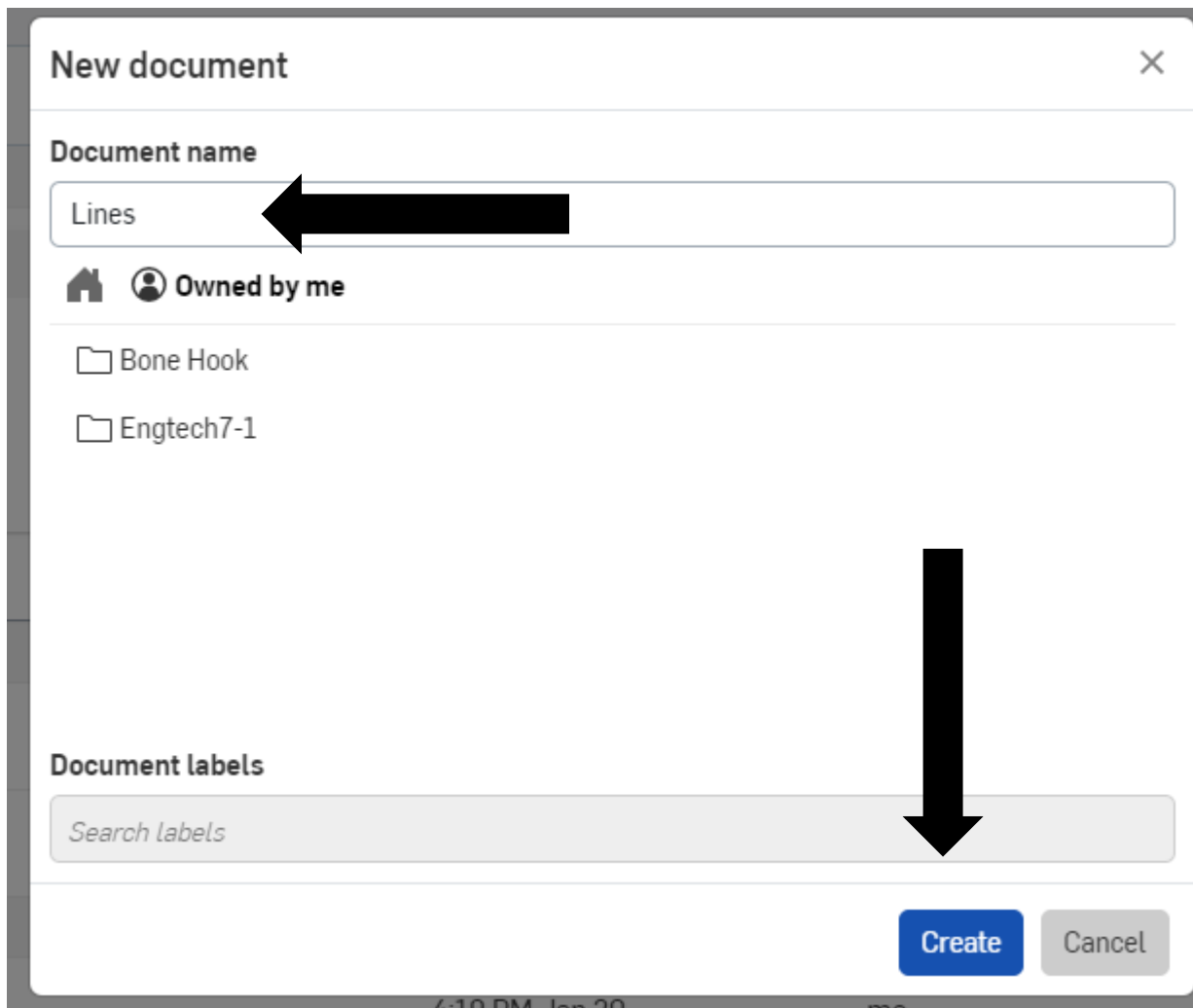
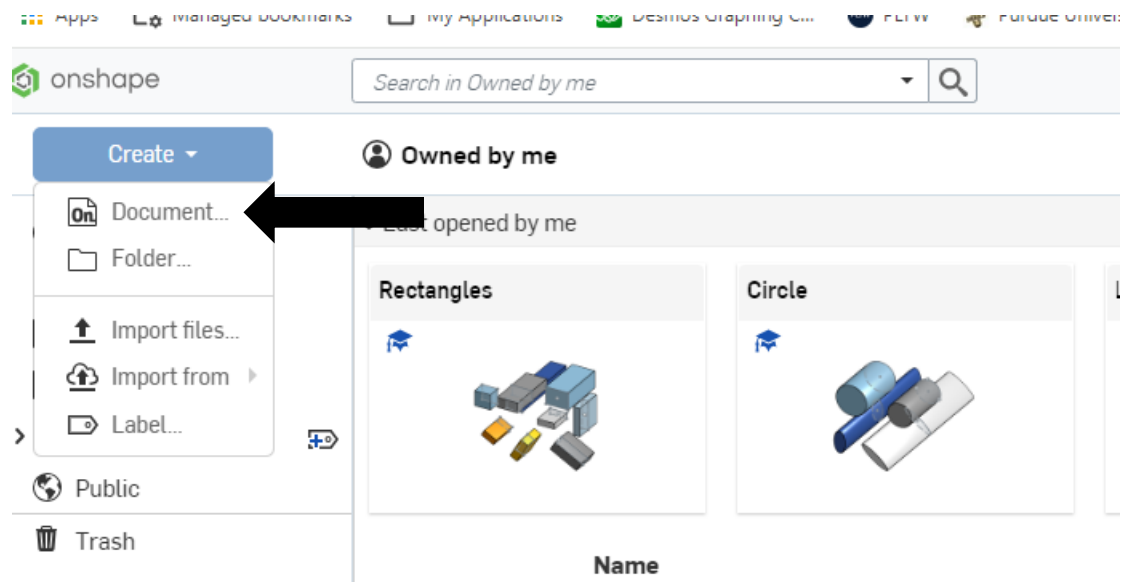
