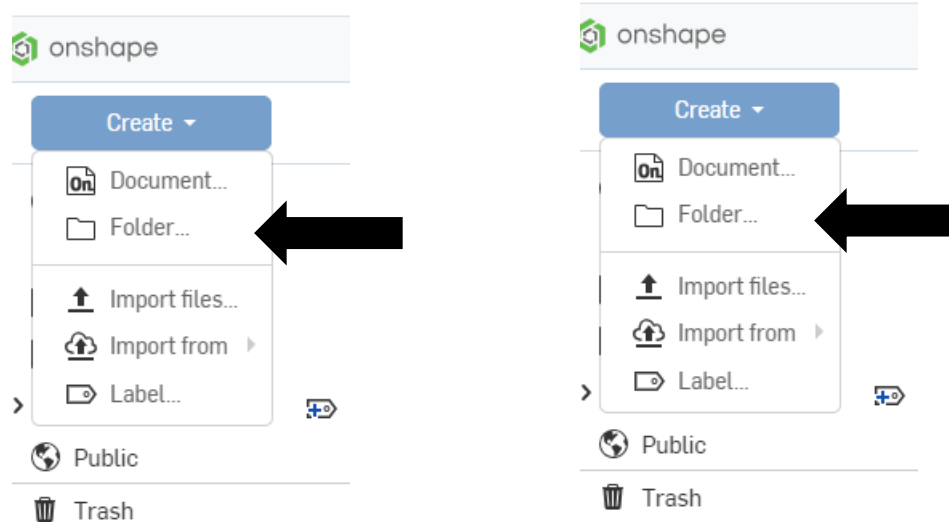


Bone Hook Lure Project



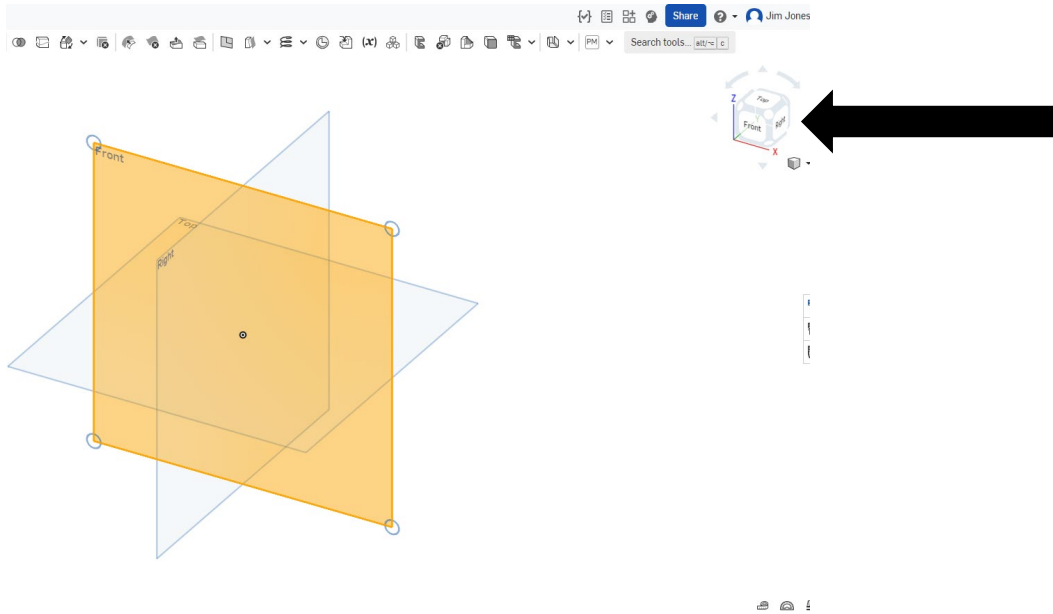
The directions will use the bone hook picture to try and duplicate the hook in 3D CAD and then make a 3D print. The overall dimensions: 1.75" tall by 1" wide. The top of the hook might also have a hole for binding.

The first step is to create a new folder in Onshape for file organization.

