

SolidWorks Example

This example demonstrates the use of SolidWorks. We will draw a 2-D plane stress structure (a steel chain link). A schematic of the steel chain link is shown below in Figure 1.

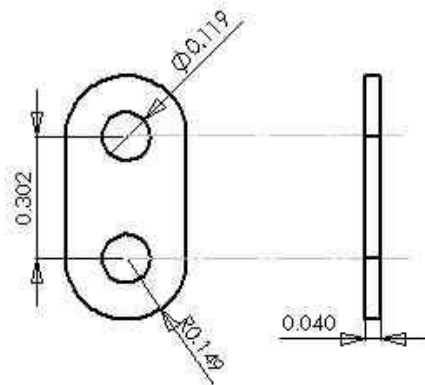
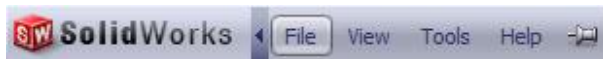


Figure1. Schematic drawing of chain link. All dimensions are in inches.

Open SolidWorks by selecting **Start**→**All Apps**→**S**→**SOLIDWORKS 2016**→**SOLIDWORKS 2016x64 Edition**. After it loads, select



File→**New...** Choose what units you would like to work with, then select **Part** and **OK**. By default in the ESIG Labs, the units are set to millimeters. Another way to change the units is to select **Tools**→**Options....** Under the **Document Properties** Tab, select **Units** (from the list on the left) and change the Unit system to **IPS** because we are working in inches. Also, change the Decimal Places to **.123** in all available places. Select **OK**.

Sketching the Part

Click on **Front** in the FeatureManager design tree (see Figure 2). The plane should now be highlighted in blue in the drawing window. Open a sketch on this plane by selecting **Insert**→**Sketch**. (You can also open a sketch by using the Sketch tool Under the **Sketch** tab on the left hand side tool bar. See Figure 3.)

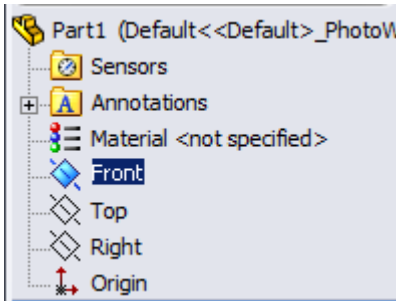
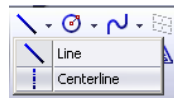


Figure 2. Select Front Plane in the FeatureManager design tree.



Figure 3. Use the Sketch tool to open a sketch on a pre-selected plane.

After you open the sketch, notice the sketch tools that are now available on the tool bar. Select **Circle** and place the mouse exactly over the origin. The pointer will turn red when it is exactly over the origin. Click and drag to draw a circle. The size is not important at this time



Select the **Centerline** tool, from the toolbar and draw a vertical centerline that extends through and beyond the origin. (Use the dotted blue inference lines to insure you are exactly under the origin.) A “90°” will appear next to the tool when you are drawing a vertical line (see Figure 4).

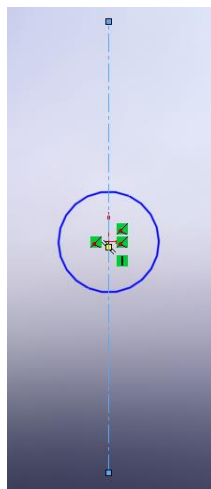
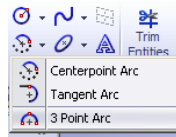


Figure 4. Draw a vertical centerline through the origin.

Draw a second circle and place its center exactly on the vertical centerline, but not precisely at the centerline's end. The exact position on the centerline and size is not important. Make sure the centerline extends some distance from the circle ends. Now draw a horizontal centerline from the center of each circle to an arbitrary point outside of the circle. (You are exactly on the center point of the circle when a red circle appears at the center of the circle.) An "180°" will appear next to the tool when you are drawing a horizontal line.



Select the **3 Point Arc** tool, and sketch a $\frac{1}{4}$ arc between the vertical centerline and the lower horizontal centerline by following these steps:

1. Place the tool exactly on the vertical centerline
2. Click and drag to the lower horizontal line. The arc might appear backwards at this point.
3. Release the mouse button and adjust the arc until it is approximately the desired curvilinear shape. Avoid all inference lines.

