Step-by-step guide

Create digital components

Explore some of Fusion 360's fundamental tools for creating and modifying solid or surface geometry.

Learning objectives:

- Create a fully dimensioned and constrained sketch.
- Create a 3D model using features.
- Modify a model with fillets and chamfers.
- Apply surface features to create or patch complex shapes.

The completed exercise

1. Open the supplied *Lofted Bodies.f3d* and *Under Defined Flange.f3d* files.
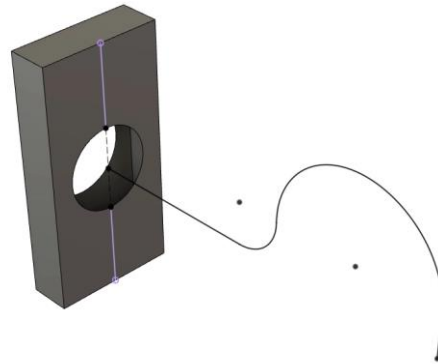

Figure 1. Open the supplied files

2. Navigate to the Under Defined Flange tab and notice that the existing body is a surface body. This body needs to be patched and converted back to a solid body.
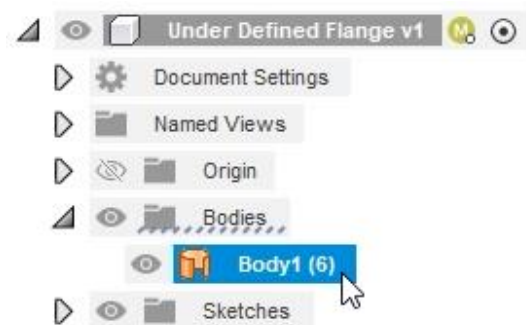

Figure 2. Inspect the Browser

3. Expand the Browser's Sketches folder and turn off the visibility for the Sweep Path sketch. To do this, click the eyeball icon next to the sketch.
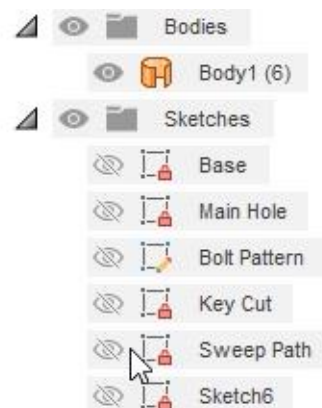

Figure 3. Hide the sketch

4. Rotate the part and notice that the opening is tapered; the opening is smaller on one side than the other. Also notice that the inside face is missing. This face needs to be re-created and the taper needs to be removed from the hole.
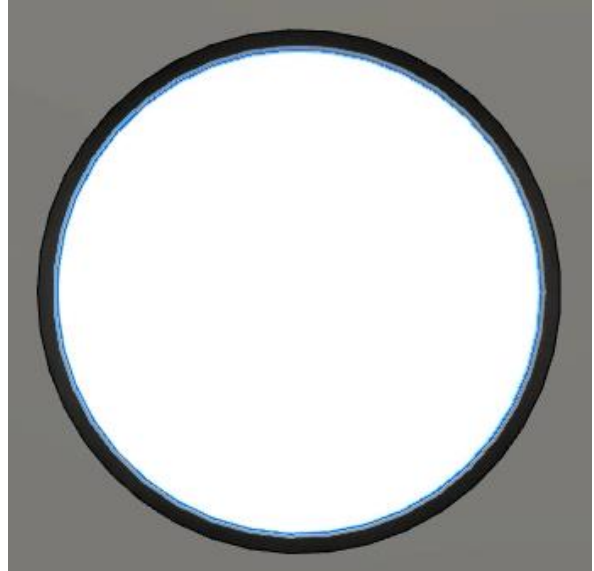

Figure 4. Inspect the body

5. Activate the Ribbon's Surface tab so that the orange surface tools are visible. Open the Extrude tool by clicking Create> Extrude or press E.

Figure 5. Open the Extrude tool

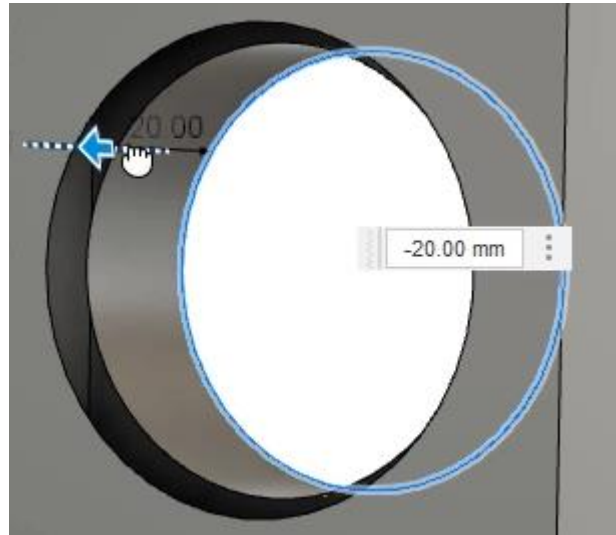**6.** Select the smaller of the two circles and extrude the geometry all the way through the body.

Figure 6. Extrude the geometry

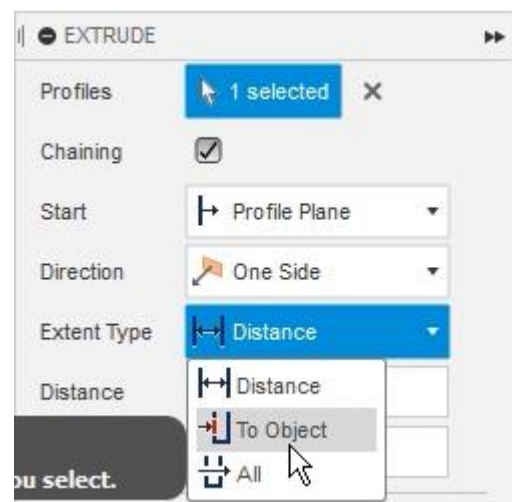**7.** Choose the To Object option from the dialog's Extent Type menu.

Figure 7. Change the extent type

8. Select the body's face and notice that the selection is extruded up to the face. If you change this face's position, the extrusion's distance will automatically change because they are linked. OK the Extrude dialog.
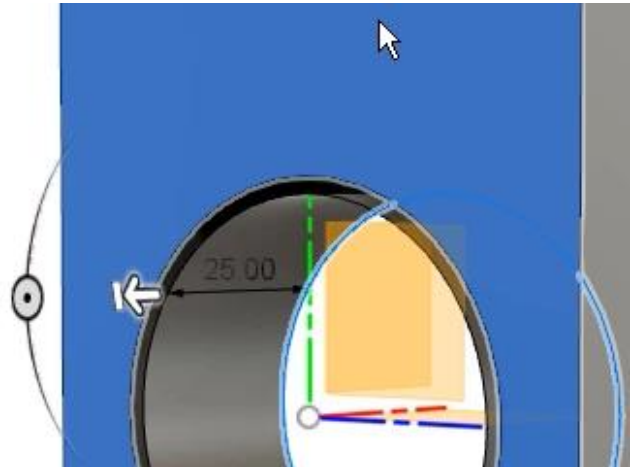

Figure 8. Extrude up to a face

9. Open the Patch tool by clicking Create> Patch.

Figure 9. Open the Patch tool

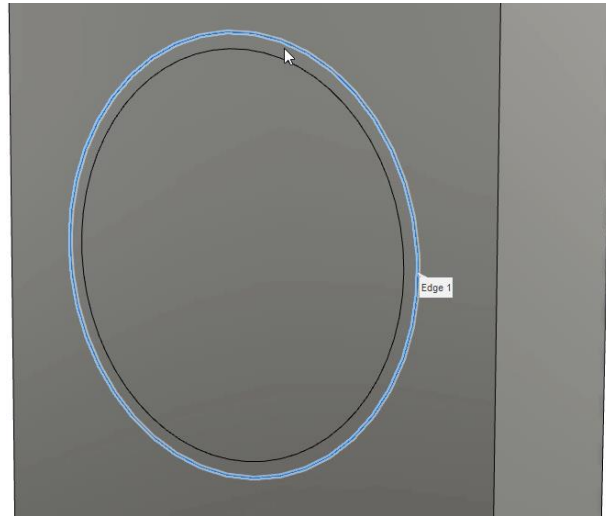**10.** Select the larger circle and notice that the hole is automatically patched.

