Department of Mechanical Engineering, MNNIT Allahabad, Prayagraj, UP

B.Tech. II Semester - Engineering Graphics (ME – 12101)

AutoCAD - Computer Aided Drafting

By

Dr. Mukul Shukla

Professor, Mechanical Engineering Department

Email – mukulshukla@mnnit.ac.in

(In association with Mr Ajeet Kumar and Dr Tarun Bhardwaj)

(April 2020)

Introduction: AutoCAD (Version 2018/19) (from Autodesk Inc., USA) is a popularly used general purpose, freely downloadable, easy to use, efficient, Computer Aided Design (CAD) and Drafting software (s/w), from which a variety of soft copy (hence portable, storable and long life) 2D drawings and 3D solid models of automobile and machine components (e.g. Fig. 1 - Spigot and Socket Joint), consumer products, buildings, electrical circuit layout, etc. can be made.

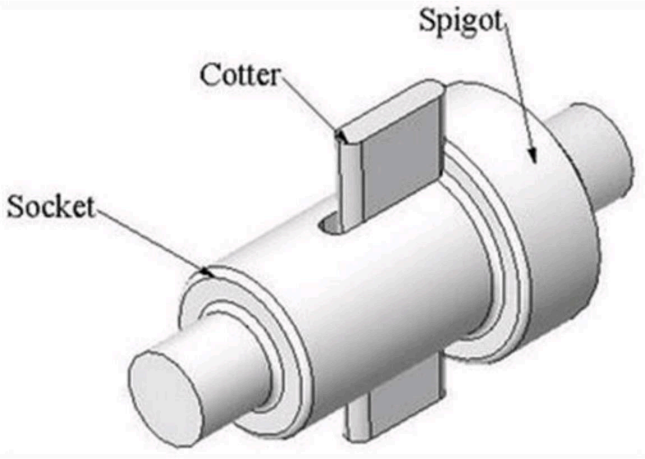


Fig. 1 - Spigot and Socket Joint

The following are some of the advantages of implementing CAD systems in real life in the companies:

1) **Increase in designer's productivity**: The CAD s/whelps designers in visualizing the final product that is to be made, its subassemblies and the constituent parts. The product can also be animated and seen how it actually works, thus helping the designer to immediately make the modifications if required. CAD s/w helps designers in synthesizing, analyzing (FEM) and documenting the design. All these factors help in significantly improving the productivity of the designer that translates into fast designing, lower designing cost and shorter project completion times.

2) **Improvement in design quality**: In a CAD s/w the designing professionals are offered a large number of tools to do thorough engineering analysis of the proposed design as also to consider large number of investigations. Since the CAD systems offer greater accuracy, the errors are reduced largely in the designed product leading to improved design. Eventually, better design helps carrying out manufacturing (or 3d Printing or Additive Manufacturing) faster and reducing the wastages that could have occurred because of the faulty design.

3) **Better communication**: After designing we make the product and component drawings for eventual manufacturing. With CAD s/w better and standardized drawings can be made easily, helping in better documentation of the design, fewer drawing errors, and greater legibility. The documentation includes geometries, shape and dimensions of the product, its subassemblies and its components, material specifications for the components, bill of materials (BOM) for the components etc. and forms a database for manufacturing.

** ***List the downstream applications of CAD! (DIY)…..***

1. **Basics:-** From the **View** menu put ON the **Standard, Draw, Modify, Object snap** and **Dimension** toolbars. (Optionally you may put ON the **View, Zoom, Solids** and **Shade** toolbars also.) Help on any of the Commands can be taken anytime by pressing the **F1** key.

