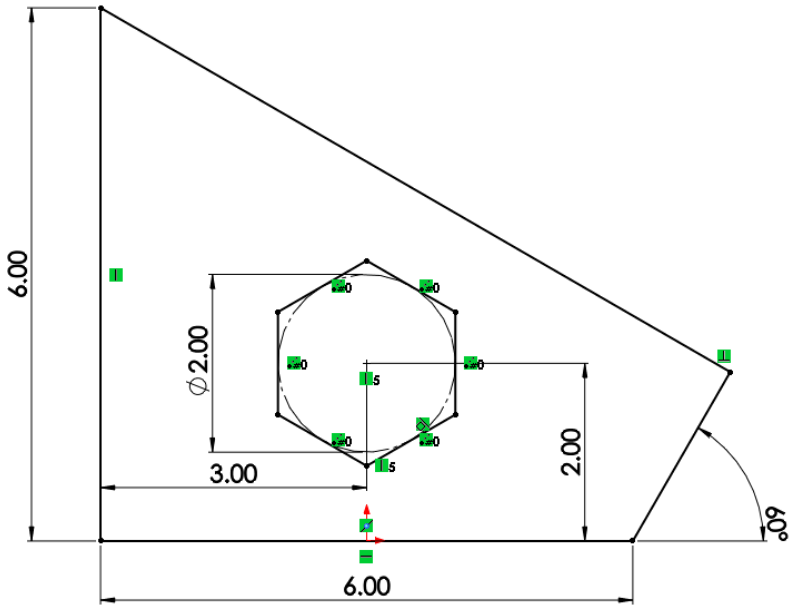
SOLIDWIZE

ONLINE SOLIDWORKS TRAINING

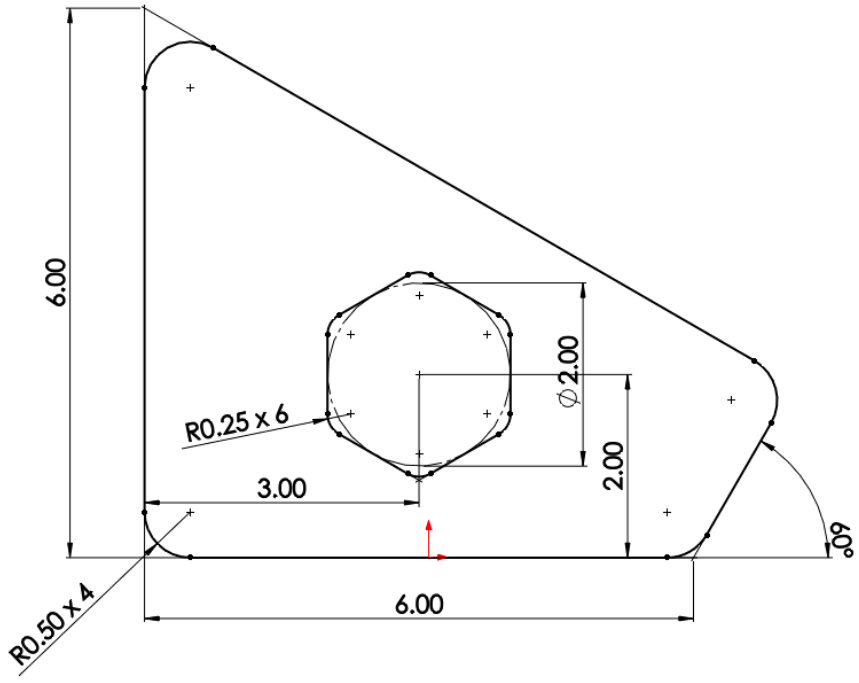
Simple Extrusion Exercise

Step 1: Creating the Body

Create the following sketch on the **Front Plane**. Notice the relations shown. You will have to create a midpoint relation between the bottom sketch line and the origin. The hexagon in the middle was created using the **Polygon** tool. Create a **vertical** relation between the bottom vertex of the hexagon and its center.