Figure 10. Select the geometry

11. Open the Trim tool by clicking Modify> Trim.

Figure 11. Open the Trim tool

12. Rotate the body and choose the cylinder as the Trim dialog's Trim Tool selection.
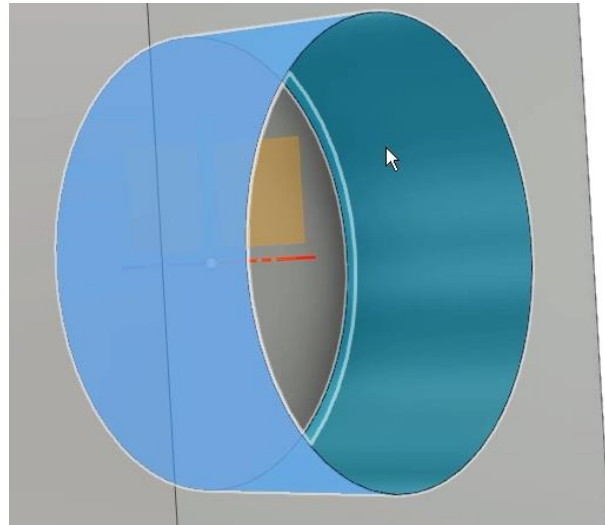

Figure 12. Choose the trim tool

13. Select the area inside the smaller circle as the area that needs to be trimmed. OK the Trim dialog.
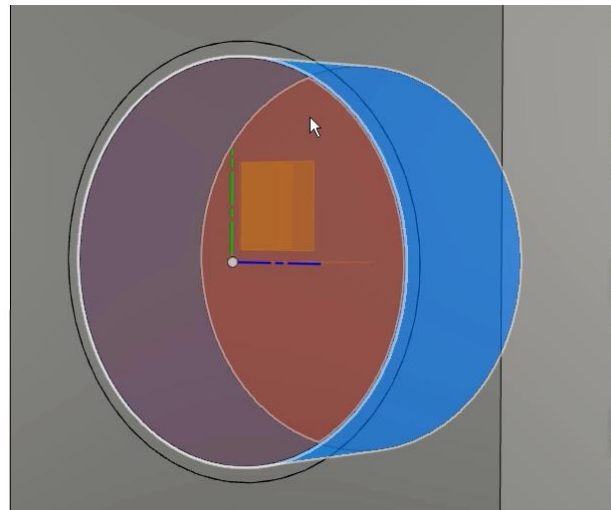

Figure 13. Choose the area to remove

14. Now that all the surfaces are created, they need to be combined to create a solid body. Open the Stitch tool by clicking Modify> Stitch.

Figure 14. Open the Stitch tool

15. Select the three surface bodies you want to stitch together. OK the Stitch dialog.
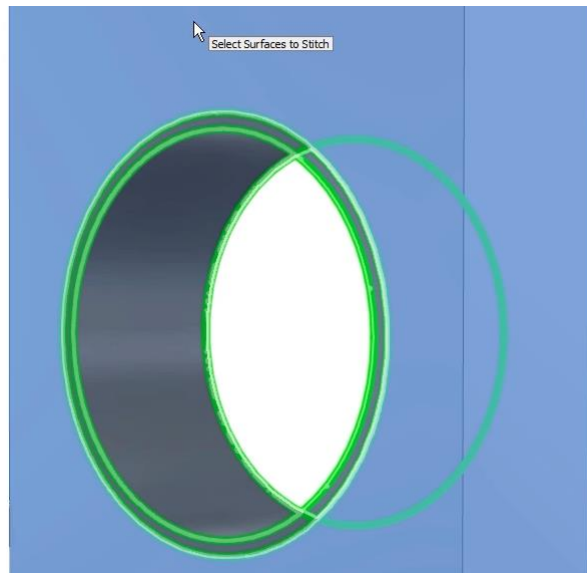

Figure 15. Select the bodies

16. Notice that the surface bodies are removed from the Browser and are replaced by a single solid body.
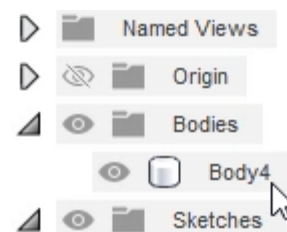

Figure 16. Inspect the Browser

17. Activate the Ribbon's Solid tab and turn on the visibility for the Bolt Pattern sketch. This sketch's icon does not have a red lock icon, which indicates the sketch is underdefined. Right-click the sketch and choose Edit Sketch from the menu.
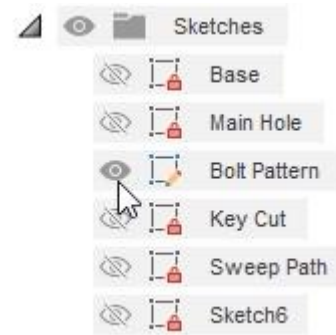

Figure 17. Show the Bolt Pattern sketch

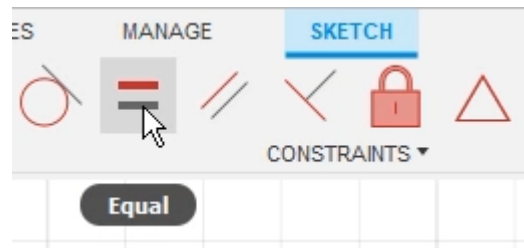**18.** Open the Equal constraint tool by clicking Constraints> Equal.

Figure 18. Open the Equal constraint tool

19. Select the top two circles to add an equal constraint between them. Notice that the right circle resizes to match the 15 mm left circle.

Figure 19. Select two circles

20. Add Equal constraints between the top left circle and each of the two bottom circles so that all of them are the same size.
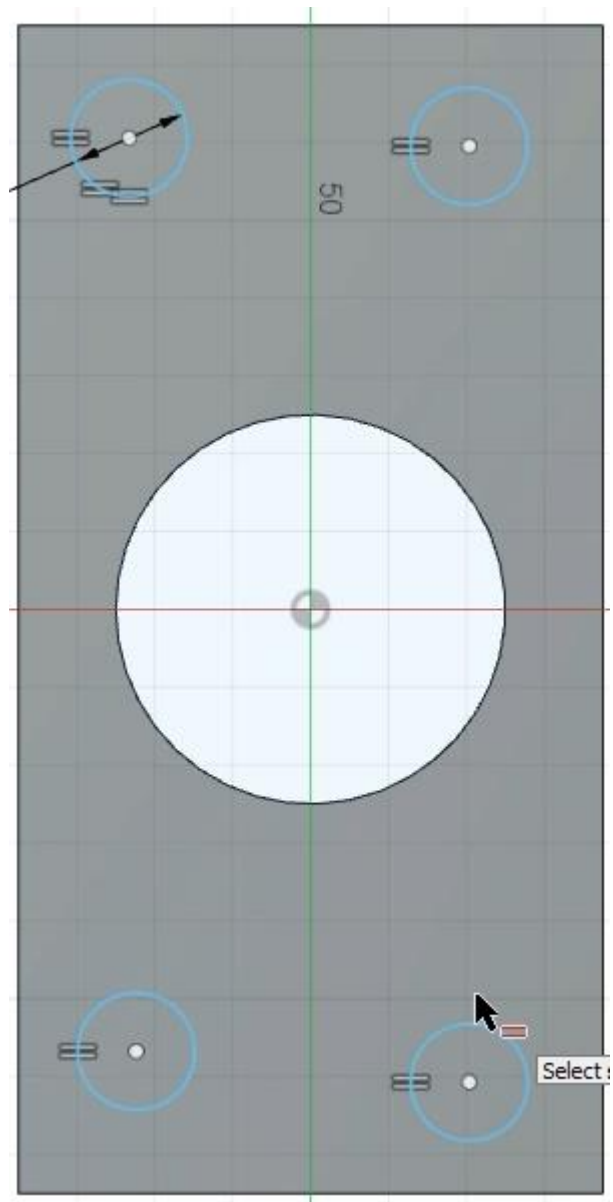

Figure 20. Continue adding Equal constraints

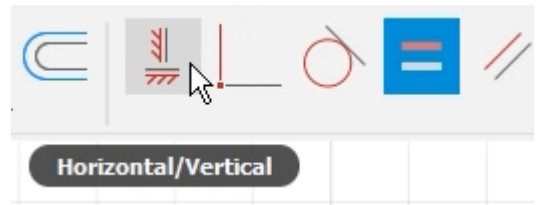**21.** Click Constraints> Horizontal/Vertical.

