Student: **** **** (Student Documentation Package)

Class: CDI-60 SolidWorks (Beginning)

Teacher: Kenneth Louie

Fall Quarter, 2010

De Anza College, Cupertino, CA

Course Objectives:

A. Learn the basic menu structure and graphical user interface used within SolidWorks.
B. Achieve competency in creation of basic sketches.
C. Apply dimensions and constraints to sketches.
D. Gain competency in extruding, sweeping and revolving basic 2-D geometry.
E. Create features such as holes, shells, fillets, and chamfers.
F. Be able to create and modify solid parts.
G. Be able to create and modify solid assemblies.
H. Create basic engineering orthographic drawings using models

What is SolidWorks?

SolidWorks is a mechanical design automation software package used to build parts assemblies and drawings in a Microsoft Windows graphical user interface.

What is Parametric Modeling?

Feature based parametric solid modeler driven by its measurements.

When a dimension is changed in a part, an assembly or drawing, all linked files are updated.

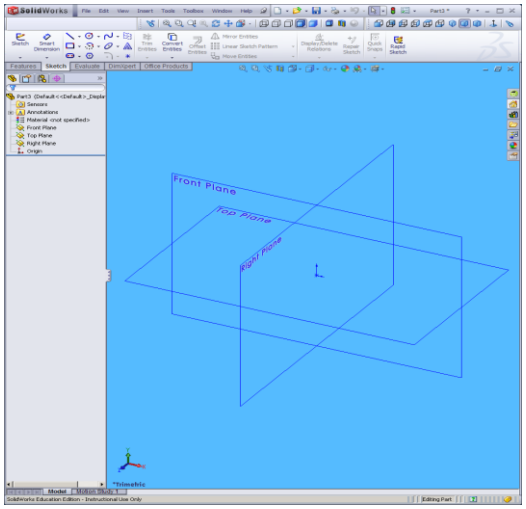


Figure 1.

I became familiar with the functions of the graphical user interface shown above.

Sketches and Parts

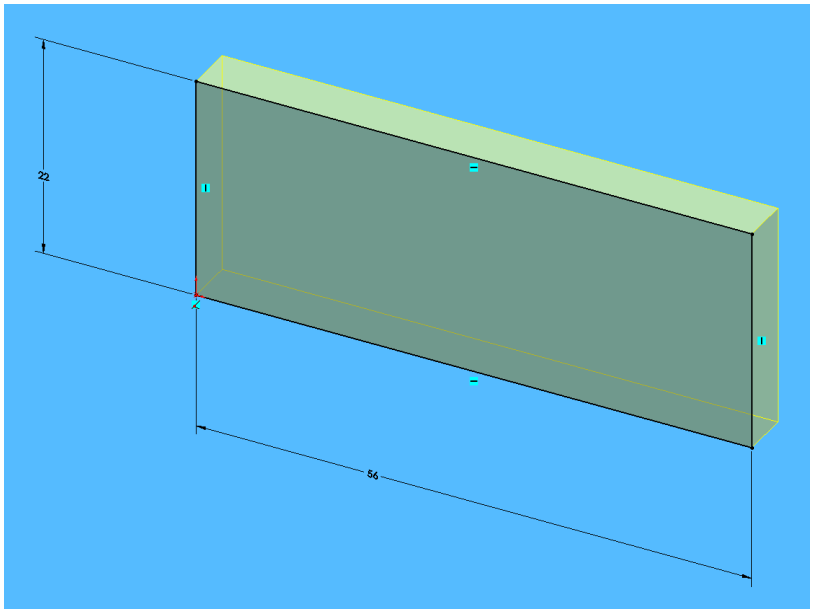


Figure 2.
I learned how to create a sketch and add dimensions so that it is fully defined.

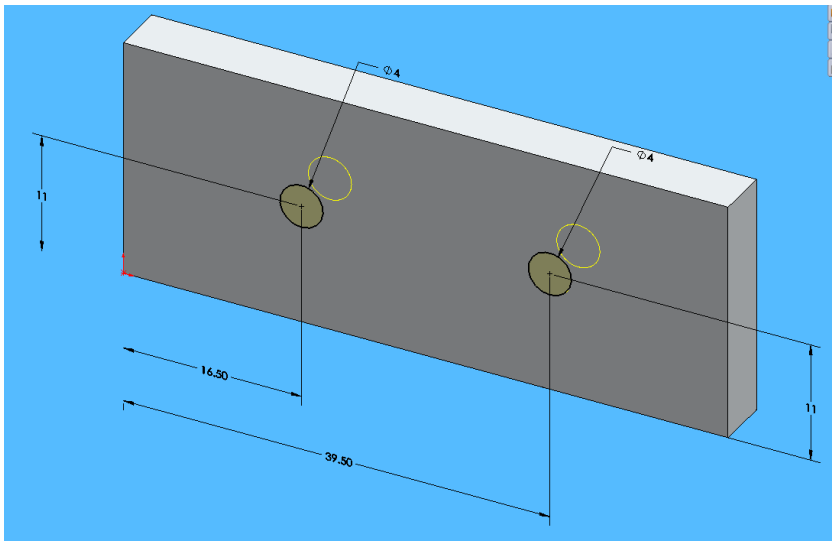


Figure 3.
I learned how to sketch holes, dimension them, and extrude cut thru a solid part.

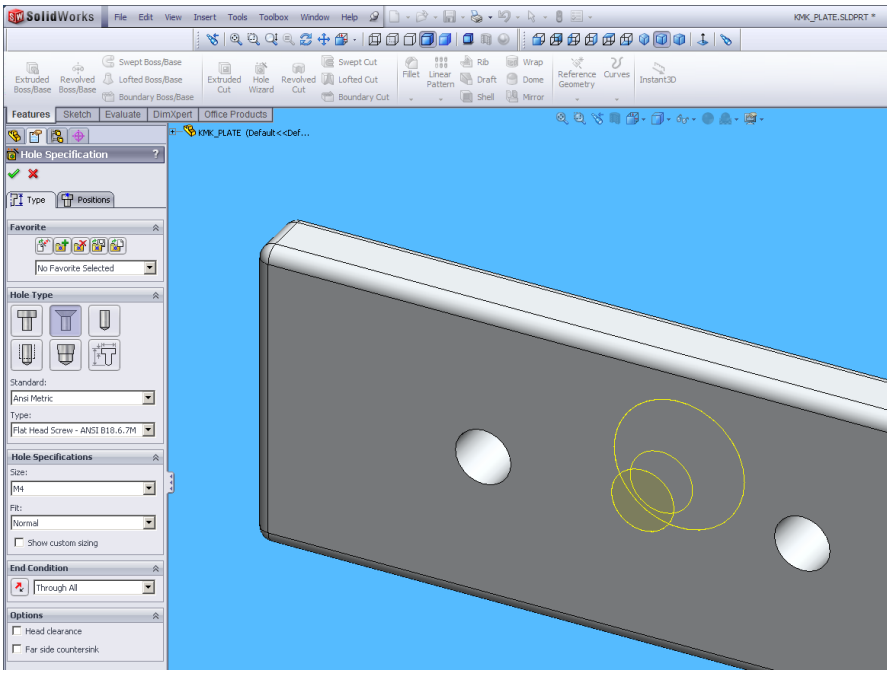


Figure 4.
Using the Hole Wizard, I learned how to insert a counter-sink hole.

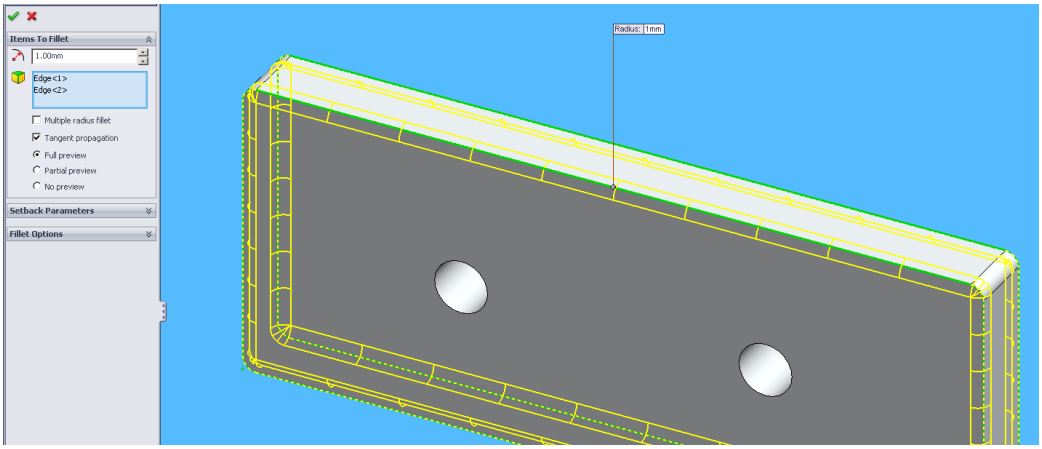


Figure 5.
Using the Fillet Command I applied a 1mm radius fillet to the selected edges.

