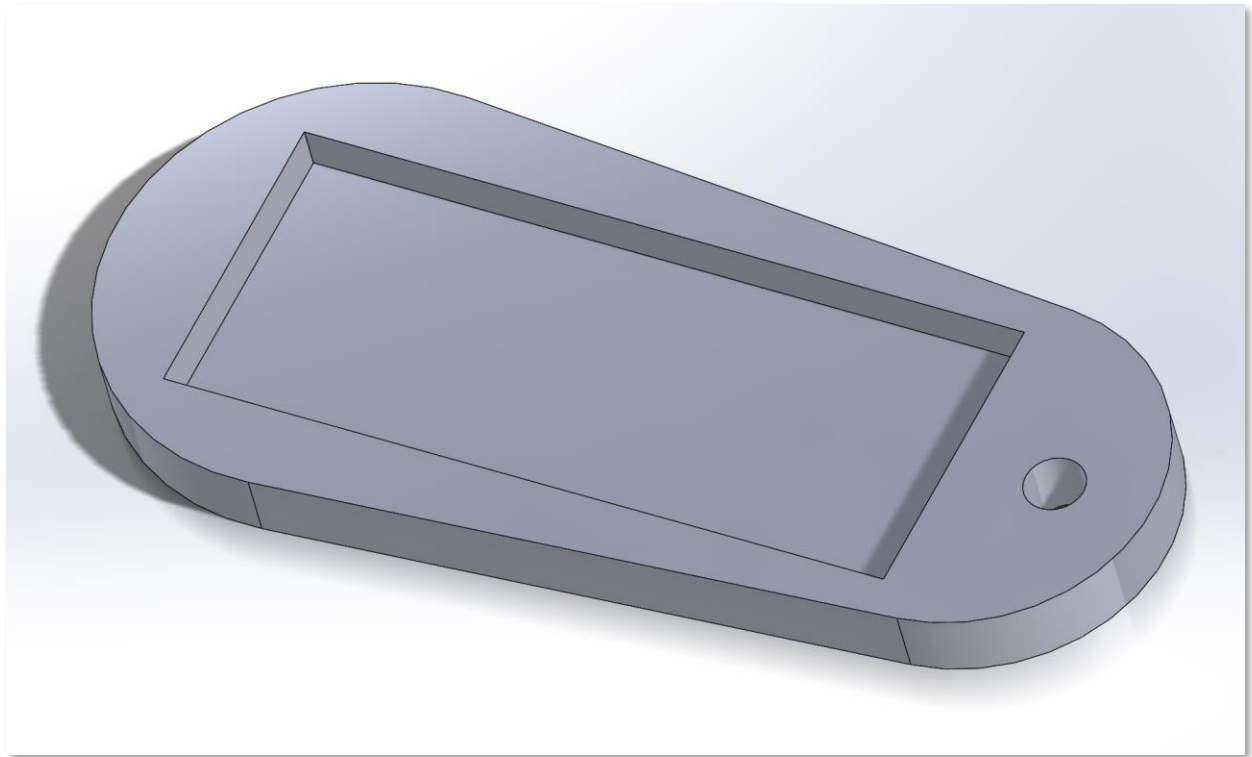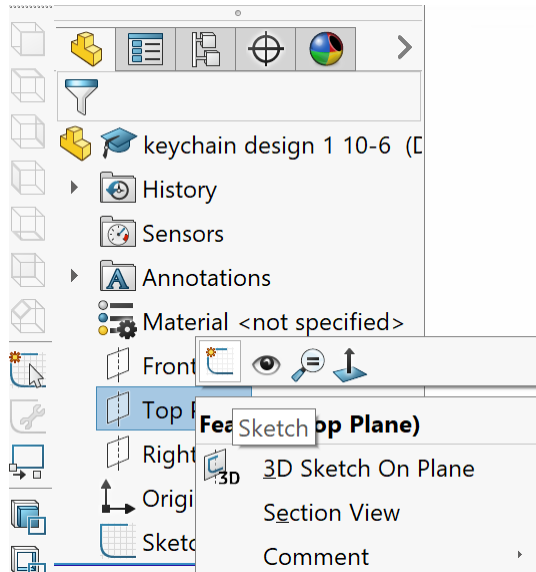# NJIT | Makerspace

Try Engineering Keychain Project CAD Instructions

1. Open up the Solidworks program on your laptop and start a new part
2. Start a new sketch on the top plane to create the profile of your keychain
   a. Right-click "Top Plane" and select the "Sketch" icon

3. Start by creating a **1.25" diameter** circle with its center at the origin
   a. Select the "circle drawing" icon from the sketch toolbar or "Command Manager" at the top of the screen, highlighted in red below
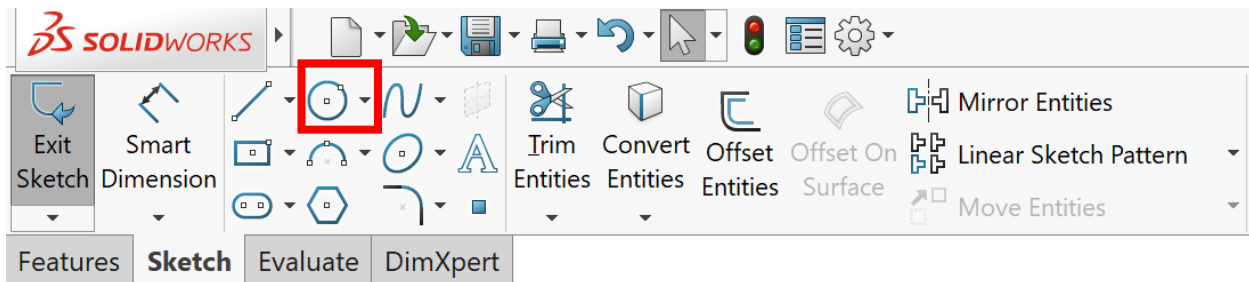
b. Select the origin as the center of the circle first by clicking on the origin, then select a point on the circle's circumference that will determine its diameter

