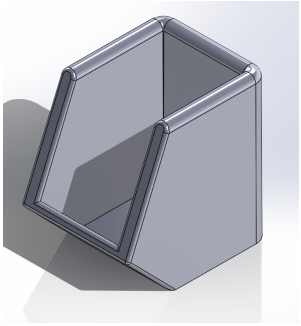
Beginner Onboarding Challenge

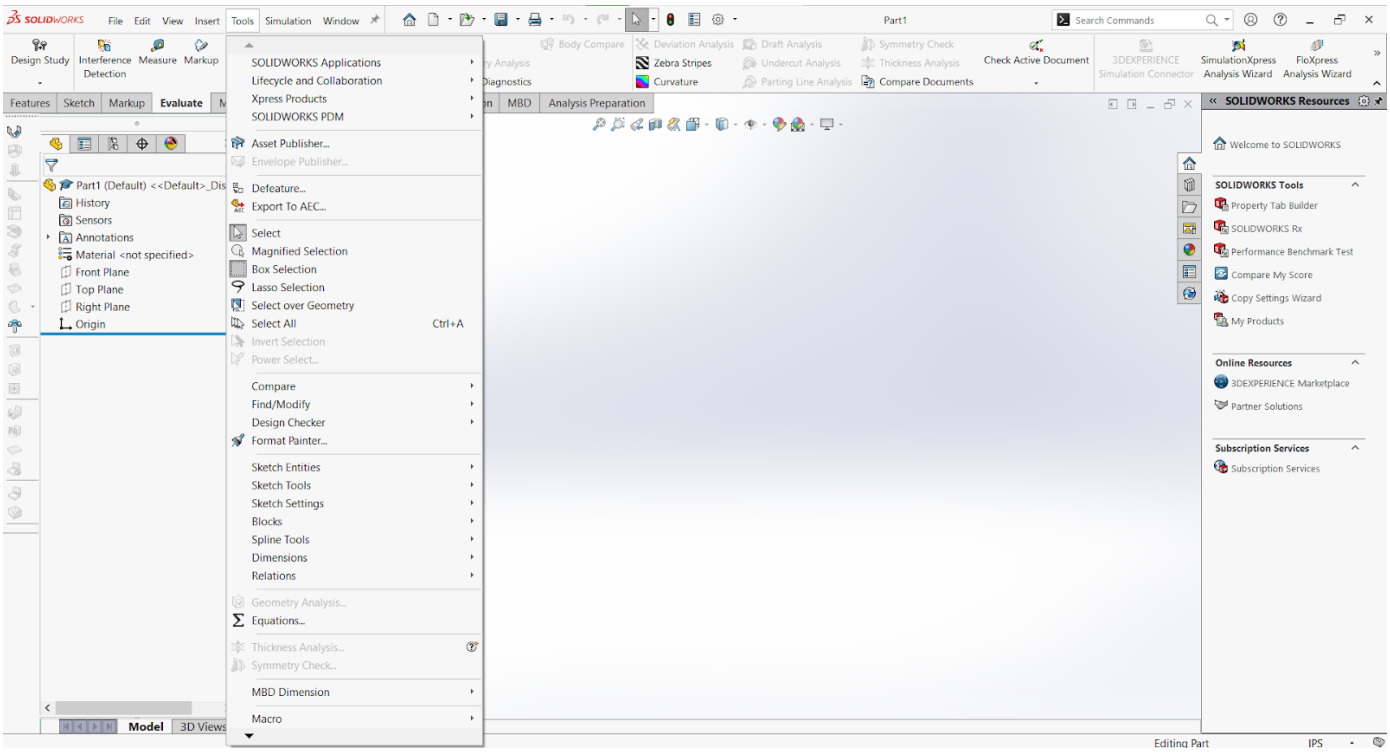


For the beginner onboarding challenge you will be walked through modeling the stackable bin pictured. This challenge is intended for members who have not used Solidworks before, if you have experience with Solidworks please attempt the intermediate onboarding challenge before this one. Feel free to message Stevie if you have trouble at any point.

