Basic Features 1

Goals:

1. Practice creating a Revolve feature using an existing sketch.

2. Practice creating an Extrude feature using an existing sketch.

3. Determine how to create an extrusion by sketching on a face.

4. Practice applying fillets to specific edges.
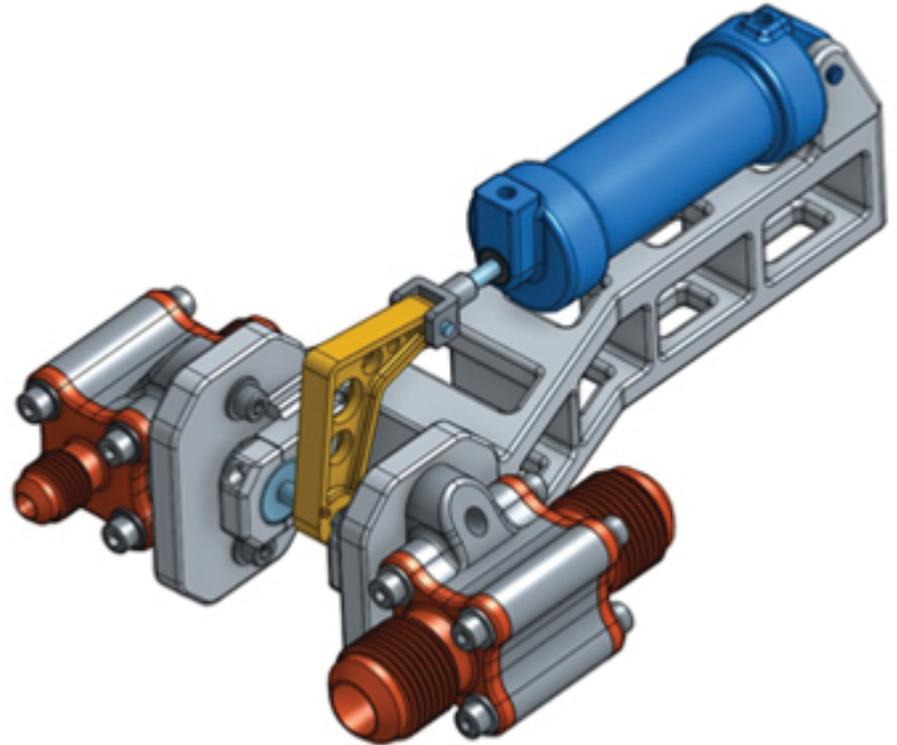
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Instructions:

1. Open the Onshape document "Onshape Instructor Kit - 2.2.1 - Basic Features 1".

2. Create the Revolve feature shown to the right using the existing sketch.

3. Create the Extrude on the left side of the part using the existing sketch. The thickness should be set to 0.25 inches.

   (Hint: Be sure to select the correct sketch faces and end types for both features.)
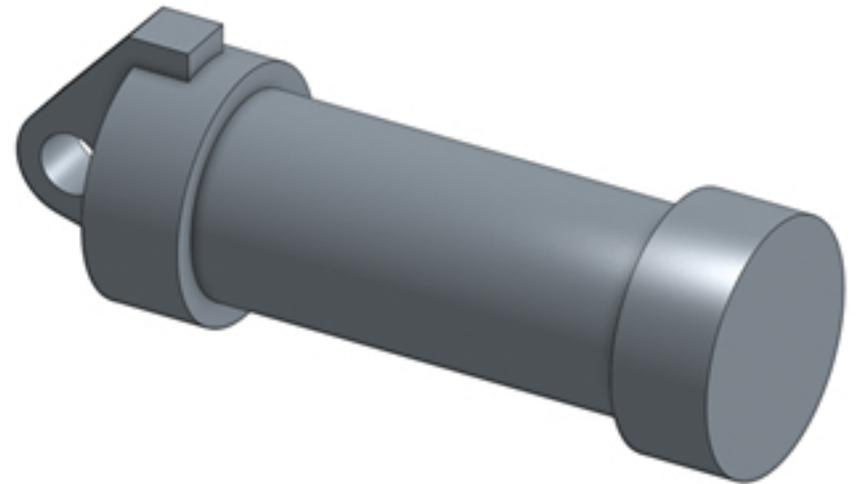
Basic Features 1

Instructions:(continued)

4. Create the sketch shown in the image to the

right on the right face of the revolved cylinder.