Figure 21. Open the Horizontal/Vertical constraint tool

22. Select the center point for each of the two top circles and notice that they move to a position horizontal to each other.
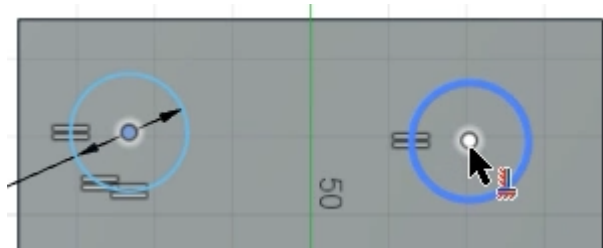

Figure 22. Select the top two circles

23. Use the same tool to add a vertical constraint between the center of the top circles and the bottom circles. After you finish, the rows of holes should be horizontal to each other and the columns of holes should be vertical to each other.
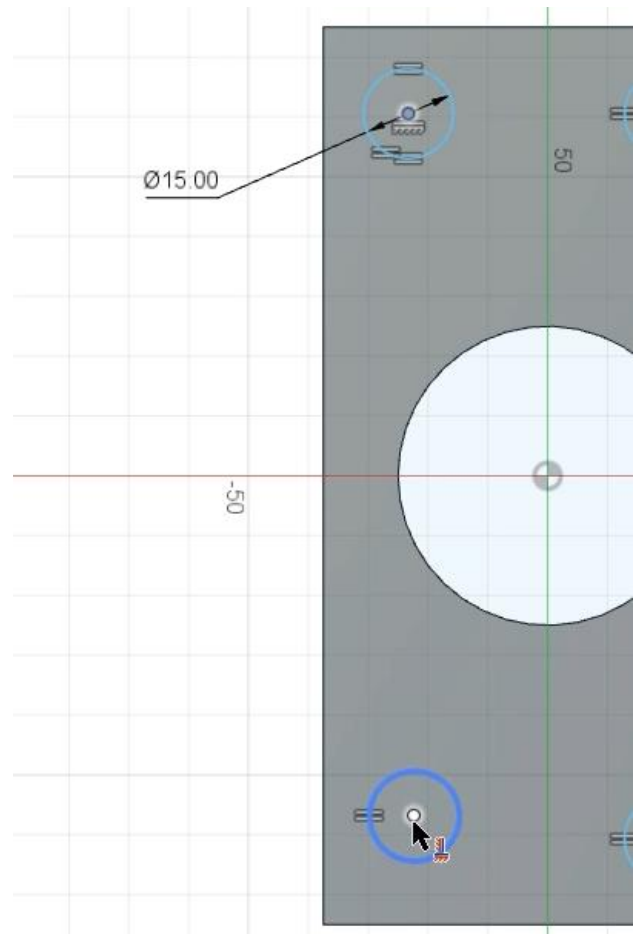

Figure 23. Apply Vertical constraints

24. Click Create> Sketch Dimension or press D to open the Sketch Dimension tool.
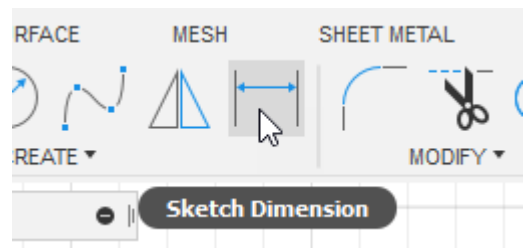

Figure 24. Open the Dimension tool

25. First, select the circle's center point. Next, select the body's edge to add a dimension between them. Click to place the dimension into the drawing, then enter a value of **15 mm**. Press enter.
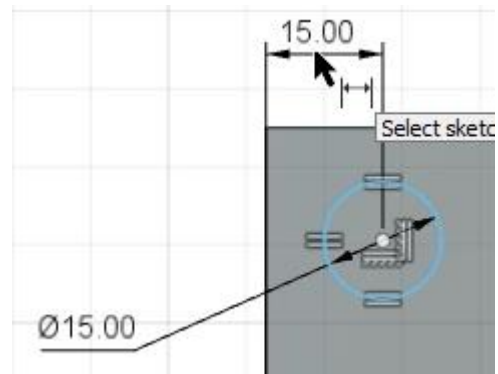

Figure 25. Dimension the circle's position

26. Create another dimension between the circle's center and the body's top edge. Instead of entering a number value, click the dimension you created in the previous Step so that its dimension automatically populates the current dimension. Notice that the new dimension is displayed with an fx: in front of it to indicate that its value is derived using an equation. Also notice that the circle turns black to indicate that it is fully defined.
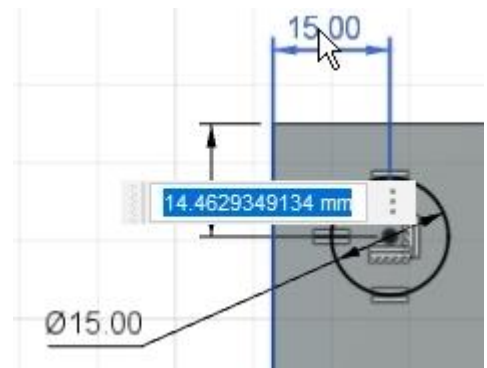

Figure 26. Link two dimensions

27. Create a dimension between the top right circle and the body's edge. Link this new dimension to the dimension you created in Step 25. Notice that this circle turns black to indicate it is fully defined.
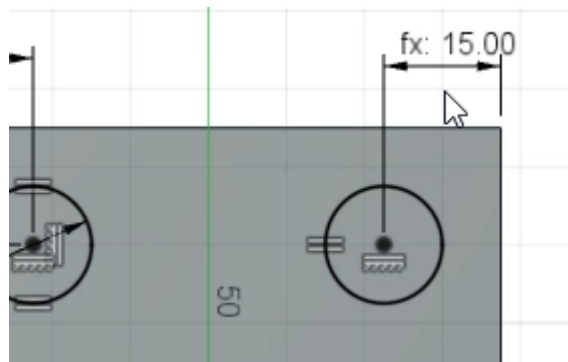

Figure 27. Create a new dimension

28. Create another dimension on one of the bottom holes and link this new dimension to the dimension you created in Step 25. Notice that all the holes are black, which indicates they are all fully defined.
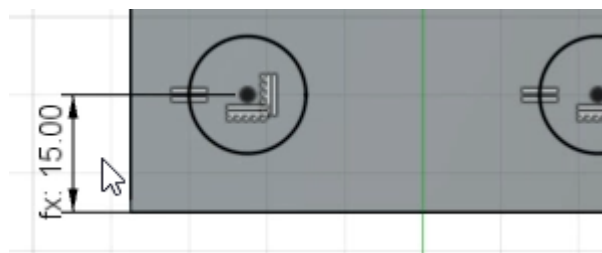