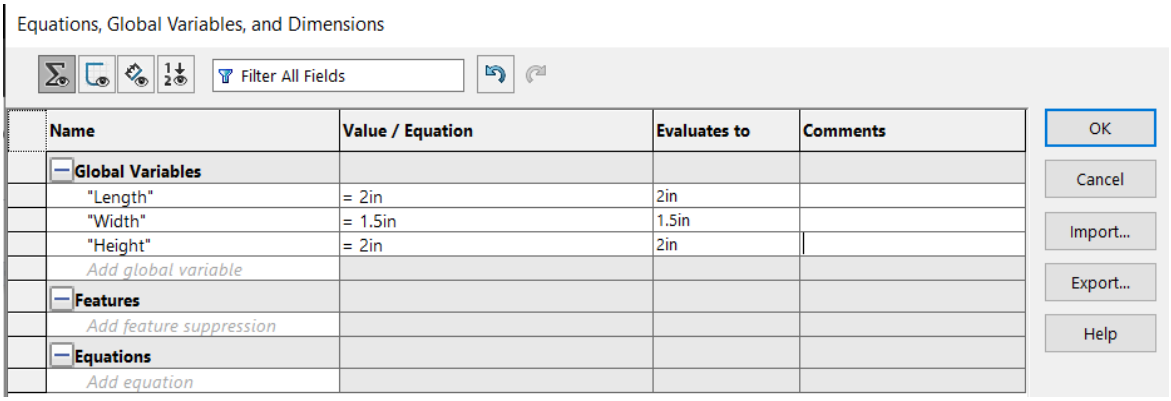
If you are brand new to Solid works, this video gives a good overview of the user interface in more detail than what is provided in the rest of this document. https://www.youtube.com/watch?v=qtgmGkEPXs8

Step 1: setting up equations

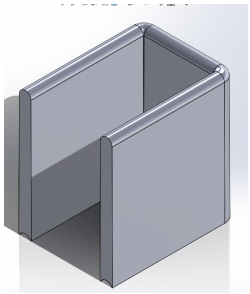
Equations allow you to quickly alter dimensions of parts by creating desired relations. First navigate to the Tools dropdown and select equations, alternatively you can use the search bar set to “search commands” and type in “equations”.



Then type in the following global variables, these will be used to dimension your part later.



Step 2: swept base

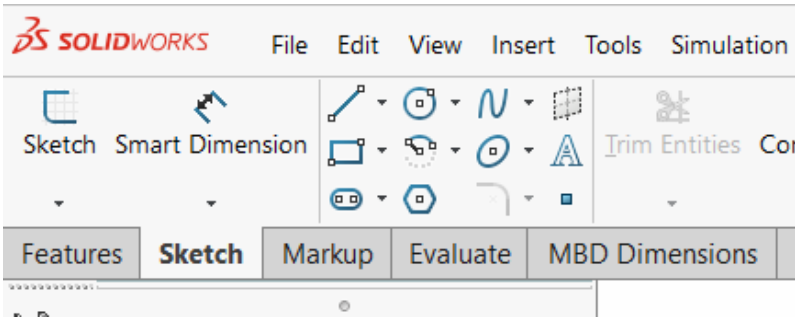


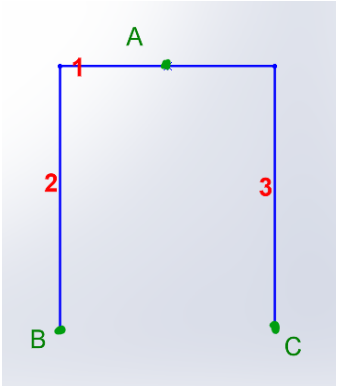
Next, use the Swept base feature to make the sides and back of the part as shown. The swept base feature requires two sketches, one for the path and one for the profile.

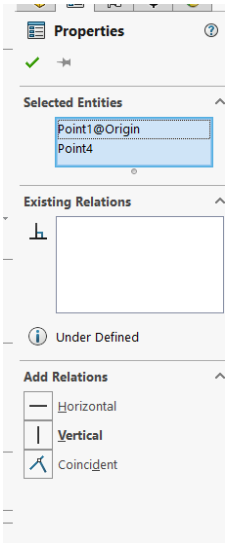
Step 2.1: swept base - path sketch

Start by creating a new sketch on the top plane

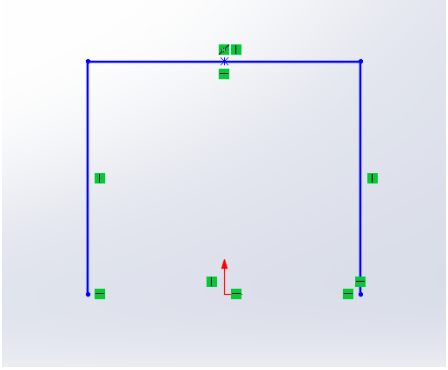
You can see the name of a sketch entity by hovering over the icon







Draw the above shape using a Midpoint Line on the top and a line on either side. To make relations CTRL select the item/items you want to make relations with and a list of possible relations will be displayed along the left side of the screen, simply click on the desired relation and hit ok.



