

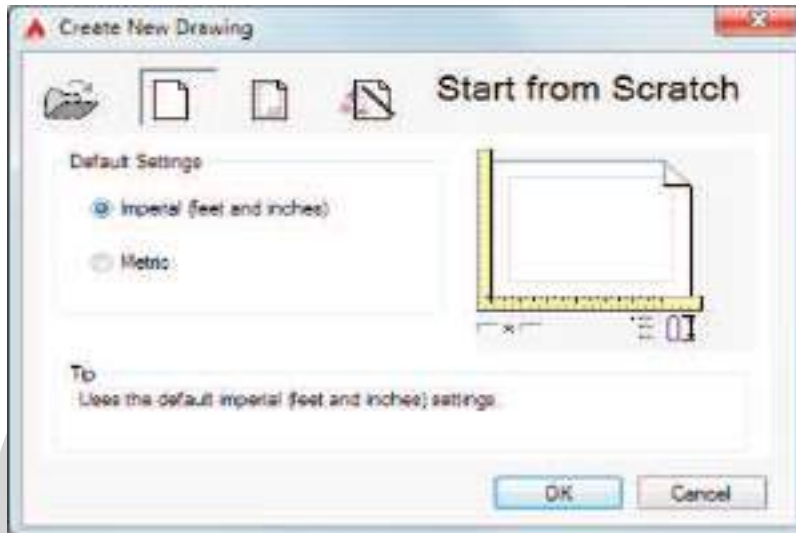
Lesson 01

Creating a Basic 3D Surface Model

Elevation & Thickness

To work in three dimensions in AutoCAD, we need to use a third axis on the rectangular (Cartesian coordinate system. This axis (defined as Z), determines the depth of an object. In this context, the X-axis will identify the WIDTH, the Y-axis LENGTH and the Z-axis determines the DEPTH of an object.

1. Start a new file from scratch. Accept all the default settings.



2. Turn the Grid and Snap ON (F7 & F9), and use the default spacing.

Command: **gird** <Enter>

Specify gird spacing (X) or [ON/OFF/Snap/Major/aDaptive/Limits/Follow/Aspect]
<0.5000>: **L** <Enter>

Display gird beyond Limits [Yes/No] <Yes>: **n** <Enter>

Command: <Enter>

Specify gird spacing (X) or [ON/OFF/Snap/Major/aDaptive/Limits/Follow/Aspect]
<0.5000>: **d** <Enter>

Turn adaptive behavior on [Yes/NO] <Yes>: **n** <Enter>

Command: <Enter>

GRID

Specify grid spacing (X) or [ON/OFF/Snap/Major/aDaptive/Limits/Follow/Aspect]
<0.5000>: **.5** <Enter>

{Press F7 to turn the grid ON}

Command: <Grid on>

Command: **z** <Enter>

ZOOM

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window] <real time>: **a** <Enter>

3. Create a new layer named object, assign the color green to it, and make it the current layer.

4. Enter the **ELEV** Command and set the new default elevation at **1"** and the new default thickness at **3"**.

Command: **elev** <Enter>
 Specify new default elevation < 0.0000 >: **1** <Enter>
 Specify new default thickness <0.0000>: **3** <Enter>

5. Begin your drawing with the **LINE** Command, and construct the figure 1. Do not be concerned about the exact sizes. Keep your drawing proportional to one shown in Figure 1.

6. Use the **VPOINT** and set it to SE Isometric.

Command: **vpoint** <Enter>
 Current view direction: VIEWDIR= 0,0000,0.0000,1.0000
 Specify a view point or [Rotate]<display compass and tripod>: **1,-1,1** <Enter> Regenerating model.

You may access this command from "View" pull down menu. (Figure 2)

Your drawing in SE Isometric will look similar to one shown in figure 3.

7. Changing the Elevation and Thickness

Command: **elev** <Enter>
 Specify new default elevation <1.0000>: **-1** <Enter>
 Specify new default thickness <3.0000>: **6** <Enter>

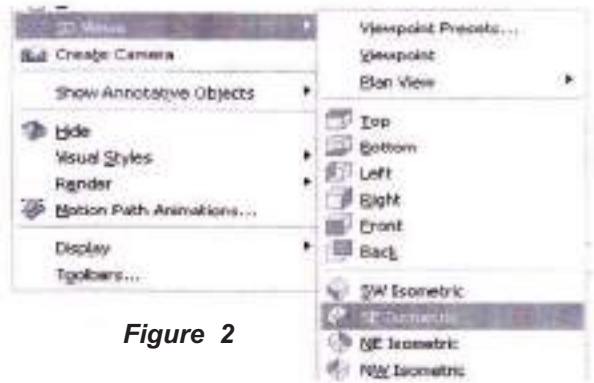


Figure 2

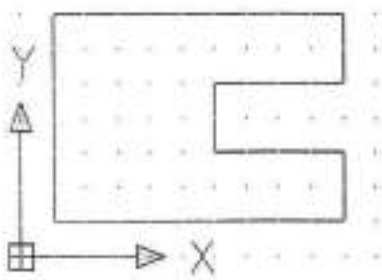


Figure 1 PLAN VIEW

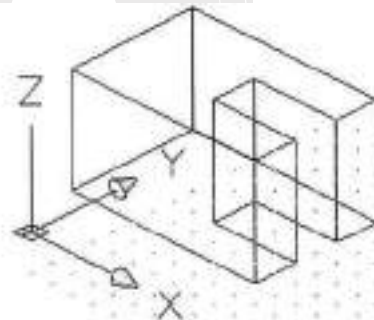


Figure 3 3D View

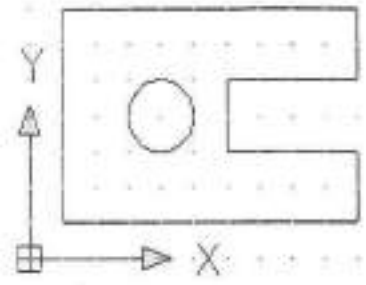


Figure 4

8. Add a circle as shown on Figure 4 and view from SE Isometric. (Figure 5)

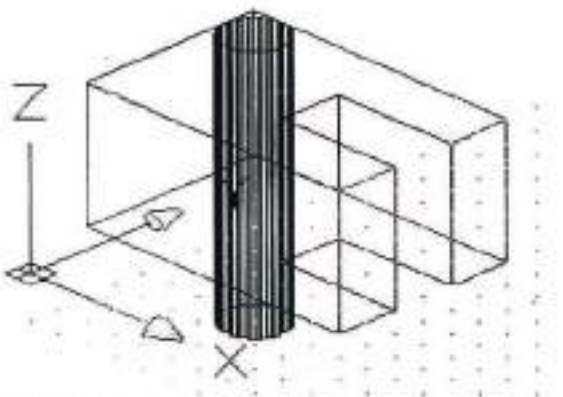


Figure 5

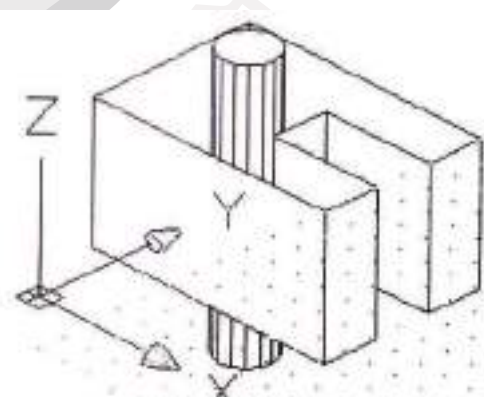


Fig. 6

8. From the "View" Pull Down menu, select "Hide", or enter the **HIDE** command from the keyboard.

Exercise 1-1

Construct a 3D solid model using Thickness and elevation change. (Figure 1-1)

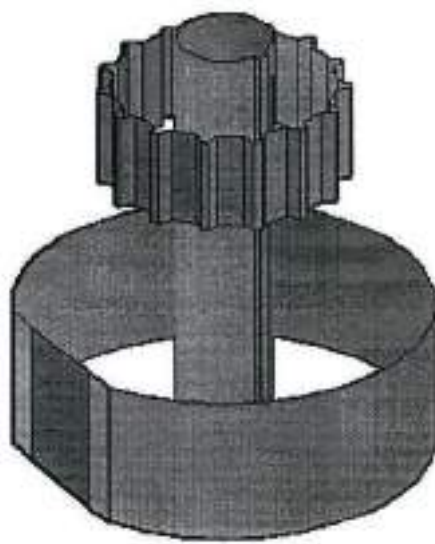
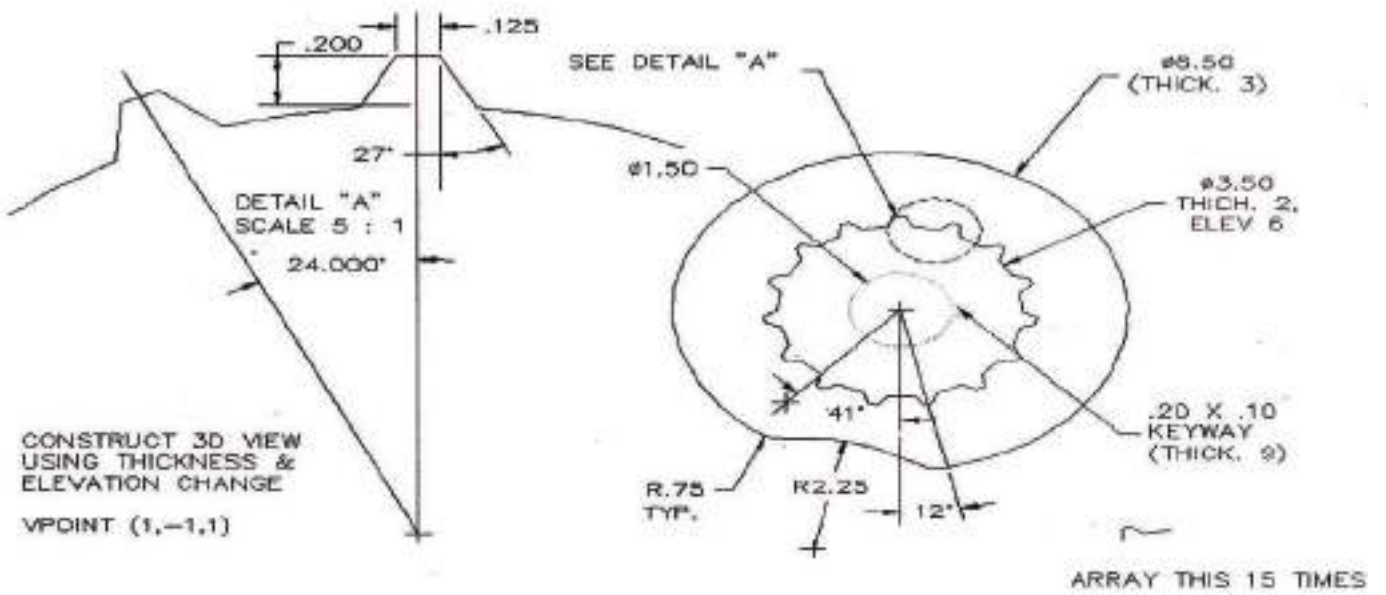


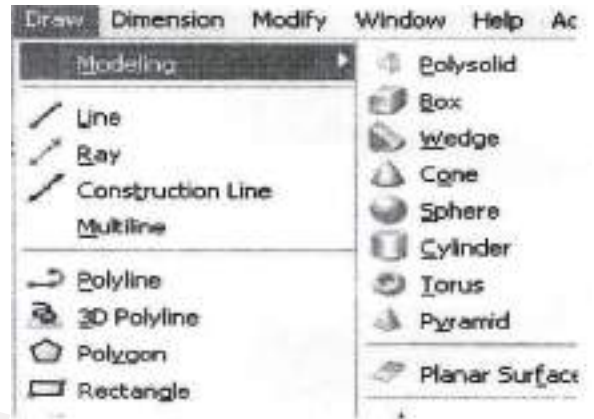
Figure 1-1

Lesson 02 Creating Primitives BOX

- Modeling toolbar: 
- Draw menu: Modeling > Box
- Command entry: box
- Dashboard: 3D Make panel, Box

Start a new file from scratch. Accept all the default setting.

1. Use the **VPOINT** and set it to SE Isometric.



2. Use the pull-down menu or the toolbar to select the desired command line.

Command: **vpoint** <Enter>

Current view direction: VIEWDIR = 0.0000,0.0000,1.0000 <Enter>

Specify a view point or [Rotate] <display compass and tripod>: **1,-1,1** <Enter> Regenerating model.

Command: **box** <Enter>

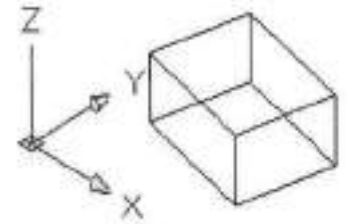
Specify first corner or [Center]: {Pick a point anywhere on screen}

Specify other corner or [Cube/Length]: **L** <Enter>

Specify length: **4** <Enter>

Specify width: **3** <Enter>

Specify height or [2 Point]: **2** <Enter>



Lesson 03 Creating Primitives **CYLINDER & WEDGE**

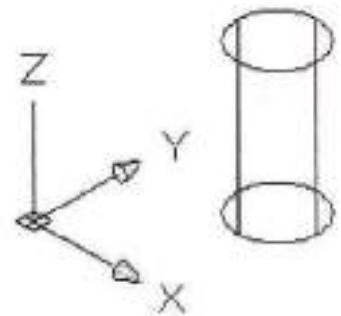


Command: **cylinder** <Enter>

Specify center point of base or [3p/2P/Ttr/Elliptical]:
{Pick a point anywhere on screen}

Specify base radius or [Diameter]: **1** <Enter>

Specify height or [2 point / Axis endpoint] <2.0000>: **4** <Enter>



Command: **wedge** <Enter>

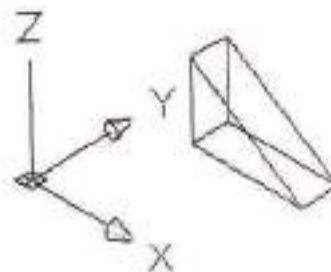
Specify first corner or [Center]: <Enter>

Specify other corner or [Cube/Length]: **L** <Enter>

Specify length <3.0000>: **3** <Enter>

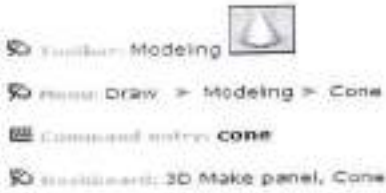
Specify width <4.0000>: **1** <Enter>

Specify height or [2 Point] <2.0000>: **2** <Enter>

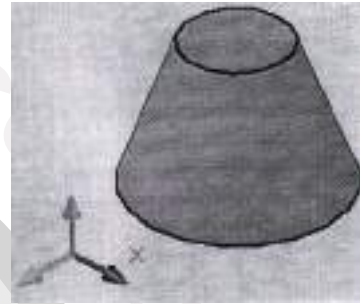
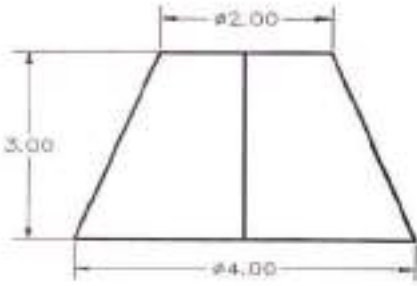


Lesson 04 Creating Primitives CONE & FRUSTUM CONE

Creates a 3D solid with a circular or elliptical base tapering symmetrically to a point or a circular or elliptical planar face



Command: `_cone <Enter>`
 Specify center point of base or [3P/2P/Ttr/Elliptical]: `5,5 <Enter>`
 Specify base radius or [Diameter] `2 <Enter>`
 Specify height or [2 Point/Axis endpoint/Top radius] `<0000>: _top`
 Specify top radius `1 <Enter>`
 Specify height or [2 Point/Axis endpoint] `3 <Enter>`

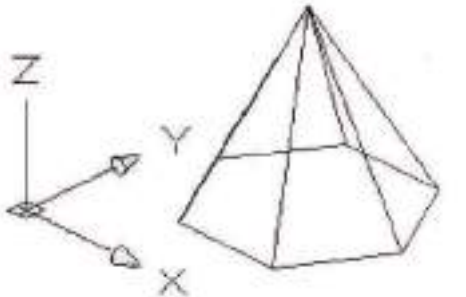
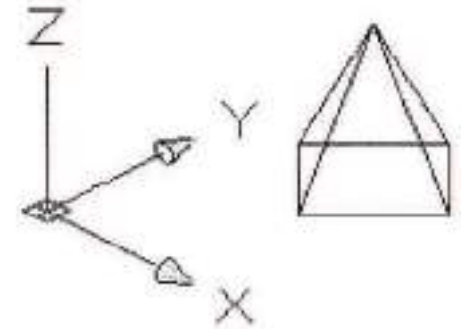


Lesson 05 Creating Primitives PYRAMID



Command: `pyramid <Enter>`
 4 sides Circumscribed
 Specify center point of base or [Edge/Sides]:
{Pick a point anywhere on screen}
 Specify base radius or {Inscribed} `<1.0000>: 1 <Enter>`
 Specify height or [2 Point /Axis endpoint/Top radius] `<2.0000>: 3 <Enter>`

Command: `pyramid <Enter>`
 4 Sides Circumscribed
 Specify center point of base or [Edge/Sides]: `S <Enter>`
{Pick a point anywhere on screen}
 Specify base radius or [Inscribed] `<1.0000>: 2 <Enter>`
 Specify height or [2 Point /Axis endpoint /Top radius] `<2.0000>: 5 <Enter>`



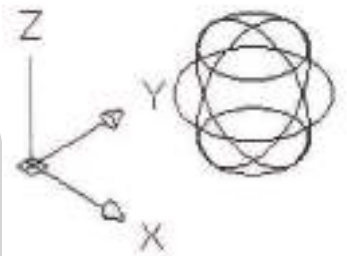
Command: **pyramid** <Enter>
 4 Sides Circumscribed
 Specify center point of base or [Edge/Sides]: **s** <Enter>
 Enter number of sides <4>: **5** <Enter>
 Specify center point of base of [Edge/Sides]: {Pick a point anywhere on Screen}
 Specify base radius of [Inscribed]: <Enter>
 Specify height of [2 Point/Axis endpoint/Top radius] <-9,2512>: **t** <Enter>
 Specify top radius <0.0000>: **.5** <Enter>
 Specify height or [2 Point/Axis endpoint] <-9.2512>: **6** <Enter>



Lesson 06
Creating Primitives
SPHERE - TORUS



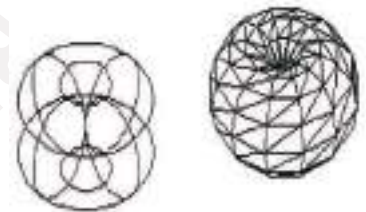
Command: **sphere** <Enter>
 Specify center point or [3P/2P/Ttr]: {Pick a point anywhere of screen}
 Specify radius or [Diameter] <2.3094>: **2** <Enter>



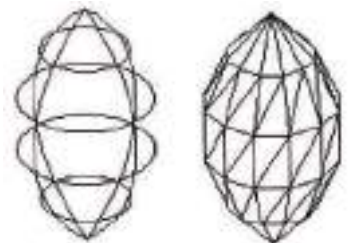
Command: **torus**
 Specify center point or [3P/2P/Ttr]: {Pick a point anywhere on screen}
 Specify radius of [Diameter] <3.3282>: **4**
 Specify tube radius or [2 Point/Diameter]: **1**



Command: **torus**
 Specify center point or [3P/2P/Ttr]: {Pick a point anywhere on screen}
 Specify radius or [Diameter] <3.0000>: **4**
 Specify tube radius or [2 Point/Diameter] <6.0000>: **4**



Command:
TORUS
 Specify center point or [3P/2P/Ttr]: {Pick a point anywhere on screen}
 Specify radius or [Diameter] <4.0000>: **-1**
 Specify tube radius of [2 Point/Diameter] <2.0000>: **1.5**



Lesson 07 Crating Primitives **POLYSOLID**

-  Modeling toolbar:
-  Draw menu: **Modeling > Polysolid**
-  Command entry: **polysolid**
-  Dashboard: **3D Make panel, Polysolid**

Command: **polysolid** <Enter>
 Specify start point or [Object/Height/Width/Justify] <Object>: **w** <Enter>
 Specify width <0.2500>: **.4** <Enter>
 Specify start point of [Object/Height/Width/Justify] <Object>: **h** <Enter>
 Specify height <4.0000>: **3** <Enter>
 Specify start point or [Object/Height/Width/Justify] <Object>:
 {Pick a point anywhere on screen}
 Specify next point or [Arc/Undo]: <**Ortho on**> **3** <Enter>
 Specify next point or [Arc/Undo]: **2** <Enter>
 Specify next point or [Arc/Close/Undo]: **3** <Enter>
 Specify next point or [Arc/Close/Undo]: **c** (Figure A)

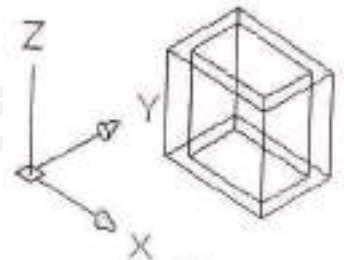


Figure A

Using the **POLYLINE** command (**PL**), to create the figure "A".
 Use **POLYSOLID** command; select "Object" option to construct Figure "C".

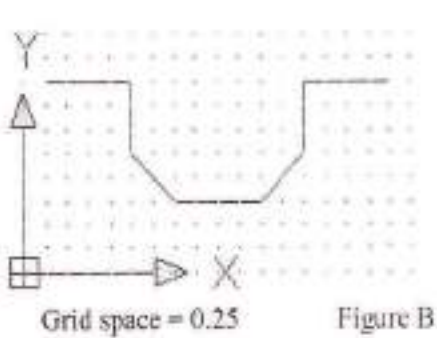


Figure B

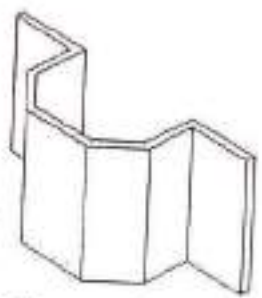
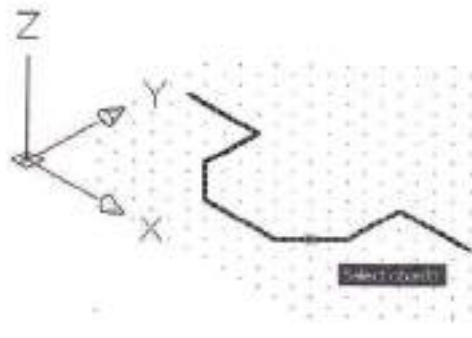
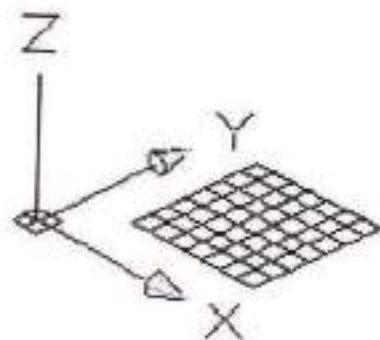


Figure C

Lesson 08 Constructing a **Planar Surface**

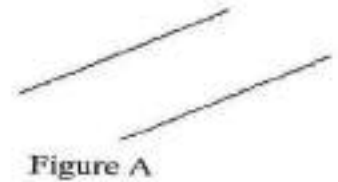
-  Modeling toolbar:
-  Draw menu: **Modeling > Planar Surface**
-  Command entry: **planesurf**
-  Dashboard: **3D Make panel, Planar Surface**

Command: **_Planesurf** <Enter>
 Specify first corner or [Object] <Object>: **4,4** <Enter>
 Specify other corner: **10,10** <Enter> (Fig. A)



(A)

Command: **1** <Enter>
 LINE Specify first point: **11,11** <Enter>
 Specify next point or [Undo]: **@ 10 <90** <Enter>
 Specify next point or [Undo]: <Enter>



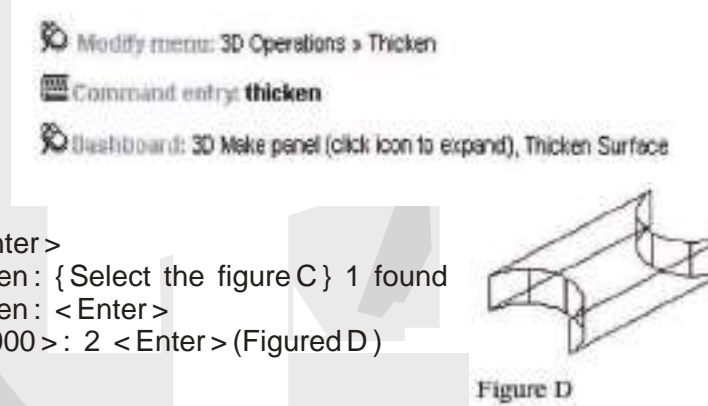
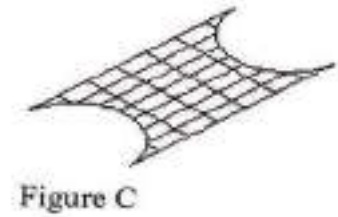
Command: **o** <Enter>
 OFFSET
 Current setting: Erase source=No Layer=Source OFFSETGAPTYPE=0
 Specify offset distance or [Through/Erase/Layer] <1.0000>: **5** <Enter>

Select object to offset or [Exit/Undo] <Exit>: {Select the line}
 Specify point on side to offset or [Exit/Multiple/Undo] <Exit>:
 {Select the Side}

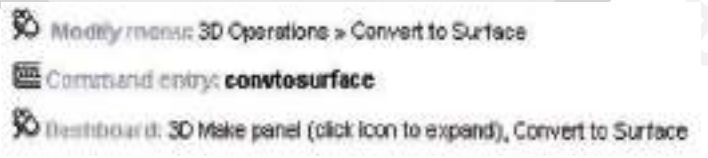
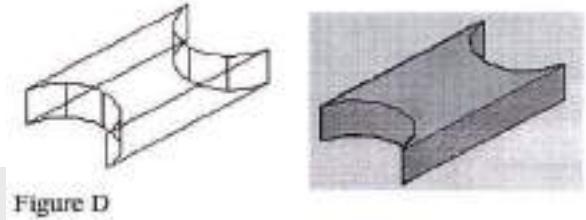


Use the ARC 3-point option to draw two arc as shown. (Figure B)

Command: **_Planesurf** <Enter>
 Specify first corner or [Object] <Object>: **o** <Enter>
 Select Objects: {Select line} 1 found
 Select Objects: {Select the line} 1 found, 2 total
 Select Objects: {Select the arc} 1 found, 3 total
 Select Objects: {Select the arc} 1 found, 4 total
 Select Objects: <Enter>



Command: **thicken** <Enter>
 Select surfaces to thicken: {Select the figure C} 1 found
 Select surfaces to thicken: <Enter>
 Specify thickness <0.0000>: **2** <Enter> (Figure D)



With the CONVTSOLID command, you can convert the following objects into extruded 3D solids:

- * Uniform-Width width polylines with thickness
- * Closed, zero-width polylines with thickness
- * Circles with thickness

Note you cannot use CONVTSOLID with polylines that contain vertices with 0 width or that contain segments of variable width.

