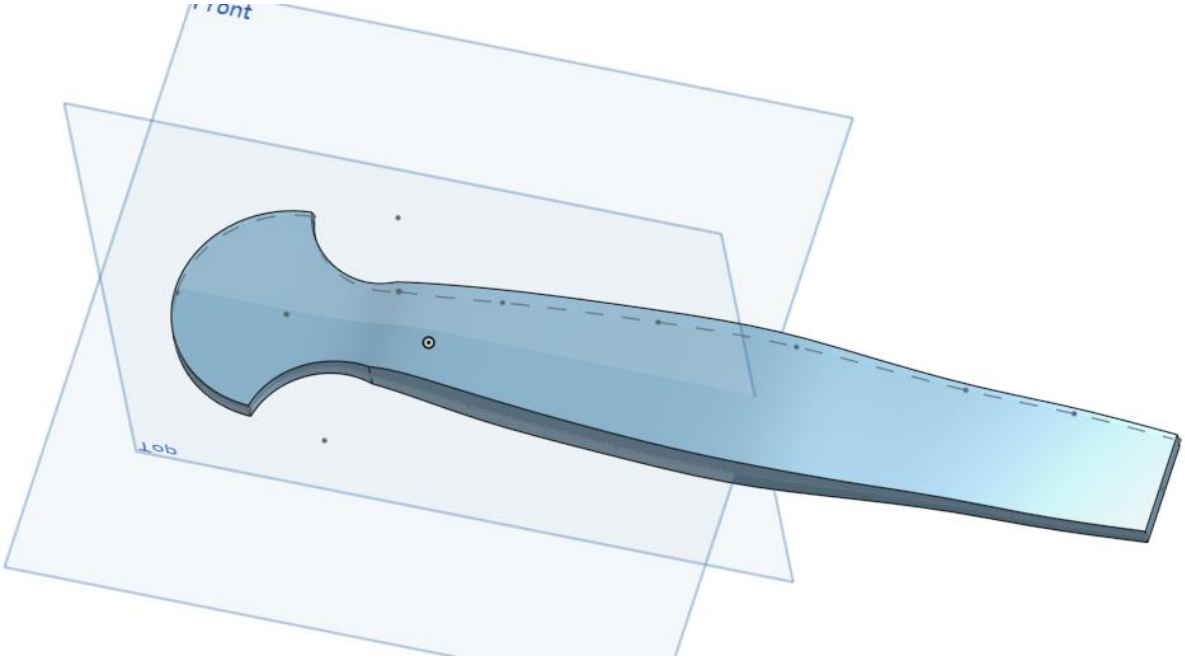
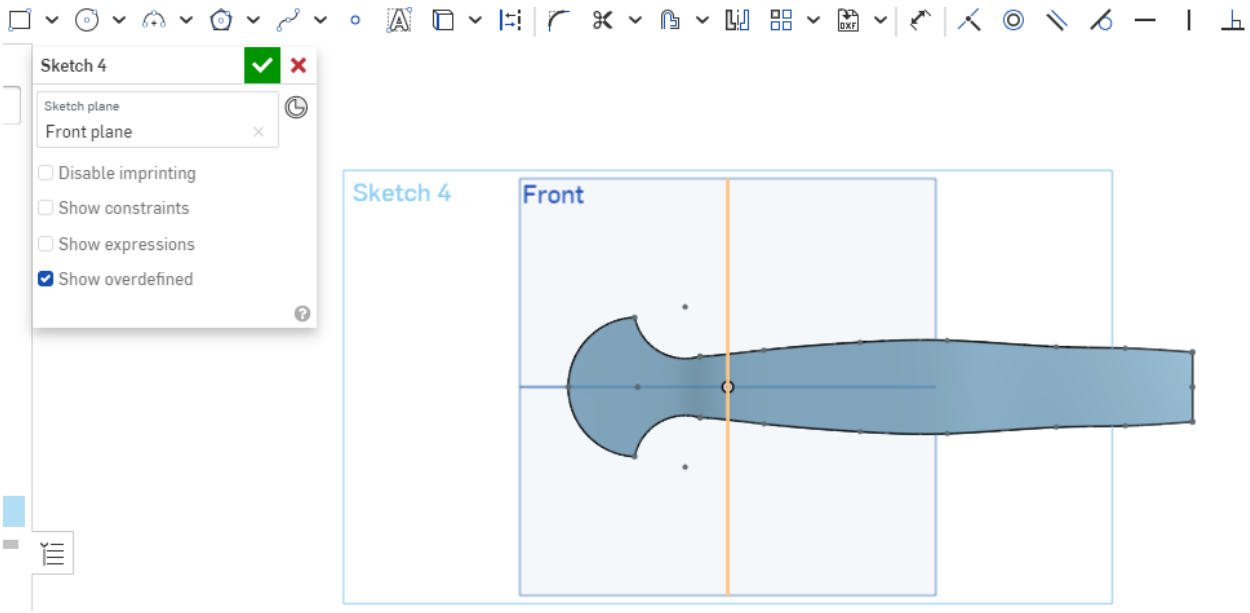
Creating a Mold in Onshape

Youtube link for more instructions. https://www.youtube.com/watch?v=TxnPs9PQeWE

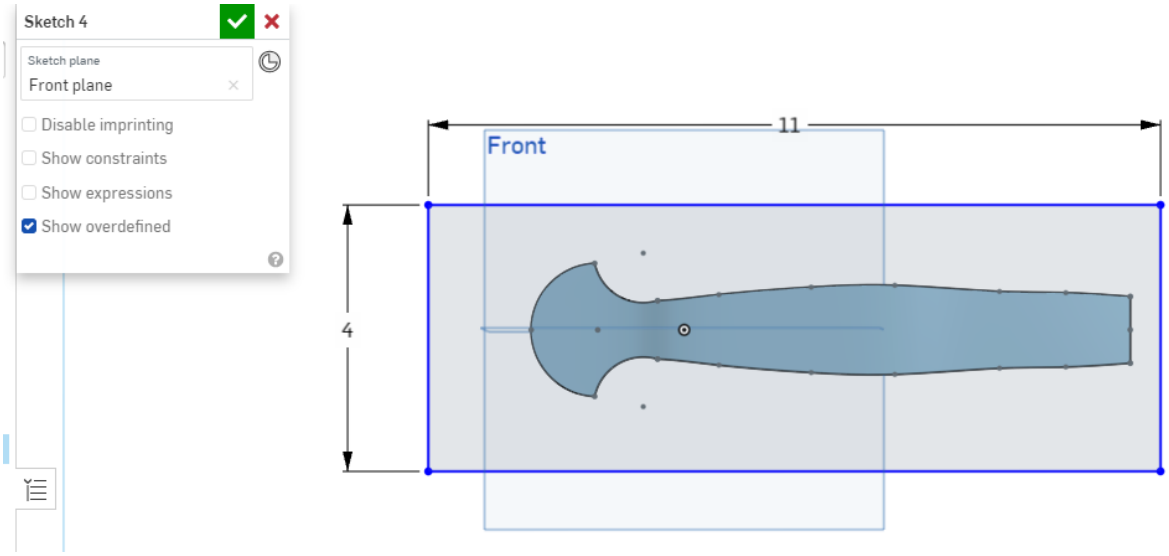
Start with a complete document (part) that needs to be in a mold for casting in plastic.



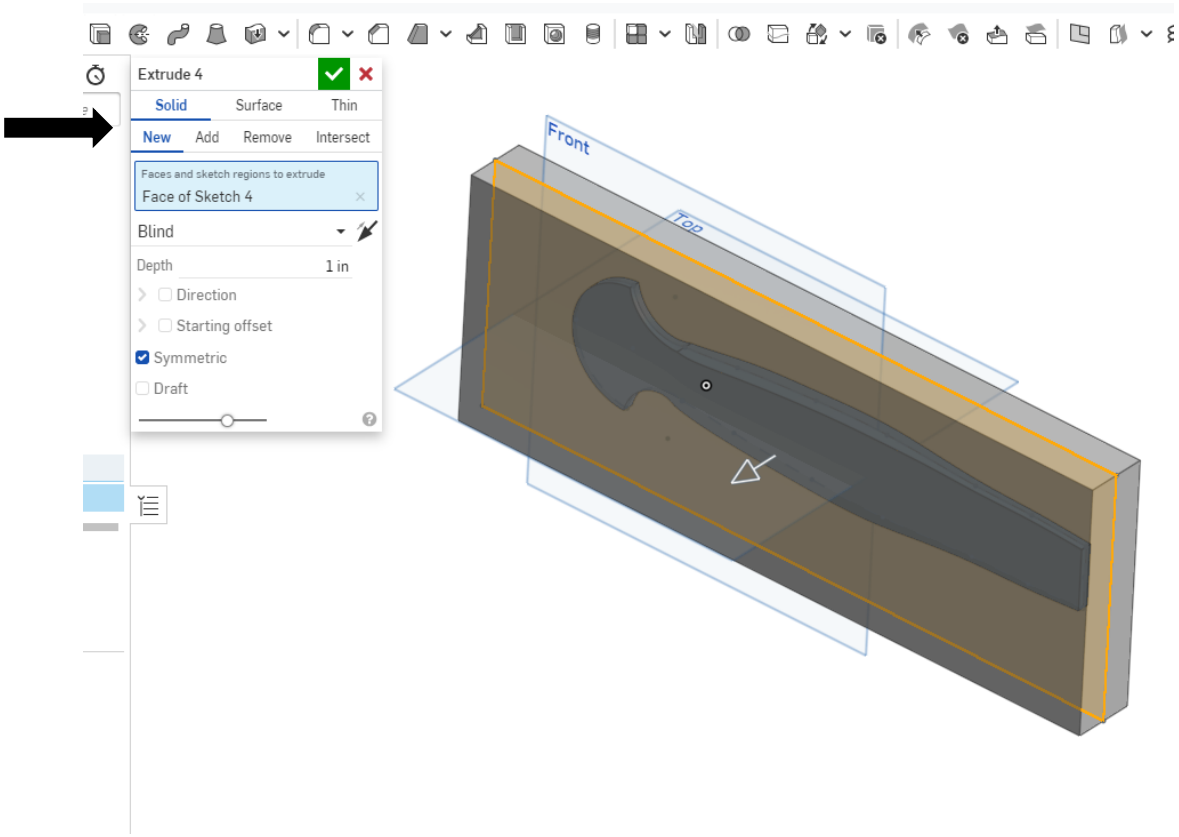
Start a new sketch with a rectangle on the plane you want to split the mold. This part will be split on the front plane.