   a) **Units:-** After opening AutoCAD, start with setting of the system of units (e.g. mm, inches etc.) along with the desired precision (1 or 2 or 3) of the dimensions and units of angle. *(Default units =??)*

   b) **Limits:-** Changes the default limits (=??) and sets the limits of the drawing that is to be made, by planning *apriori*. The origin is usually set to the Lower Left corner of the screen and the Upper Right corner is to be specified. (Avoid taking a square sized drawing area because the computer monitor resolution is rectangular, e.g. 420*297 in mm or 12*9 in inches)

   c) **Zoom all:-** Command – **zoom** (or **z**) then **all (or a)** (for implementing above changed Drawing limits)

   d) **F1 key:-** Online Help **F3 key:-** Object snap mode On/Off
   F12 key:- Dynamic Input **F7 key:-** For Grid On/Off
   F8 key:- For Ortho mode On/Off **F9 key:-** For Snap mode On/Off

   e) **Undo:-** (Shortcut – u) Reverses the most recent action.

   f) **Up / Down arrow keys –** Repeat immediate previous commands (without retyping the command)

   g) **AutoCAD can be run in 3 modes:**
      i) Command mode. Press "Ctrl+9" to open the command window. Many shortcut commands are available. (Try a, b, c, d,….. z)
      ii) Icon mode.
      iii) Pull down Menu mode.

2. **Draw tools:** They can be selected either from the *Draw* pull-down menu in the menu bar or from the Draw Toolbar. Some of the important draw commands are:-

LINE:-  Specify a point or press ENTER to continue from the last drawn line or arc.

CONSTRUCTION LINE:-  Specify a point or enter an option [Hor/Ver/Ang /Bisect/Offset]

MULTILINE:-  Specify a point or enter an option[Justification/Scale/Style].

POLYLINE:-  Specify a first point or enter an option [Arc/Close/Half width/Length/Undo/Width]

LINETYPE: Type "LINETYPE" command↵>click on "LOAD" to load **commonly used line types e.g. outline, hidden line, projection line, center line** etc.

Command	Icon	Shortcut key
Line		l
Polyline		Pline
Polygon		Polygon
Rectangle		Rectang
Arc		a
Circle	O	c
Ellipse		EL
Hatch		h

POINT:-Creates a point object.

ARC:- Creates an arc using three points.*(**Explore other options**)*

CIRCLE:-O Creates a circle using a specified radius.*(**Explore other options**)*

ELLIPSE:- Creates an ellipse using a specified center point.

POLYGON:- Enter number of sides or press ENTER Specify center of polygon or [Edge]

RECTANGLE:- Specify first corner point *or* Enter an option or specify a point[Chamfer/Elevation/Fillet/Thickness/Width].

HATCH:- (hatch) Fills / sections an enclosed area or selected objects with a **chosen hatch pattern**.

3. Object Snap tools:- These are used for proper alignment and positioning of a drawing entity with respect to another drawing entity.

Temporary Tracking Point- Creates a temporary point used by Osnaps.

Command	Icon	Shortcut
Snap from		FROM
Snap to Endpoint		ENDP
Snap to Midpoint		MID
Snap to Intersection		INT
Snap to Apparent intersect		APPINT
Snap to Extension		EXT
Snap to Center		CEN
Snap to Quadrant		QUA
Snap to Tangent		TAN
Snap to Perpendicular		PER

Snap from- Locates a point offset from a reference point within a command.

Snap to Endpoint- Snap to the closest endpoint of an object.

Snap to Midpoint- Snaps to the midpoint of an object.

Snap to Intersection- Snaps to the intersection of two objects.

Snap to apparent intersect- Snaps to the apparent intersection of two objects.

Snap to Extension- Snaps to the phantom extension of an arc or line.

Snap to Center- Snaps to the center of an arc, circle, Ellipse, or elliptical arc.

Snap to Quadrant- Snaps to a quadrant point of an arc, circle, ellipse, or elliptical arc.

Snap to Tangent- Snaps to a point tangent to an Object.

Snap to Perpendicular- Snaps to a point perpendicular to an object.

4. Modify tools:- These are tools used for the modification or editing of drawn objects.

ERASE:-Removes objects from a drawing. Select the objects to erase, right-click in the drawing area, and choose Erase.

COPY:-Duplicates objects. Select the objects to copy, right-click in the drawing area, and choose Copy Selection.

MIRROR:-Creates a mirror image copy of objects. Use an object selection method and press ENTER to finish.

