ENGR 1012 SOLIDWORKS Portfolio

Jarryd Ashby

TABLE OF FIGURES

IN CLASS EXERCISE: FLANGE

(pg 5-29)

The model in Figure 1 was created as the introductory tutorial to features within SolidWorks. The figure was generated by first sketching a circle and extruding the sketch into a disk. Then, a new plane was created offset from the original large disk, on which a new circle was sketched and extruded to the surface of the original. On the surface of the then created smaller cylinder, another circle was sketched, and an extruded cut was made through the object. On the surface of the wider cylinder, a small circle was sketched and was extruded as a cut through the larger cylinder. Then, utilizing the Circular Sketch Pattern tool, three additional circles were sketched at 90° intervals relative to the edge of the cylinders. To finish the figure, the Fillet tool was used to smooth the edges of the flange, and the Chamfer tool was used to make an angled opening to the flange. Additionally, this tutorial demonstrated features such as the Appearances and Smart Dimension tools, as well as the Draft feature of the Extrusion tools to create the taper of the smaller cylinder from the surface of the larger.

Figure 1. Flange with four bolt holes made utilizing Extrusion tools, Circular Sketch Pattern tools, and Fillet and Chamfer tools

HOMEWORK EXERCISE: STEPPED SHAFT

(P1.1)

The stepped shaft, Figure 2, was an application of the features utilized in the flange exercise (Figure 1). First, a circle was sketched, then extruded to a length of 8.00 inches. Then, upon the surfaces of the ends of this cylinder, circles were sketched of diameters 0.750 and 1.125 inches, and were extruded from the surface by 3.00, and 4.00 inches, respectively. Finally, the edges where the cylinders meet were smoothed using the Fillet tool, where the radii were 0.38 and 0.25 inches, respectively.

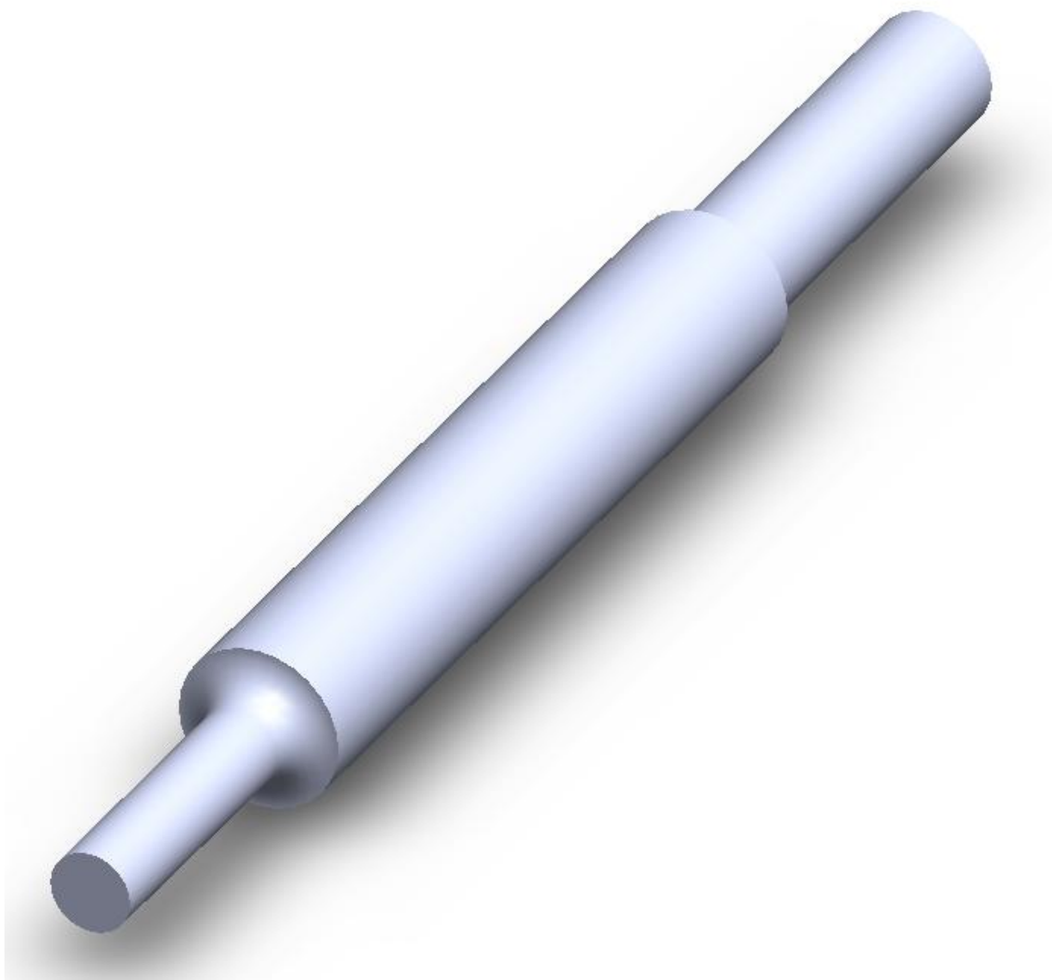

Figure 2. Stepped Shaft created using Extruded Boss/Base and Fillet tools

HOMEWORK EXERCISE: OBJECT C

(P1.2-C)

The homework object in Figure 3 was, like the stepped shaft in Figure 2, an application of the features learned in the flange exercise. Constructing the object made use of the Extruded Boss/Base and Extruded cut tools. First, a rectangle of dimensions 4 in. x 4 in. was sketched. This square was then extruded to a height of two inches. After this, a circle was sketched, centered, on the top surface of the rectangular prism, and Smart Dimensioned to a two-inch diameter. This circle was then extruded to a height of two inches relative to the surface where the circle was sketched. A 2 in. x 1 in. rectangle was then sketched on a surface of the rectangular prism perpendicular to the base of the cylinder, and centered such that it was one inch from each of the edges. Finally, this rectangle was Extrude Cut through the length of the rectangular prism to the other side.

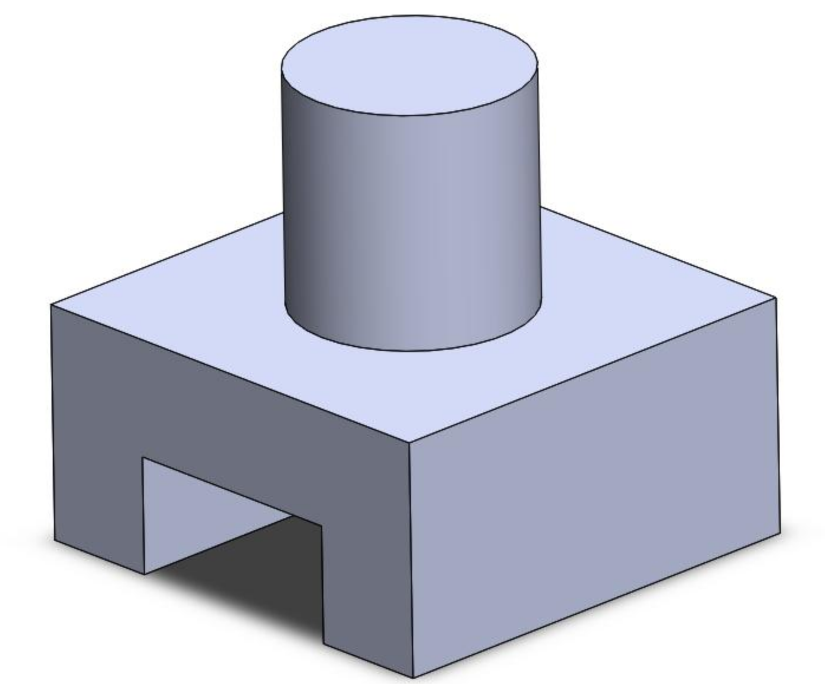

Figure 3. Homework Object made by using Extrude Boss/Base and Extrude Cut tools

IN-CLASS EXERCISE: PULLEY

(pg 35-45)

Construction of the pulley (Figure 4) was a tutorial on the use of the Revolved Boss/Base tool, as well as sketching features, such as the ability to set two lines symmetrical relative to a centerline. First, a pair of center rectangles were sketched along a sketched centerline. These rectangles were then connected by a corner rectangle that crossed the centerline. The two sides of this rectangle that ran parallel to the centerline then were set to be symmetrical which sets them equidistant from the centerline. The dimensions were adjusted by Smart Dimension, and then the Revolve Boss/Base tool was utilized to rotate this figure about the origin to generate a wheel. After, a trapezoidal sketch was constructed on the previously revolved sketch and a Revolved Cut was performed which created the groove around the circumference of the wheel. Fillets were added around the inner edges of the wheel Finally, a notch was made through the axle hole by sketching a rectangle on the surface of the axle protrusion and snapping the midpoint of the rectangle to the origin. Then, an extruded cut was made through the object. Additionally, this tutorial guides the user through the use of the Mass Properties feature of SolidWorks to find the mass of the object given construction with a specific material.

