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Learning Objective:

At the end of this course, you should be able to:

- ☐ apply all basics commands in AutoCAD
- ☐ produce 2D drawing
- ☐ produce orthographic drawing
- ☐ produce isometric drawing
- ☐ produce 3D drawing.

INTRODUCTION

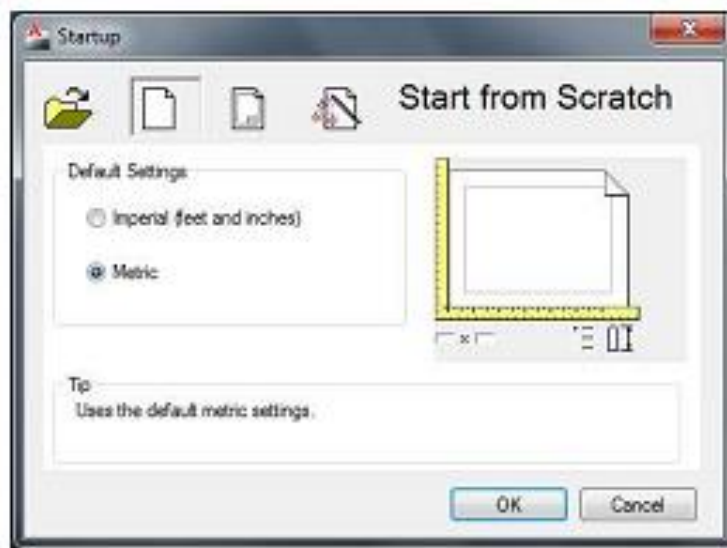
In this lab manual you are going to produce AutoCAD drawings with your computer. The AutoCAD you will learn are basic 2D drawing, orthographic drawing, isometric drawing and basic 3D drawing. You can use this lab manual as a guide for your AutoCAD laboratory practices.

1. Using the basic commands in AutoCAD

It is important for you to understand all basic commands in AutoCAD because it will help to produce an accurate and neat drawing. The basic commands that you must know are draw commands, modify commands and object snap. These commands will be used frequently in any AutoCAD drawings.

1.1 To Begin An AutoCAD Drawing

To begin an AutoCAD drawing, you first begin with “Startup Dialog Box”. Figure shows a “Startup Dialog Box”. With this dialog box, you choose the setting for your drawing. That is, Imperial or Metric system. You also can open a drawing, start a drawing from scratch, use a template or wizard. Click “OK” when you have decided your choice.



Not all the AutoCAD software will automatically generate this dialog box in your program.

You have to activate it yourself.

1.2 Starting a New Drawing

Before you start to draw, you need to decide what system of drawing units that you will use in the drawing, and then choose a drawing template file appropriate for those drawing units.

Choose Drawing Units

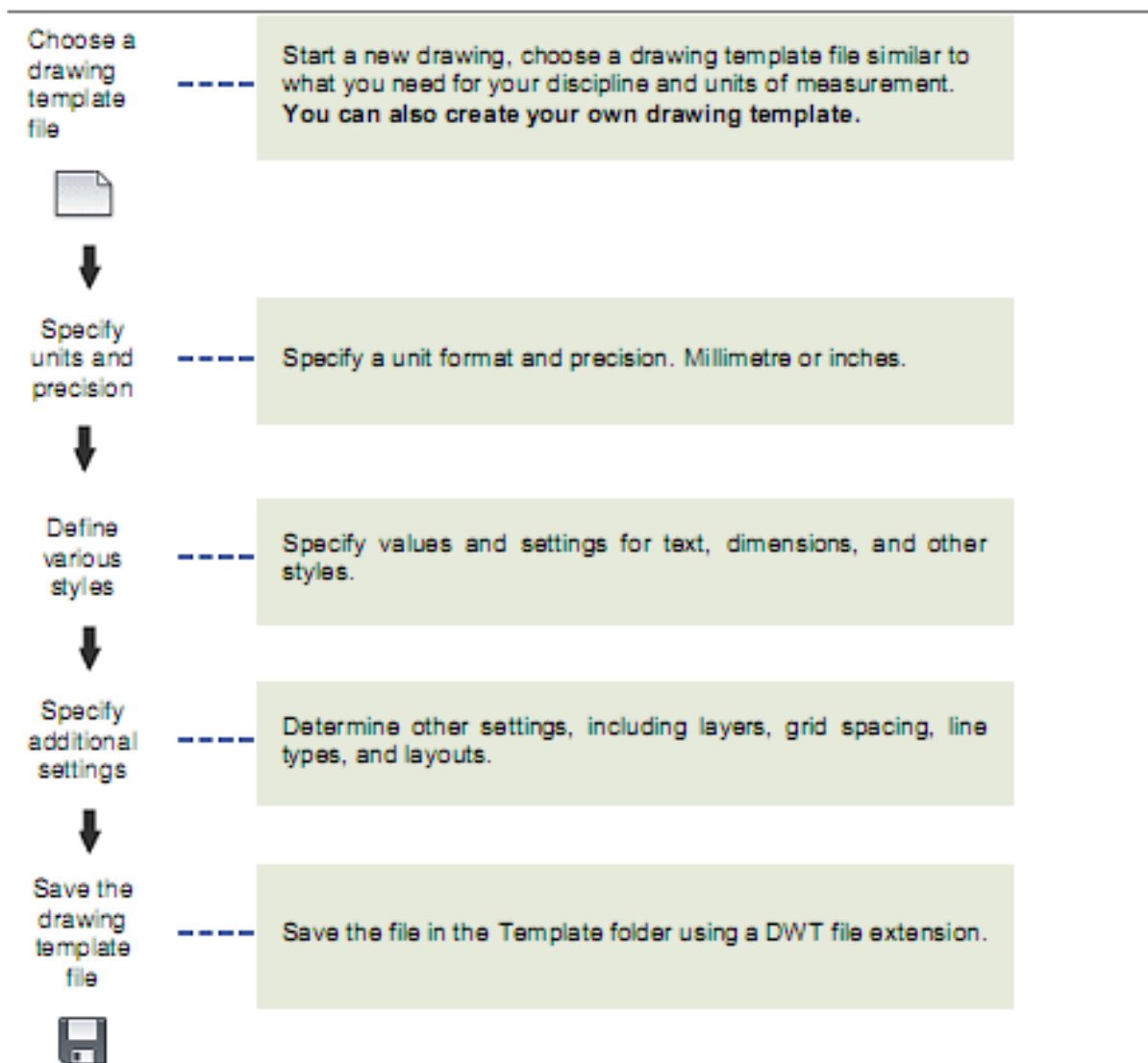
A drawing unit can equal one inch, one millimetre, or any other system of measurement.

Choose a Drawing Template File

When you start a new drawing, AutoCAD accesses a drawing template file to determine many default settings such as unit precision, dimension styles, layer names, a title block, and other settings. Many of the settings are based on whether the drawing template file is intended for use with a drawing created in inches, feet, millimetres, centimetres, or other unit of measurement.

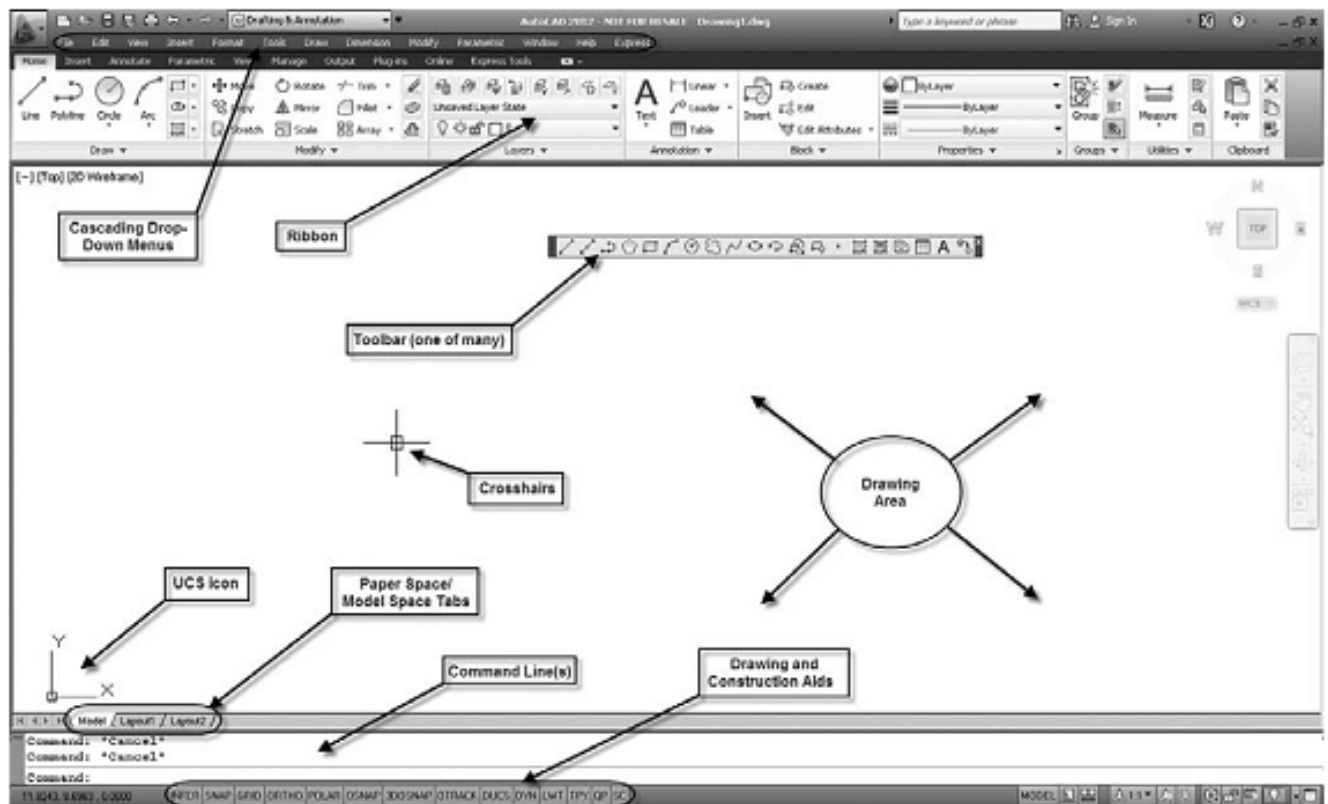
Customize a Drawing Template File

By customizing your own drawing template file, you save yourself a lot of work changing settings, and you also ensure that the settings are standardized.



You can create several drawing template files for different projects, and you can choose one when you click New.

1.3 Autocad Interface



When you start a new drawing, AutoCAD accesses a drawing template file to determine many default settings such as unit precision, dimension styles, layer names, a title block, and other settings. Many of the settings are based on whether the drawing template file is intended for use with a drawing created in inches, feet, millimetres, centimetres, or other unit of measurement.

Drawing Area. The drawing area takes up most of the screen and is colored a dark gray in the default version of the environment. This is where you work and your design appears.

It is best to change the drawing area to black to ease eye strain, as less light will radiate toward you. If you wish to change the color of the drawing area, you need to right-click into “Options...”, choose the Display tab, then Colors... . Finally, change the color from the drop-down menu on the right.

Command Line(s). Right below the drawing area is the command line or, by default, three lines. This is where the commands may be entered and also where AutoCAD tells you what it needs to continue. Always keep an eye on what appears here, as this is one of the main ways that AutoCAD communicates with you. It is colored gray and white by default but is changed (in a manner similar to the drawing area).

UCS Icon. This is a basic X-Y-Z (Z is not visible) grid symbol. It will be important later in advanced studies and 3D. The significance of this icon is great. Y axis is “up” and the X axis is “across.”

Paper Space/Model Space Tabs. These Model/Layout1/Layout2 tabs, not unlike those used in Microsoft’s Excel, indicate which drawing space you are in.

Toolbar. Toolbars contain icons that can be pressed to activate commands. They are an alternative to typing and the Ribbon, and most commands can be accessed this way. AutoCAD 2012 has dozens of them.

Crosshairs. Crosshairs are simply the mouse cursor and move around along with the movements of your mouse. They can be full size and span the entire screen or a small (fleyspeck) size. You can change the size of the crosshairs if you wish, and full screen is recommended in some cases.

Drawing & Construction Aids. These various settings assist you in drafting and modeling. We introduce them as necessary. By default, these aids are in graphical symbol form and some may be activated, which you can determine by observing their color . If they are off, they are gray; if on, then light blue.

Ribbon . This is a relatively new way of interacting with AutoCAD’s commands. The Ribbon first appeared in AutoCAD 2009 and is somewhat similar to the approach used in Microsoft Word, Excel, and PowerPoint.

Cascading Drop-down Menus . This is another way to access commands in AutoCAD.

These menus, so named because they drop out like a waterfall, may be hidden initially,

but you can easily make them visible via the down arrow at the very top of the screen, to

the right of Drafting & Annotation . A lengthy menu appears. Select “Show Menu Bar” toward the bottom, and the cascading menus appear as a band across the top of the screen, above the Ribbon.

1.4 Commands Entry and Using Short-Cuts

You have already learnt the system and basic concepts of AutoCAD. AutoCAD program offer many short-cuts for his users. These short-cut can help you to create your drawings in a simple and quicker way. Let us undertake a few commands.

Using Sub Command Object Snaps (OSNAP)

To create an accurate AutoCAD drawing, you must know how to use sub command Object Snaps (Obsnap) effectively. Object snaps become active when the object snap target box passes over an object location such as an intersection or endpoint. Depending on the complexity of your drawing, you can increase or decrease the area that is influenced by object snaps.

Here are some sub-command of Object Snaps that you will used frequently in AutoCAD drawing as follow:

(i) Endpoint

Snap to Endpoint - Snaps to the closest endpoint of an object. Figure shows a line in which a circle will be drawn at end point A. In Figure the target box with a symbol square has snap at the end point A. You just need to click at your mouse; a circle will be drawn at endpoint A as shown in Figure.

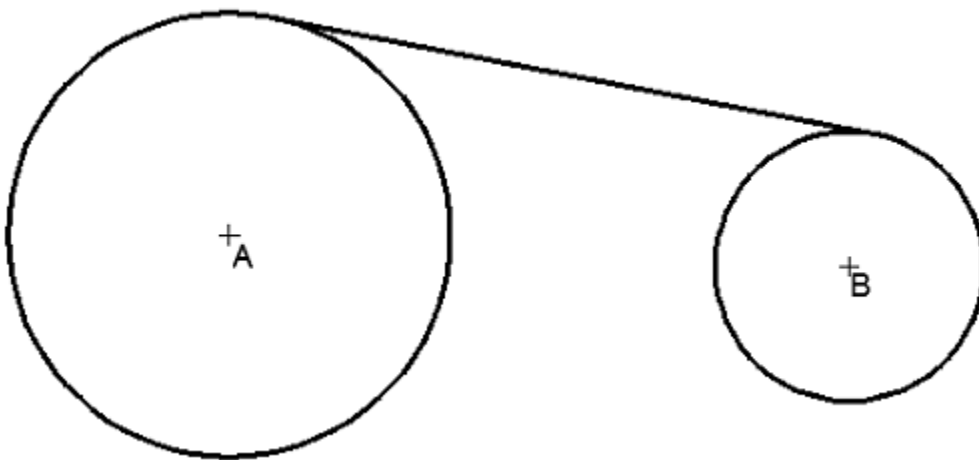


(ii) Midpoint

Snap to Midpoint mean snaps to the midpoint of an object. Figure shows a line in which a circle will be drawn at midpoint A. In Figure the target box with a symbol triangle has snap at the midpoint A. You just need to click at your mouse; a circle will be drawn at endpoint A as shown in Figure.



Figure shows how to used object snap – snap to tangent which is used for draw line tangent to two circles.



Your steps will be as follow:

Command: line

(select snap to tangent for circle A and select again snap to tangent for circle B)

Ray

A ray is a line in three-dimensional space that starts at a point you specify and extends to infinity. Unlike construction lines, which extend in two directions, rays extend in only one direction. Using rays instead

of construction lines can help reduce visual clutter. Like construction lines, rays are ignored by commands that display the drawing extents.

Command : Ray

Specify start point: -specify the coordinate -

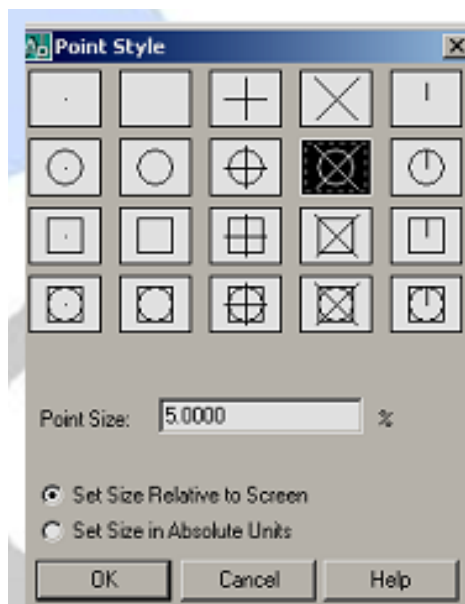
Specify through point: - specify the coordinate -

Point

Point objects are useful as nodes or reference geometry for object snaps and relative offsets. It can also be placed on existing drawings. This command can be activated by

Command : ddptype

Specify point type



Command : point

Specify start point: -specify the coordinate -

Deactivate this command by pressing – ESCAPE - button.

When user uses the Draw Menu to insert point, there are two options, single point and multiple point, which means that if user choose single point, command prompt will ask to

specify a point and then the command will be deactivate while on multiple point the command will not deactivate automatically and user has to deactivate it.

Divide

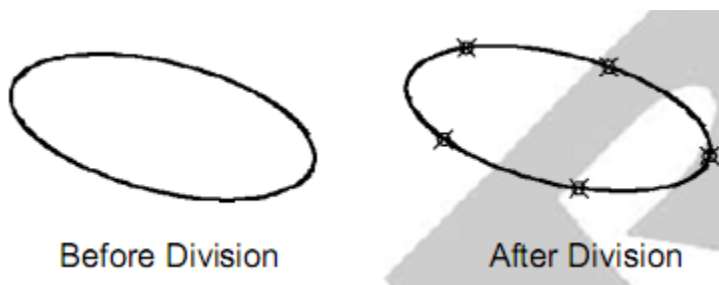
This command places evenly spaced point objects or blocks along the length or perimeter of an object. This command can be activated by

Command : divide

Select object to divide: - click on object which is to be divided -

Enter the number of segments or [Block]: - enter a numeric value -

If user has set the appropriate size and style of point than the object will seem to be divided into mentioned number of segments by placing points.



Measure

It places the point on selected object at a measure distance which is given by the user, it provides on screen information that the in how many segments, an object can be divided for a specified segment length.

Command : measure


Select object to measured: - click on object which is to be divided -

Multiline

Multiline is used to draw the piping networks, road paths etc. It draws two parallel line with the given distance. It draws a continuous line till the command is deactivated. This command can be activated by

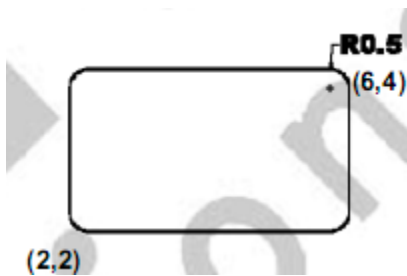
Chamfer

First draw a rectangle. When user chooses this option, it asks for chamfer distance and draw rectangle at the given chamfer.

1. Click Home tab ➤ Modify panel ➤ Chamfer. 
2. Enter **d** (Distances).
3. Enter the first chamfer distance.
4. Enter the second chamfer distance.
5. Select the lines for chamfering.

Fillet

When user chooses this option, it asks for fillet radius and draw rectangle at the given fillet.



Other Methods to draw the Rectangle

Area

This is another method to draw the rectangle, in this method AutoCAD asks the area of rectangle than a dimension and calculate other dimension using area and draw the rectangle using these dimensions, a corner fixed at a given coordinate. This option is available after entering the corner coordinate.

Command: rectang

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]:

-corner coordinate-

Specify other corner point or [Area/Dimensions/Rotation]: d

Specify length for rectangles <6.0000>: 3

Specify width for rectangles <5.0000>: 4

Specify other corner point or [Area/Dimensions/Rotation]: - click mouse above-

Ellipse

Ellipse is usually defined by its major and minor axes lengths, in AutoCAD the default way to draw an ellipse is same but there are more methods to draw the ellipse. Following is the way to activate the ellipse command and entering information.

Command: ellipse

Specify axis endpoint of ellipse or [Arc/Center]: - (1) specify end pt of major axis -

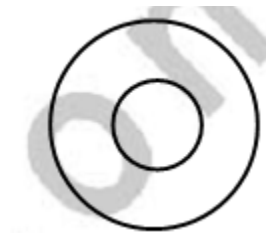
Specify other endpoint of axis: - (2) specify other end pt of major axis -

Specify distance to other axis or [Rotation]: - (3) specify end pt of minor axis -

Donut

Donut is consists of two circle which is used to mention

Some reference on drawing.



Command: donut

Specify inside diameter of donut <0.5000>: - internal circle diameter -

Specify outside diameter of donut <1.0000>: - external circle diameter -

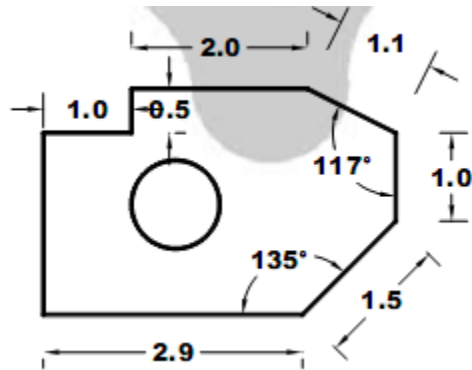
Specify center of donut or <exit>: - center of donut to be placed -

Exercise

Draw a rectangle of length 4 and width of 3 with a chamfer of 0.3.

Draw a rectangle of length 4 and width 3 with a fillet of 0.3.

Draw the following objects.



