

Take a Giant Step into the World of 3D

Tom Short, P.E.

Code GD11-2

Are you an experienced AutoCAD® 2D user who's been wondering how to get started in 3D? If so, this is the course for you, regardless of whether you work in mechanical design or in architecture. In this class, you'll learn how to create, manipulate, and display solids in AutoCAD. You'll gain a basic understanding of working in 3D as well as techniques that will get you started. This is also excellent background information for both mechanical and architectural 3D programs.

About the Speaker:

Thomas Short

Tom is a consultant and mechanical engineer. He has worked with AutoCAD® since 1983 and has taught thousands how to design in 2D and 3D. He has consulted with many companies in industries including automotive, aircraft, consumer products, and the military on optimizing their use of AutoCAD, Autodesk® Mechanical Desktop®, and Autodesk Inventor®. With his dynamic teaching style and depth of knowledge, Tom is a popular presenter at AU, and has been for every one.

Part 1

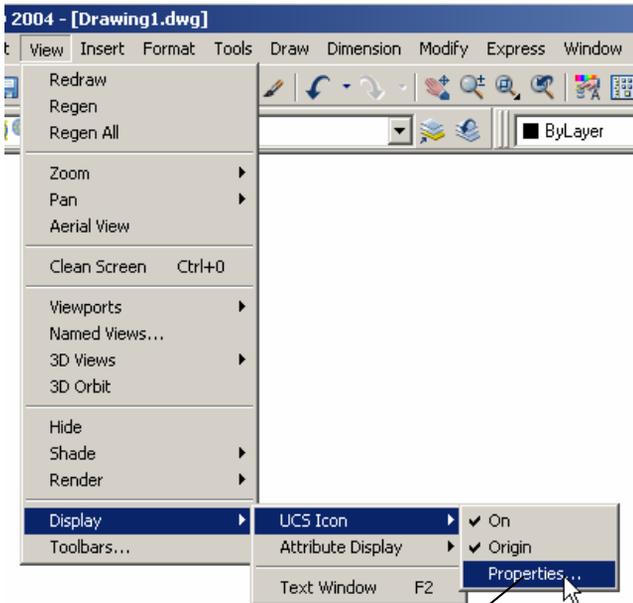
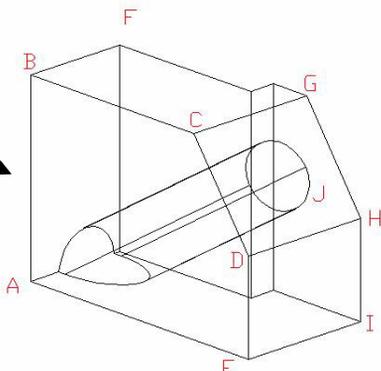
Here's Looking at you

- 1. Controlling the UCS
- 2. Controlling the Viewpoint
- 3. Controlling the Display – Hide & Shade
- 4. Controlling the View Ports
- 5. Controlling the Clipping Planes
- 6. Capturing and Inserting Images

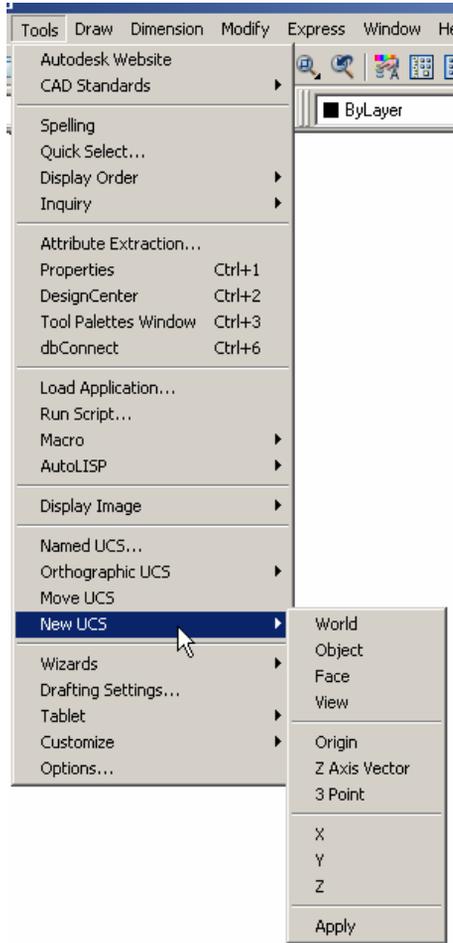
Step 1 - The User Coordinate System

Open drawing S-1-02.dwg

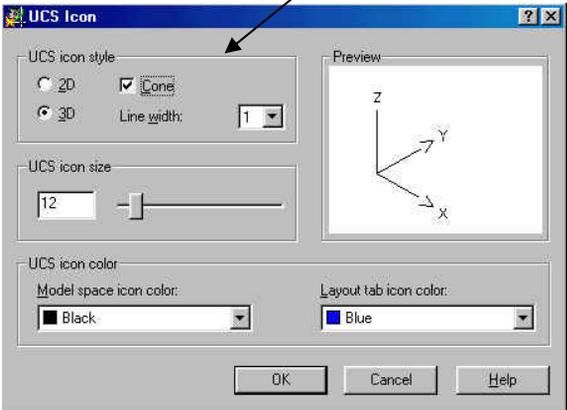
Display the UCS



Moving the UCS

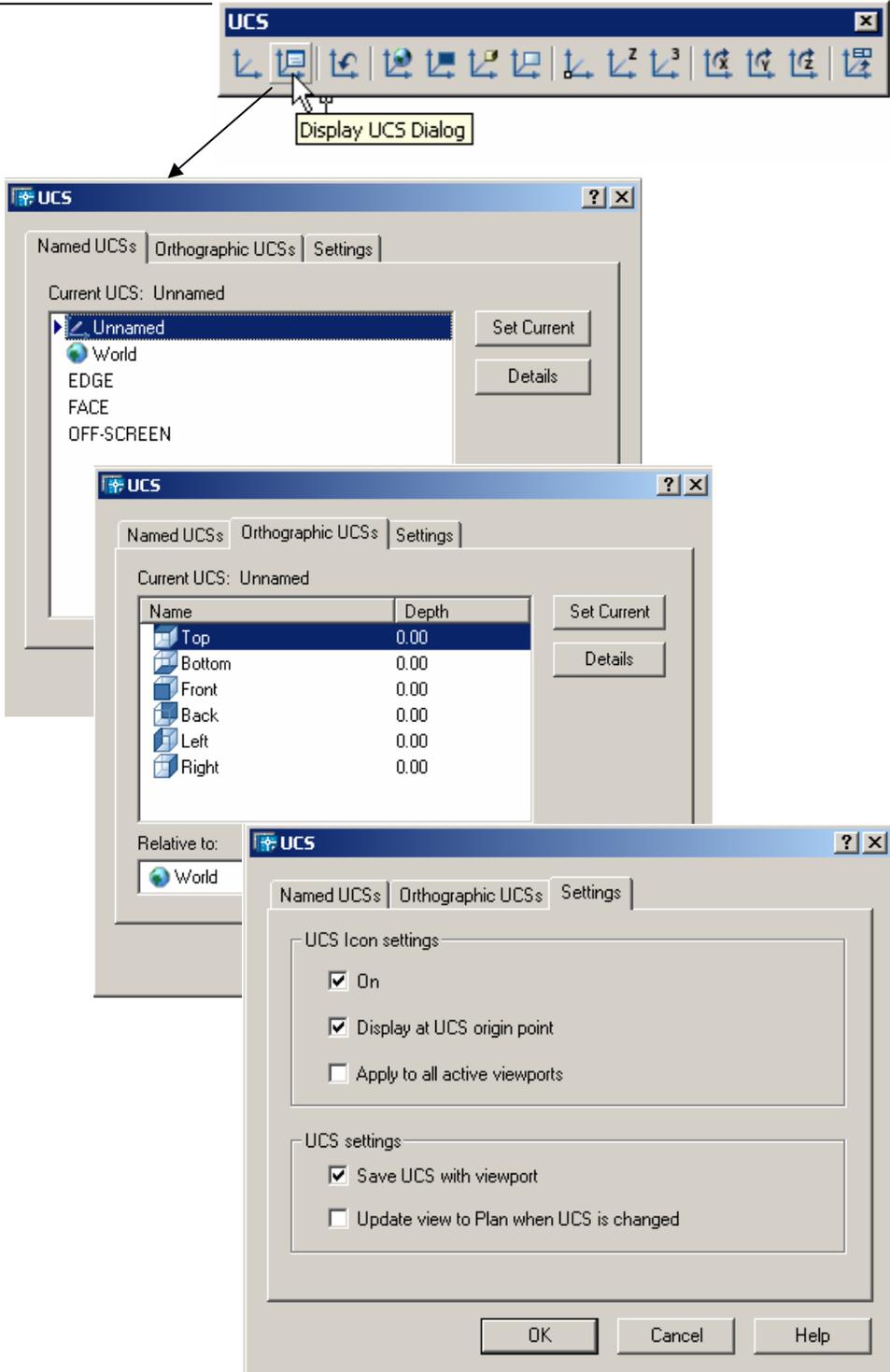


UCS Properties



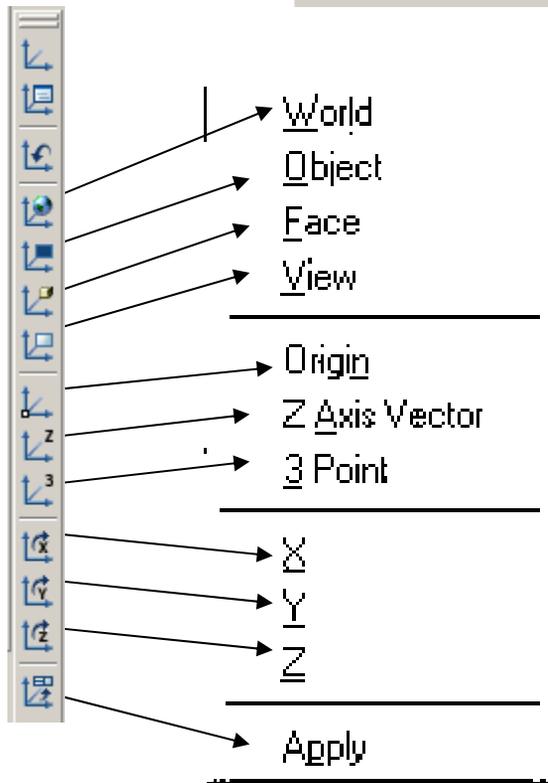
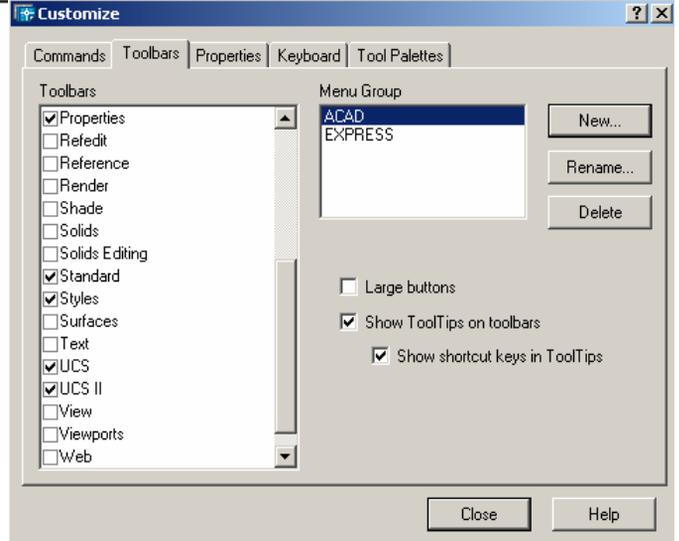
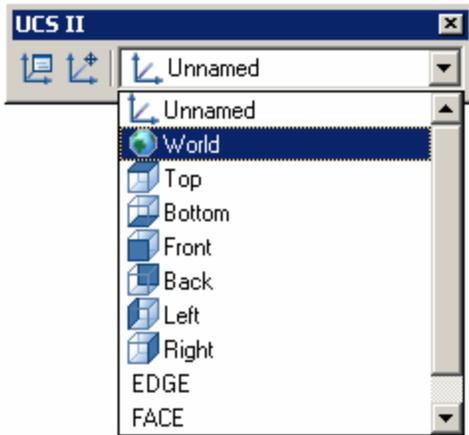
Step 1 Continued

The UCS Toolbars



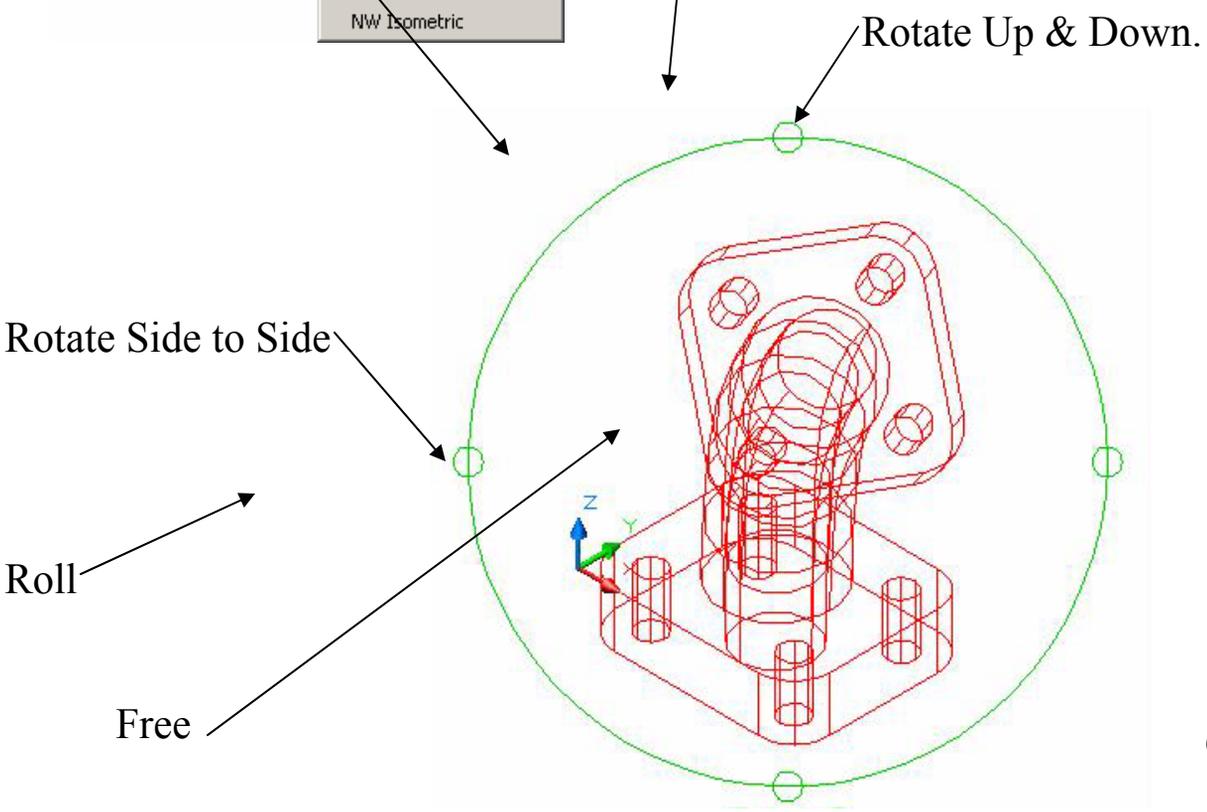
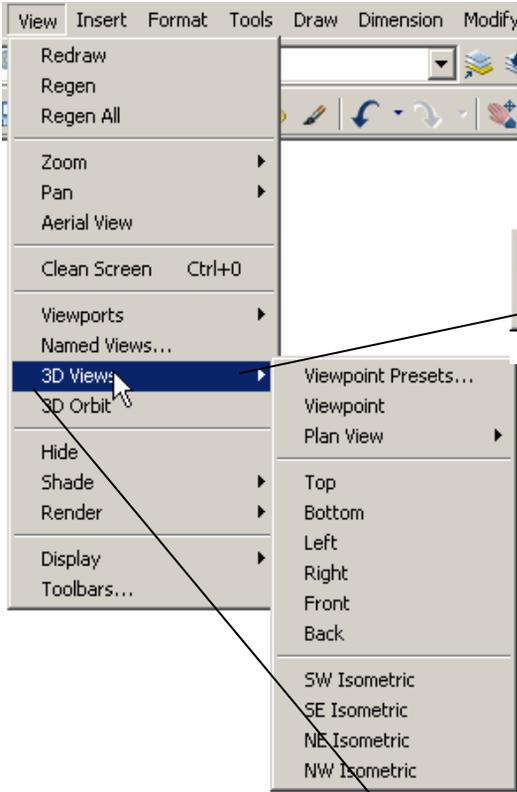
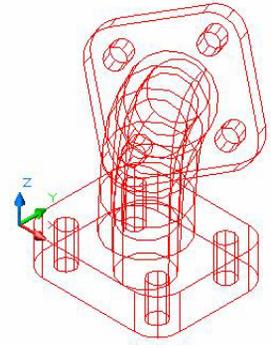
Step 1 Continued

The UCS Toolbars



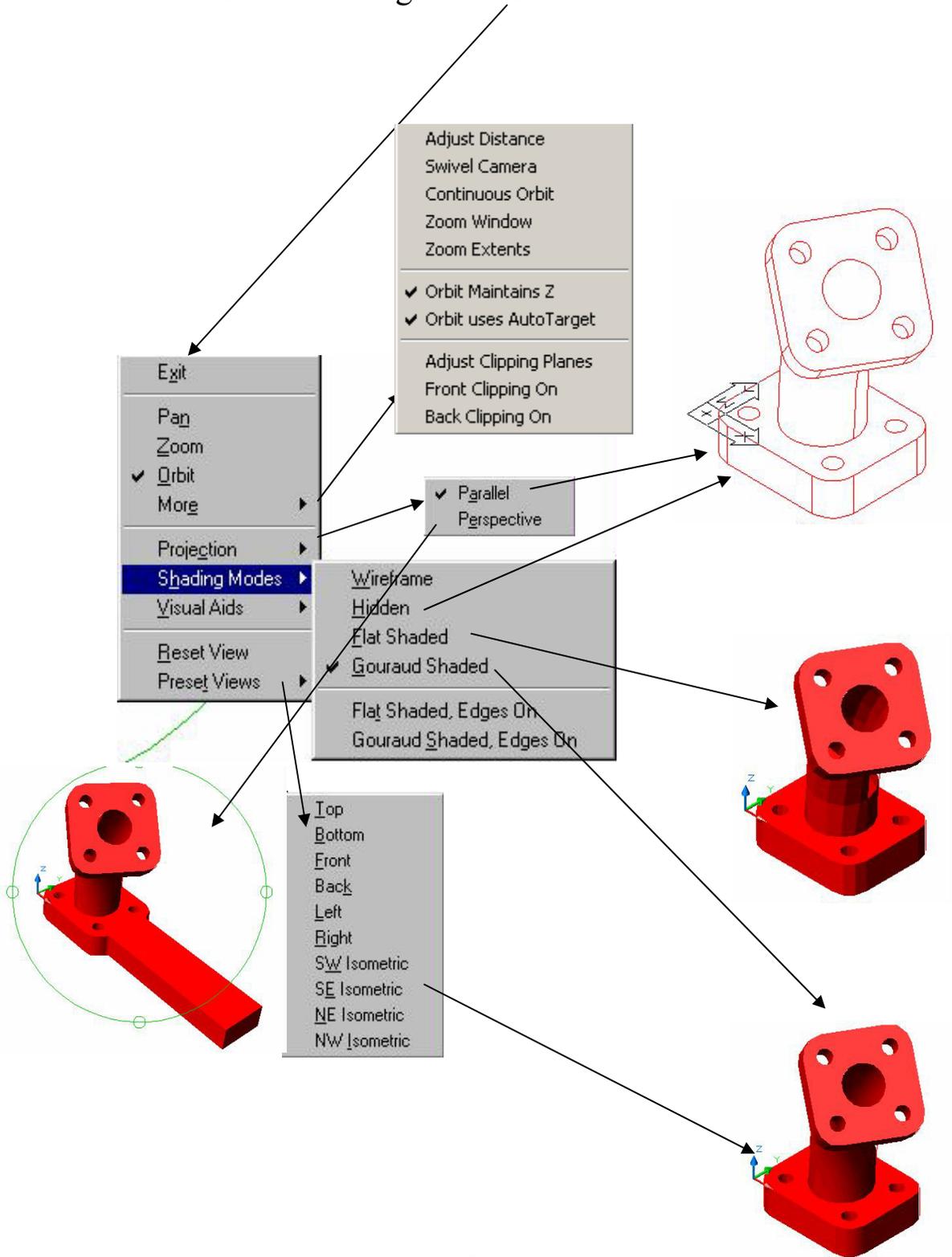
Step 2 - Controlling the Viewpoint.