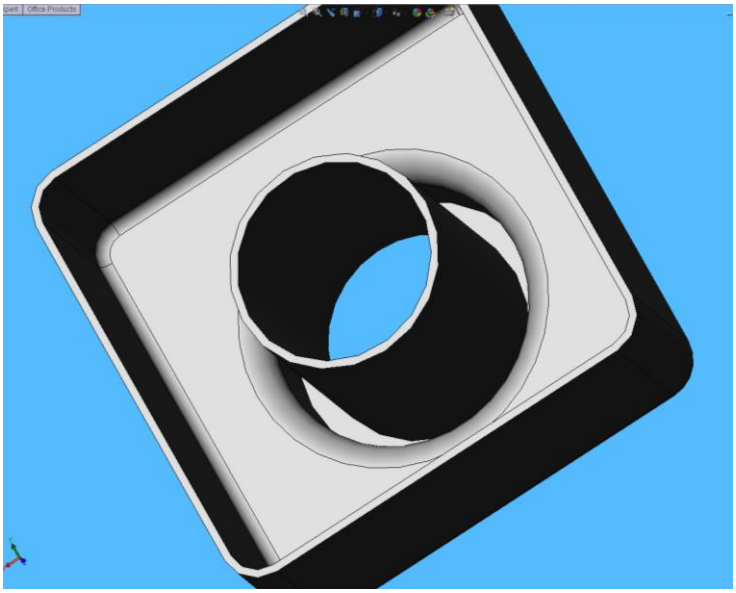


Figure 6.
I applied the Shell Command, to hollow out this part, from the SolidWorks Tutorials.

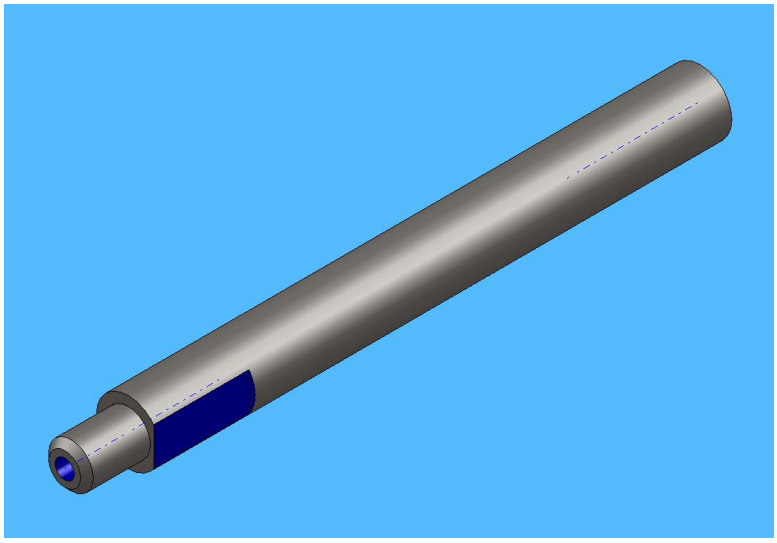


Figure 7.
I learned how to Convert Entities for use in a sketch plane to save time creating a shape for an extrude cut to create the flat surface on the side of this shaft.
I also learned how to apply color, in this case BLUE, to a surface of a part.

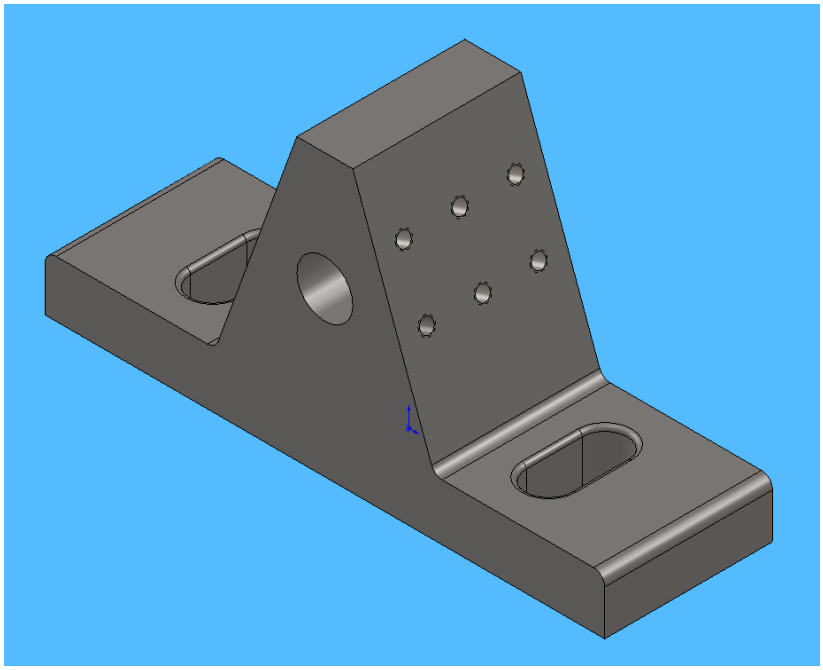
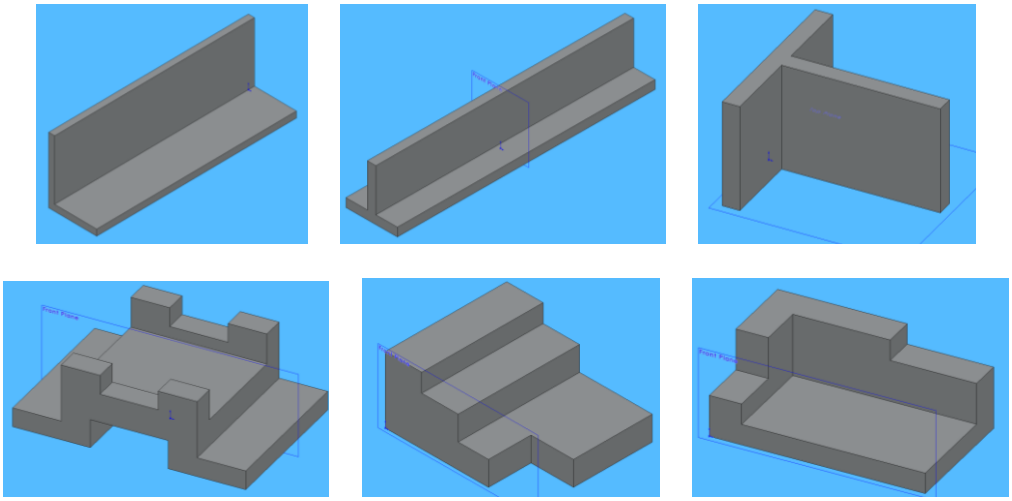


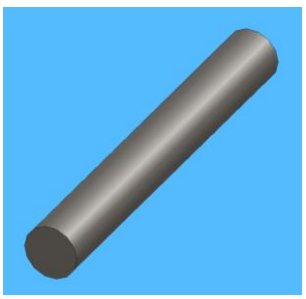
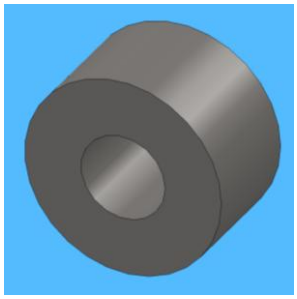
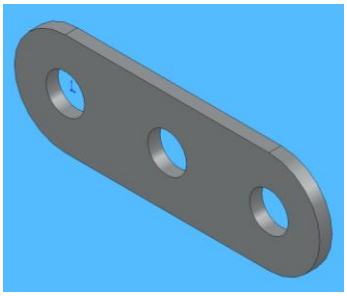
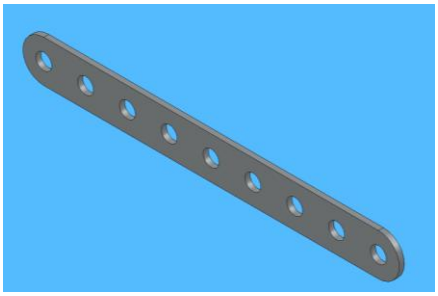
Figure 8.

In creating this part I learned how to apply Slot Sketch Tool, Extrude Cut, Dynamic Mirror, Hole Wizard, Linear pattern, and Fillet Tools. I also learned how to calculate the Mass Properties of this part. Modifications were made to the part to provide Tolerances and to correct for detected Interferences.



Figures 9 thru 14.

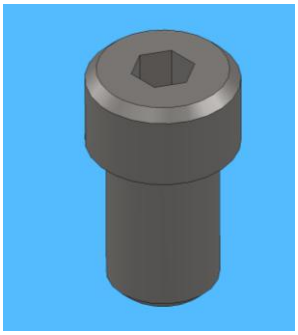
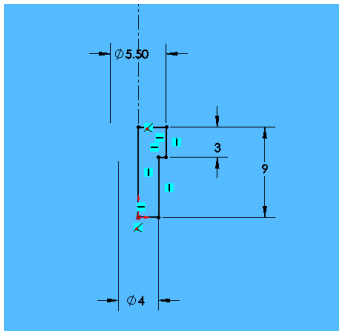
These exercises gave me practice in the use of the Sketch Tools, Dimension, Extrude Boss, and Extrude Cut Tools.