Figure 28. Create a new dimension

29. Click Finish Sketch> Finish Sketch.

Figure 29. Finish the sketch

30. Click Create> Extrude or press E to open the Extrude tool.

Figure 30. Open the Extrude tool

31. For the Extrude dialog's Profiles selection, choose the four circular regions shown in the image on right.

Figure 31. Select the profiles

32. Choose the To Object option from the dialog's Extent Type menu, then select the body's back face. The four holes will be extruded up to this selected face. OK the dialog to create the feature.
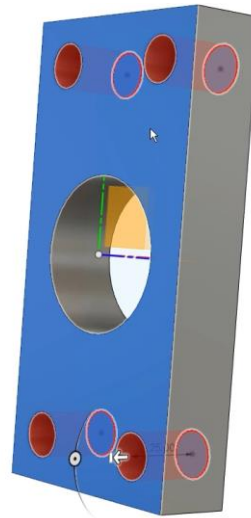

Figure 32. Choose the extrude's extent

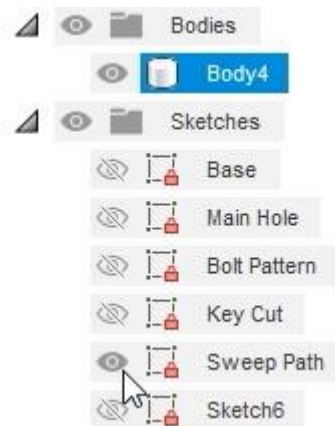**33.** Use the Browser to turn on the visibility for the Sweep Path sketch.

Figure 33. Show the sketch

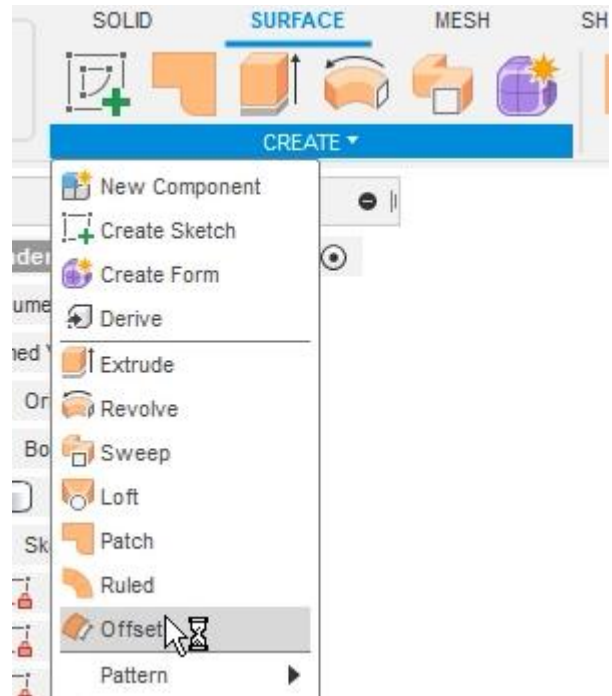**34.** Activate the Ribbon's Surface tab, then click Create> Offset.

Figure 34. Open the Offset tool

35. Select the face shown in the image on the right, then enter **5 mm** into the dialog's Distance box.
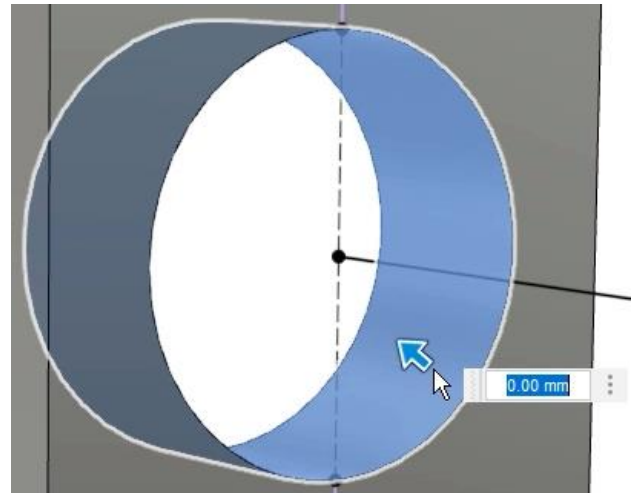

Figure 35. Select the face to offset

36. OK the dialog to create the offset
surface.

Figure 36. Create the offset surface

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37. Open the Thicken tool by clicking
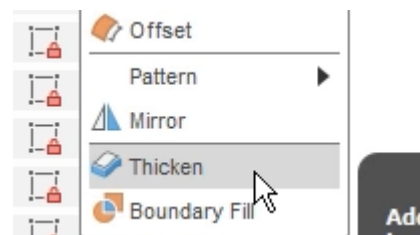Create> Thicken.

Figure 37. Open the Thicken tool

38. Select the surface body you created in Step 36, then enter **-5 mm** into the dialog's Thickness box. Make sure the New Body option is selected in the dialog's Operation menu, then OK the dialog.
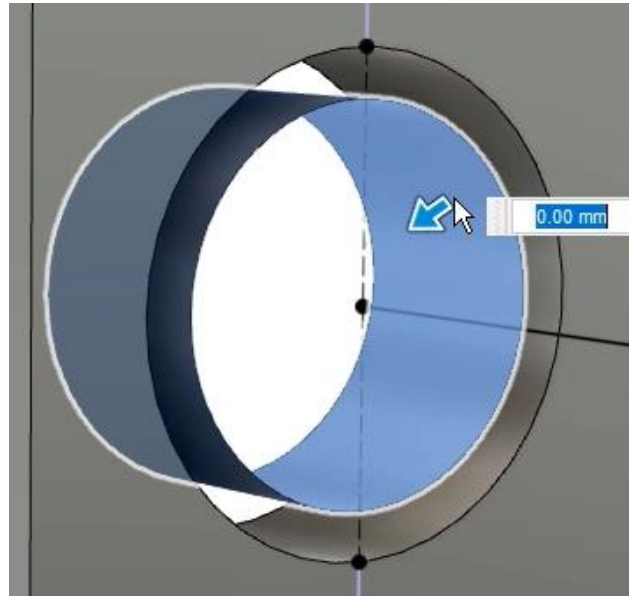

Figure 38. Thicken the surface

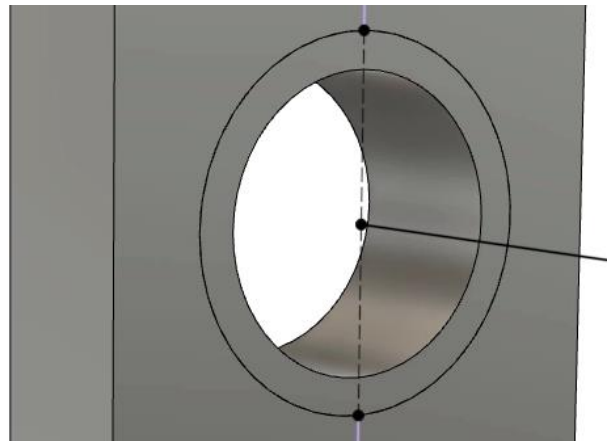**39.** The surface body is used to create a thickened solid body.