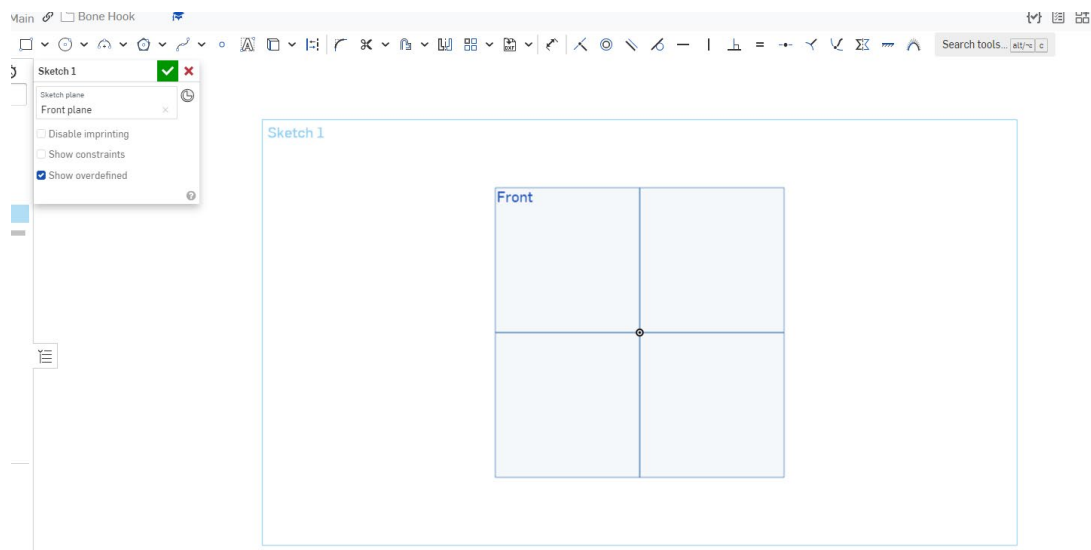
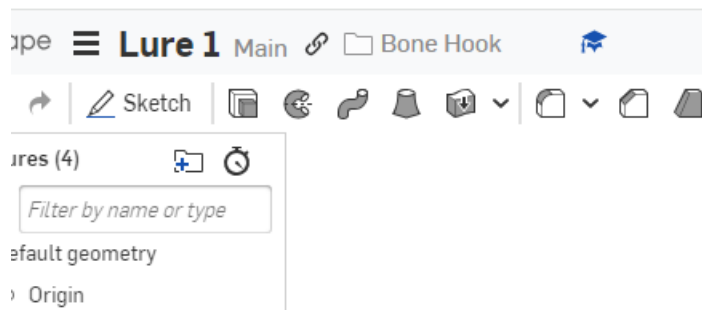


Once you have created a new folder, I named the folder Bone Hook. After making a folder you will create a new document. I named the new document Lure 1.

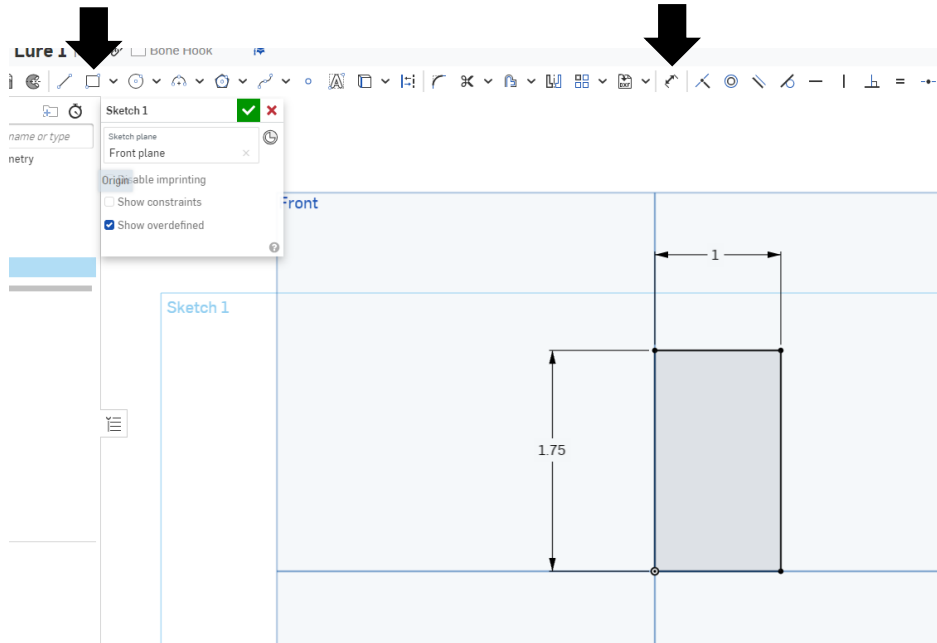
1. Select the front plane to start the sketch and then select front on the view cube.