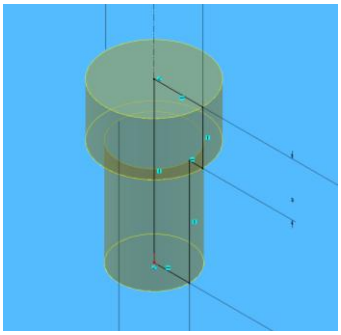





Figures 15 thru 18.

These components were created for an assembly.

The flat bars were created using the Linear Pattern Tool. The long bar was made by saving a copy of the short bar, then modifying the length and increasing the number of holes in the pattern.



Figures 19 thru 21.

Screws were created by applying a revolved base feature about an axis from a profile sketch.

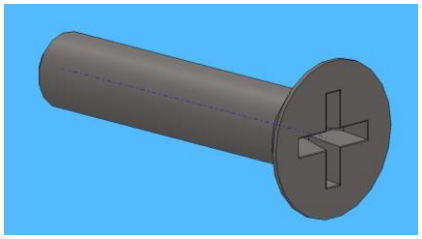


Figure 22.

Draft was applied to create the pocket in this Phillips Flat Head screw.

Assemblies and Sub-Assemblies

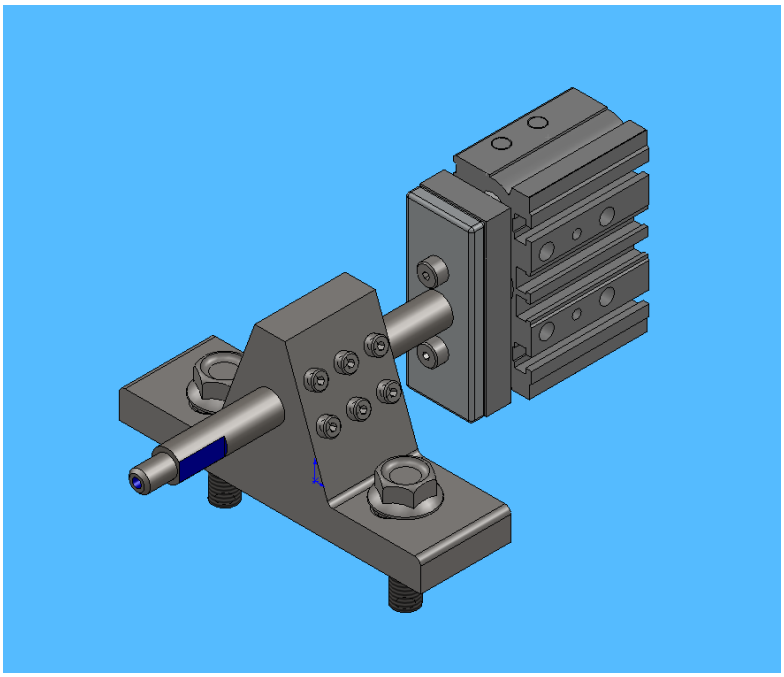


Figure 23.

Sub-Assemblies were created using parts I created, fasteners imported from the Design Library and components imported from ContentCentral.

Parts were inserted, moved and rotated, and then Mates were established to control the 6 degrees of freedom.

By suppressing fasteners, performance of the computer is enhanced during rebuilds.

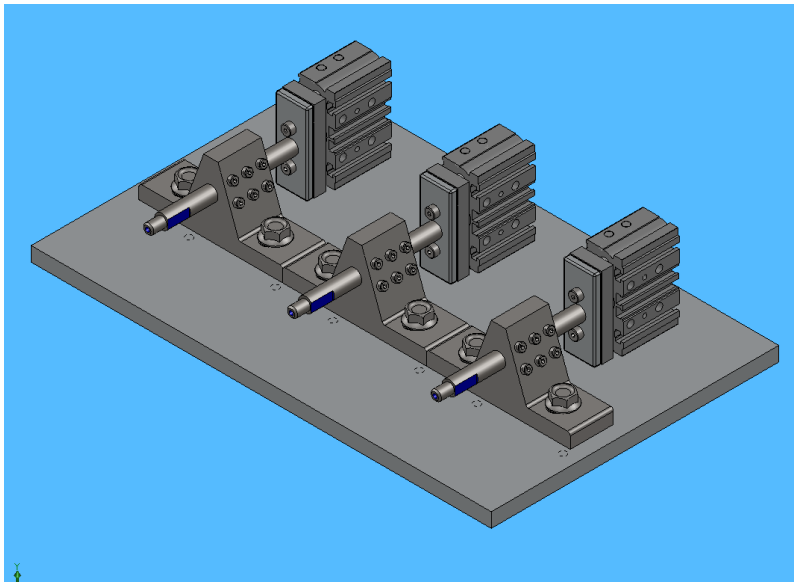


Figure 24.

Assemblies were created from Sub-Assemblies.

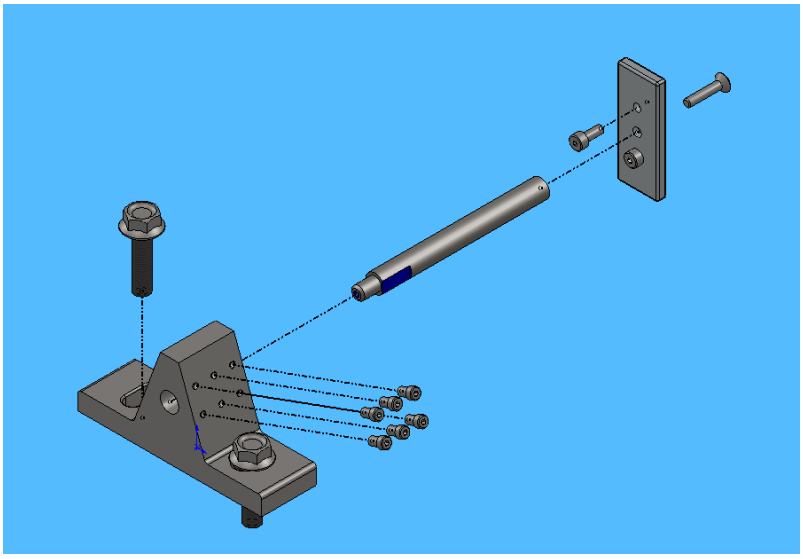


Figure 25.
Exploded Views show how parts in an assembly will go together.

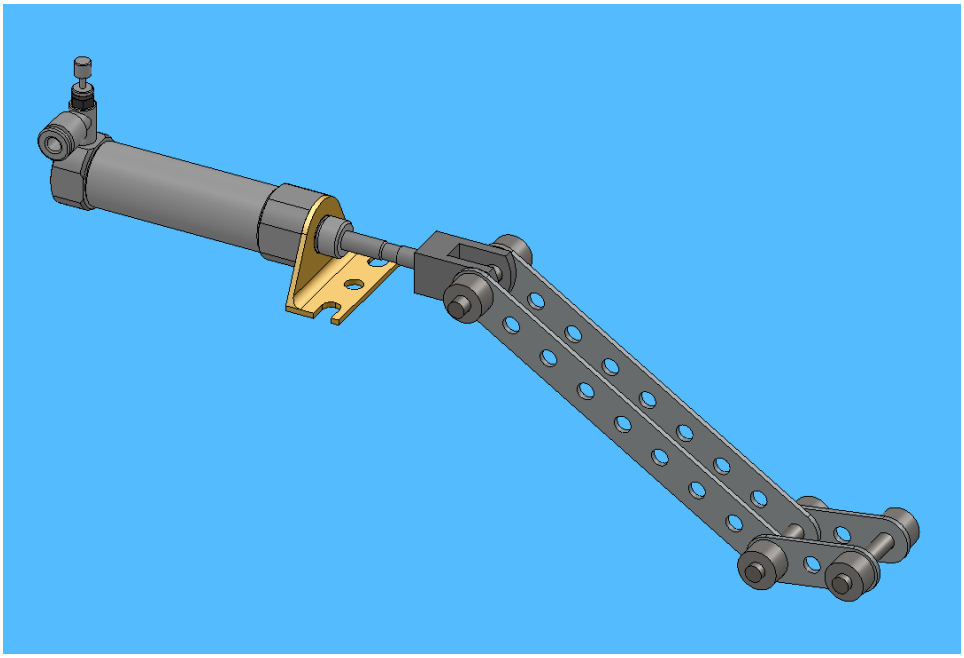


Figure 26.
This assembly utilized parts I made and an air cylinder assembly imported from ContentCentral. Mates were established to assemble the parts and control their position.