First Point

Second Point

R = 1.38
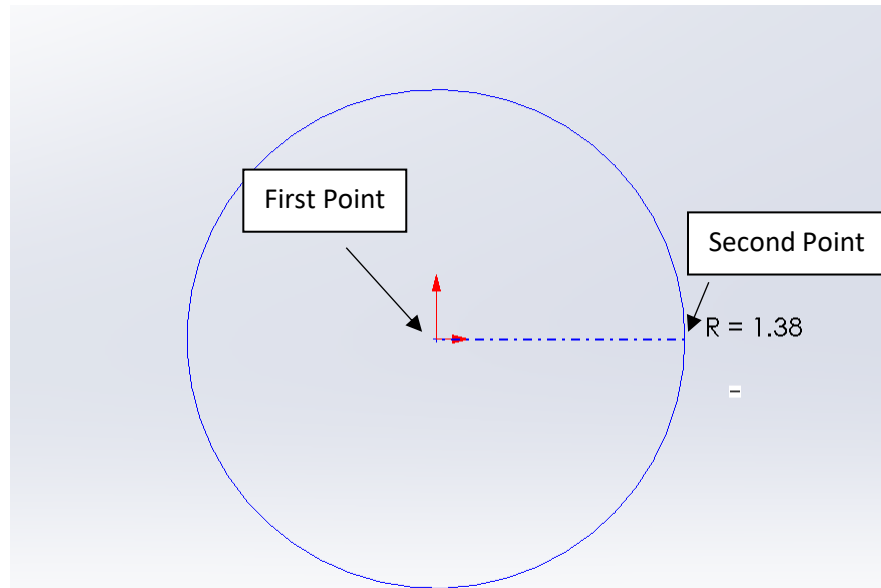
4. Use the "Smart Dimension" tool to set the diameter of the circle just drawn
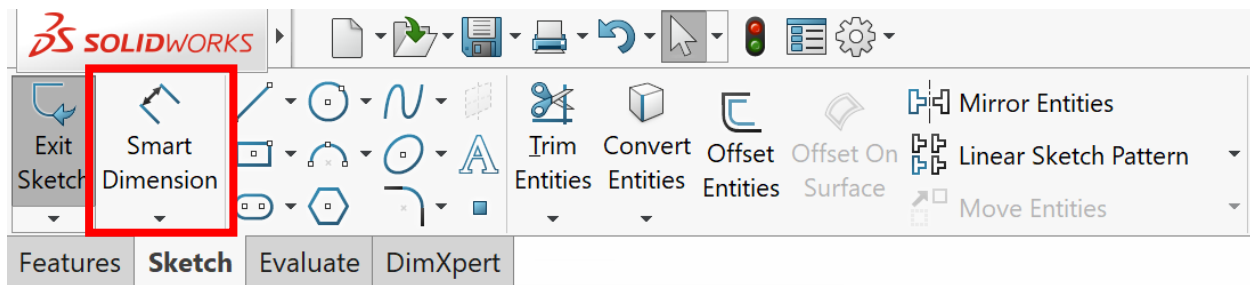
a. Click on the circle's circumference to select it for dimensioning
b. The second click will select where you want this dimension displayed

c. Finally, the third click will open up a text box to input the dimension value. In this case, we want this to be a **1.25"** diameter circle

5. Create a construction line to assist in creating the eventual length of the keychain
   a. Click the drop-down arrow to the right of the "line" tool on the sketch Command Manager and select "centerline"

   b. Select the origin in the sketch plane as the start of the centerline, and click somewhere to the right of that point to set the end-point
   i. Pressing the "escape" key exits this tool

