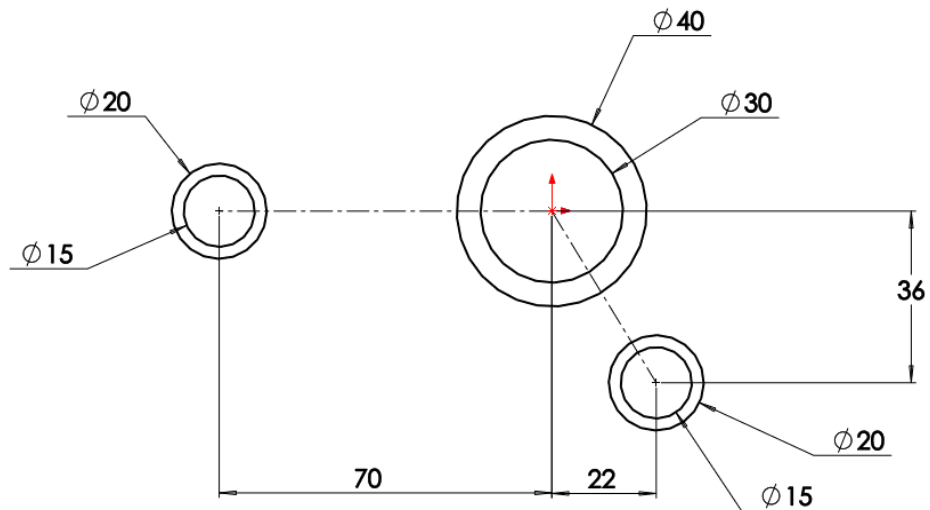# SOLIDWIZE

ONLINE SOLIDWORKS TRAINING
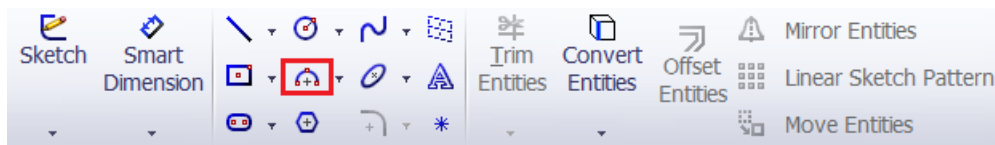
Exercise: Contour Selection
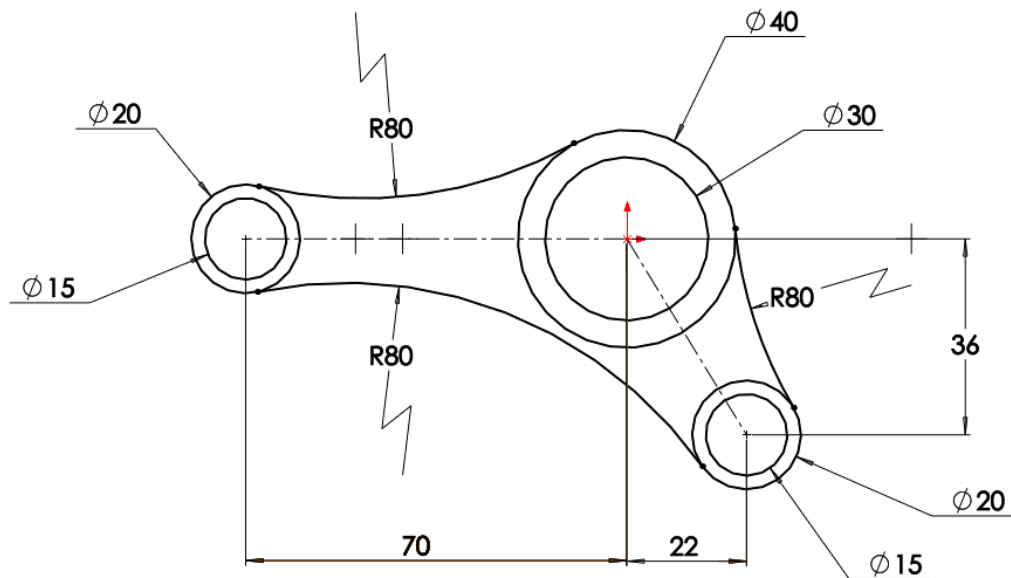
Step 1: Creating the Profile

Using **Millimeters** as the unit create the following sketch on the top plane.

