AutoCAD Workbook 3D

DRA 54
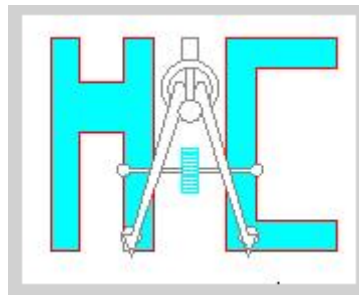
Hartnell College
**Engineering Technology**
©
Parviz D. Entekhabi

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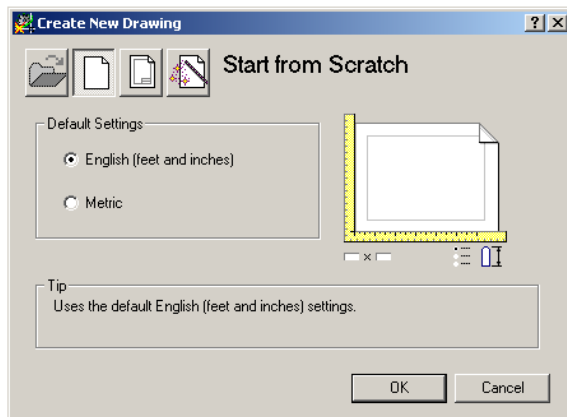
2. *Turn the Grid and Snap ON (F7 & F9), and use the default spacing.*

Command: **grid** <Enter>
Specify grid spacing(X) or [ON/OFF/Snap/Major/aDaptive/Limits/Follow/Aspect]
<0.5000>: **L** <Enter>
Display grid beyond Limits [Yes/No] <Yes>: **n** <Enter>
Command: <Enter>
Specify grid 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 you drawing with the LINE Command, and construct the figure 1. Do not be concern 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.

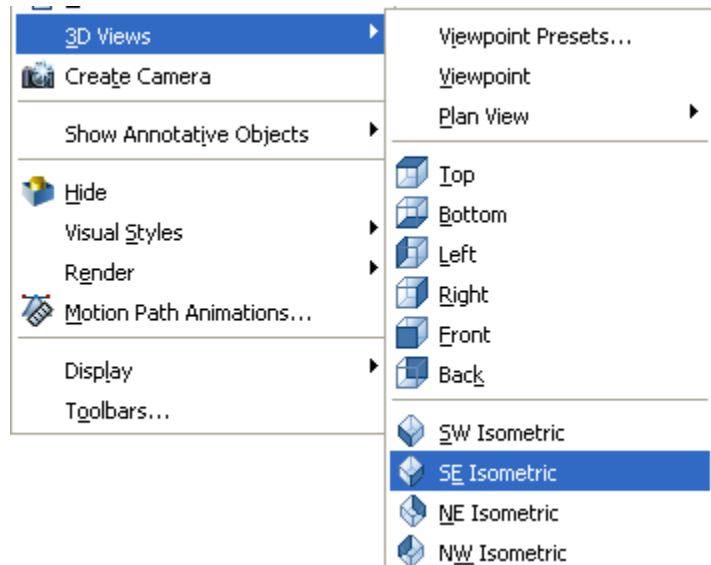*You may access this command from*
"View" pull down menu.

(Figure 2)

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>*
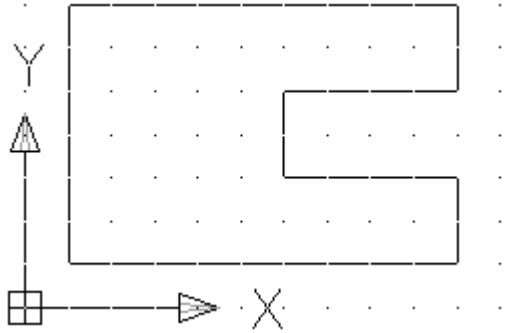*Specify new default thickness <3.0000>: **6** <Enter>*
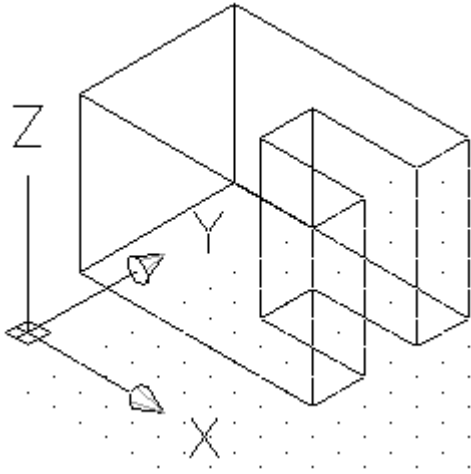
Figure 1   PLAN VIEW

Figure 3   3D View

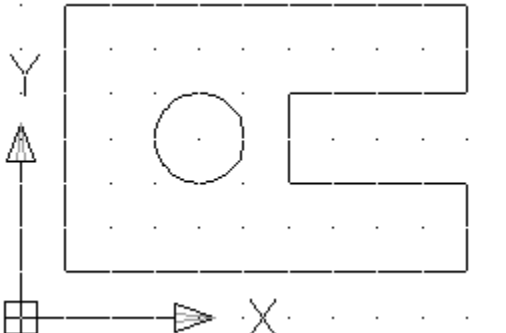8.   Add a circle as shown on Figure 4 and view from SE Isometric. (Figure 5)
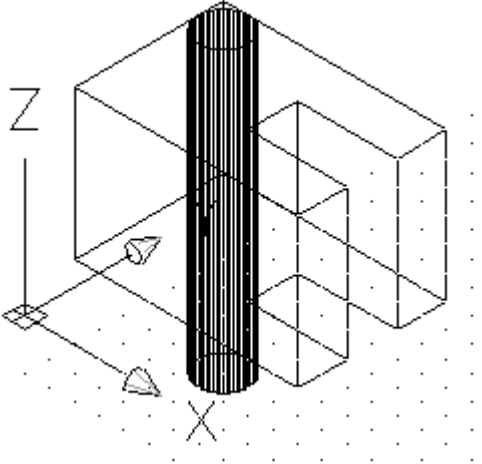
Figure 4

Figure 5

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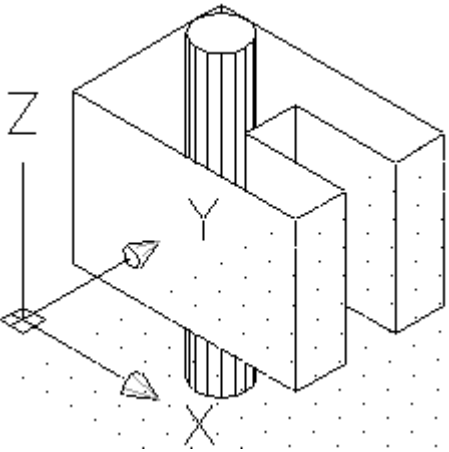*select "Hide", or enter the **HIDE***
command from the keyboard.

Fig. 6

4

Exercise 1-1

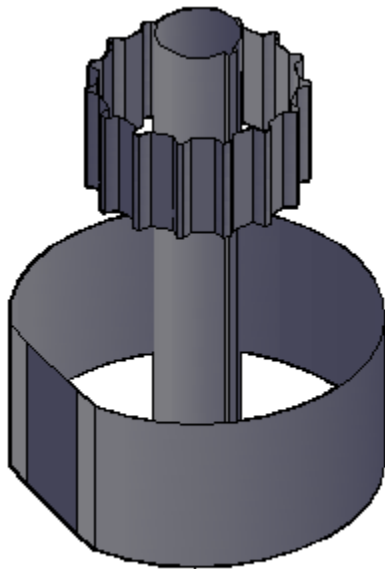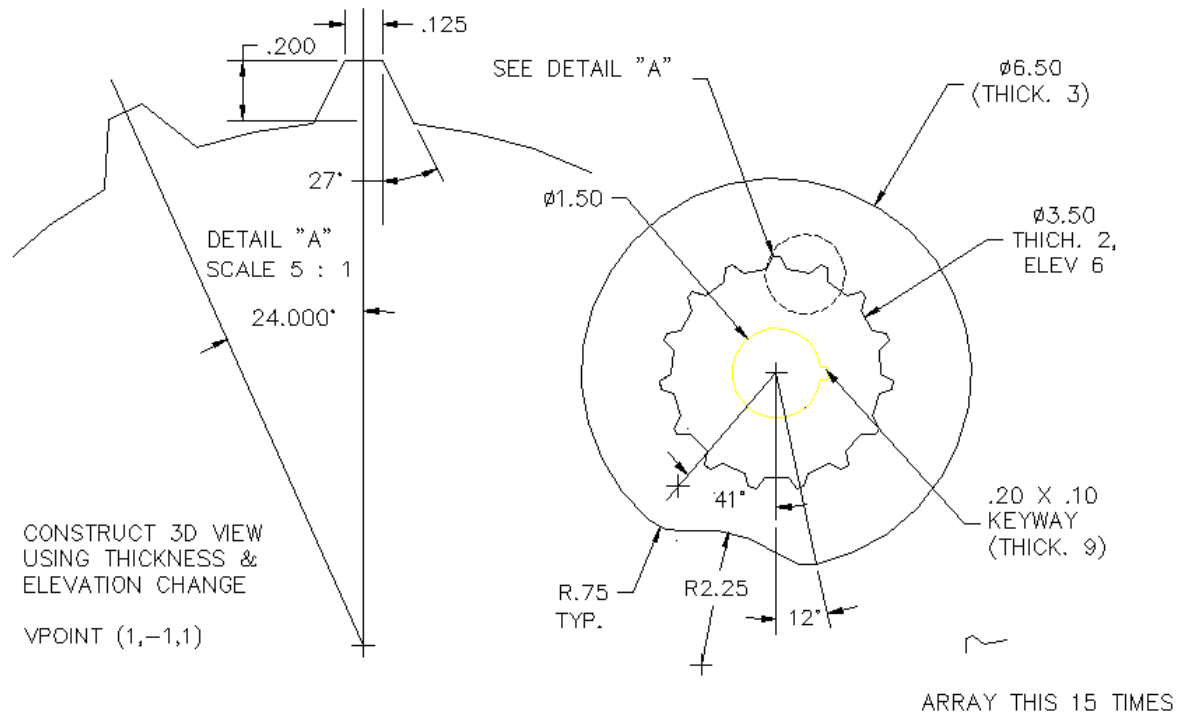*Construct a 3D solid model using Thickness and elevation change. (Figure 1-1)*

.125

.200

SEE DETAIL "A"

ø6.50
(THICK. 3)

27°

ø1.50

DETAIL "A"
SCALE 5 : 1

ø3.50
THICH. 2,
ELEV 6

24.000°

CONSTRUCT 3D VIEW
USING THICKNESS &
ELEVATION CHANGE

VPOINT (1,−1,1)

41°

.20 X .10
KEYWAY
(THICK. 9)

R.75
TYP.

R2.25

12°

ARRAY THIS 15 TIMES

Figure 1-1

Lesson 02
Creating Primitives
BOX

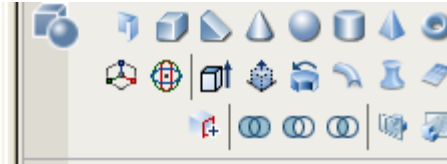⊗ **Modeling toolbar:**

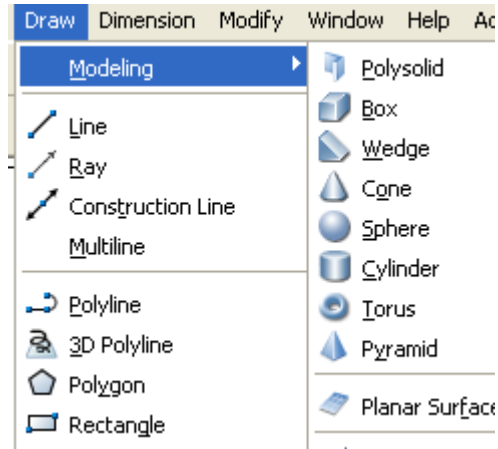⊗ **Draw menu:** Modeling » Box

⌨ **Command entry: box**

⊗ **Dashboard:** 3D Make panel, Box

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

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

2. *Use the pull-down menu or the toolbar to select the desired command. You may also type in 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>
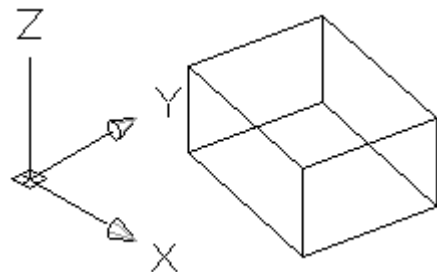Regenerating model.

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 [2Point]: **2** <Enter>

Lesson 03
Creating Primitives
CYLINDER & WEDGE

🕸 Modeling toolbar:

🕸 Draw menu: Modeling » Cylinder

⌨ Command entry: **cylinder**

🕸 Dashboard: 3D Make panel, Cylinder

🕸 Modeling toolbar:

🕸 Draw menu: Modeling » Wedge

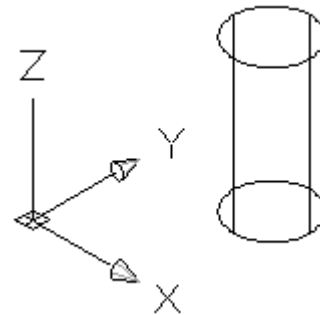⌨ Command entry: **wedge**

🕸 Dashboard: 3D Make panel, 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 [2Point/Axis endpoint] <2.0000>: **4** <Enter>

Command: wedge <Enter>
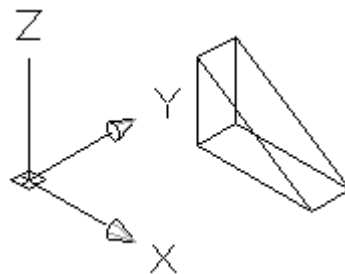Specify first corner or [Center]: <Enter>
Specify other corner or [Cube/Length]: **L** <Enter>
Specify length <3.0000>: **3** <Enter>
Specify width <4.0000>: **1** <Enter>
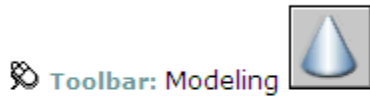Specify height or [2Point] <2.0000>: **2** <Enter>

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

⊗ Toolbar: Modeling

⊗ Menu: Draw ➤ Modeling ➤ Cone

⌨ Command entry: **cone**

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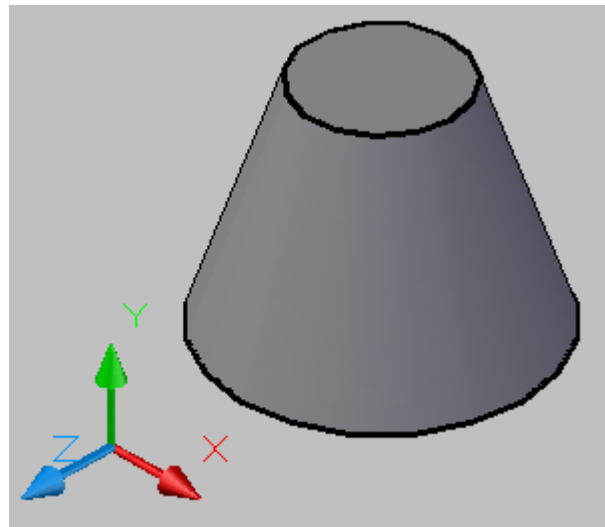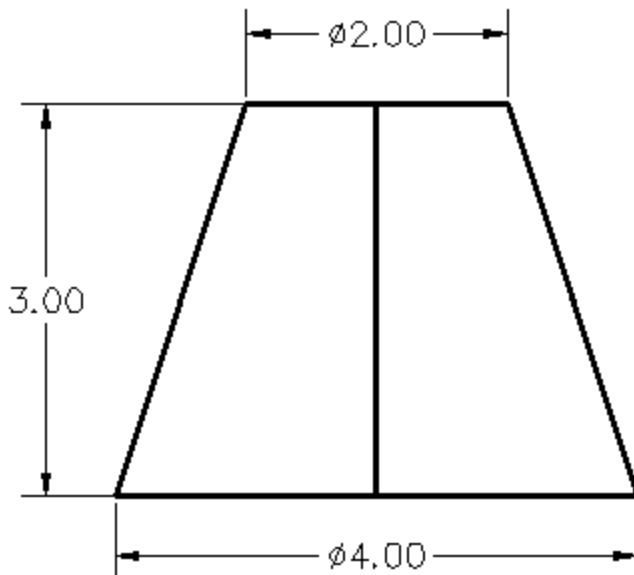
Command: _cone <Enter>
Specify center point of base or [3P/2P/Ttr/Elliptical]: **5,5** <Enter>
Specify base radius or [Diameter] **2** <Enter>
Specify height or [2Point/Axis endpoint/Top radius] <0000>: _top
Specify top radius **1** <Enter>
Specify height or [2Point/Axis endpoint] **3** <Enter>

Lesson 05
Creating Primitives
PYRAMID

▧ Modeling toolbar:

▧ Draw menu: Modeling » Pyramid

▦ Command entry: **pyramid**

▧ Dashboard: 3D Make panel, Pyramid

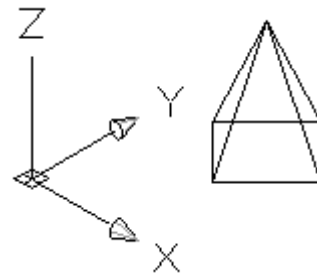Command: **pyramid** <Enter>
4 sides Circumscribed
Specify center point of base or [Edge/Sides]:
{Pick a point anywhere on screen}
Specify base radius or [Inscribed] <1.0000>: **1** <Enter>
Specify height or [2Point/Axis endpoint/Top radius] <2.0000>: **3**
<Enter>

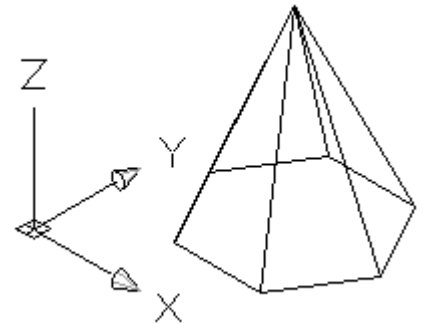Command: **pyramid** <Enter>
4 sides Circumscribed
Specify center point of base or [Edge/Sides]: **S** <Enter>
{Pick a point anywhere on screen}
Specify base radius or [Inscribed] <1.0000>: **2** <Enter>
Specify height or [2Point/Axis endpoint/Top radius] <2.0000>: **5**
<Enter>

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 or [Edge/Sides]:
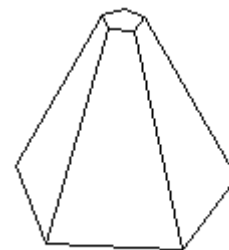*Pick a point anywhere on screen}*
Specify base radius or [Inscribed]: <Enter>
Specify height or [2Point/Axis endpoint/Top radius] <-9.2512>: **t**
<Enter>
Specify top radius <0.0000>: **.5** <Enter>
Specify height or [2Point/Axis endpoint] <-9.2512>: **6** <Enter>

9

Lesson 06
Creating Primitives
SPHERE - *TORUS*

⊗ **Modeling toolbar:**

⊗ **Draw menu:** Modeling » Sphere

⌨ **Command entry: sphere**

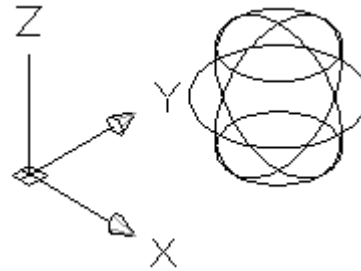⊗ **Dashboard:** 3D Make panel, Sphere

⊗ **Modeling toolbar:**

⊗ **Draw menu:** Modeling » Torus
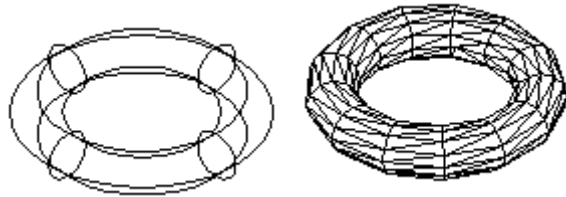
⌨ **Command entry: torus**

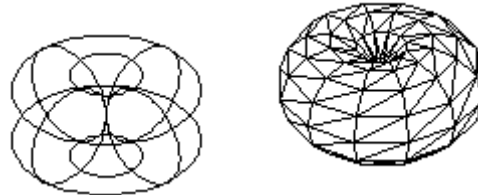⊗ **Dashboard:** 3D Make panel, Torus

Command: **sphere** <Enter>
Specify center point or [3P/2P/Ttr]:
{Pick a point anywhere on 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 or [Diameter] <3.3282>: 4
Specify tube radius or [2Point/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 [2Point/Diameter]
<6.0000>: 4
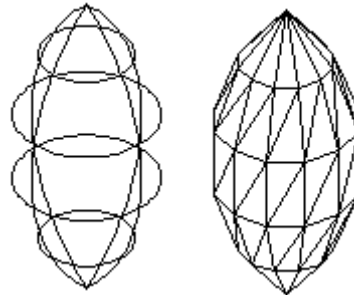
Command:
TORUS
Specify center point or [3P/2P/Ttr]:
{Pick a point anywhere on screen}
Specify radius or [Diameter] <4.0000>: -1
Specify tube radius or [2Point/Diameter] <2.0000>:
1.5

Lesson 07
Creating 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 or [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>
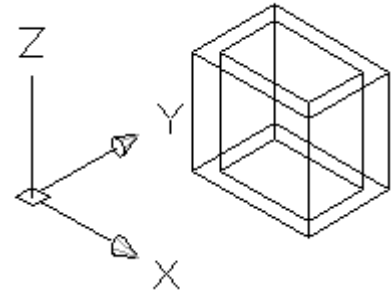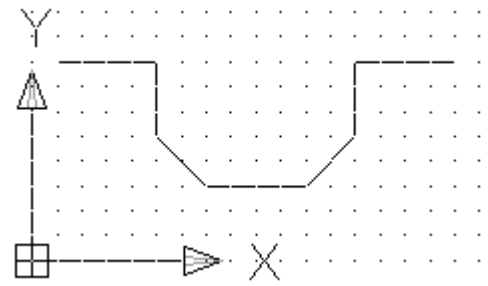Specify next point or [Arc/Close/Undo]: **c** (Figure A)

Figure A

*Using the POLYLINE command (PL), to create the
figure "A".*

*Use POLYSOLID command; select "Object" option to
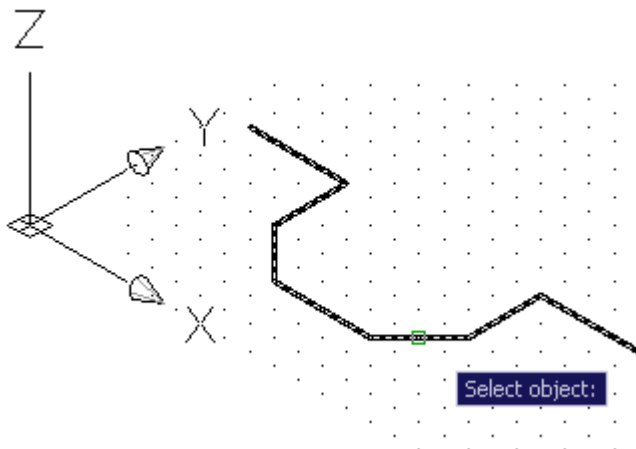construct Figure "C".*

Grid space = 0.25   Figure B

Select object:

Figure C

Lesson 08
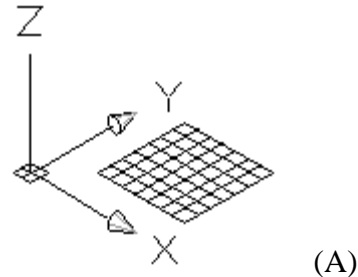## *Constructing a Planar Surface*

⊗ **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)

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

(A)

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

Figure A

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*}

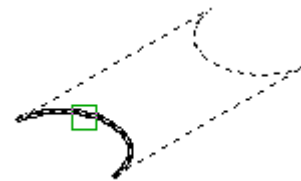*Use the ARC 3-point option to draw two arc as shown.*
(Figure B)

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
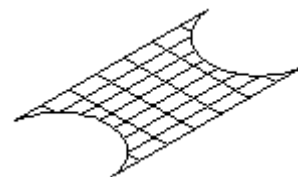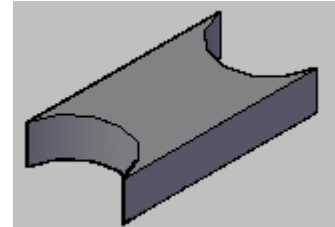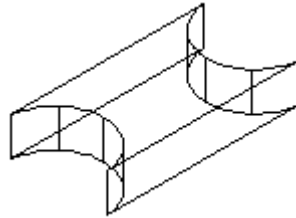Select objects: {*select the arc}* 1 found, 4 total
Select objects: <Enter>

Figure C

Modify menu: 3D Operations » Thicken

Command entry: **thicken**

Dashboard: 3D Make panel (click icon to expand), Thicken Surface

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)

Figure D

Modify menu: 3D Operations » Convert to Surface

Command entry: **convtosurface**

Dashboard: 3D Make panel (click icon to expand), Convert to Surface

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

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

Note You cannot use CONVTOSOLID 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 0.0000
Specify next point or
[Arc/Halfwidth/Length/Undo/Width]:
@5<90 <Enter>
Specify next point or
[Arc/Close/Halfwidth/Length/Undo/Width]:
a <Enter>
Specify endpoint of arc or
[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: **1**
<Enter>
Specify endpoint of arc or
[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: **2**
<Enter>
Specify endpoint of arc or
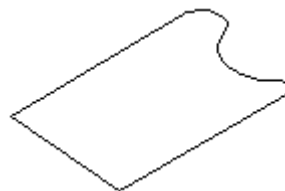
Figure E   Figure F
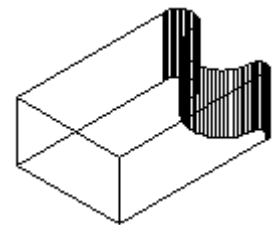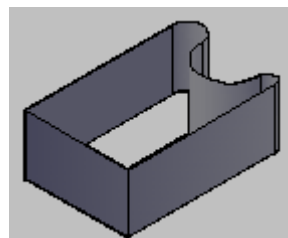
[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: **.5** <Enter>
Specify endpoint of arc or [Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: **L** <Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **@5<-90** <Enter
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **c**

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 settings.

