Making Models Using SolidWorks and the Dimension 3D Printer

1. **Preparing the Model**
   1.1. Think about how you might **modify your model to decrease volume**. Sometimes it makes sense to split the model into separate pieces and glue them together afterwards. For example, a hollow sphere will build much faster and take much less support material if you build it as two halves and orient each one cavity up (like a bowl). Building it as one piece will require the center to be filled with support material. Hollows with no exterior outlet make it impossible to remove the support material.
2. Decide **how many separate parts you want to make**. Generally, for communication purposes it's best to create an STL of the assembly. You can also slice it to make a section view that shows internal detail. If the parts need to move to communicate the concept then you need separate parts. However, you can create a single STL file for multiple parts by simply creating an assembly with none of the pieces actually connected to each other. Just constrain each part so that it is in the position that you want during the build; mostly likely lying flat on the top plane. You could also put some of the parts in the assembly in a fully constrained position and some "floating" on the top plane which will be physically assembled after being built. Make sure that you leave at least .010" clearance between parts that are supposed to move relative to each other.
3. **Create the STL file (not available on student version – you'll have to do this in the ME cluster.)**
   3.1. In SolidWorks - File > Save As | choose STL from "Save as type" dropdown. Click on Options. A popup will appear.
   3.2. Set the Options
      3.2.1. Select the file type: the Dimension 3D Printer will read either Binary or ASCII. Binary files take less space.
      3.2.2. Units can be set to either inches or mm.
      3.2.3. The resolution is important if you want to avoid the "faceted look" you need to set the deviation to less than 0.001" and the Angle tolerance to less than 5 degrees. As you move the sliders to smaller values it makes the size of the STL file larger since more triangles are created but it will take the same amount of time to build. You can see the number of triangles and size the file just below the sliders.
      3.2.4. Check the boxes to show the STL file and to preview.
      3.2.5. If you are doing an assembly as a single STL file you need to check the box next to "save all components of an assembly".
      3.2.6. Note that if you wish to 3D print a section view as shown in Figure 1 you will have to create a SW feature to slice away the material as the standard section view tool will not work.

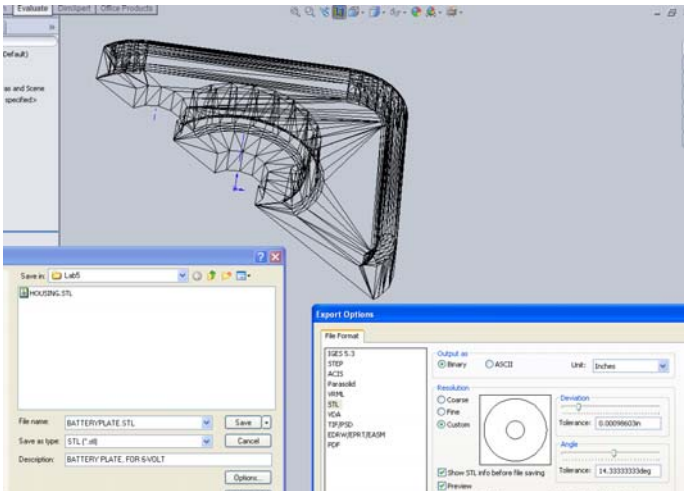

Figure 1 Shows STL file of sectioned Battery Cap from Lab 4

4. **Submit the Assignment** – Create a ZIP file to submit to using the Final Project assignment on BB. The ZIP file should contain all of the SolidWorks files used to make the model, the STL file and a README.TXT file which will contain the name and email of the contact person on the team, the volume of the model and support material, the X and Y extents of the model in the pack, the build time and the scale of the model. If all goes well the STL file can be printed directly but if there are problems they can be rectified as long as I have this information. Name the ZIP as NN_RP.ZIP where NN is your team number. The models will be built by clustering as many part models as will fit on a single plate. It is a rather slow process and might take 24 hours to complete a single build.
5. Model is Built – It's great fun to watch it being built, one layer at a time.
6. Post processing – support structure is dissolved in the hot water bath; this takes a few more hours.
7. Add color – A "Sharpie" can be used to color different parts of your model.

You can get more information about the Dimension RP machine at www.dimensionprinting.com

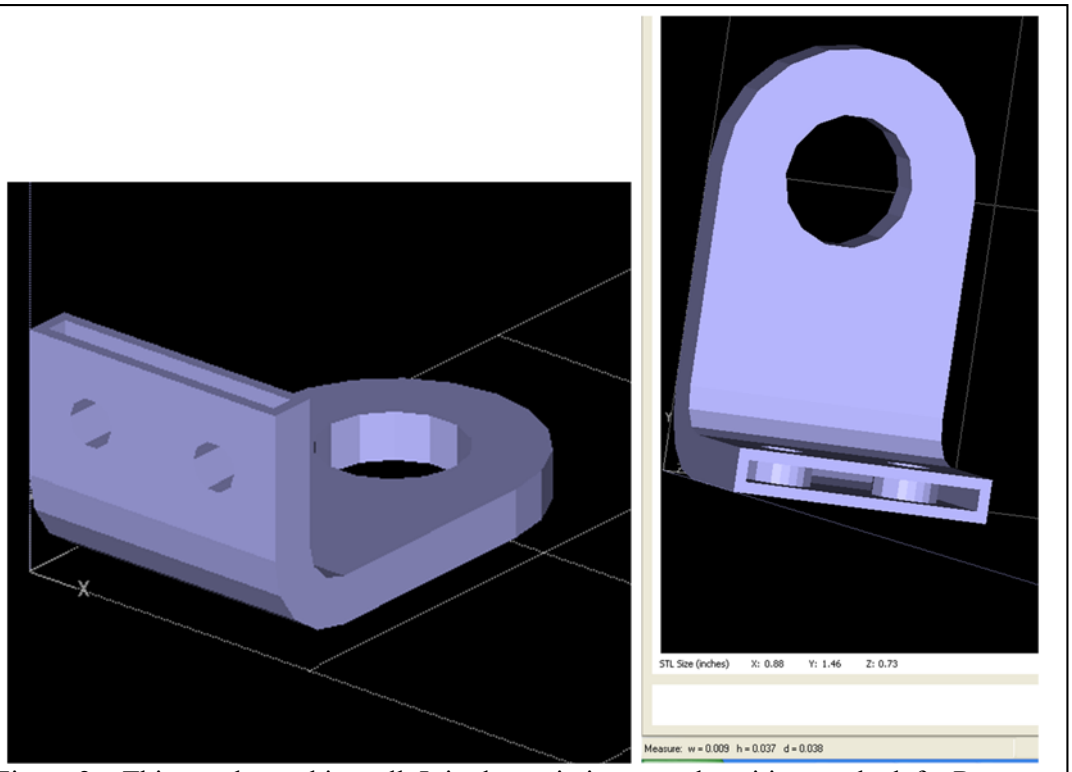

Figure 2 – This part has a thin wall. It is shown in its normal position on the left. Rotate (LMB and drag) into a position where the thin wall is close to perpendicular to the view, then RMB and choose "measure". Drag across the wall. The "d" dimension shown at the very bottom will be pretty close to the actual thickness of the wall. It should be at least 0.04", preferably 0.06". Thinner sections will end up with voids in them. If necessary, go back to your original SolidWorks model and make the original dimensions proportionately larger so that the size will be big enough after the scaling is applied.