Circular Arc

Simple arc is a part of circumference of a circle, so to draw an arc, user require some parameters most of them are same as that of circle. Below is the default method to draw an arc, known as 3 point arc which fixes as arc on given three points. Arc command can be activated by

3 Point

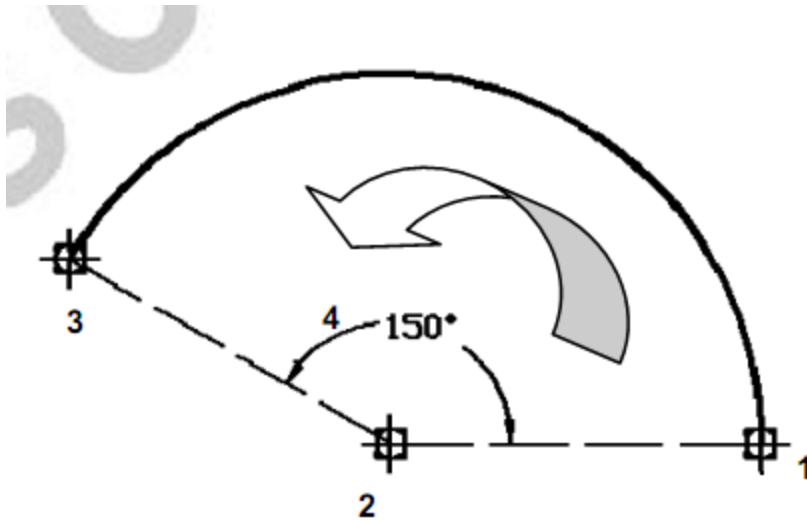
Specify start point of arc or [Center]: - (1) enter 1st point on curve -

Specify second point of arc or [Center/End]: - (2) enter 2nd point on curve -

Specify end point of arc: - (3) enter 3rd point on curve -

Start, Center, End

This method ask for starting point of arc, center point of arc and than end point but user has to keep in mind that if user enters the starting point at right side of center than arc will be drawn in such a way that the angle subtended from center will always be anti-clockwise and from east, so the arc will be upside of center while if starting point is at left side, than arc will be in downward.



Specify start point of arc or [Center]: c

Specify center point of arc: - (1) enter center coordinate -

Specify start point of arc: - (2) enter start point coordinate -

Specify end point of arc or [Angle/chord Length]: - (3) enter end point coordinate -

Start, Center, Angle

This method is same as that of above method but here, user has to enter the angle instead of end point. In this method the same method is adopt to measure the angle from starting point, angle measured is from East in Anti-clockwise direction.

Command: arc

Specify start point of arc or [Center]: - (1) enter start coordinate -

Specify second point of arc or [Center/End]: c

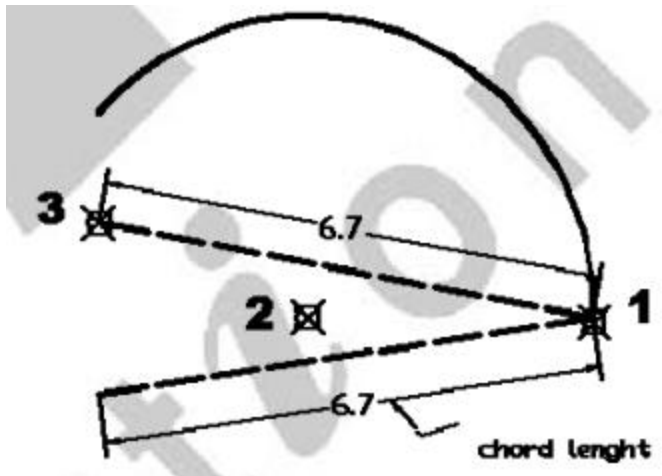
Specify center point of arc: - (2) enter center coordinate -

Specify end point of arc or [Angle/chord Length]: a

Specify included angle: - (3) enter Angle -

Start, Center, Length

In this method, AutoCAD ask for starting point, than the center point of arc than the length of chord, which is the projection of the curve. Increasing the length of curve, increase the circumference of arc.



Command: arc

Specify start point of arc or [Center]: - (1) enter start coordinate -

Specify second point of arc or [Center/End]: c

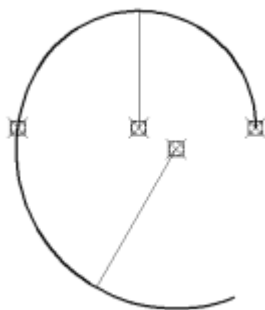
Specify center point of arc: - (2) enter center coordinate -

Specify end point of arc or [Angle/chord Length]: L

Specify length of chord: - (3) enter length or coordinate of end point -

Continue (Arc Drawing Method)

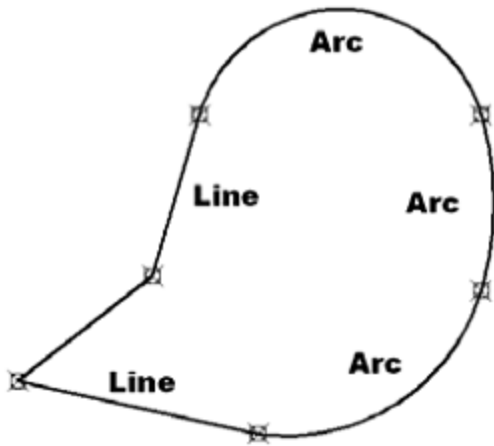
This method is basically used to continue the last drawn arc, this method can be activated by selecting the previously drawn arc



Draw > Arc > Continue –

Polyline

Polyline is a single entity which allows user to draw multiple items i.e. line, arcs etc and at the end of Drawing AutoCAD joins them as a single entity, but user has to keep in mind that the multiple items are drawn in such a way that each new segment is drawn at the end of previous segment.



Command: pline

Specify start point: - specify the starting coordinate -

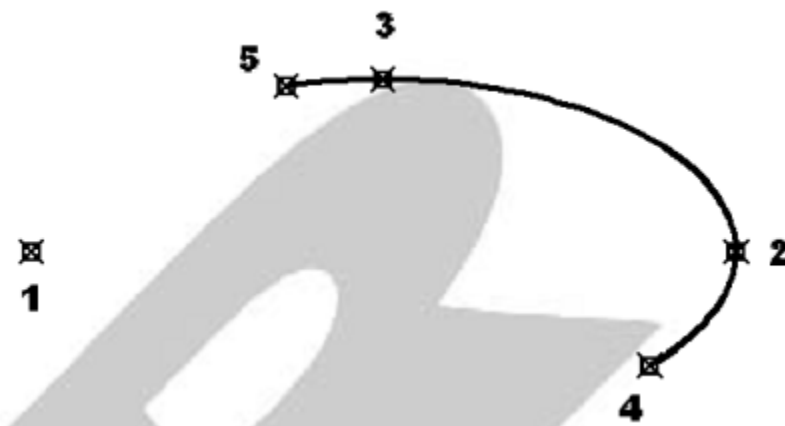
Current line-width is 6.3782

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: - specify next coordinate or switch to other options. -

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: - specify the last coordinate -

Elliptical Arc

This option draws an elliptical arc, off course this is a part of circumference of an arc. This type of arc can be drawn in the same way as that of ellipse, the difference is only to define start and end point of arc which is to be taken from the ellipse.



Command: ellipse

Specify axis endpoint of ellipse or [Arc/Center]: a – type 'a' to activate arc option -

Specify axis endpoint of elliptical arc or [Center]: - (1) Enter Major axis end point coordinate -

Specify other endpoint of axis: - (2) Enter Major axis other end point coordinate -

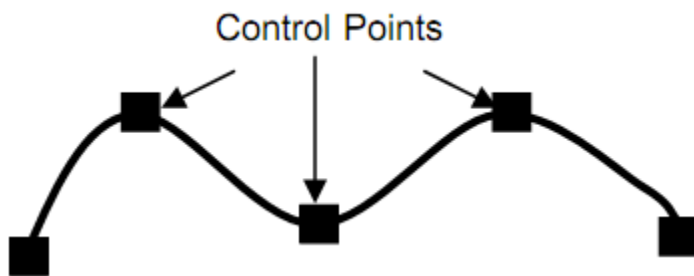
Specify distance to other axis or [Rotation]: - (3) Enter Minor axis end point coordinate -

Specify start angle or [Parameter]: - (4) Enter angle subtended by arc start from point 2 -

Specify end angle or [Parameter/Included angle]: - (5) Enter angle subtended by arc end from point 2 -

Spline

A spline is a smooth curve that passes through or near a given set of points. You can control how closely the curve fits the points. It fits a smooth β -spline curve on given control points. Spline can be smoothen by increasing the control points on spline, which are editable. More control points will increase the Order of polynomial of equation and hence the more system resources are required to draw.



Command: spline

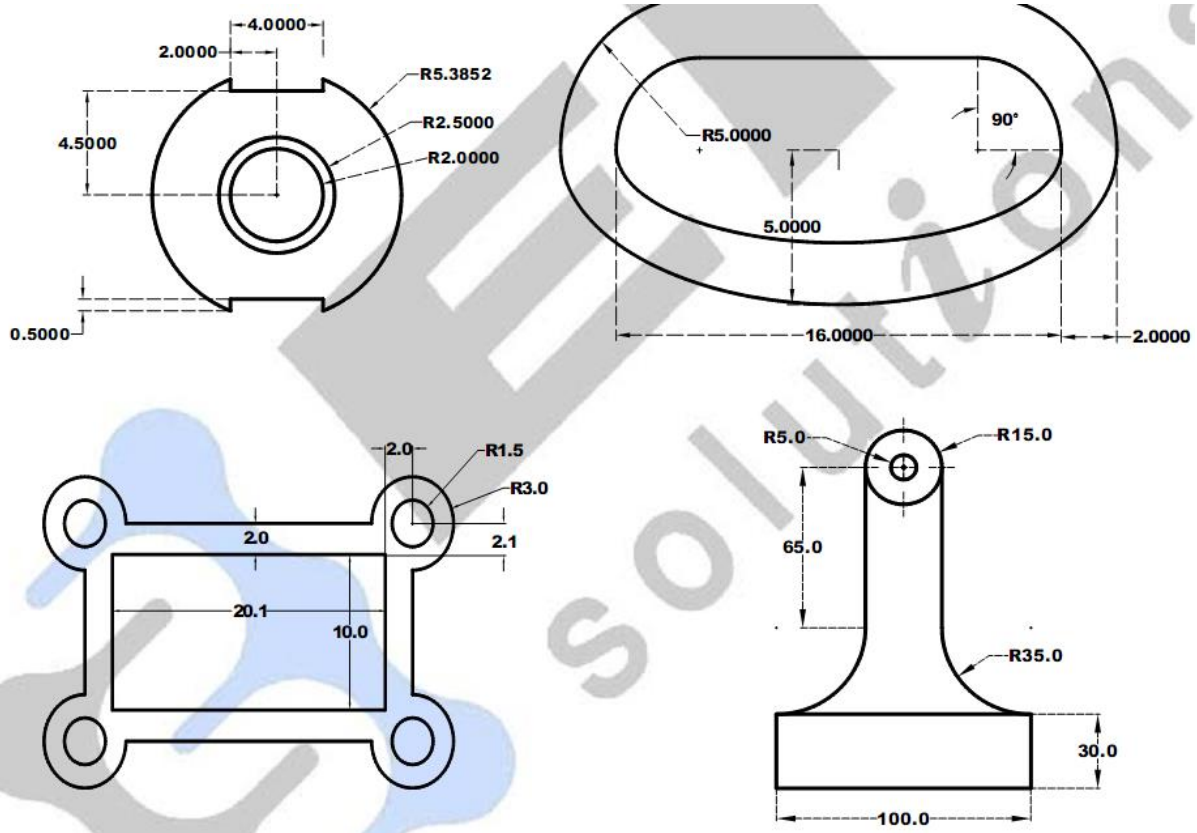
Specify next point: - specify start point -

Specify next point or [Close/Fit tolerance] <start tangent>: - specify other control point -

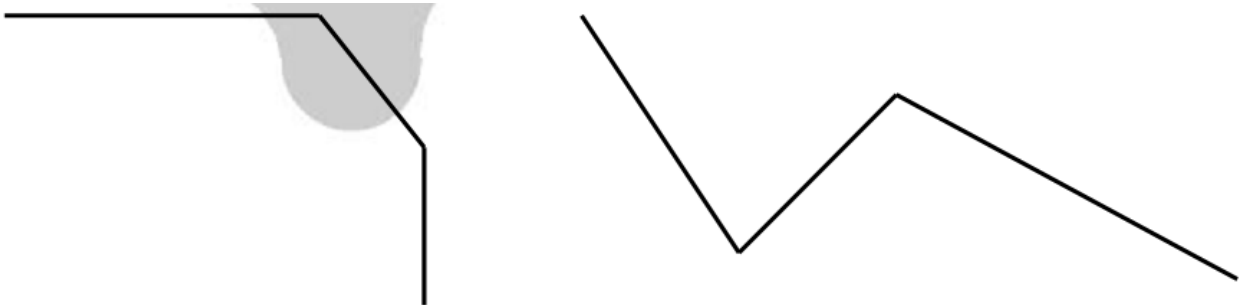
Specify next point or [Close/Fit tolerance] <start tangent>:- specify other control point or

press c to close the curve.-

Draw the following objects.



Convert the following polylines into spline

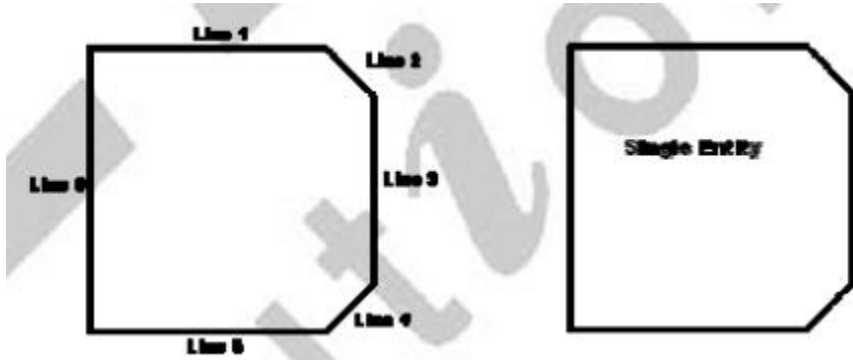


Region, Hatches & Text

Creating Regions

Regions are 2D enclosed areas which are created from closed shapes called loops. A loop is a curve or a sequence of connected curves that defines an area on a plane with a boundary that does not intersect itself. Loops can be combination of lines, polyline, circles, arcs, ellipse arcs, splines, 3D faces, trace and solids. You cannot form open objects that interest to form a closed area for example, intersecting arcs or self-intersecting curves.

To perform the region command, It is necessary to first draw a closed loop, when user draw the closed loop, the entities which draws it will be selected individually



Command: region Select objects: 1 found

Select objects: 1 found, 2 total

Select objects: 1 found, 3 total

Select objects: 1 found, 4 total

Select objects: 1 found, 5 total

Select objects: 1 found, 6 total

Select objects:

1 loop extracted.

1 Region created.

User has to keep in mind that there is a major difference between polyline joining and region, that is, Polyline joining, joins the line and curves together to form the polyline either the resultant shape is opened or a closed loop, while 'region' creates the area inside the closed loop.

Hatch & Color Gradient

Hatch and color gradient is used to fill an area (either circle, rectangle or other closed loop region) by consecutive lines, brick or colors to show the cross section of a part or other notation. It is a very useful tool to differentiate the objects in drawing. To perform the hatch and gradient, it is necessary to convert the closed loop into region using the 'region' command. To draw the hatches or color gradient, the steps are as follows

Command: Hatch

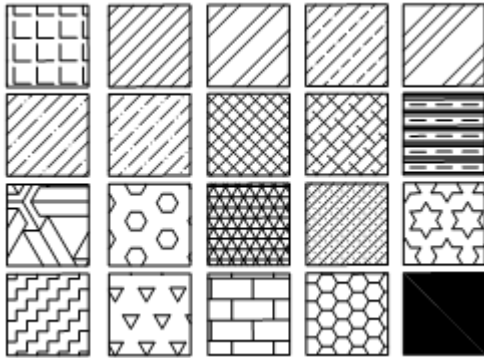
At the most basic level, AutoCAD's hatch command requires only four steps:

Step 1. Pick the hatch pattern you want to use.

Step 2. Indicate where you want the pattern to go.

Step 3. Fine-tune the pattern by adjusting scale and angle (if necessary).

Step 4. Preview the pattern and accept it if OK.



Select object

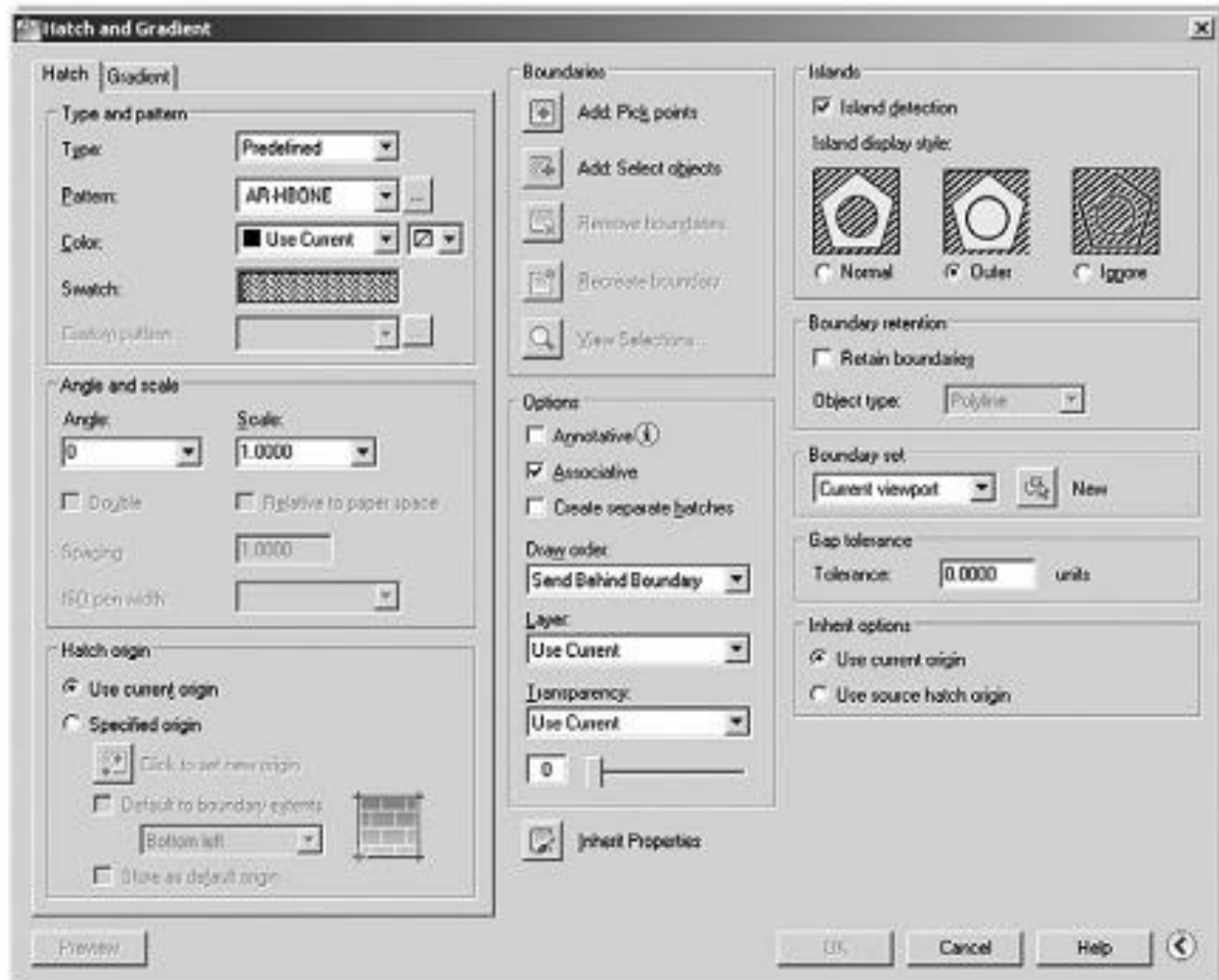
Upper ribbon will show settings that can be changed.

Gradient is similar to Hatch.

Inserting Text

Text in drawing is used for annotations. It is used to write important information and notes on drawing.

There are two categories in this feature which are Multi-line Text and Single Line text. Single line text is an old method used in AutoCAD but through multi-line text there are several new features added to modify text same as that in any word processor packing. 'Insert text' command can be activated by



Fine-Tune the Pattern by Adjusting Scale and Angle (If Necessary)

TEXT

This is your basic text creation command, and it creates a field anywhere you click on the screen, into which you can type whatever text you need. It does have a carriage return that goes to another line upon pressing Enter, so you need to press Enter twice to get out of the field. While an unlimited number of lines of text can be typed, typically regular text is used for only a single line, as multiple lines of text are not joined together in paragraph form and cannot be formatted to any significant extent.

Command: mtext

Specify first corner: - specify a point -

Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]:

- specify other corner to draw a rectangle inside which text is to be written-

A field will appear on working area as defined by the user and a property tool bar for text will appear which contains the properties of text as that of in word processors.

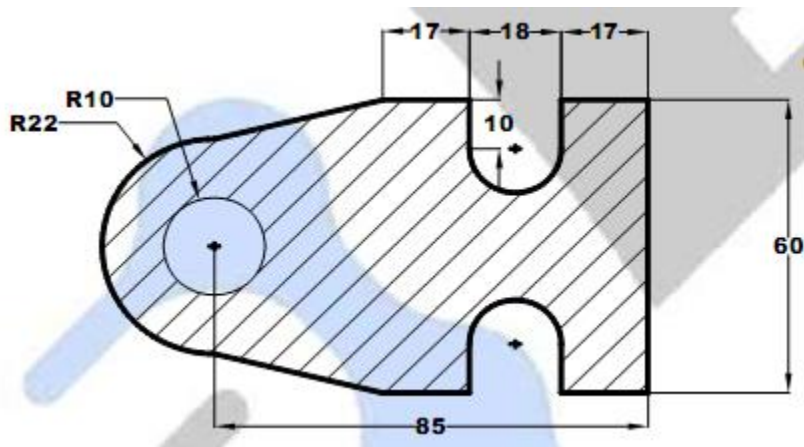
You can edit text with **mtedit** command.