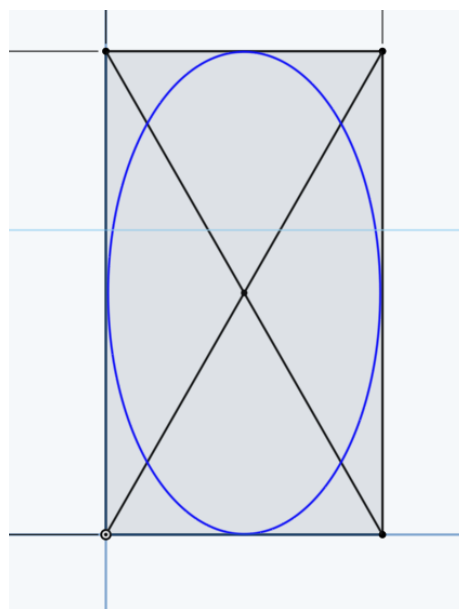
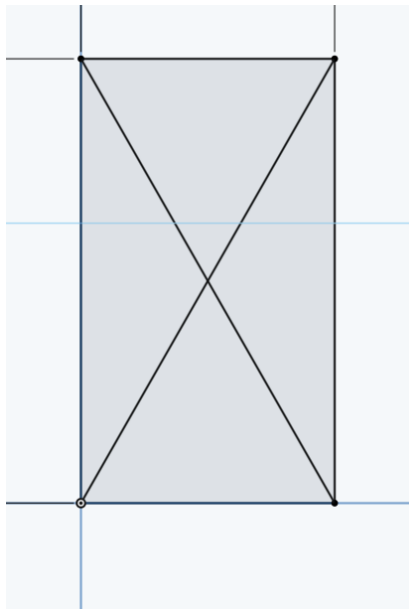
2. After selecting Front click on Sketch near the top left ribbon and then the sketch 1 plane will appear. The Sketch 1 plane will be the drawing plane to start the hook.



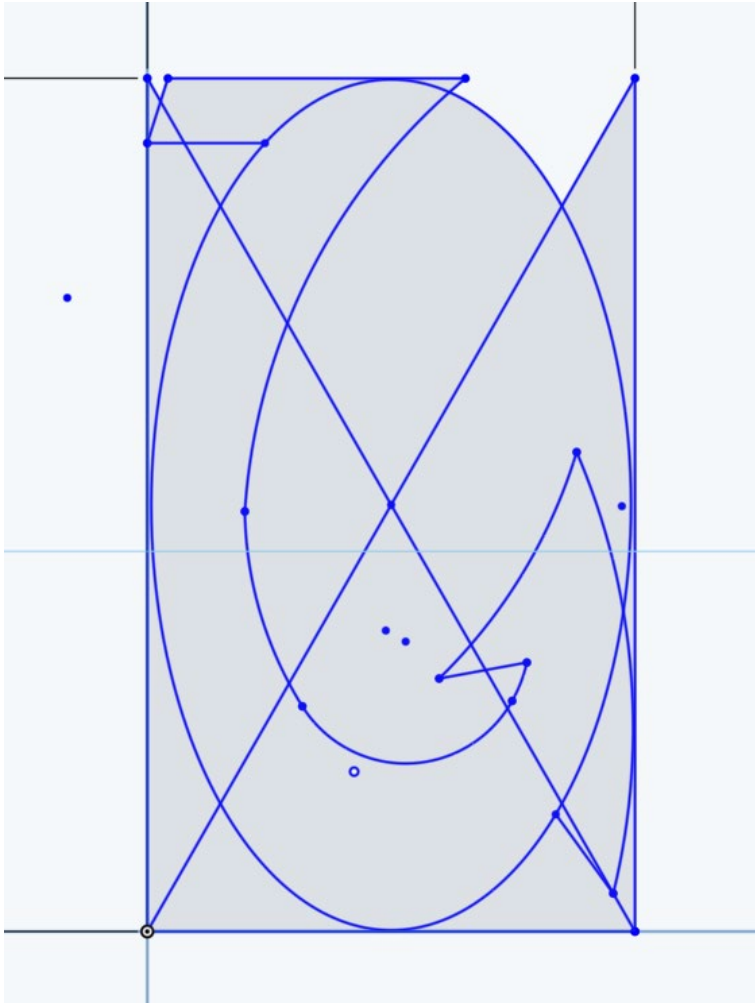
3. Sketch a rectangle and dimension it 1.75 tall and 1 wide. The rectangle will be the guide for sizing the hook.



