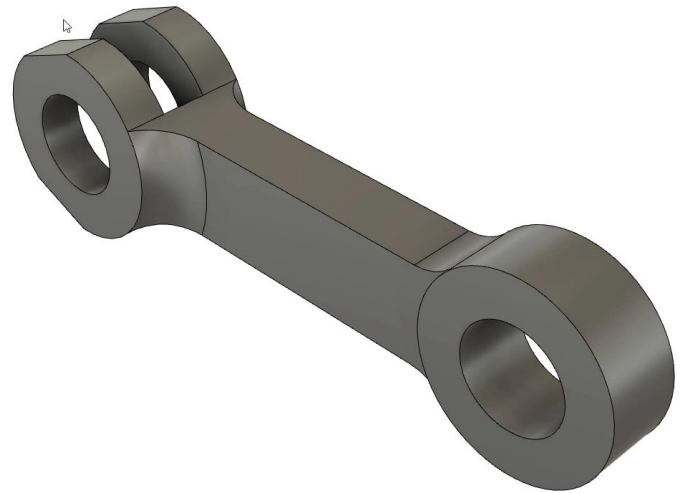


## Create a 3D mechanical link

In this module, you'll re-create an imported model so that it has features that can be modified.

### Learning objectives:

- Disable capture design history.
- Create a fully defined sketch.
- Use Extrude and Fillet.



The completed exercise

1. Upload and open the supplied *defeature link.f3d* file.

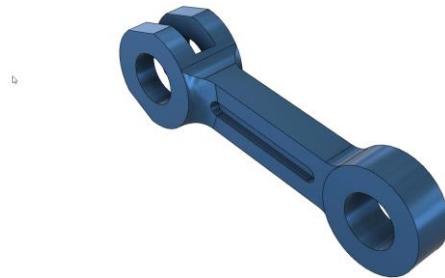


Figure 1. Open the supplied file

2. Notice that the timeline only has two features because this model was imported from a different CAD program.



Figure 2. Inspect the timeline

3. Click the gear icon in the window's bottom right corner and notice that you have the ability to toggle whether the design history is captured. Click Do not capture Design History.

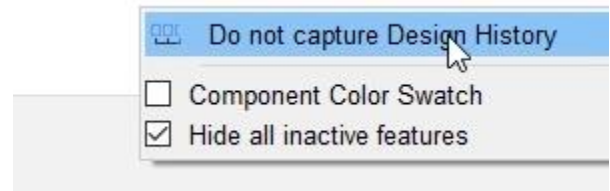


Figure 3. Note how to toggle the design history

4. Click Continue in the warning dialog. Notice the timeline is removed from the bottom of the window.

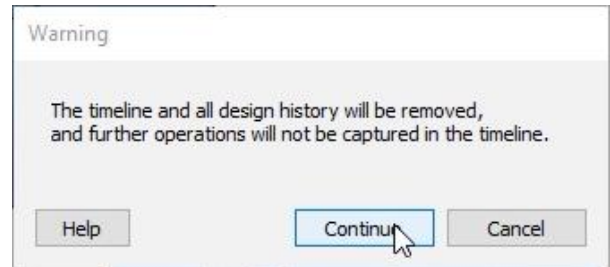


Figure 4. Click Continue

5. Expand the Create group's drop-down menu and notice that there are two options that were not available before: Find Features and Fluid Volume. Click Create > Find Features.