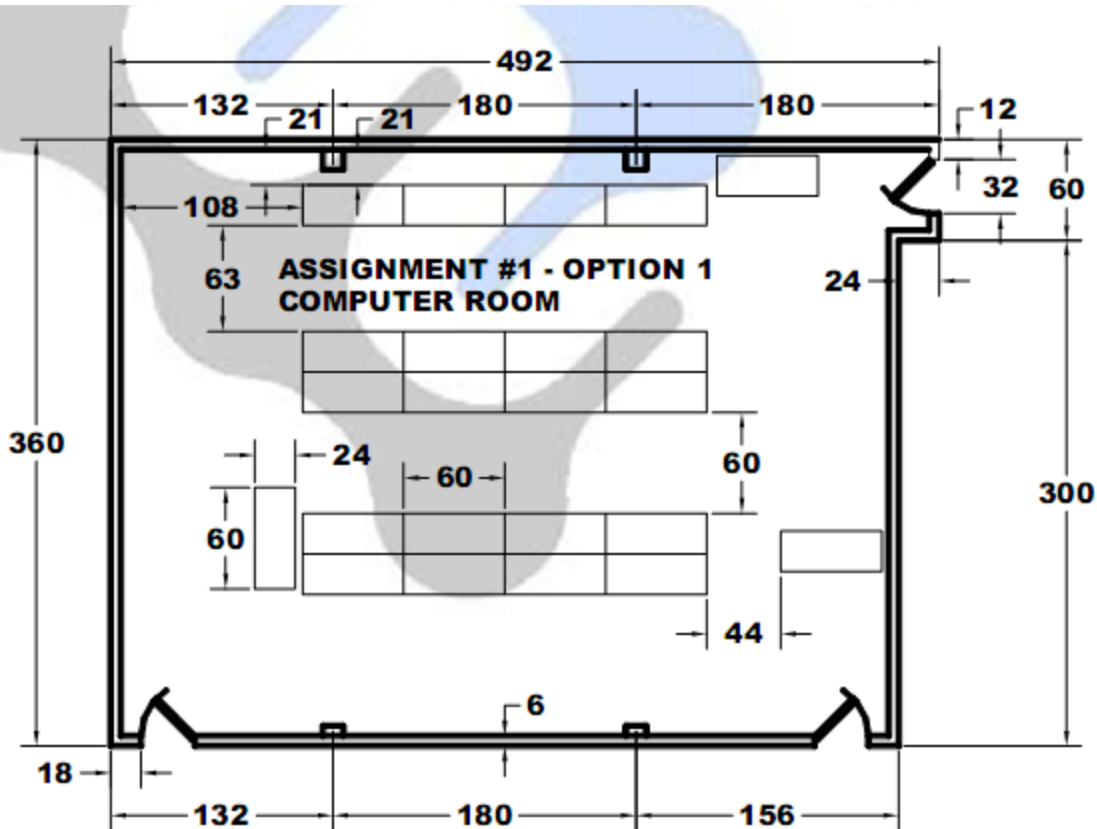
Inserting Table

Table is an important tool in drawing to show the BOMs (Bill of Materials) and other data. It is also used to show the legends on drawing. Table can be inserted in drawing by

Command: table

When the 'table' command will be executed, a window will appear on screen to enter the various properties of table which includes number of rows, number of columns, height, width or to either insert the table by drawing it on screen or on a certain coordinate.





Erase

Erase command is used to delete a single or multiple items, which depends on the selection window that after activation of this command, what part/parts/object or objects are selected. The command can be activated by

Step 1. Begin the erase command via any of the preceding methods.

○ AutoCAD says: Select objects:

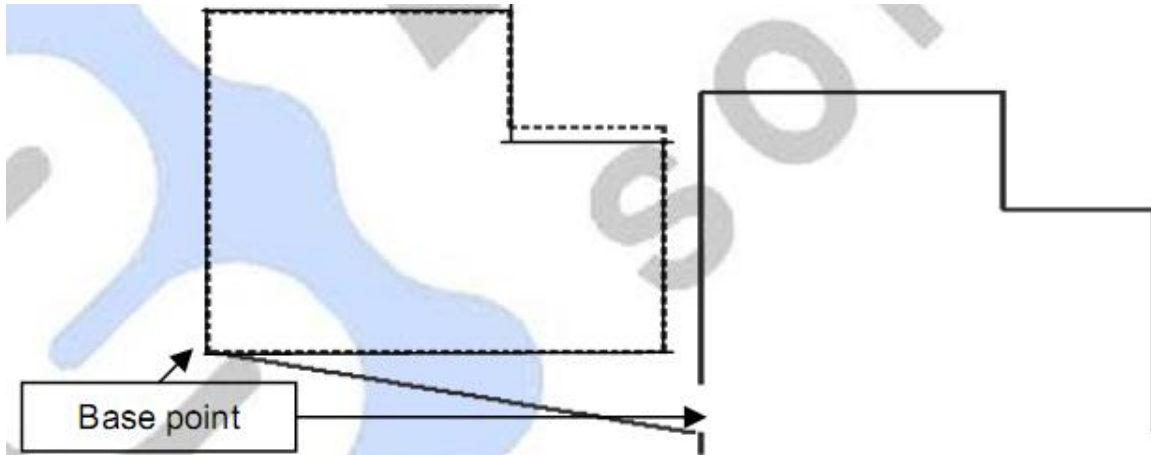
Step 2. Select any, positioning it over that object (not in the empty space), and left-clicking once.

○ AutoCAD says: Select objects: 1 found. The object becomes dashed. AutoCAD continues to ask you: Select objects: .

Step 3. Press Enter and the object will disappear.

Copy

This command is used to copy an object to another position, but it differs from traditional copy methods, in AutoCAD copy method, when user activated the 'copy command', software asks for base point, this base point is the reference point from where the part is to be take, consider the figure below.



Step 1. Begin the copy command via any of the preceding methods.

- AutoCAD says: Select objects:

Step 2. Select an object by positioning the mouse over that object and left-clicking once.

- AutoCAD says: Select objects: 1 found. The object will become dashed.
- AutoCAD then asks you again: Select objects:

Step 3. Unless you have more than one object to copy, you are done, so press Enter.

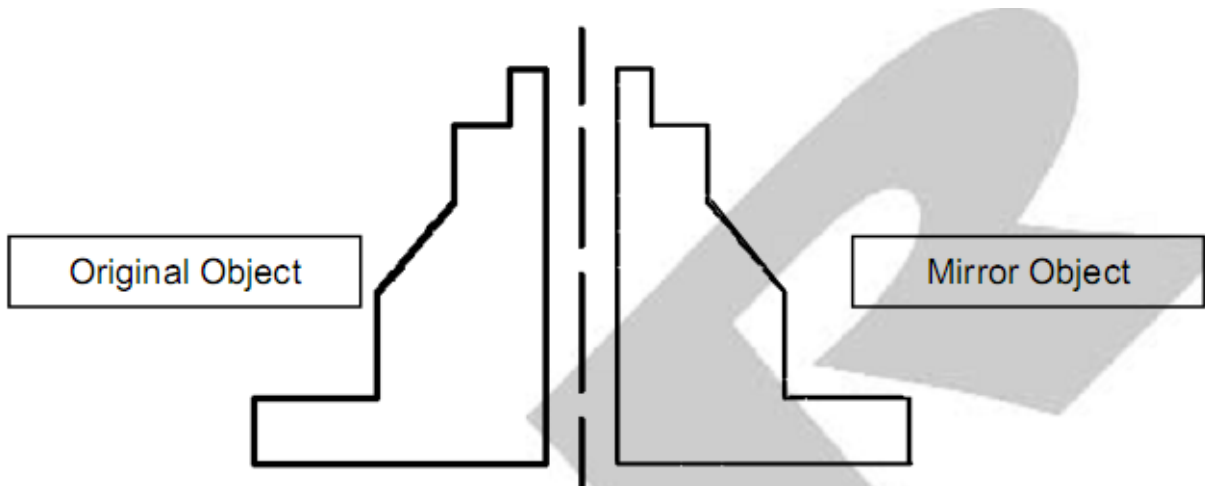
- AutoCAD says: Specify base point or [Displacement/mOde] <Displacement>:

Step 4. Left-click anywhere on or near the object to “pick it up”; this is where you are copying it from

- AutoCAD says: Specify second point or <use first point as displacement>:

Mirror

This command is useful to create the symmetrical objects, in this case, user has to draw the objects half part before the mirror line than activate this command, selecting the object, specifying the mirror line, the mirror will be created.



Command: mirror

Select objects: - Select single or multiple objects -

Specify first point of mirror line: - Select Specify mirror line -

Erase source objects? [Yes/No] <N>: -yes to keep original object -

Offset

This command is used to create the parallel of line, segment or closed loop.

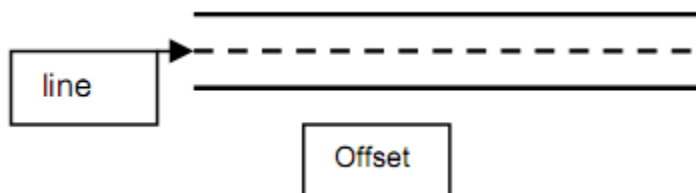
Command: offset

Specify offset distance or [Through/Erase/Layer] <Through>: - distance -

Select object to offset or [Exit/Undo] <Exit>: - Select single or multiple objects -

Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: -side-

Select object to offset or [Exit/Undo] <Exit>: e



Array

This modify command is useful to create several type of pattern, either linear, angular or circular. This command used when repetitive objects are to be drawn in drawing, some examples of linear, angular and circular array are shown

Ribbon: Home tab→Modify→Polar Array



Step 1. Start up the array command via any of the preceding methods.

- AutoCAD says: Select objects:

Step 2. Select the some block.

- AutoCAD says: 1 found

Step 3. Press Enter.

- AutoCAD says: Type = Polar Associative = Yes

Specify center point of array or [Base point/Axis of rotation]:

Step 4. You successfully selected the object to array . You now need to specify the center point of the array . Simply select the center of the large circle using the CENTER OSNAP.

- AutoCAD says: Enter number of items or [Angle between/Expression] <4>:

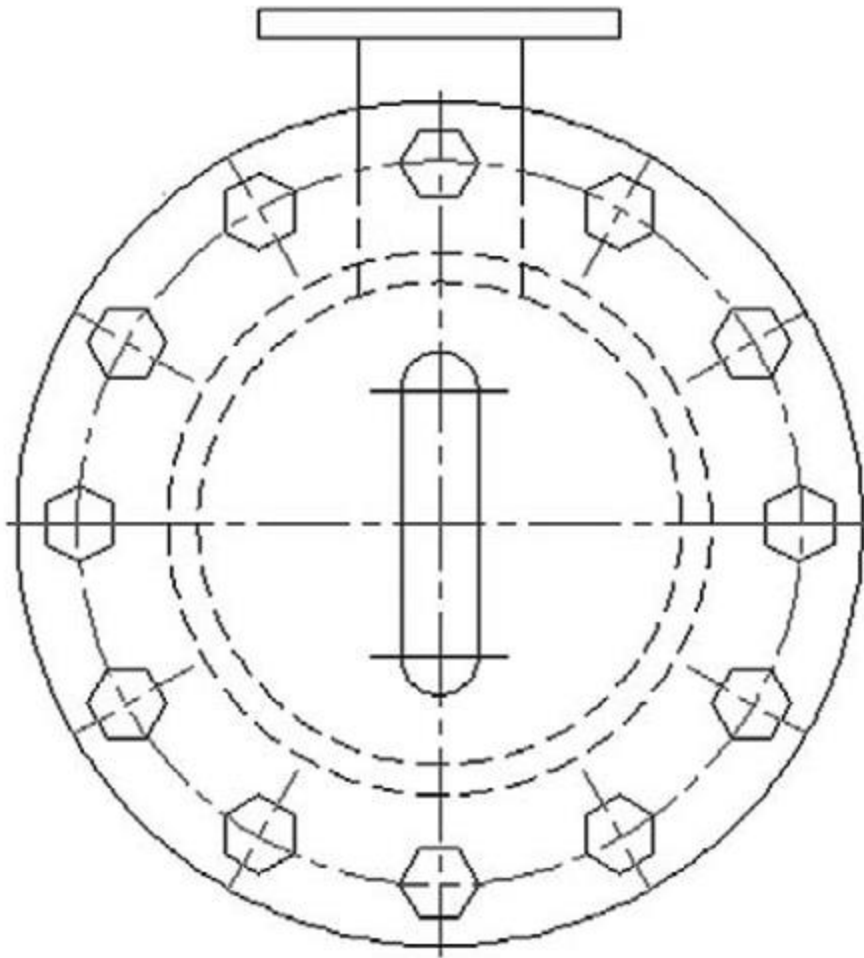
Step 5. Here you need to state how many objects to create . Enter 10 , noting that you can dynamically do this as well by moving the mouse around the circle and watching new object appear.

- AutoCAD says: Specify the angle to fill (=ccw, =cw) or [EXpression] <360>:

Step 6. Here, you need to state if you want the chair pattern to go all the way around the circle or part thereof. In this case, we fill up the circle, so just press Enter. You could have just as easily entered a numerical degree value for any part of a circle.

- AutoCAD says: Press Enter to accept or [ASsociative/Base point/Items/Angle between/Fill angle/ROWS/Levels/ROTate items/eXit]<eXit>

Step 7. Your array is done, so just press Enter to accept.



RECTANGULAR ARRAY

The idea behind a rectangular array is not fundamentally different from a polar one. We are looking to replicate objects, but this time in rows (horizontal) and columns (vertical). In 3D, you can also do levels, which we do not address here. An example, as shown in Fig, may be columns of a warehouse. If they are spread out evenly, then this is the perfect tool for this. You simply draw one I-beam column, make a block out of it, and array it up and across the appropriate area.

Step 1. Start up the array command via any of the preceding methods.

○ AutoCAD says: Select objects:

Step 2. Select the I-beam column block.

○ AutoCAD says: 1 found

Step 3. Press Enter.

- AutoCAD says: Type = Rectangular Associative = Yes

Specify opposite corner for number of items or [Base point/

Angle/Count] <Count>:

Step 4. You have now successfully selected the object to array. Now you need to specify the number of rows, columns, and of course the distance between them. You immediately notice that you can do this “dynamically” by moving the mouse around. A diagonal path is most effective

Step 5. This dynamic method is not really appropriate for setting distances between the columns, which are random values for now, but it does make creating the pattern easy. If you want to specify actual values, you need to enter c for Count

- AutoCAD says: Enter number of rows or [Expression] <4>:

Step 6. Enter a value for the rows: 5 in this example.

- AutoCAD says: Enter number of columns or [Expression] <4>:

Step 7. Enter a value for the columns: 5 in this example.

- AutoCAD says: Specify opposite corner to space items or [Spacing] <Spacing>:

Step 8. Enter s for Spacing. This indicates that you want to manually enter the appropriate distances. Note that, once again, you could do this dynamically.

- AutoCAD says: Specify the distance between rows or [Expression] <39.000>:

Step 9. Enter a value for the row distance, such as 60. Knowing the I-beam sizing pays off here, because a value significantly less than 60 would have had the rows sitting on top of each other.

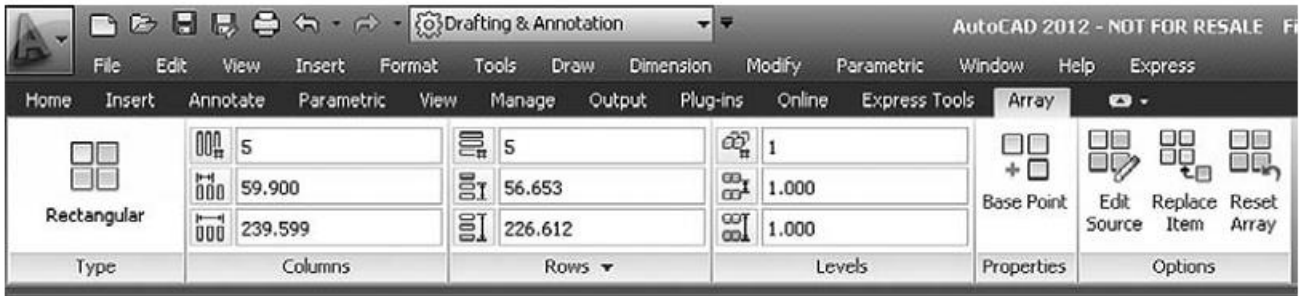
- AutoCAD says: Specify the distance between columns or [Expression] <30.000>:

Step 10. Enter a value for the column distance, such as 60 again. Knowing the I-beam sizing pays off here as well, because a value significantly less than 60 would have had the same “crowding” effect on the columns.

- AutoCAD says: Press Enter to accept or [ASsociative/Basepoint/

Rows/Columns/Levels/eXit]<eXit>:

Step 11. Press Enter to accept the array as is.



PATH ARRAY



Step 1. Start up the array command via any of the preceding methods.

- AutoCAD says: Select objects:

Step 2. Select the rectangle.

- AutoCAD says: 1 found

Step 3. Press Enter.

- AutoCAD says: Type = Path Associative = Yes

Select path curve

Step 4. You successfully selected the object to array . Now, select the path by clicking on the arc.

- AutoCAD says: Enter number of items along path or [Orientation/

Expression] <Orientation>:

Step 5. Here, you are asked to specify how many items you wish to fit along the path. You can do this dynamically via the mouse or type in a value, let us say 10.

- AutoCAD says: Specify the distance between items along path or

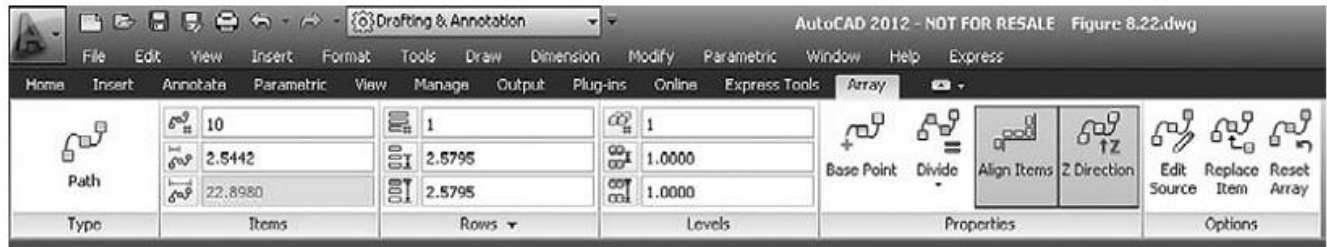
[Divide/Total/Expression] <Divide evenly along path>:

Step 6. Here, you can specify the distance between objects, but usually you just want to accept the default choice of dividing evenly along the path, so press Enter.

○ AutoCAD says: Press Enter to accept or [Associative/Basepoint/

Items/Rows/Levels/Align items/Z direction/eXit]<eXit>

Step 7. This final menu introduces some options, but we explore them separately in a moment so just press Enter once again.



Move

This command is used to move an object from one coordinate to another from a fixed base point called base point, this command can be activated by

Command: move

Select objects: - Select single or multiple objects –

Specify base point or [Displacement] <Displacement>: - Select base point -

<use first point as displacement>: - select target coordinate -

Rotate

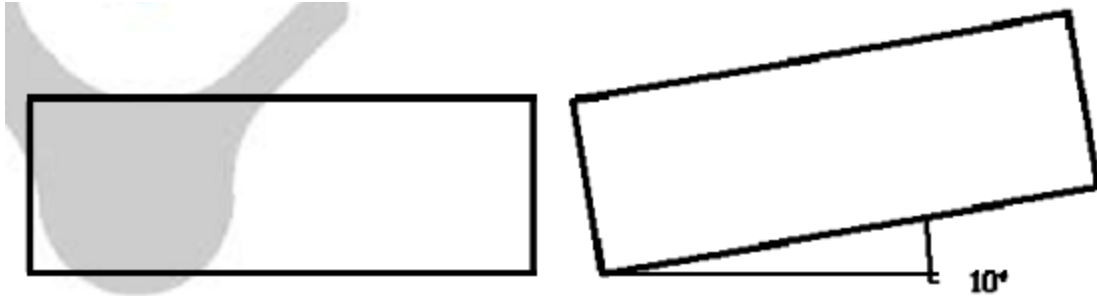
This command is used to rotate object or object about a fixed point (base point). This command is activated by

Command: rotate

Select objects: - Select single or multiple objects –

Specify base point: - Select base point -

Specify rotation angle or [Copy/Reference] <0>: - specify angle -



Scale

This command is used to increase the size of object by a certain factor or dimension. This command can be used by

Command: scale

Select objects: - Select single or multiple objects –

Specify base point: - Select base point -

Specify scale factor or [Copy/Reference] <2.0000>: - scale factor or 'c'/'r' -

Stretch

This command is used to stretch the object or distort its original shape by replacing its grip. This command can be activated by

Command: stretch

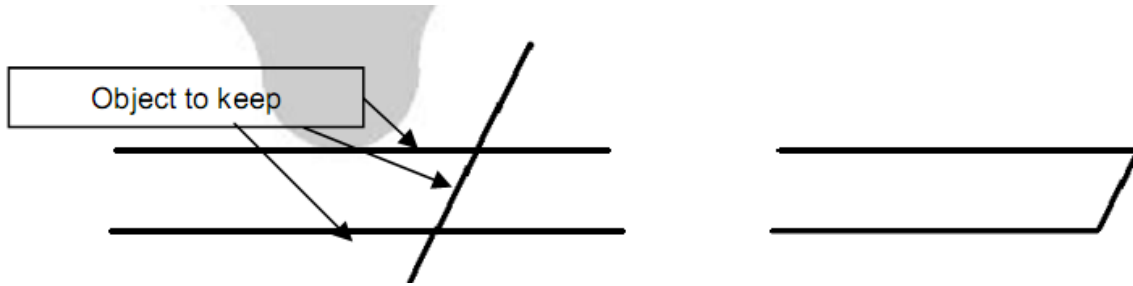
Select objects: - Select single objects -

Specify base point or [Displacement] <Displacement>:- Select base point -

Specify second point or <use first point as displacement>:- select target coordinate -

Trim

This command is used to shorten or lengthen an object to meet the target object. Following figure illustrate the concept of trim.