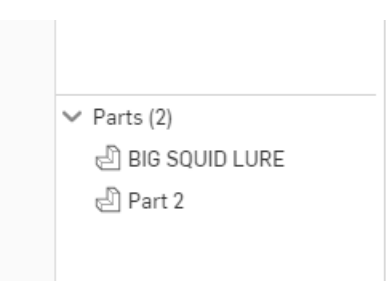
Draw a rectangle bigger than the part. Dimension the rectangle so it is big enough to meet the needs of the casting. This mold will only have one part, some will have more.

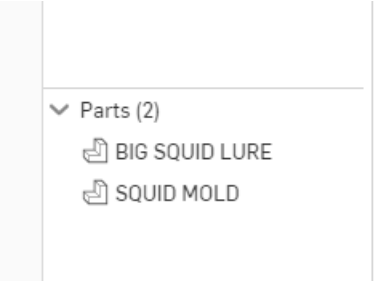


Complete the sketch the sketch and extrude symmetrically at 1 inch total. The extrude will need to be sized to the part so there is enough material around the part. Be sure to select NEW in the extrude menu to get a new part.

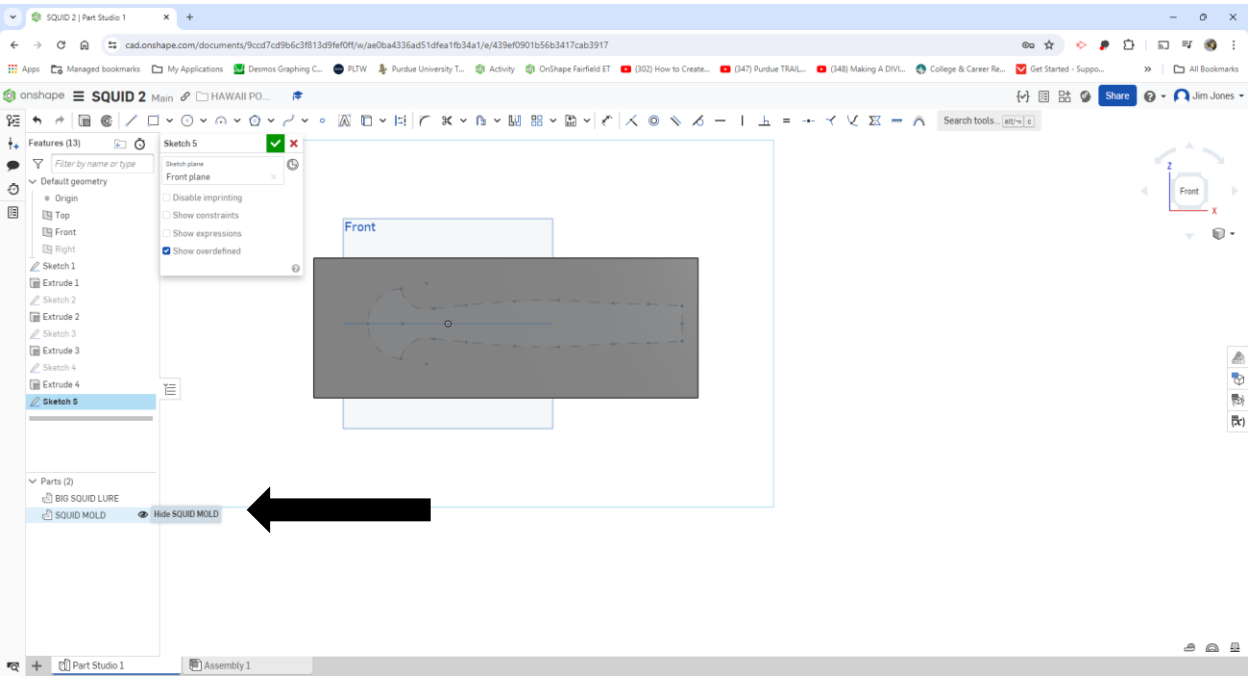


Rename the parts in the window on the left side of the browser if you want to. This does help when hiding and showing the parts.

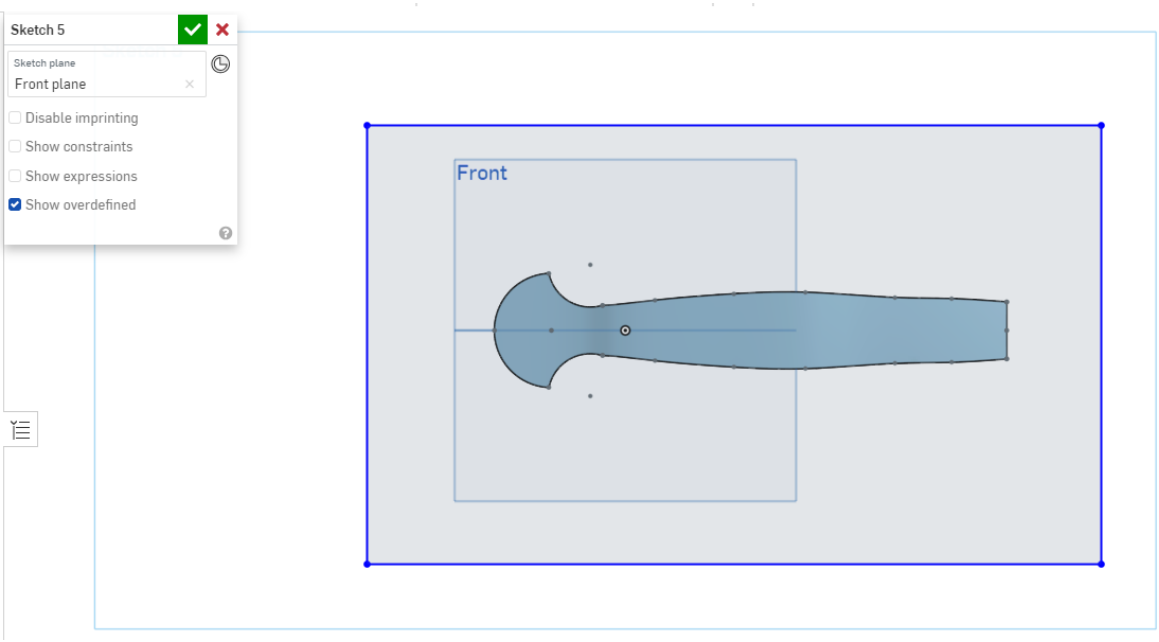




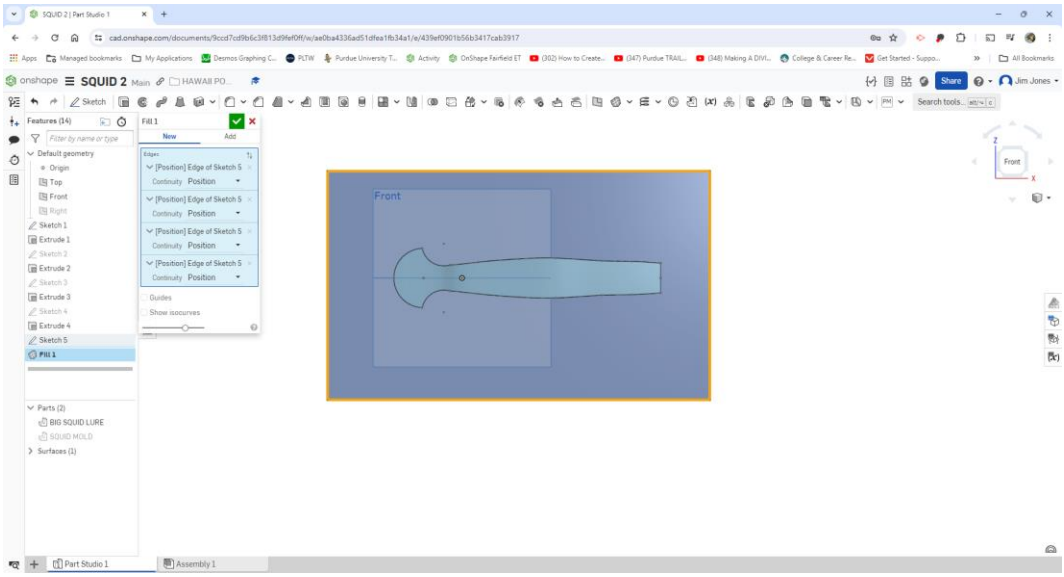
Hide the Mold by clicking on the eye on the right side of the mold name.



