

Fusion Installation

1. Navigate to <https://engineering.purdue.edu/ECN/Support/KB/Docs/AutodeskFusion360>
2. Follow the instructions to create an Autodesk account
 - At the bottom of the page is an option to install Autodesk software
 - Select the 2nd option "Fusion"
3. Once Fusion is installed, sign-in using the Autodesk account you created and activated

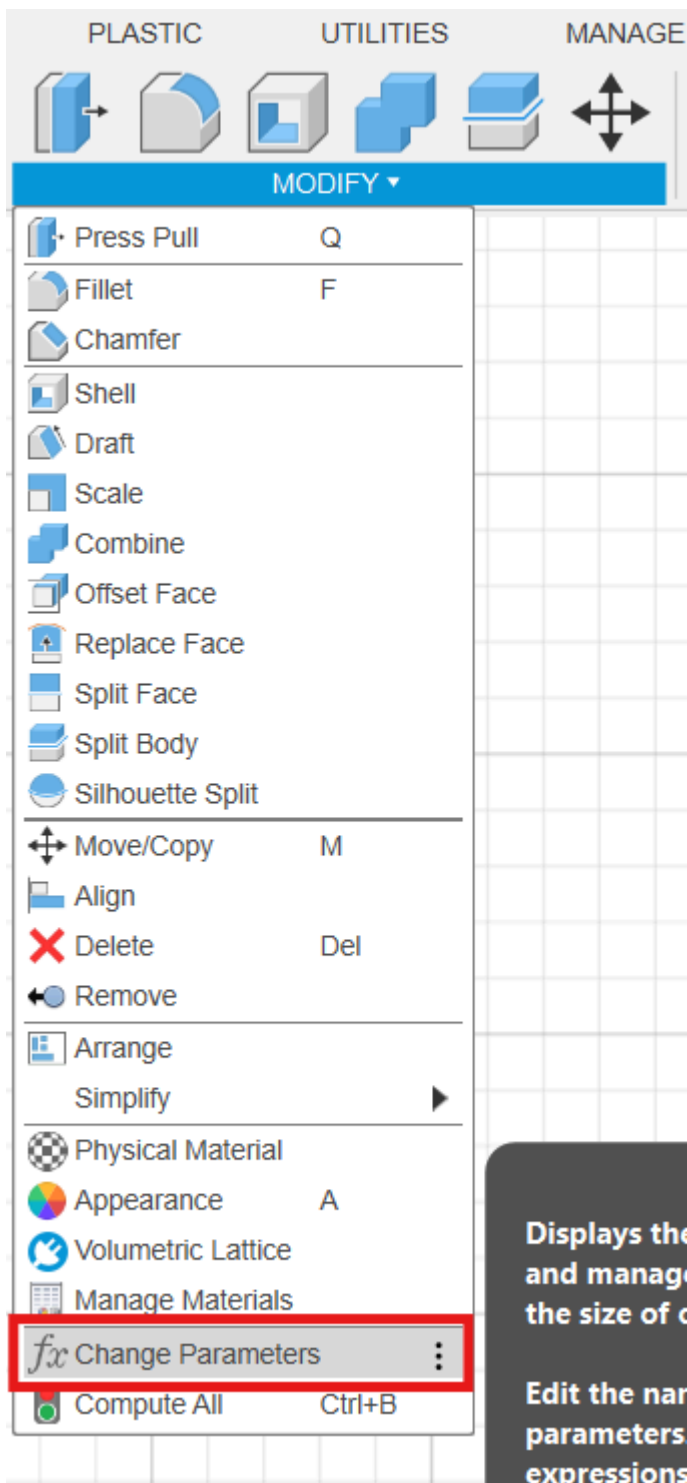
Palm Design

Step 0

- Open Fusion on your computer
 - This should open a blank, untitled workspace

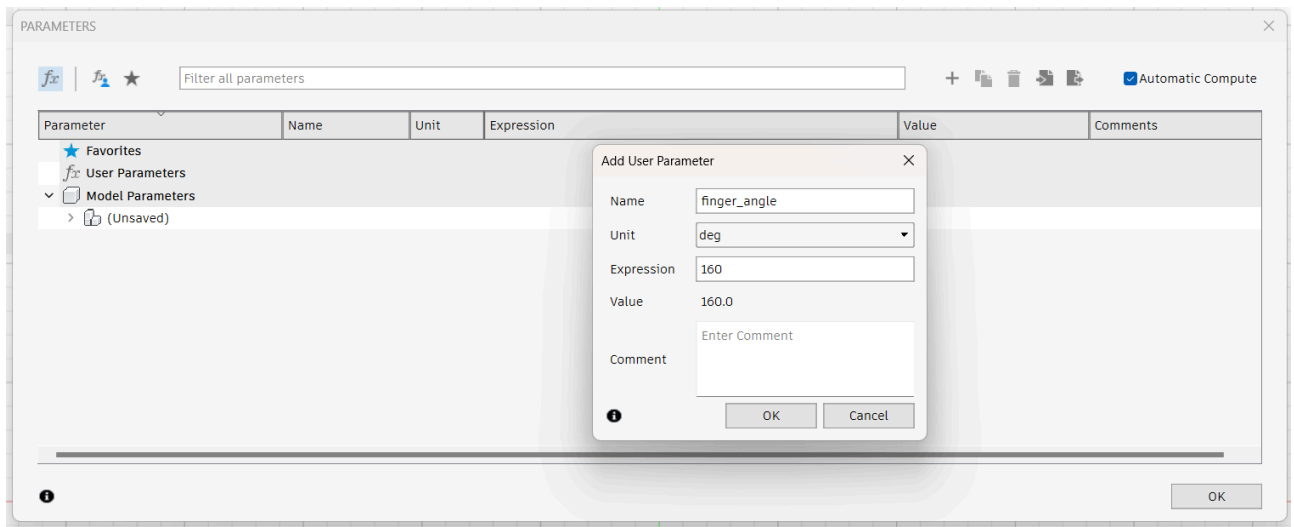
Step 1

- Within the "Modify" menu, select "Change Parameters" to add variables/parameters to the workspace



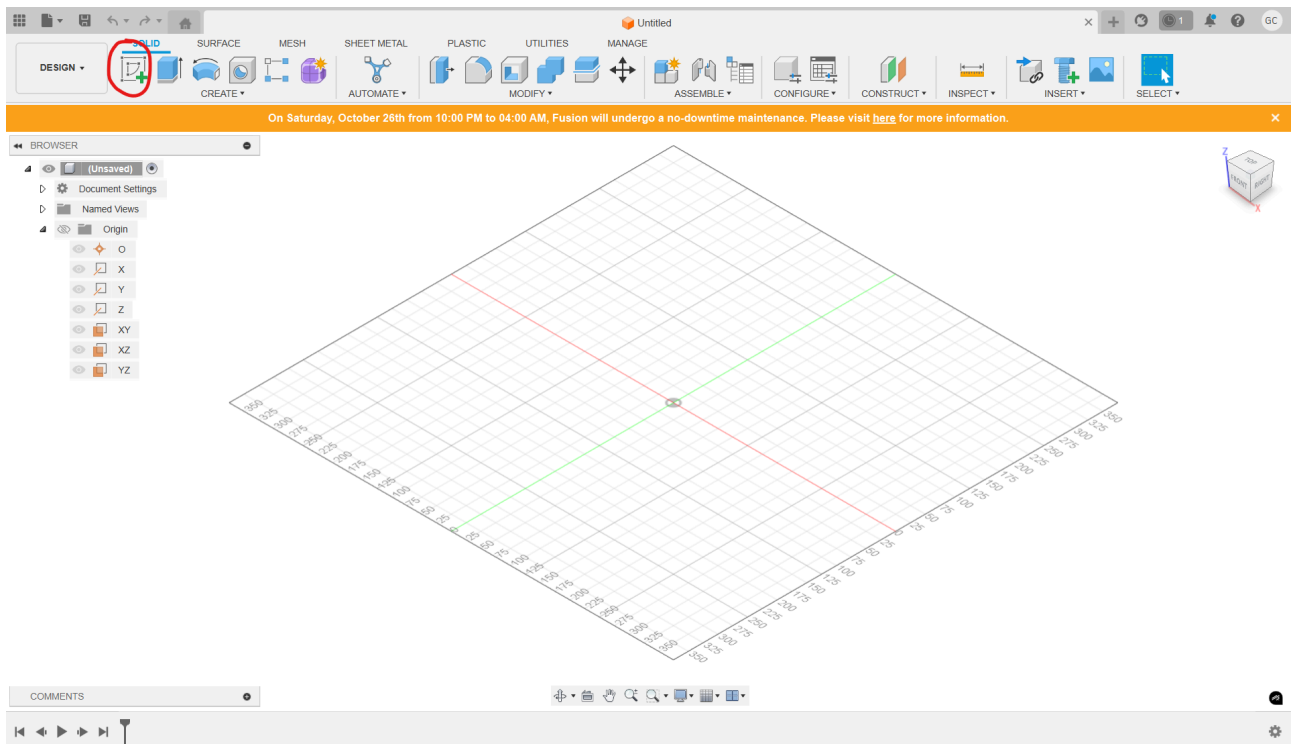
Step 2

- Click the "+" button and make parameters for the following quantities
 - scalefactor: 1 (unitless)
 - finger_angle: 160 degrees
 - knuckle_width: 25mm * scalefactor
- The definition of knuckle_width should include scalefactor so that in the model design, knuckle_width won't need to be multiplied by scalefactor



Step 3

- Once all parameters have been made, exit the menu by pressing "OK"
- Click on the "Create Sketch" button

