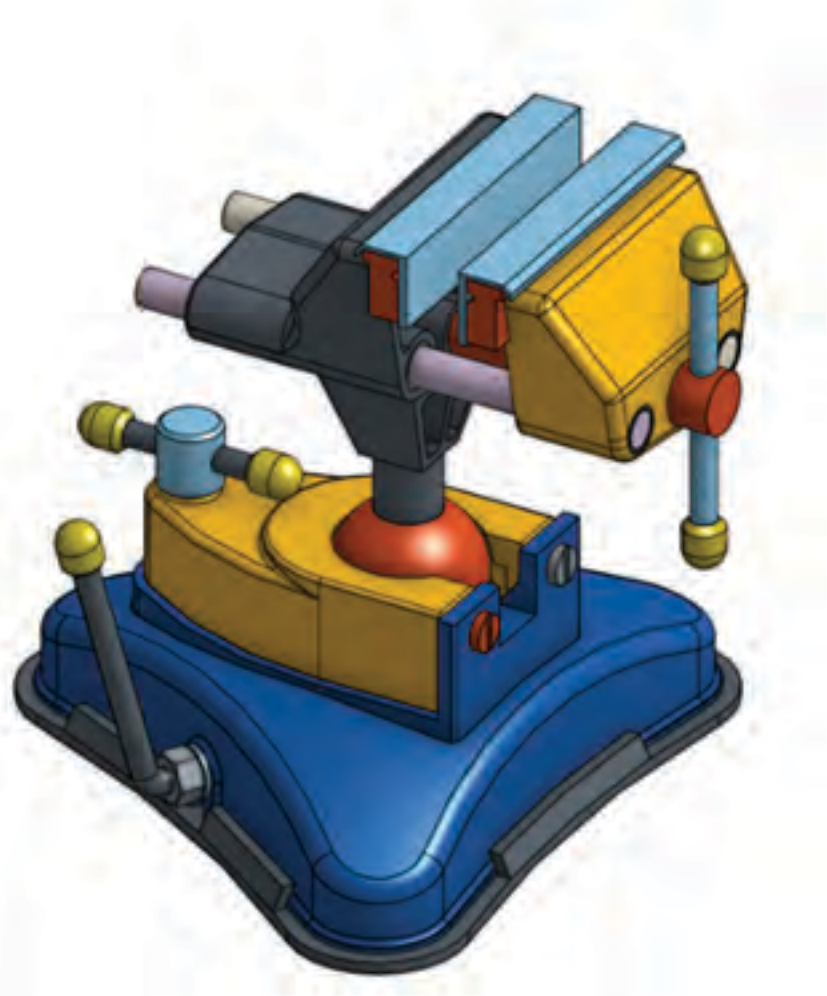
In-Context Part Design 2

Goals:

1. Practice creating a multipart design in the context of other parts.

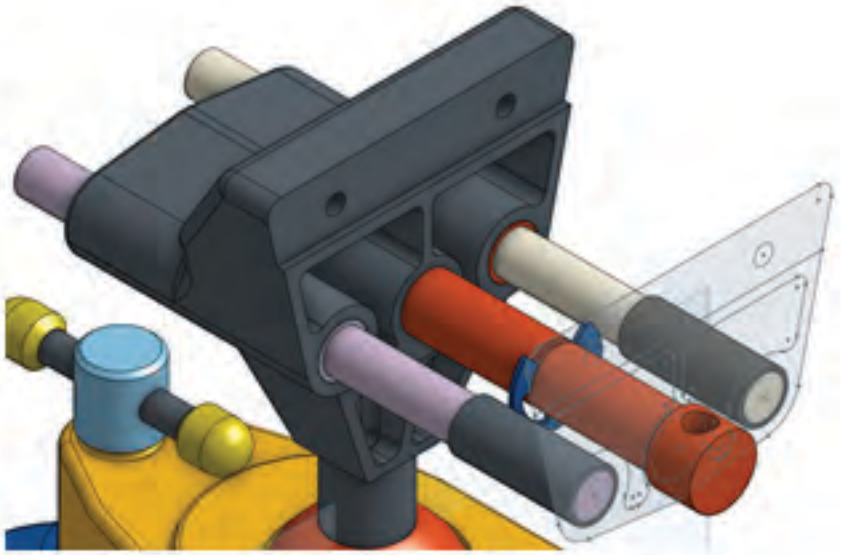
2. Determine which faces and edges of the existing parts to “Use” to create the new parts.