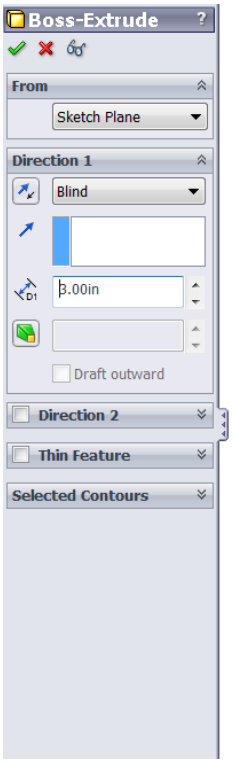


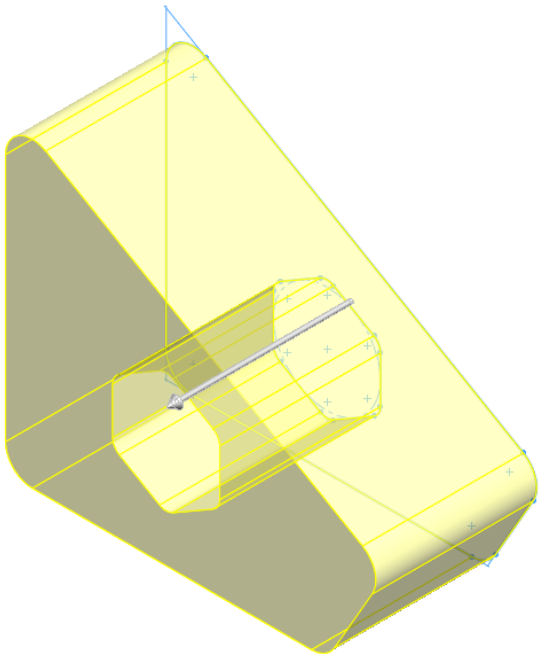
Using the **Sketch Fillet** tool, add **0.50in.** fillets to the outer edges and **0.25in.** fillets to vertices of the hexagon as shown below:



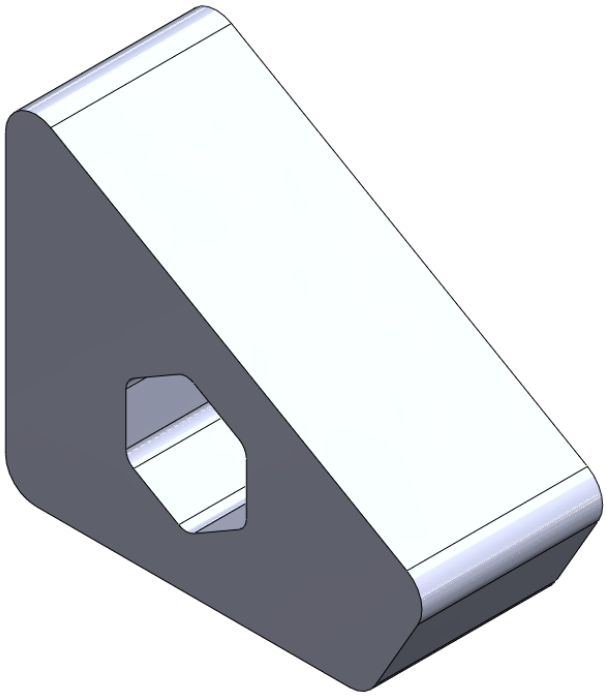
Step 2: Extruding the Body

Extrude the sketch created in **step 1**, using a **blind** extrude with an extrusion length of **3.00in.**



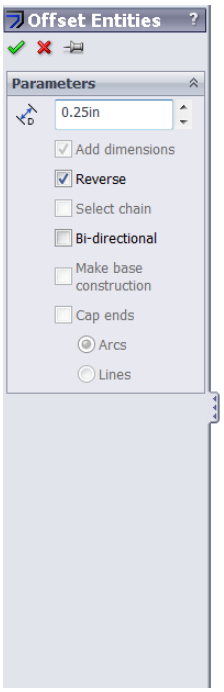


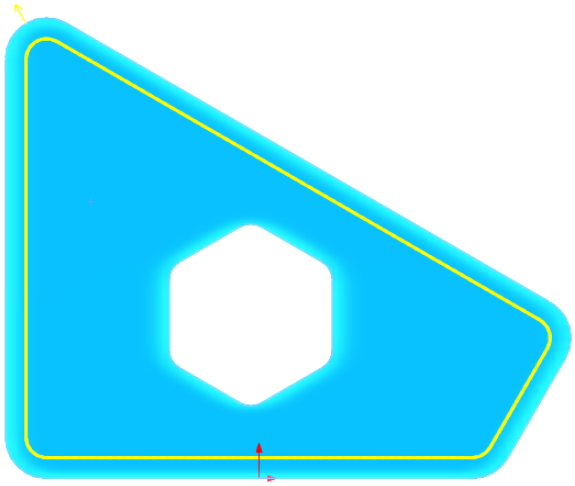
The resulting body should look like this:



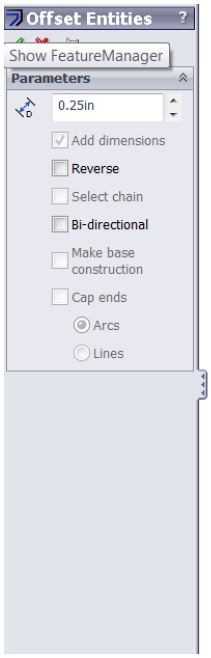
Step 3: Offsetting Entities

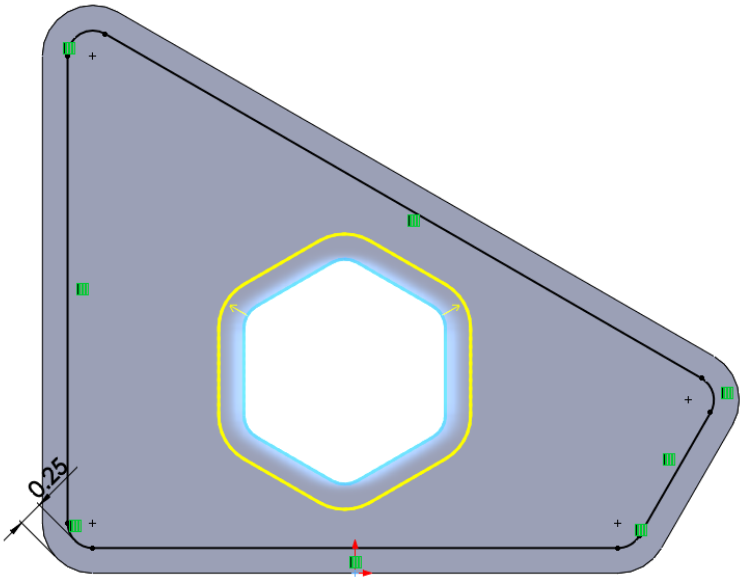
Select one of the planar side faces and under the **sketch** tab, select **Offset Entities**. This will automatically create a sketch with the entities at an offset. Set offset distance to **0.25in.**



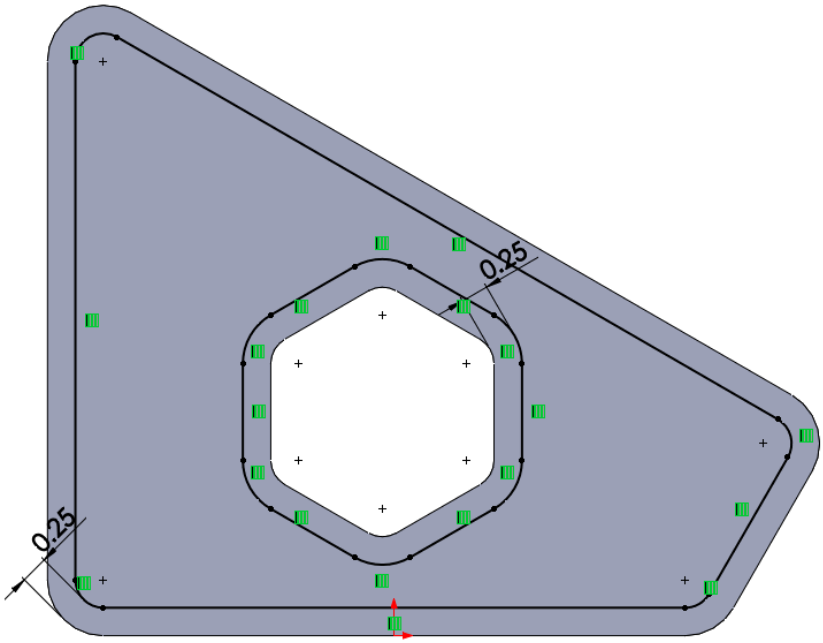


We will want to offset the hexagon also. Instead of manually selecting the edges for offset, right click one of the edges and select **"Select Tangency"**. This automatically selects the desired edges. Click **Offset Entities,** and use the same offset of **0.25in**.



