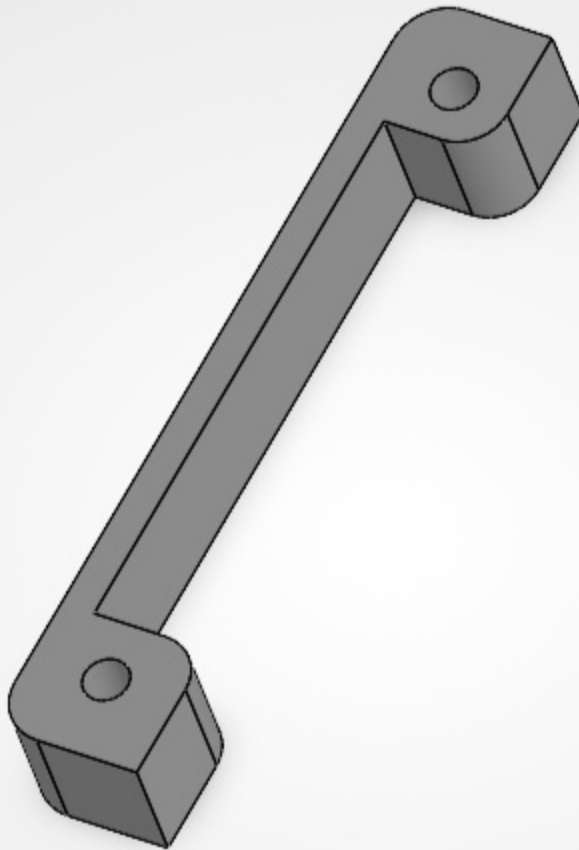




SolidWorks Basics

This guide will just barely scratch the surface on creating shapes and extruding them in SolidWorks.

Written By: Matt P. Lamparter



INTRODUCTION

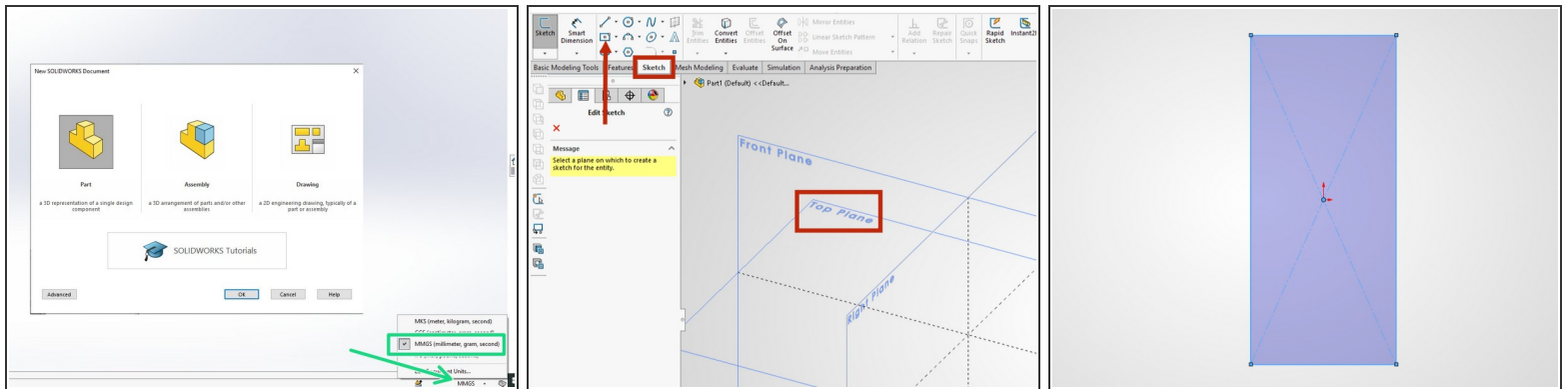
This guide will provide you with the bare minimum needed to create shapes and extrude them in SolidWorks. Concepts/commands covered will include creating 2D shapes, dimensioning them, extruding them, and making cuts. The end goal is to have an .STL file for 3D printing. The tutorial is based on a component used in the ECEG 201 DAMNED project.