Figure 5. Open the Find Features tool

6. For the dialog's Faces/Body/Components selection, choose Body1 inside the Browser.

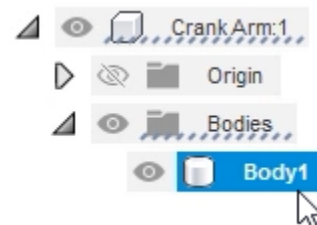


Figure 6. Choose Body1

7. Make sure all the options are activated inside the Find Features dialog, then OK the dialog.

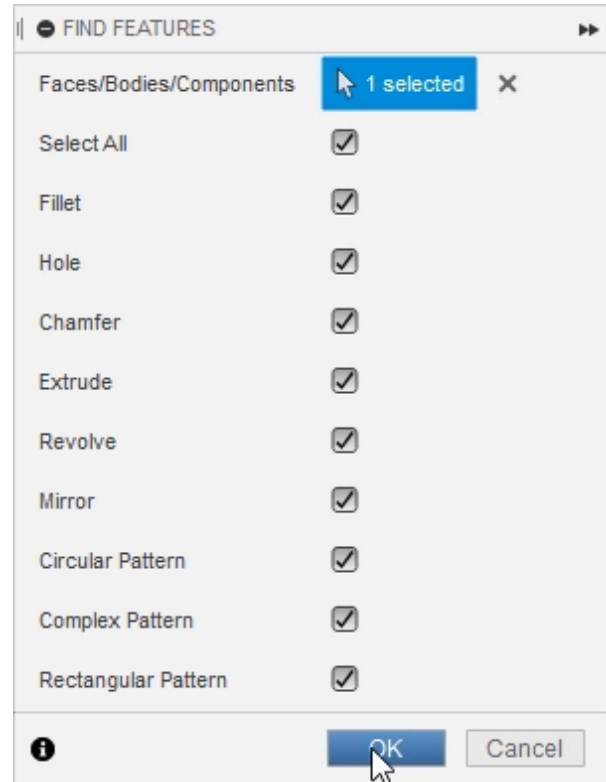


Figure 7. OK the dialog

8. Notice that a history of various features is added to the Browser.

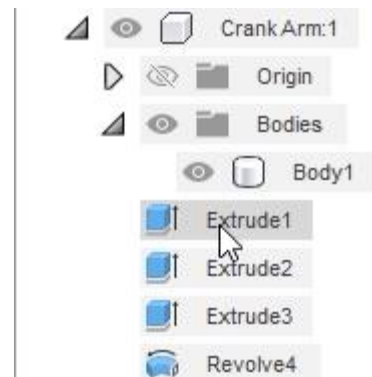


Figure 8. Inspect the Browser

9. Notice that some of the features added to the Browser can be edited the way you would edit a normal feature. However, most of the features cannot be edited.

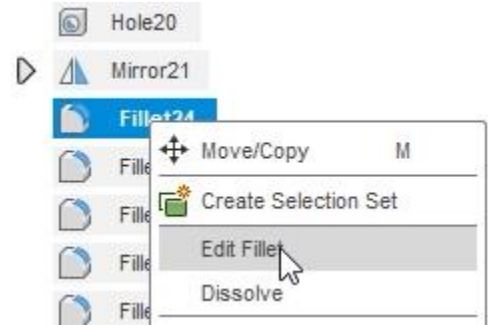


Figure 9. Note that some features can be edited

10. You can manually defeature the part to make it easier to re-create; select the four faces that create the pocket shown in the image on the right, then press Delete. The pocket is deleted and the geometry is patched. Repeat this process for the link's other side.

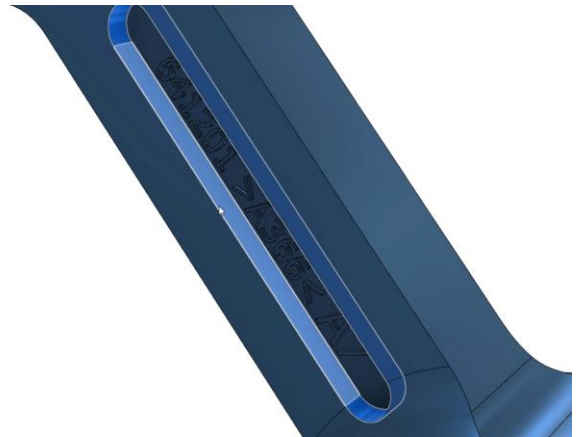
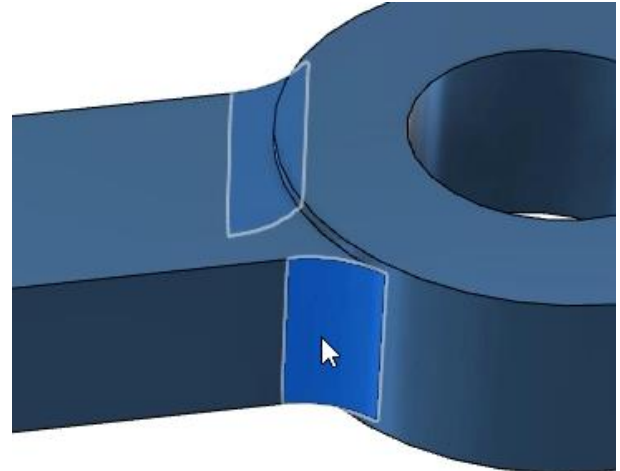


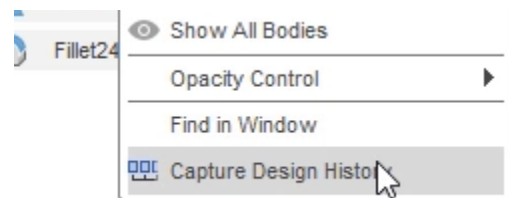
Figure 10. Delete faces

**11.** Fillets can also be deleted. Select the two fillets shown in the image on the right and press Delete.



*Figure 11. Delete the fillets*

**12.** Right-click the Browser's top level and choose Capture Design History.



*Figure 12. Activate the Capture Design History option*

**13.** Notice the timeline has only one group inside of it. Click the plus icon to expand the group and notice that there are only two features: a component and a base feature.



*Figure 13. Inspect the timeline's group*

14. To begin re-creating the link, a new component needs to be created. Click Assemble> New Component.



Figure 14. Create a new component

15. Name the new component **Link**, then OK the New Component dialog.

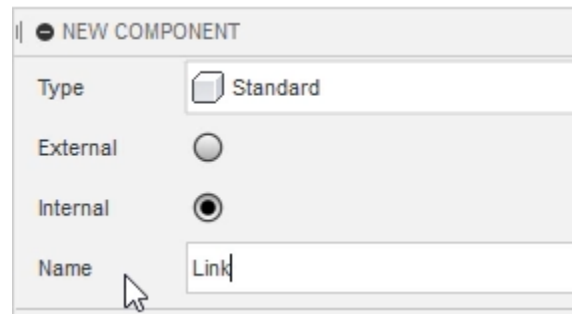


Figure 15. Name the new component

16. Expand the Browser's new Link component and turn on the visibility for the Origin folder. Notice that the origin is not located on the model. Use the Browser to hide the origin.