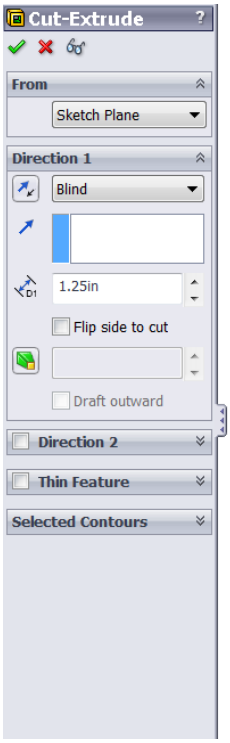


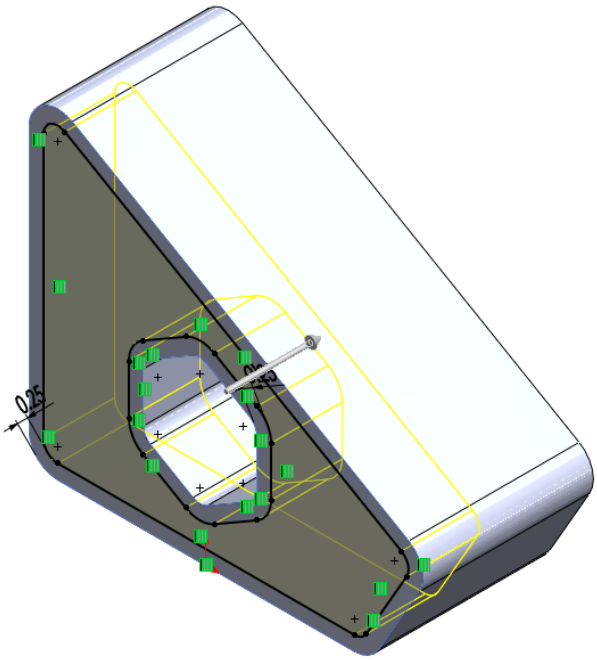
The resulting sketch should look like the following:



Step 4: Cut Extruding the Sides

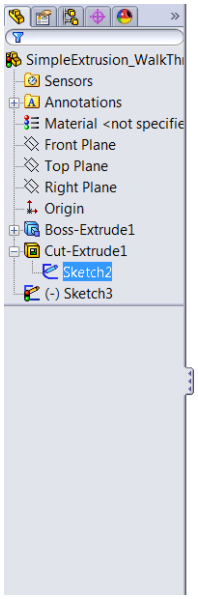
To cut extrude the sides, use the **Extruded Cut** tool under the **Features** tab. Use a **blind** cut with a depth of **1.25in.**

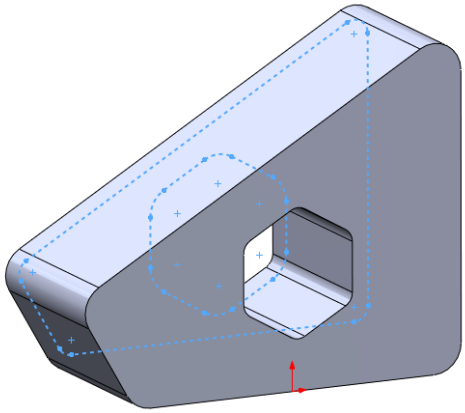




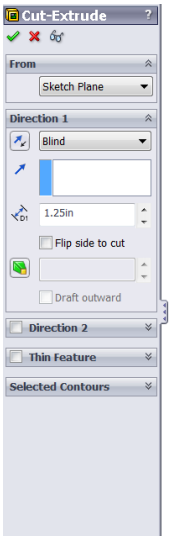
Step 5: Cut Extruding, Part 2

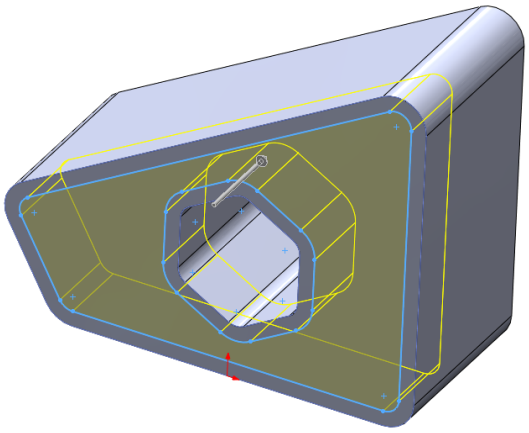
Using the **Convert Entities** tool, create a sketch on the **un-cut side planar surface** with the same sketch. To do so, click on the surface and create a sketch. Then, in the design tree located on the left of the display pane, expand the **Cut-Extrude1** feature by clicking on the **"+"** sign next to its name. Then select **Sketch2** and press **Convert Entities**. This will copy the sketch over.



