"Fusion 360" Lab

Lecture 4

Lebanese University - Faculty of Engineering – Branch 3

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Contents

Chap9: Additional Features and Operations

You can use numerous combinations of features to create a design. These can include standard sketched features and pick and place features. Drafts, splits, shells, ribs, and threads are advanced features that can be incorporated into a design to create geometry.

Learning Objectives in this Chapter

- Create a draft where the draft pull direction is normal to a selected plane or face.
- Create a shell feature that removes faces and assigns uniform wall thickness to the remaining faces in a design.
- Create a rib feature from a sketched section.
- Use the Split tool to split a face based on a split reference.
- Use the Scale tool to resize geometry in a design.
- Use the Thread tool to add threaded geometry to faces in a design.
- Use the Press Pull tool to efficiently create new features in a design.

9.1. Draft

A draft creates sloped surfaces and is often required to remove a part from a mold or casting. You can apply a draft over an entire profile, to one face, or to different faces, as shown in Figure 9–1.
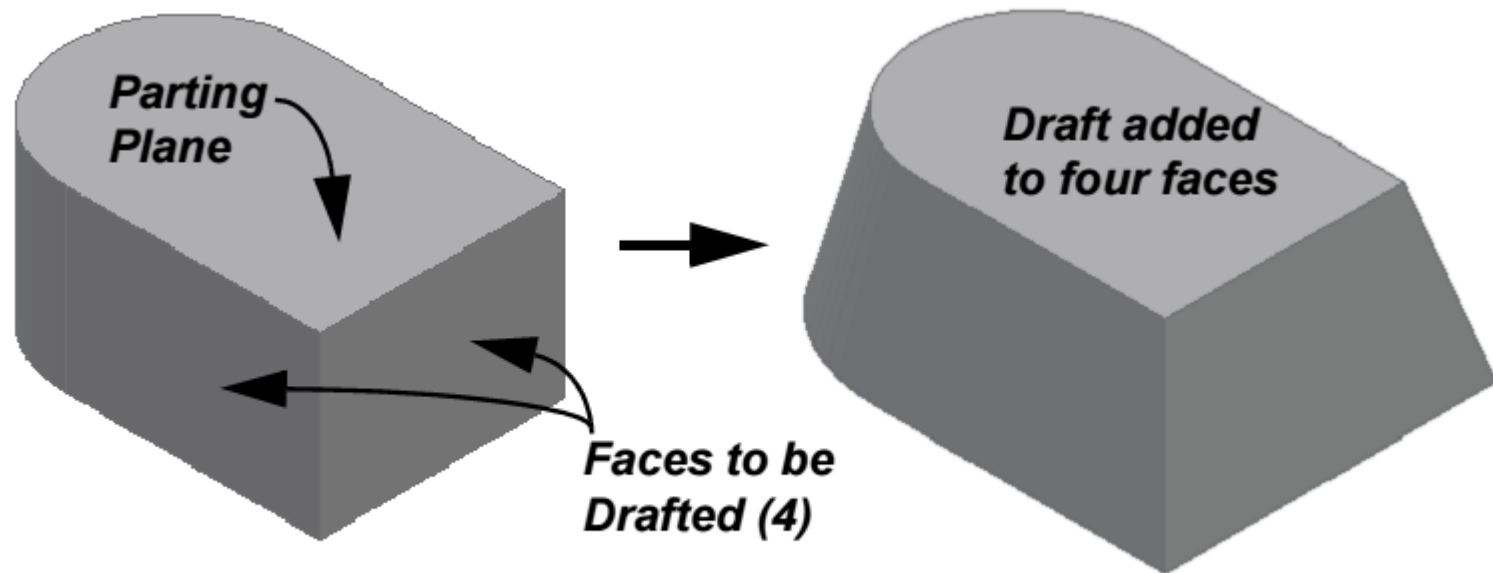

Figure 9–1

9.1. Draft

To create a split draft, you can use a parting plane, which is a construction plane that intersects the part, as shown in Figure 9–2.
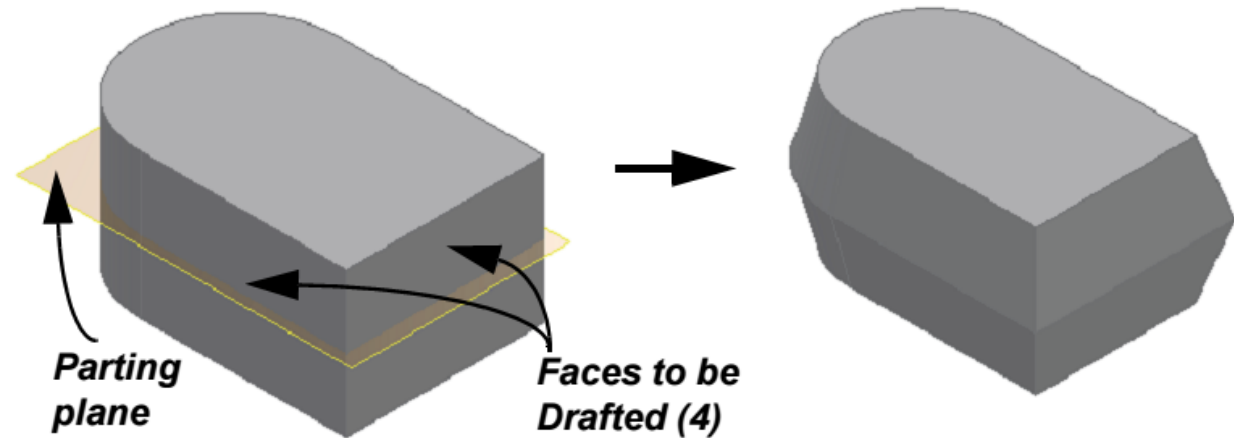

Parting plane

Faces to be Drafted (4)

Figure 9–2

Use either of the following methods to start the **Draft** tool:

- In the MODIFY panel, click (Draft).

- In the graphics window, right-click and select **Modify>Draft**.

Contents
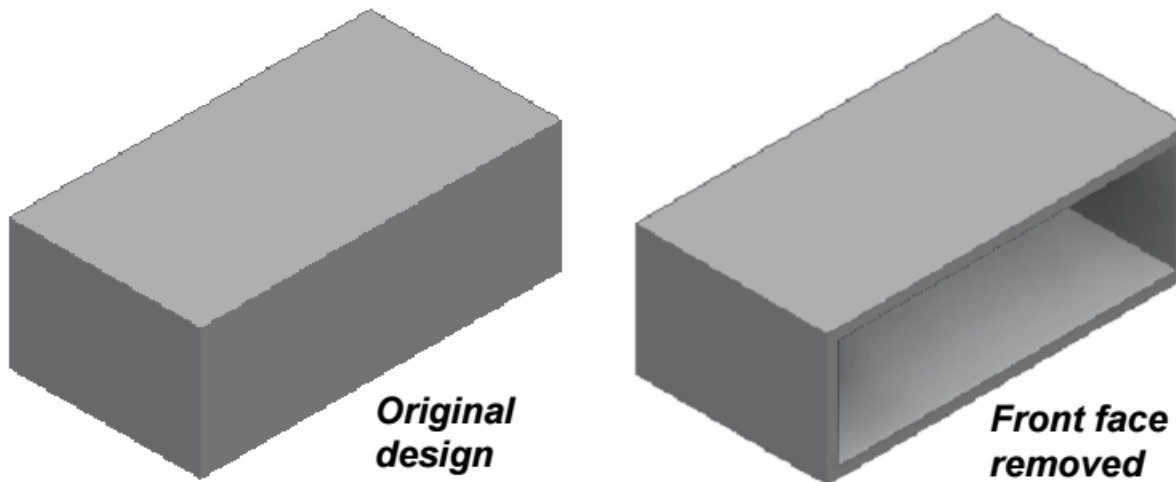
9.2. Shell

A shell operation hollows a solid, leaving a constant wall thickness. The design shown in Figure 9–3 has a shell feature added to it that removed the front face.

