

Foundational CAD concepts

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

Learning objectives:

- Understand basic sketch creation.
- Use Dimensions and Constraints to define a sketch.
- Create a 3D model using basic tools.



The completed exercise

1. Open a new untitled design in Fusion 360. Expand the Browser's Document Settings and click the Change Active Units.

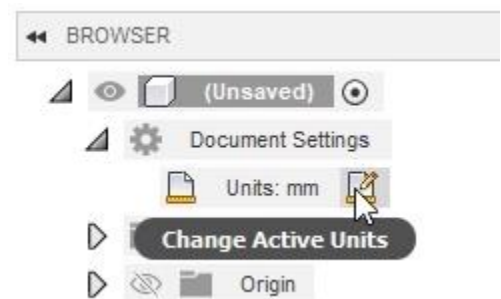


Figure 1. Change the active units

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

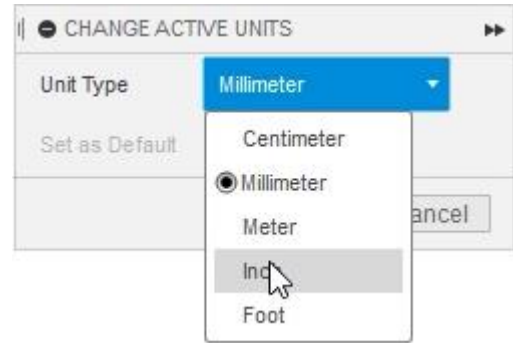


Figure 2. Change the active units

3. If most of your files need to be created using a different unit than the default unit, this can be configured in your user preferences. Click on your photo or initials in the screen's top right corner, then choose Preferences from the menu.

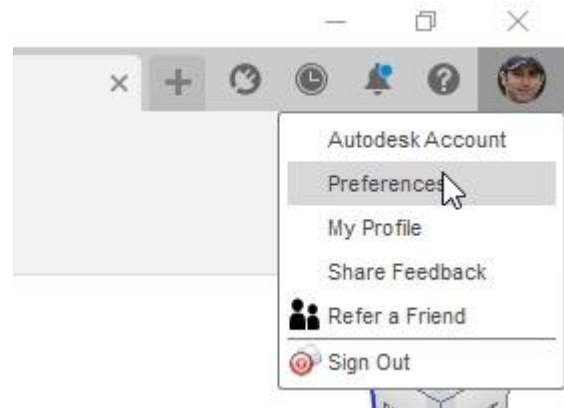


Figure 3. Open your user preferences

4. The options in the Default Units section allow you to customize your default units for the Design, Electronics, Manufacture, or Simulation workspaces. Make any changes you want before closing the Preferences Dialog.

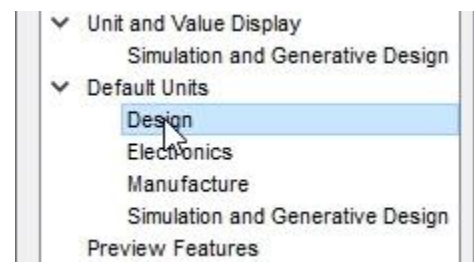


Figure 4. Open the Default Units preferences

5. Create a new sketch by clicking Create> Create Sketch.



Figure 5. Create a new sketch

6. Choose the plane that you want to sketch on. In the image on the right, the XZ plane is selected.

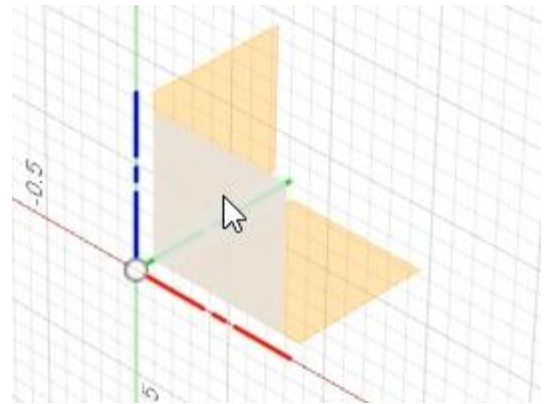


Figure 6. Choose a plane

7. Expand the Create group's menu and notice the many types of sketches you can create, then choose the Line tool. Notice that the keyboard shortcut for the Line tool is displayed next to it in the drop-down menu.