Open drawing *S-1-07.dwg*.



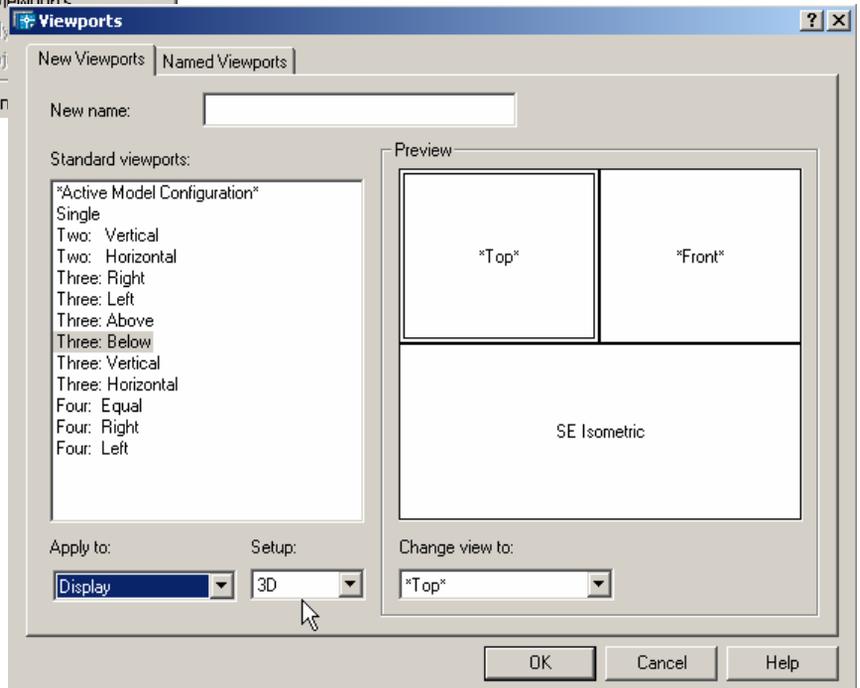
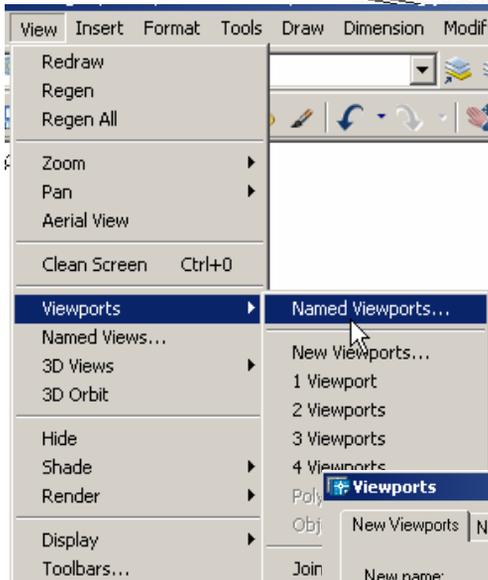
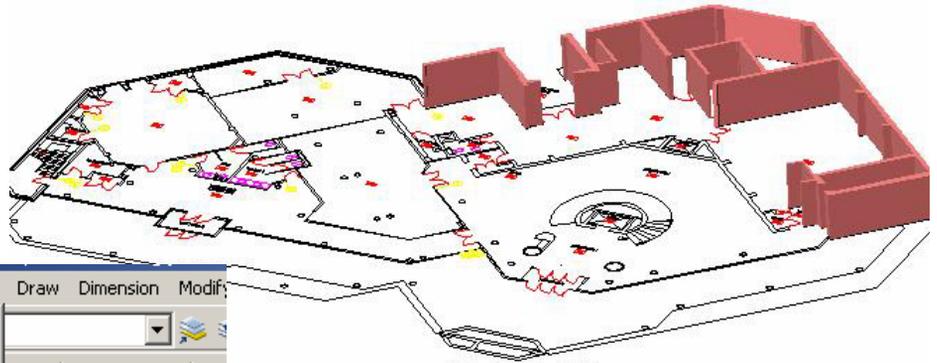
Step 3 – Controlling the Display, Hide & Shade

Click on Orbit then Right Mouse



Step 4 - Controlling the Viewports.

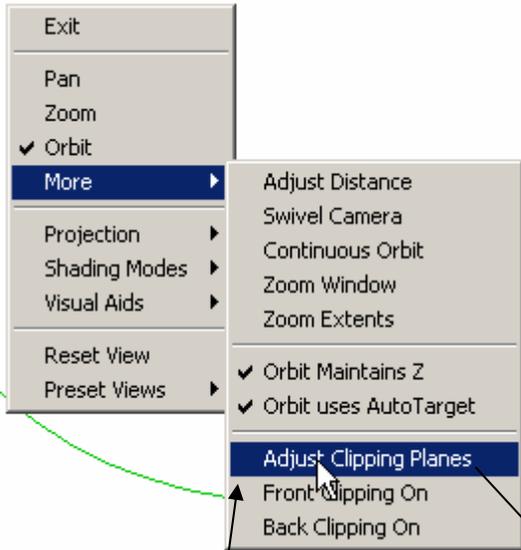
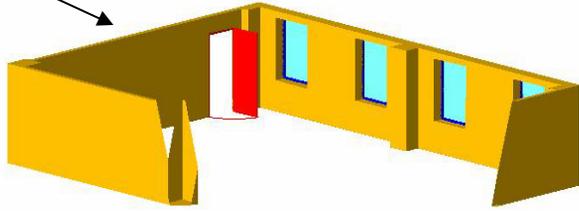
Open drawing
S-1-12.dwg.



Step 5 - Controlling the Clipping Planes.

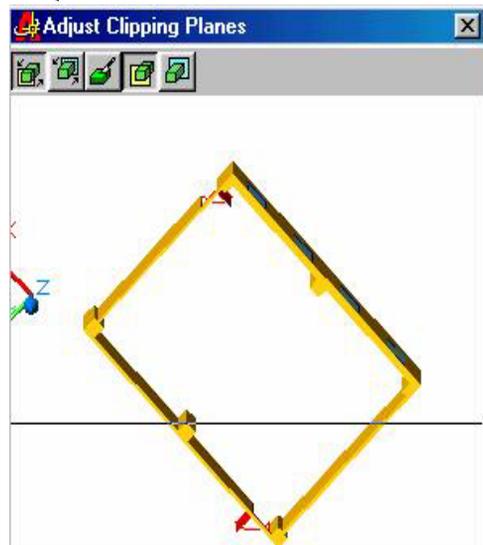
Open drawing *S-1-14.dwg*.

Click on Orbit then
Right Mouse



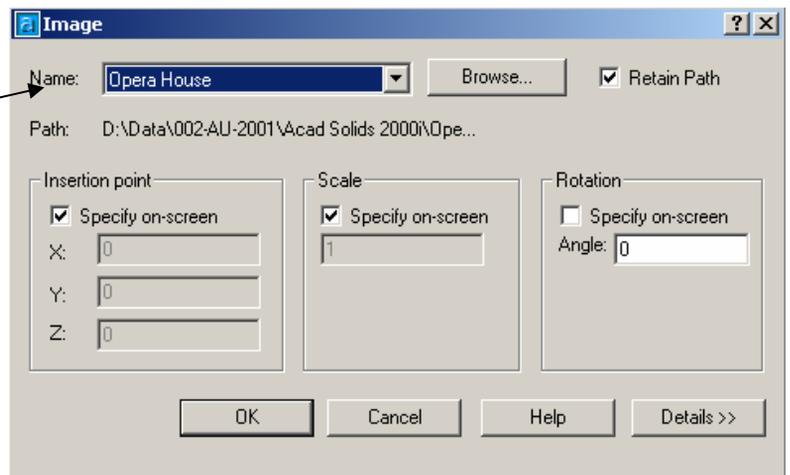
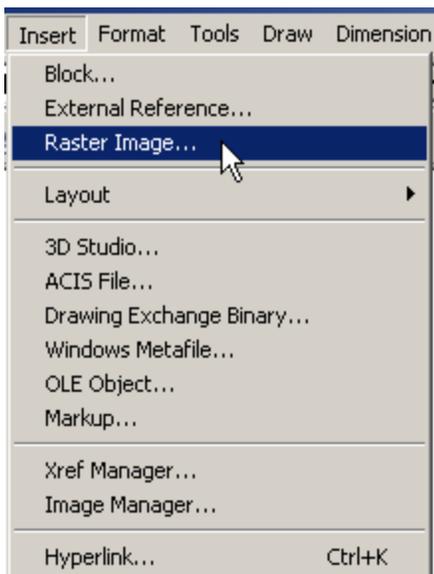
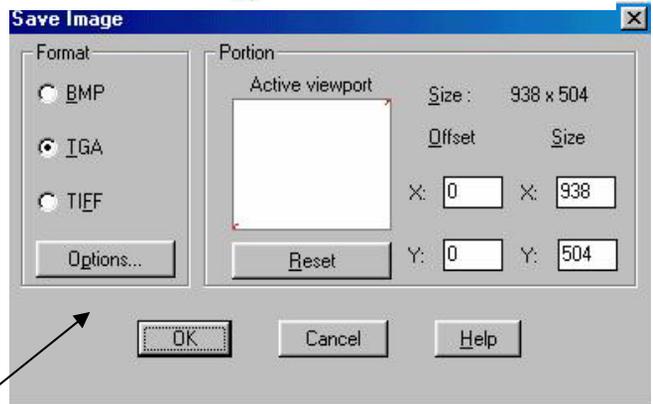
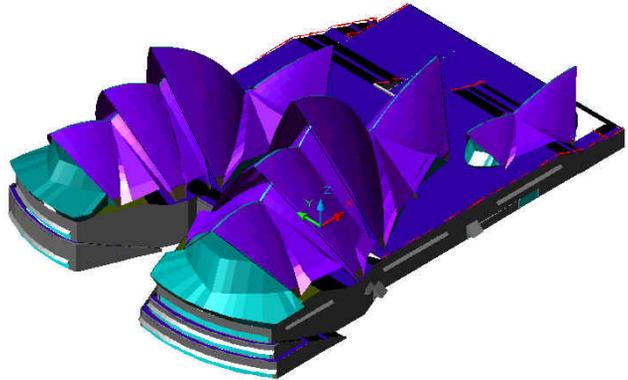
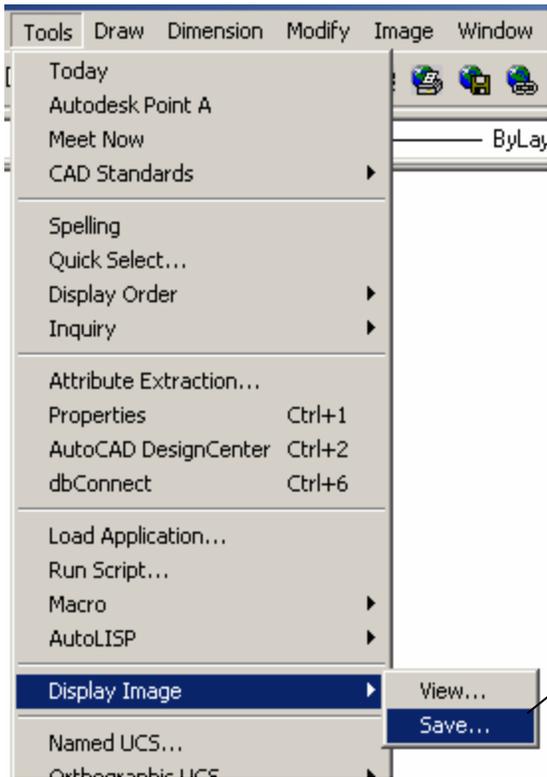
The front clipping
plane, which is
parallel to the
screen is on

Adjusts position of
clipping plane



Step 6 – Capturing & inserting Images

Open drawing *S-1-15.dwg*.



Part 2

Like a rock

1. Creating and Combining Boxes
2. Creating and Combining Cylinders
3. Using the Torus Primitive
4. The Shell Command
5. Mass and Geometric Properties

Step 1 - Creating and Combining Boxes



Start a new file

Command: box
Specify corner of box or [CEnter] <0,0,0>:
Specify corner or [Cube/Length]: 4,4
Specify height: 1.5

Command: box
Specify corner of box or [CEnter] <0,0,0>: 0,4,0
Specify corner or [Cube/Length]: 4,5,6

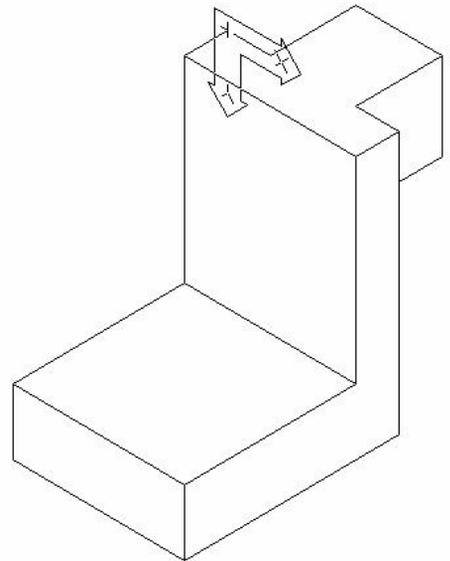
Set view to SE Isometric

Command: ucs
Current ucs name: *WORLD*
Enter an option [New/Move/orthoGraphic/Prev/Restore/Save/Del/Apply/?/World]
<World>: x
Specify rotation angle about X axis <90>: -90

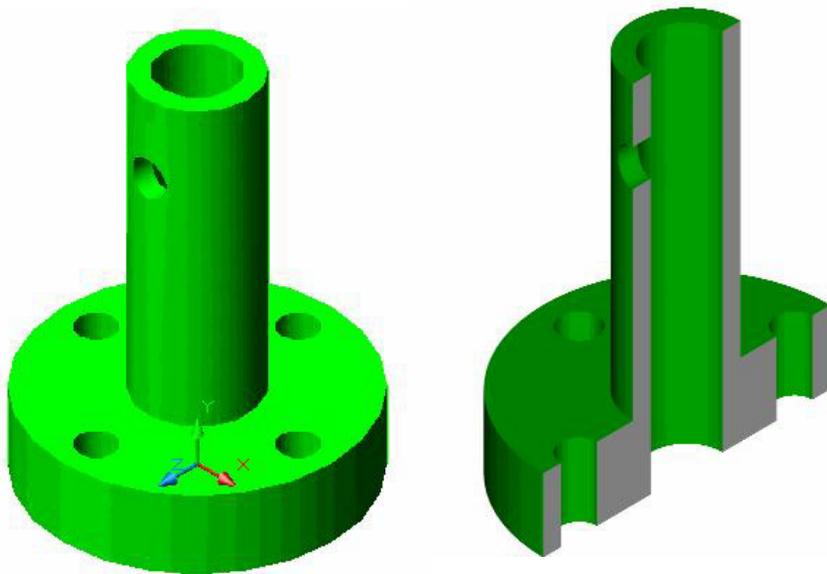
Command: ucs
Current ucs name: *NO NAME*
Enter an option [New/Move/orthoGraphic/Prev/Restore/Save/Del/Apply/?/World]
<World>: o
Specify new origin point <0,0,0>: end of
Pick upper back corner of second box

Command: box
Specify corner of box or [CEnter] <0,0,0>: 1,0,0
Specify corner or [Cube/Length]: c
Specify length: 2

Command: union
Select objects: all
3 found



Step 2 - Creating and Combining Cylinders



Start a New Drawing

CYLINDER Number 1

Center point: 0,0,0 Radius: 2 Height of Cylinder: 1

CYLINDER Number 2

Center point: 0,0,0 Radius: 0.75 Height of Cylinder: 5

Union Cylinders 1 and 2

CYLINDER Number 3

Center point: 0,0,0 Radius: 0.5 Height of Cylinder: 5

Subtract Cylinder 3 from the combination of 1 and 2, we'll call this the solid.

CYLINDER Number 4

Center point: 1.5,0,0 Radius: .25 Height of cylinder: 1

Use ARRAY to make copies: Array - Polar

Subtract the four holes from the solid.

CYLINDER Number 5

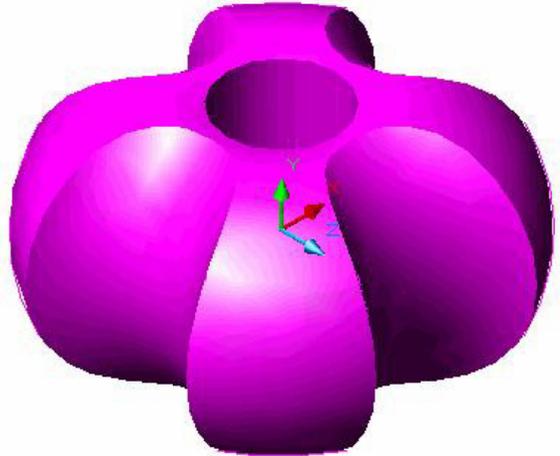
Now put a hole through the upper end of the shaft parallel to the Y axis.

Rotate the UCS 90 degrees about the X axis.

Create cylinder Number 5 with center at 0,4,0 and a 0.25 radius with a height of 1.0

Subtract cylinder 5 from the solid.

Step 3 - Using the Torus Primitive



The torus or doughnut requires 3 dimensions: the center, the radius of the torus, and the radius of the tube.

