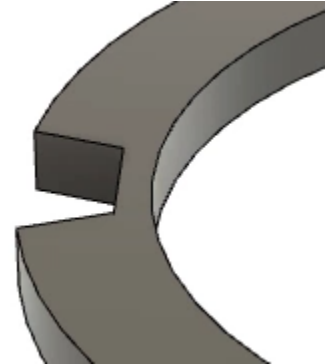


## Create and edit a sketch

In this module, you'll create sketches and extrude solid geometry.

### Learning objectives:

- Create a sketch.
- Apply a sketch dimension.
- Apply a sketch constraint.
- Use Center Point Rectangle.
- Use Trim.



The completed exercise

1. Open a new untitled document in Fusion 360. In the Browser, expand the Document Settings and note the file's units. If the units are set to millimeters, click Change Active Units.

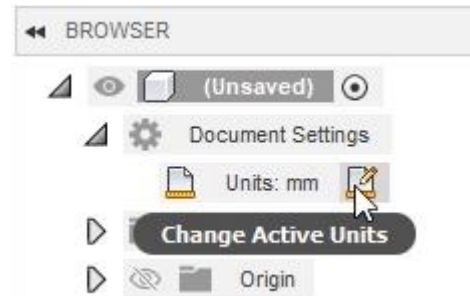


Figure 1. Change a file's units

2. Choose Inch from the dialog's Unit Type menu, then OK the dialog.

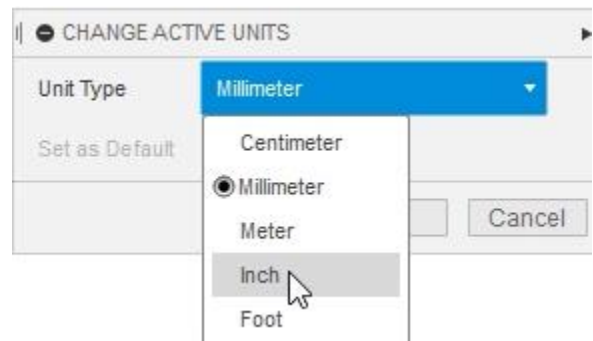


Figure 2. Choose the Inch units

3. In the Toolbar's Create group, click Create> Create Sketch.



Figure 3. Create a new sketch

4. Choose to sketch on the XY plane and notice that the camera automatically snaps to a view normal to the selected plane.

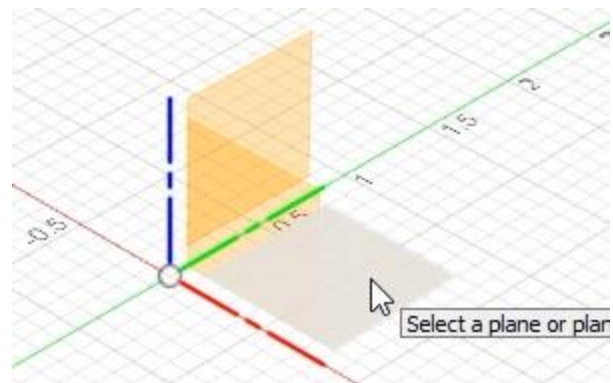


Figure 4. Click the XY plane

5. Click Create> Circle> Center Diameter Circle. Also notice that there is a C next to this choice in the menu. This is the keyboard shortcut for this tool.

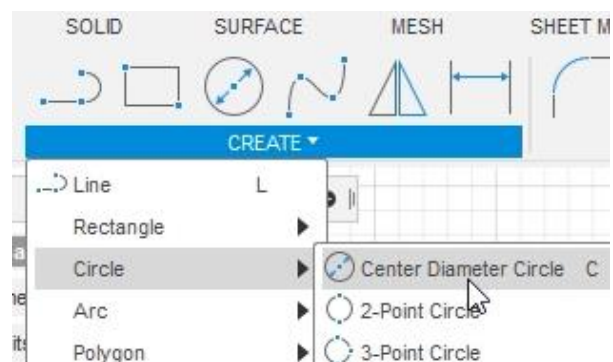


Figure 5. Open the Circle tool

6. Notice that you can change the type of circle using the options inside the dialog's Circle section. Choose the Center Diameter Circle option.