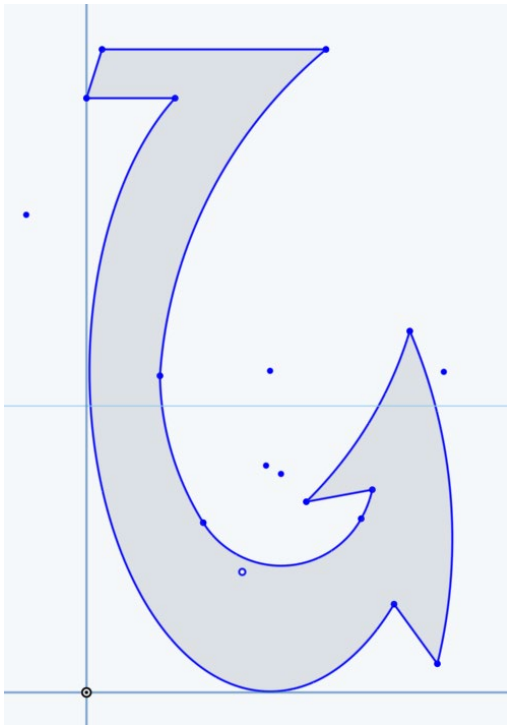
4. Start sketching with diagonal lines from the four corners to locate the center of the rectangle. Sketch an ellipse using the center intersection of the diagonal lines. The rectangle has black lines because it is fully-defined because it has dimensions. The ellipse is blue because there are no dimensions on the ellipse so it is under-defined meaning it has no dimensions and basically lines up with the rectangle. This ellipse is a guide for the outline of the hook. Right click the mouse after drawing to cancel drawing that part.



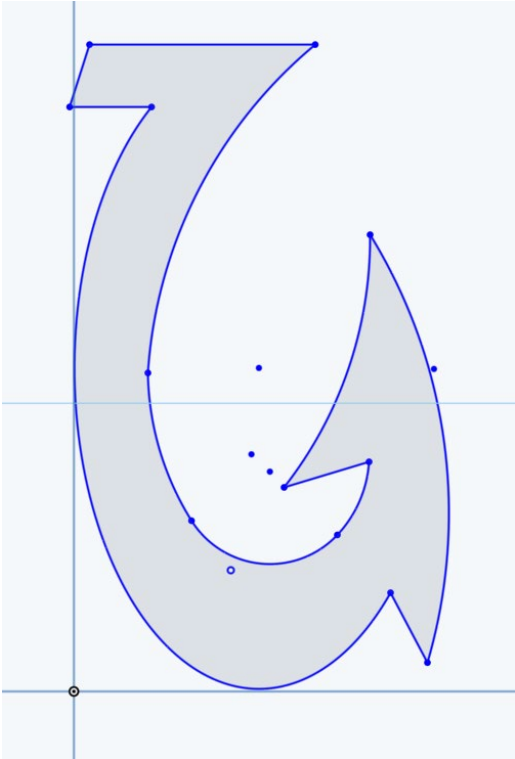
5. Use lines, 3 point arcs and fillets to get the basic shape of the hook. Practice drawings should be done before drawing the bone hook so you can experiment with lines, 3 point arcs, circles, ellipses and any of the other drawing tools.