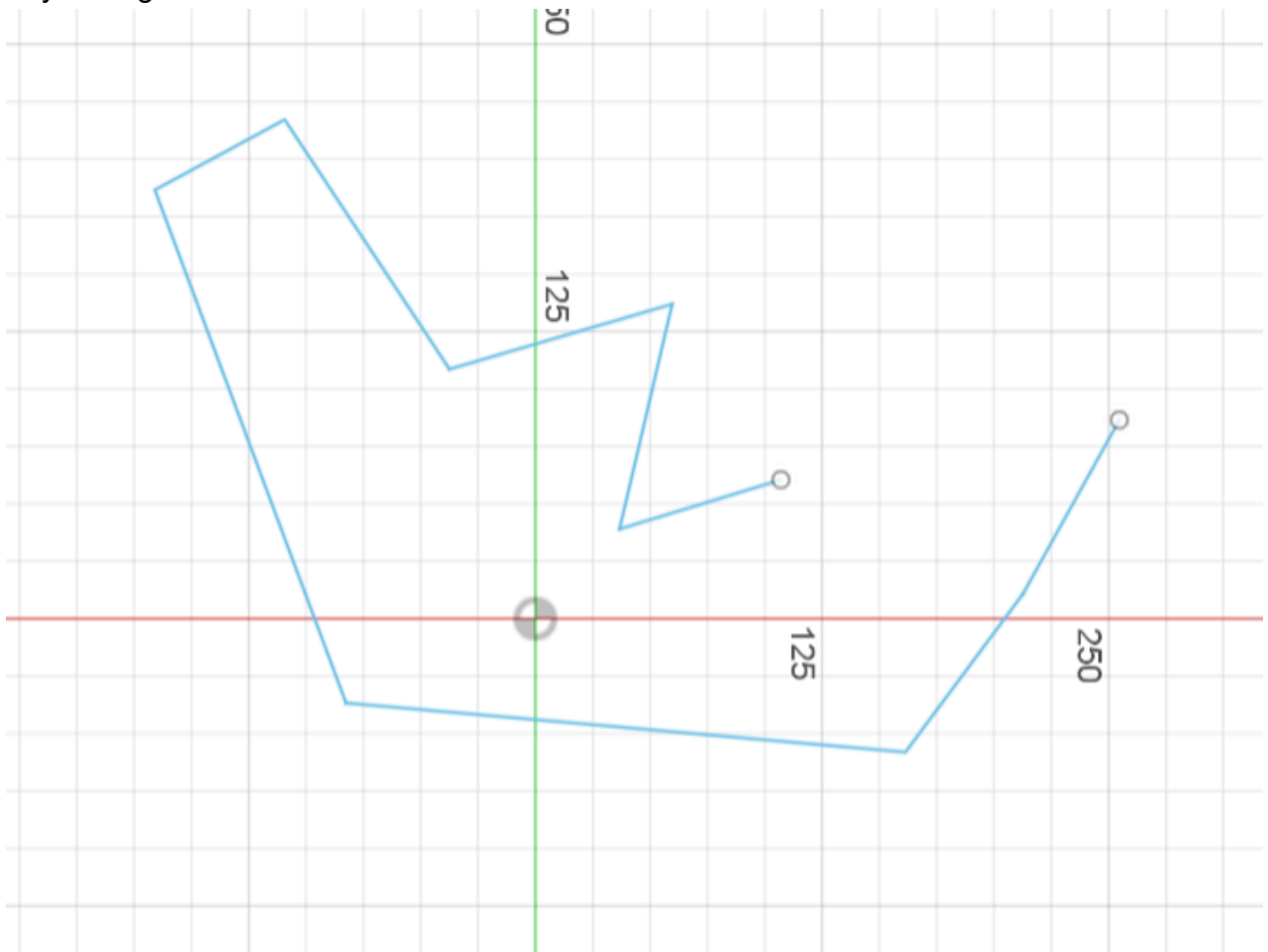


Step 2

- 4



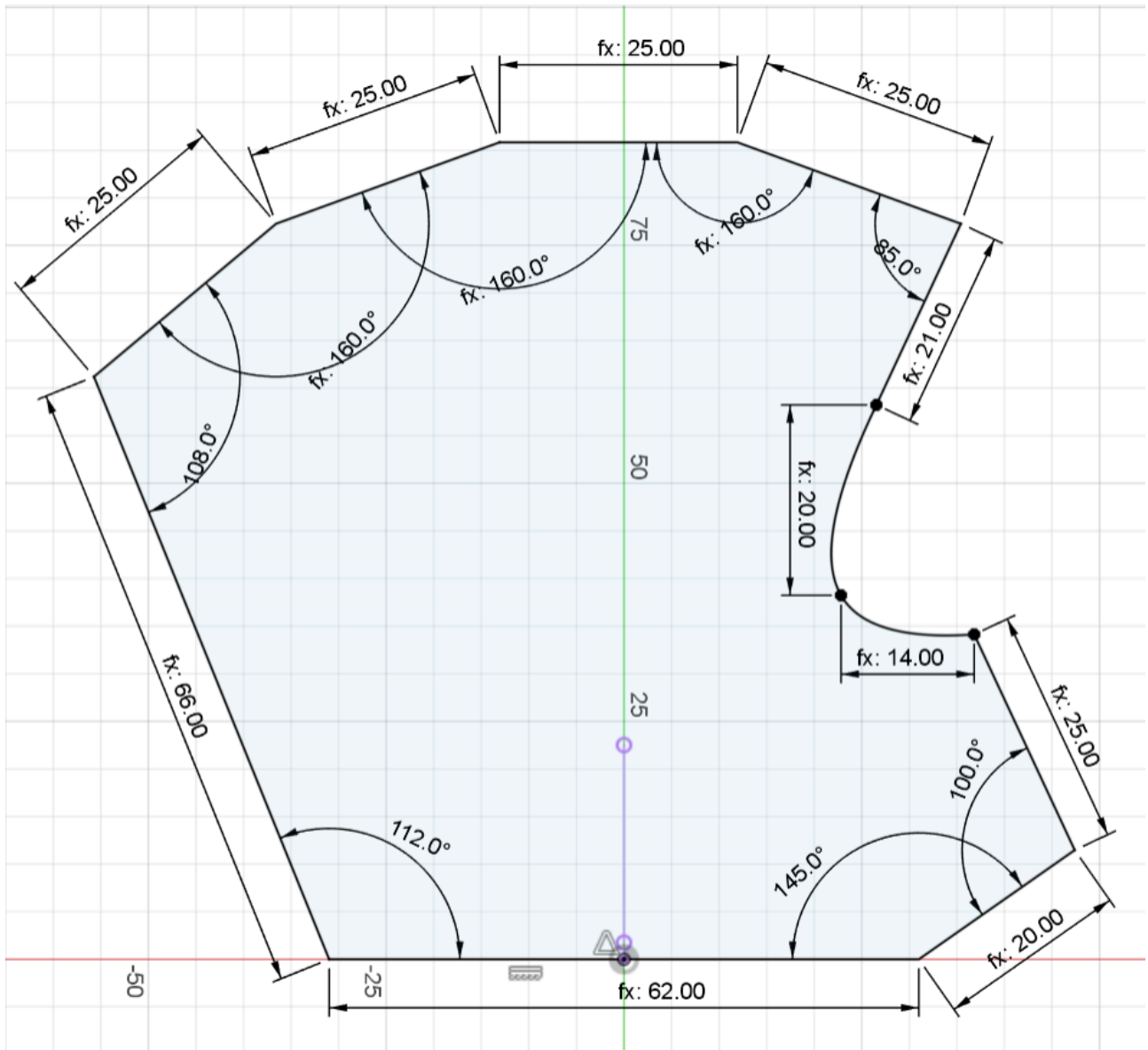
- The above picture is not correct, just a general depiction of random lines that don't have any auto-generated constraints



Step 5

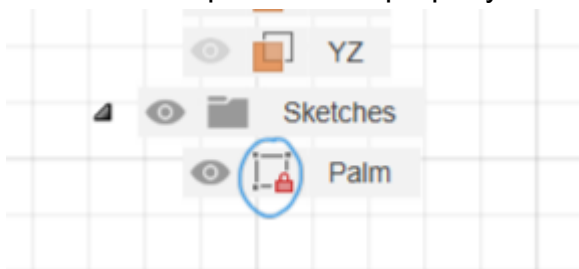
- Match the following dimensions
- Use the horizontal/vertical constraint to make the bottom line parallel to the x-axis
- Use the midpoint constraint to center the sketch on the x-axis
- Remember to use the variables defined earlier
 - All measurements that are 25mm should use the `knuckle_width` variable
 - All 160 degree measurements can use the `finger_angle` variable
 - All dimensions marked with "fx:" should be multiplied by `scalefactor` (other than those made with `knuckle_width`)

- You will need to add a spline between two straight lines on the left to complete the shape



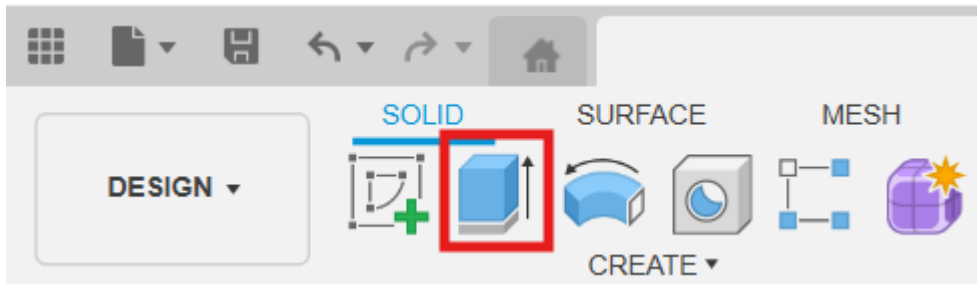
NOTE

- When you are finished, you should rename the sketch based on its purpose
- There should be a lock icon next to the sketch as well to indicate its fully constrained
- All lines should also be black to indicate they are fully constrained
- To test if the design is scalable, change the value of "scalefactor" in the parameters menu and see if the palm scales properly

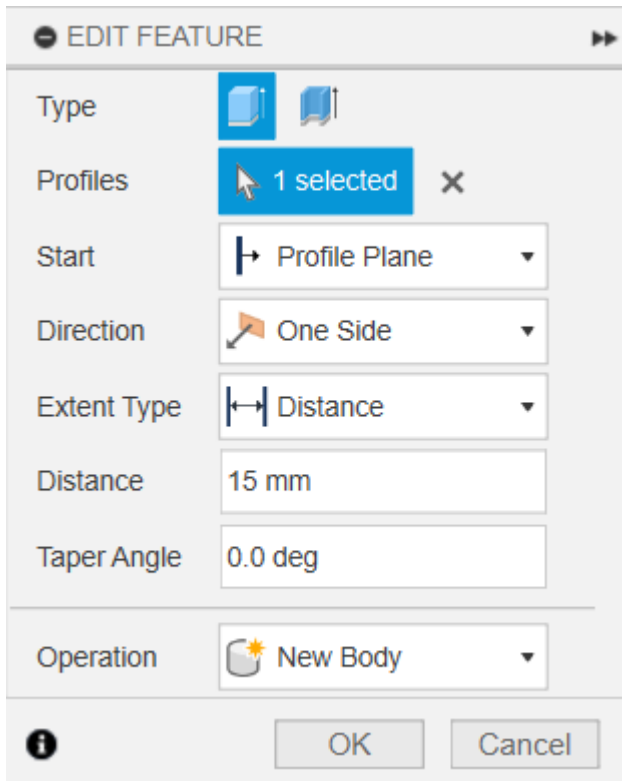


Step 6

- Click on the "Finish Sketch" button in the top right corner
- Click on the "Extrude" tool



- Enter "15mm" for the distance, everything else should default to the correct values



Step 7

- Save your current model after creating a new project
 - This could be done by using the keyboard shortcut "CTRL-S"