Using the POLYLINE command, construct the figure E, and then cover it to a solid model.
 Command: **PLINE** <Enter>

Specify start point: **25,25** <Enter>
 Current line - width is 0.000
 Specify next point or [Arc / Half width / Length / Undo / Width]: **@ 5 <90 <Enter>**
 Specify next point or [Arc / Half width / Length / Undo / Width]: **a** <Enter>
 Specify endpoint of arc or
 [Angle / CEnter / CClose / Direction / Half width / Line / Radius / Second pt / Undo / Width]: **1** <Enter>
 Specify endpoint of arc or
 [Angle / CEnter / CClose / Direction / Half width / Line / Radius / Second pt / Undo / Width]: **2** <Enter>
 Specify endpoint of arc or
 [Angle / CEnter / CClose / Direction / Half width / Line / Radius / Second pt / Undo / Width]: **.5** <Enter>
 Specify endpoint of arc or
 [Angle / CEnter / CClose / Direction / Half width / Line / Radius / Second pt / Undo / Width]: **L** <Enter>
 Specify next point or
 [Arc / Close / Half width / Length / Undo / Width]:
@ 5 <-90 <Enter>
 Specify next point or
 [Arc / Close / Half width / Length / Undo / Width]: **c**



Figure E

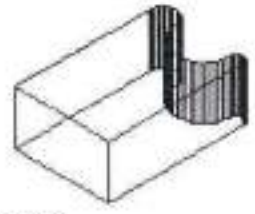


Figure F

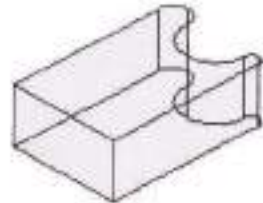
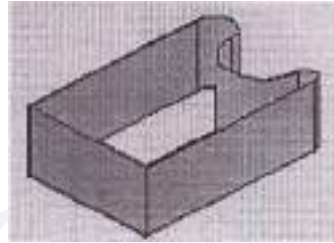
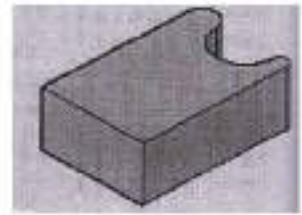


Figure G



Change the Thickness of the figure “E” from zero to 2. {Figure F}

Command: **convtosolid** <Enter>
 Select objects: 1 found {Select the figure F}
 Select objects: <Enter>

Lesson 09
 Creating Composite Solids
 (Boolean Operations : **UNION & SUBTRACT**)

Start AutoCAD and use default setting.

Command: **vpoint** <Enter> then type **1, -1, 1**. This is the same as selecting “SE Isometric” from the pull down menu “view”.

Command: **box** <Enter>
 Specify corner of box or [Center] <0,0,0>: **4,3.5** <Enter>
 Specify corner or [Cube/Length]: **L** <Enter>
 Specify Length: **3.5** <Enter>
 Specify Width: **.5** <Enter>
 Specify height: **1.5** <Enter>

Command: **z**

ZOOM

Specify corner of windows, enter a scale factor (nX or nXP), or [All / Center / Dynamic / Extents / Previous / Scale / Windows] <real time>: **1** <Enter>

Command: **cylinder** <Enter>
 Current wire frame density: ISOLINES=4
 Specify centerpoint ofr base of cylinder or [Elliptical] <0,0,0>: **5.75, 3.75, 1.5** <Enter>

Specify radius of base for cylinder of [Diameter]: **d** <Enter>
 Specify diameter for base of cylinder: **2** <Enter>
 Specify height of cylinder or [Center of other end]: **-3** <Enter>

Command: **cone** <Enter>
 Current wire frame density: ISOLINES=4
 Specify center point for base of cone or [Elliptical] <0,0,0>: **5.75,3.75,1.75** <Enter>
 Specify radius for base of cone or [Diameter]: **.75** <Enter>
 Specify height of cone or [Apex]: **-3** <Enter>

Your drawing at this time should look like the figure shown:

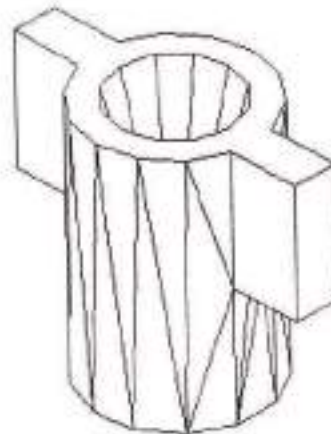
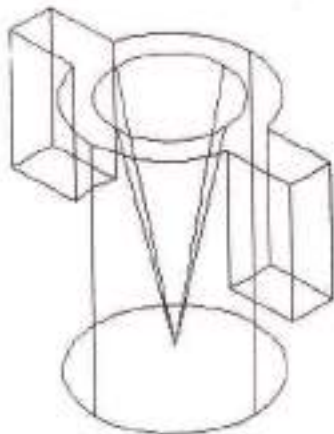
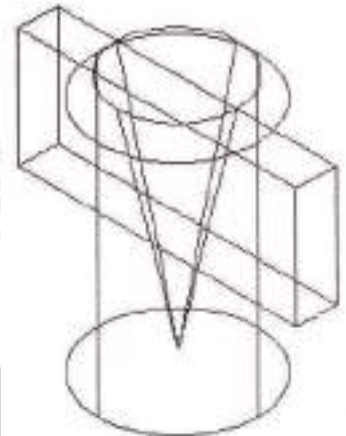
Perform the boolean operation. { A (cylinder) + B (box) - C (cone) }

Command: Union <Enter>
 Select objects: (left click on cylinder A) 1 found
 Select objects: (left click on box B) 1 found, 2 total
 Select objects: <Enter>

Command: subtract <Enter>
 Select solids and regions to subtract from. (left click on box of cylinder)
 Select objects: 1 found
 Select objects: <Enter>
 Select solids and regions to subtract.. (left click on cone)
 Select objects: 1 found
 Select objects: <Enter>

Command: **Hide** <Enter>

Your drawing should look like the figure shown:



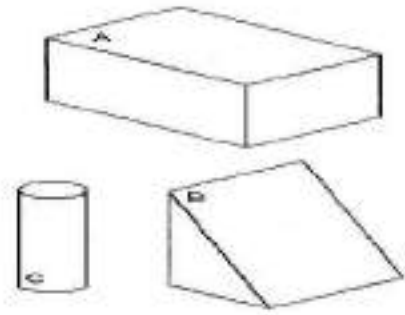
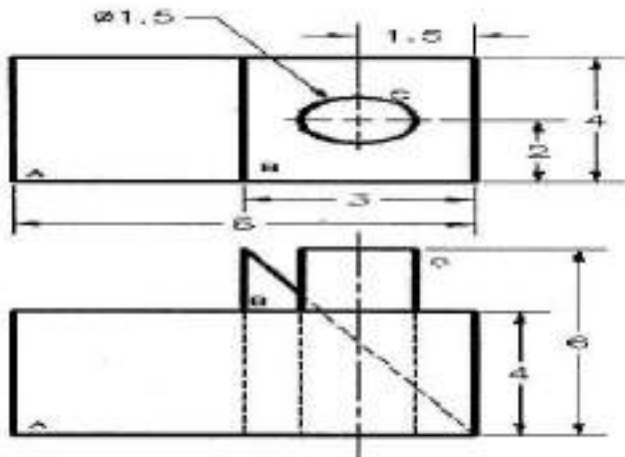
hide

Calculate the volume hide

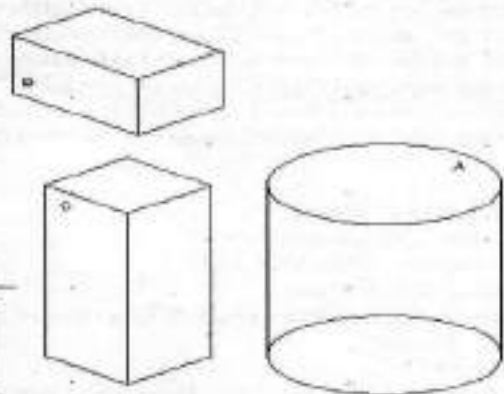
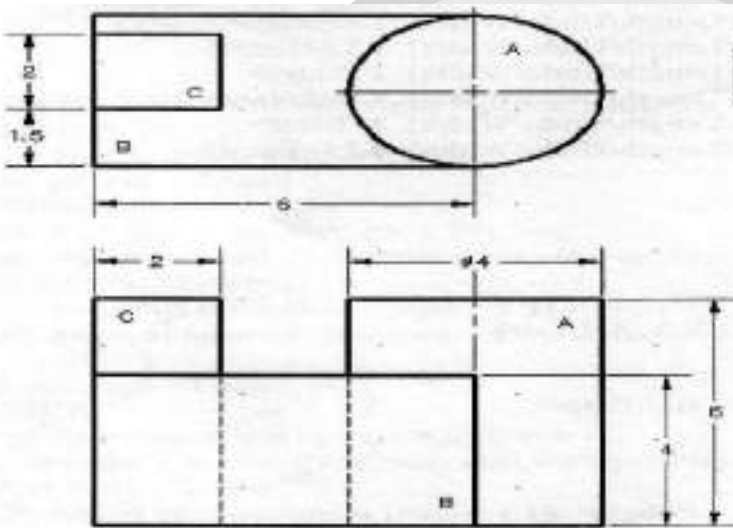
Command: Massprop <Enter>
 Select objects: (left click on the objects) 1
 Select objects: <Enter> **1.**

Mass: 9.2044
 Volume: 9.2044

Exercise 9-1 Perform the Boolean operations $\{(A + B) - C\}$, and calculate the Volume :



Exercise 9-2 Perform the Boolean operations $\{(A + B) - C\}$:



Lesson 10
Creating composite Solids
(Boolean Operation : INTERSECT)

2. Instruction for Boolean Operation Problem

Start AutoCAD and use default settings.

Command: `vpoint <Enter>`
then type 1, -1,1. This is the same as selecting "SE Isometric" from the pull down menu "view".

Select the FRONT UCS :

Command: `ucs <Enter>`

Current ucs name: *WORLD*

Specify origin of UCS or [Face/Named/OBject/Previous/View/World/X/Y/Z/ZAxis] <World> :

G <Enter>

Enter an option [Top/Bottom/Front/BAck/Left/Right] <Top>: **F <Enter>**

Set ORTHO to "ON" (F8), use PLOYLINE Command and draw the Front View :

Command: **pl** <Enter>

PLINE

Specify start point: {pick any point on screen, begin with upper left corner and draw clockwise}

Current line - width is 0.0000

Specify next point of [Arc/Half width/Length/Undo/Width]: < Ortho on> **1.5** <Enter>

Specify next point of [Arc/Close/Half width/Length/Undo/Width]: **1** <Enter>

Specify next point of [Arc/Close/Half width/Length/Undo/Width]: **1.13** <Enter>

Specify next point of [Arc/Close/Half width/Length/Undo/Width]: **1** <Enter>

Specify next point of [Arc/Close/Half width/Length/Undo/Width]: **1.5** <Enter>

Specify next point of [Arc/Close/Half width/Length/Undo/Width]: **2** <Enter>

Specify next point of [Arc/Close/Half width/Length/Undo/Width]: **4.13** <Enter> Close

Select the RIGHT UCS :

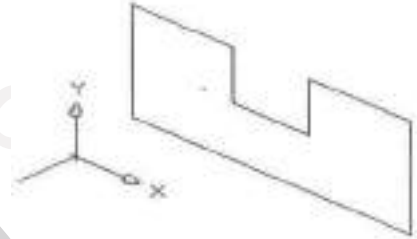
Command: **ucs** <Enter>

Current ucs name: *WORLD*

Specify origin of UCS or [Face/Namde/OBject/Previous/View/World/X/Y/Z/ZAxis]<World>: **G** <Enter>

Enter an option

[Top/Bottom/Front/BAck/Left/Right/] <Top>: **R** <Enter>



Make sure the ORTHO is "ON" (F8), use PLOYLINE Command and draw the Side view

Command: **pl** <Enter>

PLINE

Specify start point: {pick any point on screen, begin with upper left corner and draw clockwise}

Current line - width is 0.0000

Specify next point or [Arc/Half width/Length/Undo/Width]: < Ortho on> **2.25** <Enter>

Specify next point or [Arc/Close/Half width/Length/Undo/Width]: **2** <Enter>

Specify next point or [Arc/Close/Half width/Length/Undo/Width]: **.5** <Enter>

Specify next point or [Arc/Close/Half width/Length/Undo/Width]: **1.25** <Enter>

Specify next point or [Arc/Close/Half width/Length/Undo/Width]: **1.25** <Enter>

Specify next point or [Arc/Close/Half width/Length/Undo/Width]: **1.25** <Enter>

Specify next point or [Arc/Close/Half width/Length/Undo/Width]: **.5** <Enter>

Type "c" <Enter> to Close

EXTRUDE both front and right figures, with extrusion height of extrude 2.25 and 4.13 respectively :

Command: **ext** <Enter>

EXTRUDE

Current wire frame density: ISOLINES=4

Select objects: 1 found {Select the front view profile}

Select objects: <Enter>

Specify height of extrusion or [Path]: **2.25** <Enter>

Specify angle of taper for extrusion <0>: <Enter>

Command: **ext** <Enter>

EXTRUDE

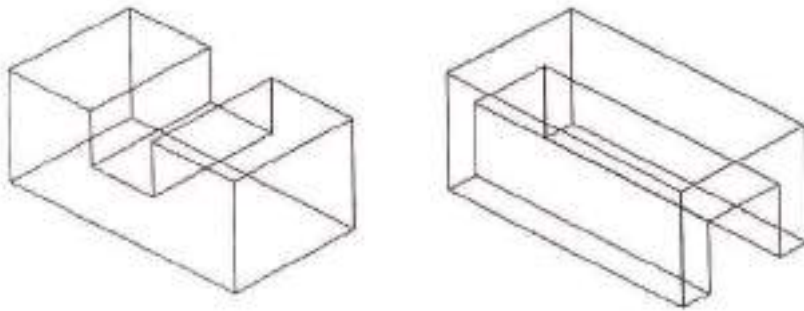
Current wire frame density: ISOLINES=4

Select objects: 1 found {Select the side view profile}

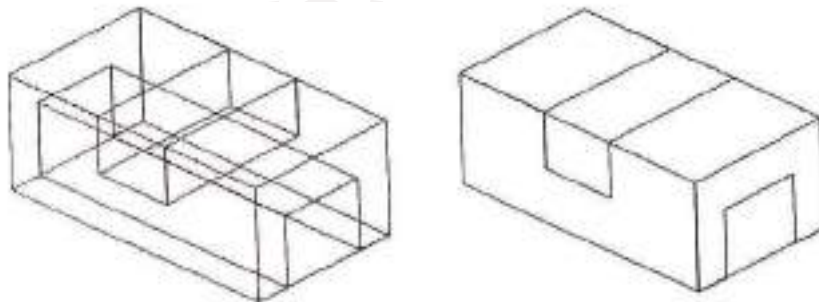
Select objects: <Enter>

Specify height of extrusion or [Path]: **4.13** <Enter>

Specify angle of taper for extrusion <0>: <Enter>



Move these two solid objects to the position as shown :

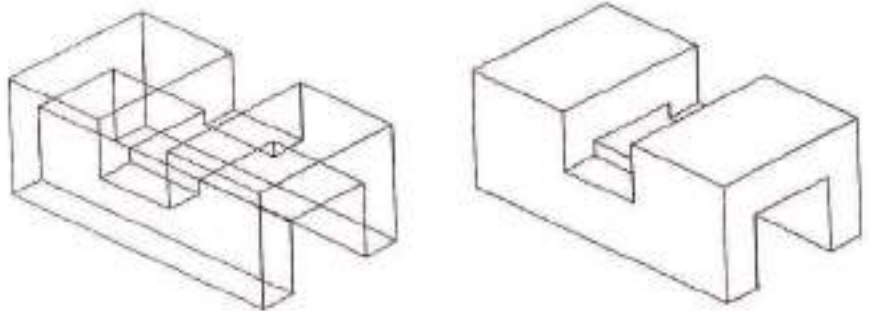


before hide

after hide

USE INTERSECT COMMAND :

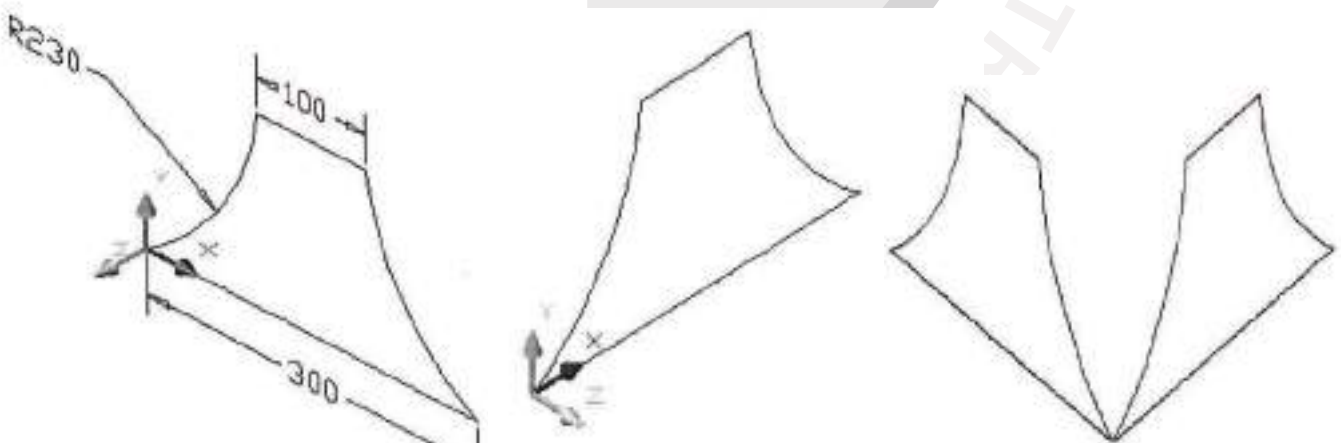
Command: **intersect** <Enter>
 Select objects: 1 found
 { Select both solid objects)
 Select objects: 1 found, 2 total
 Select objects: <Enter>



before hide

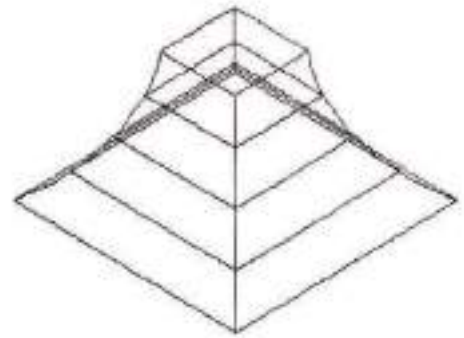
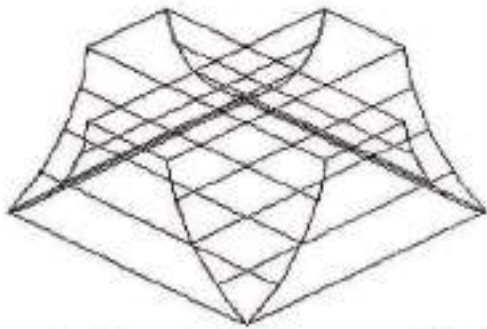
after hide

EXRCISE 10-1



1. Draw this figure on the front UCS.

2. Switch to the right UCS and draw the same figure.



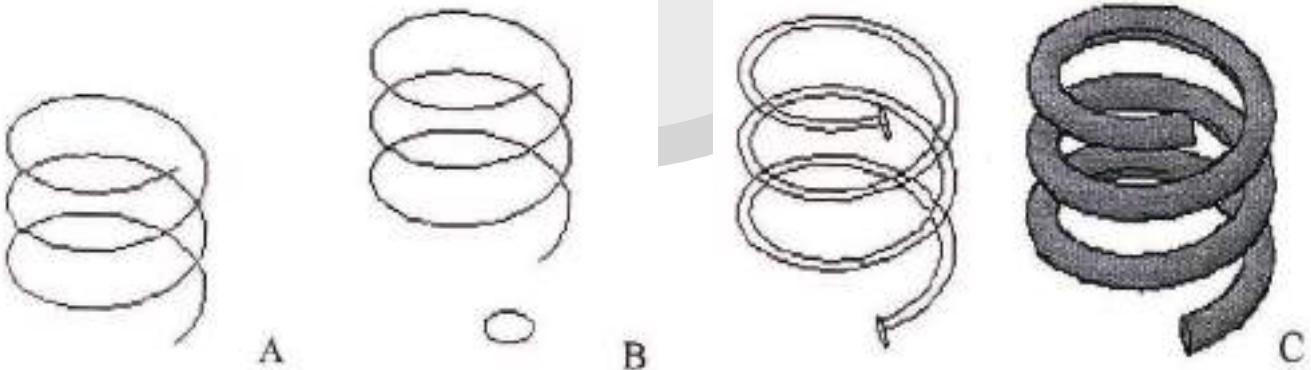
3. *Extrude both figures with 3" extrusion height.*

Lesson 11 Creating Helix



Construct a solid model of a spring. Create a centerline as shown on the following figures. Use this centerline as a path to extrude a circle through.

Command: **helix** <Enter>
 Number of turns = 3.0000 Twist = CCW
 Specify center point of base: {pick a point on screen}
 Specify base radius or [Diameter] <1.0000> : **.5** <Enter>
 Specify top radius or [Diameter] <1.0000> : **.5** <Enter>
 Specify helix height or [Axis endpoint / Turns / turn Height / Wist] <1.0000> : <Enter> (Figure A)
 Draw a circle with 0.125 diameter: (Figure B)
 Command: **_sweep** <Enter>



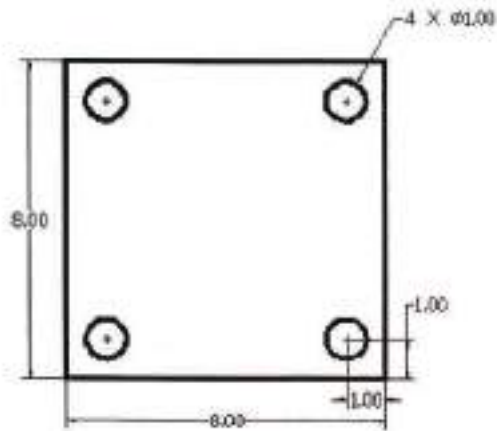
Current wire frame density: ISOLINES = 4
 Select objects to sweep: {pick the circle} 1 found
 Select objects to sweep: <Enter>
 Select objects sweep path or [Alignment / Base point / Scale / Twist]: {pick the helix}

Lesson 12

Creating 2 D Region Model



Construct the following 2D shape and convert it to 2D Region Model:



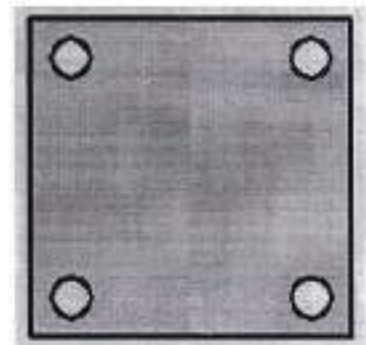
Command: **_region** <Enter>
 Select objects: 1 found
 {Select the square and four circles}
 Select objects: 1 found, 2 total
 Select objects: 1 found, 3 total
 Select objects: 1 found, 4 total
 Select objects: 1 found, 5 total
 Select objects: <Enter>
 5 loops extracted
 5 Regions created

View Conceptual:

Command: **_Vscurrent**
 Enter an option [2 dwireframe / 3dHidden / Realistic / Conceptual / Other]
 <2 dwireframe> : **_C**

Command: **subtract** <Enter>
 Select solids and regions to subtract from..
 Select objects: 1 found { select the square }
 Select objects: <Enter>

Select solids and regions to subtract..
 Select objects: { select the four circles } 1 found
 Select objects: 1 found, 2 total
 Select objects: 1 found, 3 total
 Select objects: 1 found, 4 total
 Select objects: <Enter>

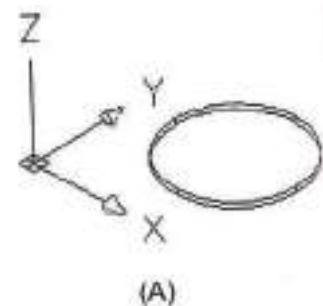


Calculate the area of the region.