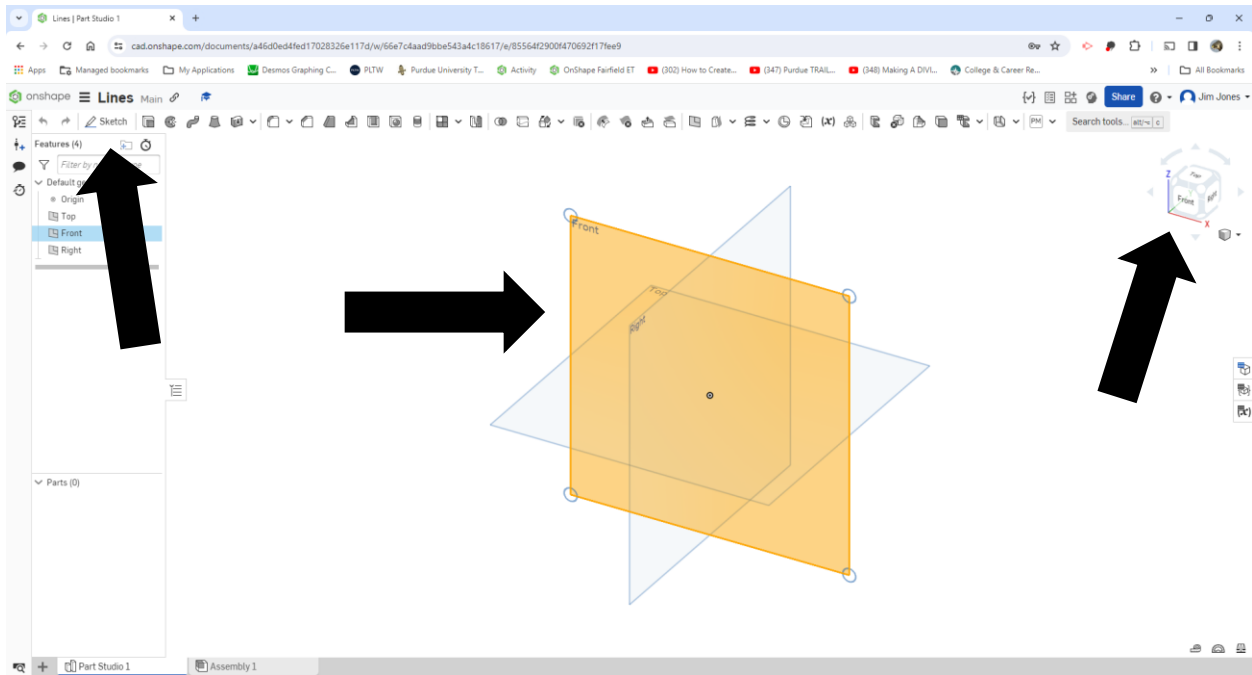