OFFSET:- Creates concentric circles, parallel lines, and parallel curves. Specify a distance, enter *t*, or press ENTER

ARRAY:-Creates multiple copies of objects in a pattern. You can create **rectangular or polar arrays** by choosing the appropriate option. ***(Explore its applications)***

MOVE:-Displaces objects a specified distance in a specified direction. Select the objects to move, right-click in the drawing area, and choose Move.

ROTATE:-Rotates objects about a base point. Select the objects to rotate, right-click in the drawing area, and choose Rotate.

Command	Icon	Shortcut
Erase		e
Copy		cp
Mirror		mi
Offset		o
Array		ar
Move		m
Rotate		ro
Scale		sc
Trim		tr
Chamfer		cha
Fillet		f

SCALE:-Enlarges or reduces objects proportionally in the X, Y, and Z directions. Select the objects to scale, right-click in the drawing area, and choose Scale

TRIM:-Trims objects at a cutting edge defined by other objects. Select one or more objects and press ENTER, or press ENTER to select all objects.

CHAMFER:-Bevels the edges of objects. Select first line or [Polyline/Distance/Angle/Trim/Method]

FILLET:-(Shortcut – f) Rounds and fillets the edges of objects. Select first object or [Polyline/Radius/Trim] (Radius 0,2,3…. etc)

5. Dimensioning Tools:- (Shortcut – d) These tools are used for dimensioning a drawing.

Set ISO Dimension:- Type ''DIMSTYLE'' command> Enter>click modify option >click Text >click ISO standard >OK

Dimension Setting:- Type ''DIMSTYLE'' command >Enter >click modify option >an option [Line/symbols and Arrow/Text/Fit/Primary unit/Tolerances]

LINEAR DIMENSION:- Specify a point or enter an option

[Mtext/Text/Angle/Horizontal/Vertical/Rotated]

ALIGNED DIMENSION:- Specify a point for manual extension lines, or press ENTER for automatic extension lines.

RADIUS DIMENSION:- Specify a point or enter an option[Mtext/Text/Angle].

DIAMETER DIMENSION:- Specify a point or enter an option[Mtext/Text/Angle].

Command	Icon	Shortcut
Linear dimension		Dimlinear
Aligned dimension		Dimaligned
Radius dimension		Dimradius
Diameter dimension		Dimdiameter
Angular dimension		Dimangular
Quick dimension		Qdim

ANGULAR DIMENSION:- Select an arc, circle, or line, or press ENTER to create the angular dimension by specifying three points.

QUICK DIMENSION:- Select the objects you want to dimension and press ENTER an option [Continuous/Staggered/Baseline/Ordinate/Radius/Diameter/datumPoint/Edit].

For coloring a drawing the following dimension variables are used:-

1. Dimclrd: Controls the color of dimension lines and arrows.
2. Dimclre: Controls the color of dimension extension lines.
3. Dimclrt: Controls the color of dimension text.

6. View tools:- (Shortcut – v) These tools help in visualizing a drawing from different views for better interpretation e.g. **Zoom** (Shortcut – z) to increase or decrease the drawn object's size.

REGENERATE:- Command –**Regen** to regenerate a drawing after making any modifications to it.

ZOOM REALTIME:- With no objects selected right-click in the drawing area and choose Zoom to zoom in real time.

ZOOM WINDOW:- Zooms local portions of a drawing selected within a rectangular window.

ZOOM IN:-Increase the size of the drawing.

ZOOM OUT:-Decreases the size of the drawing.

Command	Icon
Zoom Window	
Zoom IN	
Zoom OUT	
Zoom All	

ZOOM ALL:-…….. ***(Self study)*** **ZOOM EXTENTS:-** …….. ***(Self study)***

PAN: -  (Shortcut – p) with no objects selected, right-click in the drawing area and choose pan.

PAN REALTIME:- ……

3d ORBIT:-…… ***(Self study)***

7. <u>3D Solid Modeling tools</u>:- These are advanced features of AutoCAD which enable 3-d object modeling using a Building-block approach. The following solids / primitives are commonly used:

BOX: - Specify corner of box or press ENTER for corner of box, or enter *c* for center.