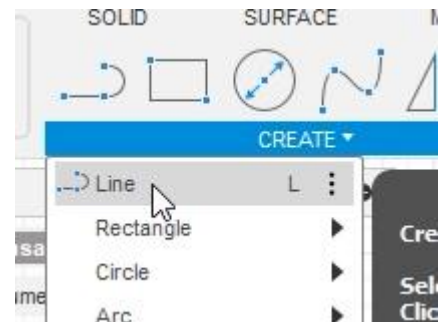


Figure 7. Open the Line tool

8. Hover your cursor near the sketch's origin and notice that it automatically snaps to the origin.

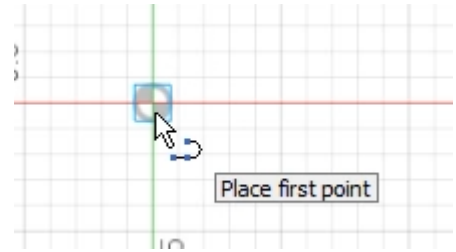


Figure 8. Notice the snap feature

9. If you want your cursor to snap to the smaller lines of the grid, click Display and Settings Bar > Grids and Snaps > Snap to Grid. Choose whichever option you prefer.

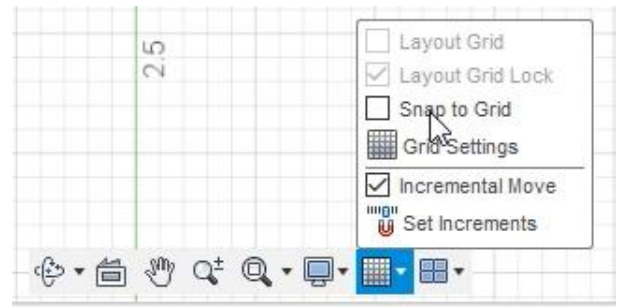


Figure 9. Toggle the Snap to Grid option

10. Click to place the line's first point at the sketch's origin. Move your cursor away from the origin and notice that the line's dimension and angle are automatically displayed.

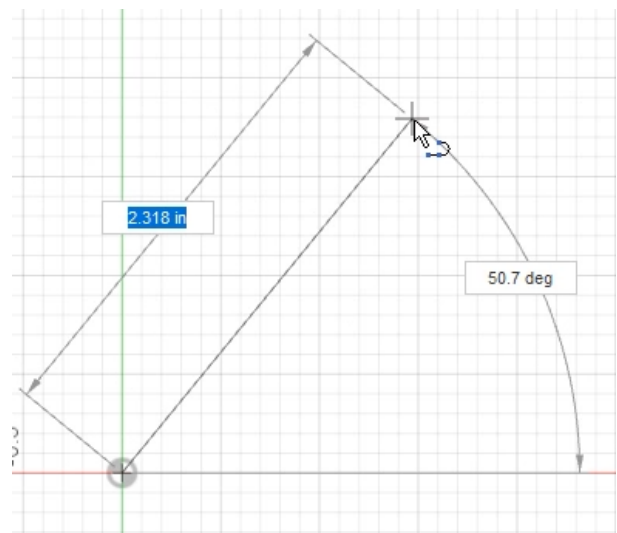


Figure 10. Place the line's first point

- 11.** Click to place the line's second point directly above the origin.

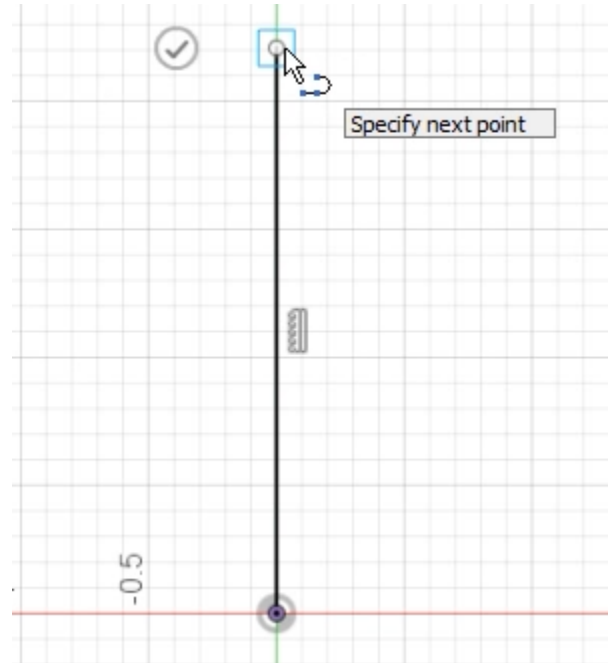


Figure 11. Place the line's second point

- 12.** Move the cursor and notice that the Line tool is still active. Type **3** into the dimension box and notice that the line automatically extends to 3 inches long.