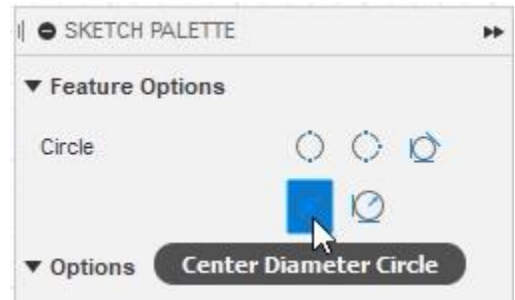


Figure 6. Explore the options in the Circle section

7. Click the sketch's origin to place the circle's center point.

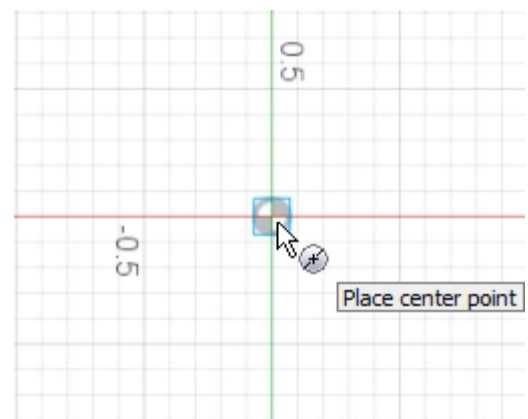


Figure 7. Place the circle's center point

- Press **5** to enter this value into the circle's dimension box, then press Enter. You have now drawn a 5-inch circle concentric with the sketch's origin. Notice that the circle's geometry is black, indicating that it is fully defined. Press Esc to leave the Circle tool.

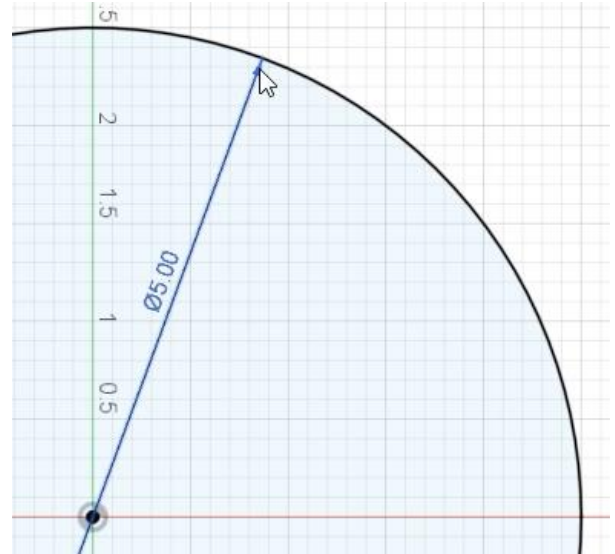


Figure 8. Draw the circle

- Press **C** to open the Circle tool again. Place the new circle's center at the sketch's origin, then click a second time to create a circle smaller than the first circle you drew. The new circle is blue because it is underdefined. The center is fixed at the sketch's origin but the diameter can change.