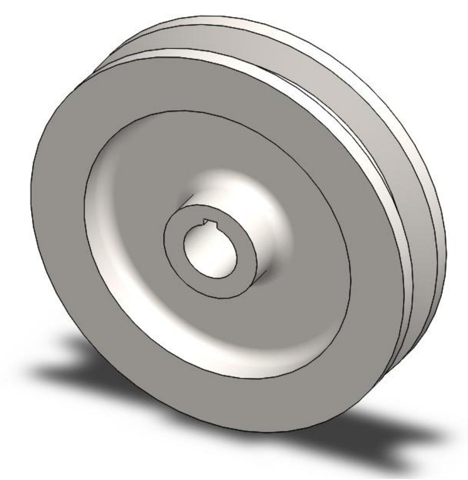

Figure 4. Pulley constructed using the Revolved Boss/Base, Revolved Cut, Extruded Cut and Fillet tools

HOMEWORK EXERCISE: LAMP BASE

(P1.4)

Generating the lamp base was an application of the skills utilized in the construction of the pulley (Figure 4), as well as the Fillet tool in Figure 1. The lamp base was constructed utilizing a series of connected line segments set parallel to a centerline, and then revolved about that centerline. Additionally, the Fillet tool was utilized to smooth edges where line segments meet. The Materials tool was used to set the material of the base to pine wood.

Figure 5. Pine lamp base made by Revolved Boss/Base and Fillet tools

HOMEWORK EXERCISE: BRAKE ROTOR

(P1.6)

The brake rotor was an application of the Circular Sketch Pattern, Fillet, and Extrusion tools. First a series of eight concentric circles were sketched and dimensioned, four of which were set to "For Construction" such that they would not affect the final extrusion. Then along three of the concentric circles three small circles were drawn at an angle to one another, and the circles were patterned around the ring by the Circular Sketch Pattern feature. On the inner ring, this function was utilized to produce six holes. To produce the spokes, line segments were drawn from the inner ring to the outer parallel to centerlines and set to be symmetrical to one another. This was also patterned around the wheel to make four spokes. Finally, the completed sketch was extruded and fillets were then added where the wheels and spokes meet to produce the finalized brake rotor.

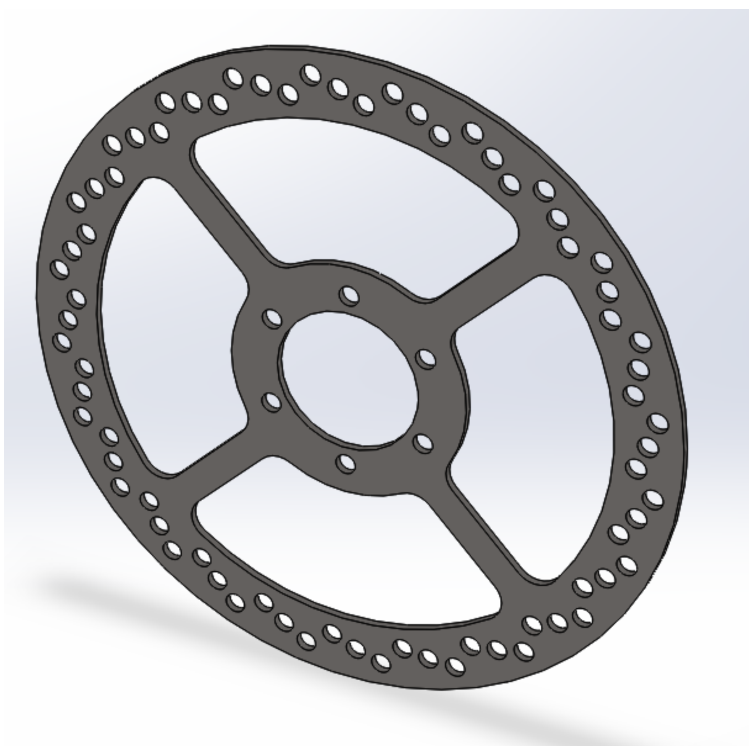

Figure 6. Brake rotor created through Circular Sketch Pattern, Fillet, and Extrusion tools

IN-CLASS EXERCISE: ENGINEERING DRAWING OF FLANGE

(pg 55-79)

This exercise was a tutorial on generating Engineering Drawings using SolidWorks, as well as how to construct a template for engineering drawings. The drawing in Figure 7 is a simplified (without the draft on the smaller cylinder extrusion). The drawing was made by importing the flange part and rendering top, front, right, and trimetric views of the flange part. A feature of SolidWorks allows parts to be modified from drawings and vice-versa. Additional features utilized to produce the drawing include the Detail View feature, which allows for the viewing of zoomed in portions of the part shown, as well as the Section View, which generates a cross-sectional view at a desired position. The sheet itself was generated by modifying a preexisting template without its features and adding rectangles to contain information about the sketch, and the Note tool to input that information, along with the Link to Property feature to set the scale on the drawing automatically.

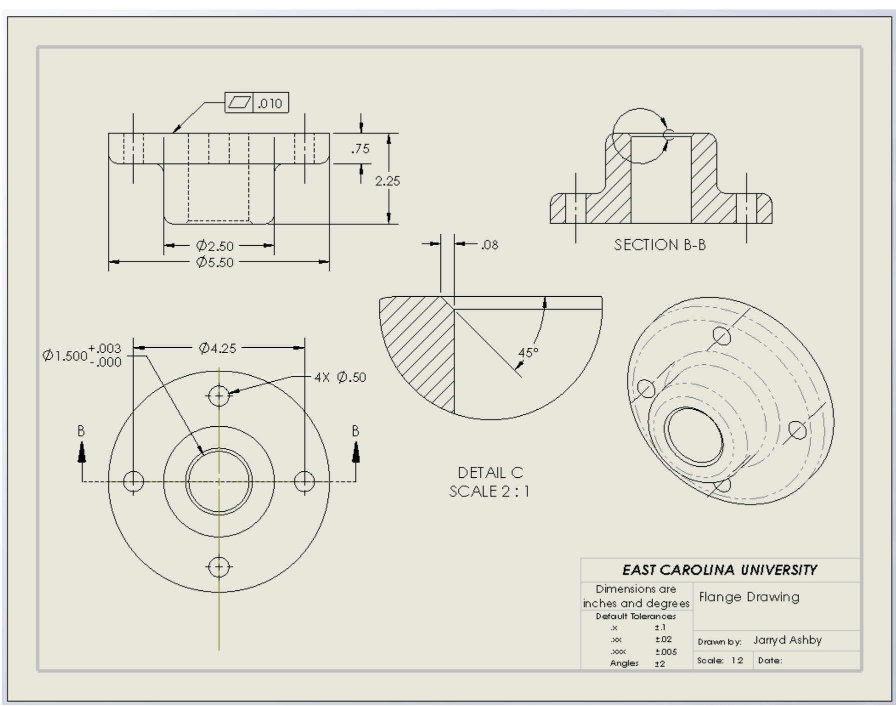

Figure 7. Engineering drawing of flange (*Produced in **Figure 1***)

HOMEWORK EXERCISE: ENGINEERING DRAWING

(COURSE PACK MV1)

The engineering drawing in Figure 8 applies the skills used to make the drawing in Figure 7. To construct this drawing, first, object MV1 from the Course Pack was constructed as a part, and then imported into a new drawing on the template created for Figure 7. Due to the Link to Property feature of the Note tool, the scale automatically changed to 1:2. The dimensions of the views were imported directly from the fully defined sketches used to make the object originally.

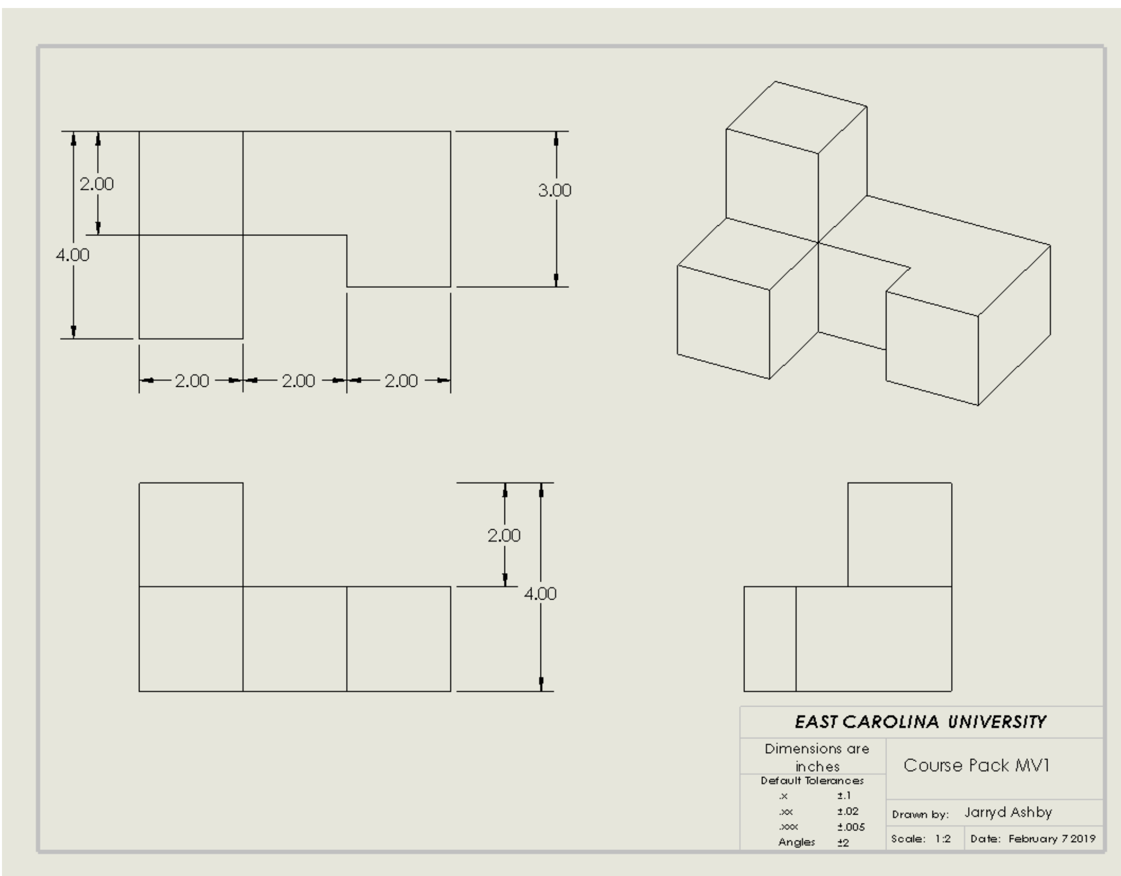

Figure 8. Engineering drawing of Course Pack Problem: MV1

IN-CLASS EXERCISE: BEAM SEGMENTS

(pg 84-91)