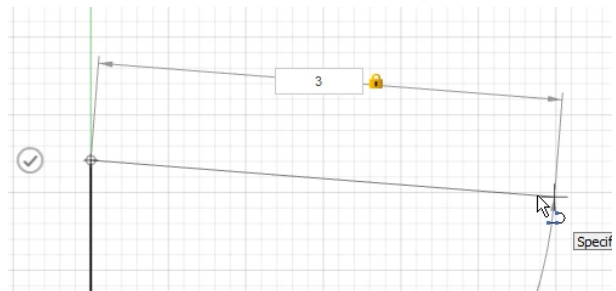


Figure 12. Dimension the length of the line

- 13.** Press Tab to move to the angle dimension and enter 85. Press Enter to create the geometry and end the Line tool. Notice the newer line is blue but the original line is black; black lines are fully defined and blue lines are underdefined. The first line's origin and angle are specified but the length could change if you click and drag its endpoint.

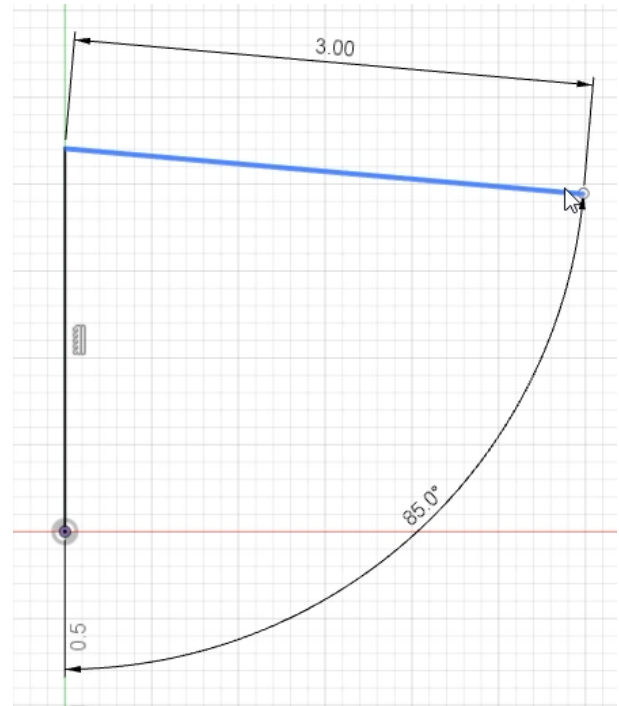


Figure 13. Dimension the line's angle

- 14.** Click Create> Sketch Dimension to open the Dimension tool.

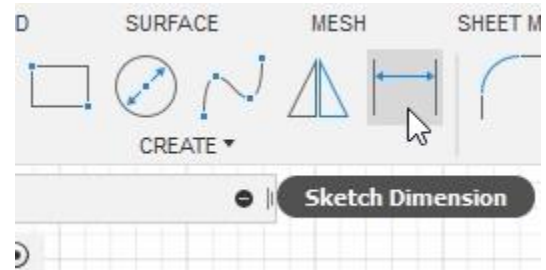


Figure 14. Open the Dimension tool

- 15.** Select the first line you drew to add a dimension to it, then enter **2.5** into the dimension box.

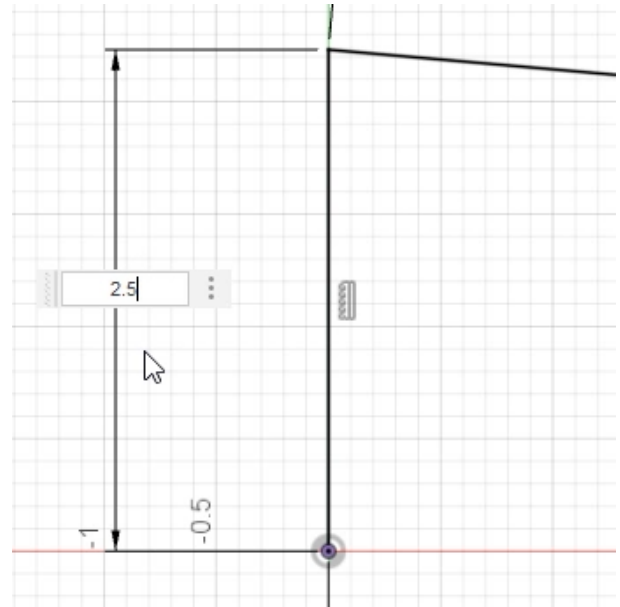


Figure 15. Dimension the line

- 16.** Press Enter and notice that both lines turn black to indicate that they are fully defined and cannot be moved without changing their dimensions or constraints. Press Esc to leave the Dimension tool.