Step 1 — The basics



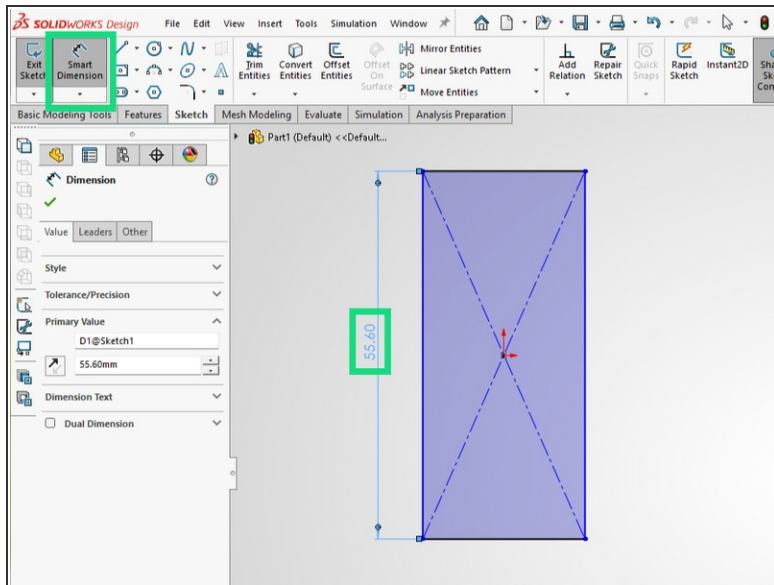
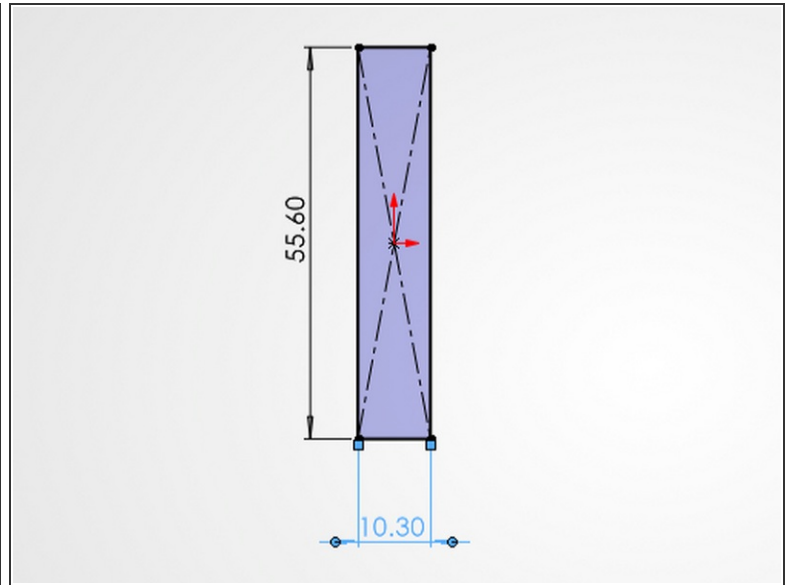
- Launch SolidWorks from any computer in the engineering buildings. You may also install it on your personal Windows machine by following the instructions [here](#).
- The basic idea in SolidWorks is to create a 2D shape (a sketch), add dimensions to it, and then extrude it. In other words you "pull" the 2D shape into a third dimension and give it depth.
- Once you have a basic 3D shape then you can add additional 2D shapes on to it and further extrude these.
- Alternatively, you can add a 2D shape onto a 3D part and use the 2D shape to create a cut.
- There are **many** more things you can do in SolidWorks, but for the purposes of this tutorial that summarizes everything.