Add **3-point arcs** between the three circular profiles. The **3-point** arc tool can be found in the sketch tab.
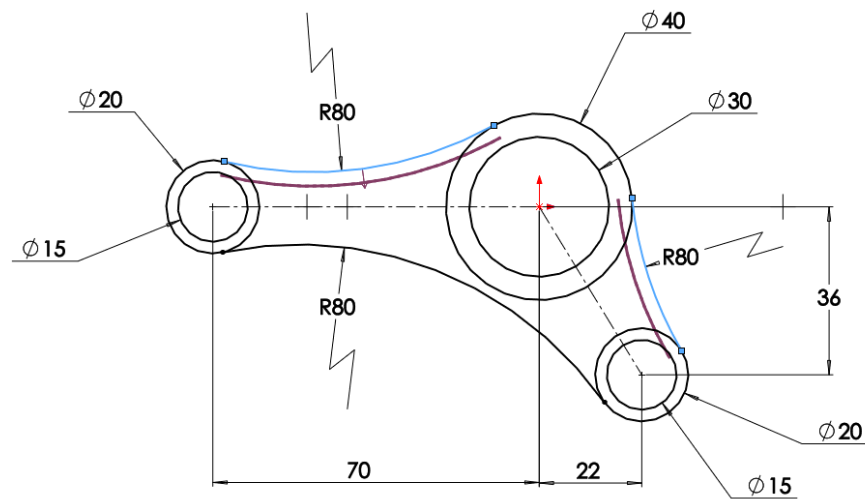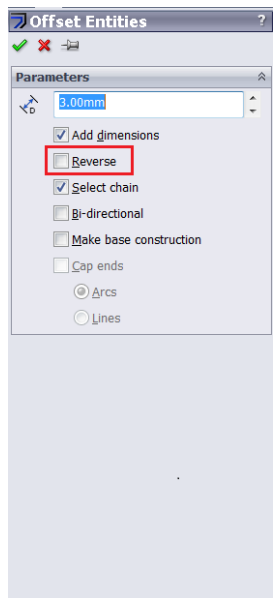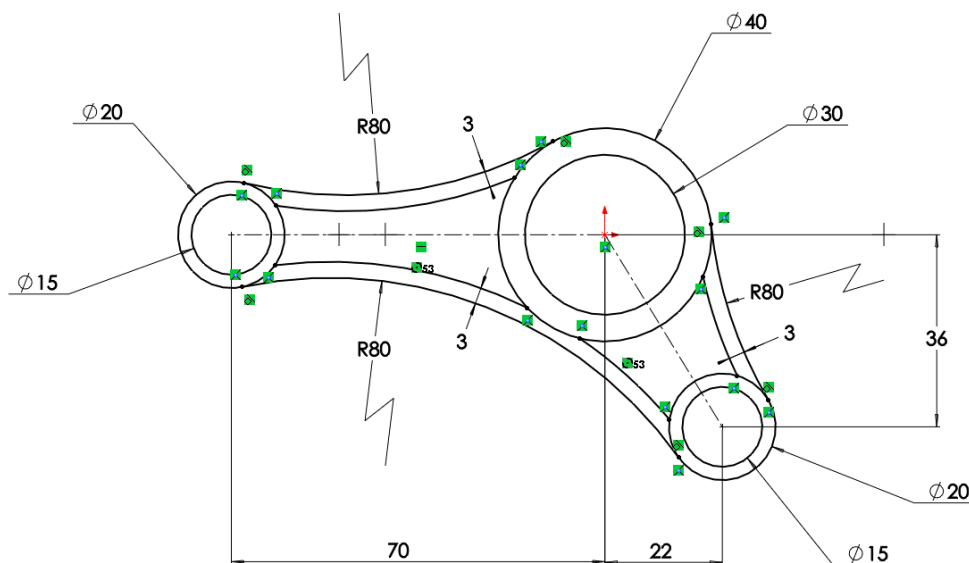

Add the arcs as shown below.

Using the offset tool, create an offset of the created arcs. The offset tool can be found in the sketch tab also.