3. Determine which end condition to use for each feature.



In-Context Part Design 2

Instructions:

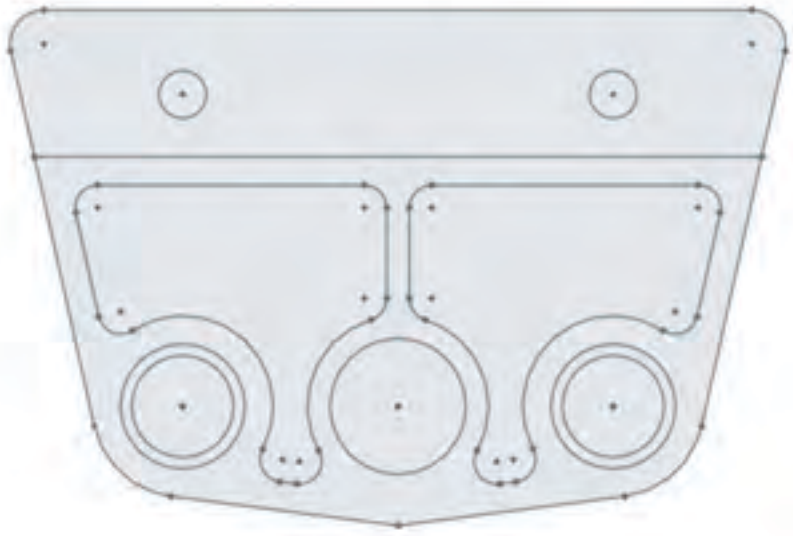


1. Open the Onshape document "Onshape Instructor Kit - 3.2.2 - In-Context Part Design 2".

2. Begin a sketch on the end face of the left or right rod.

3. Create the sketch shown to the right by projecting existing geometry onto the sketch plane.

   (Hint: This will require converting edges from the clamp as well as the sleeves around the ends of the rods.)



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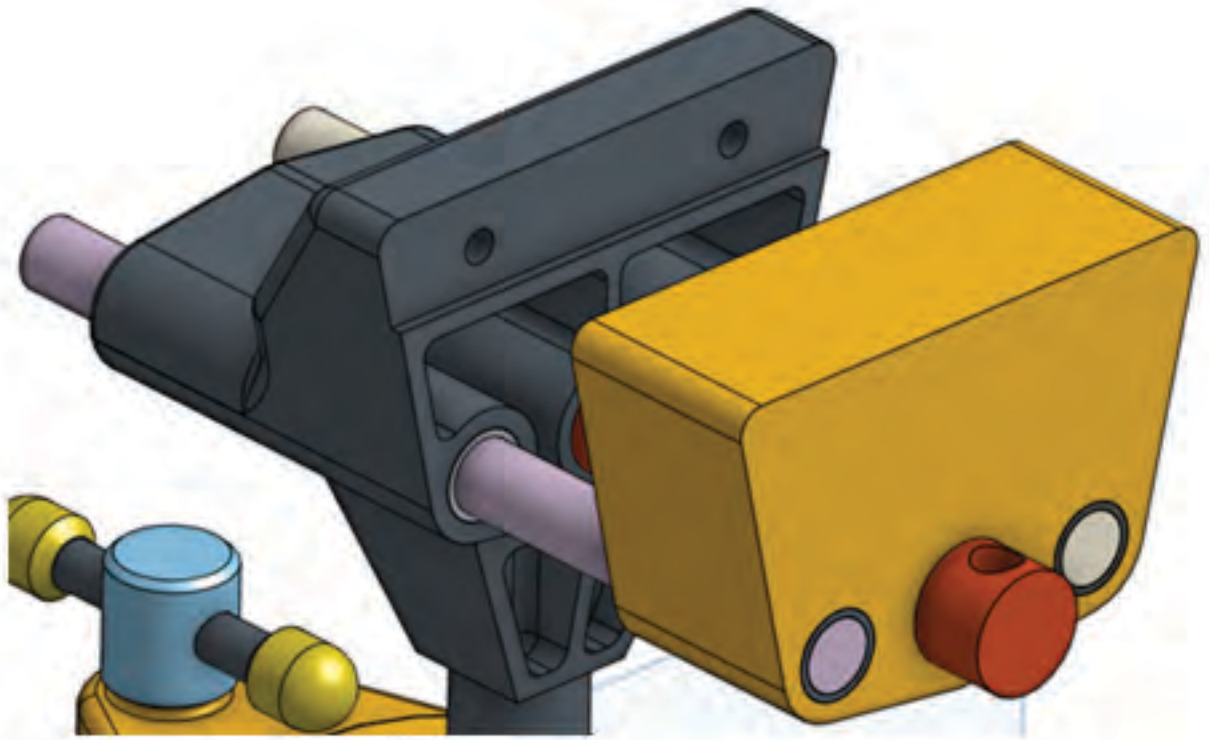
Instructions:(continued)