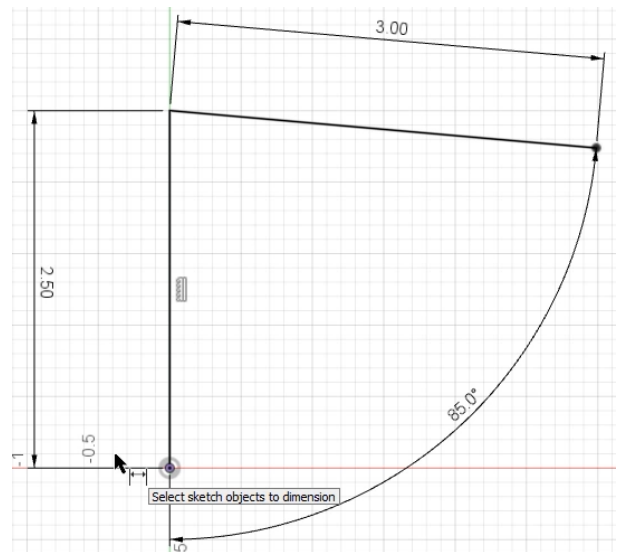


Figure 16. Inspect the defined geometry

- 17.** Press L to open the Line tool. Draw two lines and make sure they do not snap to anything by holding Ctrl (Windows) or Command (MacOS) as you place the lines. Press Esc to leave the Line tool. The two new lines are blue because they are underdefined. Hover your cursor over the joint where the two lines meet and notice that a coincident constraint is automatically created.

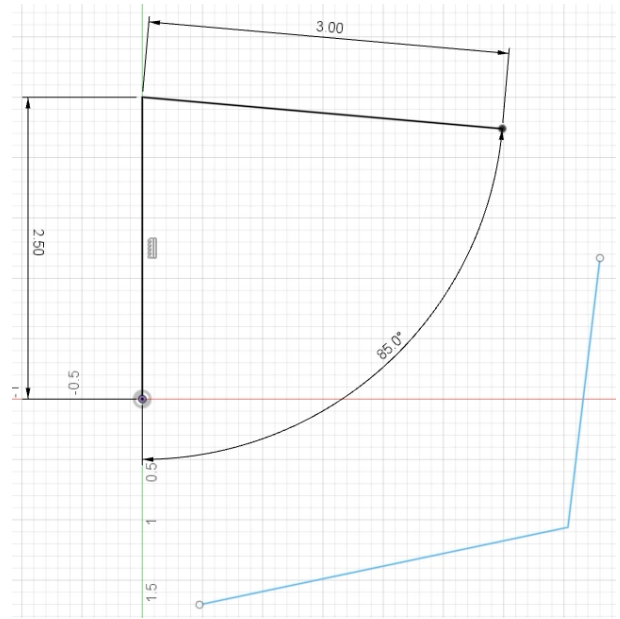


Figure 17. Draw two new lines

- 18.** Click Constraints> Perpendicular.

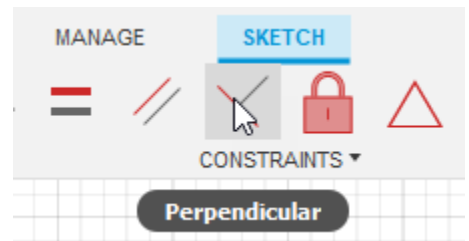


Figure 18. Open the Perpendicular tool

- 19.** Select the two new lines you drew to create a perpendicular constraint between them. Notice that the icon has changed where the two lines meet.