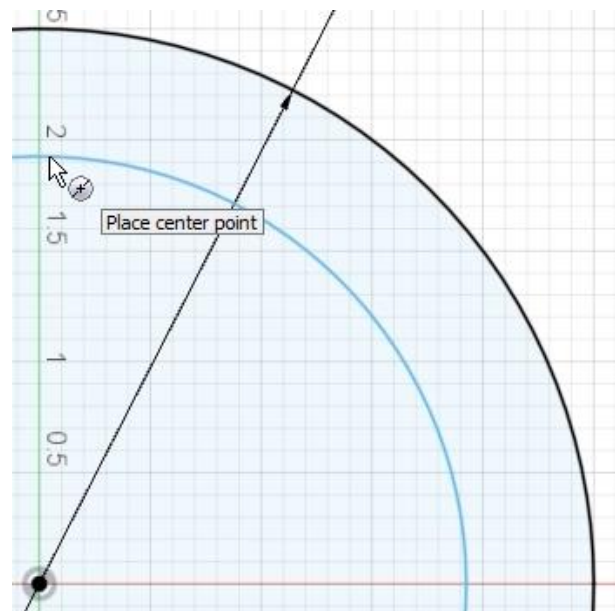


Figure 9. Create a new circle

10. Click Create> Sketch Dimension.

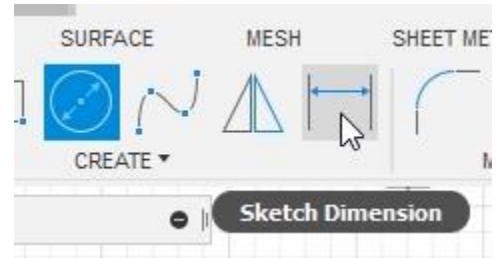


Figure 10. Open the Sketch Dimension tool

11. Select both circles you drew to create a dimension between them. Click to place the dimension in the Canvas area.

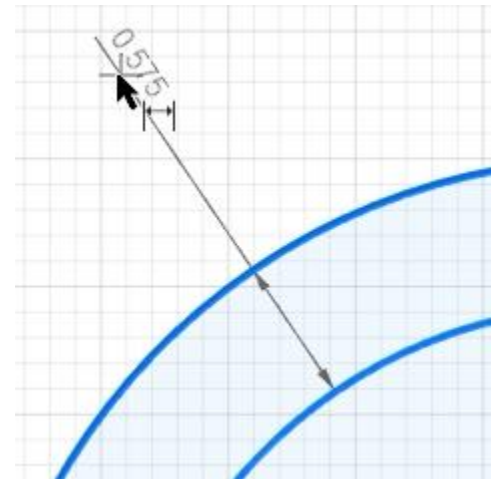


Figure 11. Select both circles

12. Type **0.5 inches** into the dimension box to make sure the two circles are 0.5 inches apart. Notice that the new circle turns black to indicate that it is now fully defined. Press Esc to leave the Circle tool.

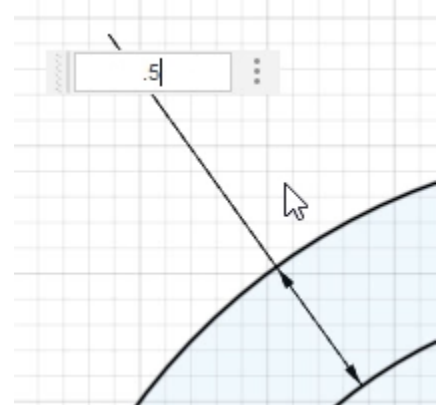


Figure 12. Create the dimension

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



Figure 13. Finish the sketch

**14.** Notice that the sketch has two selectable regions that can be selected independently.

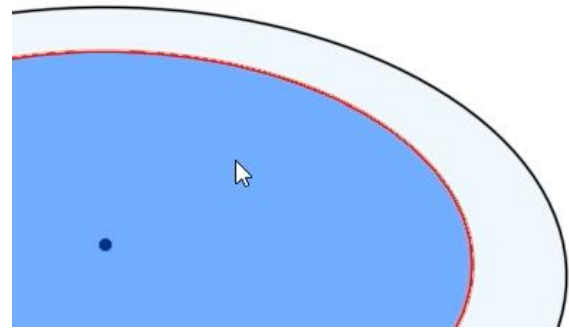


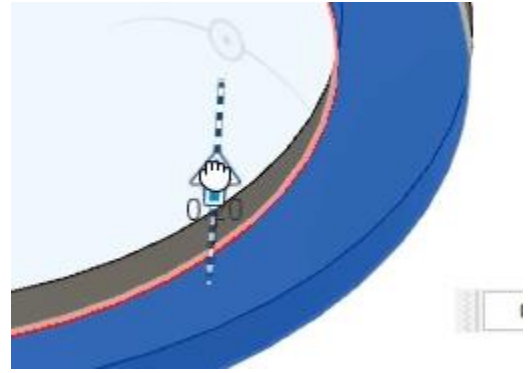
Figure 14. Notice the sketch regions

**15.** Click Create> Extrude.



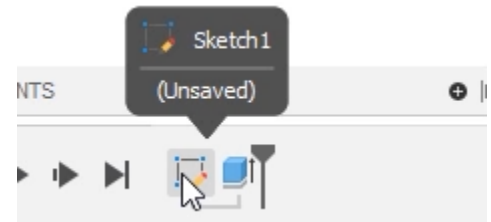
Figure 15. Open the Extrude tool

- 16.** Select the outside region and drag the on-screen manipulator up to extrude the selected region into a solid body. Choose to extrude the selection 0.25 inches, then OK the dialog.



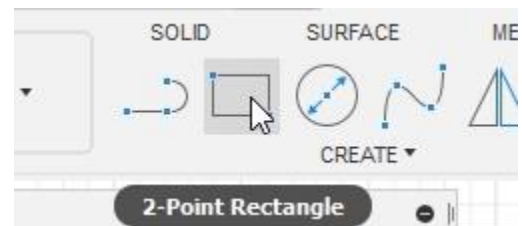
*Figure 16. Extrude the selection into a solid body*

- 17.** Notice that the sketch and extrude features are added to the timeline at the bottom of the window. Double-click the sketch icon to edit the sketch.