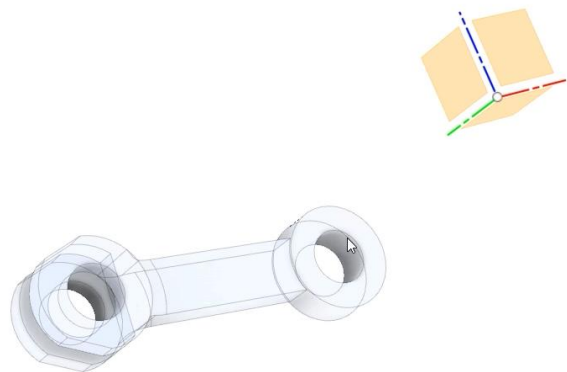
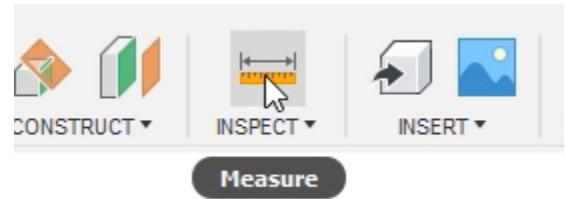


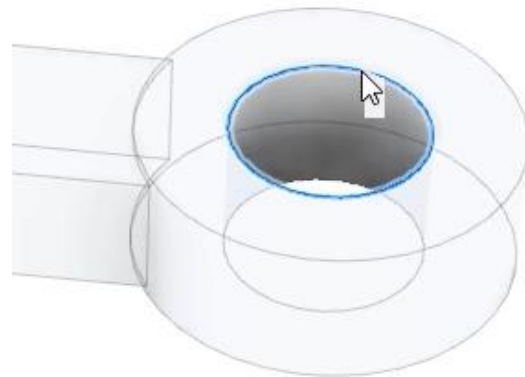
Figure 16. Note the origin's location

**17.** Open the Measure tool by clicking Inspect> Measure.



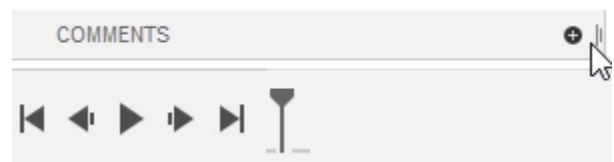
*Figure 17. Open the Measure tool*

**18.** Measure the model's various edges and note the measurements.



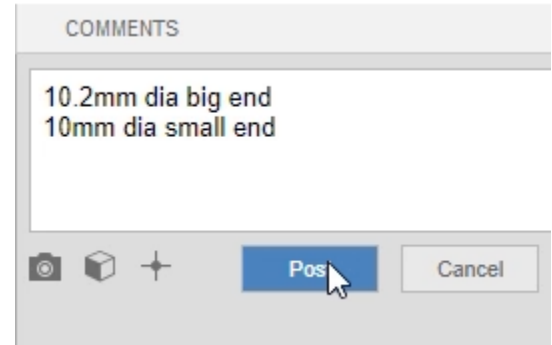
*Figure 18. Measure the model*

**19.** Click the plus icon to expand the Comments dialog.



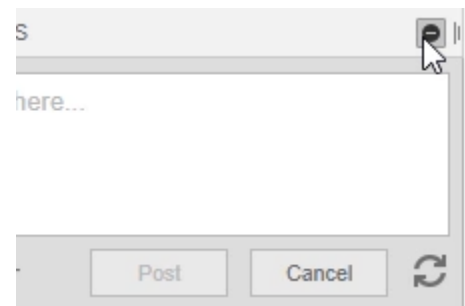
*Figure 19. Expand the Comments dialog*

**20.** Type the measurements you took into the dialog, then click the dialog's Post. This comment is now saved with the design and can be accessed at any time.



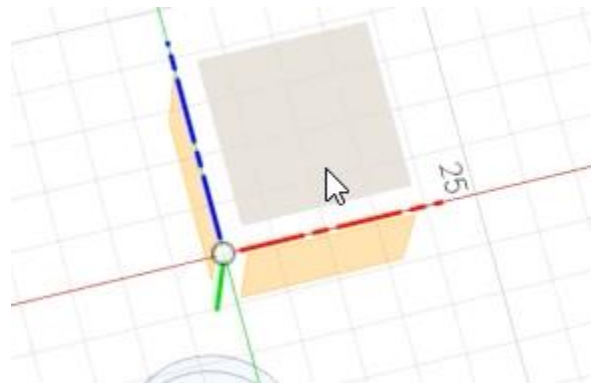
*Figure 20. Create a comment*

**21.** Continue to measure the model and make comments to capture the measurements. Minimize the Comments dialog by clicking the minus icon. Also press Esc to leave the Measure tool.