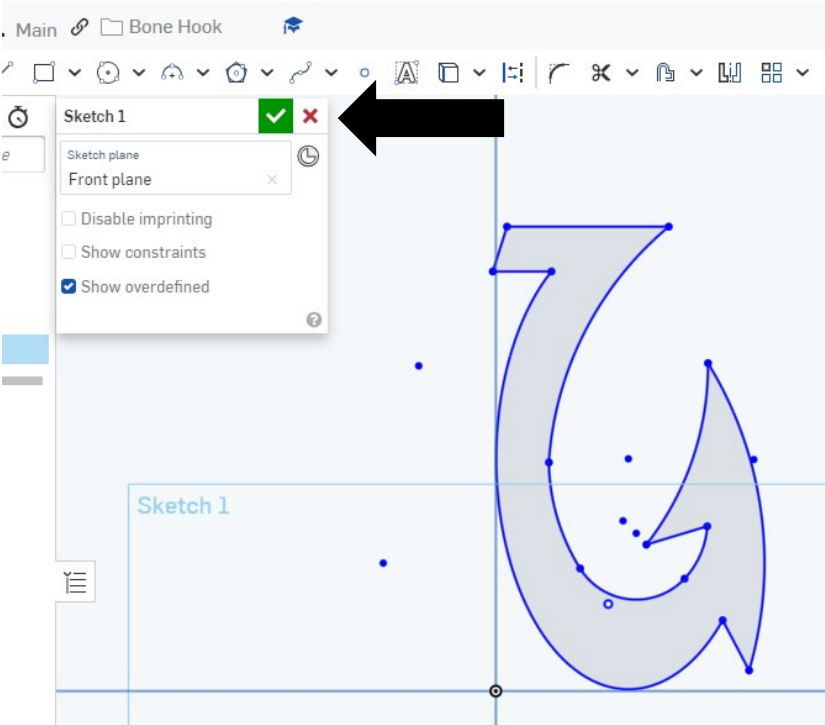
6. Delete and Trim the lines that are no longer needed. The Trim tool looks like a pair of scissors. To delete a line click on it to highlight the line and then press delete or use Delete sketch entity from the menu when you press the right mouse button.



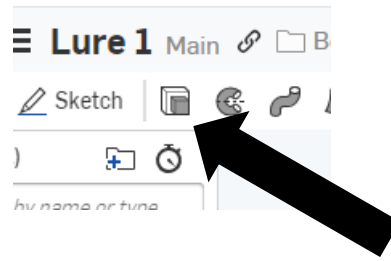
This is the basic shape of the hook. Refine the Lines and Arcs as needed. My final sketch is below. Making a CAD drawing does take a bit of practice so keep trying and get the hook shape similar to the example. It doesn't have to look exactly like the sample.



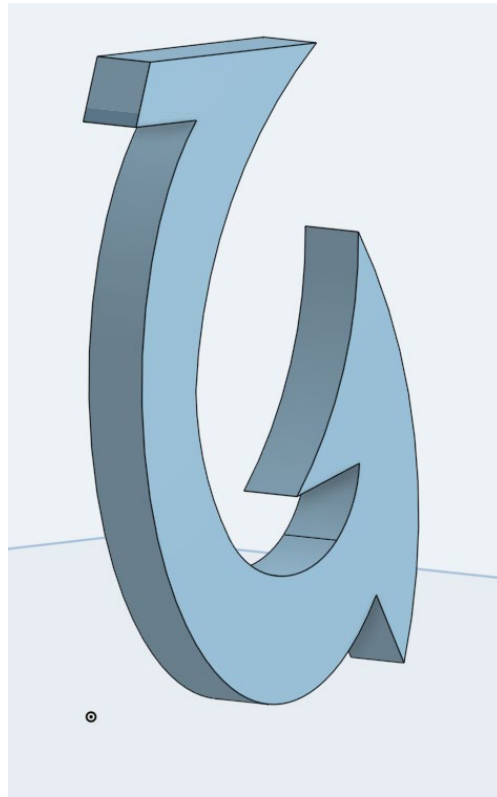
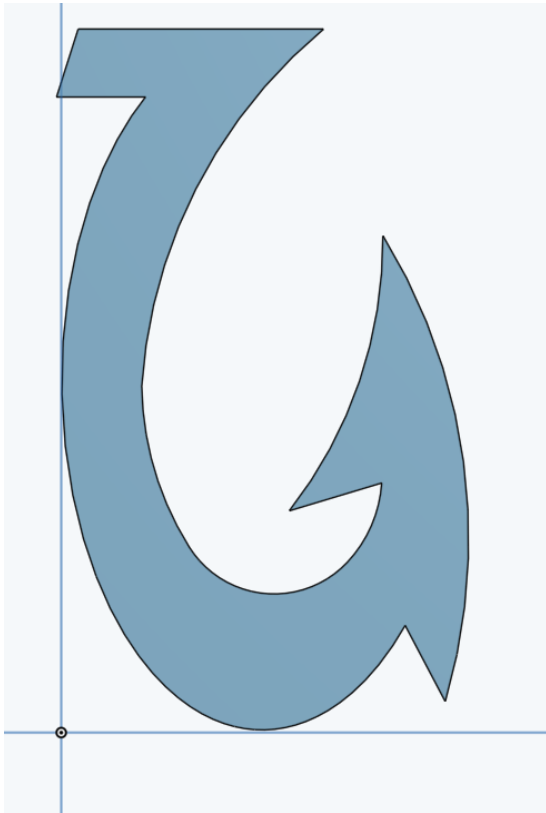
7. Click on the Green check mark when the hook sketch is done to Finish the sketch.