*Figure 17. Edit the timeline's sketch feature*

- 18.** Click Create > 2-Point Rectangle.



*Figure 18. Open the Rectangle tool*

19. Choose the Center Rectangle option from the dialog's Rectangle section.

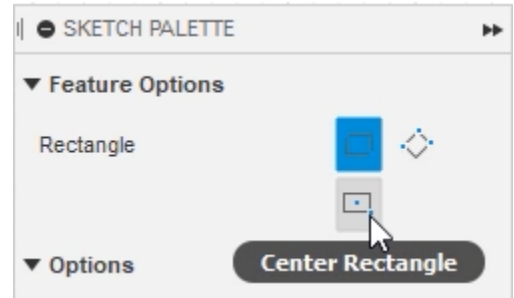


Figure 19. Choose the Center Rectangle option

20. Click to place the rectangle's center to the outside circle, then click a second time to define the rectangle's shape.

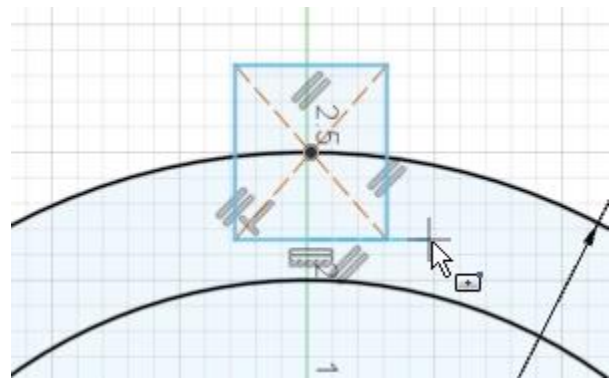


Figure 20. Draw a rectangle

21. Click Constraints > Equal to open the Equal constraint tool.

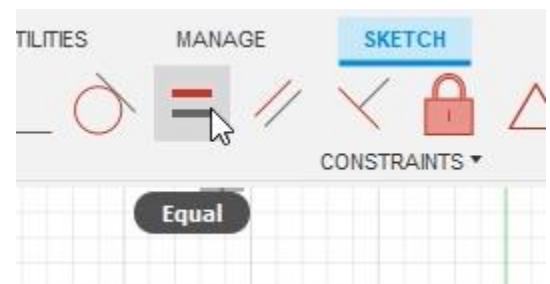


Figure 21. Open the Equal constraint tool



- 22.** Select two of the new rectangle's sides to add an equal constraint between them. If the length of one of the sides changes, the second side will automatically update to match it.

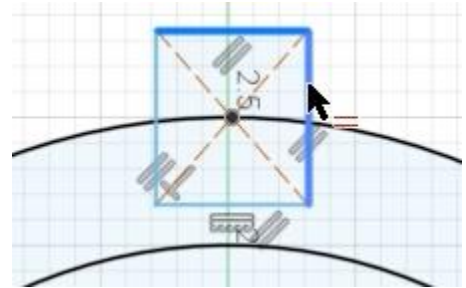


Figure 22. Add the constraint to the rectangle

- 23.** Click Constraints > Horizontal/Vertical.

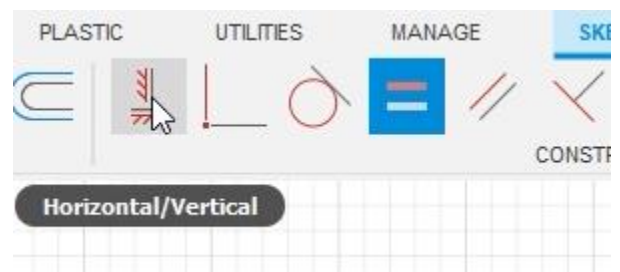


Figure 23. Open the Horizontal/Vertical tool

- 24.** Select the rectangle's center point and the sketch's origin to create a vertical constraint between them.

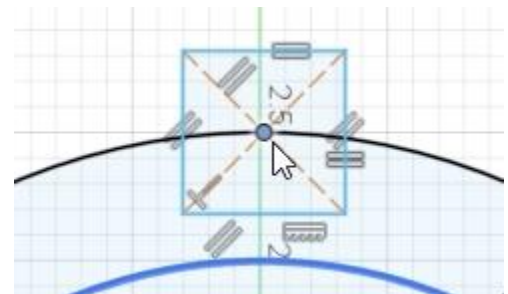


Figure 24. Add a vertical constraint between two points

**25.** Press D to open the Sketch Dimension tool. Select the rectangle's top edge and add a **0.5-inch** dimension to it. Notice that the rectangle's height automatically changes to match the rectangle's width.