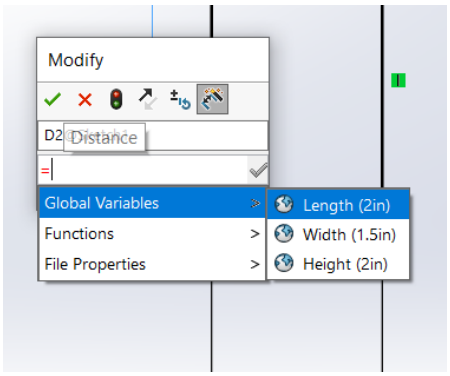
Now make the following relations:

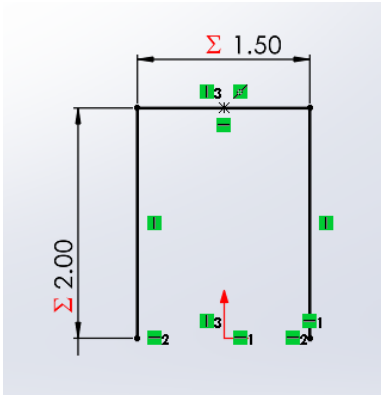
- Line 1 is Horizontal
- Lines 2 and 3 are both Vertical
- Point A and origin are Vertical
- Points B, C and origin are Horizontal

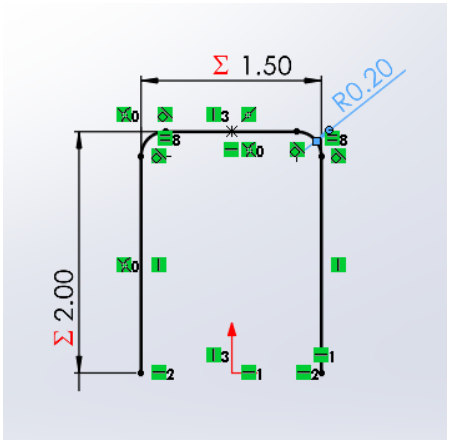
Use the smart dimension tool to set the dimensions of the sketch, select the feature you want to dimension (typically a line or pair of points) to bring up the dialogue box. To use global variables for dimensions, simply type “=” in the dialogue box and hover over global variables.

Add the following dimensions:

- Line 1 = Width
- Line 2 = Length





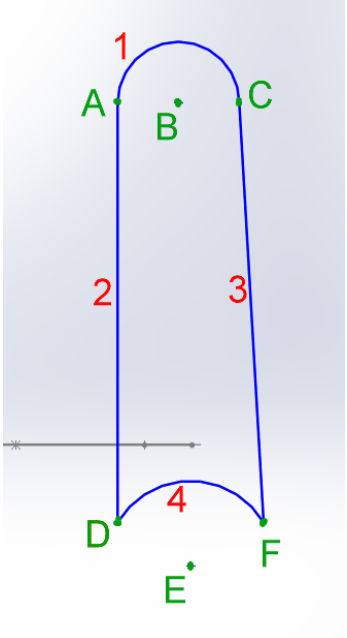


Lastly, add a 0.20in radius fillet to each of the two top corners using the sketch fillet feature.

Step 2.2: Swept base - profile sketch

Start a new sketch on the front plane, use a centerpoint arc on the bottom and top and a line on each side. Add the following relations:

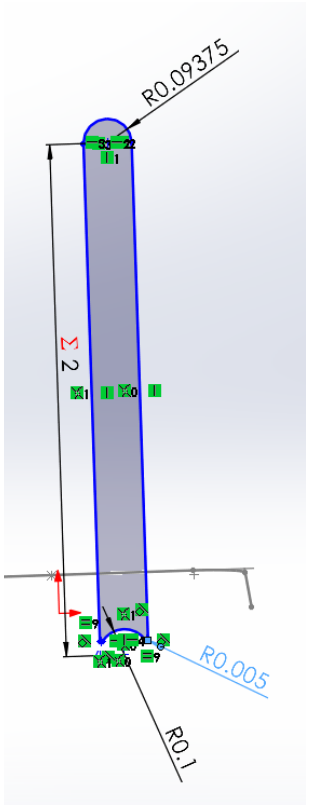
- Lines 2 and 3 are Vertical
- Points B and E are Vertical
- Points A, B, and C are Horizontal
- Points D and F are Horizontal