Step 2 — Create a rectangle



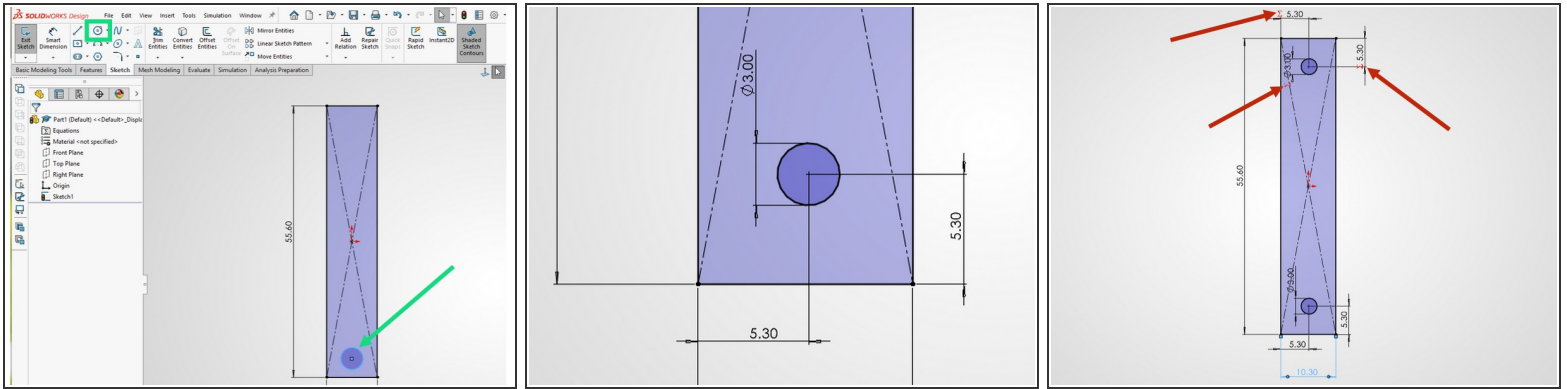
- After launching SolidWorks, begin by creating a new part.
- Once you have clicked OK to create a new part, be sure to choose metric units of measurement (MMGS) in the bottom right corner of the screen as shown here in green.
- Create a center rectangle by clicking the **Center rectangle** button within the **Sketch** tab.
- Choose to create your sketch in the **Top Plane** by hovering your mouse over the correct plane until it highlights orange, then left clicking.
- Click the origin of the sketch (small red axes near the center of the screen) to anchor the center of your rectangle there.
- Move your pointer out away from the center of the rectangle. Create a rectangle of arbitrary size (but roughly skinny and vertical) by clicking again.
- ❗ Rotating the scroll wheel of your mouse will zoom in or out on the drawing.

Step 3 — Dimension the rectangle



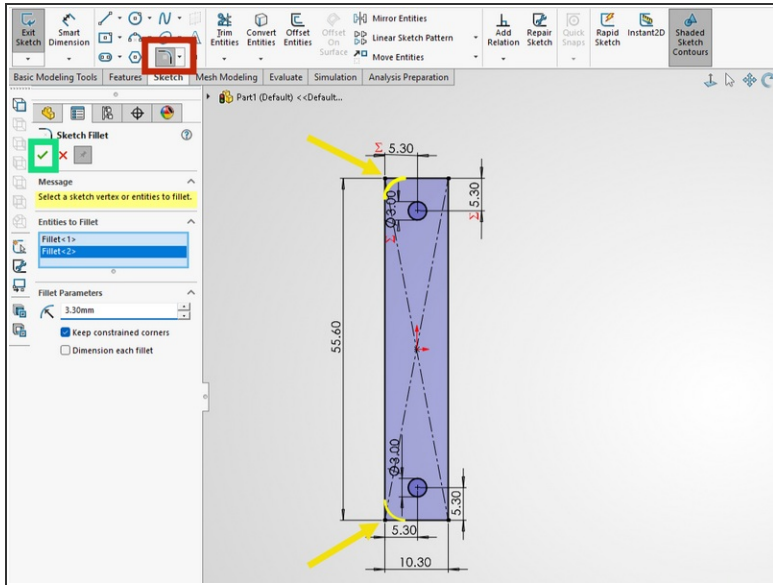
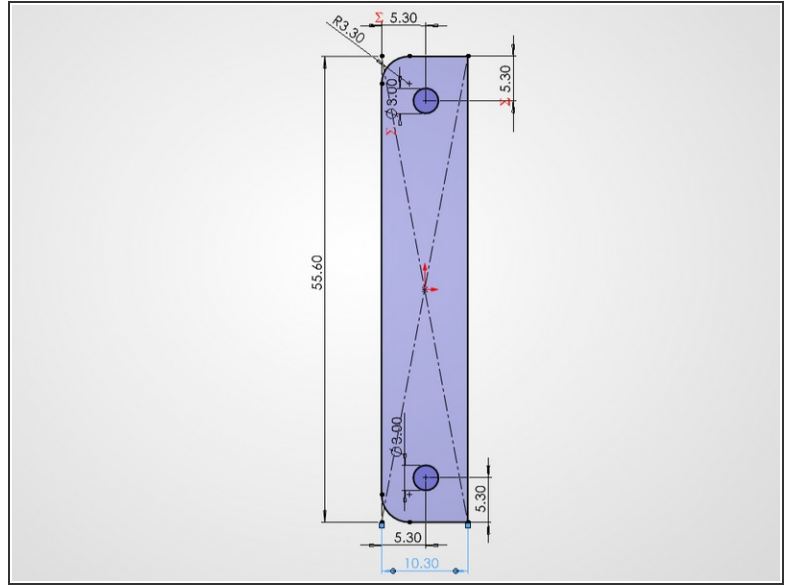
- Click on the **Smart Dimension** button and then click on the long edge of the rectangle you just drew. Drag the dimension out and click to anchor it. A pop up box will appear and allow you to enter a value. Enter **55.6**.
- Repeat this process for the narrow dimension and set the value equal to **10.3**.