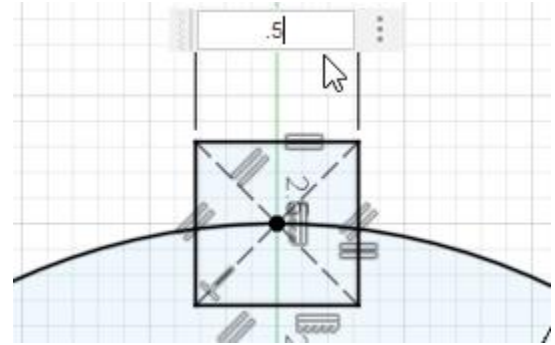


Figure 25. Dimension the rectangle's top edge

**26.** Finish the sketch.



Figure 26. Finish the sketch

**27.** Fusion 360 automatically hides sketches after they are used to create the extrudes. Expand the Browser's Sketches folder and click the eyeball icon to make Sketch1 visible.

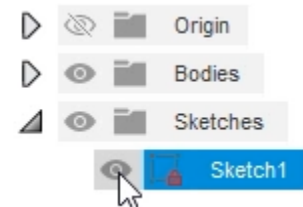


Figure 27. Show Sketch1

- 28.** Press E to open the Extrude tool. Notice that it is difficult to select the rectangle's region inside the extrude. Hover your cursor over the intended selection, hold the left mouse button, then choose Profile from the menu. The rectangle's region inside the extrude is now selected.

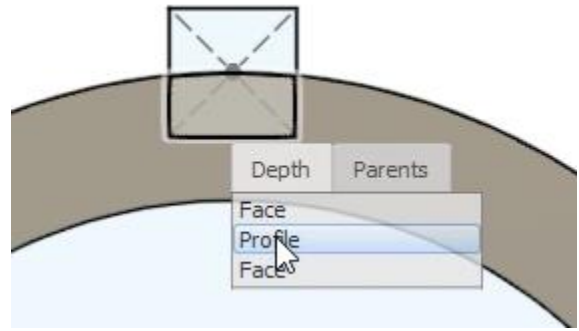


Figure 28. Select the rectangle's region

- 29.** Hold Ctrl and select the rectangle's second region.

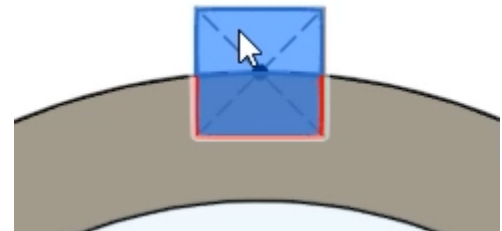


Figure 29. Select the rectangle's second region

- 30.** Drag the on-screen manipulator's arrow to extrude the selections through the solid body. Notice that the geometry is automatically selected as an extrude cut which removes material from the solid body.

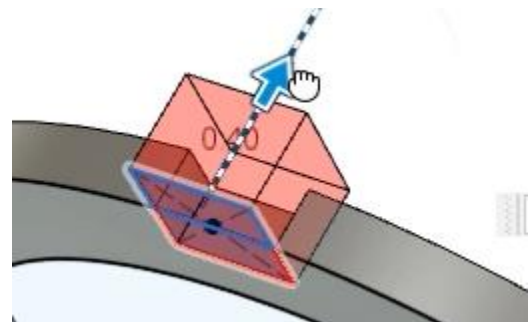


Figure 30. Extrude the selection

- 31.** Drag the selection down and notice that the extrude type is changed from cut to join.

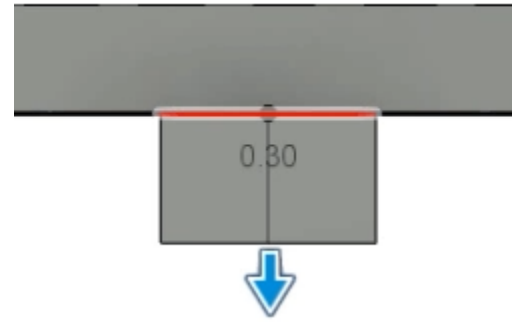


Figure 31. Begin extruding the selection

- 32.** Drag the selection up so that the extrude type is removing material from the solid body, then explore the various options inside the Extrude dialog's Operation menu. These options allow you to manually override the default Operation type. Choose the Cut option, then OK the dialog.