# Onshape Introduction Lines

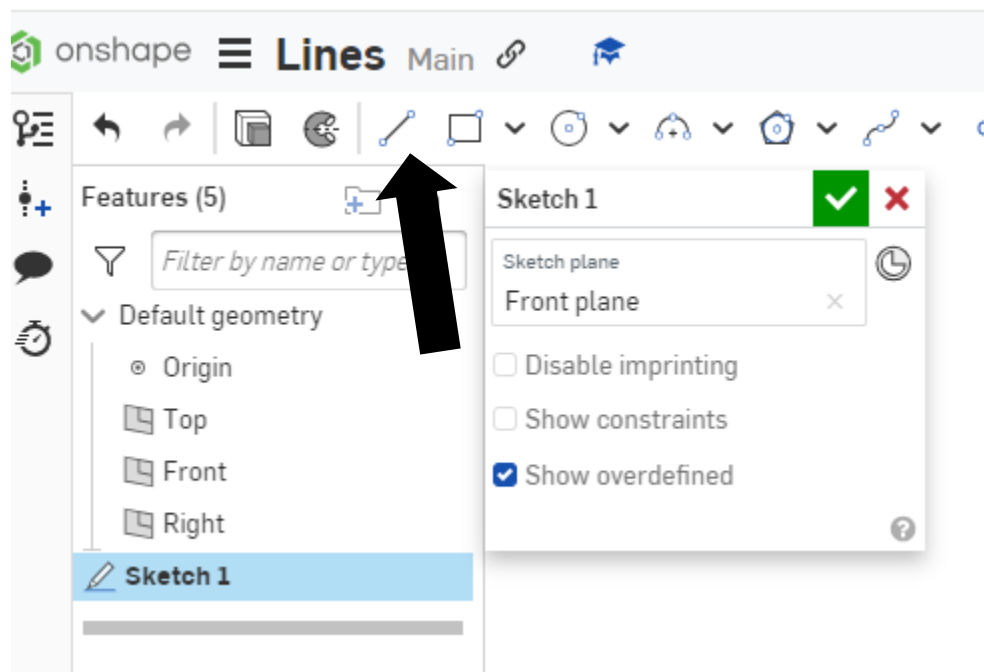
Create a new document and name it Lines.



Select the Front plane and Front on the view cube to rotate the drawing. Left click on Sketch to start the drawing lines.



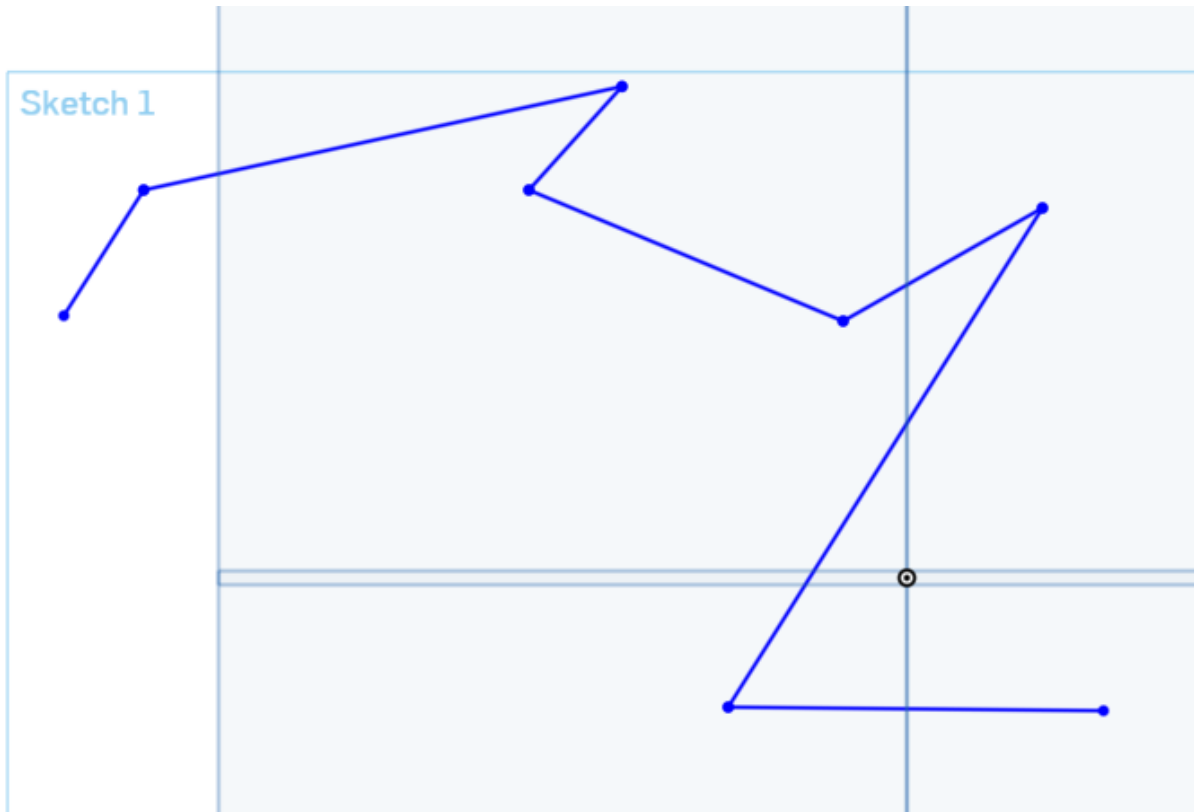
The Line icon is the angled line with a little circle on each end.