Command: trim

Select objects or <select all>: - (1) Select objects which are to be keep -

Select object to trim or shift-select to extend or

[Fence/Crossing/Project/Edge/eRase/Undo]: - (2) Select segments which are to be trim -

Extend

This command is used to extend a line till another line or extent. Extend command can be activated by

Command: extend

Select objects or <select all>: - (1) Select objects which are to be keep -

Select object to extend or shift-select to trim or

[Fence/Crossing/Project/Edge/Undo]: (2) Select segment which is to be extend -

Join

This command is used to join the broken object, i.e. it join the line with another line or arc which can be coincide with it. This command can be activated by

Command: join

Select source object: - select first part of segment to be join -

Select lines to join to source: - select 2nd part of segment to be join –

Select lines to join to source: - right click to deactivate -

Chamfer

This command convert the edge into chamfer, it can be activated by

Command: chamfer

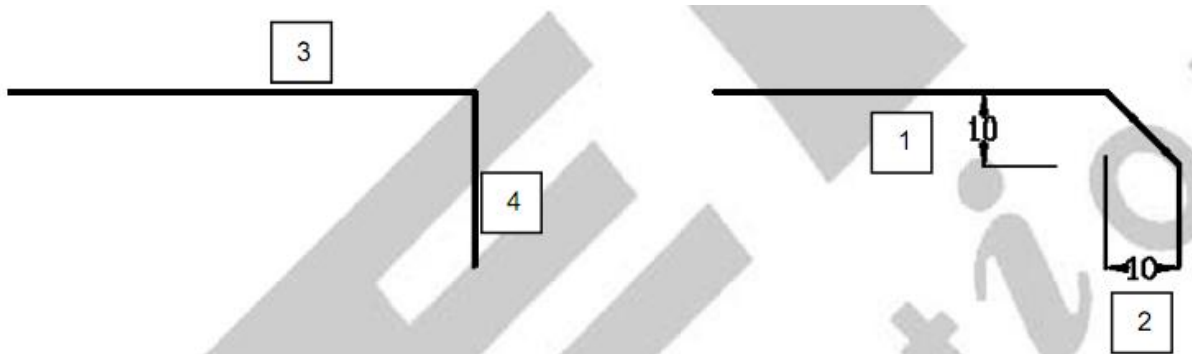
Select first line or [Undo/Polyline/Distance/Angle/Trim/ mEthod/Multiple]: d

Specify first chamfer distance <0.0000>: (1) – enter value -

Specify second chamfer distance <10.0000>: (2) – enter value -

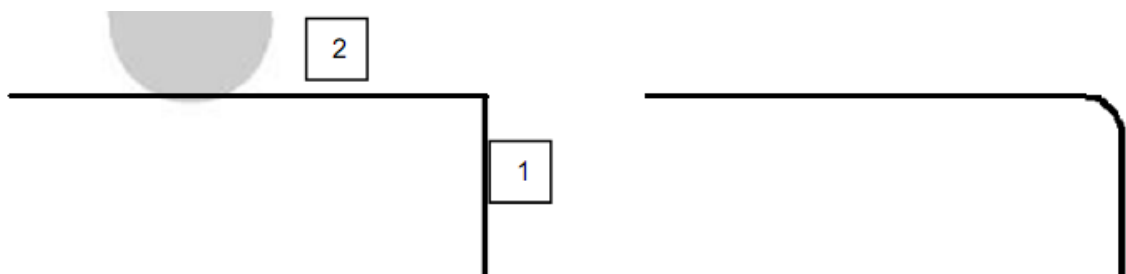
Select first line or [Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple]: -(3) select 1st line-

Select second line or shift-select to apply corner: -(4) select 2nd line-



Fillet

This command is used to replace the edge with round with a specific radius defined by user as shown in figure, this command can be activated by



Command: fillet

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: r

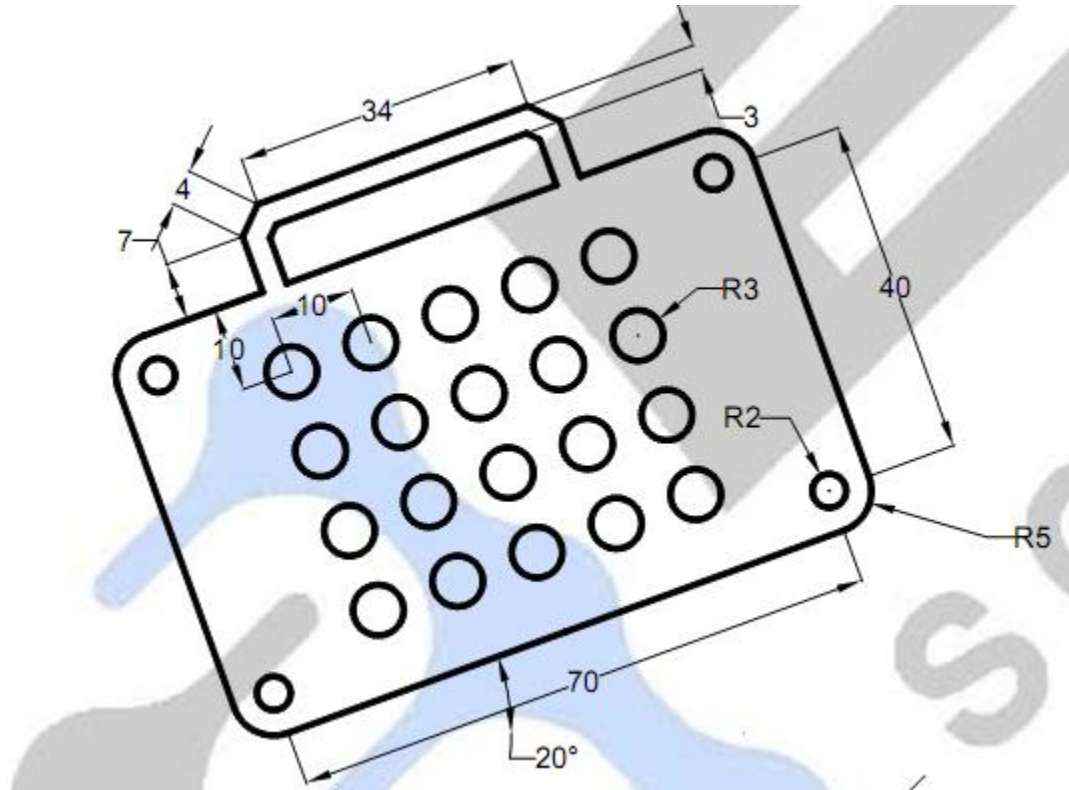
Specify fillet radius <0.0000>: (3) – enter value -

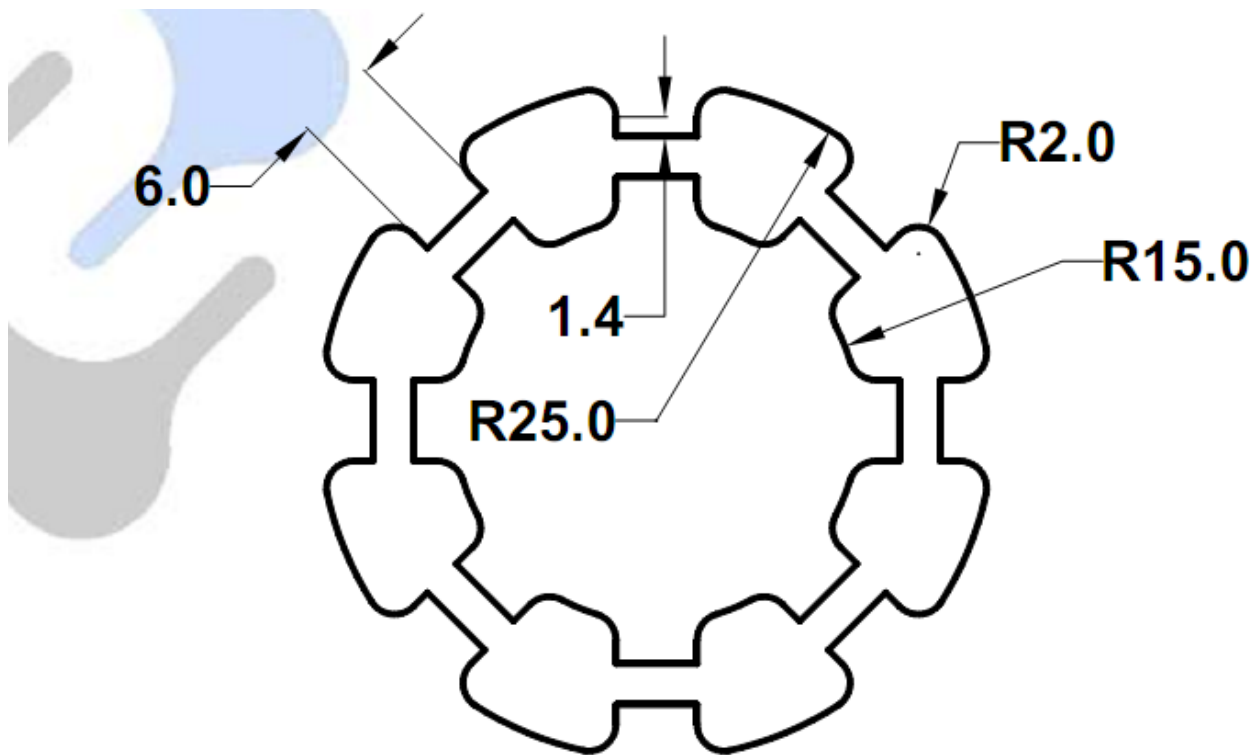
Select first object or [Undo/Polyline/Radius/Trim/Multiple]: -(1) select 1st line-

Select second object or shift-select to apply corner: -(2) select 2nd line-

Explode

This command is used to break the object into smallest possible number of segment in which it can be divided.





Dimensions

Dimension is an important annotation on drawing or draft, it reflects the data of drawing in the form of magnitude as well as directions. Dimensions are of different types, linear, aligned, marks, radius or diameter etc but it also includes tolerances

Linear Dimension

Linear dimension command is used to specify the horizontal and vertical dimensions in drawing. This command can be activated by

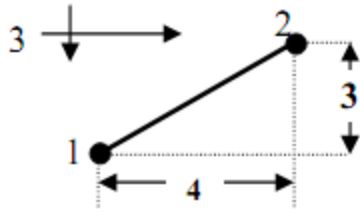
Command: dimlinear

Specify first extension line origin or <select object>: - (1) click on 1st

coordinate of object side -

Specify second extension line origin: - (2) click on 2nd coordinate of object side -

Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]: (3)



Command: dimlinear

Specify first extension line origin or <select object>: - (1) click on 1st coordinate of object side -

Specify second extension line origin: - (2) click on 2nd coordinate of object side - Specify dimension line location or

[Mtext/Text/Angle/Horizontal/Vertical/Rotated]: (3)

Aligned Dimension

Aligned dimension is used to specify the dimension of tilted or inclined line to show its original length.



Command: dimaligned

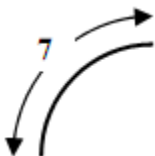
Specify first extension line origin or <select object>: - (1) click on 1st coordinate of object side -

Specify second extension line origin: - (2) click on 2nd coordinate of object side -

Specify dimension line location or [Mtext/Text/Angle]:

Arc Length

This command is used to specify the arc length as dimension. It can be activated by



Command: dimarc

Select arc or polyline arc segment: - select the arc or segment -

Specify arc length dimension location, or [Mtext/Text/Angle/Partial/Leader]: - click where dimension should shown -

Radius

As the command name describes, this command is used to show the radius of an arc, segment or circle as a dimension. This command can be activated by

Command: dimradius

Select arc or circle: - click on arc or circle -

Specify dimension line location or [Mtext/Text/Angle]: -specify position-

Diameter

This command is used to show the diameter of an arc, segment or circle as a dimension. This command can be activated by

Command: dimdiameter

Select arc or circle: - click on arc or circle -

Specify dimension line location or [Mtext/Text/Angle]: -specify position-

Angular Dimension

This command is used to show the angle between two lines as dimension. This command can be activated by

Command: dimangular

Select arc, circle, line, or <specify vertex>:- (1) select first line or arc -

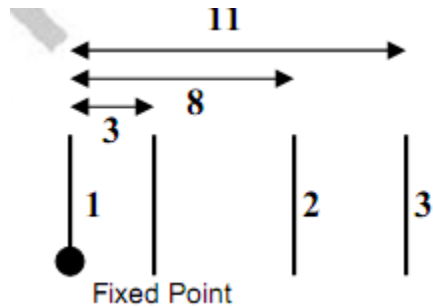
Select second line: - (2) select second line or arc to which angle to be measured -

Specify dimension arc line location or [Mtext/Text/Angle]: - (3) select location of dimension -

Baseline Dimension

This dimension is used to measure the dimension from a fixed datum, hence when more than one dimension is required, this command gives the dimensions of those points with respect to a point. This command can be activated by

dimbaseline



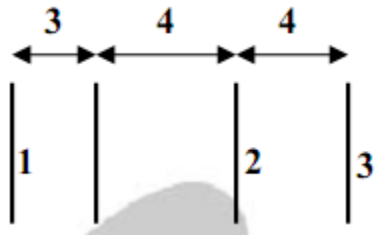
Continue Dimension

This dimension is used to measure the dimension continuously, at the end of first dimension, next dimension is to be start. This command also need a already drawn dimension.

Command: dimcontinue – after activation of command AutoCAD will ask for second point-

Specify a second extension line origin or [Undo/Select] <Select>: - now click on 2nd point -

Specify a second extension line origin or [Undo/Select] <Select>: - now click on 3rd point -



Leader

This commands is used to insert the leader to show the relation between text and dimension.

Command: leader

Specify leader start point: - click on starting point -

Specify next point: - click on another point to specify end point -

Specify next point or [Annotation/Format/Undo] <Annotation>: a

Enter first line of annotation text or <options>: - write text here and than click right -



Center Mark

This command is used to specify the center mark of circle, arc or ellipse in drawing. It can be activated by

Command: dimcenter

Dimension Style

Command: dimstyle

This feature is used to modify the dimension style, i.e. dimension font, color, line type, arrow style, offset and extension line style etc. Using this feature user can either modify the all dimensions at once or one at a time. To access this feature perform one of the following method. A dialogue box will be appeared where several options are available, with a list of existing dimension styles which can be set to current by the user. User can make a new profile by clicking on 'New' , a dialogue box will ask new name and description and the profile will be created. There are Many button are in this dialogue box.

SET TO CURRENT This is used to activate the existing dimension profile, saved by the user.

NEW To create a new profile of dimensions.

MODIFY To modify the dimension profile which will be implemented to all dimensions in the drawing.

OVERRIDE It is used to modify a single dimension which will make differs from the existing profile dimension made by modify method.

This override button also exist in Dimension Menu.

COMPARE This button compares the user made dimension profile with the standard.

When user click on modify button a new windows opens for individual setting of each component of dimension as shown below. Using this windows, user can modify the line style, symbol and arrow, text, primary unit, secondary unit, tolerance etc.

Dimension modification tool dialogue box is shown below. To modify the dimension, -click on modify button –A new dialogue box will appear.

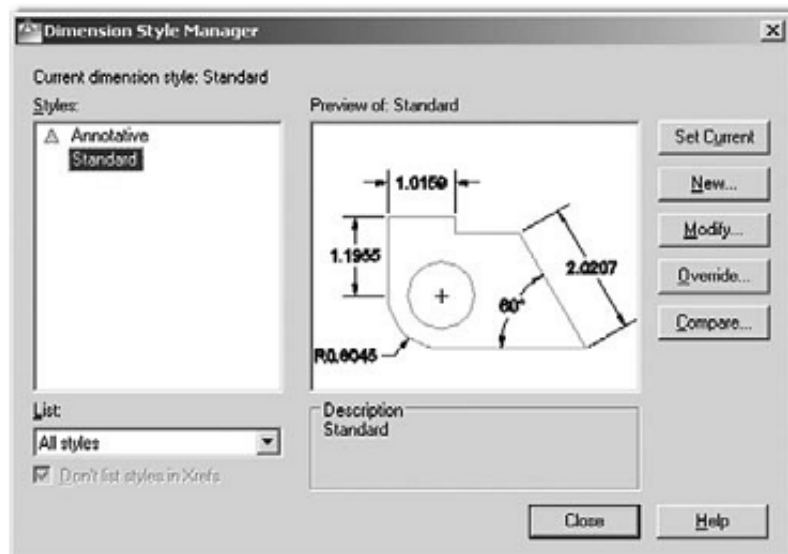
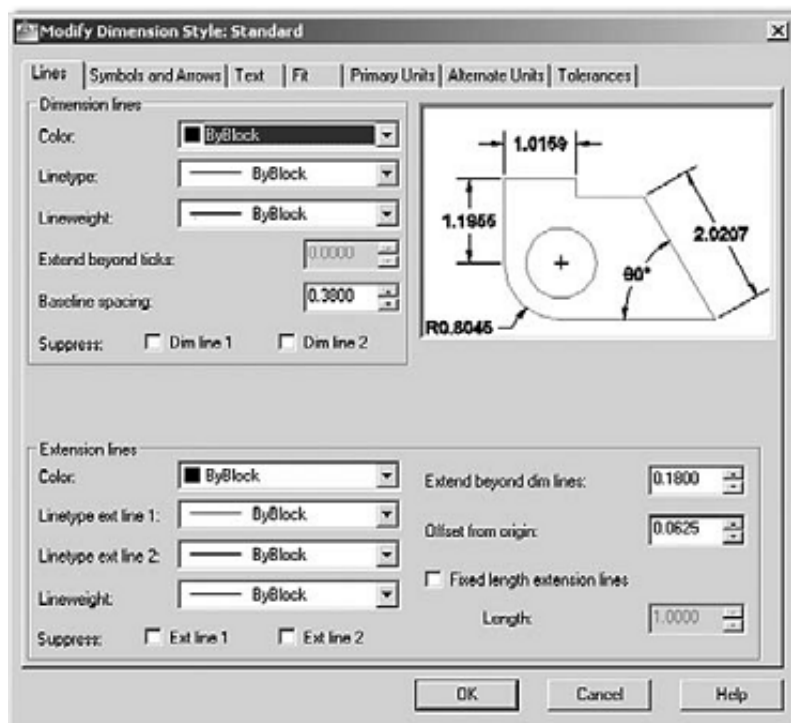
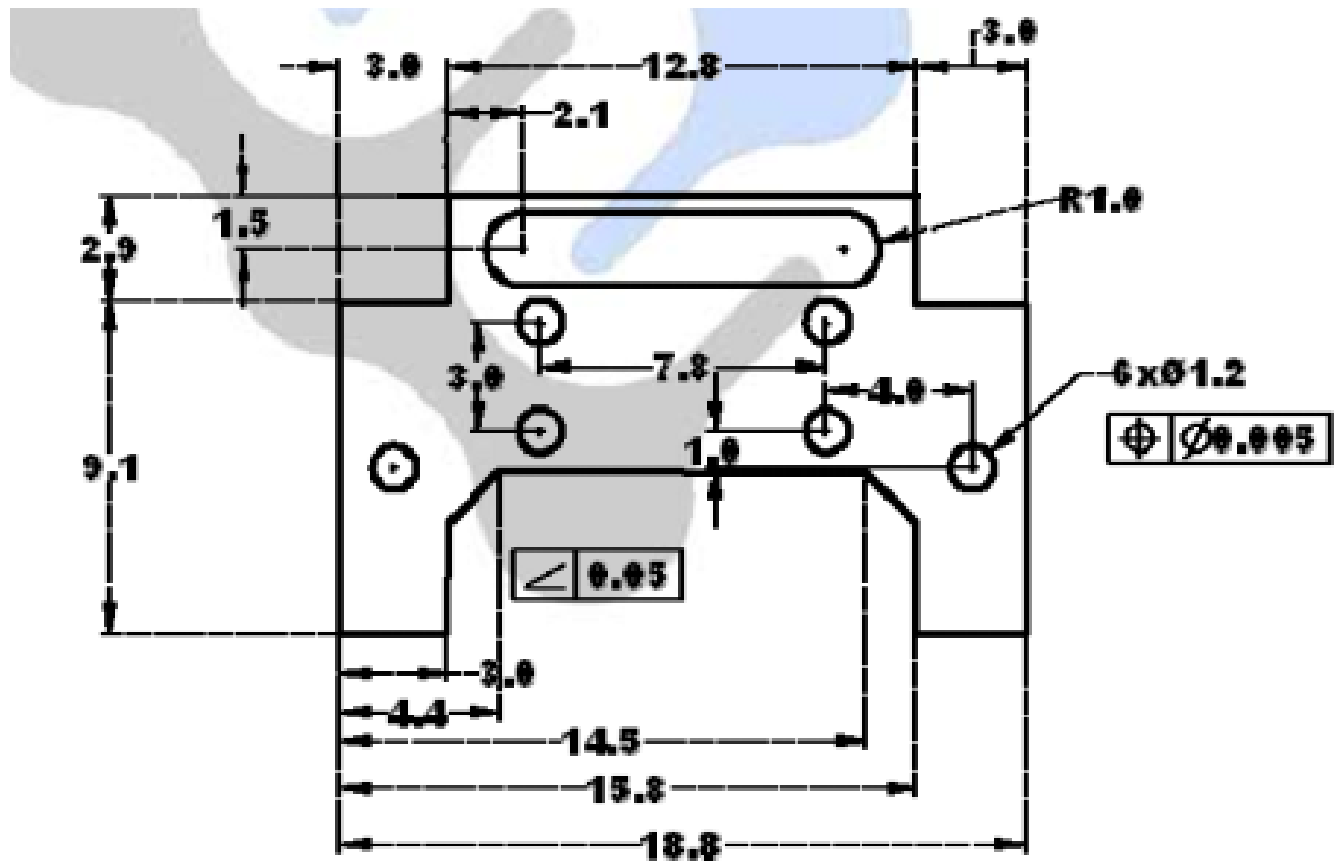
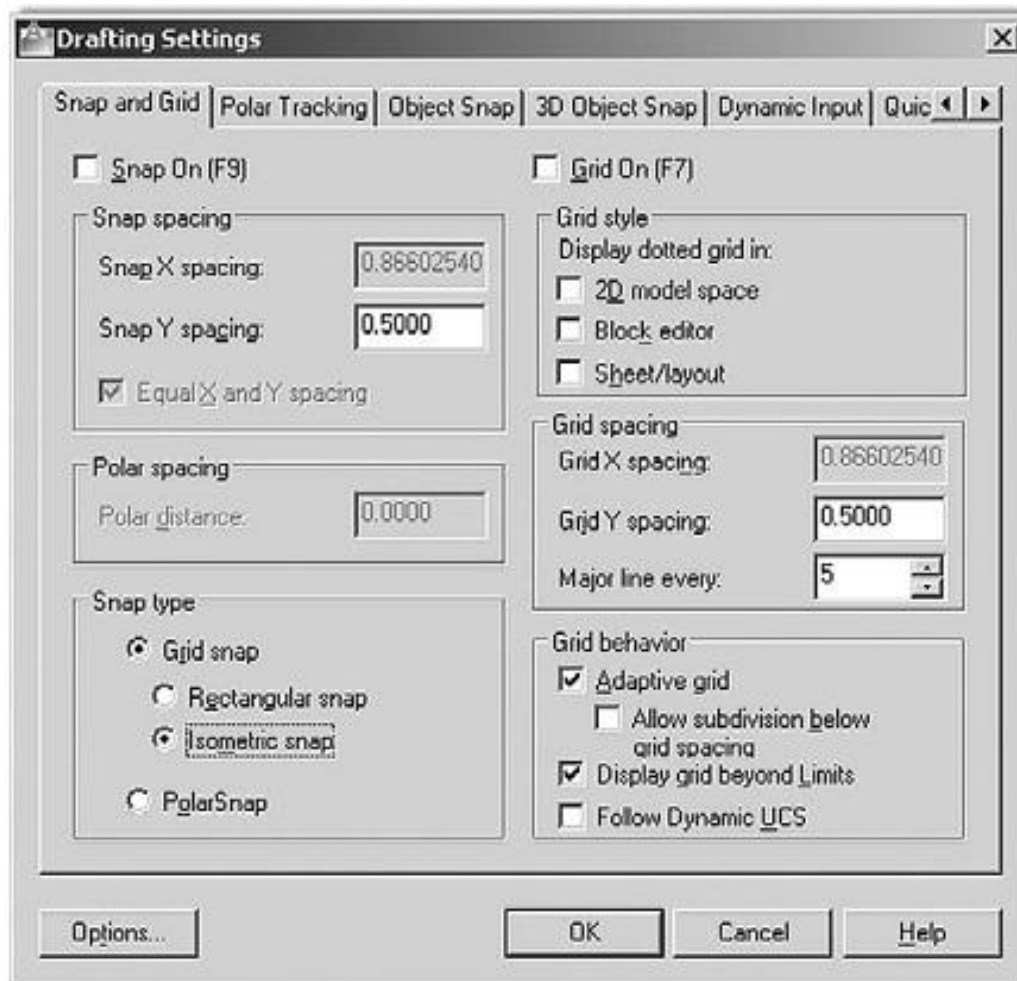


FIGURE 6.29
Dimension Style Manager.