4. Extrude the appropriate sketch faces until they reach the ends of the sleeves.

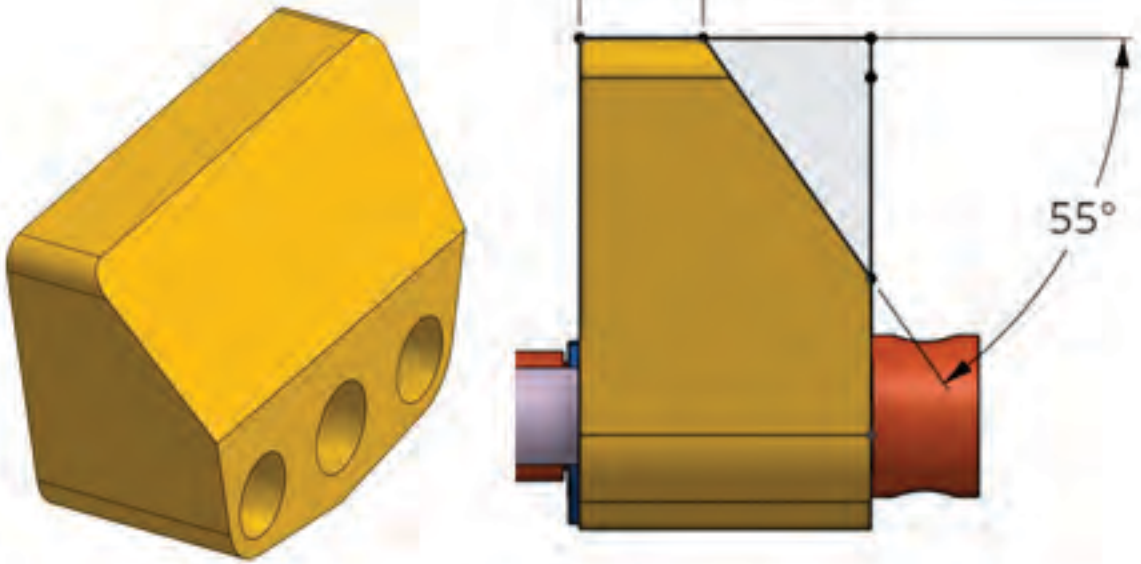


In-Context Part Design 2

Instructions:(continued)

5. Cut out the material so that the clamp is angled on one side as shown to the right. To do this, create the sketch shown on the Front plane and create a symmetric Extrude that removes material.