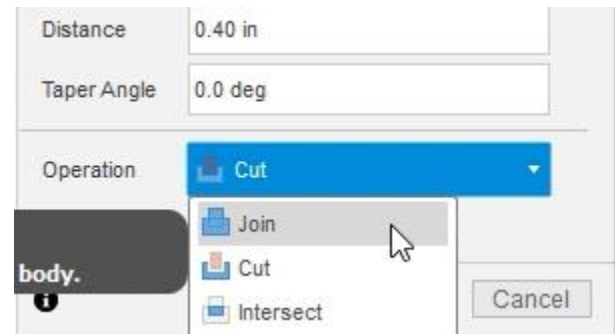


Figure 32. Choose a new operation type

- 33.** Notice the material is removed from the solid body.

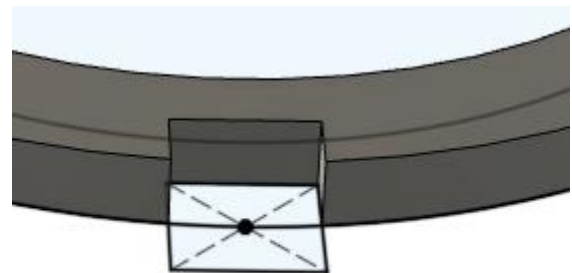


Figure 33. Inspect the result

**34.** Hide the sketch by clicking Sketch1's eyeball icon in the Browser.

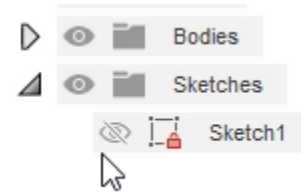


Figure 34. Hide the sketch

**35.** Edit Sketch1 by right-clicking it and choosing Edit Sketch from the menu.

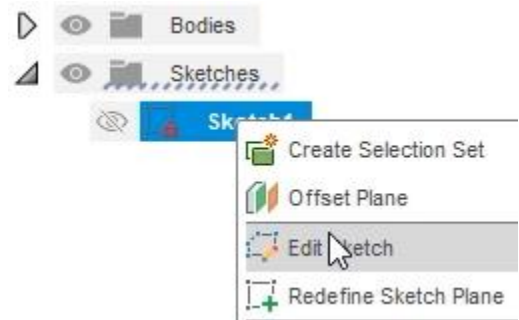


Figure 35. Edit Sketch1

**36.** Click Create> Line.

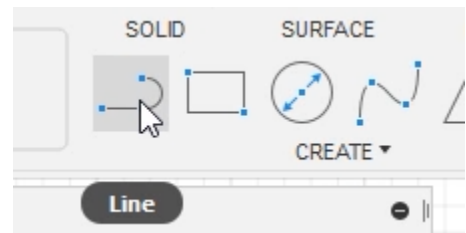


Figure 36. Open the Line tool

**37.** Draw three lines similar to the image on the right, then click the green checkmark to end the lines. Press Esc to leave the Line tool.