c. Dimension the centerline using the dimensioning method we used for the circle, and set this line length as **1.75"**
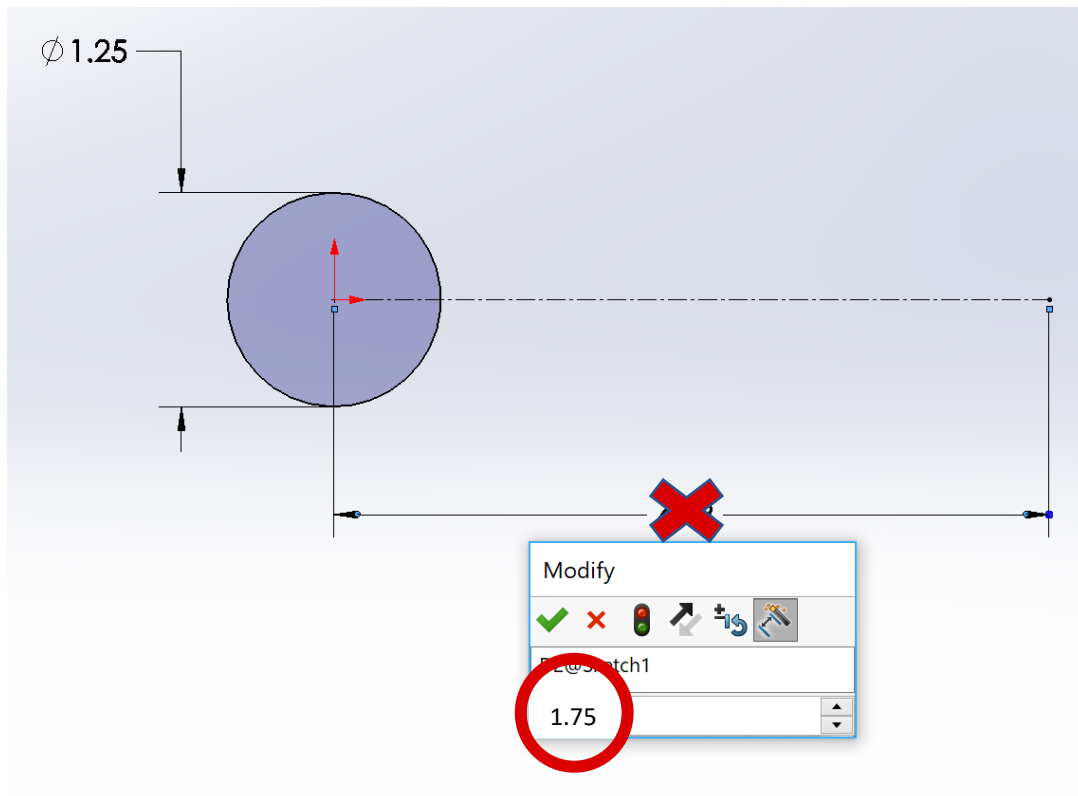
6. Using the end-point of the construction line just created, draw a second circle with a diameter of **1.0".** Its center should be at the endpoint of the construction line just drawn.
   a. (Essentially, repeat steps 3 & 4)
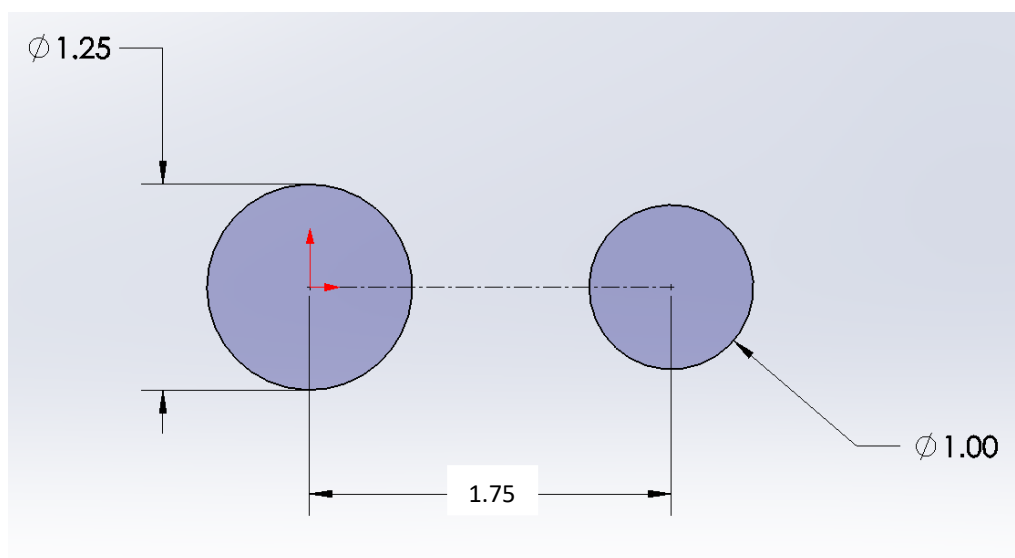
7. We will now make the outer boundary of the keychain by creating a line tangent to both circles. Then we will mirror that line
   a. First, select the line tool from the sketch Command Manager
   b. Click somewhere on the circumference of the left circle to set the first point of the line to be coincident with the left circle
      i. Be sure that the start point of this line **does not** snap into place at 12, 3, 6 or 9 o'clock on the circle!
   c. Click somewhere on the circumference of the right circle to set the second/end point of the line to be coincident with the right circle
      i. Be sure that the start point of this line **does not** snap into place at 12, 3, 6 or 9 o'clock on the circle!
   d. Press "esc" on the keyboard to exit the line tool
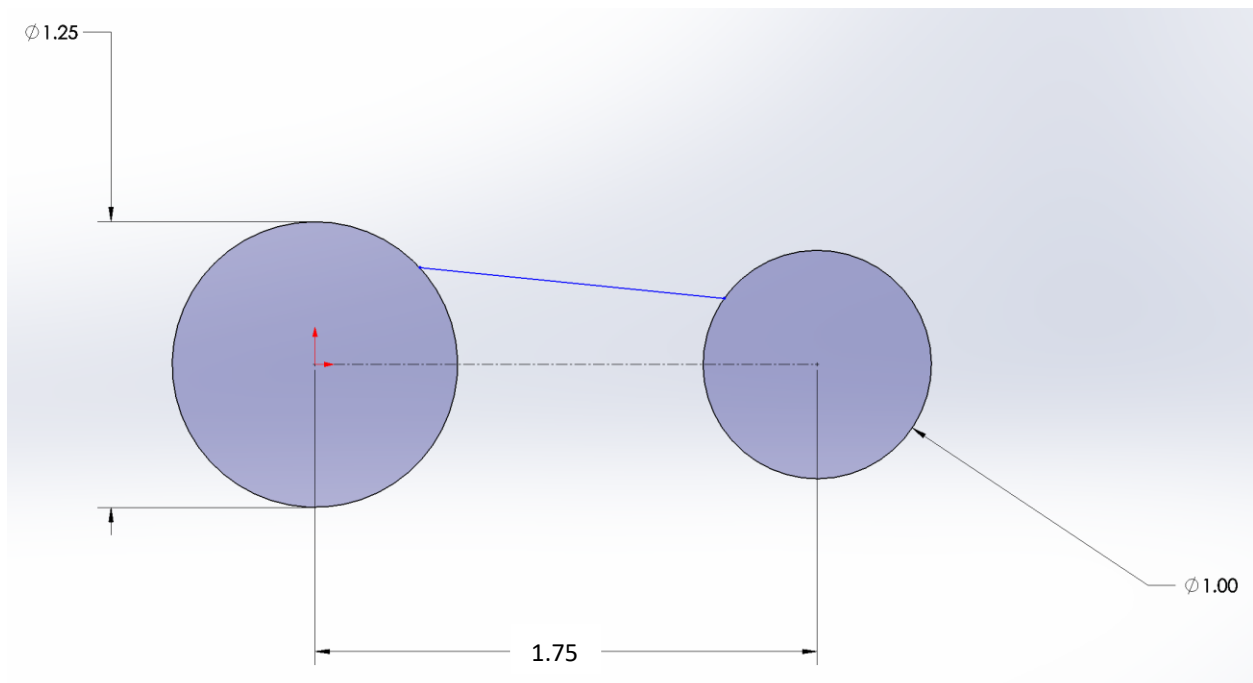
   e. Hold the "control" key on your keyboard and select the line just drawn **and** the left circle. This will open the "Properties" pane on the left side of the screen. Here, we can add "constraints" to our drawing

f. In this case, we want this line to be tangent to the left circle's circumference, so select the "Tangent" relation
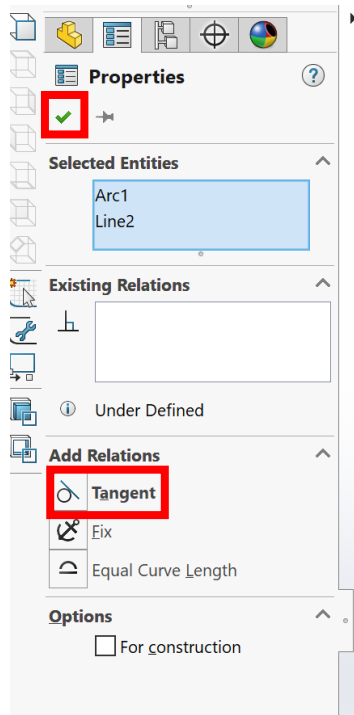