Lesson 13

3D Coordinates

Command: **cylinder** <Enter>
 Specify center point of base or [3P/2P/Ttr/Elliptical]: **10,10** <Enter>
 Specify base radius or [Diameter] <6.0000>: **6** <Enter>
 Specify radius or [2point /Axis endpoint] <-3.0000>: **.5** <Enter>



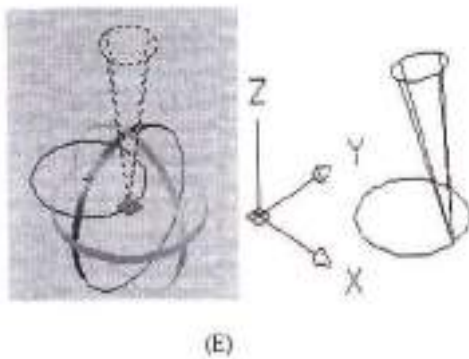
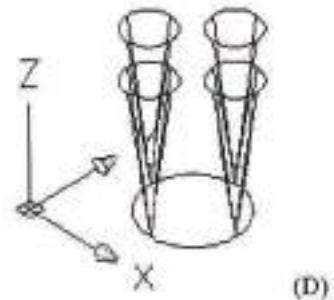
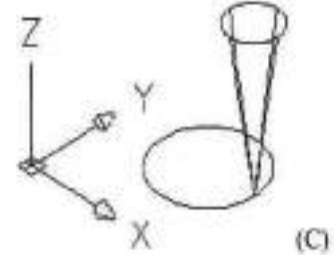
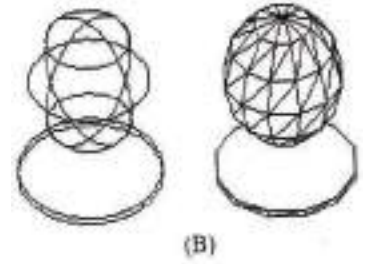
Command: **sphere** <Enter>
 Specify center point or [3P/2P/Ttr]: **10,10,8** <Enter>
 Specify radius or [Diameter] <6.0000>: **5** <Enter>

Command: **c** <Enter>
 CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
10,10,0.5 <Enter>
 Specify radius of circle of [Diameter] <6.0000>: **2** <Enter>

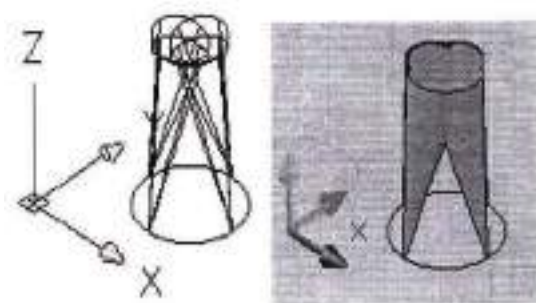
Command: **cone** <Enter>
 Specify center point of base or [3P/2P/Ttr/ Elliptical]: **10,12,3.5** <Enter>
 Specify base radius or [Diameter] <0.5000>: **.5** <Enter>
 Specify height or [2Point /Axis endpoint/Top radius] <-3.0000>: **-3** <Enter>

Copy the cone to each to each quadrant of 4" diameter circle. (Figure D)

Rotate each cone 15 from the vertical axis. (Figure E)

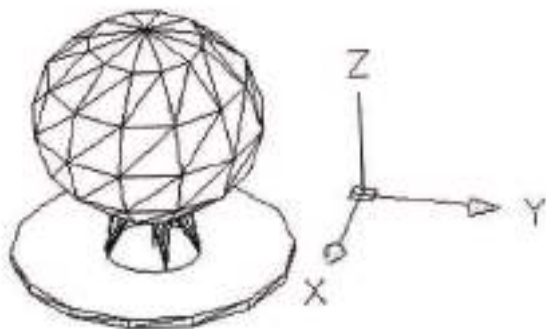


Command: **_3drotate** <Enter>
 Current positive angle in UCS :
 ANGDIR = counter lockwise ANGBASE = 0
 Select objects: { select the cone } 1 found
 Select objects: <Enter>
 Specify base point: { snap to the endpoint } : Figure E
 Pick a rotation axis: { left click on the green circle to rotate along Y axis }
 Specify angle start point: 15 <Enter> Regenerating model.



Command: **vpoint** <Enter>

Current view direction: VIEWDIR=1.0000, -1.0000, 1.10000
 Specify a view point or [Rotate] <display compass and tripod>: **R** <enter>
 Enter angle in XY plane from X axis <315>: 15 <Enter>
 Enter angle from XY plane <35>: <Enter>
 (Figure F)



Lesson 14

Cylindrical Coordinates

Command: **cylinder** <Enter>
 Specify center point of base or [3P/2P/Ttr/Elliptical]:
{select any point on screen}
 Specify base radius or [Diameter] <0.5000>: **2.5** <Enter>
 Specify height or [2Point/Axis endpoint]<0.0000>: **-7.5** <Enter>

Command: <Enter>
 CYLINDER
 Specify center point of base of base or [3P/2P/Ttr/Elliptical]:
{snap to center of the upper cylinder}
 Specify base radius or [Diameter] <2.5000>: **3** <Enter>
 Specify height or [2Point /Axis endpoint]<-7.5000>: **-8** <Enter>

(Figure A)

Command: **ucs** <Enter>
 Current ucs name: *WORLD*
 Specify origin of UCS or
 [Face /NAmEd/OBject/Previous/View/World/X/Y/Z/ZAxis]
 <World>: **o** <Enter>
 Specify new origin point <0,0,0>: *{Snap to center of the base}*

Change the UCS (User Coordinate System) position from "C" to "D".

Command: **ucs** <Enter>
 Current ucs name: *NO NAME*
 Specify origin of UCS or [Face/NAmEd/OBject/Previous/View/World/X/Y/Z/ZAxis]
 <World>: **n** <Enter>

Specify origin of new UCS or [ZAxis/3point /OBject/Face/View/ X/Y/Z] <0,0,0>: **3** <Enter>

Specify new **origin** point <0,0,0>: **3,0,1** <Enter>

Specify point on positive portion of **X**-axis <4.0000,0.0000,1.0000>: @ **3<90** <Enter>

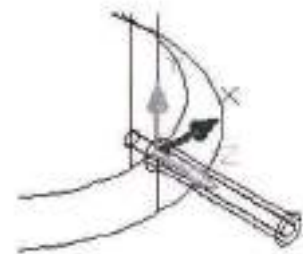
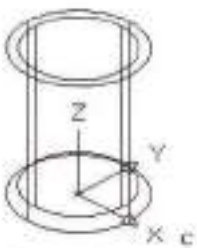
Specify point on positive-**Y** portion of the UCS XY plane <2.0000,0.0000,1.0000>: @ **0,0,3** <Enter>



(A)



(B)



E

Command: **cylinder** <Enter>
 Specify center point of base or [3P/2P/Ttr/Elliptical]: **0,0** <Enter>
 Specify base radius or [Diameter] <0.00>: **.25** <Enter>
 Specify height or [2Point/Axis/endpoint]<5.000>: **2.5** <Enter>

Command: cylinder <Enter>
 Specify center point of base or [3P/2P/Ttr/ Elliptical]:
 {snap to the center of the .25" diameter cylinder}
 Specify base radius or [Diameter] <0.25000>: .125 <Enter>
 Specify height or [2Point/Axis endpoint] <2.5000>: -3 <Enter>



(Figure E)

Subtract these two cylinder to create .125 diameter tube. (Figure F)

Text and Dimensions in 3D

- Creating Text with Thickness
- Dimensioning in 3D

3. Exercise 14-1

Construct the following 3D solid model, add a single TEXT with default height and thickness of .125. Complete the dimensions as shown. (Figure 1)
 You need to switch the work plane as necessary, using UCS "3point" or "Face" option.

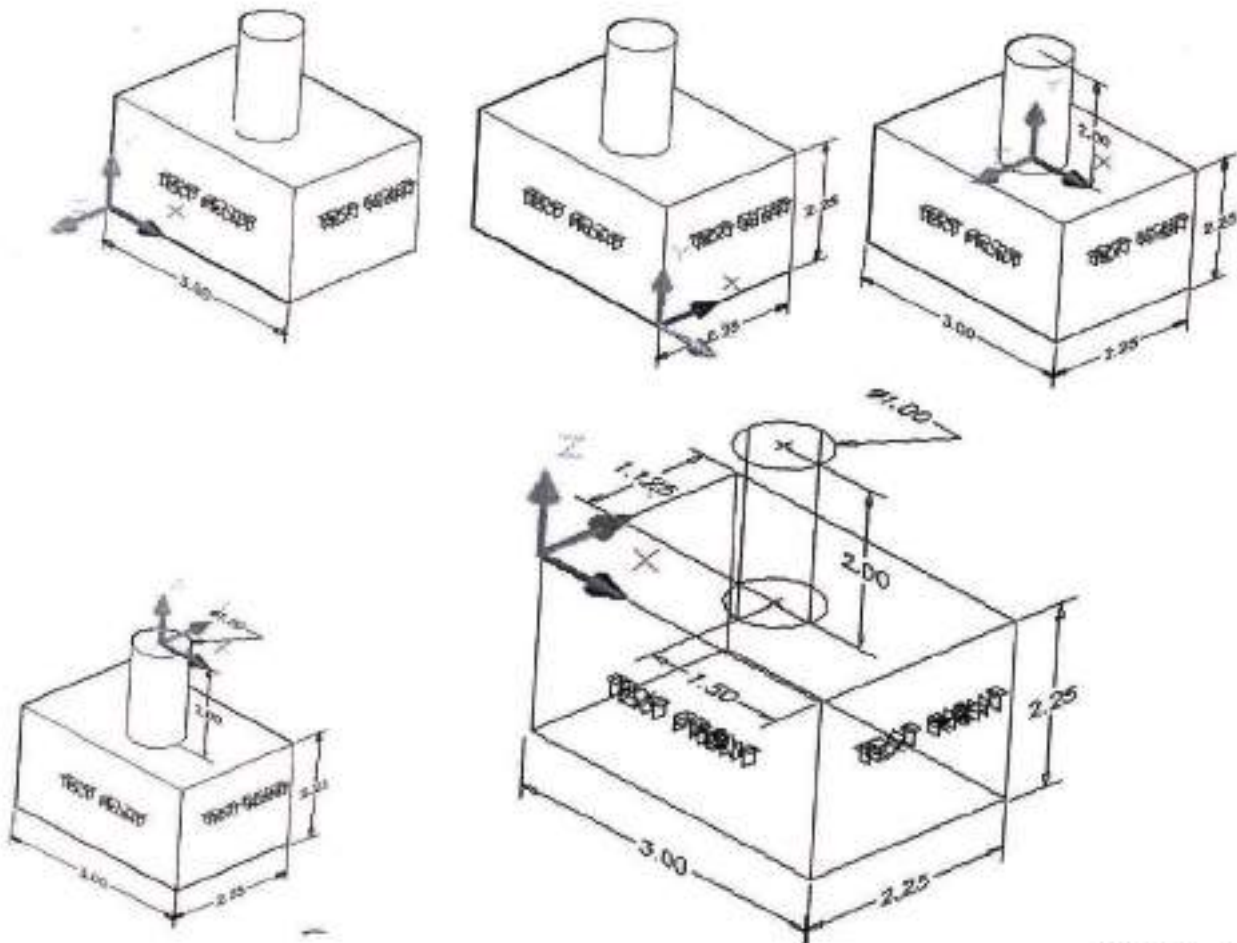






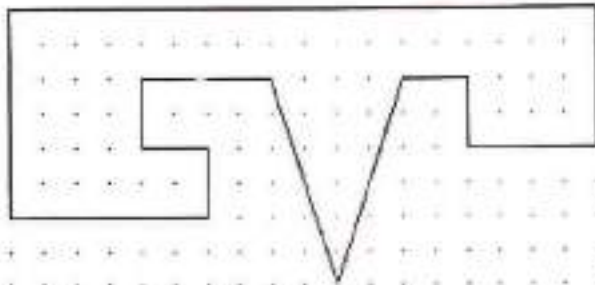
Figure -1

Lesson 15

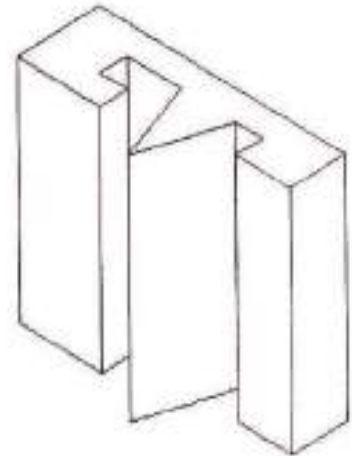
Extrude

-  Modeling toolbar:
-  Draw menu: Modeling > Extrude
-  Command entry: **extrude**
-  Dashboard: 3D Make panel, Extrude

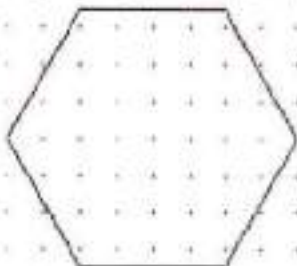
1. Start the AutoCAD and use the default setting.
2. Construct the 2D drawing as shown below to form a closed polyline.
3. Select the SE Isometric view.
4. Extrude (4") to form the solid object.
5. Calculate the volume.



GRID SPACE = 0.25



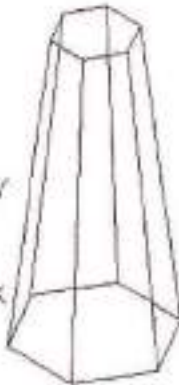
6. Start the AutoCAD and use the default setting.
7. Construct the 2D polygon as shown below.
8. Select the SE Isometric view.
9. Extrude to form the solid object. Height=4, Taper=6
10. Calculate the volume.



Grid space = 0.25



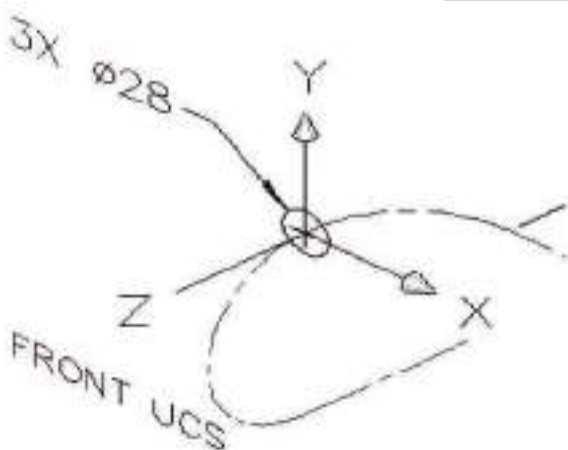
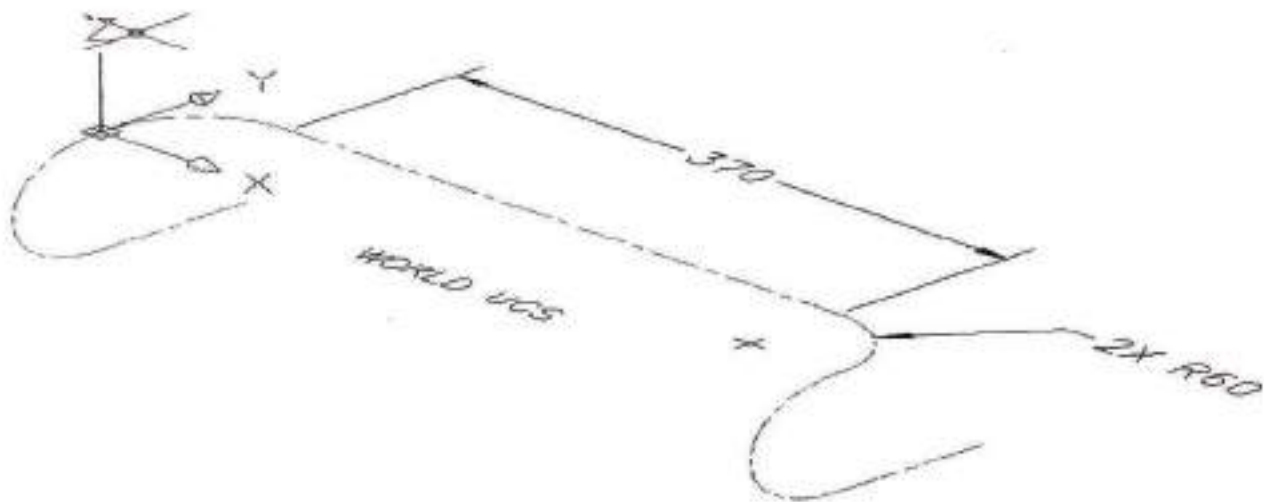
Wireframe



Hide

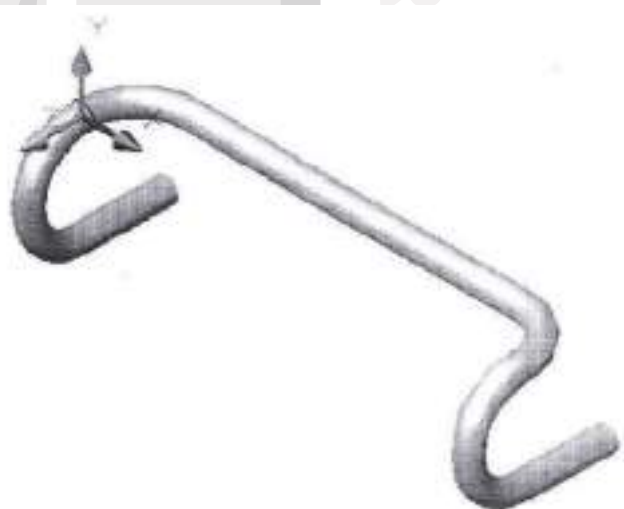
EXTRUDE (Path)

Construct a solid model of a bicycle handle bar. Create a centerline as shown on the following figures. Use this centerline as a path to extrude a circle through. Three mutually perpendicular coordinate systems are required: the WORLD, the SIDE and the FRONT. The centerline path consists of three separate **plines**.



1. Pline with 370 length pline (arc option) or ARC with 60 radii. On each end on WCS.
2. Create three circles on the FRONT UCS.
3. Create the drop portion of the bar using the pline command on the SIDE UCS.
4. Extrude each circle along each pline.

Plot the drawing using Shadeplot option as shown. Make sure the color set to "gray # 9) on your object layer. Save your project.

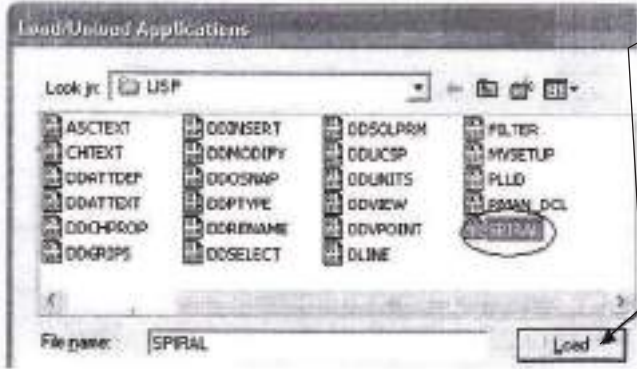


EXTRUDE (path)

Construct a solid model of a spring. Create a centerline as shown on the following figures. Use this centerline as a path to extrude a circle through. To create this path you must use 3D polyline command and define a path with the 3D coordinate system. Easier alternative would be using a LISP file. This file must be loaded before it can be used. Follow the instruction below to load and use this file:

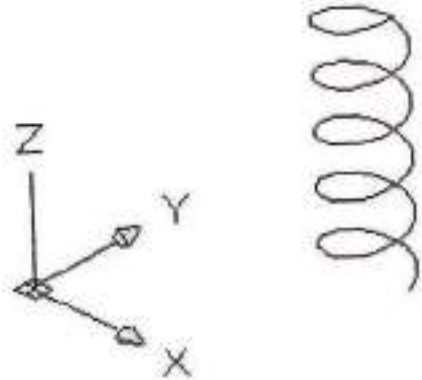
Note: In AutoCAD version 2008, you may use Helix command (lesson 11), use data on step 8 and skip to step 9.

5. From the "Tool" pull down menu select "Load Application..".
From the server (G: drive), student access, "symbols-title". under the "LISP" folder search for the file "Spiral".



6. Highlight "SPIRAL" then click on "load" tab
7. From View, Select SE Isometric :
8. On Command line type 3d spiral, and follow the step below :

Command: 3dspir **<Enter>**
 Center point: **<pick a point>**
 Number of rotations: **5 <Enter>**
 Starting radius: **.5 <Enter>**
 Horizontal growth per rotation: **0 <Enter>**
 Vertical growth per rotation: **.75 <Enter>**
 Points per rotation **<30>**: **<Enter>**
 Command:



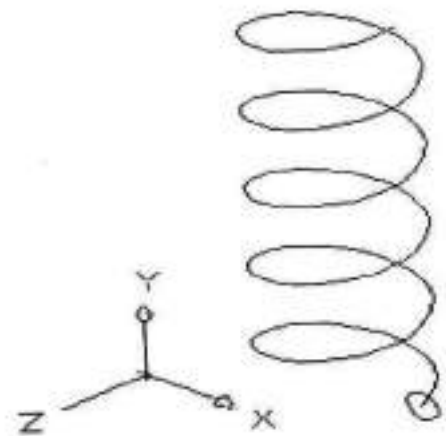
Your drawing should look like the figure

9. At this time you need to change you UCS to the FRONT view and draw a circle with center point at **one end of the path** and radius of **0.125** (See the figure).

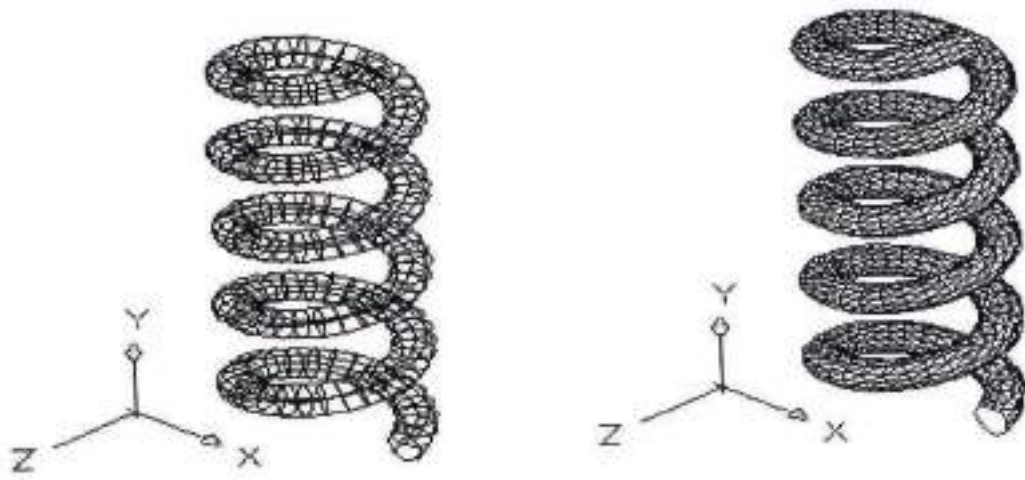
Command: ucs **<Enter>**
 Current ucs name: *WORLD*
 Enter an option
 [New / Move / orthoGraphic / Prev / Restore / Save / Del / Apply / ? / World] **<world>**: **g <Enter>**
 Enter an option [Top / Bottom / Front / BAcK / Left / Right] **<Top>**: **f <Enter>**

Command: c **<Enter>**
 CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: {snap to the end point of the path}
 Specify radius of circle of [Diameter]: .125 **<Enter>**

On the FRONT UCS draw diameter 0.25



10. Extrude the 0.25 diameter circle along the spiral path.
 11. Determine the volume and print your file. (refer to printing 3D handout)
 12. Save your drawing.



Exercise 15-1

Construct the 3D solid model shown on Figure 15-1

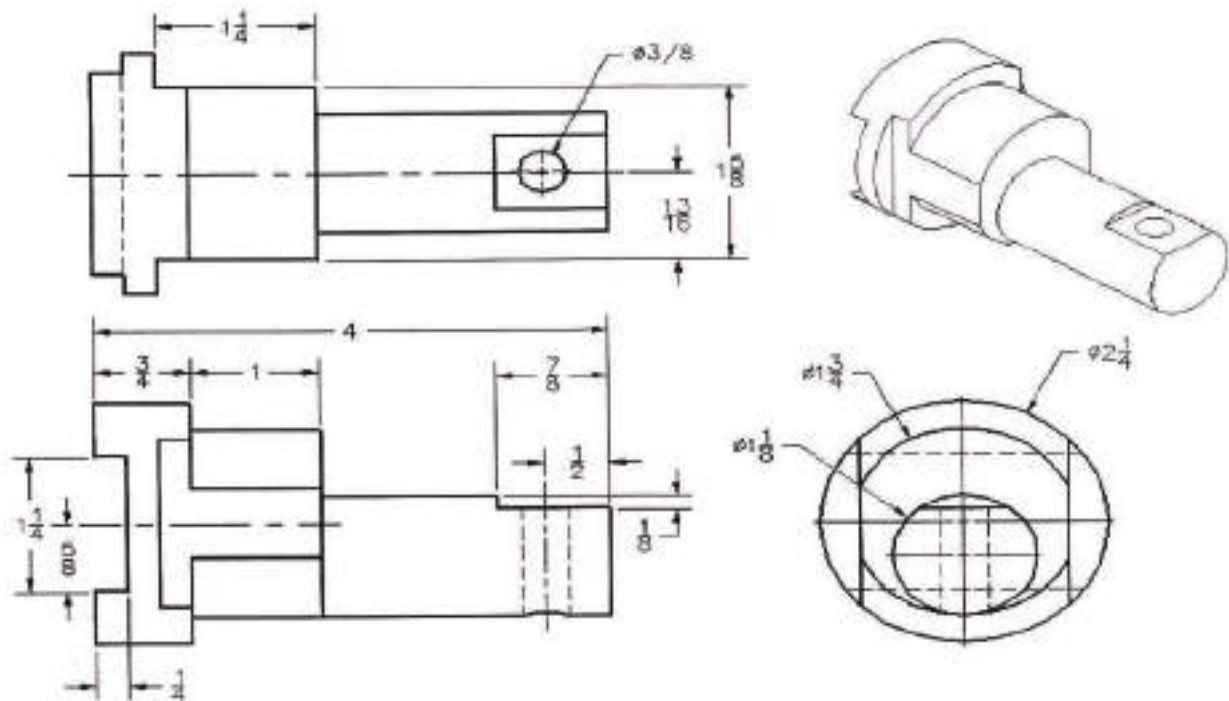


Figure 15-1

Tutorial Exercise 15-2

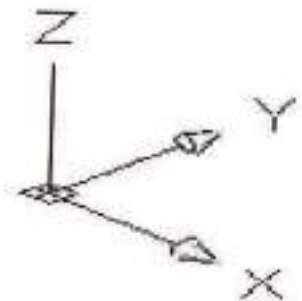
Start AutoCAD with the default settings.