Finger Design

- Create a new part by adding a tab in Fusion near the top of the screen

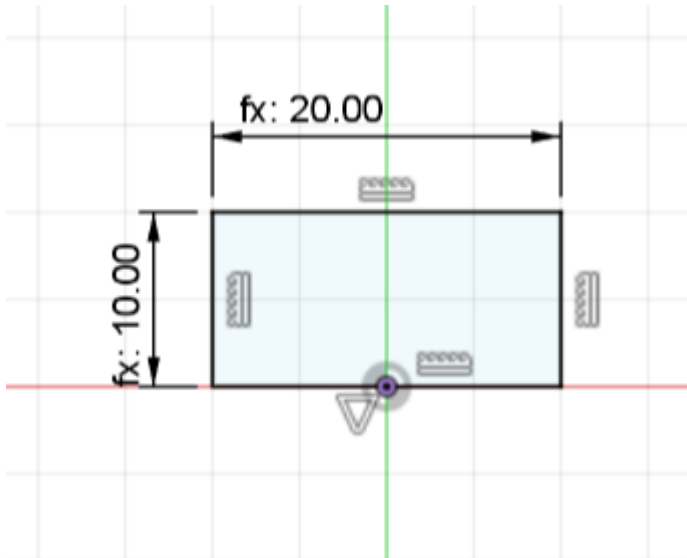
Step 1

- Add new variables to this part following the same process as before
 - finger_width: 20mm
 - finger_height: 10m

- finger_length: 30mm

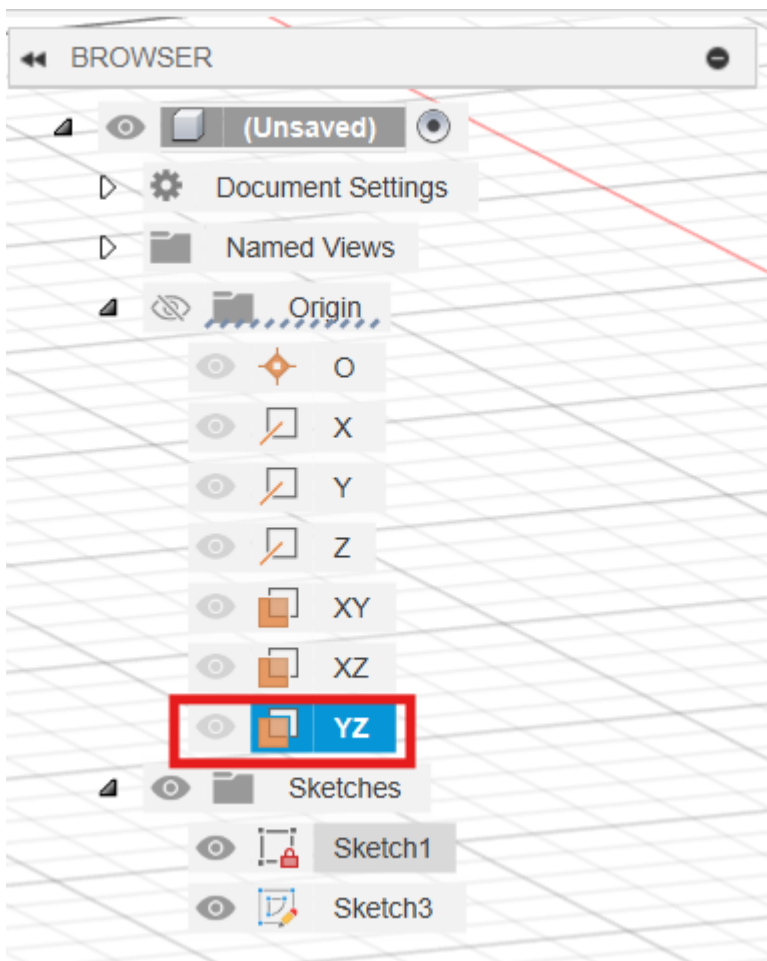
Step 2

- Create a sketch in the "front" or XZ plane
- Use the rectangle tool, the midpoint constraint, and the variables defined earlier to obtain the below sketch



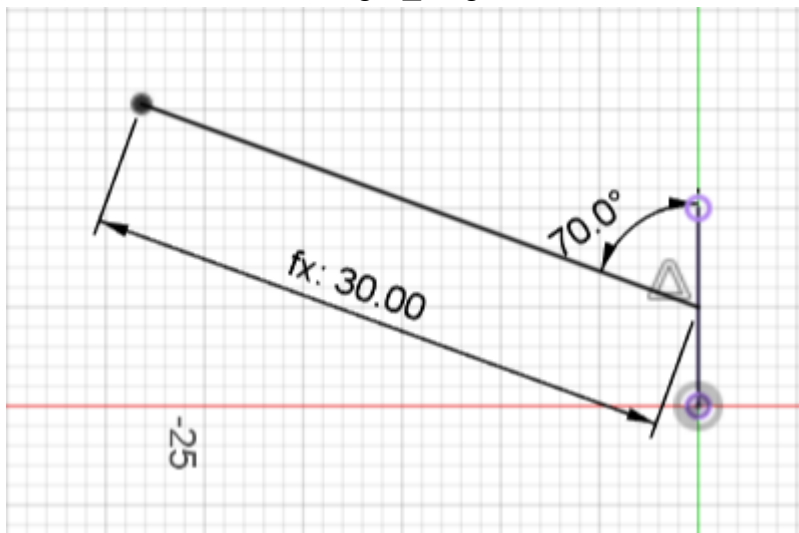
Step 3

- Once the sketch is confirmed by pressing the "Finish Sketch" button, click on the YZ plane on the left of the screen and then create a new sketch



Step 4

- Create a line constrained to the midpoint of the rectangle following the above dimensions
- Remember to use the `finger_length` variable

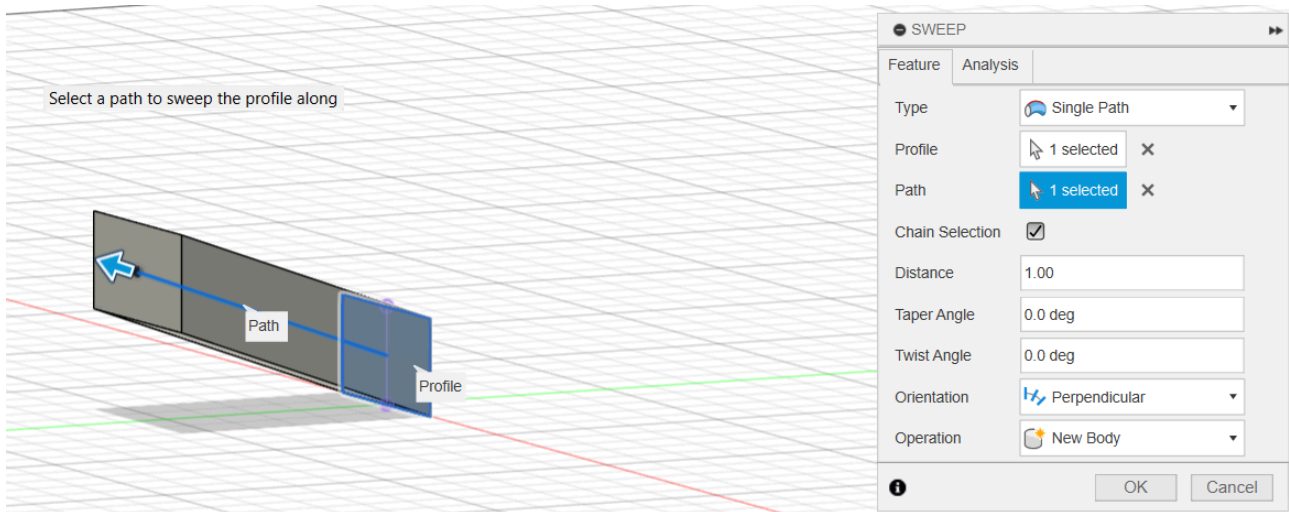


Step 5

- Once the sketch is confirmed, select sweep from the options near the upper left of the screen

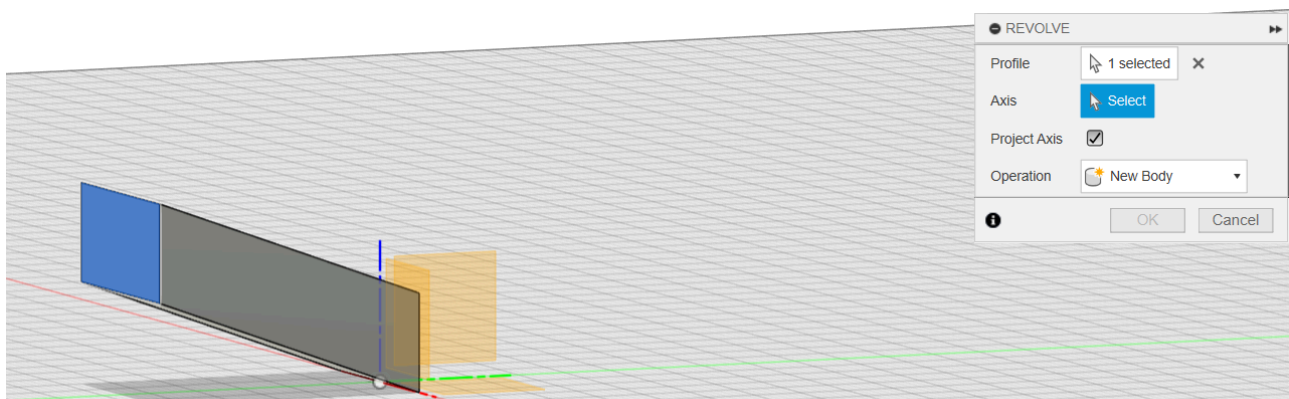


- Once sweep is selected, choose the face of the original rectangle as the profile and the line from the second sketch as the path

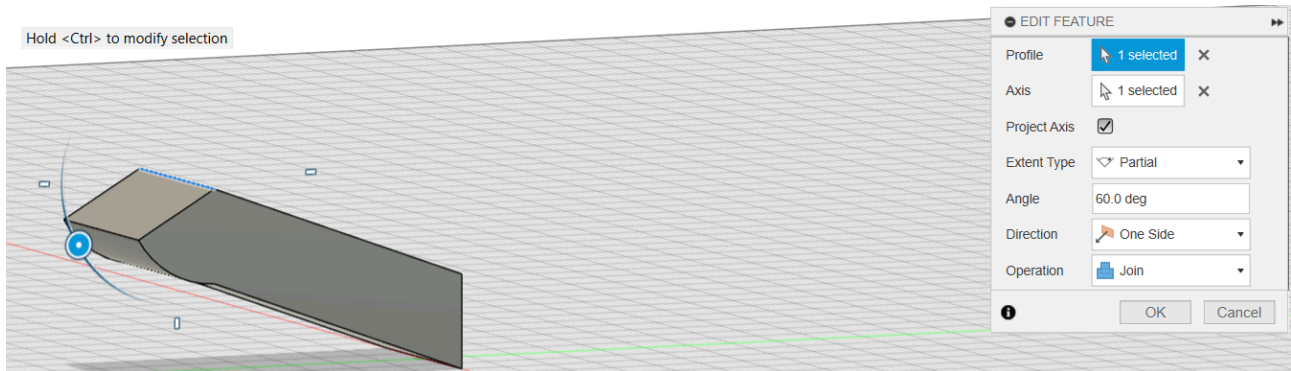


Step 6

- Once the sweep is confirmed, select the face created by the sweep and select "Revolve" from the same menu that included "Sweep"



