Engr 210 – Lab #17
Advanced Solids Features

Solid Modeling Display Variables

By default, AutoCAD solids are displayed in wireframe representation. Four variables control the display of solids for wireframe, hidden, and meshed representations.

ISOLINES controls the number of lines that are used to visually define curved surfaces. DISPSILH can be toggled on or off to display silhouette lines for wireframe models. FACETRES controls the density of the mesh apparent with *Hide*. FACETRATIO creates a two-dimensional mesh for cylinders and cones.

Analyzing Solid Models

MASSPROP is a command that automatically computes a variety of mass properties associated with a solid. The command produces a text screen displaying the following results of calculations:

Mass	Displayed numerical value is the same as volume with an assumed density of 1.
Volume	The amount of space occupied by the solid.
Bounding Box	These lengths specify the extreme width, depth, and height of the selected solid.
Centroid	The geometric center of the solid. For a homogeneous solid (uniform density), the centroid is also the center of mass and the center of gravity.
Moments of Inertia	Moments convey how the mass is distributed around the X, Y, and Z axes, and are measures of the solid's resistance to angular acceleration.
Products of Inertia	Measures of the solid's resistance to angular acceleration with respect to two axes at a time.
Radii of Gyration	Useful for computing moments of inertia.
Principal Moments of Inertia	Maximum and minimum moments of inertia.

Creating Solids of Revolution

With REVOLVE a solid can be created by revolving a 2D shape about the X or Y-axis of the current UCS, using a specified angle. The object can also be revolved about a line, polyline, or

specified points. Similar to EXTRUDE, REVOLVE is useful for objects that contain fillets or other details that would other wise be difficult to reproduce in a common profile. The 2D shape to revolve can be a Pline, Polygon, Circle, Ellipse, or a Spline. Only one object at a time can be revolved. 3D objects, objects contained within a block, polylines that have crossing or intersecting segments, or polylines that are not closed cannot be revolved.

Creating 2D Drawings from 3D Models

Several AutoCAD features can be used to create a 2D drawing from a 3D model: *Mvsetup, Solview, Soldraw,* and *Solprof.* These commands are applicable in creating multiview drawings. Depending on the type of 3D model you have (wireframe, or surface, or solid model), a particular command or combination of commands would be applicable.

MVSETUP

Mvsetup is a program that assists you in setting up paper space viewports. It can be used for any type of 3D model (wireframe, surface or solid model). The *Standard Engineering* option can be used to set up a 3D model in viewports, each viewport having a different standard view.

Typical Steps for Using the *Standard Engineering* option of *Mvsetup*

1. Create the 3D-part geometry in model space.
2. Create a layer for viewports and a layer for titleblock (named VPORTS and TITILE, for example). Set the viewports layer current.
3. Activate a *Layout* tab and use the *Page Setup* dialog box to set the *Plot device* and *Paper size*.
4. Invoke *Mvsetup* and use *Options* to set the *Mvsetup* preferences.
5. Use the *Create* option to make the paper space viewports. Select the *Standard Engineering* option from the list.
6. Use *Scale viewports* to set the viewport scale factor (*Zoom XP* factor) for the model geometry displayed in the viewports.
7. The model geometry that appears in one viewport can be aligned with the model in adjacent viewports if necessary using the *Align* option.
8. If desired, use *Title block* to *Insert* one of many AutoCAD-supplied or user-supplied borders or titleblocks.
9. From this point, other AutoCAD commands must be used to create dimensions for the views or to convert some lines to “invisible” lines.

Using SOLVIEW and SOLDRAW to Create Multiview Drawings

The ***Solview*** command creates new layers and new views in paper space viewports for the existing 3D model space geometry. The ***Soldraw*** command creates new 2D objects on the new layers. It projects the model geometry onto a 2D plane with the appropriate continuous or hidden linetype. *Soldraw* can only operate with viewports that have been created with *Solview* (not Mvsetup). Solview and Soldraw operate for solid models only.