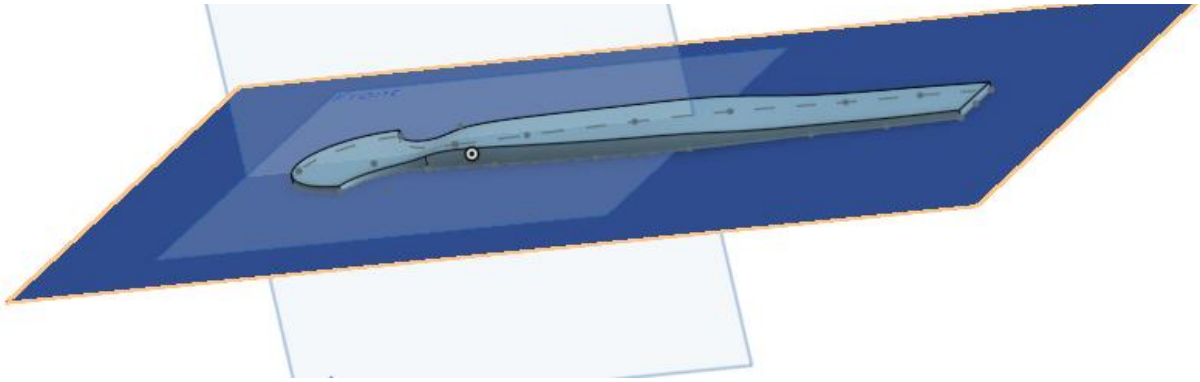
Start a new sketch on the same plane as the mold and draw a rectangle around the part. Click the green check mark to finish the sketch.

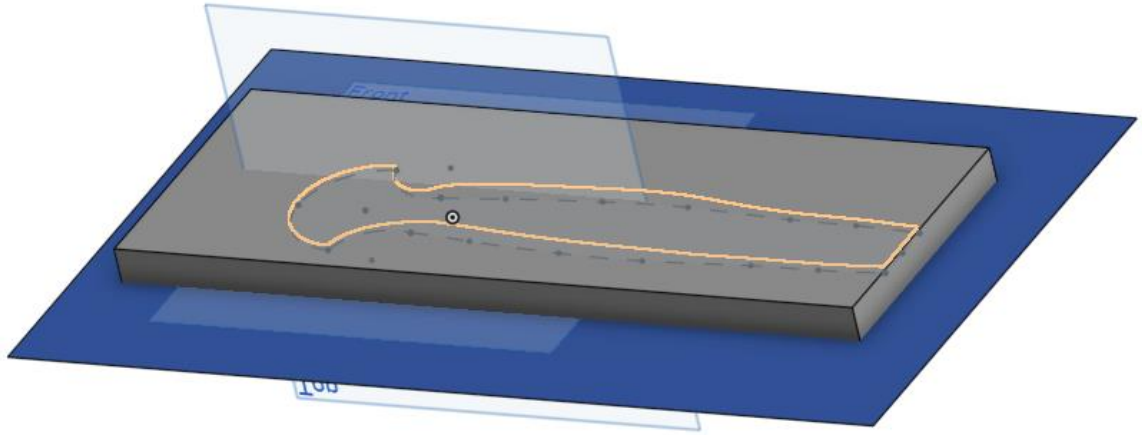


Click the drop down arrow on the Offset surface icon and select fill. Click each line of the rectangle and it will turn yellow and highlight in blue. Click the green check mark.

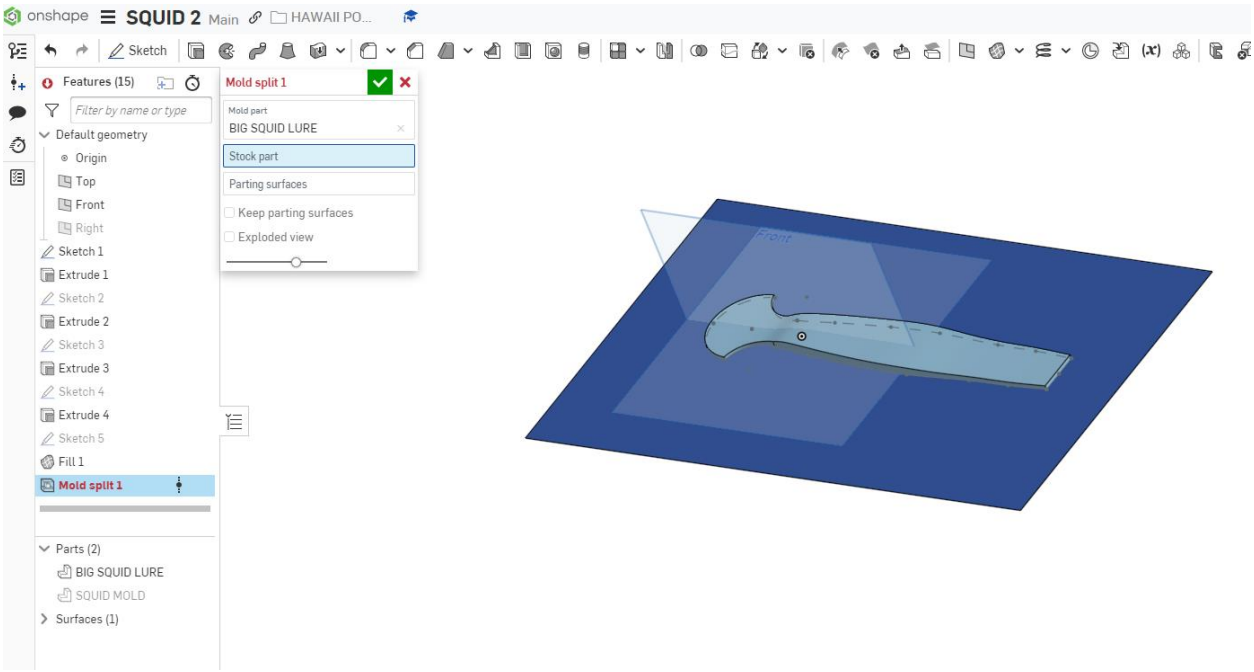


A blue surface will stay on the plane and now turn the mold visibility back on.