*Figure 21. Minimize the dialog*

**22.** Create a new sketch on the XZ plane.



*Figure 22. Create a new sketch*



- 23.** Press C to open the Circle tool and draw two circles concentric with the sketch's origin.

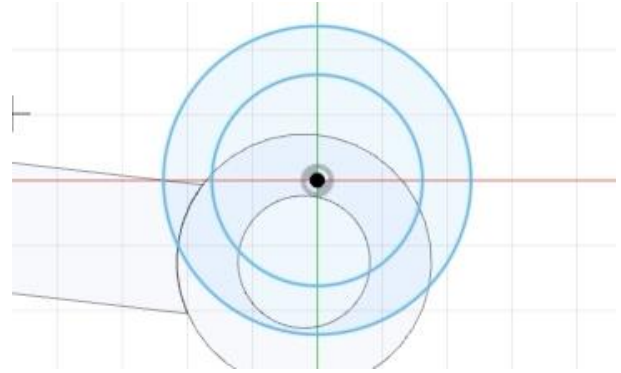


Figure 23. Draw two circles

- 24.** Draw two additional circles to the left of the origin and make sure they do not snap to any of the model's geometry.

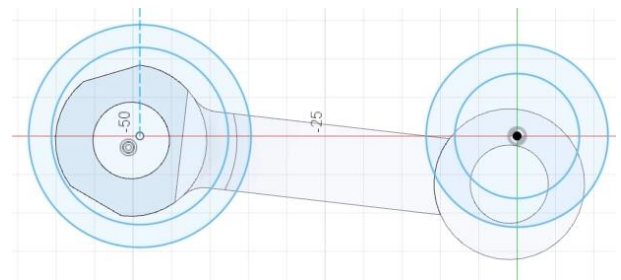


Figure 24. Draw two circles

- 25.** Select the centers of the two groups of circles and click Constraints > Horizontal/Vertical to add a Horizontal constraint between them.

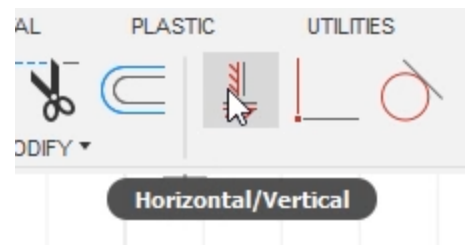


Figure 25. Constrain the circles

**26.** Press D to open the Dimension tool and dimension the first group of circles you drew.

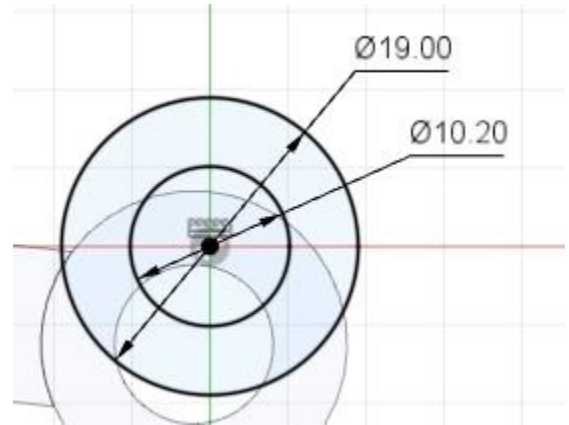


Figure 26. Dimension the circles

**27.** Continue to add dimensions to the geometry until the sketch is fully defined.

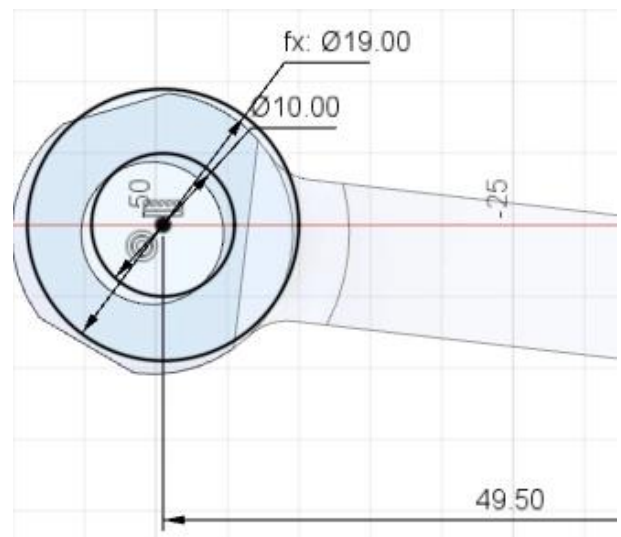
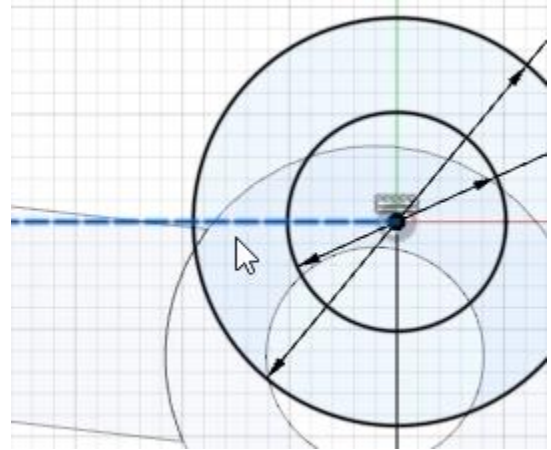


Figure 27. Fully define the sketch

**28.** Press L to open the Line tool and connect the center of the two circles with a line. Select the line and press X to convert it to construction geometry.