SPHERE: - Specify radius of sphere or [Diameter] and Specify a distance or enter d.

CYLINDER: - Specify center point for base of cylinder or [Elliptical] or press ENTER.

CONE: - Specify center point for base of cone or [Elliptical] or press ENTER.

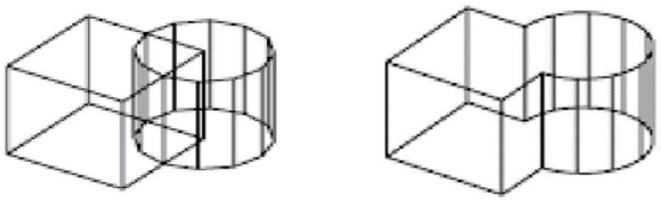
EXTRUDE: - Select objects and Specify height of extrusion or [Path].

REVOLVE: - Specify start point for axis of revolution or define axis by [Object/X (axis)/Y (axis)]: Specify a point or enter an option.

By a combination of **Union, difference (Subtract)** and **Intersect** operations on a combination of the above solids, any complicated 3d object can be created.

Command	Icon	Shortcut Key
Union		**UNI**
Intersect		**INT**
Subtract		**SU**
Press pull		**PRESS**
Loft		**LOF**

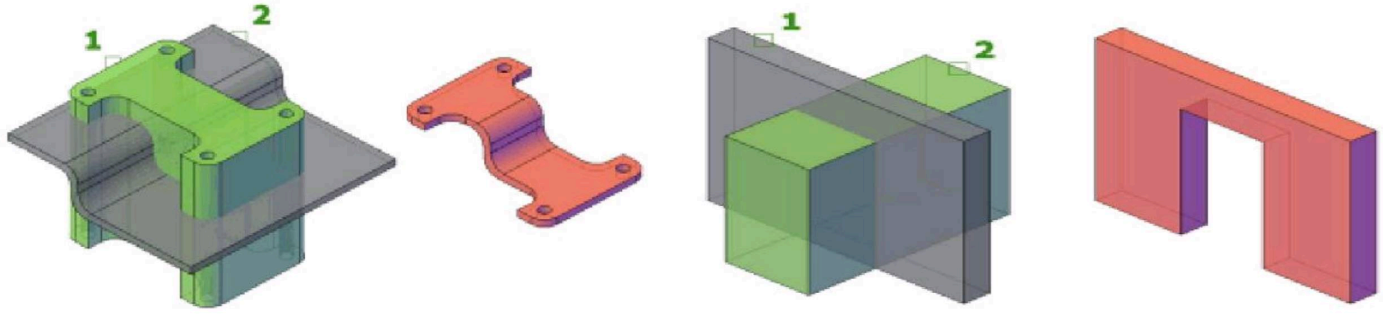
Union: Combines (or adds) two or more 3D solids, surfaces, or 2D regions into a single, composite 3D solid, surface, or region.



Solids before Union **Solid after Union**

Intersect: Creates a 3D solid, surface, or 2D region from overlapping solids, surfaces, or regions.

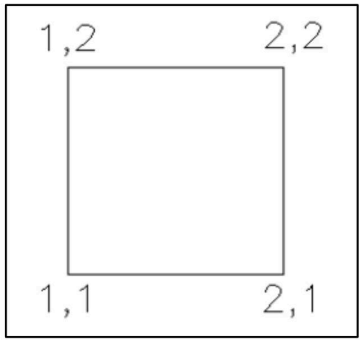
Subtract: Creates a new object by subtracting one overlapping region or 3D solid from another.