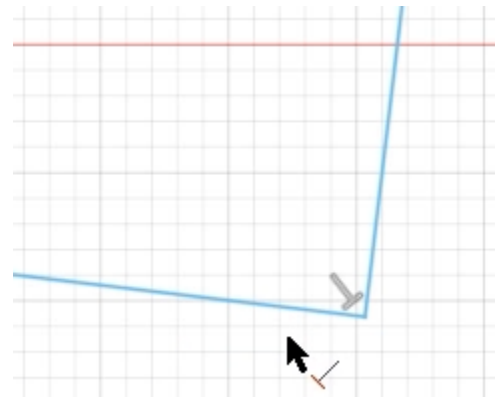


Figure 19. Make the two lines perpendicular

20. Click Constraints> Horizontal/Vertical.

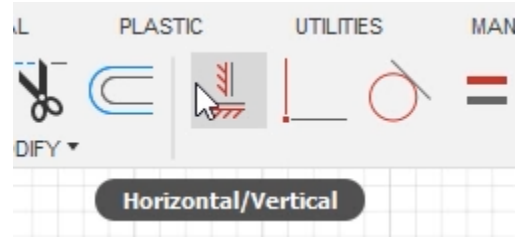


Figure 20. Open the Horizontal/Vertical tool

21. Select the line that is closest to being horizontal to add a horizontal constraint to it.

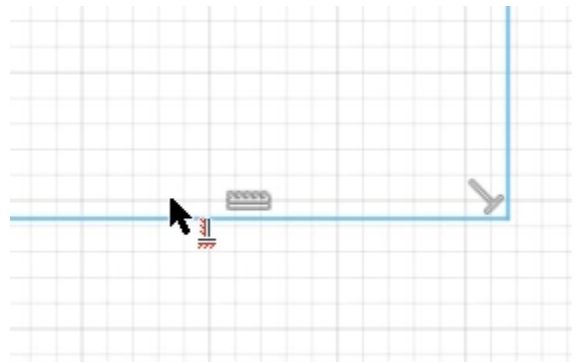


Figure 21. Make one of the lines horizontal

22. Click Constraints> Coincident.



Figure 22. Open the Coincident tool

- 23.** Select the free end of the horizontal line and the vertical line to create a coincident constraint between them. The new line will automatically snap into the correct position after the second selection is made.

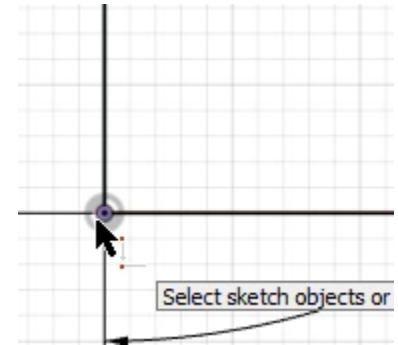


Figure 23. Create a Coincident constraint

- 24.** Create a second coincident constraint to join the remaining two free ends together. Press Esc. Notice that the area inside the shape turns blue; this indicates that the shape is closed and the inside area is selectable. This closed area is called a sketch profile.

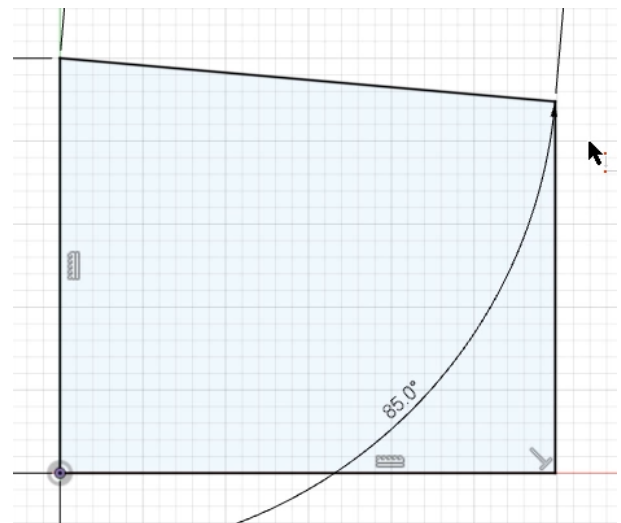


Figure 24. Add a second coincident constraint

- 25.** Constraints can also be accessed by selecting geometry, right-clicking it, then choosing the applicable constraint from the drop-down menu.

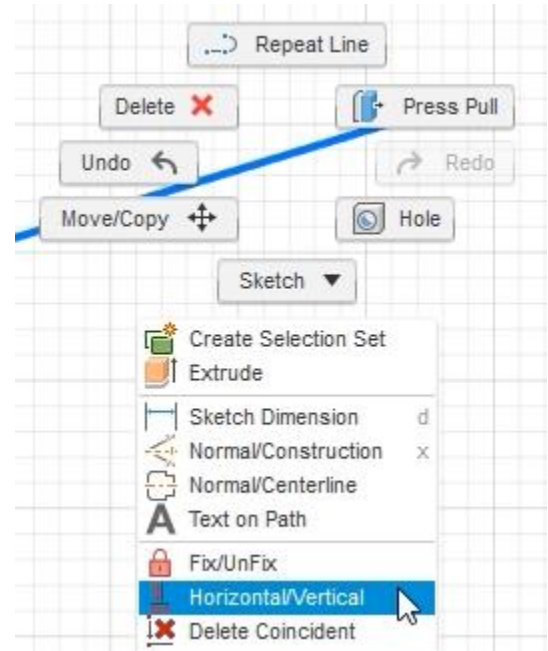


Figure 25. Explore alternate ways to apply constraints

- 26.** Press L to open the Line tool and draw a line connecting the shape's bottom left corner to its top right corner. Press Esc. This line currently divides the profile into two separate regions.