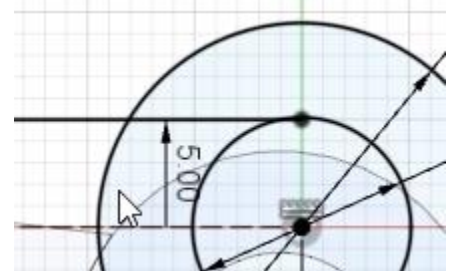
*Figure 28. Create a construction line connecting the two circles*

**29.** Click Modify > Offset.



*Figure 29. Open the Offset tool*

**30.** Select the construction line and offset it 5 mm.



*Figure 30. Offset the construction line*

**31.** Click Create> Mirror.

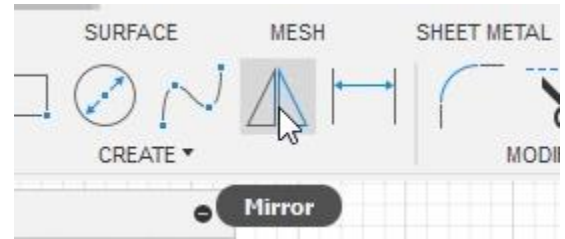


Figure 31. Open the Mirror tool

**32.** For the dialog's Objects selection, choose the offset line. For the Mirror Line selection, choose the construction line. OK the dialog to mirror the line.

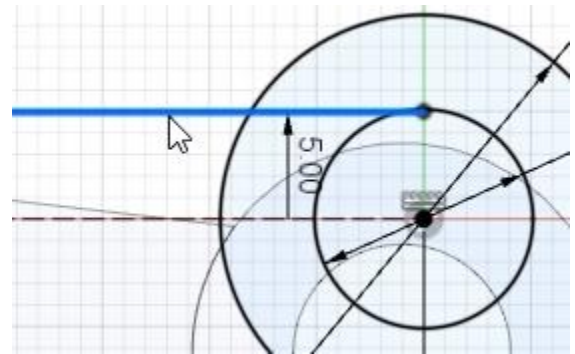


Figure 32. Mirror the offset line

**33.** Next, the flats need to be created on the one end of the link. Open the Line tool and draw a line that is coincident with the circle at both ends to create one of the flats. Press D to open the Dimension tool and add a dimension of 22.5° between the new line and the construction line.

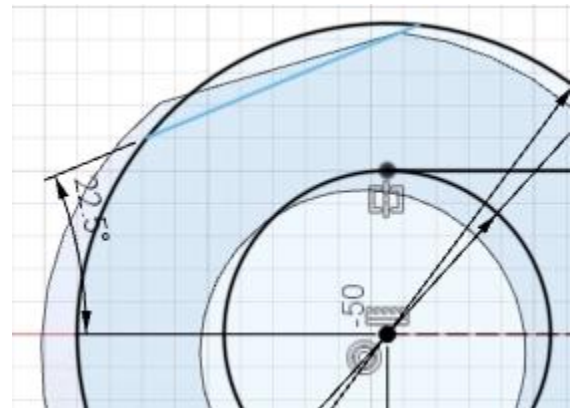


Figure 33. Draw a line

- 34.** Click Constraints> Horizontal/Vertical and add a Vertical constraint between one end of the line and the center of the circle. The line should now be fully defined.

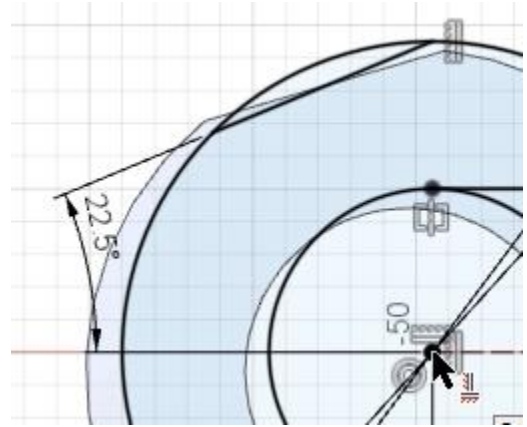


Figure 34. Constrain the line

- 35.** Click Create> Mirror, select the new line you drew as the dialog's Objects selection, then choose the construction line as the dialog's Mirror Lines selection. OK the dialog to mirror the line.