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>
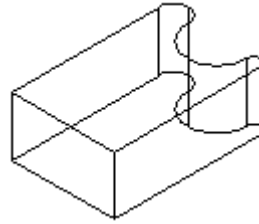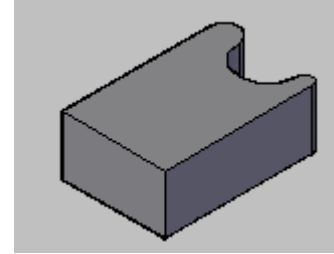Specify length: **3.5**   <Enter>
Specify width: **.5**   <Enter>
Specify height: **1.5**   <Enter>
Command: **z**   <Enter>
ZOOM
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window] <real time>: 1 <Enter>

Command: **cylinder** <Enter>
Current wire frame density: ISOLINES=4
Specify center point for base of cylinder or [Elliptical] <0,0,0>: **5.75,3.75,1.5**   <Enter>
Specify radius for base of cylinder or [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)}*
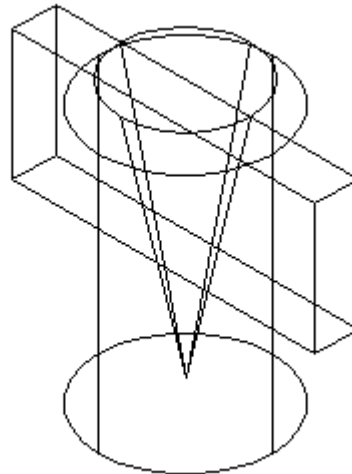
Command: union   <Enter>
Select objects: *(left click on cylinder A)* 1 found
Select objects: *(left click on box B)* 1 found, 2 total
Select objects: <Enter>

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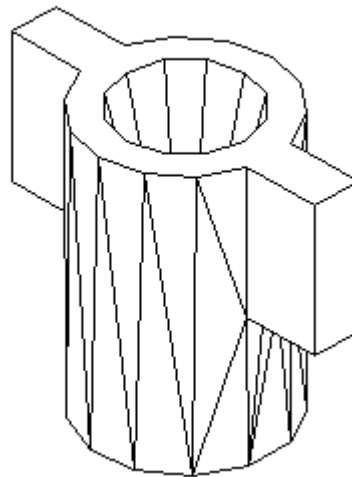Select objects: <Enter>
Select solids and regions to subtract ..   (left click on cone)
Select objects: 1 found
Select objects: <Enter>

Command: **Hide**   <Enter>

Your drawing should look like the figure shown:
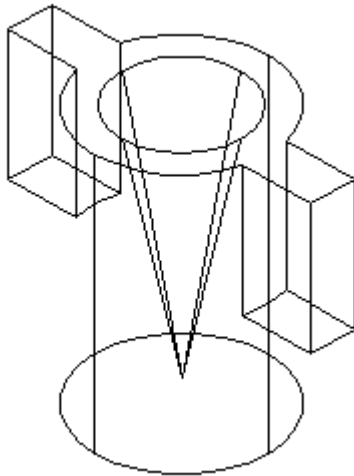
                                                                                                                    hide

Calculate the volume:

Mass:	9.2044
Volume:	9.2044

Command: massprop <Enter>
Select objects: (left click on the object) 1
Select objects: <Enter>

     1.

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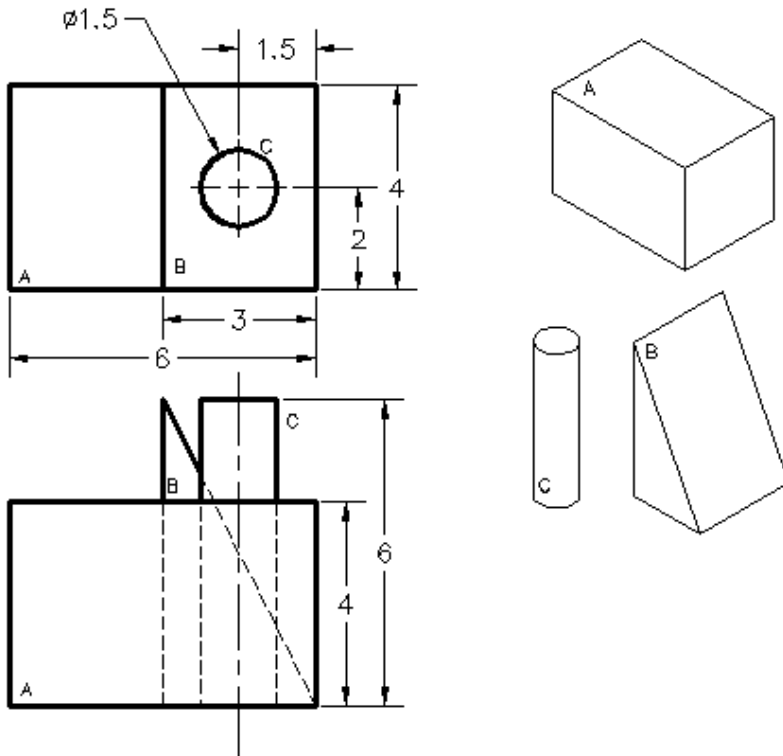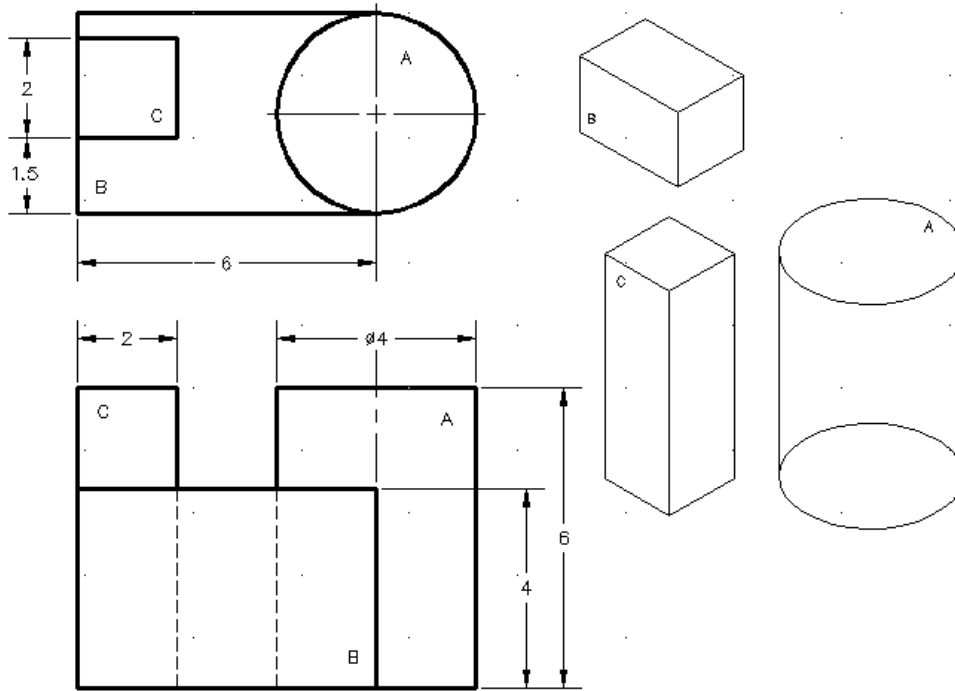

Exercise 9-2   *Perform the Boolean operations {(A + B) - C}:*

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 <u>upper left</u> corner and draw clockwise}*
Current line-width is 0.0000
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: <Ortho on> **1.5** <Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **1** <Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **1.13**<Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **1**<Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **1.5**<Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **2**<Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **4.13**<Enter>
Close

Select the RIGHT UCS:
Command: **ucs** <Enter>
Current ucs name: *WORLD*
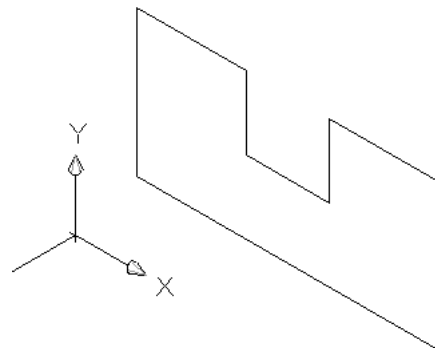Specify origin of UCS or
[Face/NAmed/OBject/Previous/View/World/X/Y/Z/ZAxis]
<World>: **G**<Enter>
Enter an option
[Top/Bottom/Front/BAck/Left/Right]<Top>: **R** <Enter>

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 <u>upper left</u> corner and draw clockwise}*
Current line-width is 0.0000
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: <Ortho on> **2.25** <Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **2** <Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **.5**<Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **1.25**<Enter>
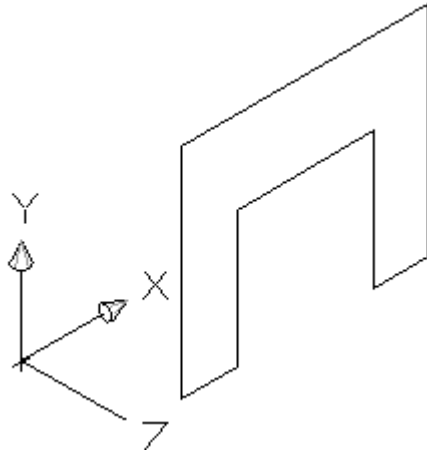Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **1.25**<Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **1.25**<Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: **.5**<Enter>
 Type **"c"** <Enter> to Close

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>
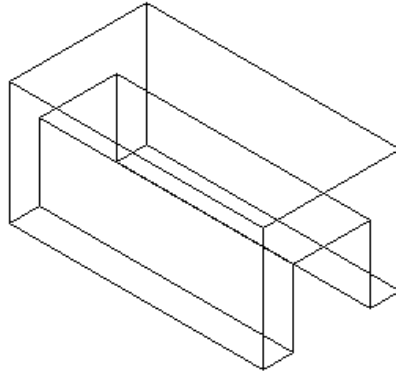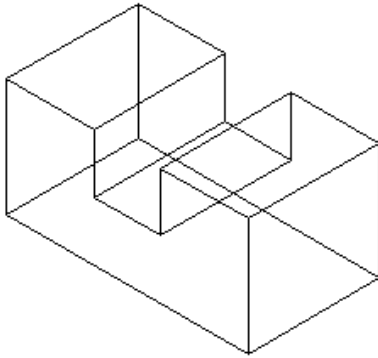
Command: **ext** <Enter>
EXTRUDE
Current wire frame density: ISOLINES=4
Select objects: 1 found {Select the side view profile)
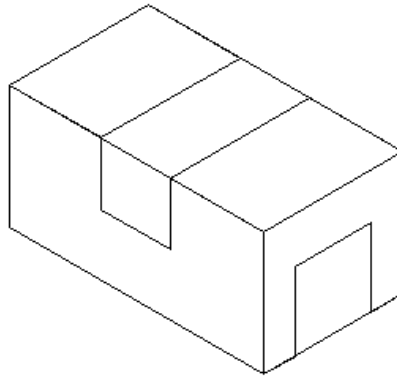Select objects: <Enter>
Specify height of extrusion or [Path]: **4.13** <Enter>
Specify angle of taper for extrusion <0>: <Enter>

Move these two solid objects to the position as shown:
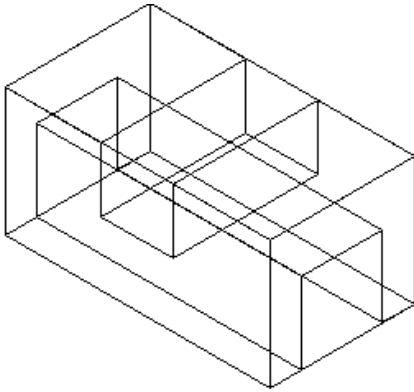
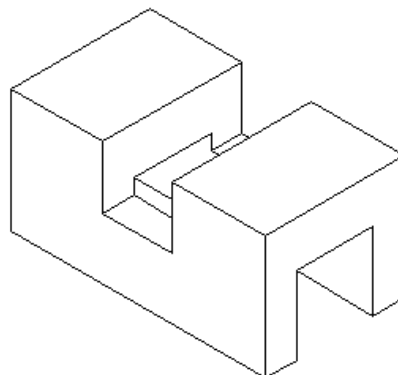before hide   after hide
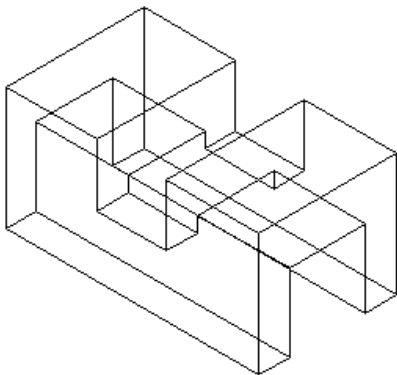
USE INTERSECT COMMAND:
Command: **intersect** <Enter>
Select objects: 1 found {Select both solid objects)
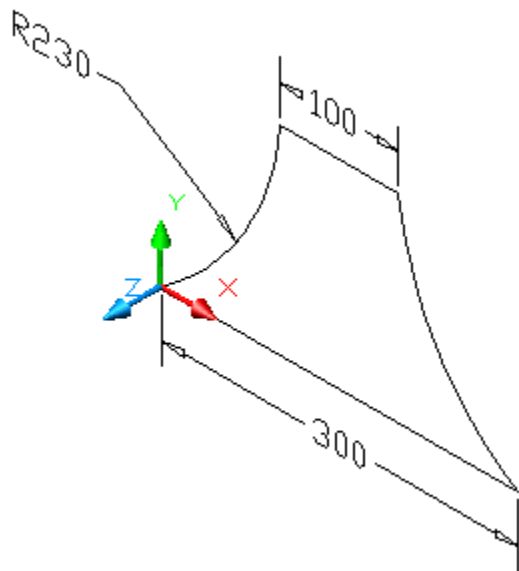Select objects: 1 found, 2 total
Select objects: <Enter>
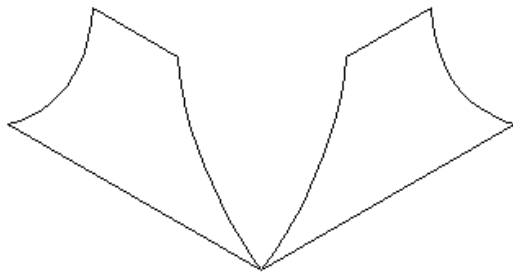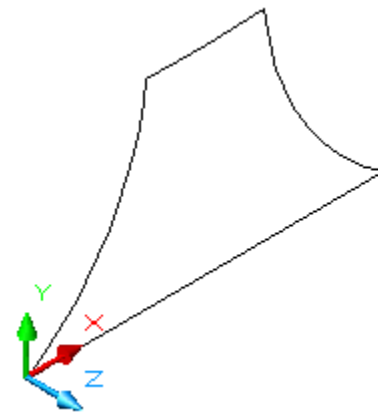
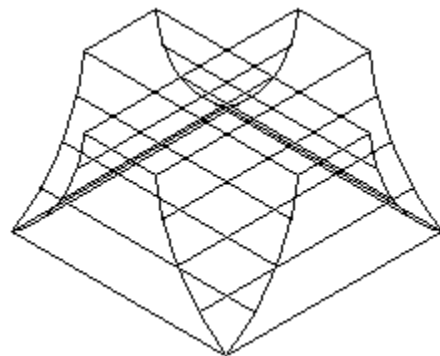before hide   after hide

EXERCISE 10-1
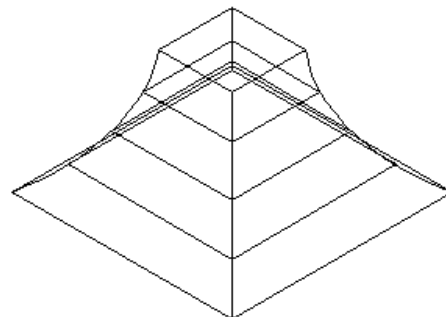

1. *Draw this figure on the front UCS.*

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

3.

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

Lesson 11
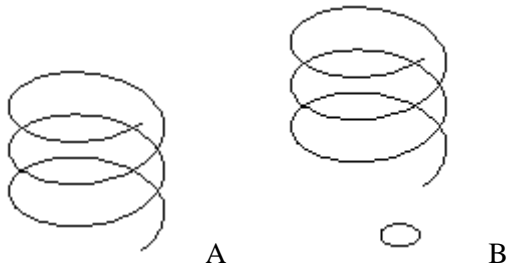## *Creating Helix*

Modeling toolbar

Command entry: **HELIX**

Dashboard: 3D Make panel (click icon to expand), 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 careen}*
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/tWist] <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 sweep path or [Alignment/Base point/Scale/Twist]:   {pick the helix}

A   B

C
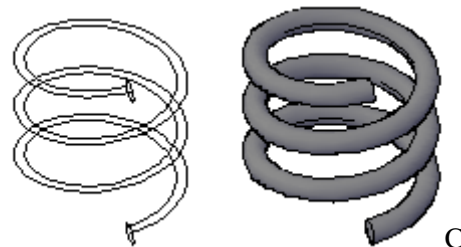
Lesson 12
Creating 2D Region Model

🔊 **Draw toolbar:**

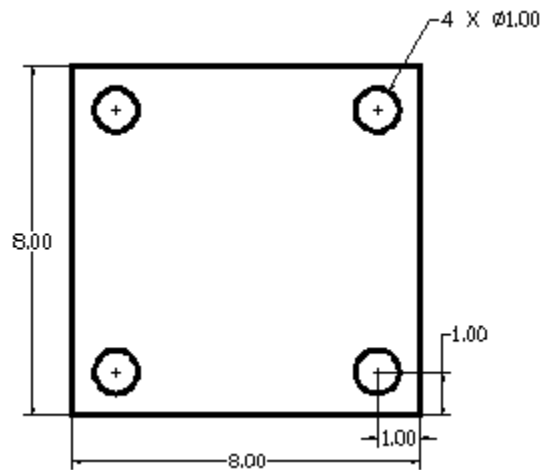🔊 **Draw menu:** Region

⌨ **Command entry: region**

Construct the following 2D shape and covert 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
[2dwireframe/3dwireframe/3dHidden/Realistic/Conceptual/Other]
<2dwireframe>: _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
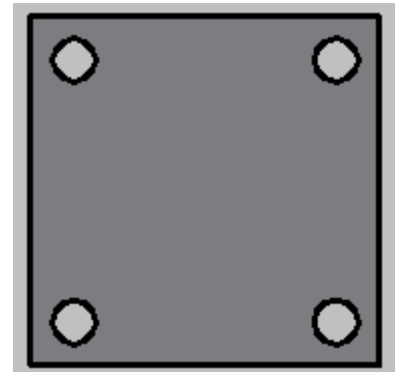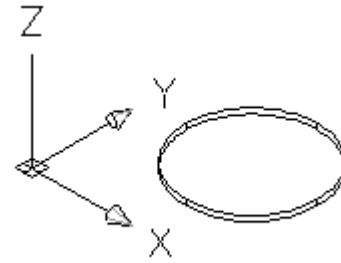Select objects: <Enter>

Calculate the area of the region.

Lesson 13
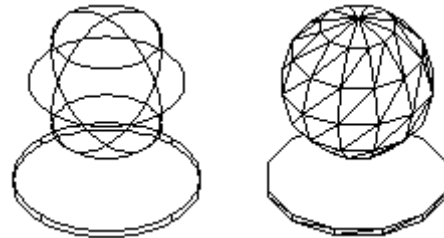## *3D Coordinates*

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 height or [2Point/Axis endpoint] <-3.0000>: **.5** <Enter>

(A)

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

(B)

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 or [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>
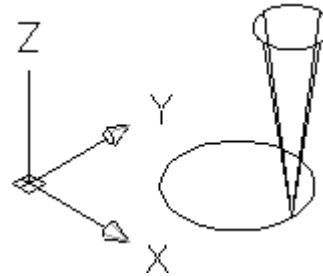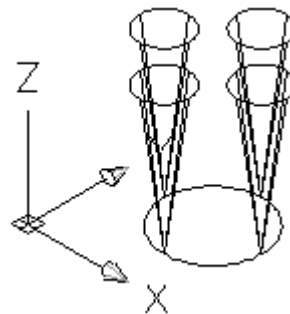Specify height or [2Point/Axis endpoint/Top radius]
 <-3.0000>: **-3** <Enter>

(C)

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

*Rotate each cone 15˚ from the vertical axis.
 (figure E)*

(D)

Command: _3drotate <Enter>
Current positive angle in UCS:
ANGDIR=counterclockwise
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.