Use the VPOINT Command to change the viewing option:

Command: vpoint <Enter>

Current view direction: VIEWDIR=0.0000,0.0000,1.0000

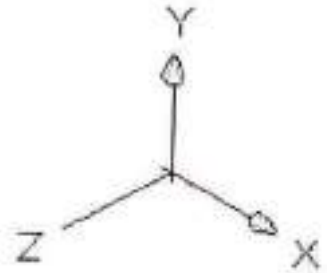
Specify a view point or [Rotate] <display compass and tripod>: 1,-1,1 <Enter> Regenerating model.



This is the same as selecting "SE Isometric" from the pull down menu "view".

Change UCS to have XY plane to be parallel to front view :

Command: **ucs** <Enter>
Current ucs name: *WORLD*
Specify origin of UCS or [Face/Named/Object/Previous/View/World/X/
Y/Z/Zaxis] <World>: **x** <Enter>
Specify rotation angle about X axis <90>: <Enter>



Use POLYLINE command and draw the figure 6-1:

Command: **pL** <Enter>
PLINE
Specify start point: 5,5 <Enter>
Current line-width is 0.0000
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: @3,0 <Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: @.5<90 <Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: @-1,.5 <Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: @-1,.5 <Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: @.5<90 <Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: @3<180 <Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: c <Enter>

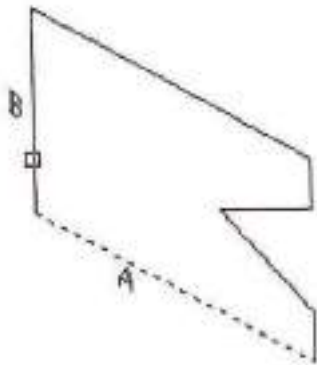


Figure 6-1



Figure 6-2

Command: **cha**
CHAMFER
(TRIM mode) Current chamfer Dist 1 = 0.0000
Select first line or [Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple]: **d**
Specify first chamfer distance <0.0000>: **.5** <Enter>
Specify second chamfer distance <0.5000>: <Enter>
Select first line or [Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple]:
{ left click on line "B" Figure 6-1 }
Select second line or shift-select to apply corner: {left click on line "B" Figure 6-1}
Your drawing should look like the one shown in figure 6-2.

Use POLYLINE command and draw the figure 6-3:

Command: **pL** <Enter>
PLINE
Specify start point: 2,2 <Enter>
Current line-width is 0.0000

Specify next point or [Arc/ Halfwidth/ Length/ Undo/ Width]: @ 3,0 <Enter>
 Specify next point or [Arc/ Close/ Halfwidth/ Length/ Undo/ Width]: @ .5 <90 <Enter>
 Specify next point or [Arc/ Close/ Halfwidth/ Length/ Undo/ Width]: @ 1,5 <Enter>
 Specify next point or [Arc/ Close/ Halfwidth/ Length/ Undo/ Width]: @ 1,5 <Enter>
 Specify next point or [Arc/ Close/ Halfwidth/ Length/ Undo/ Width]: @ .5 <90 <Enter>
 Specify next point or [Arc/ Close/ Halfwidth/ Length/ Undo/ Width]: @ 1 <180 <Enter>
 Specify next point or [Arc/ Close/ Halfwidth/ Length/ Undo/ Width]: @ .5 <270 <Enter>
 Specify next point or [Arc/ Close/ Halfwidth/ Length/ Undo/ Width]: @ -1.5,-.5 <Enter>
 Specify next point or [Arc/ Close/ Halfwidth/ Length/ Undo/ Width]: @ -.5,.5 <Enter>
 Specify next point or [Arc/ Close/ Halfwidth/ Length/ Undo/ Width]: @ -.5,.5 <Enter>
 Specify next point or [Arc/ Close/ Halfwidth/ Length/ Undo/ Width]: c <Enter>

Command: **cha** <Enter>

(TRIM mode) Current chamfer Dist 1 = 0.5000, Dist 2 = 0.5000

Select first line or [Polyline/ Distance/ Angle/ Method]: {left click on line "A" Figure 6-3}

Select second line: {left click on line "B" Figure 6-3}

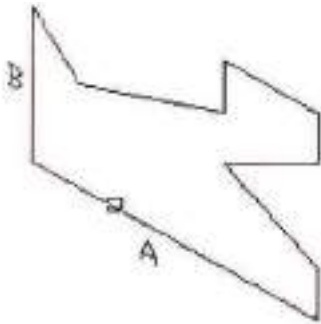


Figure 6-3

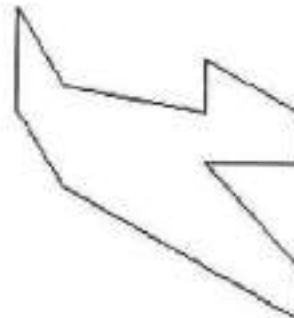


Figure 6-4

Use Extrude Command to create the model:

Command: **extrude** <Enter>

Current wire frame density: ISOLINES = 4

Select objects: {left click any place on profile Figure 6-1} 1 found

Select objects: {left click any place on profile Figure 6-3} 1 found, 2 total

Select objects: <Enter>

Specify height of extrusion or [path]: **.5** <Enter>

Specify angle of taper for extrusion <0>: <Enter>

Use MOVE Command with help of object snap to move these models to their proper positions:

Command: **m** <Enter>

MOVE

Select objects:

{left click any place on model "A" Figure 6-5} 1 found

Select objects: <Enter>

Specify base point or displacement:

end of {left click at point "1" part "A" Figure 6-5}

Specify second point of displacement or

<use first point as displacement>:

end of {left click at point "2" part "B" Figure 6-5}

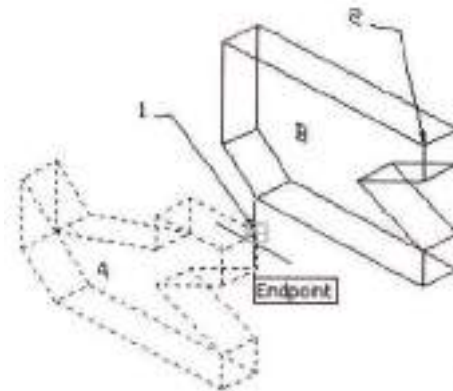


Figure 6-5

Command: **hide** <Enter> Regenerating model.

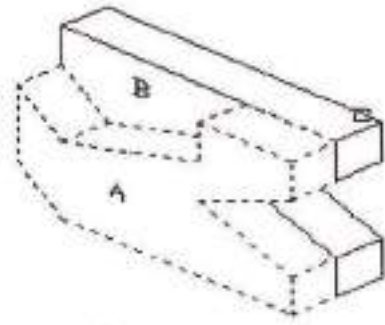


Figure 6-6

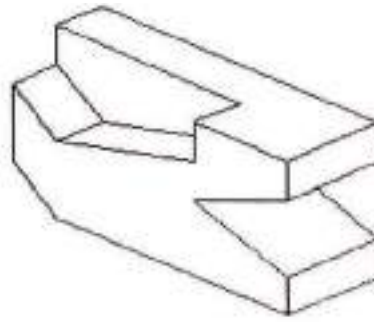


Figure 6-7

Command: **union** <Enter>

Select objects: {left click on part "A" Figure 6-6} 1 found

Select objects: {left click on part "B" Figure 6-6} 1 found, 2 total

Select objects: <Enter>

Your final drawing shown in figure 6-7

Use MASSPROP Command to calculate the volume.

Use TEXT Command and write the result on your final printed drawing.

Note: Refer to instruction on "Printing 3D Model".

4. Tutorial Exercise 15-3

Start AutoCAD with the default setting.

Use the VPOINT Command to change the viewing option:

Command: **vpoint** <Enter>

Current view direction:

VIEWDIR = 0.0000,0.0000,1.0000

Specify a view point or [Rotate] <display compass and tripod>: **1,-1,1**<Enter>

Regenerating model.

This is the same as selecting "SE Isometric" from the pull down menu "view".

Change UCS to have XY plane to be right side view:

Command: **ucs** <Enter>

Current ucs name: *WORLD*

Specify origin of UCS or [Face/Named/OBJECT/Previous/View/World/X/Y/Z/ZAxis] <World>: **x**<Enter>

Specify rotation angle about X axis <90>: <Enter>

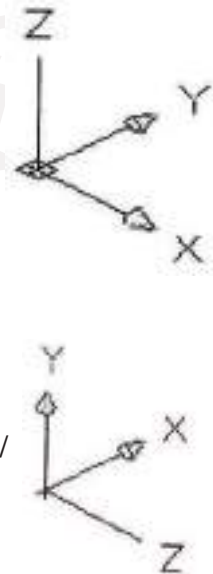
Command:

UCS

Current ucs name: *NO NAME*

Specify origin of UCS or [Face/Named/OBJECT/Previous/View/World/X/Y/Z/ZAxis] <World>: **y**<Enter>

Specify rotation angle about Y axis <90>: <Enter>



Use POLYLINE command and draw the figure 6-1:

Command: **pl** <Enter>
PLINE
Specify start point: **5,5** <Enter>
Current line-width is 0.0000
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **@ 1 <0** <Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **@ 2 <270** <Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **@ 2 <180** <Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **@ 1 <90** <Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **@ .5 <0** <Enter>

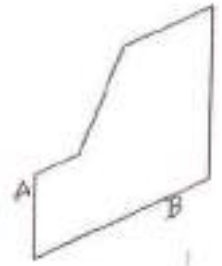


Figure1

Command: **cha** <Enter>
CHAMFER
(TRIM mode) Current chamfer Dist1 = 0.0000, Dist2 = 0.0000
Select first line or [Undo/Polyline/Distance/Angle/Trim/mEtho/Multiple]: **d** <Enter>
Specify first chamfer distance <0.0000>: **.5** <Enter>
Specify second chamfer distance <0.5000>: <Enter>
Select first line or [Undo/Polyline/Distance/Angle/Trim/mEtho/Multiple]:
{left click on line "A" Figure 6-1}
Select second line or shift-select to apply corner:
{left click on line "B" Figure 6-1}
Your drawing should look like the one shown in figure-2.

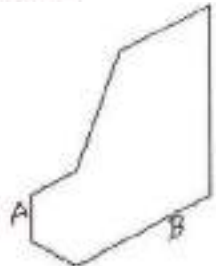


Figure2

Use POLYLINE & COPY commands and draw the figure-3:

Command: **pl** <Enter>
PLINE
Specify start point: {snap to the end point 1}
Current line-width is 0.0000
Specify next point or [Arc/Halfwidth/Length/Undo/Width]:
{snap to the end point 2}
Specify next point or [Arc/Halfwidth/Length/Undo/Width]:
{snap to the end point 3}
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **c** <Enter>

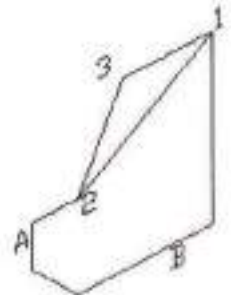


Figure-3

Move this profile to a new location (Figure 4)

Command: **m** <Enter>
MOVE
Select objects: **L** <Enter> 1 found
Select objects: <Enter>
Specify base point or [Displacement]<Displacement>: {snap to the end point 1}
Specify second point or <use first point as displacement>: **@ 0,0,-1** <Enter>

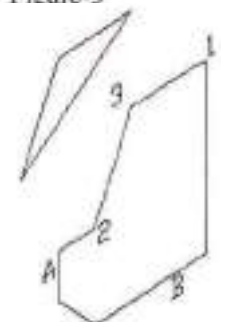
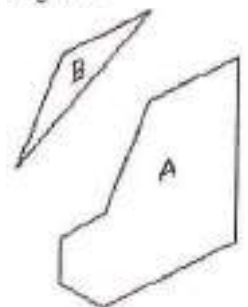


Figure-4

Use EXTRUDE Command, to create the model:

Command: **ext** <Enter>
EXTRUDE
Current wire frame density: ISOLINES = 4
Select objects to extrude: {select the profile "A"} 1 found
Select objects to extrude: <Enter>
Specify height of extrusion or [Direction/Path/Taper angle] <3.0000>: **-3** <Enter>



Command: **ext** <Enter>

EXTRUDE

Current wire frame density: ISOLINES=4

Select objects to extrude: {select the profile "B"} 1 found

Select objects to extrude: <Enter>

Specify height of extrusion or [Direction/Path/Taper angle] <3.0000>: **-1** <Enter> Figure 5

Command: **subtract** <Enter>

Select solids and regions to subtract from.. {left click on the base part A}

Select objects: 1 found

Select objects: <Enter>

Select solids and regions to subtract.. {left click on the small part B}

Select objects: 1 found

Select objects: <Enter> {Figure-7}

Command: **hide** <Enter> Regenerating model.

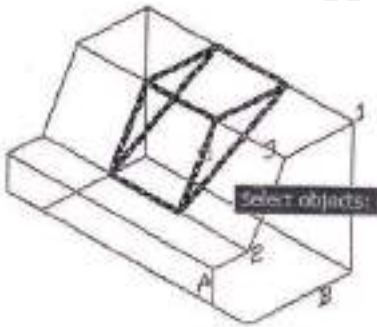


Figure 5

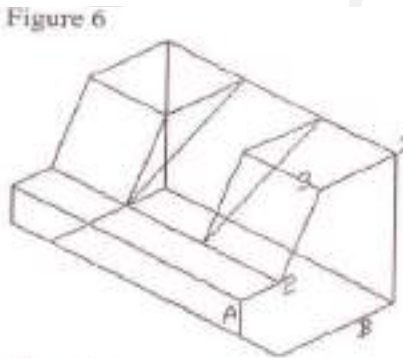


Figure 6

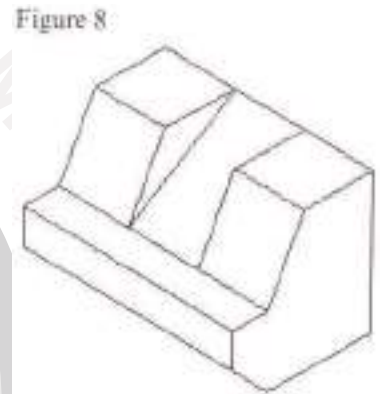


Figure 8

Use **MASSPROP** Command to calculate the volume.

Use **TEXT** Command and write the result on your final printed drawing.

Note: Refer to instruction on "Printing 3D Model".

Lesson 16 Revolve Command

Modeling toolbar:



Draw menu: Modeling > Revolve

Command entry: **revolve**

Dashboard: 3D Make panel, Revolve

Draw a rectangle as shown on figure 1:

Command: **pl** <Enter>

PLINE

Specify start point:

Current line - width is 0.0000

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **1.5** <Enter>

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **.625** <Enter>

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **1.5** <Enter>

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **c**

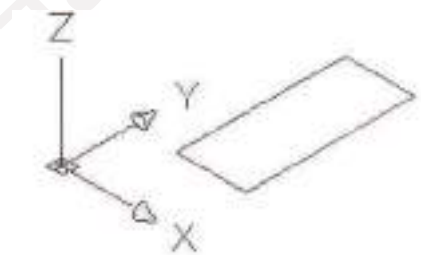


Figure 1

Draw the line "A" with an approximate length and distance of 1.5 from the rectangle "B". (Figure 2)

Command: **revolve** <Enter>
Current wire frame density: ISOLINES=4
Select objects to revolve: 1 found
Select objects to revolve: (click on the polyline B)
Specify axis start point of define axis by [Object/X/Y/Z] <Object>: **o** <Enter>
Select an object: (click on the line "A" towards end: figure 2)
Specify angle of revolution or [Start angle] <360>: **90** <Enter>

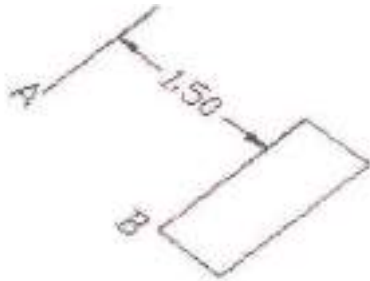


Figure 2



Figure 3



Exercise 16-1

Construct the 3D solid model shown on Figure 16-1. Use the following data & steps:

```
VPOINT 1, -1, 1  
CIRCLE Cntr 0,6 & R=1  
CIRCLE Cntr 0,6 & R=1.3125  
REVOLVE  
    (both circles 90* on X)  
SUBTRACT:  
    (both circles form each other)  
CIRCLE Cntr 0,6 & R=3  
CIRCLE Cntr 0,6 & R=1  
CIRCLE Cntr @ 2.375<90 & R=.3125  
ARRAY L; P; 4  
  
EXTRUDE:  
    (select all 6 circles ) height = .625  
SUBTRACT  
    (5 circles from a 3" Rad. circle)
```

```
COPY: L; 0,6; @  
ROTATE 3D: L; X; 0,0; 45  
COLOR: 2  
CIRCLE CNR 0,6,.625 & R=2  
ROTATE 3D: L; X; 0,0; 45  
COLOR: by layer
```

```
SLICE: (select pipe): Object;  
    { select yellow circle }  
UNION: All  
ROTATE 3D; L; z; 0,0; -60
```

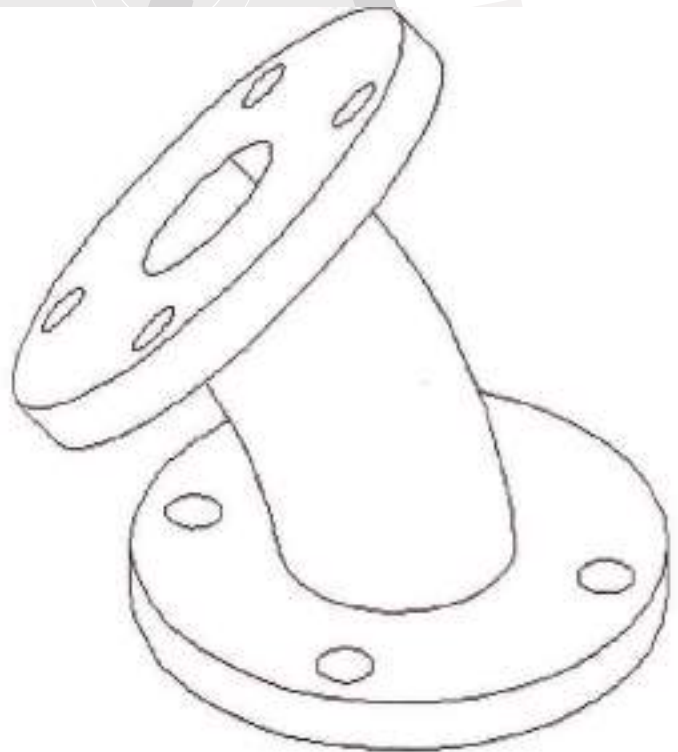


Figure 16-1

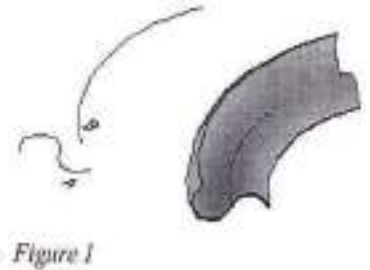
Lesson 17 Sweep Command



With the **SWEEP** command, you can create a new solid of surface by sweeping an open or closed planar curve (profile) along an open or closed 2D or 3D path. **SWEEP** draws a solid or surface in the shape of the specified profile along the specified path. You can sweep more than one object, but they all must lie on the same plane

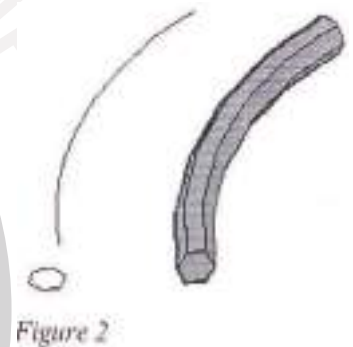
- a. Sweeping the open shape profile creates a SURFACE.**
Draw the figure similar to A & B (Figure 1), using **ARC** or **POLYLINE**.

Command: **sweep** <Enter>
Current wire frame density: ISOLINES=4
Select objects to sweep: {select the arc A} 1 found
Select objects to sweep: <Enter>
Select sweep path or [Alignment / Base / point / Scale / Twist] :
{select the arc B}



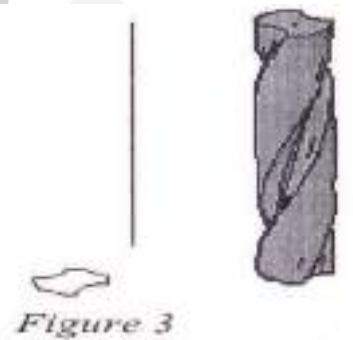
- b. Sweeping the close shape profile creates a SOLID.**
Using **POLYGON** & **ARC**, draw the objects similar to Figure 2

Command: **sweep** <Enter>
Current wire frame density: ISOLINES=4
Select objects to sweep: {select the polygon} 1 found
Select objects to sweep: <Enter>
Select sweep path or [Alignment / Base point / Scale / Twist] :
{select the arc}



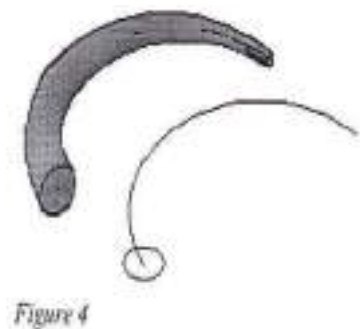
- c. Sweeping the close shape profile with Twist option (Figure 3).**

Command: **sweep** <Enter>
Current wire frame density: ISOLINES=4
Select objects to sweep: {select the profile} 1 found
Select objects to sweep: <Enter>
Select sweep path or [Alignment / Base point / Scale / Twist] : **T** <Enter>
Enter twist angle or allow banking for a non-planar sweep path [Bank]
<0.0000>: **270** <Enter>
Select sweep path or [Alignment / Base point / Scale / Twist] :
{select the line} (Figure 5)



- d. Sweeping the close shape profile with Scale option (Figure 4).**

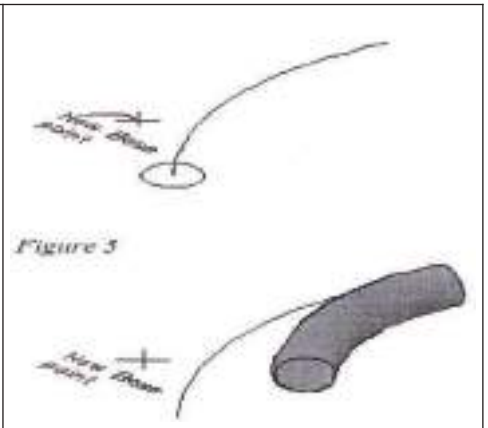
Command: **sweep** <Enter>
Current wire frame density: ISOLINES=4
Select objects to sweep: {select the circle} 1 found
Select objects to sweep: <Enter>
Select sweep path or [Alignment / Base point / Scale / Twist] : **S** <Enter>
Enter scale factor or [Reference] <1.0000>: **.25** <Enter>
Select sweep path or [Alignment / Base point / Scale / Twist] :
{select the arc} (Figure 5)



e. Sweeping the close shape profile with Base point option (Figure 4).

Command: sweep <Enter>
 Current wire frame density: ISOLINES=4
 Select objects to sweep: {select the circle} 1 found
 Select objects to sweep: <Enter>
 Select sweep path or [Alignment /Base point/Scale/Twist]:
 B <Enter>
 Select sweep path or [Alignment /Base point/Scale/Twist]:
 {select the arc}

(Figure 5)



Lesson 18
Loft Command

With the LOFT command, you can create a new solid or surface by specifying a series of cross sections. The cross sections define the profile the (shape) of the resulting solid or surface. Cross sections (generally, curves or lines) can be open (for example, an arc) or closed (for example, a circle). LOFT draws a solid or surface in the space between the cross sections. You must specify at least two cross sections when you use the LOFT command.

- Modeling toolbar: 
- Draw menu: Modeling > Loft
- Command entry: loft
- Dashboard: 3D Make panel, Loft

a. Controlling the Loft with **Cross Sections**:

Command: **c** <Enter>
 CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:
 Specify radius of circle or [Diameter] <1.5000>: **1.5** <Enter>

Command:
 CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:@ **0,0,3** <Enter>
 Specify radius of circle or [Diameter] <1.5000>: **.5** <Enter>

Command: **polygon** <Enter>
 Enter number of sides <4>: **5** <Enter>
 Specify center of polygon or[Edge]: @**0,0,2** <Enter>
 Specify center of polygon or [Inscribed in circle/Circumscribed about circle] <I>: <Enter>
 Specify radius of circle: **1** <Enter> (Figure 1)

Command: loft <Enter>
 Select cross-sections in lofting order: specify opposite corner:
 Select cross-sections in lofting order:
 {select the large circle} 1 found
 Select cross-sections in lofting order: {select the large circle}
 1 found, 2 total
 Select cross-sections in lofting order: {select the large circle}
 1 found, 3 total
 Select cross-sections in lofting order: <Enter>
 Enter an option [Guides/Path/Cross-sections only] <Cross-sections only>: <Enter> (Figure 2)



b. Controlling the Loft with Guide Curves :

Command : **loft** <Enter>

Select cross -sections in lofting order: {select the circle} 1 found

Select cross -sections in lofting order: {select the circle}

1 found,2 total

Select cross -sections in lofting order: <Enter>

Enter an option [Guides / Path / Cross - sections only] < Cross - sections only > : **g** <Enter>

Select guide curves: {select the curve} 1 found

Select guide curves: {select the curve} 1 found, 2 total

Select guide curves: {select the curve} 1 found, 3 total

Select guide curves: {select the curve} 1 found, 4 total

Select guide curves: {select the curve} 1 found, 5 total

Select guide curves: <Enter>

c. Controlling the Loft with a Path :

Draw the Figure 5, using the Helix for the path with an approximate size. Use the Circle, Polygon, and Point command to draw cross-sections for the loft. Make sure the path curve is intersecting with all the cross sections objects.

Command : **loft** <Enter>

Select cross -sections in lofting order: {select the end circle} 1 found

Select cross -sections in lofting order: {select the end circle} 1 found, 2 total

Select cross -sections in lofting order: {select the end circle} 1 found, 3 total

Select cross -sections in lofting order: {select the end circle} 1 found, 4 total

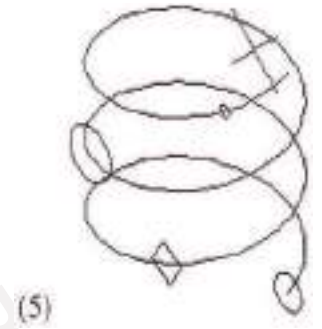
Select cross -sections in lofting order: {select the end circle} 1 found, 5 total

Select cross -sections in lofting order: <Enter>

Enter an option [Guide / Path / Cross - sections only] < Cross - Sections only > :

p <Enter>

Select path curve: {select the helix}



Lesson 19

3dalign Command



You can specify either one, two or three points for the source object. Then, you can specify one, two, or three points for the destination. The selected object is moved and rotated so that base points, and the X and the Y axes of the source and destination align in 3D space. 3DALIGN works with dynamic UCS (DUCS), so you can dynamically drag the selected objects and align them with the face of a solid object.