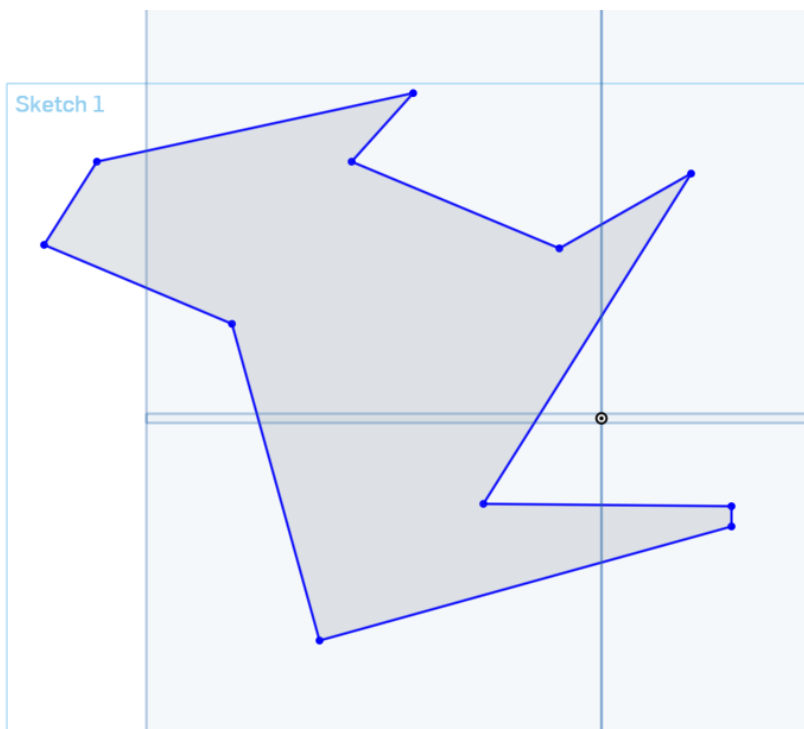
A small crosshair will show up and you can left click twice to make a line. Do not click the same location twice. The first line takes two left clicks and then you can make more lines just left clicking. All the lines are attached to the last point as you are making a new line. Right click and then left click on Escape line to stop making lines or just press the ESC key.



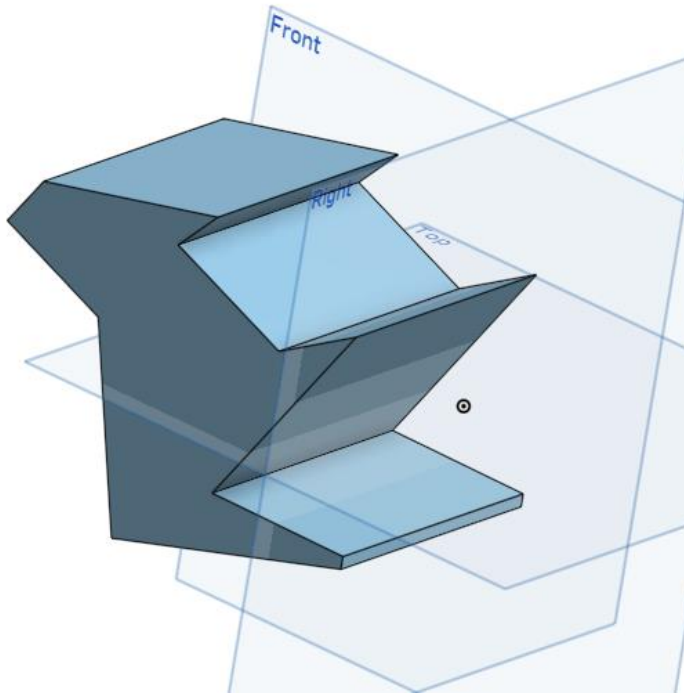
You can manipulate the lines by left clicking on the intersection dot and moving the point.