Due to the locations of the arcs, you can only offset the top two arcs together with the same direction. If the bottom arc is also selected, it will create a down-ward offset which is not what we want. To change the direction of the offset, select the **"reverse"** option.
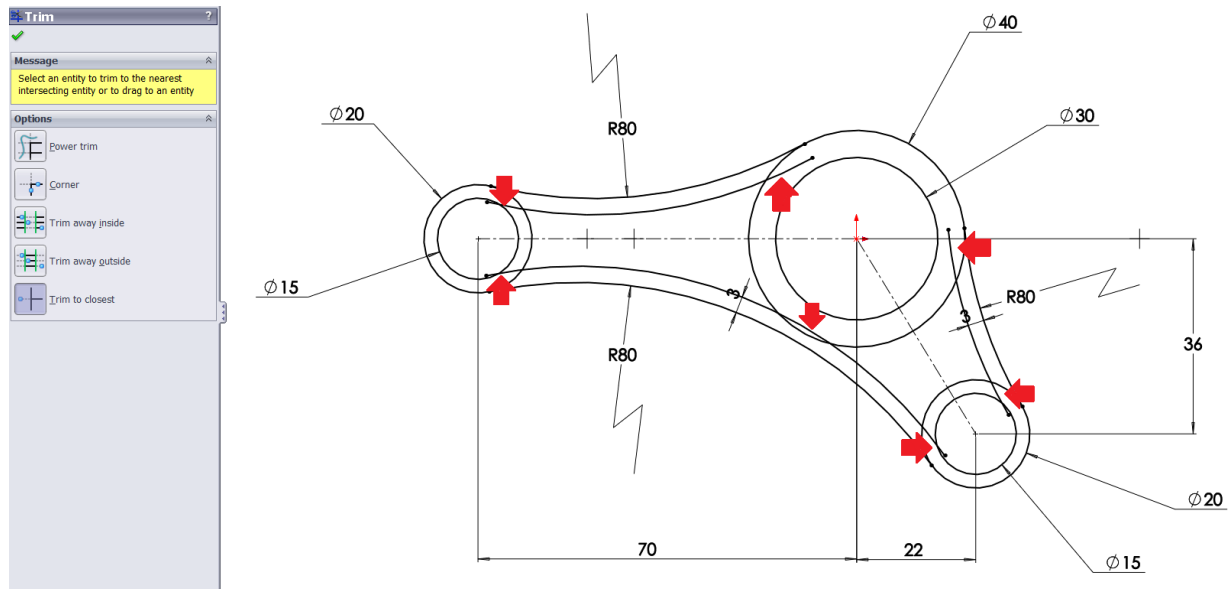

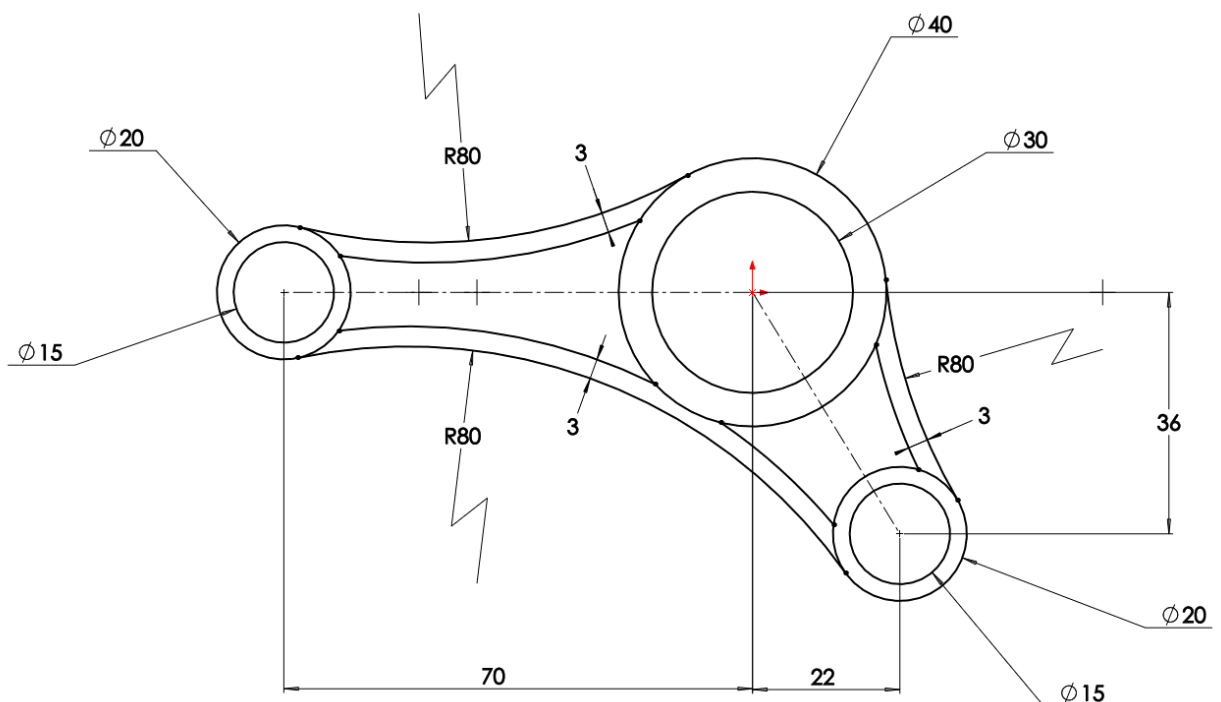The resulting sketch should look like the following:

To trim the unneeded sketch entities, select the **trim entities** tool from the **sketch** tab.