You can hollow a body without removing a face by selecting the body in the BROWSER.

Original design

Front face removed

Figure 9–3

Use either of the following methods to start the **Shell** tool:

- In the MODIFY panel, click [icon] (Shell).

- In the graphics window, right-click and select **Modify>Shell**.

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9.3. Rib

The **Rib** tool enables you to create solid extrusions with open geometry. The sketch in Figure 9–4 consists of a single line. In Figure 9–5, a rib feature was created so that the space up to the adjacent surfaces is filled across the thickness of the rib. In Figure 9–6, a depth is defined across the thickness of the rib so that there is a open space between it and the adjacent surfaces.
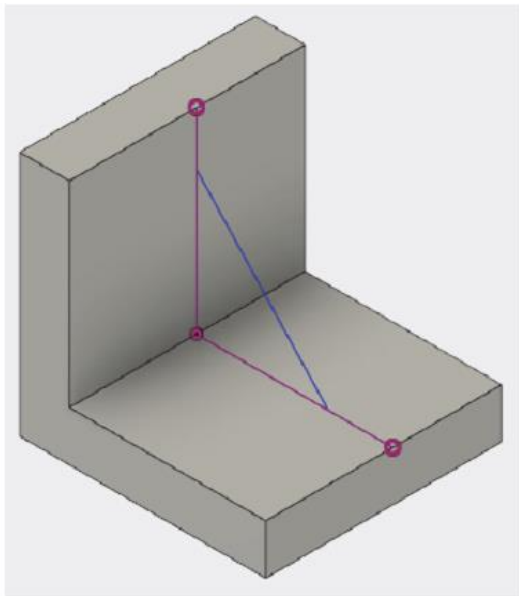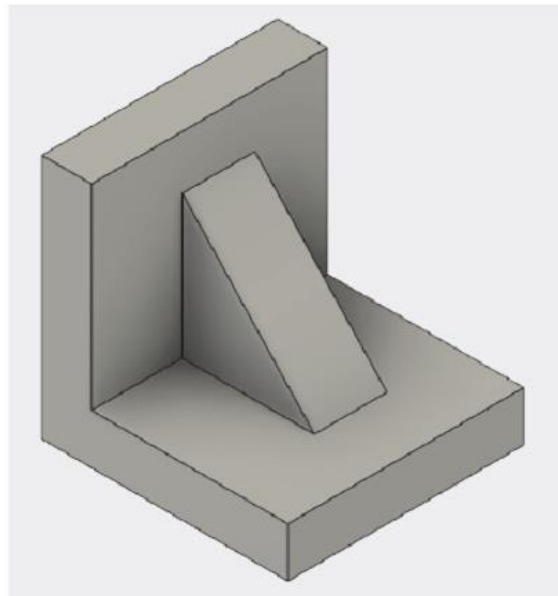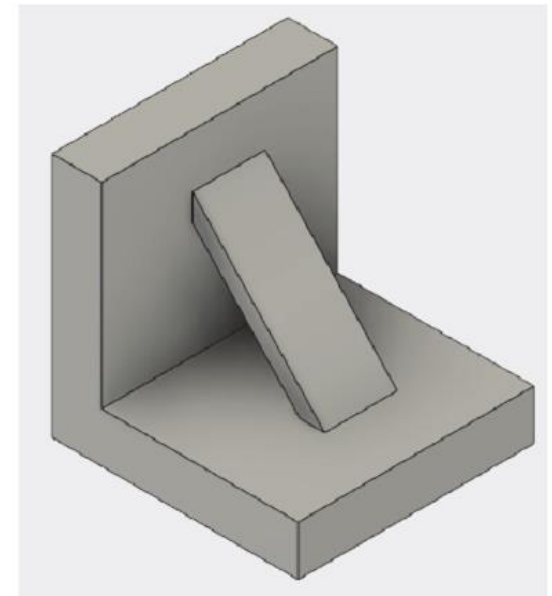

Figure 9–4

Figure 9–5

Figure 9–6

9.3. Rib

Use either of the following methods to start the **Rib** tool:

- In the CREATE panel, click (Rib).

- In the graphics window, right-click and select **Create>Rib**.

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9.4. Split face

You can split a face and manipulate each portion of the face independently. For example, a face can be split by a sketch so that a draft can be applied to one portion of it, as shown in Figure 9–7. You can use a face, construction plane, edge or sketch profile as the splitting tool.
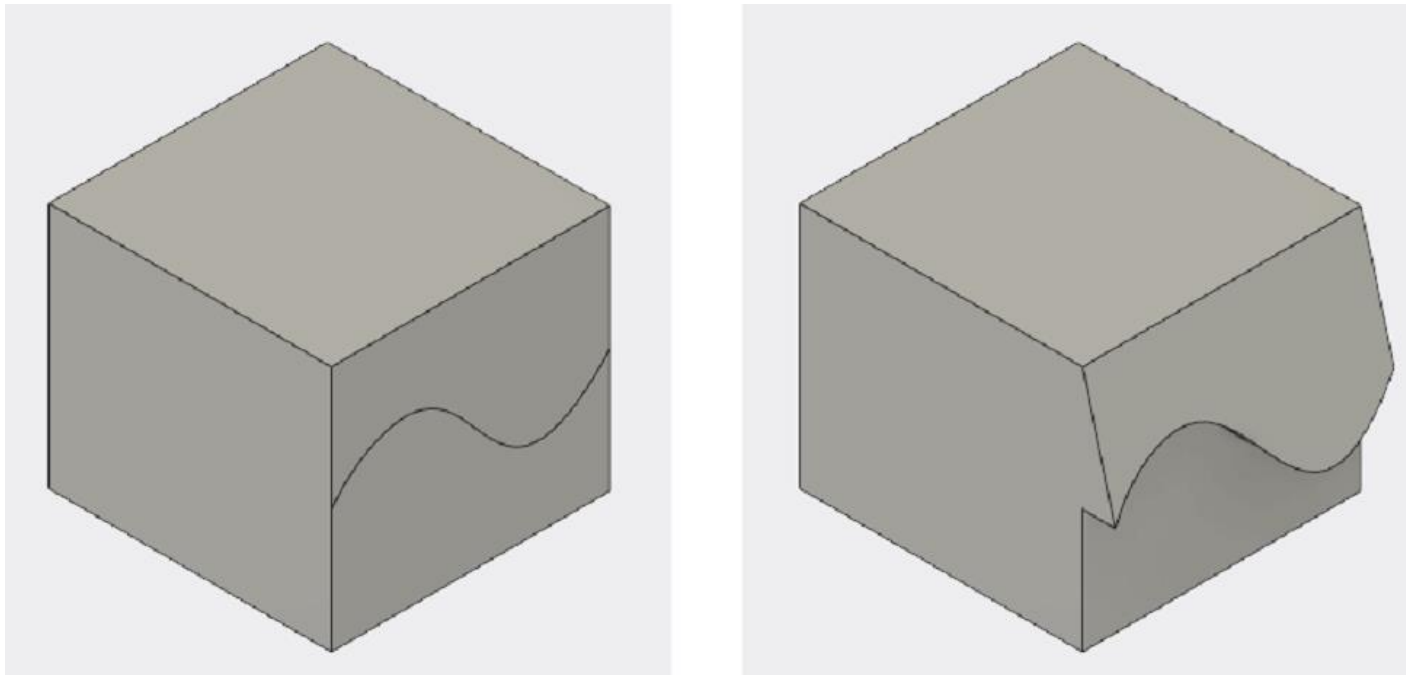
Figure 9–7