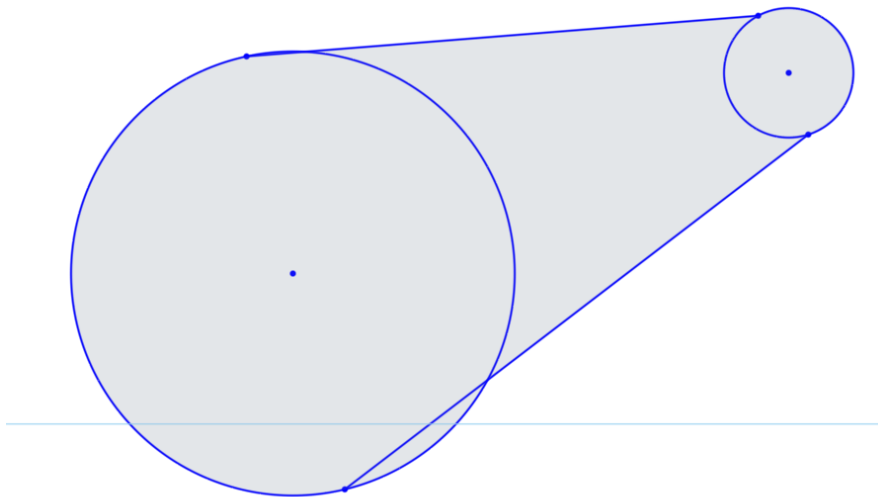
If you add a few more lines and connect the end point of the last line with the first point of the first line you should have a closed drawing. This will allow you to extrude the part.

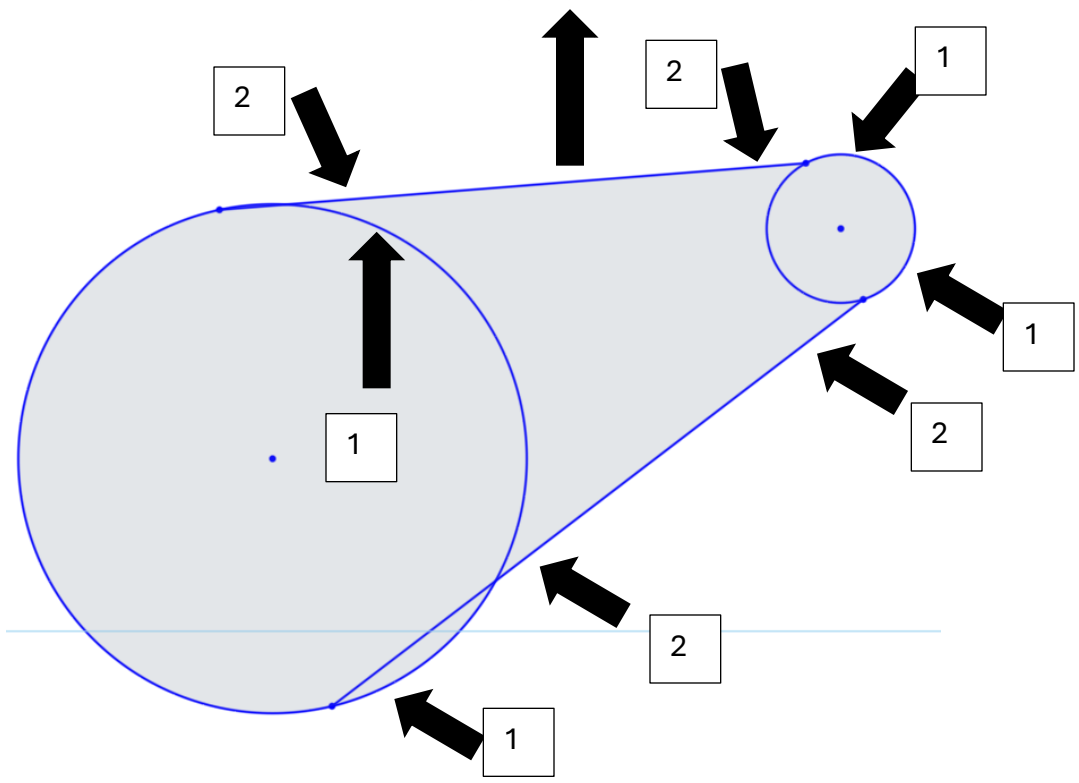


Left click on the green check mark to finish the sketch. Then extrude the part. Lines are great for sketching but you must make sure the end points are connected to make an extrude.