- The axis should be the top line of the rectangular face chosen to be revolved



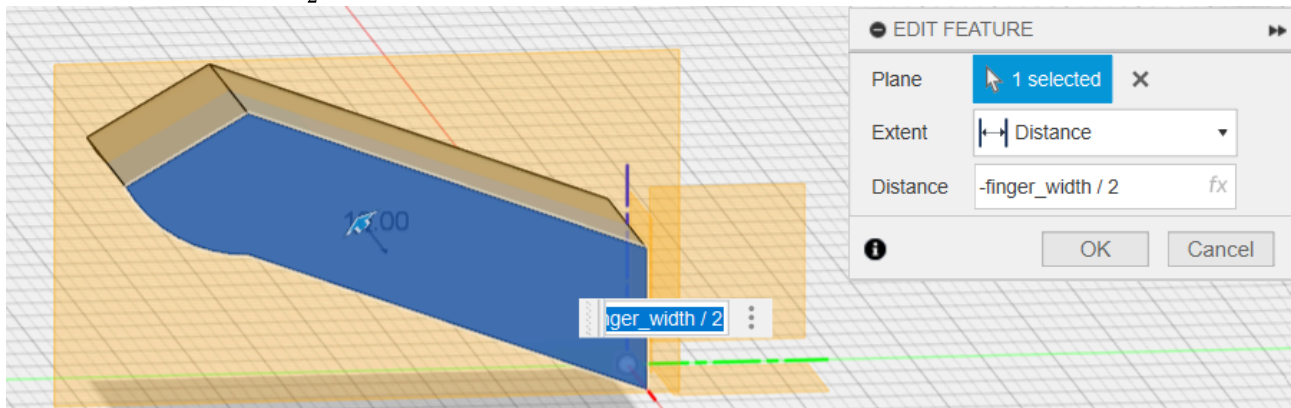
- The angle may need to be adjusted from 360 degrees to 60 degrees
- Confirm the revolve once the model looks like the above image

Step 7

- Create a new offset plane under the construct menu



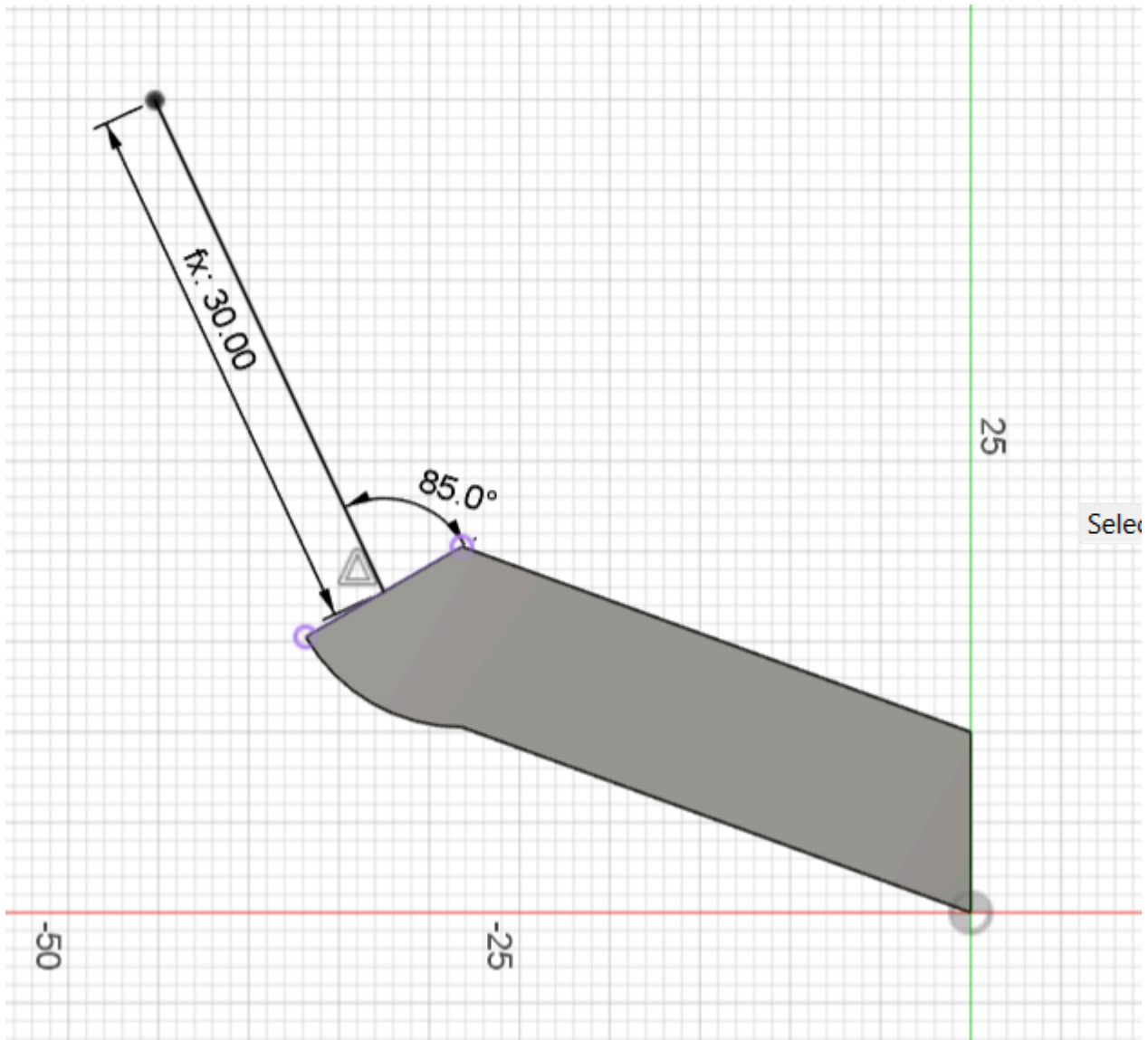
- Select the side of the body that was swept/revolved
- Enter the offset as $-\frac{1}{2}$ finger_width



Step 8

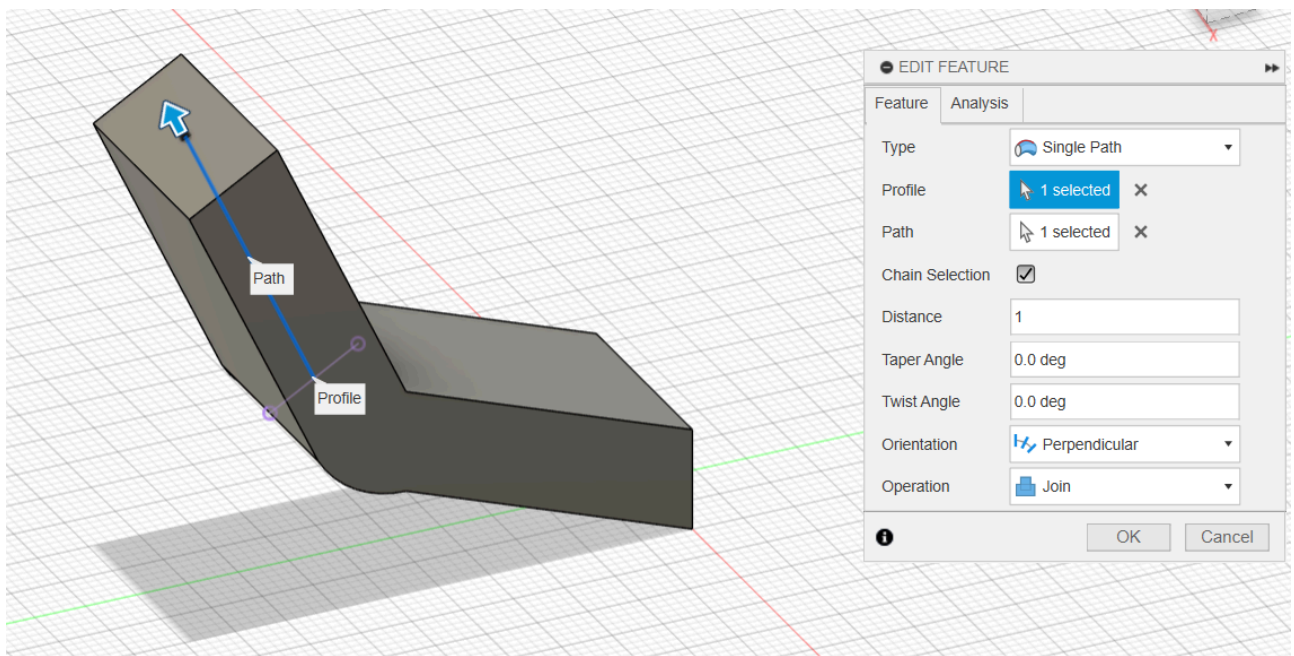
- Create a sketch on the new plane matching the below specifications

- Ensure to use the finger_length variable



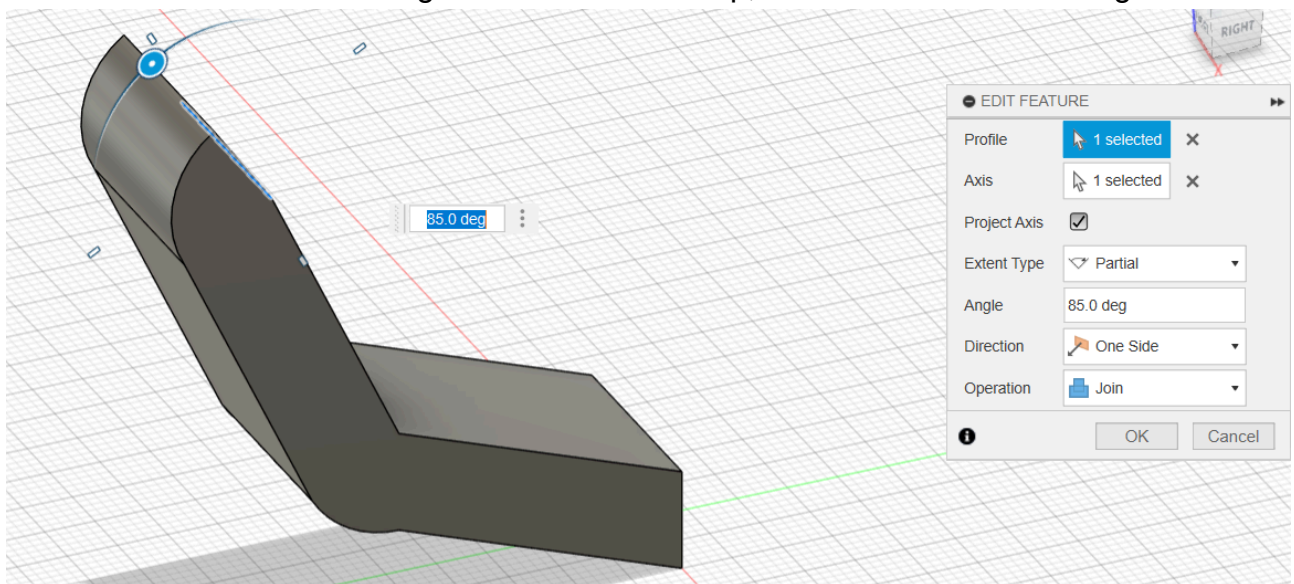
Step 9

- Perform another sweep using the line created in step 15 and the face of the revolve created by step 14