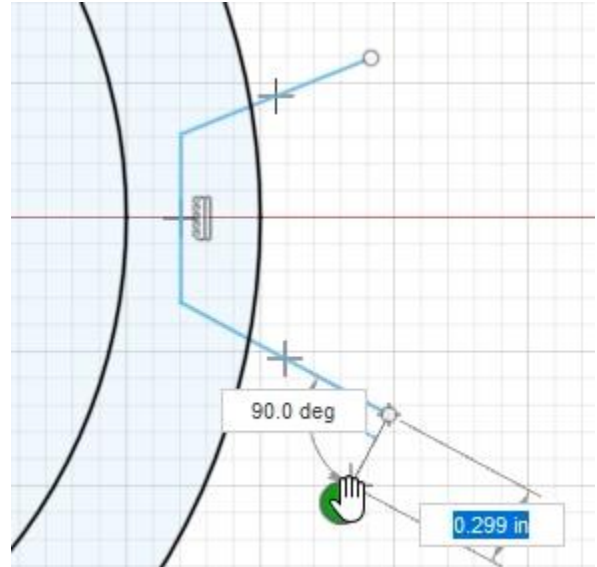


Figure 37. Draw three lines

**38.** Drag one of the lines' endpoints and snap it to the circle.

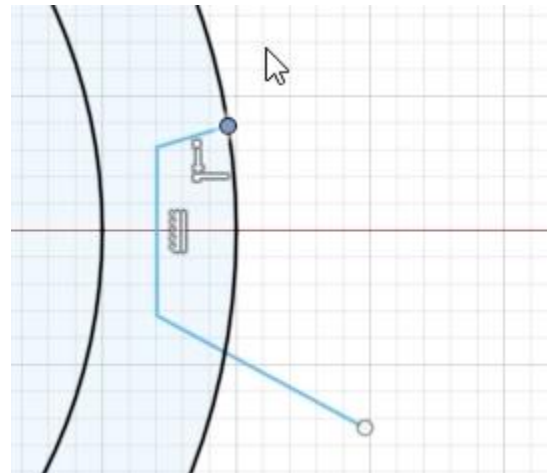


Figure 38. Adjust one of the lines.

**39.** Other tools can be used to adjust the lines you just drew; click Modify> Trim.



Figure 39. Open the Trim tool

**40.** Select the line's section that sticks out of the circle. The Trim tool trims the line down to the circle.

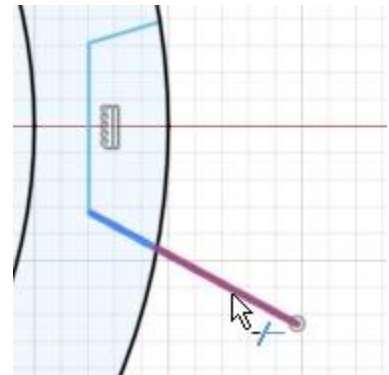


Figure 40. Trim the line

**41.** Click the circle to trim the section shown in the image on right. Press Ctrl + Z to undo this trim and notice that the circle becomes whole again.

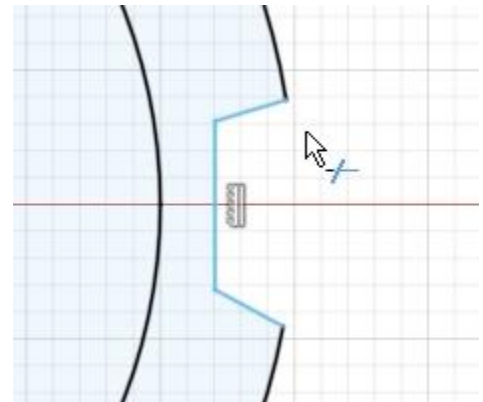


Figure 41. Trim the circle

- 42.** Press L to open the Line tool and connect the sketch's origin to the vertical line you drew. Make sure the triangle icon appears as you connect the new line to the vertical line. This indicates that you're snapping the line's endpoint to the vertical line's midpoint. Press Esc to leave the Line tool.

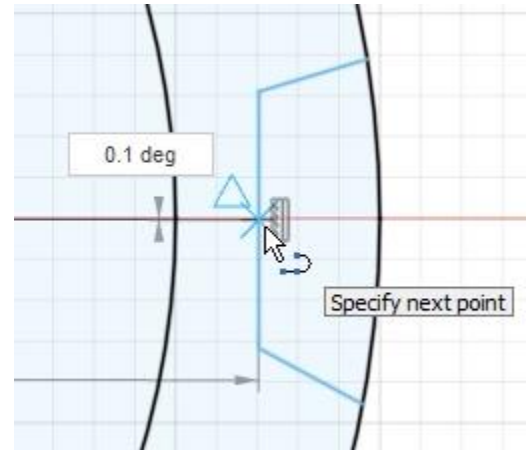


Figure 42. Draw a new line

- 43.** Select the line you just drew and click Constraints> Horizontal/Vertical. A horizontal constraint is applied to the new line. Press Esc to leave the constraint tool. Select the new line and change it to construction geometry by selecting the Construction option in the dialog's Line Type section.

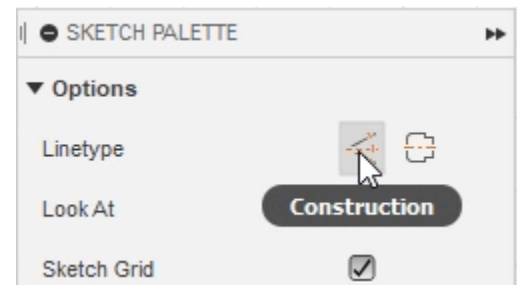


Figure 43. Convert the new line to construction geometry



- 44.** The line now has a dashed appearance to indicate it is construction geometry. Construction lines are not used to create sketch regions.