Start a New Drawing

Create Torus Number 1 with center at 0,0,0; radius = 1.125; radius = 1.5

Create a Cylinder with center at 0,0,-2; radius = 0.75; height = 4.0

Subtract the cylinder from the torus. This is the "Solid"

Rotate the UCS about the X axis 90 degrees

Create Torus Number 2 with center at 0,0,0; radius = 3.0; radius = 1.25

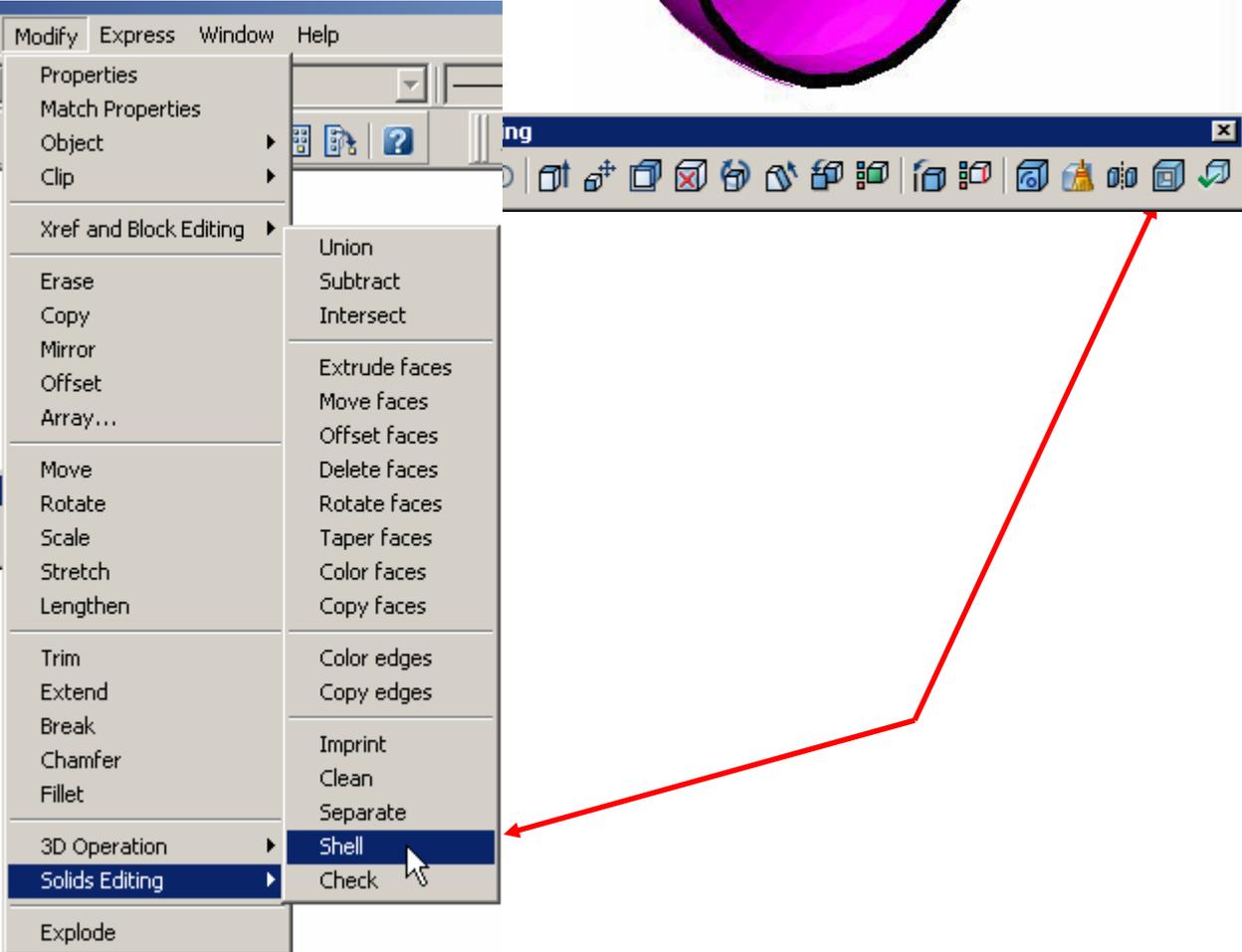
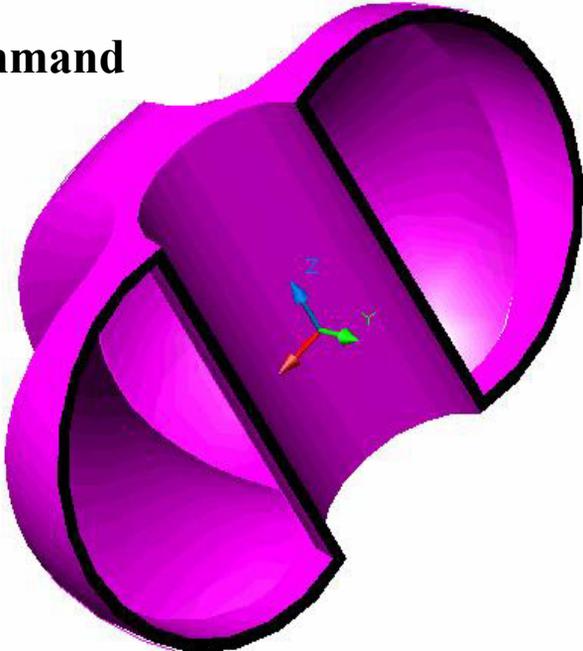
Subtract Torus Number 2 from the Solid

Rotate the UCS about the Y axis 90 degrees

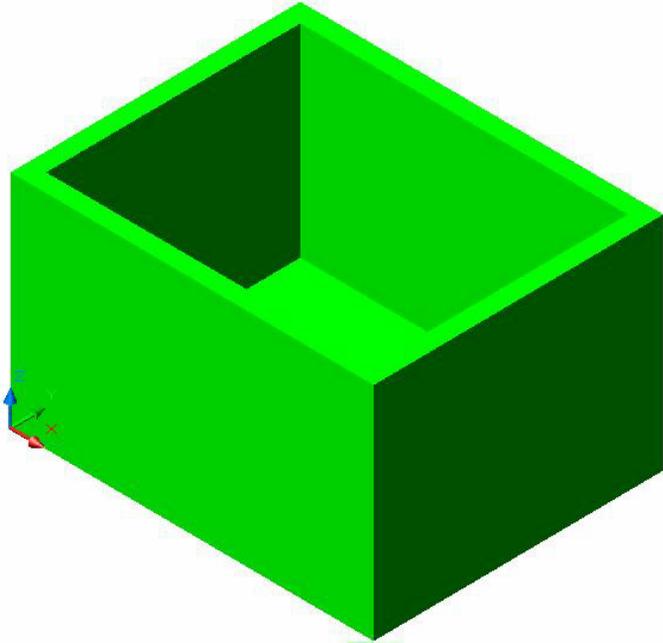
Create Torus Number 3 with center at 0,0,0; radius = 3.0; radius = 1.25

Subtract the Torus Number 3 from the Solid

Step 4 - The Shell Command



Step 4 - The Shell Command Continued

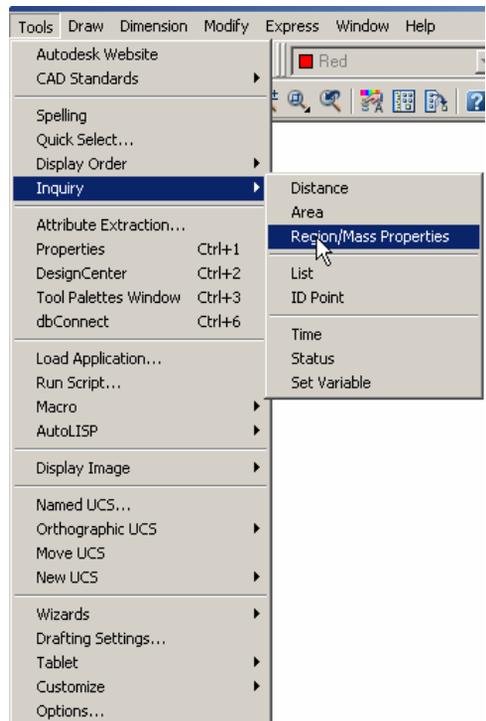
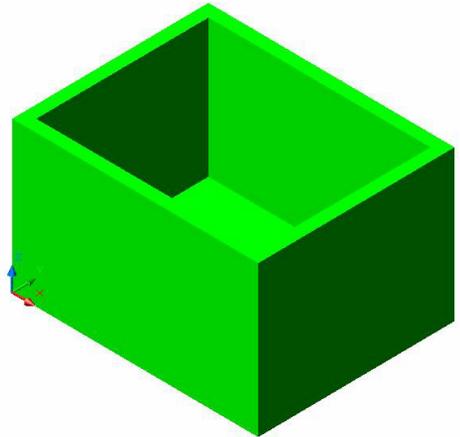


- Start a New Drawing
- Box from 0,0,0 to 5,4,3
- Select Shell Icon
- Select Box
- Select Top Face (no indication of selection is shown)
- Enter
- Input wall thickness of 0.25 (a negative value makes box bigger)
- Enter Enter

Step 5 - Mass and Geometric Properties

----- SOLIDS -----

Mass: 16.6875
 Volume: 16.6875
 Bounding box: X: 0.0000 -- 5.0000
 Y: 0.0000 -- 4.0000
 Z: 0.0000 -- 3.0000
 Centroid: X: 2.5000
 Y: 2.0000
 Z: 1.1756
 Moments of inertia: X: 140.8672
 Y: 194.5391
 Z: 258.7422
 Products of inertia: XY: 83.4375
 YZ: 39.2344
 ZX: 49.0430
 Radii of gyration: X: 2.9054
 Y: 3.4143
 Z: 3.9377
 Principal moments and X-Y-Z directions about centroid:
 I: 51.0560 along [1.0000 0.0000
 0.0000]
 J: 67.1810 along [0.0000 1.0000
 0.0000]
 K: 87.6953 along [0.0000 0.0000
 1.0000]



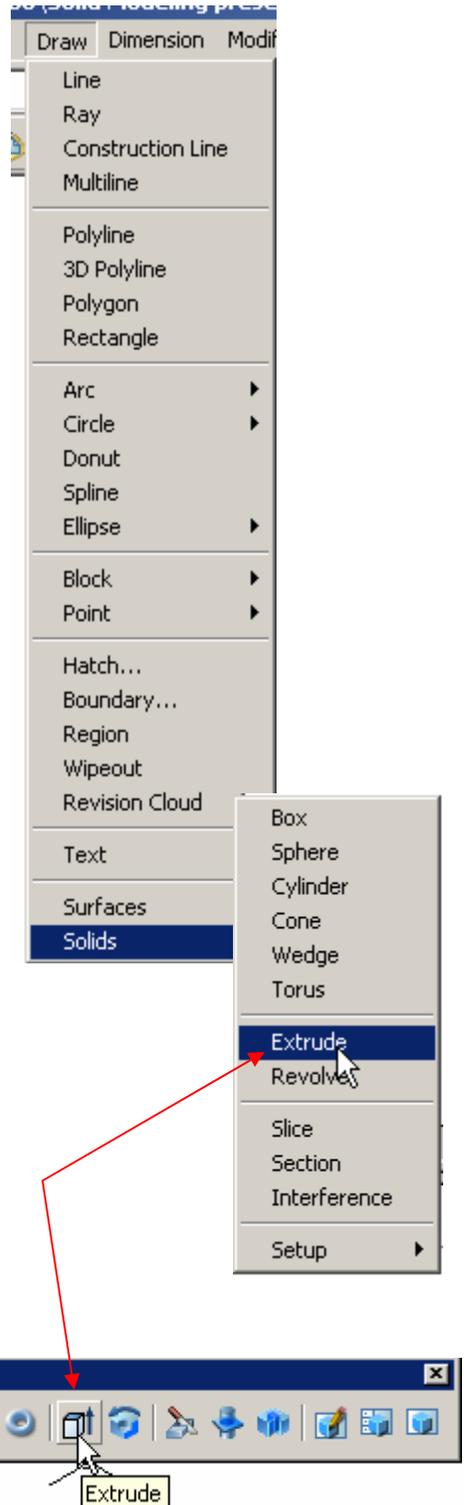
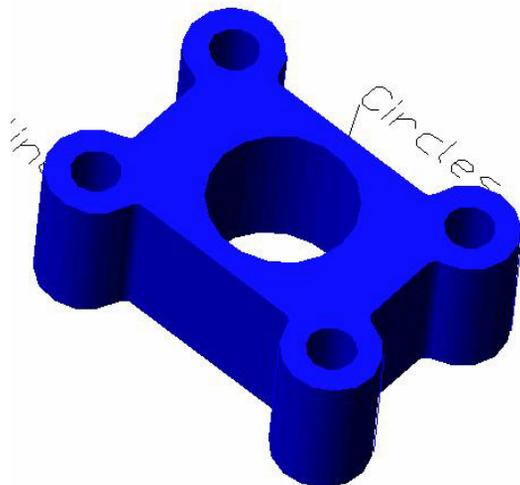
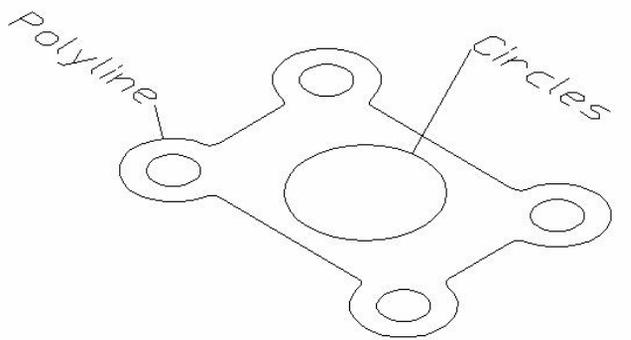
Part 3

We Have Liftoff

1. The Extrude Command
2. The Draft Angle
3. Changing Face Draft Angle & Face Colors
4. Extrude Along a 2D Path
5. Extrude Along a 3D Path
6. Extrude a Face
7. The Intersect Command
8. The Boundary Command
9. The Revolve Command

Step 1 - The Extrude Command

- Open Part S-3-01.dwg
- Extrude the Polyline and Circles 3 units.
- Subtract the extruded circles.



Step 1 – Continued

What Shapes Can You Extrude?

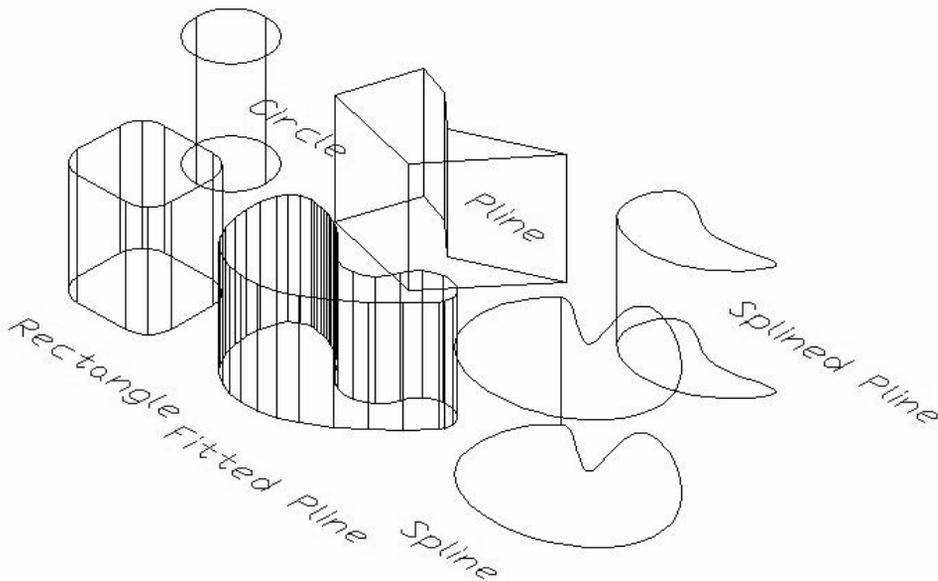
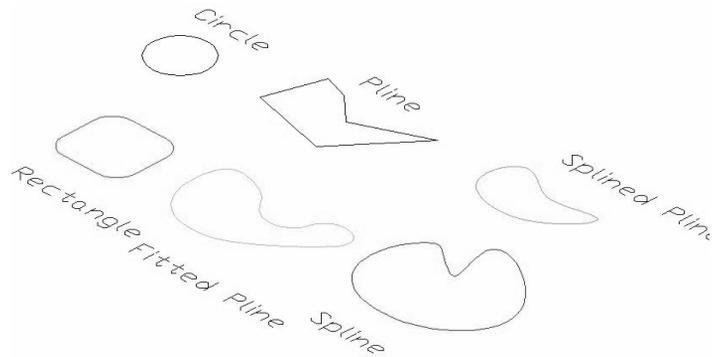
Open drawing *S-3-02.dwg*

Any shape that is:

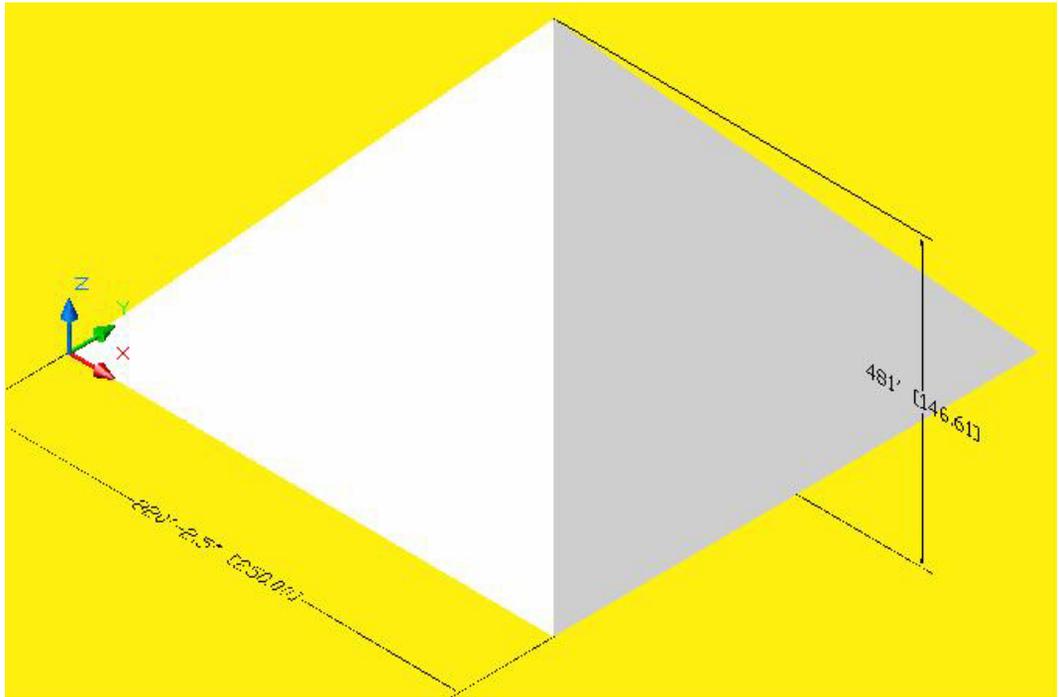
- Closed
- Flat (planar)
- Does not cross its self
- Contains no islands

can be extruded

You can extrude several shapes in one command.