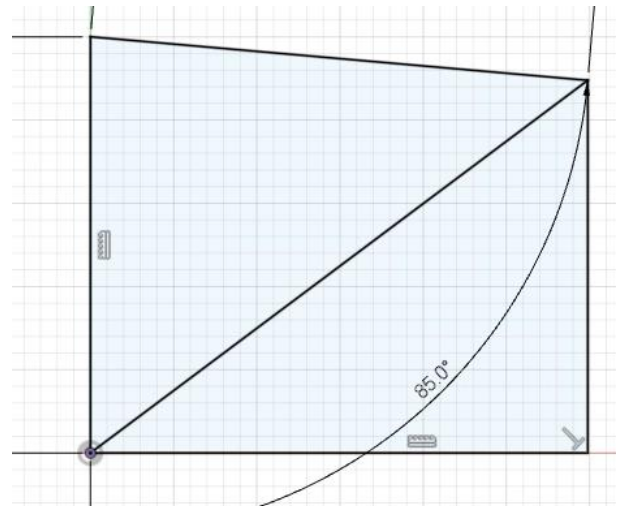


Figure 26. Draw a line

- 27.** Select the new line, then click the Construction option in the dialog's Line Type section.

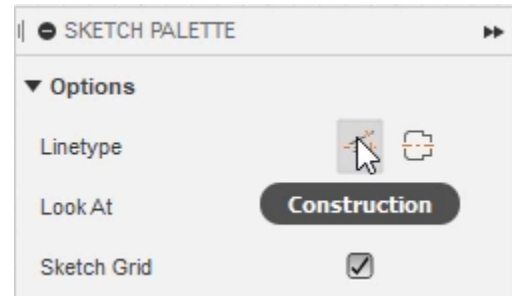


Figure 27. Convert the line to construction geometry

- 28.** The line now has a dashed appearance to indicate that it is construction geometry and the region is no longer divided into two separate regions.

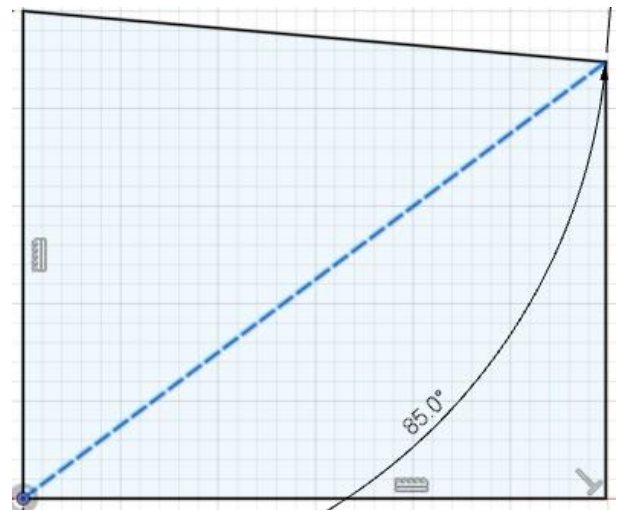


Figure 28. Inspect the construction geometry

29. Construction geometry can also be activated before you draw a line. Activate the dialog's Construction option, then press L to open the Line tool. Connect the profile's other two corners and notice that the line is created as construction geometry and does not need to be converted. Press Esc and notice that the Construction option is still selected in the Sketch Palette dialog. The next geometry you draw will also be created as construction geometry. Toggle off the Construction option.

