limited. This limit prevents memory and performance problems. However, you can change the maximum number of hatch lines with the [HPMAXLINES](#) system variable.

Choose from several methods to specify the boundaries of a hatch.

- Specify a point in an area that is enclosed by objects.
- Select objects that enclose an area.
- Specify boundary points using the -HATCH Draw option.



Gradient:

Fills an enclosed area or selected objects with a gradient fill.

Access Methods

Tool Set: Drafting tab > Hatch panel > Gradient. 

Menu: Draw > Gradient.

The Gradient visor or Hatch and Gradient dialog box is displayed. If you prefer using the Hatch and Gradient dialog box, set the HPDLGMODE system variable to 1.

A gradient fill creates a smooth transition between one or two colors.



LAB SESSION 5

To construct a geometry using basic modify Tools like Erase, Copy, Move, Rotate, Offset, Mirror

Learning Objective:

At the end of this study, the student will be able to construct a geometry using basic modify Tools like Erase, Copy, Move, Rotate, Scale, Align, Offset, Mirror, Stretch, Lengthen, Break, Trim, Extend.

Introduction:


This lab manual will introduce students to some basic modifying commands which will aid the user to generate some complex geometric shape and engineering drawing. Students shall learn to use basic modifying commands to create complex geometric shapes and generate their first orthographic projection of a simple engineering part using first-angle projection method.

MODIFY TOOLBAR

MOVE

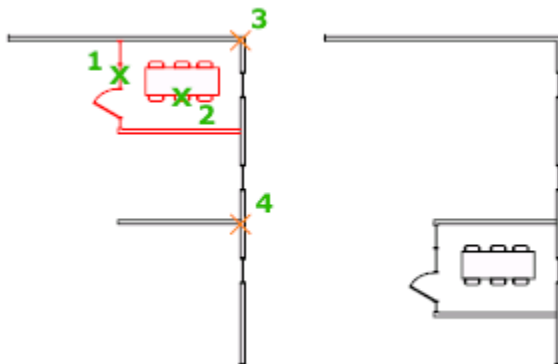
Moves objects a specified distance in a specified direction.

Access Methods

Tool Set: Drafting tab > Modify panel > Move. 

Menu: Modify > Move.

Use coordinates, grid snaps, object snaps, and other tools to move objects with precision.



The following prompts are displayed.

Select objects

Specifies which objects to move.

Base point

Specifies the start point for the move.

Second point

In combination with the first point, specifies a vector that indicates how far, and in what direction, the selected objects are moved.

If you press Enter to accept the Use first point as displacement value, the first point is interpreted as a relative X,Y,Z displacement. For example, if you specify **2,3** for the base point and press Enter at the next prompt, the objects move 2 units in the X direction and 3 units in the Y direction from their current position.

Displacement

Specifies a relative distance and direction.

The two points you specify define a vector that indicates how far from the original the copied objects are to be placed and in what direction.

ROTATE

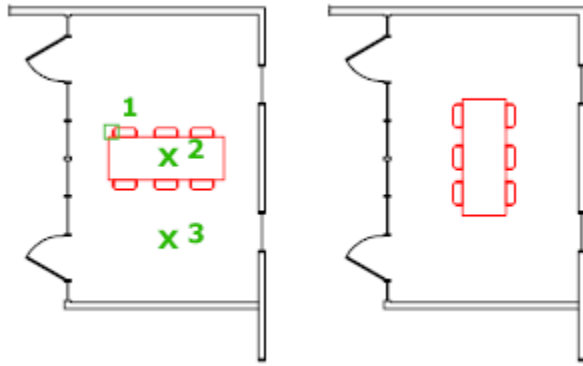
Rotates objects around a base point.

Access Methods

Tool Set: Drafting tab > Modify panel > Rotate. 

Menu: Modify > Rotate.

You can rotate selected objects around a base point to an absolute angle.



The following prompts are displayed.

Select objects

Use an object selection method and press Enter when you finish.

Specify base point

Specify a point.

Specify rotation angle

Enter an angle, specify a point, enter **c** , or enter **r**.

- **Rotation Angle.** Determines how far an object rotates around the base point. The axis of rotation passes through the specified base point and is parallel to the Z axis of the current UCS.
- **Copy.** Creates a copy of the selected objects for rotation.
- **Reference.** Rotates objects from a specified angle to a new, absolute angle. When you rotate a viewport object, the borders of the viewport remain parallel to the edges of the drawing area.

ERASE

Removes objects from a drawing.

Access Methods

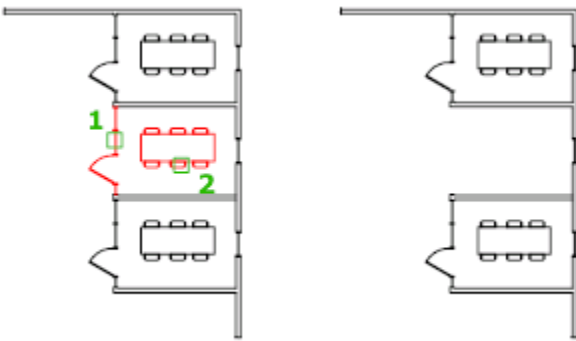
Tool Set: Drafting tab > Modify panel > Erase. 

Menu: Modify > Erase.

You can erase selected objects from the drawing. This method does not move objects to the Clipboard, where they can then be pasted to another location.

If you are working with 3D objects, you can also erase subobjects such as faces, meshes, and vertices.


Instead of selecting objects to erase, you can enter an option, such as **L** to erase the last object drawn, **p** to erase the previous selection set, or **ALL** to erase all objects. You can also enter **?** to get a list of all options.



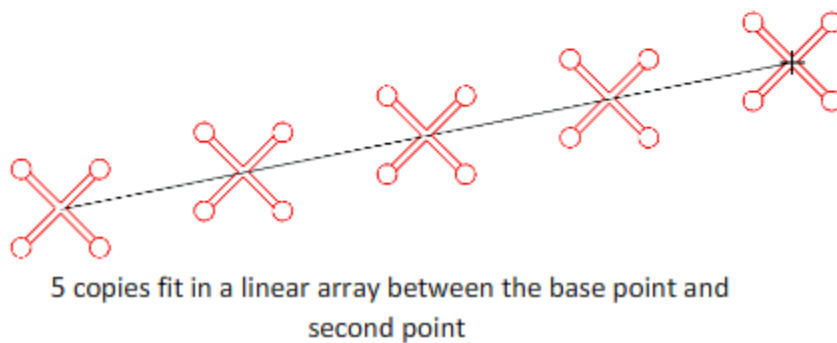
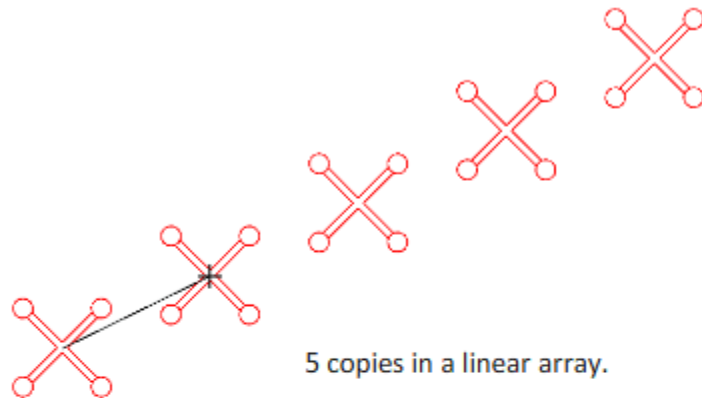
COPY

Copies objects a specified distance in a specified direction.

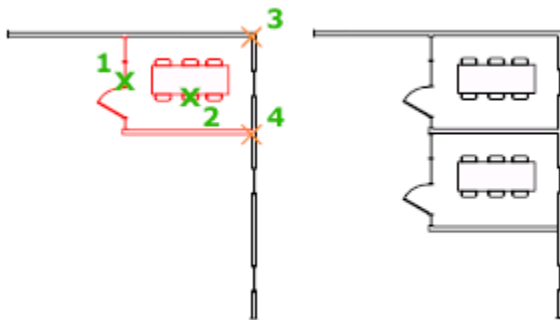
Access Methods

Tool Set: Drafting tab > Modify panel > Copy. 

Menu: Modify > Copy.



With the COPYMODE system variable, you can control whether multiple copies are created automatically.



The following prompts are displayed.

Select objects: *Use an object selection method and press Enter when you finish*

Specify base point or [Displacement/mOde/Multiple] <Displacement>: *Specify a base point or enter an option*

Specify second point or [Array] <use first point as displacement>: *Specify a second point or enter an option*

Displacement

Specifies a relative distance and direction using coordinates.

The two points you specify define a vector that indicates how far from the original the copied objects are to be placed and in what direction.

If you press Enter at the Specify Second Point prompt, the first point is interpreted as a relative X,Y,Z displacement. For example, if you specify **2,3** for the base point and press Enter at the next prompt, the objects are copied 2 units in the X direction and 3 units in the Y direction from their current location.

Mode

Controls whether the command repeats automatically (COPYMODE system variable).

Single

Creates a single copy of selected objects and ends the command.

Multiple

Overrides the Single mode setting. The COPY command is set to repeat automatically for the duration of the command.

Array

Arranges a specified number of copies in a linear array.