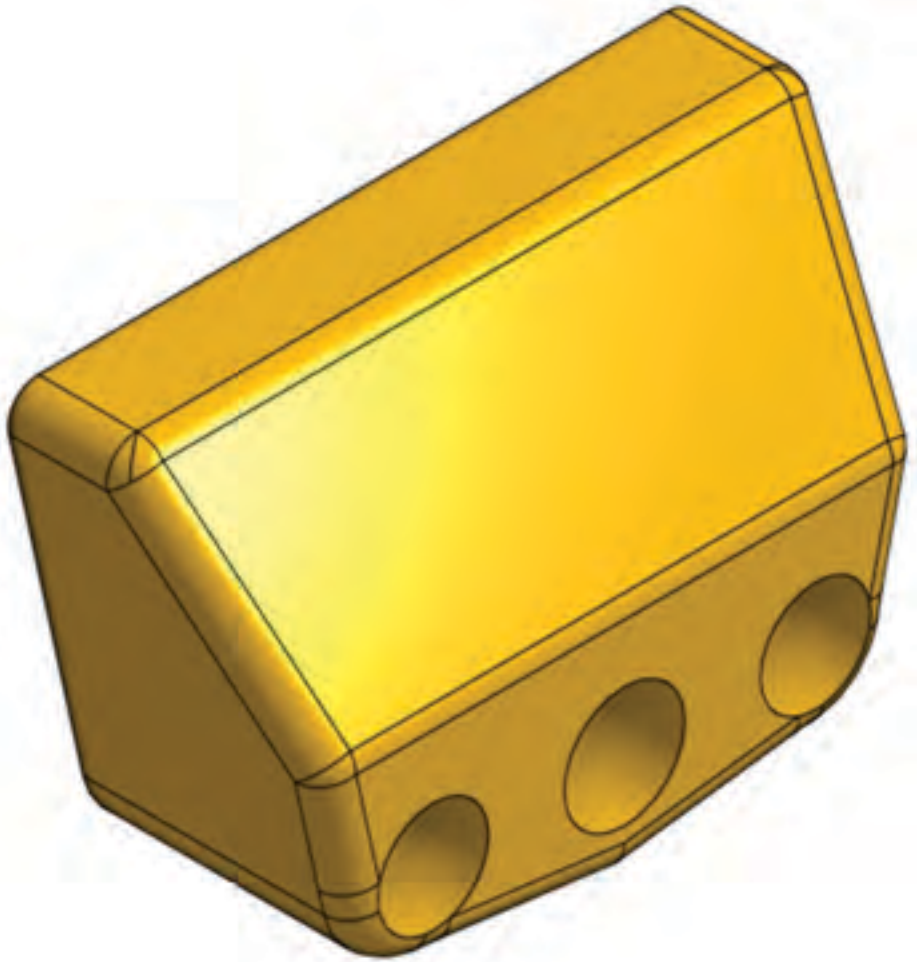
   (Hint: Project the edges of the clamp with the Use tool and constrain your geometry to the projected lines.)



In-Context Part Design 2

Instructions:(continued)