Step 2 - The Draft Angle.



Open the drawing *S-3-03.dwg*

Draw a square 250 x 250 (that's meters!)

Extrude 146 meters with a draft angle of:

Extrusion taper angle <0>: 'cal

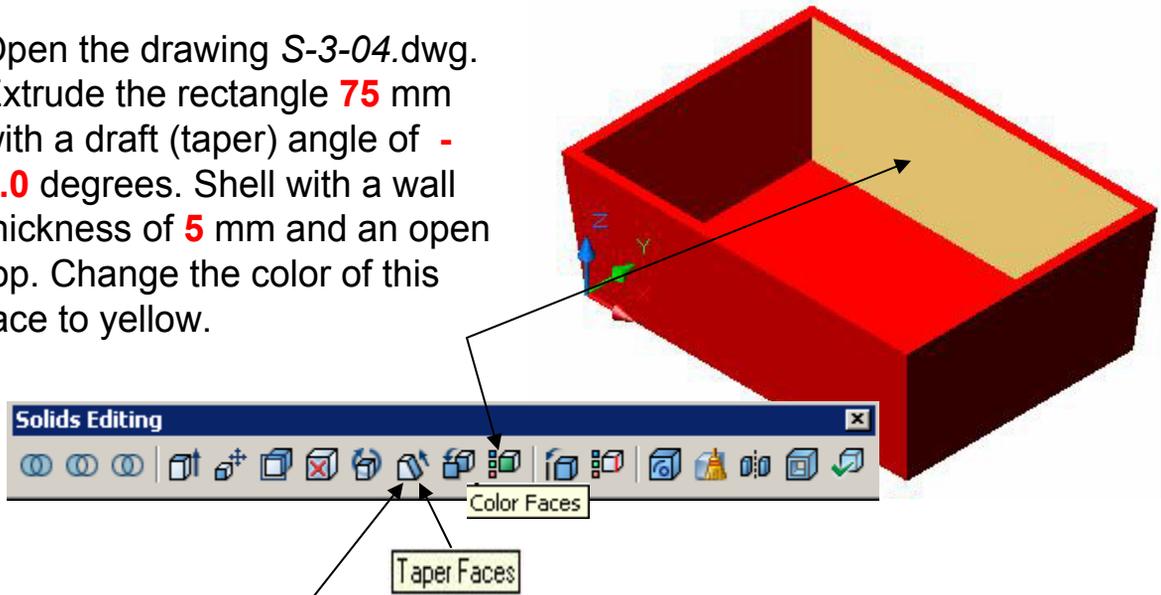
Initializing...>> Expression: atan(125/146)

What happens if the angle is smaller, 10 degrees?

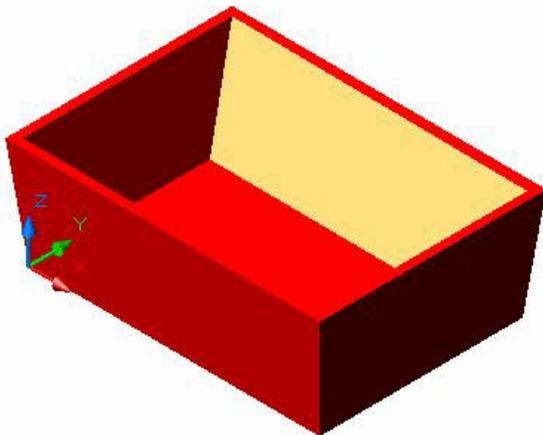
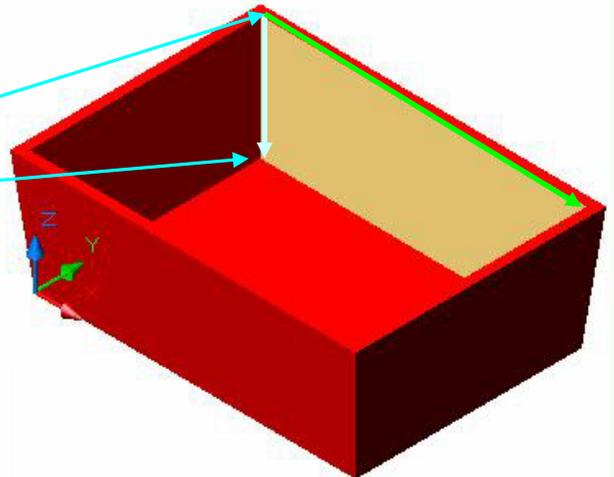
What happens if the angle is larger, 40 degrees?

Step 3 - Changing the Draft Angle on a Face and Face Colors.

Open the drawing S-3-04.dwg. Extrude the rectangle **75** mm with a draft (taper) angle of **-3.0** degrees. Shell with a wall thickness of **5** mm and an open top. Change the color of this face to yellow.



Select Taper Faces and select the yellow face. Pick end point B as the Base point and end point P as another point. Input an angle of **-12.0** degrees.

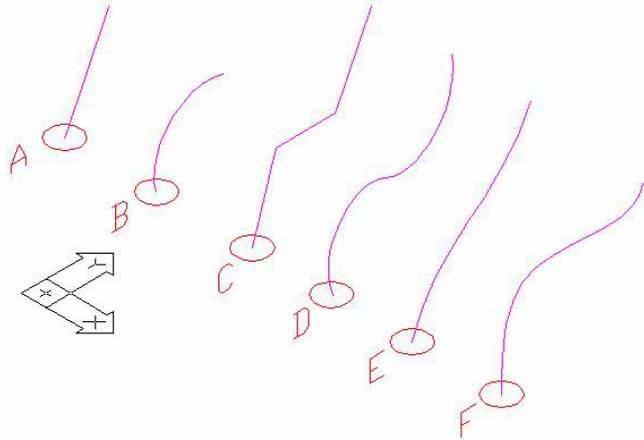


The face is rotated about the green arrow. Positive rotation is the curl of your fingers when you put your right thumb along the green arrow and your first finger along the light blue.

Step 4 - Extruding along 2D Path.

Open the drawing S-3-05.dwg. There are six circles and six 2D paths each starting at the center of the circle. The paths are a:

- A. Line
- B. Arc
- C. Polyline
- D. Fitted Polyline
- E. Splined Polyline
- F. Spline



The procedure is:

Command: Extrude

Current wire frame density: ISOLINES=4

Select objects: 1 found

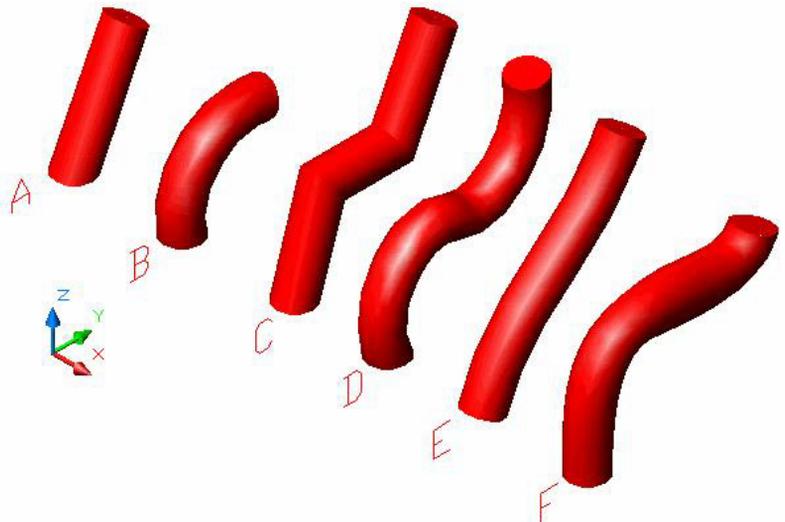
Select objects:

Specify height of extrusion or [Path]: Path

Select extrusion path:

The extruded shape can be any of the shapes that could be extruded a distance.

The shell command will turn the rod into tube. Only one shell per solid. Negative values for the offset shells to the outside of the part.



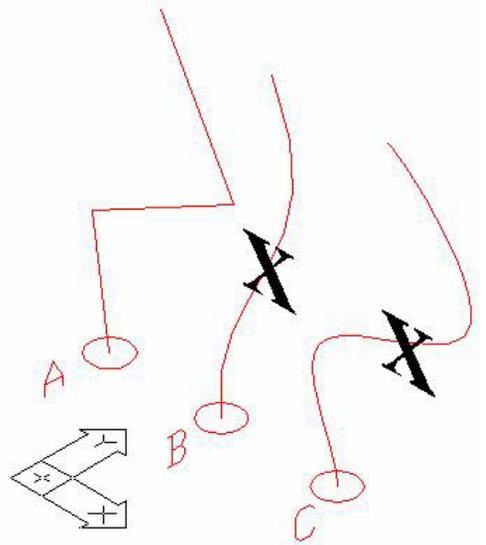


Step 5 - Extruding along a 3D Path.

Open the drawing S-3-06.dwg. There are three circles and three 3D paths each starting at the center of the circle. The paths are a:

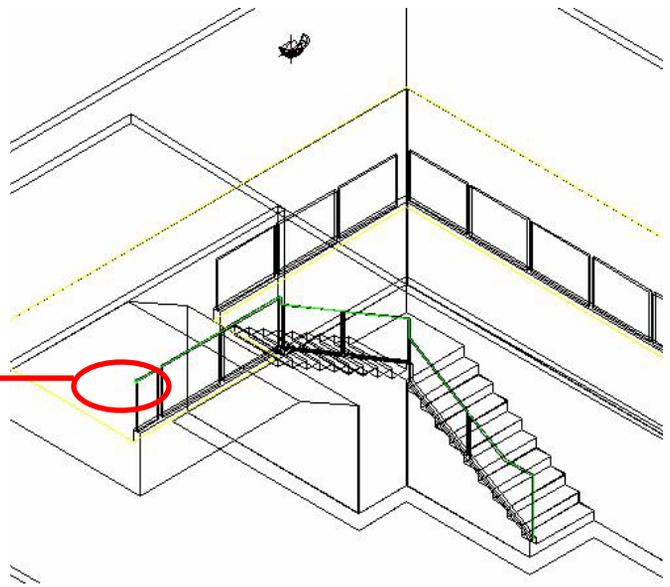
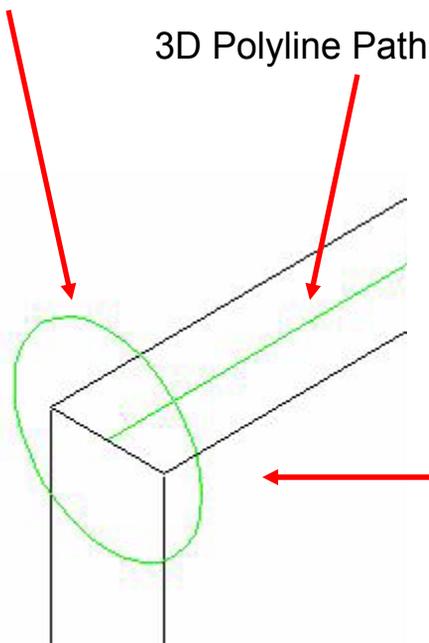
- A. 3D Polyline
- B. Splined 3D Polyline
- C. 3D NURBS Spline

Only A, the 3D Polyline containing lines, can be used as a path!



Open the drawing S-3-07.dwg. Extrude a rail for the stairway. Detail shows:

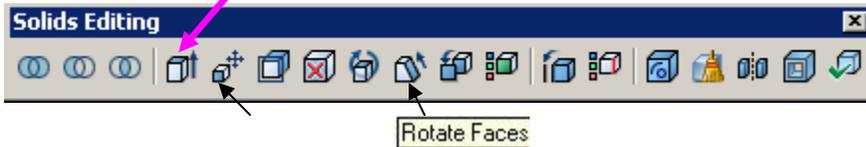
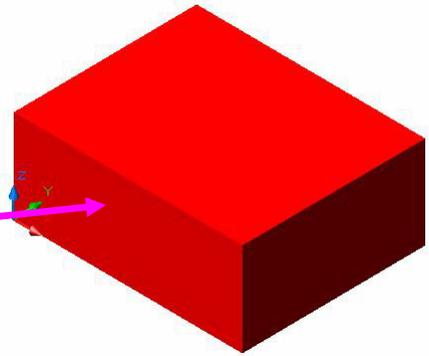
Object



Step 6 - Extrude a Face

Start a new drawing. Construct a Box from 0,0,0 to 5,4,2. Select Extrude Faces,

pick this face,
a distance of -1.0

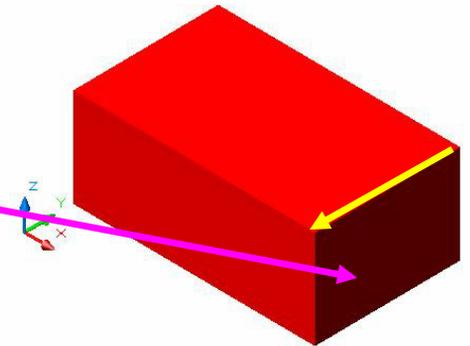


Select Rotate Faces.

pick this face,

Pick 2 endpoints at the yellow arrow for
an axis,

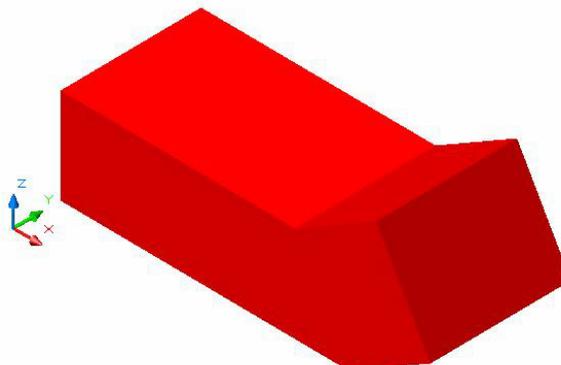
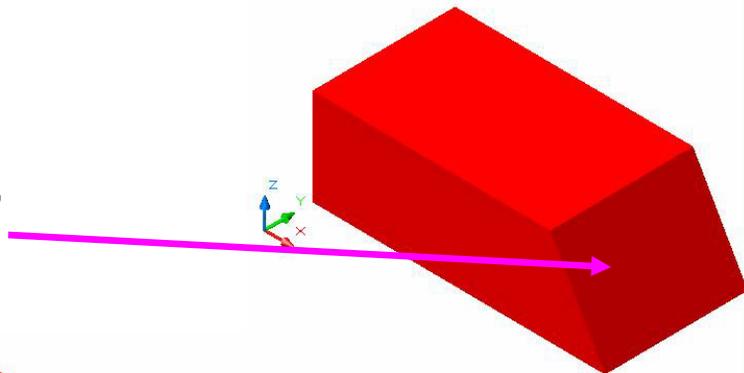
at an angle of **+30**



Select Extrude Faces,

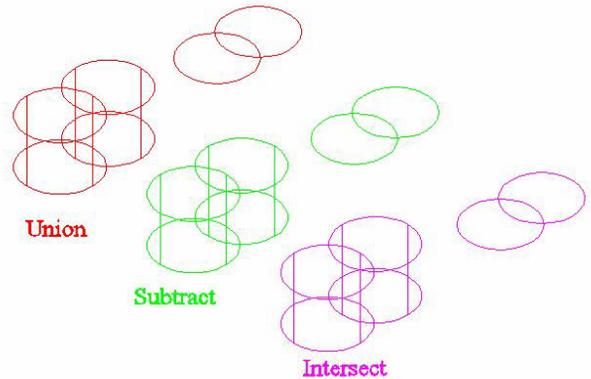
pick this face,

a distance of **+2.0**

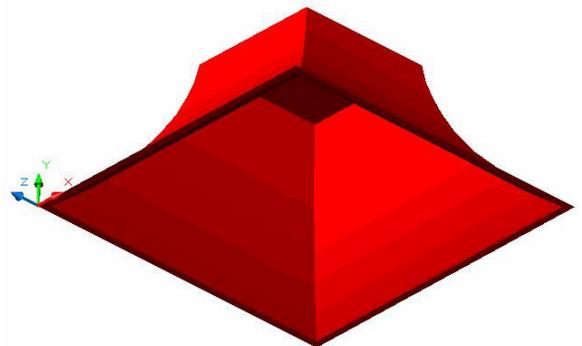
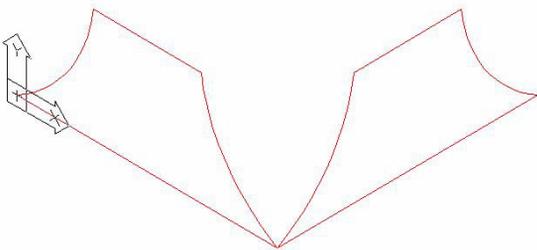
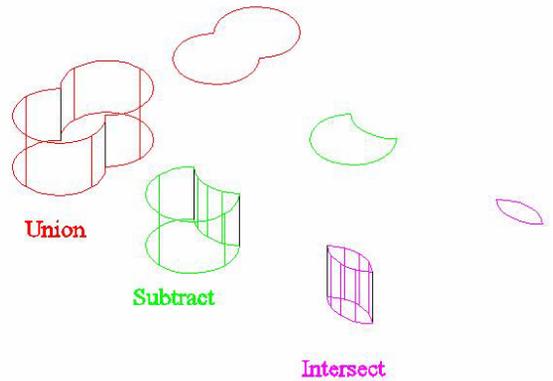


Step 7 - The Intersect command

Open the drawing *S-3-08.dwg*. Union, Subtract, and Intersect are much like a 3D version of trim. There are 3 ways to trim the circles and three ways to combine the solids.