Command : **3dalign** <Enter>

Select objects: {select box A} 1 found

Select objects: <Enter>

Specify source plane and orientation...

Specify base point or [Copy]: {pick endpoint 1}

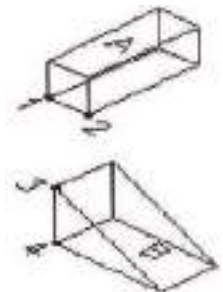
Specify second point or [Continue] <C>: {pick endpoint 2}

Specify third point or [Continue] <C>: <Enter>

Specify destination plane and orientation...

Specify first destination point: {pick endpoint 5}

Specify Second destination point or [eXit] <X>: {pick endpoint 6}



Specify third destination point or [eXit] <X>: <Enter>

Command: <Enter>

3DALIGN

Select objects: {select the wedge B} 1 found

Select objects: <Enter>

Specify source plane and orientation...

Specify base point or [Copy]: {pick endpoint 3}

Specify second point or [Continue] <C>: {pick endpoint 4}

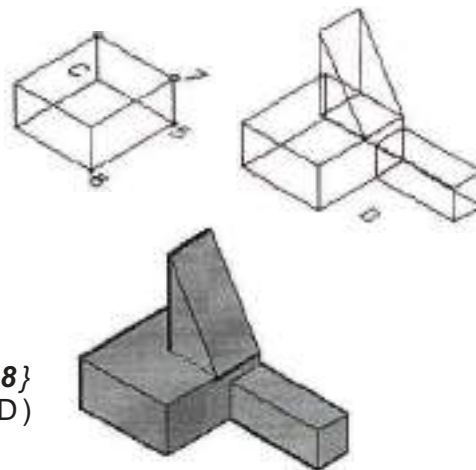
Specify third point or [Continue] <C>: <Enter>

Specify destination plane and orientation...

Specify first destination point: {pick endpoint 7}

Specify Second destination point or [eXit] <X>: {pick endpoint 8}

Specify third destination point or [eXit] <X>: <Enter> (Figure D)



Lesson 20

Mirror3D - 3dAarray

a. Draw a box and a wedge with an approximate size as shown in figure 1.

Command: _mirror3d <Enter>

Select objects: {select the wedge} 1 found

Select objects: <Enter>

Specify first point of mirror plane (3 points) or [Object/Last/Zaxis/View/XY/YZ/ZX/3points]<3point>: <Enter>

Specify first point on mirror plane: {select the modepoint 1}

Specify second point on mirror plane: {select the modepoint 2}

Specify third point on mirror plane: {select the modepoint 3}

Delete source objects? [Yes/No] <N>: N <Enter>

(Figure 2)

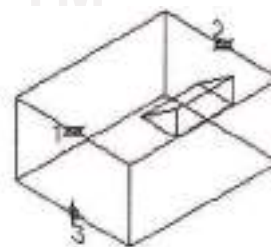


Figure 1

b. Create two boxes A & B and move to the position as shown in figure C. (you may use the 3dalign or move command).

c. Use the array or copy command to construct the figure D.

d. Use 3DARRAY to create the figure E.

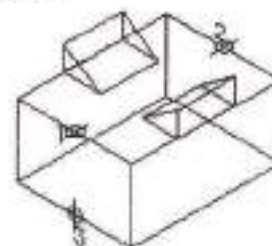
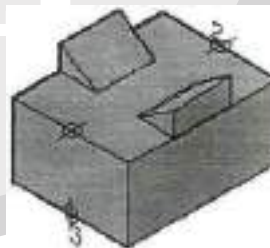


Figure 2



Note: Use SE Isometric view



Command: **BOX** <Enter>

Specify first corner or [Center]: {pick a point onscreen}

Specify other corner or [Cube/Length]: **L** <Enter>

Specify length: <Ortho on> **4** <Enter>

Specify width: **2** <Enter>

Specify height or [2Point] <4.1326>: **.25** <Enter>



Command: **_box** <Enter>

Specify first corner or [Center]: {pick a point onscreen}

Specify other corner or [Cube/Length]: **L** <Enter>

Specify length <4.0000>: <Ortho on> **.25** <Enter>

Specify width <2.0000>: **.25** <Enter>

Specify height or [2Point] <-0.2500>: **2** <Enter>



Command : **3darray** <Enter>

Select objects: 1 found {select all the boxes}

Select objects: 1 found, 2 total

Select objects: 1 found, 3 total

Select objects: 1 found, 4 total

Select objects: 1 found, 5 total

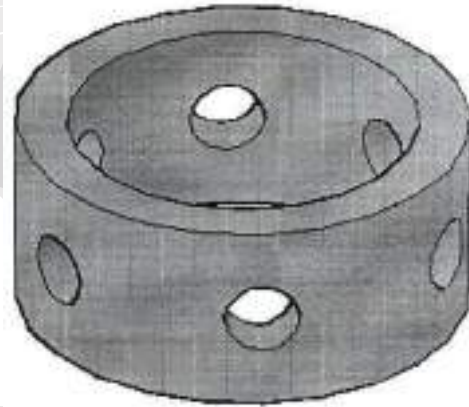
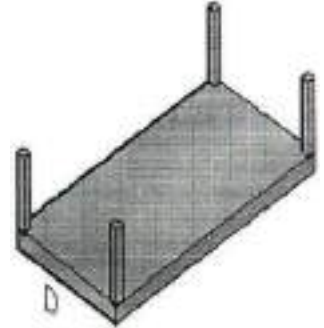
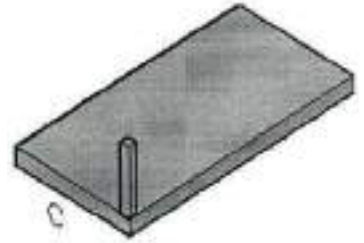
Select objects: <Enter>

Enter the type of array [Rectangular/Polar] <R>: **R** <Enter>

Enter the number of rows (---) <1>: <Enter>

Enter the number of columns (|||) <1>: **4** <Enter>

Specify the distance between levels (...): **2** <Enter>

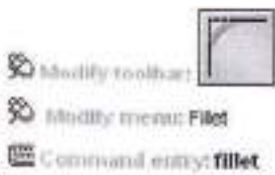


Exercise 20 - 1

Use the 2DARRAY (polar option), to create the model shown. Use your own dimensions proportion to this figure.



Lesson 21 Fillet - Chamfe



Construct a solid model shown in figure a With an approximate size. (2 boxes with union)

Command : **f** <Enter>

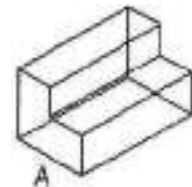
Current setting: Mode = TRIM, Radius = 0.0000

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

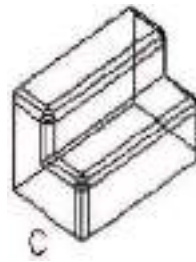
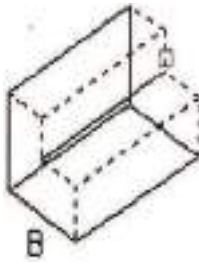
Enter filler radius <0.0000>: **.25** <Enter>

Select an edge or [Chair / Radius]:

{select all the edges as highlighted in figure B}



Select an edge or [Chair/Radius]:
 Select an edge or [Chair/Radius]:
 Select an edge or [Chair/Radius]:
 Select an edge or [Chair/Radius]:
 Select an edge or [Chair/Radius]:
 Select an edge or [Chair/Radius]:
 Select an edge or [Chair/Radius]:
 Select an edge or [Chair/Radius]:
 10 edge (s) selected for fillet.



Command: **cha** <Enter>

(TRIM mode) Current chamfer Dist=0.000, Dist2=0.0000

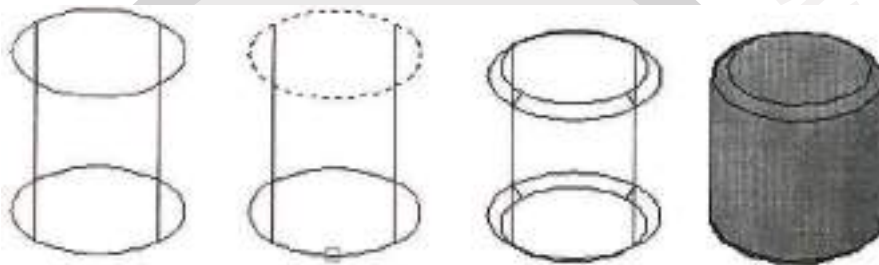
Select first line or [Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple]:

Base surface selection ...

Specify base surface chamfer distance <0.0000>: .5

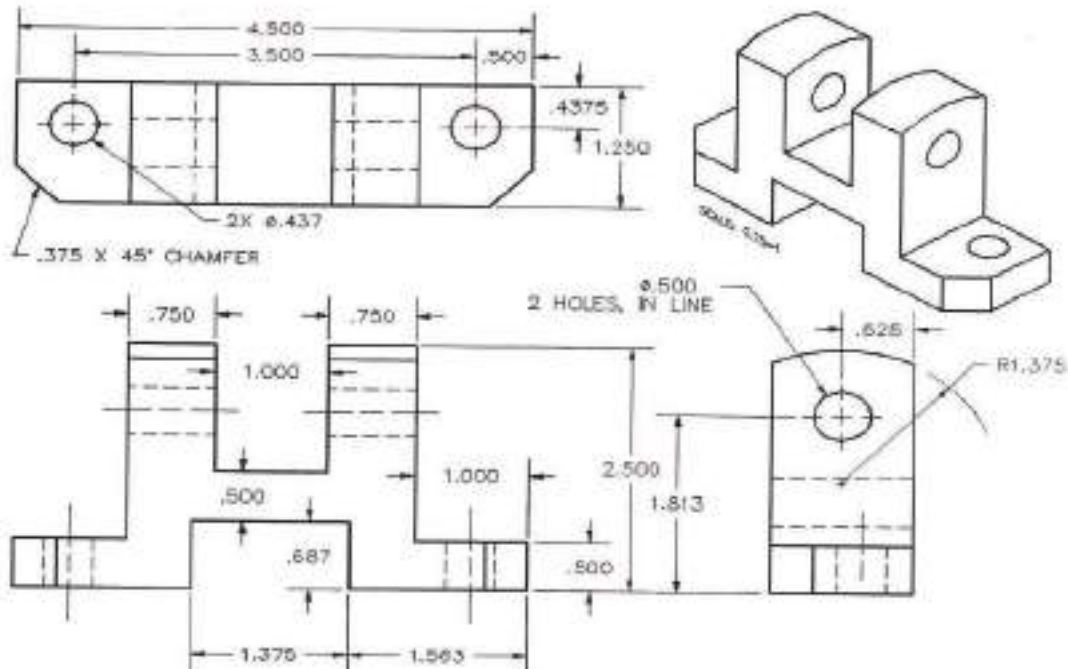
Specify other surface chamfer distance <0.0000>: .5

Select an edge or [Loop]: Select an edge or [Loop]: Select an edge or [Loop]:






Exercise 21-1

Construct a solid model of this figure, using the dimensions given on the orthographic views. {Hints: Draw the front view and extrude}



Lesson 22

Slice Command

-  Mobility menu: 3D Operations > Slice
-  Command entry: slice
-  Transmittent: 3D Make panel (click icon to expand), Slice

1. Draw a cone with the following specifications :

Command: **cone** <Enter>
 Specify center point of base or [3P/2P/Ttr/Elliptical]:
 Specify base radius or [Diameter]: **d** <Enter>
 Specify diameter: **38** <Enter>
 Specify height or [2Point/ Axis endpoint/Top radius]: **50** <Enter>
 (Figure 1)

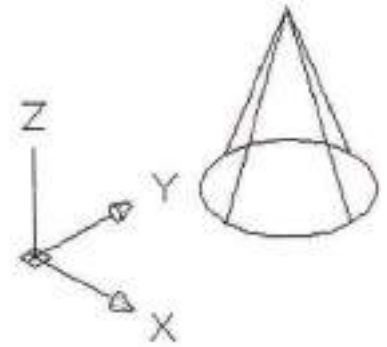


Figure 1

2. Change the UCS to the front view and set the color to green and draw the following construct lines:

Line "A" is exactly 33 mm, line "B" is a horizontal line with an approximate length, line "C" is connected from the quadrant of the base to the vertex of the cone and line "D" is 30°, begins from the intersection of two lines with approximate length. (Figure 2)

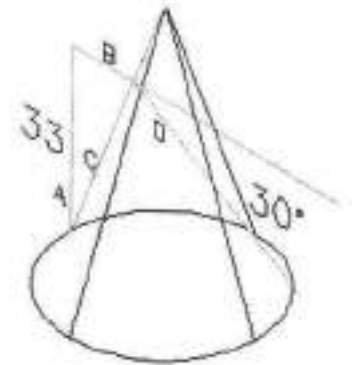


Figure 2

3. Use the COPY Command to copy the line "D" to both sides with the approximate distance as shown in figure 3.

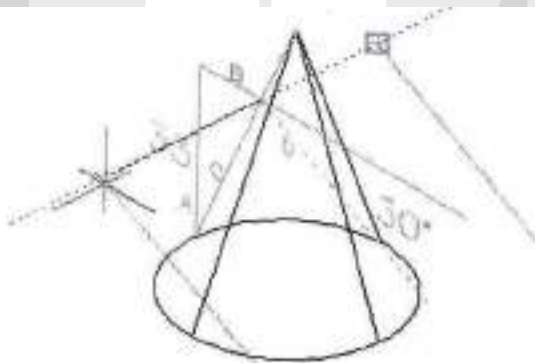


Figure 3

4. Use the SLICE Command with the "3 point" option :

Command: **slice** <Enter>
 Select objects to slice: {select the cone} 1 found
 Select objects to Slice: <Enter>
 Specify Start point of slicing plane or [planar Objects/Surface/Zaxis/View/XY/YZ/ZX/3point]
 <3points>: <Enter>
 Specify first point on plane: {pick the endpoint 1} (Figure 4)
 Specify second point: {pick the endpoint 2}

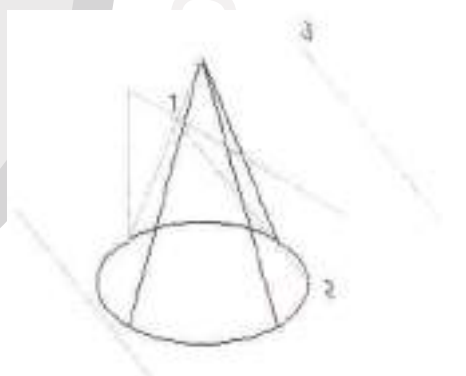
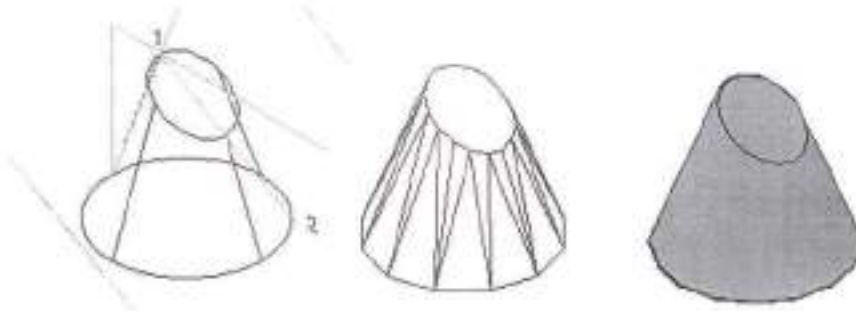


Figure 4

Specify third point: {pick the endpoint 3}

Specify a point on desired side or [keep Both Sides] <Both>: {pick the center of the base circle}



Exercise 22-1_ (Adjusting Guide)

Use the VPOINT Command to change the viewing option:

Command: vpoint <Enter>

Current view direction: VIEWDIR=0.0000,0.0000,1.0000

Specify a view point or [Rotate] <display compass and tripod>:
1,-1,1<Enter> Regenerating model.

This is the same as selection "SE Isometric" from the pull down menu "view".

Command: BOX <Enter>

Specify corner of box or [Center] <0,0,0>: {pick any point on screen}

Specify corner or [Cube/Length]: L <Enter>

Specify length: 110 <Enter> (turn the Ortho on and place the mouse on the positive X direction)

Specify width: 20 <Enter>

Specify width: 20 <Enter>

Command: Z <Enter>

ZOOM

Specify corner of window, enter a scale factor (nX or nXP), or

[All/ Center/ Dynamic/ Extents/ Previous/ Scale/ Window/ Object]
<real time>: .05 <Enter>

Command: 3DARRAY <Enter>

Select objects: 1 found {select the box you just created}

Select objects: <Enter>

Enter the type of array [Rectangular/Polar] <R>: R

Enter the number of rows (---) <1>: 3 <Enter>

Enter the number of columns (|||) <1>: <Enter>

Enter the number of levels (...) <1>: 3 <Enter>

Specify the distance between rows (...): 20 <Enter>

Specify the distance between levels (...): 20 <Enter> (Figure 1)

Erase 3 of the boxes and use the HIDE command. Your drawing should be similar to Figure-2

Use the UNION Command and select all six boxes:

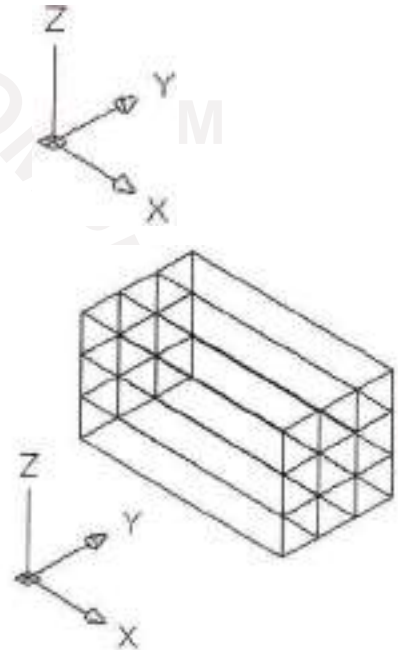
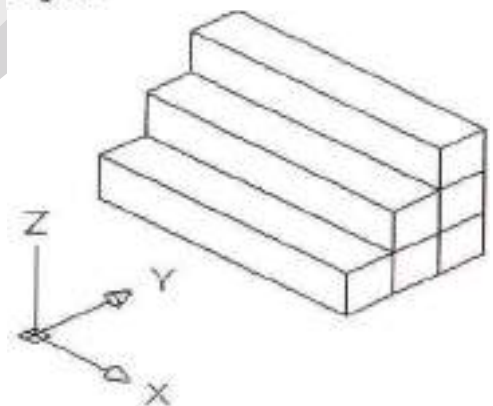


Figure 1



Command: **union** <Enter>

Select objects: 1 found (select the each box)

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: <Enter>

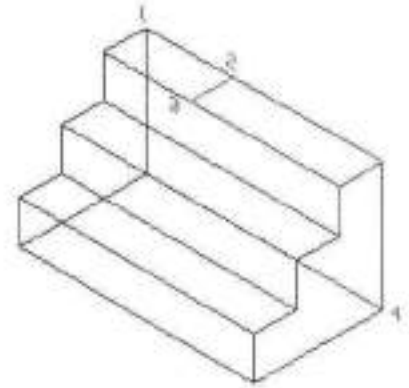
Change the color to "red" and draw a construction line as follows:

Command: **L** <Enter>

LINE Specify first point: _ from Base point: {under OSNAP menu pick "from" then click at point 1} <Offset>: **@40<0** <Enter>

Specify next point or [Undo]: <Ortho off> **@20<270** <Enter>

Specify next or [Undo]: <Enter>



Command: **slice** <Enter>

Select objects: 1 found {select the model}

Select objects: <Enter>

Specify first point on slicing plane by [Object/Zaxis/View/XY/YZ/ZX/3points] <3point>: <Enter>

Specify first point on plane: {pick point 2} Figure 3

Specify second point on plane: {pick point 3} Figure 3

Specify third point on plane: {pick point 4} Figure 3

Specify a point on desired side of the plane or [keep Both sides]: **b** <Enter>

Erase the right side of the figure 4 to complete the project, (Figure 5)

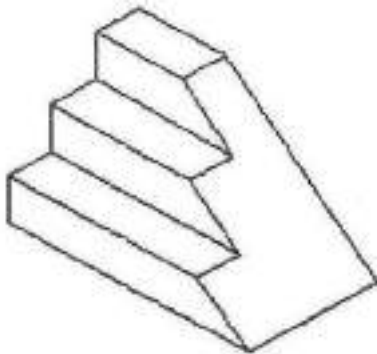
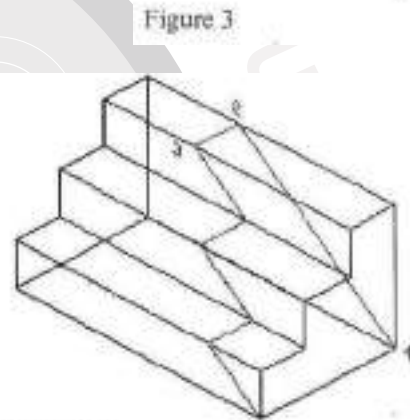
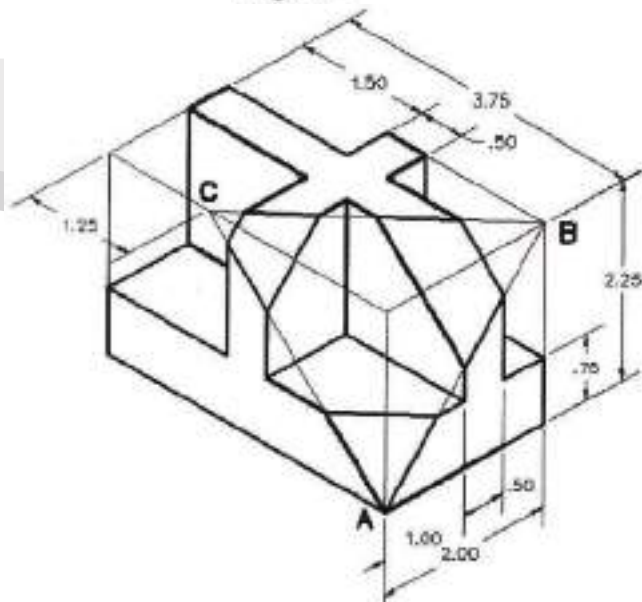


Figure 5 - (Adjusting Guide)

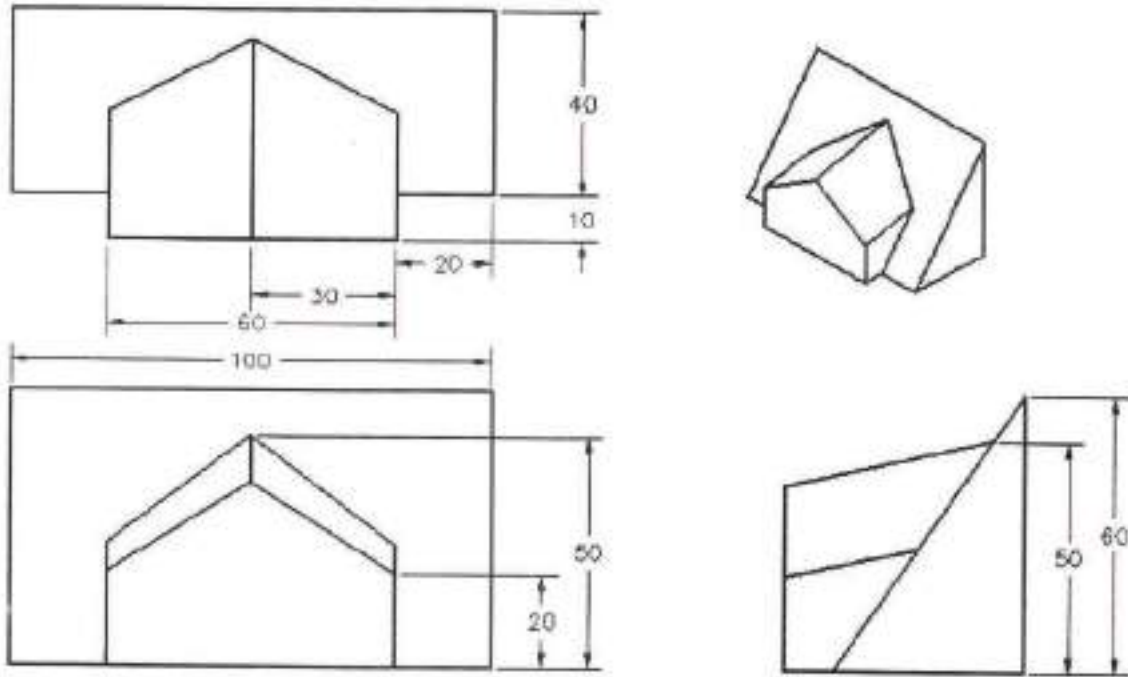
Figure 4



Exercise 22-2

Construct a solid model of these figure:

Exercise 22-3



Lesson 23 Solid Editing

☒ Menu: Modify > Solid Editing > Imprint Edges

☒ Command entry: **imprint**

☒ Dashboard: 3D Make panel, Imprint

Imprints an object on the selected solid. The object to be imprinted must intersect one or more faces on the selected solid in order for imprinting to be successful. Imprinting is limited to the following objects: arcs, circles, lines, 2D and 3D polylines, ellipses, splines, regions, bodies, and 3D solids.

Create the solid box, and draw an ellipse and a polygon (with the approximate sizes) as shown in Figure 2.