(E)
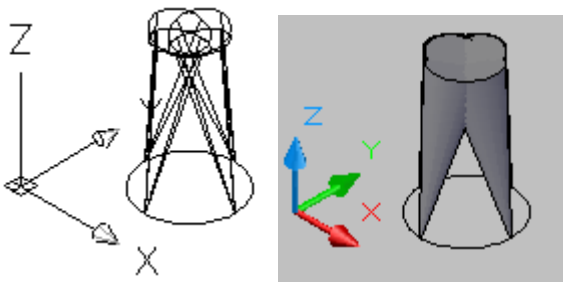
Command: **vpoint** <Enter>

Current view direction: VIEWDIR=1.0000,-
1.0000,1.0000
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)

(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 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>

(A)

(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}
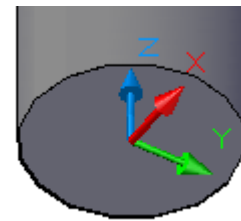
(B)

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>

C

D
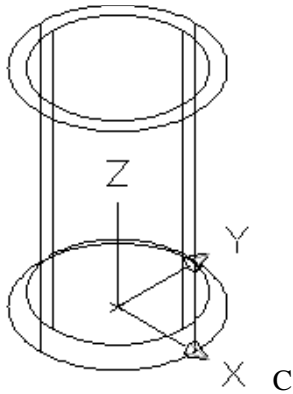
Command: cylinder <Enter>
Specify center point of base or [3P/2P/Ttr/Elliptical]: **0,0** <Enter>
Specify base radius or [Diameter] <0.00>: **.25**<Enter>
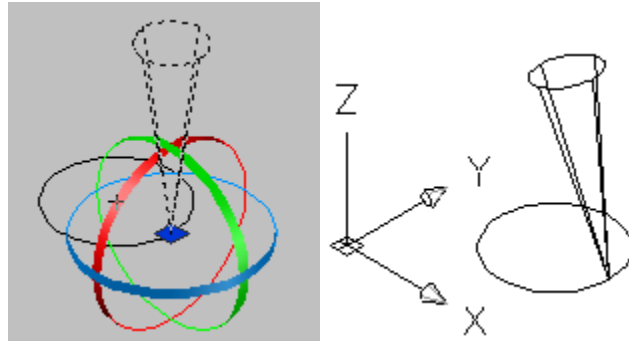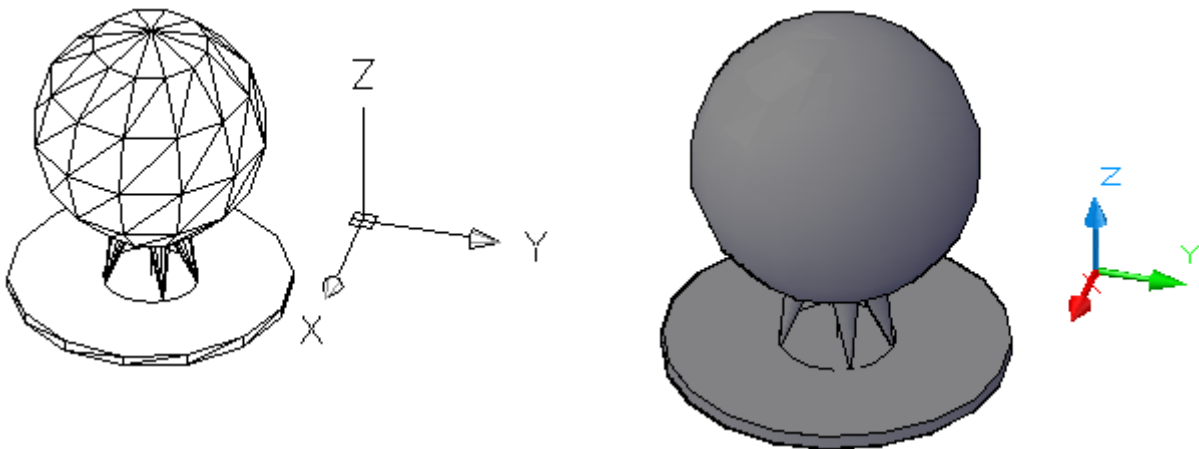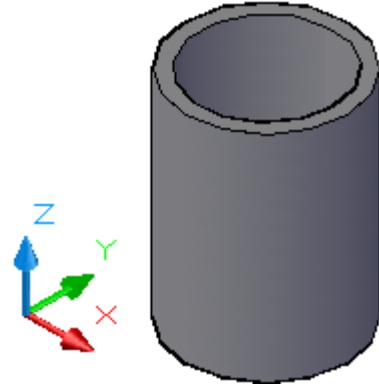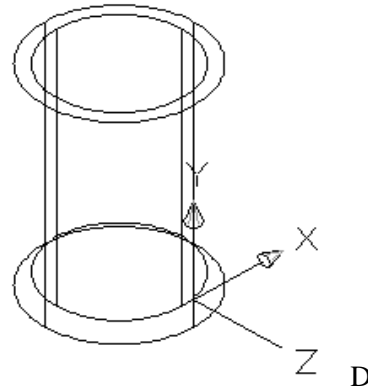Specify height or [2Point/Axis endpoint] <5.0000>: **2.5** <Enter>

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] <0.2500>: **.125** <Enter>
Specify height or [2Point/Axis endpoint] <2.5000>: **-3** <Enter>
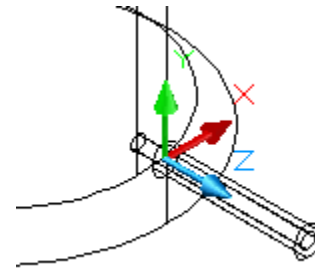
(Figure E)

E

Subtract these two cylinders to create .125 diameter tube. {Figure F)

F
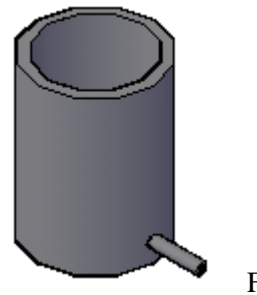
Text and Dimensions in 3D

-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.
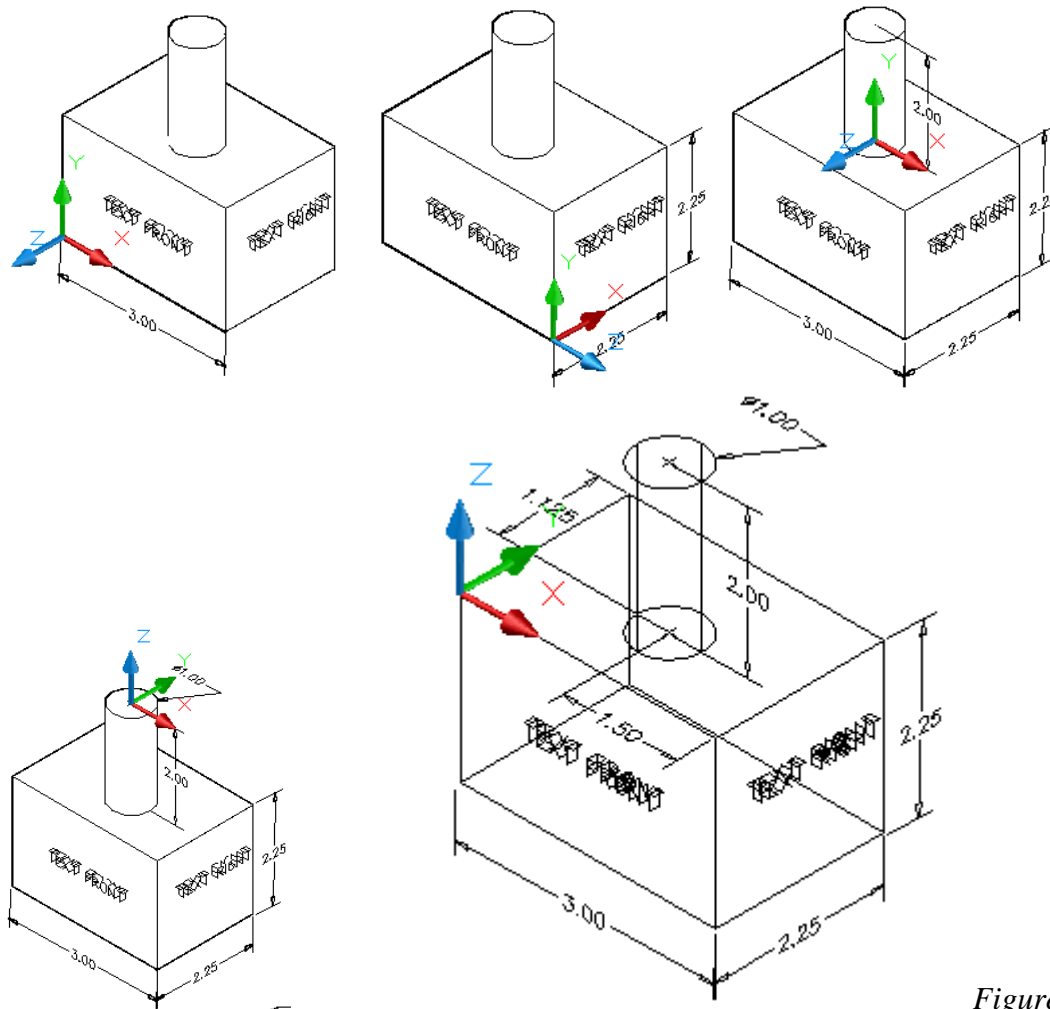
Figure -1

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.
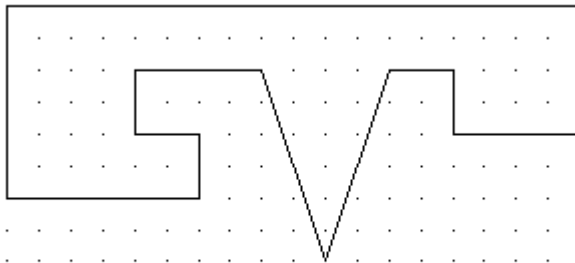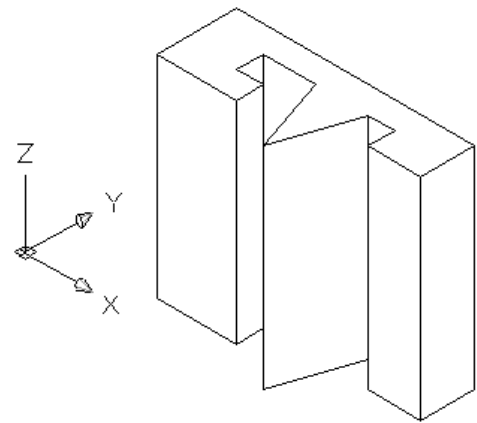
GRID SPACE= 0.25

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.
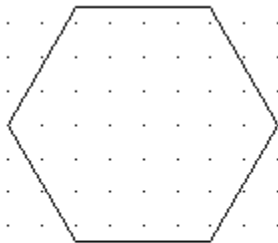
Grid space = 0.25
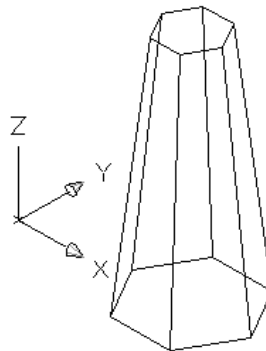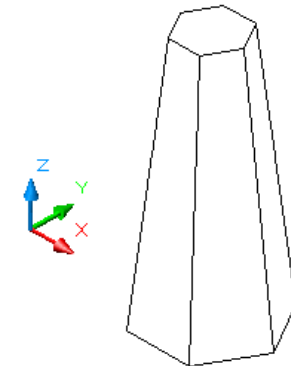
Wireframe   Hide

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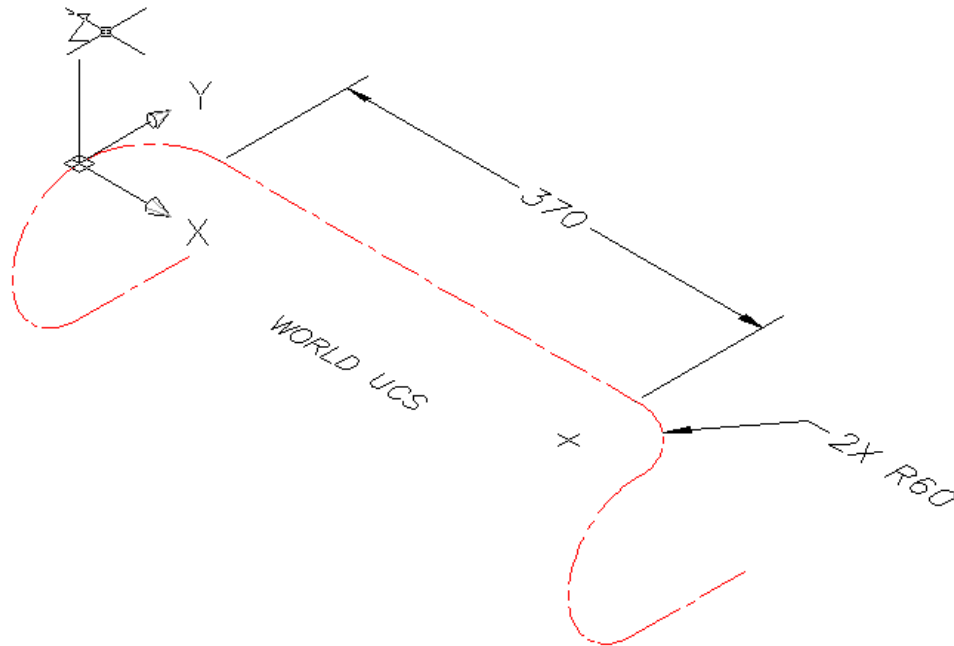
1. Pline with 370 length pline (arc option) or ARC with 60 radii. On each end on WCS.
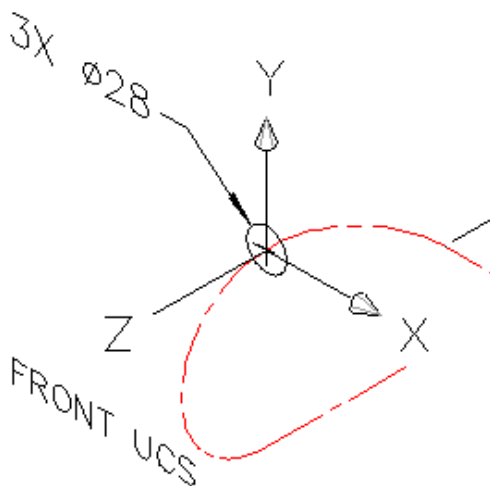
2. Create three circles on the FRONT UCS.
3. Create the drop portion of the bar using the pline command on the SIDE UCS.
4. Extrude each circle along each pline.

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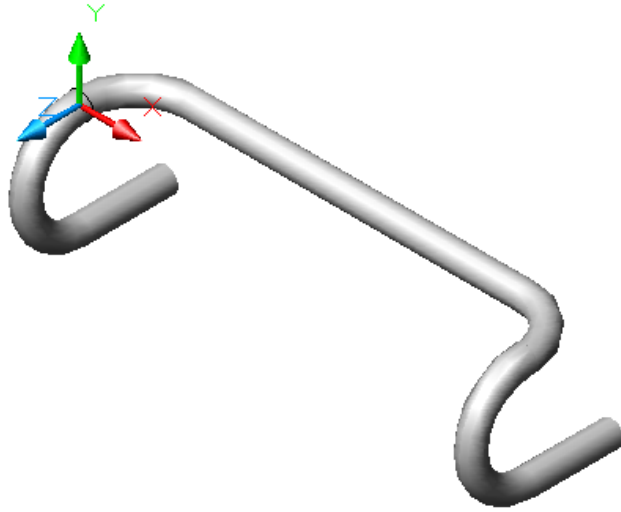
EXTRUDE (*path*)

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 step9.

     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".
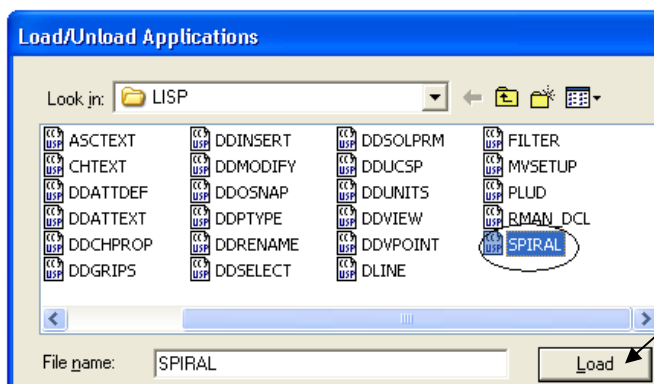
6. *Highlight "SPIRAL", then click on "load" tab*

7. *From View, select SE Isometric:*
8. *On Command line type 3d spiral, and follow the step below:*

Command: 3dspiral <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
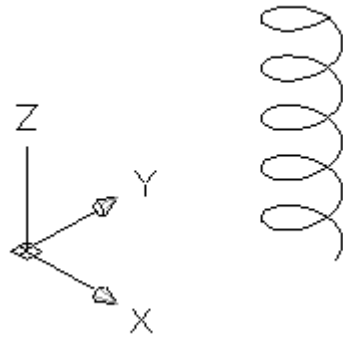
9. *At this time you need to change you UCS to the FRONT view and draw a circle with center point at **one end of the path** and radius of **0 .125** (See the figure).*

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 or [Diameter]: .125 <Enter>

On the FRONT UCS draw diameter 0.25
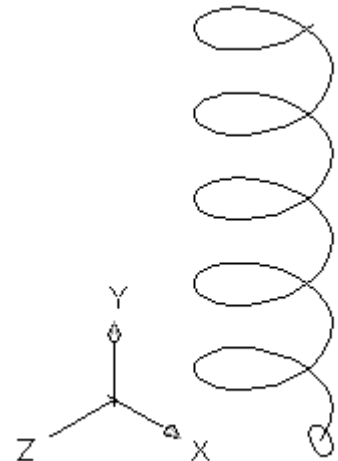
10. Extrude the 0.25 diameter circle along the spiral path.
11. Determine the volume and print your file. (refer to printing 3D handout)
12. Save your drawing.
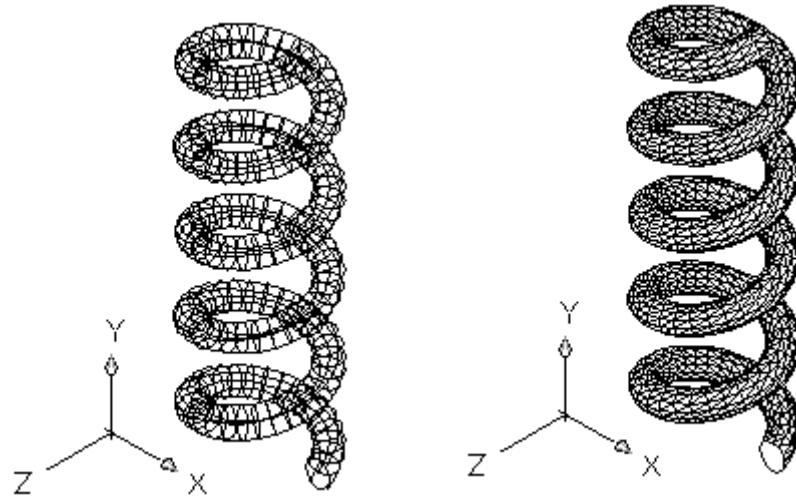
Exercise 15-1

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

Figure 15-1

33

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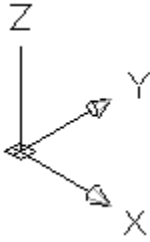
This is the same as selecting "SE Isometric" from the pull down menu "view".