Figure 39. Inspect the result

40. Activate the Ribbon's Solid tab, then click Create> Sweep.
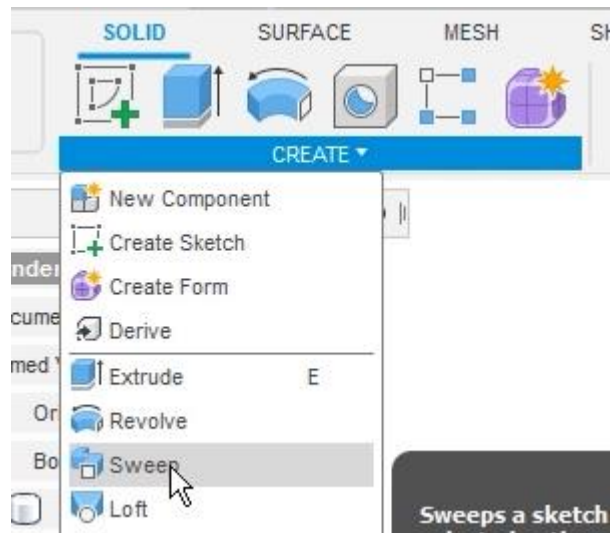

Figure 40. Open the Sweep tool

41. Select the region shown in the image on the right as the Sweep dialog's Profile selection.
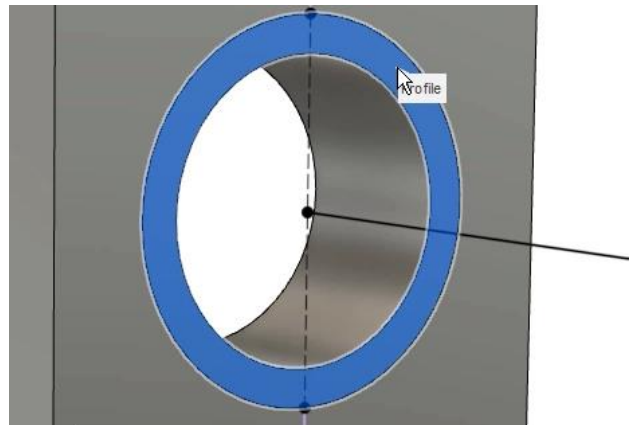

Figure 41. Select the profile

42. For the dialog's Path selection, select the Sweep Path sketch. OK the dialog to sweep the solid body.
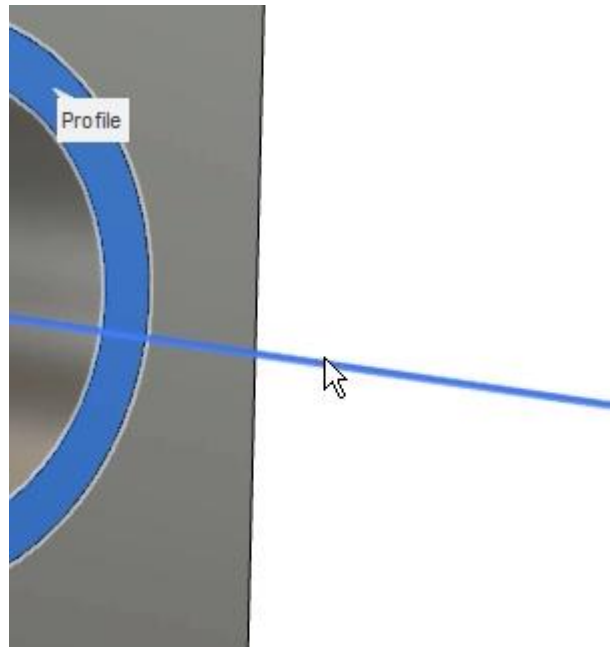

Figure 42. Select the path

43. Right-click the Sweep Path sketch and choose Show Dimension from the menu.
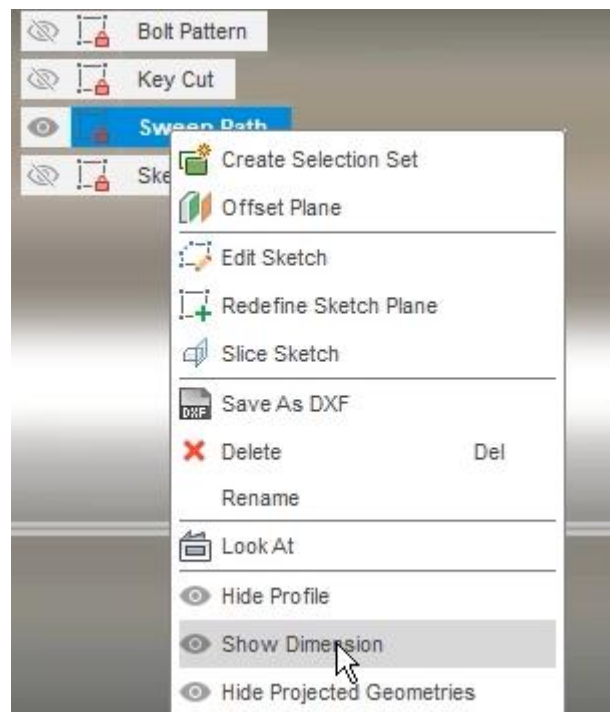

Figure 43. Show the sketch's dimensions

44. The sweep created some tight geometry inside one of the curves. Click the 25 mm dimension and increase it to **30 mm**. Press Enter. Notice that the sweep's geometry automatically updates to match the new sketch geometry. Use the eyeball icon to turn off the visibility for the Sweep Pass sketch.
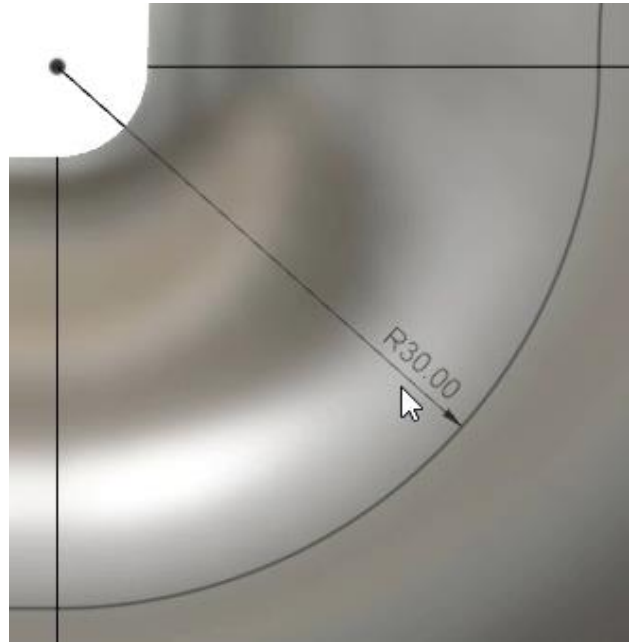

Figure 44. Adjust the sketch

45. Notice that the solid bodies were combined into a single solid body during the sweep feature. You can control whether the bodies are combined by using the options in the Sweep dialog's Operation menu.
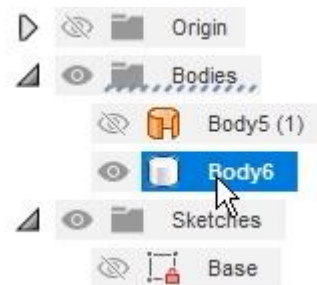

Figure 45. Explore the Browser

46. Double-click the timeline's Sweep feature to edit it.

Figure 46. Edit the timeline feature

47. Instead of using the options in the Operation menu to control whether the bodies are combined, turn off the visibility for the rectangular body. OK the dialog to update the feature.
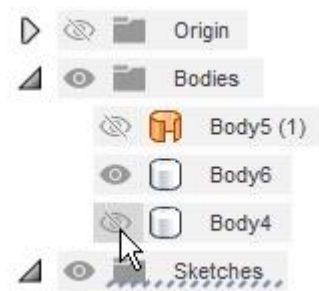

Figure 47. Hide the body

48. Notice that the sweep is created and is not joined to the original rectangular body.
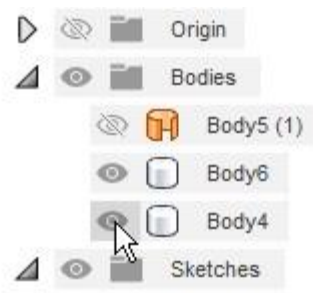

Figure 48. Inspect the result

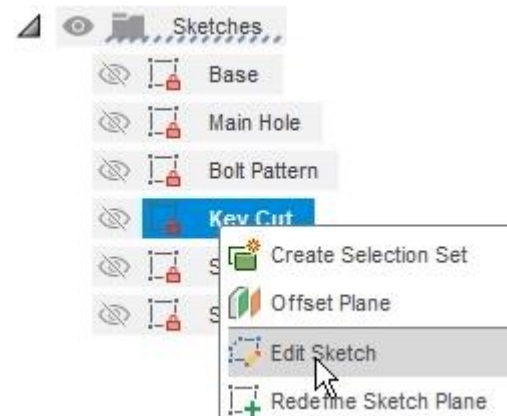**49.** In the Browser's Sketches folder, right-click the Key Cut sketch and choose Edit Sketch from the menu.