Typical Steps for using Solview and Soldraw

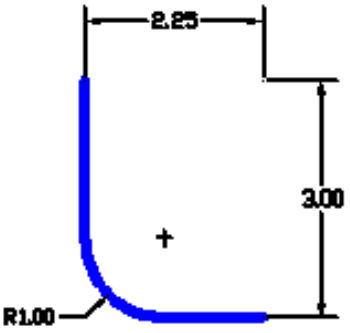
1. Create the part geometry in model space. Set up a UCS parallel to the desired profile (front) plane of the object. Also ensure that the HIDDEN linetype is loaded.
2. Activate a *Layout* tab. Use the *Page Setup* dialog box to set the *Plot device* and *Paper size*.
3. Type *Solview* at the command prompt.
4. Use the *UCS* option to create the profile (front) view. You can select the location and scale for the view.
5. Use the *Ortho* option to create the other principal orthographic views. Usually a top and/or a side view is needed.
6. If a section or auxiliary 2view is desired, use the *Section* or *Auxiliary* option to create the view in the desired location.
7. Type *Soldraw* on the command line. *Soldraw* is used to project the :views: to a 2D plane, thus creating new 2D geometry on the appropriate layers (created by *Solview*).
8. To display the new 2D geometry, use the *Layer Properties Manager* dialog box to control the visibility of the layers. *Freeze* the MODEL layer (or whatever layer you used for the model geometry). Also, *Freeze* the VPORTS layer to prevent the viewport borders from being displayed.
9. If desired, create a new layer for the titleblock.

Laboratory #17

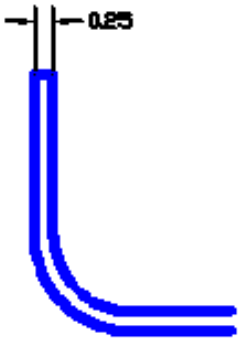
Solids of Revolution, Rendering and 2D Views from 3D Models

Part 1: Solids of Revolution:

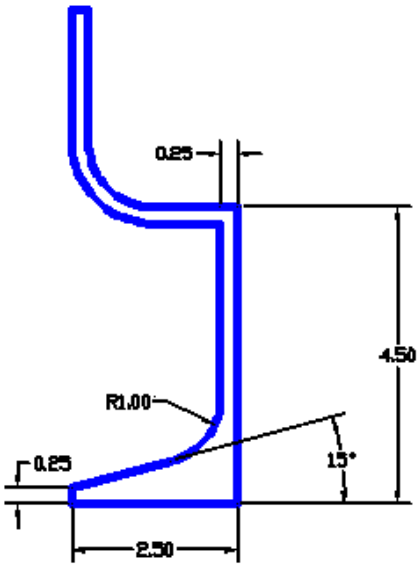
A solid of revolution with an axis along the vertical can be created by first creating a two-dimensional Region, Pline, Polygon, Circle, Ellipse, or a Spline, and then rotating about the Zaxis.



1. Start by changing your ***Workspace*** to ***3D Modeling***.
2. Open a new drawing. In the *Select Template* manager, **left-mouse-click** on the ***triangular*** button to the right of the ***Open*** button, and select ***Open with no Template – Imperial***. Start a new drawing by creating the geometry shown in the top figure. All dimensions are inches.
3. Change the ***3D view*** and the ***UCS*** to ***Front***. (You can do this by selecting tab ***Visualize → UCS, Names UCS → Named UCSs → Orthographic UCSs → front → Set Current***.)
4. Create the two lines and the arc as shown in the figure to the right. You may also create the arc using the ***Fillet*** command.
5. Use the ***Offset*** command icon in the *Modify* toolbar to edit the drawing as shown. The *offset distance* is **0.25**, and the copies of the objects are offset to the left and below the original entities.

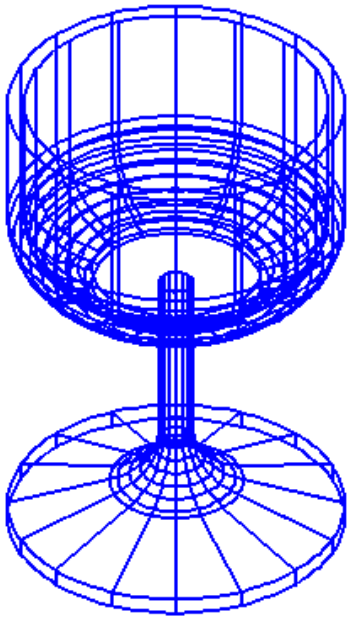


6. Finish creating the two-dimensional drawing in the frontal plane with the dimensions shown.
7. Select the ***Home → Draw → Region*** command icon.
8. At the prompt "*Select objects:*" select all the objects in the drawing by creating a selection window enclosing the entire drawing
9. ***Right-mouse-click*** to accept selection. This converts all the objects into a single region.
10. Switch to the ***SE Isometric*** view.