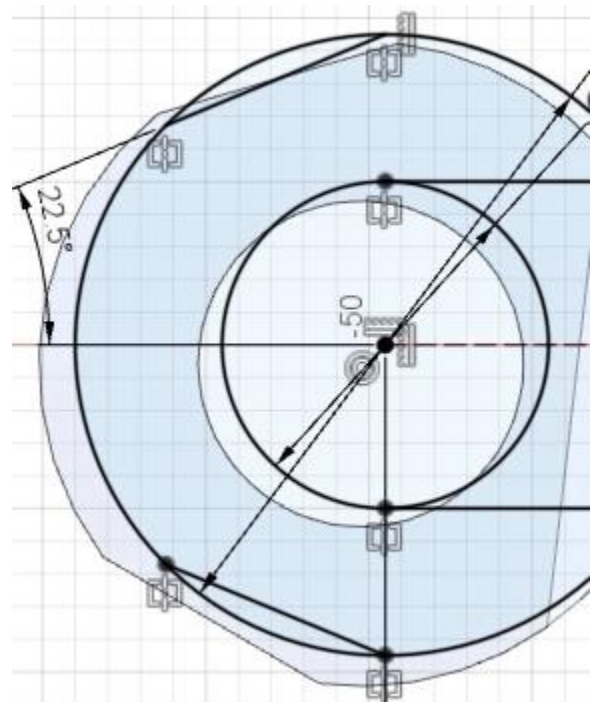


Figure 35. Mirror the new line

- 36.** Open the Circle tool and add a 13 mm circle concentric with the circles that you added the flats to. Select the new circle and press X to convert it to construction geometry.

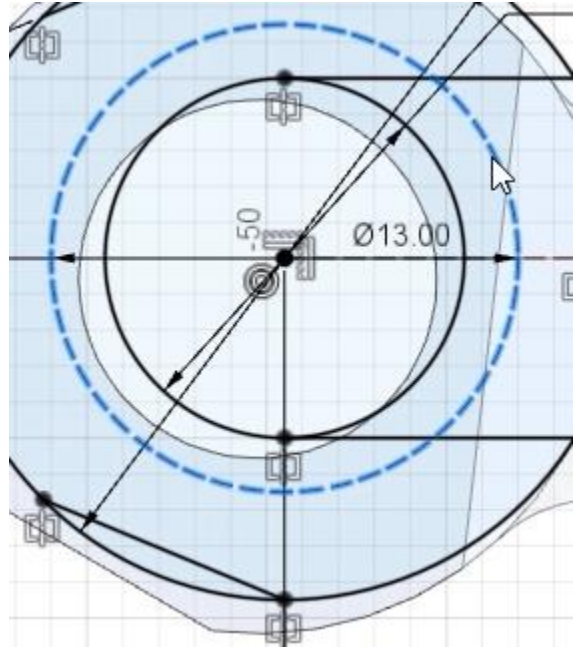


Figure 36. Draw a new circle

- 37.** Open the Line tool and draw a vertical line to match the image on the right. Click Constraints> Tangent and add a tangent constraint between the construction circle and the new line.

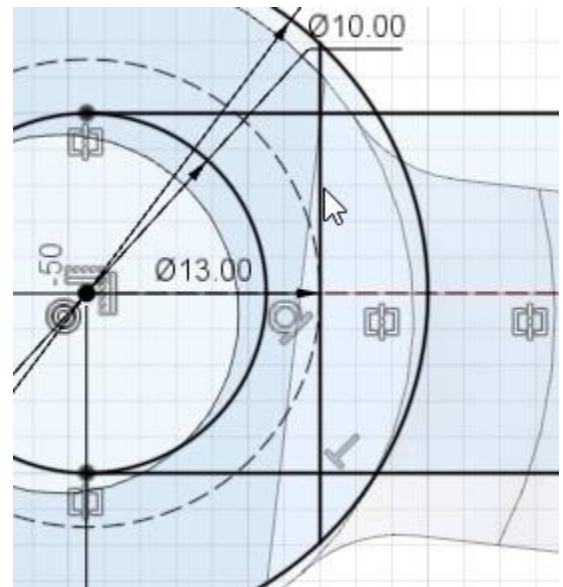


Figure 37. Draw a line

**38.** Finish the sketch by clicking Finish Sketch> Finish Sketch.



Figure 38. Finish the sketch

**39.** Press E to open the Extrude tool and select the two regions shown in the image on the right as the dialog's Profiles selection.

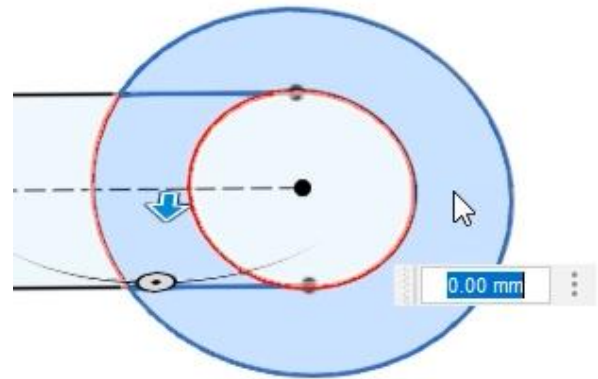


Figure 39. Select the region to extrude

**40.** Choose the Symmetric option from the dialog's Direction menu, activate the Whole Length option in the Measurement section, then enter **7.45mm** into the Distance box. OK the dialog to extrude the geometry.