Step 10

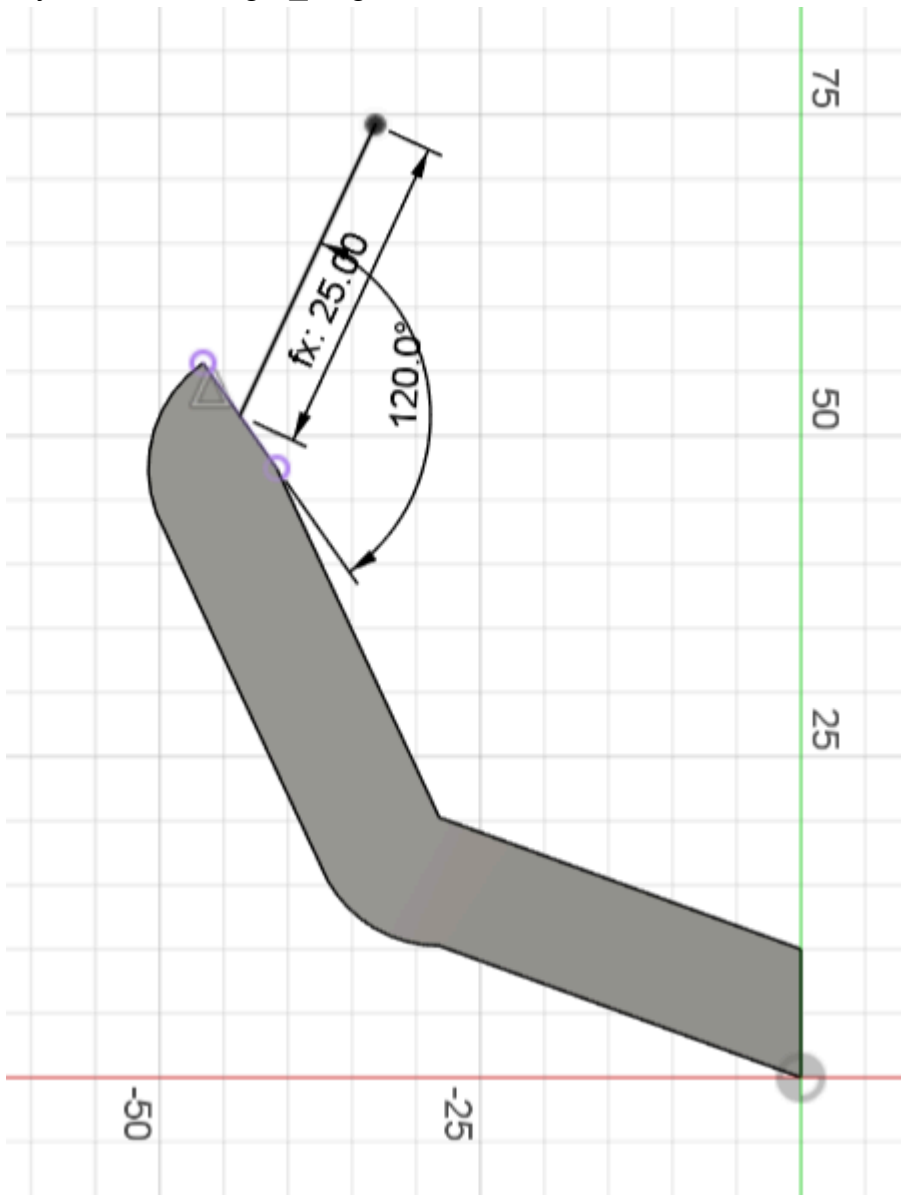
- Perform another revolve using the face of the sweep, this one should be 85 degrees



Step 11

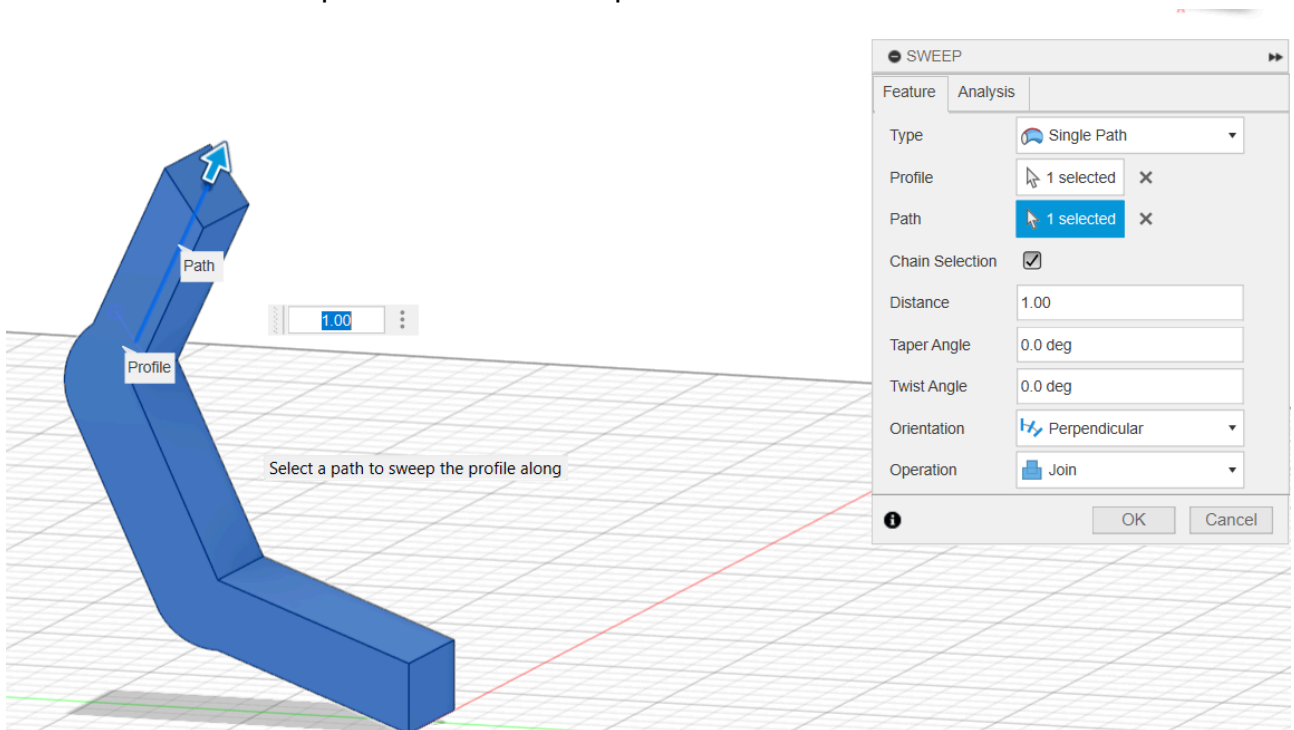
- Using the plane created from earlier, create another line pointing away from the face of the new revolve

- Try to use the finger_length variable to create the distance from the image below