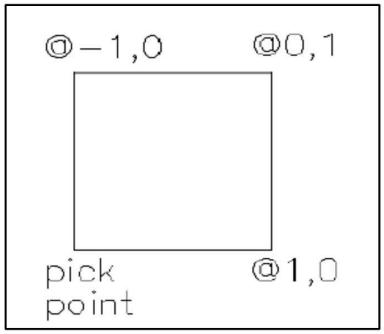


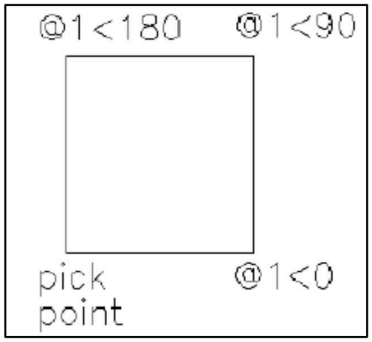
Output of Intersection of two solids (1 and 2) **Output of Subtraction of two solids (1 and 2)**

<u>Coordinate System</u>:-

DYN= ALWAYS OFF DYN =Dynamic Input (F12)







Absolute Coordinates: Input: X,Y Coordinates w. r. to origin	Relative Rectangle Coordinates from pick point. Input: Coordinates	Polar Coordinates From pick point. Input: Distance and angle

Absolute Coordinate system:- Type "Line'' Command, Enter, coordinate (1,1) then (2,1) then (2,2) then (1,2) then (1,1) and so on.

Relative Rectangle coordinate system:- Type "Line'' Command, Enter, pick point, coordinate @ (1,0) then @ (0,1) then @ (-1,0) then @ (0, -1) and so on.

Polar coordinate system:- Type "Line'' Command, Enter, pick point, coordinate @ (1<0) then @ (1<90) then @ (1<180) then @ (1<270) and so on.

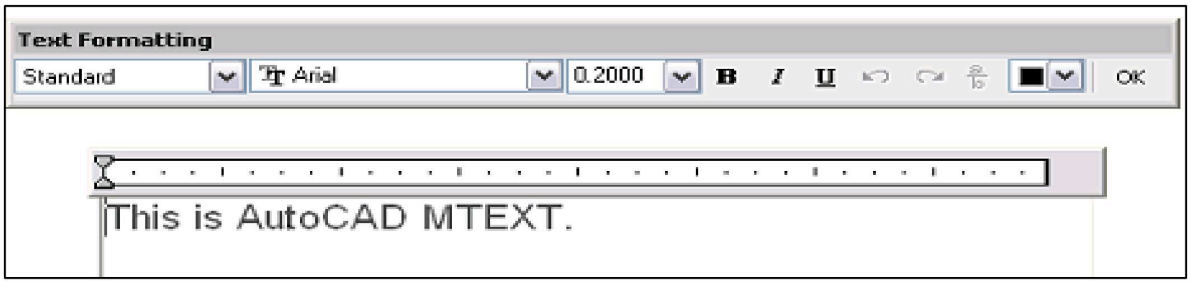
Text Command:

Text Style: You can specify the current text style to determine the appearance of all new text. A text style includes the font, size, oblique angle, orientation, and other text characteristics.

Editing Text: Edits single-line texts, dimension text, attribute definitions, and feature control frames.

Command	Icon	Shortcut Key
Text		**TEXT**
Multiline Text		**MTEXT**
Text Style		**STYLE**
Editing Text		**DDEDIT**

Mtext Editor:



***Supplementary Reading**:* The concept and usage of **Layers** (e.g. to group objects in a drawing by function and to enforce standards for colour, line type, line weight and other properties. **Command: LAYER (or LA)),**