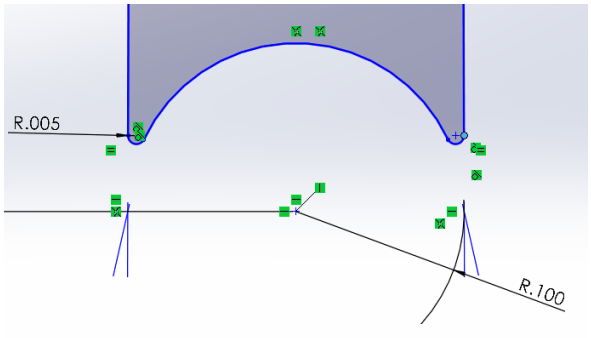


Now add the following dimensions:

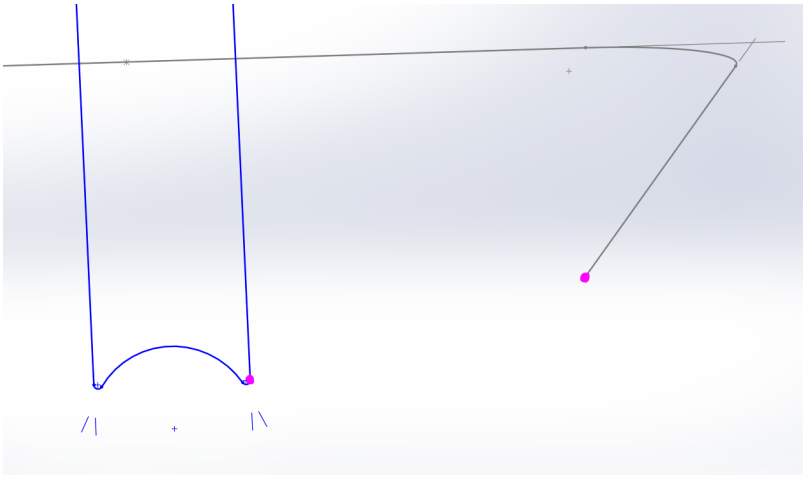
- Radius of arc 1 = 3/32”
- Radius of arc 4= 0.1”
- Distance between points B and E = height

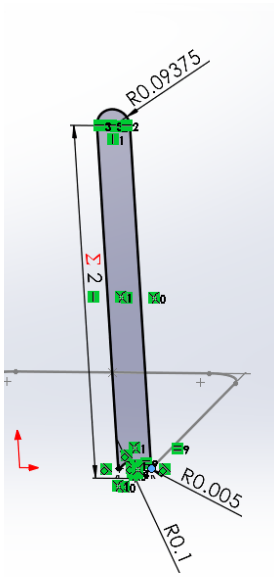
Then add a 0.005” radius fillet to both of the bottom corners.



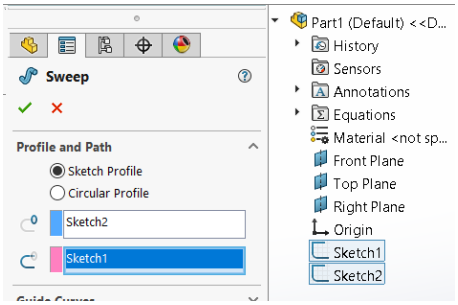


Finally, make the highlighted point coincident with sketch one as shown.





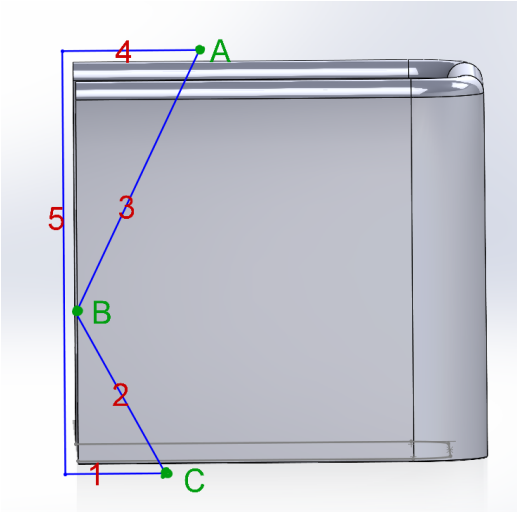
Now that both of the sketches are complete, you can make the swept base feature. Use sketch one as the path and sketch two as the profile.