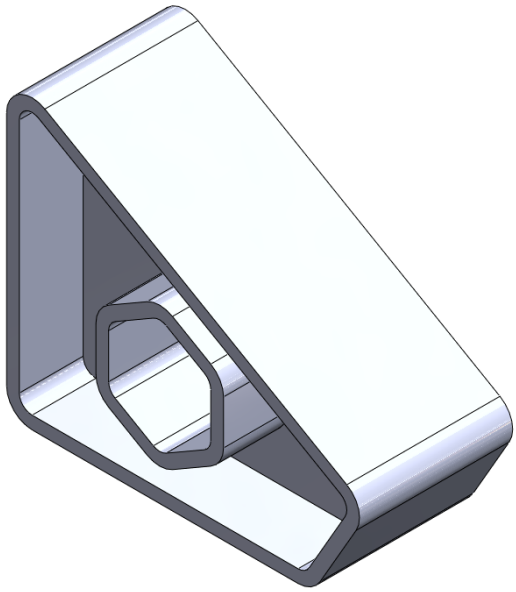


Then extrude cut the sketch like before, using the **Blind** option, with a depth of **1.25in.**



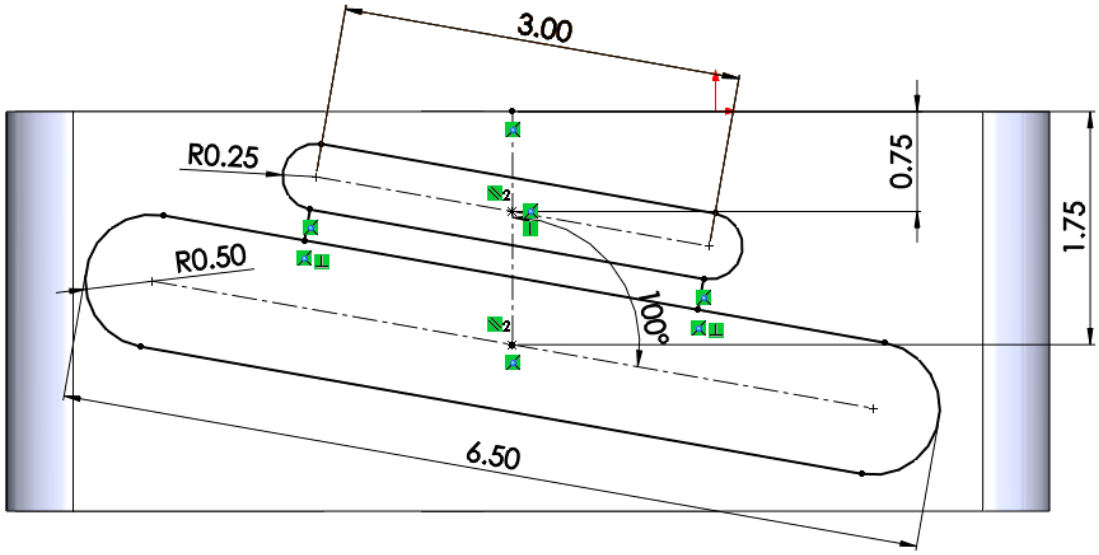


The resulting cut should look this:



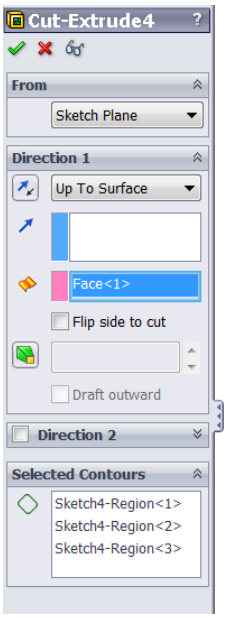
Step 6: Creating the Top Slots

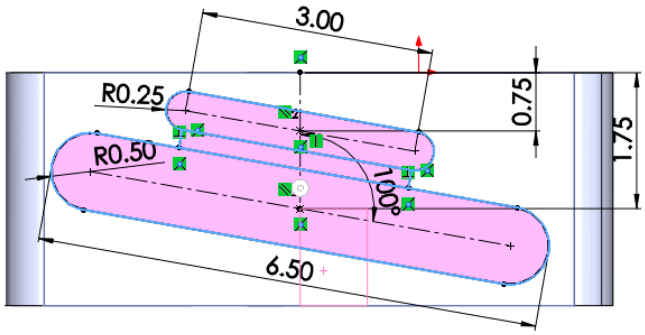
On the **Slanted Planar Surface**, create a sketch as shown below. The sketch has to slots created using the **Straight Slot** tool found in in the sketch tool menu. Make sure to have all relations shown. The larger slot has an angle of **100 degrees** between its centerline and the vertical construction line, while the slots have a **parallel** relation between their center lines. The centerline's midpoints are coincident to the vertical construction line. Notice the two small lines connecting the two slots. These lines have **Perpendicular** relation to the slots.