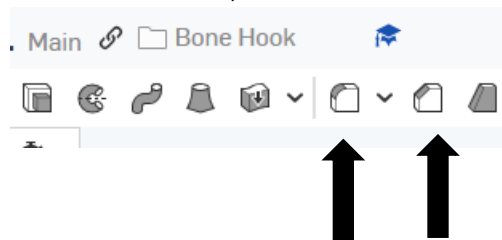
8. Left click on Extrude



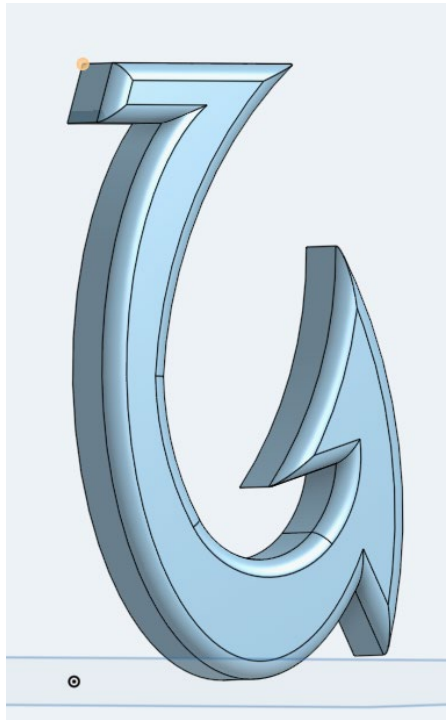
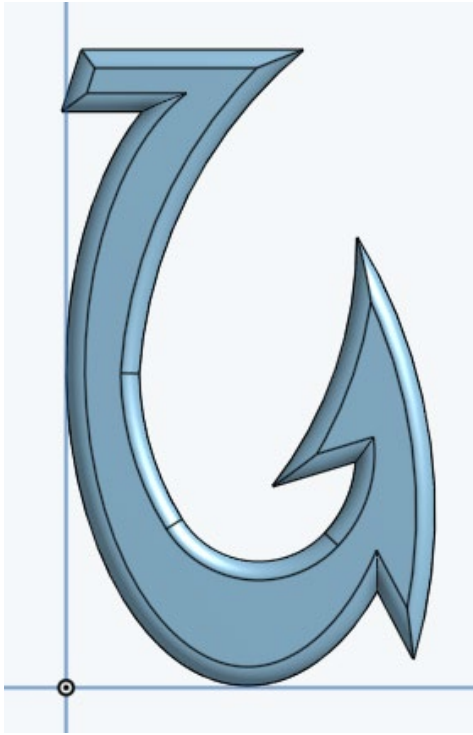
and left click the profile of the hook, not on the edge. Set the depth at .2" and click the Green check mark. Use the right mouse button to spin the hook to see the depth.