The beam segments in Figure 9 were generated from a tutorial on producing similar parts of multiple configurations with the Configuration Manager. The first beam segment was generated by sketching a pair of rectangles and connecting them with line segments, and then making them symmetrical along centerlines extending from the origin along the top and right planes. Using the now introduced Trim Entities tool, segments of the rectangles were cut such that an I shaped figure was formed. The figure was extruded to form the beam, the Material tool was used to make the beam ASTM A36 steel. The Configuration Manager was used to change the dimensions of the beam, and save the new configuration as the same part.

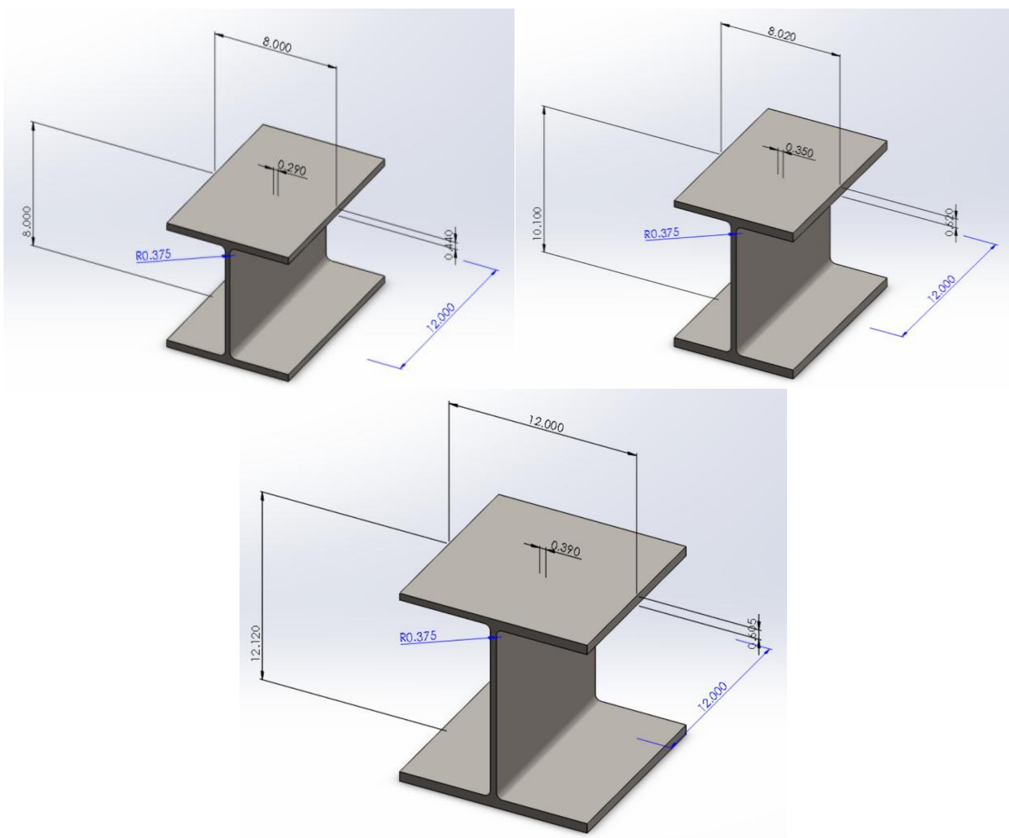

Figure 9.   Beam Segments (*Clockwise* from *top-left: W8x31, W10x45, W12x65)*

IN-CLASS EXERCISE: BRACKET

(pg 92-105)

The bracket (Figure 10) is a tutorial that demonstrates additional uses for the Reference Geometry tool with regards to planes, the tangent arc sketching feature, and the rib tool. The four bolt holes were created with the Linear Pattern tool. The triangular appendage extending from the rectangular mount was created by generating a new plane using the Reference Geometry tool. Then a centerline was sketched on the new plane after which an angled line segment was sketched. Finally a tangent arc was made from the line segment to the centerline, and the lines were mirrored to complete the shape, which was tuned with Smart Dimension. The raised cylinder and hole at the tip of the bracket were created with the Extruded Boss/Base tool and Extruded Cut tool, respectively. The central rib was created by Mid Plane extrusion of a sketched shape made from a line segment that meets with the rectangular prism and curved triangular piece. The other smaller ribs were generated by "moving" the Right Plane to generate a new plane, sketching a line segment on the new plane in a manner similar to the central rib, and, by using the Rib tool, generating a rib. The other was formed by using the Mirror tool about the Right Plane.

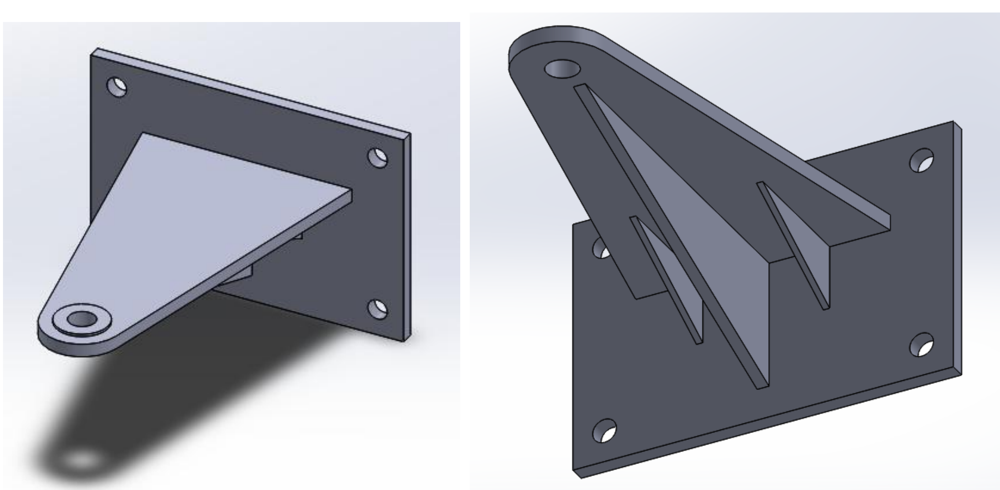

Figure 10. Bracket created with Extrusion tools, Rib tools, and Linear Pattern tools

(Trimetric View: Left; Right view shown for ease of viewing ribs)

HOMEWORK EXERCISE: BEAM WITH HOLE PATTERN

(P3.3)

The beam shown in Figure 11 was an application of Linear Pattern and Mirror tools. The beam was created by sketching line segments constituting the edges of one-quarter of the cross-section reaching to perpendicular centerlines extending from the origin. These line segments were then mirrored about the two centerlines to great the completed I-shape. The I-shape was then extruded to 36-inches. A small circle was then sketched on the face of the beam web and the Extruded Cut tool was used to make a hole through the web. The Linear Sketch Pattern tool was then used to generate a total of six holes, which were then repositioned in distance from the edge of the beam using Smart Dimension. A centerline was then drawn perpendicular to the web of the beam at the midpoint. Finally, the Mirror tool was used about that centerline to generate six tools at the opposite end of the beam.

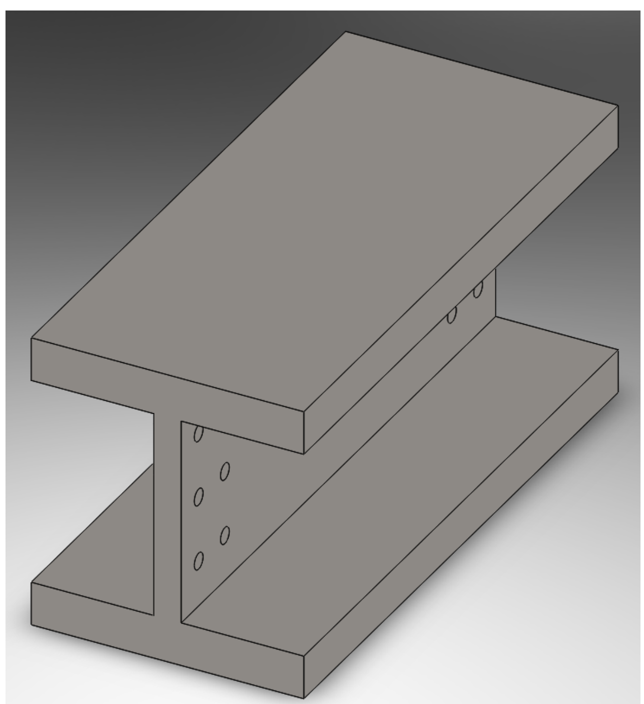

Figure 11. Beam with hole pattern created with Mirror, Linear Pattern, and Extruded Cut tools

HOMEWORK EXERCISE: MUFFIN PAN

(P3.7)

The muffin pan (Figure 12) was an application of the Linear Sketch Pattern, Extruded Boss/Base, and Fillet tools. First a rectangle was sketched and extruded to form the tray. Then a Center Rectangle was sketched at the corner of the face of the top of the tray coincident with the edge of the tray to create a center point for sketching the circles that would form the well. A centerline was sketched from opposite corners of this square to find a midpoint. Two concentric circles were sketched from this midpoint. The larger was extruded downward from the tray to create the base of the well and then a cut was extruded from the smaller circle to make the muffin well. Both the extrusion and the extruded cut had a draft of ten degrees to create a tapered well. Using the Linear Sketch Pattern tool, the well was duplicated in the 4x3 pattern shown below. Finally, the corners of the tray were smoothed with the Fillet tool.