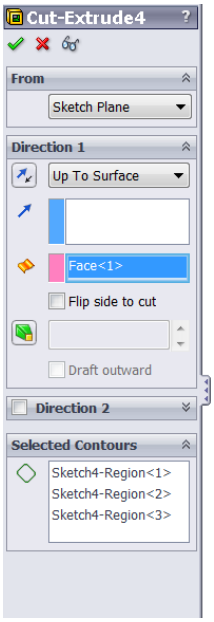
Step 7: Cut Extruding the Slots

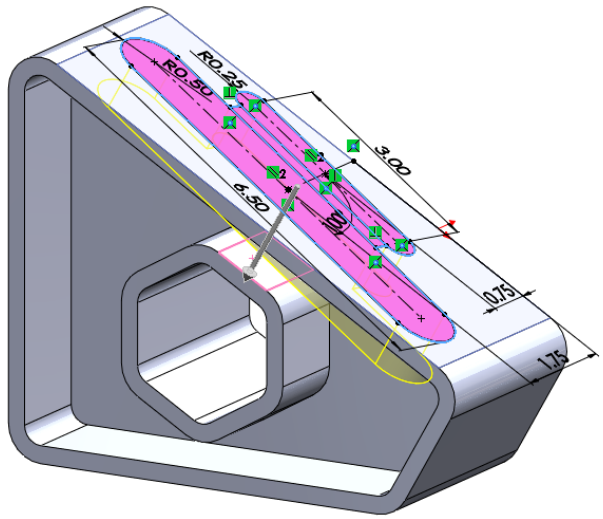
Since the slots have multiple closed areas (areas created by the two slots and the area between the slots bordered by the perpendicular lines), the extrude cut tool will not know which area to use to cut the body. You will have to select each area manually. With the sketch selected, click **Extruded Cut**, from the Features tab. Under **"Selected Contours"** in the property manager, select the three areas as shown below. Selected areas appear in pink



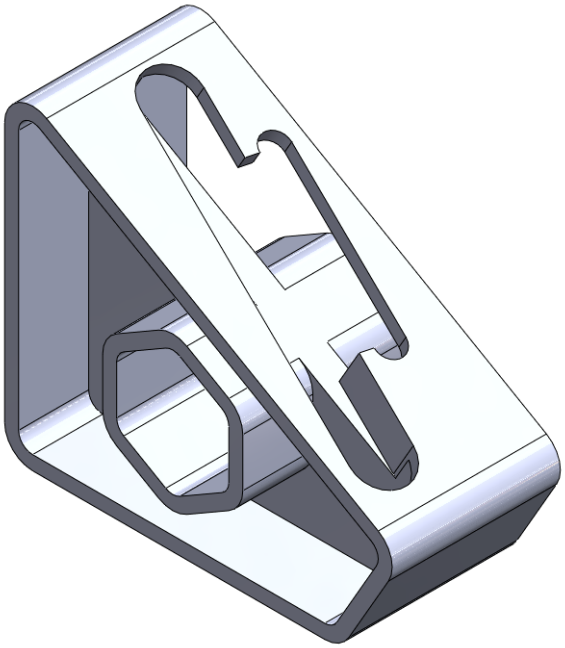


Under **Direction 1**, change the cut type to **"Up To Surface"** by selected from the drop down menu. Then select the **Top Face** of the outer Hexagon as shown below. The selected face is highlighted in pink.



