**Auxiliary Views**

**Commands: solview, soldraw**

When drawing objects with incline planes or sides that are not defined on standard orthographic planes, it is necessary to produce another view, an auxiliary view that will show the true size and shape of the features. When inclined surfaces are presented in a standard view, they are said to be foreshortened. Measurements on the plane of interest are shorter than the true size. Creating a view that is perpendicular to the incline develops an auxiliary view.

The solview command has an option for auxiliary views. The plane is identified, and projected into a new view from the view that defines the angle of the incline. The solview command will produce a full auxiliary view.

To demonstrate this command and the organization of the subsequent layout, a solid model is built. For this purpose the following wedge is presented.

To begin the construction of the wedge, start a new drawing using a template drawing that has layers created, and at least the Hidden linetype loaded. Most of the definition for this wedge is in the front view, so, start the drawing there.
Draw the profile for the part in the front view using the polyline command. Be sure to close the object. If the profile is disjoint, use the polyline edit command (pedit) to ‘join’ all the segments together.

This profile, as a single object, is extruded along the negative z-axis to a depth of –3.5”, as shown in this isometric view. To create the hole feature, a solid (primitive) cylinder will be placed on the incline, and then subtracted from the solid wedge.

A drawing plane must be defined to work on this incline. A good method for this definition is the UCS command’s three-point option. The new origin for this plane is the lower left corner of the incline, the lower right corner defines the direction for a positive x-axis, and the upper left corner defines a direction for the positive y-axis.
Draw a construction line dividing the incline plane and use the solid cylinder command to produce an R=1” cylinder two inches into (-z-axis) the plane.

The cylinder passes through the bottom of the wedge. Use [Modify-Solids Editing-Subtract] to produce the hole.
Coloring and shading the wedge will help in determining if it is drawn correctly.

**The Layout**
Set up a blank layout for a local printer or plotter.
The solview command will be used to create the views for this layout. The solview command [Draw-Solids-Setup-View] will be used to create the top view based on the world coordinate system. The other orthographic views will be generated, orthographically, from the top, and then the front view.

First select the ucs option, and world. The scale used in the example is .25. Position the view near the top left quadrant of the paper. Draw a viewport around the view. Name this view, top.

Using the ‘ortho’ option of the solview command, select the bottom of the top viewport and drag to position the front view. When positioned, use two points to create a viewport boundary around this view and name it ‘front’.

Using the ortho option again, select the right side of the front viewport and draw to position the right view. The command will align the views and maintain the same scale for each. Draw a boundary around the right view and name it ‘right’. The command created the views and created three layers for each view. One each for visible line, hidden lines, and dimensions.

The dialog below shows the prompts and responses for creating the top view.

The auxiliary view is created using the auxiliary option of the solview command. To identify the incline for the command, two points must be selected from the front view. Then a side is selected for the view to be generated.
(The incline is defined by 1\textsuperscript{st} point and 2\textsuperscript{nd} point. The side of projection is defined with a third pick)

Position the auxiliary view between the top and right views. Draw a viewport around the aux view and give it the name ‘aux’.

The result is a standard three-view layout with a fourth view defining and displaying the true shape and size of the incline plane.

The soldraw command will produce two dimensional geometry, both solid lines visible in a view, and hidden lines showing features not visible in a view, based on the solids orientation in each view. With the soldraw command active, select each viewport then press enter.
The objects in these viewports are two-dimensional. The model is still complete, on the layer of creation.

These views can be dimensioned. For each view there is a dimension layer named appropriately. For example, set the dimension layer for top, called ‘top-dim’, activate floating model space for the top view (double click inside the viewport) and dimension the view. The dimensions will only appear in that view. Continue by setting the dimension layer for each view, and dimension as shown.

The layer properties manager dialog shows that thirteen layers were created by the solview command. Three layers each for the top, front, right and auxiliary views and a layer for the viewport boundary called ‘vports’.
Freezing the ‘vports’ layer removes the boundaries but leaves the views.

A fifth view for an isometric view is shown. This isometric view was created with solview by using a named ucs. An ucs option called view is set while displaying an isometric view in model space. Save the ucs (with the ucs command, save option) That named ucs is entered while running the solview command.