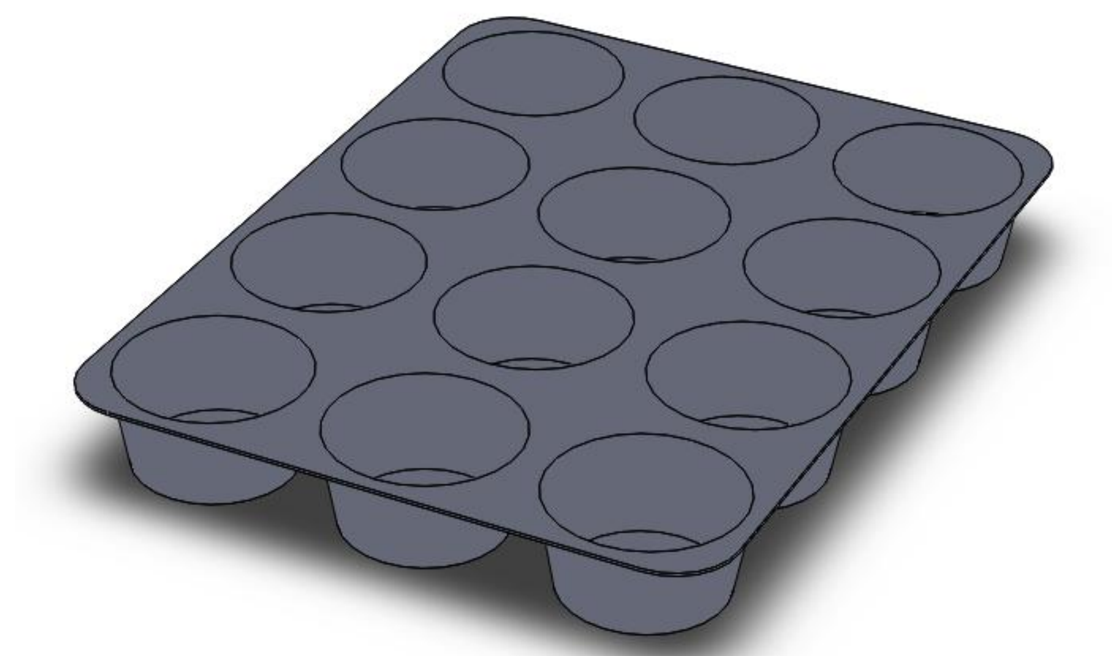

Figure 12. Twelve muffin pan made using Extrusion, Linear Pattern, and Fillet Tools

IN-CLASS EXERCISE: CARD HOLDER

(pg 117-125)

The card holder, Figure 13, was produced by sketching a rectangle and then creating an offset reference plane from the rectangle. On that plane, a straight slot was sketched, and then the Lofted Base tool was used to connect the two sketches as a sold piece. After this, a straight slot was drawn on the top portion of the figure and an extruded cut was made to create the hole for the cards to sit. Fillets were made to smooth the edges and then the Shell tool was used to hollow the underside of the card holder. The text tool was then used to write "Part #18122" on the bottom of the card slot, and this text was extruded to create a textured surface.

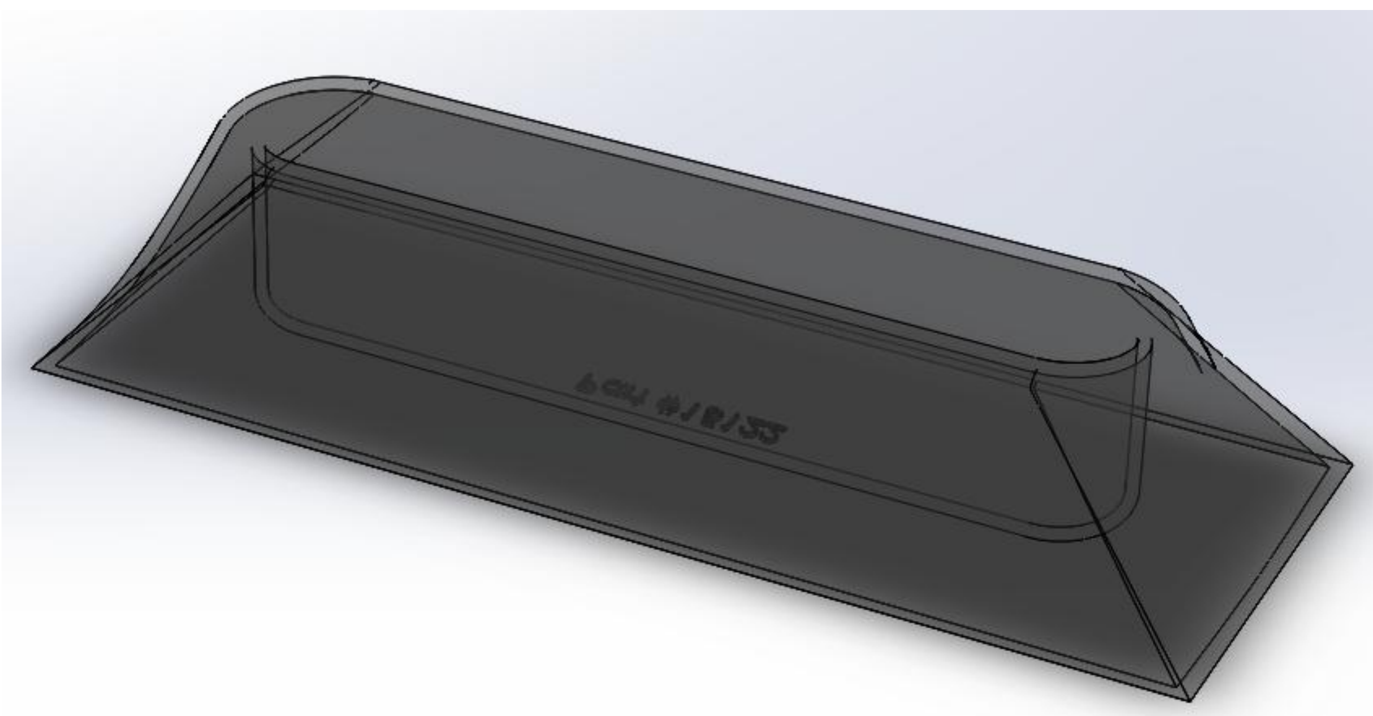

Figure 13. Card holder made using straight slot, lofted base, and shell tools

IN-CLASS EXERCISE: BENT TUBING

(pg 127-129)

The bent tubing shown in Figure 14 was made by sketching a line, followed by sketching a tangent arc, followed by a third line, and then using the Smart Dimension tool to adjust the angle of two lines and the curvature of the arc. After this, concentric circles of diameter .40" and .50" were drawn on the endpoint of the line such that the centers of the circles were on the line endpoint and the arcs of the circles were perpendicular to the line path. Finally, using the Sweep tool the tube shape was formed.