Number of Items to Array

Specifies the number of items in the array, including the original selection set.

Second Point

Determines a distance and direction for the array relative to the base point. By default, the first copy in the array is positioned at the specified displacement. The remaining copies are positioned in a linear array beyond that point using the same incremental displacement.

OFFSET

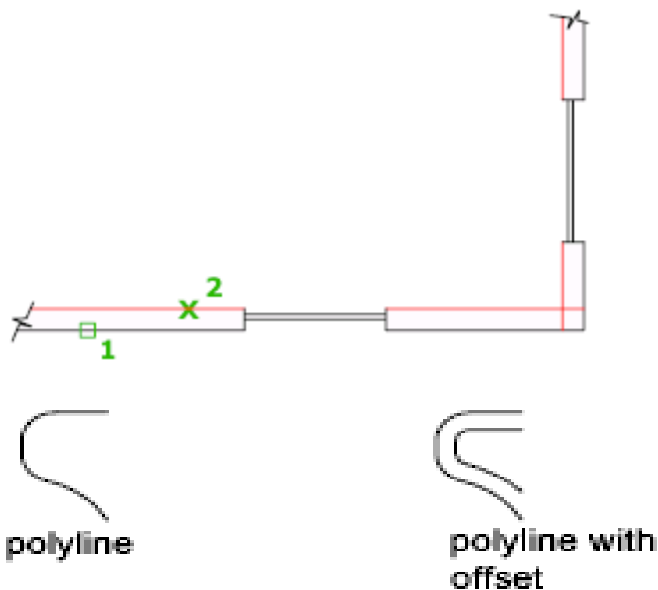
Creates concentric circles, parallel lines, and parallel curves.

Access Methods

Tool Set: Drafting tab > Modify panel > Offset. 

Menu: Modify > Offset.

You can offset an object at a specified distance or through a point. After you offset objects, you can trim and extend them as an efficient method to create drawings containing many parallel lines and curves.

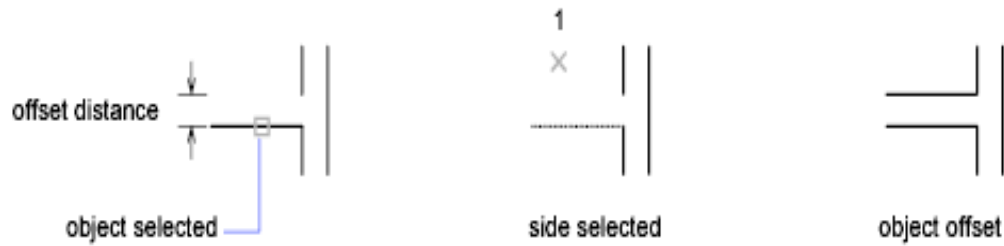


The OFFSET command repeats for convenience. To exit the command, press Enter.

The following prompts are displayed.

Offset Distance

Creates an object at a specified distance from an existing object.



Exit

Exits the OFFSET command.

Multiple

Enters the Multiple offset mode, which repeats the offset operation using the current offset distance.

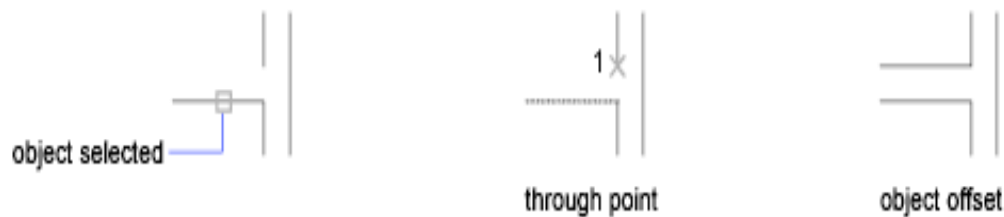
Undo

Reverses the previous offset.

Through

Creates an object passing through a specified point.

Note: For best results when you offset a polyline with corners, specify the through point near the midpoint of a line segment, not near a corner.



- Exit
- Multiple
- Undo

Erase

Erases the source object after it is offset.

Layer

Determines whether offset objects are created on the current layer or on the layer of the source object.

MIRROR

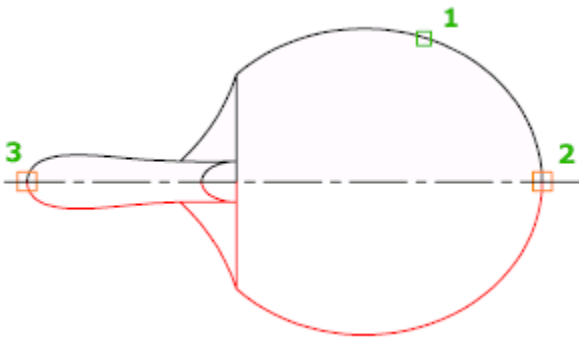
Creates a mirrored copy of selected objects.

Access Methods

Tool Set: Drafting tab > Modify panel > Mirror. 

Menu: Modify > Mirror.

You can create objects that represent half of a drawing, select them, and mirror them across a specified line to create the other half.



Note: By default, when you mirror a text object, the direction of the text is not changed. Set the MIRRTEXT system variable to 1 if you do want the text to be reversed.



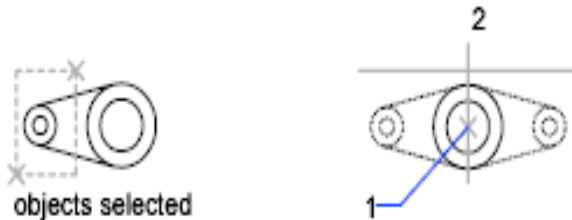
The following prompts are displayed.

Select objects

Use an object selection method to select the objects to be mirrored. Press Enter to finish.

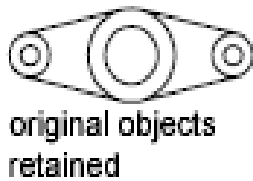
Specify first point, second point of mirror line

The two specified points become the endpoints of a line about which the selected objects are mirrored. For mirroring in 3D, this line defines a mirroring plane perpendicular to the XY plane of the user coordinate system (UCS) containing the mirror line.



Erase source objects

Determines whether the original objects are erased or retained after mirroring them.




LAB SESSION 6

Introduction to stretch, break, trim, extend, explode.

STRETCH

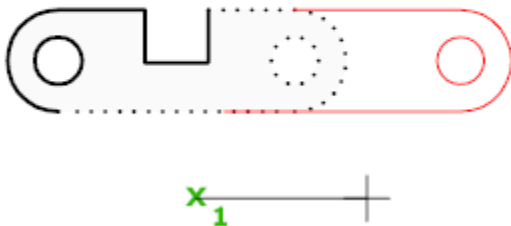
Stretches objects crossed by a selection window or polygon.

Access Methods

Tool Set: Drafting tab > Modify panel > Stretch. 

Menu: Modify > Stretch.

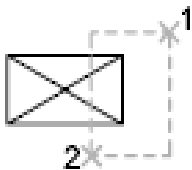
Objects that are partially enclosed by a crossing window are stretched. Objects that are completely enclosed within the crossing window, or that are selected individually, are moved rather than stretched. Some types of objects such as circles, ellipses, and blocks, cannot be stretched.



The following prompts are displayed.

Select objects

Specifies the portion of the object that you want to stretch. Use the cpolygon option or the crossing object selection method. Press Enter when the selection is complete.



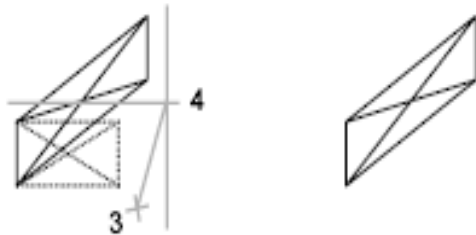
STRETCH moves only the vertices and endpoints that lie inside the crossing selection, leaving those outside unchanged. STRETCH does not modify 3D solids, polyline width, tangent, or curve-fitting information.

Base Point

Specifies the base point from which the offset for the stretch is calculated. This base point can be outside the area being stretched.

Second point

Specifies a second point that defines the distance and direction of the stretch. The distance and direction of this point from the base point defines how far the and in what direction the selected portions of the object will be stretched.



Use first point as displacement

Specifies that the stretch distance and direction will be based on the distance and direction of the base point you specified from the 0,0,0 coordinates in the drawing.

Displacement

Specifies the relative distance and direction of the stretch.


- To set a displacement based on the relative distance from the current location, enter distances in X,Y, Z format. For example, enter 5,4,0 to stretch the selection to a point that is 5 units along the X axis and 4 units along the Y axis from the original point.
- To set the displacement based on the distance and direction from the 0,0,0 coordinates in the drawing, click a location in the drawing area. For example, click a point at 1,2,0 to stretch the selection to a point that is 1 unit along the X axis and 2 units along the Y axis from its current location.

BREAK

Breaks the selected object between two points.

Access Methods

Tool Set: Drafting tab > Modify panel > Break drop-down > Break. 

Tool Set: Drafting tab > Modify panel > Break drop-down > Break at Point. 

Menu: Modify > Break.

You can create a gap between two specified points on an object, breaking it into two objects. If the points are off of an object, they are automatically projected on to the object. BREAK is often used to create space for a block or text.