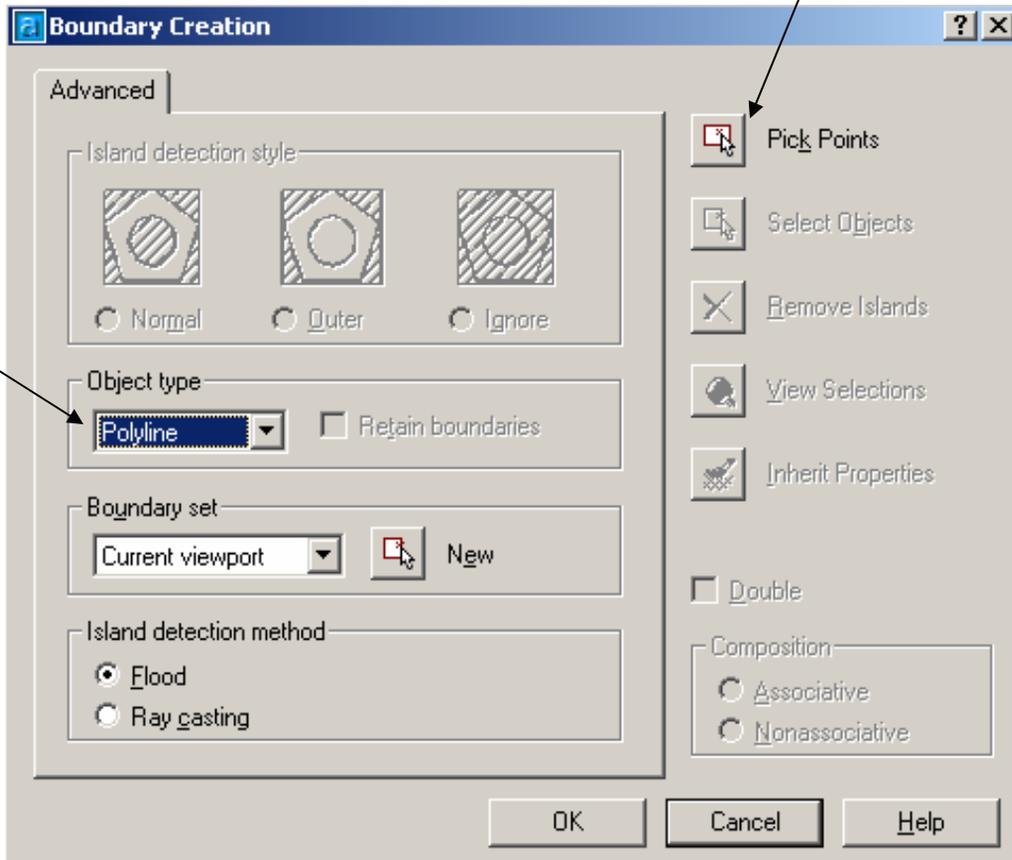
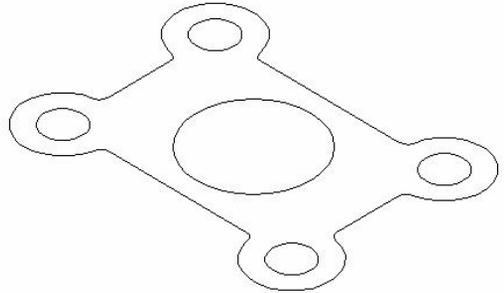


Open the drawing *S-3-09.dwg*. Extrude the left shape -20' and the right face +20' and then intersect the two solids. The pagoda roof can be shelled. To exclude the bottom face, rotate the part and pick in the face. The thickness is 3".



Step 8 - The Boundary Command

Open the drawing *S-3-10.dwg*. The Boundary command is much like Batch. Pick a point inside the shape but outside the circles and six “Boundaries” are created. These are sort of 2D solids that have area but no thickness. They can be Unioned and Subtracted and can be extruded into 3D Solids.

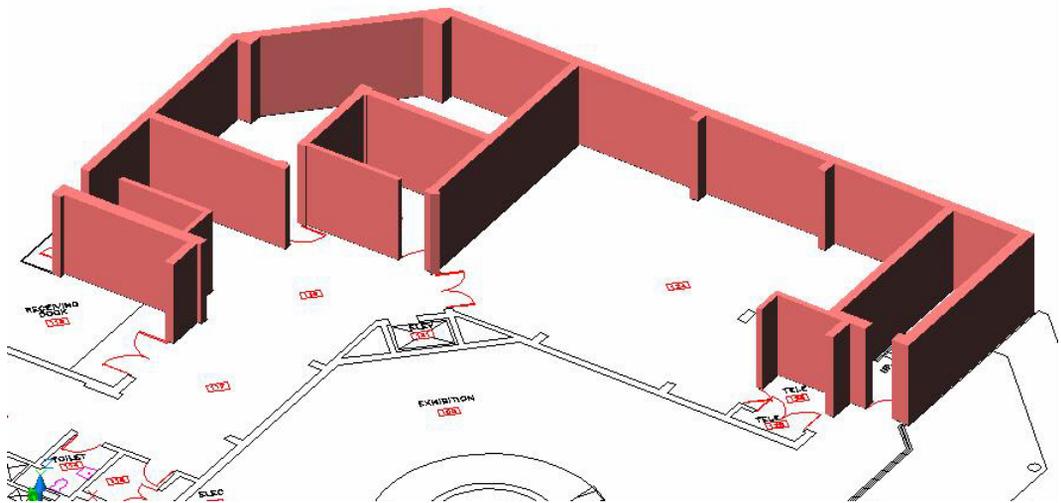
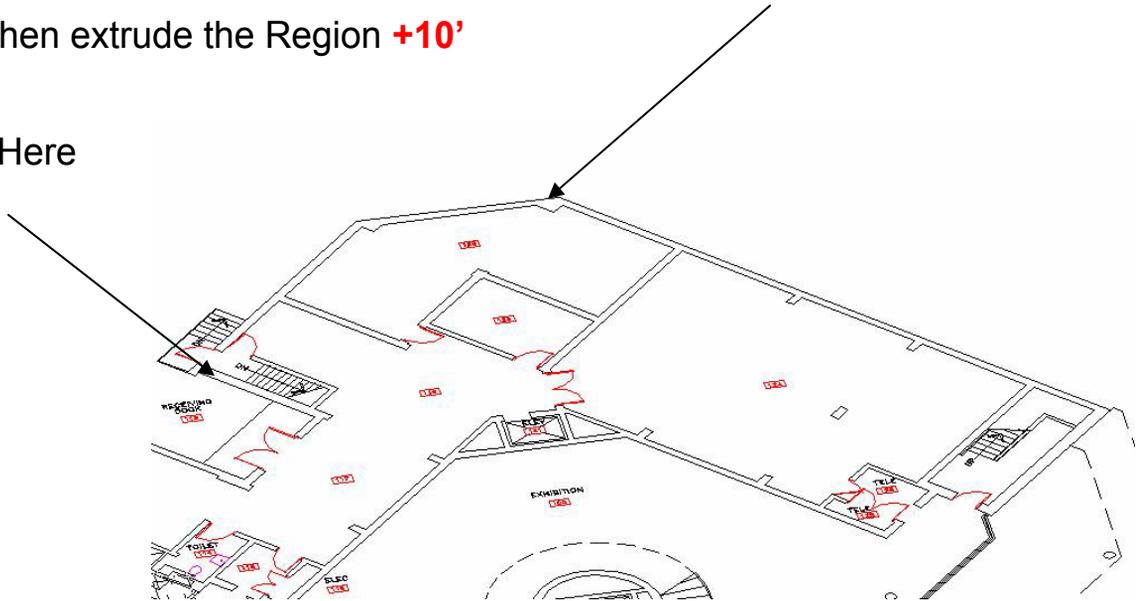


Step 8 - The Boundary Command Continued

Open the drawing *S-3-11.dwg*. Use the Boundary command to build a 3D wall.
Pick a point inside the wall
and then extrude the Region **+10'**

Pick Here

Pick Here

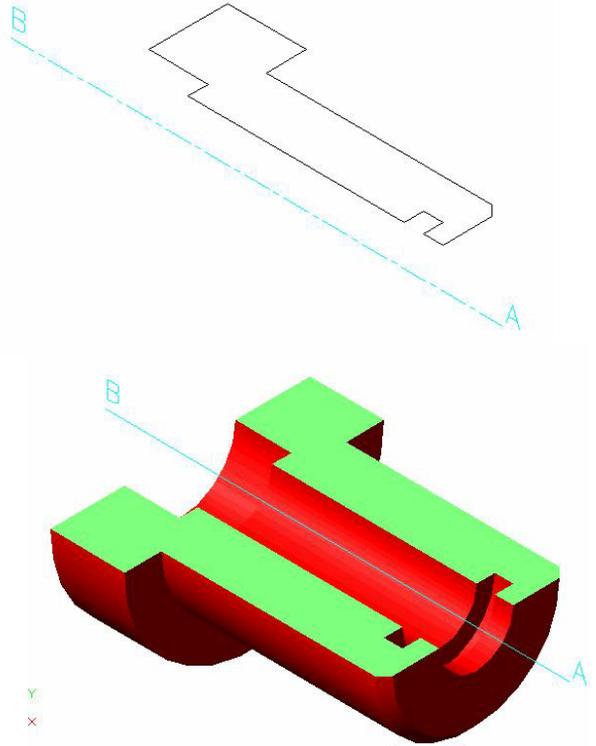


Step 9 - The Revolve Command

Open the drawing S-3-12.dwg.

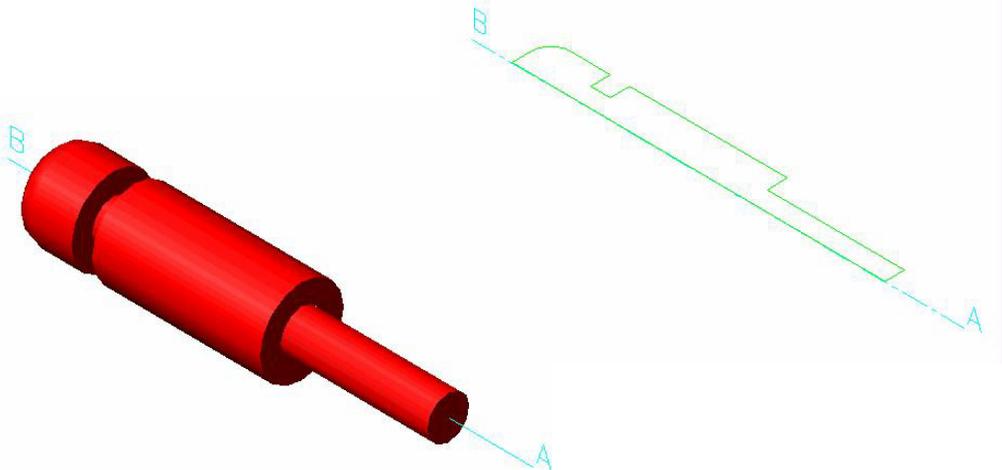
Any closed shape can be revolved by any angle about any axis, including an edge, that does not pass through the part.

Revolve the shape 180 degrees about the axis A. Which way the part rotates depends on which end of AB you pick first, the right hand rule. Change the color of two faces.



Open the drawing S-3-13.dwg

Revolve the shape about axis AB or it's edge for 360 degrees.



Part 4

The Cad-O-Matic It Slices, It Dices

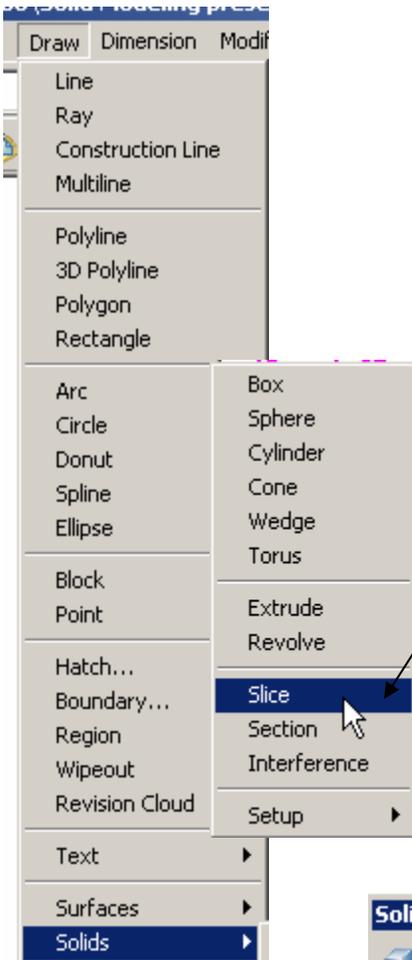
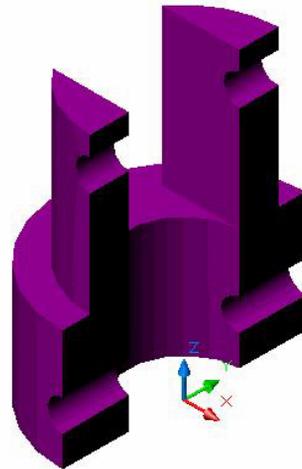
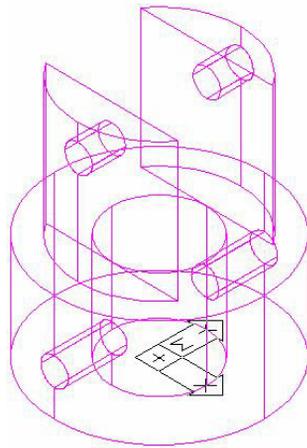
- 1. The Slice Command**
- 2. The Section Command**
- 3. The Separate Command**
- 4. The Imprint Command**
- 5. Move and Offset Faces**
- 6. Copy Faces and Edges**

Step 1 - The Slice Command

Open the drawing *S-4-01.dwg*.

The Slice command will divide a part into two parts by cutting along a plane. You can keep either one or both of the new parts.

We'll slice along the YZ plane through 0,0,0 and keep the left part by picking a point with a negative X.



Command: slice

Select objects: L

1 found

Select objects:

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

<3points>: YZ

Specify a point on the YZ-plane <0,0,0>:

Specify a point on desired side of the plane or [keep Both sides]: -1,0,0



Step 2 - The Section Command

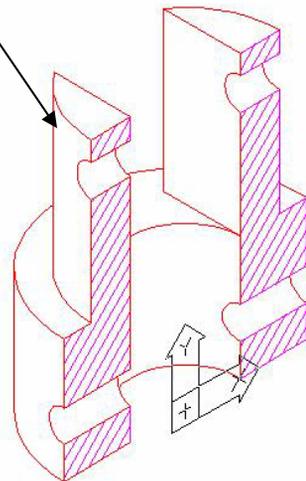
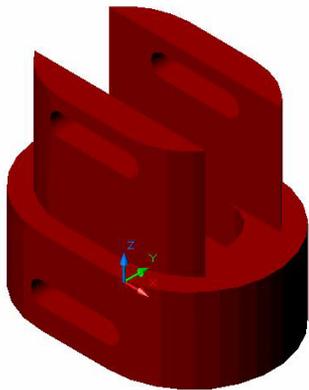
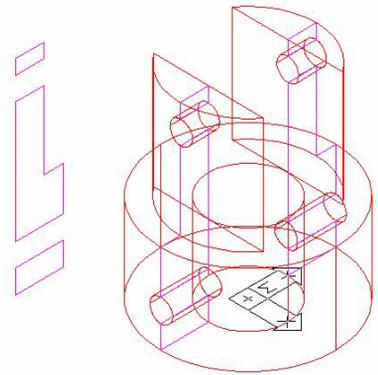
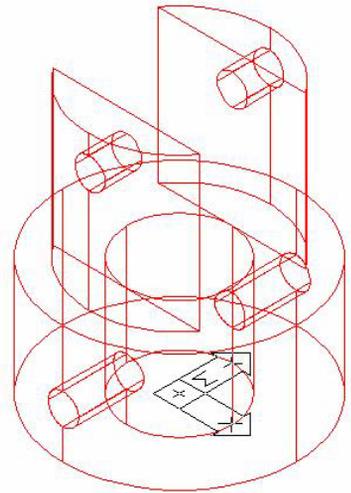
Open the drawing *S-4-02.dwg*.

The Section command will create a 2D section of a part on a plane through the part. The section is one object and is a Region.

We'll section along the YZ plane through 0,0,0 .

The section is copied to show it is one object. The part is sliced, the UCS rotated into the plane of the section and the section crosshatched to demonstrate one application.

In the second example the part is sliced keeping both halves. One half is moved 0.25 units, the section is extruded 0.25 units and all three points unioned.



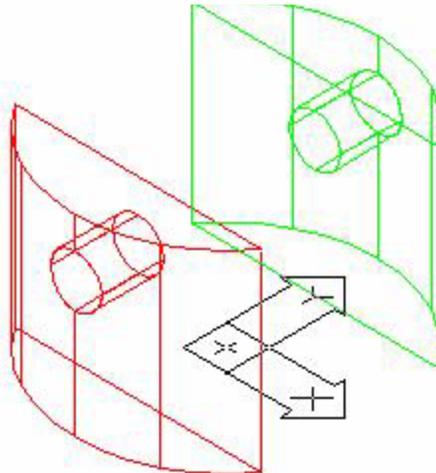
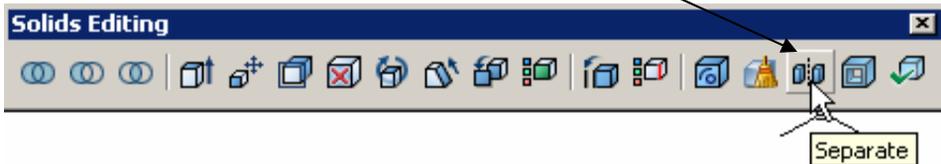
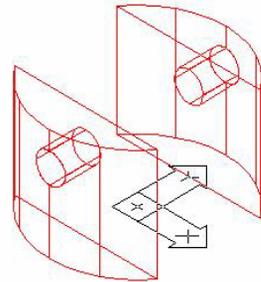
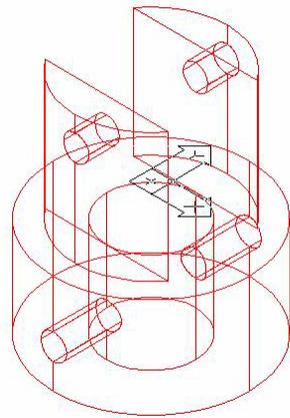
Step 3 - The Separate Command

Open the drawing *S-4-03.dwg*.

We'll slice along the XY plane through 0,0,0 and keep the top part by picking a point with a positive Z.

This looks like it is two parts but it is only ONE!

To change it into two parts use the Separate command. This changes any part that has a separation by space into individual parts. (it does NOT separate a solid into it's primitives.)

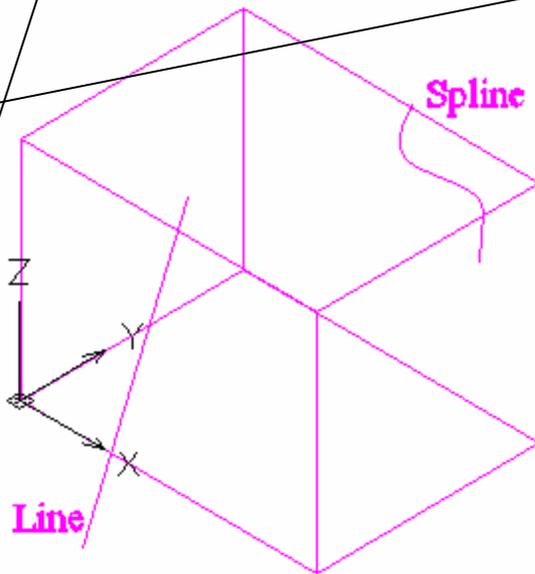


Step 4 - The Imprint Command



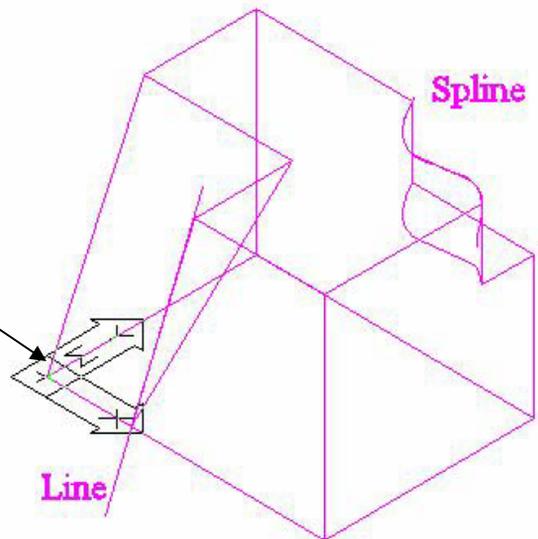
Open the drawing
S-4-04.dwg.

