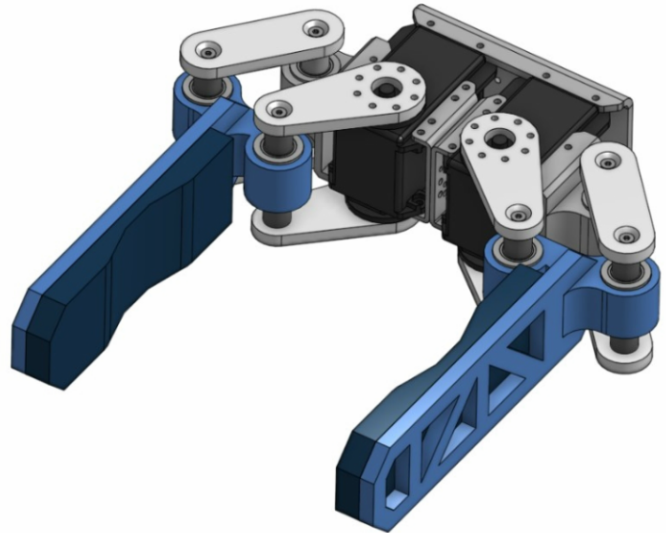


In-Context Part Design 1

Goals:

1. Create a new part within the context of a part studio.
2. Learn to assess which faces and/or planes to use for sketching.
3. Practice creating extrudes, fillets, and chamfers from sketches and features made by you.

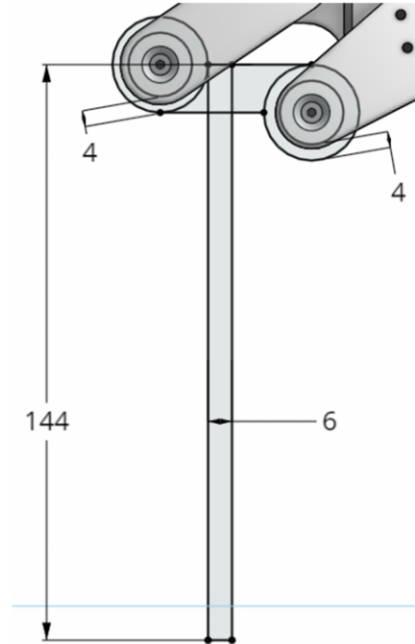


In-Context Part Design 1

Instructions:

1. Open the Onshape document "[Onshape Instructor Kit - 3.2.1 - In-Context Part Design 1](#)".
2. Begin a sketch on the face under the lip of the top bearing on the left.
3. Create the sketch shown in the image to the right.

(Hint: In order for the geometry to fit the bearing, some sketch entities must be created by projecting existing geometry onto the sketch plane.)

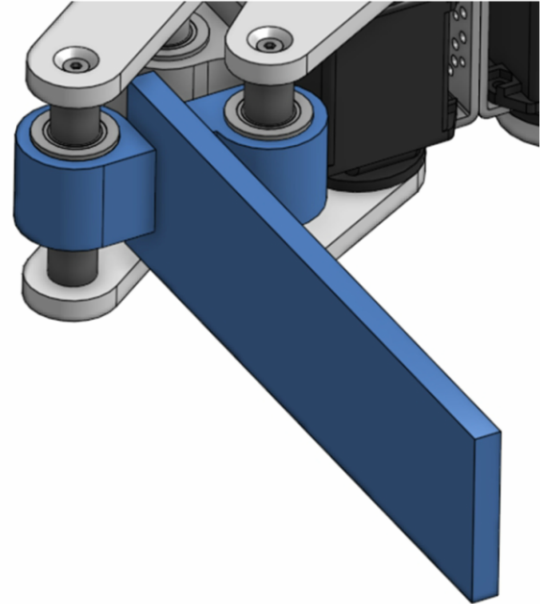


In-Context Part Design 1

Instructions:(continued)

4. Using the same sketch, extrude the correct sketch faces to make sure the arm fits between the top and bottom bearings.
5. Extrude the side plate of the arm so that it extends 10 mm above and below the end piece that connects to the bearings.