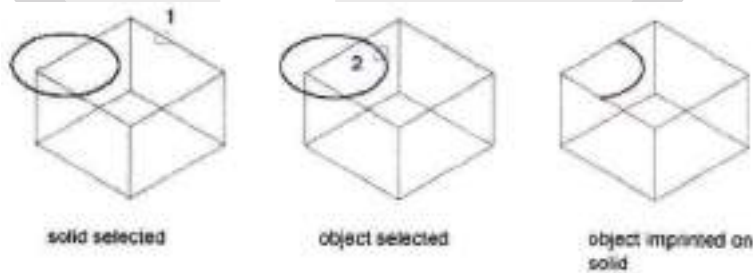


Figure 1

Command: `_imprint`
 Select a 3D solid:
 Select an object to imprint:
 Delete the source object [Yes/No] <N>: y <Enter>
 Select an object to imprint:
 Delete the source object [Yes/No] <N>: y <Enter>
 Select an object to imprint: <Enter>

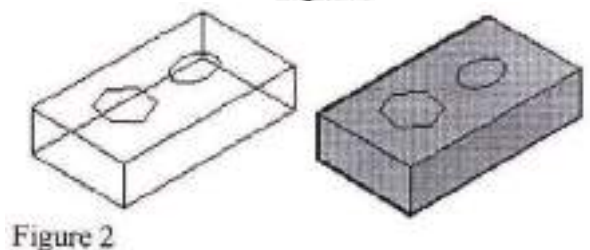


Figure 2

Command: `_solidedit`
 Solids editing automatic checking: `SOLIDCHECK = 1`
 Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: `_face`
 Enter a face editing option
 [Extrude/Move/Rotate/Offset/Taper/Delete/Copy/CoLor/mAterial/Undo/eXit]
 <eXit>:



Figure 3

`_extrude`
 Select faces or [Undo/Remove/ALL]: { pick the top face figure 3 } 2 faces found.
 Select faces or [Undo/Remove/ALL]: **R** <Enter>
 Remove faces or [Undo/Add/ALL]: { pick the top face figure 3 } 2 faces found,
 1 removed.
 Remove faces or [Undo/Add/ALL]: Enter >
 Specify height of extrusion or [Path]: `-3` <Enter>
 Specify angle of taper for extrusion <0>: <Enter>
 Solid validation started.
 Enter a face editing option
 [Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/mAterial/Undo/eXit] <exit >
X
 Solids editing automatic checking: `SOLIDCHECK=1`
 Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: **X**
 Solids editing automatic checking: `SOLIDCHECK=1`
 Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: **X**

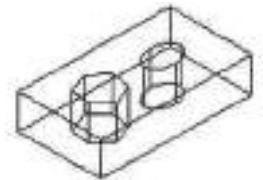


Figure 4



SHELL

Shelling creates a hollow, thin wall with a specified thickness. You can specify a constant wall thickness for all the faces. You can also exclude faces from the shell by selecting them. A 3D solid can have only one shell. New faces are created by offsetting existing ones outside their original positions.

Select a 3D solid: Select an object

Remove faces or [Undo/Add]: Select one or more faces or enter an option

The descriptions of the Undo, Remove, Add, and All options match the descriptions of the corresponding options under Extrude. Afer you select faces or enter an option, the following prompt is displayed.

Remove faces or [Undo/Add/ALL]: Select a face (1), enter an option, or press **ENTER**. Enter the shell offset distance: Specify a distance



face selected



shell offset=0.5



shell offset=-0.5

Specifying a positive value creates a shell to the inside perimeter of the solid; specifying a negative value creates a shell to the outside perimeter of the solid.

“Modify “Pull Down menu > Solid Editing > Shell :

Command : `_solidedit <Enter>`

Solids editing automatic checking: `SOLIDCHECK=1`

Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit> : `_body`

Enter a body editing option

[Imprint/seParate solids/Shell/cLean/Check/Undo/eXit] <eXit> : `_shell`

Select a 3D solid:

Remove faces or [Undo/Add/ALL]: 2 faces found, 2 removed.

{Remove the desired faces as illustrated in the following figures:}

Remove faces or [Undo/Add/ALL]: 2 faces found, 1 removed.

Remove faces or [Undo/Add/ALL]:

Enter the shell offset distance: `.25 <Enter>`

Solid validation started.

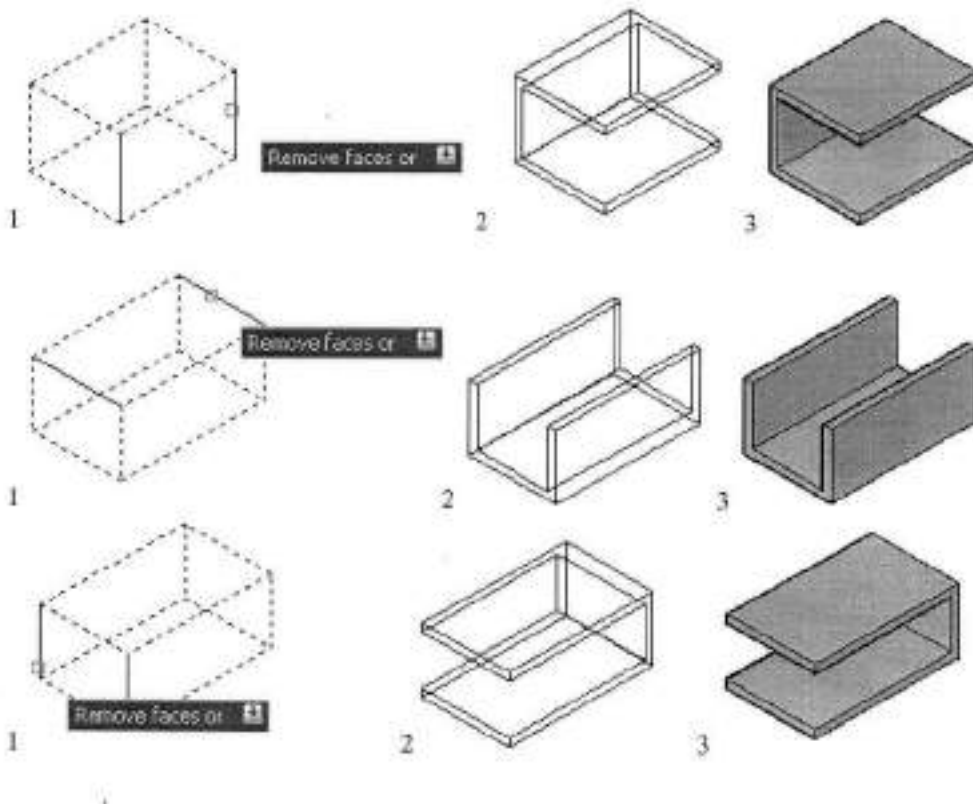
Solid validation completed.

Enter a body editing option

[Imprint/seParate solids/Shell/cLean/Check/Undo/eXit] <eXit> : `X <Enter>`

Solid editing automatic checking: `SOLIDCHECK=1`

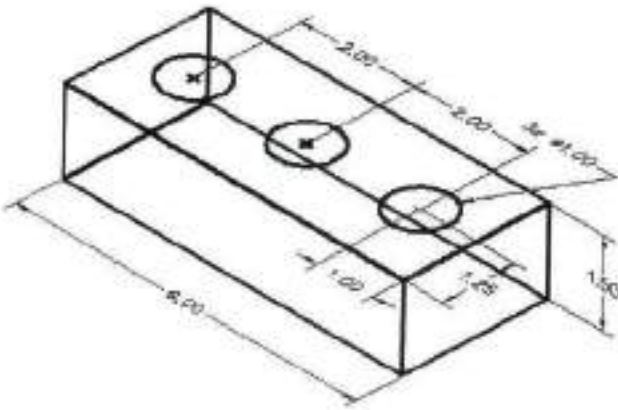
Enter a solids editing option [Face/Body/Undo/eXit] <eXit> : `X <Enter>`



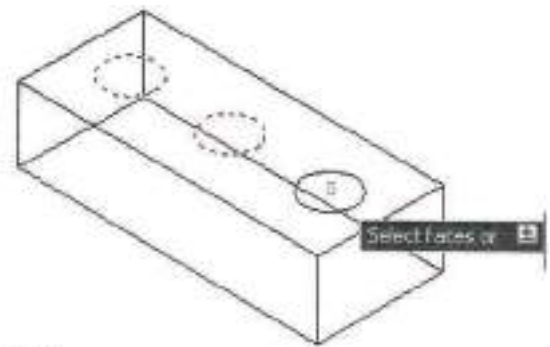
EXERCISE 23-1

1. Draw a solid box using the dimension shown in Figure 23-1
2. On the top surface, locate a single 1.0 diameter circle and copy or array the circle to other locations shown.
3. Use the Imprint option to imprint the circles onto the solid box. Delete the source object.

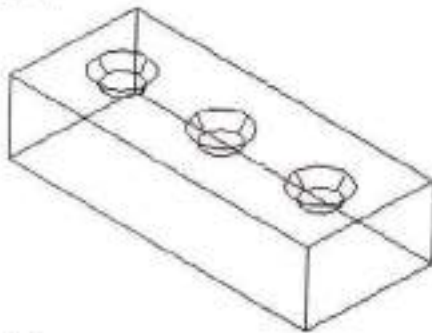
4. Extrude the imprinted circles to create three countersink holes as shown in Figure 23-3. (NOTE: if you pick the inside the circles, only the circle are selected and highlighted. In either case by sure to remove the top face, or any other faces id selected. (Figure 23-2) Extrusion height = -1.25 & Taper angle = 35°



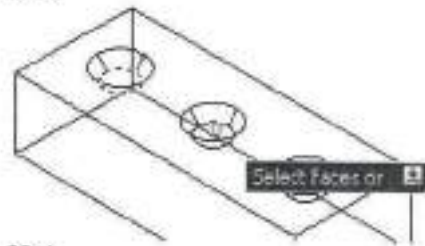
23-1



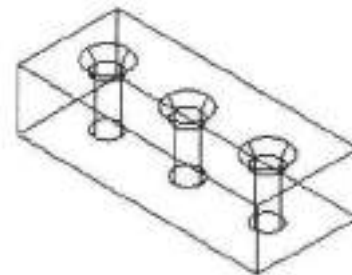
23-2



23-3



23-4



23-5

5. Extrude the small diameter of the three tapered holes so they intersect the bottom of the solid box. Select the holes by picking the inside of the small holes (figure 23-4). Use the negative value larger than thickness of the thickness of the solid box, for the height of the extrusion. (ex. -2)



Coping Faces

“Modify “Pull Down menu > Solid Editing > Copy Faces :

Copies face as a region or a body: If you specify two points, SOLIDEDIT uses the first point as a base point and places a single copy relative to the base point. If you specify a single point (usually entered as a coordinate) and then press ENTER, SOLIDEDIT uses the coordinate as the new location.

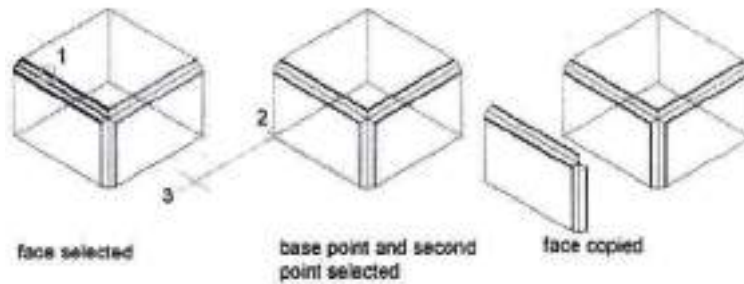
Select faces or [Undo/Remove]: Select one or more faces

The descriptions of the Undo, Remove, Add, and All options match the descriptions of the corresponding options under Extrude. After you select faces or enter an option, the following prompt is displayed:

Select faces or [Undo/Remove/ALL]: Select one or more faces (1), enter an option, or press ENTER

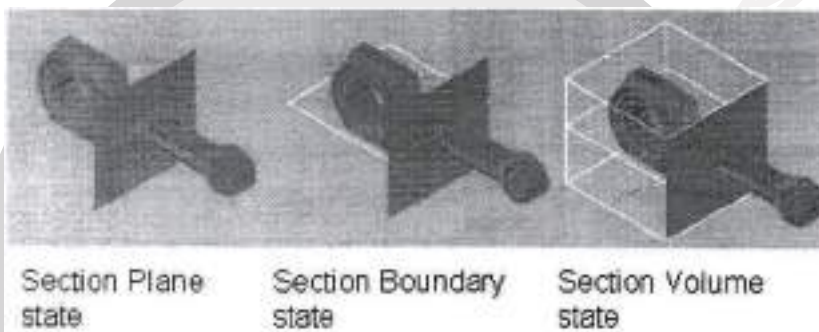
Specify a base point or displacement: Specify a base point (2)

Specify a second point of displacement: Specify a point (3)



Lesson 24 Creating Section Planes

Section objects have three states: Section Plane, Section Boundary, and Section Volume. Depending on the state you choose, you can view the cutting plane as a 2D plane, a 2D box, or a 3D box. Grips allow you to make adjustments to the length, width, and height of the cutting area.



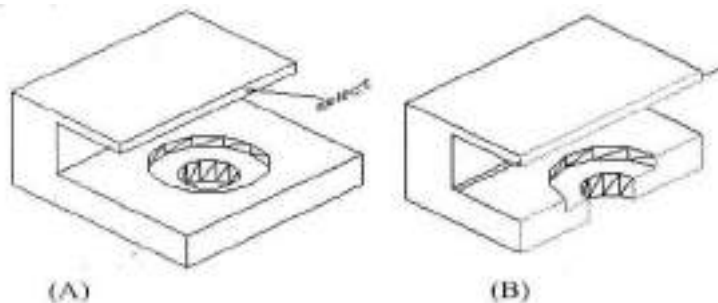
- ✳ **Section Plane.** The section line and transparent section plane indicator display. The cutting plane extends infinitely in all directions.
- ✳ **Section Boundary.** A 2D box shows the XY extents of the cutting plane. The cutting plane along the Z axis extends infinitely.
- ✳ **Section Volume.** A 3D box shows the extents of the cutting plane in all directions.

a. Face Option

Construct the figure "A" to an approximate size. Use SECTION PLANE Command to create figure "B"

Command: sectionplane <Enter>

Select face or any point to locate section line or [Draw Section/Orthographic]: { Select the face }



b. Two-Points Option

Construct a 3D solid model of the given views. Use the SECTION PLANE command to create a full-section model as shown.

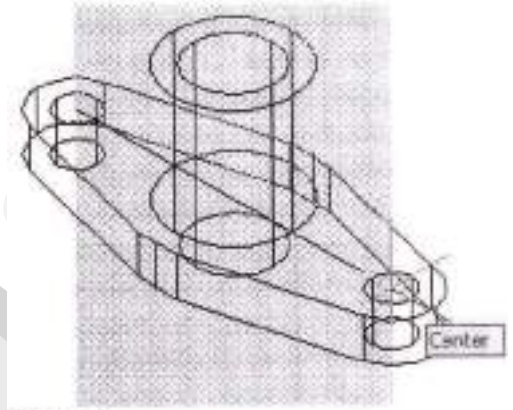
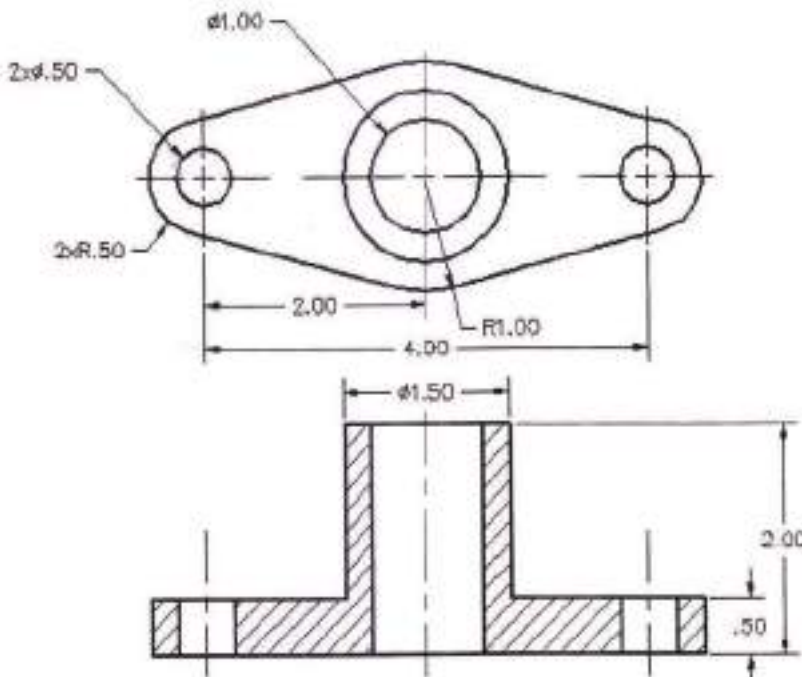


Figure 1

Command: sectionplane <Enter>

Select face or any point to locate section line or [Draw section/ Orthographic]: **D** <Enter>

Specify start point: {select the center of the small circle on the left}

Specify next point: {select the center of the small circle on the right} Figure 1.

Specify next point or ENTER to complete: <Enter>

Specify point in direction of section view: {select a point on the opposite side of the section object, using object snap if necessary} Figure "2"

Note:

Select the section plane and right-click to turn on the "live sectioning" (Figure 3). Also make sure the section plane is selected by clicking on the red triangle on the section plane. (Figure 4)

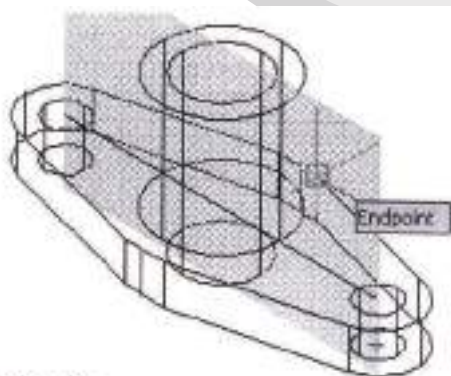


Figure 2



Figure 3

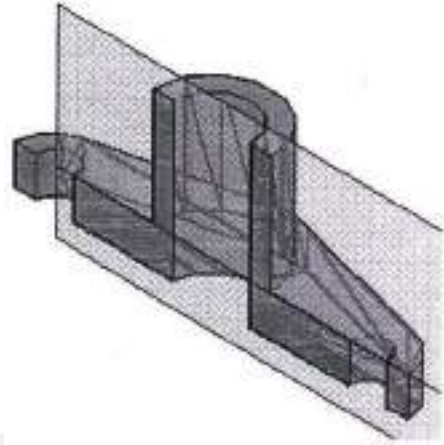
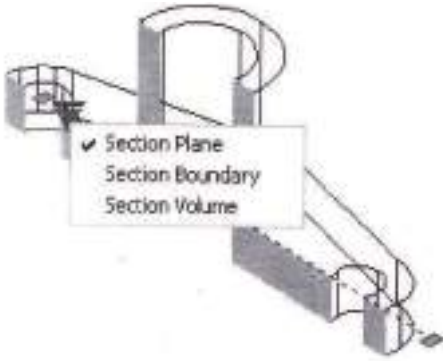
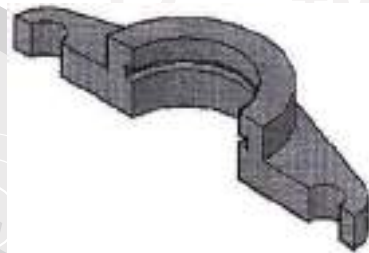
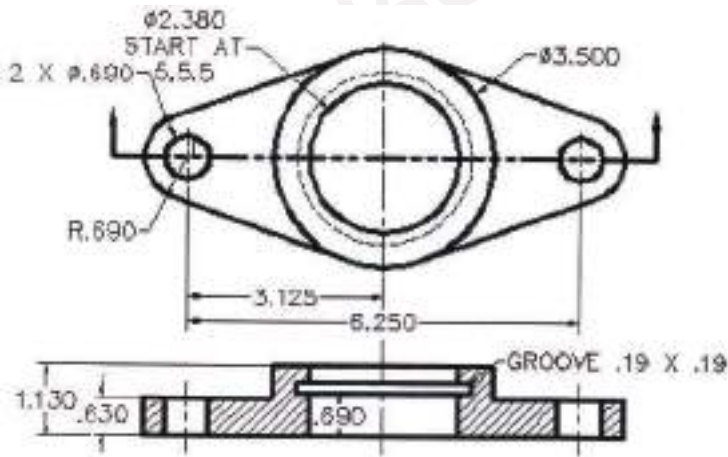


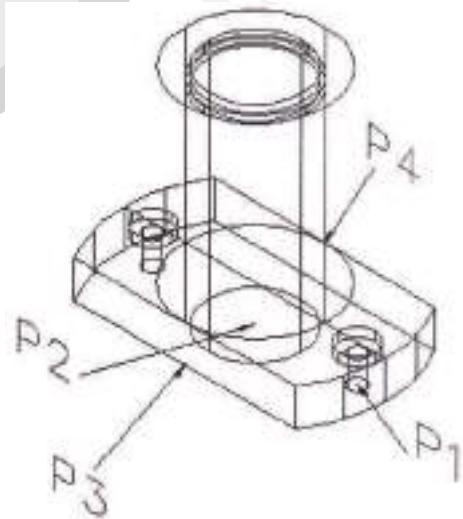
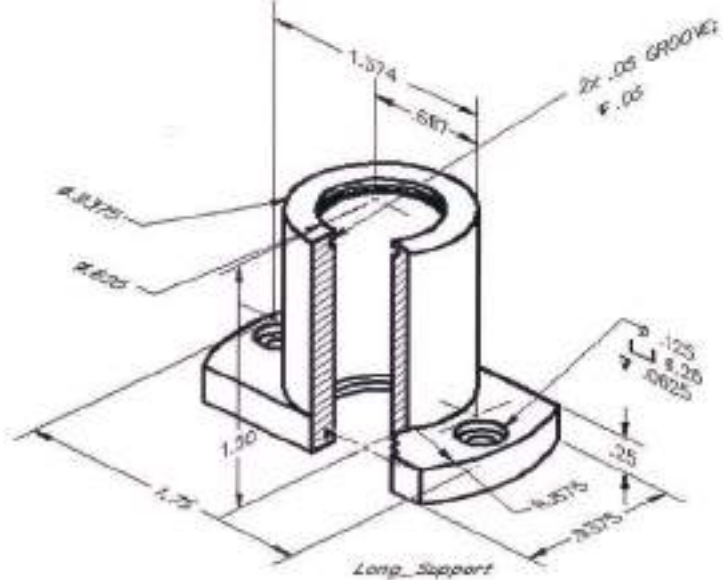
Figure 4

Exercise 24-1



c. Multiple Points Option

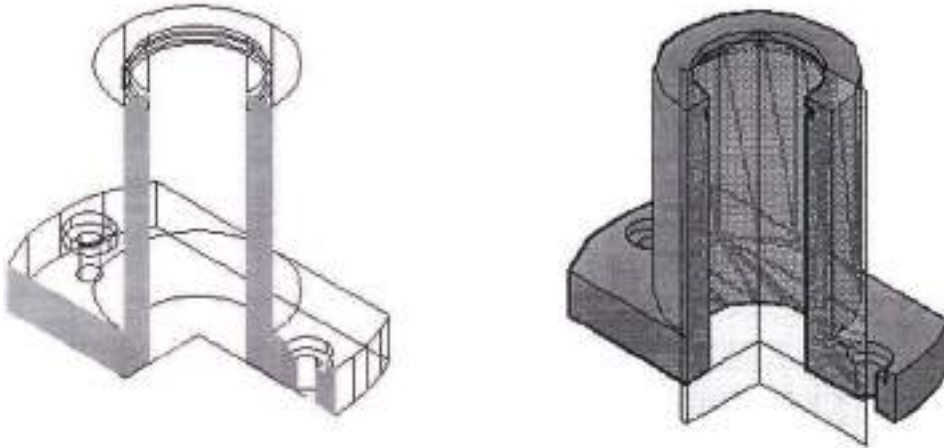
Construct a 3D solid model of the figure below. Use the SECTION PLANE command on create a Half-section model as shown



Command: sectionplane <Enter>
 Select face or any point to locate section line
 or [Draw section /Orthographic]: **D** <Enter>
 Specify start point: {select point p1}
 Specify next point: {select point p2}
 Specify next point or ENTER to complete: {select point p3}
 Specify next point or ENTER to complete: <Enter>

Specify point in direction of section view: {select point p4}

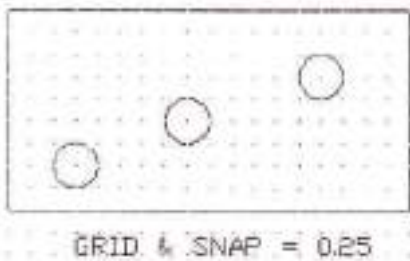
(Use object snap as needed)



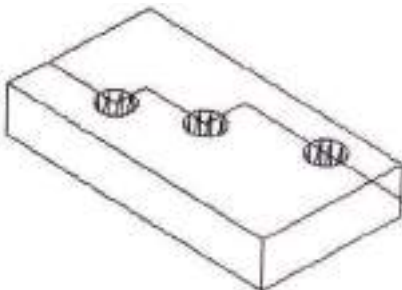
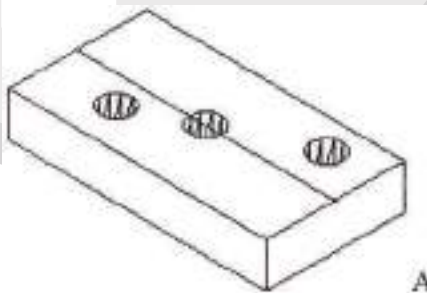
d. Adding Jogs to Section (Offset Section)

Select section object: Select as section object Specify a point on the section line to add jog: Pick a point on the section line

A jog is created on the section line. The jogged segment is created at a 90-Degree angle to the section line

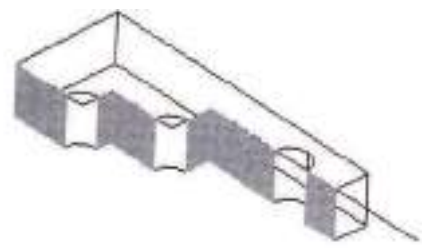
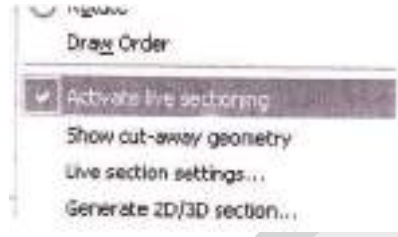
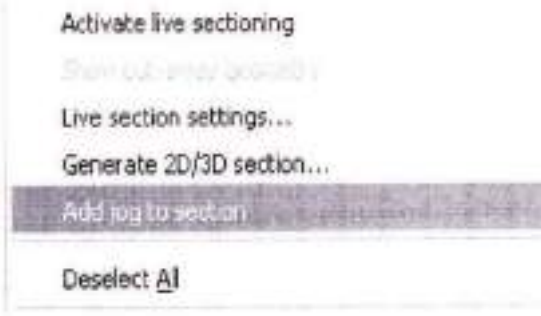
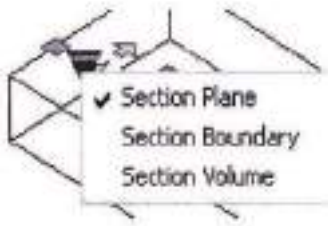


a. Draw a section plane from mid point



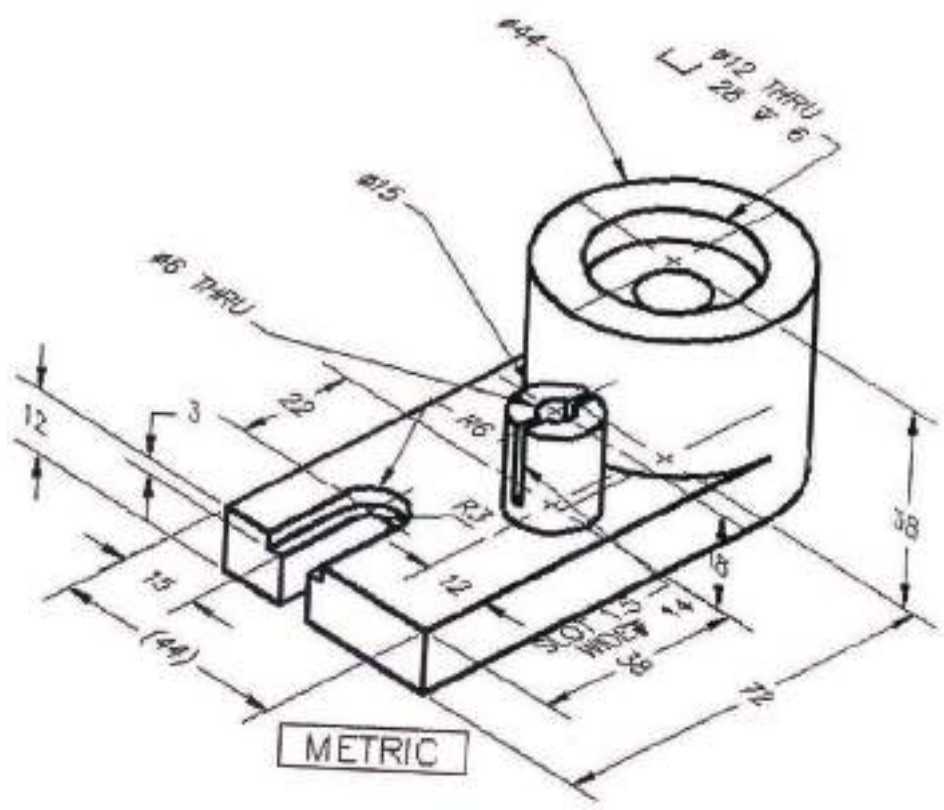
b. Select the section line and right-click to add jog to section Make sure the section plane is also checked.