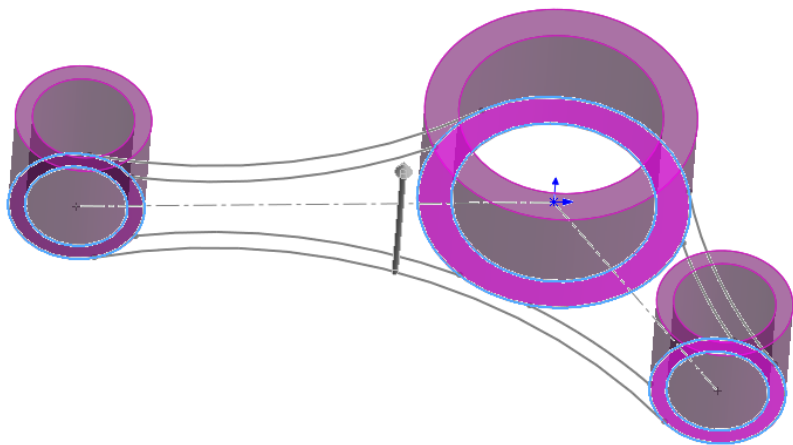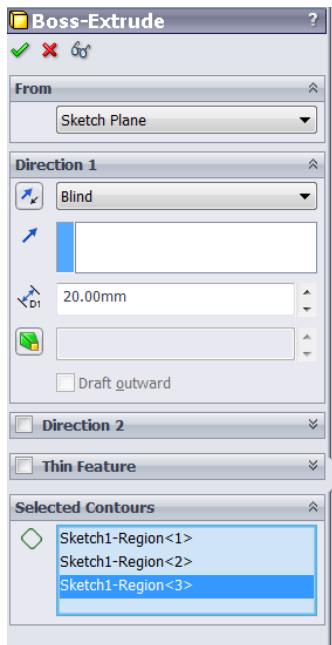Trim away the entities shown using the **trim to closest** option.

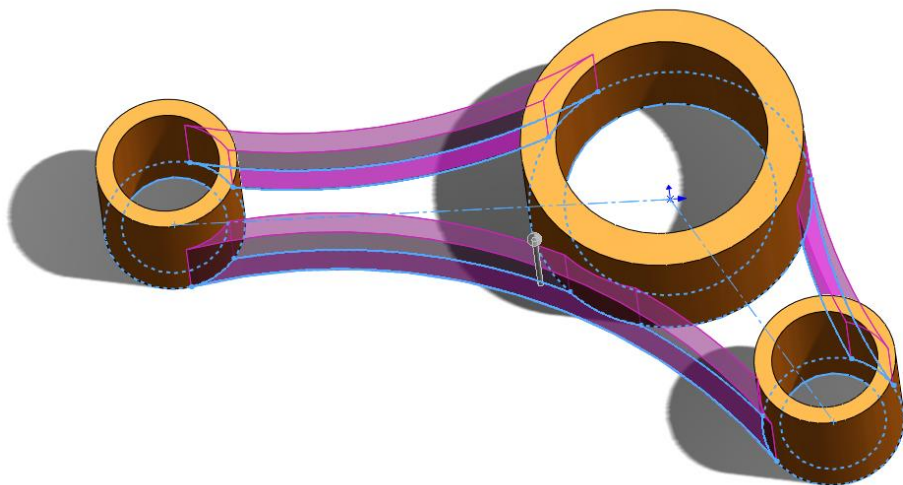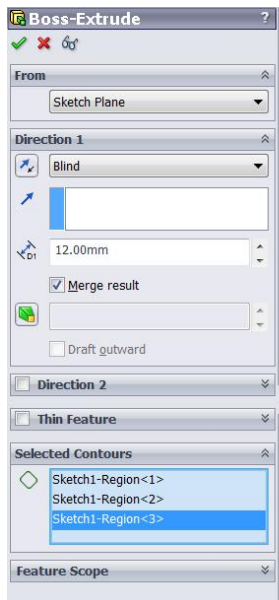The fully trimmed sketch is shown below:
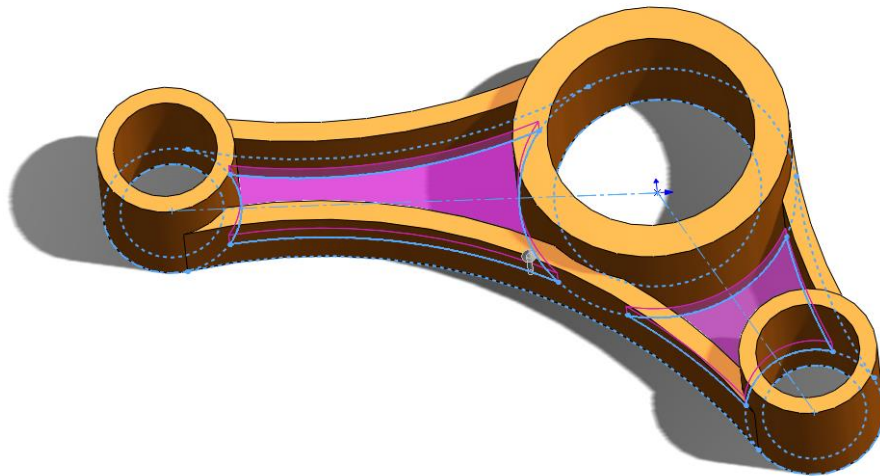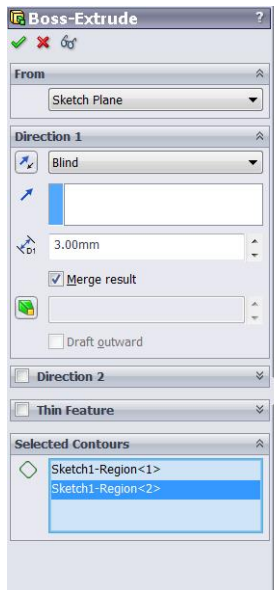
Step 2: Extruding the Profiles