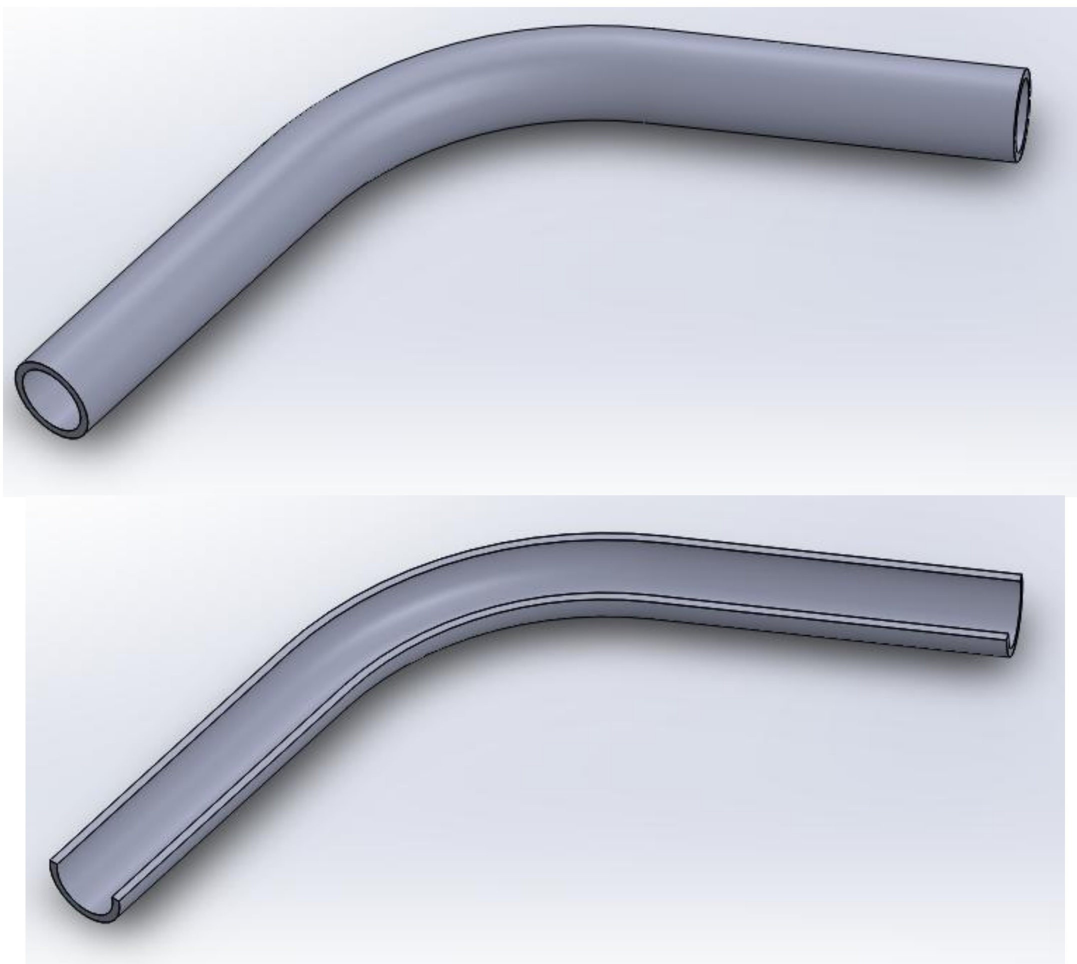

Figure 14. Bent tube and section view (*bottom*) made using sweep tool

IN-CLASS EXERCISE: HELICAL SPRING

(pg 129-131)

Figure 15 was also a demonstration of the Sweep tool, as well as the helix tool. First a circle was sketched of diameter 2.00". Then, the Helix tool to draw a helix from that circle of the same diameter. After this, a new reference plane was created that was incident with the endpoint of the helix and perpendicular to the helical path. Then a circle was sketched on that plane similar to how Figure 14 was done, and using the Sweep tool the helical spring was generated.

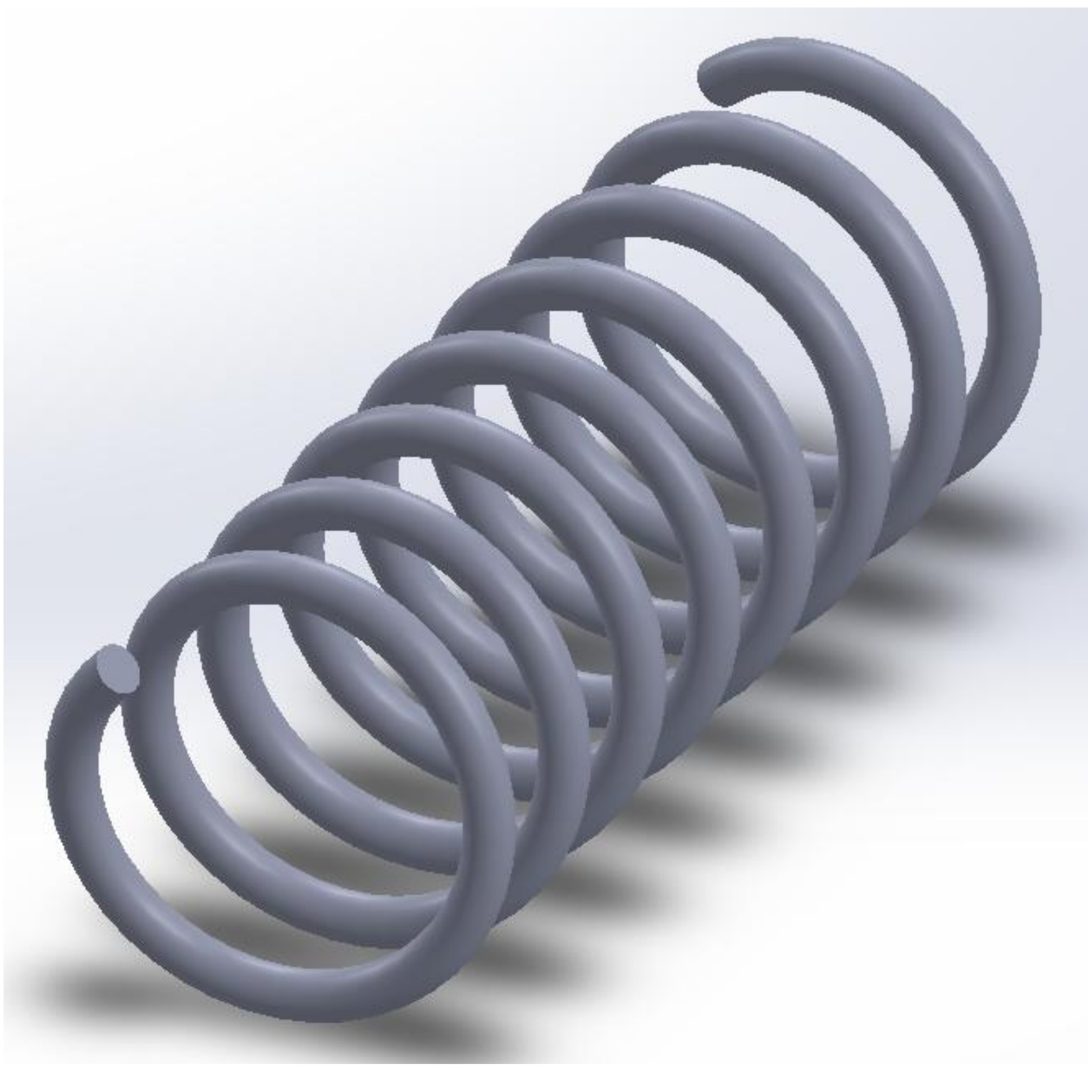

Figure 15. Helical Spring made using helix and sweep tools

IN-CLASS EXERCISE: HANDLEBARS

(pg 131-137)

The handlebars were made by making a 3d sketch of a series of lines to define the general shape, which were smoothed by the Sketch Fillet tool. Then at the starting point, i.e. what would be the center of the handlebar, two concentric circles were sketched and then swept down the length of the 3d sketch to make half a handlebar. Finally, the Mirror Entities tool was used to make the other half. as seen in Figure 16.

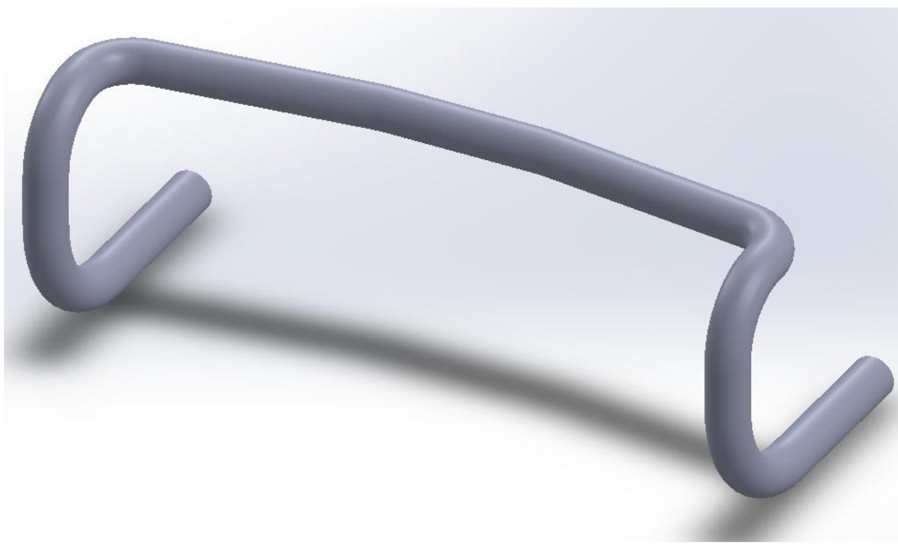

Figure 16. Handlebars made using a 3d sketch, linear fillets, and the sweep tool

HOMEWORK EXERCISE: BOTTLE

(P4.5)

To make the bottle shown in Figure 17, an ellipse was sketched. From there, a reference plane was generated using the Reference Geometry tool, and a second ellipse was sketched. After this a second reference plane was generated followed by a circle. Using the Lofted Base tool, the ellipses and circle were connected as one entity to make the general bottle shape, but still filled with solid. On the top plane(the circle), a circle coincident with the original circle was sketched and then extruded upward to make the bottleneck. Fillets were added to smooth the bottle at the neck/bottle junction, and then the Shell tool was used to hollow out the bottle.