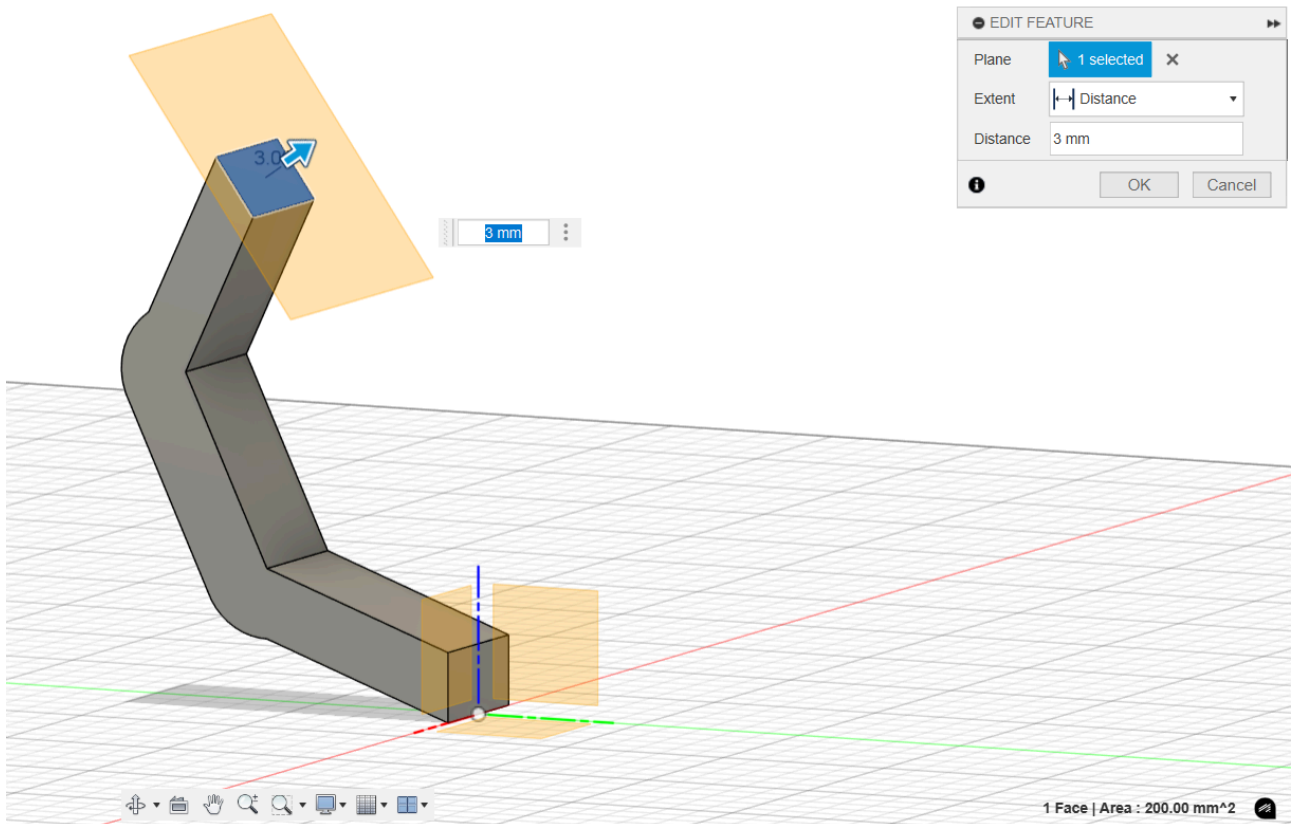
Step 12

- Perform another sweep as described in steps 13 & 16



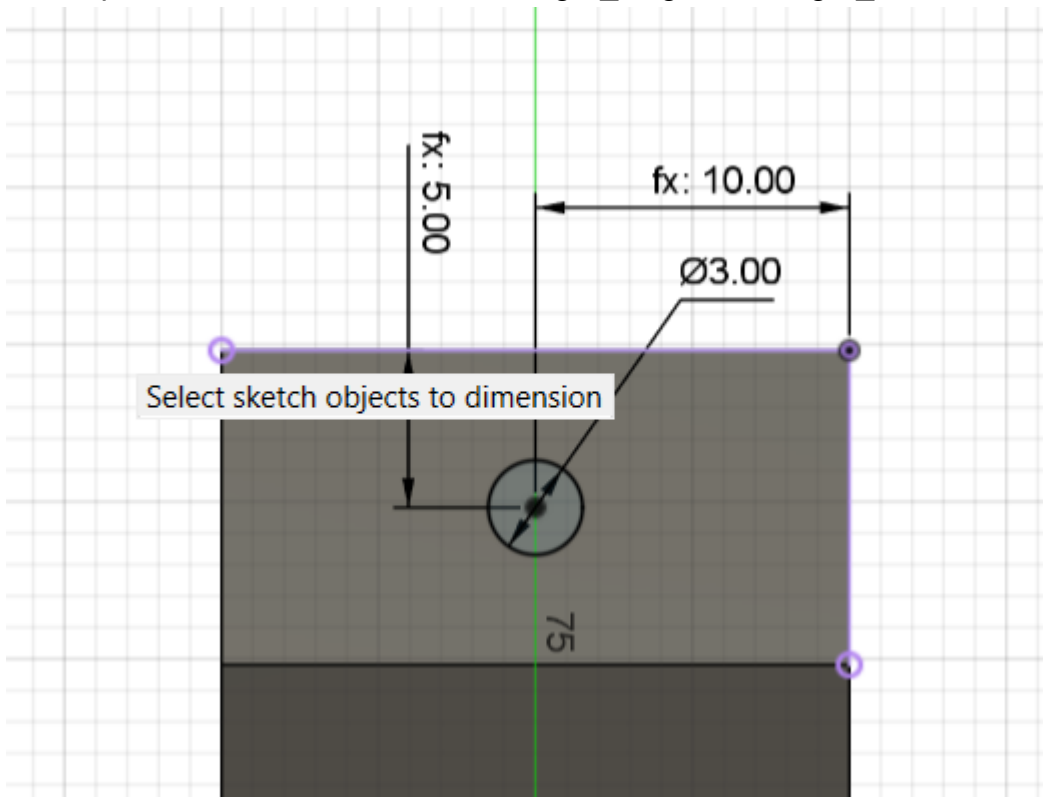
Step 13

- Create a plane 3mm offset from the face of the sweep from step 19



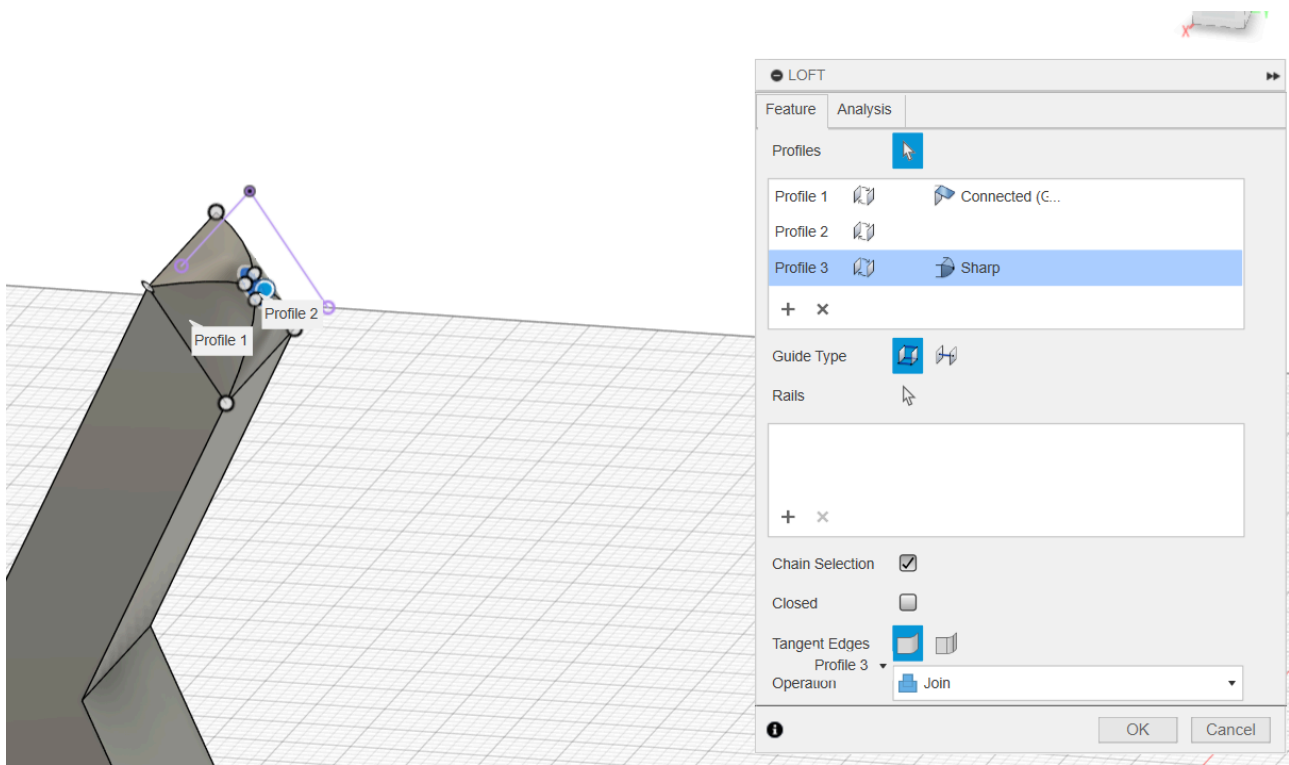
Step 14

- Using the new plane, start a sketch of a circle centered relative to the rectangular face of the sweep from step 19
- To keep the circle centered, use the finger_height and finger_width variables



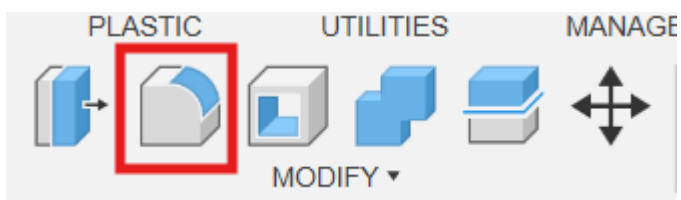
Step 15

- Once, the sketch on the new plane is confirmed, select the "Loft" tool
 - Should be located under the same menu as sweep and revolve
- Select the face of the sweep (rectangle), the outside of the circle, and the circle's center point to create a "fingertip"

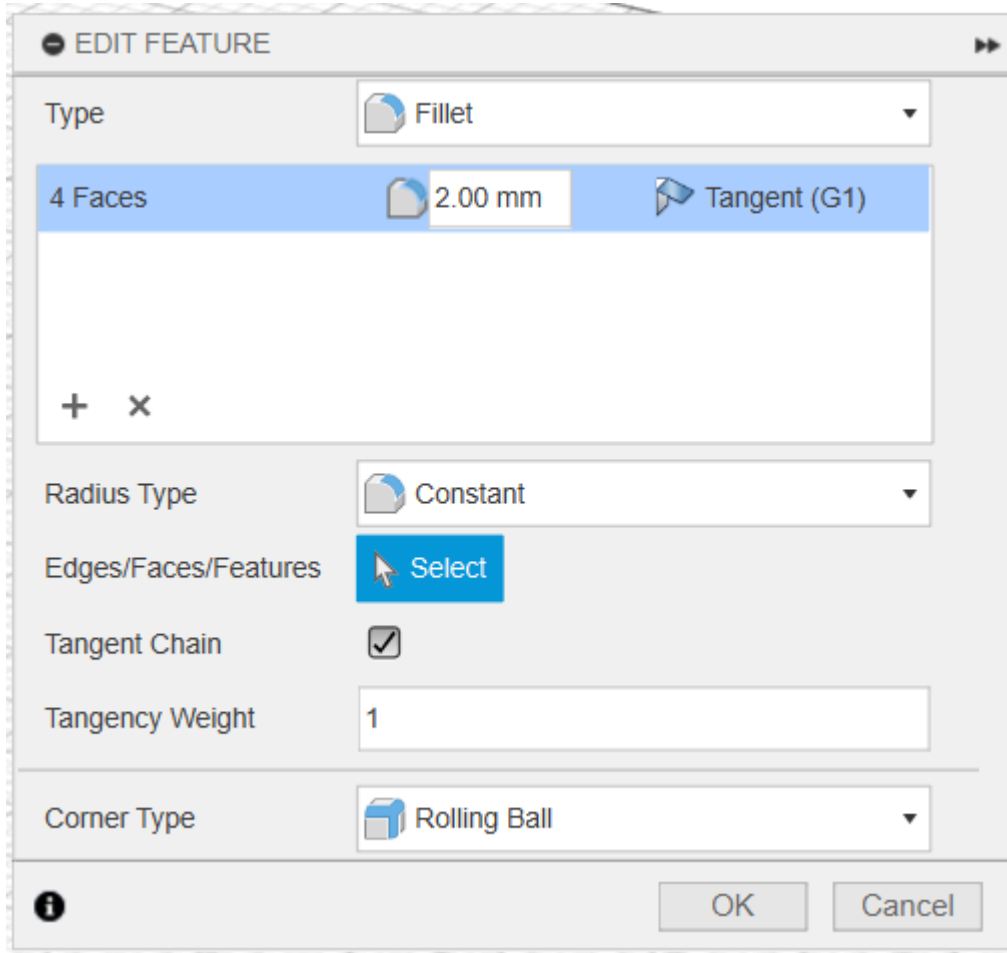
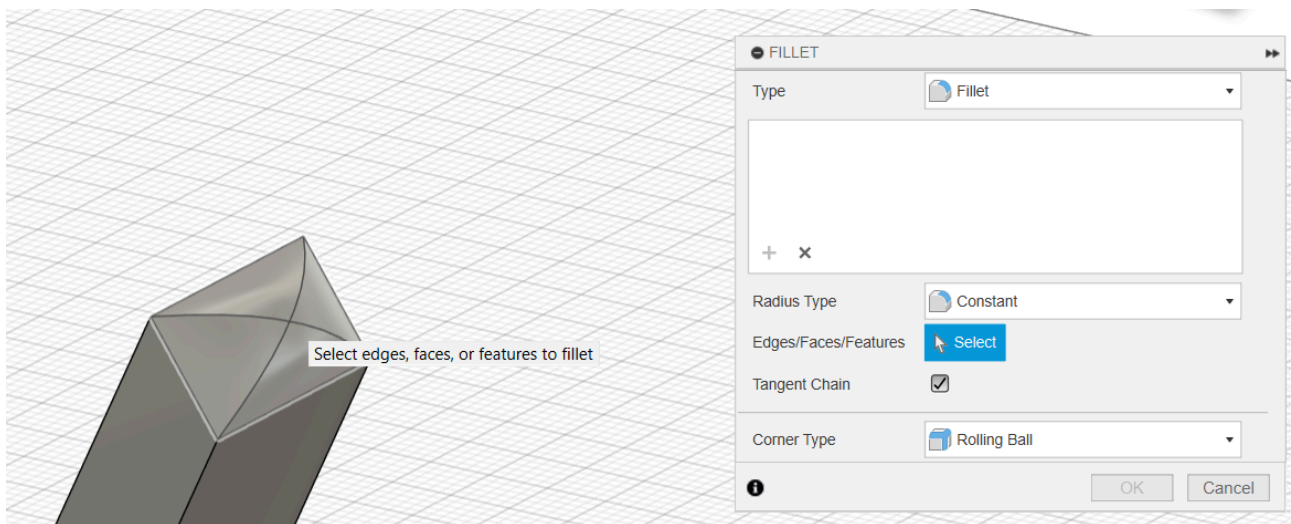