Imprint lets you
“imprint” a Line,
Arc, Spline, or
Pline onto a face
which divides the
face into two
faces.



Imprint the Line on the vertical
face and the Spline on the top
face. Rotate the new vertical
face 25 degrees. Base Point.

Extrude the new face from the
Spline by -20'.

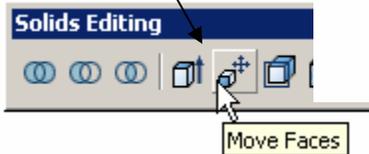
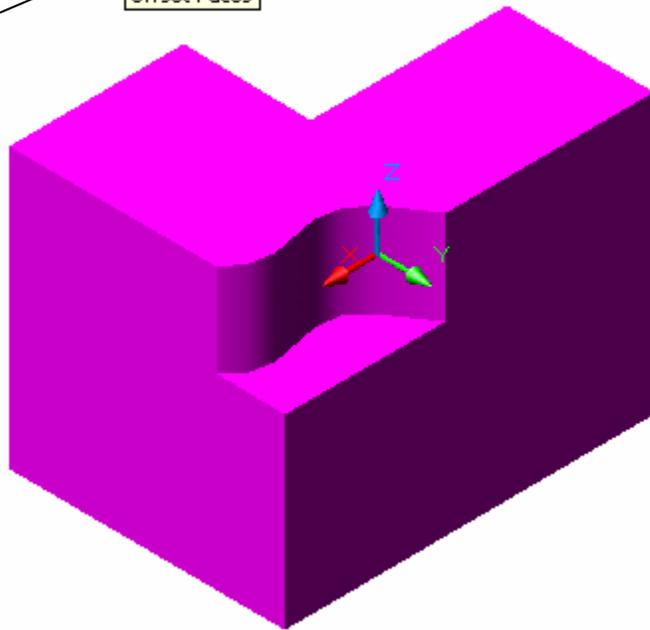


Step 5 - Move and Offset Faces

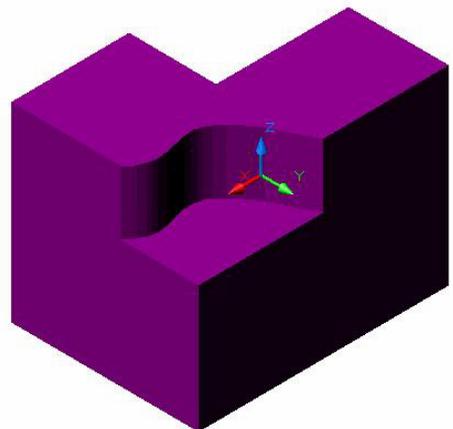
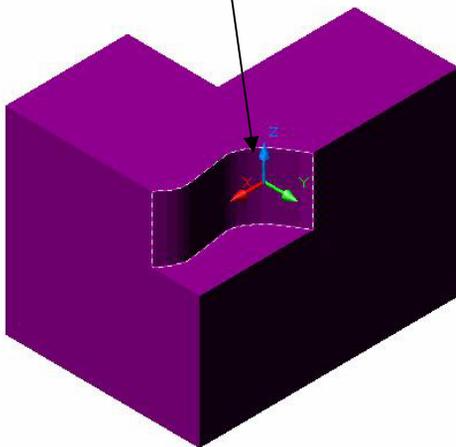


Open the drawing *S-4-05.dwg*.

The Offset and Move Faces commands work very similar to the 2D Move and Offset.

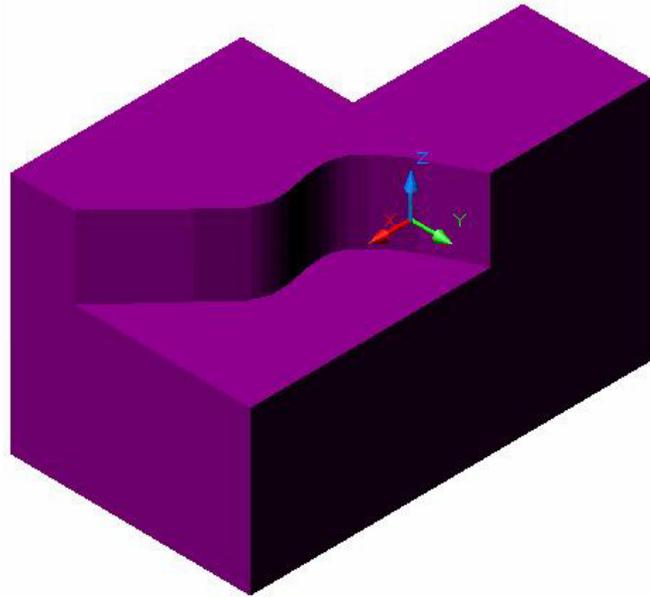
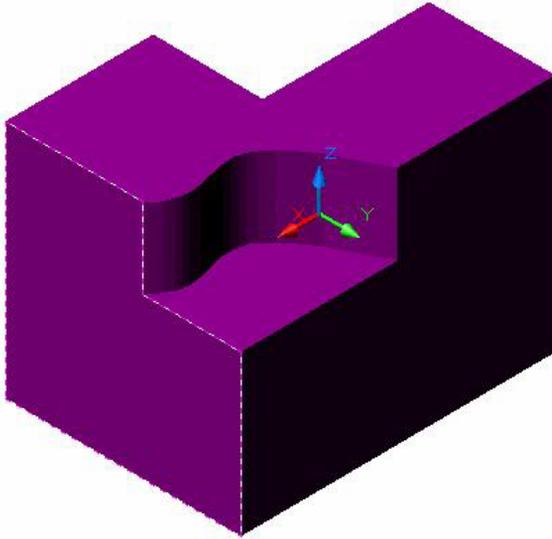


Offset this face a -10' (minus numbers make the part smaller.)

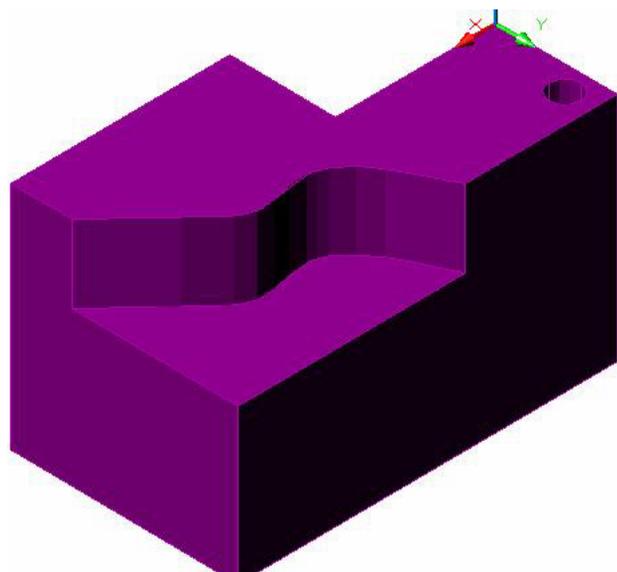
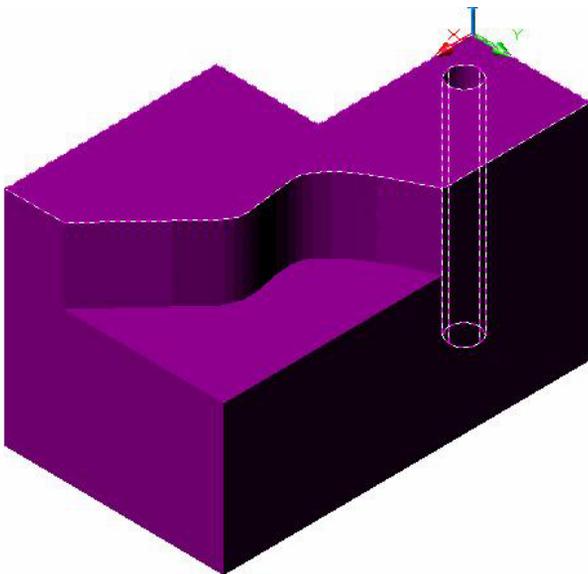


Step 5 - Move and Offset Faces Continued

Move this face a distance of **20',0,0** just like the 2D move command you can input a distance or a base point and a second point.



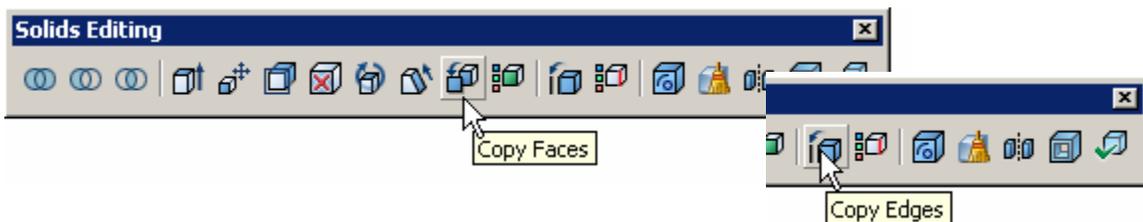
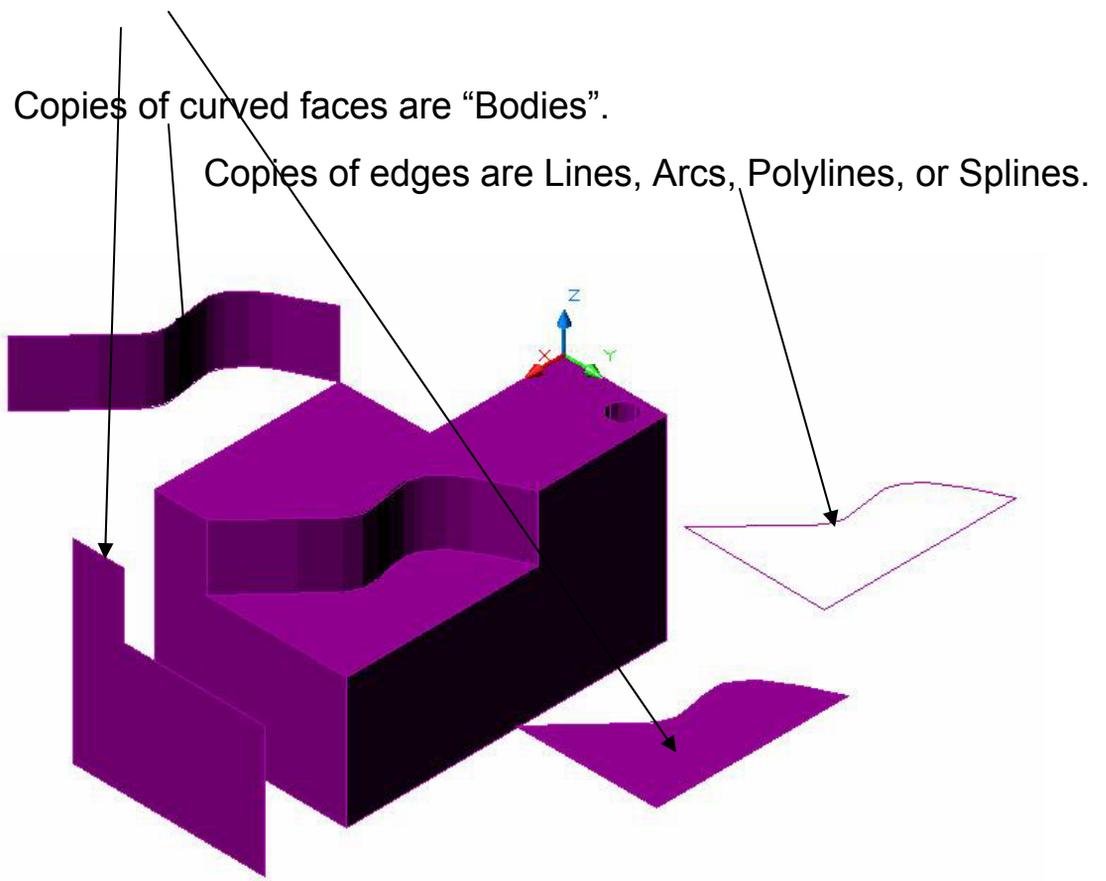
Moving holes is one of the great uses of the Move Faces command.



Step 6 - Copy Faces and Edges

You can copy any faces and any edges.

The copies of flat faces are regions which can be extruded or revolved.



Part 5

Round and Round We Go

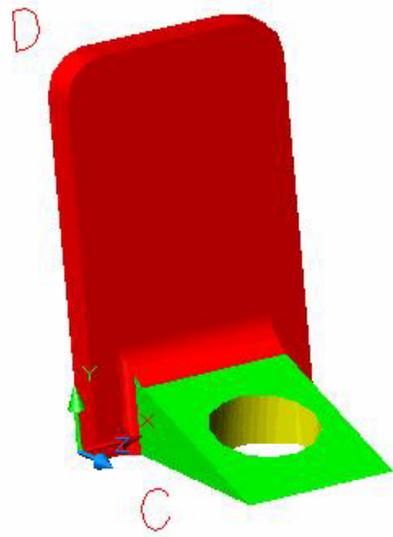
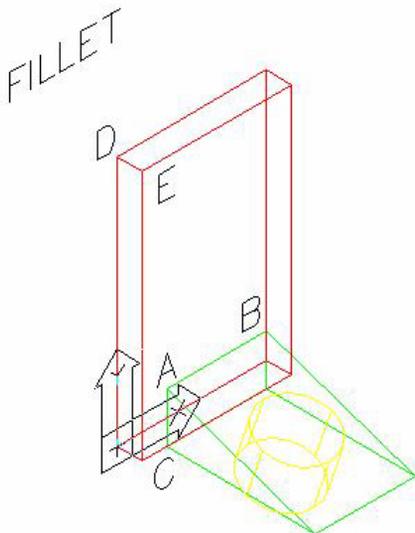
- 1. Creating Fillets**
- 2. When Fillets Fail**
- 3. Creating Chamfers**
- 4. Finding Interference
between Solids**

Step 1 - Creating Fillets

Open the drawing *S-5-01.dwg*.

You can put FILLETS on the inside edges and the outside edges (these may be called “rounds”) of solids. It is NOT necessary to move the UCS into the plane of the fillet before creating. There is no special solid fillet command just use the Fillet command in the Modify pulldown. But the command does have special features for solids. To put a fillet at point E in 2D you would select the two lines at point E in the ABCE plane. In 3D you select the edge or edges of the solid you want filleted, in this case edge DE.

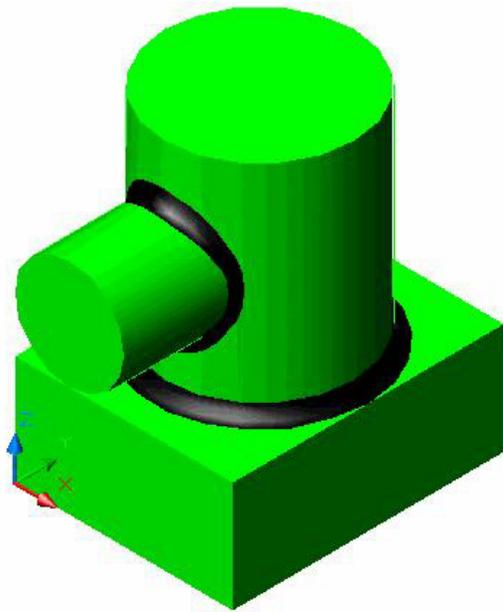
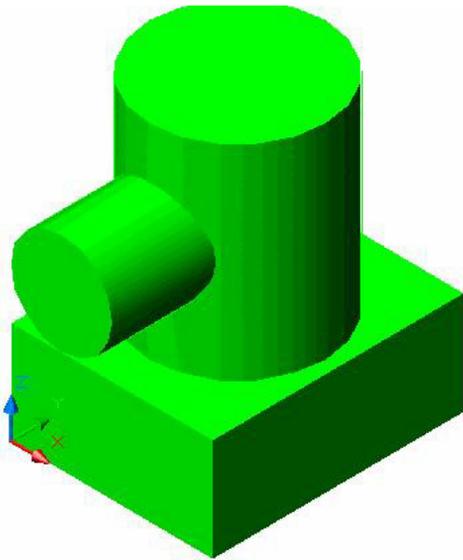
First Union the box and the wedge and Subtract the hole. Then put a 1.0 radius fillet at AB and DE and then a 0.3 radius fillet at AC.



Step 1 - Creating Fillets Continued

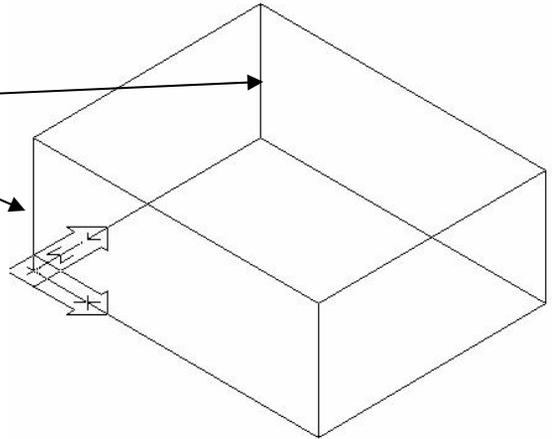
Open the drawing *S-5-02.dwg*.

Put a 0.25 fillet at the intersection of the cylinder and the block and the intersection of the two cylinders. Note the system color.



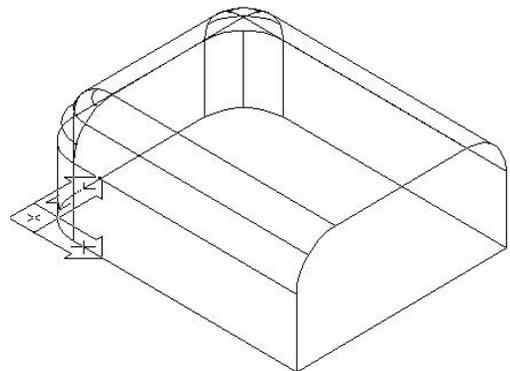
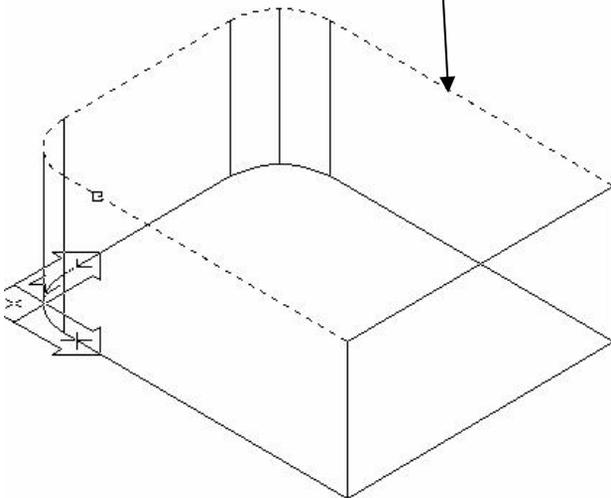
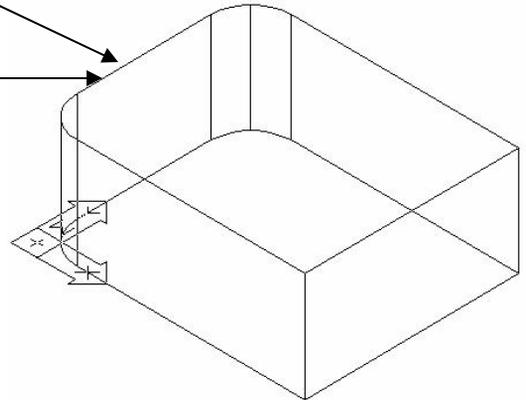
Step 1 - Creating Fillets Continued

Open the drawing *S-5-03.dwg*.
Put a **0.75** fillet on these two edges.