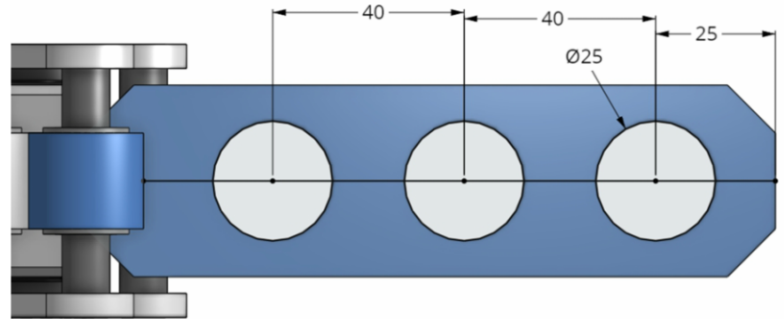
(Hint: Multiple features may need to be used to create the side plate.)



In-Context Part Design 1

Instructions:(continued)

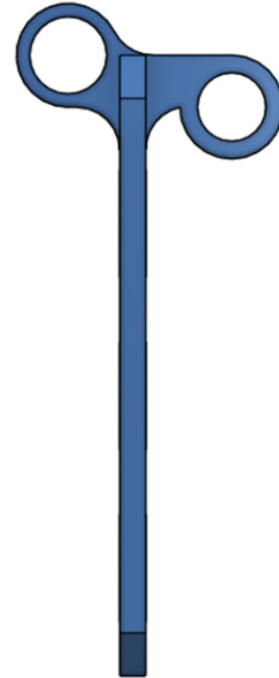
6. Add 10 mm equal distance chamfers to all four corners of the side plate.
7. Create three circular cuts through the side plate of the arm using the dimensions shown in the image to the right. All three circles have the same diameter.



In-Context Part Design 1

Instructions:(continued)

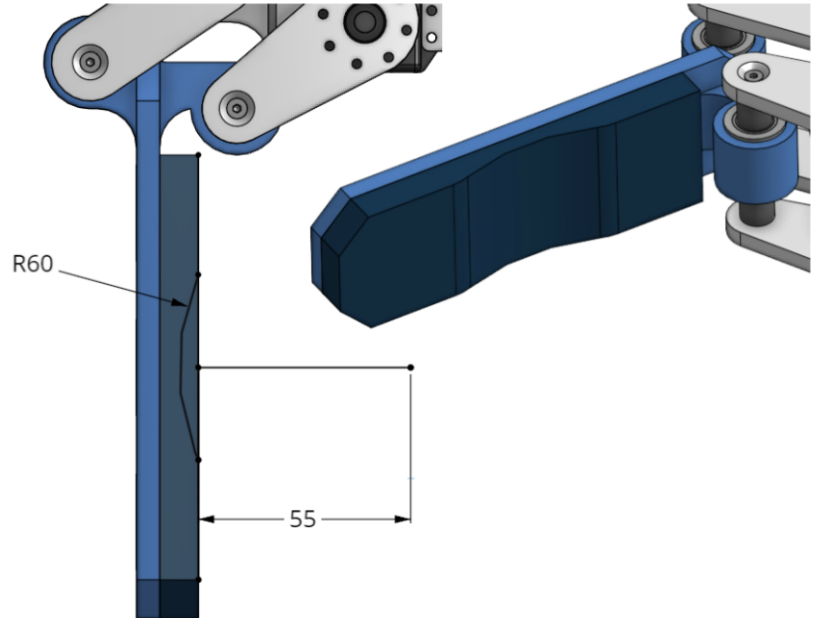
8. Add three fillets where the end piece meets the side plate of the arm using a radius of 10 mm.



In-Context Part Design 1


Instructions:(continued)

9. Create the pad on the inside of the arm as a new part. The total length of the pad is 120 mm and the thickness is 10 mm.
10. Create the curved cutout using the sketch dimensions shown to the right. The center of the curve is located at the midpoint of the edge of the pad.
11. Add 15 mm fillets to the sharp corners of the curved cutout on the pad.



In-Context Part Design 1

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 (mm³)?