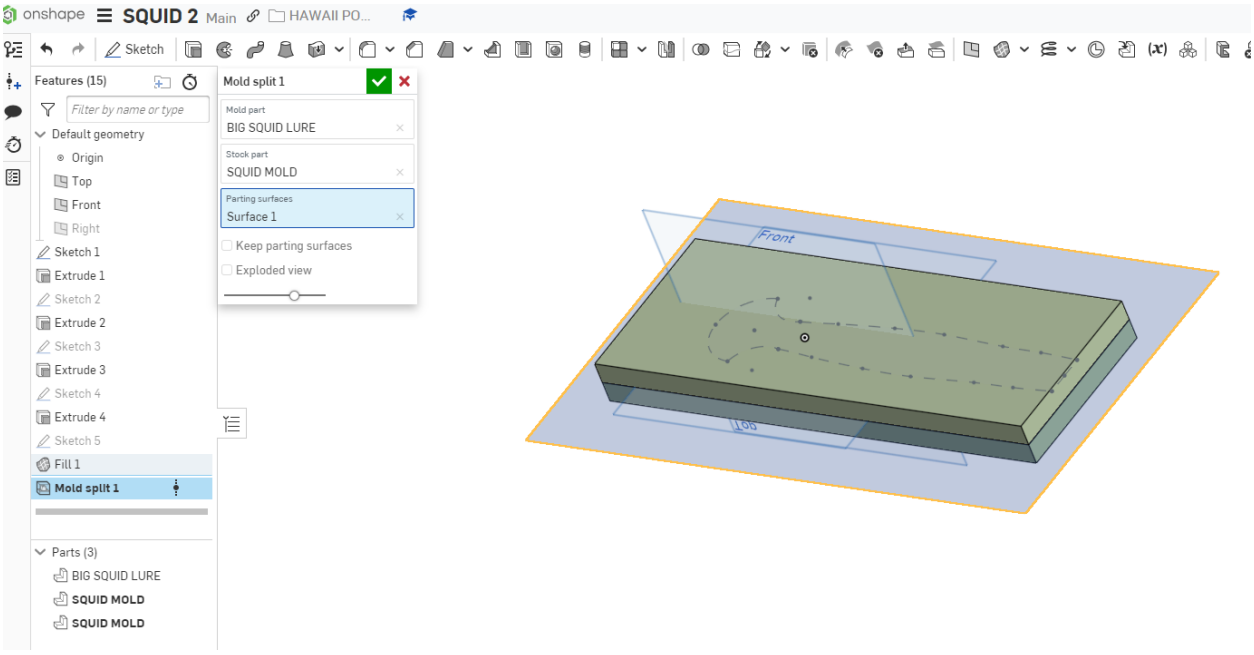




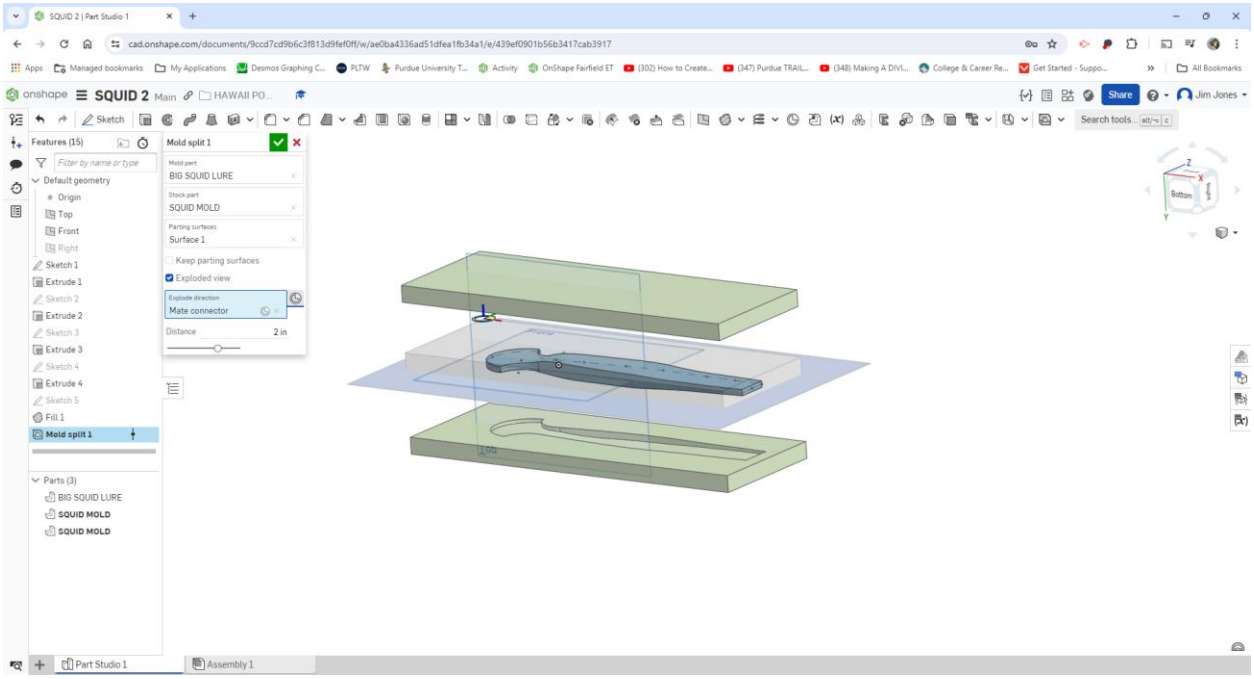
Select Mold Split on the menu bar. You may need to use the Search tools to find it. In the Mold Split menu box, pick the lure as the as the Mold part. It may help to hide the mold.



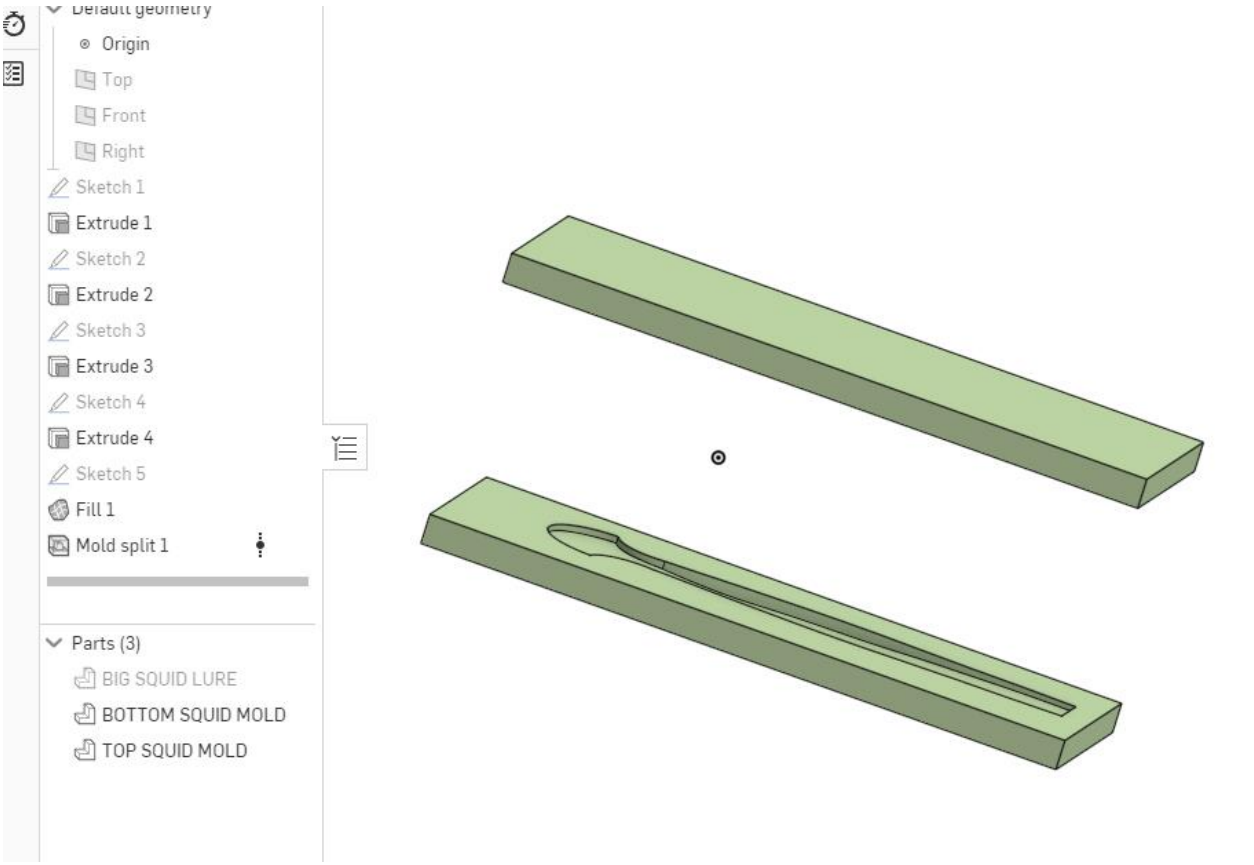
Unhide the mold and select the mold as the Stock part and then select the blue surface as the Parting surface.



Select the Exploded view view box and set a distance and them click on the top surface of the mold to get an exploded view. The molds now have a cavity that will be where the plastic is injected. Click the green check mark and inspect your work.



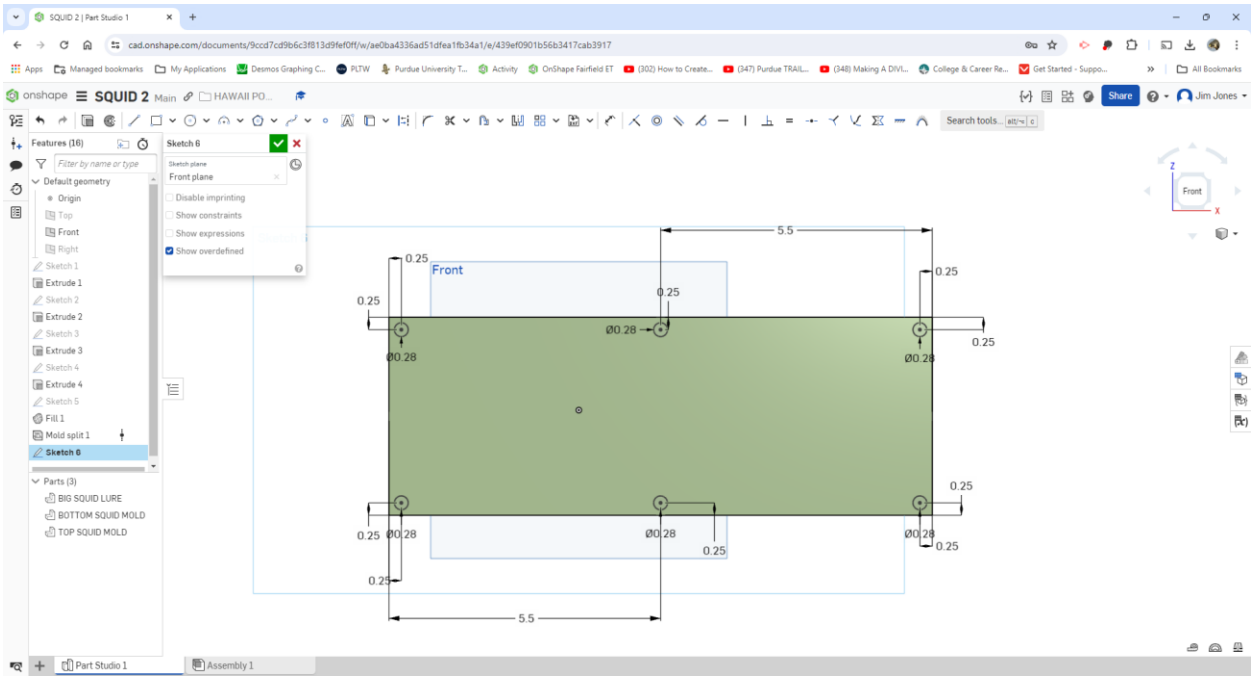
Hide the work planes and hide and unhide the three parts.