9.4. Split face

Use the following steps to split a face:

1. In the MODIFY panel, click (Split Face). Alternatively, in the graphics window, right-click and select **Modify>Split Face**. The SPLIT FACE palette opens as shown in Figure 9–8.
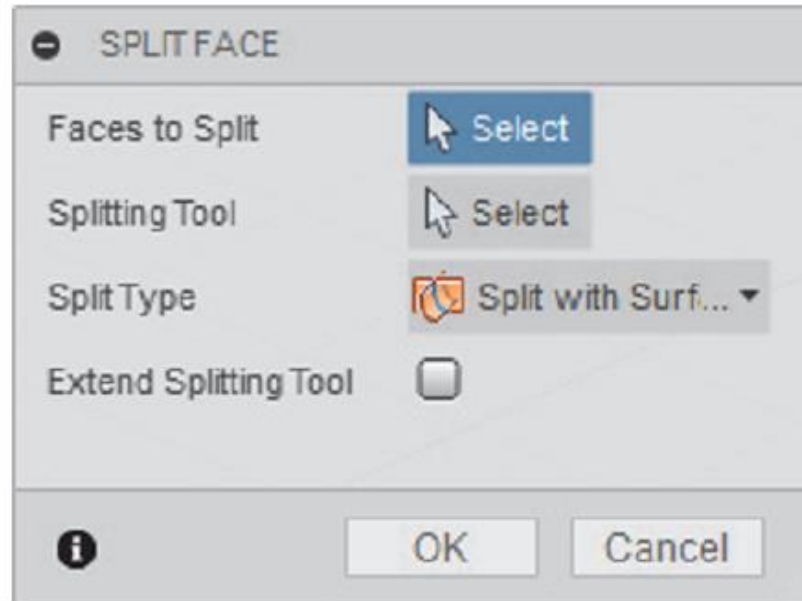

Figure 9–8

9.4. Split face

2. Select the face or faces you want to split.
3. Select the *Splitting Tool* field to activate it and select the splitting tool that will be used as the reference for splitting.

4. If you select a sketch or face as the Splitting Tool, you must select the *Split Type*. The options include:
   - **Split with Surface:** Splits the selected faces at the actual or extended intersection with the splitting tool reference.
   - **Along Vector:** Splits the selected faces along a defined vector relative to the splitting tool reference.
   - **Closest Point:** Splits the selected faces at the closest point to the splitting tool reference.

5. Use the **Extend Splitting Tool** option to extend the tool through the design, if required.
6. Click **OK**.

9.4. Split face

If the split with the selected faces does not generate a closed loop after the split, the split fails to create.

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9.5. Scale

You can use a scale operation to modify the size of sketch objects, bodies, or components. The scale can be done uniformly in all directions, or scaled independently in the design's X-, Y-, and Z-directions. A point is selected on the design from which the scaling will be applied.

Use either of the following methods to start the **Scale** tool:

- In the MODIFY panel, click (Scale).

- In the graphics window, right-click and select **Modify>Scale**.

Contents
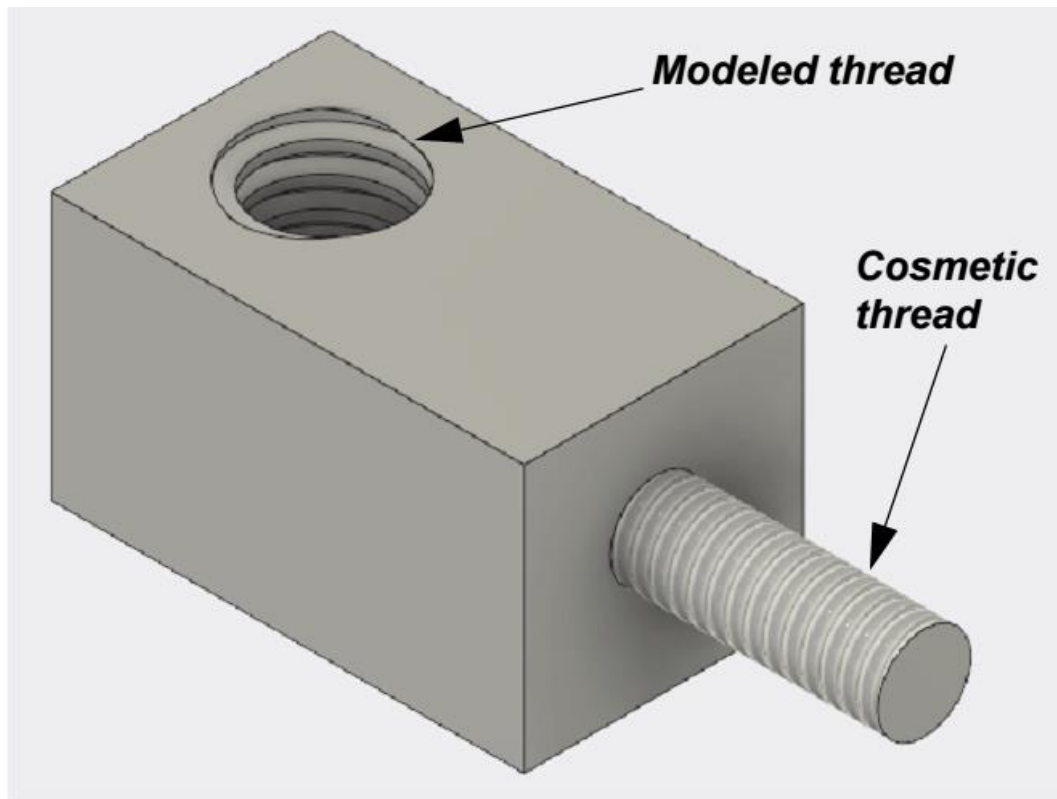
9.6. Thread

You can use the Thread tool to add a thread to cylindrical faces, as shown in Figure 9–9. The thread geometry can be modeled on the design, or it can be represented by a cosmetic feature.

Modeled thread

Cosmetic thread

Threaded geometry cannot be added to a conical face.

Figure 9–9

9.6. Thread

Use the following steps to create a thread feature:

1. In the CREATE panel, click (Thread). The THREAD palette opens as shown in Figure 9–10.
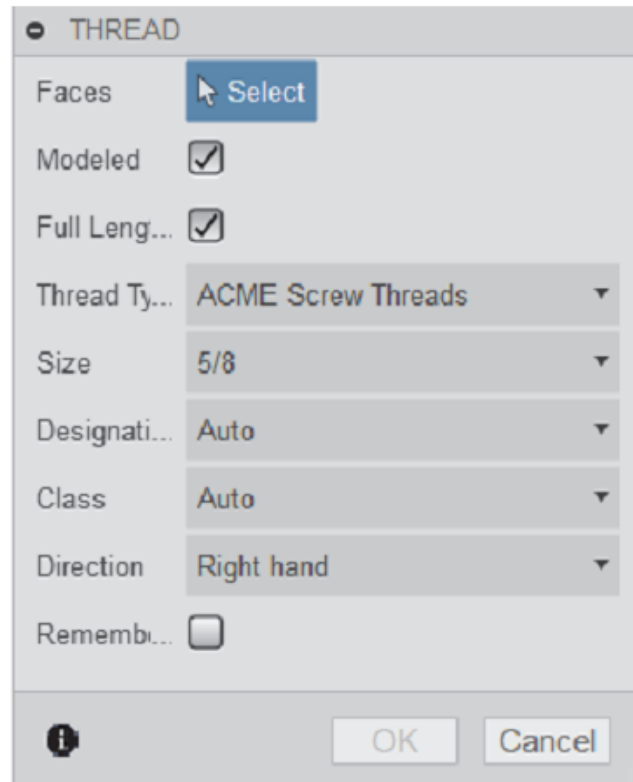
THREAD	
Faces	▶ Select
Modeled	☑
Full Leng...	☑
Thread Ty...	ACME Screw Threads ▼
Size	5/8 ▼
Designati...	Auto ▼
Class	Auto ▼
Direction	Right hand ▼
Rememb...	☐
❶	OK Cancel

Figure 9–10

2. Select a cylindrical face on which to create the thread. Note that a thread can only be created on one face at a time.