Step 4 — Add bolt holes



- Click the **Circle** button from the **Sketch** tab and place a circle near the bottom of the rectangle.
- Use the **Smart Dimension** tool to give the circle a diameter of 3.0 mm and position it 5.3 mm from the left and bottom edges. See the second image.
 - Clicking on the edge of the circle will allow you to define a diameter.
 - Clicking on the center of the circle and then an adjacent side will allow you to define the distance from the circle center to that side.
- Create a second circle at the top of the rectangle. Give it the same diameter and spacing as the first circle you created.
 - Since this circle should match the bottom circle you can use the = (equals) sign when dimensioning it, followed by clicking an existing dimension you wish to copy. This creates a pointer in SolidWorks so if the original dimension changes, the pointers will update automatically.
 - Note the three red Σ (sigma) symbols in front of the top three dimensions in the final image shown here. This indicates these values are pointing to other existing dimensions.

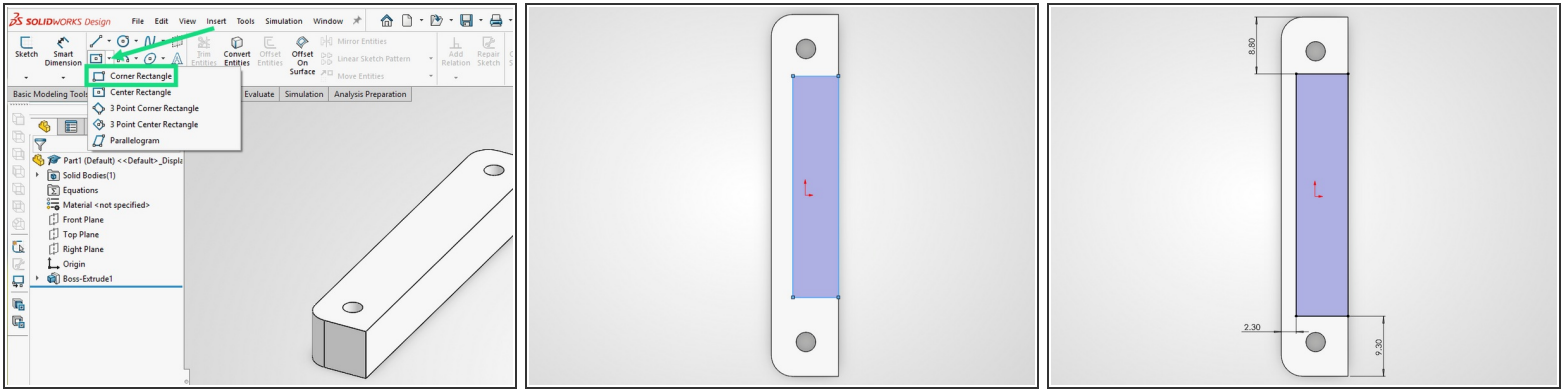
Step 5 — Fillet the corners



- Click on the **Sketch fillet** button in the **Sketch** tab. A fillet is a rounding of an interior or exterior corner of a sketch or part.
- Click on the top left and bottom left corner points to add them to the list of points to fillet. You should see a fillet preview appear in yellow. Enter a value of 3.3 mm for the fillet radius.
- Click the green check mark twice to accept these fillets.
- Your sketch should look like the final image shown here.