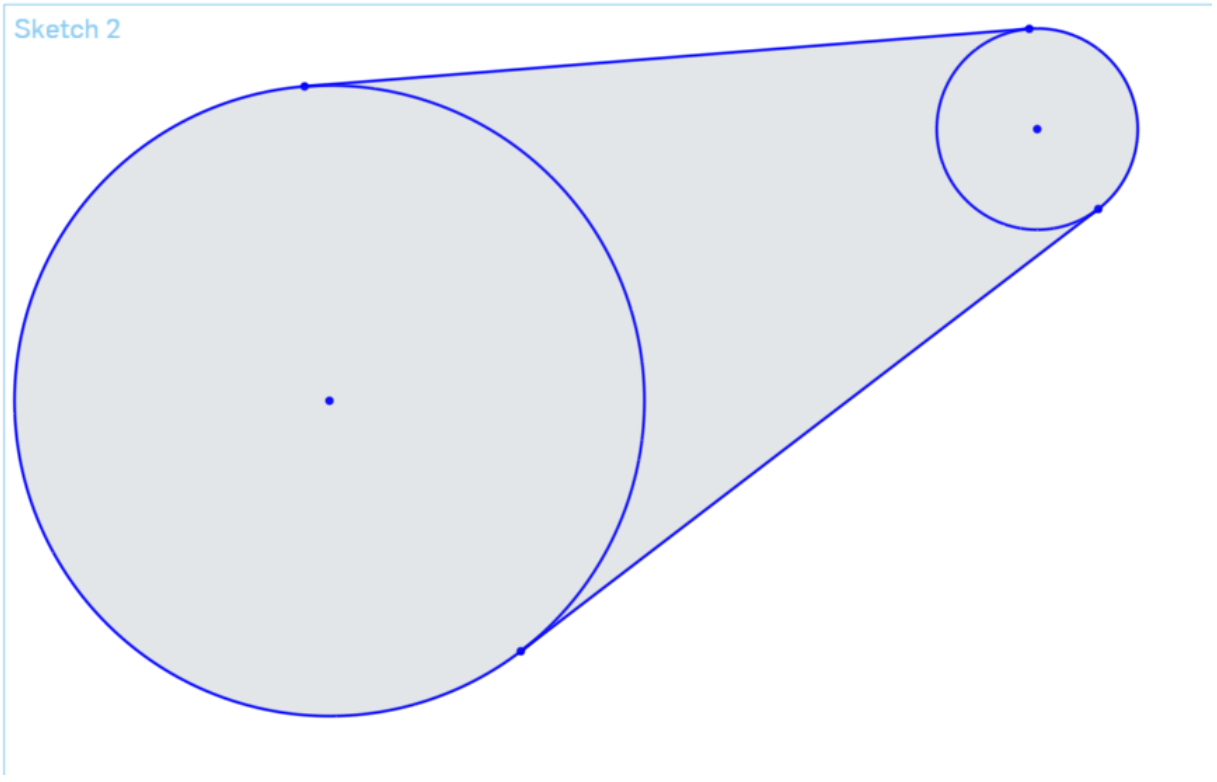


Lines can be used with other shapes to make custom shapes as seen below. Two circles and two lines. The use the Target constraint to correctly attach the lines to the circles.