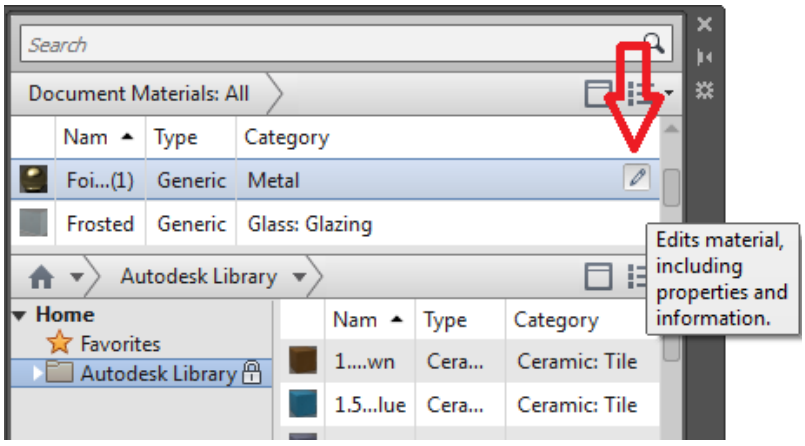
Before revolving the region about its axis, we will increase the density of isolines to be used when displaying the model as a wireframe.

11. Select the ***Revolve*** command icon in the *Modeling* toolbar.
12. At the prompt "*Select objects:*" ***left-mouse-click*** the region.
13. ***Right-mouse-click*** to accept selection.
14. At the prompt "*Specify start point for axis of revolution ordefine axis by [Object/X (axis)/Y (axis)]:*" select one end of the axis of revolution (the vertical line of length 4.50).
15. At the prompt "*Specify endpoint of axis:*" select the other end of the axis of revolution.
16. At the prompt "*Specify angle of revolution <360>:*"
17. Enter: **360 [ENTER]**
18. Save this drawing as **Lab17a.dwg**.



Part 2: Using RENDER to Shade Solids

1. Select the ***Visualize*** tab.
2. Select the ***Materials Browser*** in the *Materials* toolbar. The Materials dialog box appears.
3. The *Autodesk Materials Library* shows a list of available materials.
4. To apply any of the materials to your model, drag the materials swatch into the model.
5. To display the model with the applied material, click on ***Render***. Experiment with different materials.
6. To edit the settings of a material, click on the ***Edit*** icon. (See image to the right.
7. Explore the effects of the materials settings by changing the values of the various parameters. Use the ***Render*** command to view the effects of the changes you make.



8. Adjust the parameters "*Shininess, Opacity, Refraction Index, Translucency*, and *Self-Illumination*" as desired.

Next, explore the changes you can make by adjusting rendering parameters.

Before closing the drawing, we will investigate its *Mass Properties*.

9. In the pull-down menus, select:
   [Tools] => [Inquiry] => [Region/Mass Properties].
10. At the command prompt "*Select objects:*" select the model.
11. ***Right-mouse-click*** to accept the selection.
12. The *AutoCAD Text Window* appears and shows a list of properties related to the model. Note that the values of centroidal coordinates, and moments of inertial are with respect to your.
13. Press **ENTER** twice to close the *AutoCAD Text Window*.

Assignment for Part 1:

1. Create a solid of revolution of your own. Be creative.
2. Attach appropriate materials, and lighting to your model.
3. Save a rendered version of your solid of revolution as **Lab17b.dwg**.

Part 2: Creating multiview drawings from solid models.

1. Starting from the 3D solid model created in Lab #16, and previously saved as **Lab16a.dwg**, create a 2D layout drawing showing the front, right, and top views using Solview and Soldraw. Save this layout drawing as **Lab17c.dwg**.

2. Starting from the 3D solid model created in Lab #16, and previously saved as **Lab16b.dwg**, create a 2D layout drawing showing the top and front views, and the auxiliary view showing the true size of the inclined plane. Save this layout drawing as **Lab17d.dwg**.