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:
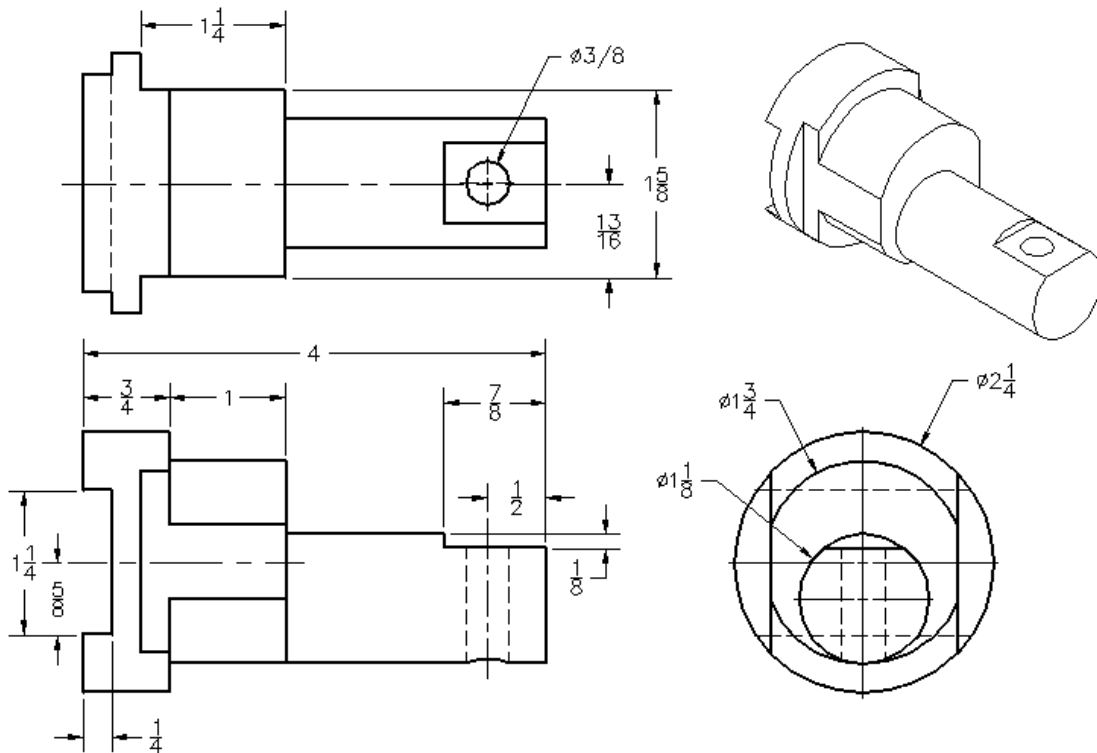
Command: **pL**   <Enter>
PLINE
Specify start point:5,5   <Enter>
Current line-width is 0.0000
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: @3,0   <Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: @.5<90   <Enter>
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: @-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>
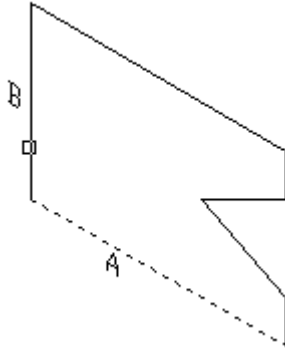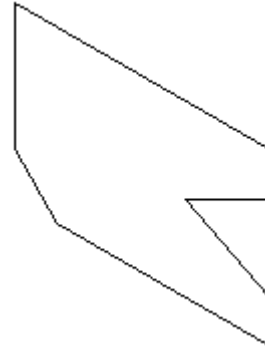
Figure 6-1 ˈ ∨   Figure 6-2

Command: **cha**
CHAMFER
(TRIM mode) Current chamfer Dist1 = 0.0000, Dist2 = 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 "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 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 Dist1 = 0.5000, Dist2 = 0.5000
Select first line or [Polyline/Distance/Angle/Trim/Method]: *{left click on line "A"*
Figure 6-3}
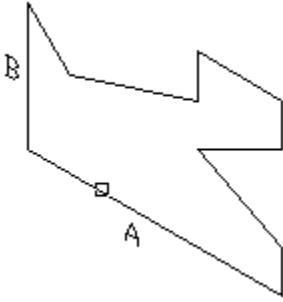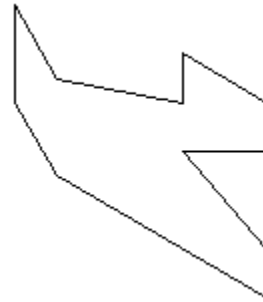Select second line: *{left click on line "B" Figure 6-3}*

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:

Figure 6-5

Command: **m**   <Enter>
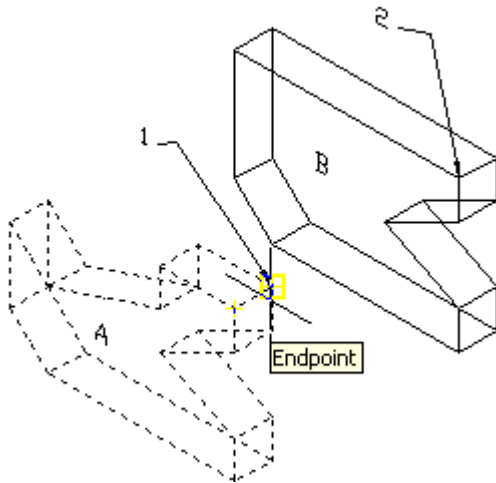MOVE
Select objects: *{ left click any place on model "A" Figure 6-5 }* 1 found
Select objects: <Enter>
Specify base point or displacement: end
of *{ left click at point "1" part "A" Figure 6-5 }*
Specify second point of displacement or <use first point as displacement>:
end of *{ left click at point "2" part "B" Figure 6-5 }*

Command: **hide** <Enter>
Regenerating model.
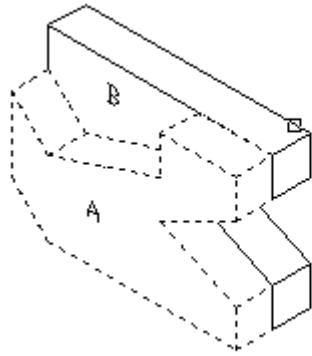
Figure 6-6   Figure 6-7

Command: **union** <Enter>
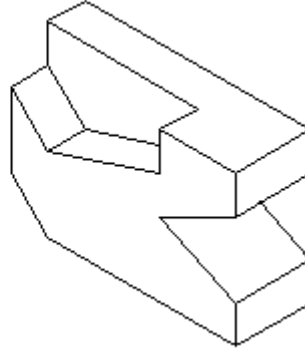Select objects: *{ left click on part "A" Figure 6-6 }*   1 found
Select objects: *{ left click on part "B" Figure 6-6 }*   1 found, 2 total
Select objects: <Enter>
Your final drawing shown in figure 6-7

Use MASSPROP Command to calculate the volume.
Use TEXT Command and write the result on your final printed drawing.
<u>*Note*</u>*: Refer to instruction on "Printing 3D Model" .*

4. Tutorial Exercise 15-3

Start AutoCAD with the default settings.

Use the VPOINT Command to change the viewing option:
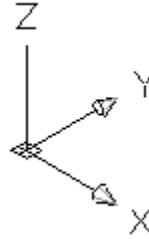Command: **vpoint**   <Enter>

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 right side view:

ommand: **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>
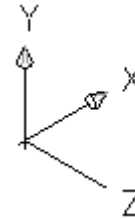Command:
UCS
Current ucs name: *NO NAME*
Specify origin of UCS or
[Face/NAmed/OBject/Previous/View/World/X/Y/Z/ZAxis]
<World>: **y** <Enter>
Specify rotation angle about Y axis <90>:<Enter>

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>

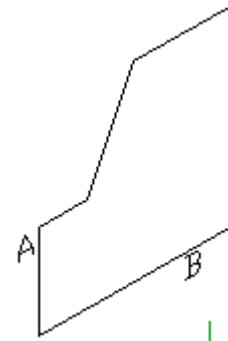Figure1

Command: **cha** <Enter>
CHAMFER
(TRIM mode) Current chamfer Dist1 = 0.0000, Dist2 = 0.0000
Select first line or [Undo/Polyline/Distance/Angle/Trim/mEthod/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/mEthod/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)}
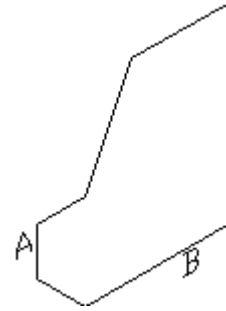Your drawing should look like the one shown in figure -2.

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>
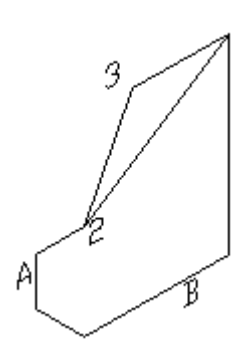
Figure-3

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**
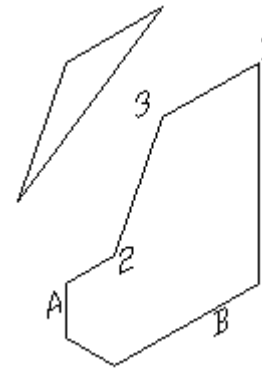<Enter>

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>:
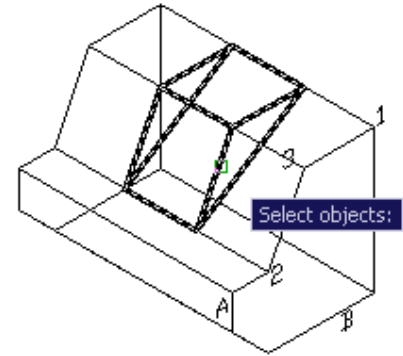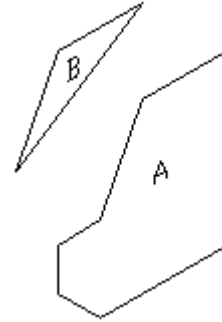**-1** <Enter>   Figure 5

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}
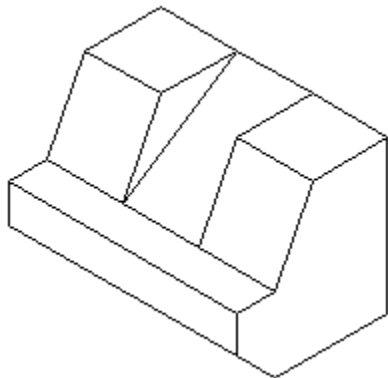
Figure 5

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

Figure 6

Figure 8

Use MASSPROP Command to calculate the volume.
Use TEXT Command and write the result on your final printed drawing.
<u>*Note*</u>*: Refer to instruction on "Printing 3D Model".*
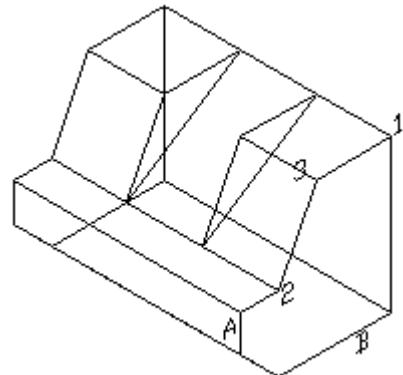
Figure 7

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 or define axis by [Object/X/Y/Z]
<Object>: **o** <Enter>
Select an object: *(click on the line "A" towards end; figure 2)*
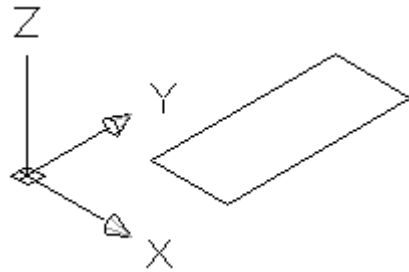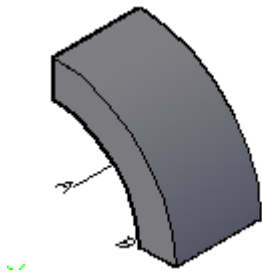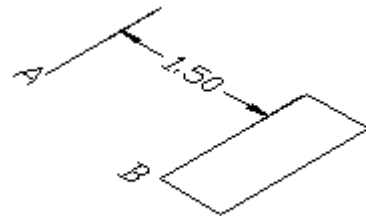Specify angle of revolution or [STart angle] <360>: **90**
<Enter>

Figure 2
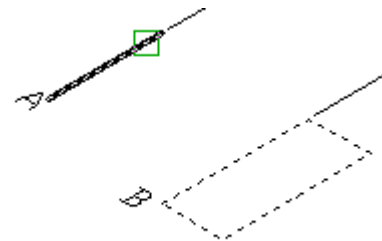
Figure 3

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 from 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; @
ROTATE3D: L;X; 0,0; 45
COLOR: 2
CIRCLE CNR 0,6,.625 & R=2
ROTATE3D: L; X; 0,0; 45
COLOR: by layer

