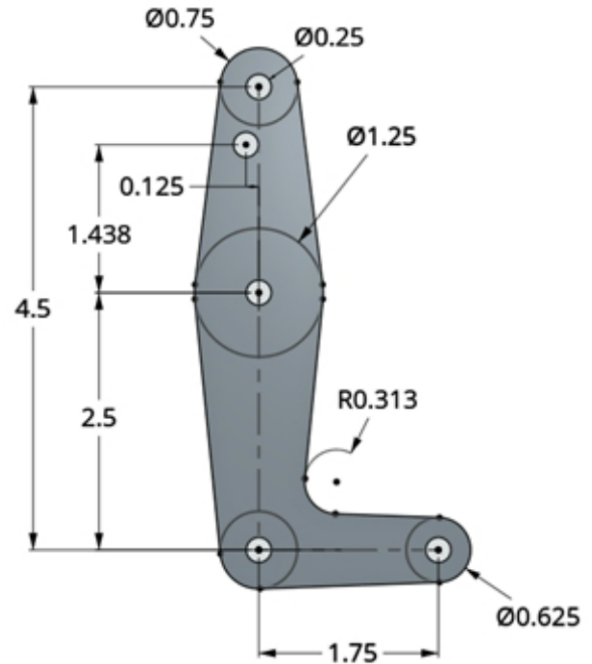# Basic Sketching

Goals:

1. Practice sketching using a variety of sketch tools.

2. Practice applying constraints to sketch entities.

3. Learn when to apply a constraint vs. a dimension to a sketch entity.

4. Create a fully defined sketch.

5. Create a basic part by extruding the correct sketch face.

Basic Sketching

Instructions:

1. Open the Onshape document "[Onshape Instructor Kit - 1.2.1 - Basic Sketching](#)".

2. Create a new part studio tab and create a new sketch on the Front Plane.

3. Use the Line tool and Circle tool to create the sketch profile shown on the right. Be sure that the circle at the center is not sketched at the midpoint of the vertical line.

   *(Hint: Note the location of the origin).*
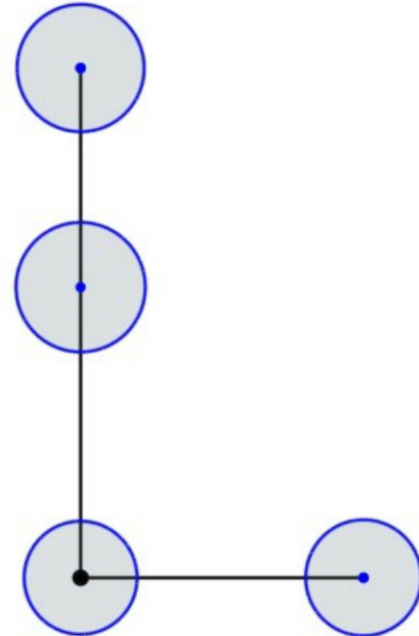
Basic Sketching

Instructions:(continued)

4. Add dimensions to fully define the sketch geometry using the Dimension tool. Experiment by selecting line segments, or combinations of lines and points as you place each dimension, and type in the values shown here.