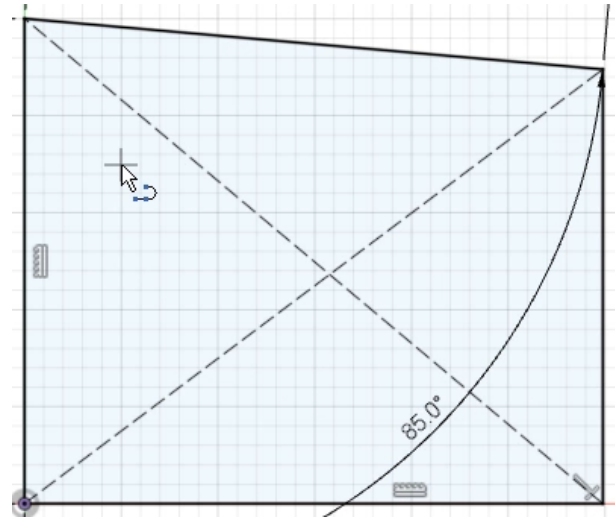


Figure 29. Draw a construction line

30. Click Create> Center Diameter Circle.

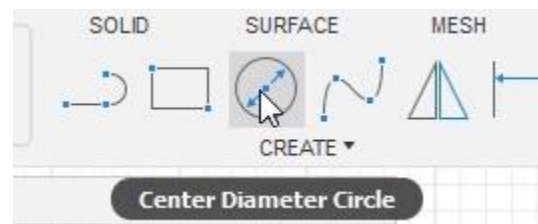


Figure 30. Open the Circle tool

31. Click one of the construction lines to place the circle's center on it, then click a second time to determine the circle's diameter. Press Esc. Click the circle's center and notice that it can be dragged to a new location.

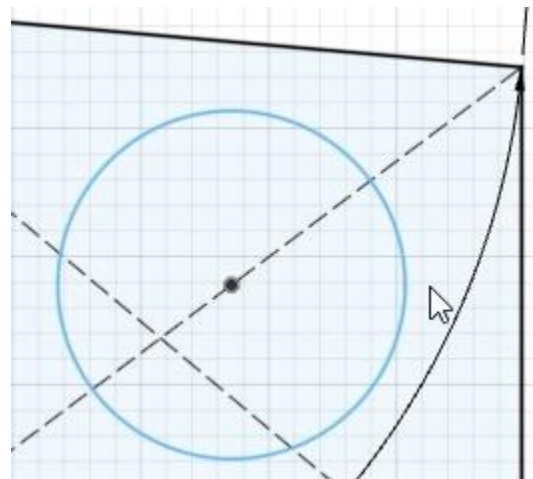


Figure 31. Draw a circle

- 32.** Click Constraints> Coincident, then Select the Circle's Center and the other construction line. After the second selection is made, the circle will snap to a position where it is coincident with the selected construction line.

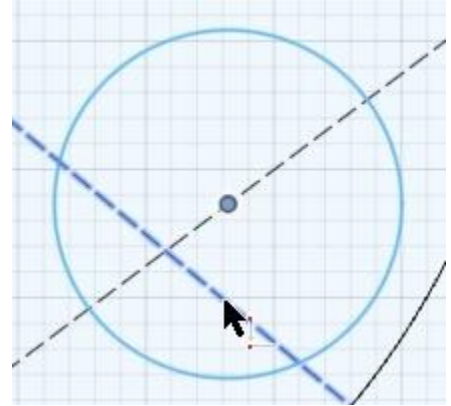


Figure 32. Add a coincident constraint

- 33.** Press D to open the Dimension tool and add a 1.25-inch dimension to the circle. Press Enter.

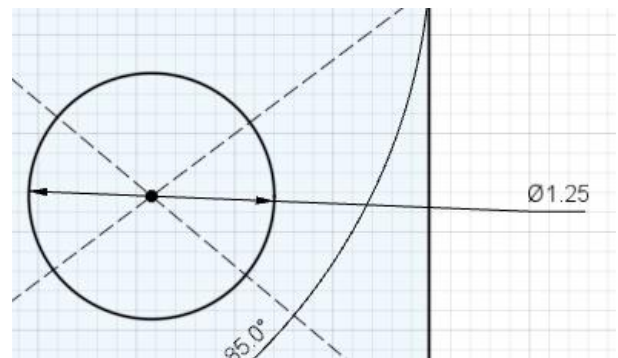


Figure 33. Dimension the circle

- 34.** Finish the current sketch by clicking Finish Sketch> Finish Sketch.



Figure 34. Finish the sketch

- 35.** Open the Extrude tool by clicking Create> Extrude.



Figure 35. Open the Extrude tool

- 36.** Select the region shown in the image on the right, then use the on-screen manipulator to extrude it backwards.

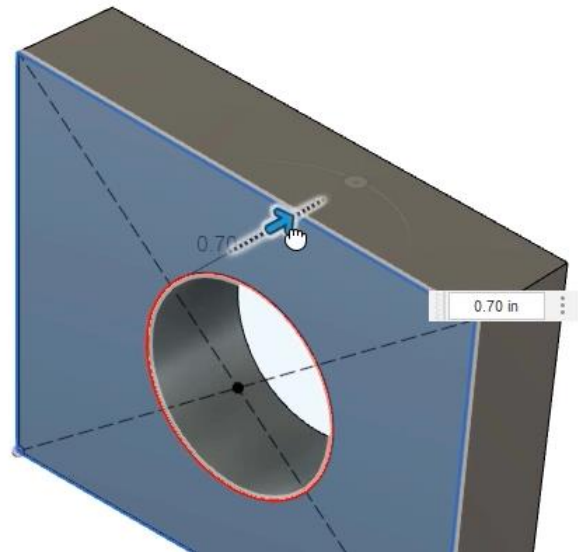


Figure 36. Extrude the profile

37. Enter **1.5** into the dialog's Distance box, then choose the Symmetric option from the Direction section's menu. Notice that the profile is extruded 1.5 inches in both directions. OK the dialog.