Step 16

- Select the "Fillet" tool

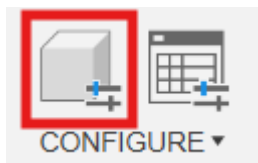


- In the normal fillet (not rule or full) click on the middle of the loft to select all faces and set the radius to 2mm



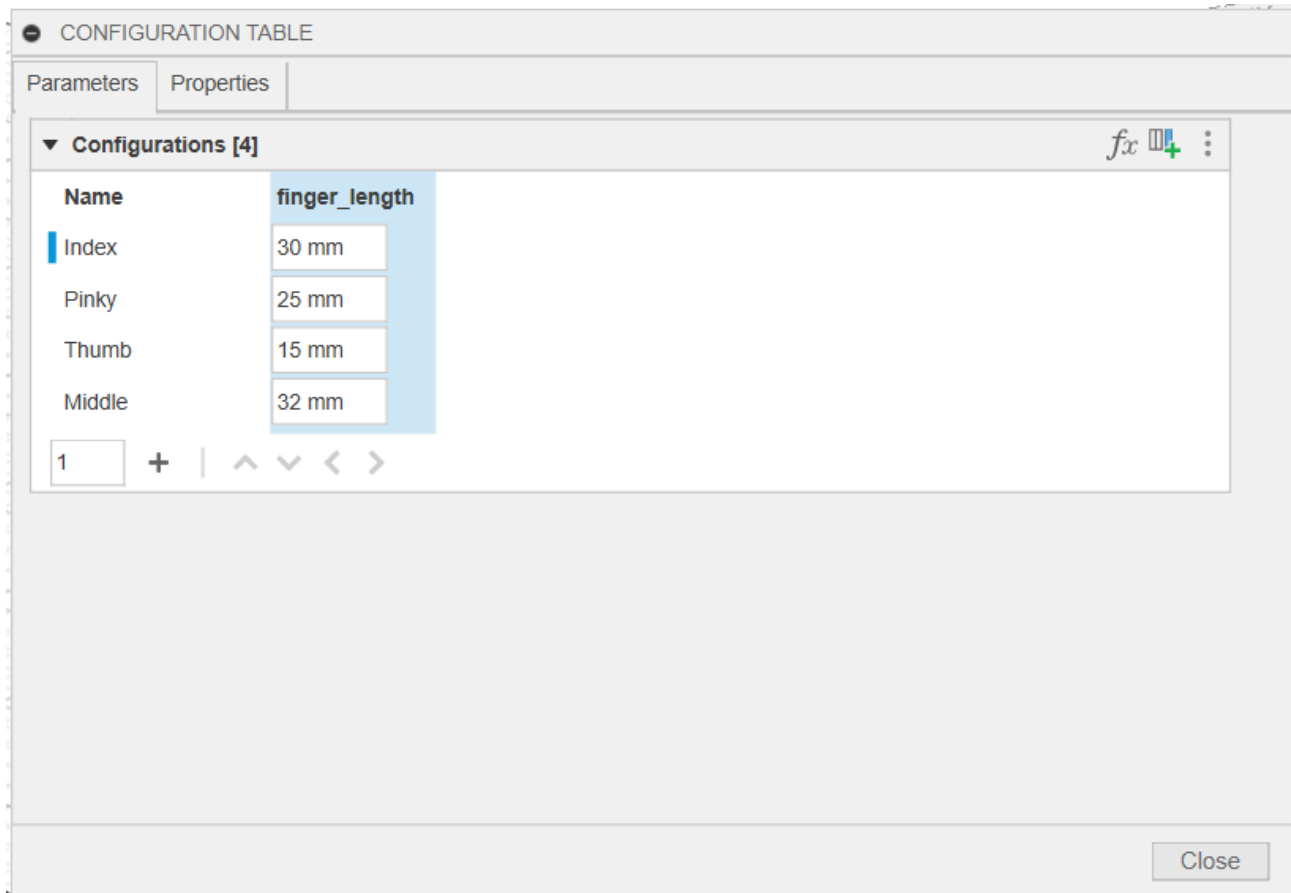
Step 17

- Save this new part as "Finger" in the same project once the fillet is complete
- Now click on "Configure" in the toolbar



- Add the following finger configurations

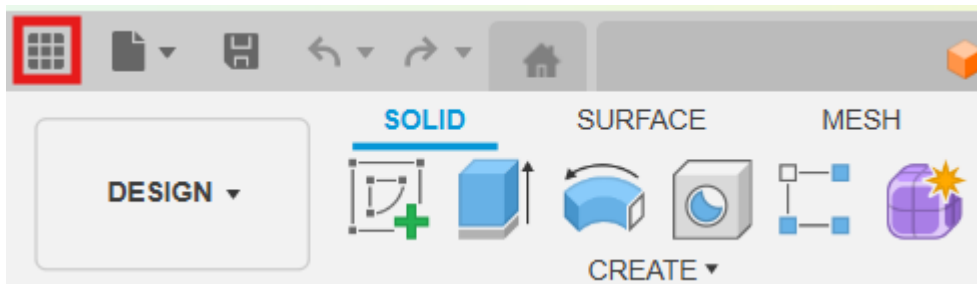
- Use the fx in the upper right to add the finger_length variable to customize each configuration