3. Define the thread using the following options:

   - Select **Modeled** to create the thread geometry. Clear the option to create a cosmetic version.

   - Select **Full Length** to set the thread length to cover the entire length of the selected face. Clear the **Full Length** option to set the thread length. A non-full length thread starts from one end or at the required *Offset* value. Enter a value to define the *Length* of the thread.

9.6. Thread

4. Set the thread specifications. The options are as follows:

Thread Type	Sets the series of threads to apply.
Size	Sets the diameter size of the selected thread type.
Designation	Sets the number of threads based on the unit of length and bolt size.
Class	Sets the class of threads.
Direction	Sets the direction (right-hand or left-hand,) in which the threads should be applied.

5. Select **Remember** to reuse the same size for the next thread feature.

6. Click **OK** to complete the thread

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9.7. Press Pull

Press Pull is a tool that adapts the feature type based on the reference entity you select. You can use Press Pull to offset faces, extrude sketched profiles, and create fillets on edges, as shown in Figure 9–11.

Faces do not need to be planar.

Select face and create offset

30.00 mm

9.7. Press Pull

Select profile and create extrude

30.00 mm ▼
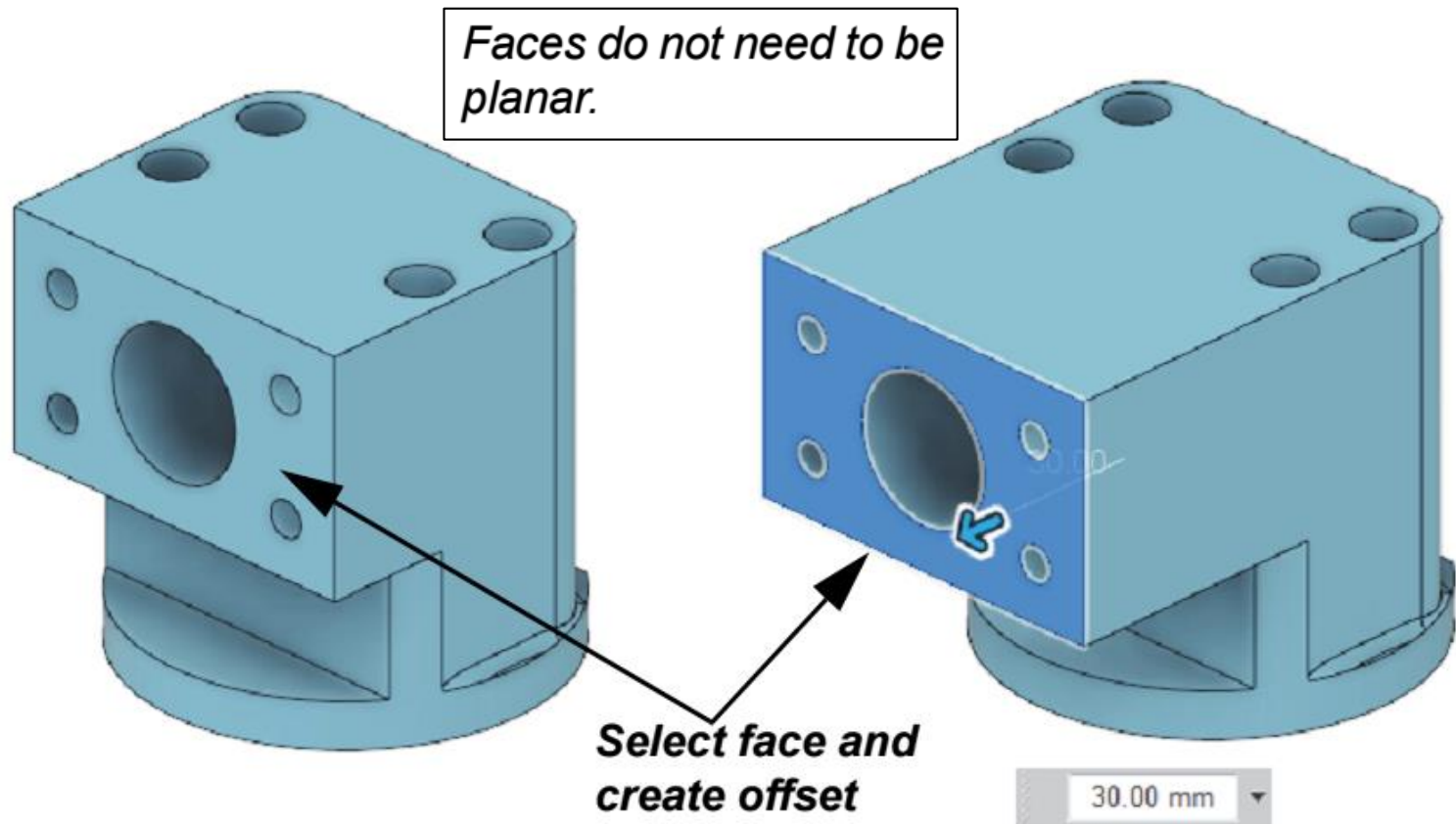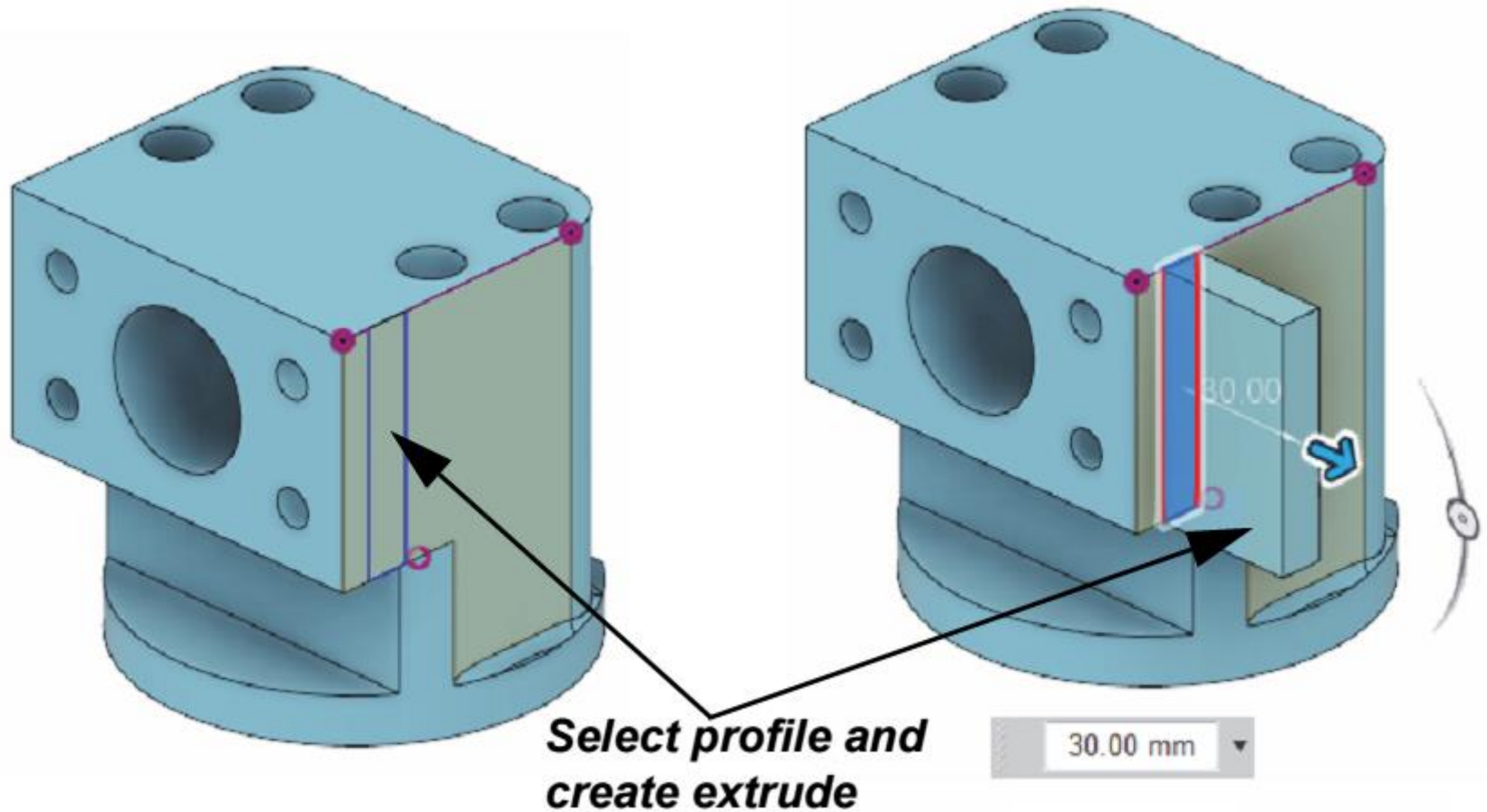
Offsets and Extrudes created with Press Pull can add or remove material.