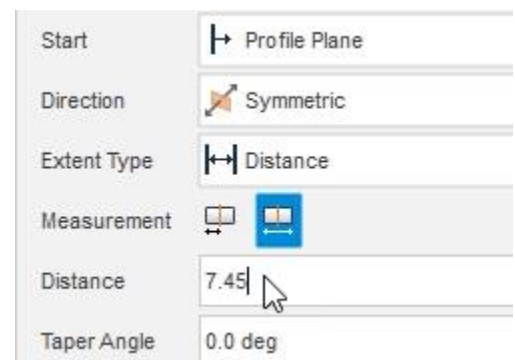


Figure 40. Extrude the geometry

41. Use the Browser to turn on the visibility for Sketch1.

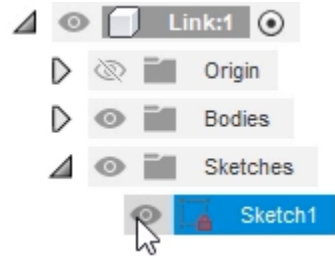


Figure 41. Show Sketch1

42. Open the Extrude tool and select the five regions shown in the image on the right.

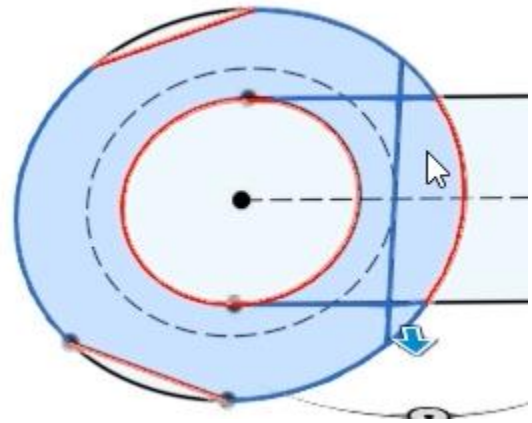


Figure 42. Open the Extrude tool and select the regions to extrude

43. Use the image on the right as a guide to extrude the new geometry. Instead of being 7.45 mm tall, the new section is **13 mm** tall.

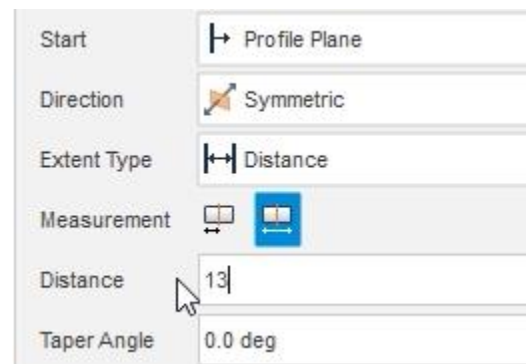


Figure 43. Extrude the new geometry



44. Open the Extrude tool and select the region shown in the image on the right.

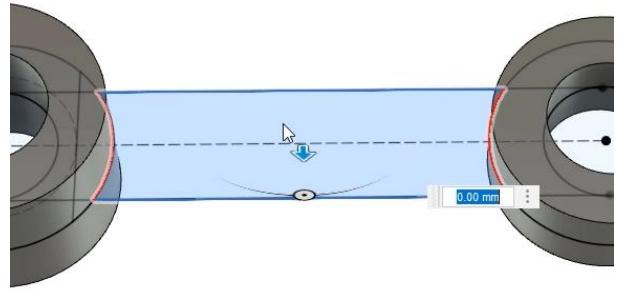


Figure 44. Select the region to extrude

45. Use similar settings to the previous two extrudes, but enter **6.5 mm** into the Distance box. OK the dialog.

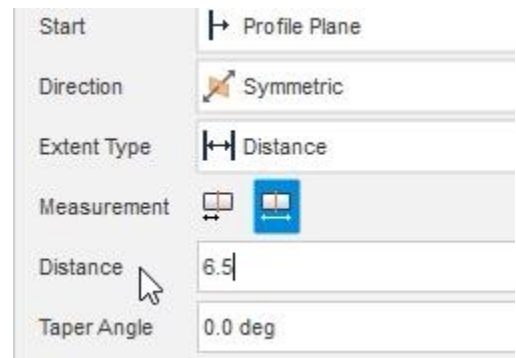


Figure 45. Create the extrude

46. Open the Extrude tool and select the two regions shown in the image on the right.

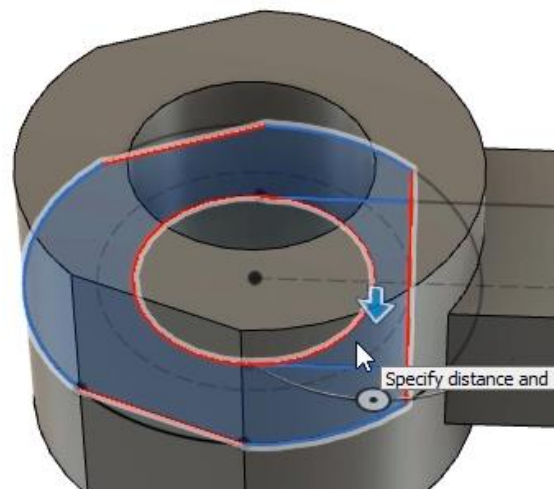


Figure 46. Open the Extrude tool

**47.** Configure the new extrude similar to the previous extrudes except enter **4.65 mm** into the Distance box. Make sure the Cut option is selected in the dialog's Operation menu, then OK the dialog.