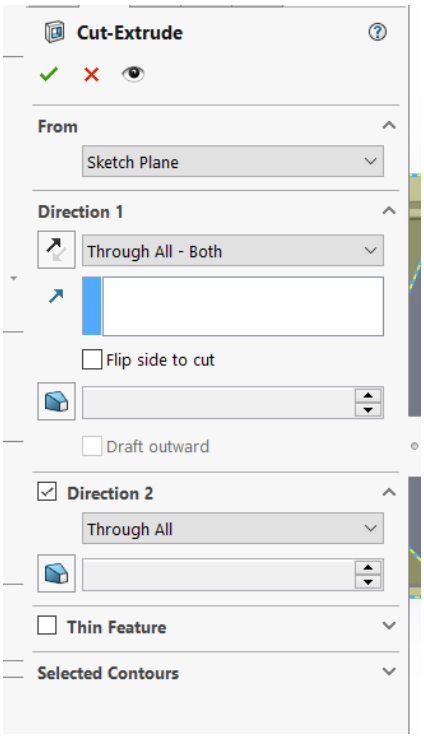
Step 3: Extruded cut

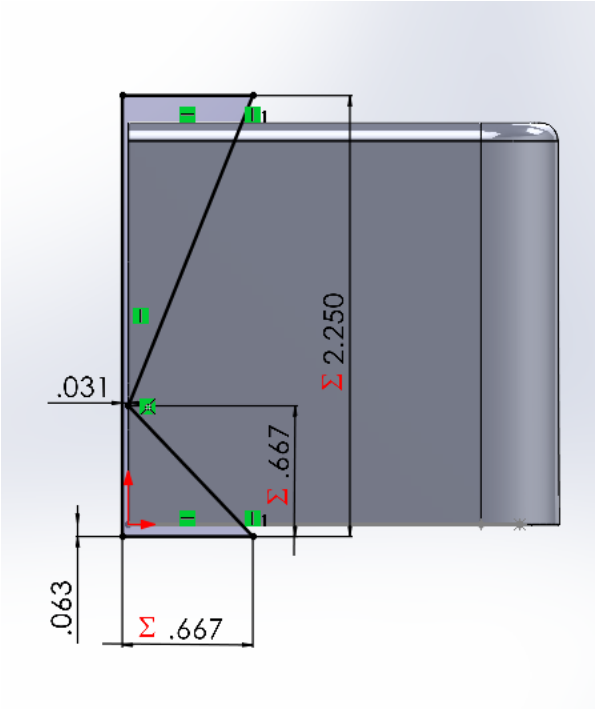
Next add the extruded cut as shown. Start by selecting the extruded cut feature and choosing the right plane for the sketch plane. Now make the following shape using a series of lines. Add the following relations and dimensions:

- Lines 1 and 4 are both Horizontal
- Line 5 is Vertical
- Point B is Coincident with the front plane
- Point A and C are Vertical
- From Line 1 to Top Plane = 1/16”
- From Line 1 to Point B = height/3
- Length of Line 1 = Length/3
- Length of Line 5 = Height + ¼”
- From Line 5 to Point B = 1/32”



Your sketch should now look something like this, you can now exit the sketch. To finish the extruded cut, select the Through All - Both option for the cut dimension.