Orthographic Drawings and Templates

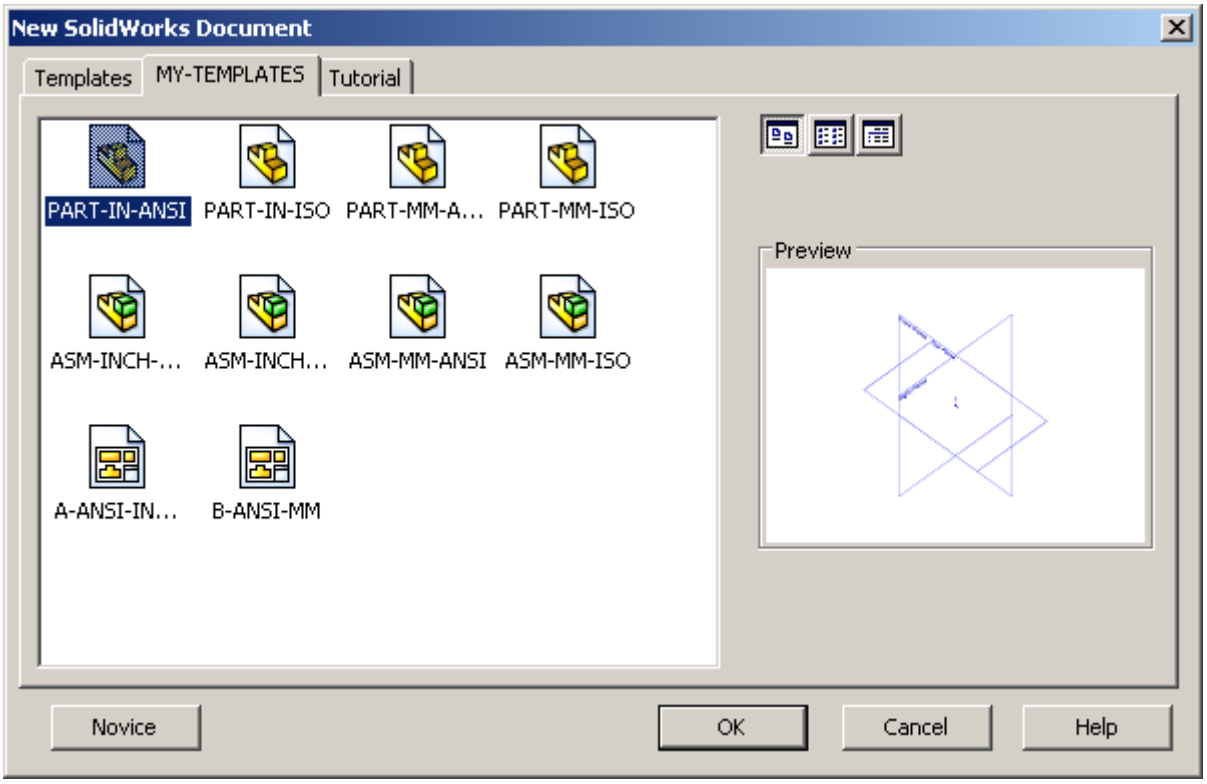
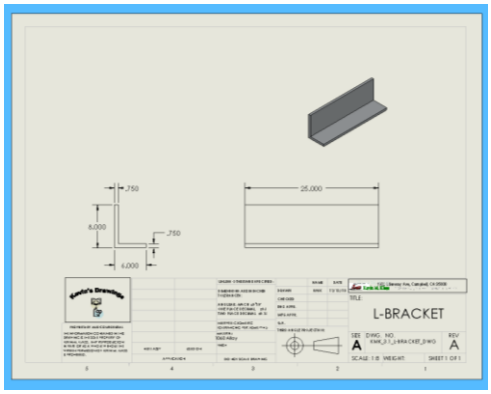
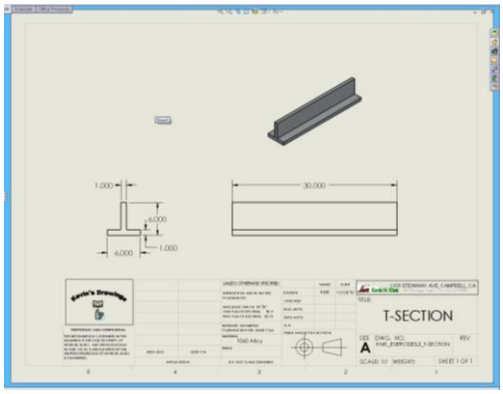


Figure 27.

Here are ten Templates I created for Parts, Assemblies and Drawings.
The document options have been set for dimensioning preferences, drafting standards and units of measure.
The drawings contain Sheet Formats, which have title block details and a personal Logo.





Figures 28 and 29.

These drawings were created with the Drawing template A-ANSI-INCH, using the Parts from Figures 9 and 10.

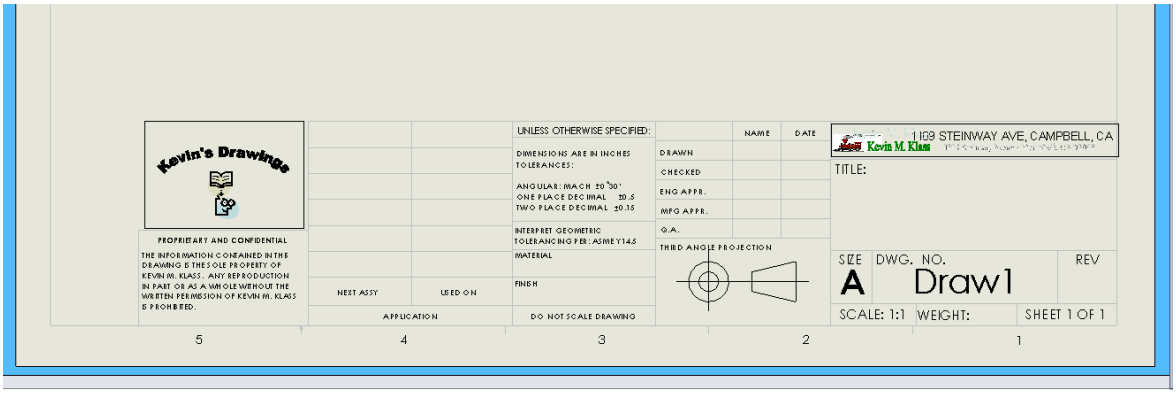


Figure 30.

This shows the custom LOGO and title block notes I added to my A-size drawing Sheet Format.

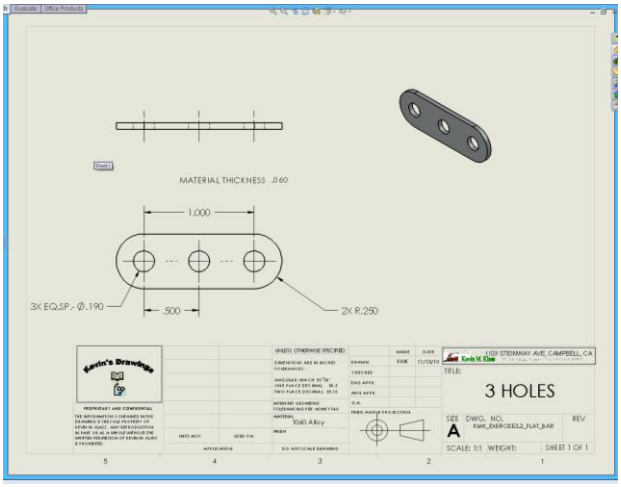


Figure 31.

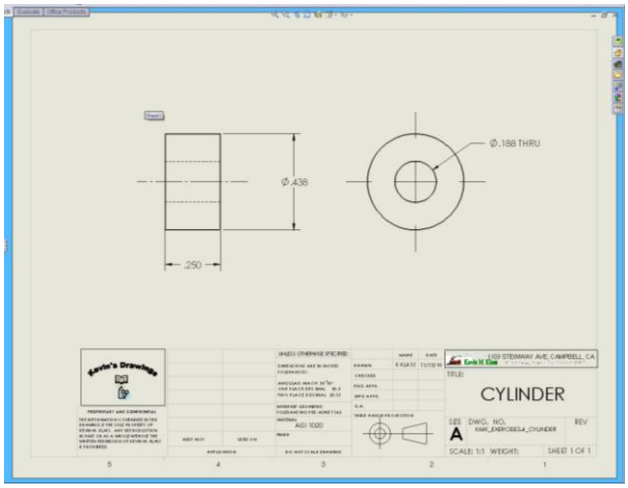


Figure 32.

Here are two drawings I created using my Drawing Template A-ANSI-INCH, from parts in Figure 16 and 17. I learned to import dimensions from the model with the Model Items Tool.

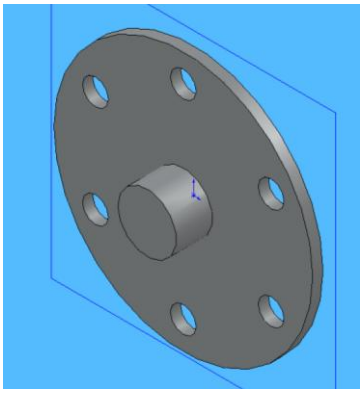


Figure 33 – PRESSURE PLATE.