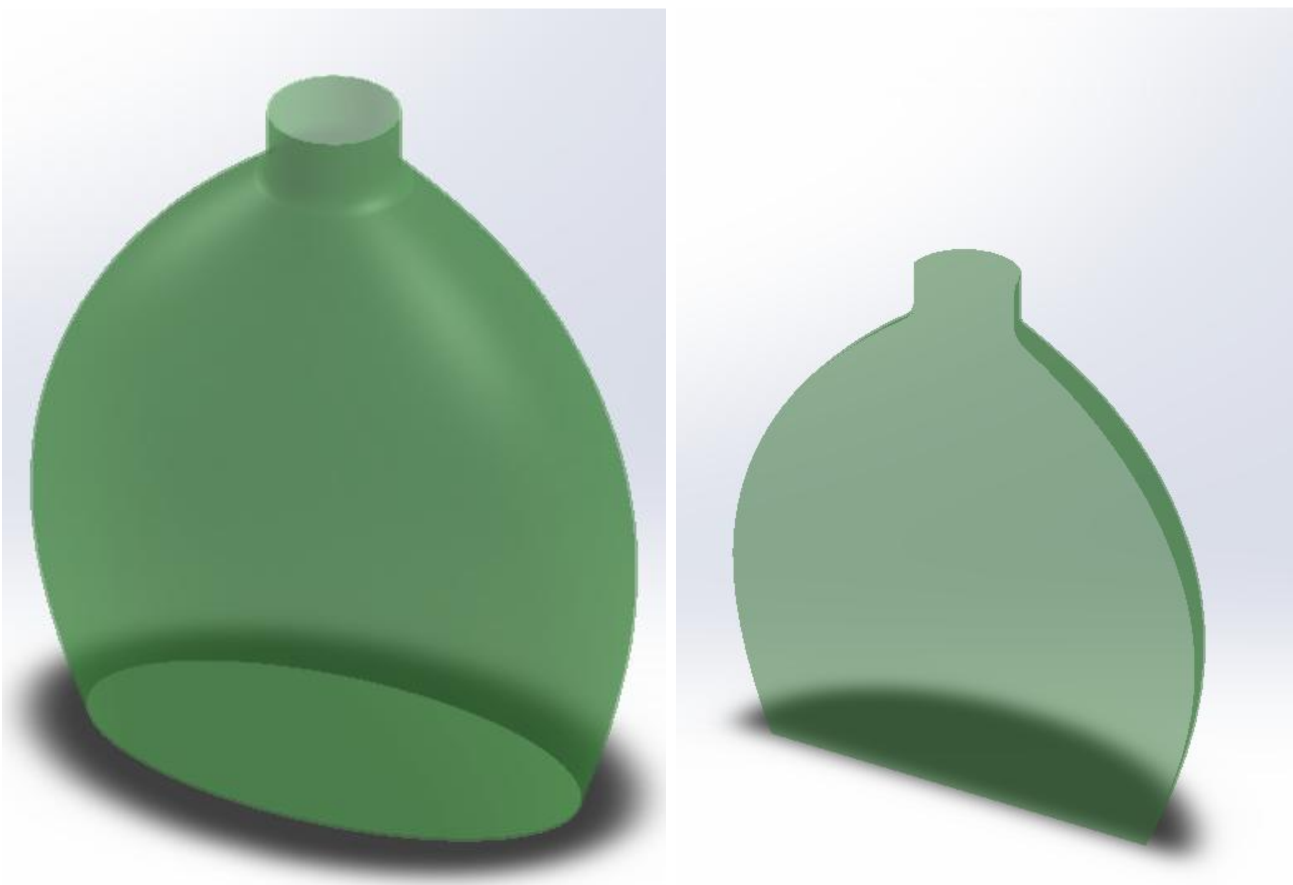

Figure 17. Green glass bottle and section view (*right*) made using reference planes, lofted base, and shell tools

IN-CLASS EXERCISE: DOOR ASSEMBLY

(pg 189-216)

The assembly in Figure 18 was first made by constructing the individual parts used to make it. The hinge was made using the Thin Feature portion of the Extrusion tool on a horizontal line and tangent arc, which gave bulk to the line to make it a full object. Extruded cuts were used to make the openings and the Hole Wizard feature was used to make the screw holes in the hinge. The "door" part of the door was made by simple extrusion of two squares. The screws were made by using the Revolved Base tool and Revolved cut tools on a cylinder to make the top and bottom parts, respectively. Finally, the door was assembled by opening a new assembly and then using the Insert Components tool adding the parts and using the Mate tool mating the parts to the appropriate locations relative to one another. The screws, however, were added using the Linear Component Pattern tool to add an additional seven screws rather than add each individually. Sketching a point and using the Hole Wizard tool the screw holes were added to the door.

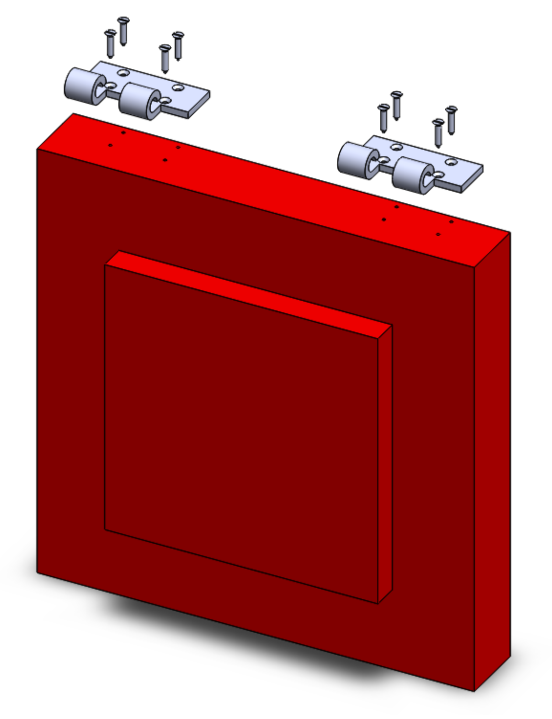

Figure 18. Door Assembly

IN-CLASS EXERCISE: HATCH ASSEMBLY

(pg 227-247)

The final hatch assembly, Figure 19, was made in the same manner as Figure 18; however, rather than add individual parts, this was made by making a new assembly and then adding the door assembly and additional parts and mating them. The frame was made by extruding a square and then cutting a square of the same dimensions as the door. The hinge pin was made by extrusion of two circles for the long part and the cap. All the parts were brought together using the Mate tool.

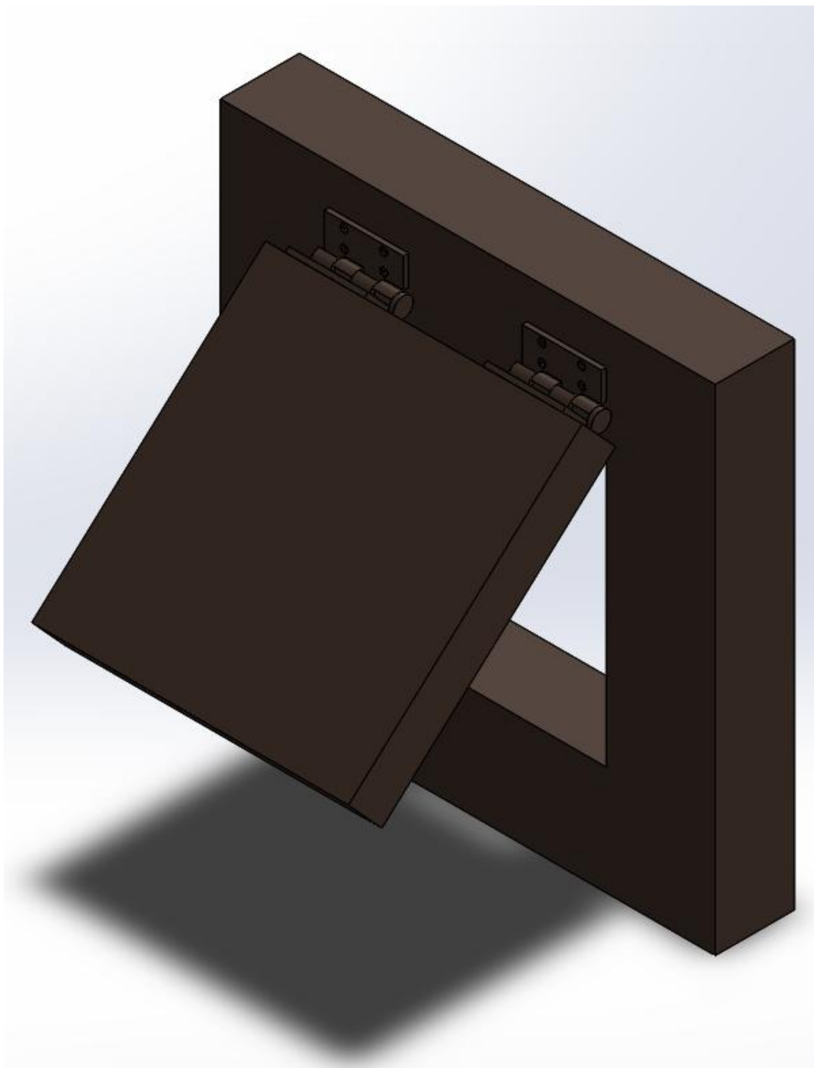

Figure 19. Hatch Assembly

HOMEWORK EXERCISE: PULLEY-SHAFT-KEY ASSEMBLY

(P6.4)

The pulley-shaft-key assembly, Figure 20, was made using the pulley from Figure 4 and then creating additional parts (the key and shaft). The key is a simple rectangle extrusion. The shaft is a pair of extruded circles from which a rectangle was cut using the Cut Extrude tool of the same size as the key. The pieces were then fitted together using concentric mates. This homework also highlights the Exploded View feature that allows the assembly to show the parts separately, as well as how they come together.