The prompts that are displayed depend on how you select the object. If you select the object by using your pointing device, the program both selects the object and treats the selection point as the first break point. At the next prompt, you can continue by specifying the second point or by overriding the first point.

First point

Overrides the original first point where you selected the object with a new point that you specify.



Second point

Specifies a second point. The portion of the object is erased between the two points that you specify. If the second point is not on the object, the nearest point

on the object is selected; therefore, to break off one end of a line, arc, or polyline, specify the second point beyond the end to be removed.

To split an object in two without erasing a portion, enter the same point for both the first and second points. You can do this by entering @ to specify the second point.

Lines, arcs, circles, polylines, ellipses, splines, donuts, and several other object types can be split into two objects or have one end removed.

The program converts a circle to an arc by removing a piece of the circle starting counterclockwise from the first to the second point.



You can also break selected objects at a single point with the Break at Point tool.

Valid objects include lines, open polylines, and arcs. Closed objects such as circles cannot be broken at a single point.



TRIM

Trims objects to meet the edges of other objects.

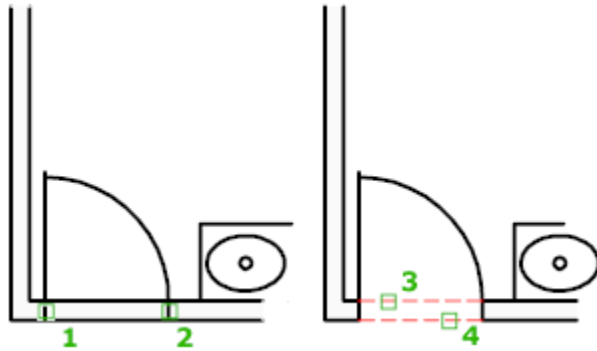
Access Methods

Tool Set: Drafting tab > Modify panel > Trim.



Menu: Modify > Trim.

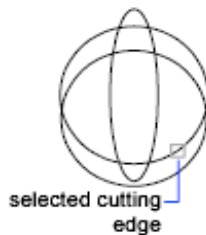
To trim objects, select the boundaries and press Enter. Then select the objects that you want to trim. To use all objects as boundaries, press Enter at the first Select Objects prompt.



The following prompts are displayed.

Select cutting edges

Specifies one or more objects to be used as a boundary for the trim. TRIM projects the cutting edges and the objects to be trimmed onto the XY plane of the current user coordinate system (UCS).



Note: To select cutting edges that include blocks, you can use only the single selection, Crossing, Fence, and Select All options.

Select objects

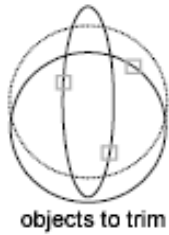
Specifies objects individually.

Select all

Specifies that all objects in the drawing can be used as a trim boundary.

Object to Trim

Specifies the object to trim. If more than one trim result is possible, the location of the first selection point determines the result.



Shift-Select to Extend

Extends the selected objects rather than trimming them. This option provides an easy method to switch between trimming and extending.

Fence

Selects all objects that cross the selection fence. The selection fence is a series of temporary line segments that you specify with two or more fence points. The selection fence does not form a closed loop.

Crossing

Selects objects within and crossing a rectangular area defined by two points.

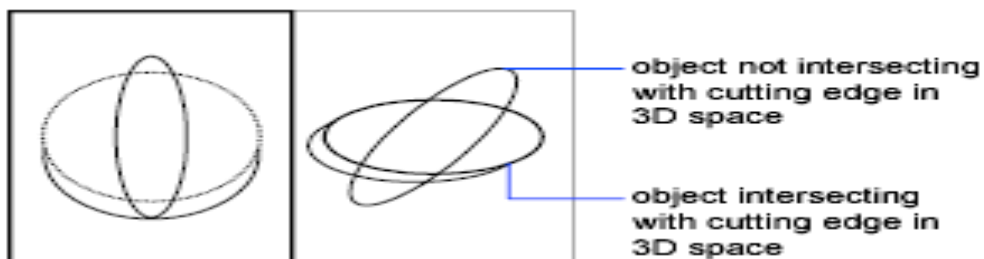
Note: Some crossing selections of objects to be trimmed are ambiguous. TRIM resolves the selection by following along the rectangular crossing window in a clockwise direction from the first point to the first object encountered.

Project

Specifies the projection method used when trimming objects.

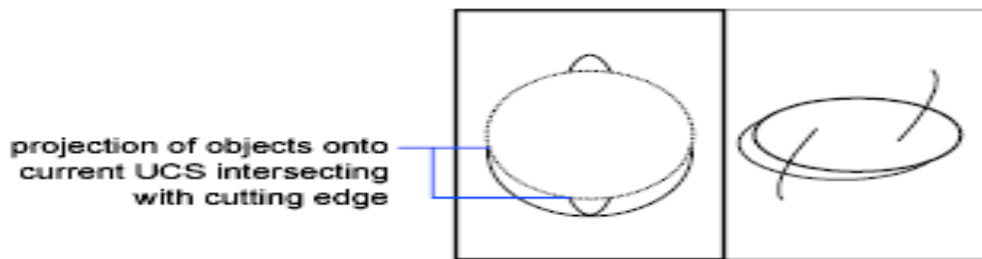
None

Specifies no projection. The command trims only objects that intersect with the cutting edge in 3D space.



UCS

Specifies projection onto the XY plane of the current UCS. The command trims objects that do not intersect with the cutting edge in 3D space.



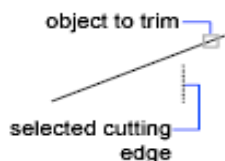
View

Specifies projection along the current view direction. The command trims objects that intersect the boundary in the current view.



Edge

Determines whether an object is trimmed at another object's extrapolated edge or only to an object that intersects it in 3D space.



EXTEND

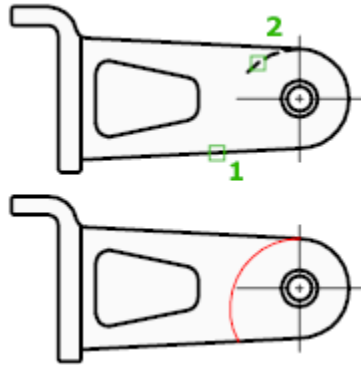
Extends objects to meet the edges of other objects.

Access Methods

Tool Set: Drafting tab > Modify panel > Extend. 

Menu: Modify > Extend.

To extend objects, first select the boundaries. Then press Enter and select the objects that you want to extend. To use all objects as boundaries, press Enter at the first Select Objects prompt.



The following prompts are displayed.

Current settings: Projection = *current*, Edge = *current*

Select boundary edges...

Select objects or <select all>: *Select one or more objects and press Enter, or press Enter to select all displayed objects*

Select object to extend or shift-select to trim or
[Fence/Crossing/Project/Edge/Undo]: *Select objects to extend, or hold down SHIFT and select an object to trim, or enter an option*

Boundary Object Selection

Uses the selected objects to define the boundary edges to which you want to extend an object.

Object to Extend

Specifies the objects to extend. Press Enter to end the command.

Shift-Select to Trim

Trims the selected objects to the nearest boundary rather than extending them. This is an easy method to switch between trimming and extending.

Fence

Selects all objects that cross the selection fence. The selection fence is a series of temporary line segments that you specify with two or more fence points. The selection fence does not form a closed loop.

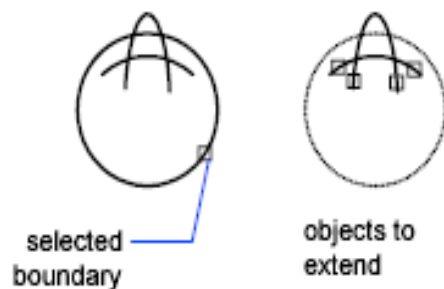
Crossing

Selects objects within and crossing a rectangular area defined by two points.

Note: Some crossing selections of objects to be extended are ambiguous. EXTEND resolves the selection by following along the rectangular crossing window in a clockwise direction from the first point to the first object encountered.

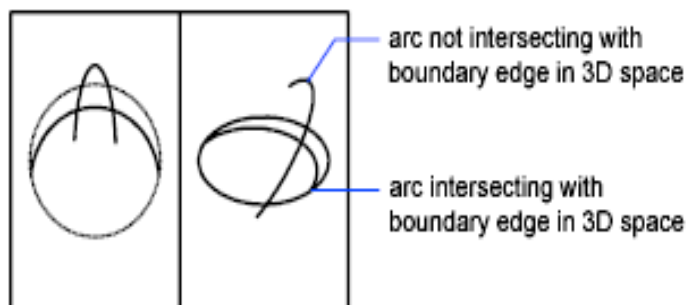
Project

Specifies the projection method used when extending objects.



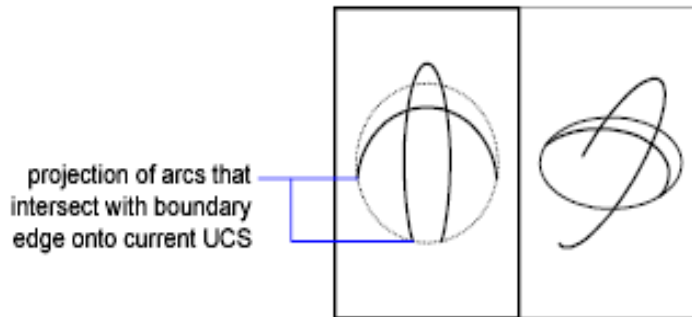
None

Specifies no projection. Only objects that intersect with the boundary edge in 3D space are extended.