SLICE: (select pipe): Object;
    (select yellow circle)
UNION: All
ROTATE3D ; L; z: 0,0; -60
```


Figure 16-1

Lesson 17
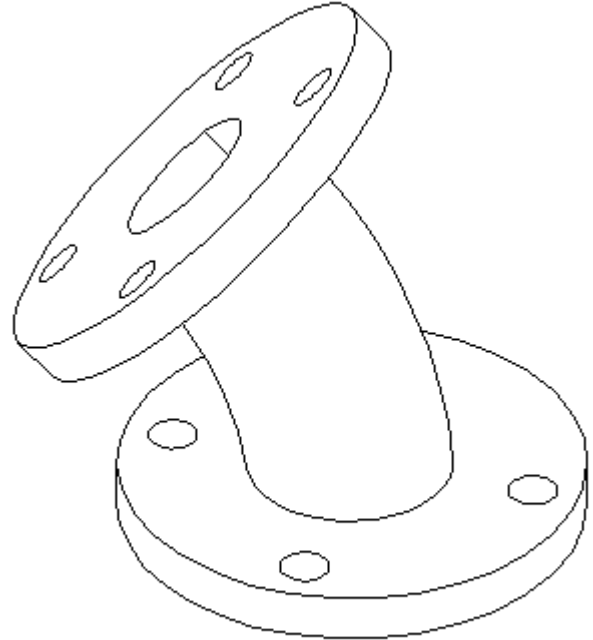## *Sweep Command*

📎 Modeling toolbar:

📎 Draw menu: Modeling » Sweep

⌨ Command entry: **sweep**

📎 Dashboard: 3D Make panel, Sweep

With the SWEEP command, you can create a new solid or 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 figures 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}*

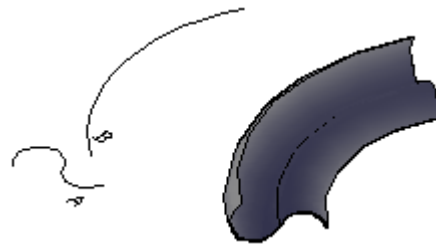*Figure 1*

   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>
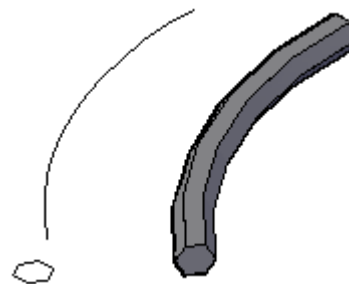Select sweep path or [Alignment/Base point/Scale/Twist]:
{select the arc }

Figure 2

   c.   *Sweeping the close shape profile with **Twis**t 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)

Figure 3

   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
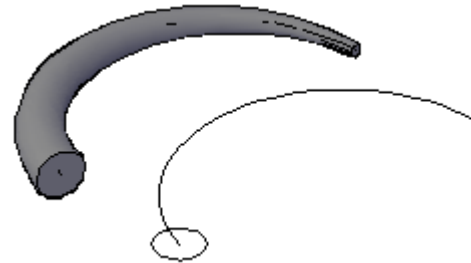 [Alignment/Base point/Scale/Twist]: *{select the arc}*   *Figure 4*

(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>
Specify base point: _nod of {*select the point*}
Select sweep path or [Alignment/Base
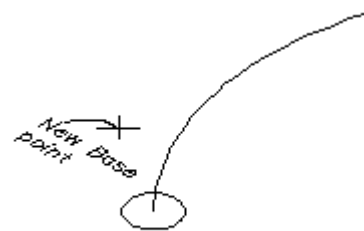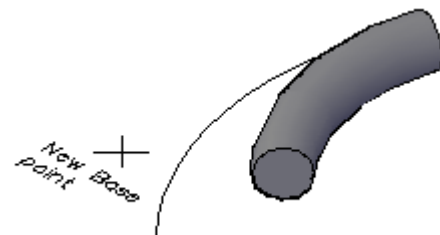point/Scale/Twist]: {*select the arc*}

(*Figure 5*)

Figure 5

Lesson 18
Loft Command

† Modeling toolbar:

† Draw menu: Modeling » Loft

“ Command entry: **loft**

† Dashboard: 3D Make panel, Loft

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 (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.

> *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>   (1)

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

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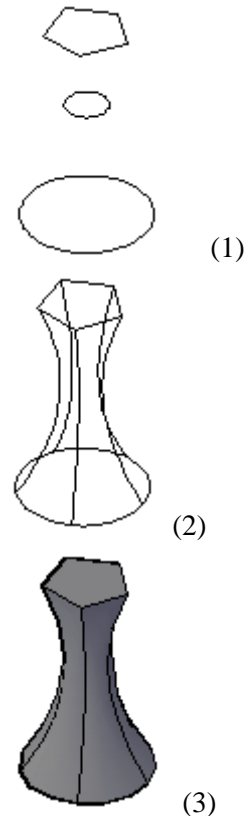1 found, 2 total
Select cross-sections in lofting order: *{select the large circle}*
 1 found, 3 total
Select cross-sections in lofting order: <Enter>
Enter an option [Guides/Path/Cross-sections only]
<Cross-sections only>: <Enter>   (Figure 2)
                                                       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

*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
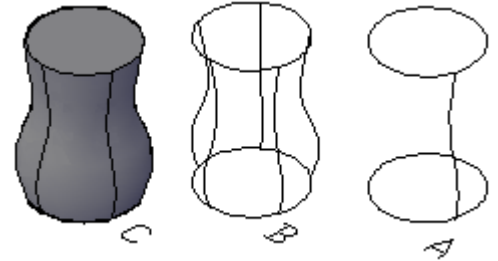only>: **g** <Enter>
Select guide curves: *{select the curve}*1 found
Select guide curves: *{select the curve}*1 found, 2 total
Select guide curves: *{select the curve}*1 found, 3 total
Select guide curves: *{select the curve}*1 found, 4 total   (4)
Select guide curves: *{select the curve}*1 found, 5 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-section 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 square}*1 found, 3 total
Select cross-sections in lofting order: *{select the large circle in the
middle}*1 found, 4 total
Select cross-sections in lofting order: *{select the small circle}*1 found, 5
total   (5)
Select cross-sections in lofting order: <Enter>
Enter an option [Guides/Path/Cross-sections only] <Cross-sections
only>: **p** <Enter>
Select path curve:*{select the helix}*
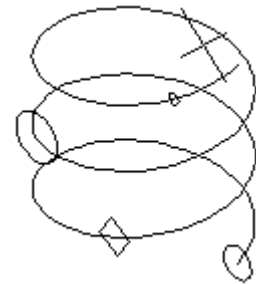
(6)

Lesson 19
3dalign Command

⊗ Modeling toolbar

⊗ Modify menu: 3D Operations » 3D Align

⌨ Command entry: **3dalign**

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 the base points, and the *X* and *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)
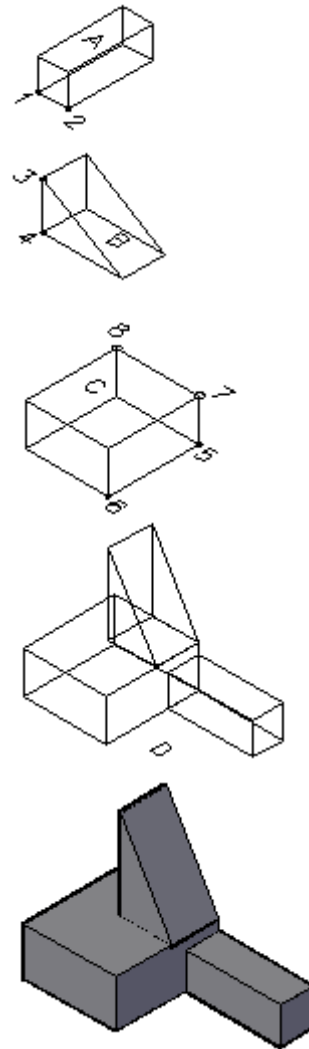
Lesson 20
Mirror3D- 3dAarray

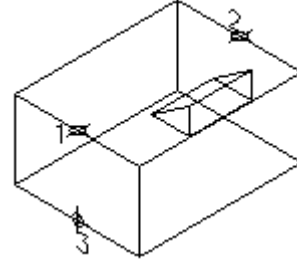   *a. Draw a box and a wedge with an approximate size as shown in figure 1.*

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] <3points>:
<Enter>
Specify first point on mirror plane: *{select the modepoint1}*
Specify second point on mirror plane: *{select the modepoint2}*
Specify third point on mirror plane: *{select the modepoint3}*
Delete source objects? [Yes/No] <N>: N <Enter>
(Figure 2)
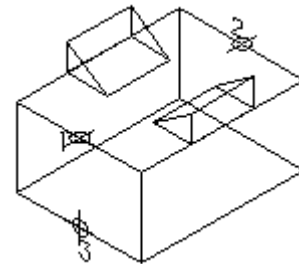
Figure 1

    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
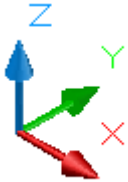
Note: Use SE Isometric view

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>
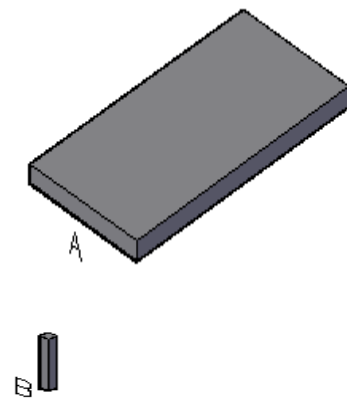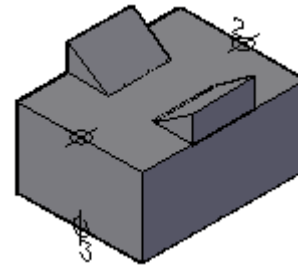
Select objects: 1 found   *{select all the boxes}*
Select objects: 1 found, 2 total
Select objects: 1 found, 3 total
Select objects: 1 found, 4 total
Select objects: 1 found, 5 total
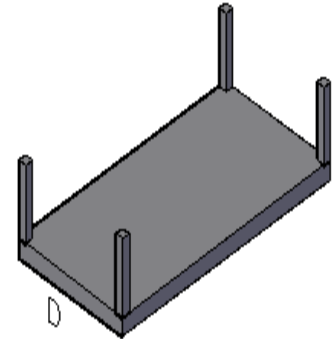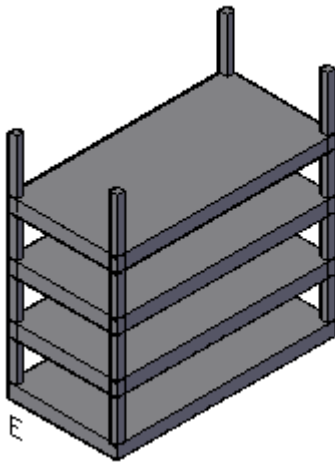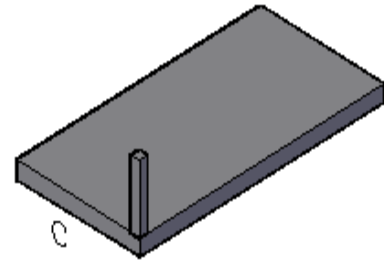Select objects: <Enter>

Enter the type of array [Rectangular/Polar] <R>: **R** <Enter>
Enter the number of rows (---) <1>: <Enter>
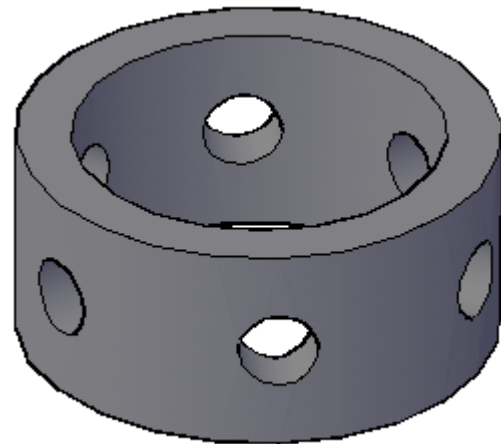Enter the number of columns (|||) <1>: <Enter>
Enter the number of levels (...) <1>: **4** <Enter>
Specify the distance between levels (...): **2** <Enter>

Exercise 20-1
Use the 3DARRAY (polar option), to create the model shown. Use your own dimensions proportion to this figure.

Lesson 21
Fillet- Chamfer