The Nearest object snap is turned on to ensure you pick the line. It is not critical that you pick the exact location on the line. Grips allow you adjust the section plane location later. (Figure B)



EXERCISE 24 - 2

Create an offset sectional model of the figure below:



Lesson 25 FLATSHOT Command

Command entry: flatshot

Default panel: 3D Mate panel, Flatshot

With the FLATSHOT Command, you can create a flattened view of all the 3D solids and regions in the current view.

The resulting view is a block that is a flattened representation of the 3D model and is projected onto the XY plane. The process is similar to taking a “snapshot” of the entire 3D model with a camera and then laying the photograph flat. After inserting the block, it can be modified because the flattened view is comprised of 2D geometry. This feature is particularly useful for creating technical illustrations.

EXERCISE _ open any of your 3D model drawing and practice with command.

Try to use the “Exports to a file option”. And then open this file and explore!



Lesson 26 Creating and Using Multi view Layouts SOL VIEW & SOLD RAW Commands

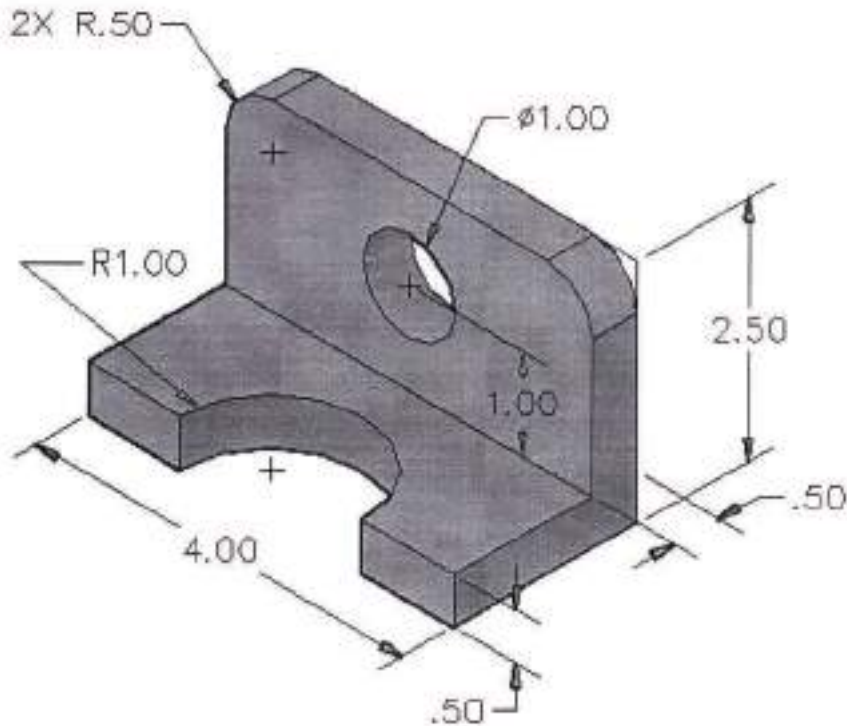
Menu: Draw > Modeling > Setup > View

Command entry: solview

Menu: DRAW > Modeling > Setup > Drawing

Command entry: soldraw

Construct the following 3D Model, and then generate a 2D orthographic view using the steps outlined below: Do not add dimension to the model view. Dimensioning will be done on the orthographic views.



Steps to generate orthographic views from a 3D Model

Note : After completing your model, make sure to save your work before continuing on the following steps :

Command: ucs <Enter>

Current UCS name: *WORLD*

Enter an option [New/Move/orthoGraphic/Prev/Restore/Save/Del/Apply/?/World] <world> : V <Enter>

Command: <Enter>

UCS

Current ucs name: *WORLD*

Enter an option [New/Move/orthoGraphic/Prev/Restore/Save/Del/Apply/?/World] <world> : S <Enter>

Enter name to save current UCS or [?]: ISO <Enter>

Go back to UCS default position (world) by pressing <Enter> twice:

Command: <Enter>

Enter an option [New/Move/orthoGraphic/Prev/Restore/Save/Del/Apply/?/World]<world> :<Enter>

Make sure your model relation to the user coordinate system is similar to the figure 1.

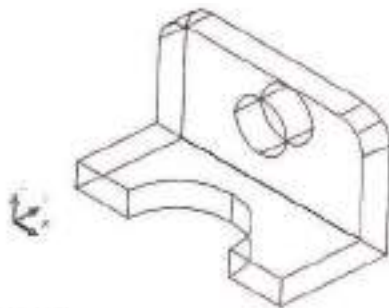
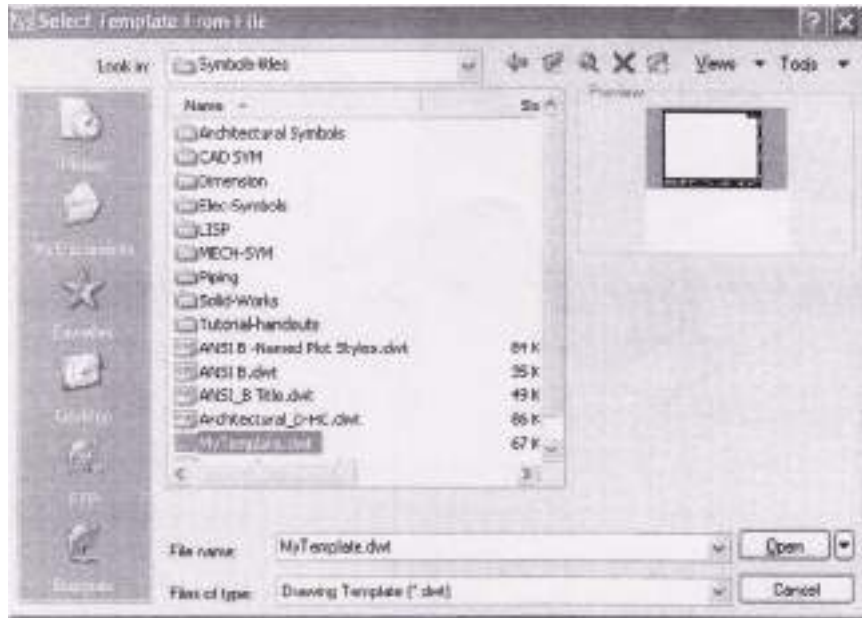


Figure 1



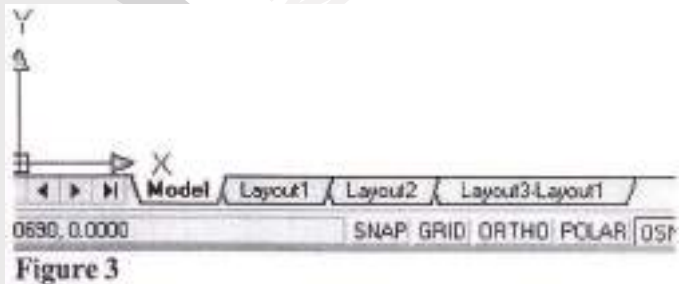
Figure 2

From the "Pull - Down Menu" Insert >Layout> Layout from brows through the CAD lab server insert "Mytemplate. dwt" file.



Select the "Layout3 tab-Layout "tab. (Figure 3)

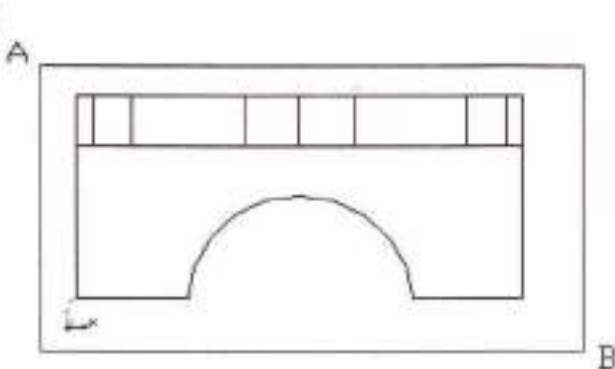
Command: **solview** <Enter>
 Enter an option [Use/Ortho/Auxiliary/Section]: u
 (for ucs)
 Enter an option [Named/World/?/Current]:
 <Enter>



Enter view scale <1.0000>: <Enter>

Specify view center: {pick approximate point for the center of the top view, continue until the view is in a desired location} <Enter>
 Specify first corner of viewport: {pick approximate point on the upper left corner of the top view (point A)}
 Specify opposite corner of viewport: {pick approximate point on the lower right corner of the top view (point B)} Figure 5.

Figure 5



Enter view name: top <Enter>

UCSVIEW=1 UCS will be saved with view
 Enter an option [Ucs/ortho/Auxiliary/Section]: **O**
 Enter >

Specify side of viewport to project: {pick a point on the lower portion of the rectangle port.}
 Specify view center: {pick approximate point for the center of the front view} <Enter>
 Specify first corner of viewport: {pick approximate point on the upper left corner of the front view}
 Specify opposite corner of viewport: {pick

approximate point on the lower right corner of the front view }

Enter view name: front <Enter>
 UCVIEW=1 UCS will be saved with view
 Enter an option [Ucs/Ortho/Auxiliary/Section]: **O** <Enter>

{ Repeat the above steps to create a side view }

Specify side of viewport to project: (Pick a point on the right side of the front rectangle)

Specify view center: {pick approximate point for the center of the side view, continue until the view is in a desired location then press the <Enter> key }

Specify view center <specify viewport>: {pick a point on the lower portion of the rectangle port.} <Enter>

Specify first corner of viewport: {pick approximate point on the upper left corner of the side view }

Specify opposite corner of viewport: {pick approximate point on the lower right corner of the side view}. Figure 6

Enter view name: side <Enter>

UCSVIEW=1 UCS will be saved with view

Enter an option (Ucs/Ortho/Auxiliary/Section]: <Enter>

Command: **soldraw** <Enter>

Select viewports to draw.

Select objects: {select anywhere on the viewports }

Select objects: {press Enter key to perform the Soldraw operation }

One solid selected.

One solid selected.

One solid selected.

Command:

Use the following steps to generate an isometric view from your 3D solid model:

Command: **solview** <Enter>

Regenerating layout.

Regenerating model-caching viewports.

Enter an option [Use/Ortho/Auxiliary/Section]: **U** <Enter>

Enter an option [Named/World/?/Current] <Current> : **N** <Enter>

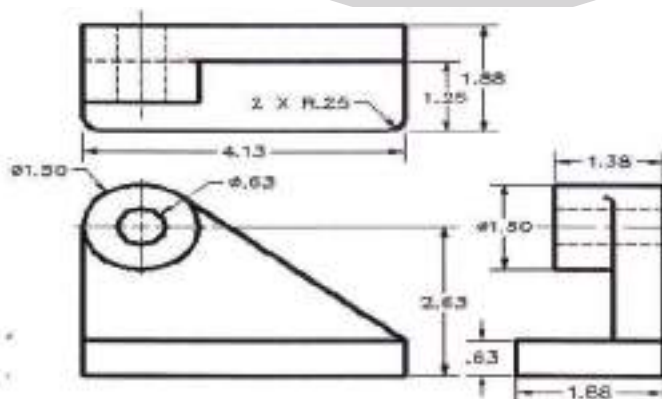
Enter name of UCS to restore **iso** <Enter>

(Repeat all the steps above to complete the isometric view)

Before printing, freeze (or select "no print icon on the layer dialog box) for the ISO -HID and VPORTS layers.

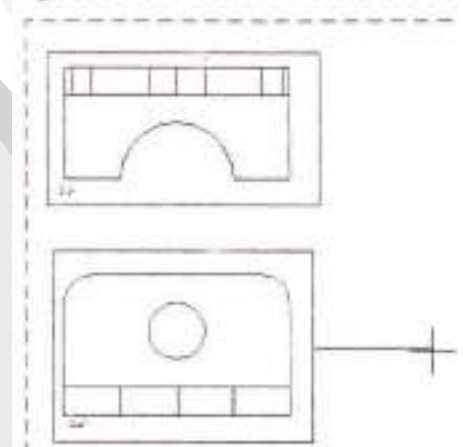
EXERCISES:

26-1 Create a solid model of the given figures. After completion, use SOLVIEW and SOLDRAW to generate the necessary orthographic views (as shown), add dimension and print the drawing.

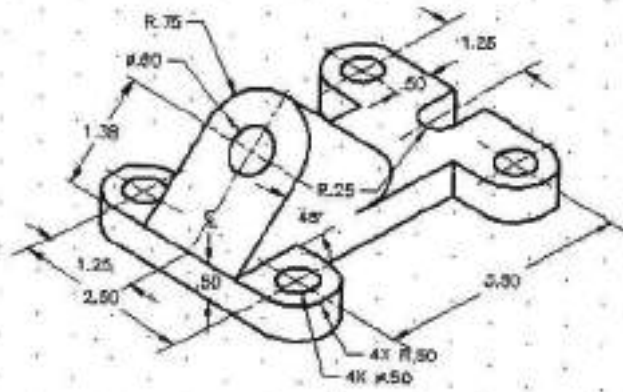


EX-26-1_ Shaft support

Figure 4



Lesson 27 Auxiliary UCS



5. Instruction

1. Draw four circles with radius with radius of 0.50 as shown in figure 1
2. Follow the steps in figure 2 through 5 to create the profile as shown in figure 5.



Fig.1



Fig.2

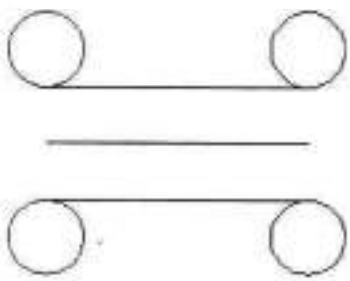


Fig.3

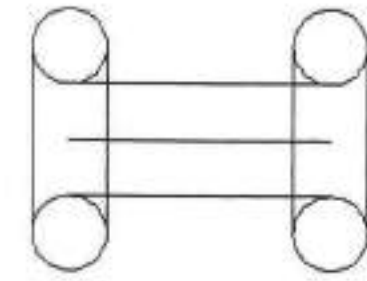


Fig.4

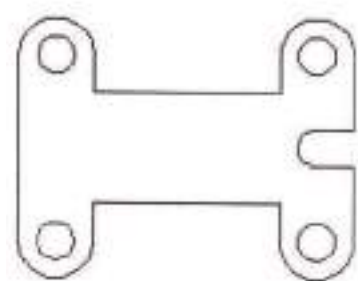


Fig.5

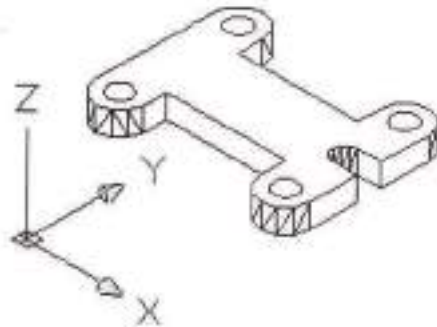


Fig.6

3. Change the view point to SE Isometric.
4. Use REGION or PEDIT to join the lines and arcs.
5. EXTRUDE the profile and four circles with -.5 Extrude height. (Figure 6)
6. Subtract four small cylinders from the model. (Figure 7)
7. Change UCS to one that shows in Figure 8 and draw a 45° line.
8. Create a new auxiliary using UCS Command with "3point" option

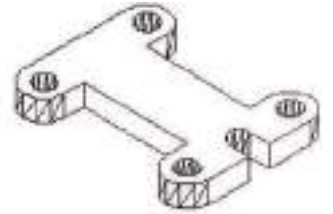


Fig.7

Command: line <Enter>
 LINE Specify first point: _mid {pick the mid point as shown in figure 8}
 Specify next point or [Undo]:
 >>Enter new value for ORTHOMODE <0>: Resuming LINE command.
 Specify next point or [Undo]: @1.38 <24 <Enter>

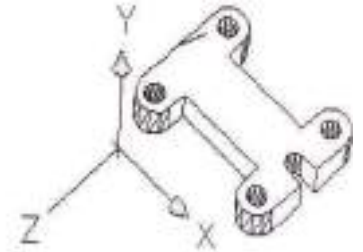


Fig.8

Command: ucs <Enter>
 Current ucs name: *WORLD*
 Enter an option
 [New/Move/orthoGraphic/Prev/Restore/Save/Del/Apply/?/World]
 <World>: n <Enter>

Specify origin of new UCS or
 [Zaxis/3point/Object/Face/View/X/Y/Z] <0,0,0>: 3 <Enter>

Specify new origin point <0,0,0>:{pick the end point 1; figure 9}
 Specify point on positive portion of X-axis
 <-2.7475,7.5789,-0.9901>: {pick the end point 2; figure 9}
 Specify point on positive -Y portion of the UCS XY plane
 {pick the end point 3; figure 9}<-2.7475,7.5789,-0.9901>:

Your drawing orientation should look like the one shown in Figure 10

9. Change the view point to NE Isometric: (Figure 11)

Command: vpoint <Enter>
 *** Switching to the WCS ***
 10. Current view direction: VIEWDIR = -1.0000,1.0000,1.0000 <Enter>

Create the profile on this plane and extrude.

Command: extrude <Enter>
 Current wire frame density: ISOLINE = 4
 Select objects: 1 found {select the profile}

Select object: <Enter>
 Specify height of extrusion or [Path]: =3 <Enter>
 Specify angle or taper for extrusion <0>: <Enter>

Command: hide <Enter> Regenerating model.

Union first then subtract the last hole.
 Use the **SLICE** Command to remove the bottom portion. Draw the last hole on the auxiliary plan.

Use SOLVIEW and SOLDRAW to generate the necessary orthographic view including one auxiliary view.

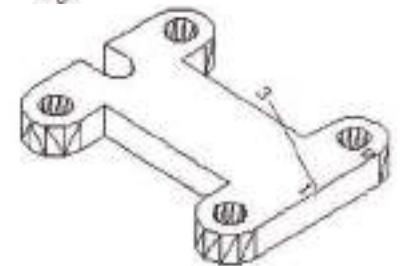


Fig.9

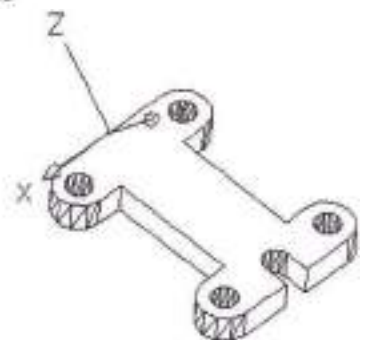


Fig.10

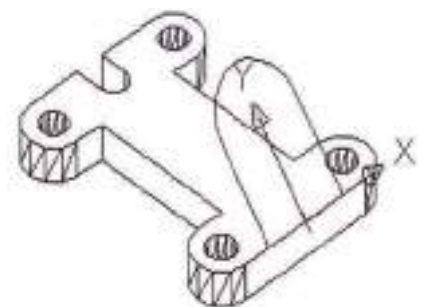


Fig.11

Note: Refer to instruction on "Printing 3D Model"

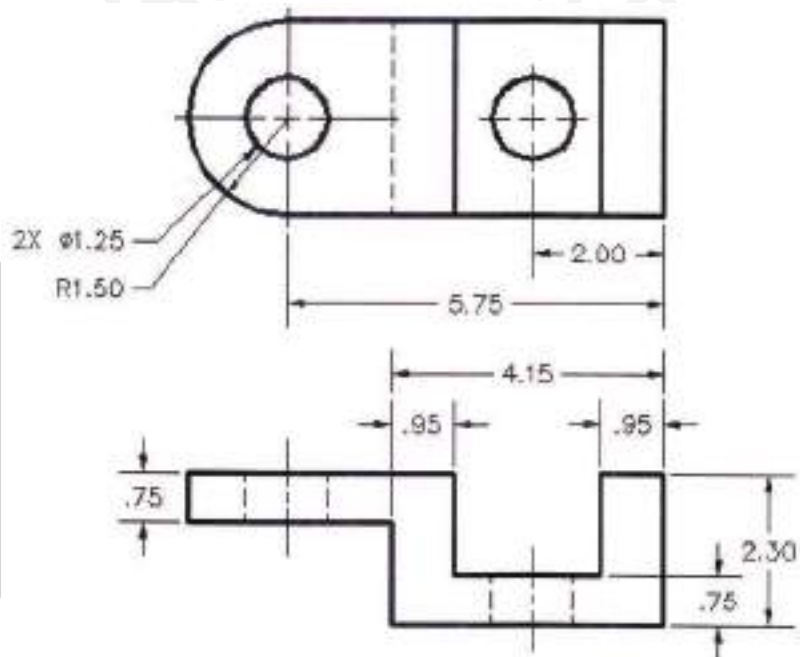


Fig.12

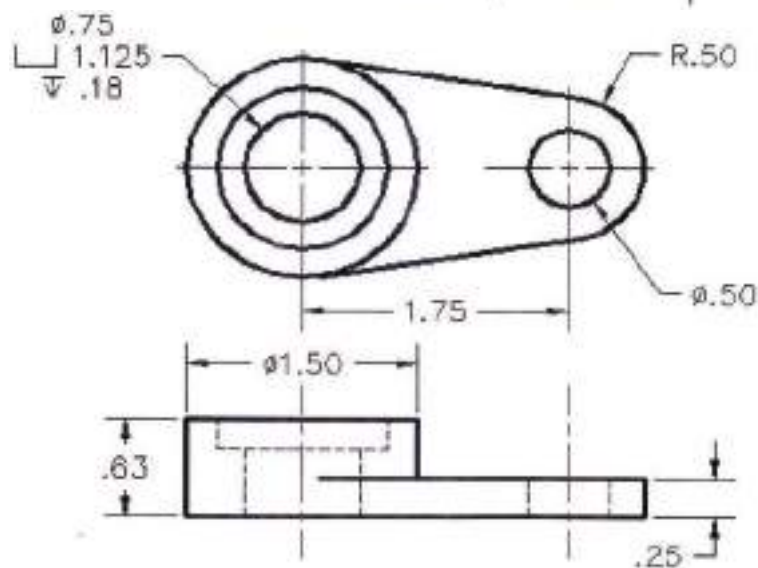
Additional Problems

Construct a 3D solid model of the given orthographic views.