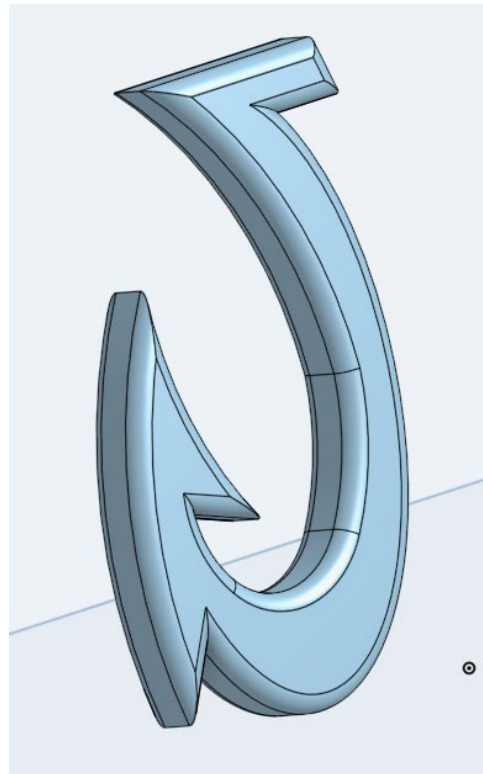
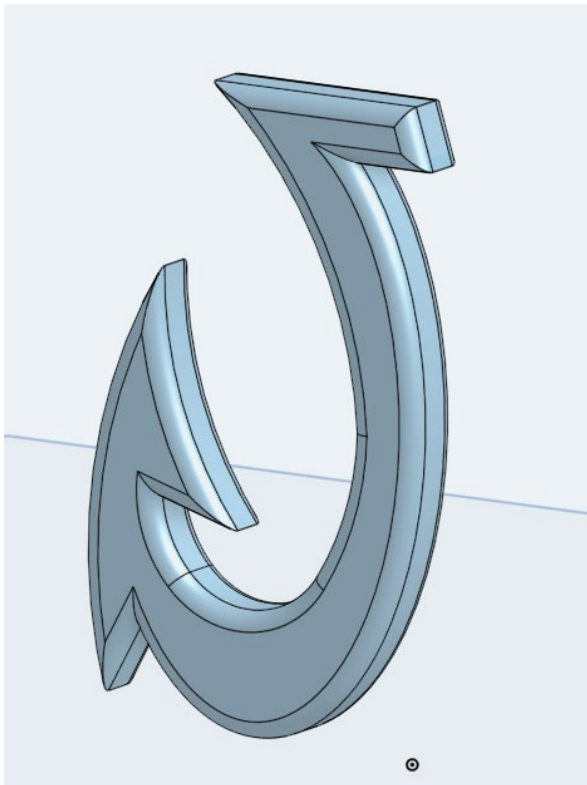
9. Select either Fillet, Chamfer or a combination of the two to adjust the top edges of the hook.



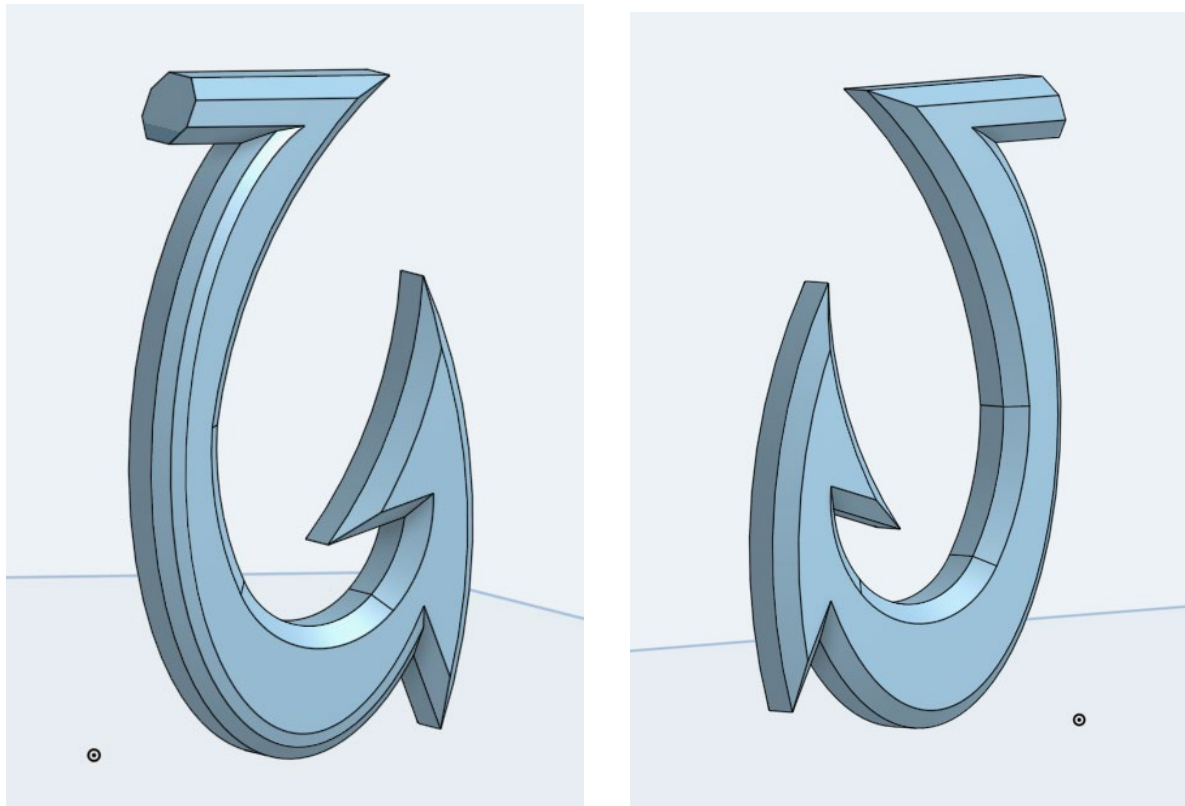
The picture shows a .05 fillet on all the top edges.