Figure 49. Edit the sketch

50. Use the Browser to hide the surface body.

Figure 50. Hide the surface body

51. The vertical line on the right side of the sketch is a line that has been projected from the surface body. Even though the sketched geometry is not closed, the projected sketch line can be used to close the sketch region because they overlap. Click inside the sketch region and notice that it is selectable and turns blue. The sketch region can be used for extrudes and other features. Click Finish Sketch> Finish Sketch.
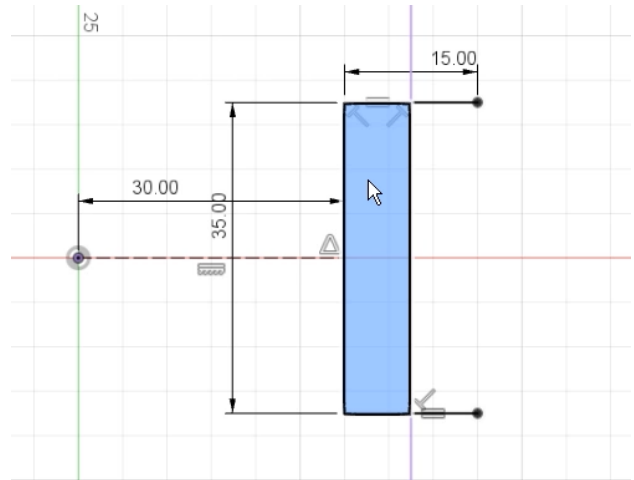

Figure 51. Explore the sketch geometry

52. Save the design.

Figure 52. Save the design
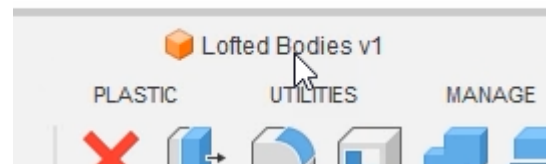
53. Navigate to the Lofted Bodies tab.

Figure 53. Navigate to the Lofted Bodies tab
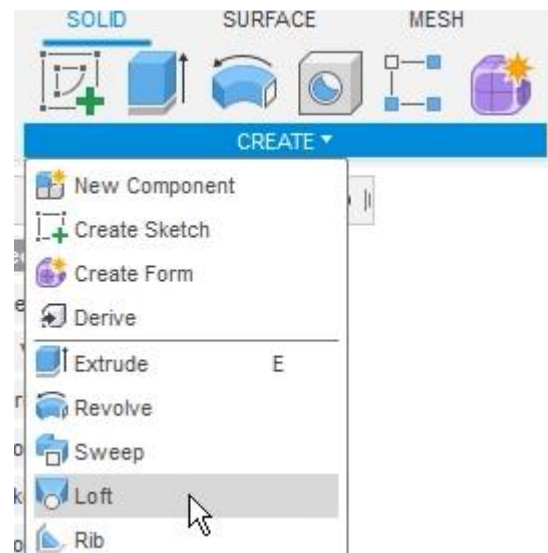
54. Click Create> Loft.

Figure 54. Open the Loft tool

55. For the dialog's Profile selections, first select the bottom region shown in the image on right, then select the top region.
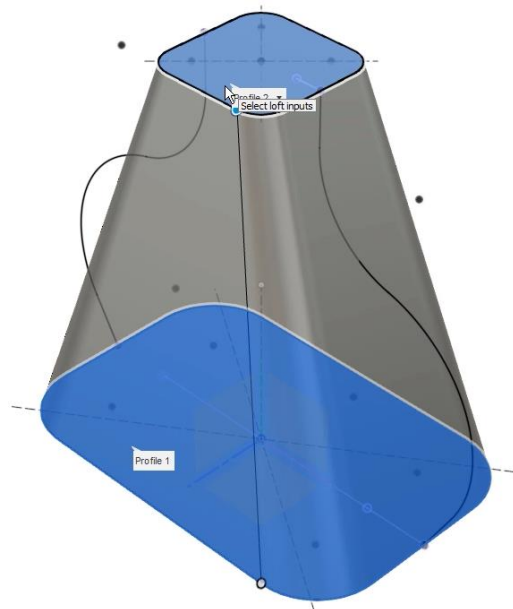

Figure 55. Select the profiles

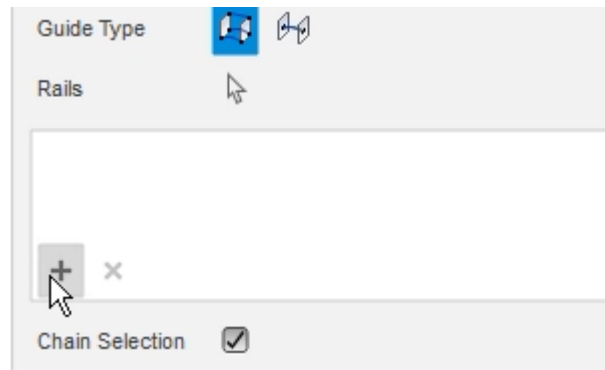**56.** In the dialog's Rails section, click the plus icon to select a new rail.

Figure 56. Click the plus icon

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57. Select the spline shown in the image on the right as the first rail selection and notice the loft's shape.
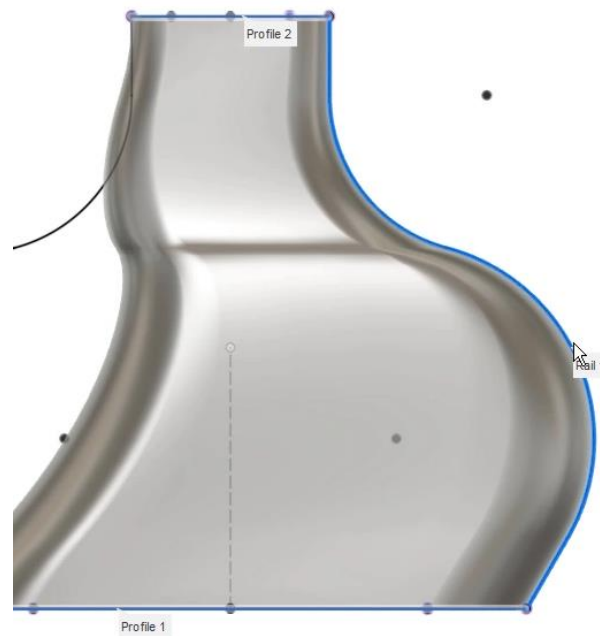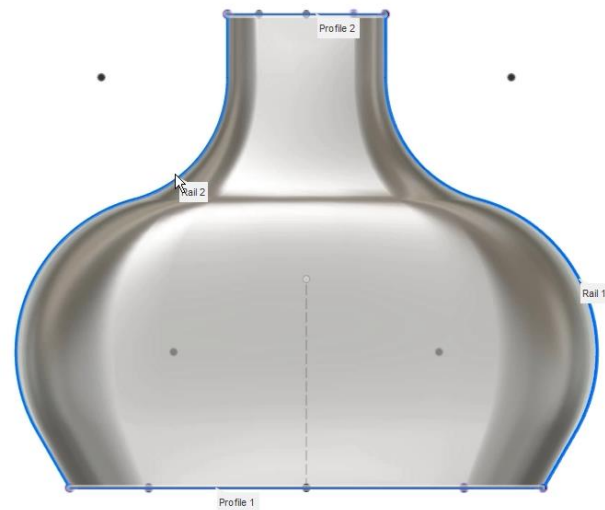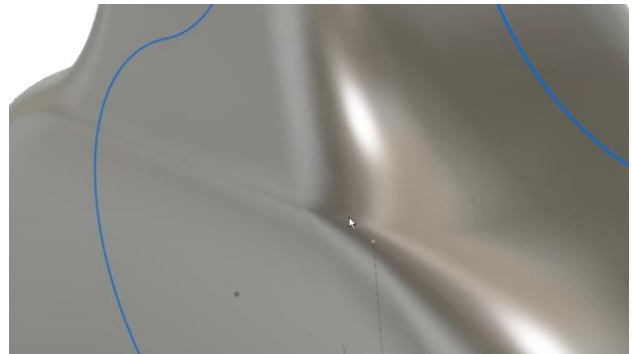

Figure 57. Select the rail

58. Select the second spline and notice
how the loft updates.

Figure 58. Select a second rail

59. Inspect the loft and notice that the
geometry could be adjusted to remove
the small crease. OK the dialog to
create the loft.