Select Fillet again and pick this edge and accept the radius of 0.75.

Touch "C" for chain and pick the same edge again. Note that the chain selects edges until it gets to a sharp corner where it stops.



Step 2 - When Fillets Fail.

Open the drawing S-5-04.dwg.

Fillet is too large:

Try to Place a **1.00** fillet on this edge.

In AutoCAD 2000i this would work, in 2002 it fails as the largest fillet is 0.25, in 2004 it works.

Open the drawing S-5-04-1.dwg.

The plate is 2.0 units thick. Try to place a 2.25 unit fillet on the top edge using chain..

FILLET

Current settings: Mode = TRIM, Radius = 1.0000

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

Enter fillet radius <1.0000>: 2.25

Select an edge or [Chain/Radius]: c

Select an edge chain or [Edge/Radius]:

Select an edge chain or [Edge/Radius]:

8 edge(s) selected for fillet.

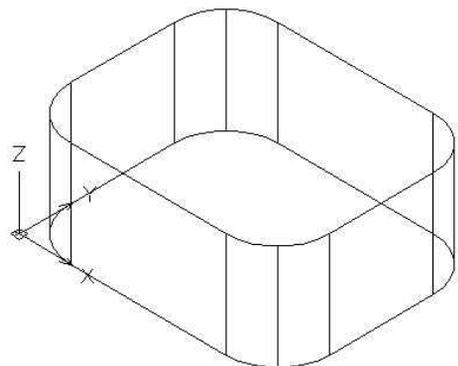
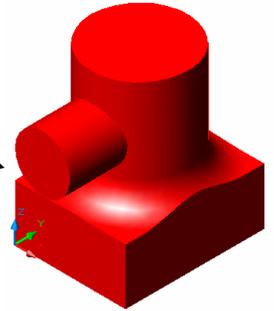
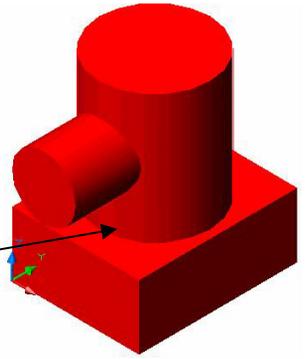
Modeling Operation Error:

Blends too complex to reorder.

Failed to perform blend.

Failure while filleting.

The first thing to try is a smaller fillet!

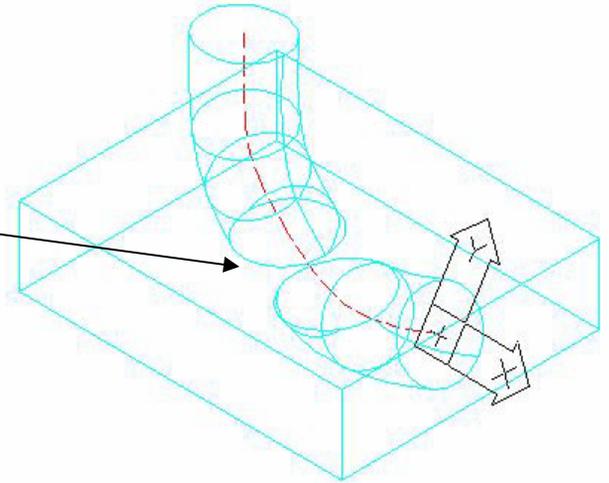


Step 2 - When Fillets Fail.

Open the drawing *S-5-05.dwg*.
Union the box and the tube. Try to
put a **5.0** fillet at the intersection of
the tube and the top.

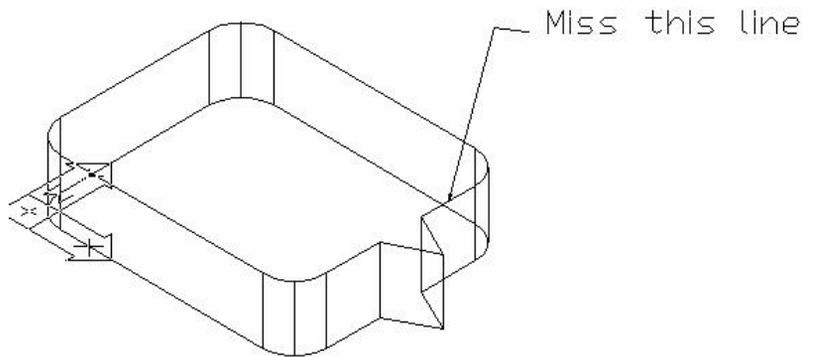
**Fillet is too complex for current
modeler:**

*Whoops! This one works now as
well if you use chain.*



Open the drawing *S-5-05A.dwg*. Put
a 10 unit fillet around the top. Select
the edges one at a time but miss one
of them.

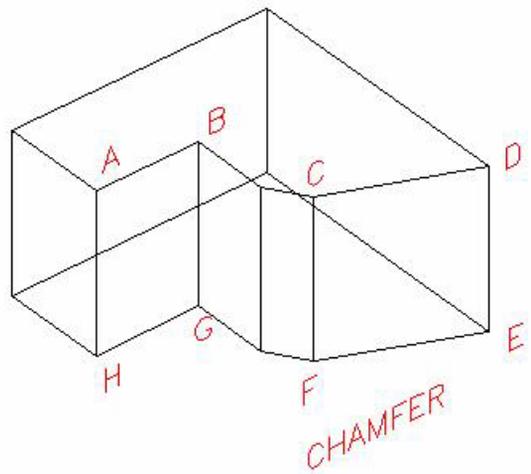
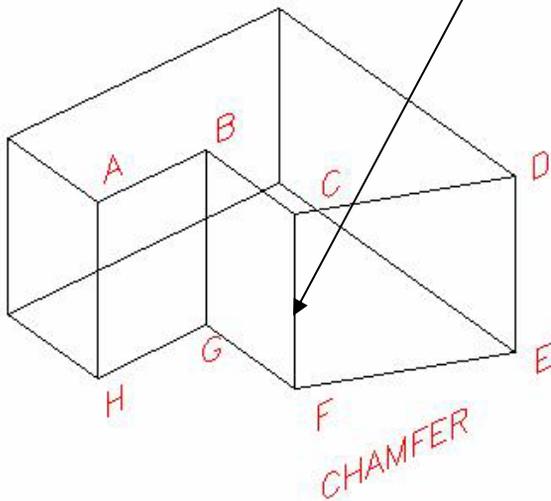
It is better to use chain.



Step 3 – Creating Chamfers

Open the drawing *S-5-06.dwg*.. Solids could be chamfered by creating a solid wedge and subtracting it from the solid. These two steps are done automatically with the CHAMFER command. As in solid fillets, it's not necessary to reorient the UCS. As in 2D, chamfers are specified by two distances.

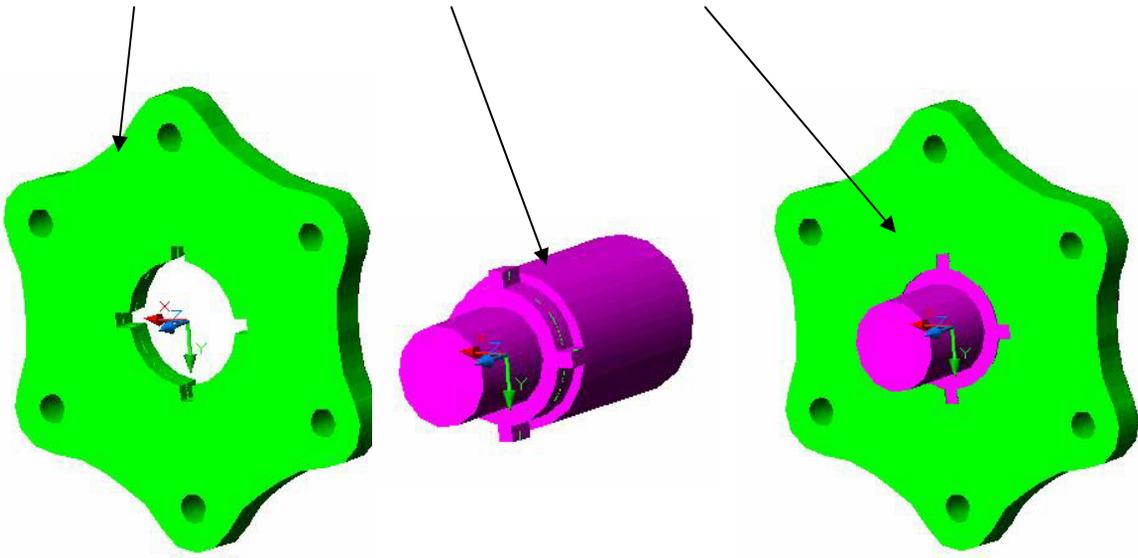
Put a 45 degree chamfer on the edge CF at 0.5" measured perpendicular to the edge. This edge is on the corner of the two surfaces CDEF and BCFG.



Step 4 - The Interference Command

Open the drawing *S-5-07.dwg*.

The star wheel and the shaft are assembled.



Test to see if there is any interference between the two parts. Use the interference command.



Command: **csgwb**

Interfere...

Select objects: 1 found

Select objects:

Select second set of solids:

Select objects: 1 found

Select objects:

(Enter)

(Enter)

Comparing 1 solid against 1 solid.

Step 4 - The Interference Command Continued

Interfering solids (first set): 1

(second set): 1

Interfering pairs : 1

Create interference solids? [Yes/No] <N>: **YES**

Command: **Move**

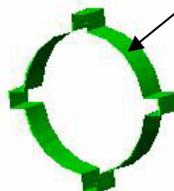
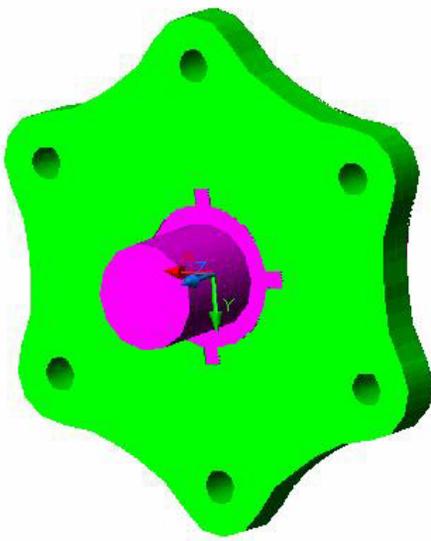
Select objects: **Last**

1 found

Select objects:

Specify base point or displacement: Specify second point of displacement or

<use first point as displacement>:



This tells you there is interference.

This is the interference solid moved for clarity. It can be measured or could be subtracted from the hub.

Part 6

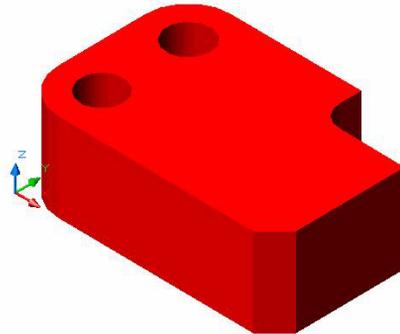
And for my last trick!

- 1. Creating 2D Drawings from the 3D Model with Solview and Soldraw.**
- 2. The Render Command.**
- 3. Applying Materials to Solids for Render.**
- 4. Lights, Shadows, People, and Plants.**

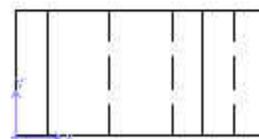
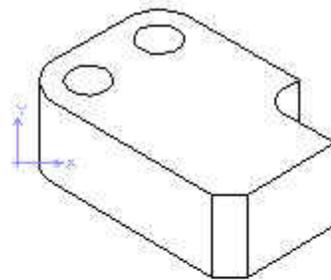
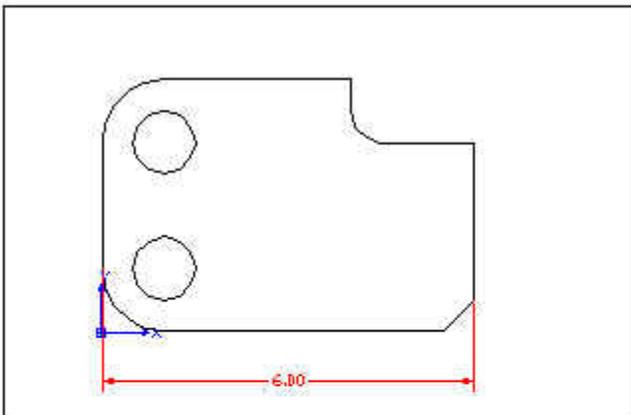
Step 1 - 2D Drawings from the 3D Model

Open the drawing S-6-02.dwg. It is possible to create 2D drawings from the model using the commands Solview and Sldraw.

The command line from this is on the next pages and involves an understanding of Paper Space!



Sldraw Solview Solprof



Step 1 - 2D Drawings from the 3D Model Continued

Command: SOLVIEW

Enter an option [Ucs/Ortho/Auxiliary/Section]: UCS

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

Enter view scale <1.0000>: .5

Specify view center: (pick point)

Specify view center <specify viewport>:

Specify first corner of viewport: (pick point)

Specify opposite corner of viewport: (pick point)

Enter view name: TOP

UCSVIEW = 1 UCS will be saved with view

Enter an option [Ucs/Ortho/Auxiliary/Section]: ORTHO

Specify side of viewport to project: (pick side)

Specify view center: (pick point)

Specify view center <specify viewport>:

Specify first corner of viewport: (pick point)

Specify opposite corner of viewport: (pick point)

Enter view name: FRONT

UCSVIEW = 1 UCS will be saved with view

Enter an option [Ucs/Ortho/Auxiliary/Section]: ORTHO

Specify side of viewport to project: (pick side)

Specify view center: (pick point)

Specify view center <specify viewport>:

Specify first corner of viewport: (pick point)

Specify opposite corner of viewport: (pick point)

Enter view name: SIDE

UCSVIEW = 1 UCS will be saved with view

Enter an option [Ucs/Ortho/Auxiliary/Section]: (Escape)

Step 1 - 2D Drawings from the 3D Model Continued

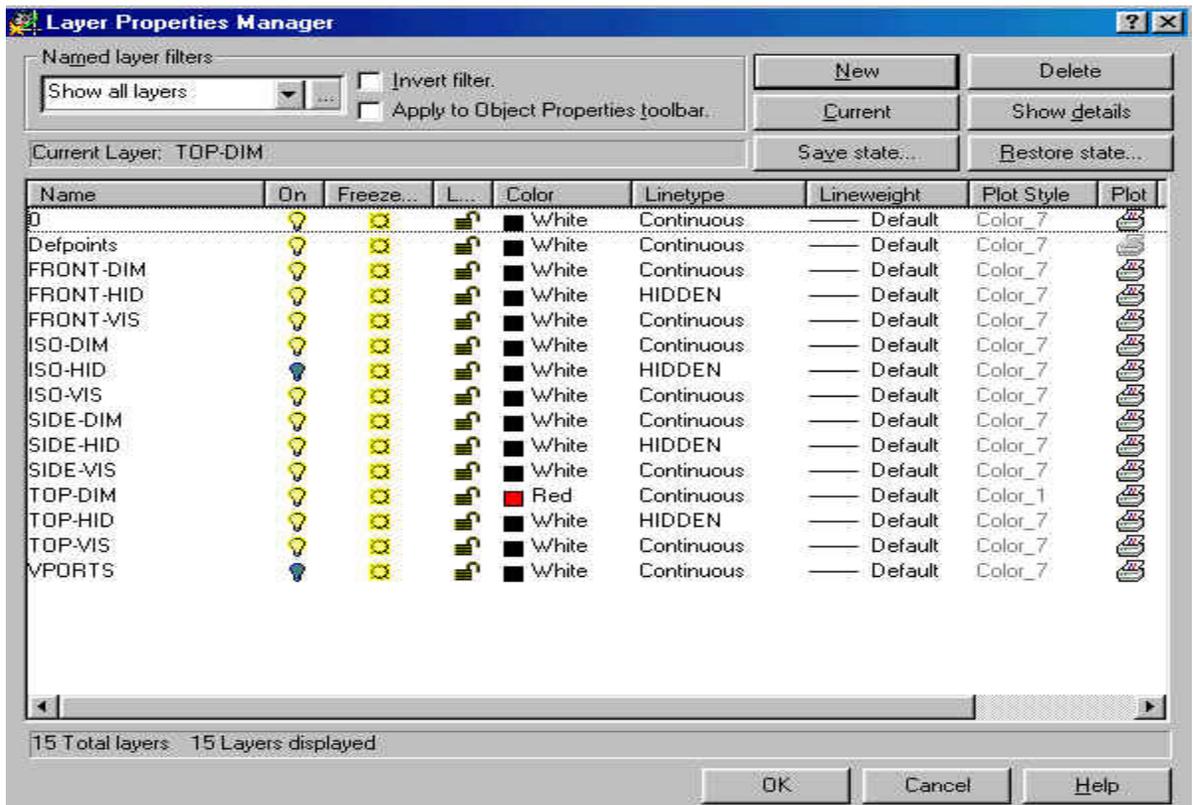
Command: <Switching to: Model>
Regenerating model.
Command: UCS
Current ucs name: *WORLD*
Enter an option
[New/Move/orthoGraphic/Prev/Restore/Save/Del/Apply/?/
World] <World>: View

Command: SOLVIEW
Regenerating layout.
Regenerating model.
Enter an option [Ucs/Ortho/Auxiliary/Section]: UCS
Enter an option [Named/World/?/Current] <Current>:
Enter view scale <1.0000>: .5
Specify view center: (pick point)
Specify view center <specify viewport>:
Specify first corner of viewport: (pick point)
Specify opposite corner of viewport: (pick point)
Enter view name: ISO

UCSVIEW = 1 UCS will be saved with view
Enter an option [Ucs/Ortho/Auxiliary/Section]:

Command: SOLDRAW
Select viewports to draw..
Select objects: 1 found
Select objects: 1 found, 2 total
Select objects: 1 found, 3 total
Select objects: 1 found, 4 total

