

Download the Drawing template file: "pav_dwg_template.DRWDOT". Save it to this folder:
C:\Program Data\SolidWorks\SolidWorks 2010\templates.

Before you start, you may want to set the scroll wheel zoom so that it works as it does in every other modeling program in the universe. Click the "Options" icon (Top, last icon). In the list, click on "View" and check the 'Reverse mouse wheel zoom direction' box. Sheesh.

Zoom: Scroll wheel (or shift and hold scroll wheel)

3d Rotate: press and hold the scroll wheel

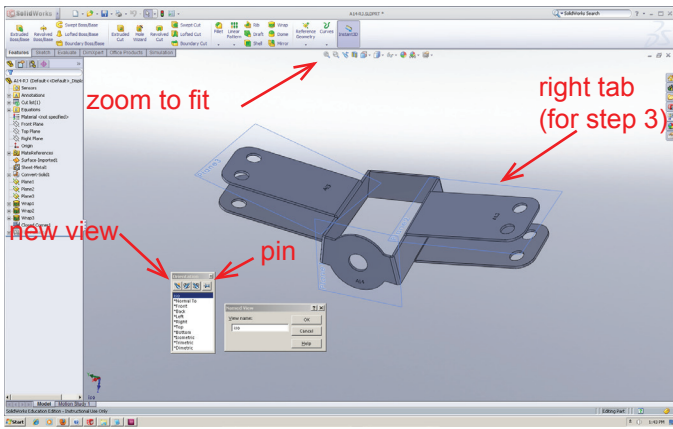
Pan: Ctrl + press scroll wheel

2d rotate: Alt + press scroll wheel

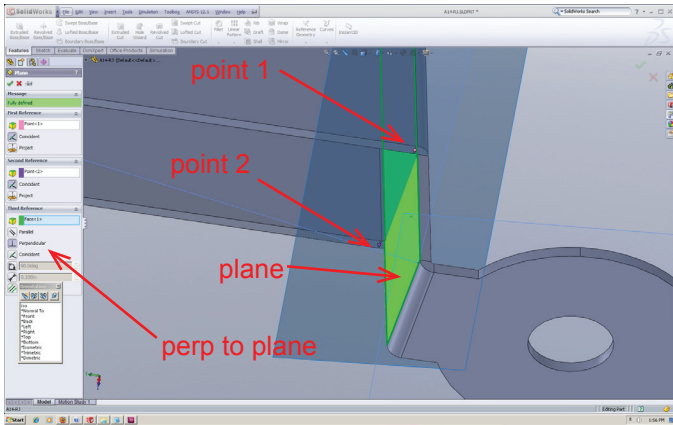
Also, in the Options menu > "System Options" tab > Drawings > uncheck "automatically scale new drawing views".