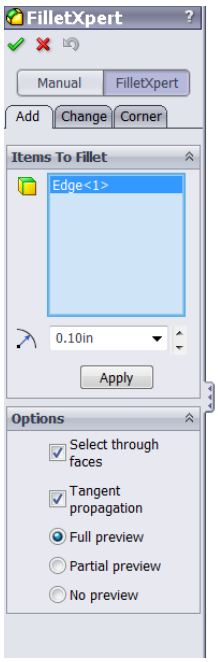


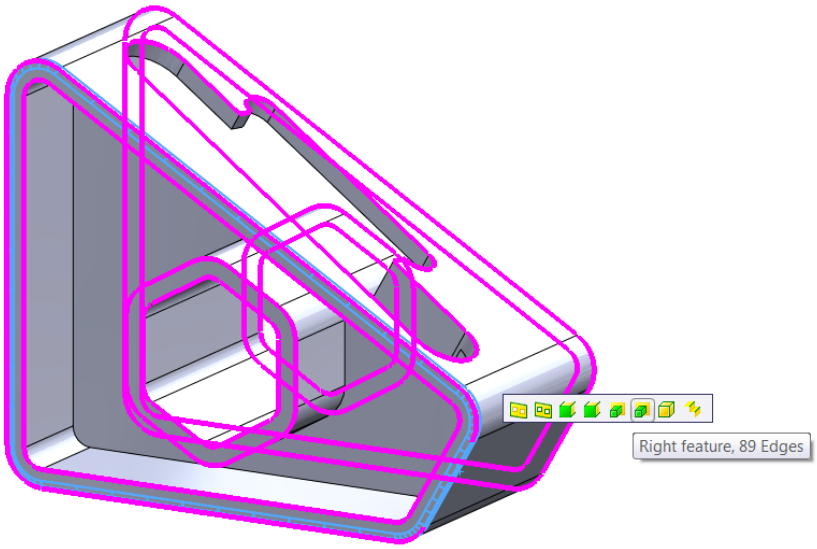
The resulting extrude cut should look like this:



Step 8: Adding the Fillets, Part 1

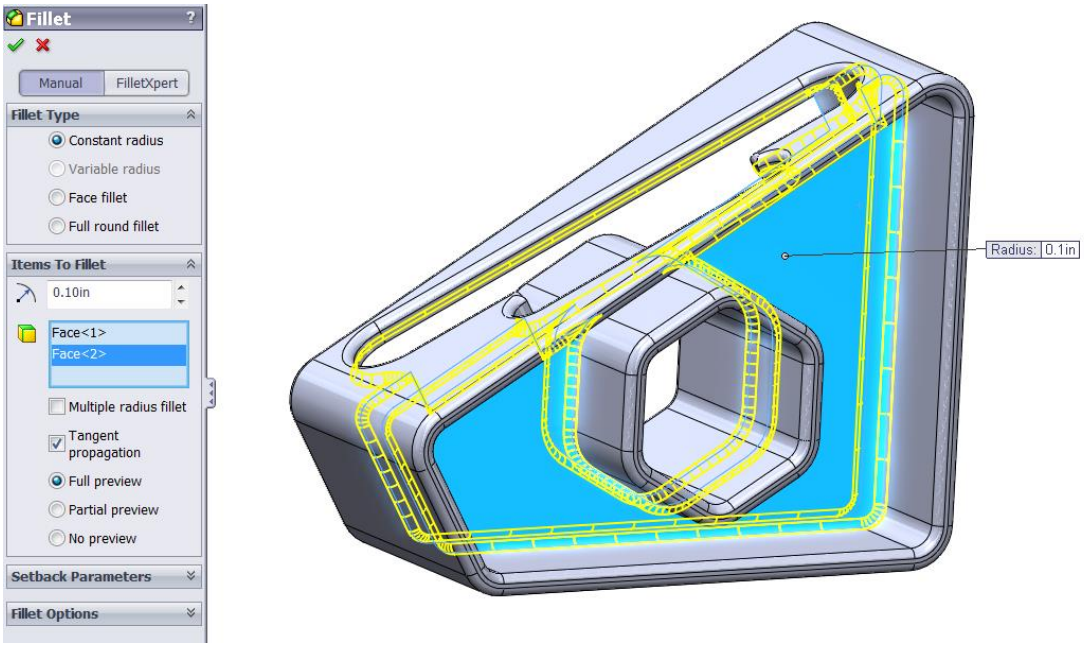
Using **FilletXpert,** we will now add some fillets to the part. Select the **Fillet** tool from the **Features** tab and select **FilletXpert**. Under **"Items To Fillet**"**,** select one of the outer edges of the part. Before moving the cursor away, an options menu should appear next to your cursor. This allows you to choose multiple fillet options. Pick **"Right Feature"** which will select all the desired edges. Use a fillet radius of **0.10in.** Click the check mark to accept the fillet.