g. Click the green checkmark at the top of this pane to complete setting this constraint/relation
h. Repeat steps e,f,g to ensure that this same line is also tangent to the smaller circle on the right
i. Now that this line is properly constrained, we will mirror it to the bottom half of the sketch
j. Click the "Mirror Entities" option from the sketch Command Manager
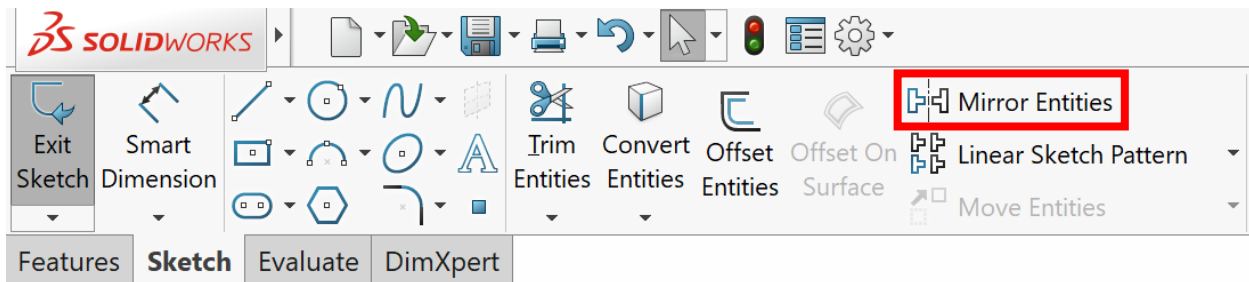
k. First select the line we would like to mirror (the line we just constrained) to mark it as an entity to mirror

l. In the "mirror" pane on the left of the screen, make sure that the "copy" box is checked, then select the white box under "Mirror about:"
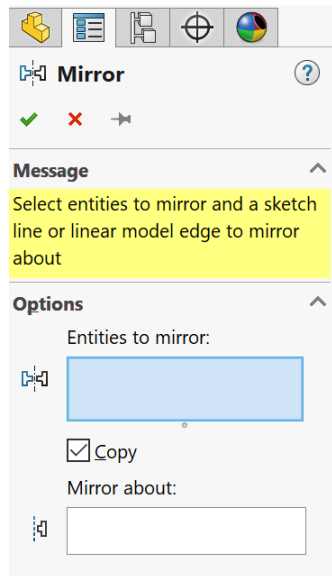

m. Select the horizontal center/construction line we created earlier as the line to mirror about. You should see a mirrored entity appear in yellow

n. Click the green check mark to complete the mirroring process

8. Use the "Trim Entities" tool to remove lines from the interior of the closed sketch we have just created

a. Select the "Trim Entities" tool from the sketch Command Manager
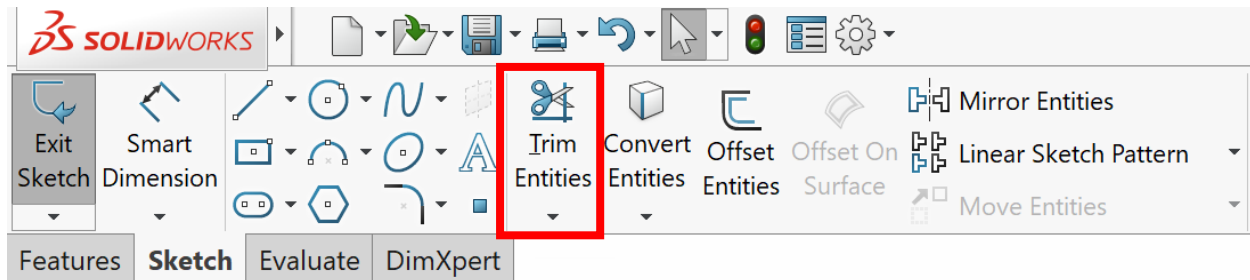
b. With this tool selected, click and drag across the line segment you would like to remove. This starts a trimming line that will eliminate lines between two points. Use this to eliminate the parts of the circumferences that are "inside" our closed sketch so it looks like this:
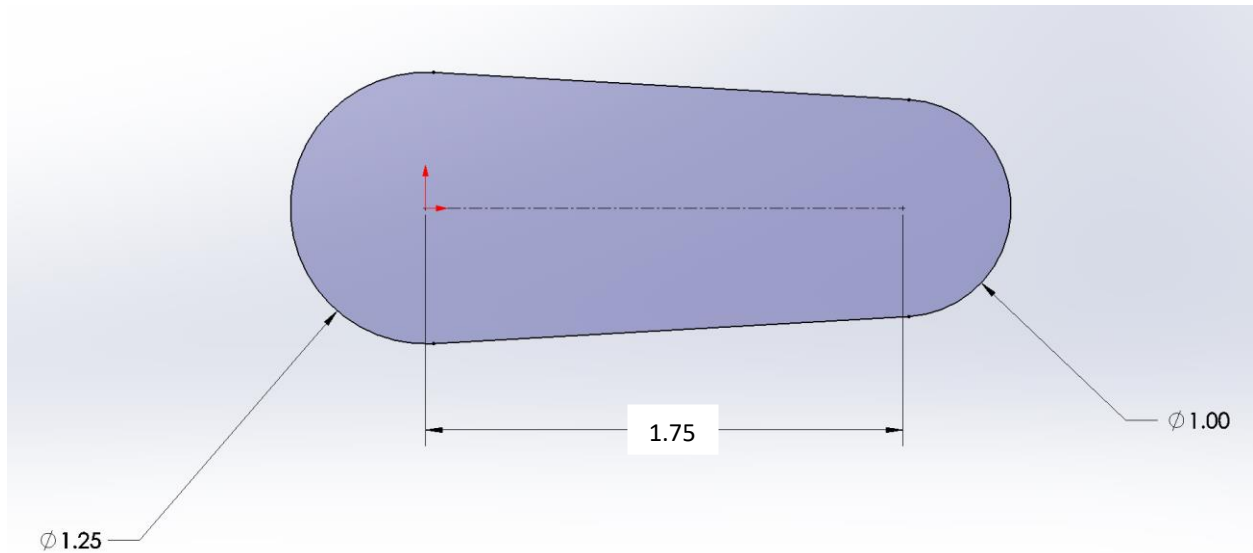
9. Finally, we will create a hole for our keychain in this sketch using the tools previously outlined. Make the diameter of this hole **5/32"**