Problem 1

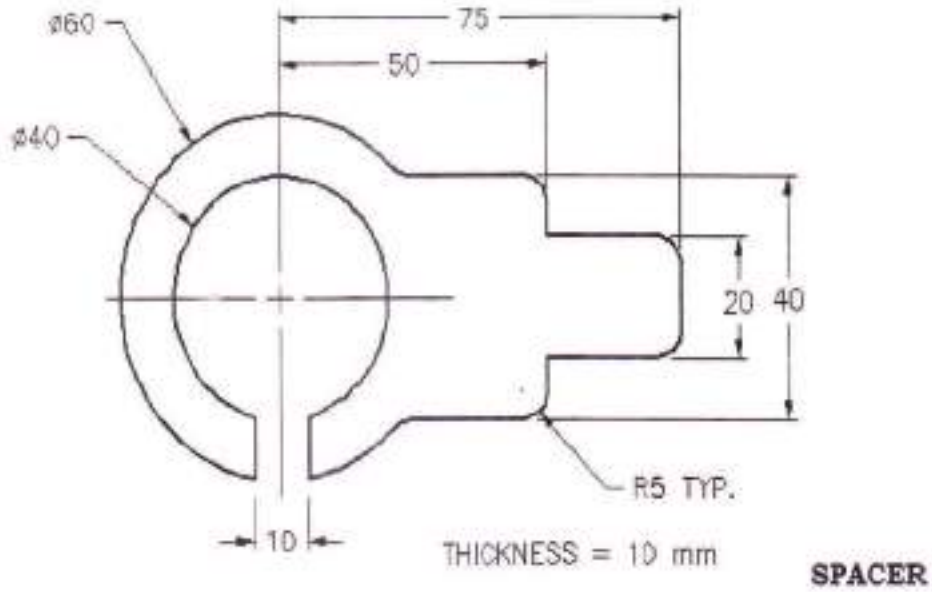


Problem 2



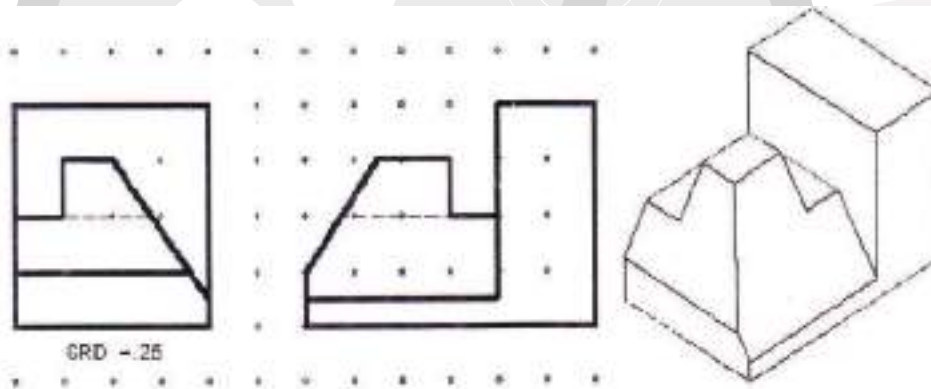
Problem 3

Given front view of the figures below, construct the 3D solid model with 10 mm thickness.

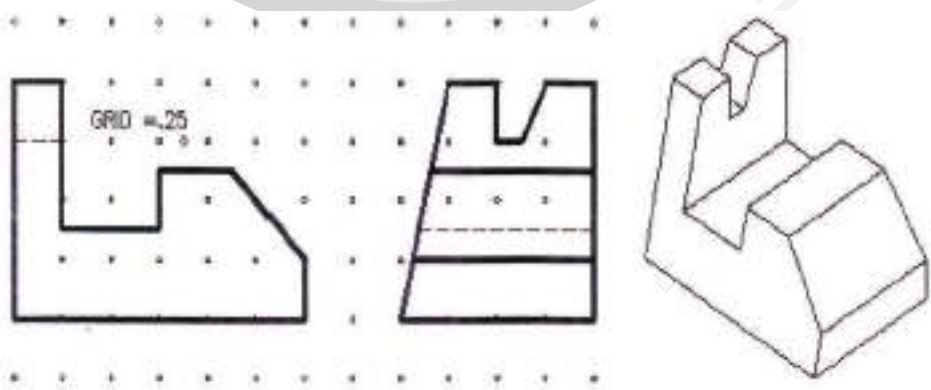


Problem 4

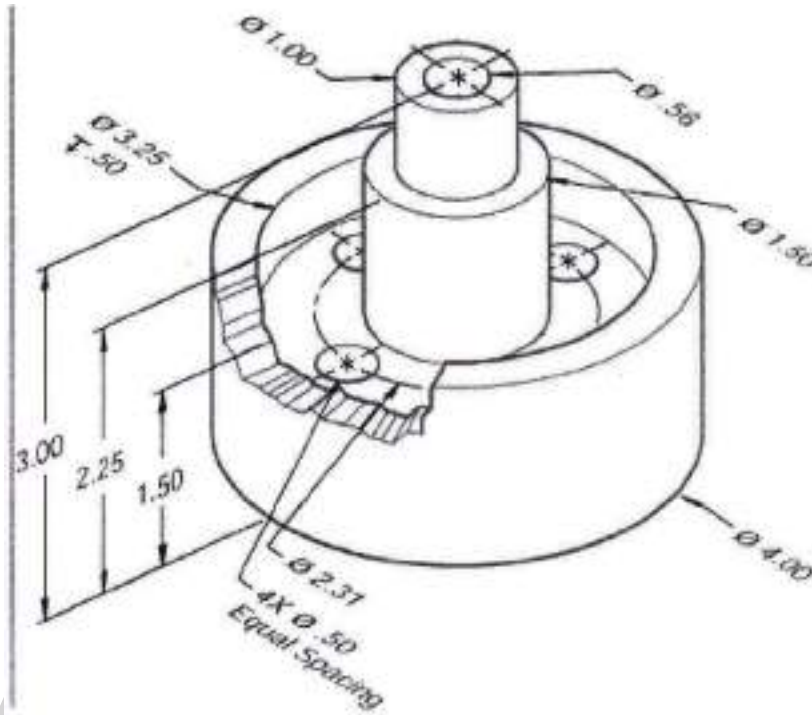
Given front and right side views of the figures below, construct the 3D solid models:



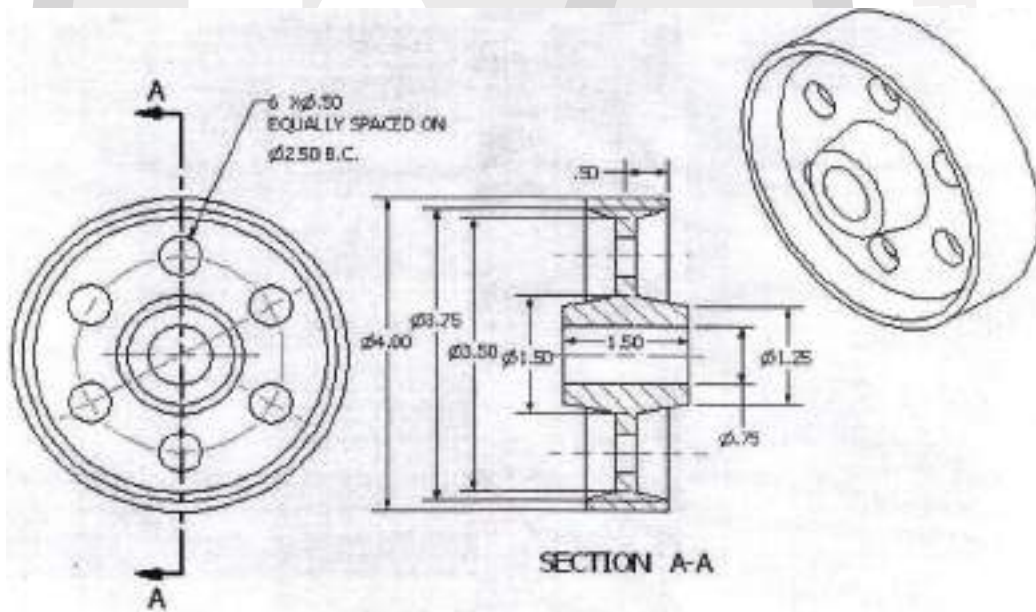
Problem 5



Problem 6

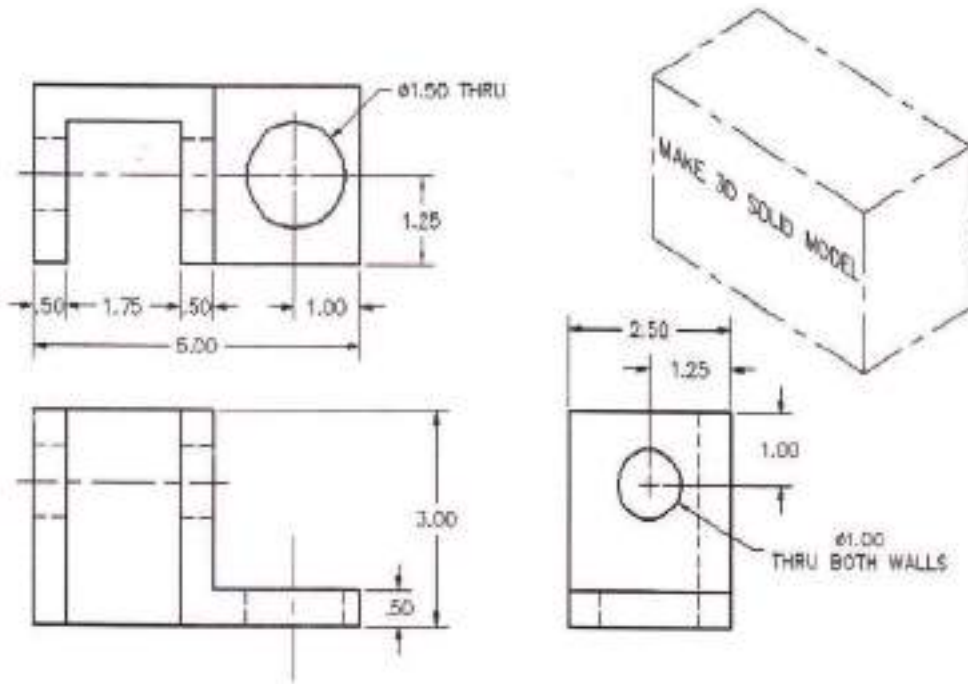


Problem 7

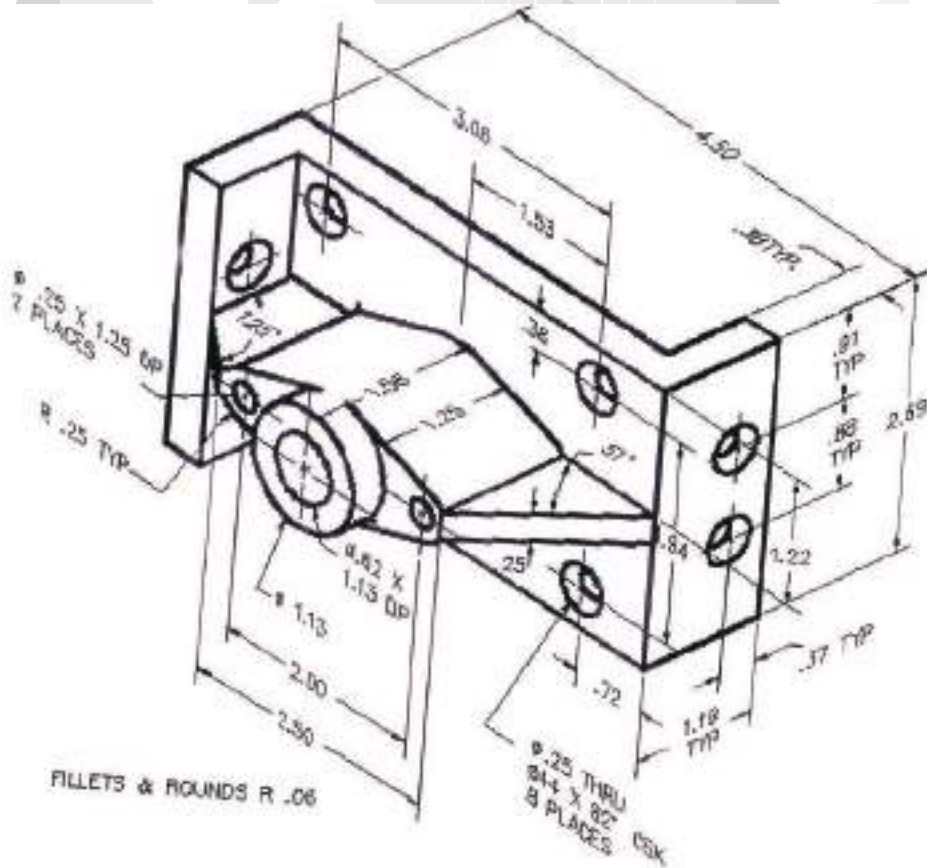


Problem 8

- Using the AutoCAD's modeling features, create 3D model of the given 3 views (TOP, FRONT & RIGHT SIDE).
- Calculate the VOLUME.
- Generate the orthographic views as shown including an isometric view.
- Print space 1 = 1.



Problem 9



Printing 3D Models

Insert your template file :

After your model is completed, from command line or from pull-down menu insert your template drawing as shown :

Command : **layout** <Enter>

Enter layout option [Copy / Delete / New / Template / Rename / SA / veas / Set / ?] <set> : **t** <Enter>

Locate your template file (called "My template" see Figure 2).

Note: For outside of the CAD Lab, I recommend that you save this file on a diskette or go to the course web site www.engrgraphics.pageout.net, "Course Contents "Course Materials" My Template. dwt.

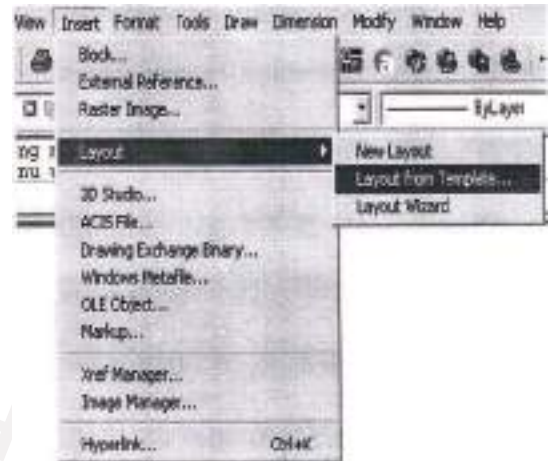


Figure 1

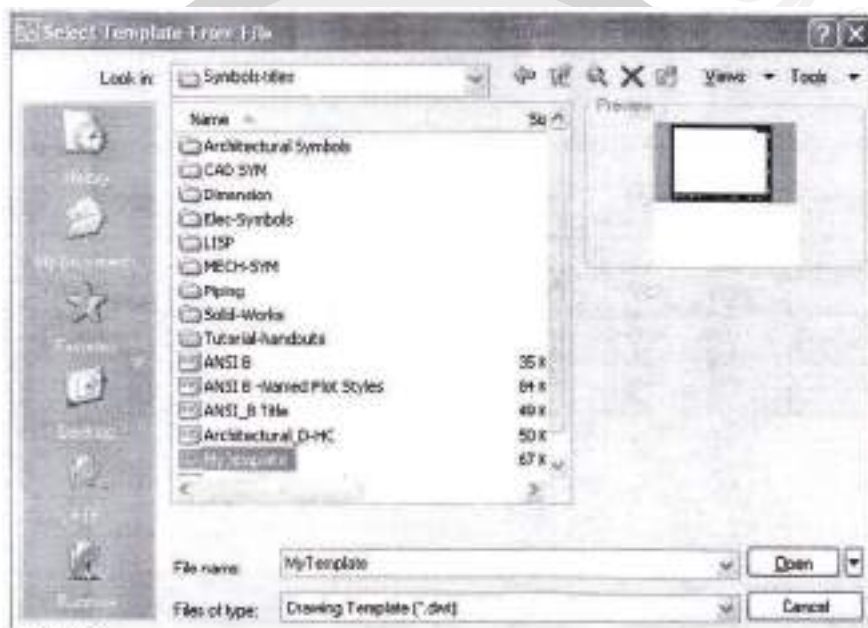


Figure 2

Left click "OK" tab.

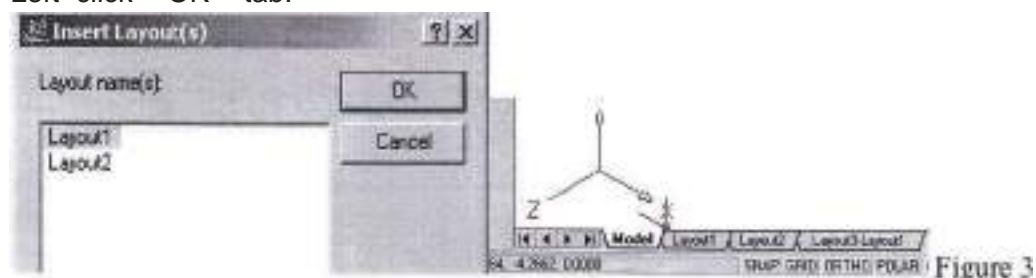


Figure 3

Command : < Switching to : Layout 3 - Layout 1 >
Regenerating layout

Restore your drawing :

Command : **mview** <Enter>

Specify corner of viewport or [On/OFF/Fit/Hideplot/Lock/Object/Polygonal/Restore /2/3/4] <fit> :
R <Enter>

Enter viewport configuration name or [?] <*Active> : <Enter>

Specify first corner or [Fit] <Fit> : **.5,.75** <Enter>

{NOTE : instead of typing, you may left click on the approximate points within the active drawing area}

Specify opposite corner : **10,7.25** <Enter>

Regenerating model.

Left click on "PAPER" tab or double click any where inside the active drawing area. Figure 4.
"PAPER" tab should switch to "MODEL". Figure 5

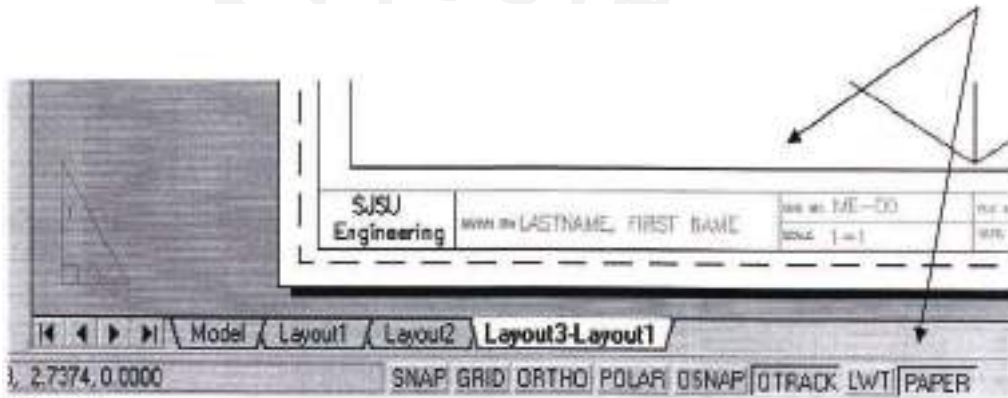


Figure 4

Scale your Drawing :

Command : **z** <Enter>

ZOOM

Specify corner of window, enter a scale factor (nX or nXP), or [All/Center/Dynamic/Extents/Previous/Scale/Window] <real time> : **1xp** {for 1=1 scale}

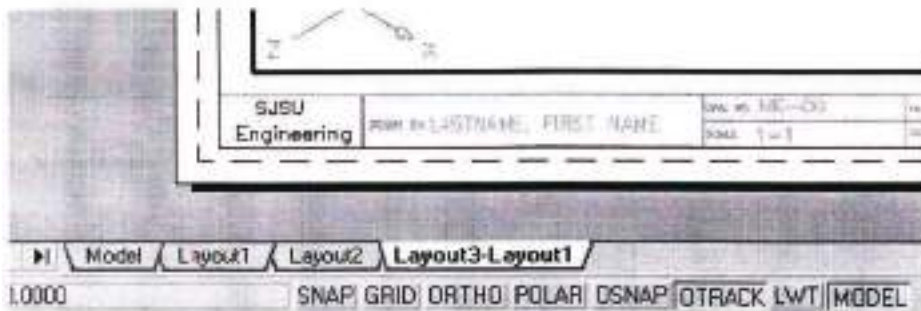


Figure 5

Click on "MODEL" tab to switch back to "PAPER" mode.

Perform Hide Plot:

Left click once on the edge of the floating view port to select it. Right click to display the short_cut menu as shown in Figure 6. Select Hidden option.

Create a new layer :

Select the layer from “Format” pull - down menu, or from command :

Command : **layer** <Enter>

Select the “New” tab. Figure 7. While “Layer 1” is highlighted, click on “Freeze” icon to turn to gray. Click on “Current” tab.

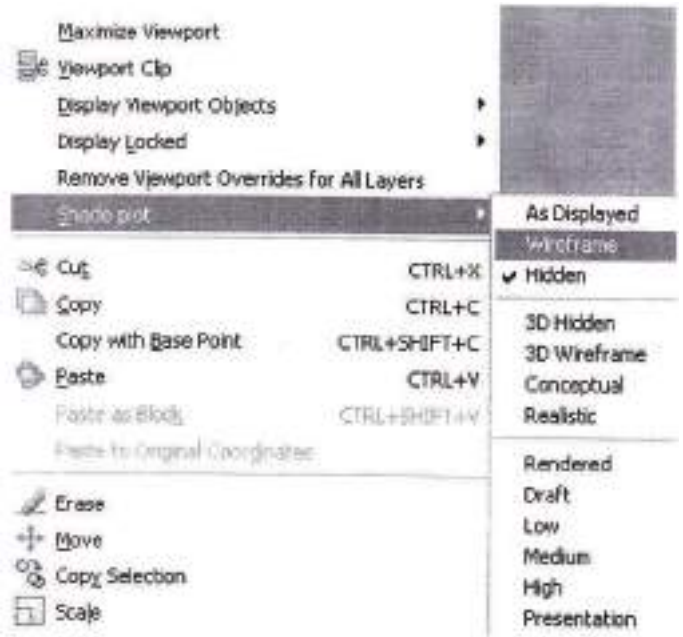


Figure 6

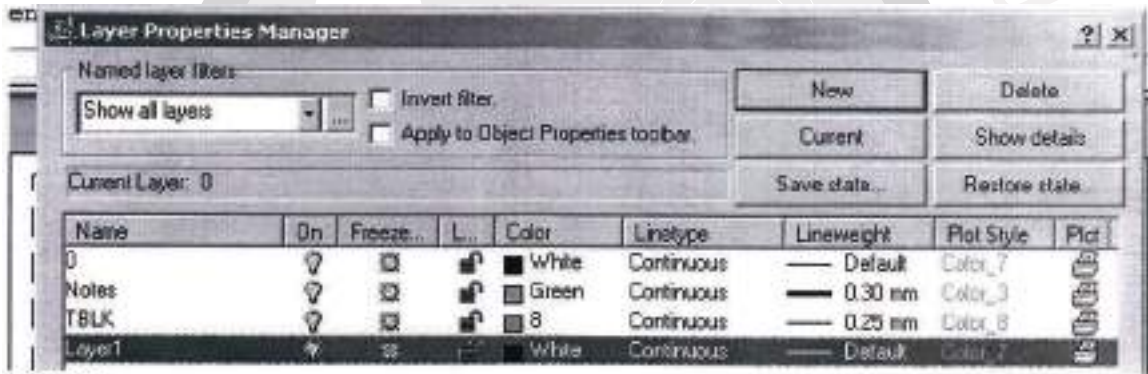


Figure 7

Assign the port to Layer 1 :

Click on the edge of the floating view port high light the layer 1 from the layer window and click “OK”. Figure 8:



Figure 8

Print the drawing :

Access the plot command from “File” pull - down menu, or from the command line or from the printer icon. On “Plot dialog box” click on the “plot device” tab ; select

the printer your lab. On plot style table” tab, select monochrome.ctb. Click on the “plot settings” tab, and choose the appropriate scale for you drawing. See the following illustrations :

- a. Select your printer
- b. Select settings
- c. Full preview (Figure 9)

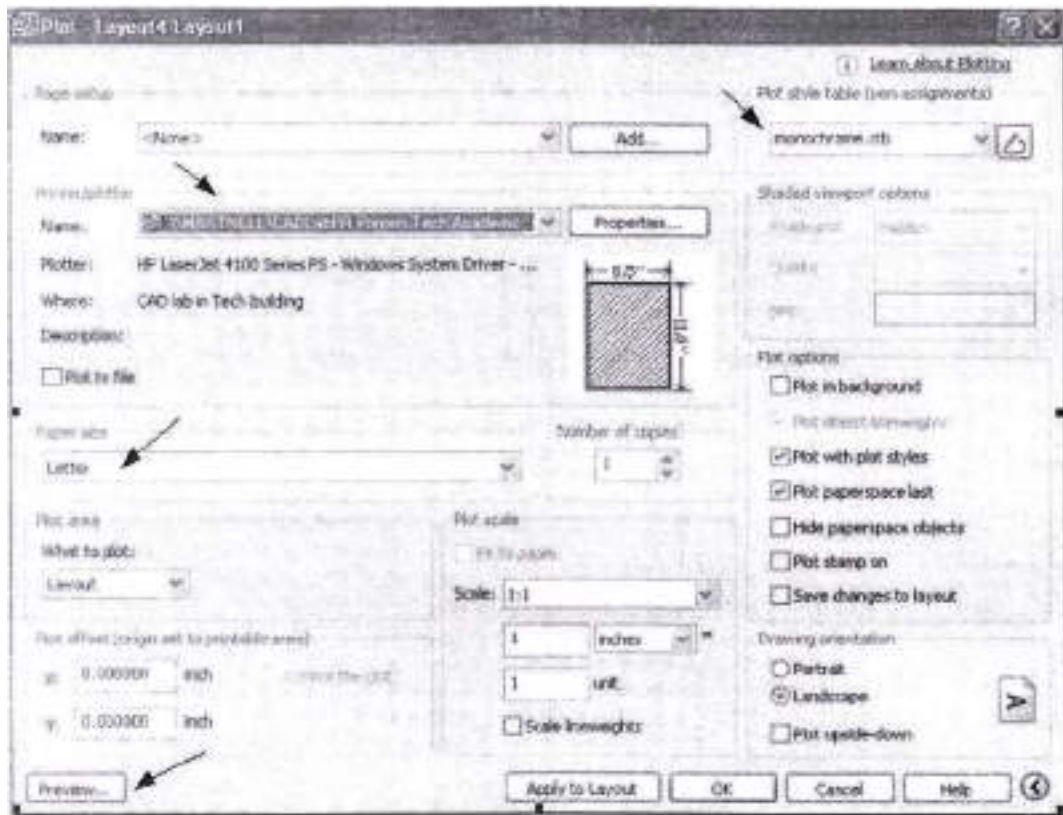


Figure 9

Note :

Select Full Preview : to check your work before sending to the printer.

If you print preview looks correct, right click and select “plot” from the short-cut menu. Otherwise exit, check all the print settings and fix the problem.