6. Add fillets to the outside exterior edges of the new vise head using a radius of 0.12 inches (excluding the holes).



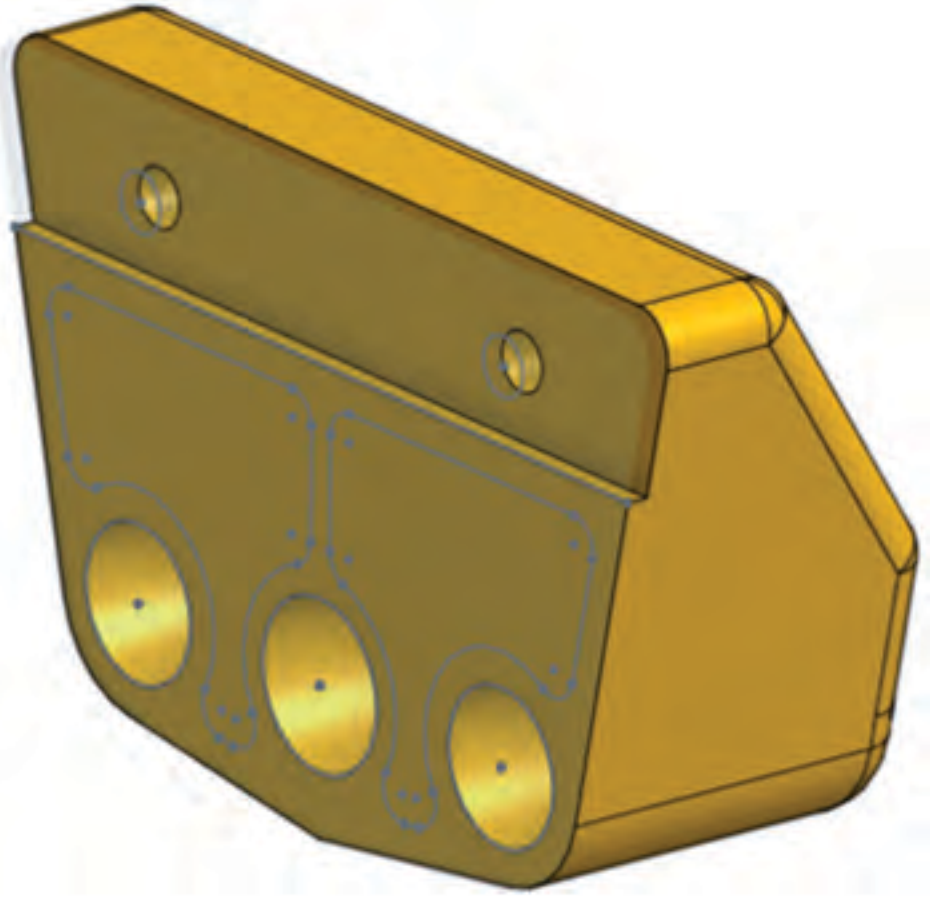
In-Context Part Design 2

Instructions:(continued)

7. Project all of the sketch entities from the initial sketch onto the inside face of the new vise head.

8. Create the cut inset into the top of the part using a depth of 0.06 inches.

9. Add the holes to this new inset face using a depth of 0.4 inches away from the sketch plane.



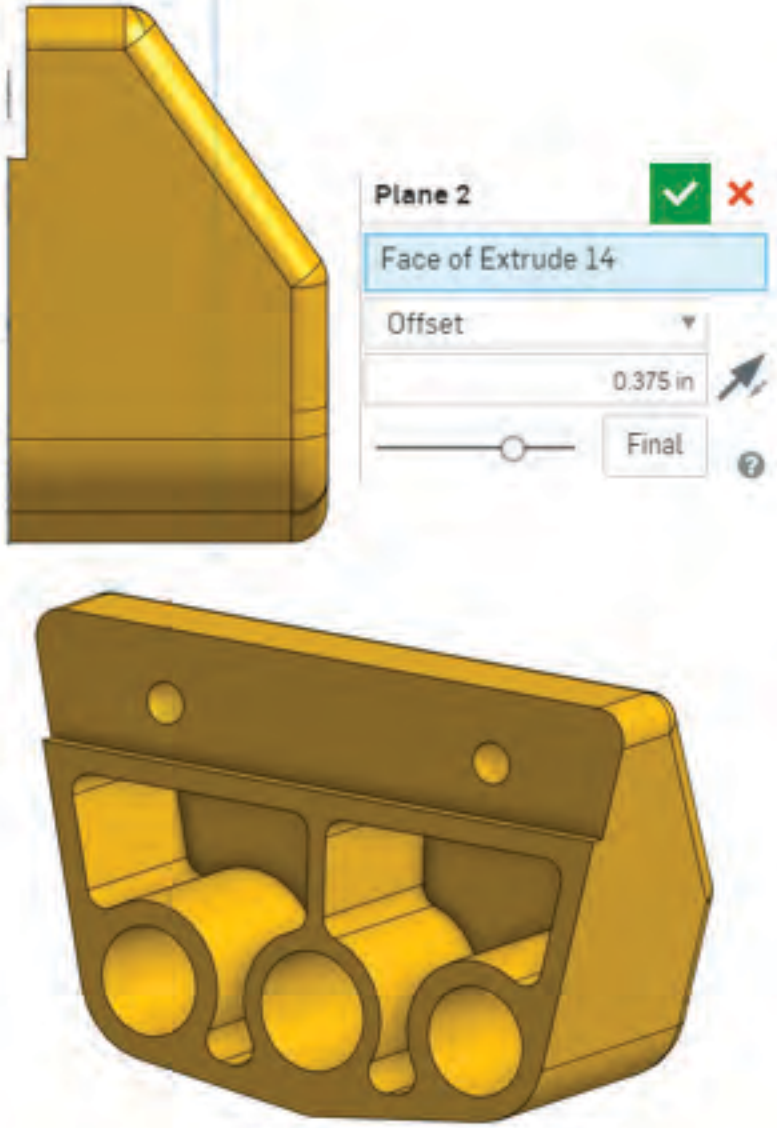
In-Context Part Design 2

Instructions:(continued)

10. Add a plane that is offset 0.375 inches from the right side of the part (in the Front view).

11. Create the cut around on the inside face of the vise that surrounds the three holes for the rods to pass through.

    (Hint: This requires a specific end type to be used to create the correct geometry.)



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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)?