   a. Make sure the center of this hole is **0.23"** from the center of the right-most arc on the keychain
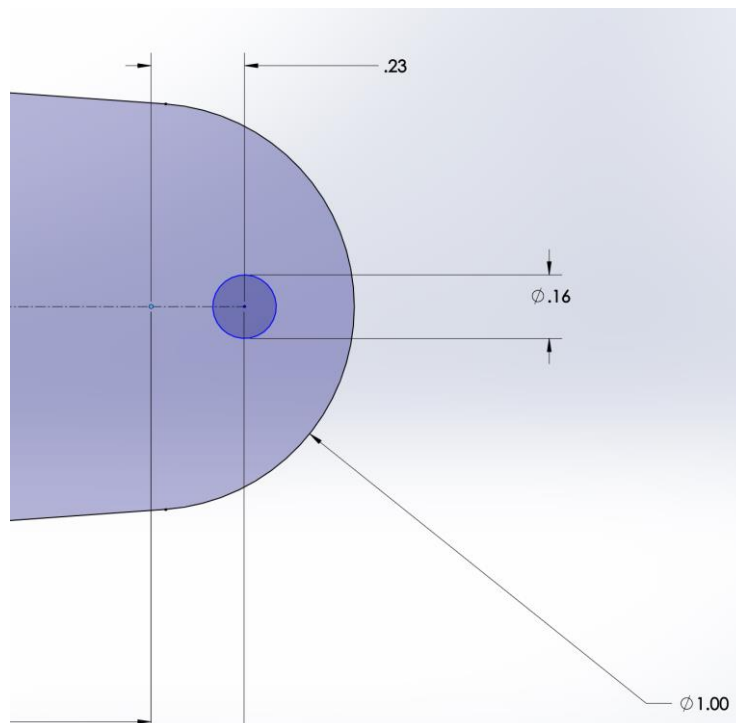
10. Click "Exit Sketch" to return to our 3D model space
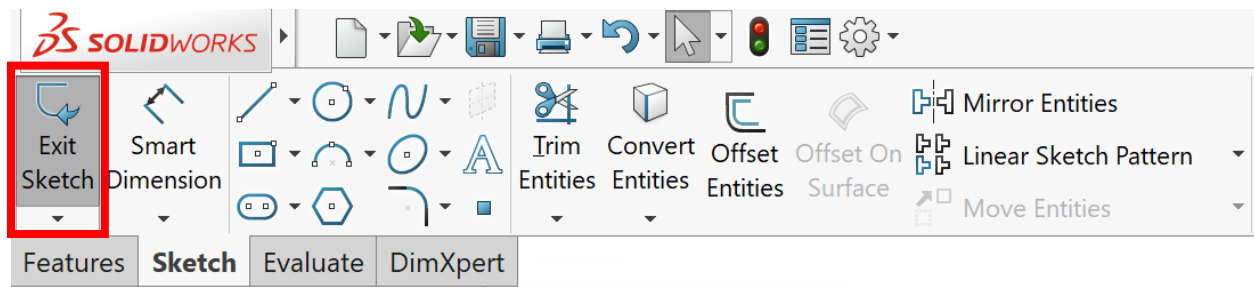
11. We will extrude the created 2D sketch to create the basis of our solid 3D model of our keychain
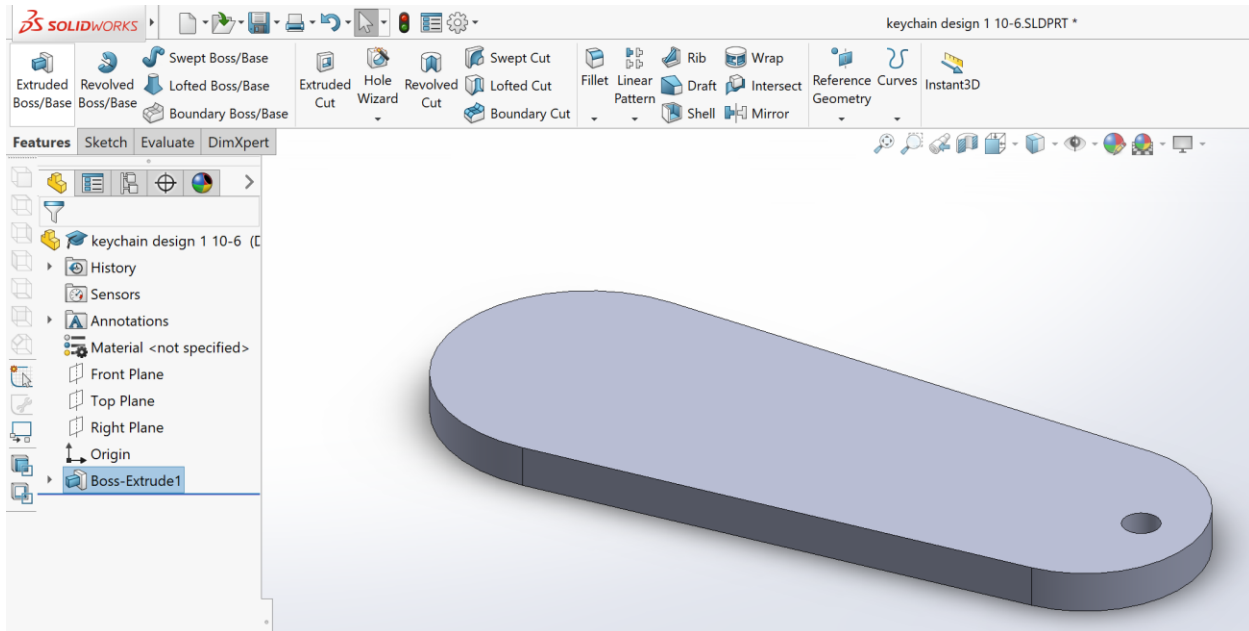   a. Select the previously created sketch from the Feature Manager in the left pane and click "Extruded Boss/Base" from the "Features" Command Manager
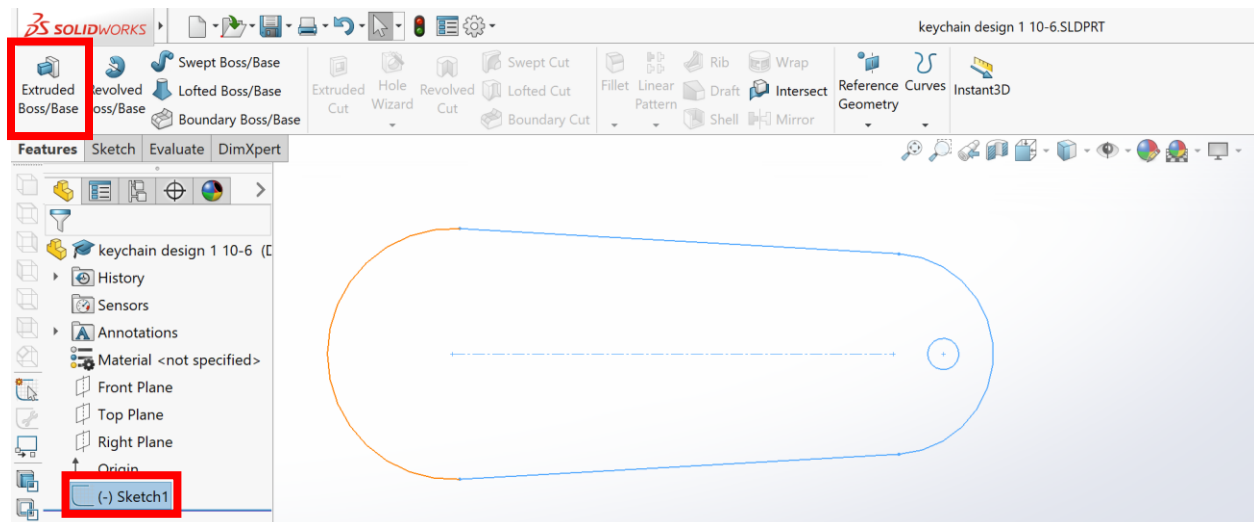

