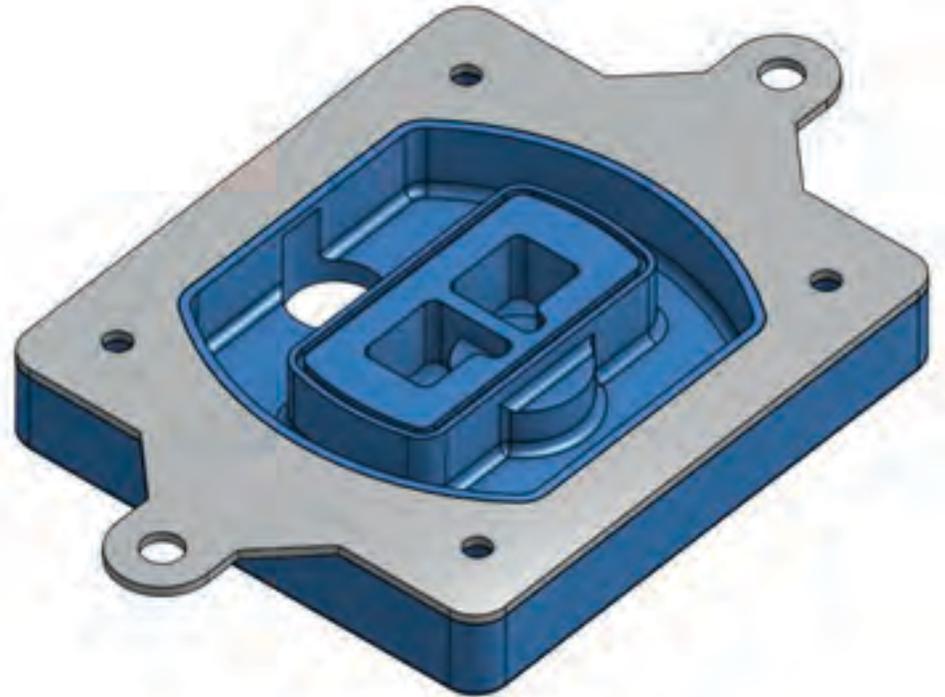


Sketching In-Context

Goals:

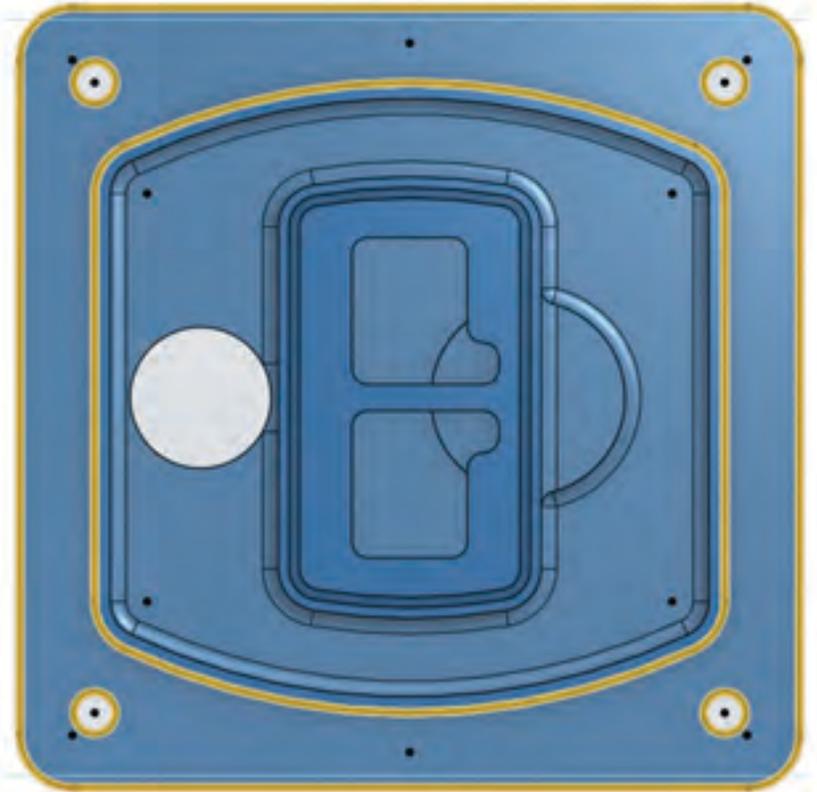
1. Create a sketch profile by utilizing geometry from an existing part.
2. Apply a variety of sketch constraints.
3. Learn how to speed up the design process when a part is symmetric.
4. Enable the new part to update when the existing part is modified.



Sketching In-Context

Instructions:

1. Open the Onshape document "[Onshape Instructor Kit - 1.2.2 - Sketching In-Context](#)".
2. Choose the tab "Bracket" and create a new sketch using the large top face of the part.
3. Select the appropriate edges as shown in the image to the right and Use them in the new sketch.