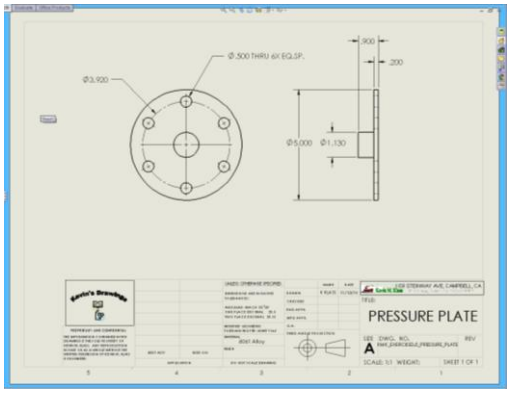


Figure 34 – Drawing of PRESSURE PLATE.

Detailing with Annotations and Tables

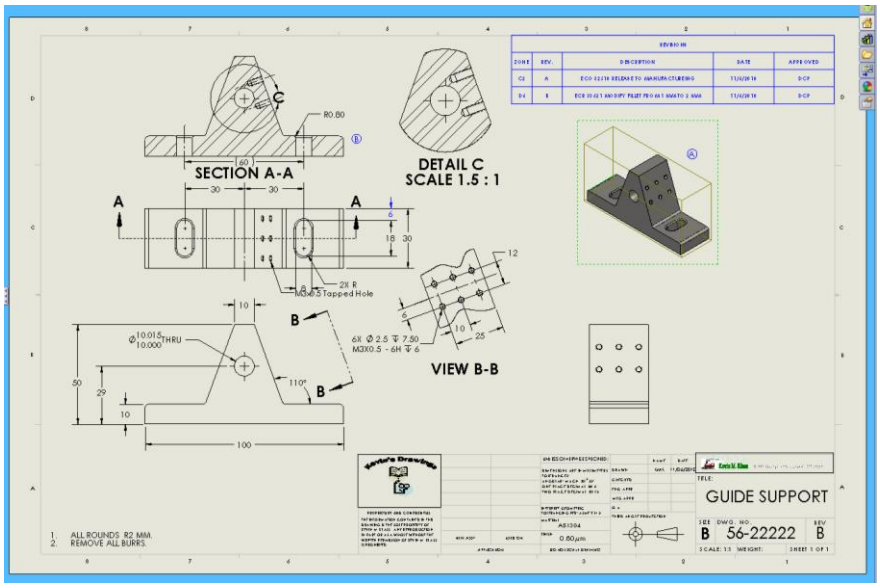


Figure 35.

While creating this drawing, I learned to apply the revision block, import dimensions, move dimensions around and create a section view A-A and detail C in a different scale. I also included an Auxiliary View B-B. I learned how to link dimensions Parametrically to notes, so that dimensions in the notes will update automatically when changes are made to the part model.

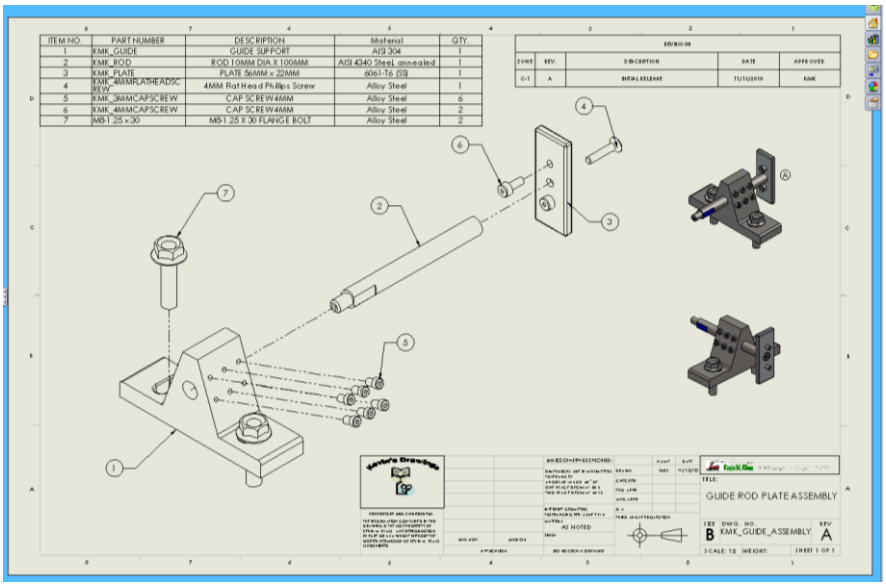


Figure 36.

A bill of materials was added to this assembly drawing with Bubble leaders to identify each component.

The Flashlight Project

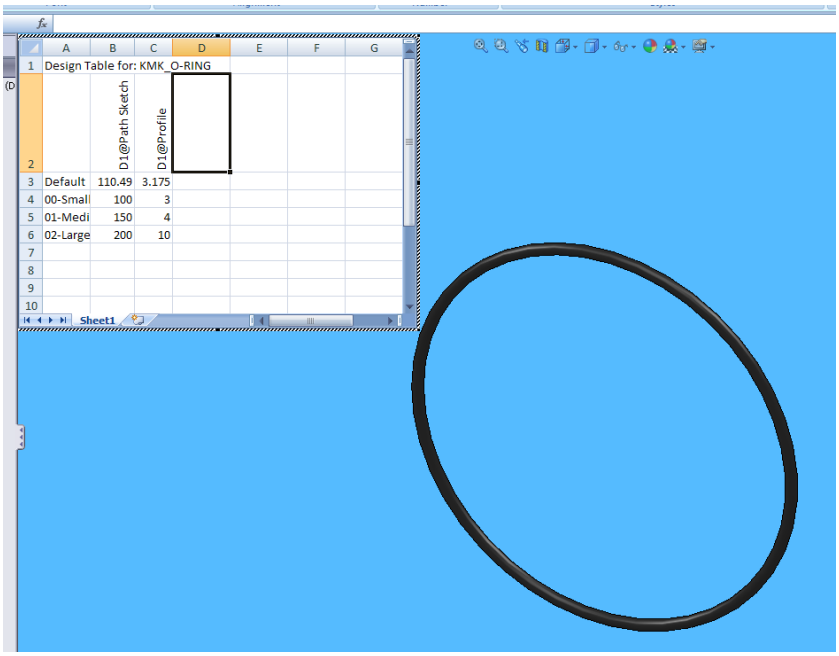


Figure 36 – O-RING.

I Created and edited an O-RING with a Design Table for various sizes.
The design table allows for the creation of a family of parts of various sizes from one model.

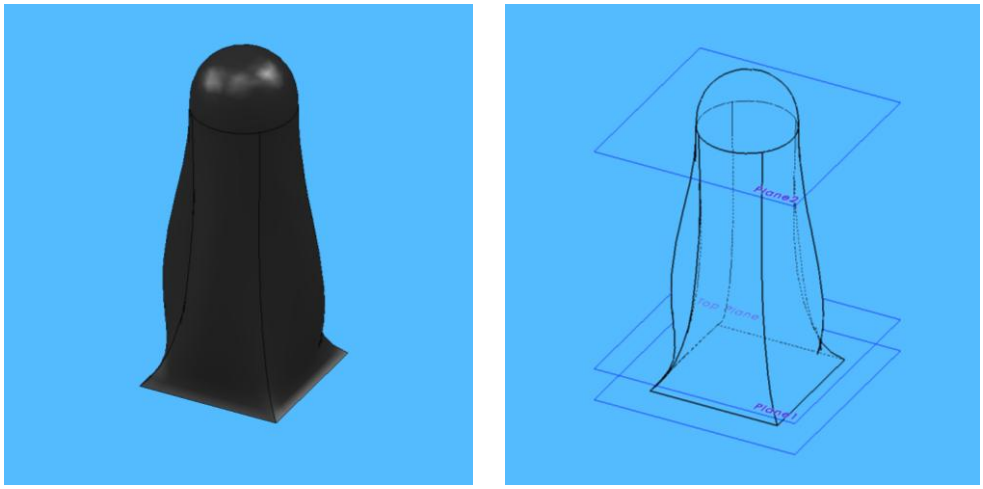


Figure 37 - SWITCH.

I learned how to use the Loft Feature with multiple sketch planes to create this switch.
The Dome feature added the dome.