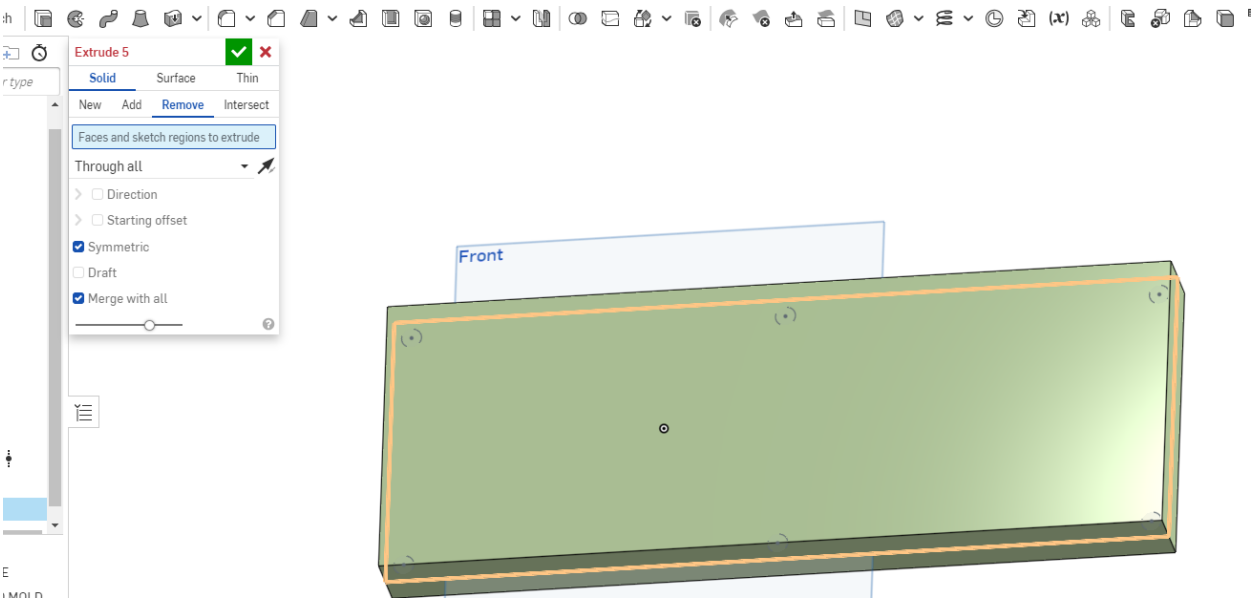
There are now two Squid molds and you can rename them as you see fit. I will have a top and bottom mold.

It is now possible to Edit the Mold split and set the distance to 0 to continue work on the molds. For a two part mold you will need bolt holes, vent holes and inlet hole for the plastic to be injected.

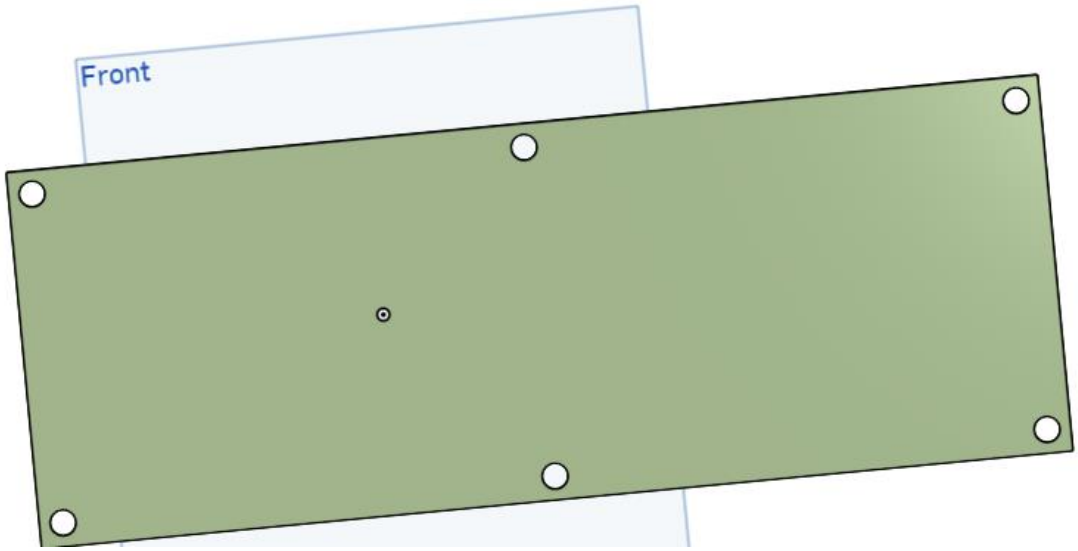
Start a new sketch plane you used for the mold rectangle. For this demo it is the front plane. Draw six circles at .28 diameter. Six holes are needed because this is a large mold. Most molds will only need corner holes. The bolts will be ¼" x 20 x 1 ½" bolts with ¼" Wing nuts to clamp the mold shut. Dimension the circle centers .25 from each edge. Finish the sketch. Click the green check mark to finish the sketch.



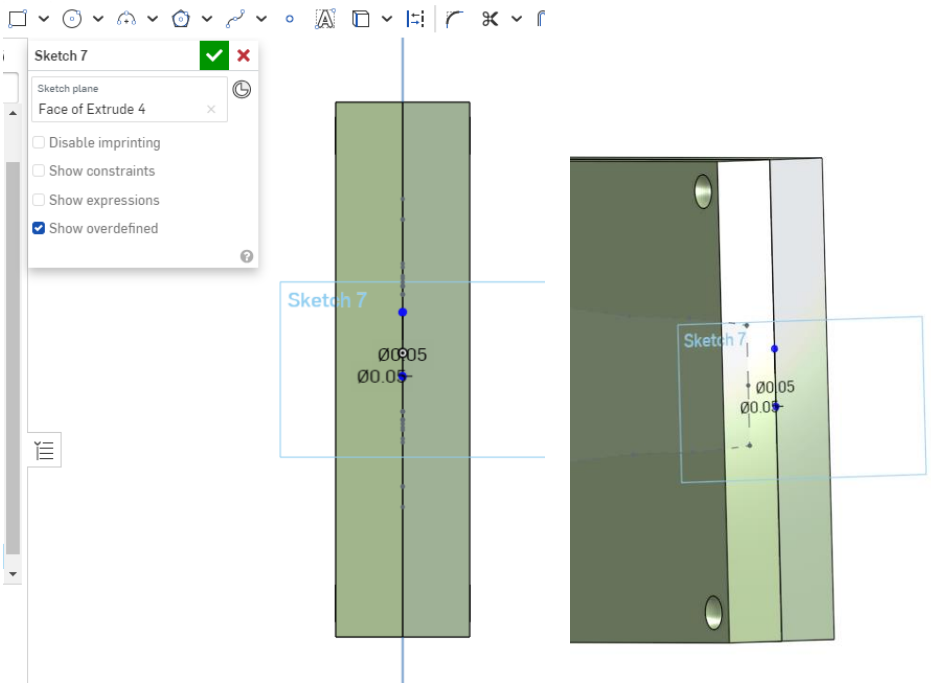
Extrude the holes Symmetrically to go through both molds and set Remove on the menu to create the holes through both molds. Click on Sketch 6 for the holes and the six holes will highlight. Select Merge with all from the Extrude menu and select through all in the drop down menu Click the green check mark.