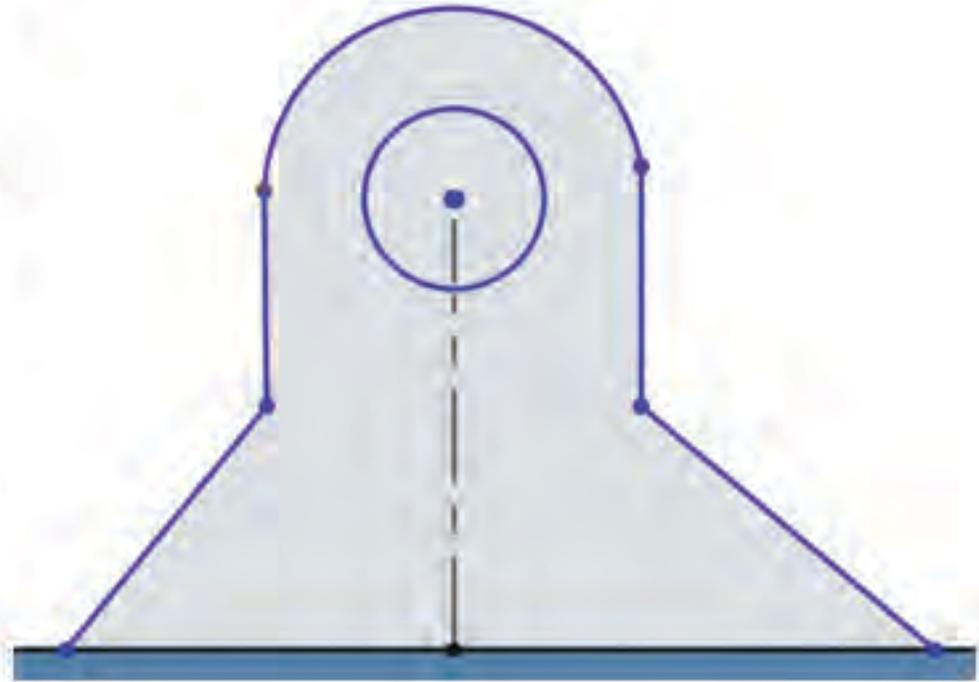


Sketching In-Context

Instructions:(continued)

4. Sketch the profile shown to the right along the top edge. This includes **Three-Point Arcs**, **Circles**, **Lines**, and **Construction lines**. Be sure that the vertical construction line connects to the midpoint of the horizontal edge.

(Hint: Be sure to make the large arc, the circle, and the vertical construction line coincident at the same point.)

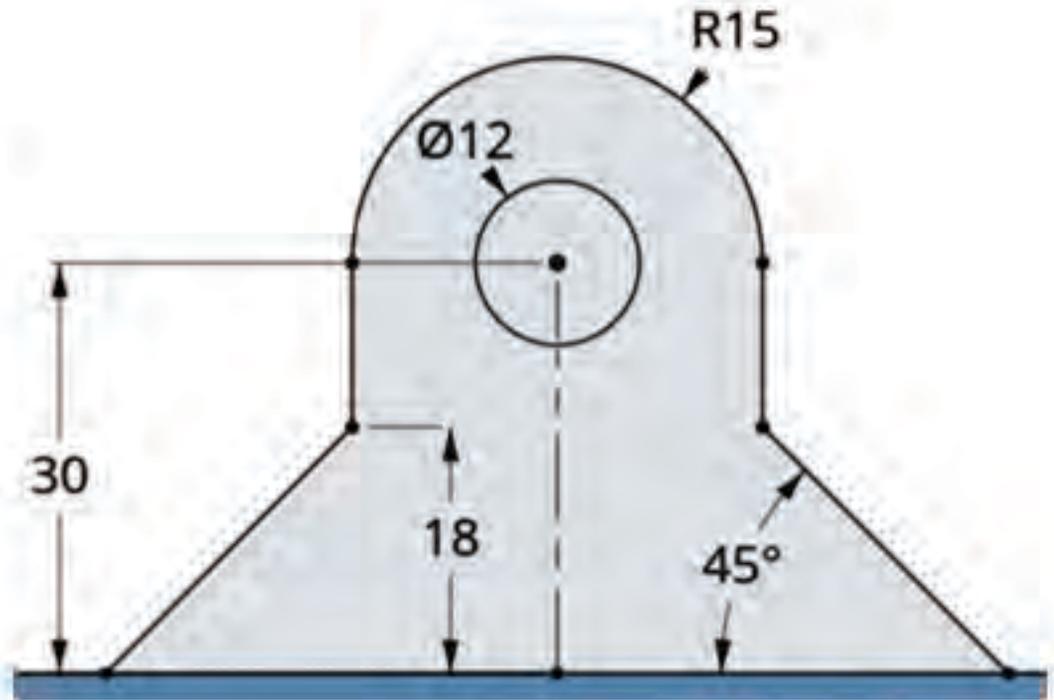


Sketching In-Context

Instructions:(continued)