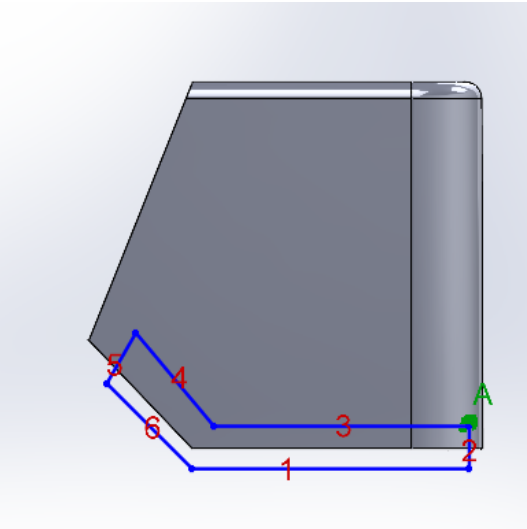




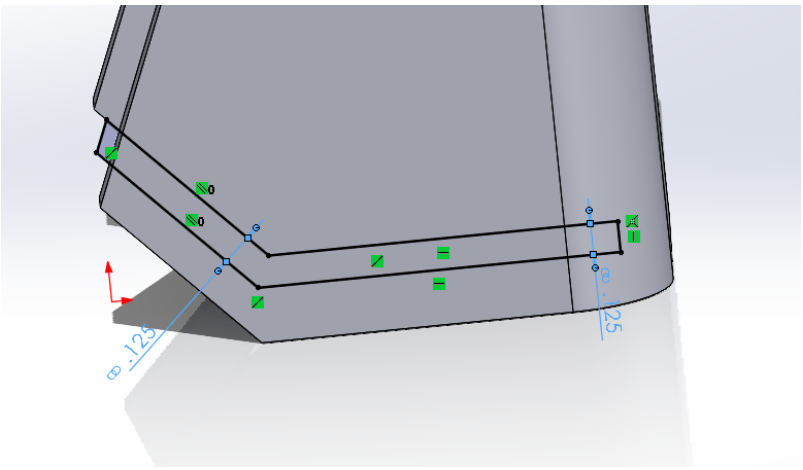
Step 4: Extruded Boss

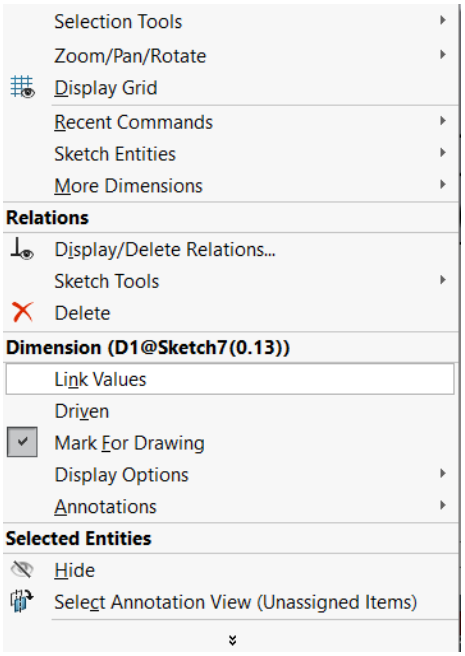
Now use the extruded boss to make the bottom and front edge. Start by selecting the Extrude Boss/Base feature and choose the Right Plane as the sketch plane, use Lines to draw the pictured shape, then add the following relations and dimensions:

- Lines 1 and 3 are both Horizontal
- Line 2 is vertical
- Lines 4 and 6 are parallel
- Line 1 Collinear with Top Plane
- Line 6 Collinear with the corresponding front edge
- Line 4 parallel with the corresponding front edge
- Line 5 Collinear with top front edge
- Point A Coincident with the inner back edge
- Line 1 to Line 3 = ⅛”
- Line 4 to Line 6 = ⅛”