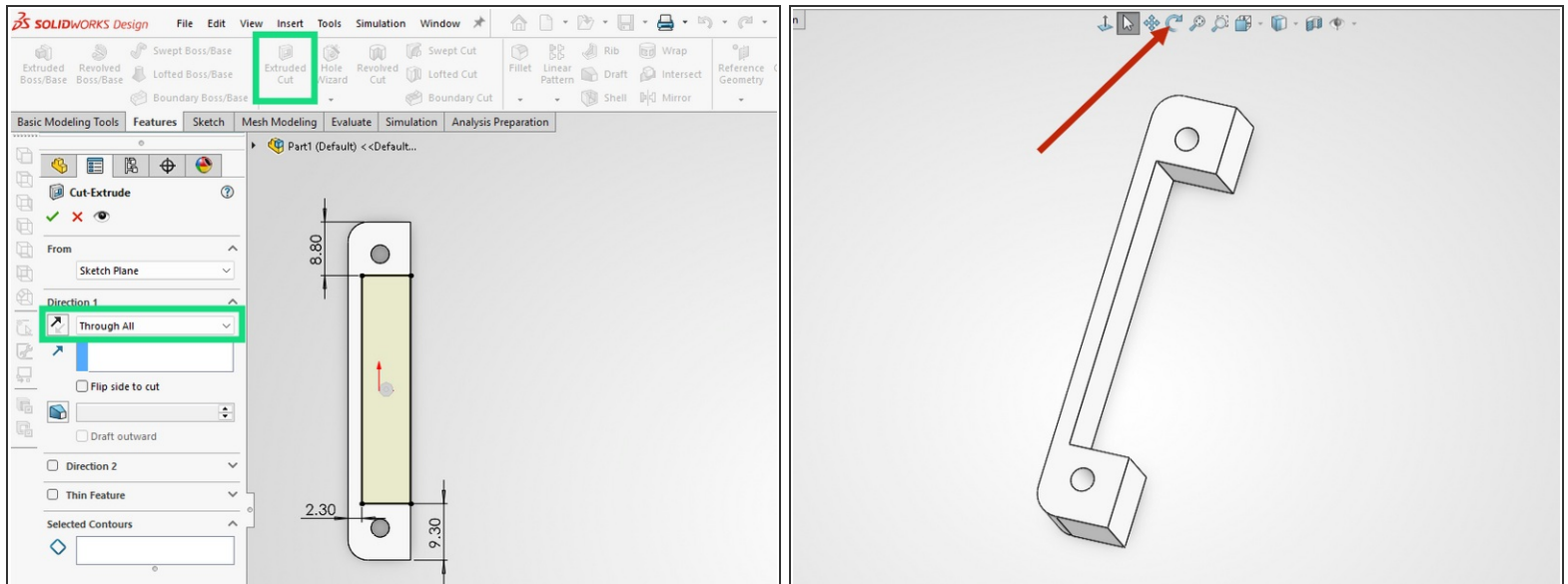
- Click the **Extruded Boss/Base** button from the **Features** tab. Your sketch should rotate and a yellow preview of the extrusion will appear.
- Enter a **depth** value of 8.19 mm and then click the green check mark to accept this value and extrude the sketch.
- The part should now be shown as a 3D dimensional object.