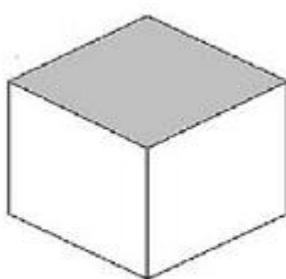




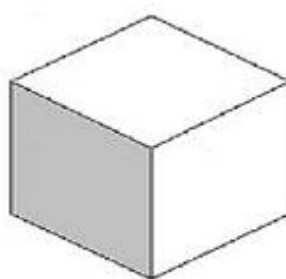
Isometric Drawing



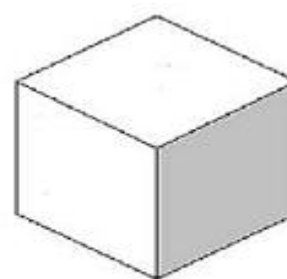
In isometric drawing, one is always drawing in one of three available planes: top, right, or left (see Figure). Because you are using a 2D pointing device (a mouse), which exists only in a 2D world, you need to be able to easily move from one plane to the other to be able to draw on it. To toggle between planes, press the F5 key. Your crosshairs line up accordingly, and now you just draw using the line command. Here is one final rule: You almost always have Ortho on while in isometric mode, otherwise, the lines are not straight and rotating them 30° is meaningless. Let us put it all together and draw an isometric box.



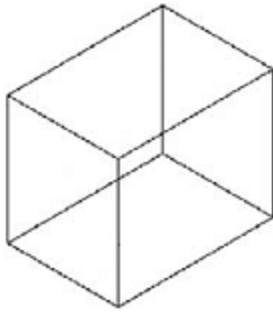
TOP PLANE



LEFT PLANE

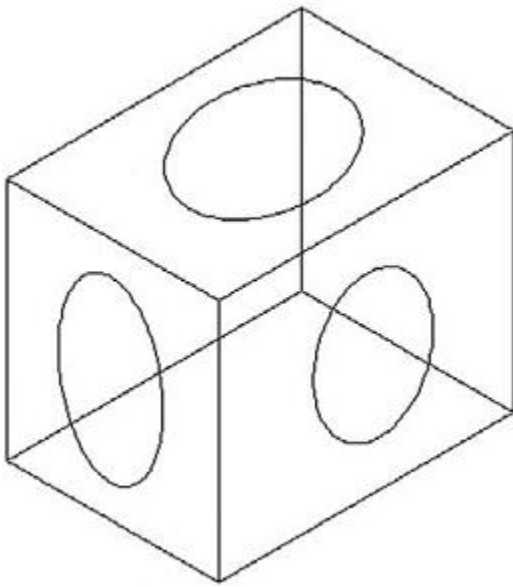


RIGHT PLANE



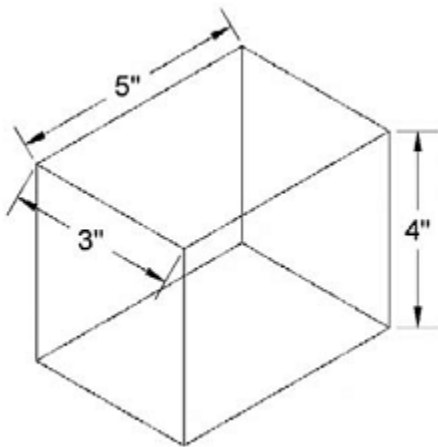
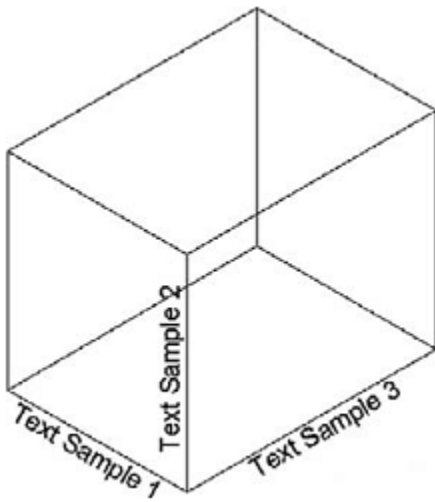
Isometric drawing

- To draw circles in isometric perspective use ellipses
- Always stay in Ortho mode unless you need to deliberately angle a line at some odd angle
- Text can be angled up or down by 30° to align with the isoplanes.



TEXT AND DIMENSIONS IN ISOMETRIC DRAWING

Text can easily be aligned with isoplanes by rotating it plus or minus 30° , or other values depending on the plane.



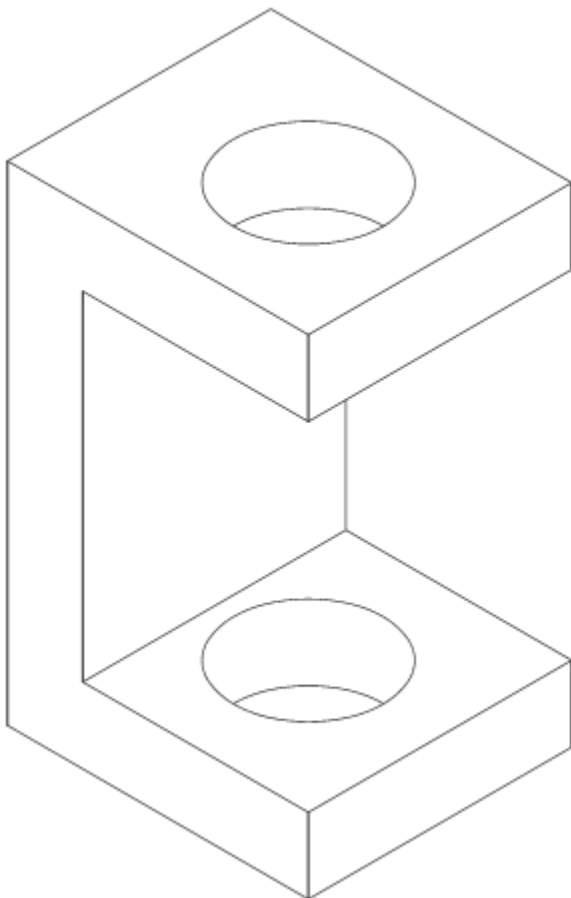
Command: **DIMEDIT**

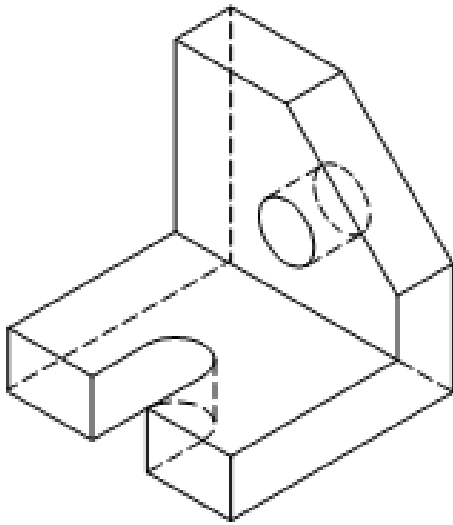
Enter type of dimension editing [Home/New/Rotate/Oblique] <Home>: **O**

Select objects: 1 found

Select objects:

Enter obliquing angle (press ENTER for none): **-30**





3d spring

Command helix

step 2:
specify the center point of base of the spring your going to draw
step 3:
specify the base radius or Diameter of the spring
step 4:
specify the top radius of spring.. it'll same value unless your spring is of conical shape
step 5:
specify height of the spring
step 6:
to see the spring you have drawn in isometric views.. type

Properties select

Command **3dorbit**

and enter.. or click the icon showing 3d orbit..
now rotate the drawing with help of your mouse..

draw a circle of required pitch you wanted.. using command C or circle or clicking the icon..

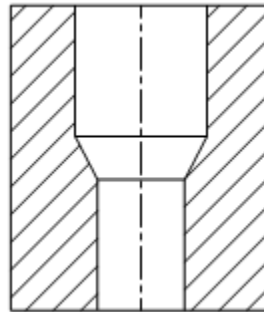
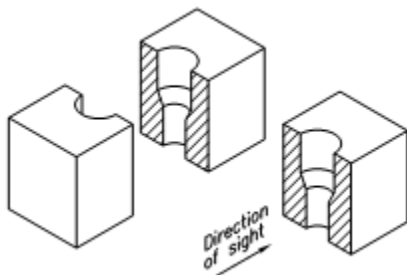
to see the 3D SPRING in shaded view.. type SHADE in command box and enter.. you will get the 3d spring.. :)

command SWEEP and then select the circle (for Object to sweep)..

Command **SHADE** you will get the 3d spring.. :)

Hatch

Sectional Views





Layers

Layers are the primary tool used in AutoCAD and other software to differentiate different groups of object at a place and enable and disable their display as per requirement.

Default layer in AutoCAD is named by '0' which can not be deleted from list. Layers are controlled by Layer Tool Bar shown with Properties Tool bar on screen.

Layers are like overlays in drawing, in which same groups of objects are set in the same layer to form a group, a single layer consists of same visual setting like same line type, line weight and color which differentiate the layer from other layers.

Default layer in AutoCAD is named by '0' which can not be deleted from list. Layers are controlled by Layer Tool Bar shown with Properties Tool bar on screen.

<i>Keyboard:</i> Type in layer or la and press Enter	
<i>Cascading menus:</i> Format→Layer...	
<i>Toolbar icon:</i> Layers toolbar	
<i>Ribbon:</i> Home tab→Layers	



Layer Properties

Layer Properties button is used to create new layer, delete existing layer, make filter in layer groups etc. A dialogue appears which asks several settings. This dialogue box can also be accessed by the button shown above in Layer Tool Bar and can also be accessed by

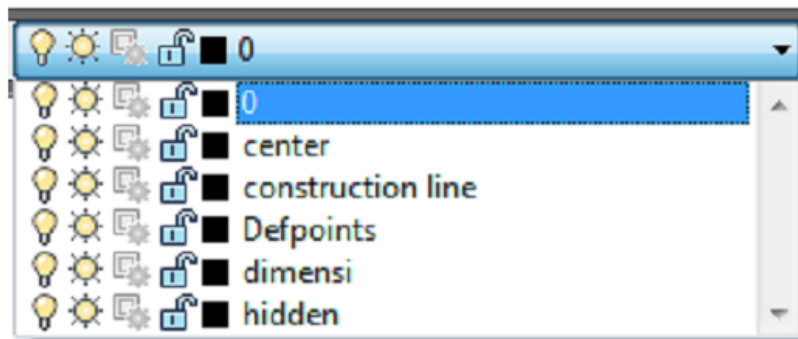
Command layer

Right click newlayer

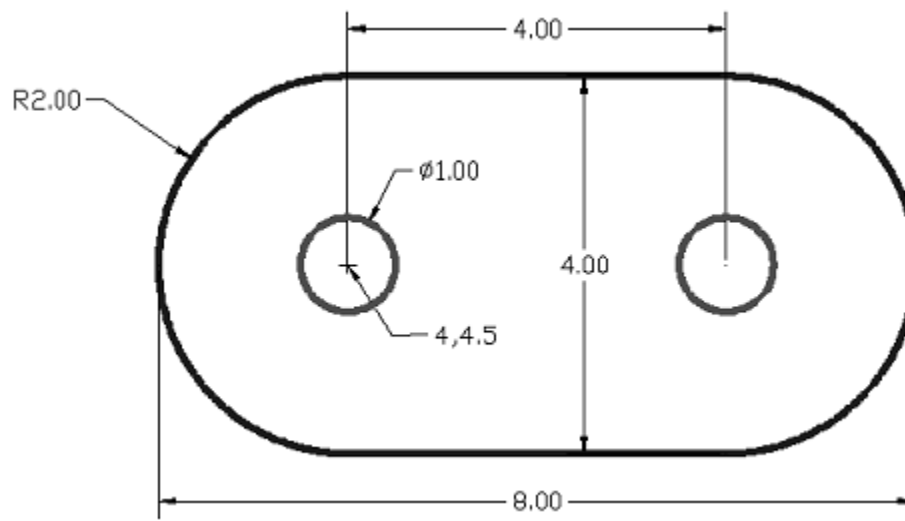
You can edit the current layer only

Format your drawing layer as follow:

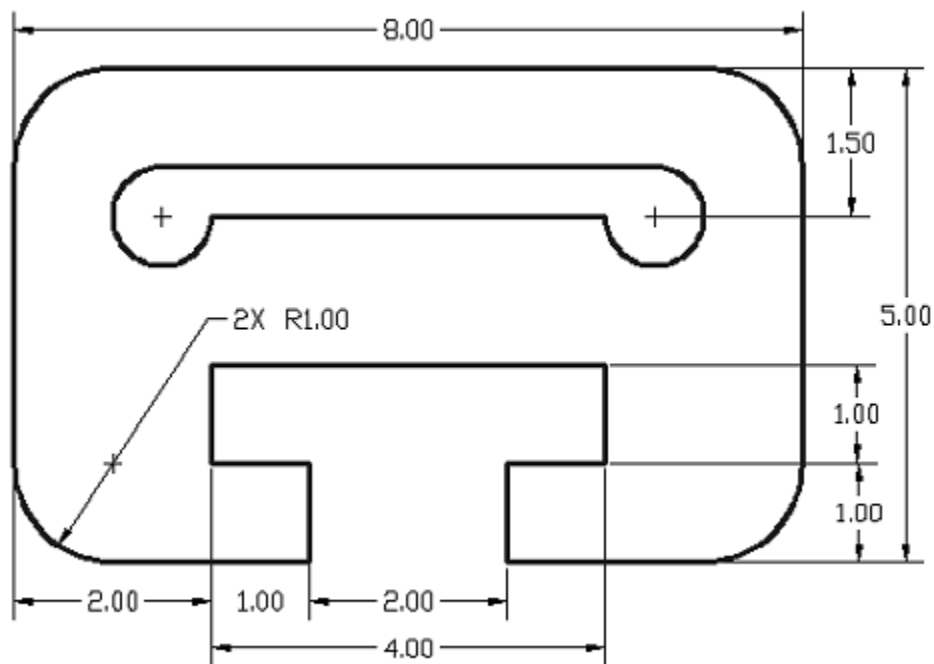
Layer name	Line t type	Line weight
0	Continuous	0.4
hidden	hidden	default
center	center	default
dimension	Continuous	default
Construction line	Continuous	default