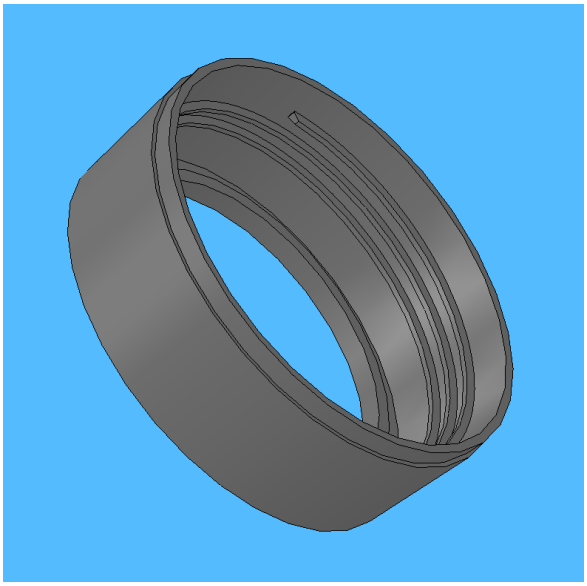


Figure 38 - LENSCAP.

The shell feature was used to hollow out the part.
The thread path was generated with the Insert, Curve, Helix/Spiral tool.
The Thread profile sketch was attached to the path with the Pierce relation at 90 degrees to the path.
Then, the Sweep Boss/Base tool generated the thread.

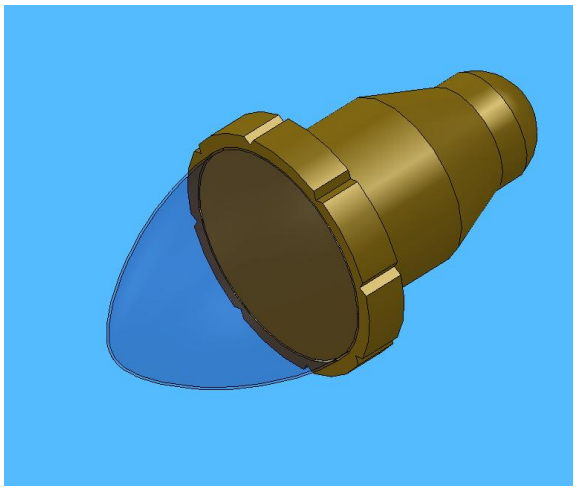


Figure 39 – BULB.

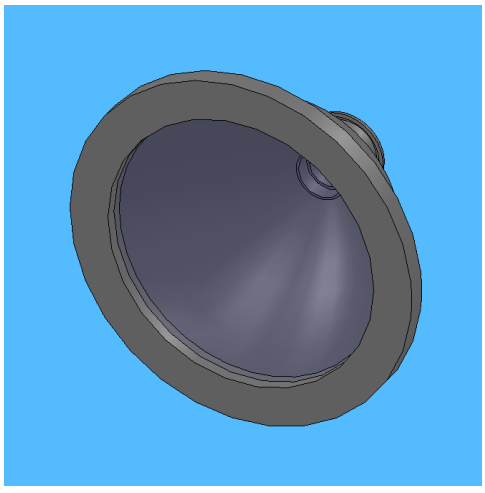


Figure 40 - LENS.

The Tangent Arc tool was used to create the bulb glass.
Both the bulb and lens have a blue tinted clear color applied to the glass.
The brass material applied to the bulb part gives it a golden color.

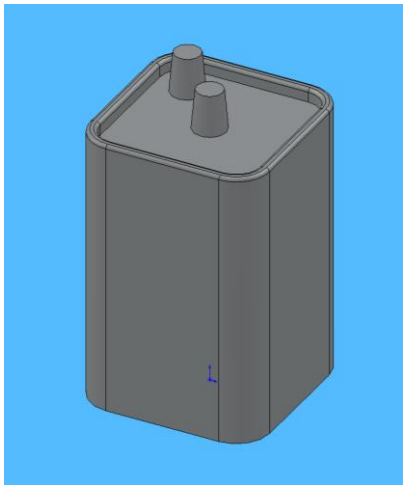


Figure 41 – BATTERY.

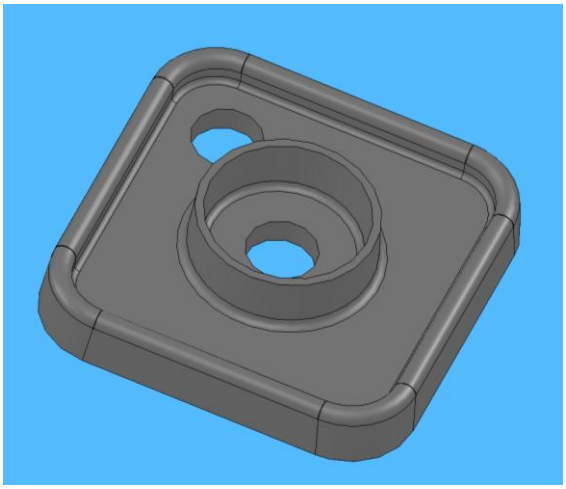


Figure 42 – BATTERY PLATE.

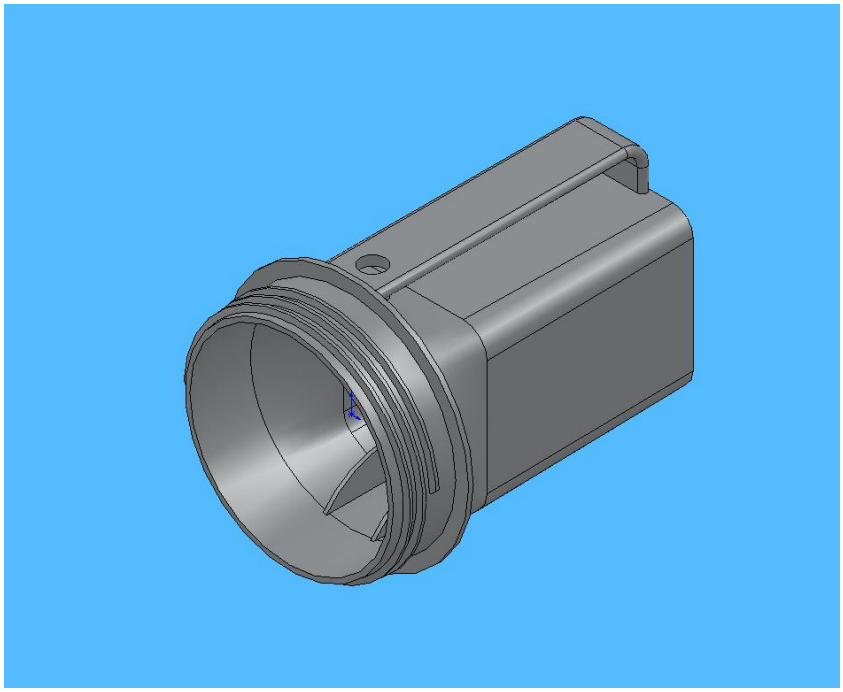


Figure 43 – HOUSING.

The Loft feature was used to generate the transitions from square to round shapes.
The Path Sketch, Profile sketch and Pierce relation was used to generate the handle.
A seed ribs was created with the Rib feature tool. The Linear Pattern tool was used to add more ribs.

Three sub-assemblies and one assembly

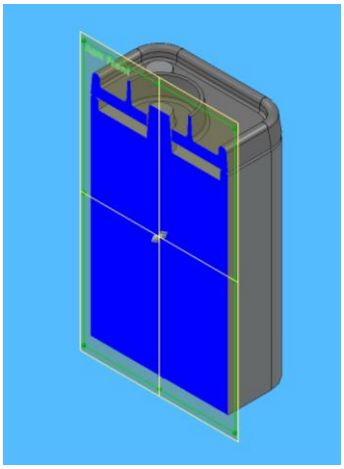


Figure 44 – BATTERY AND PLATE.

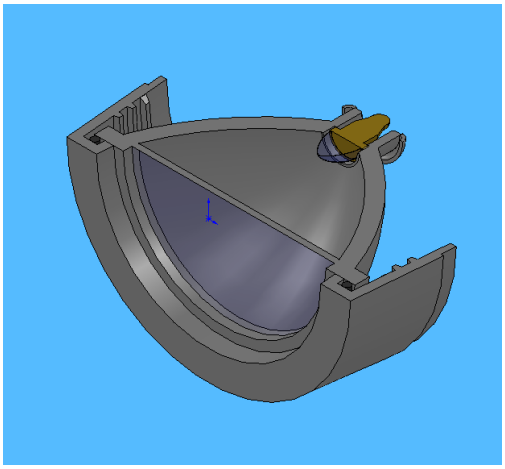


Figure 45 – LENS AND LENSCAP.

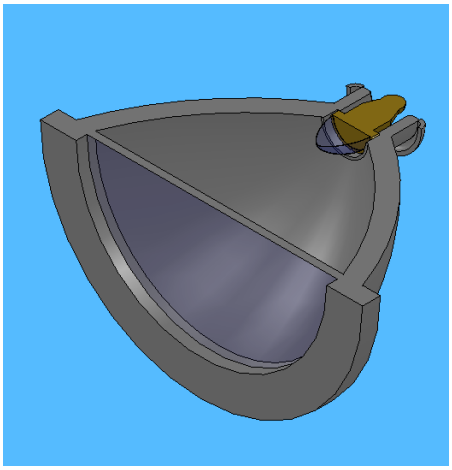


Figure 46 - LENS AND BULB.