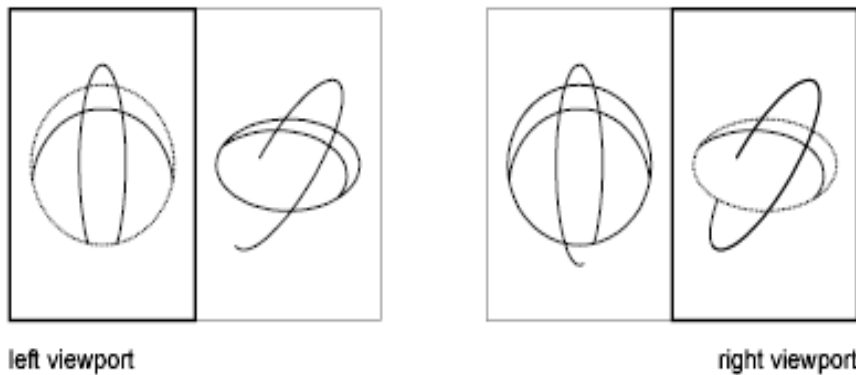
UCS

Specifies projection onto the *XY* plane of the current user coordinate system (UCS). Objects that do not intersect with the boundary objects in 3D space are extended.



View

Specifies projection along the current view direction.



Edge

Extends the object to another object's implied edge, or only to an object that actually intersects it in 3D space.



Extend

Extends the boundary object along its natural path to intersect another object or its implied edge in 3D space.



No Extend

Specifies that the object is to extend only to a boundary object that actually intersects it in 3D space.



Undo

Reverses the most recent changes made by EXTEND.

EXPLODE

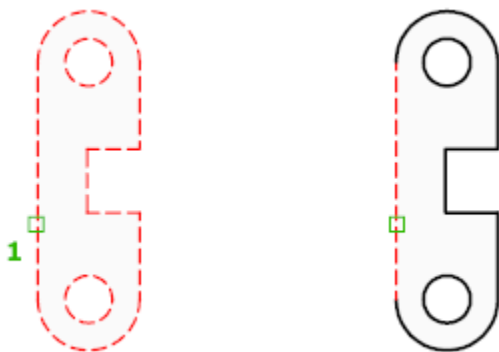
Breaks a compound object into its component objects.

Access Methods

Tool Set: Drafting tab > Modify panel > Explode.

Menu: Modify > Explode.

Explodes a compound object when you want to modify its components separately. Objects that can be exploded include blocks, polylines, and regions, among others.



The color, linetype, and lineweight of any exploded object might change. Other results differ depending on the type of compound object you're exploding. See the following list of objects that can be exploded and the results for each.

To explode objects and change their properties at the same time, use XPLODE.

Note: If you're using a script or an ObjectARX[®] function, you can explode only one object at a time. (Not applicable to AutoCAD LT.)

Here are the results of EXPLODE for each of the following types of objects:

2D Polyline

Discards any associated width or tangent information. For wide polylines, the resulting lines and arcs are placed along the center of the polyline.



Annotative Objects

Explodes the current scale representation into its constituent parts which are no longer annotative. Other scale representations are removed.

Arc

If within a nonuniformly scaled block, explodes into elliptical arcs.

Array

Explodes an associative array into copies of the original objects.

Block

Removes one grouping level at a time. If a block contains a polyline or a nested block, exploding the block exposes the polyline or nested block object, which must then be exploded to expose its individual objects.

Blocks with equal X, Y, and Z scales explode into their component objects. Blocks with unequal X, Y, and Z scales (nonuniformly scaled blocks) might explode into unexpected objects.

When nonuniformly scaled blocks contain objects that cannot be exploded, they are collected into an anonymous block (named with a “*E” prefix) and referenced

with the nonuniform scaling. If all the objects in such a block cannot be exploded, the selected block reference will not be exploded. Body, 3D Solid, and Region entities in a nonuniformly scaled block cannot be exploded. (Not available in AutoCAD LT.)

Exploding a block that contains attributes deletes the attribute values and redisplay the attribute definitions.

Blocks inserted with external references (xrefs) and their dependent blocks cannot be exploded.

Blocks inserted with MINSERT cannot be exploded. (MINSERT is not available in AutoCAD LT.)

Body

Explodes into a single-surface body (nonplanar surfaces), regions, or curves.

Circle

If within a nonuniformly scaled block, explodes into ellipses.

Leaders

Explodes into lines, splines, solids (arrow heads), block inserts (arrow heads, annotation blocks), multiline text, or tolerance objects, depending on the leader.

Mesh Objects

Explodes each face into a separate 3D face object. Color and materials assignments are retained. (Not available in AutoCAD LT.)

Multiline Text

Explodes into text objects.

Multiline

Explodes into lines and arcs.

Polyface Mesh

Explodes one-vertex meshes into a point object. Two-vertex meshes explode into a line. Three-vertex meshes explode into 3D faces.

Region

Explodes into lines, arcs, or splines.

LAB SESSION 7

Develop geometry of planar machine parts by using Advanced Editing Commands like Fillet, Chamfer, Array

Learning Objective:

At the end of this study, the student will be able to develop geometry of planar machine parts by using Advanced Editing Commands like Fillet, Chamfer, Array, Splinedit, Pedit, etc

Advanced Editing Commands

FILLET

Rounds and fillets the edges of objects.

Access Methods

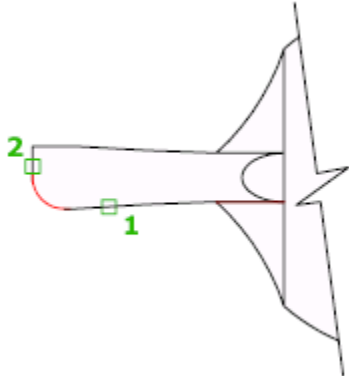
Tool Set: Drafting tab > Modify panel > Fillet drop-down > Fillet. 

Menu: Modify > Fillet.

A round or fillet is

- an arc that is created tangent between two 2D objects.
- a curved transition between two surfaces or adjacent faces on a 3D solid.

In this example, an arc is created tangent to the selected lines, which are trimmed to meet the endpoints of the arc.



Create 2D Fillets

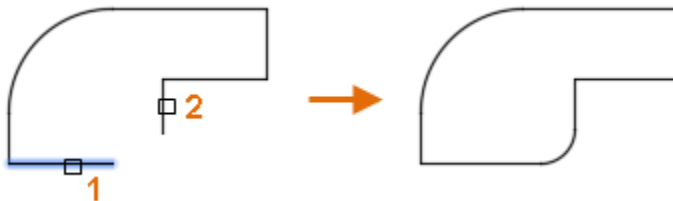
A round or fillet can be created between two objects of the same or different object types: 2D polylines, arcs, circles, ellipses, elliptical arcs, lines, rays, splines, and xlines.

If the two selected objects are on the same layer, the arc defined is created on that layer. Otherwise, the arc is created on the current layer. The layer affects object properties including color and linetype.

The following prompts are displayed when creating a 2D fillet.

First Object

Select the first of two objects or the first line segment of a 2D polyline to define the fillet.



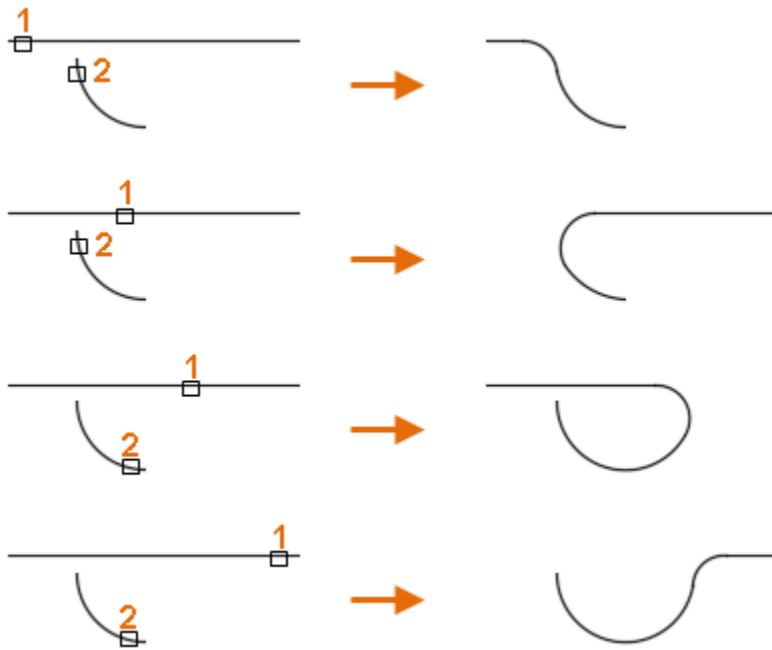
Second object or shift-select to apply corner

Select the second object or line segment of a 2D polyline to define the fillet.