    b. The view in the graphics area will change to show a preview of extruded base
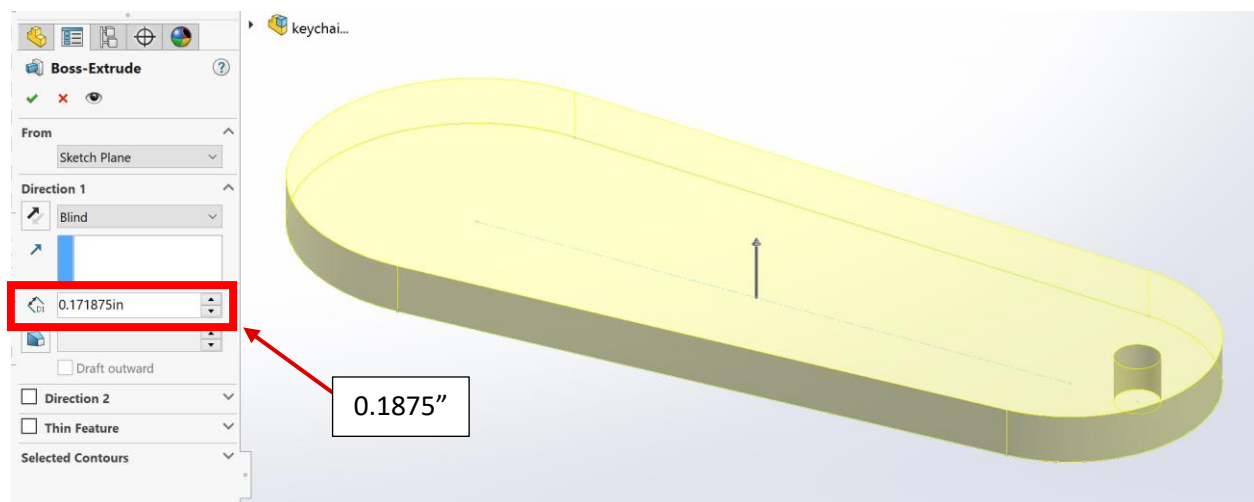
0.1875"

c. Make sure that the thickness of the extrusion is set as **3/16"** or **0.1875"** then click the green check mark


12. To complete the keychain, we will create a pocket for our CO_2 laser cut and engraved name plate using the "Extruded Cut" command

a. First, right-click the top face of the extruded base and select "Sketch" in the pop-up window. This will begin a new 2D sketch on the top face of the keychain base
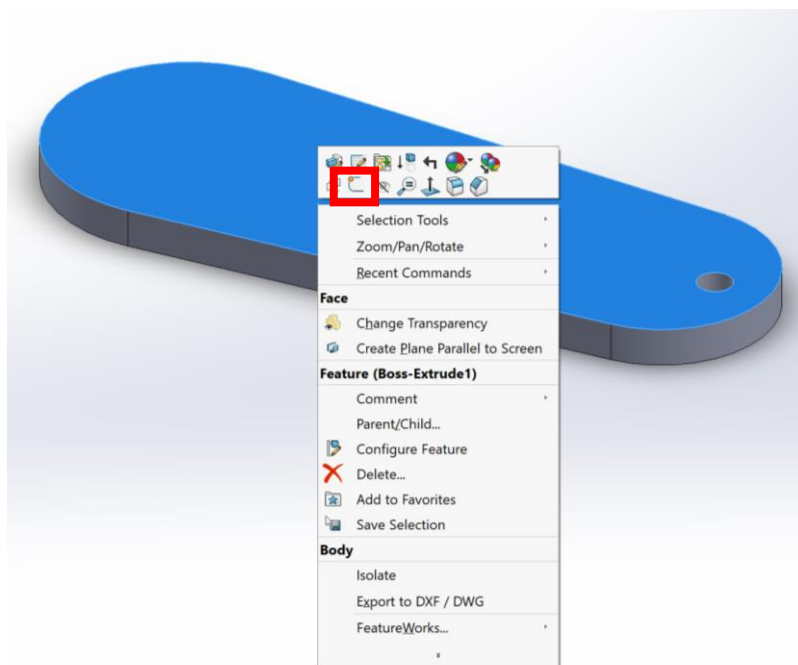