Step 1 - 2D Drawings from the 3D Model Continued

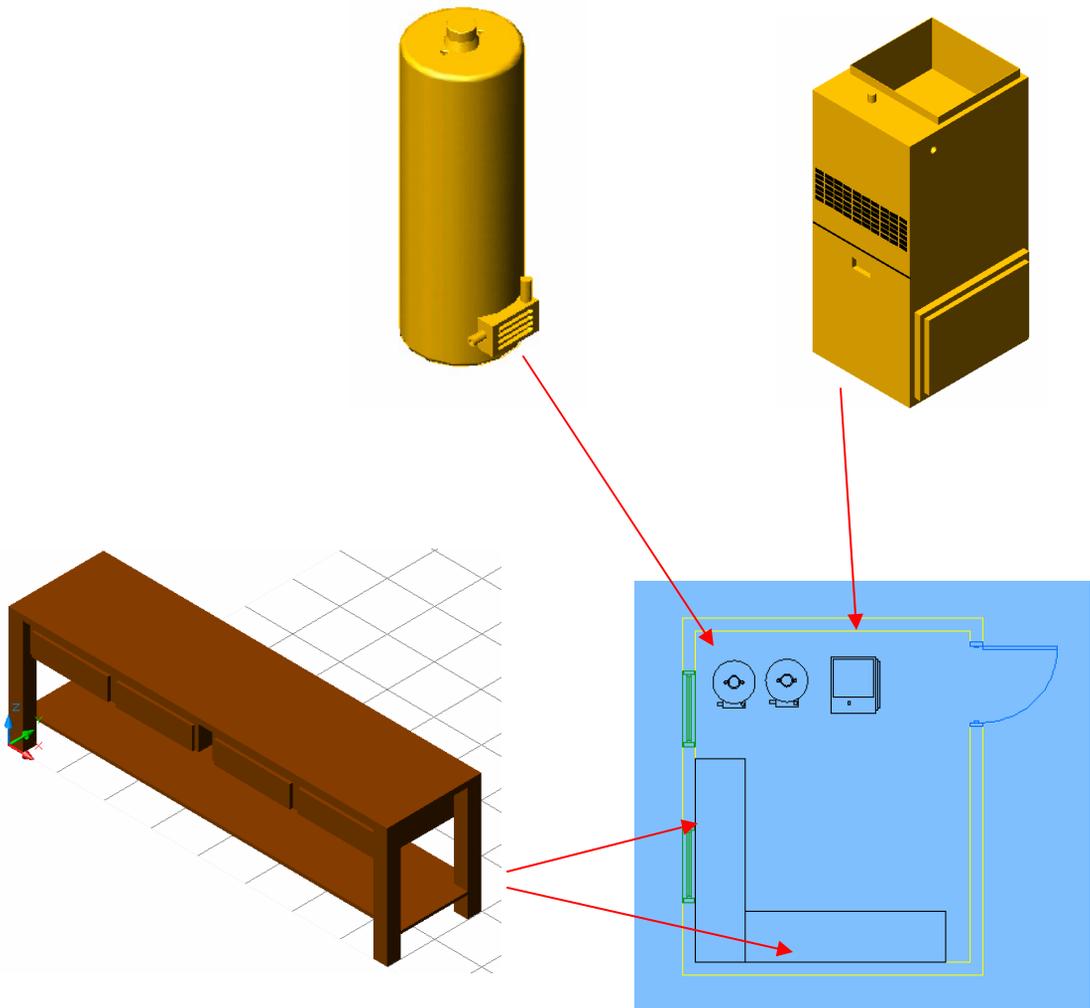


These layers were generated by SolidWorks then some changes were made manually. The Linetype for the layers with names ending in "HID" was changed to Hidden. The layer ISO-HID was turned off hiding the hidden lines in the Iso view. The Vports layer was also turned off hiding the viewport frames.

The color for the layer for the dimensions in the top view, TOP-DIM was changed to Red. This layer is frozen in the other three viewports automatically by SolidWorks.

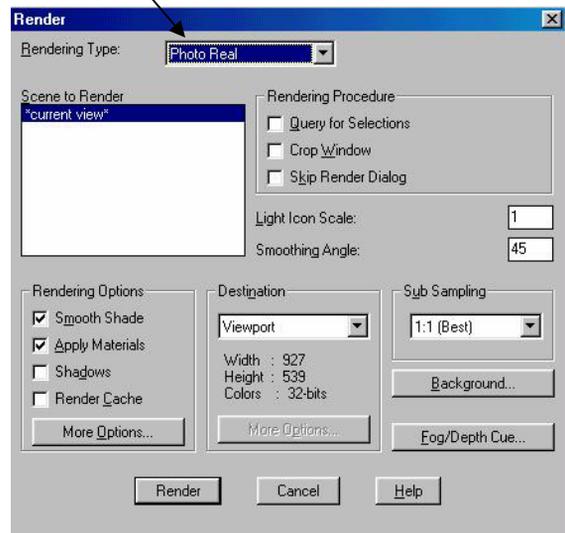
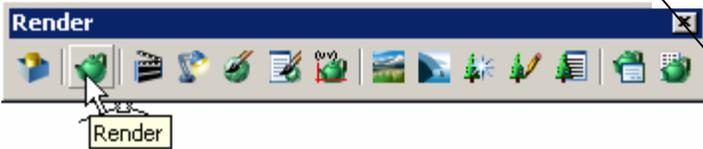
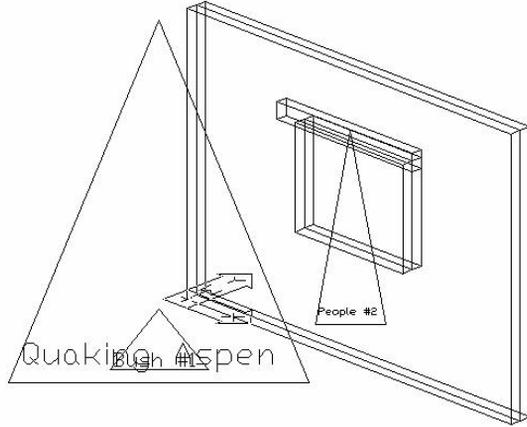
Step 1 – Example in Architectural Desktop

In Architectural Desktop you can use this technique to create “Multi-View Blocks” that are solids with attached 2-D views. When you place the Block in a building and look at the 3D view you see the solid but when you look at a 2D view you only see the outline without hidden lines. The figure shows solid models of a furnace, water heater, and work bench; all multi-view blocks. The plan view of these blocks placed in a room only shows their top views. ADT comes with a large number of blocks such as sinks, appliances but with this technique you can create your own custom blocks.



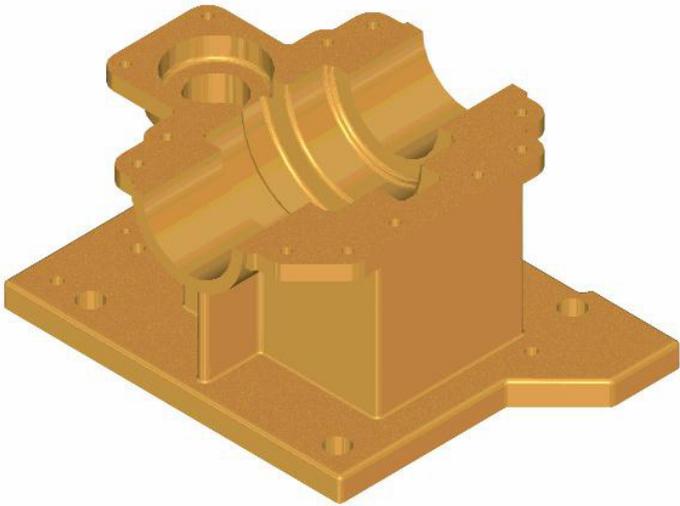
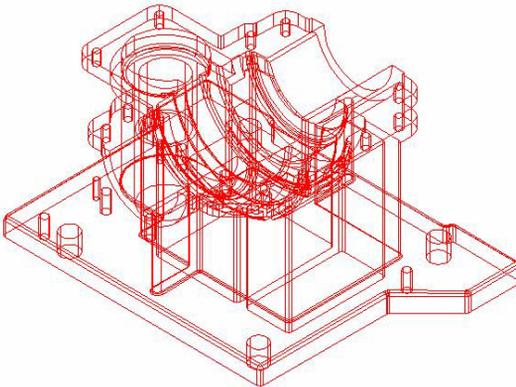
Step 2 – The Render Command

Open the drawing *S-6-03.dwg*. You can assign a material to solids and insert objects, plants and people. When you render the viewport you see the materials and the objects. Photo Real rendering takes a little longer but gives the best results.



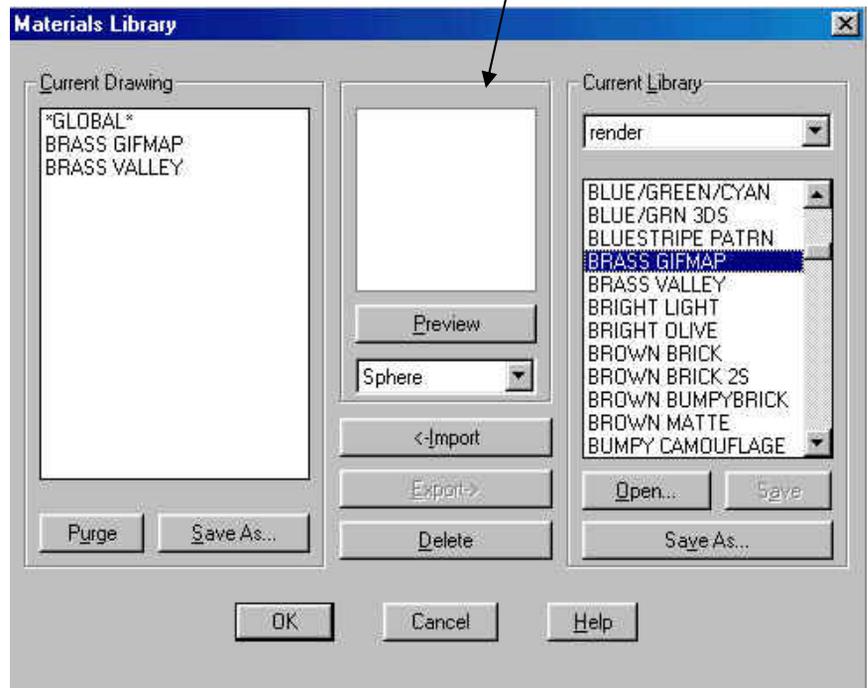
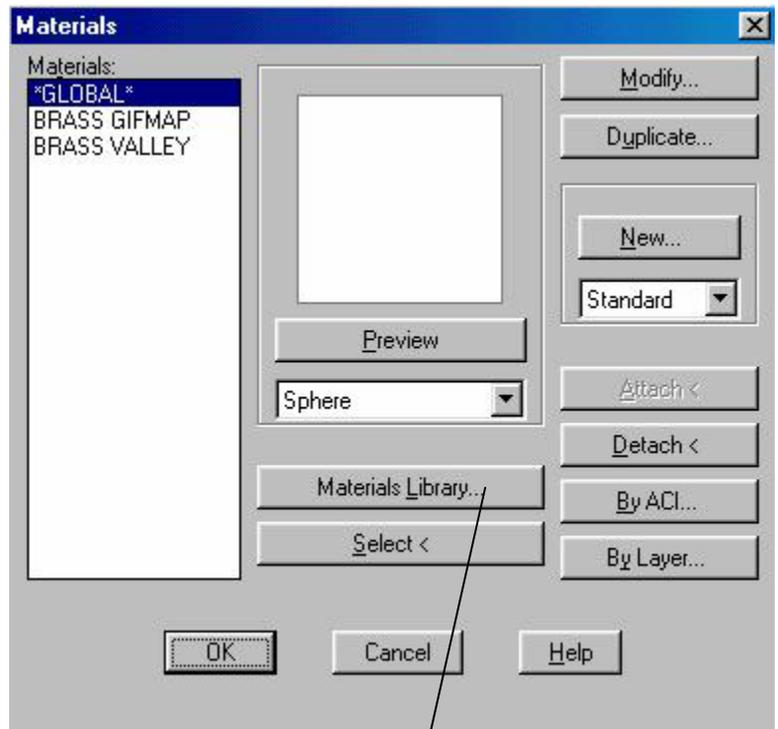
Step 3 – Applying Materials to Solids for Render

Open the drawing *S-6-04.dwg*. This part has the material BRASS GIFMAP attached to it.



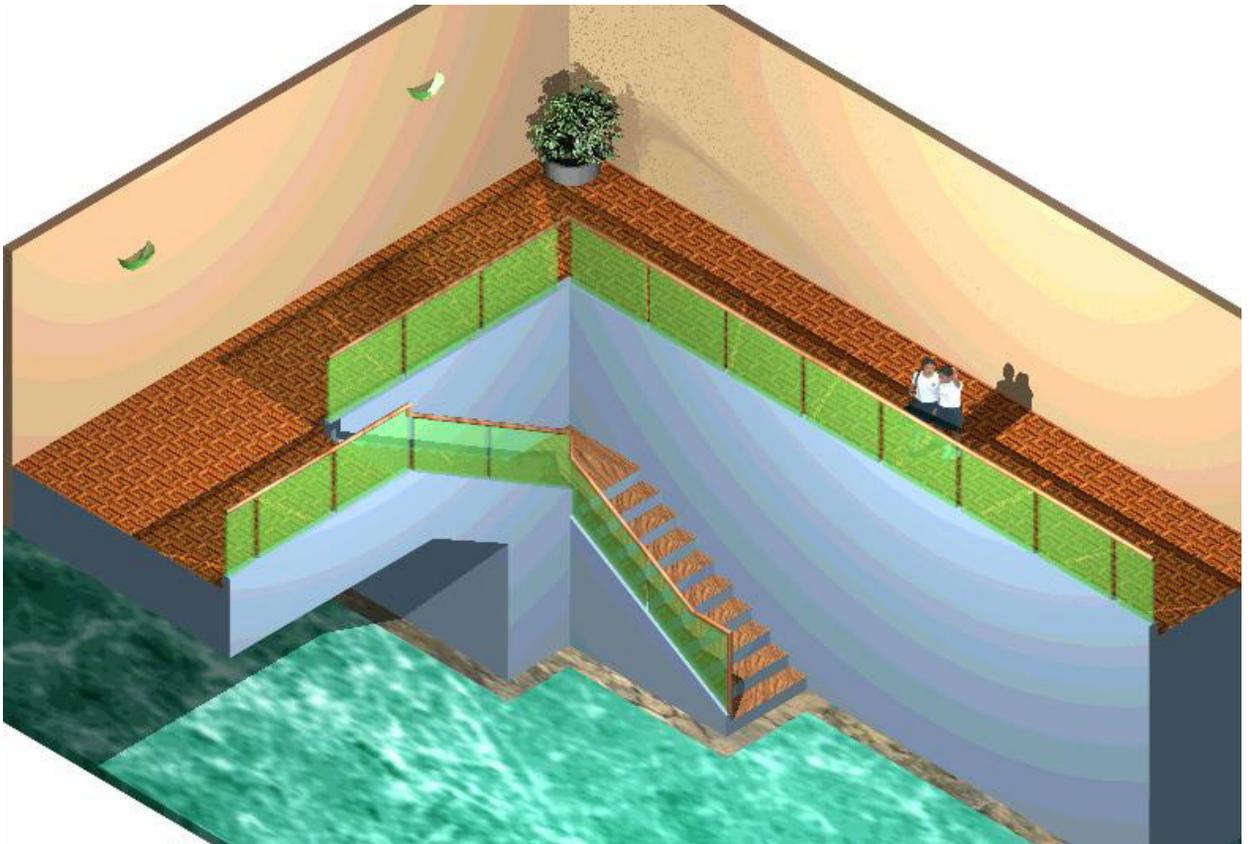
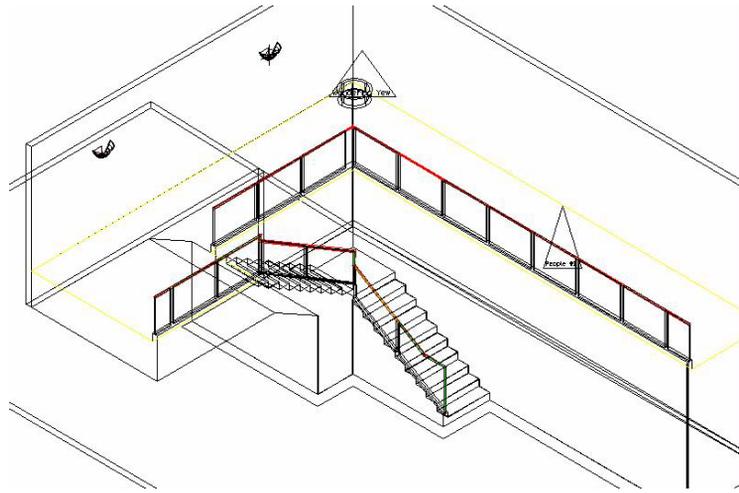
Step 3 – Applying Materials to Solids for Render Continued

The Materials Library dialog box Imports material files. The Materials dialog box Attaches the material to the solid.



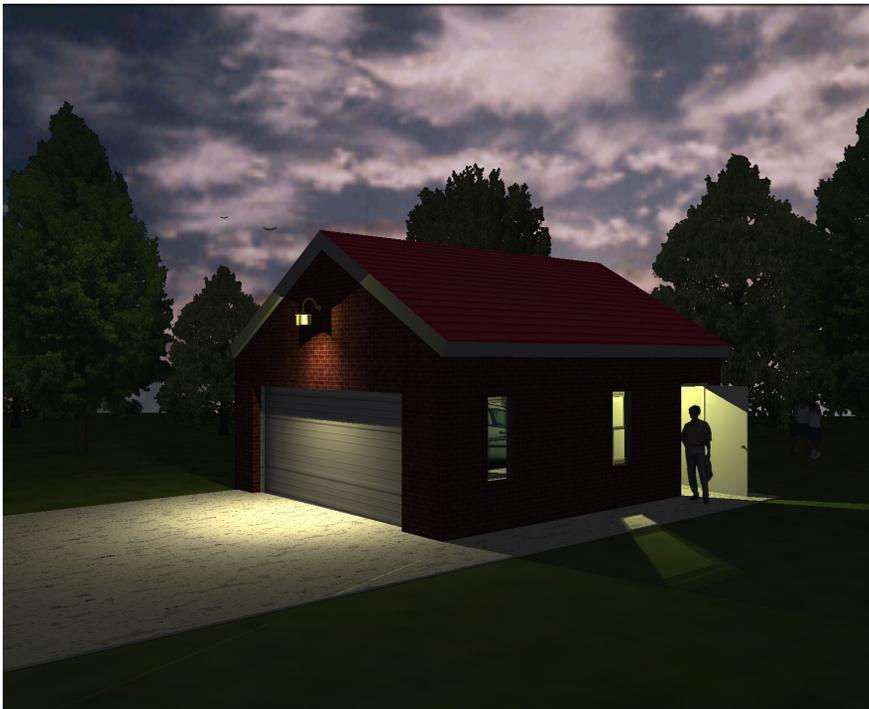
Step 4 – Lights, Shadows, People, and Plants

Open the drawing *S-6-07.dwg*.
This part has the materials, lights with shadows, and people and plants.



Step 4 – Lights, Shadows, People, and Plants Continued

Garage modeled and rendered in Architectural Desktop.
Image of the car was taken from a digital photo.



Step 4 – Lights, Shadows, People, and Plants Continued

This barge was designed in AutoCAD solids and rendered by Jerry McNaughton.