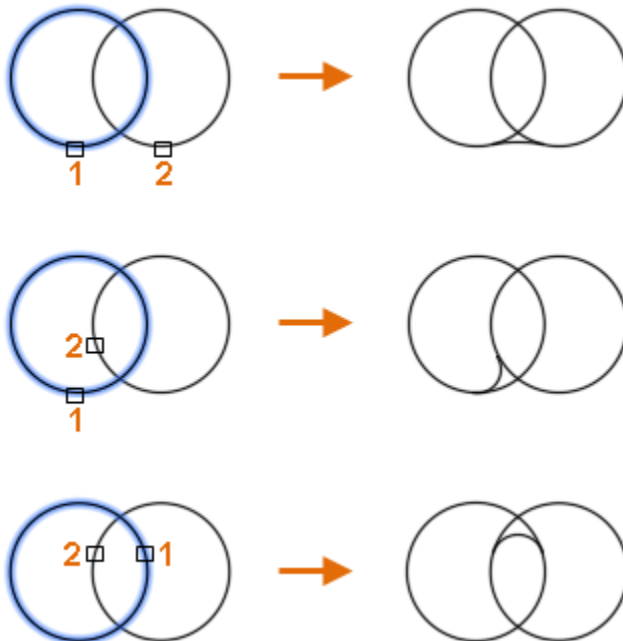
You can also hold down the Shift key before selecting the second object or line segment of a 2D polyline to extend or trim the selected objects to form a sharp corner. While Shift is held down, a temporary value of zero is assigned to the current fillet radius value.

If the selected objects are straight line segments of a 2D polyline, the line segments can be adjacent to each other or separated by one other segment. When the selected segments are separated by a segment, the segment that separates them is removed and replaced with the fillet.

The direction and length of the arc created is determined by the points picked to select the objects. Always select an object closest to where you want the endpoints of the fillet to be drawn.



When a circle is selected, the circle is not trimmed; the fillet drawn meets the circle smoothly.



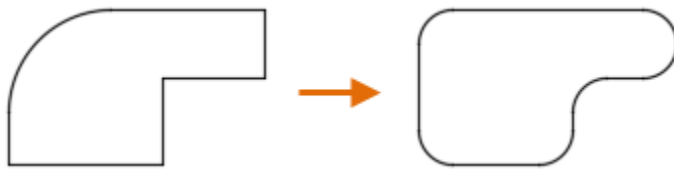
Note: Adding a fillet or round to a hatch boundary that was defined with individual objects results in the removal of hatch associativity. If the hatch boundary was defined from a polyline, associativity is maintained.

Undo

Reverses the previous action in the command.

Polyline

Inserts a fillet at each vertex of a 2D polyline where two straight line segments meet. The fillets become new segments of the polyline, unless the Trim option is set to No Trim.



Select 2D polyline

Select the 2D polyline to insert fillets at each vertex.

If an arc segment separates two straight line segments, the arc segment is removed and replaced with the fillet.

Note: Line segments that are too short to accommodate the fillet radius are not modified.

Radius

Sets the radius for subsequent fillets; changing this value does not affect existing fillets.

Note: A radius value of zero can be used to create a sharp corner. Filleting two lines, rays, xlines, or line segments of a 2D polyline with a radius of zero extends or trims the objects so they intersect.

Trim

Controls whether the selected objects are trimmed to meet the endpoints of the fillet.

- **Trim.** Selected objects or line segments are trimmed to meet the endpoints of the fillet.
- **No Trim.** Selected objects or line segments are not trimmed before the fillet is added.

The current value is stored in the TRIMMODE system variable.

Multiple

Allows for the rounding of more than one set of objects.

CHAMFER

Bevels or chamfers the edges of two 2D objects or the adjacent faces of a 3D solid.

Access Methods

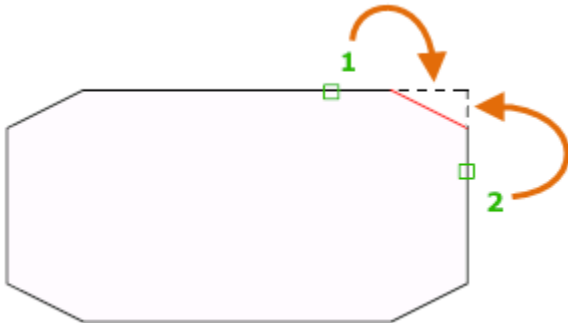
Tool Set: Drafting tab > Modify panel > Fillet drop-down > Chamfer. 

Menu: Modify > Chamfer.

A bevel or chamfer is

- an angled line that meets the endpoints of two straight 2D objects.
- a sloped transition between two surfaces or adjacent faces on a 3D solid.

The distances and angles that you specify are applied in the order that you select the objects.



Create 2D Chamfer

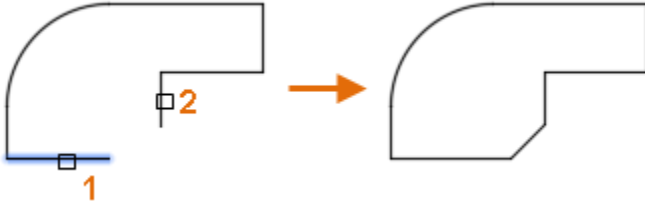
A bevel or chamfer can be defined by selecting two objects of the same or different object types: lines, polylines, rays, and xlines.

If the two selected objects are on the same layer, the line defined is created on that layer. Otherwise, the line is created on the current layer. The layer affects object properties including color and linetype.

The following prompts are displayed when creating a 2D chamfer.

First Line

Select the first of two objects or the first line segment of a 2D polyline to define the chamfer.



Second line or shift-select to apply corner

Select the second object or line segment of a 2D polyline to define the chamfer.

You can also hold down the Shift key before selecting the second object or line segment of a 2D polyline to extend or trim the selected objects to form a sharp corner. While Shift is held down, a temporary value of zero is assigned to the current chamfer distance and angle values.

If the selected objects are straight line segments of a 2D polyline, the line segments can be adjacent to each other or separated by one other segment. When the selected segments are separated by a segment, the segment that separates them is removed and replaced with the chamfer.

Note: Adding a chamfer or bevel to a hatch boundary that was defined with individual objects results in the removal of hatch associativity. If the hatch boundary was defined from a polyline, associativity is maintained.

Undo

Reverses the previous action in the command.

Polyline

Inserts a chamfer line at each vertex of a 2D polyline where two straight line segments meet. The chamfer lines become new segments of the polyline, unless the Trim option is set to No Trim.

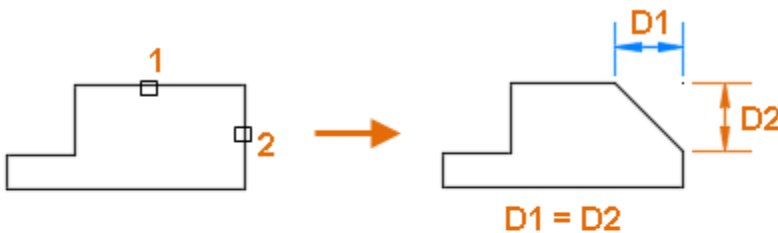


Note: Line segments that are too short to accommodate the chamfer distance are not modified.

Distance

Sets the chamfer distances from the intersecting points of the first and second objects.

If both distances are set to zero, the selected objects or line segments are extended or trimmed so they intersect.



Angle

Sets the chamfer distance from the intersecting point of the selected objects and the XY angle from the first object or line segment.

If both values are set to zero, the selected objects or line segments are extended or trimmed so they intersect.



Trim

Controls whether the selected objects are trimmed to meet the endpoints of the chamfer line.

- **Trim.** Selected objects or line segments are trimmed to meet the endpoints of the chamfer line. If the selected objects or line segments do not intersect with the chamfer line, they are extended or trimmed before the chamfer line is added.
- **No Trim.** Selected objects or line segments are not trimmed before the chamfer line is added.

Note: The current value is stored in the TRIMMODE system variable.

Method

Controls how the chamfer line is calculated from the intersecting point of the selected objects or line segments.

- **Distance.** Chamfer line is defined by two distances.
- **Angle.** Chamfer line is defined by a distance and an angle.

The current value is stored in the CHAMMODE system variable.

Multiple

Allows for the beveling of more than one set of objects.

ARRAY

Maintains legacy command line behavior for creating nonassociative, 2D rectangular or polar arrays.

List of Prompts

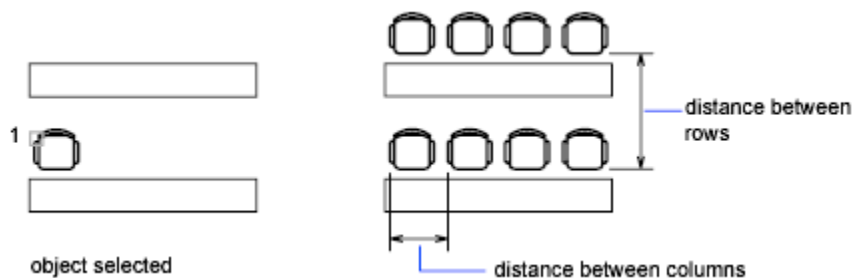
The following prompts are displayed.

Select objects: *Use an object selection method*

Enter the type of array [Rectangular/Polar] <current>: *Enter an option or press Enter*

Rectangular

Creates an array of rows and columns of copies of the selected objects.



Enter the number of rows (---) <1>: *Enter a nonzero integer or press Enter*

Enter the number of columns (|||) <1>: *Enter a nonzero integer or press Enter*

If you specify one row, you must specify more than one column and vice versa.