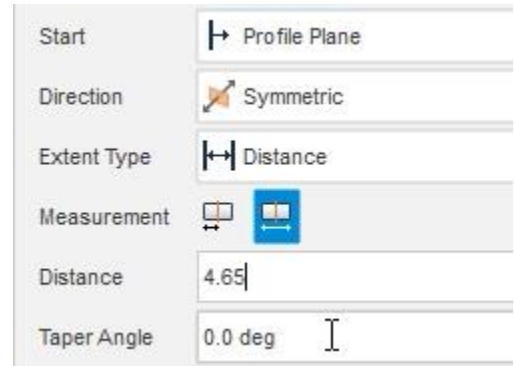


Figure 47. Create the extrude cut

**48.** Use the Browser to hide the Crank Arm component and Sketch1.

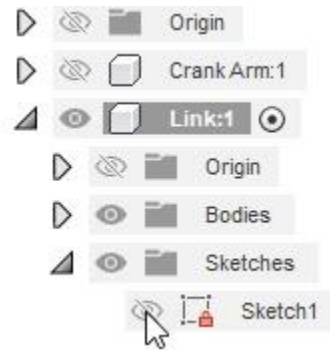


Figure 48. Hide the component and sketch

**49.** Click Modify > Fillet.

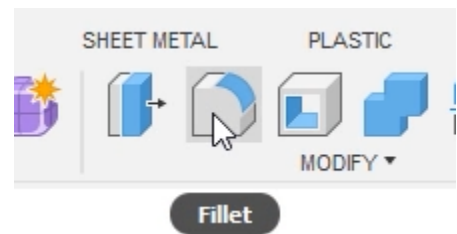


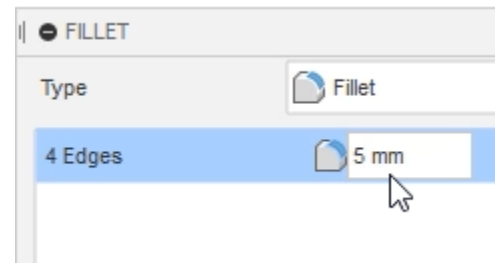
Figure 49. Open the Fillet tool

**50.** Select the four edges in the image on the right to add a fillet to them.



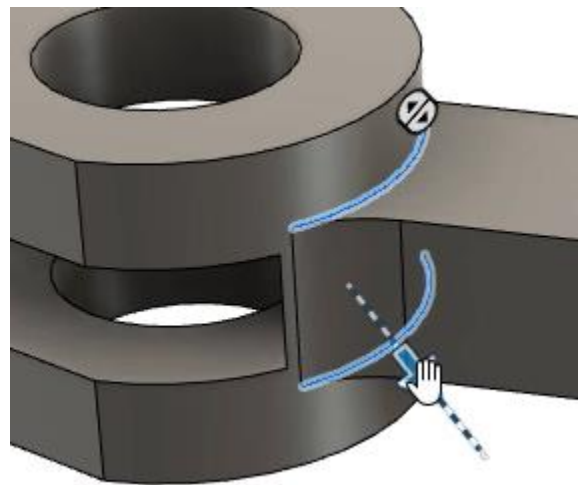
*Figure 50. Select the edges*

**51.** Enter **5 mm** into the dialog's box to create a 5 mm fillet on the edges you selected. OK the dialog.



*Figure 51. Create the fillet*

**52.** Open the Fillet tool again, then select the two edges shown in the image on the right. Use the on-screen manipulator to add a 5.5 mm fillet to the selected edges, then OK the dialog.



*Figure 52. Add a 5.5 mm fillet*

**53.** Inspect the timeline and notice that the original component has been re-created with features that can be easily modified if the geometry needs to change.

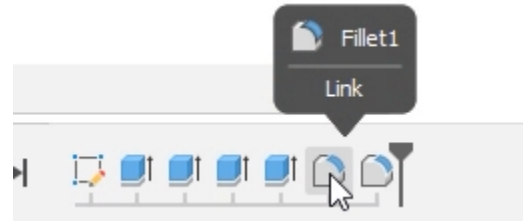


Figure 53. Inspect the timeline

**54.** Click File > Save As.



Figure 54. Save the file

**55.** Name the file **parametric link** and click the Save dialog's Save. Continue to the next module.

## Save As

Name:

parametric link

Figure 55. Save the file