Modify toolbar:
Modify menu: Fillet
Command entry: **fillet**

Modify toolbar:
Modify menu: Chamfer
Command entry: **chamfer**

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

Command: **f** <Enter>
Current settings: Mode = TRIM, Radius = 0.0000
Select first object or [Undo/Polyline/Radius/Trim/Multiple]:
Enter fillet radius <0.0000>: **.25** <Enter>
Select an edge or [Chain/Radius]:
{select all the edges as highlighted in figure B}
Select an edge or [Chain/Radius]:
Select an edge or [Chain/Radius]:
Select an edge or [Chain/Radius]:
Select an edge or [Chain/Radius]:
Select an edge or [Chain/Radius]:
Select an edge or [Chain/Radius]:
Select an edge or [Chain/Radius]:
Select an edge or [Chain/Radius]:
Select an edge or [Chain/Radius]:
10 edge(s) selected for fillet.

Command: **cha** <Enter>

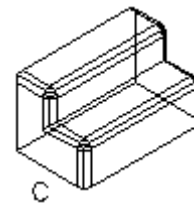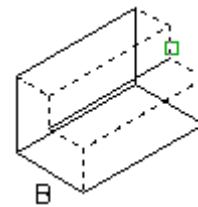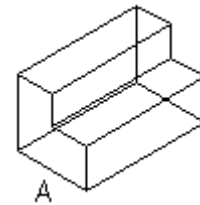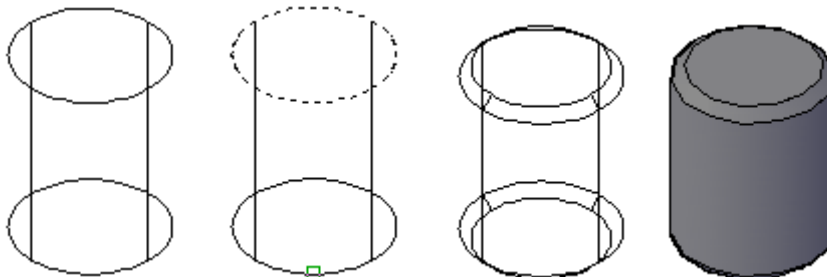(TRIM mode) Current chamfer Dist1 = 0.0000, Dist2 = 0.0000
Select first line or [Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple]:
Base surface selection...
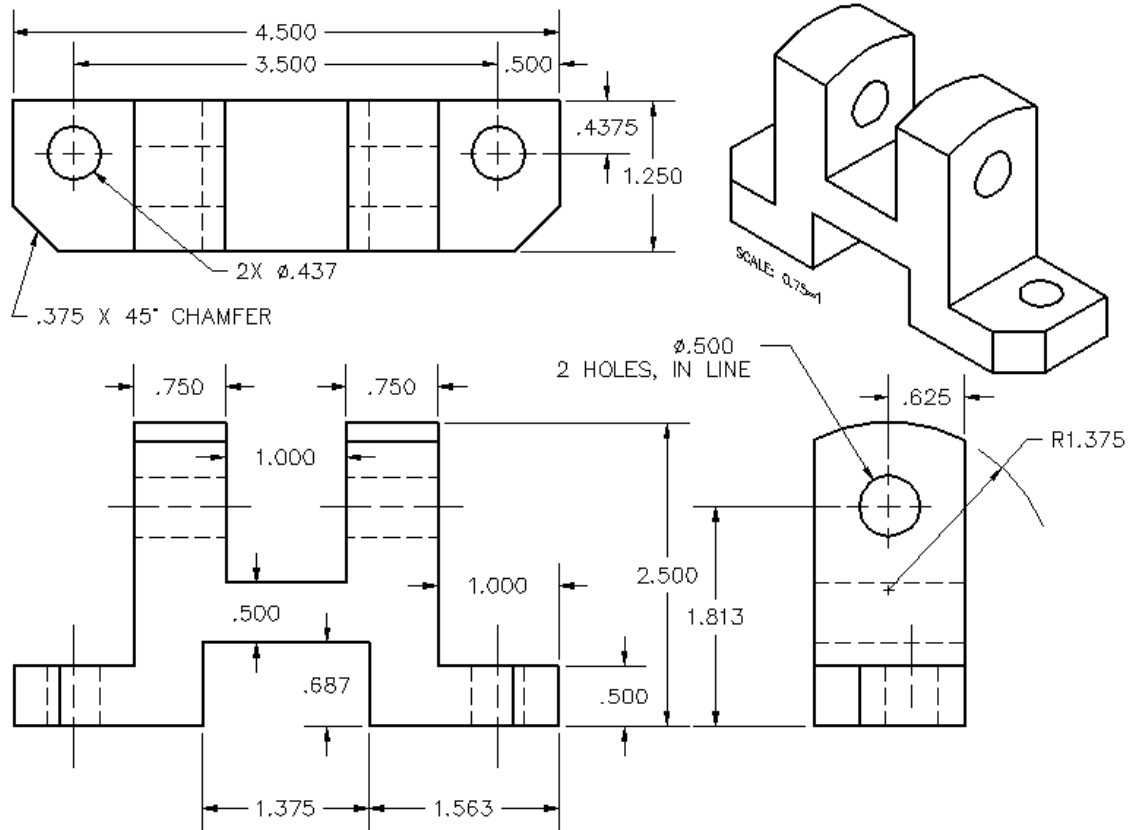Specify base surface chamfer distance <0.0000>: .5
Specify other surface chamfer distance <0.0000>: .5
Select an edge or [Loop]: Select an edge or [Loop]: Select an edge or
[Loop]:

50
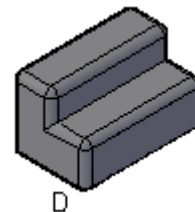
Exercise 21-1

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

Lesson 22
Slice Command

📎 **Modify menu:** 3D Operations » Slice

⌨ **Command entry: slice**

📎 **Dashboard:** 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*)

2.   *Change the UCS to the front view and set the color to green and draw the following construction 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)*

3.   *Use the COPY Command to copy the line "D" to both sides with the approximate distances as shown in figure 3.*
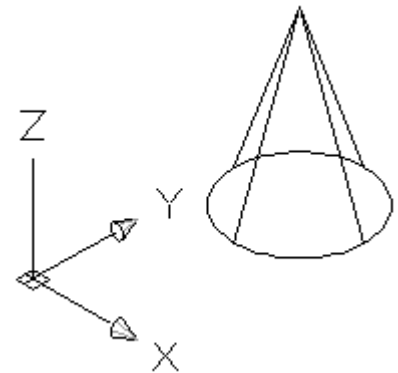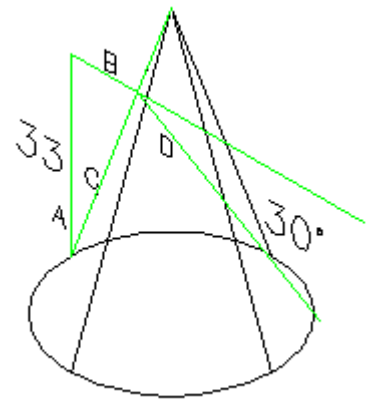
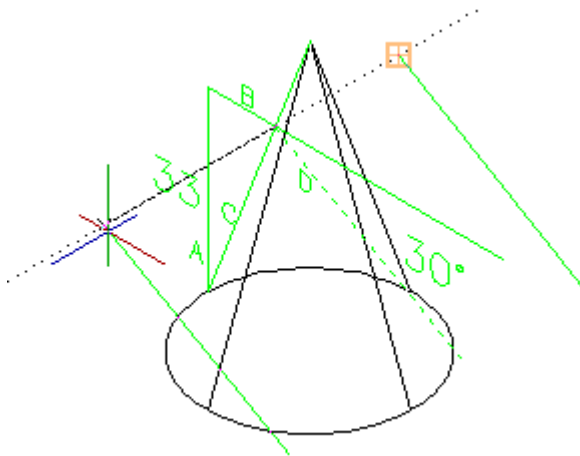Figure 1

Figure 2

Figure 3

4.   Use the SLICE Command with the "3 points" 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
Object/Surface/Zaxis/View/XY/YZ/ZX/3points] <3points>:
<Enter>
Specify first point on plane: {*pick the endpoint 1*}
(Figure 4)
Specify second point on plane: {*pick the endpoint 2}*
Specify third point on plane: {*pick the endpoint 3*}
Specify a point on desired side or [keep Both sides]
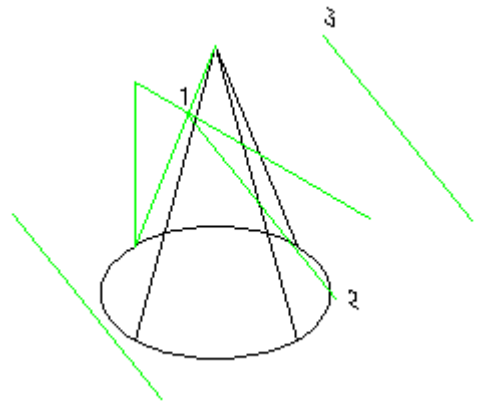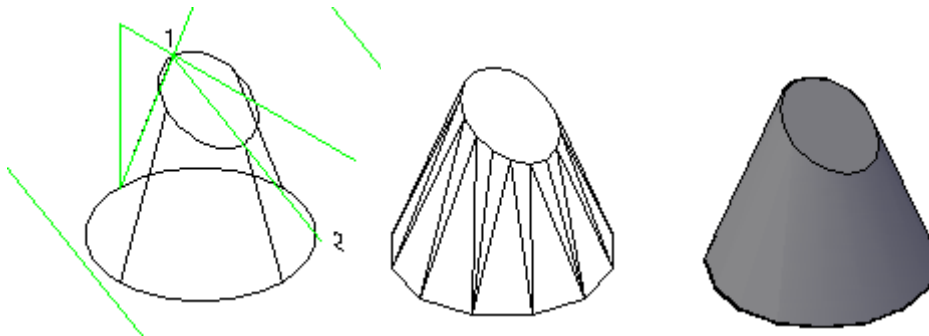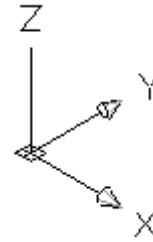<Both>: {*pick the center of the base circle*}

Figure 4

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 selecting "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 height: **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>
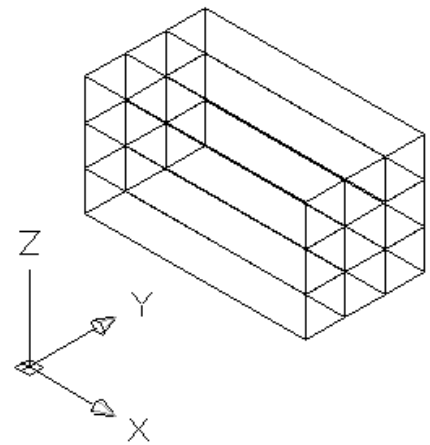
Figure1

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:

Command: **union** <Enter>
Select objects: 1 found   *(select the each box)*
Select objects: 1 found, 2 total
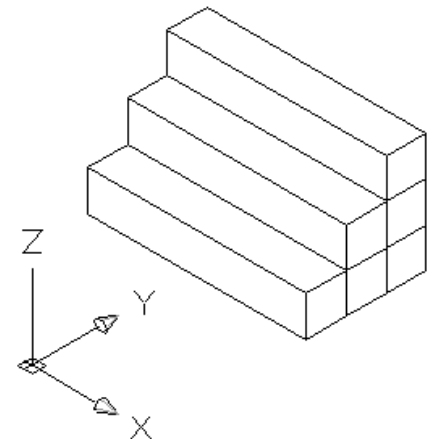Select objects: 1 found, 3 total

Figure 2
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 point 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] <3points>: <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>

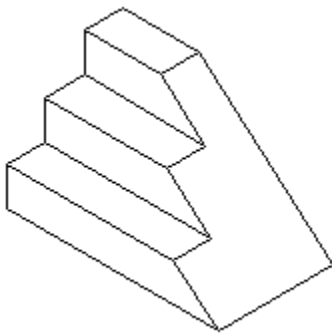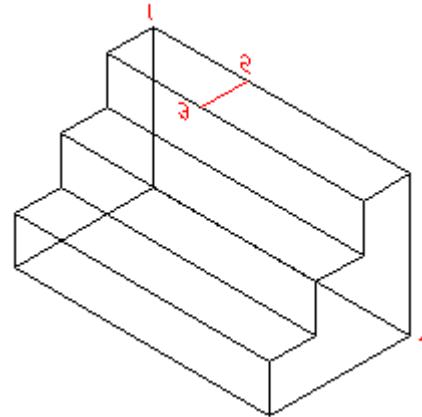*Erase the right side of the figure 4 to complete the project. (Figure 5)*
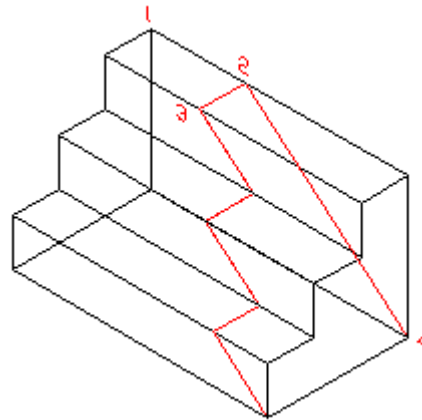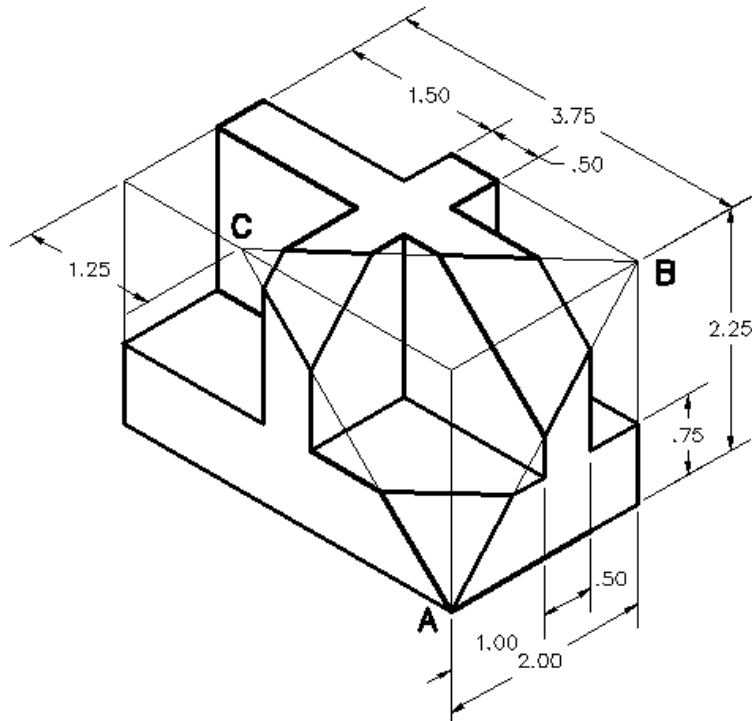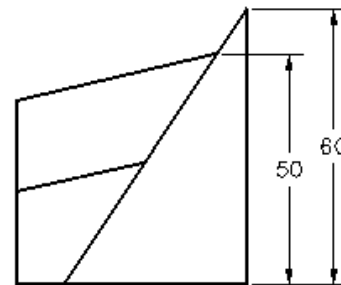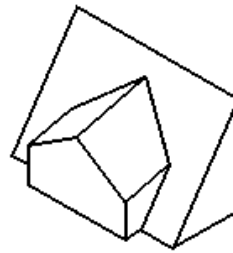

Figure 3

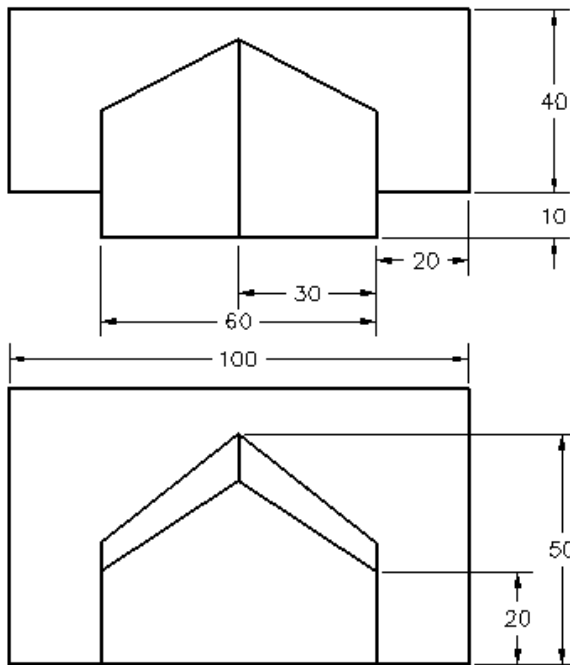Figure 5 - (Adjusting Guide)

Figure 4

Exercise 22-2

Construct a solid model of these figures:

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.
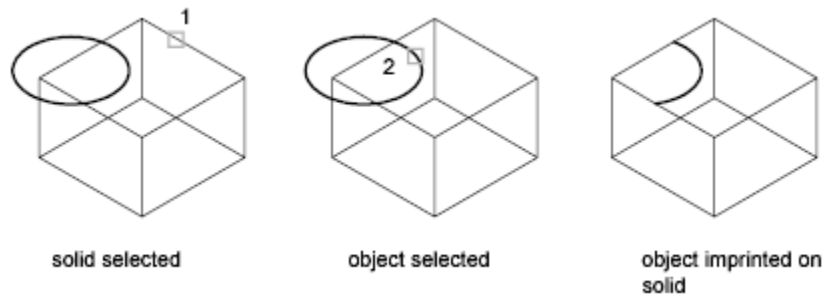

solid selected   object selected   object imprinted on solid

Figure 1

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

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>
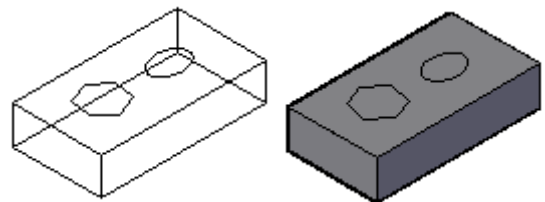Select an object to imprint: <Enter>

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>:
_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
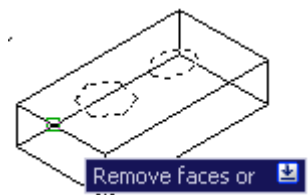

Figure 3

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.
Solid validation completed.
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
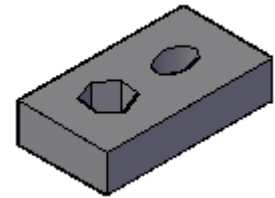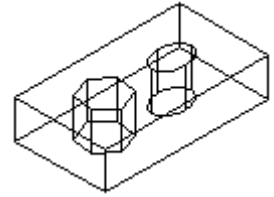Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: X

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. After 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>
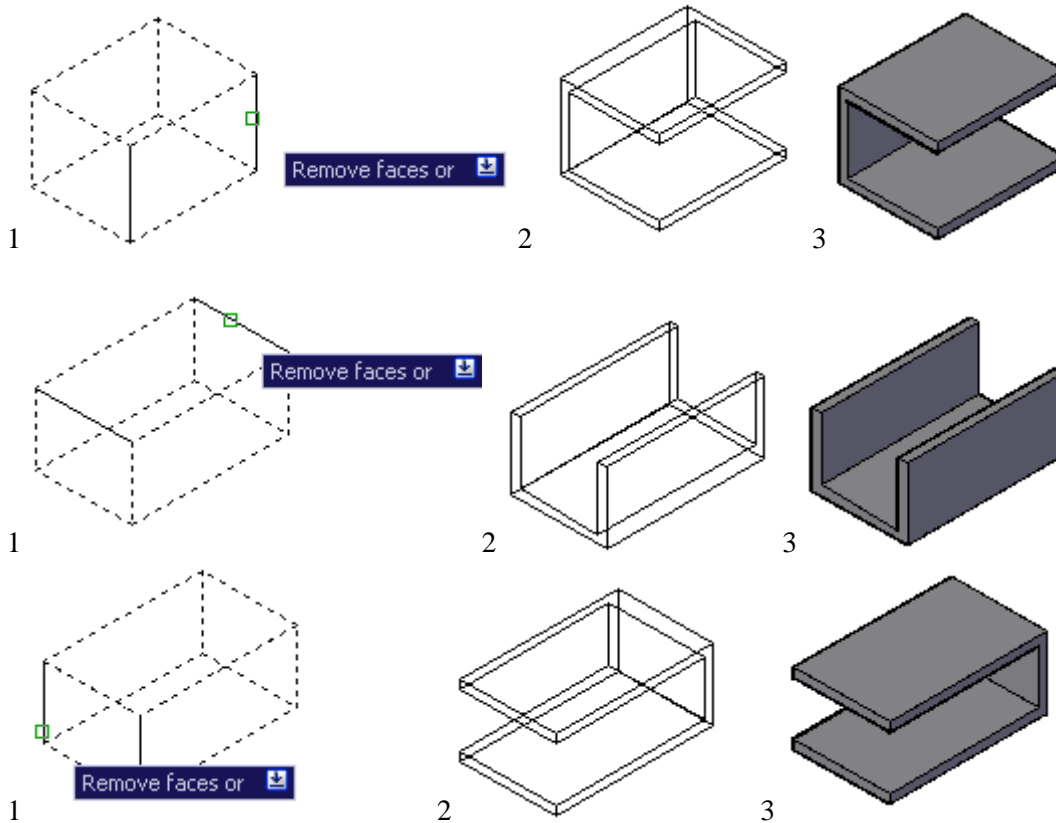Solid validation started.
Solid validation completed.
Enter a body editing option
[Imprint/seParate solids/Shell/cLean/Check/Undo/eXit] <eXit>: X <Enter>
Solids editing automatic checking: SOLIDCHECK=1
Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: X <Enter>

1   2   3

1   2   3

1   2   3

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 be sure to remove the top face, or any other faces if selected. (Figure 23-2)*
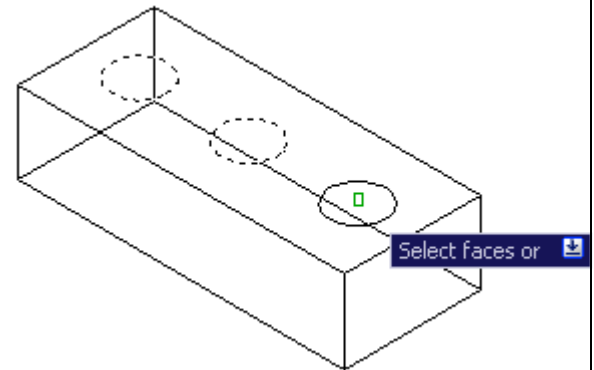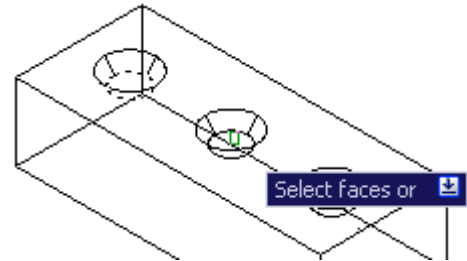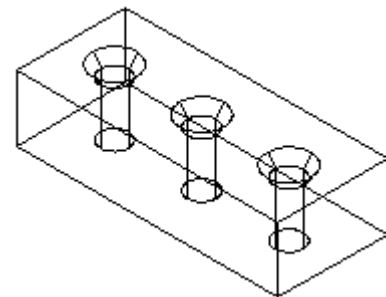   Extrusion height = -.25 & Taper angle =35°

23-1

23-2

23-3

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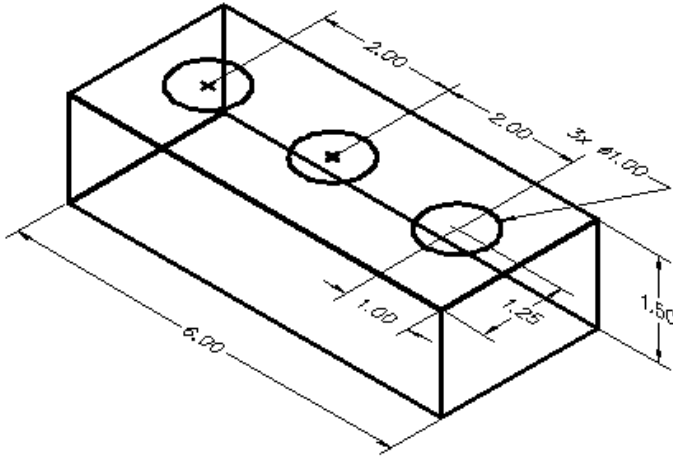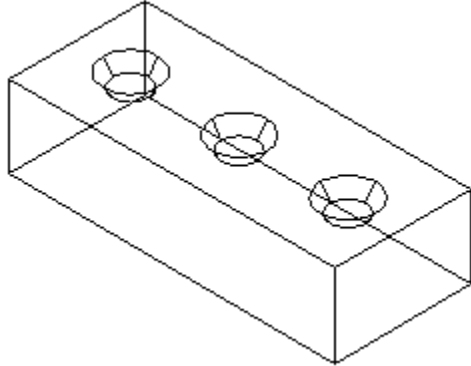
5. *Extrude the small diameter of the three tapered holes so they intersect the bottom of the solid box. Select the holes by picking the inside of the small holes (figure 23-4). Use the negative value larger than thickness of the solid box, for the height of the extrusion. (ex. –2)*

23-5

Copy faces   **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)*

face selected   base point and second   face copied
                       point selected
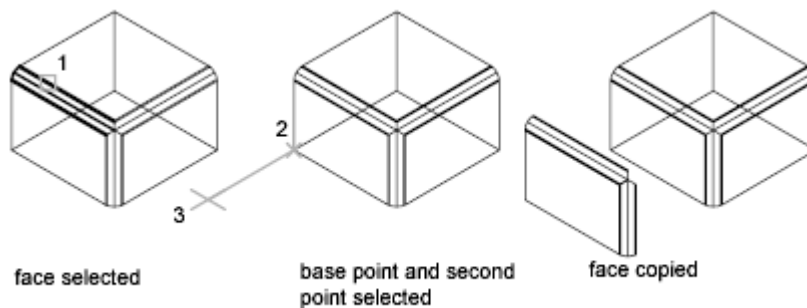
Lesson 24
Creating Section Planes

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.
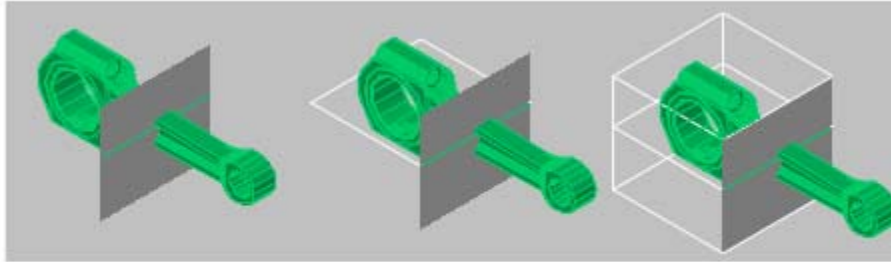
Section Plane   Section Boundary   Section Volume
state   state   state

- *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.
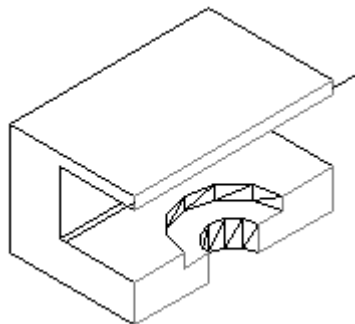
a. Face Option

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}*
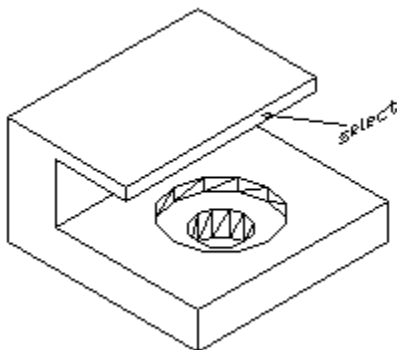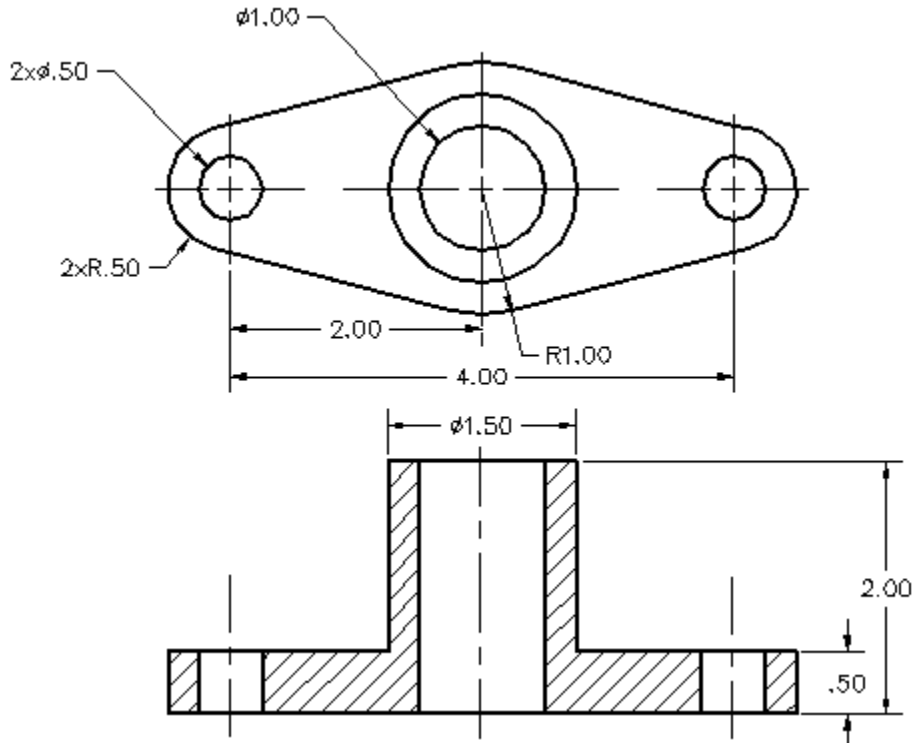

(A)   (B)

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.

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"
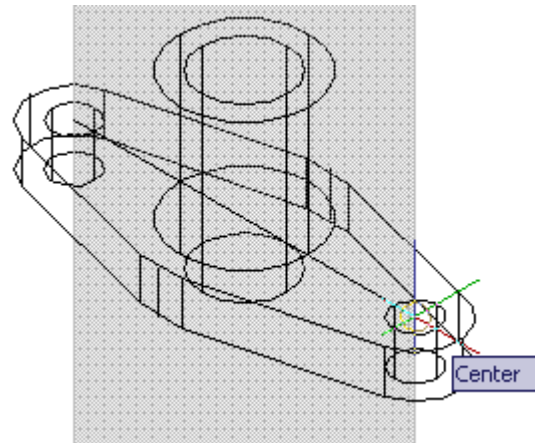
Figure 1

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)
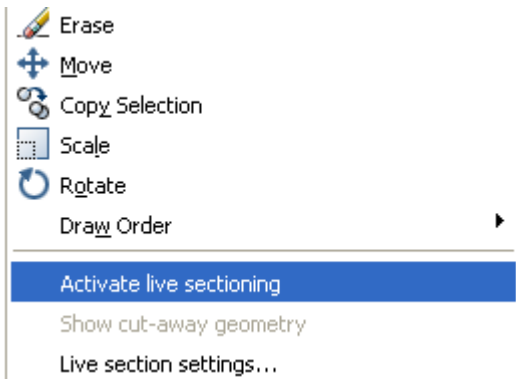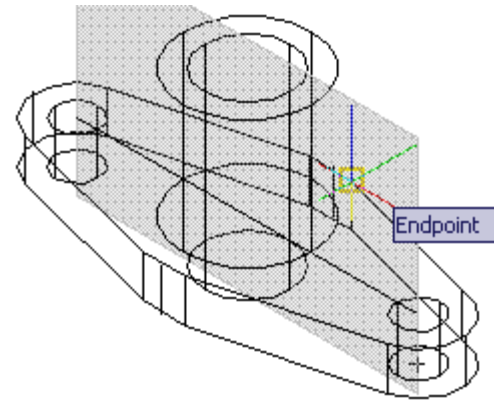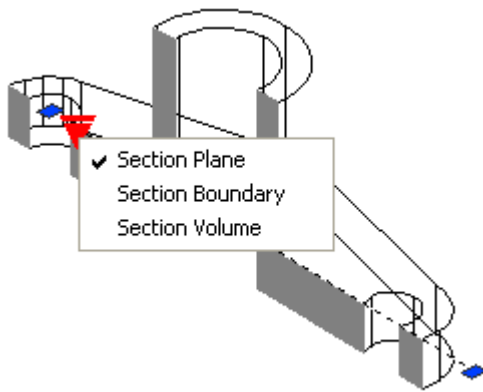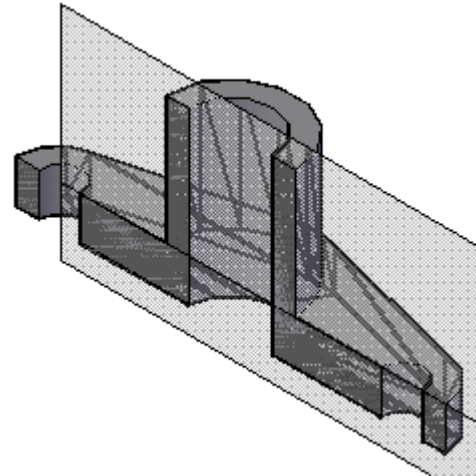
Erase
Move
Copy Selection
Scale
Rotate
Draw Order   ▶

Activate live sectioning

Show cut-away geometry

Live section settings...

Figure 2

Figure 3

✓ Section Plane
   Section Boundary
   Section Volume

Figure 4

Exercise 24-1

Ø2.380
START AT
2 X Ø.690 5,5,5
                                              Ø3.500

R.690

3.125
6.250

GROOVE .19 X .19

1.130 .630   .690
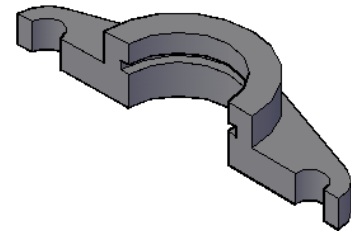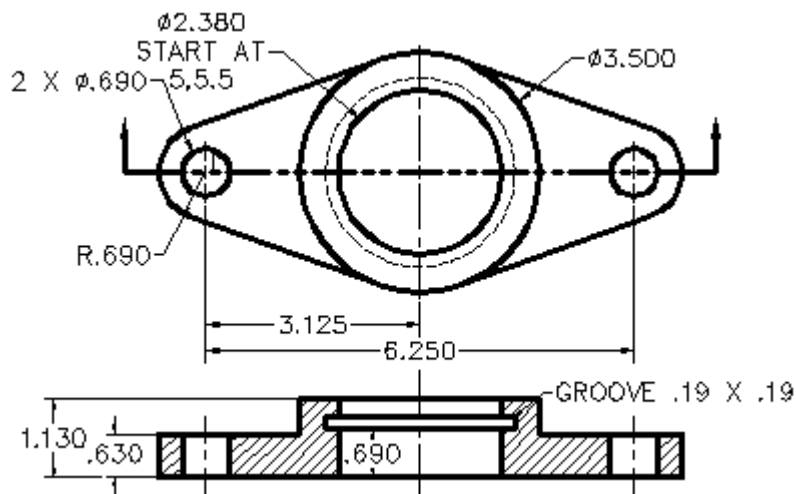
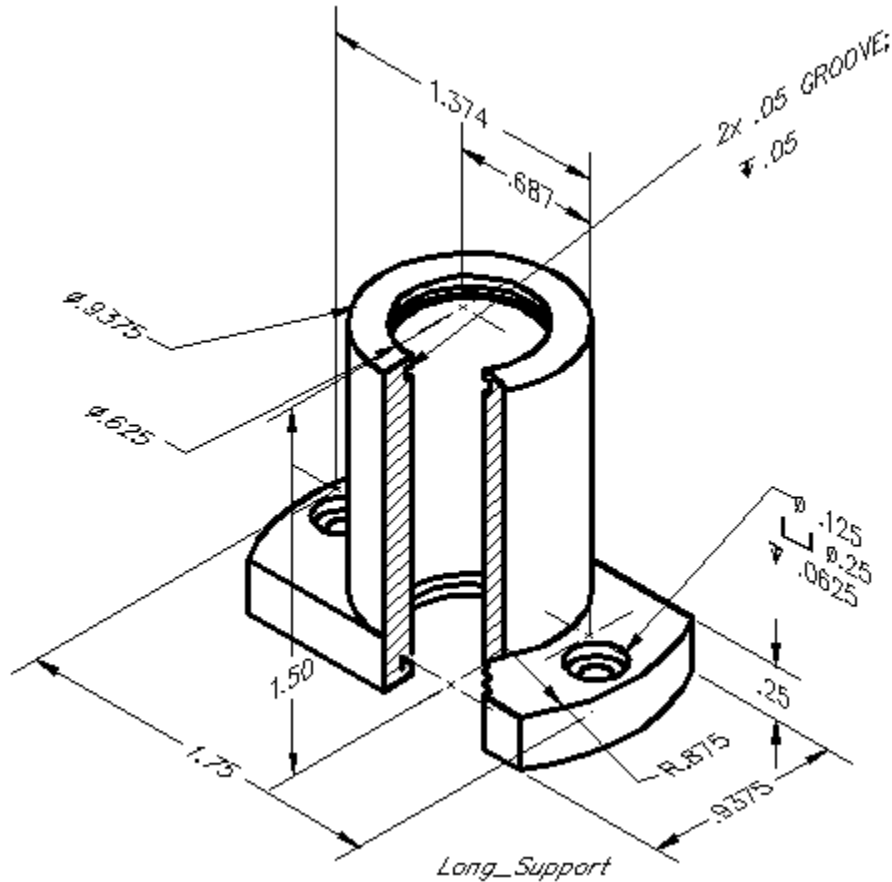### c. <u>**Multiple Points Option**</u>

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


Long_Support

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}
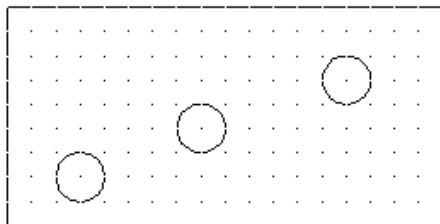
(Use object snap as needed)
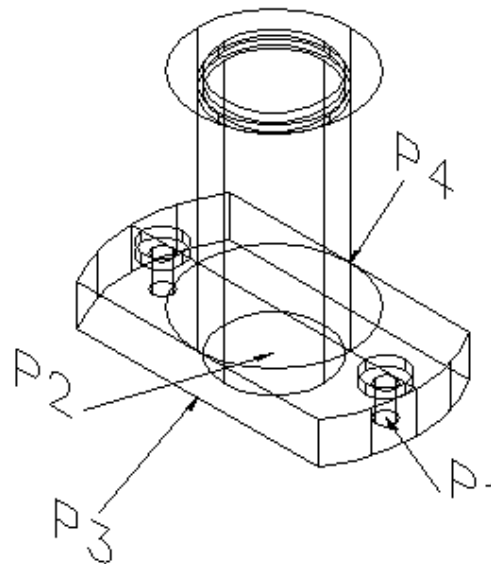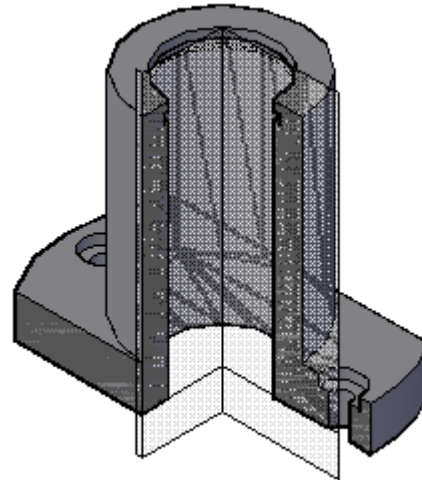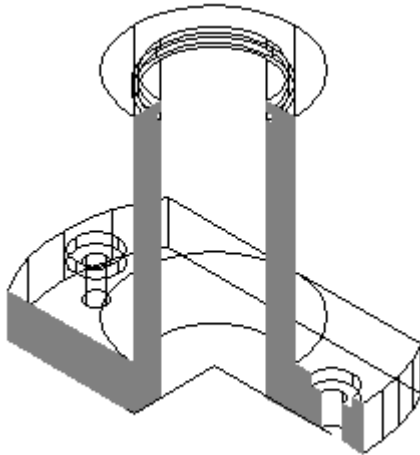
d.   <u>Adding Jogs to a Section</u> (Offset Section)
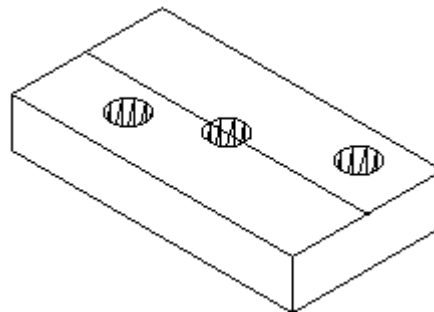
⌨ Command entry: jogsection

Select section object: Select a 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

GRID & SNAP = 0.25

a. Draw a section plane from mid point

A

B

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)

Activate live sectioning
Show cut-away geometry
Live section settings...
Generate 2D/3D section...
Add jog to section
Deselect All

✓ Section Plane
Section Boundary
Section Volume

Draw Order
✓ Activate live sectioning
Show cut-away geometry
Live section settings...
Generate 2D/3D section...

C

EXERCISE 24-2

Create an offset sectional model of the figure below:

METRIC

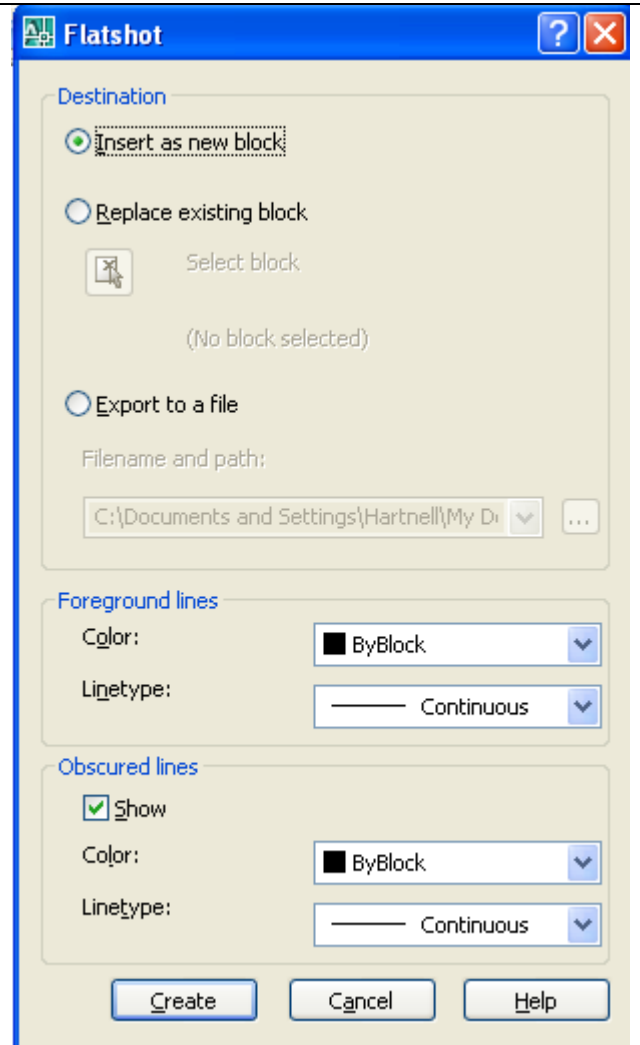# Lesson 25
FLATSHOT Command

⌨ Command entry: **flatshot**

❖ Dashboard: 3D Make 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!

Flatshot

Destination
- ⦿ Insert as new block
- ◯ Replace existing block

  🔲 Select block

  (No block selected)

- ◯ Export to a file

  Filename and path:

  C:\Documents and Settings\Hartnell\My D: [...]

Foreground lines
  Color: ■ ByBlock
  Linetype: —— Continuous

Obscured lines
  ☑ Show
  Color: ■ ByBlock
  Linetype: —— Continuous

  [Create]   [Cancel]   [Help]

.

Lesson 26
## Creating and Using Multiview Layouts
SOLVIEW & SOLDRAW 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.
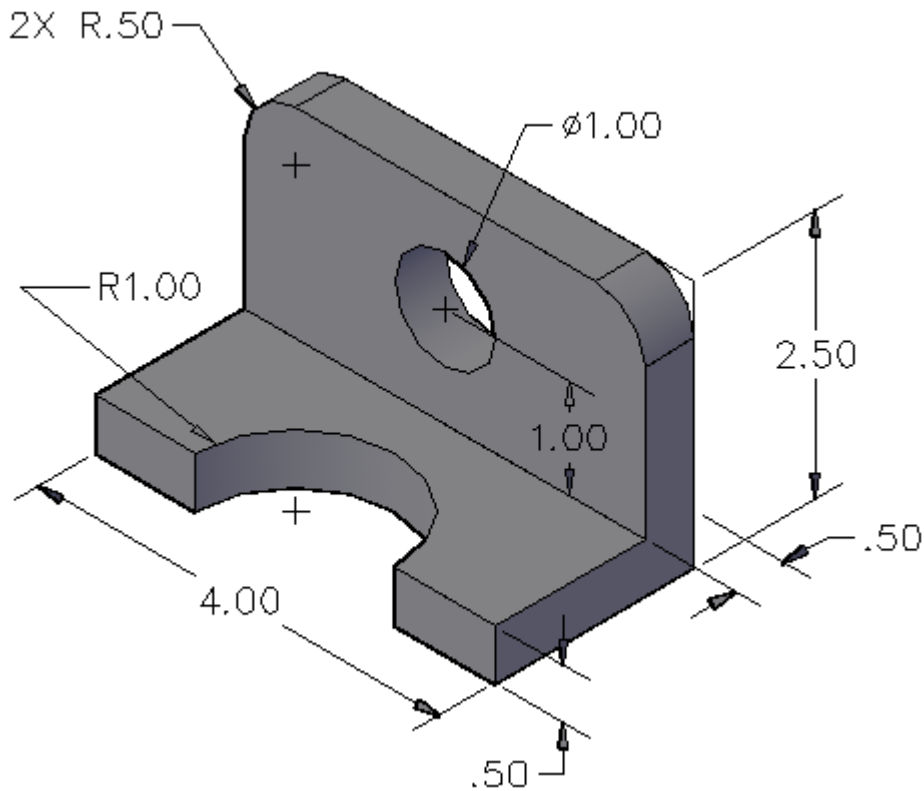
Steps to generate orthographic views from a 3D Model

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>

69

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.*

*From the "Pull-Down Menu" Insert>Layout > Layout from,
brows through tha CAD lab server insert "Mytemplate.dwt"
file.*
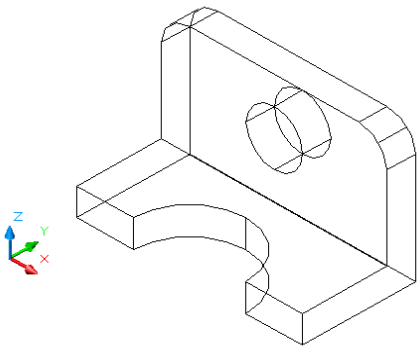
Figure 1

Figure 2
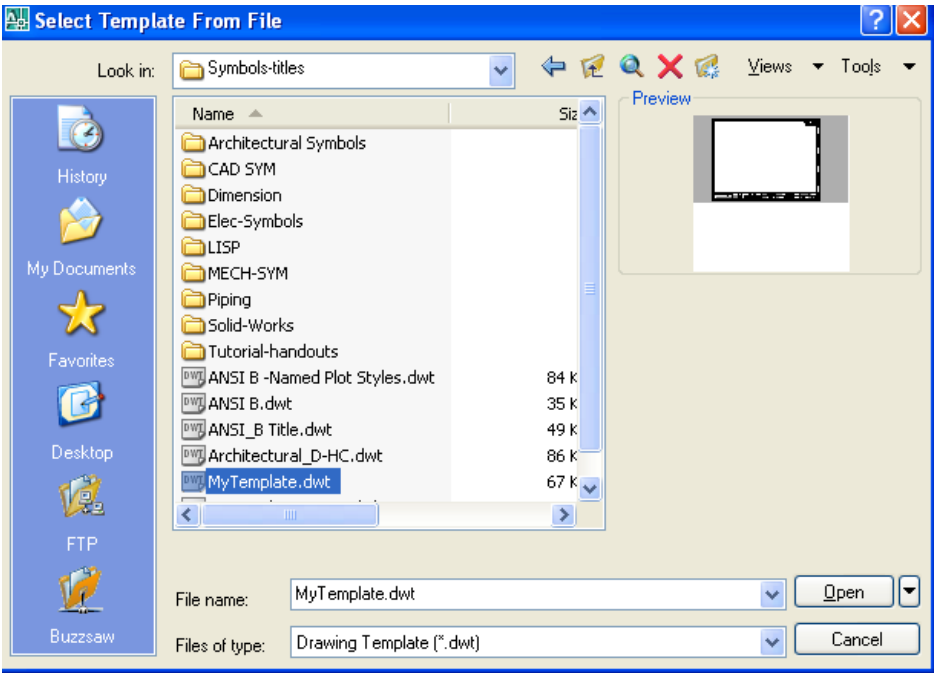
Select the "Layout3 tab-
Layout1" tab. (Figure 3)
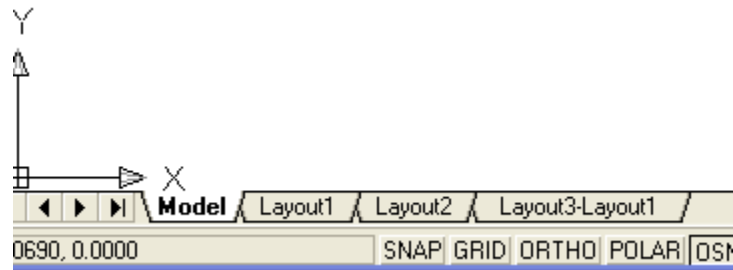
Figure 3

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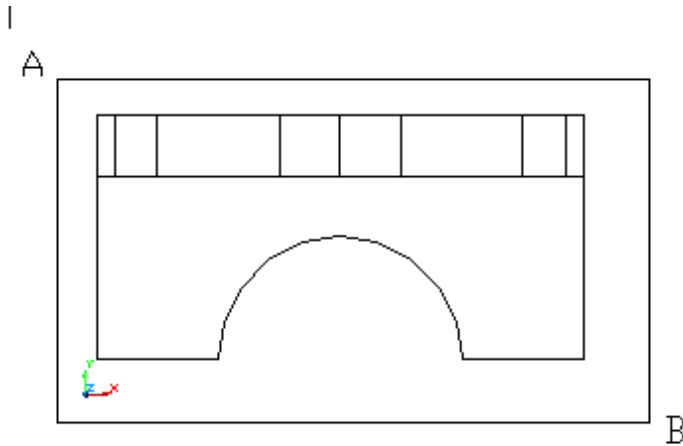
Enter view name: top   <Enter>

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>