The selected object, or cornerstone element, is assumed to be in the lower-left corner, and generates the array up and to the right.

The specified distance between the rows and columns includes the corresponding lengths of the object to be arrayed.

Enter the distance between rows or specify unit cell (---):

To add rows downward, specify a negative value for the distance between rows. ARRAY skips the next prompt if you specify two points for the opposite corners of a rectangle.

Specify the distance between columns (|||):

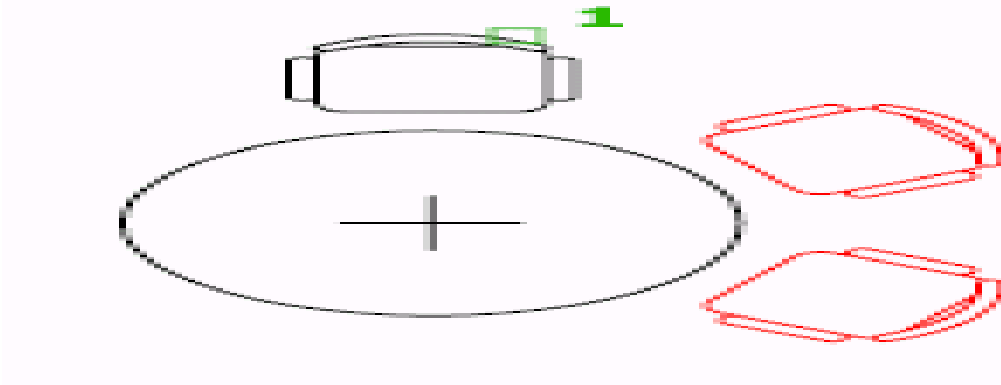
To add columns to the left, specify a negative value for the distance between columns. Rectangular arrays are constructed along a baseline defined by the current snap rotation. This angle is normally 0, so the rows and columns are orthogonal with respect to the X and Y drawing axes. The Rotate option of the [SNAP](#) command changes the angle and creates a rotated array. The [SNAPANG](#) system variable stores the snap rotation angle.

If you specify a large number of rows and columns for the array, it might take a while to create the copies. By default, the maximum number of array elements that you can generate in one command is 100,000. How you change this limit depends on the product:

- Most AutoCAD-based products: The limit is set by the MAXARRAY setting in the registry. To reset the limit to 200,000, for example, enter (setenv "MaxArray" "200000") at the Command prompt.
- AutoCAD LT: You can change the maximum number of array elements by setting the MaxArray system registry variable using the SETENV command.

Polar

Creates an array by copying the selected objects around a specified center point.



Specify center point of array or [Base]: *Specify a point or enter **b** to specify a new base point*

Center Point

Creates an array defined by a center point.

Base

Specifies a new reference (base) point relative to the selected objects that will remain at a constant distance from the center point of the array as the objects are arrayed.

Enter the number of items in the array: *Enter a positive integer or press Enter*

If you enter a value for the number of items, you must specify either the angle to fill or the angle between items. If you press Enter (and do not provide the number of items), you must specify both.

Specify the angle to fill (+ccw, -cw) <360>: *Enter a positive number for a counterclockwise rotation or a negative number for a clockwise rotation*

You can enter **0** for the angle to fill only if you specify the number of items.

If you specify an angle to fill without providing the number of items, or if you specify the number of items and enter **0** as the angle to fill or press Enter, the following prompt is displayed:

Angle between items: *Specify an angle*

If you specified the number of items and entered 0 as the angle to fill or pressed Enter, ARRAY prompts for a positive or negative number to indicate the direction of the array:

Angle between items (+=ccw, -=cw): *Enter a positive number for a counterclockwise rotation or a negative number for a clockwise rotation*

ARRAY determines the distance from the array's center point to a reference point on the last object selected. Reference points include locations such as the center point of a circle or arc, the insertion base point of a block, the start point of text, or the endpoint of a line.

Rotate arrayed objects? <Y>: *Enter y or n, or press Enter*

In a polar array, the reference point of the last object in the selection set is used for all objects. If you defined the selection set by using window or crossing selection, the last object in the selection set is arbitrary. Removing an object from the selection set and adding it back forces that object to be the last object selected. You can also make the selection set into a block and replicate it.

Path

Evenly distributes copies of the selected object along a path or a portion of a path (same as the ARRAYPATH command).

LAB SESSION 8

Draw the orthographic views of 3D machine parts and organizing your work by dimensioning, managing with layers

Learning Objective:

At the end of this study, the student will be able to draw the orthographic views of 3D machine parts and organizing the work by dimensioning, managing with layers and creating blocks in AutoCAD

Layout Managing Tools

Dimension

Creates multiple types of dimensions within a single command session.

Access Methods

Tool Set: Drafting tab > Dimension panel > Dimension. 

Menu: Dimension > Dimension.

When you hover over an object for dimensioning, the DIM command automatically previews a suitable dimension type to use. Select objects, lines, or points to dimension and click anywhere in the drawing area to draw the dimension.

The supported dimension types range from vertical, horizontal, aligned, and rotated linear dimensions, to angular dimensions, to radius, diameter, jogged radius, and arc length dimensions, to baseline and continued dimensions. If required, you can change the dimension type using command line options.

First extension line origin

Creates a linear dimension when you select two points.

Angular

Creates an angular dimension showing the angle between three points by or the angle between two lines (same as the DIMANGULAR command).

- **Vertex.** Specifies the point to use as the vertex of the angle.
- **Specify first side of angle.** Specifies one of the lines that form the angle.
- **Specify second side of angle.** Specifies the other line that forms the angle.

- **Angular dimension location.** Specifies where to place the dimension arc line. Depending on the position of the dimension, the quadrant for the dimension changes.
 - **Mtext.** Displays the Text Editor contextual tab, which you can use to edit the dimension text.
 - **Text.** Customizes the dimension text at the Command prompt. The generated dimension is displayed within angle brackets.
 - **Text angle.** Specifies the angle of the dimension text.
 - **Undo.** Returns to the previous prompt.
- **Undo.** Returns to the previous prompt.

Baseline

Creates a linear, angular, or ordinate dimension from the first extension line of the previous or selected dimension (same as the DIMBASELINE command).

Note: By default, the last created dimension is used as the base dimension.

- **First extension line origin.** Specifies the first extension line of the base dimension as the extension line origin for the baseline dimension.
- **Second extension line origin.** Specifies the next edge or angle to dimension.
- **Feature Location.** Uses the endpoint of the base dimension (ordinate dimension) as the endpoint for the baseline dimension.
- **Select.** Prompts you select a linear, ordinate, or angular dimension to use as the base for the baseline dimension.
- **Offset.** Specifies the offset distance from which the baseline dimensions are created.
- **Undo.** Undoes the last baseline dimension created.

Continue

Creates a linear, angular, or ordinate dimension from the second extension line of a selected dimension (same as the DIMCONTINUE command).

- **First extension line origin.** Specifies the first extension line of the base dimension as the extension line origin for the continued dimension.
- **Second extension line origin.** Specifies the next edge or angle to dimension.
- **Feature location.** Uses the endpoint of the base dimension (ordinate dimension) as the endpoint for the continued dimension.
- **Select.** Prompts you select a linear, ordinate, or angular dimension to use as the base for the continued dimension.
- **Undo.** Undoes the last baseline dimension created.

Align

Aligns multiple parallel, concentric, or same datum dimensions to a selected base dimension.

- **Base dimension.** Specifies a dimension to use as basis for the dimensions alignment.

- **Dimensions to align.** Selects the dimensions to align to the selected base dimension.

Distribute

Specifies the method on how to distribute a group of selected isolated linear or ordinate dimensions.

- **Equal.** Equally distributes all selected dimensions. This method requires a minimum of three dimension lines.
- **Offset.** Distributes all selected dimensions at a specified offset distance.

LAYER

Manages layers and layer properties.

Access Methods

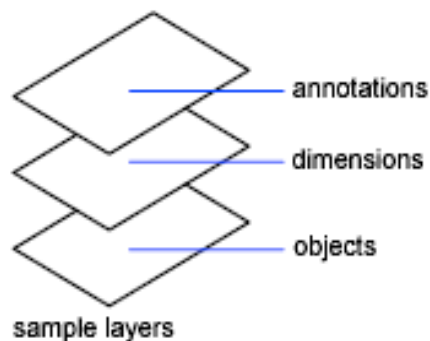
Menu: Format > Layers.

Command entry: 'layer for transparent use

Summary

The [Layers Palette](#) is displayed.

If you enter **-layer** at the Command prompt, [options are displayed](#).