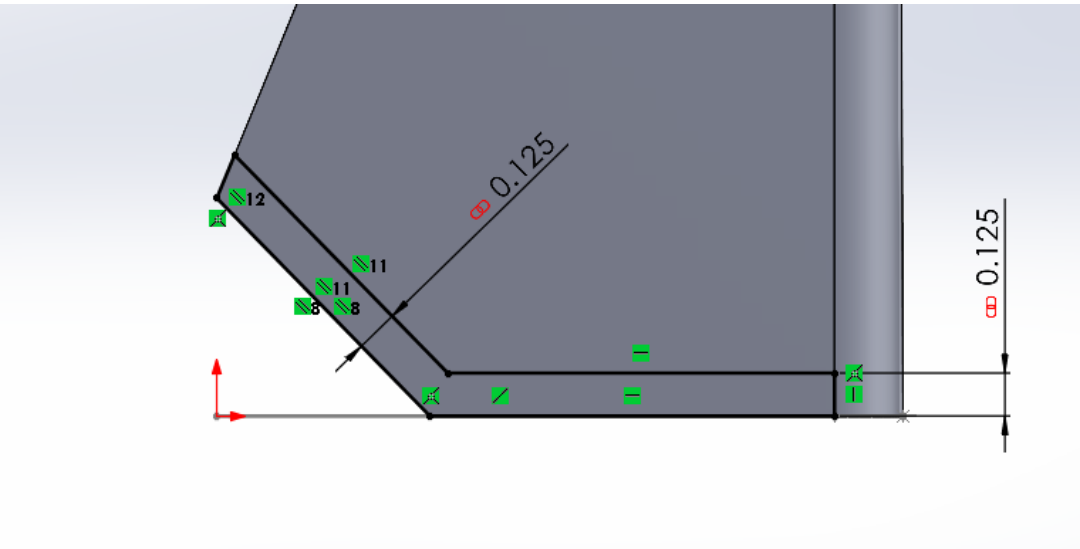


To link the base thickness dimensions, CTRL and select both of the ⅛” dimensions then right click and select link values. Name the linked dimension Base Thickness.

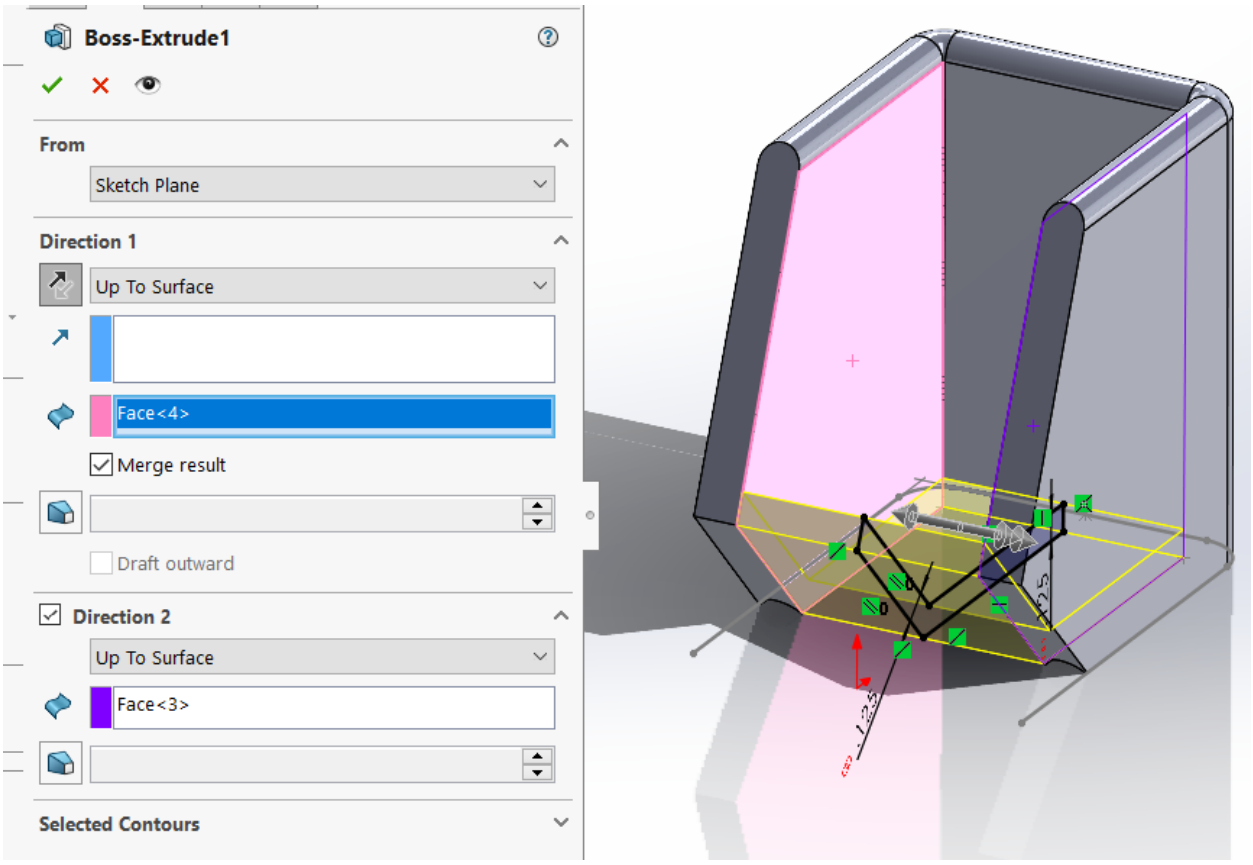




Your finished sketch should look something like this, you can now exit the sketch