Repeat this process for the upper arc.

Select the line tool and draw a line that connects the ends of both arcs. A red circle will appear below your tool when you are exactly on the arc end point (see Figure 5). The line doesn't necessarily have to be perfectly vertical at this time.

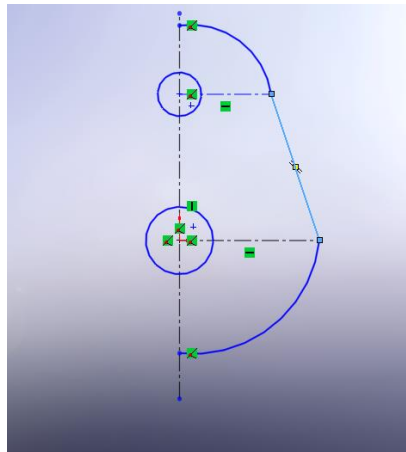
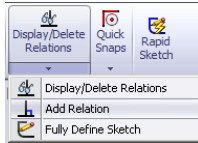




Figure 5. Connect the two $\frac{1}{4}$ arcs with a line. The dotted lines are for reference only.

Adding Relations and Dimensions

We add relations to constrain the drawing as well as simplify the dimensions. Switch to the **Arrow** tool. With the **Ctrl** button held down, select the **upper** and **lower** $\frac{1}{4}$ arcs.




Select the **Add Relation** tool, . Then select the **upper** and **lower** ¼ arcs, Arc 3 and Arc 4 should now be in the Selected Entities field. You can add or remove items from this field by clicking on the object in the drawing window. Select **Equal**, and press .

Add these other relations: (The new objects with relations can be selected with the ADD Geometric Relations dialog box open.)

- Add a **Concentric** relation with the **lower** ¼ arc and the **lower circle**.
- Add a **Concentric** relation with the **upper** ¼ arc and the **upper circle**.
- Add an **Equal** relation with the **upper** and **lower circle**.



Select the **Dimension** tool, , and click one of the circles. The dimension will be displayed, click to place the dimension, and it will ask you if you want to change the dimension. Dimension the circle to **0.119** inches (diameter). Both circles may disappear. We will zoom in at a later time.

Now dimension one of the ¼ arcs to **0.149** inches (radius).

Again, select the **Dimension** tool and click on one of the circle centers. Now click on the other center and place the dimension. Dimension this distance to **0.302** inches. To zoom in on the sketch, use the plain magnifying glass on the tool bar or click anywhere in the drawing window (with no tool selected) and then use the center wheel on your mouse.



The **Arrow** tool alone will allow zooming with the center mouse wheel. However, you may need to click once on the screen.

At this point, the sketch should be fully defined. If it is not, go back and make sure all the steps are correct. You can view relations by using the **Display/Delete Relations** tool.

Mirroring


We will take advantage of the symmetry of this part and mirror it about the vertical centerline. Hold down the Ctrl key and select the both ¼ arcs, the vertical line, and the vertical centerline. *With the Ctrl key held down*, select Tools→Sketch Tools→Mirror. It will automatically create the mirror about the vertical centerline. The part is mirrored about the center and is still fully defined.

Extruding

Select the **Extruded Boss/Base** tool from the **Features** tool bar, (see Figure 7).



Figure 7. Extruded Boss/Base tool in the Features tool bar.

Set the type to **Blind**, and the Depth to **0.040** inches and press .

The part is now complete (see Figure 8).

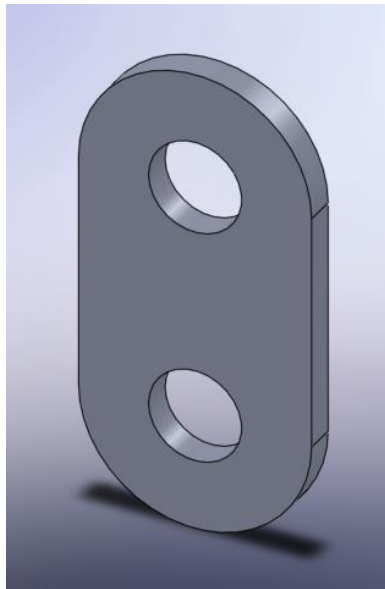


Figure 8. Completed chain link in SolidWorks.