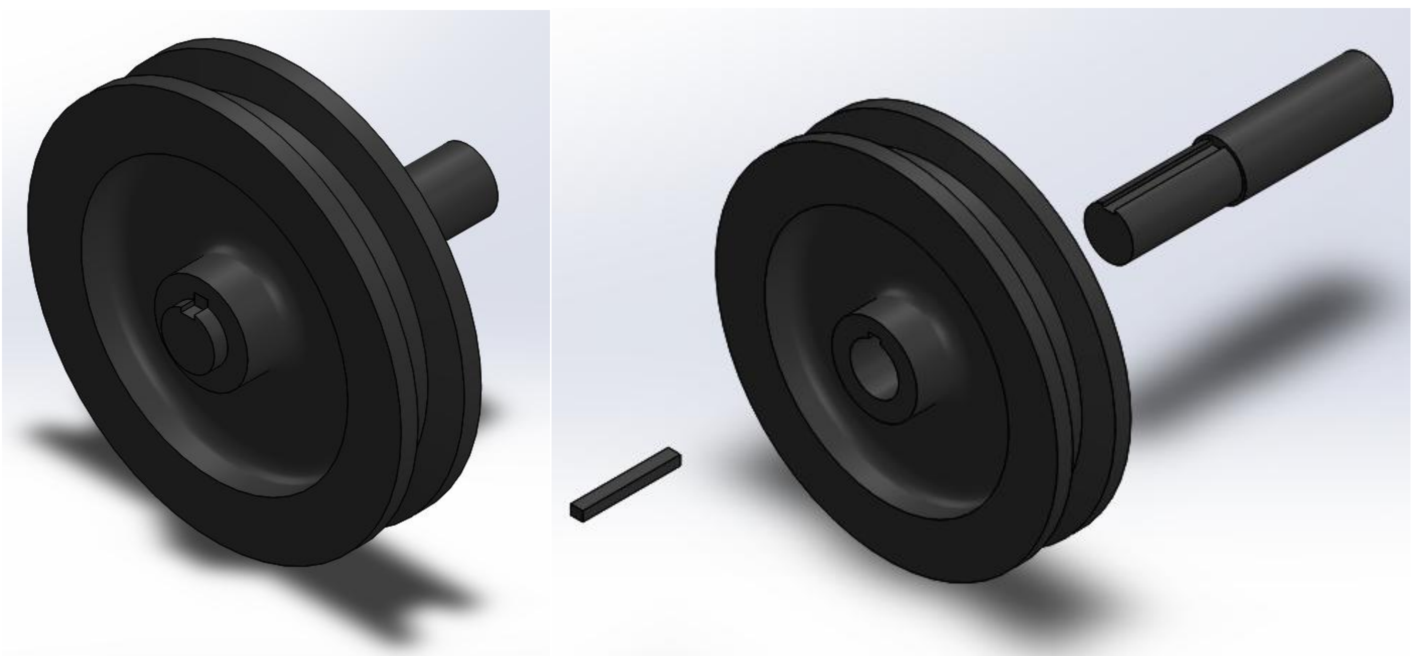

Figure 20. Pulley-Shaft-Key Assembly and Exploded View (*Right*)

IN-CLASS EXERCISE: ASSEMBLY DRAWING (*DOOR*)

(pg 239-247)

Figure 21, shows an engineering drawing of the door assembly in Figure 18. This exercise highlights an important aspect of assembly drawings: dimensions in assembly drawings only show how parts sit relative to one another and not dimensions of the parts themselves. Additionally, the assembly drawing provides special instructions for making parts fit together, such as drilling holes in one part to fit another into it. Finally, a Bill of Materials was added that lists the materials as well as their quantities and also provides a legend for the materials on the drawing using balloon annotations.

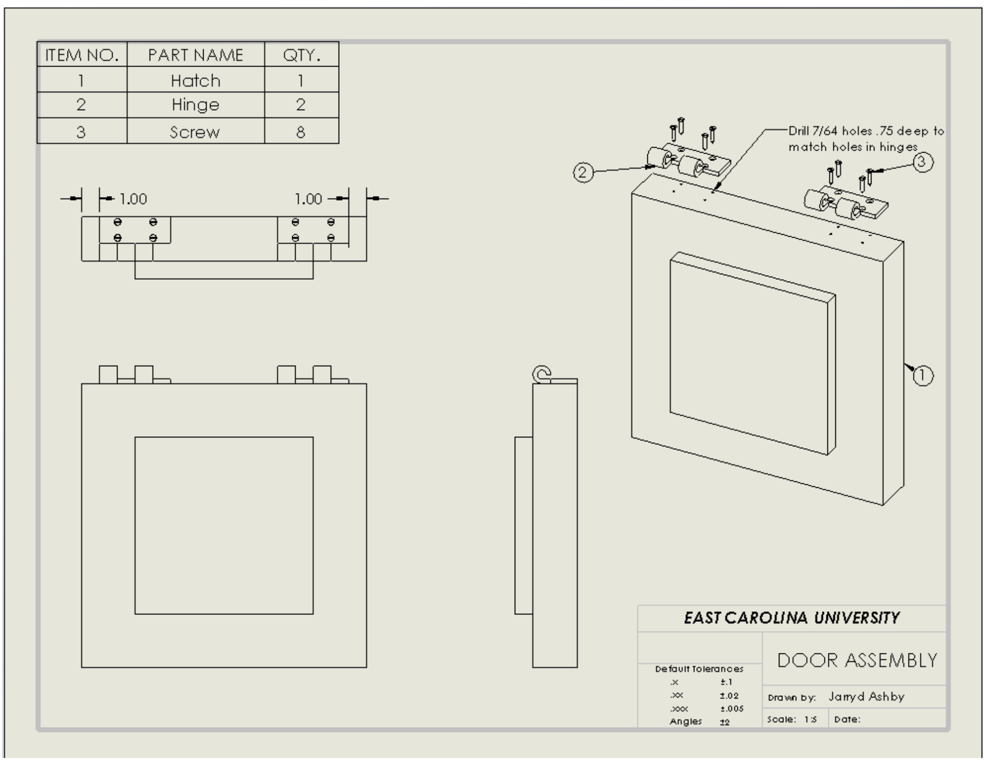

Figure 21. Drawing of Door Assembly (**Figure 18**)

HOMEWORK EXERCISE: ASSEMBLY DRAWING (*PULLEY-SHAFT-KEY*)

(P8.3)

The pulley-shaft-key assembly drawing homework exercise, Figure 22, reinforces the same principles shown in Figure 21. One thing it does make apparent that the previous does not, is the principle that three views are not always necessary to describe a part or assembly. This is especially true when the figure is circular like the pulley is, with the exception of the notch. The part would otherwise be completely shown, however, with only two of those views, since the right and top are the same.

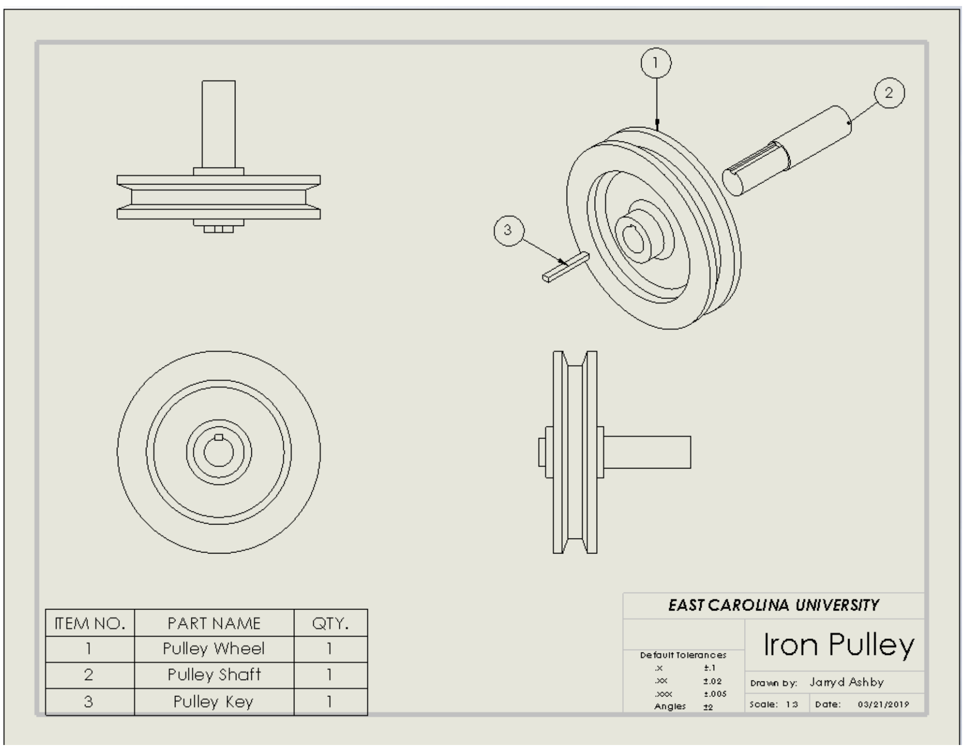

Figure 22. Drawing of Pulley (Error! Reference source not found.)