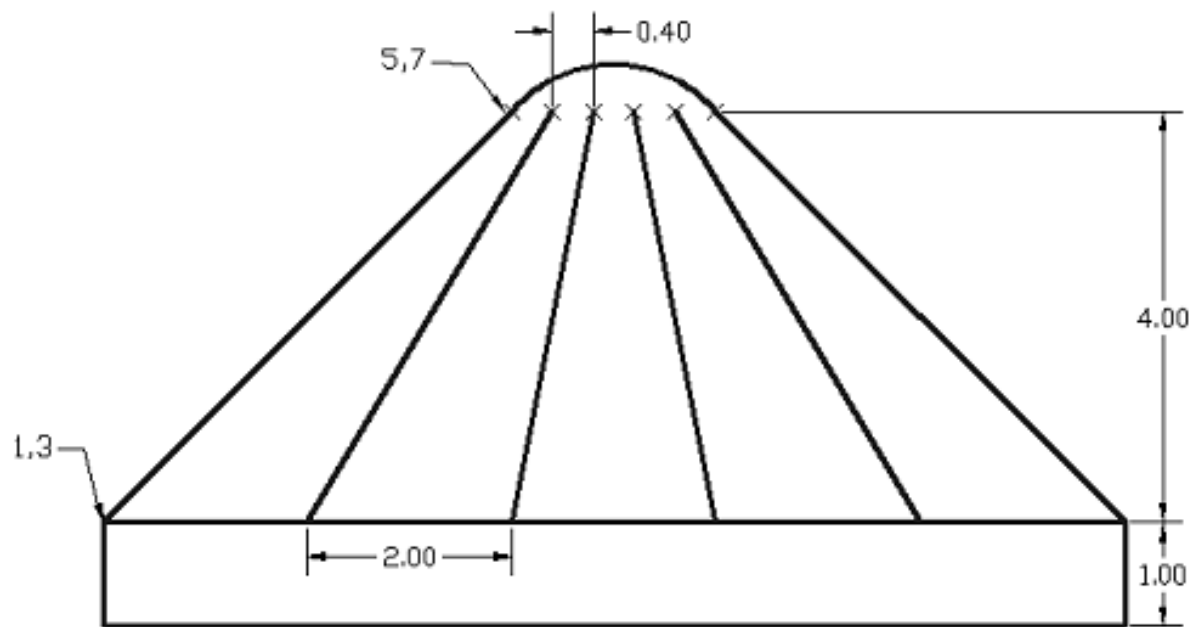
Exercise 1



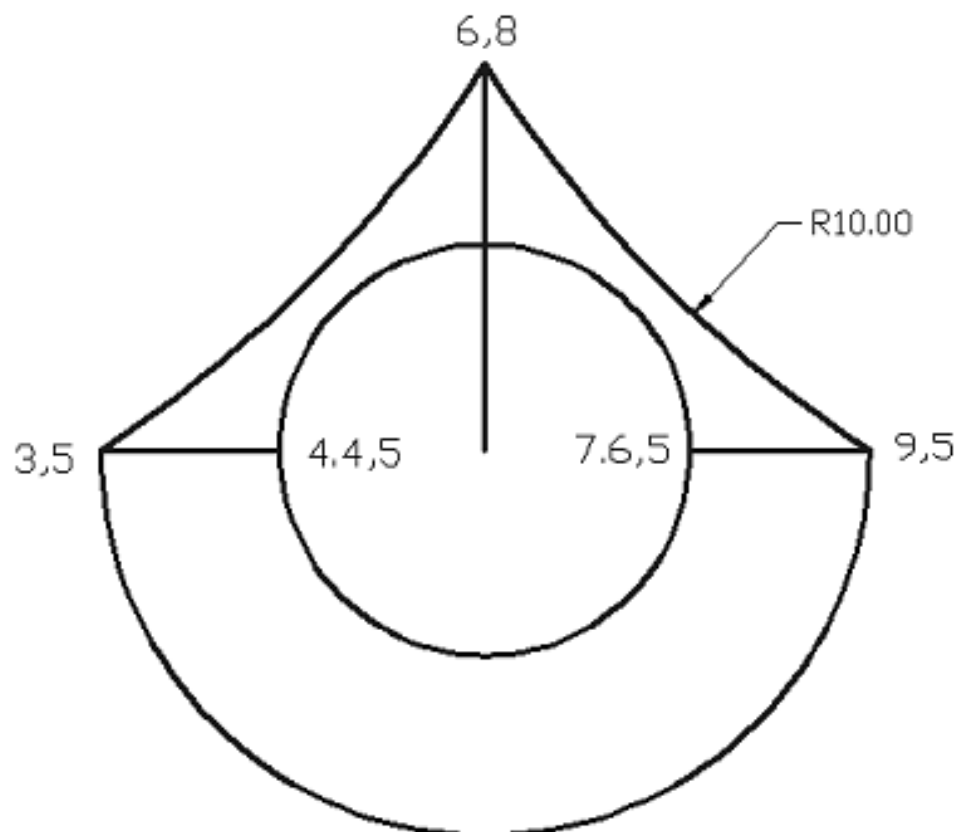
Exercise 2



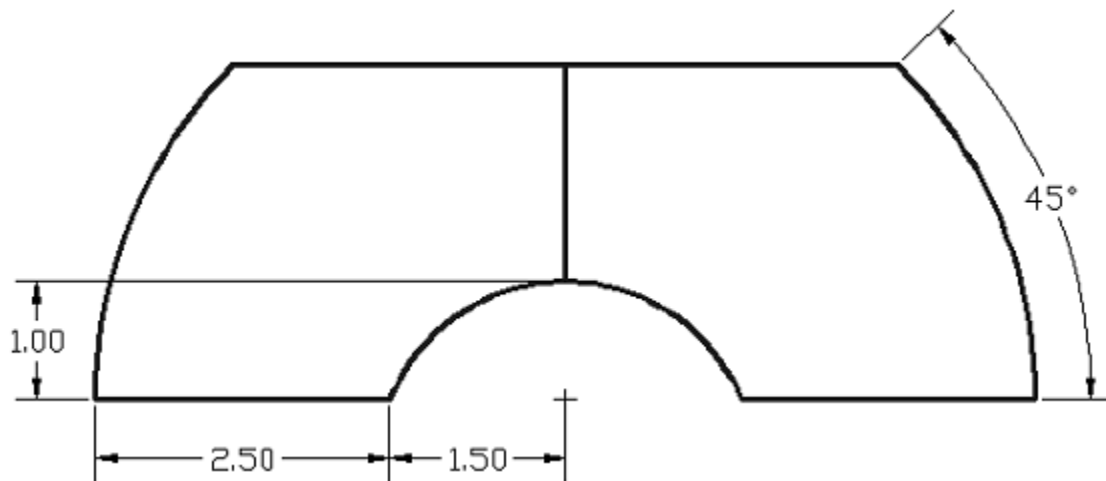
Exercise 3



Exercise 4

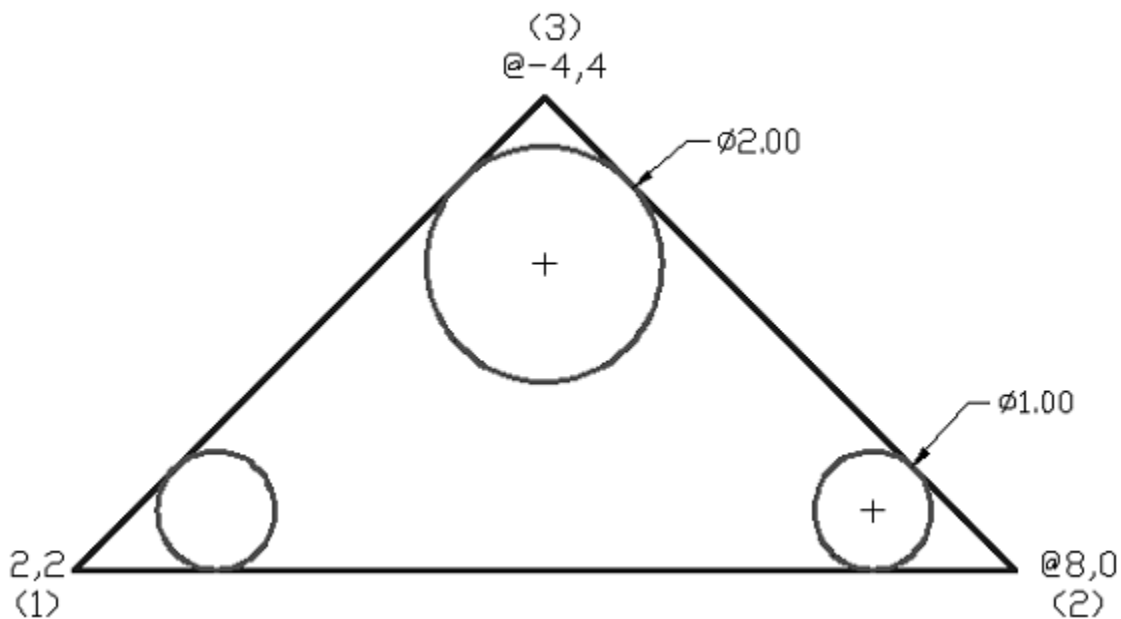


Exercise 5



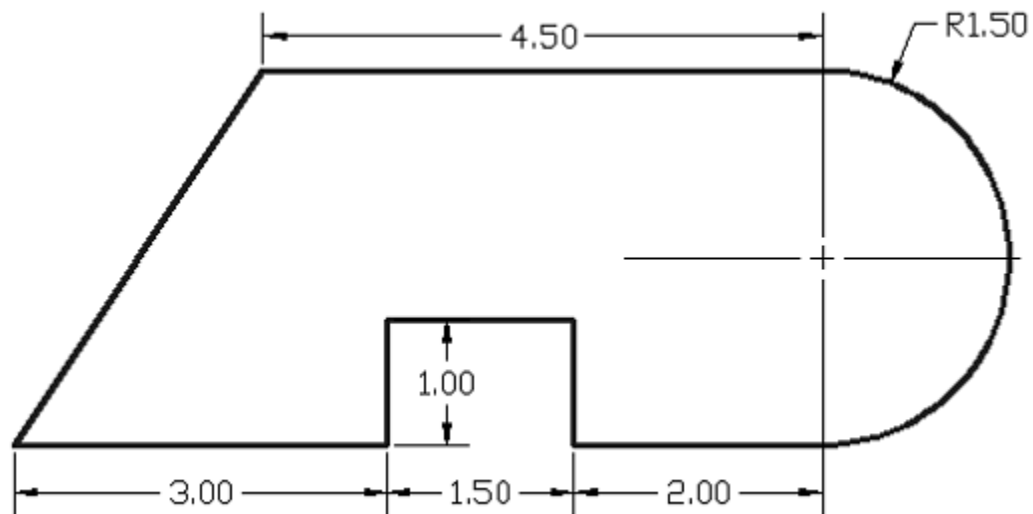
To make this object, start with the line, then the arcs. The center of the arc in 45° is the endpoint of the location 1.50.

Exercise 6



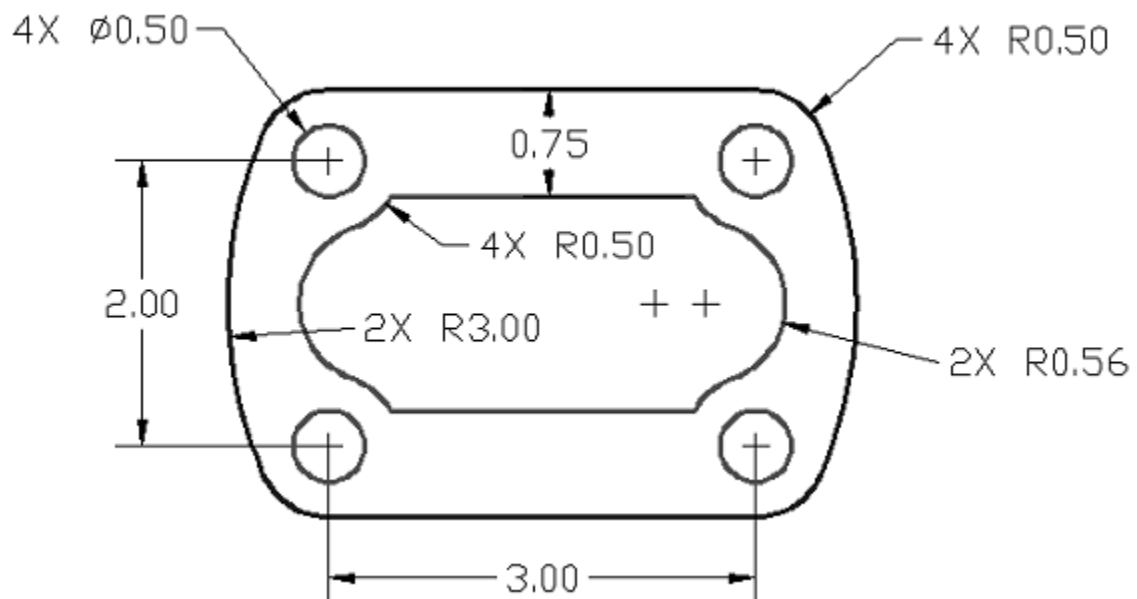
This drawing again has absolute coordinate locations to follow. The triangle is made by following those commands given in numbered order. To circle inside the triangle is made through circle-ttr command.

Exercise 7



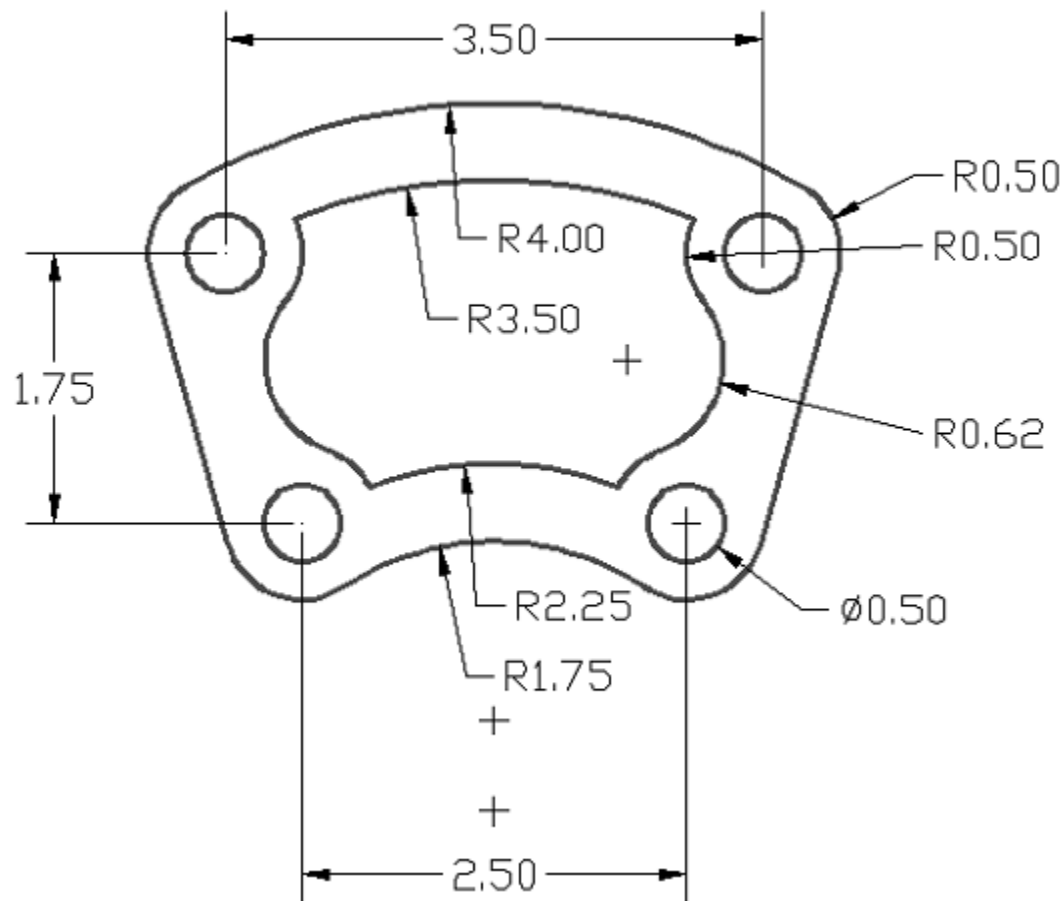
Though this object can be made through lines and arcs, the objective is to make this one using polylines wherein the lines and arcs are connected.

Exercise 8



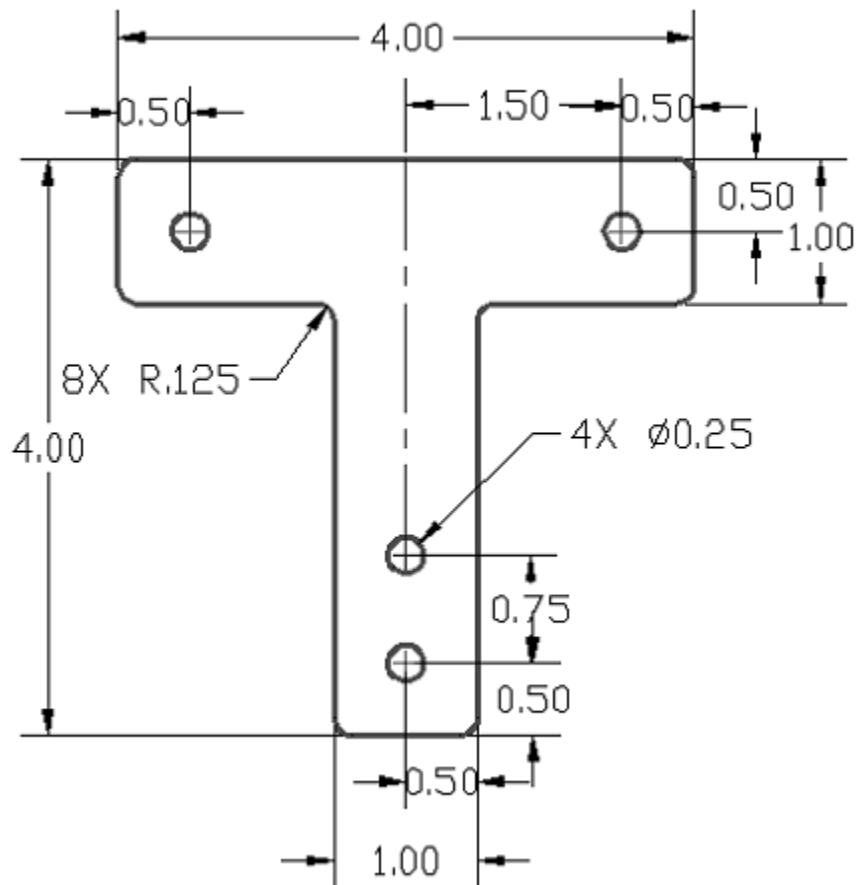
This object starts with the smaller circles with the diameter or 0.50 and the rest follows. A circle-ttr command is more useful than using arc to make the curves.

Exercise 9



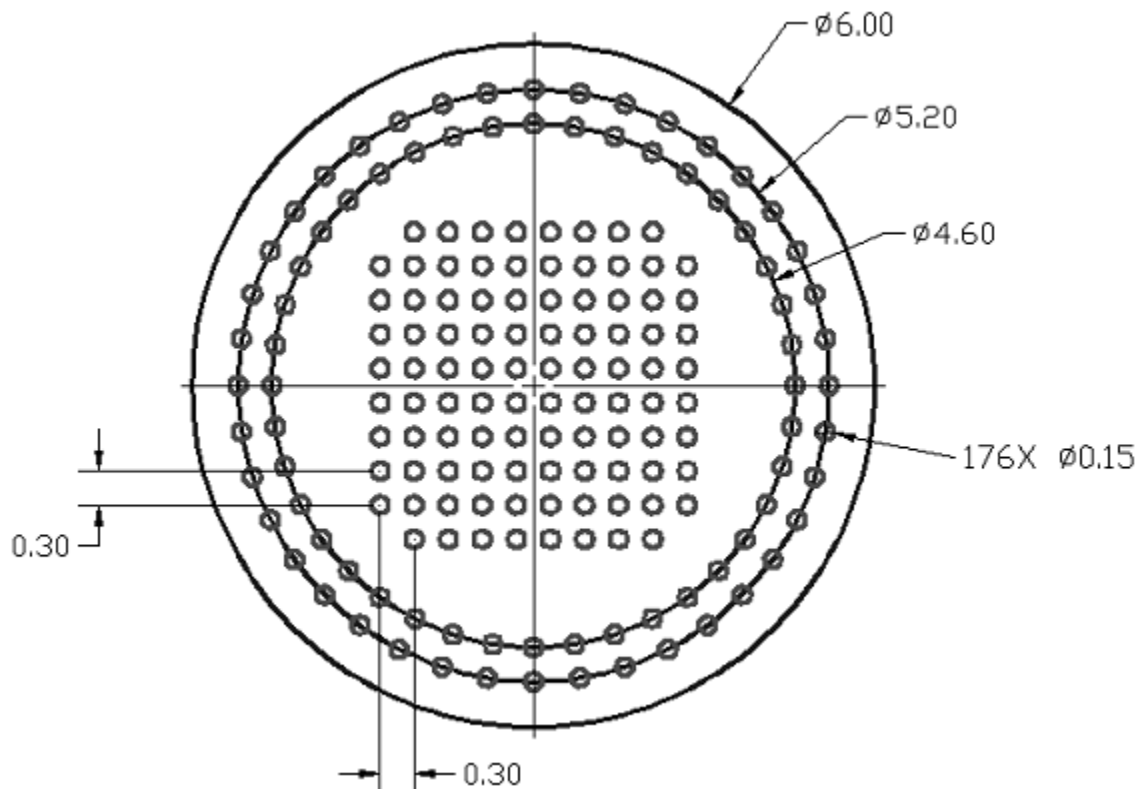
This object is almost similar to previous figure which also starts with the smaller circles. Again a circle-ttr command is used here than arcs to make the curves. To copy the arc inside or outside at a different radius, the offset command could be useful.

Exercise 10



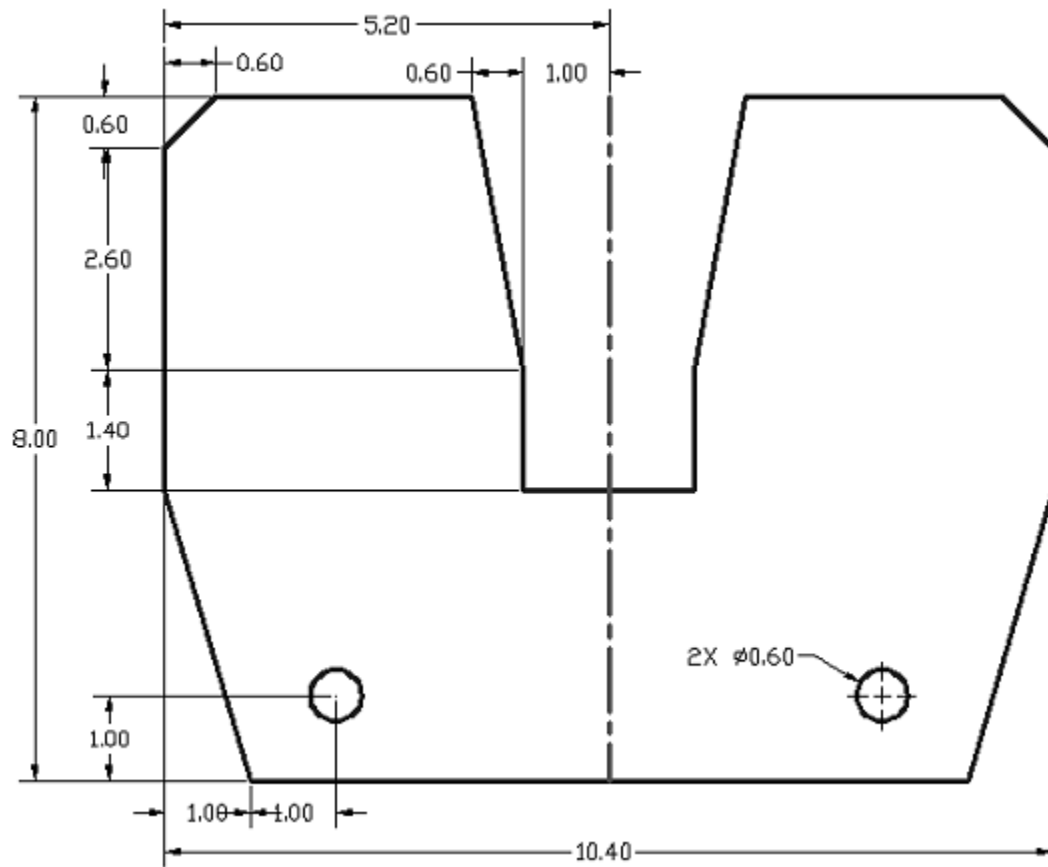
In this objects the direct distance entry could be useful to make the perimeter. For the curves on the edge, fillet command could also be used. Relative rectangular entry with snap-from command could locate the circles' center. To make the dash-dot center line you need to load the line in the linetype box.