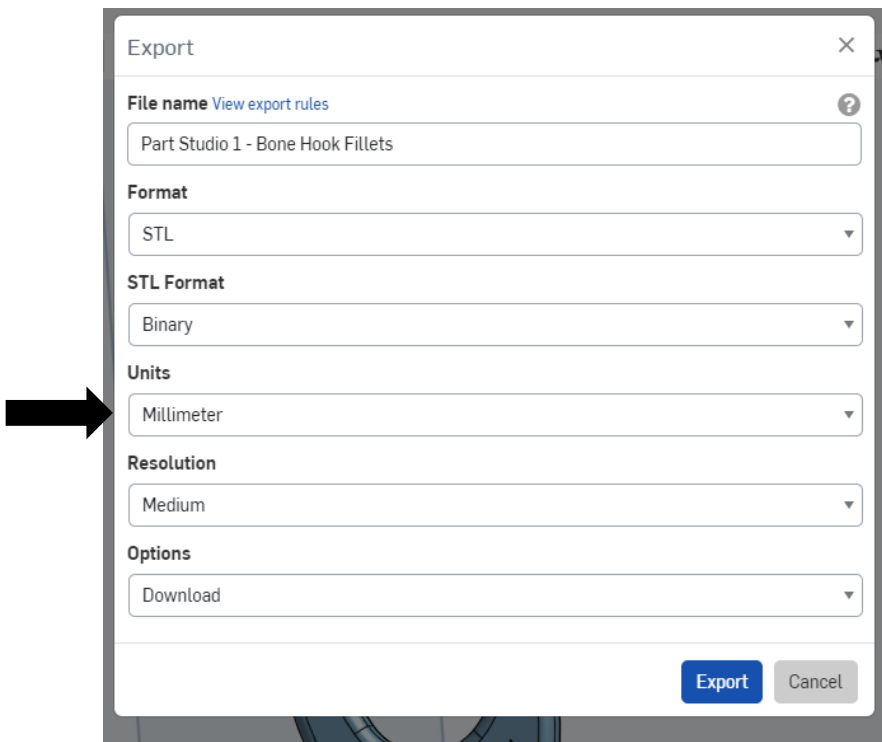
10. Rotate the hook and put the .05 fillet on the back side.



If you would like to use Chamfers instead of Fillets, repeat the process with Chamfer at .05" or what ever looks good to you.



To make the STL file, right click on Part 1 in the browser and left click on Export. Set your Units to millimeters and click on Export.



The STL file will go to your Downloads. Cut and paste the file to your secure saving location. The Onshape CAD file will be saved in the Cloud.

The STL file is now ready to be sliced depending on your slicing software.

Good Luck.