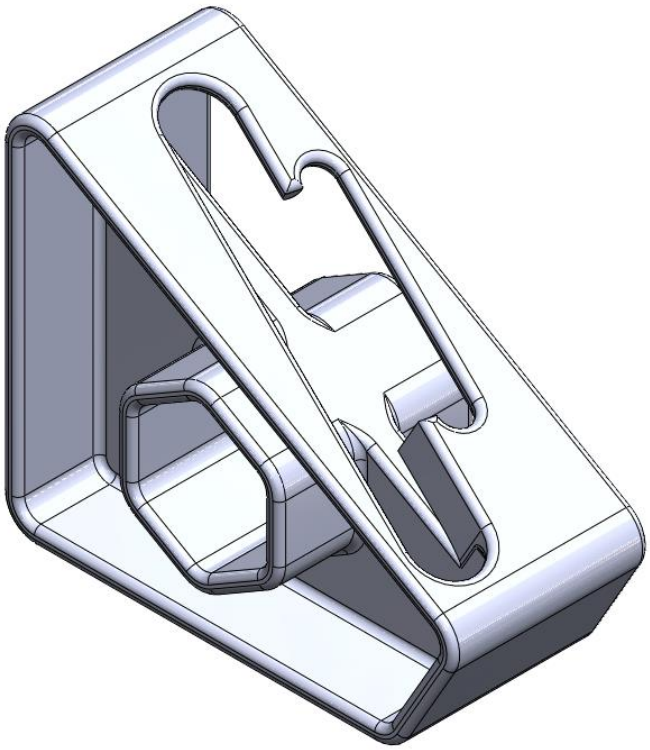


Step 9: Adding the Fillets, Part 2

Now with the **Manual Fillet** option, select the two inner planar side faces. Use a **Constant Fillet** type with fillet radius of **0.10in**. The preview should look like the following:

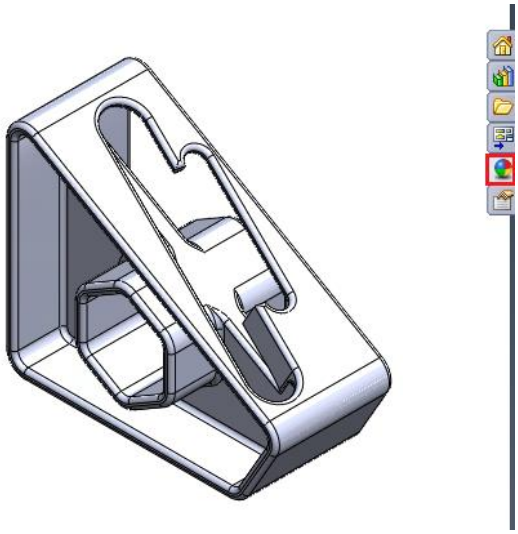


The resulting part should look like this:

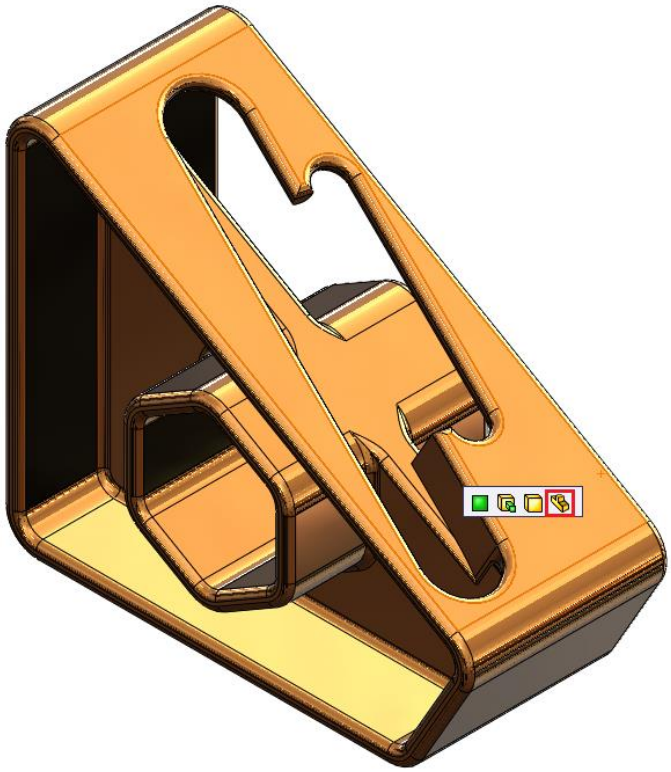


Step 10: Adding Appearance

To add the appearance shown on the cover page, select the **Appearance** tab from the right hand side of the display pane as shown below:



Then under **Metals,** select **Bronze > Polished Bronzed**. Drag the color icon onto the part. A small menu should appear next to the cursor. Select the last option as shown below. The options allow you to choose which level you want the appearance applied to. From left to right, the options are: Face, Feature, Body, and Part. We will apply the appearance on the Part level as shown below:



The final part should look like this:

Step 11: Save and Close

Do not forget to save and close the part! Save the part as **Simple_Extrusion.sldprt** and exit.