Select the **Extruded Boss/Bass** feature from the **features** tab. Notice that the sketch profile created has multiple closed contours. SolidWorks allows you to choose which contour you want to extrude. To select the contours, select the sketch contour surrounding the area, or the area encompassed by the sketch. Select the three circular contours as shown below. Extrude the contour **20.00 mm** upwards.

Create another extrude feature and select the areas encompassed by the arcs. Extrude these contours a distance of **12.00 mm** upwards.
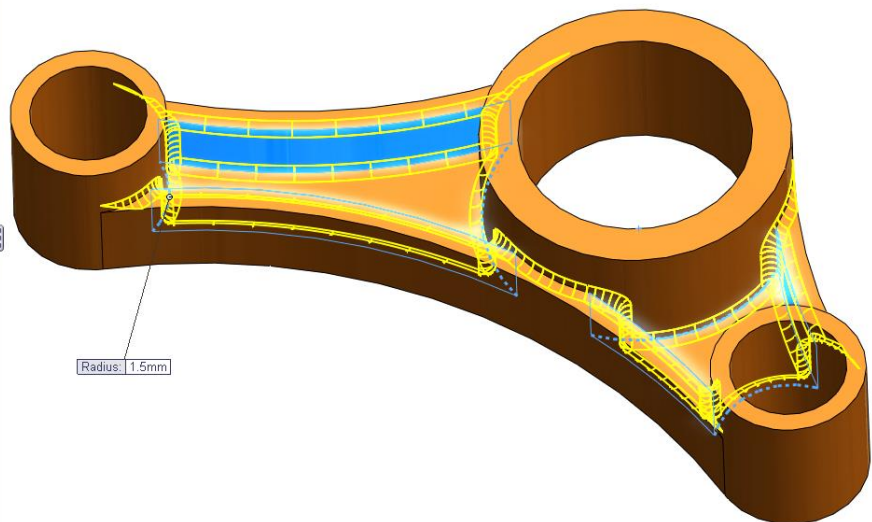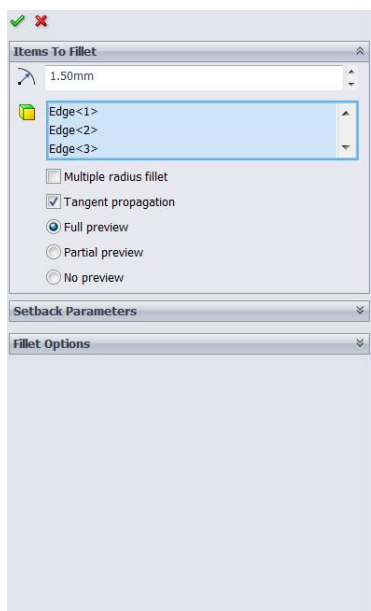
Lastly, create another extrude feature using the remaining closed contours. Extrude a distance of
3.00mm.

Step 3: Adding Fillets

Add **1.5mm** fillets to all the edges shown.

Step 4: Save and Exit

The final part should look like the following:

Save the part as **Contour_Selection.sldprt** and exit the part.