Step 7 — Add a sketch for a PCB components cutout



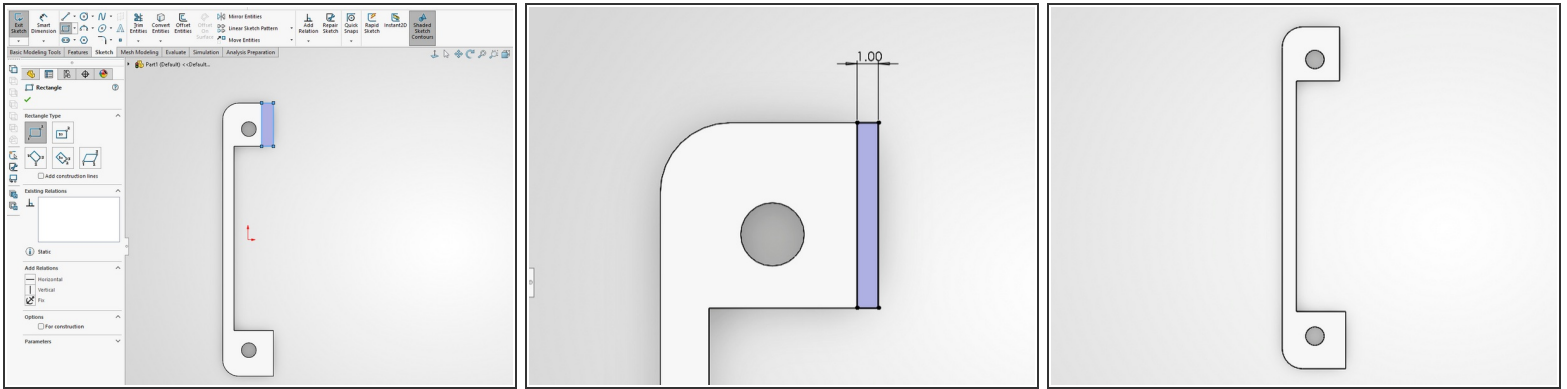
- Return to the **Sketch** tab and choose to sketch a **Corner rectangle**. You may need to click the small arrow to the right of the **Center rectangle** tool to display a drop down menu where you can choose **Corner rectangle**.
- SolidWorks will prompt you to select a *plane, planar face, or an edge on which to create a sketch for the entity*. Click on the top face of the part.
- Anchor the top right corner of the rectangle on the right edge of the face and anchor the bottom left corner of the rectangle some distance below this but not touching the left edge of the face. See the second image.
- Use the **Smart Dimensions** tool to give the rectangle the three dimensions shown in the final image.

Step 8 — Cut rectangle from the extrusion



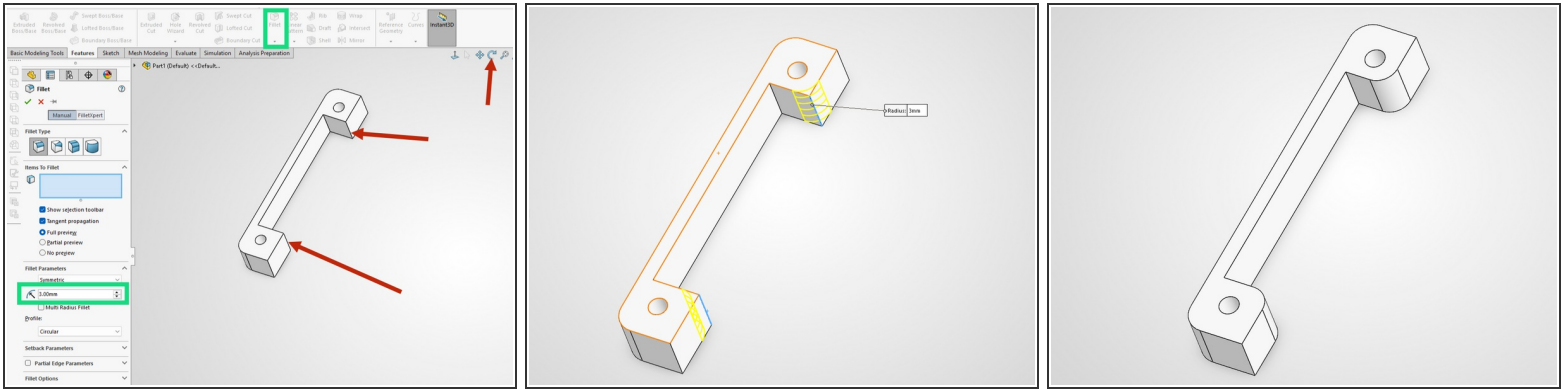
- Return to the **Features** tab and choose **Extruded Cut**.
- Change the **End condition** drop down menu to **Through All**. This tells SolidWorks to make this cut pass through the entirety of the extrusion, regardless of how deep it is.
- Click the green check mark to accept the cut.
- There should now be a rectangular cutout in the part. You can use the **Rotate View** tool to click and drag in order to rotate your part in three dimensions to confirm the cutout went all the way through the extrusion.