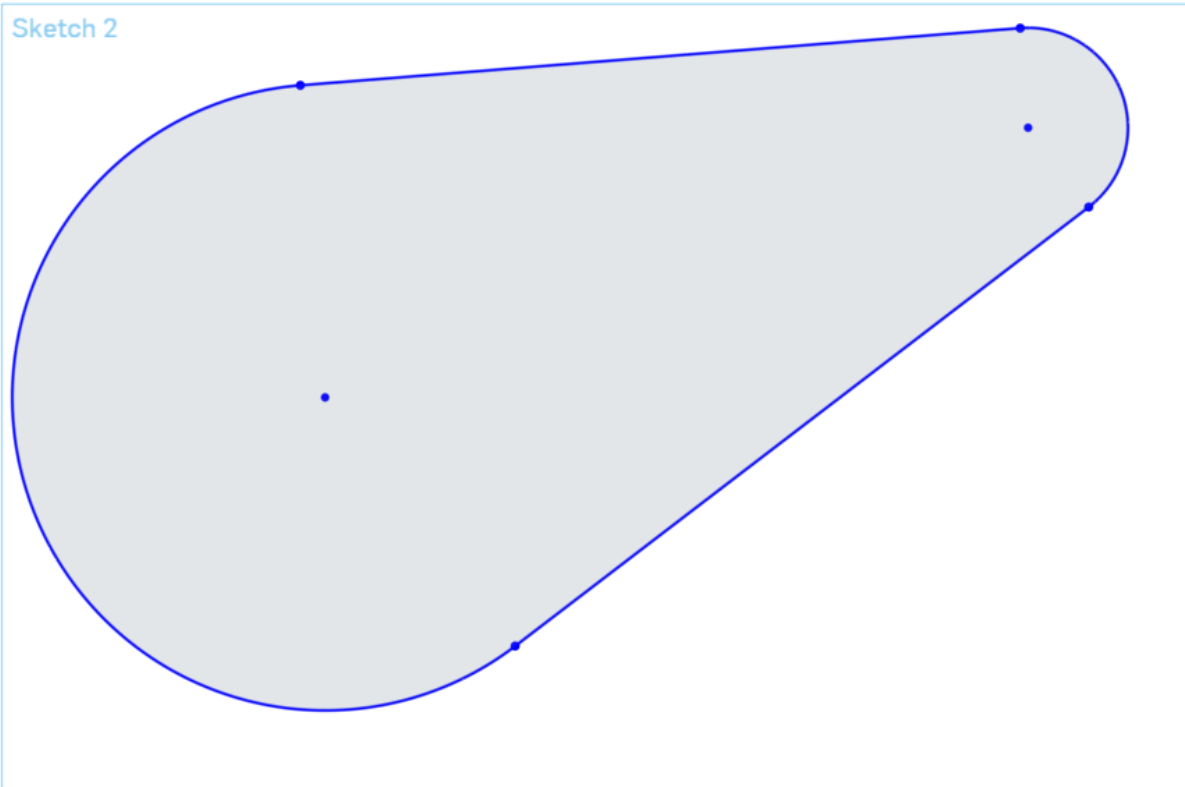
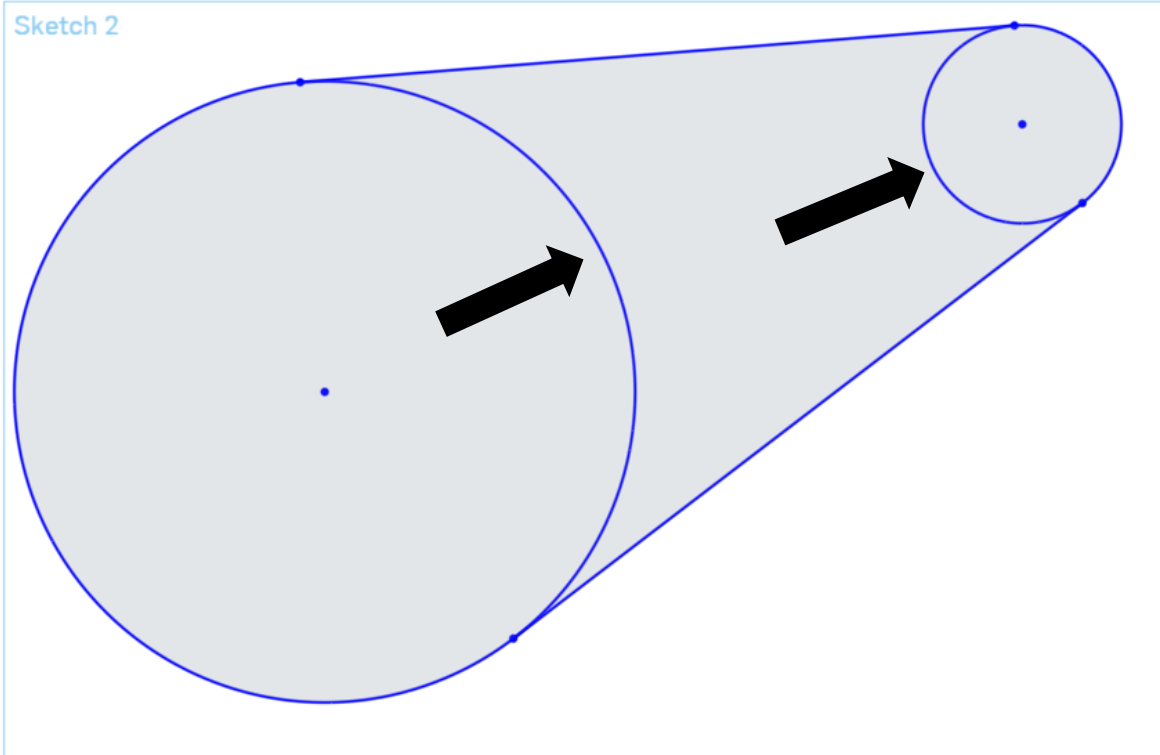