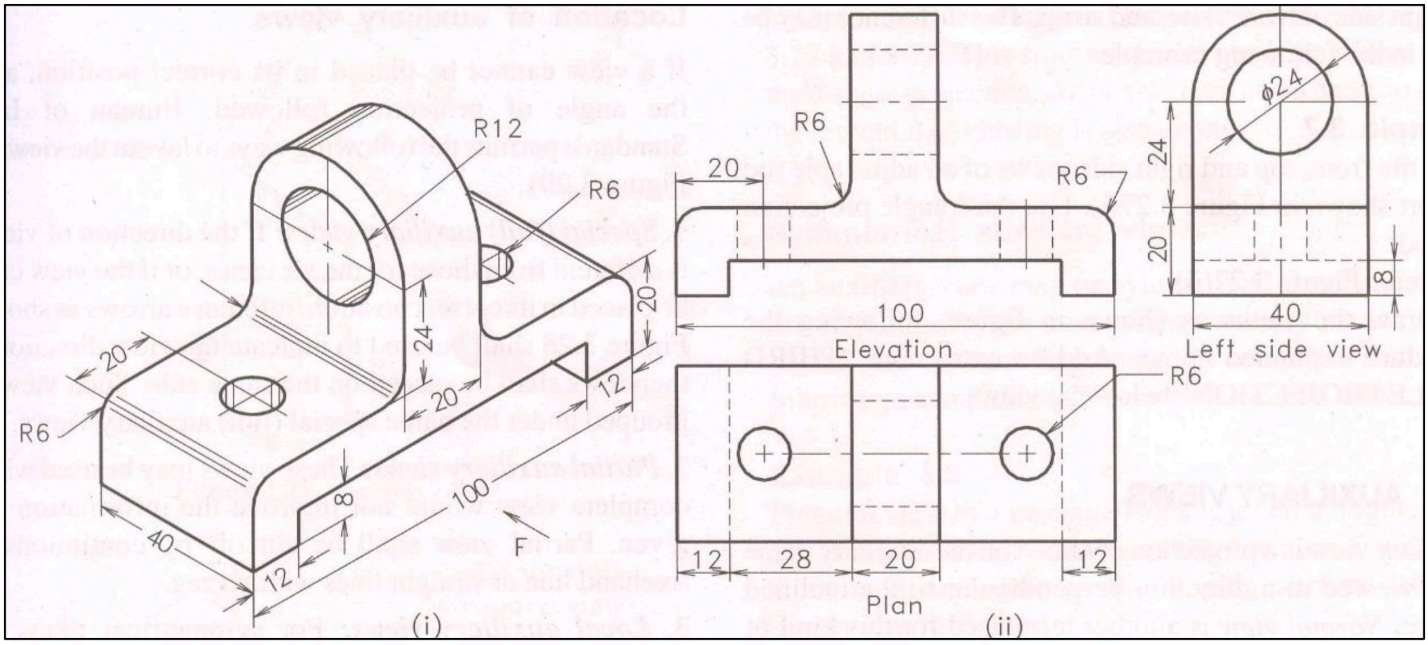
Also try –

- ✓ **Block** (shortcut – b)**,**

- ✓ **Isometric views,**
- ✓ **Export file** (formats), **dxf file, Spline,**
- ✓ **Viewport,**
- ✓ **UCS,**
- ✓ **Hide, Render, Shade,** and
- ✓ **Surfaces** in AutoCAD???

- - - - - - - ** - - - - - - -

Example 1: Draw unsectioned front, top and left side views of the object shown below.



Front view;

Step-1:Type"Line''command>Enter>Pick 1st point>Pick 2nd point

Step-2:Type "Fillet'' command>Enter>Type "Radius''>Enter>give radius value>Enter>click 1st line>2nd line

Step-3: Type "Offset''command, or Enter "O">give distance value enter>select line>give direction

Step-4:Type "Mirror''command >Enter>select object>Enter>pick mirror point>give direction>Enter

Top view-:

Step-1: Click Rectangle icon (X=100,Y=40)Enter

Step-2:Select Rectangle>click explodes

Step-3:Type "Offset'' command>Enter>give offset distance value enter>select line>click give direction

Step-: Click circle icon>specify centre point for circle>give radius value enter

Left Side view-:

Step-1:Type "Line'' command>Enter>Pick 1st point>Pick 2nd point

Step-2:Type"Mirror'' command>Enter>select object>Enter>pick mirror point>give direction>Enter