Exercise 11



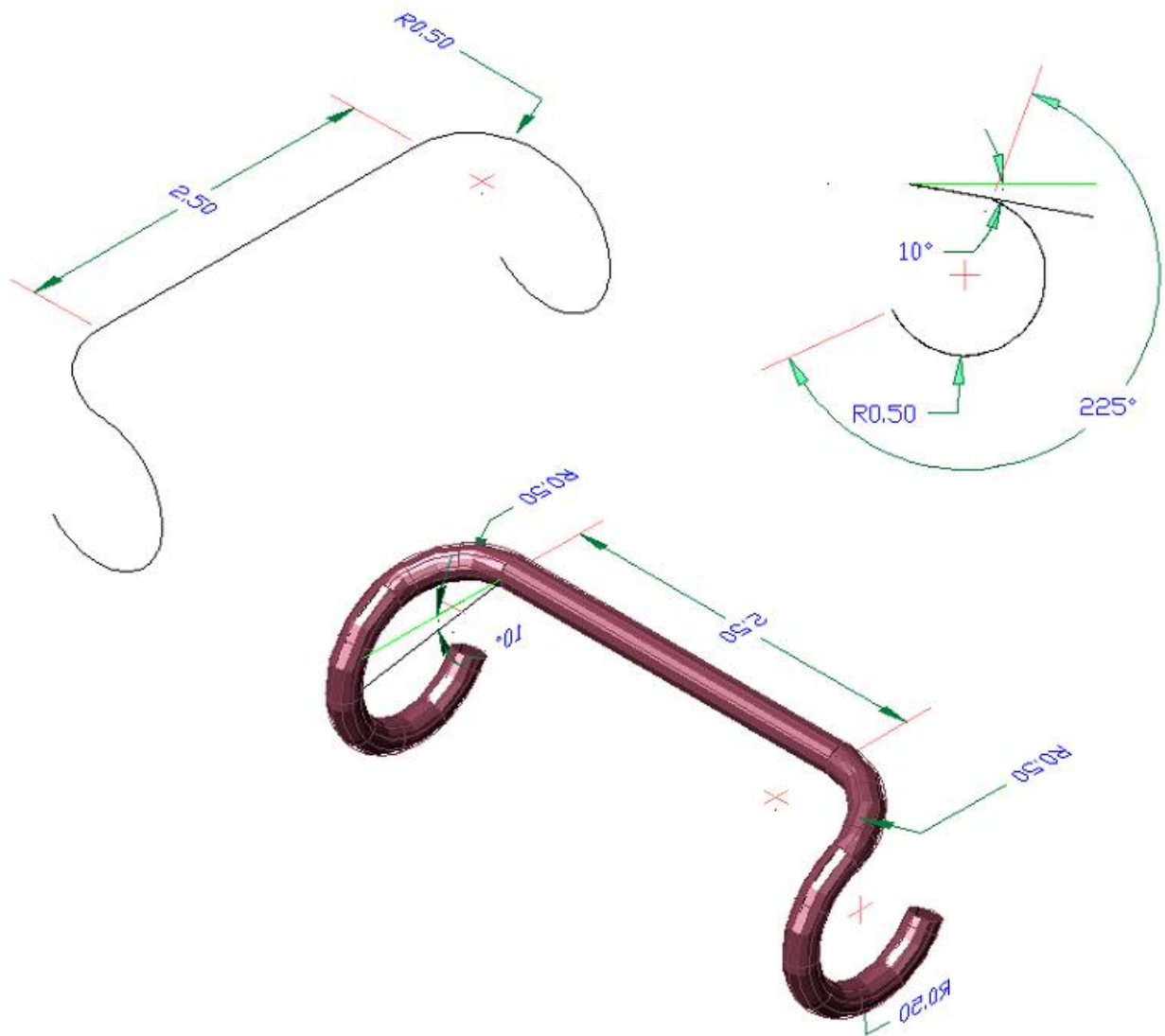
The objective of making this object is to learn the array (rectangular and polar) command. There are a total of 176 smaller circles. There are 40 circles around each inner circles and 96 circles inside.

Exercise 12

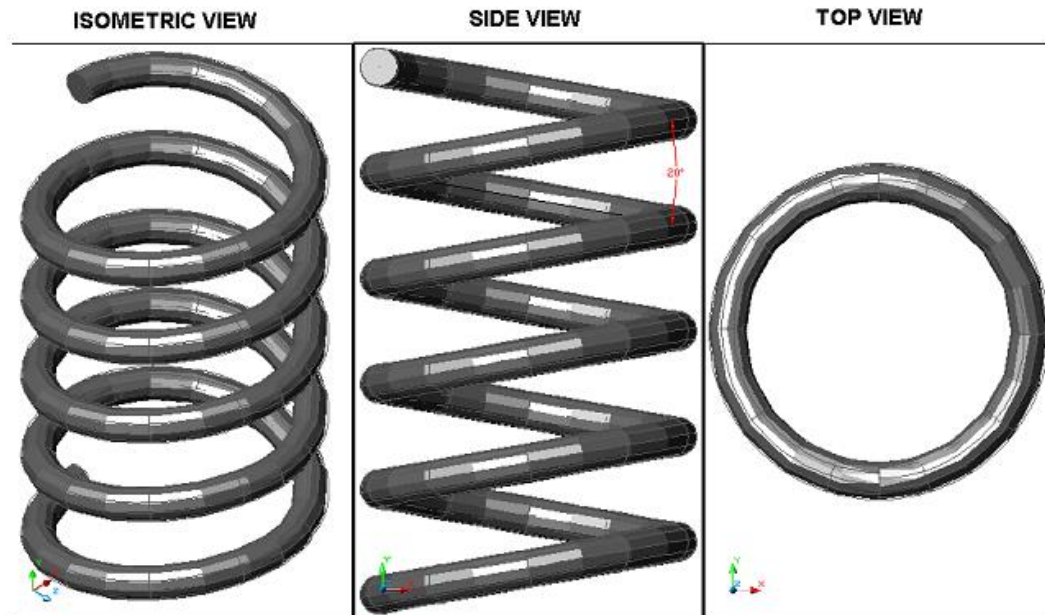


This exercise will help you use the chamfer command which will help you make the cut on the edge. The mirror command is also used to reflect an object to its opposite side.

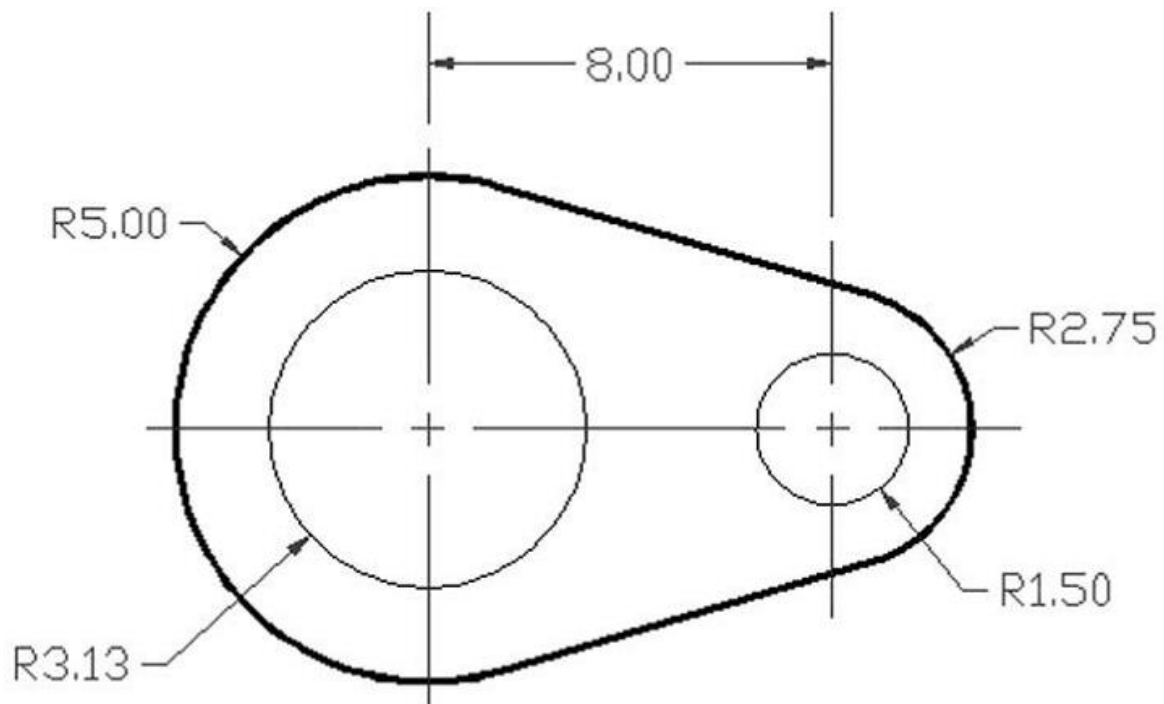
Exercise 13



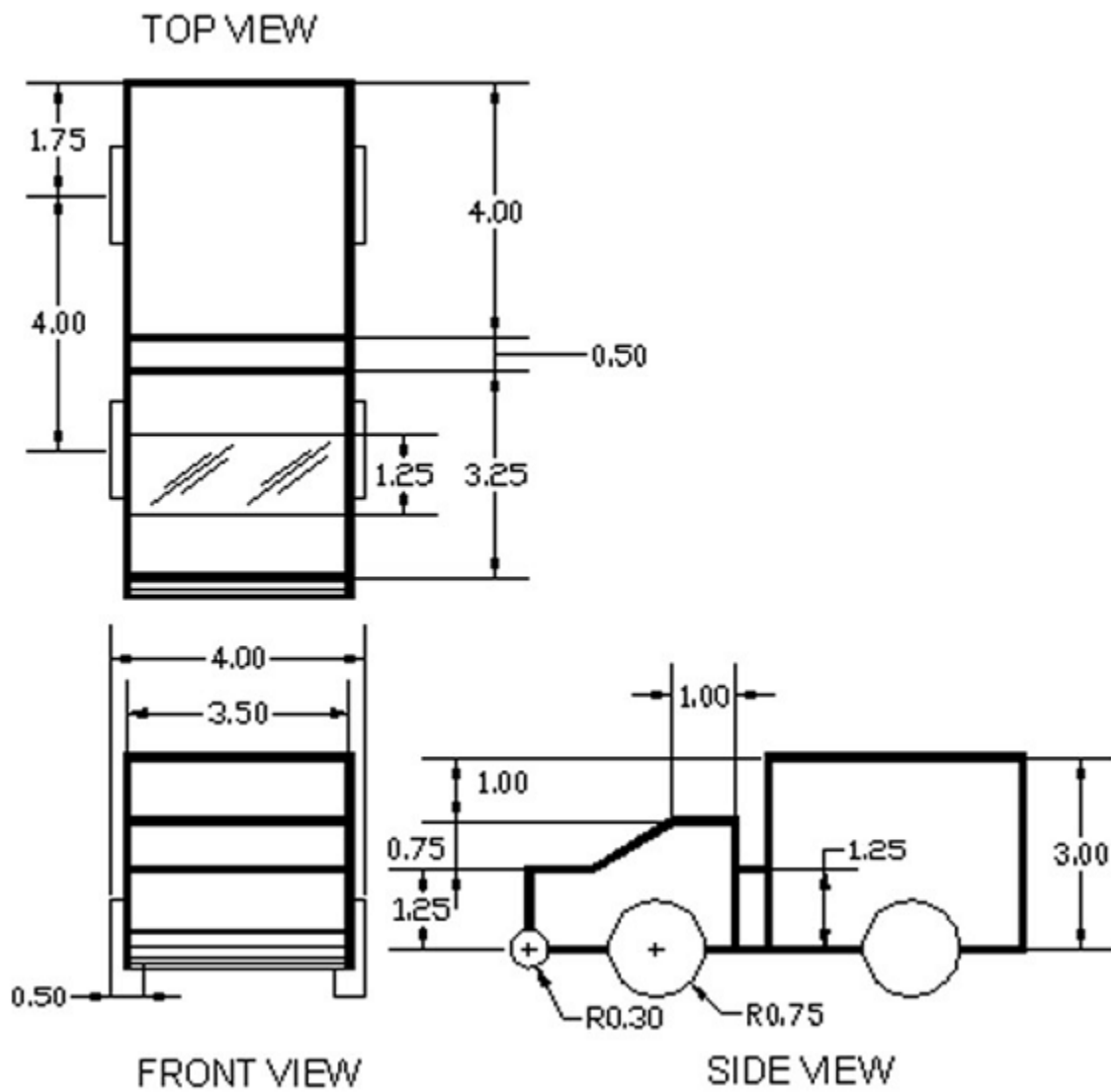
THE SPRING



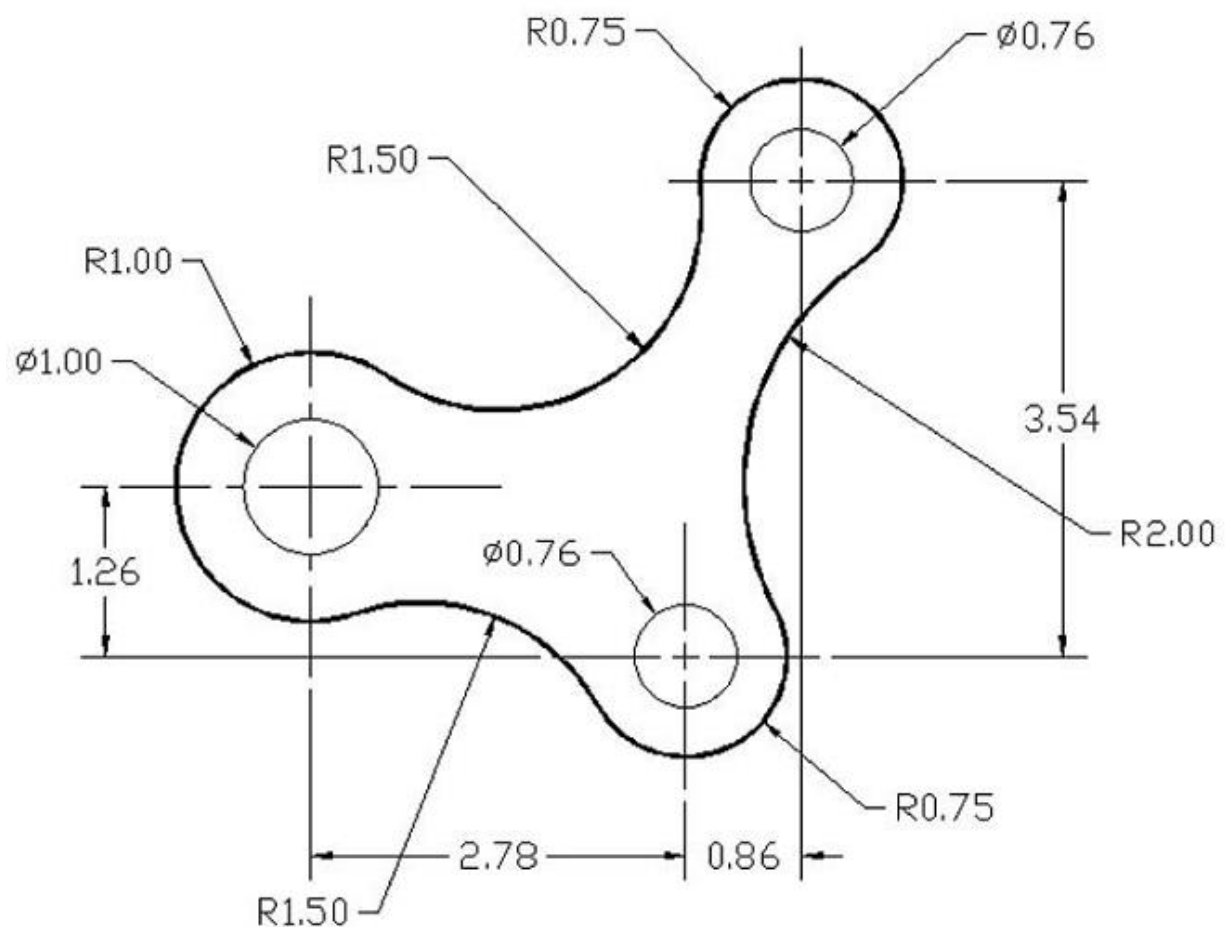
Exercise 14



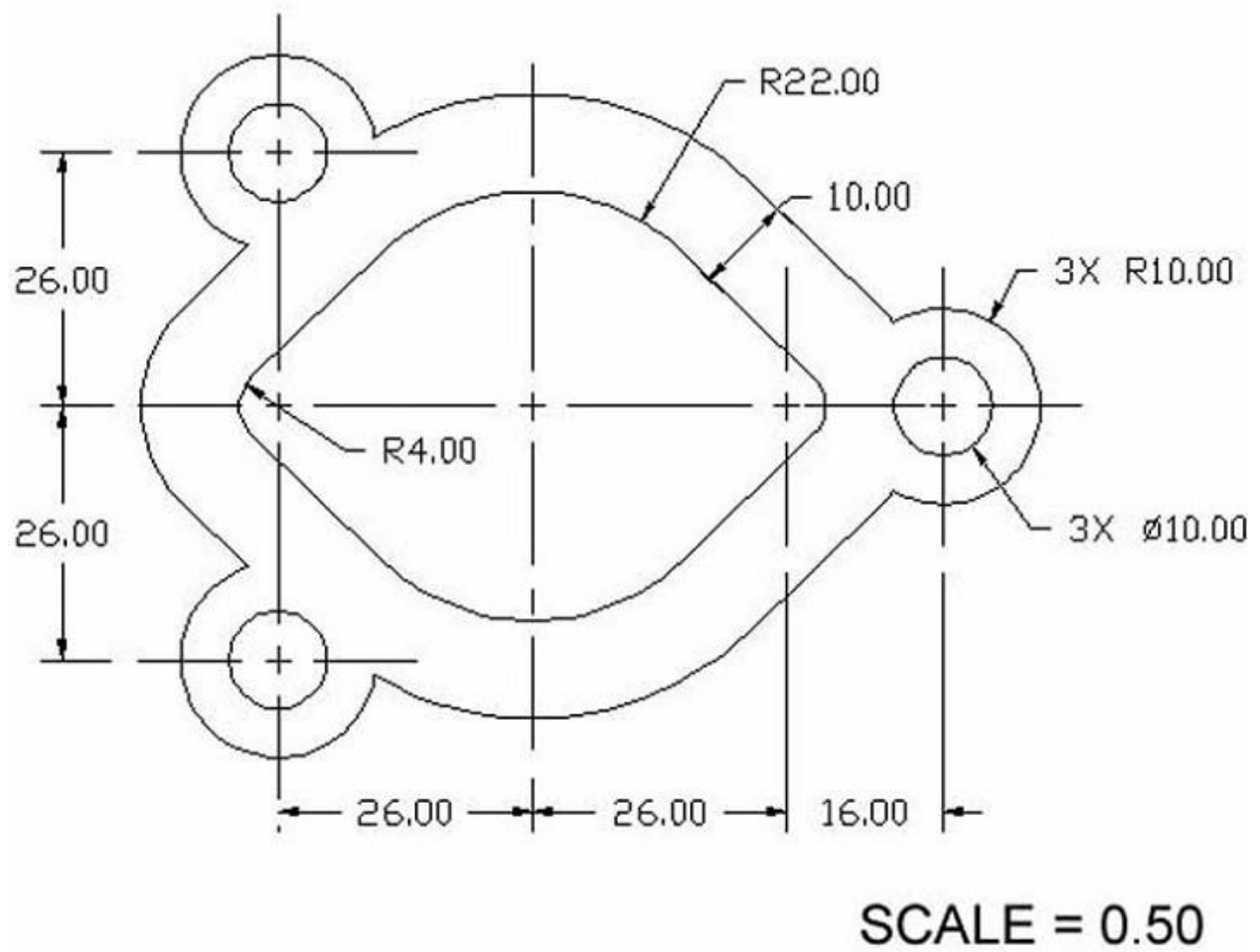
Exercise 15



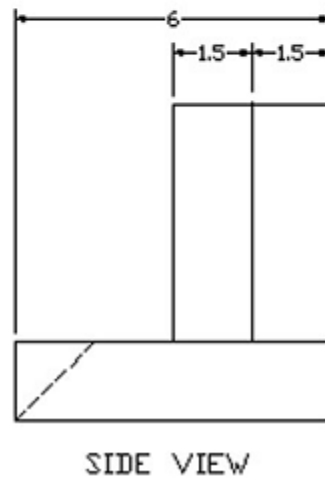
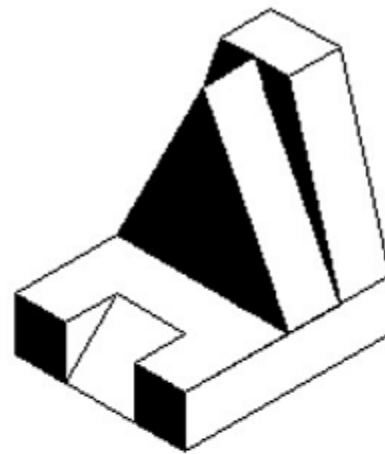
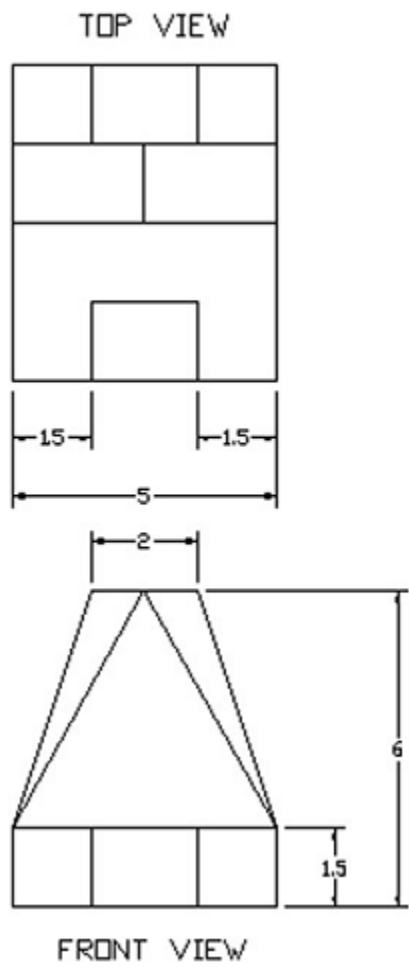
Exercise 16