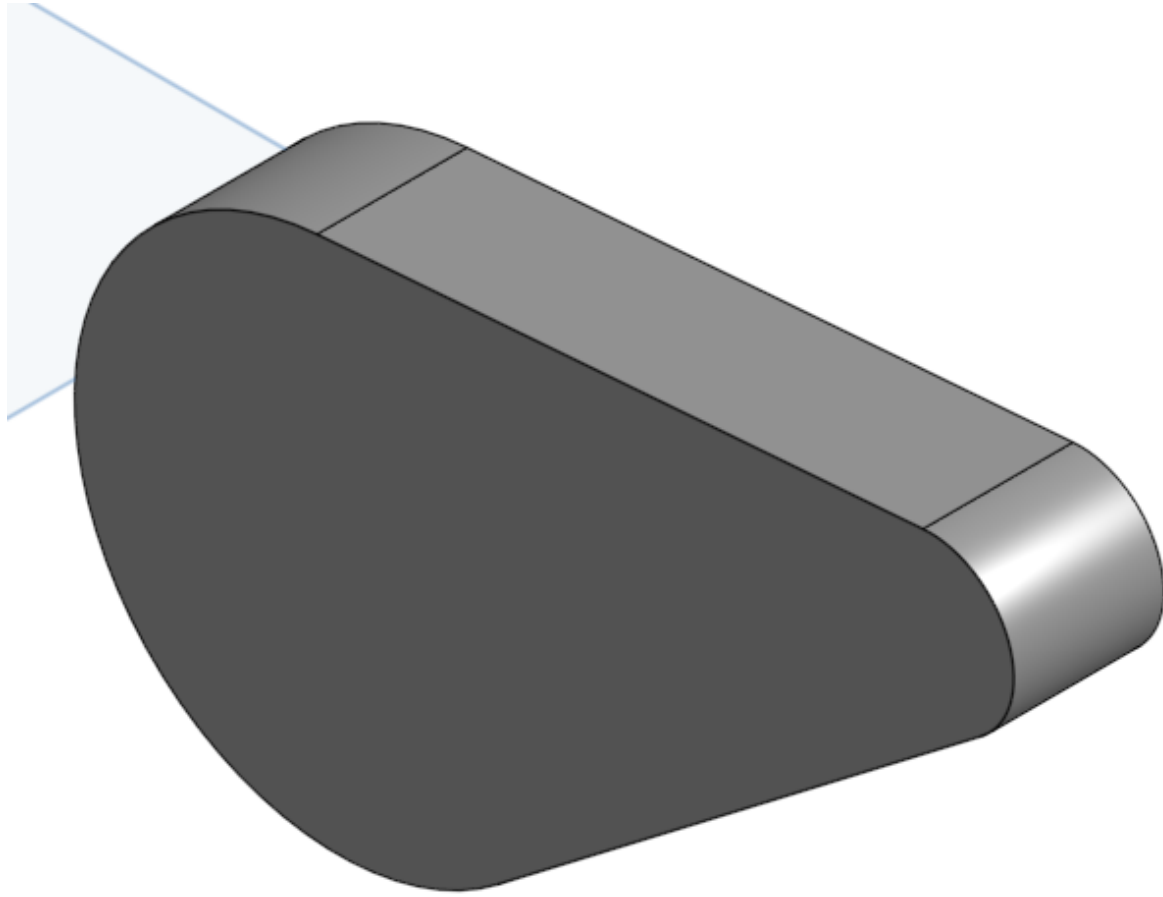
Left click on the circle then near the line intersection with the circle.



Select the Scissor icon to Trim the inside arcs of the circles to form the new part.



The arcs are gone so left click on the green check mark to finish the sketch. Use the Isometric icon to rotate the part and then extrude to 1”.



Practice making shapes with lines and combining lines with rectangles and circles to make custom shapes.