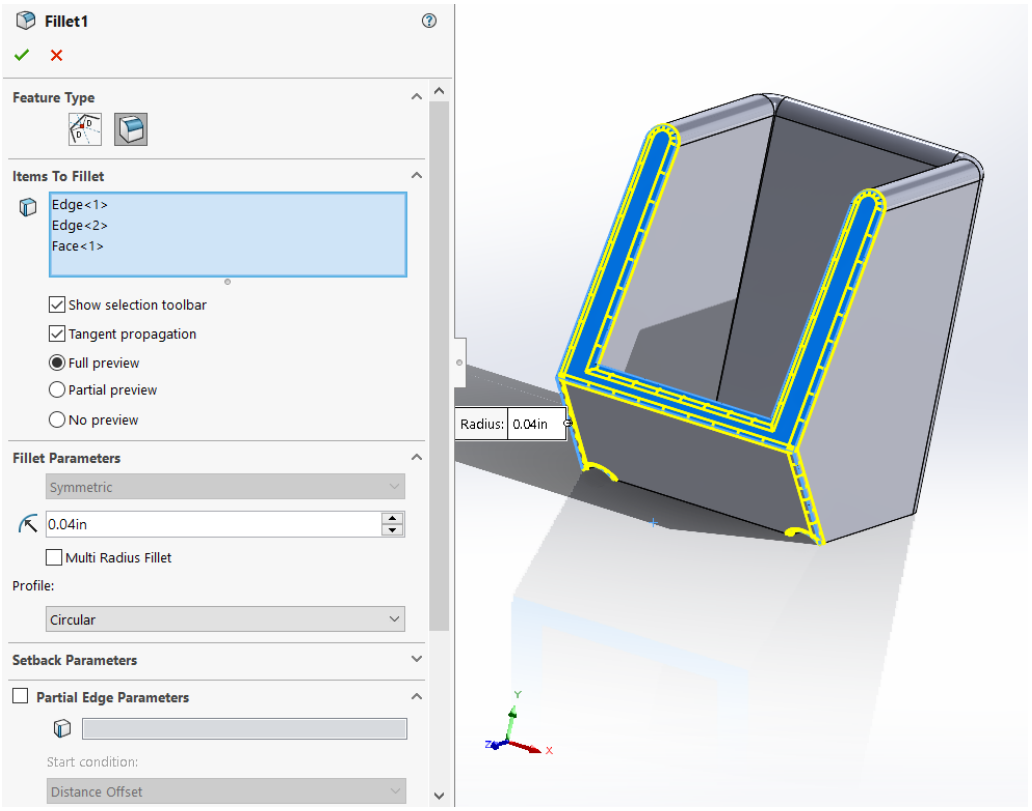


To finish off the extruded boss, choose Extrude to Surface for both directions and select the inner faces of the bin as shown

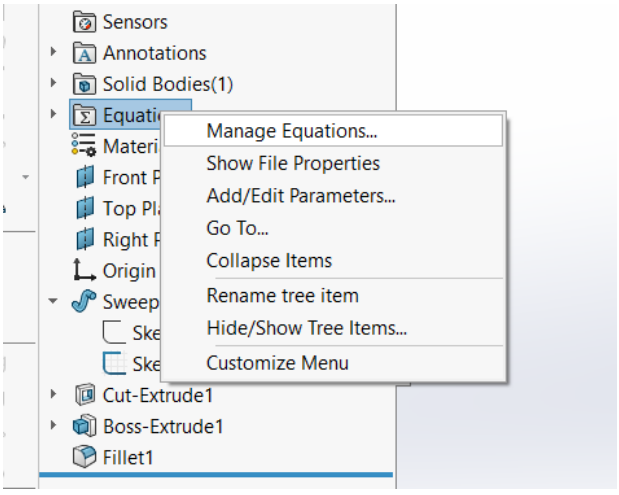


Step 5: Fillets

The last feature you'll need to add is a fillet, this feature can be used to smooth sharp internal or external corners of parts. Select the Fillet feature, then select the top-front face and two front edges. Set the fillet radius as 0.04in and click ok.



You can change the size of the part by right clicking on equations and selecting manage equations



Now that the part is fully modeled, save the part file using the naming convention **Onboarding_2022_Firstname_Lastname** and send it to Stevie.