Select edge and create fillet

2.00 mm
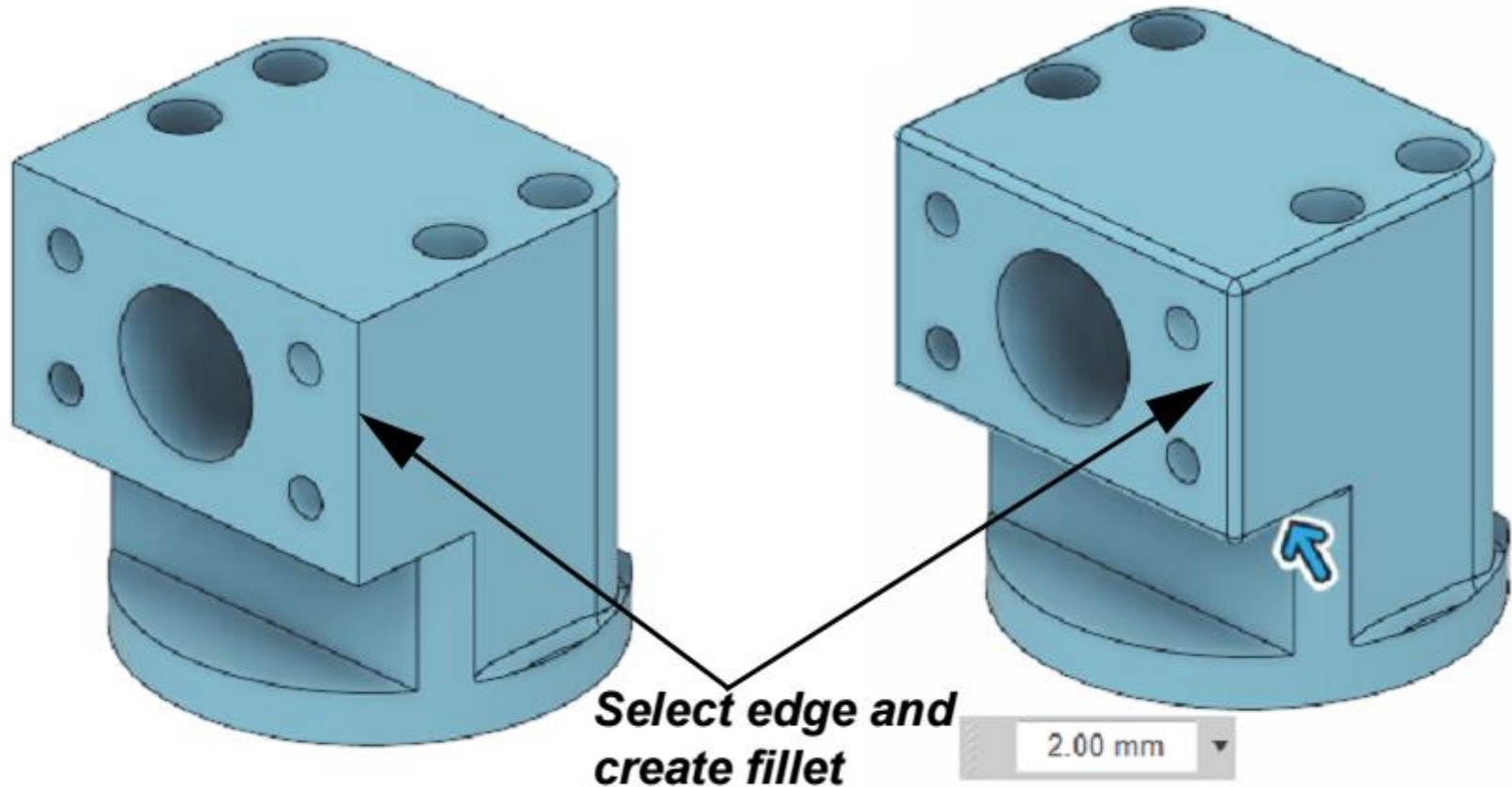
Figure 9–11

9.7. Press Pull

When you select faces for use with Press Pull, you can choose between creating a new offset or modifying the existing feature that the face belongs to.

Use one of the following methods to start the **Press Pull** tool:

- In the MODIFY panel, click (Press Pull).

- In the graphics window, right-click and in the marking menu, select **Press Pull**.

- In the graphics window, right-click and select **Modify>Press Pull**.

- Press <Q>.

Command Summary

Button	Command	Location
	Draft	• **Ribbon:** *Model* Workspace>MODIFY panel • **Context Menu**: Right-click in the graphics window and select **Modify**.
	Press Pull	• **Ribbon:** *Model* Workspace>MODIFY panel • **Context Menu**: Right-click in the graphics window and select **Modify**. • **Marking Menu**: Right-click in the graphics window.
	Rib	• **Ribbon:** *Model* Workspace>CREATE panel • **Context Menu**: Right-click in the graphics window and select **Create**.

Command Summary

	Scale	• **Ribbon:** *Model* Workspace>MODIFY panel • **Context Menu**: Right-click in the graphics window and select **Modify**.
	Shell	• **Ribbon:** *Model* Workspace>MODIFY panel • **Context Menu**: Right-click in the graphics window and select **Modify**.
	Split Face	• **Ribbon:** *Model* Workspace>MODIFY panel • **Context Menu**: Right-click in the graphics window and select **Modify**.
	Thread	• **Ribbon:** *Model* Workspace>CREATE panel • **Context Menu**: Right-click in the graphics window and select **Create**.

Chap 9
Practice

Creating Shells and Ribs

Practice Objectives

- Create a Shell feature that removes required geometry from the design.
- Create a Rib feature from a sketched section.