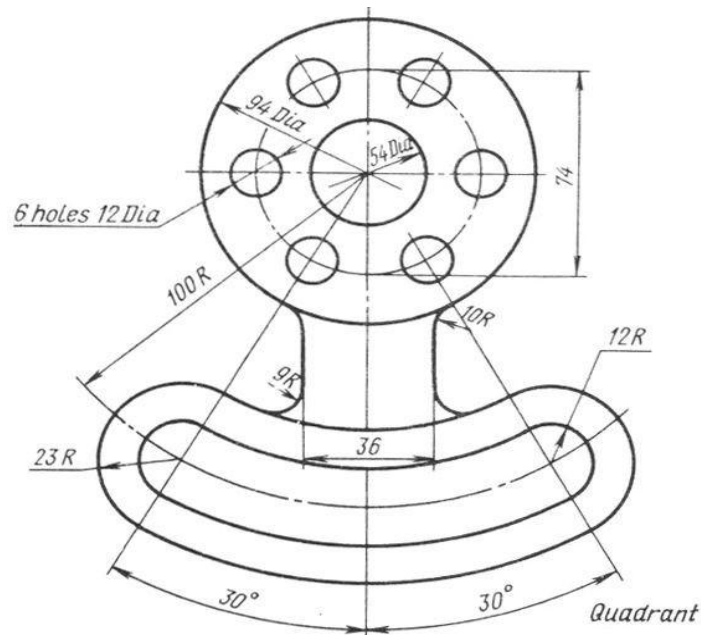
Use layers to control the visibility of objects and to assign properties such as color and linetype. Objects on a layer normally assume the properties of that layer. However, you can override any layer property of an object. For example, if an object's color property is set to BYLAYER, the object displays the color of that layer. If the object's color is set to Red, the object displays as red, regardless of the color assigned to that layer.

Exercise – 1

Aim : To create a 2D view of the given diagram using Auto CAD.

Procedure:

1. Type limits in command menu & set value to 297,290.
2. Change the units to millimeters from inches and also precision to 0 by clicking format -> units ->ok.
3. To set the paper size type zoom -> enter and type a -> enter in Command bar.
4. Draw the 3 concentric circles with diameters 94, 74 & 54
5. Draw the two axis lines from center of circles
6. Draw the vertical line from the center of circle
7. From the modify tool bar, use the array command to draw the 6holes with 12dia from center of circles
8. Now draw the 300 line by use the vertical line
9. Then mirror the 300 line, with vertical line
10. Again draw the concentric of radius 100 from center of circle
11. From the modify toolbar, use the offset command to draw the 12 & 23 distance circle.
12. Draw 2 circles. With radius 23 & 12 on the 100R circle where the 300 line co-inside.
13. From modify toolbar, mirror these circle to represent the another side.
14. And offset vertical line from center of circle with a distance both side of vertical line.
15. From modify toolbar, use the fillet command to represent fillet of radius 10 &9 to the offset line.
16. Trim the unwanted lines to get required 2D drawing



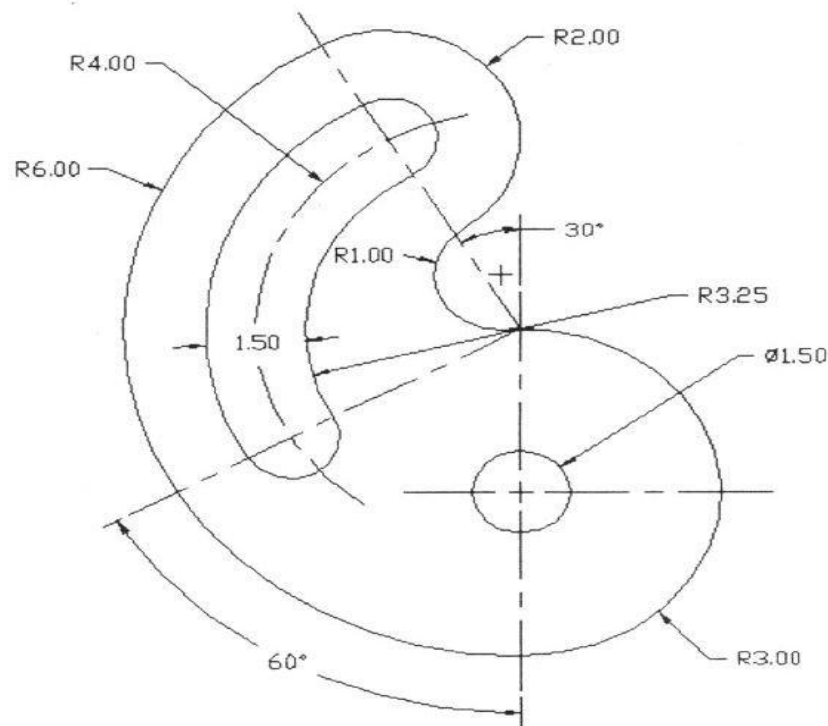
Result: Hence the required 2D diagram is created using Auto CAD.

Exercise – 2

Aim: To create a 2D view of the given diagram using Auto CAD.

Procedure:

1. Type limits in command menu & set values to 200,200.
2. Change the units to millimeters from inches and also precision to 0 by clicking format -> units -> ok.
3. To set the paper size type zoom -> enter and type a -> enter in command bar
4. Draw 2 lines as the axes and draw concentric circles of specified diameter.
5. Draw a ray of 30° angle to the +ve Y axis as shown in the figure and a ray of angle 60° to the -ve Y axis as shown in the figure.
6. Draw concentric circles from the point where the circle meets the positive Y axis as show.
7. Now trim the circles to get appropriate shape.
8. Continue with the design until the AutoCAD drawing his complete.
9. Give the dimensions from the dimension tool bar as in diagram.



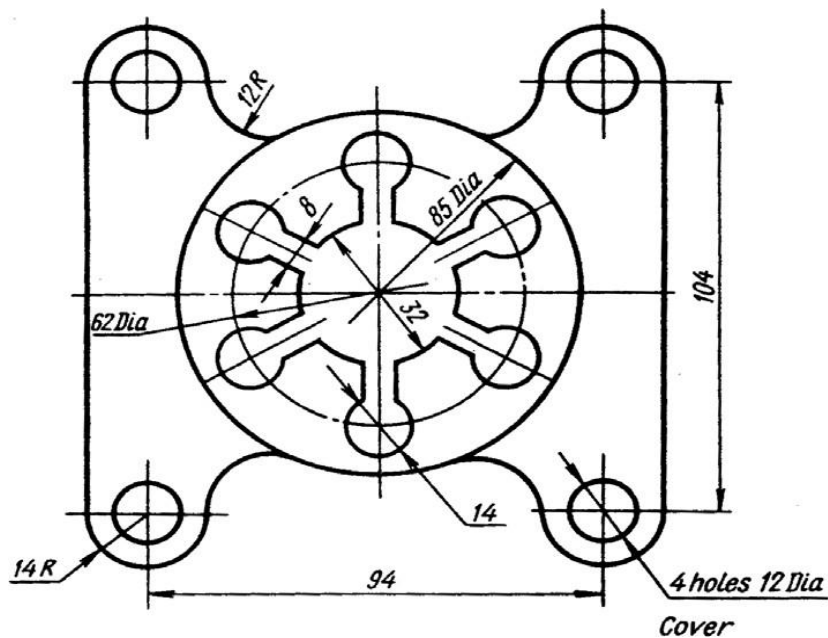
Result: Hence the required 2D diagram is created using Auto CAD.

Exercise – 3

Aim: To create a 2D view of the given diagram using Auto CAD.

Procedure:

1. Type limits in command menu & set values to 45,45
2. Change the units to millimeters from inches and also precision to 0 by clicking format -> units -> ok.
3. To set the paper size type zoom -> enter and type a -> enter in command bar
4. Draw the 3 concentric circles of diameters 85,62,32
5. Draw the 2 axes lines from the center of the circles
6. Draw the circle with 14dia on 62dia of circle and offset of the vertical line with distance 4 to both sides of the vertical line
7. Then trim the unwanted lines
8. Use the array command from modify tool bar to represent the 6 holes with 14 dia of center of the circles
9. Offset the vertical and horizontal axes with 47 and 52 distance
10. And draw the 2 circles with 14 radius and 12 dia at coincide of the offset axes
11. From the modify tool bar select the fillet command to represent the 12R fillet
12. Then mirror this to require the 2D drawing
13. Finally trim the unwanted lines and circles