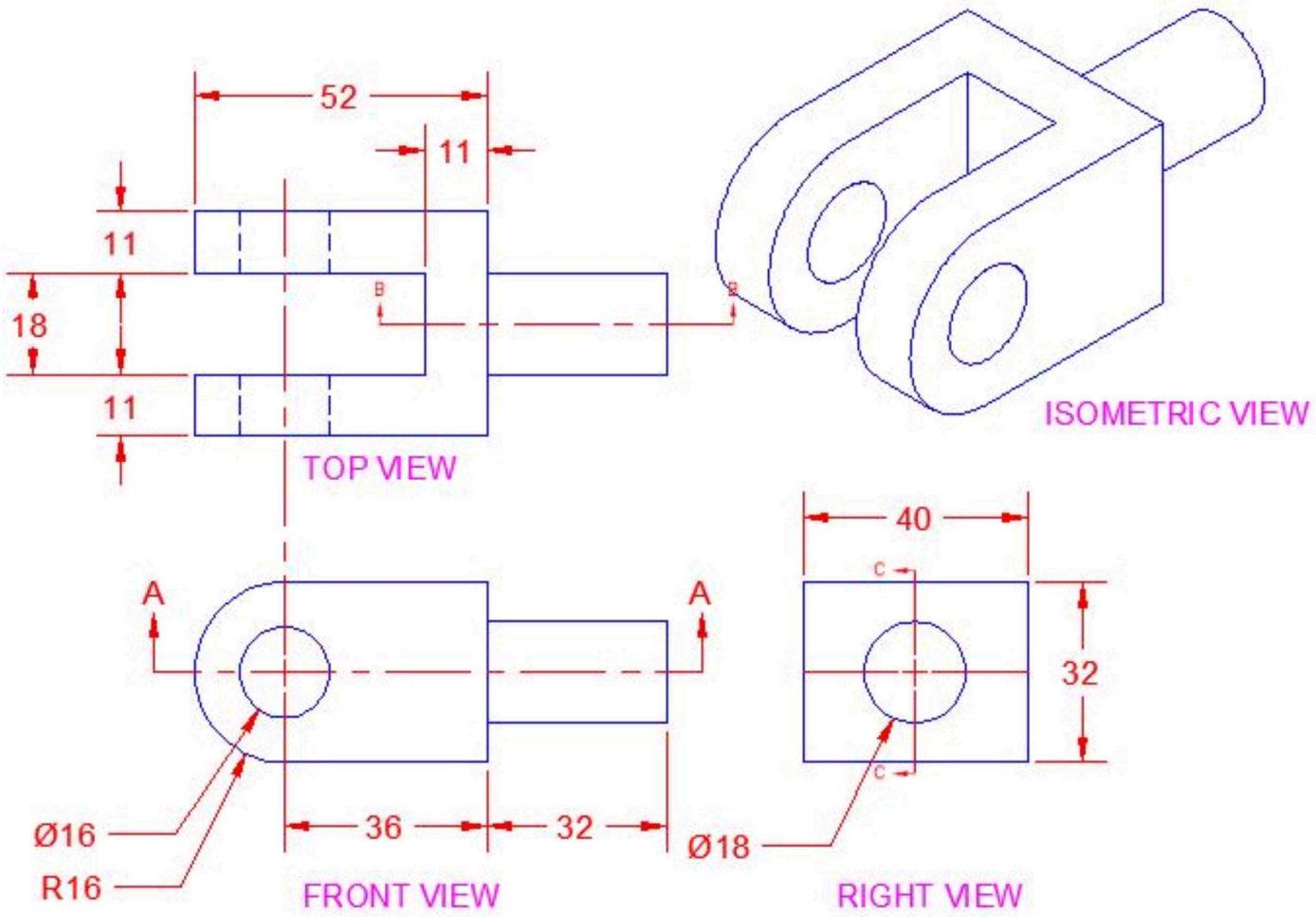
Step-3:Type "offset'' command, or Enter "O">give distance value enter>select line>give direction

Step-4: Click circle icon>specify centre point for circle>give radius value enter