UCSVIEW = 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 ina 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}*
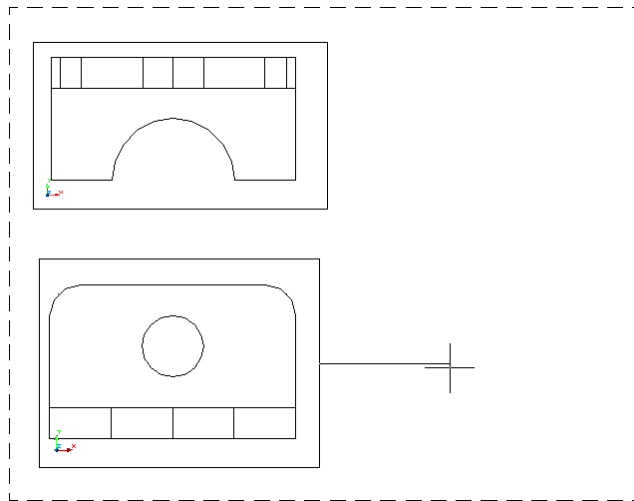Specify opposite corner of viewport: {*pick approximate point on the lower right corner of the side view}*. Figure 6
Enter view name: side   <Enter>
UCSVIEW = 1 UCS will be saved with view
Enter an option [Ucs/Ortho/Auxiliary/Section]: <Enter>

Figure 4

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 [Ucs/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.
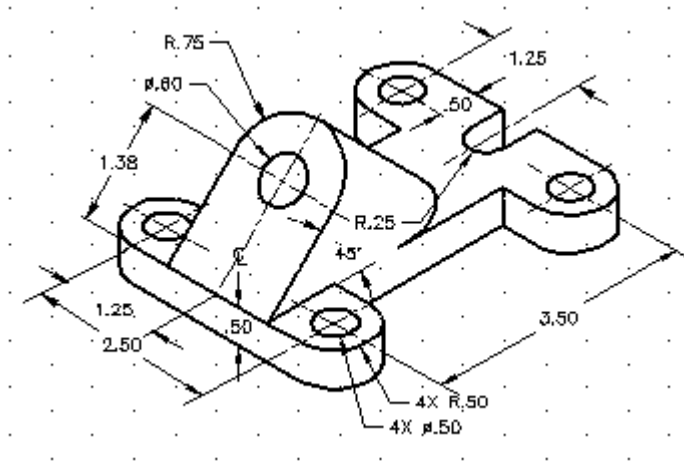
EXERCISES:

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.
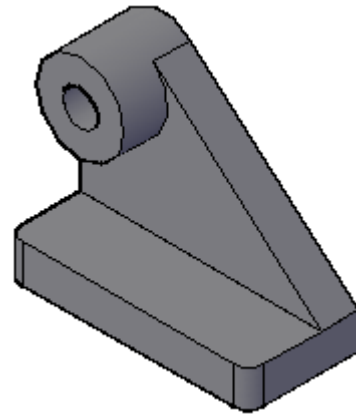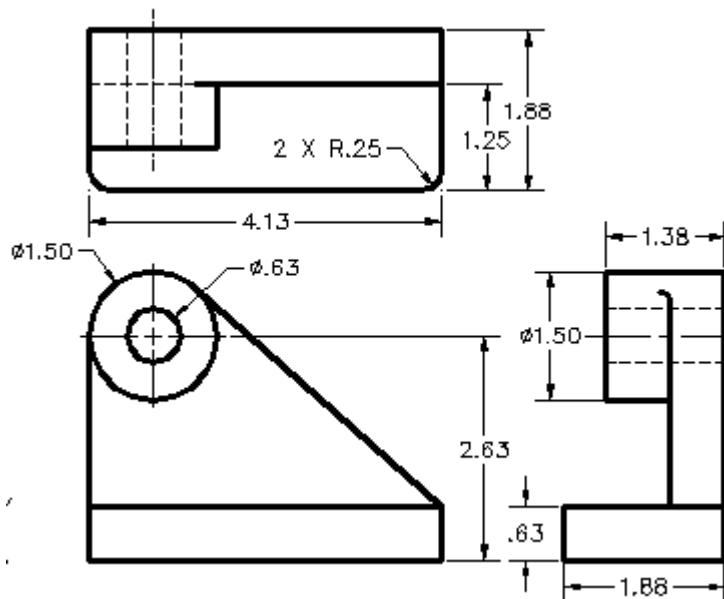
EX-26-1_ Shaft support

Lesson 27
Auxiliary UCS

Automatic Stop

5. Instruction

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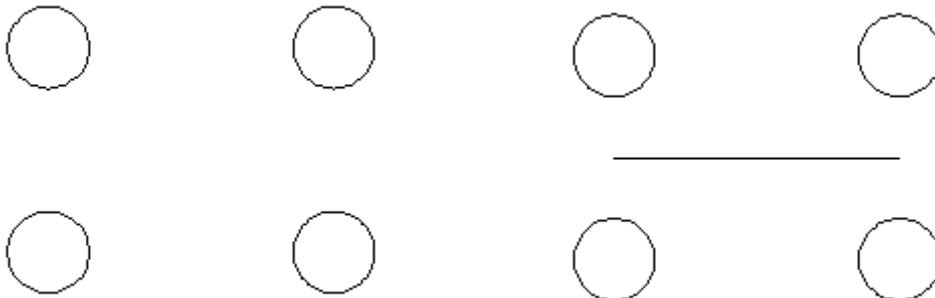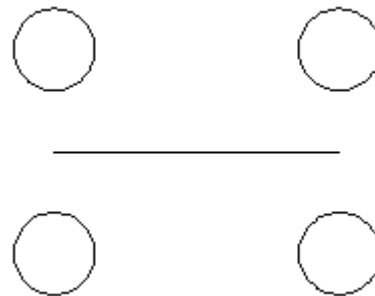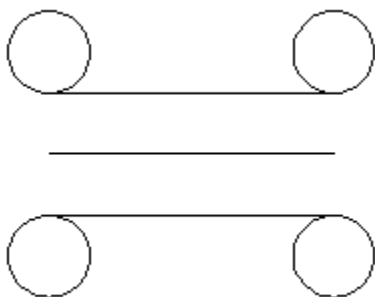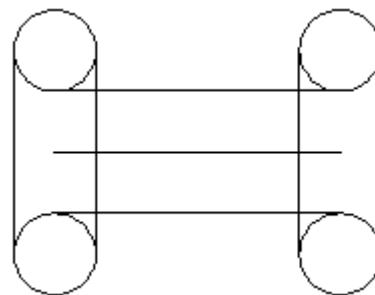2.   Follow the steps in figures 2 through 5 to create the profile as shown in figure 5.

Fig.1   Fig.2

Fig.3   Fig.4
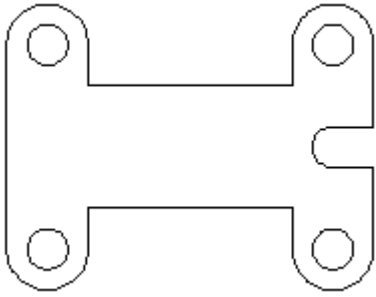
Fig .5
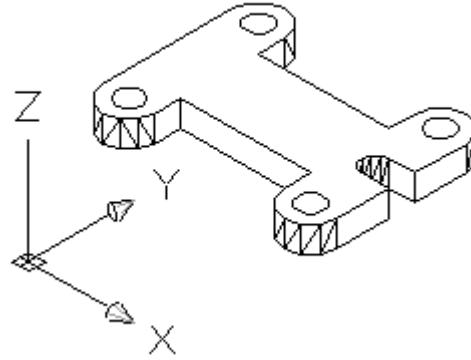
Fig.6

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.
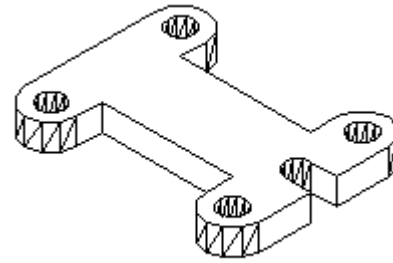8. Create a new auxiliary using UCS Command with "3point" option

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<45** <Enter>
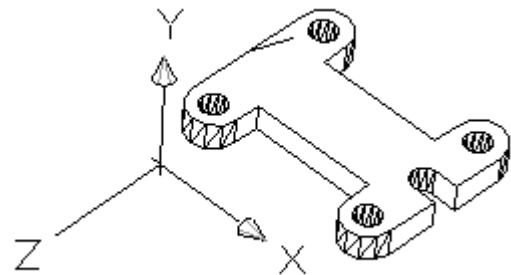
Fig.8

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}
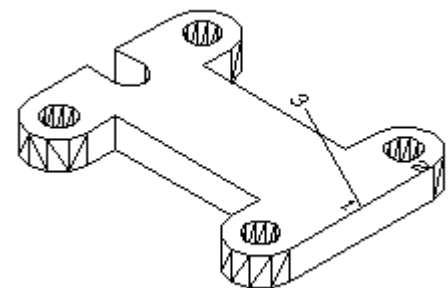<-2.7475,7.5789,-0.9901>:

Fig.9

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: ISOLINES=4
Select objects: 1 found *{select the profile}*

Select objects: <Enter>
Specify height of extrusion or [Path]: -3 <Enter>
Specify angle of 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 views including one auxiliary view.
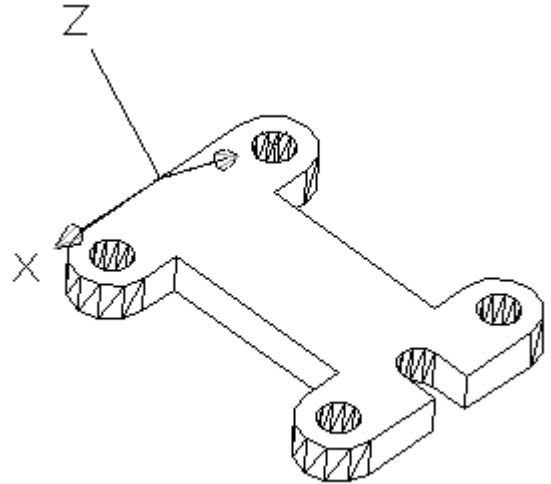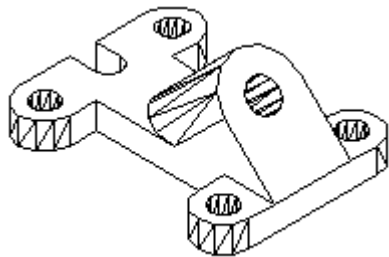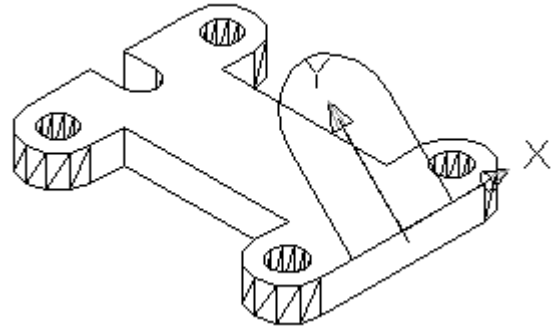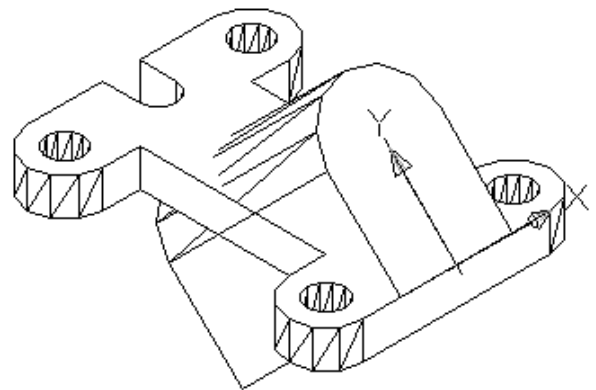*<u>Note</u>: Refer to instruction on "Printing 3D Model"*
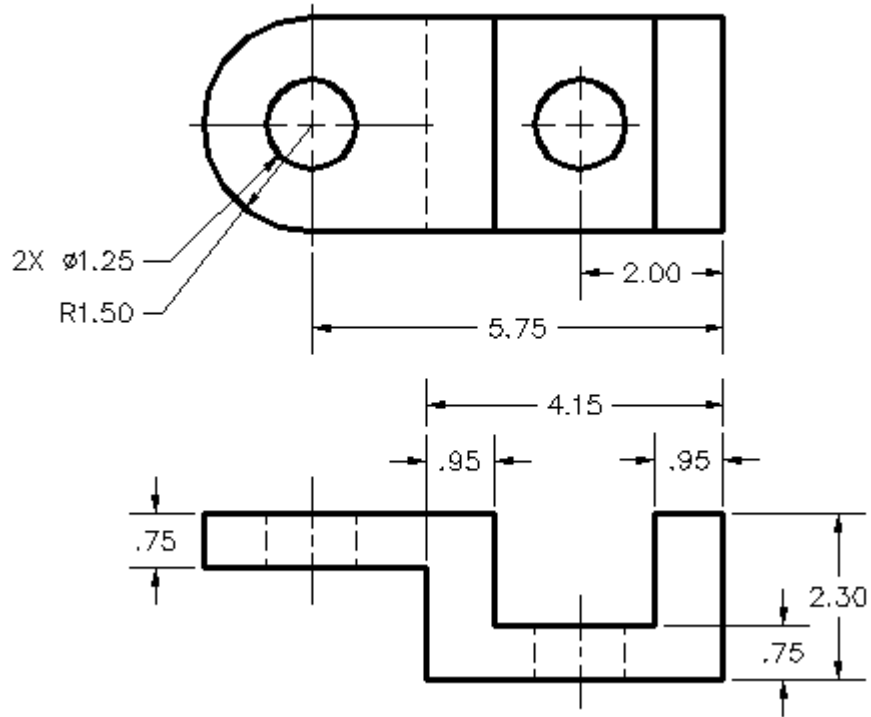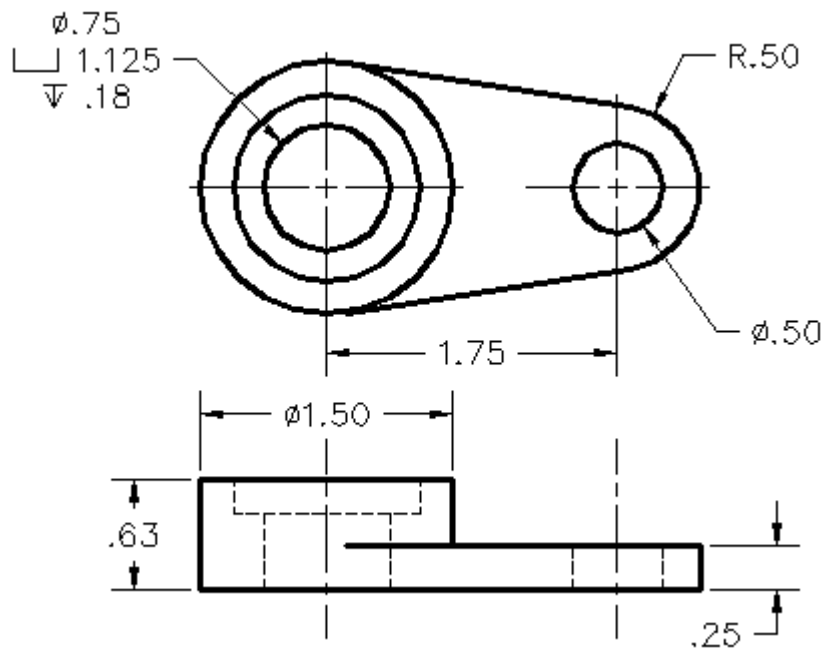

Fig.10

Fig.11

Fig.12

Additional Problems

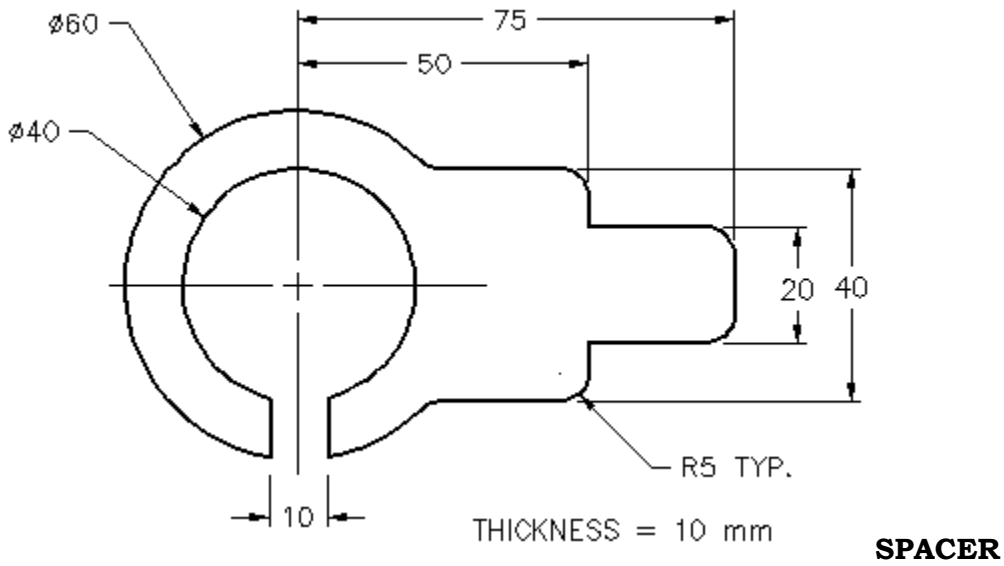Construct a 3D solid model of the given orthographic views.
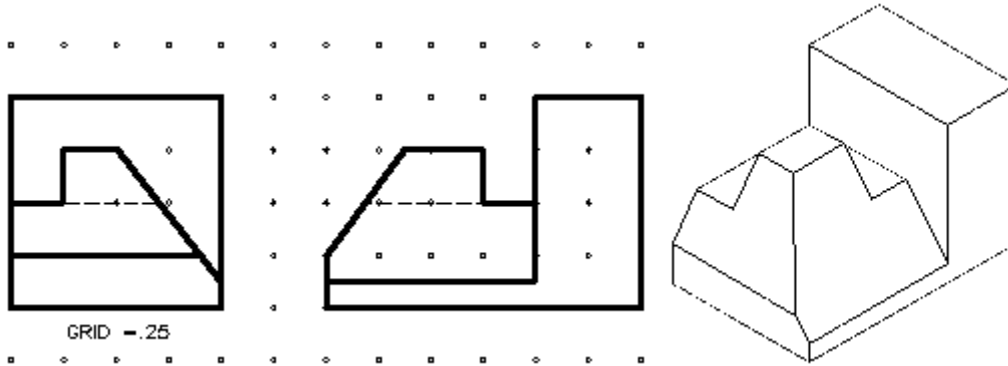
Problem 1

Problem 2

Problem 3

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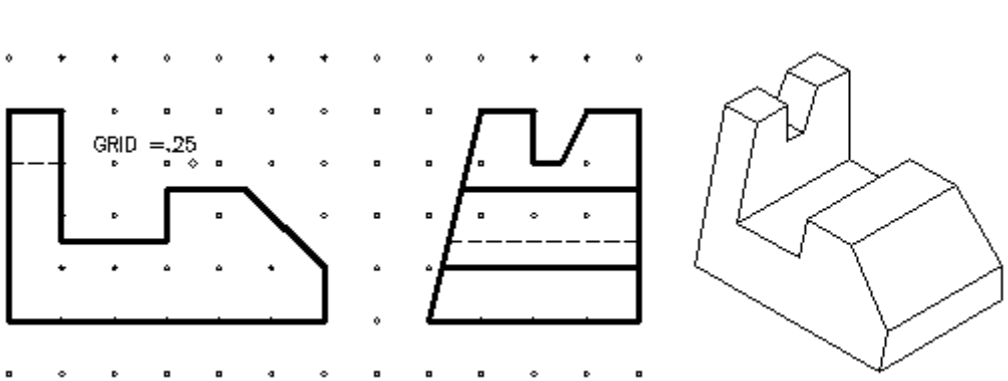

SPACER

Problem 4

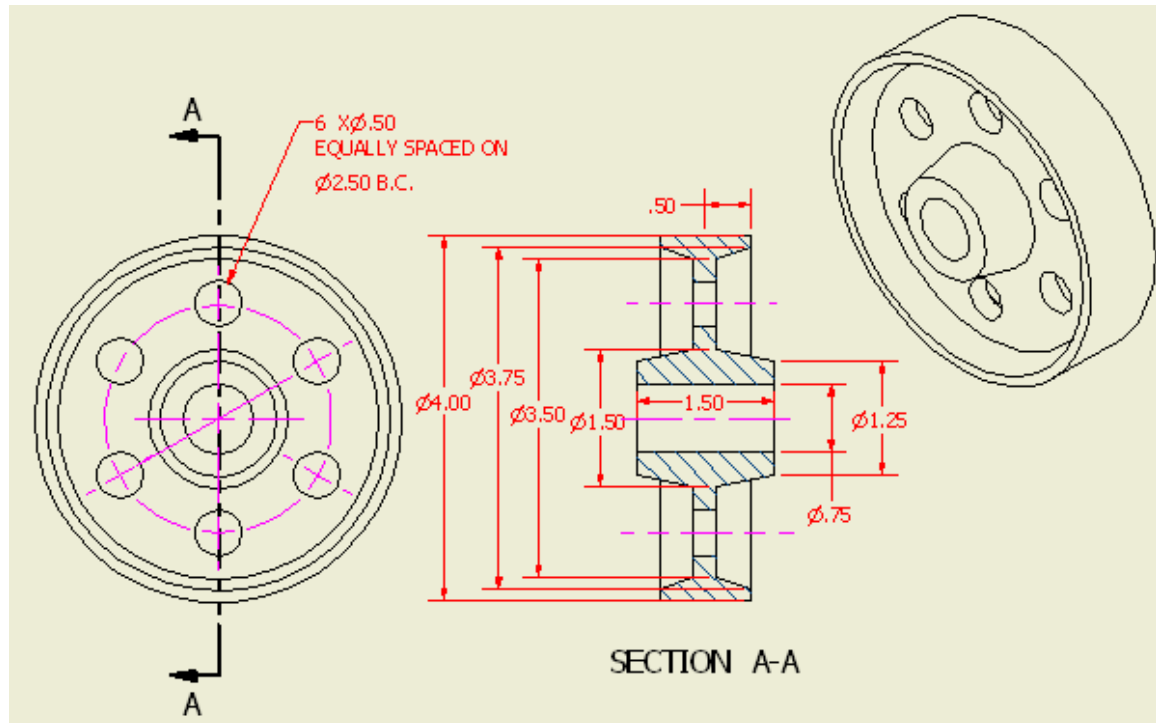*Given front and right side views of the figures below, construct the 3D solid models:*

Problem 5

Problem 6

Ø 1.00
Ø .56
Ø 3.25
▼ .50
Ø 1.50
3.00
2.25
1.50
Ø 2.31
4X Ø .50
Equal Spacing
Ø 4.00

Problem 7

A

6 XØ.50
EQUALLY SPACED ON
Ø2.50 B.C.

.50

Ø3.75
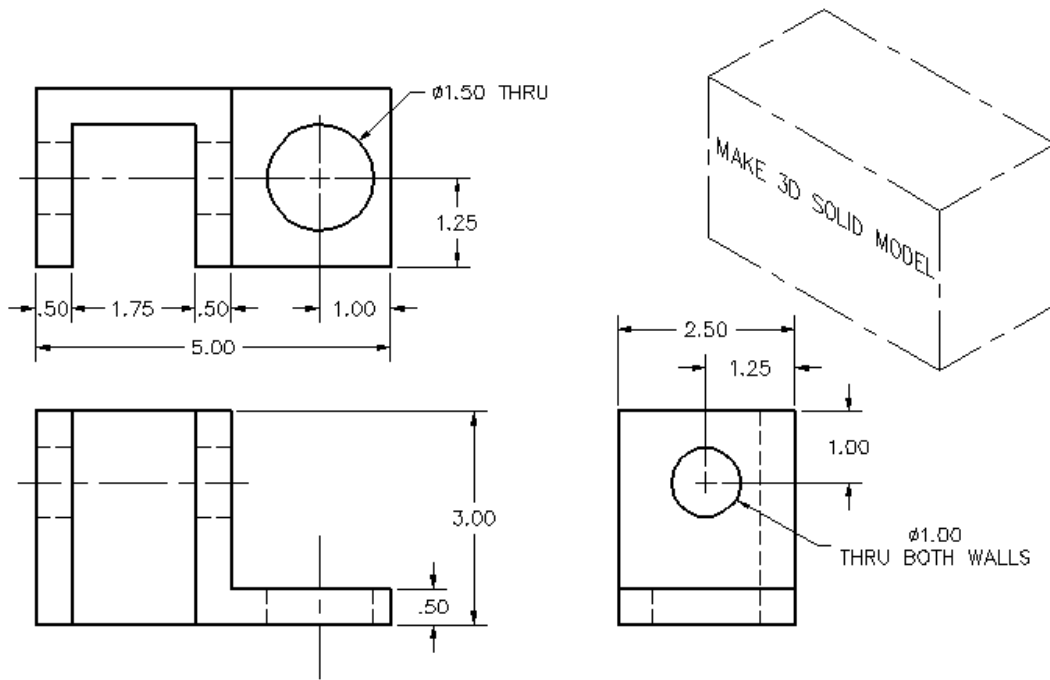Ø4.00
Ø3.50 Ø1.50
1.50
Ø1.25
Ø.75

A

SECTION A-A
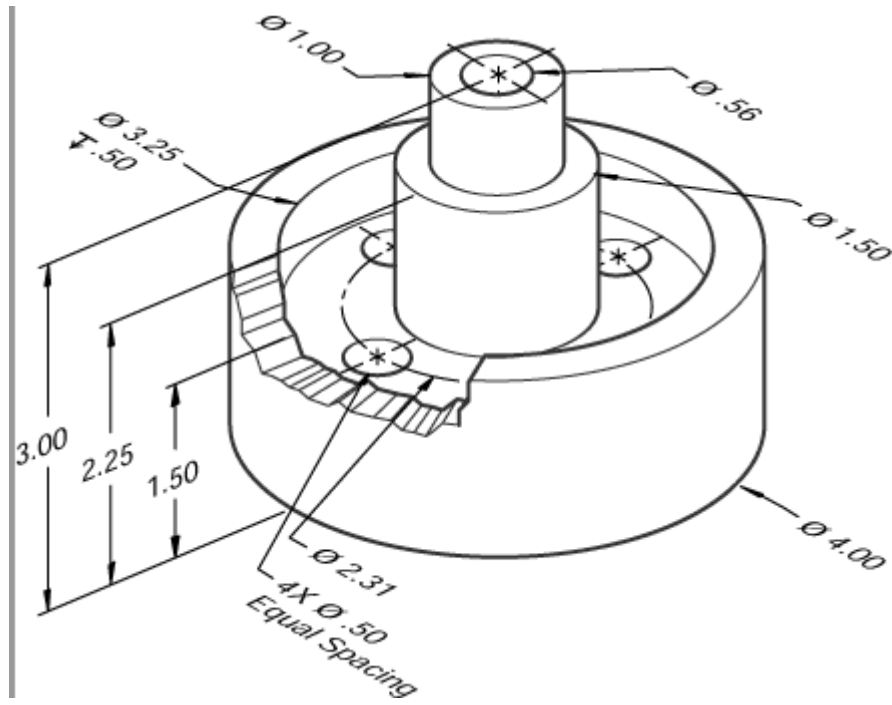
Problem 8

    a. Using the AutoCAD's modeling features, create 3D model of the given 3 views (TOP, FRONT & RIGHT SIDE).

    b. Calculate the VOLUME.

    c. Generate the orthographic views as shown including an isometric view.

    d. Print space scale 1=1.

Problem 9

FILLETS & ROUNDS R .06

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/SAveas/Set/?] <set>: **t** <Enter>
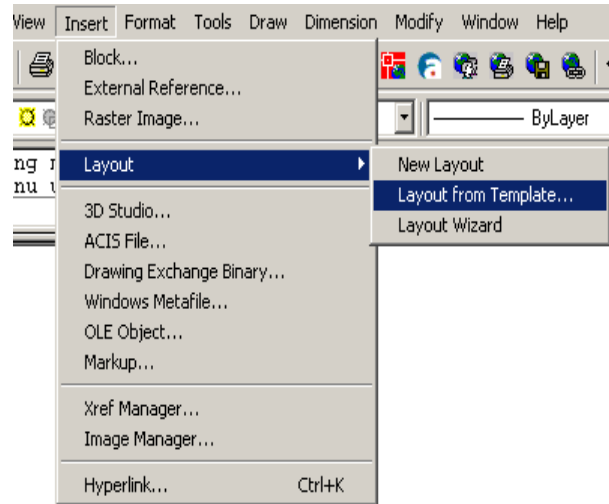*Locate your template file (called "MyTemplate" see Figure 2.)*

<u>*Note*</u>*: 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" MyTemplate.dwt.*

Figure: 1

Figure 2

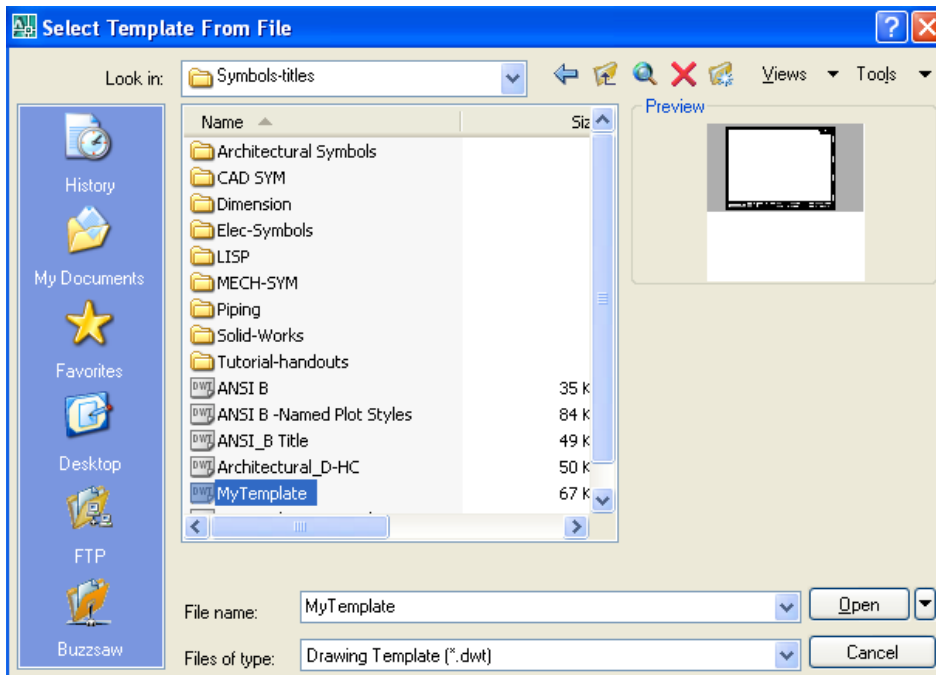Left click "OK" tab.   Click on "Layout3-Layout1 "tab
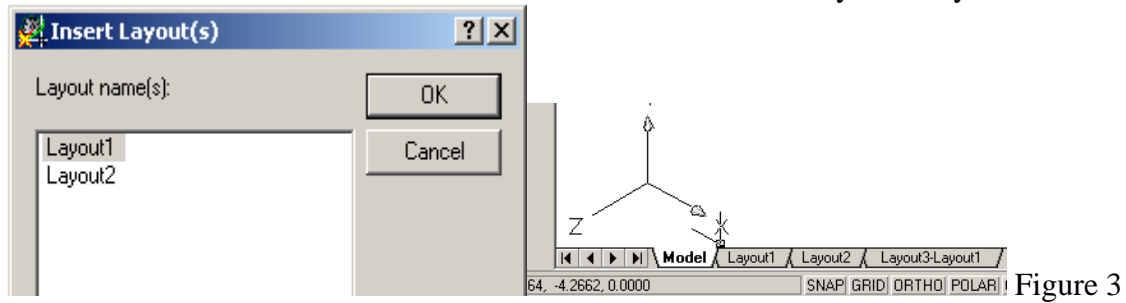

Figure 3

Command: <Switching to: Layout3-Layout1>
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.
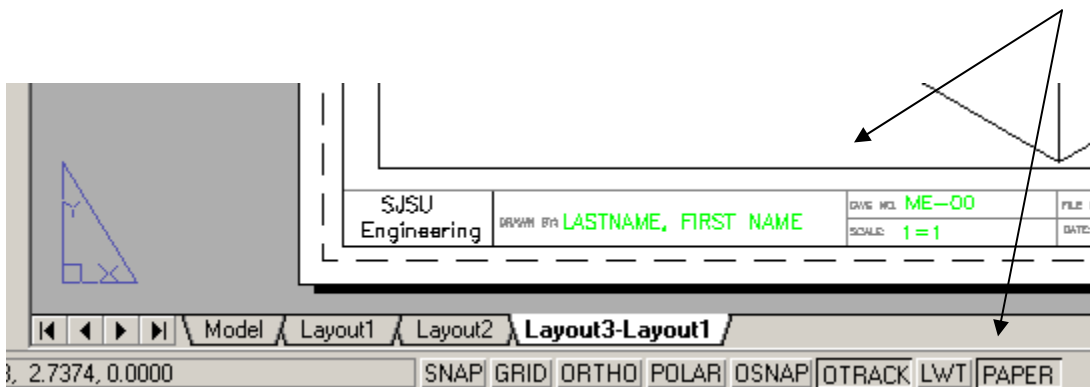*"PAPER" tab should switch to "MODEL". Figure 5*

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}*
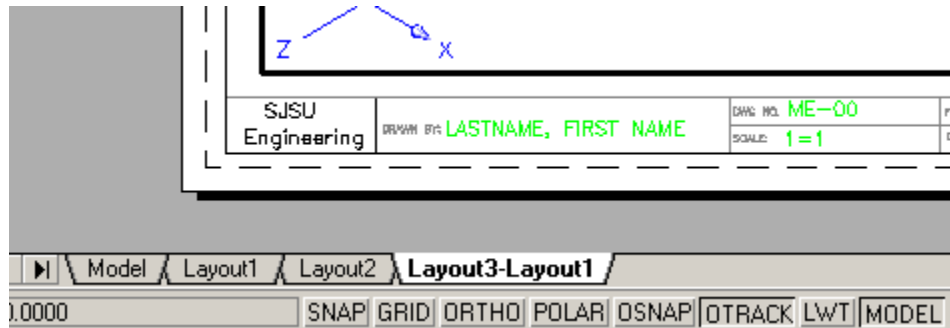
Perform Hide Plot:

Figure 5

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

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:
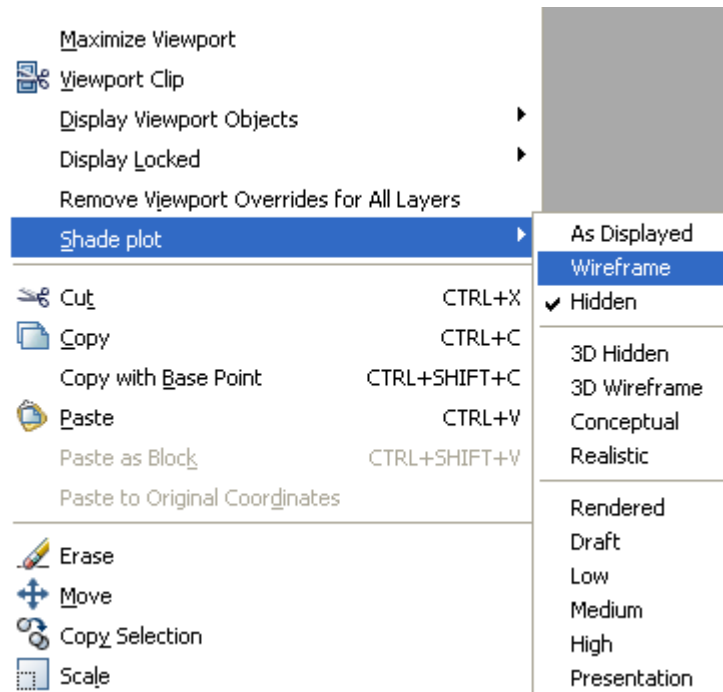
Command: **layer** <Enter>

Figure 6

*Select the "**New**" tab. Figure 7.*
*While "Layer1" is highlighted, click on "**Freeze**" icon to turn to gray. Click on "**Curren**t" tab.*
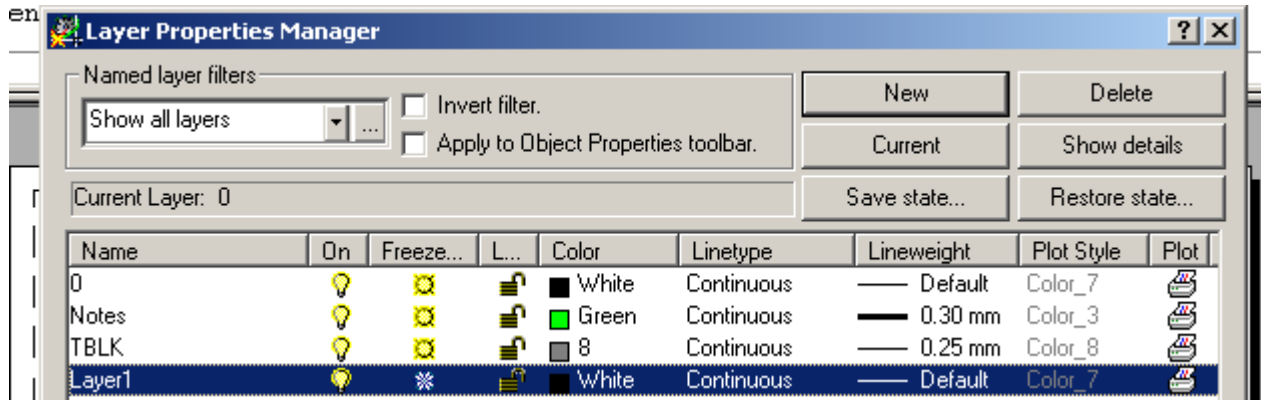
Figure 7

Assign the port to Layer1:

Click on the edge of the floating view port high light the layer1 from the layer window and click "OK".
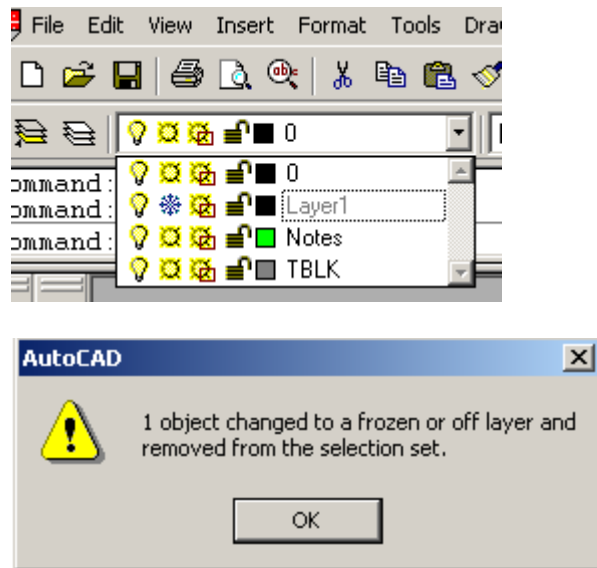*Figure 8:*

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 for 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**
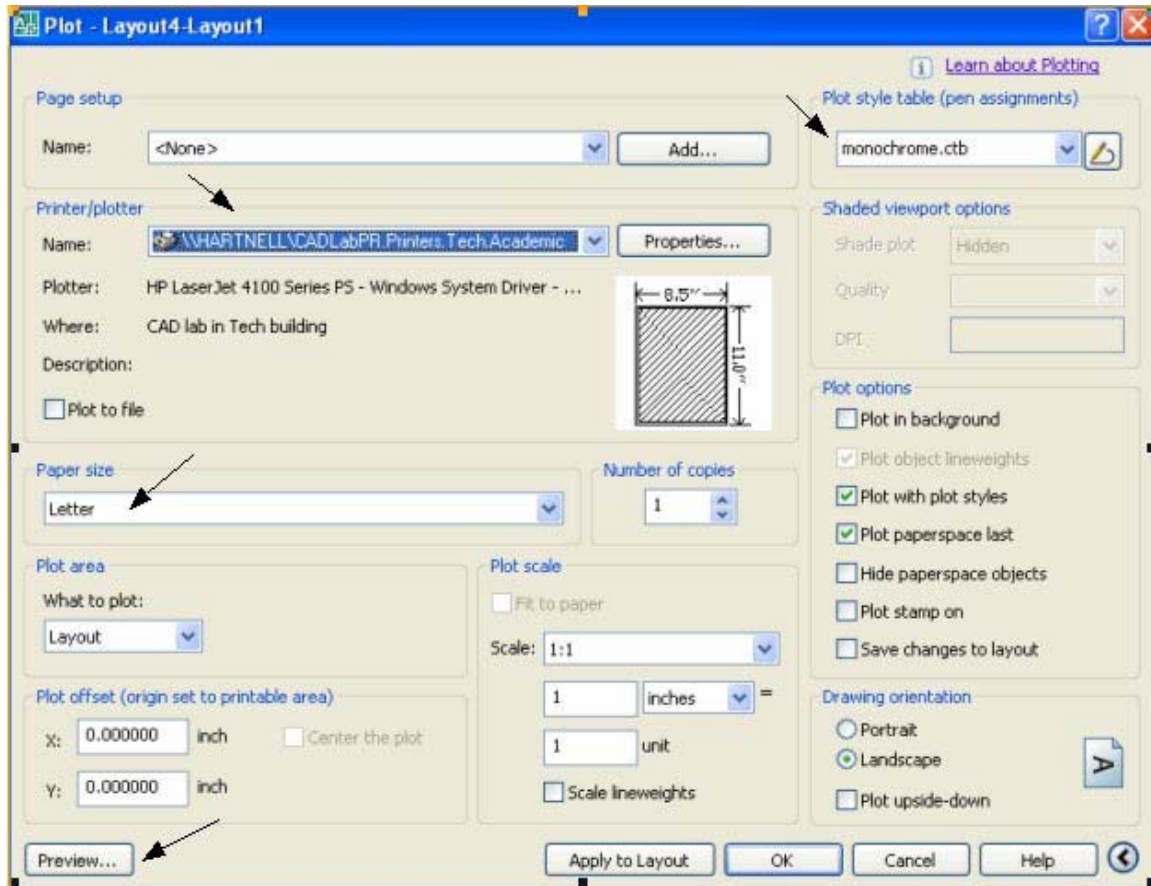   *c.*   **Full preview**   *(Figure 9)*

Figure 9

Note:

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

If your 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.