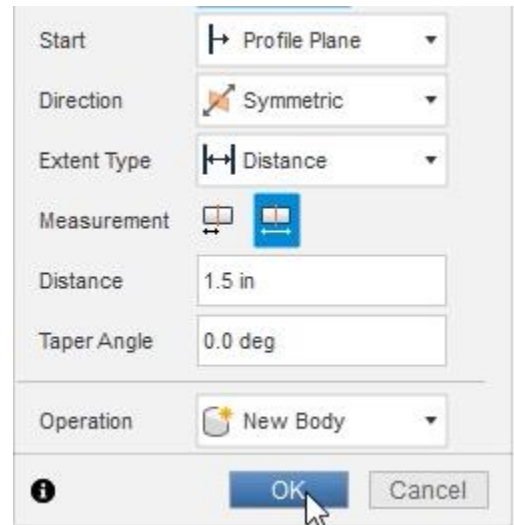


Figure 37. Create the extrude

38. Click Modify > Fillet.

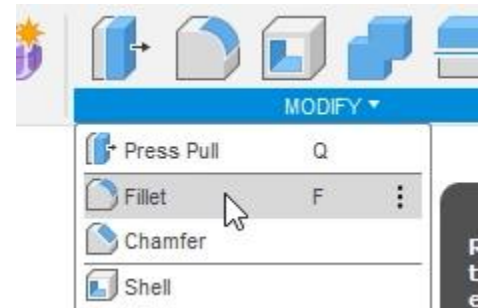


Figure 38. Open the fillet tool

39. Select two of the model's edges and drag the on-screen manipulator to add a fillet to them. Right-click in an open area of the Canvas to open the right-click Marking Menu and choose OK to end the Fillet tool.

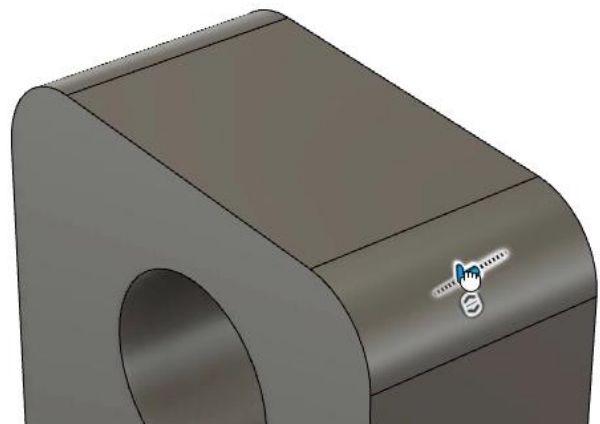


Figure 39. Fillet two of the edges

- 40.** Right-click to open the Marking Menu and choose the Repeat Fillet option. This is an efficient way to reopen the tool you most recently used.

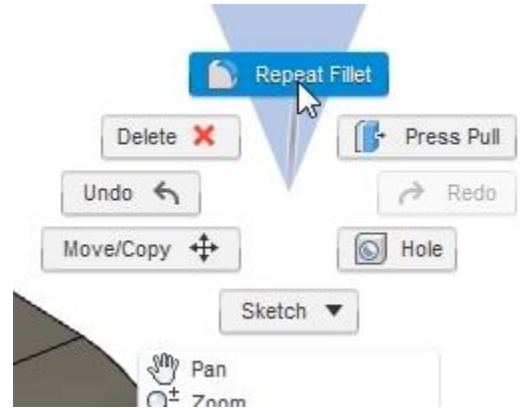


Figure 40. Reopen the fillet tool

- 41.** Select one of the edges and notice that additional edges might be automatically selected if they are tangent to the edge you selected. Drag the on-screen manipulator to fillet the selected edges. OK the Fillet dialog to accept the new fillet.



Figure 41. Fillet more edges

42. Save the file and name it **Intro to CAD.**

Click the Save dialog's Save, then continue to the next module.

Save

Name:

Intro to CAD

Location:

22 Intro to CAD CAM Practical CNC > CAD

Figure 42. Save the file