0.6

Ø0.12

Ø0.4
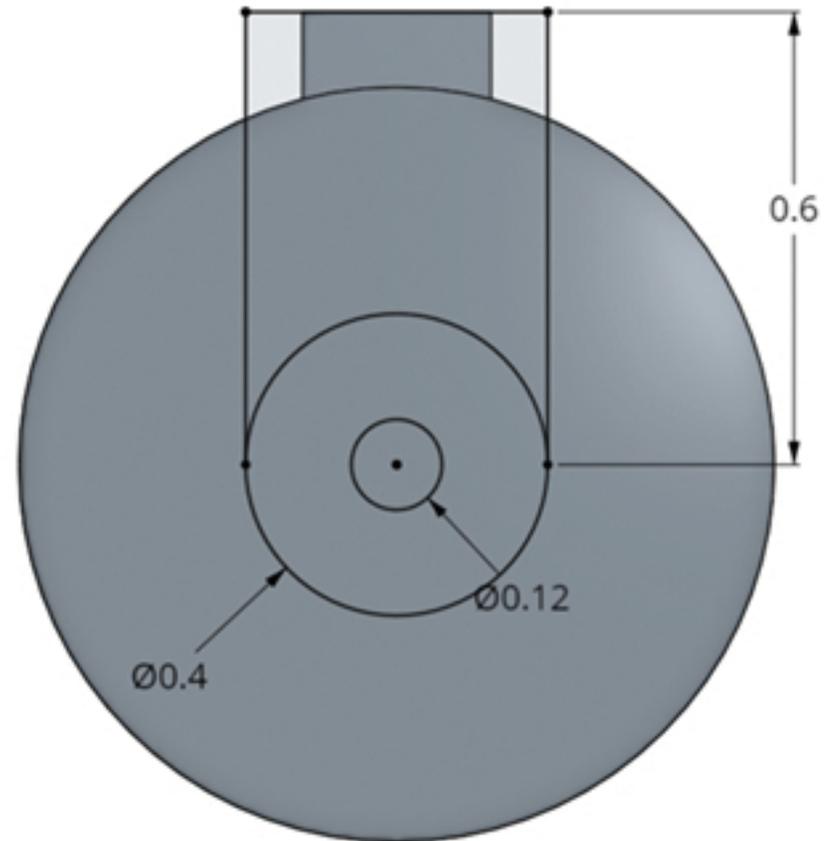
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Instructions:(continued)

5. Create the extrusions on the right side of the part. The total thickness of the extrusion is 0.3 inches, and it extends into the part by 0.05 inches.

   (Hint: You will need to use multiple features to create the geometry.)

6. Create a hole on the right side of the part that extrudes up to the face shown in the image on the right.
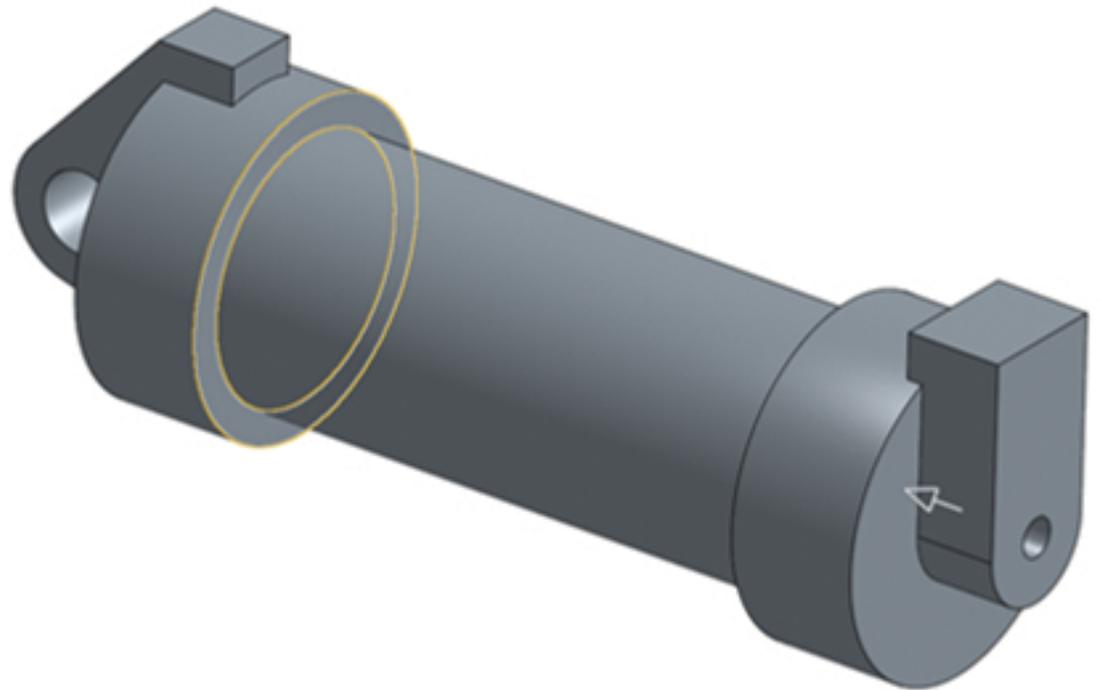
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Instructions:(continued)

7. Create the sketches for the two holes on the top faces of the part using a diameter of 0.12 inches for both holes. Make sure that the holes are centered on the faces.