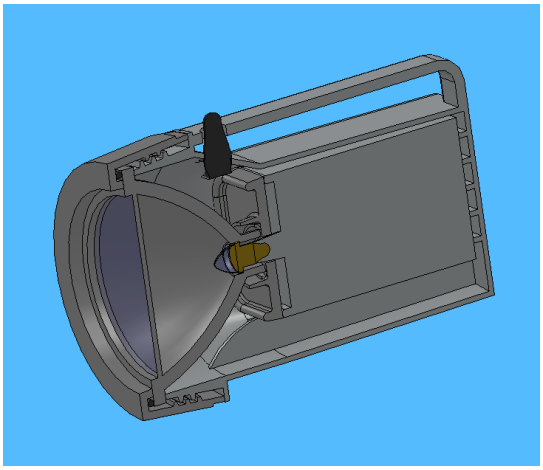


Figure 47 – FLASHLIGHT Assembly.

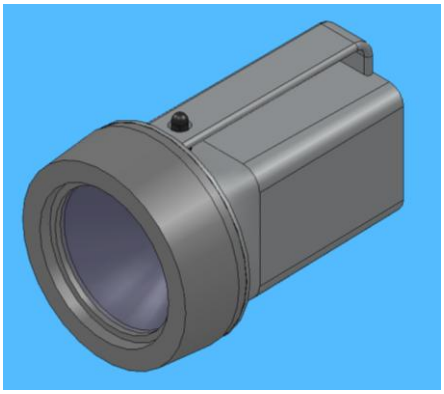


Figure 48 - FLASHLIGHT.

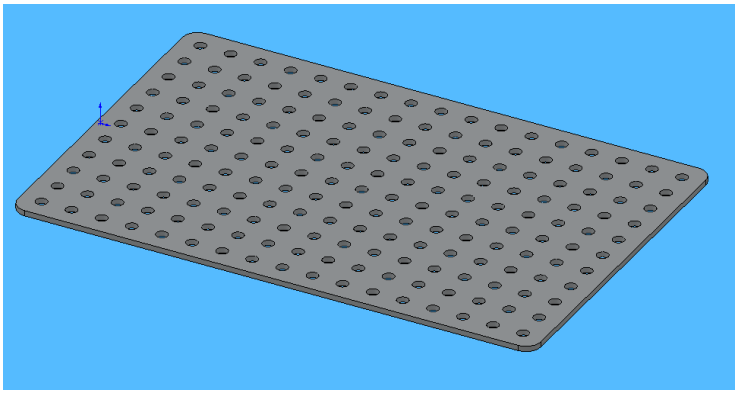


Figure 49.

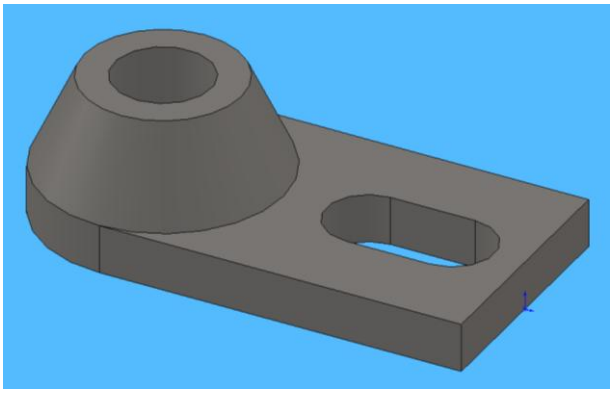


Figure 50.

Figure 51.

Top-Down Design Project.

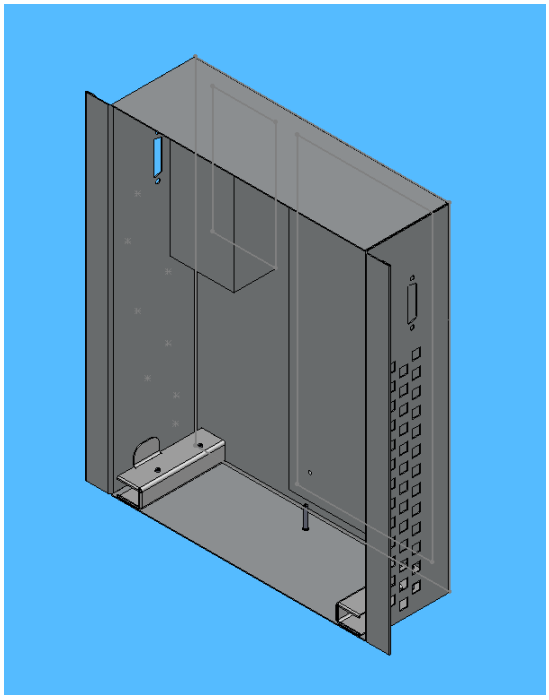


Figure 52. –BOX ASSEMBLY

Electrical Box with Power Supply and Motherboard was created in 3 different sizes.

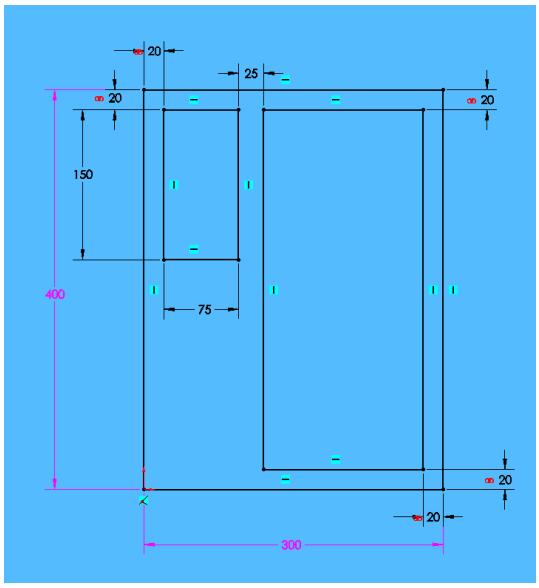


Figure 53.

Dimensions in the base sketch were linked so that gaps were maintained as the overall size changed.

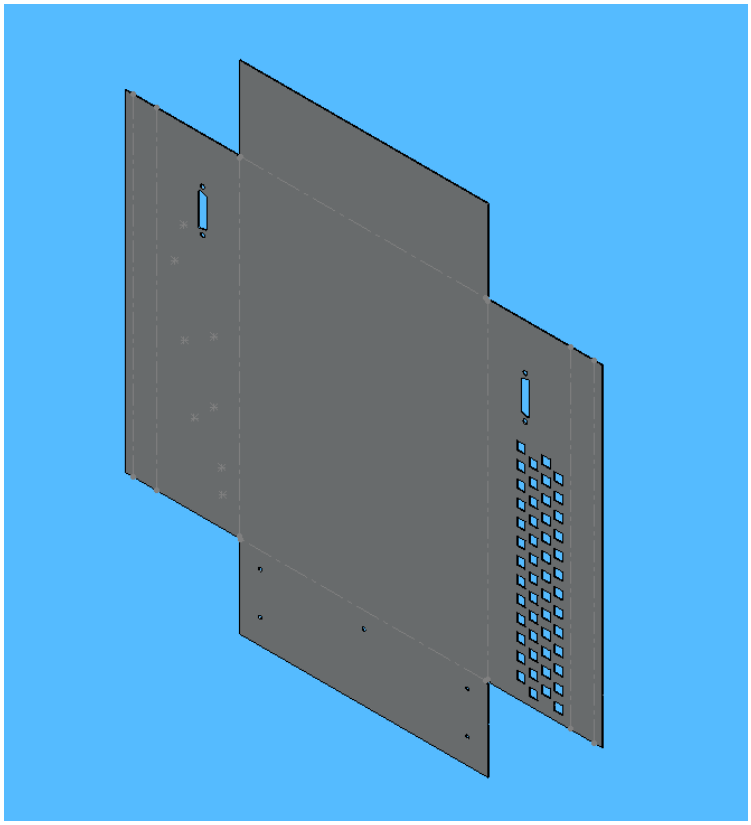


Figure 54.
Flat patterns are generated to show the sheet metal pattern as it is cut before bending.

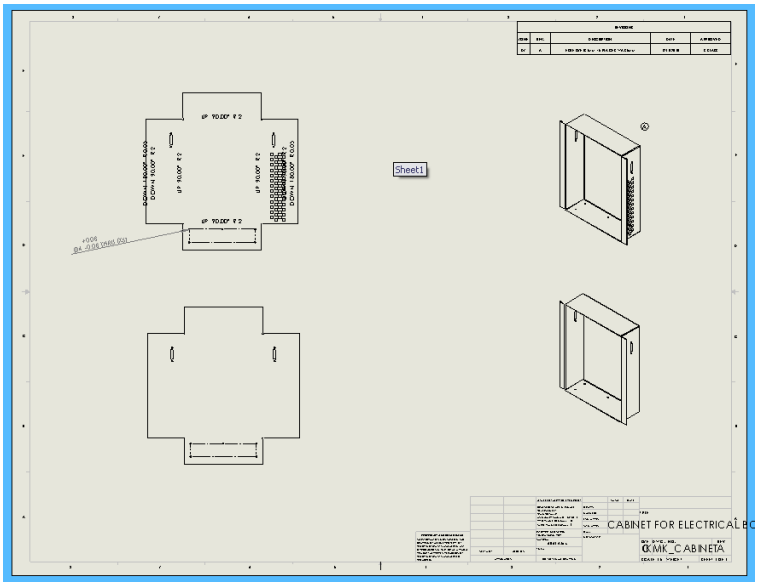


Figure 55.
Four different configurations of the CABINET were created. Two and shown in this drawing.