5. Add the necessary constraints and dimensions as shown in the image to the right to fully define the sketch.

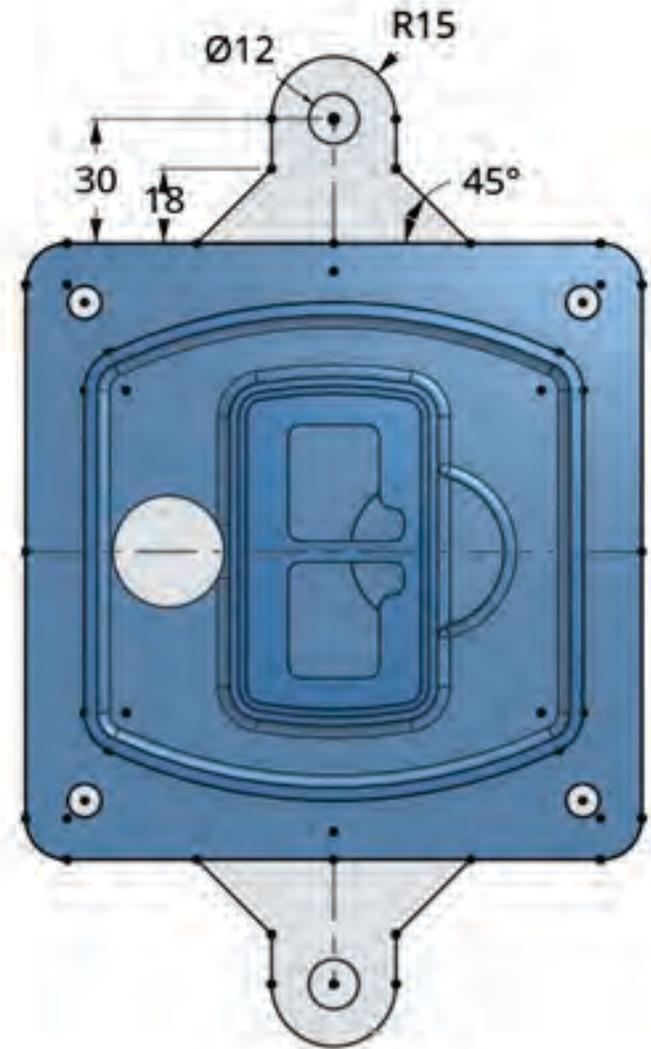
*(Hint: Relations such as **Tangent**, **Perpendicular**, and **Vertical** may be required to fully define the sketch.)*



Sketching In-Context

Instructions:(continued)

6. Create a construction line that connects the midpoints of the two vertical lines on the sides of the profile.
7. **Mirror** all of the sketch entities of the tab profile about the construction line.