IN-CLASS EXERCISE: FLANGE WITH RIBS AND EQUATIONS

(pg 148-164)

The flange with ribs and equations, Figure 23, introduces parametric modeling techniques. The flange was made by extruding a pair of concentric circles, the smaller of which has a 3 degree draft. Then a hole was extrude cut from the top with a two degree draft such that the drafts both go out toward the larger base. Fillets were added to smooth the flange. Then the fillets at the top and bottom parts of the flange were selected an silhouette incident sketch arcs were added to them using the Convert Entities tool. A right triangle was drawn to these points and then a midpoint extrusion was made to make the first flange. The Trim Entities tool was then used to smooth the flange along with a rectangle and the Revolved Cut tool. These tasks were then patterned using the Circular Pattern tool. Circular holes were then added by using centerlines and a concentric construction circle and then cutting and patterning the circle around at the same intervals as the flange. Using the Equations tool. The angle of the ribs relative to one another, and the angle of the holes, and the number of holes, were all made relative to one another. For example, the equation is set such that the angle between each rib is 180 degrees divided by the number of ribs so that the ribs are equidistant.

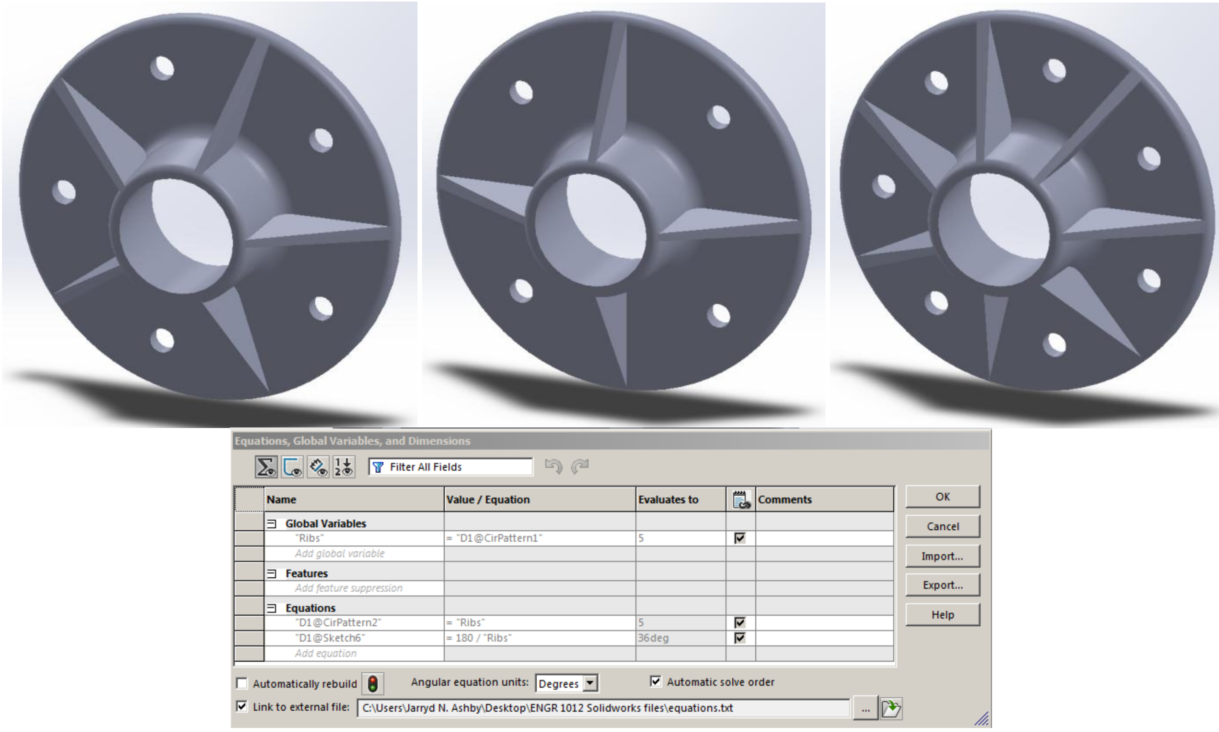

Figure 23. Ribbed Flange in (*L-R*) 5,4,7 ribbed configurations and Equations generating configurations (*below*)

HOMEWORK EXERCISE: CAP SCREW

(pg165-178)

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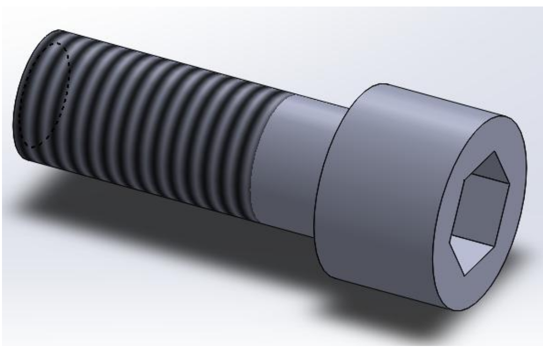

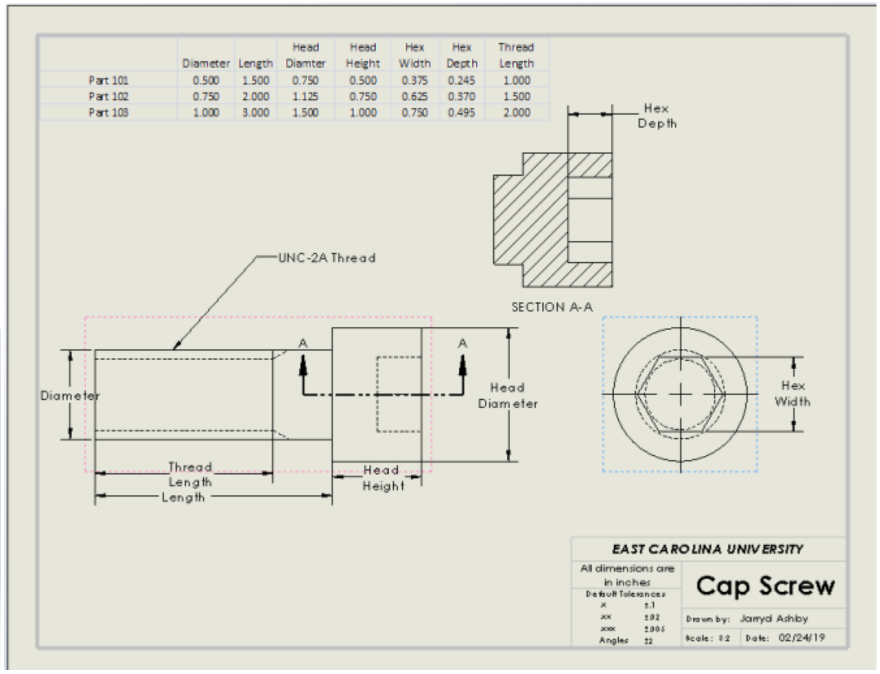

Figure 24. Cap Screw and Drawing (*Right*). Drawing used to show multiple configurations due to visual similarity of cap screw configurations

IN-CLASS EXERCISE: FOUR-BAR LINKAGE

(pg 295-312)

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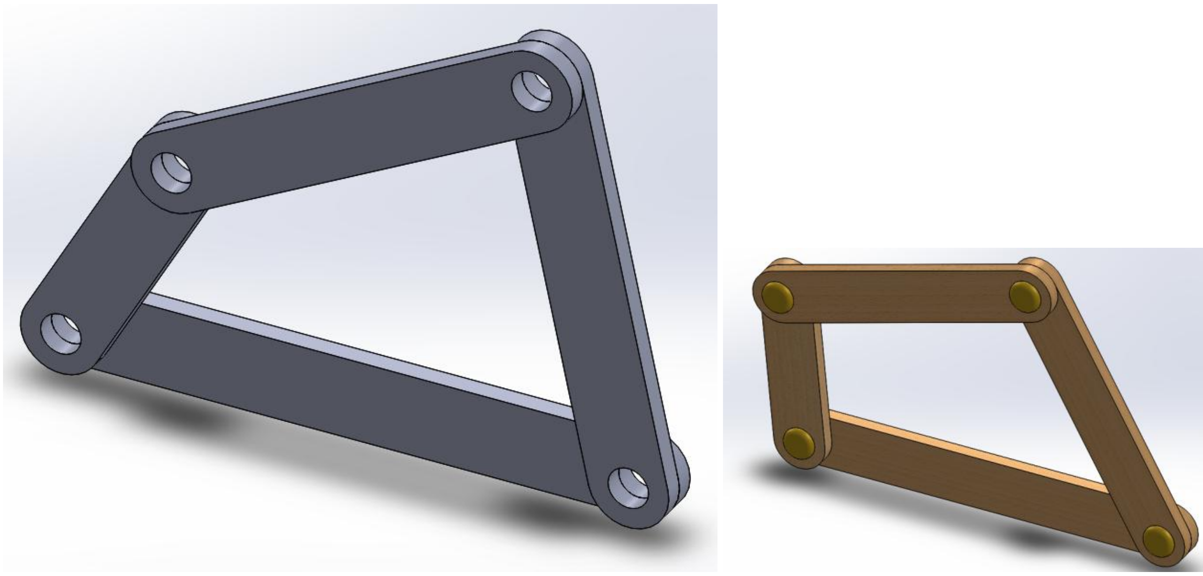

Figure 25. Four-Bar Linkage and Aesthetically Improved version (*Right*)