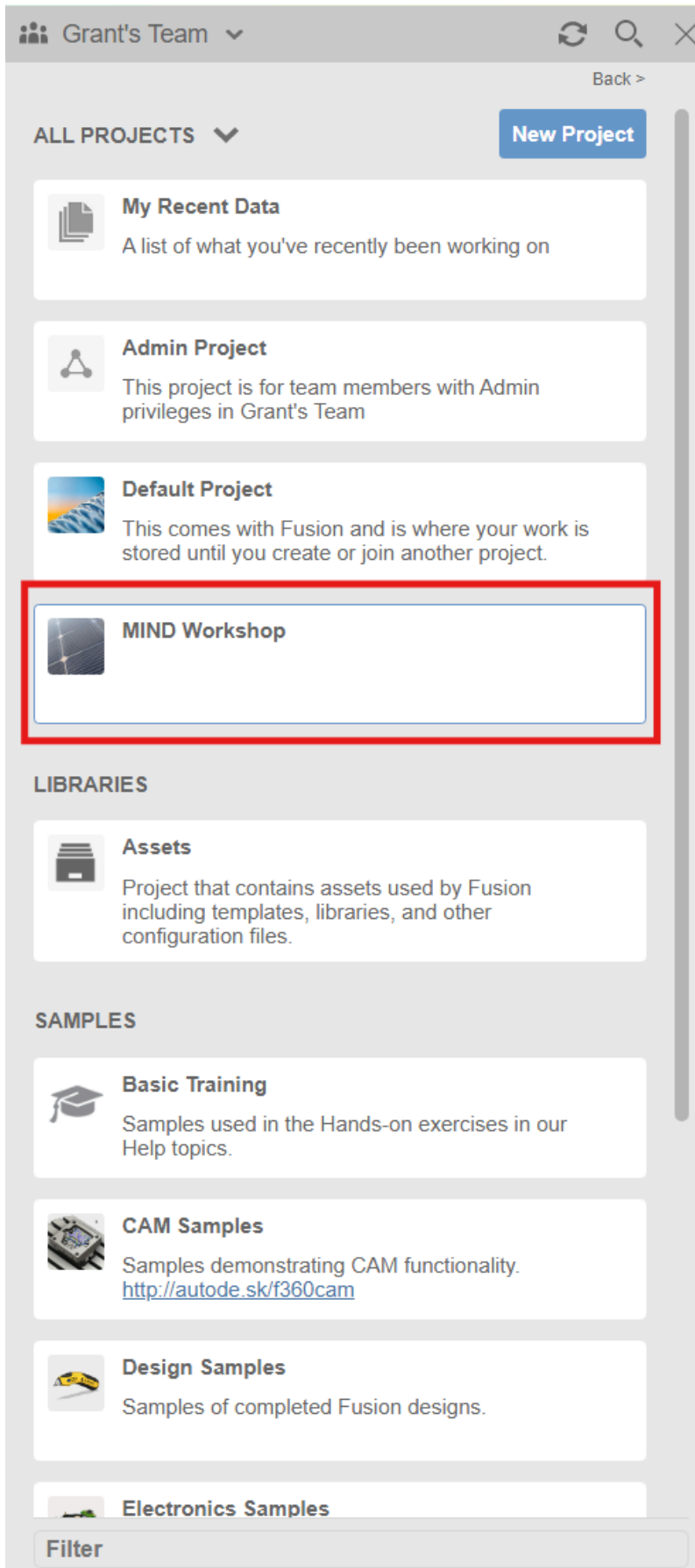
Hand Assembly

Step 1

- Click on the 3x3 dot matrix in the upper left corner



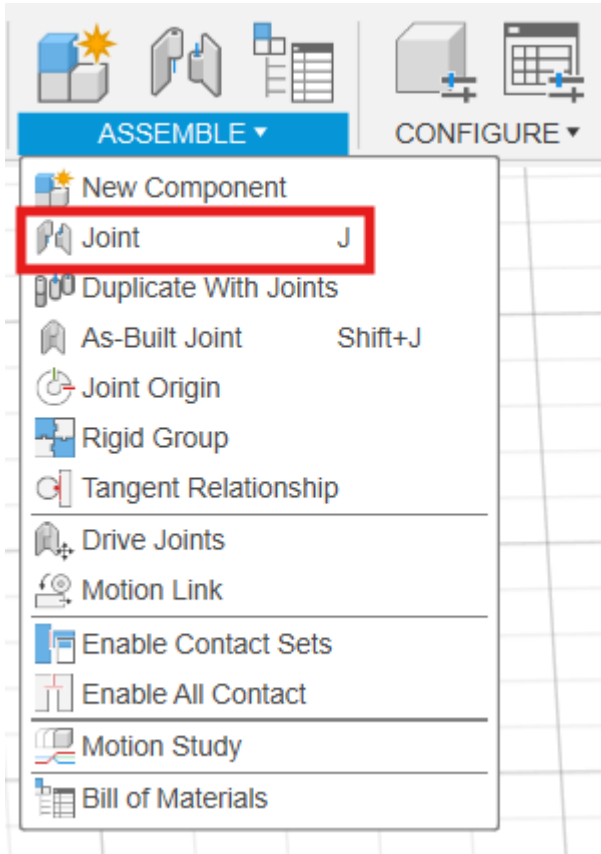
- Select the project you made



- From here drag in the palm model and 1 finger with whatever configuration you want to attach first

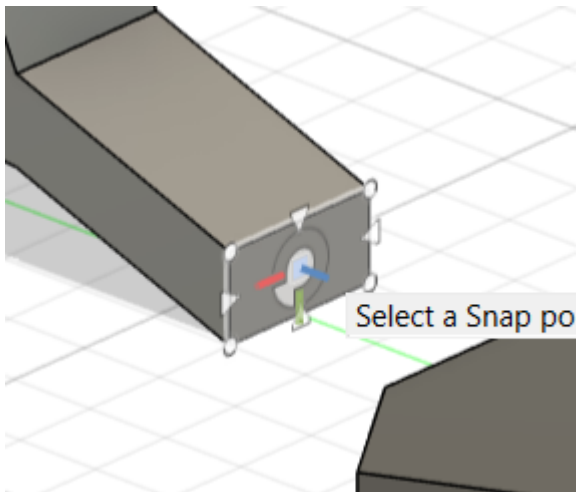
Step 2

- Under the "Assemble" menu click on "Joint"



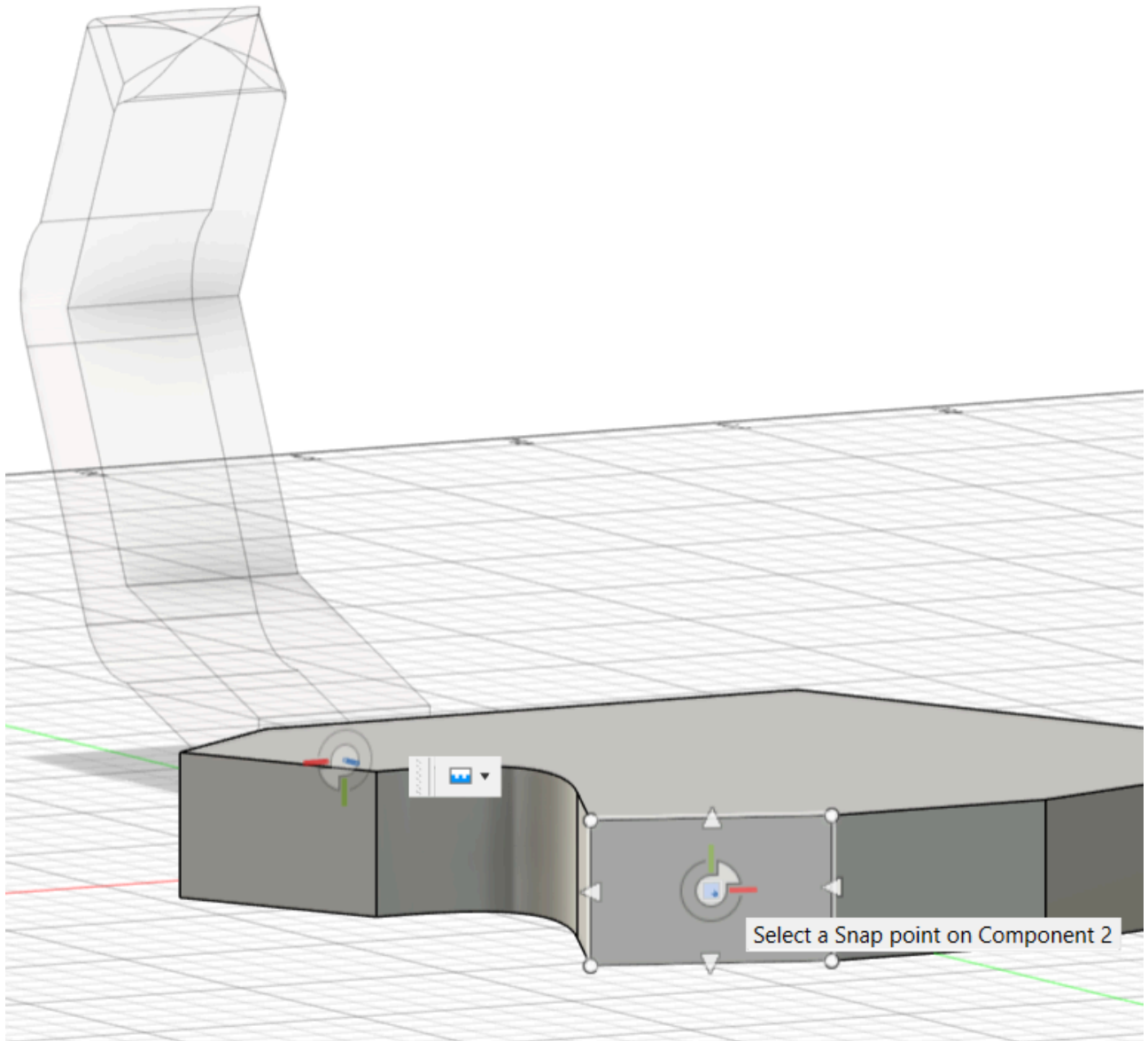
Step 3

- Click the center of the rectangle that was the first sketch in creating your finger

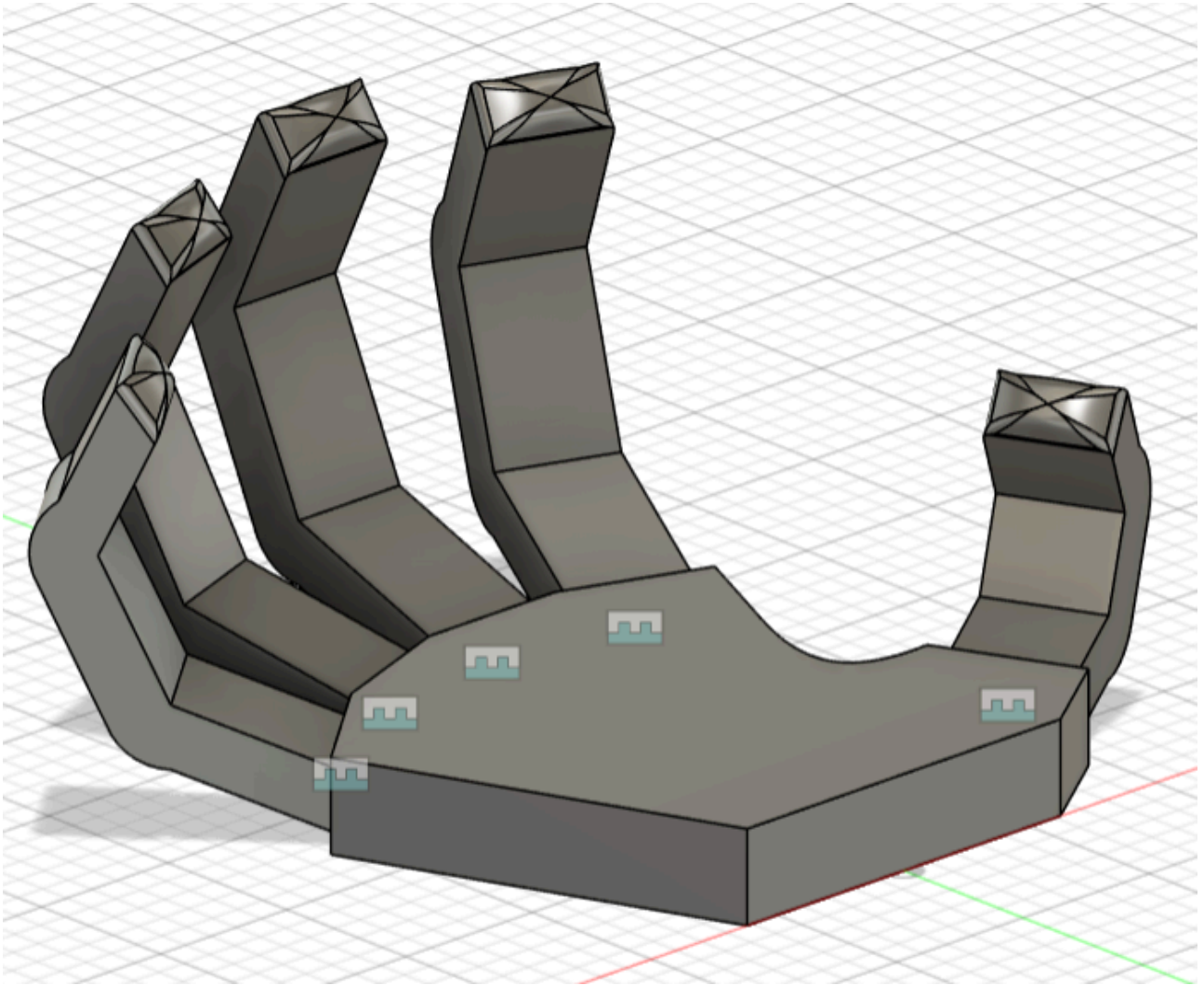


- Click on the where a finger would be attached to the base of the hand
- There should now be 1 finger attached to the hand

- Repeat this step for all fingers with their proper configuration



- Save the final model, it should look similar to this



Export to STL (optional)

- STL is the file type splicers use to create instructions for 3D printers to print designs
- To export the assembly, click on "File" next to the 3x3 dot matrix from earlier
- Next, click "Export"
- Under "File Type" scroll to the bottom and selected "STL files"
- Then click "Export" and your STL file should download to your computer