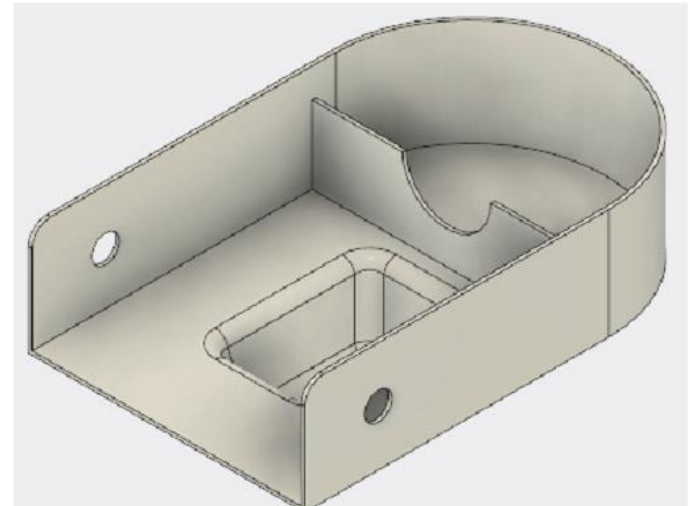In this practice, you will add a Shell and a Rib feature to create the geometry shown in Figure 9–12.

Figure 9–12

Practice 1

Task 1 - Create a new design from file.

1. Click > **Open**. In the Open window, click **Open from my computer**.

2. In the Open dialog box, navigate to the *C:\Autodesk Fusion 360 Practice Files* folder, select **shell.f3d**, and click **Open**. The design displays as shown in Figure 9–13.
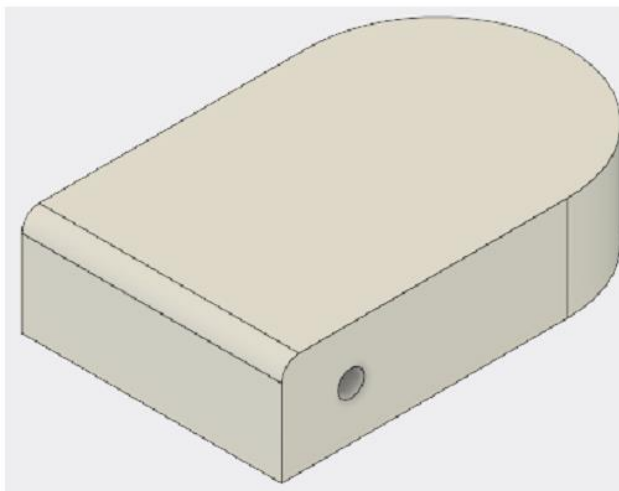
Figure 9–13

Practice 1

Task 2 - Shell the design.

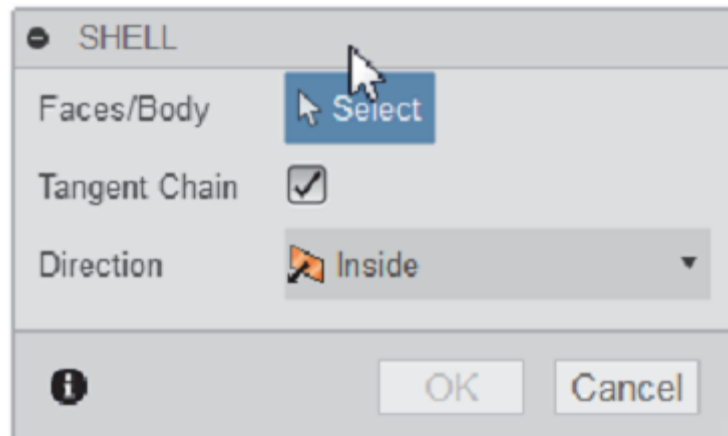1. In the MODIFY panel, click ⬛ (Shell). The SHELL palette opens, as shown in Figure 9–14.

Figure 9–14

2. Ensure that the **Tangent Chain** option is selected and then select the face shown in Figure 9–15 as the face to remove.

Select this face to remove

0.00 mm
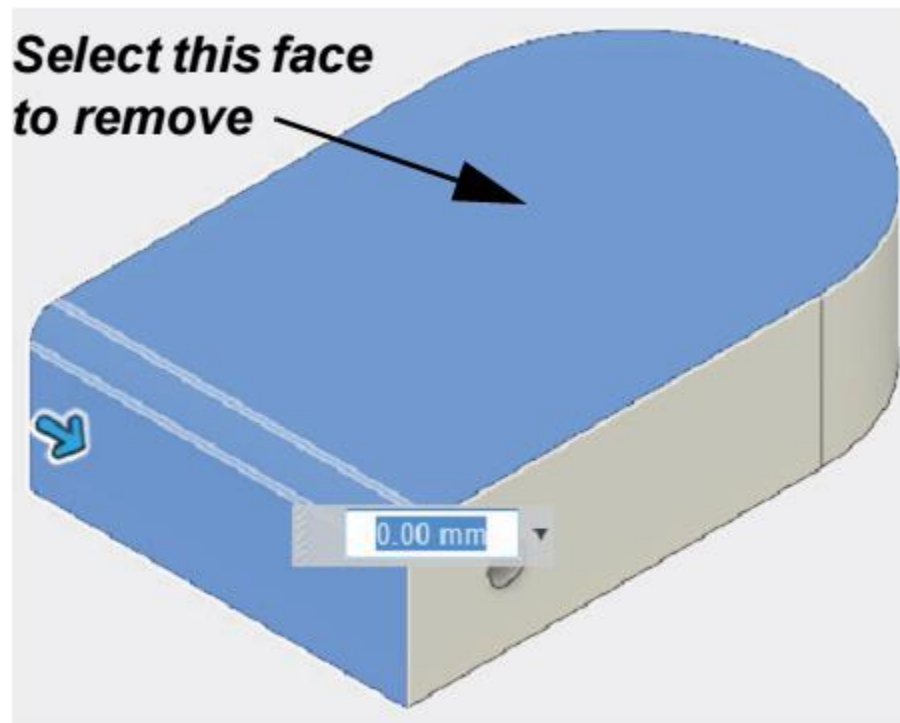
Figure 9–15

Practice 1

3. Ensure that the *Direction* option is set to **Inside**. Set the *Inside Thickness* value to **1**.

4. Click **OK** to complete the shell. The design displays as shown in Figure 9–16. The hole in the design did not shell as expected because its face was not selected for removal.
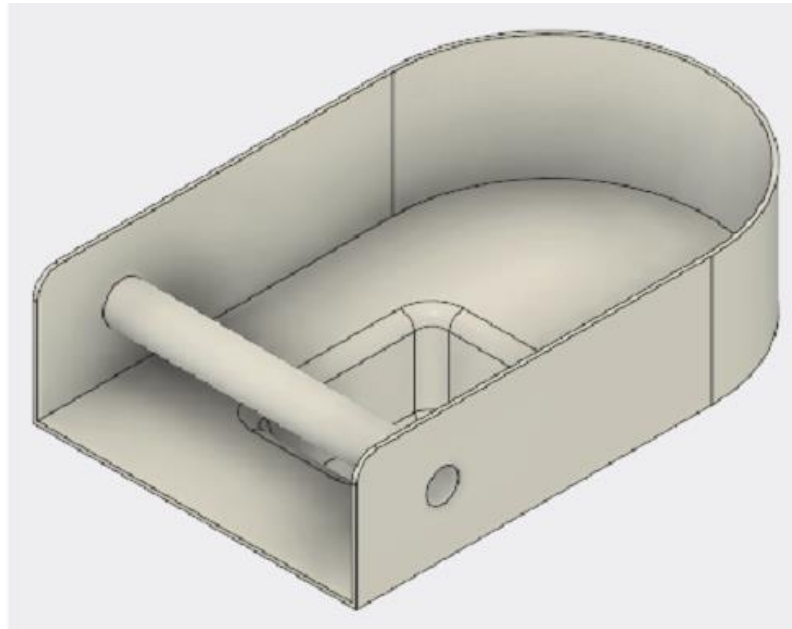
Figure 9–16