   (Hint: Be sure to select the top and bottom circles along the vertical line and make them equal.)
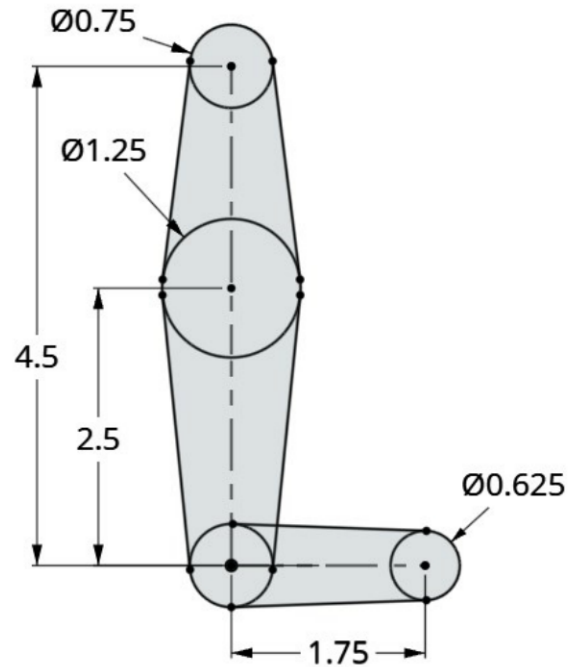
5. Change the vertical and horizontal lines to construction lines using the Construction tool.
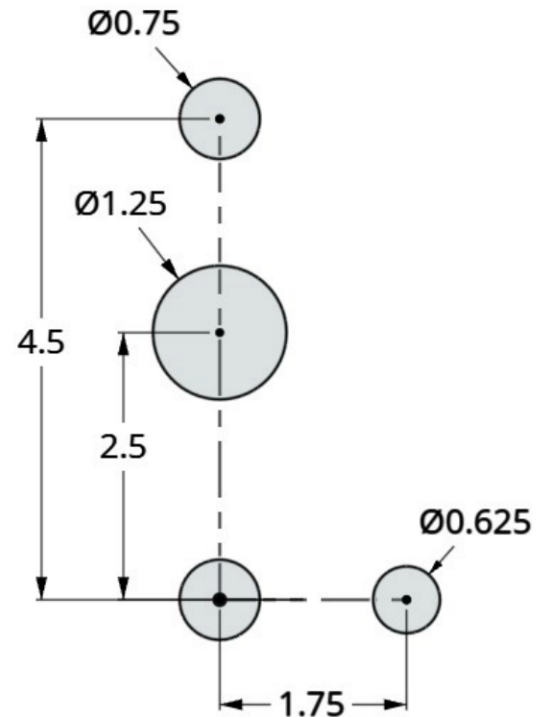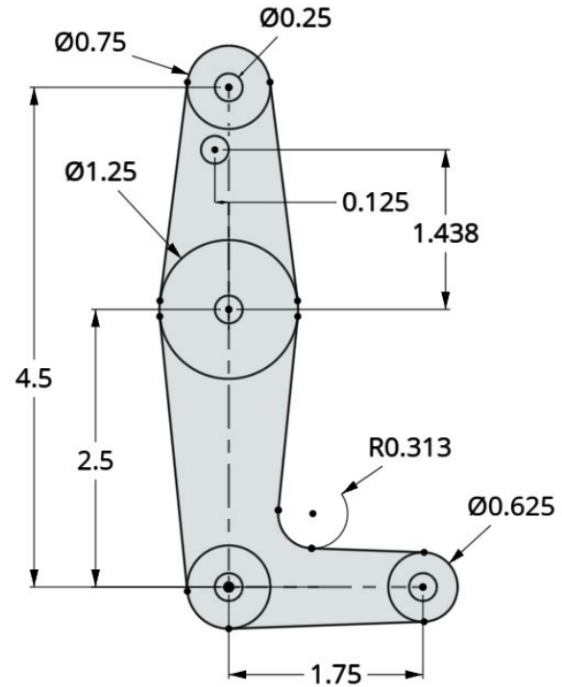
Basic Sketching

Instructions:(continued)

6. Add lines that connect the outsides of the circles as shown in the sketch profile to the right.

7. Fully define the lines by adding Tangent constraints between each line and the circle it connects to.

Basic Sketching

Instructions:(continued)

8. Create four additional circles and make them equal.

9. Create a Sketch Fillet at the inside corner of the "L" where the two tangent lines intersect.

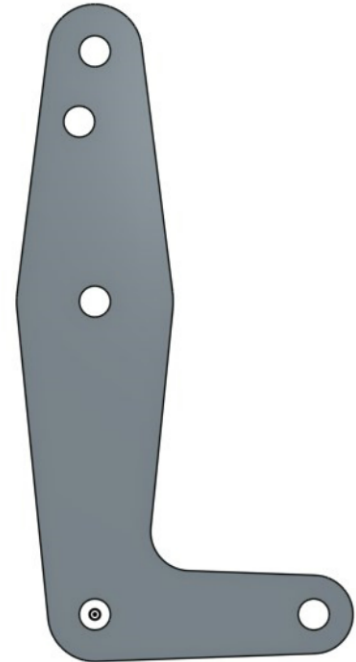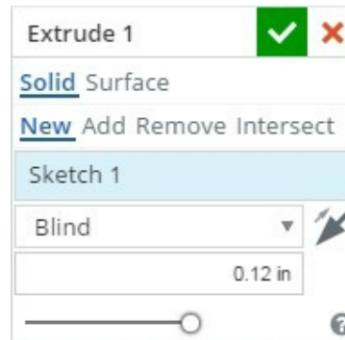10. Add the remaining dimensions to fully define the sketch.

Basic Sketching

Instructions:(continued)

11. Create a new part by extruding the correct sketch faces.

   (To create an Extrude, click on the 🗇 *icon in the toolbar, then select the settings shown on the right and click the green check.)*

Extrude 1	✓	✗
Solid Surface		
New Add Remove Intersect		
Sketch 1		
Blind ▾		
0.12 in		

Basic Sketching

Assessment:

1. Select the part in the features list.

2. Click on the ⚖ icon in the lower right corner of the Onshape interface.

What is the volume of the part (in^3)?