Step 9 — Compensate for USB C connector



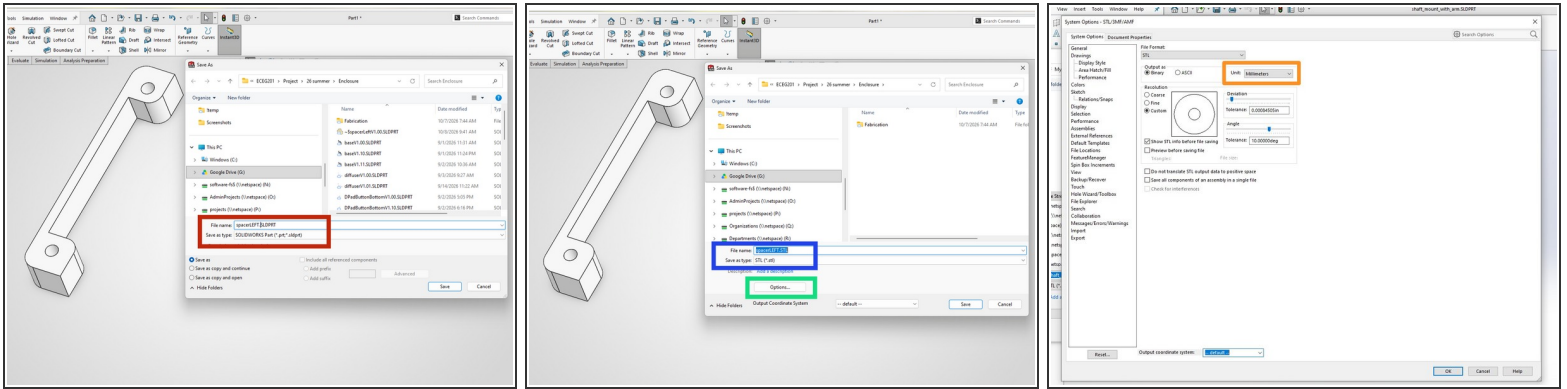
- ❗ The USB C connector is close to the edge of this spacer so we need to remove a small amount of material to compensate for this and allow clearance for the spacer to properly fit inside of the enclosure.
- Return to the **Sketch** tab and create another **Corner rectangle**. When prompted to choose a face, choose the top of the extrusion like you did in the previous step.
- Anchor the top right corner of the rectangle to the top right corner of the extruded part. Anchor the bottom left corner of the rectangle to the bottom edge of this upper portion of the extruded part. See the first image.
- Use the **Smart Dimensions** tool to give the rectangle a width of 1.0 mm.
- Return to the **Features** tab and create an **Extruded Cut** from this rectangle. Choose an **End condition** of **Through All** like you did previously.
- The top width should now be slightly more narrow than the bottom width as shown in the final image.

Step 10 — Add additional fillets



- The final step is to add two more fillets. This time we will add fillets directly to existing extrusions as opposed to adding them to a 2D sketch as we did in a previous step.
- Within the **Features** tab click on the **Fillet** button. Enter a value of 3.0 mm into the **Radius** box.
- Again use the **Rotate View** tool to rotate the part so you can clearly see the two edges indicated here in red.
- Clicking on the indicated edges should provide a preview of the fillets in yellow. Click the green check mark to accept these fillets.

Step 11 — Save your part as .sldprt and .STL



- Save your part as a .sldprt file so that you can modify it later if necessary. Give it a meaningful name. If you're working on DA7 for ECEG 201 be sure to indicate that this is the LEFT spacer so you don't confuse it with the additional RIGHT spacer you also need to create.
- Also save your part as a .STL file. You will need an .STL file to 3D print. Note that you cannot (easily) modify an .STL file.
- Be sure to click the **Options** button before saving the .STL file.
- Because many 3D printer slicing applications operate in metric units, you should confirm that the **Unit** parameter of your STL file is set to **Millimeters**.
- Click **OK** and then save your .STL file.
- SolidWorks will pop up a window telling you how many triangles your part contains. Click **Yes** to continue saving.

Congratulations! You now know enough SolidWorks to be dangerous. YouTube and the builtin SolidWorks tutorials are your friends. SolidWorks is a powerful, full featured tool. Anything you want to accomplish has already been done. Use an internet search to find a video detailing the steps necessary to accomplish anything more complex than what is provided in this tutorial.