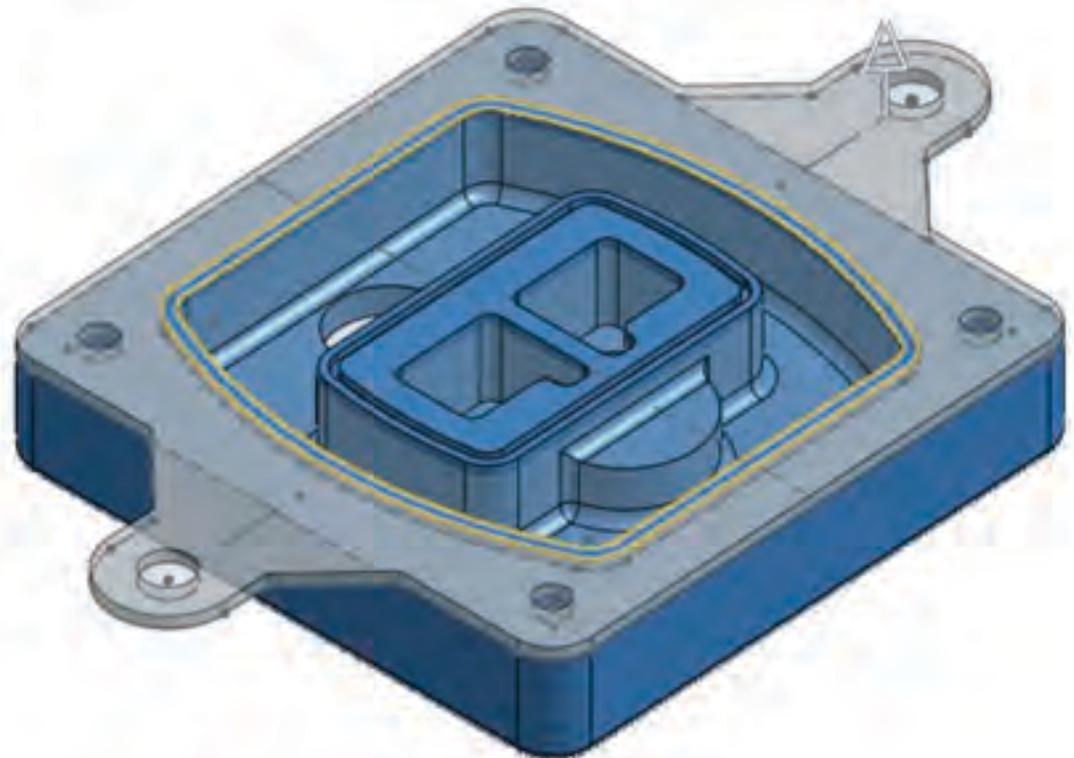
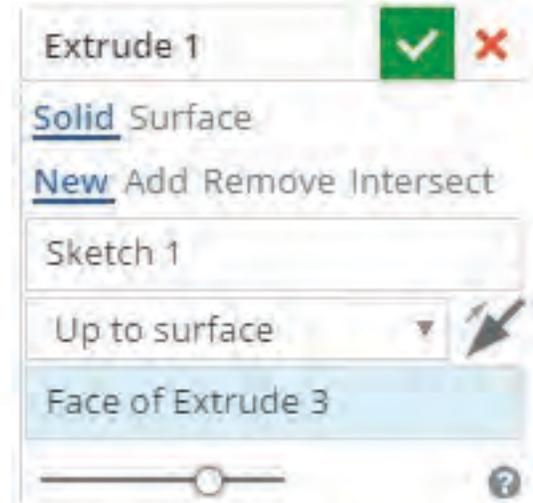


Sketching In-Context

Instructions:(continued)

8. Create a new part by extruding the correct sketch faces. Change the end condition to "Up to Surface" and select the topmost ridge of the part. This will extrude the selected sketch faces up to that ridge.

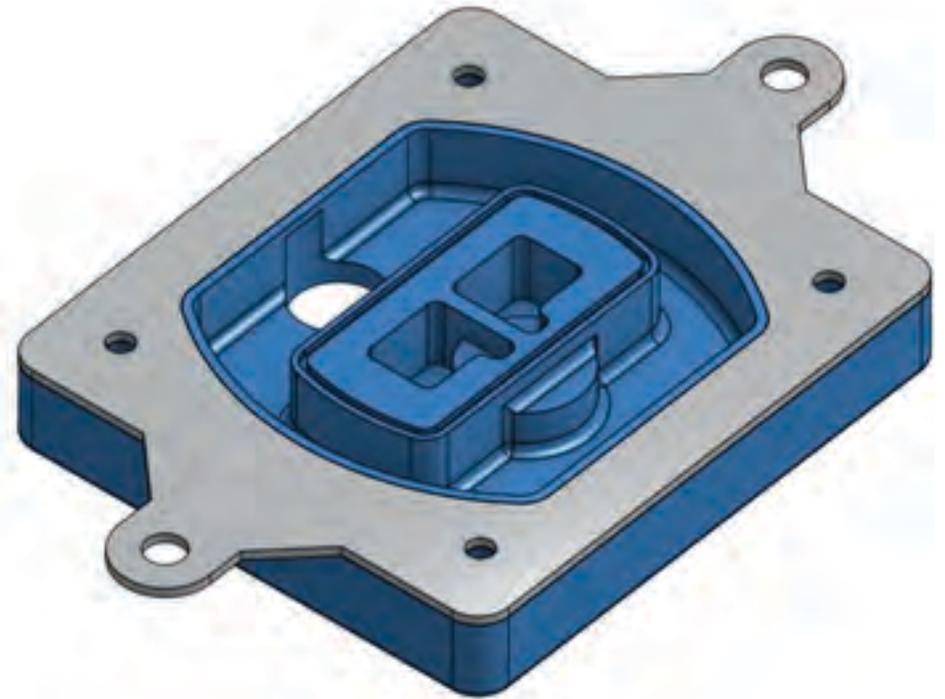
*(To create an **Extrude**, click on the  icon in the toolbar, then select the settings shown on the right and click the green check. Make sure to click **New** rather than **Add**.)*



Sketching In-Context

Assessment:

1. Right click on the first sketch in the features list and select "Edit".
2. Change the 148.5 mm dimensions to 180 mm and 160 mm as shown in the image to the right.
3. Click the green check to exit.



Sketching In-Context

Assessment:

4. Select the part in the features list.
5. Click on the  icon in the lower right corner of the Onshape interface.

What is the volume of the part (mm³)?

