ZTM112 BİLGİSAYAR DESTEKLİ ÇİZİM TEKNİĞİ

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- Ankara Üniversitesi Ziraat Fakültesi Tarım Makinaları ve Teknolojileri Mühendisliği Bölümü
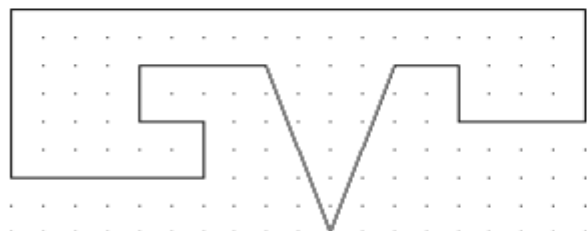
- ckoc@ankara.edu.tr
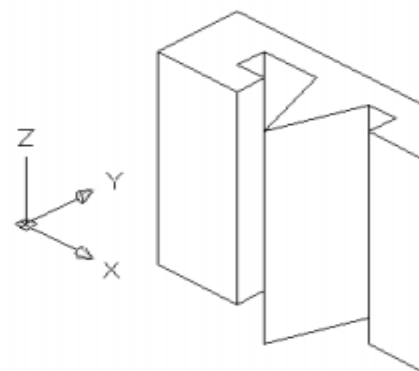
Modeling toolbar:

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.
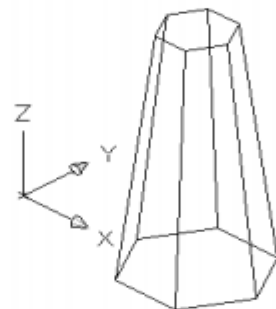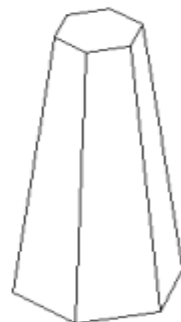5. Calculate the volume.

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.
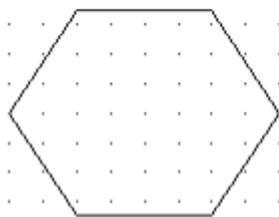

Grid space = 0.25

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.*

WORLD UCS
370
2X R60
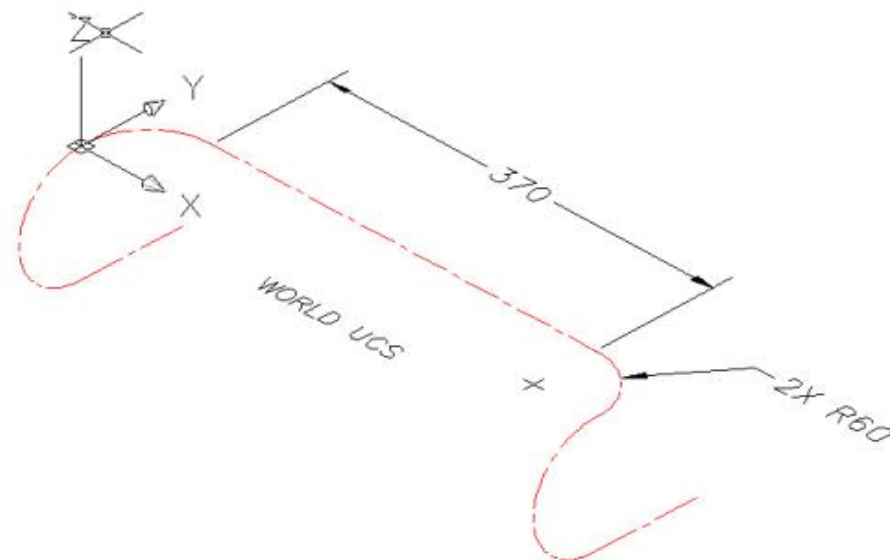
1. Pline with 370 length pline (arc option) or ARC with 60 radii. On each end on WCS.