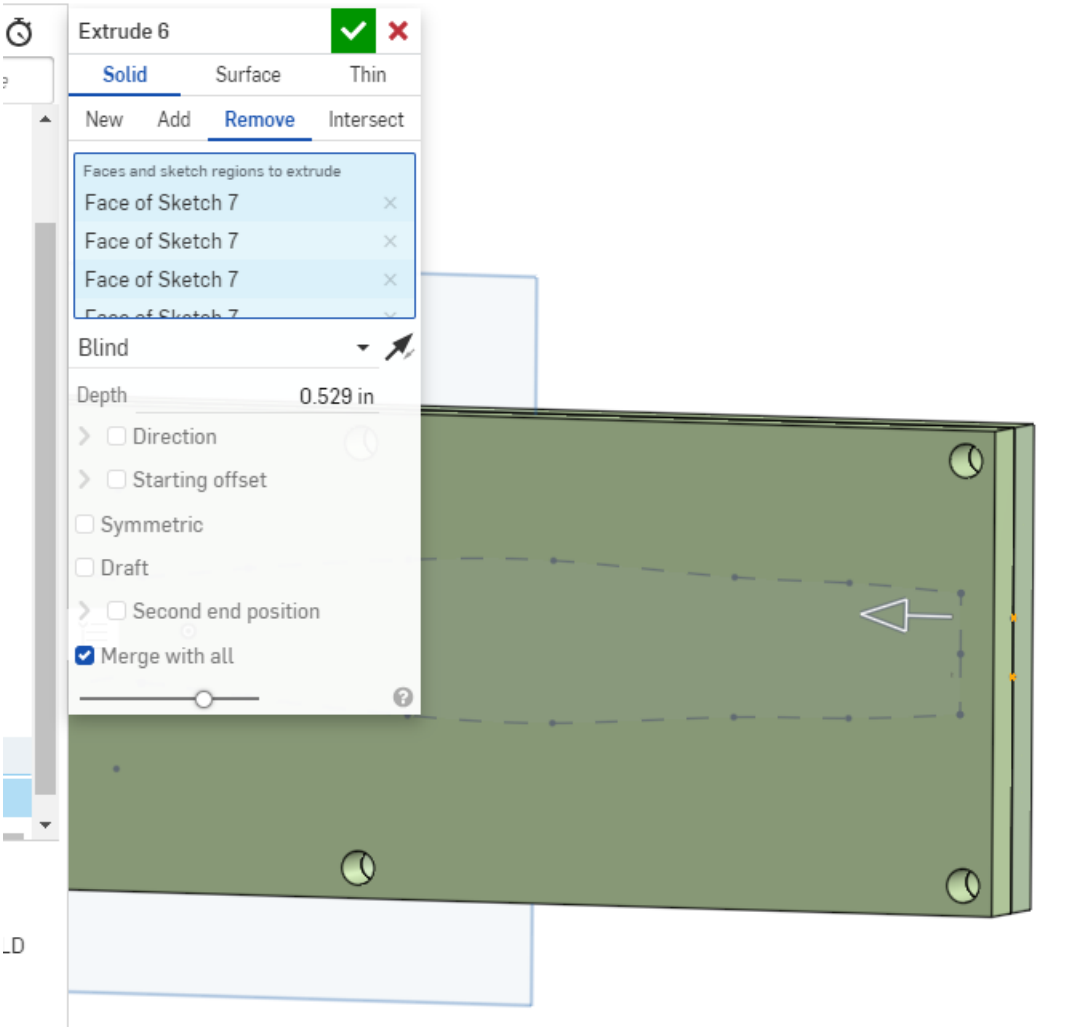


Rotate the part to confirm the holes go through both molds.

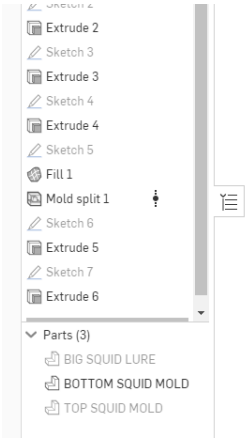


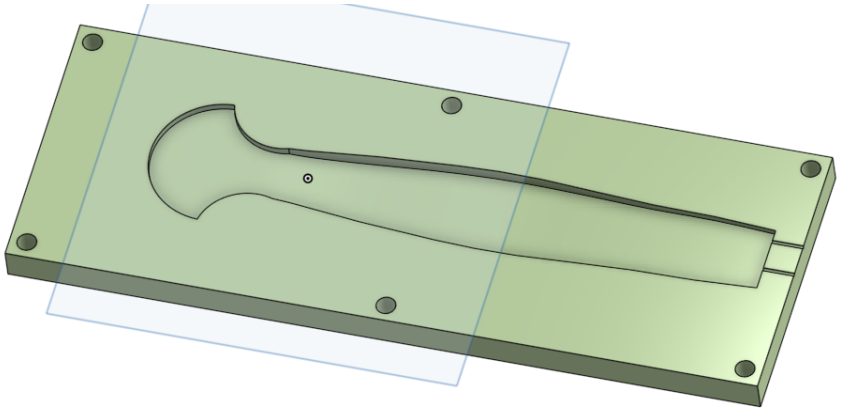
Unhide the sketch of the lure to make it easier to draw the vent hole on the end of the molds. Make a new sketch on the end of the mold on the tail end. Draw one or two circles on the end surface for vent holes. Dimension the circles at 0.05” and extrude the holes into the lure cavity. Set the distance so the extrude goes into the cavity and make sure to select Remove material and Merge with all. Click the green check mark. The holes should be on the end of both molds.

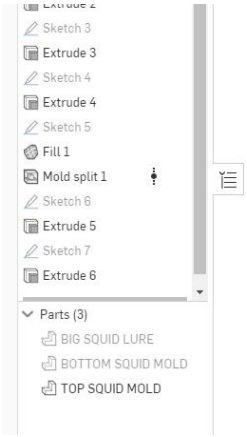

Sketch 7
Sketch plane
Face of Extrude 4
Disable imprinting
Show constraints
Show expressions
Show overdefined
Sketch 7
Ø0.05
Ø0.05
Sketch 7
Ø0.05
Ø0.05

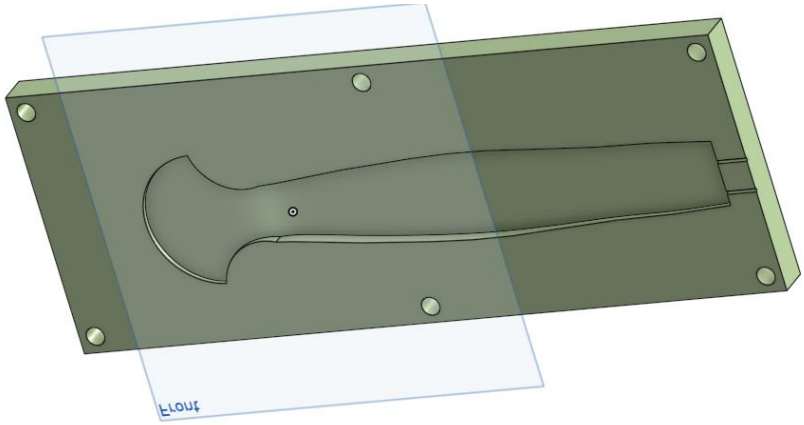

Extrude 6
Solid
Surface
Thin
New
Add
Remove
Intersect
Faces and sketch regions to extrude
Face of Sketch 7
Face of Sketch 7
Face of Sketch 7
Blind
Depth
0.529 in
Direction
Starting offset
Symmetric
Draft
Second end position
Merge with all

Hide one of the mold parts and rotate to see the small vent holes. Do the same with the other mold part to inspect the vent holes.