5. Edit the **Shell**. Hold <Ctrl> and select the surface of the hole to be removed, as shown in Figure 9–17.

Specify thickness, or hold Ctrl/CMD to modify selections
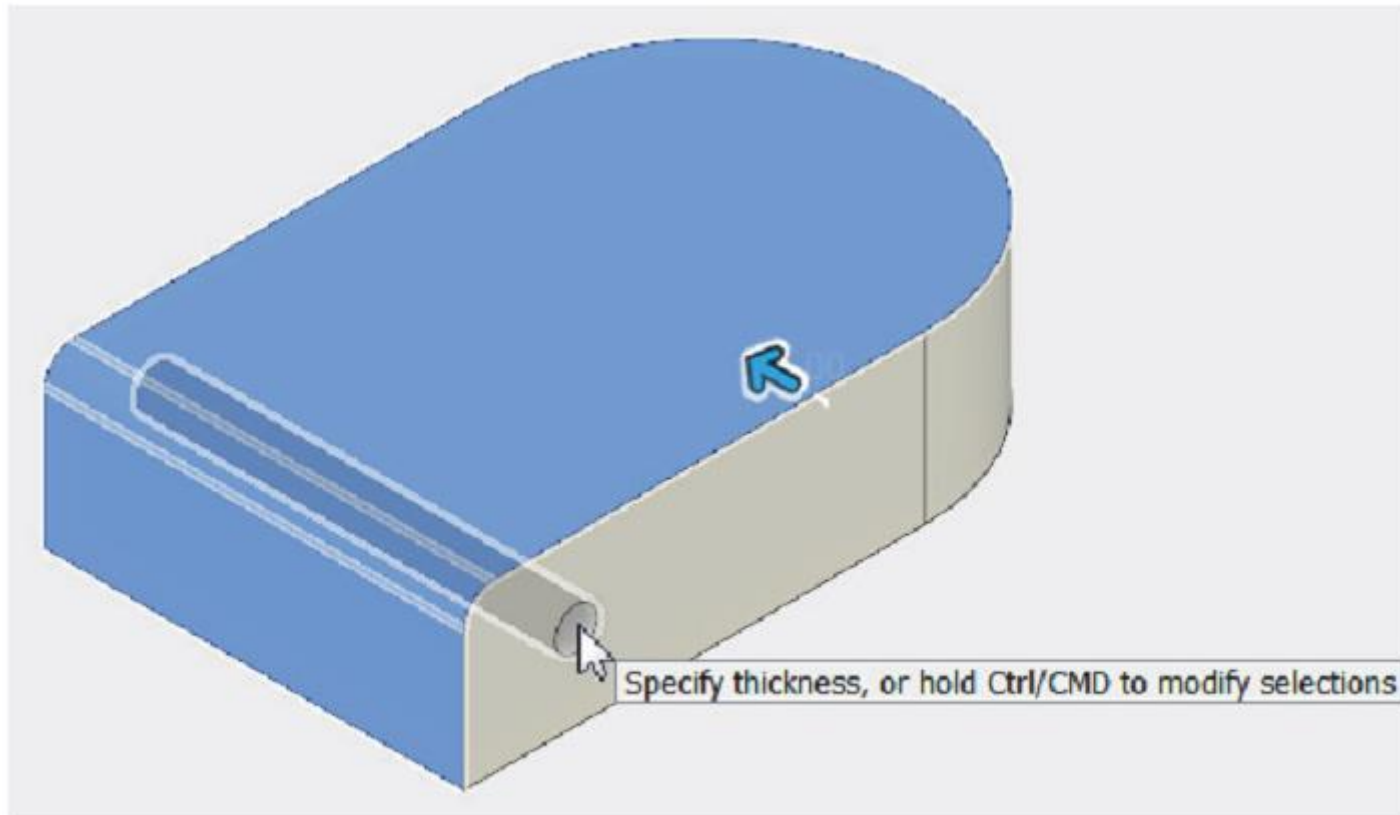
Figure 9–17

6. Complete the shell. The design displays as shown in Figure 9–18.
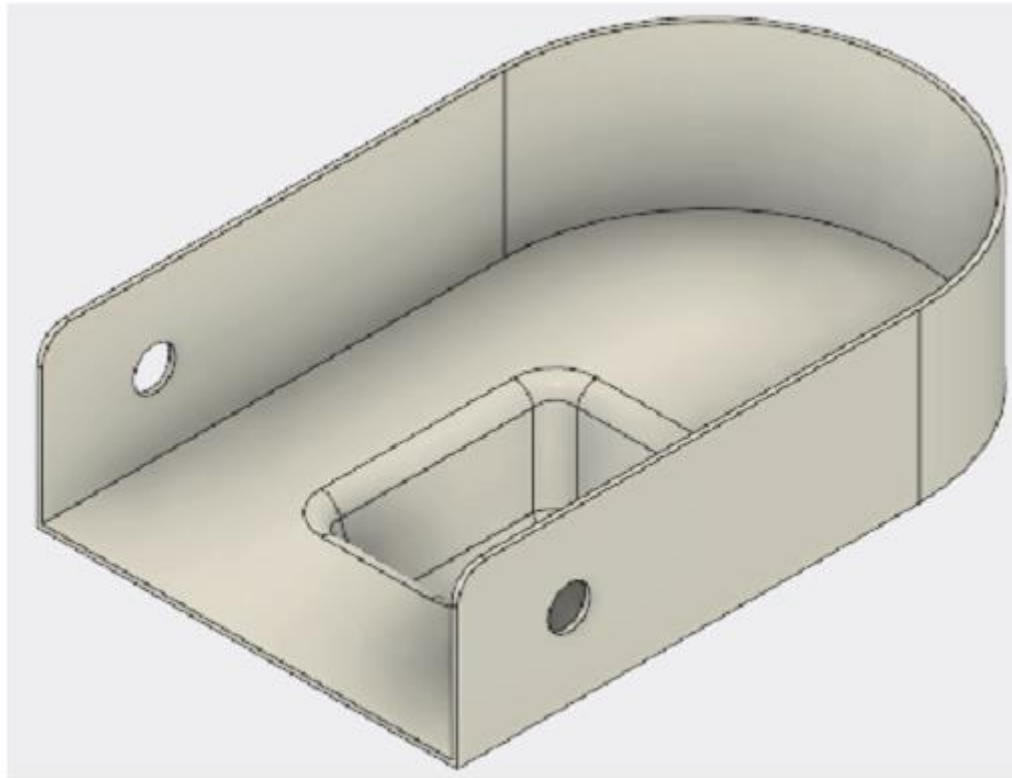
Figure 9–18

Practice 1

Task 3 - Create a rib.

1. In the BROWSER, expand the *Sketches* folder and toggle on the display of **Sketch1**. This sketch will be used to create a rib.

2. In the CREATE panel, click (Rib).

3. Select **Sketch1** as the curve for the rib feature.

4. For the *Thickness Option,* ensure that (Symmetric) is selected.

5. For the *Depth Options,* select (To Next).

> Ensure that the display of the Sketches folder is also on.