3X Ø28
FRONT UCS
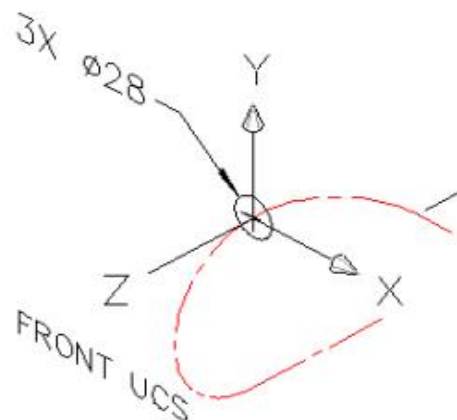
2. Create three circles on the FRONT UCS.
3. Create the drop portion of the bar using the pline command on the SIDE UCS.
4. Extrude each circle along each pline.

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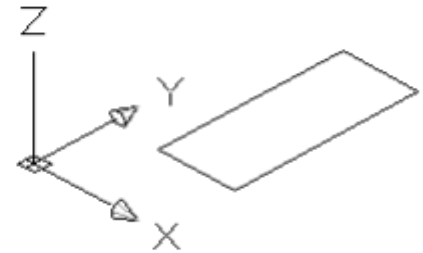Specify next point or
[Arc/Close/Halfwidth/Length/Undo/Width]: **c**

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*)
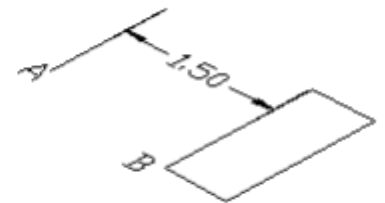Specify angle of revolution or [STart angle] <360>: **90**
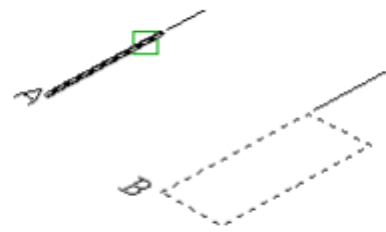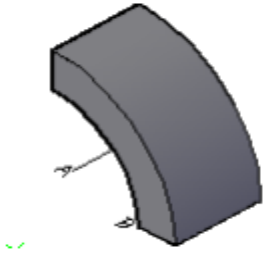<Enter>

Figure 2

Figure 3

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
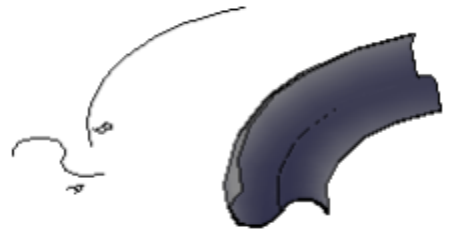point/Scale/Twist]: *{select the arc B}*

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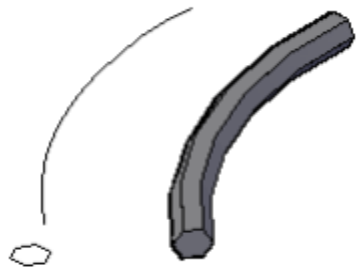Select sweep path or [Alignment/Base point/Scale/Twist]:
{select the arc }

Figure 2

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>
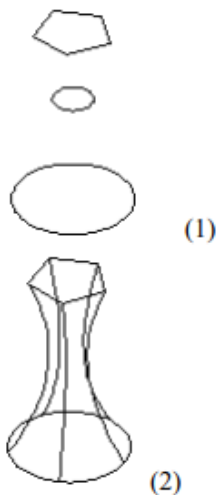Specify radius of circle or [Diameter] <1.5000>: **.5** <Enter>

(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>
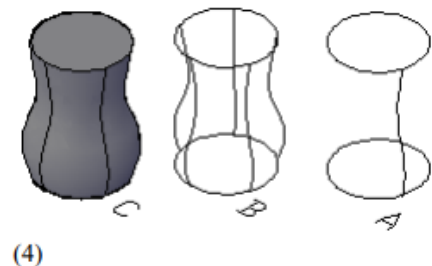Select guide curves: *{select the curve}*1 found
Select guide curves: *{select the curve}*1 found, 2 total
Select guide curves: *{select the curve}*1 found, 3 total
Select guide curves: *{select the curve}*1 found, 4 total
Select guide curves: *{select the curve}*1 found, 5 total
Select guide curves: <Enter>

(4)

    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
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}*

(5)

(6)
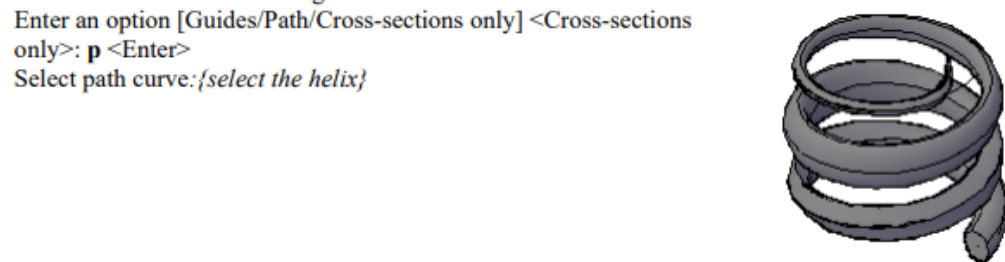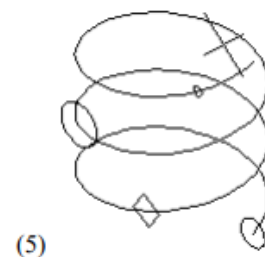
Command: _mirror3d <Enter>
Select objects: {select the wedge}1 found
Select objects: <Enter>
Specify first point of mirror plane (3 points) or
[Object/Last/Zaxis/View/XY/YZ/ZX/3points] <3points>:
<Enter>
Specify first point on mirror plane: {select the modepoint1}
Specify second point on mirror plane: {select the modepoint2}
Specify third point on mirror plane: {select the modepoint3}
Delete source objects? [Yes/No] <N>: N <Enter>
(Figure 2)

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.

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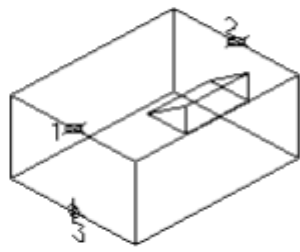Specify height or [2Point] <4.1326>: **.25** <Enter>

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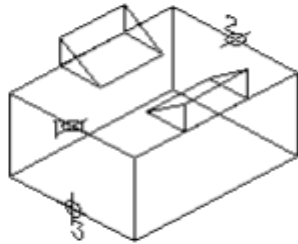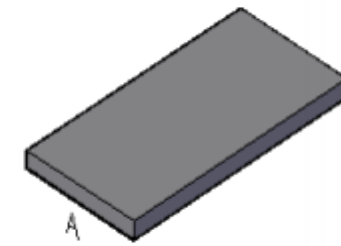Specify height or [2Point] <-0.2500>: **2** <Enter>

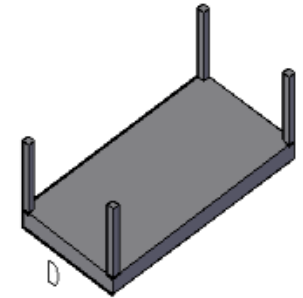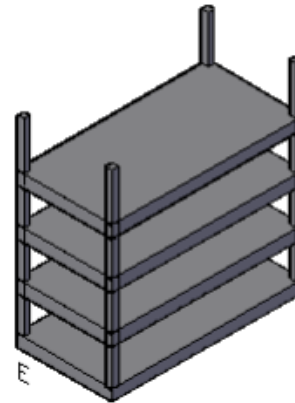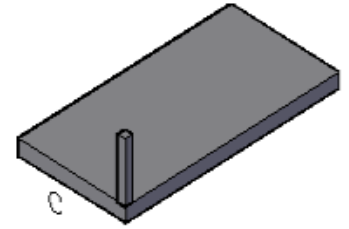Command: **3darray** <Enter>

Figure 1

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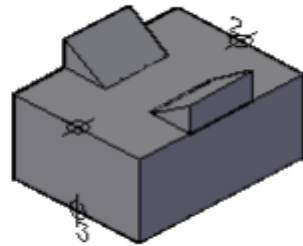
Select objects: 1 found   {select all the boxes}
Select objects: 1 found, 2 total
Select objects: 1 found, 3 total
Select objects: 1 found, 4 total
Select objects: 1 found, 5 total
Select objects: 1 found, 5 total
Select objects: <Enter>

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>

C

D

E

A

B

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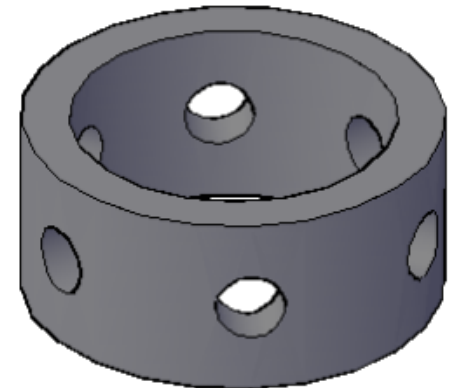
Modify toolbar:

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]:
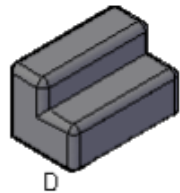*{select all the edges as highlighted in figure B}*
Select an edge or [Chain/Radius]:
Select an edge or [Chain/Radius]:
Select an edge or [Chain/Radius]:
Select an edge or [Chain/Radius]:
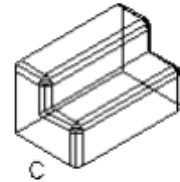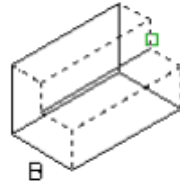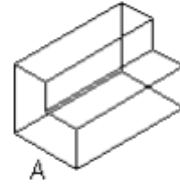Select an edge or [Chain/Radius]:
Select an edge or [Chain/Radius]:
Select an edge or [Chain/Radius]:
Select an edge or [Chain/Radius]:
Select an edge or [Chain/Radius]:
10 edge(s) selected for fillet.

Command: **cha** <Enter>

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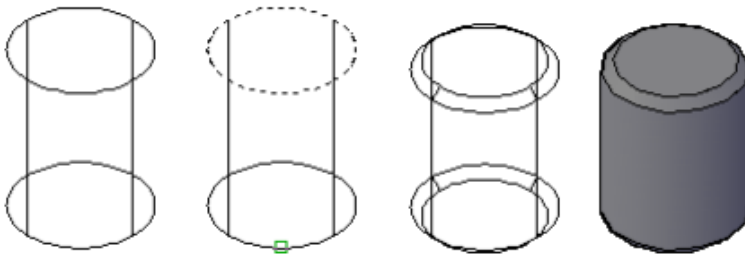Base surface selection...
Specify base surface chamfer distance <0.0000>: .5
Specify other surface chamfer distance <0.0000>: .5
Select an edge or [Loop]: Select an edge or [Loop]: Select an edge or
[Loop]:

A

B

C

D

⬙ 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.
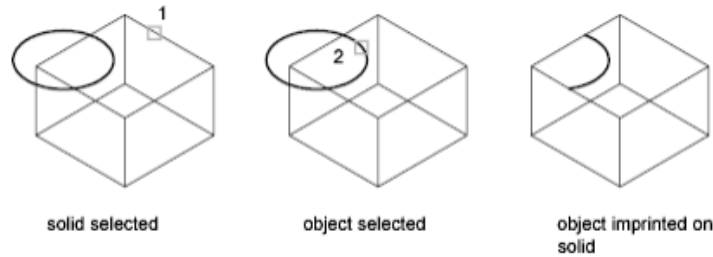
solid selected   object selected   object imprinted on solid

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>
```
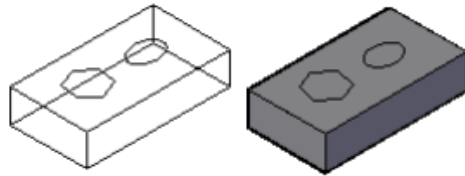
Figure 2

```
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
```
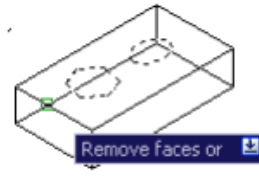
Remove faces or

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
```
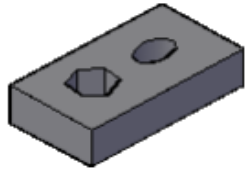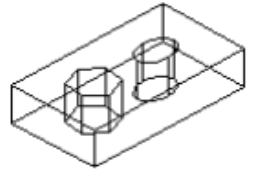
Figure 4

⬚ Shell   **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
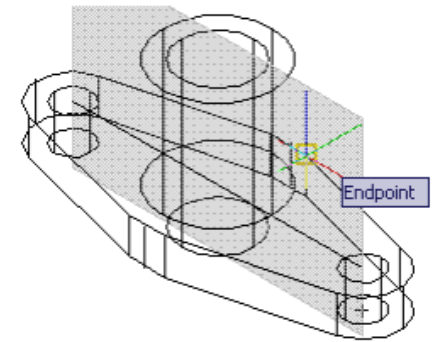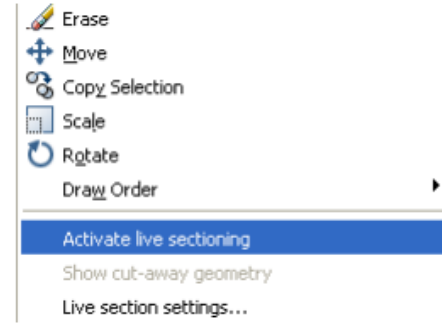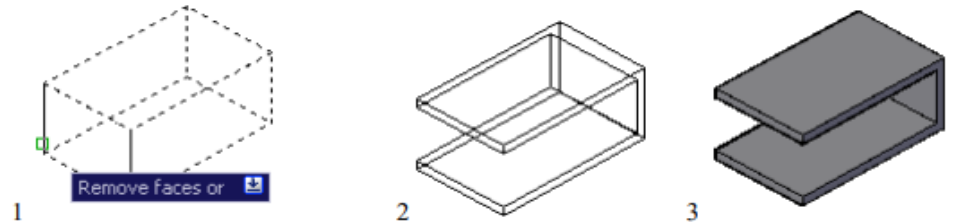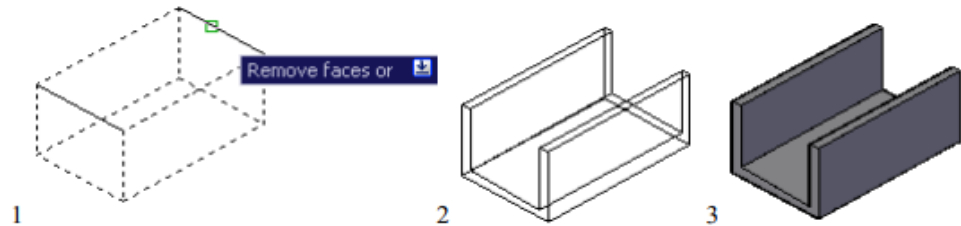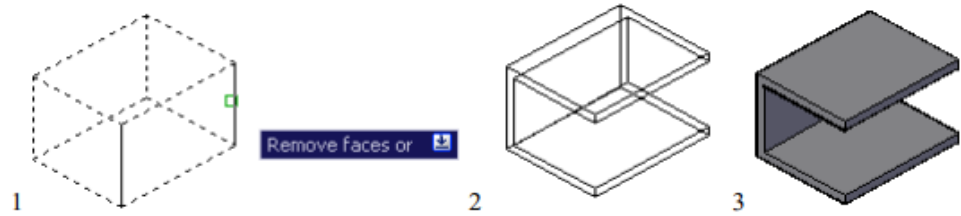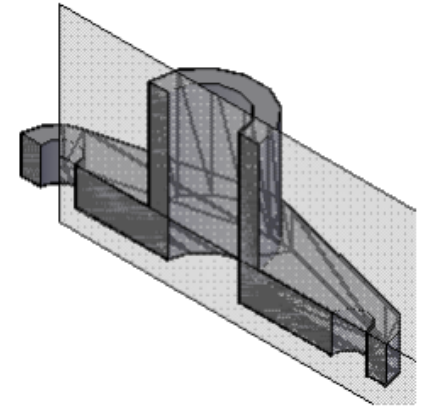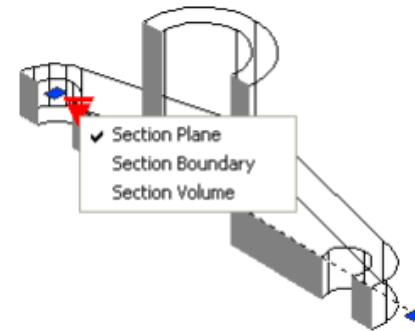Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: X <Enter>

Figure 3

Figure 2

Figure 4

Exercise 24-1