b. Press **ctrl+8** to ensure the sketch view is normal to the sketching surface

c. Start this sketch by drawing a horizontal construction line ("centerline") that passes through the x/y origin of the sketch

d. Use the drop-down arrow to the right of the "rectangle" tool to select the "corner rectangle" option
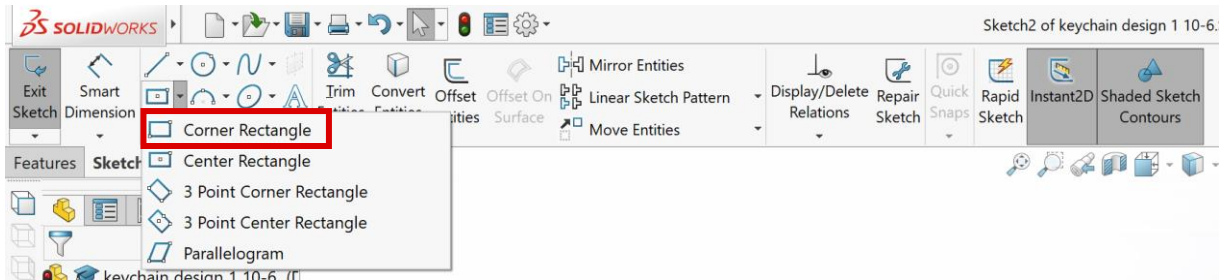

e. Draw a rectangle anywhere on the face of the keychain. We will constrain and dimension this rectangle to ensure it is centered
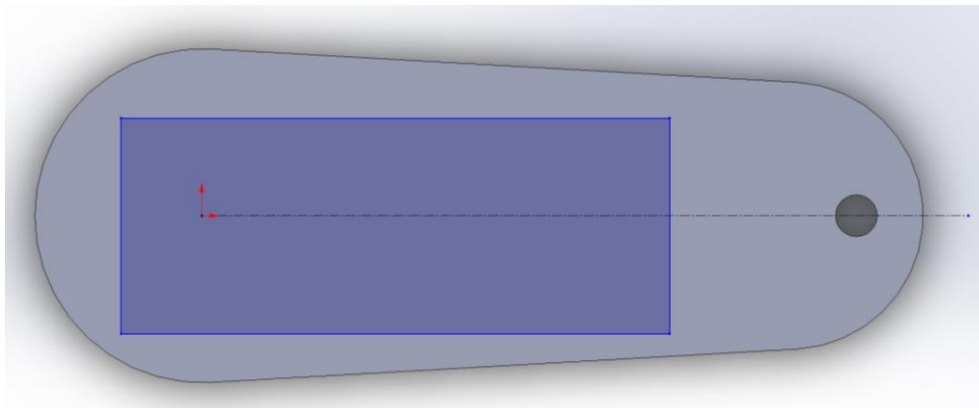

f. Hover your mouse over the line that forms the right side of the rectangle until a **yellow/orange square** appears. This square designates the midpoint of that line

g. Click that midpoint marker and **ctrl+click** the construction line to constrain these two features to be **coincident**
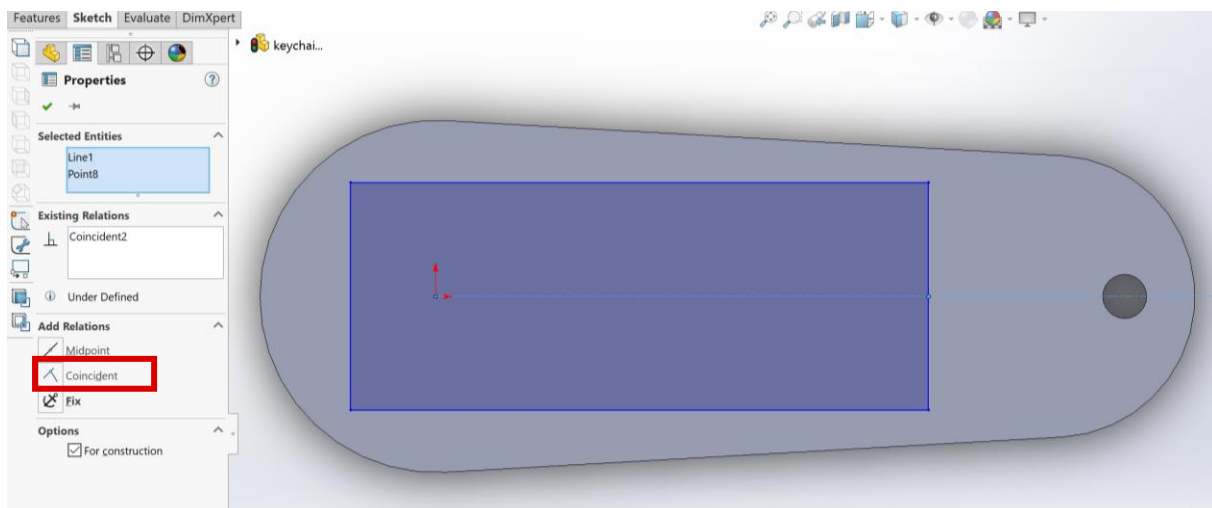
h. Dimension the rectangle to have a height of **0.8"** and a length of **2.0"**
i. Add an additional dimension to ensure that the ride side of the rectangle is **0.28"** from the center of the keychain hole. To do this, click the right side of the rectangle, click the keychain circle, then enter a dimension value
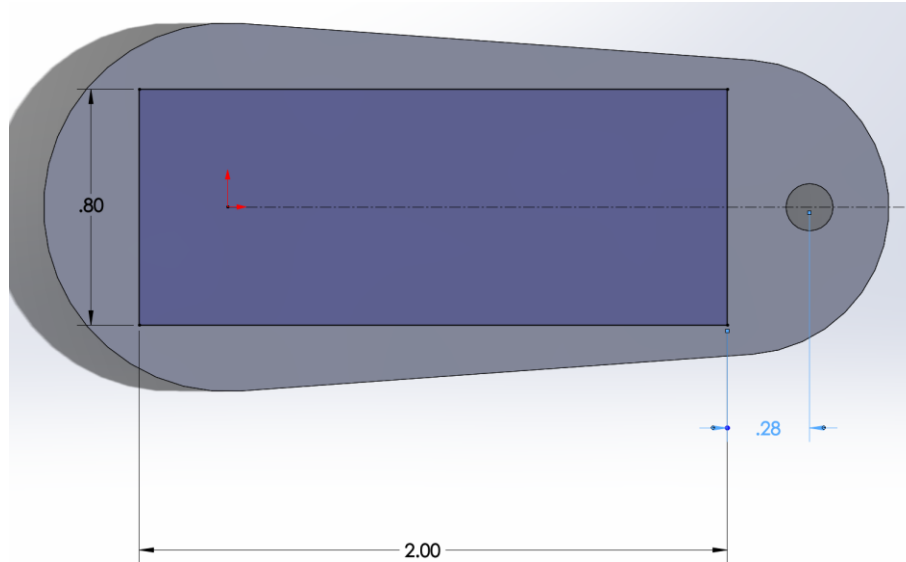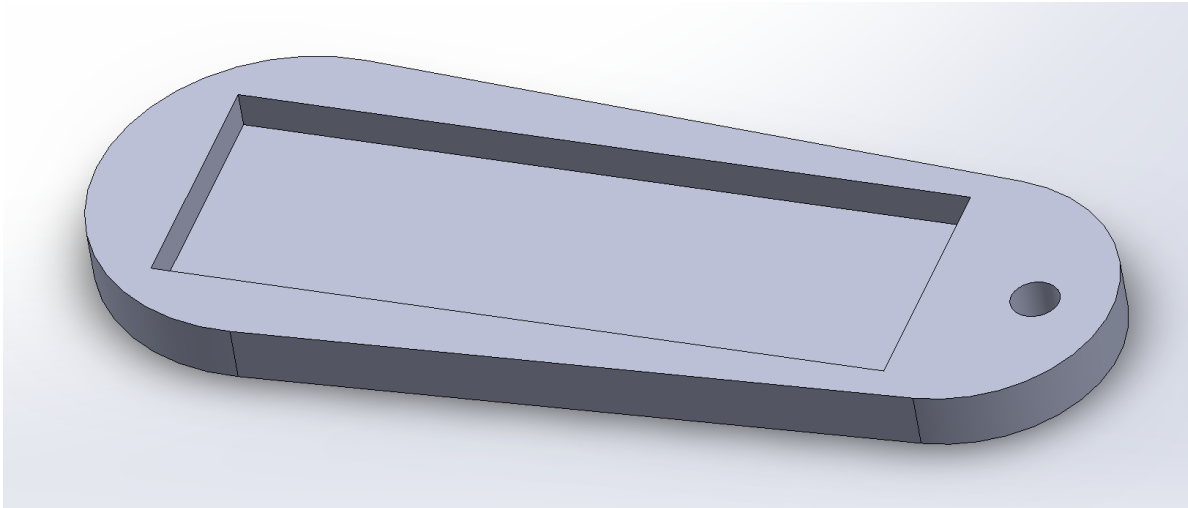

j. Exit the sketch
k. Select this sketch ("Sketch 2") from the Feature Manager and select "**Extruded Cut**" from the Command Manager
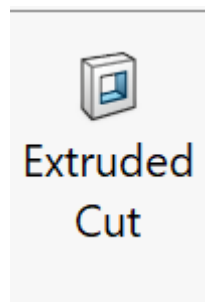
l. Extrude this cut to a depth of **1/8" or 0.125"**

m. Click the check mark. The 3D keychain model is now complete!

13. In order to 3D print this model, we will need to export the file as a .STL
   a. Click "File" at the top menu bar and select "Save As". This will open the save dialogue box
   b. In the "Save as file type:" drop-down menu, select **"STL"**
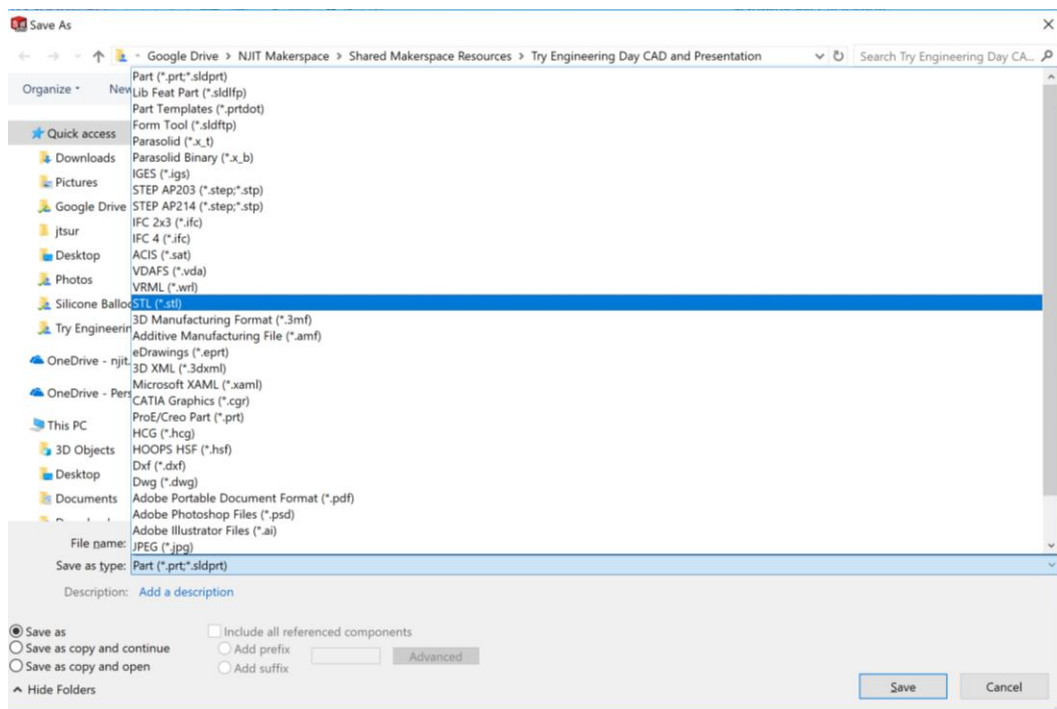
   c. Name your file and click "Save". Solidworks will ask you if this is okay, and it will ask what to export from the model. Make sure "All bodies" is selected and click "OK"