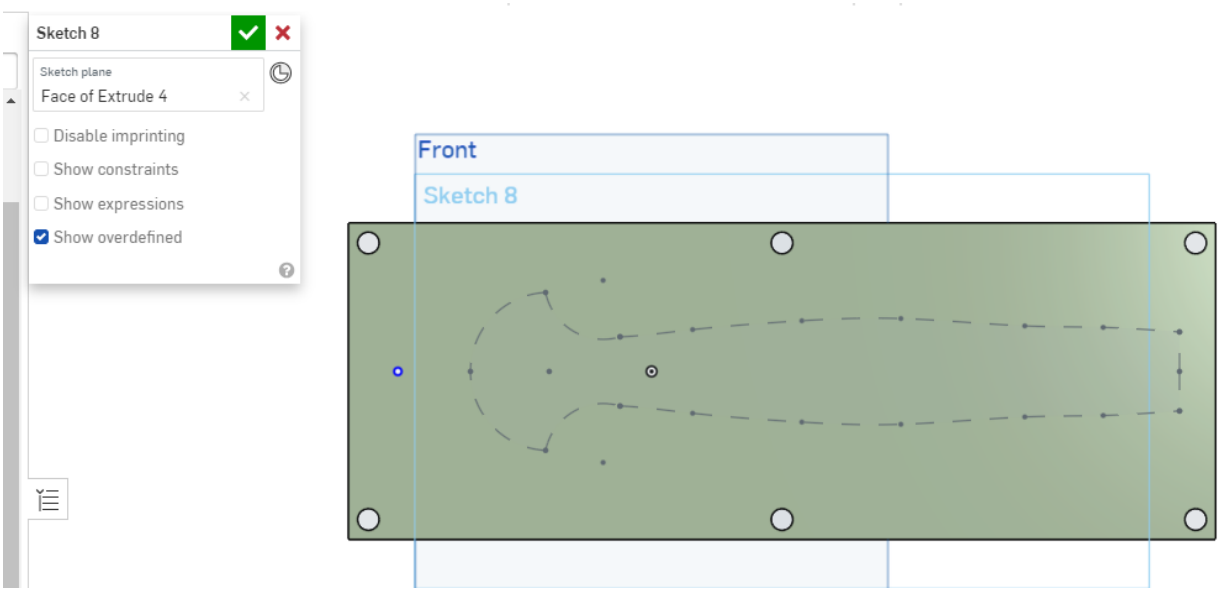




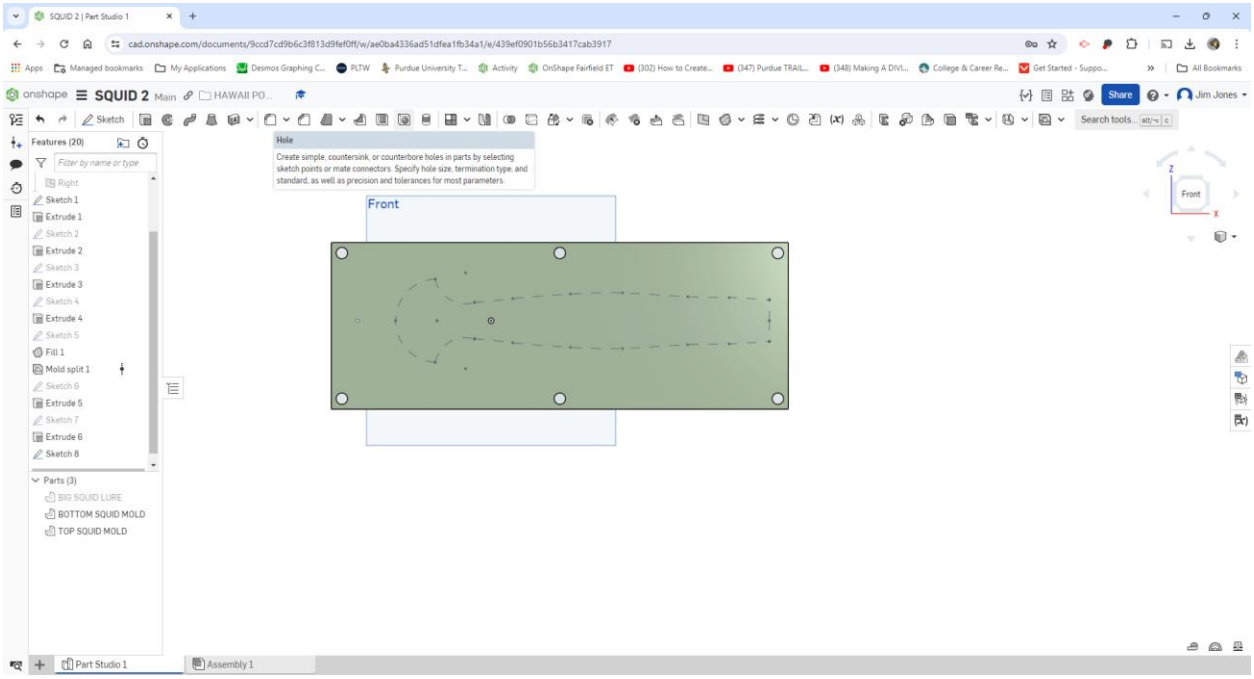


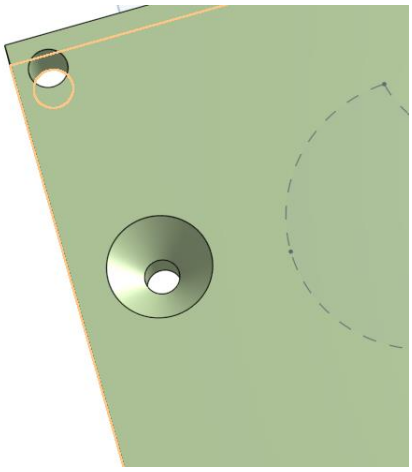


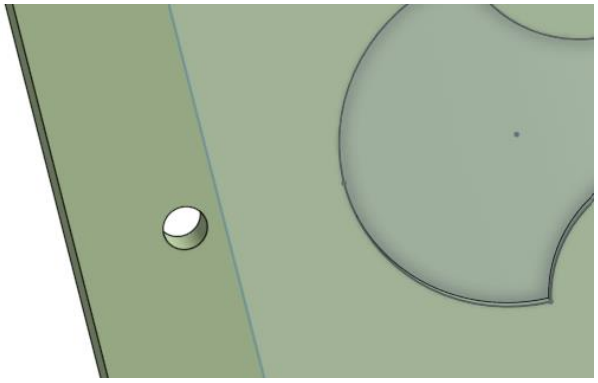
Start a new sketch on the top surface of the top mold. Draw a Point on the surface between the end of the mold and the lure. This point will be used to place a counter sink hole on the top mold as the opening for the plastic to be injected into the mold. Click the green check mark to finish the point sketch.



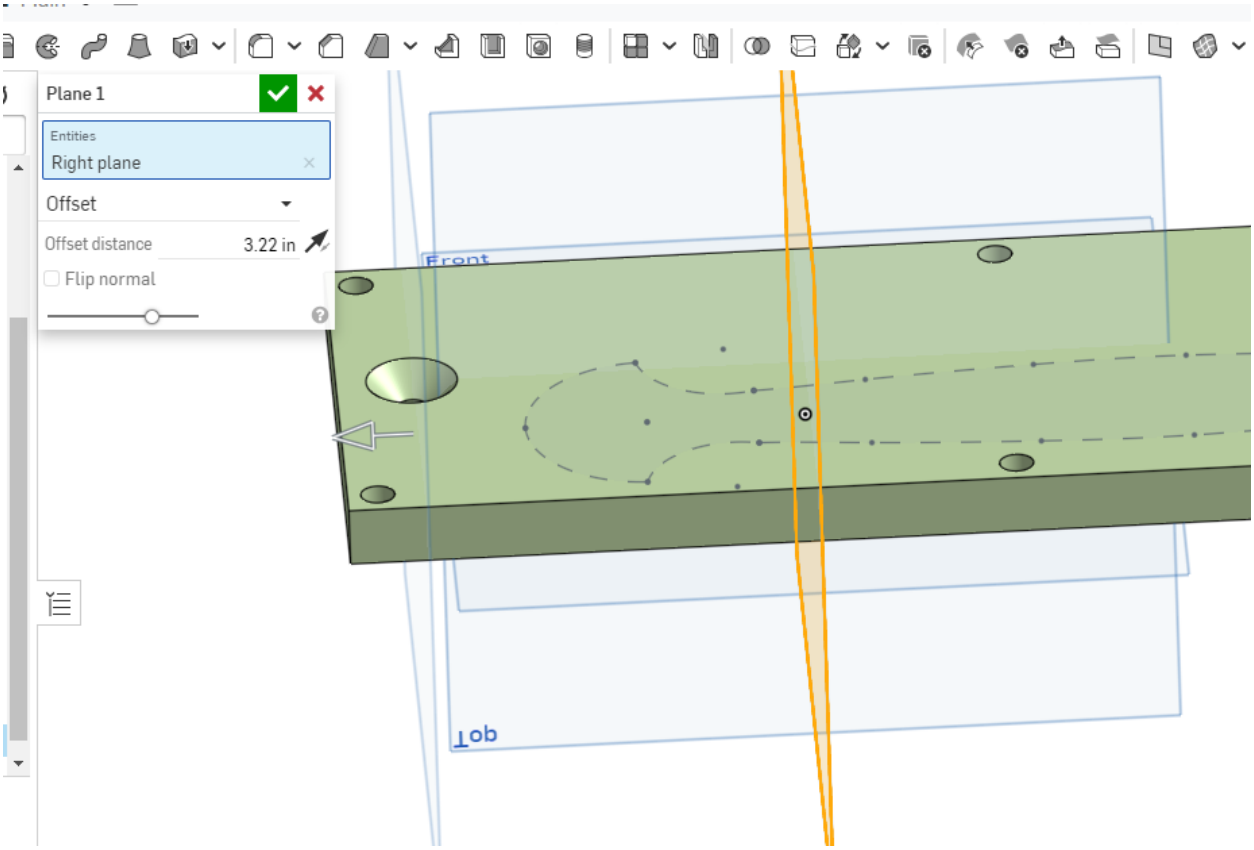
Select hole from the menu and use the counter sink with a diameter of .75”. Follow the measurements on the menu. You can hide the bottom mold to confirm the counter sink hole.