Step-4: Type "TRIM'' command >double enter>click remove line

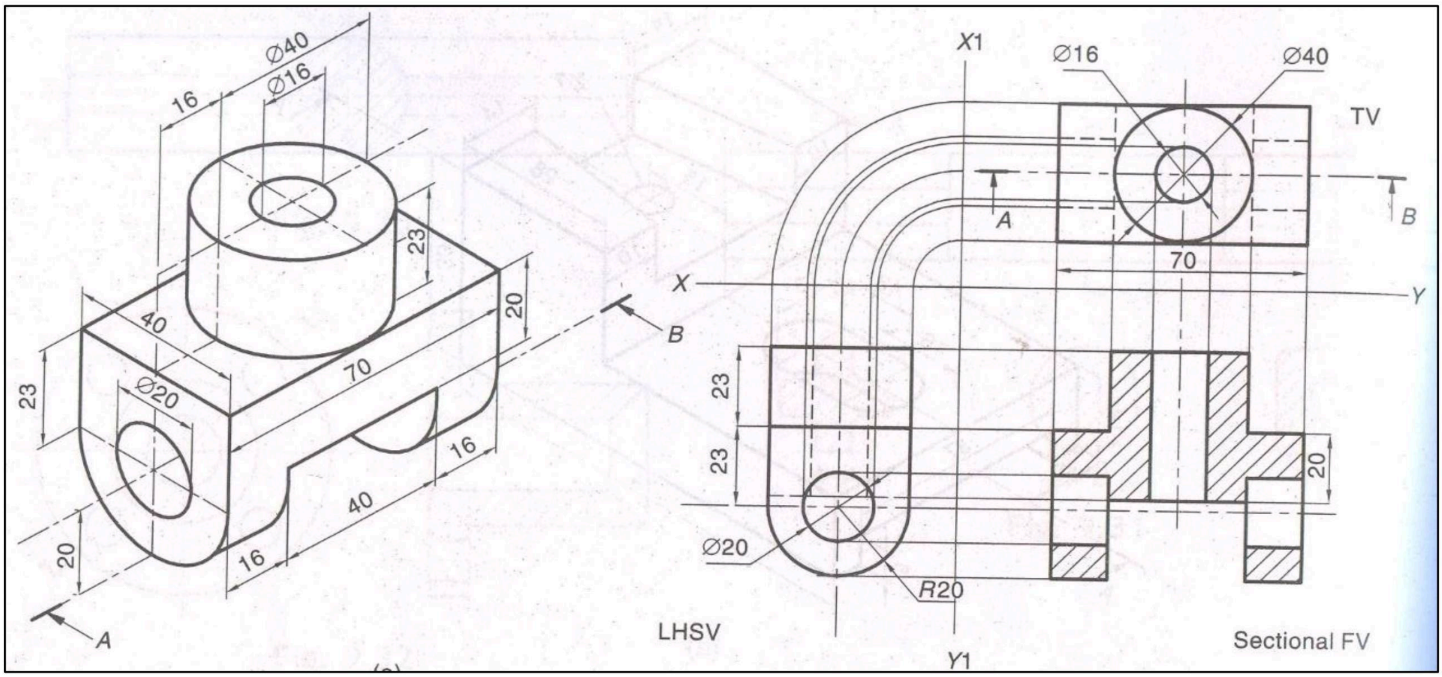
Step-5: Dimension >click 1st point & click 2nd point>give direction

Example 2: a) Draw unsectioned front, top and right view of the object shown below.
b) Also try to draw its Isometric view in AutoCAD.



Example 3: Draw the three orthographic views (as shown) of the object given below.

Sectional Front view:



Step-1: Type "Line'' command>Enter>Pick 1st point>Pick 2nd point

Step-2: Centre line, select line>by layer>choose centre line

Step-3: Type "offset'' command or Enter "O">give distance value enter>select line>give direction

Step-4: Type "Mirror'' command>Enter>select object>Enter>pick mirror point>give direction>Enter

Step-5: Type "Boundary'' command>Enter>click pick points>pick internal point>Enter

Step-6: Type "Hatch'' command>Enter>choose pattern>ok> add pick points>pick internal Point>Enter>ok

Step-7: Dimension >click 1st point & click 2nd point>give direction