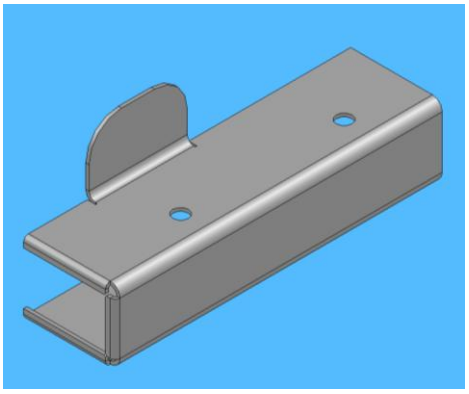


Figure 56 – LEFT BRACKET.

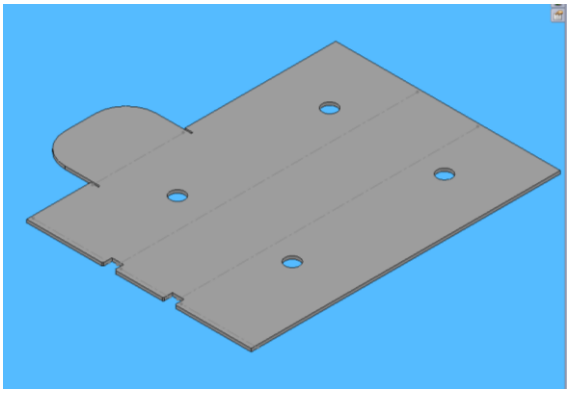


Figure 57 – LEFT BRACKET flat pattern.

The LEFT BRACKET was created “in-context” with the top level assembly to establish external relations to the CABINET.

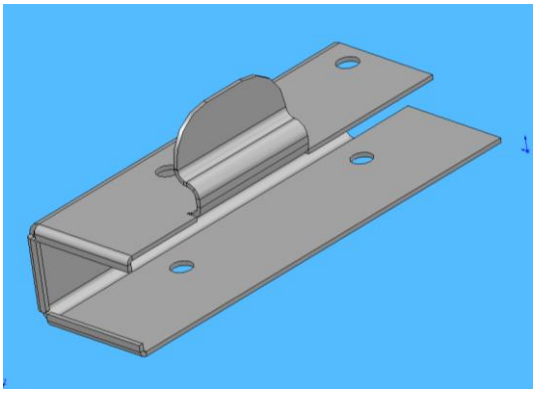


Figure 58 – RIGHT BRACKET.

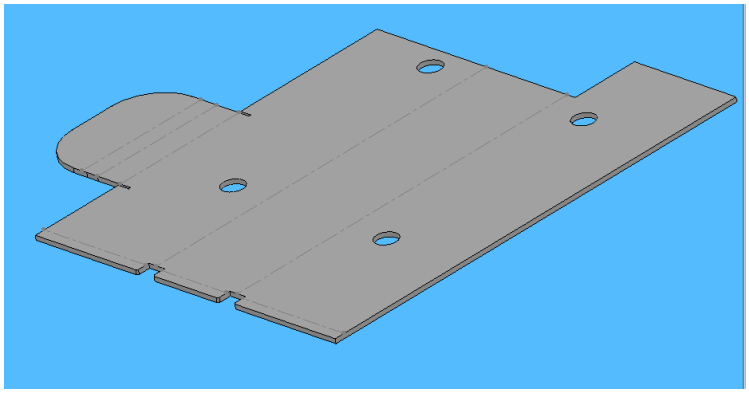


Figure 59 – RIGHT BRACKET flat pattern.

Created as a mirror of the LEFT BRACKET, the RIGHT BRACKET then was modified to suit the location.

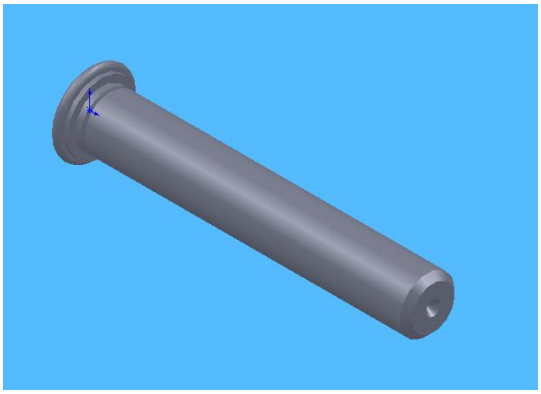


Figure 60 – 4MM PEM Fastener.

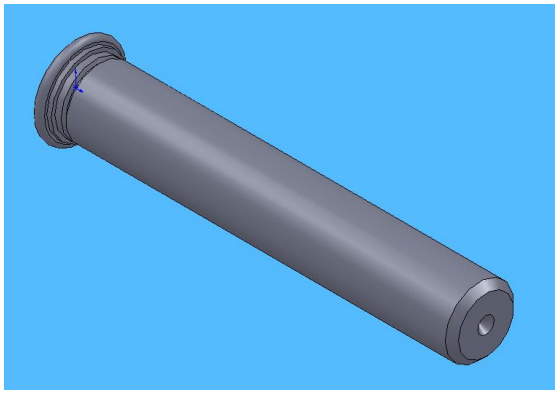


Figure 61 – 5MM PEM Fastener.

These PEM fasteners were imported from the manufacturer’s web site and applied to the Model.

Three sizes of the CABINET were created using the design table.
The space between the inner walls and the components remains constant by using linked values
The space available for the MOTHERBOARD varies as the size of the CABINET varies.
This relationship is created by establishing linked values between the base sketch in the assembly and the components that have external relations to the assembly sketch.
This is a key advantage of Parametric Modeling.

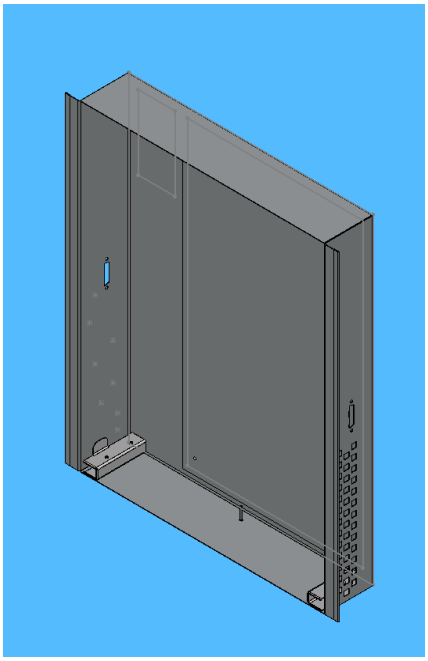


Figure 62 – large.

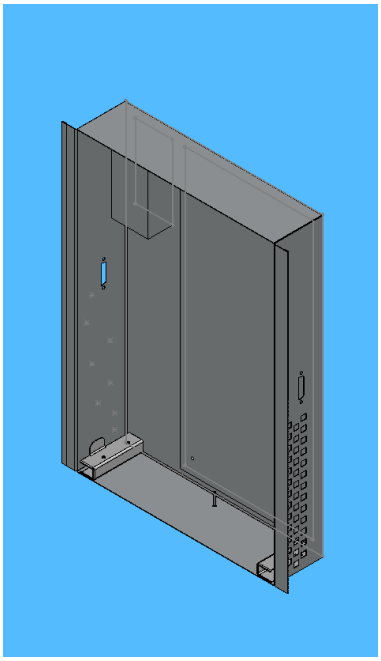


Figure 63 – medium.

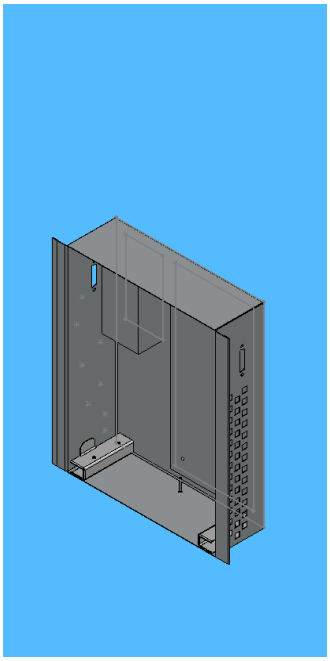


Figure 64 – small.

Acknowledgements

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