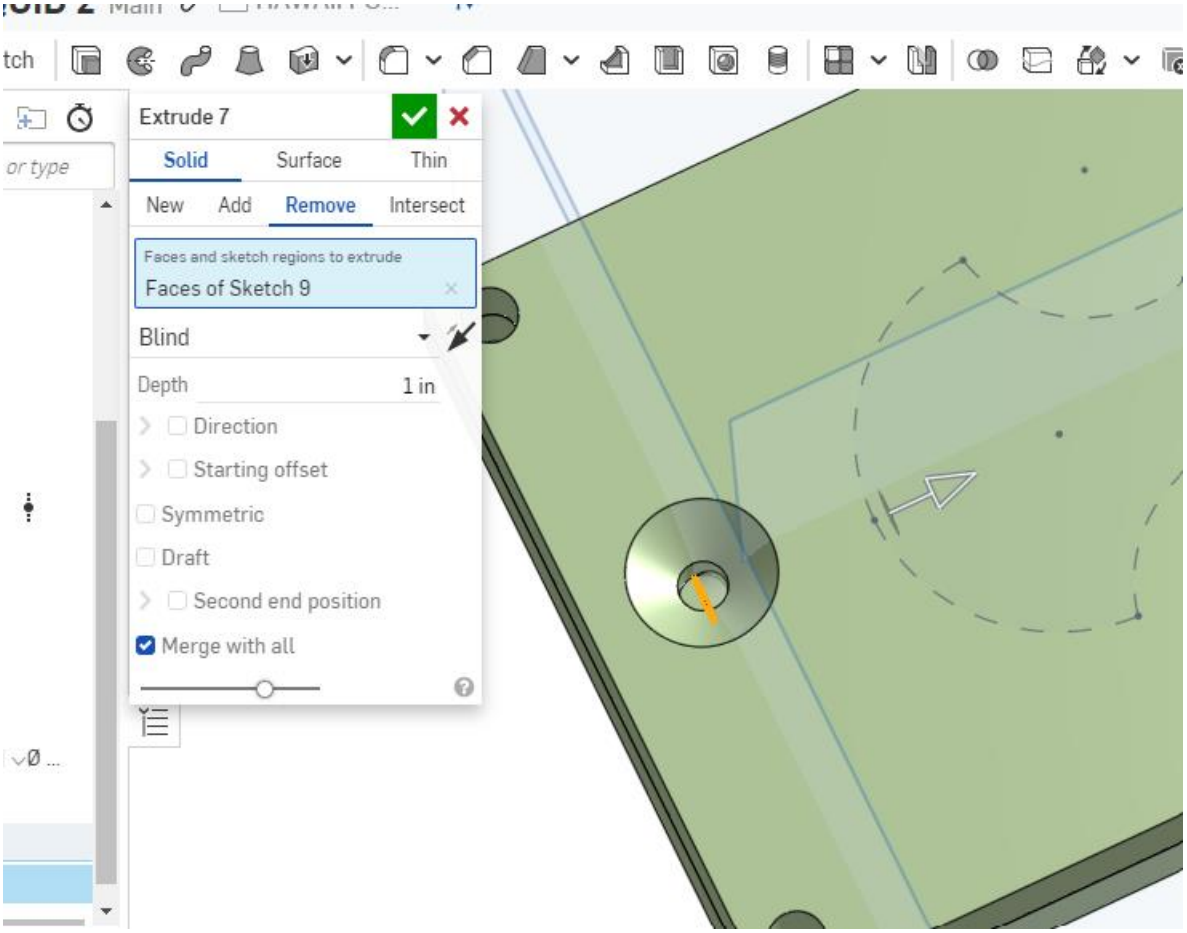




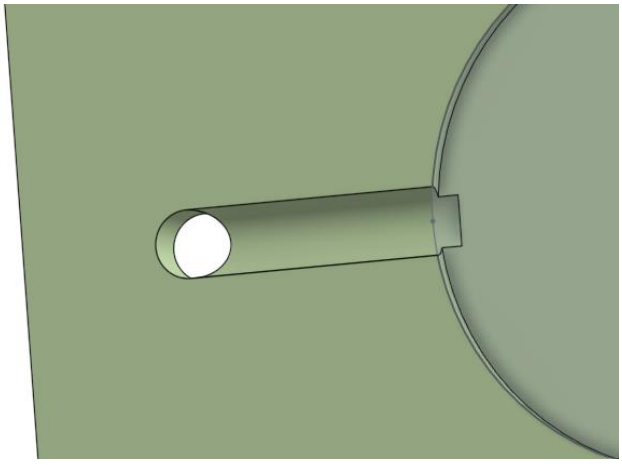
Add a construction plane to go through the hole to draw a .25” channel from the hole into the lure. This is needed to get the plastic into the lure cavity.

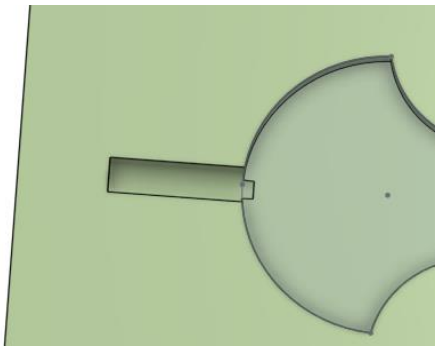


Draw a .25” circle on the plane. Finish the sketch and extrude the hole into the cavity. Follow the extrude menu to Remove and Merge with all so the extrude is in both parts of the mold.

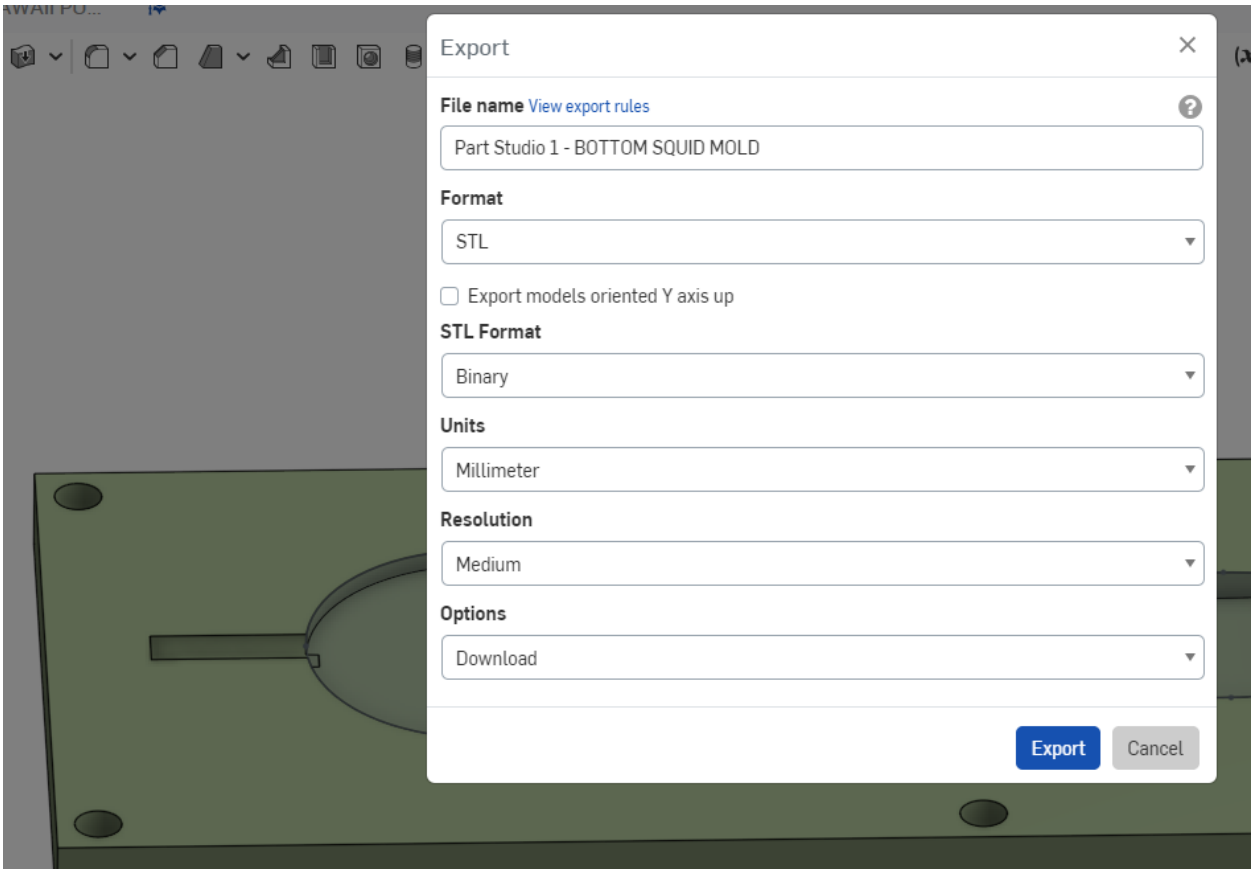


Hide the work planes and hide the bottom mold and inspect the top mold and then hide the top and inspect the bottom mold.

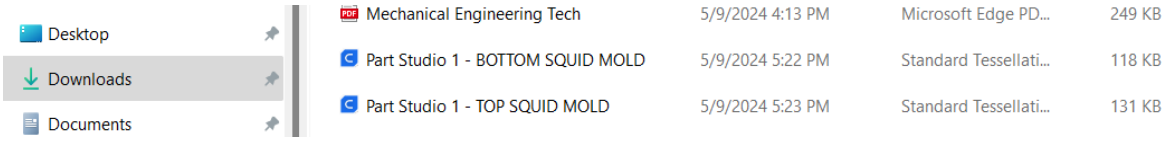




Right click on the bottom mold and select Export to create and save the bottom mold as an STL file. Do the same with the top mold.



Both exports are in the downloads file so now copy into your file to be able to slice the STL files and then 3D print them.



Great job.