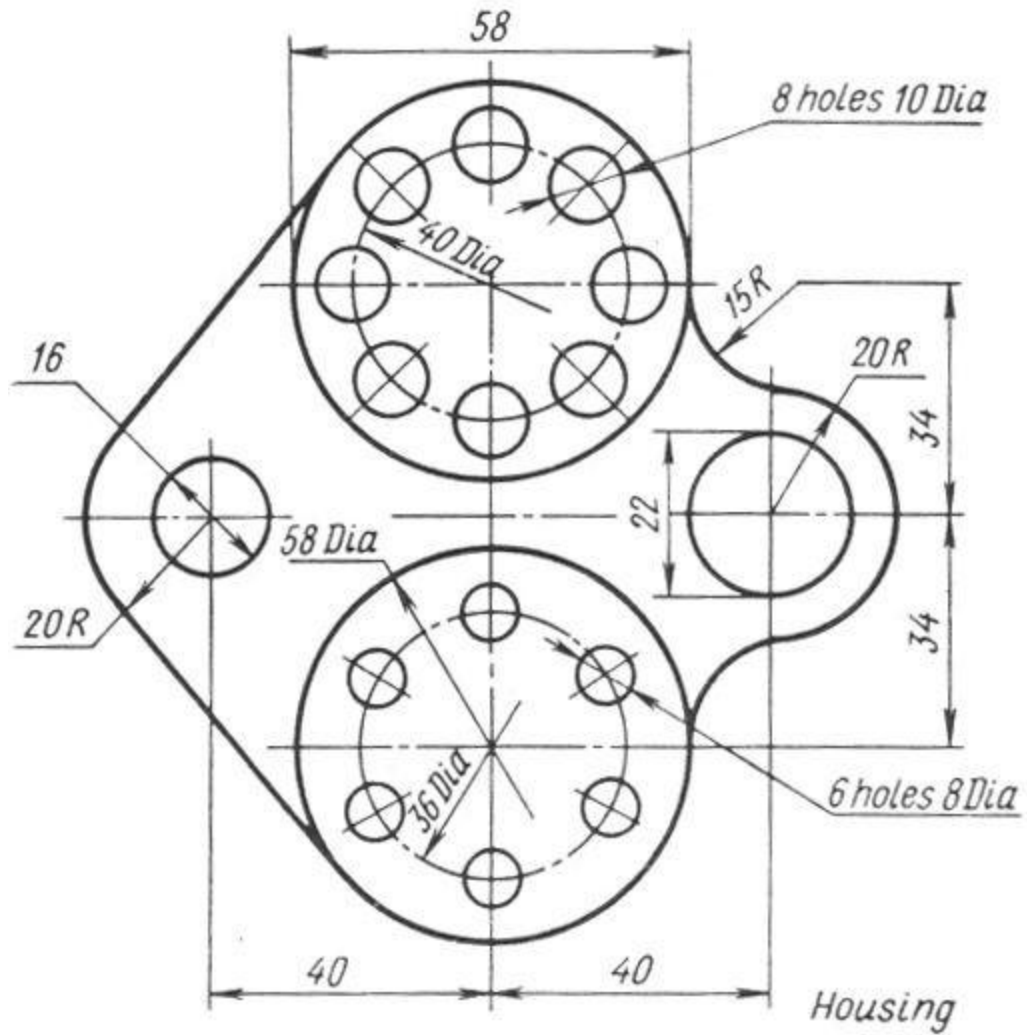
Result: Hence the required 2D diagram is created using Auto CAD.

Exercise – 4

Aim: To create a 2D view of the given diagram using Auto CAD.

Procedure:

1. Type limits in command menu & set value to 297,290.
2. Change the units to millimeters from inches and also precision to 0 by clicking format -> units -> ok.
3. To set the paper size type zoom -> enter and type a -> enter in Command bar.
4. Draw the 2 axes lines
5. Draw 2 concentric circles of diameter 58 and 40 above the axes
6. Draw the 10dia circle on the 40dia of the circle
7. And use the array command from modify tool bar to represent the 8 holes with 10dia from center of the circles.
8. Draw the 2 concentric circles of diameter 58 & 36 below the axes
9. Draw the circles of the diameter 8 on the circle of 36dia
10. Use the array command from the modify tool bar to represent the 6 holes with 8dia
11. Draw the 2 concentric circles of diameter 22 and 20 radius at the right side of the vertical line from the vertical axes.
12. Fillet the circles of radius 20 with 15 radiuses.
13. And again draw the circles of diameter 16& 20R at left side of the vertical line from the vertical axes.
14. And chamfer the circles of radius 20 with 58dia circle
15. Then trim the unwanted lines to get the required 2D drawing.



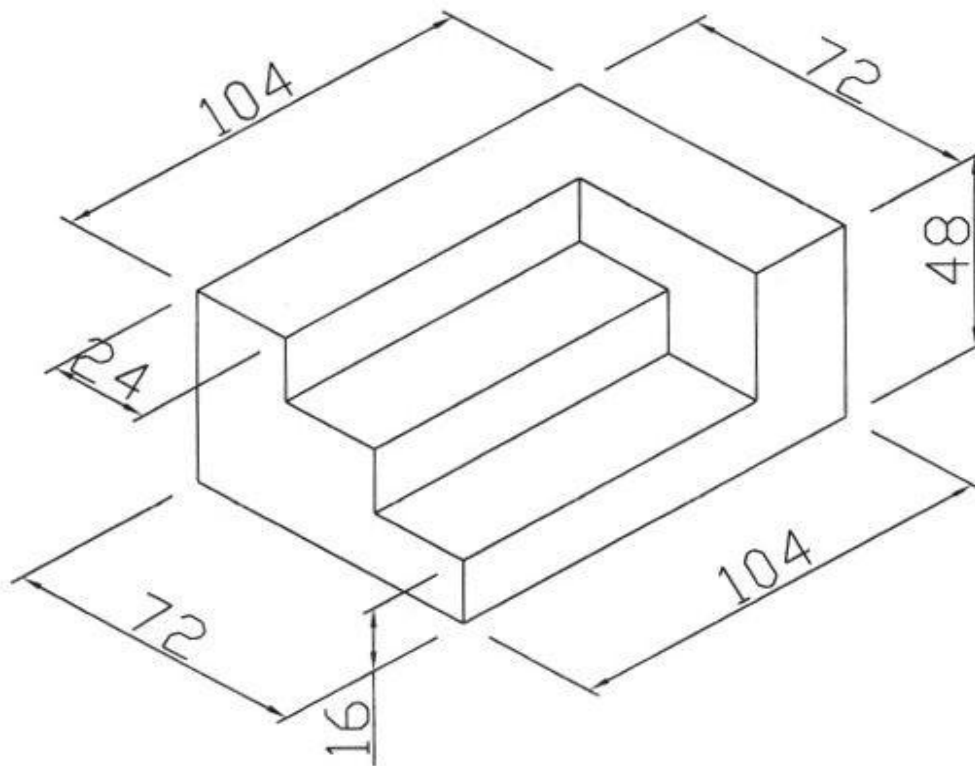
Result: Hence the required 2D diagram is created using Auto CAD.

Exercise – 5

Aim: To create a 2D isometric view of the given diagram using Auto CAD.

Procedure:

1. Type limits in command menu & set value to 297,290.
2. Change the units to millimeters from inches and also precision to 0 by clicking format -> units -> ok.
3. To set the paper size type zoom -> enter and type a -> enter in command bar.
4. Go to drafting settings and turn on isometric snap..
5. Use the F5 key to change between the views of isometric planes.
6. Start from the front view and draw the the line of length of line 104 using the F8 key (O snap key) and continue with the 48 length line.
7. Change to top plane and draw the 72mm line.
8. Continue in the same fashion to complete the whole figure.
9. Give the dimensions from the dimension tool bar as in diagram.



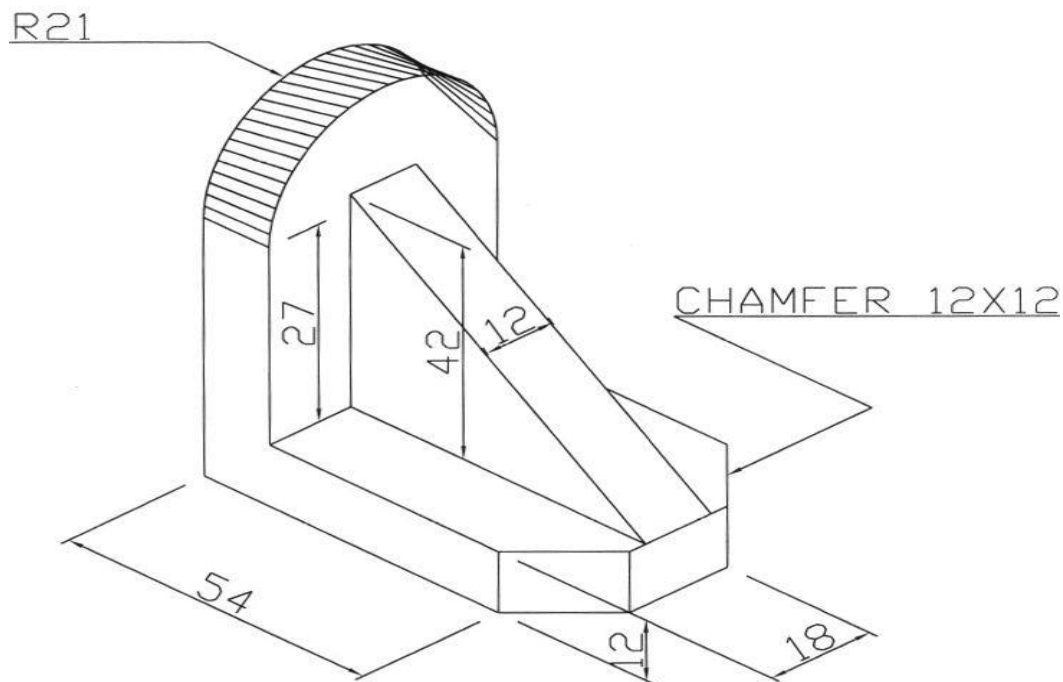
Result: Hence the required 2D isometric diagram is created using Auto CAD.

Exercise – 6

Aim: To create a 2D isometric view of the given diagram using Auto CAD

Procedure:

1. Type limits in command menu & set value to 297,290.
2. Change the units to millimeters from inches and also precision to 0 by clicking format -> units ->ok.
3. To set the paper size type zoom -> enter and type a -> enter in Command bar.
4. Go to drafting settings and turn on isometric snap. Use the F5 key to change between the views of isometric planes.
5. Start from the front view and draw the the line of length of line 54.
6. Draw the semi circle using the Iso circle option from the ellipse command.
7. Continue drawing using F5 and F8 snap keys.
8. Give proper dimensions to the figure and practice at home.



Result: Hence the required 2D isometric diagram is created using Auto CAD.