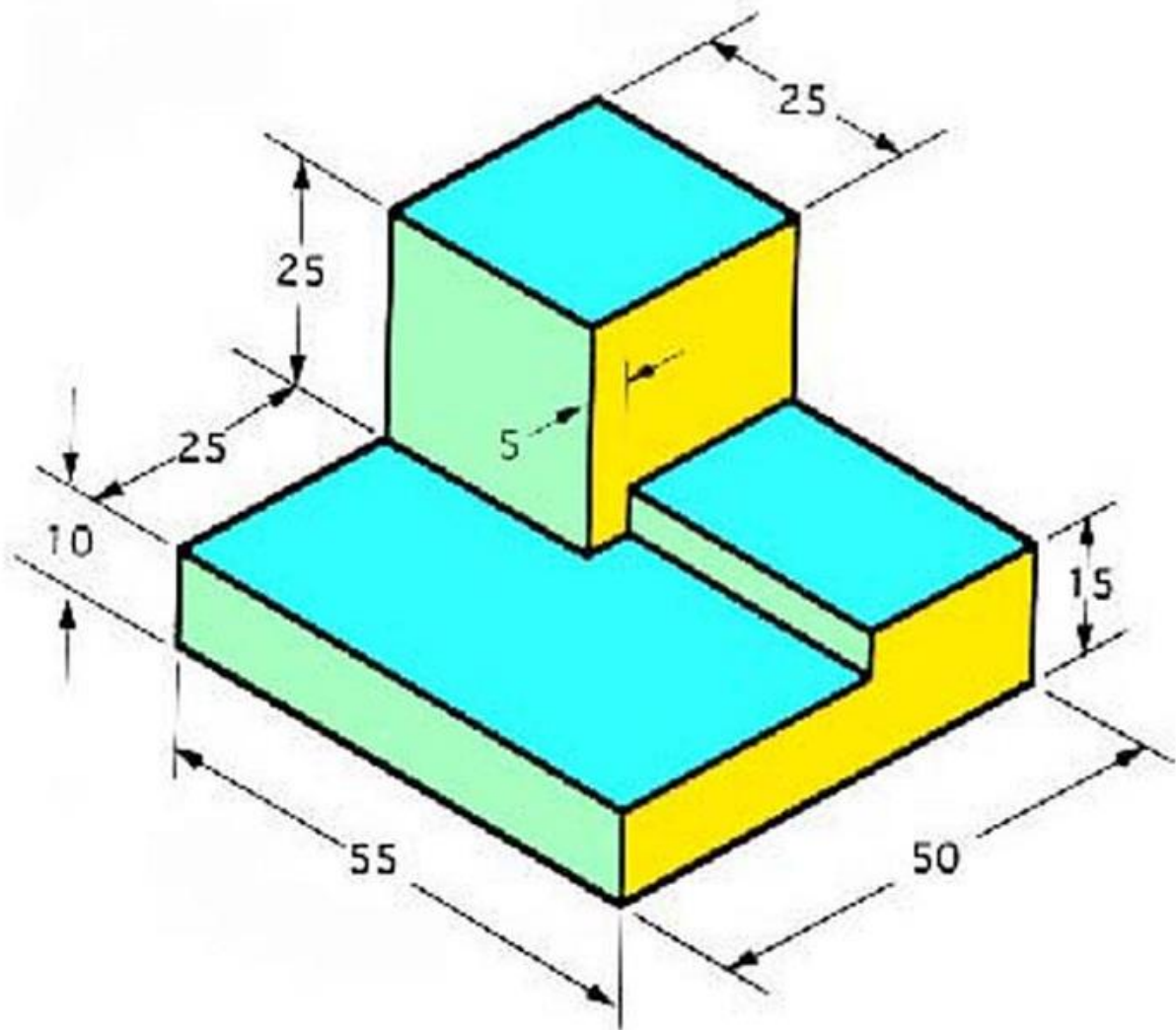
Exercise 17



Exercise 18

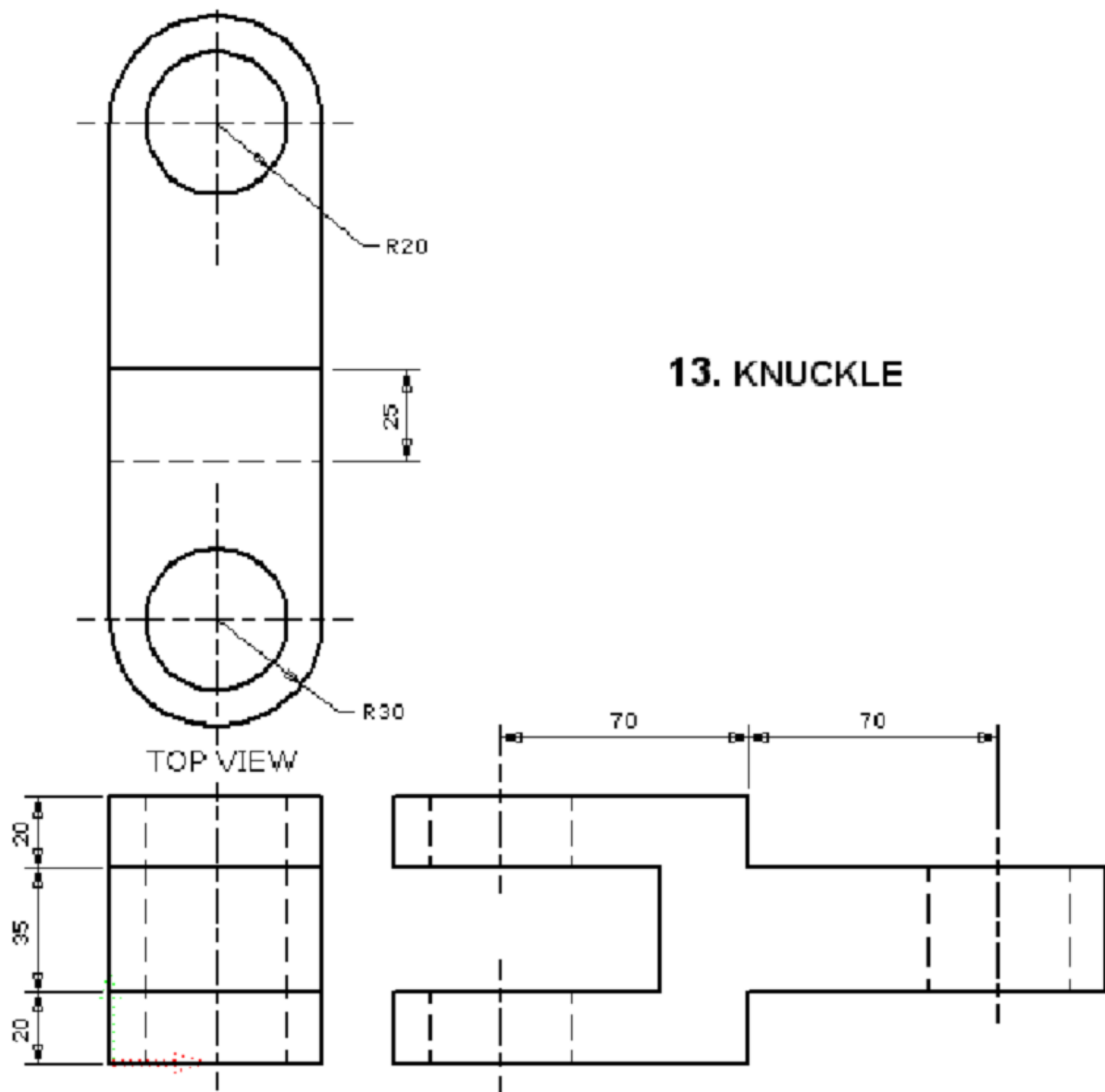


Exercise 19

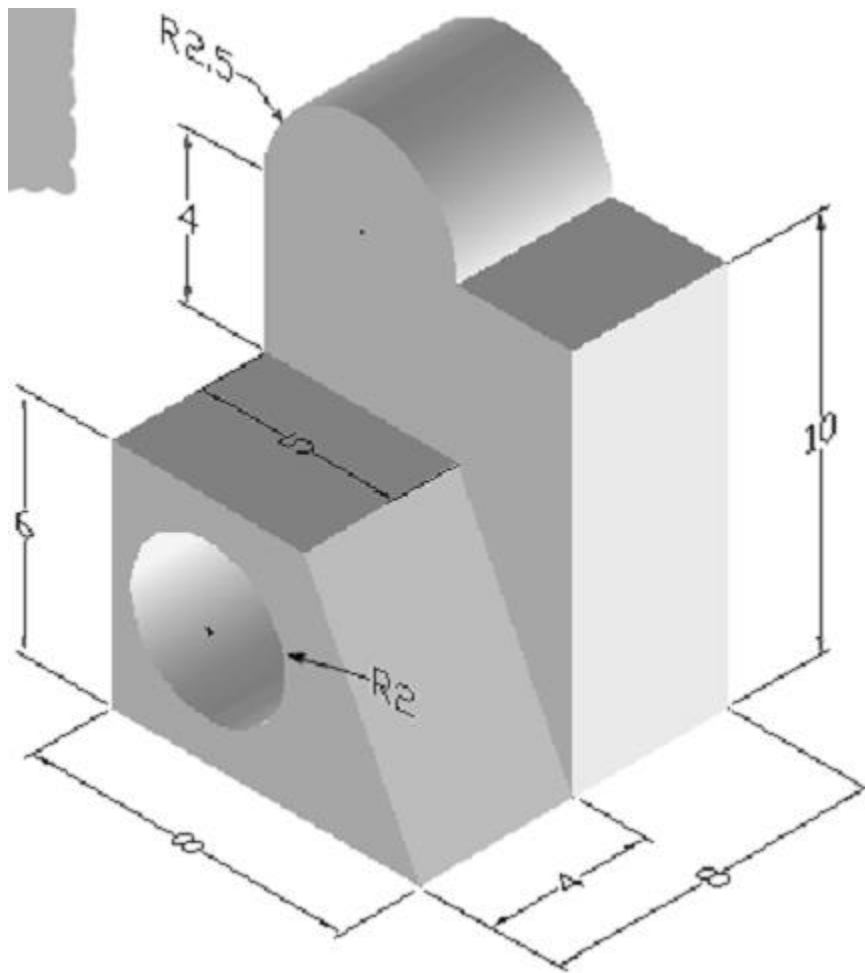


Make Front Side and plan along with Isometric view

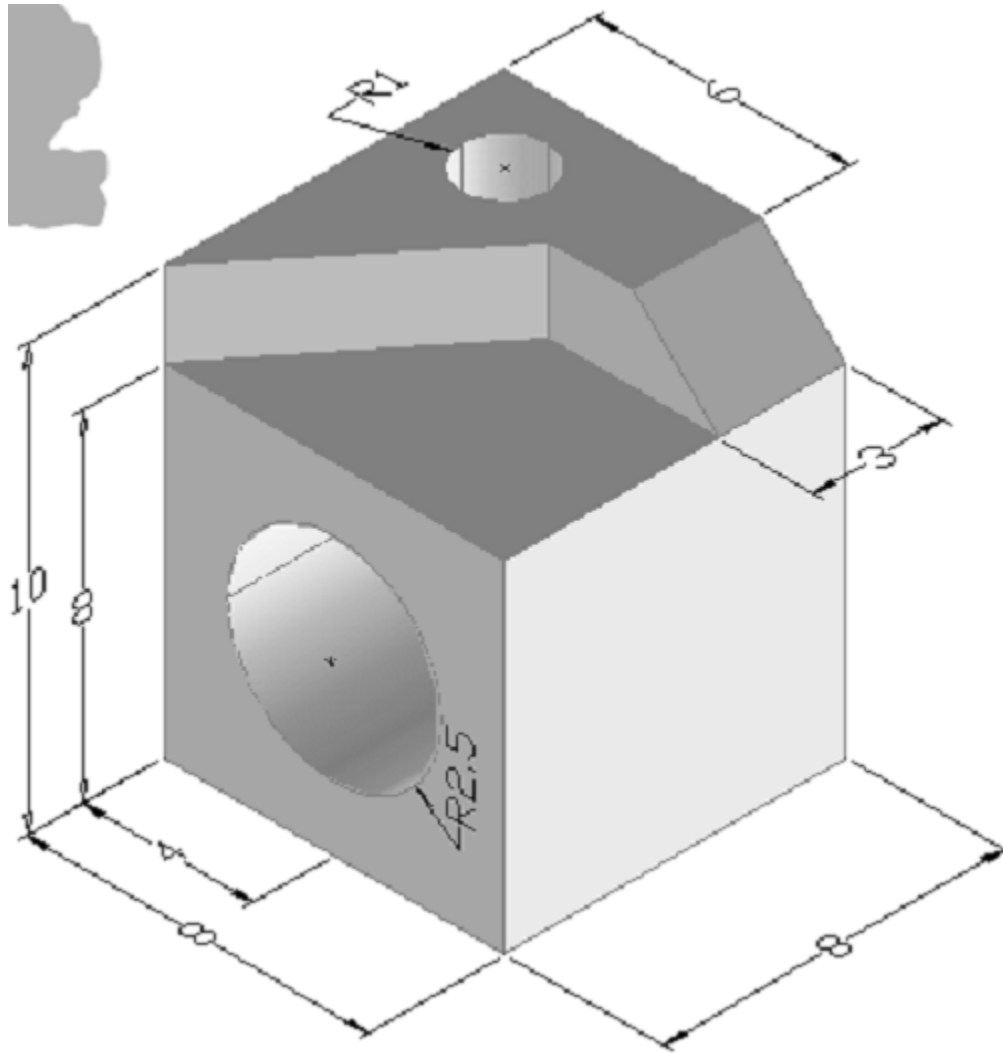
Exercise 20



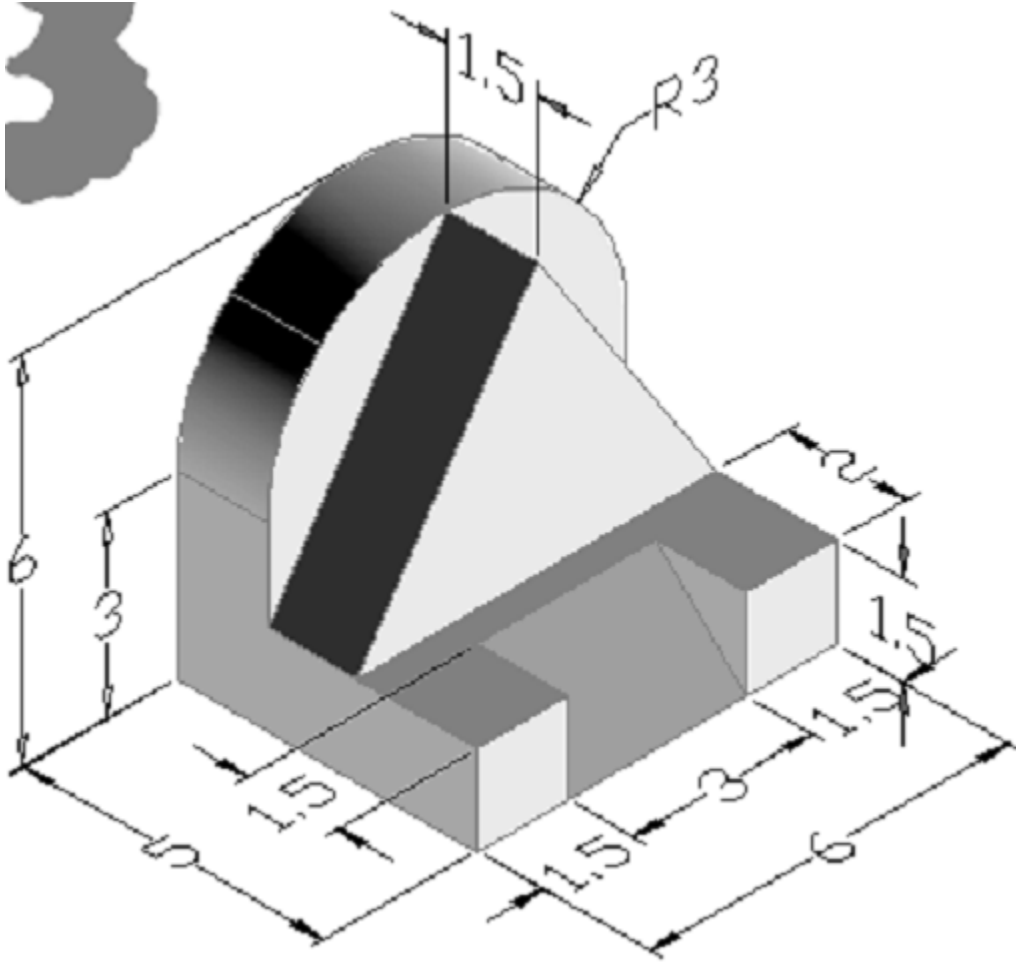
Exercise 21



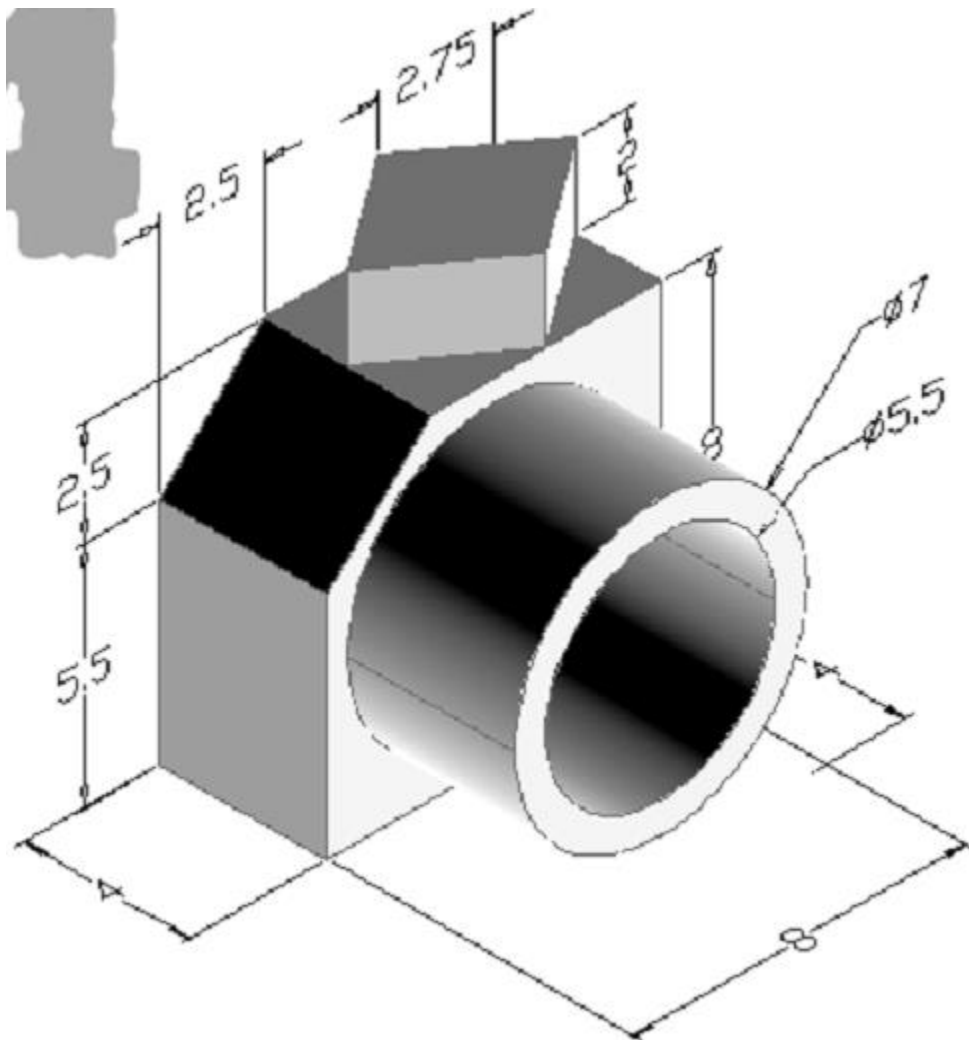
Exercise 22



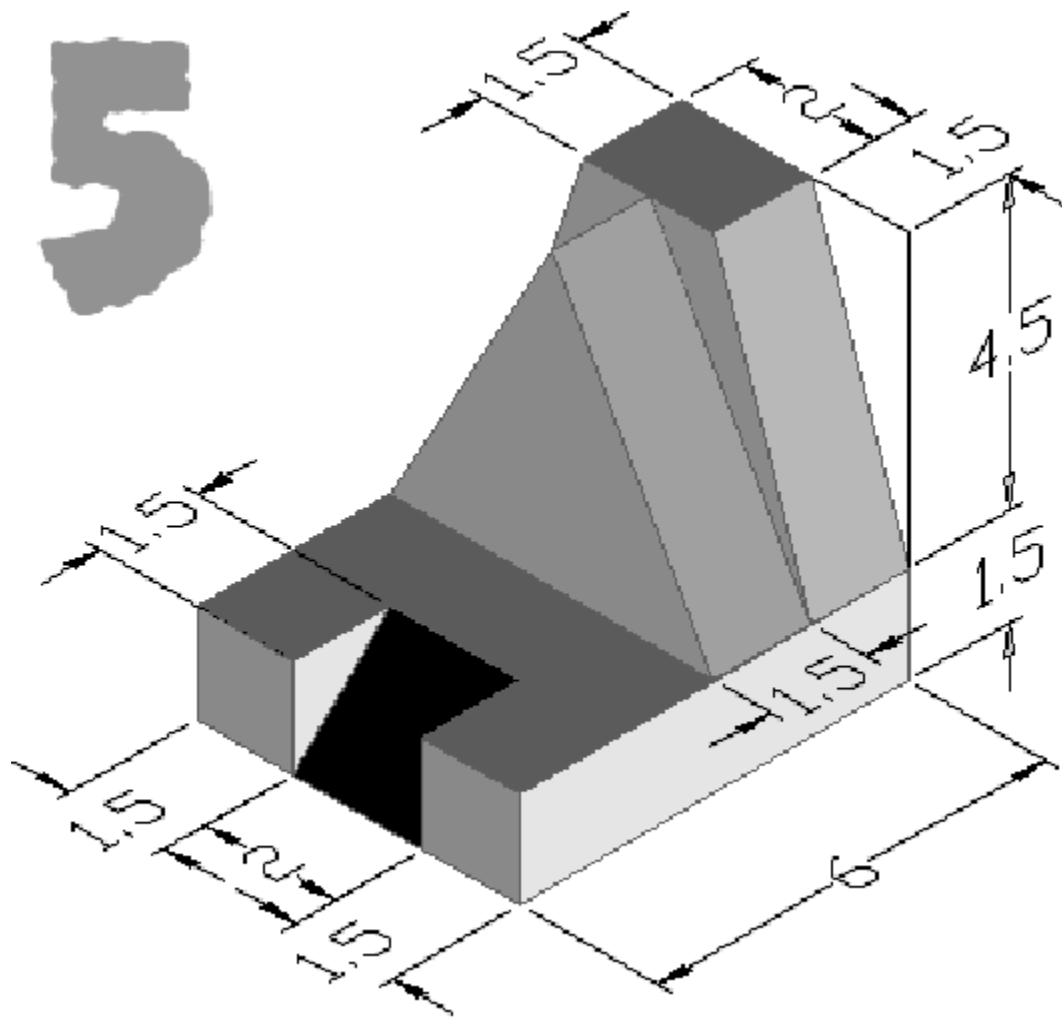
3



Exercise 24



Exercise 25



Exercise 26