PART 1: SET UP VIEWS

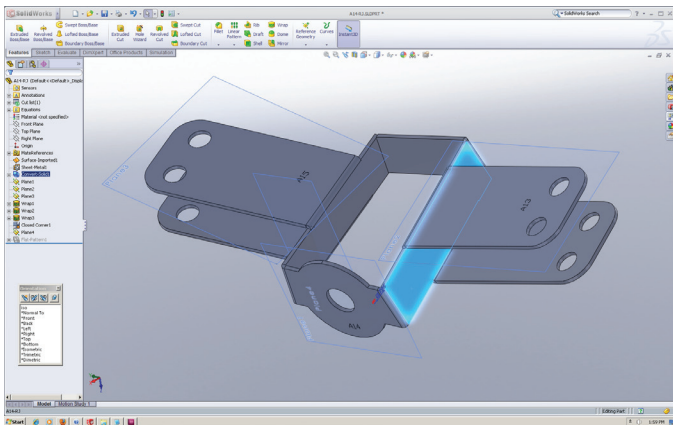
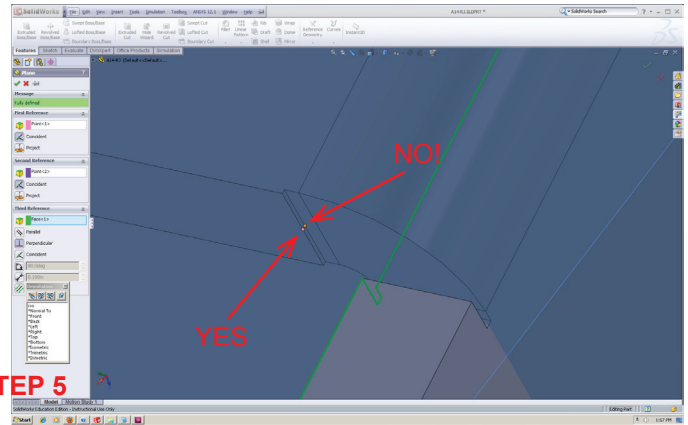
1. Open part in solidworks.
2. Set to a nice iso view that shows as many faces as possible and all three sets of labels. Try to keep all views as 'landscape' as possible so that they fit on the drawing later. Hit Spacebar, and click the first icon "New View". Save view as "**iso**". Click the pin icon to keep the views window open.
3. Click on the edge of the plane sitting on the right tab. Once it's highlighted, right-click and click on the "Normal To" icon. 2d-rotate to make it horizontal and zoom to fit. Click the "New View" icon again, and save as "**face1**".
4. Repeat step 3 for the left tab, saving as "**face2**".
5. Now you must define a new reference plane that will let you see one set of tabs in 'elevation'. Near the top right, click the "Features" tab, then the "Reference Geometry" icon in the menu along the top, and select Plane. You are going to give it two points and a plane to be perpendicular to. The points are the midpoints shown in the screenshots. You'll need to zoom in nice and close to make sure you're getting the outer points, not the ones inside the notch. Then select the side of the ring between the tabs, and make sure that the "Perpendicular" button is on in the menu at the left. Your new plane should look like the screenshot. Accept this plane by clicking the green checkmark at the top right. Set view normal to this plane, rotate and zoom to fit. If you can only see the edges of the tabs, you picked the points right and you can save this view as "**tab_elev**".
6. Select the front face (the one with the bolt hole). Set view normal to this, and save as "**face3**".
7. Going counterclockwise around the ring, select outer faces and save views as "**face4**" (for the right), "**face5**" (for the back), and "**face6**" (for the left).
8. Select the top edge of the back face, and set view normal to this small plane. You should notice that both the front and back faces are seen edge-on. Save this view as "**top_view**".



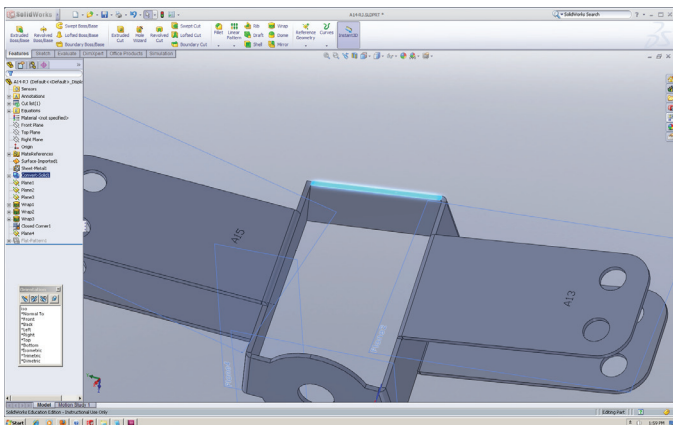
STEP 2



STEP 5



STEP 7 - plane to create "face4"



STEP 8 - plane to create "top_view"

PART 2: CREATE DRAWING

1. From the top menu, click the arrow next to the “New” icon, select “**Make drawing from part/assembly**”. Click the “advanced” option at the bottom of the new window. You should see the pavilion template file in the list, because you put it there earlier. Double-click it. You should see 4 of your views on sheet 1 (change by clicking tabs at the bottom left).
2. Find the “view palette” icon at the right of the screen. Drag and drop the **flat pattern** view into the space on sheet 1. If the angle annotations are all jumbled, click on the drawing so that it is highlighted in orange, then at the left, under “Import Option” check “Import Annotation” and then “Design Annotations”. Now the labels should be aligned to the fold lines.
3. Go to sheet 2. In the menu at the left, select all the drawing names, right-click and select insert model. Ensure that the right model is selected (by default this is the one you have open) then click the green check. All the views should be updated.

PART 3: DIMENSIONING

1. Use the “Note” command again to **label the iso** views appropriate to each page. This means faces 1 & 2 on page 1, faces 3 to 6 on page 2.
2. From the “Annotation” menu, use “**Smart Dimension**” to dimension the parts as per the example drawing. If you are dimensioning a bolt hole, you need to sketch (from the sketch menu) a circle over the polyline (“Perimeter circle” works best). To locate the center of the hole, you need to sketch lines over the straight parts of the tab, and dimension to those.
3. Please don’t forget to put curly brackets around the angles in the top view!!
4. Save drawing, then save as PDF.