Figure 59. Inspect the loft

60. Open the Shell tool by clicking Modify > Shell.

Figure 60. Open the Shell tool

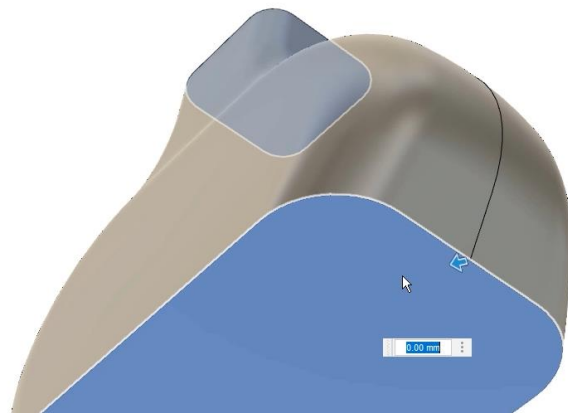**61.** Select the body's top and bottom faces, then enter **2 mm** into the dialog's Inside Thickness box. OK the dialog.

Figure 61. Configure the Shell

62. Inspect the result and notice that the body has a consistent 2-millimeter wall thickness.

Figure 62. Inspect the result

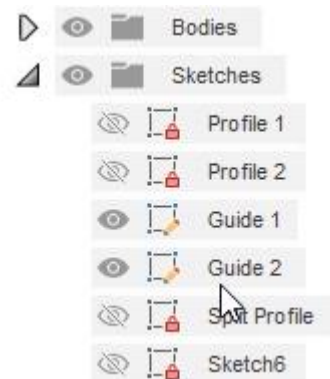**63.** Expand the Browser's Sketches folder and turn on the visibility for Guide 1 and Guide 2.

Figure 63. Show the guide sketches

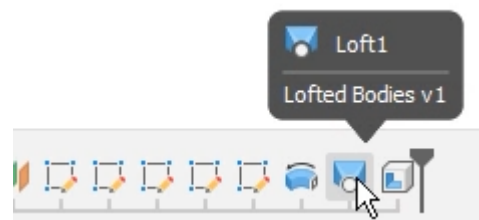**64.** Double-click the timeline's Loft feature to edit it.

Figure 64. Edit the Loft feature

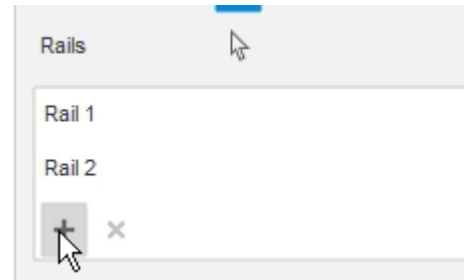**65.** Click the dialog's plus icon to add more rail selections.

Figure 65. Add more rail selections

66. Choose the two new rail sketches to add them to the rail selections. OK the dialog.

Figure 66. Choose the new rails

67. Inspect the result and notice that the crease has been mostly eliminated. Also notice that the Shell feature automatically updates and the body still has a consistent 2-millimeter wall thickness despite the updated geometry.
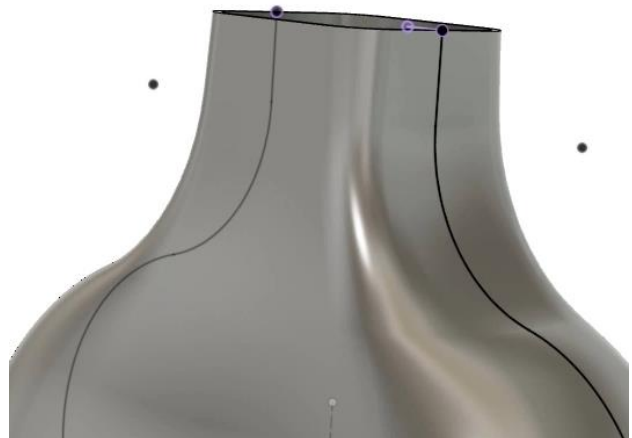

Figure 67. Inspect the result

68. Drag the timeline marker backwards
so that it excludes the Shell feature.
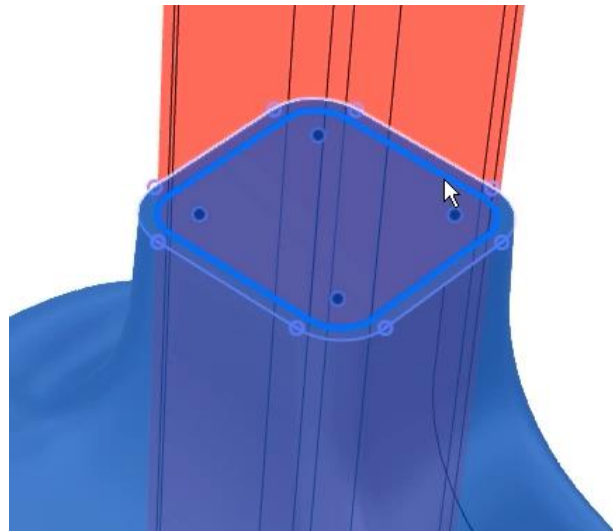

Figure 68. Drag the timeline marker backwards

69. In the Browser's Sketches folder, turn
on the visibility for the Split Profile
sketch. Click Modify> Split Body.

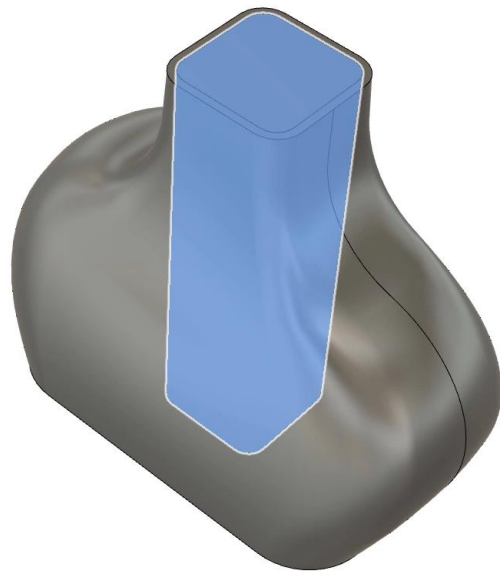*Figure 69. Open the Split Body tool*

70. For the dialog's Body to Split selection, choose the lofted body. For the Splitting Tool selection, choose the Split Profile sketch. OK the dialog.

Figure 70. Configure the Split

71. Expand the Browser's Bodies folder and notice that the body has been split into two separate bodies.

Figure 71. Inspect the result

72. Select the Split feature in the timeline, then press Delete. Notice that two separate bodies are now a single body again.