8. Extrude the hole on the left to a depth of 0.7 inches.

9. Extrude the hole on the right to a depth of 0.41 inches.
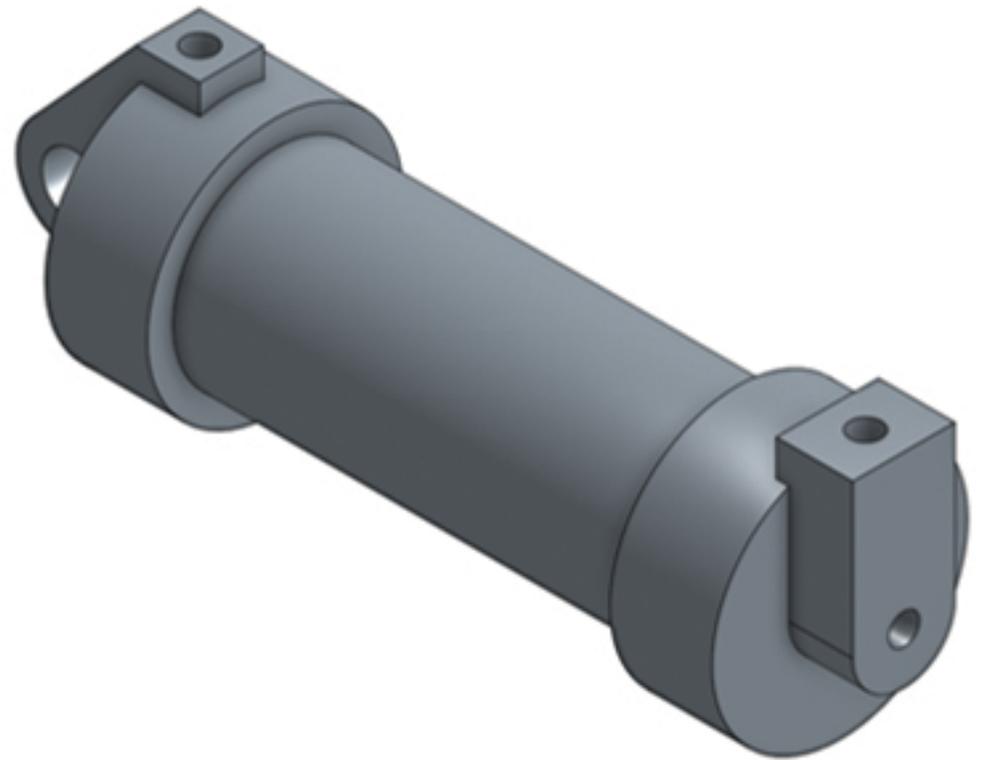
Basic Features 1

Instructions:(continued)

10. Add fillets to the edges of the cylinder using a radius of 0.05 inches.

11. Add fillets to all external edges on the left side of the part (excluding holes) using a radius of 0.05 inches.

12. Add fillets to all external edges on the right side of the part (excluding holes) using a radius of 0.05 inches.
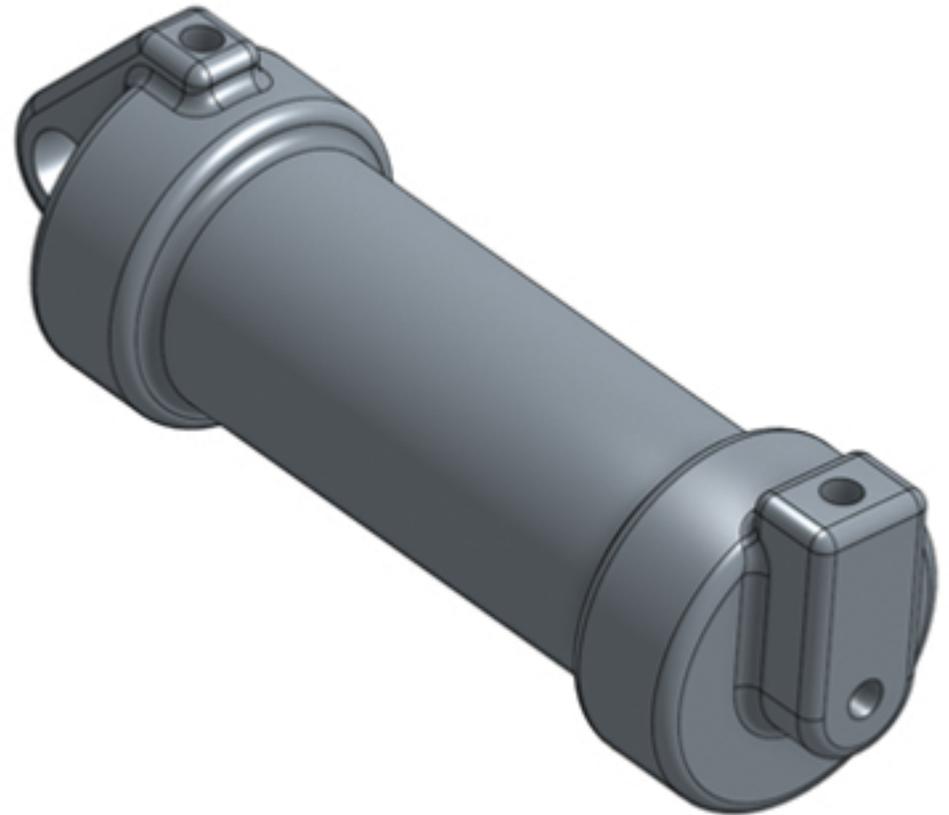
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Assessment:

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)?