Figure 44. Inspect the construction geometry

- 45.** Press D to open the Sketch Dimension tool. Use the image on the right as a guide when adding dimensions to the sketch. To create the angle dimension, select the two lines that need to have the angle dimension between them.

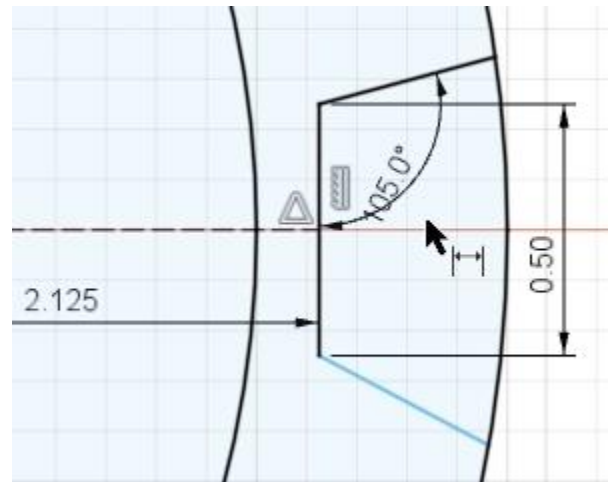


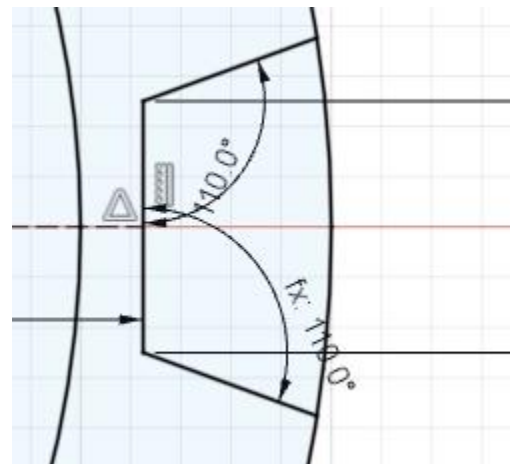
Figure 45. Dimension the sketch

- 46.** An existing dimension can be used as a reference for a different dimension. Create a new angle dimension for the geometry shown in the image on the right, then click the existing 105° dimension so that its value is automatically imported into the new dimension's box. Press Enter.



*Figure 46. Use an existing dimension as a reference*

- 47.** Modify the original dimension to 110° and notice that the second dimension automatically updates to match. Finish the sketch by clicking Finish Sketch > Finish Sketch.



*Figure 47. Update a dimension*

**48.** Turn on the visibility for Sketch1 by clicking the Browser's eyeball icon, then open the Extrude tool by pressing E. Select the sketch region you just created. Since the region is difficult to select, hover your cursor over the intended selection, hold the left mouse button, then choose the appropriate selection from the menu.

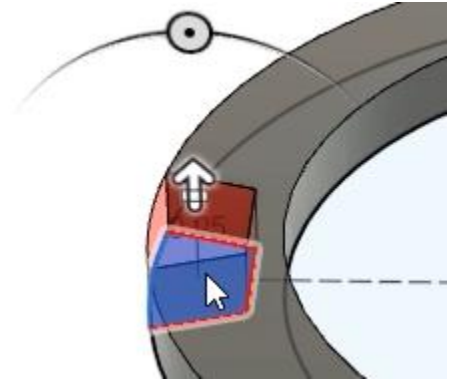


Figure 48. Choose the sketch region

**49.** Choose the All option from the dialog's Extent Type menu, then OK the dialog.

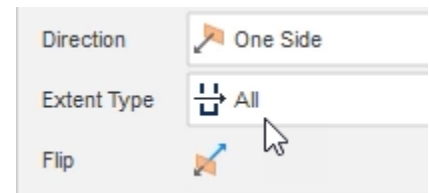


Figure 49. Choose the extrude's extent type

**50.** Hide the sketch and inspect the result. The sketch region you drew is removed from the solid body. Save the file and continue to the next module.

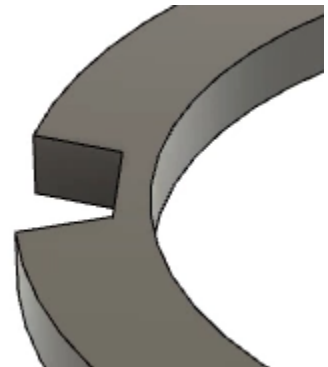


Figure 50. Inspect the result