Figure 72. Delete the Split feature

73. Make sure the lofted body and the Split Profile sketch are visible, then click Modify> Split Face. Select the top face as the dialog's Faces to Split option, then choose the sketch as the Splitting Tool. OK the dialog.
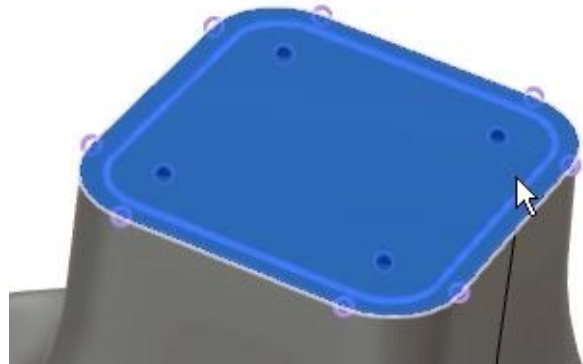

Figure 73. Configure the split

74. Use the Browser to turn off the visibility for the Split Profile sketch and notice that the lofted body's top face is split into two separate faces. However, the split face does not separate the body into two separate bodies.
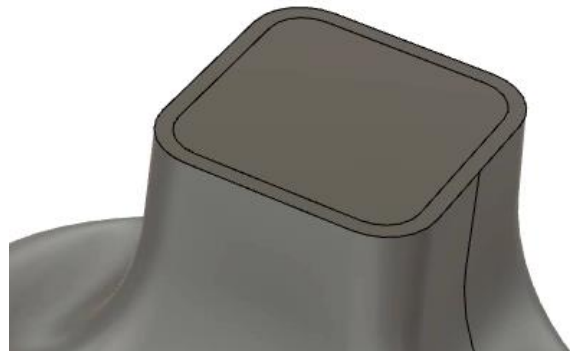

Figure 74. Inspect the result

75. Use the Browser to turn on the visibility for the Plug body, then click Modify> Combine.
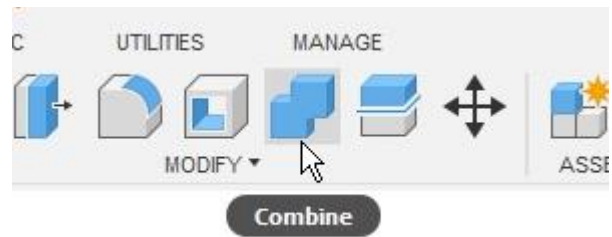

Figure 75. Open the Combine tool

76. Choose the loft body as the dialog's Target Body selection.

Figure 76. Choose the target body

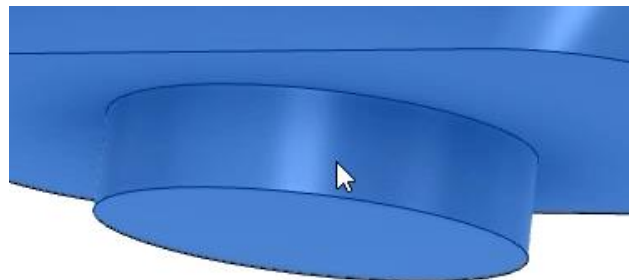**77.** For the dialog's Tool Body selection, choose the Plug body.

Figure 77. Choose the tool body

78. Choose the Cut option from the dialog's Operation section, then activate the Keep Tools option. OK the dialog.
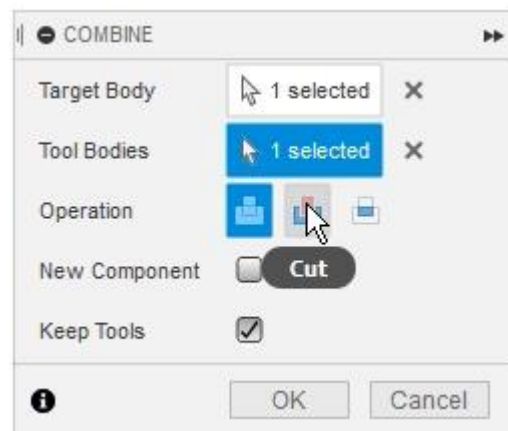

Figure 78. Configure the combine

79. Turn off the visibility for the Plug body and inspect the result. The Plug body's geometry is cut from the loft body.
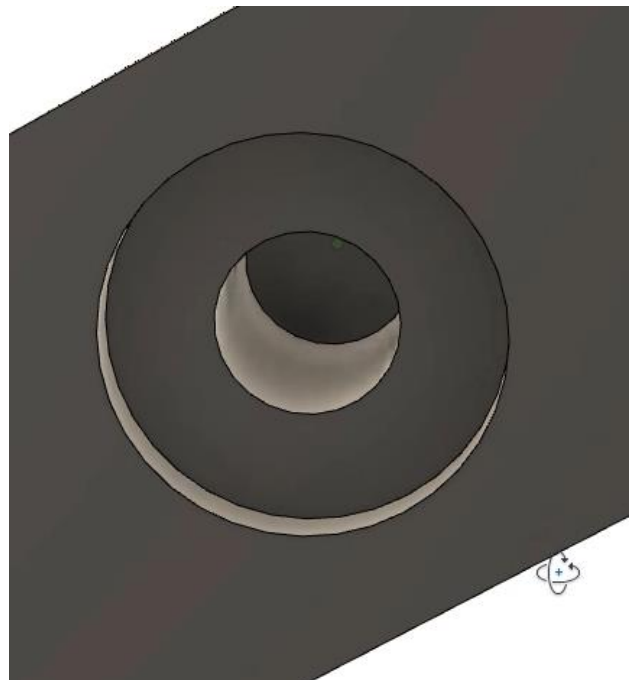

Figure 79. Inspect the result

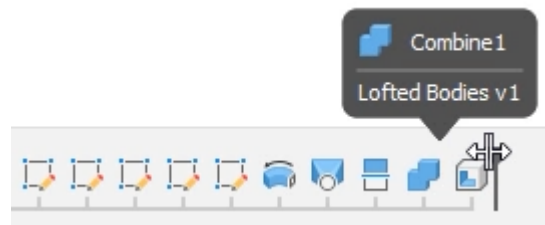80. Drag the timeline marker to the end of the timeline to reveal the Shell feature.

Figure 80. Move the timeline marker to the end of the timeline

81. Notice that the body is not shelled correctly. Double-click the timeline's Shell feature to edit it.

Figure 81. Inspect the result

82. Clear the dialog's Faces/Body selection, then select three new faces. Select the loft body's top two faces and the plug area's top face. OK the dialog.
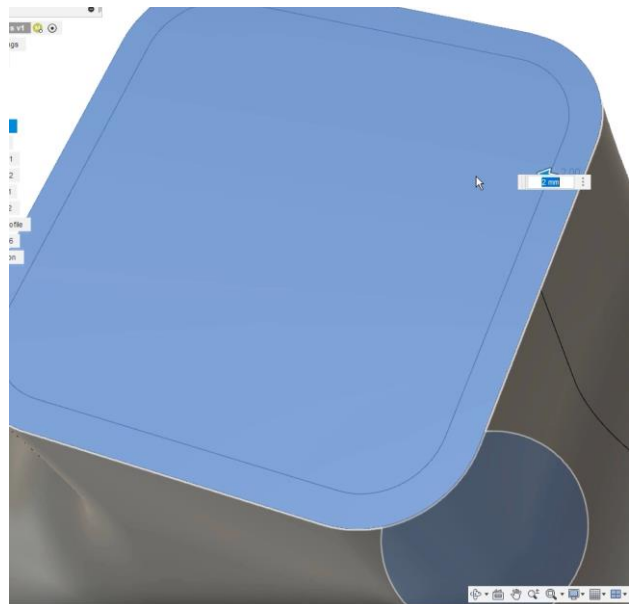
Figure 82. Reselect the faces

83. Rotate the body and inspect the result. Notice that the body maintains a consistent 2-millimeter wall thickness. Save the design.

Figure 83. Inspect the result