6. Enter a *Thickness* value of **1.00**. The palette and design display as shown in Figure 9–19. If the rib is created in the

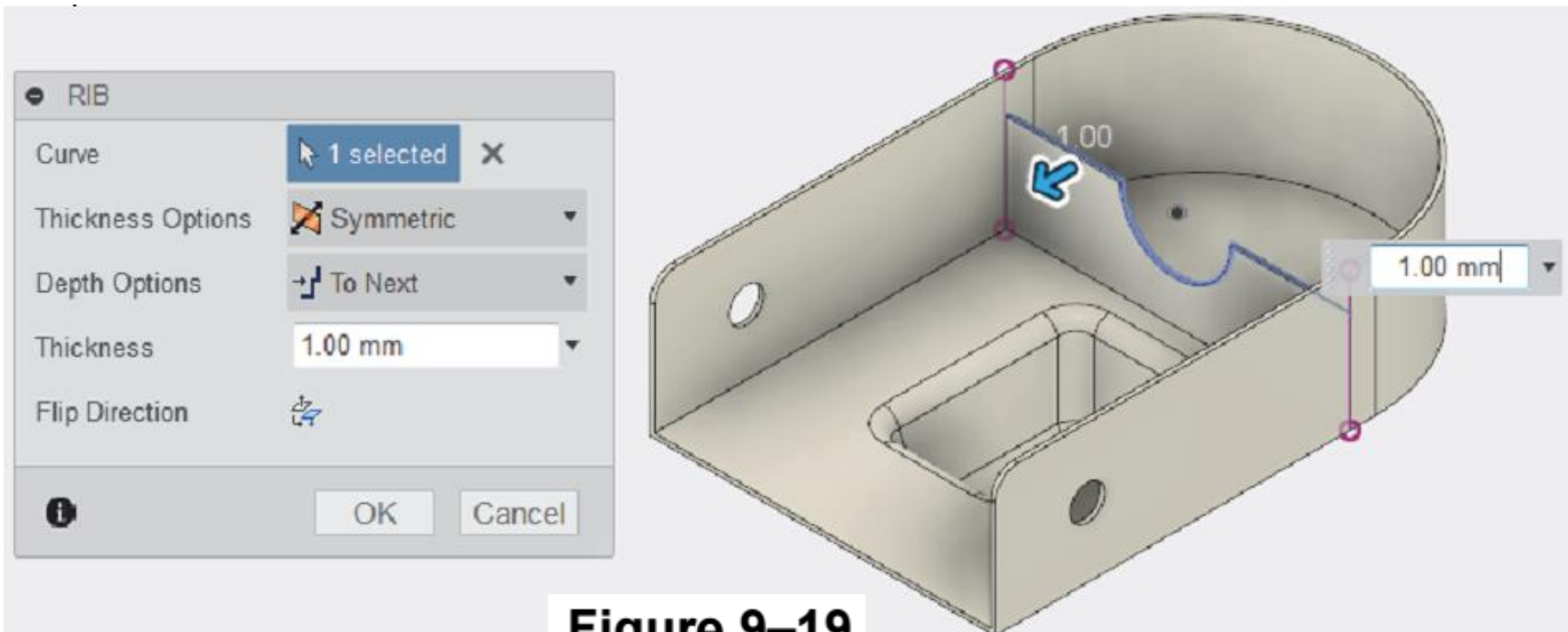   wrong direction, in the RIB palette, select [icon] to flip the direction.

Figure 9–19

7. Complete the rib. The design displays as shown in Figure 9–20.
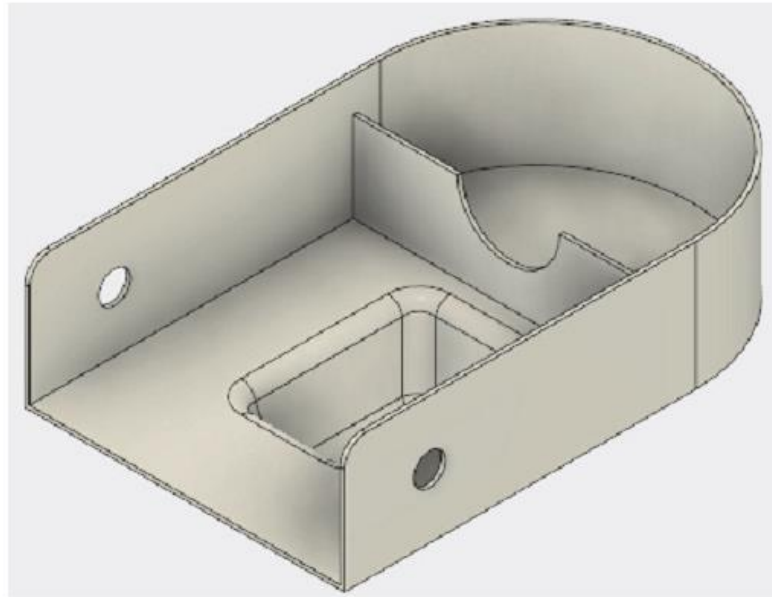

Figure 9–20

8. Save the design with the name **shell** to your *Autodesk Fusion 360 Practice Files* project.

9. Close the file.

Using Advanced Design Tools

Practice Objectives

- Create a Split feature that splits a selected face based on a sketched profile.
- Create a Draft feature to add draft to a selected face in the design.
- Create an Extrude and Fillet using the Press Pull tool.
- Create a threaded hole in the design.
- Shell the design.

In this practice, you will use various tools to create the design shown in Figure 9–21, including the **Split Face**, **Draft**, **Press Pull**, **Hole**, **Thread**, and **Shell** tools.

Practice 2

Figure 9–21

Task 1 - Create the first solid feature.

1. Start a new design with the *Unit Type* set to **Inch**.

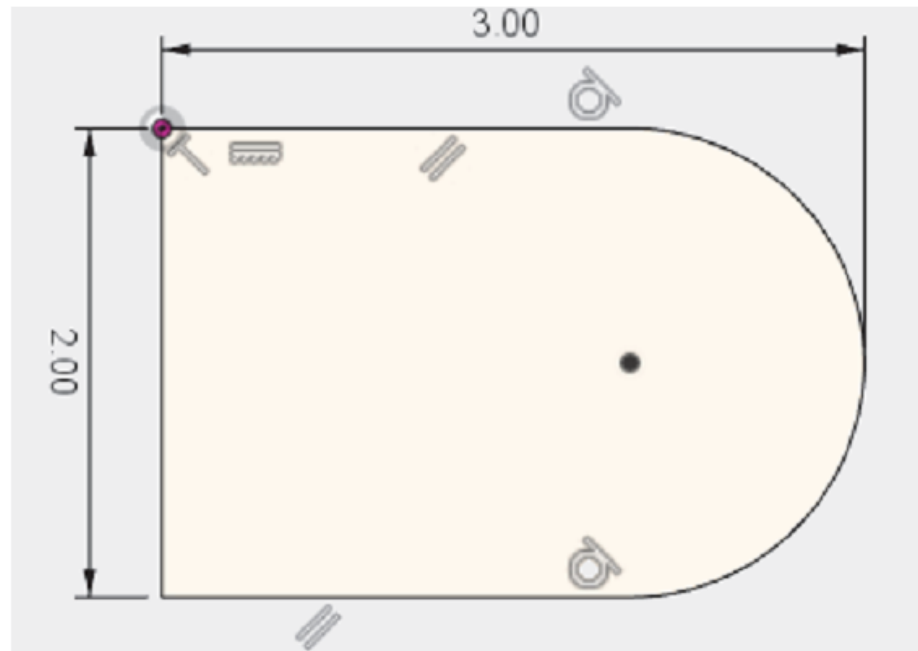2. On the XZ Origin plane, create the sketch shown in Figure 9–22.

Figure 9–22

3. Stop the sketch.

4. In the MODIFY panel, click [icon] (Press Pull).

5. Select the sketched profile. Because a profile is selected, the software recognizes that an Extrude is intended, so the EXTRUDE palette replaces the PRESS PULL palette.

6. