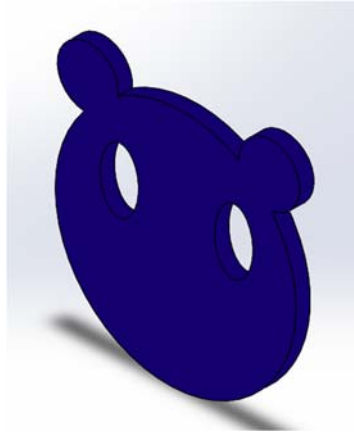# SolidWorks Intro Part 2

Dave Touretzky and Susan Finger

1. Symmetry and Reference Geometry

Today, you'll make this part and then cut it on the laser cutter:

Open SolidWorks and a create a new part using your IPS part template. Be sure you are working on the front plane and in the front view. Refer to the beginning of Lab 1b if you are having trouble remembering how to do this.

We are going to make an extrusion using what I think of as a bookend command. We will start the Extruded Base command, go into Sketch mode and sketch the cross section of the extrusion. When we exit Sketch Mode, the cross section will be extruded and the Extrude Base command will end.

From the front view, start the **Extruded Boss/Base** command which is in the Features tab. SolidWorks will switch you to Sketch mode. Make a circle centered on the origin with a radius of roughly 1 inch.
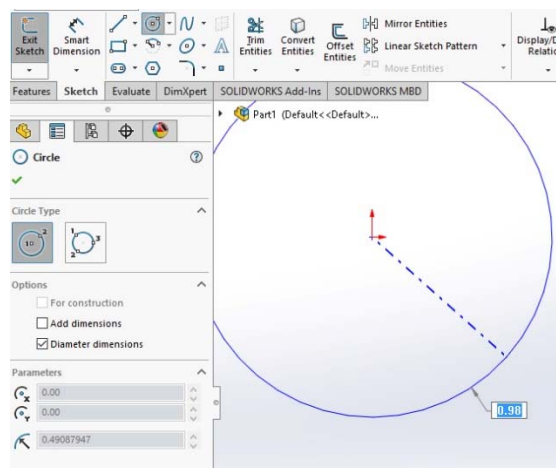
Click **Exit Sketch**. The left-hand panel now has the options for the Extrude command. Set extrude depth to 1/8 in, and click OK.
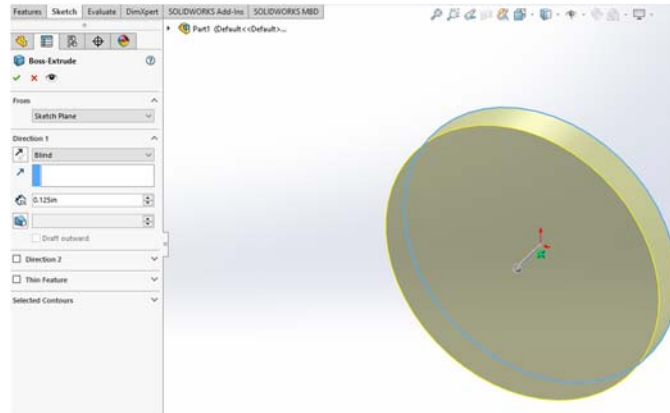
This completes the **Extruded Boss/Base** command. If you look in the Feature tree, you will see the Boss-Extrude feature based on Sketch 1.
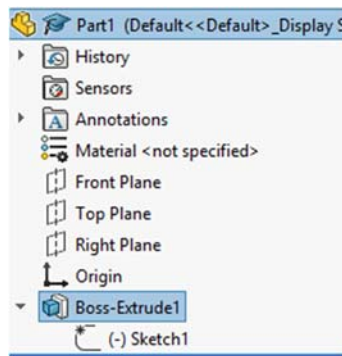

Now we want to make the ears. Be sure you are in **Front View** and then select the **Front Plane** from the part feature tree. You want to be sure your second sketch is on the same plane as the first sketch so the extrusions will match. In the screen shot below, I have done the two steps above and then rotated the part so you can see that the Front Plane is the back of the part.
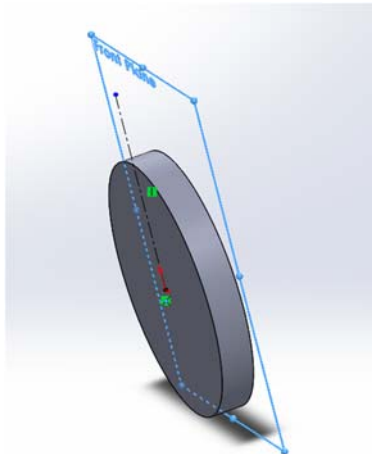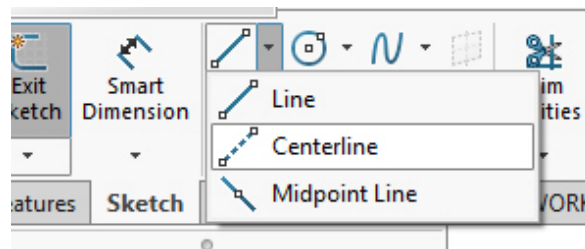
Be sure you are in the **Front View** and start the **Extrude Boss/Base** command again, which again puts you into Sketch mode.
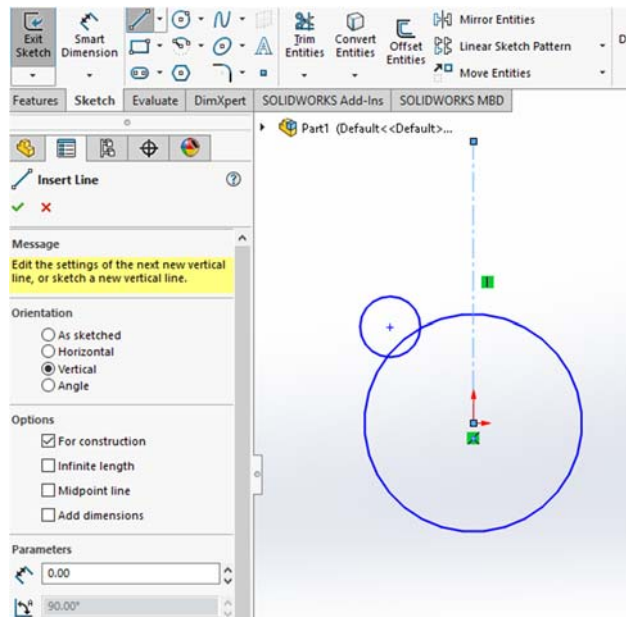
Make smaller circle above and to the left of the big one, radius roughly 0.27 inches, that intersects the somewhere in the top left quadrant.
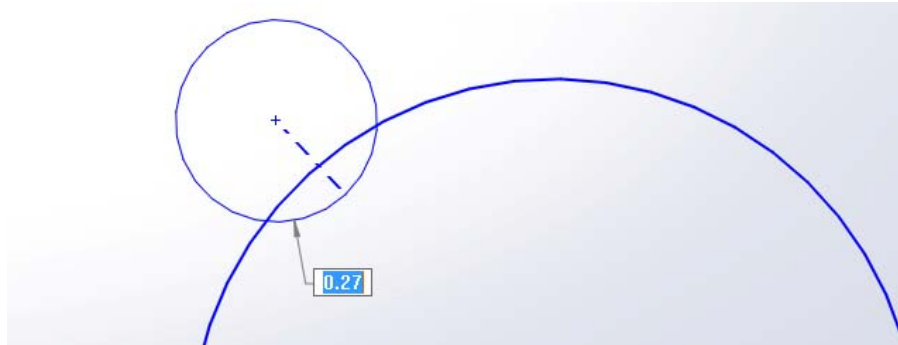

From the **Line** tool, select **Centerline**.

Draw a centerline straight up from the origin for about 1.5 inches. Check the **Vertical** Orientation in the left-hand panel to be sure your line goes straight up.
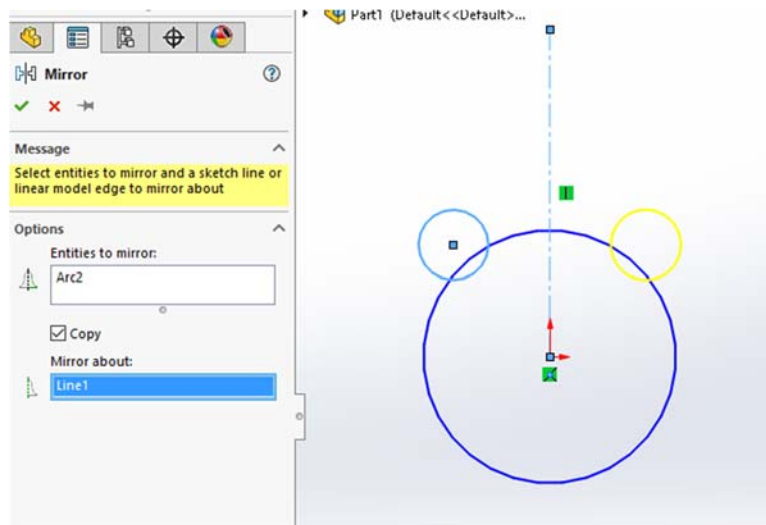
Notice the line property: **For construction** is checked; it is indicated visually as a dashed line. Click the green checkmark or press Escape to exit.

Select the **Mirror Entities** tool from the sketch toolbar.
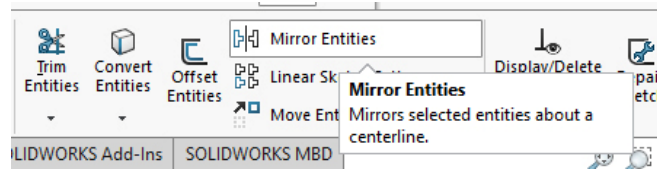
Select the small circle for **Entities to Mirror**. Then click in the **Mirror About** box and select the centerline.

Click OK to complete the mirror.

Try clicking on either circle to change its size. Click and drag on the center of either small circle to change position. This shows you how the circles are logically linked.

Once your circles are about the right size, click OK to accept the sketch, and set the extrusion depth to 1/8 in. Look at your part in a perspective view to see what it looks like so far:
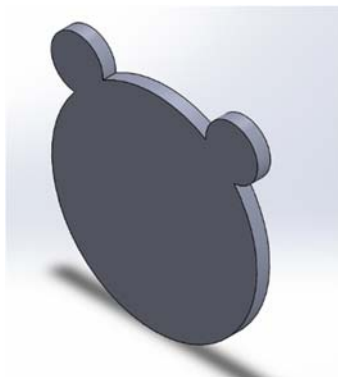
Go back to the front view. Now, we will make the eyes. Since they are cut out from the part, we'll use **Extruded Cut**. We will also draw on the top face and cut into the part

1. Select the **ellipse** from the drawing tools.
2. Click about where the center of one of the eyes would be.
3. Move the mouse down and click to set the length of the eye.
4. Move the mouse to the right to adjust the width of the eye
5. Add a centerline again. Remember to check the Vertical box.
6. Mirror the ellipse
7. Add a centerline as before.
8. Mirror the ellipse.
9. Click and drag on the bottom point of one of the ellipses to tilt the eyes.
10. Exit Sketch.
11. In the Cut Extrude dialog, change Blind to Through All.
12. Click OK.

Save your part as andrewid_Mickey.SLDPRT.

Make a SolidWorks drawing of your part and save it as andrewid_Mickey.DXF. Open your file in Inkscape and set the colors for the laser cutter. When you are done send save the part on a flash drive or email it to yourself. You will cut this part yourself on the laser cutter.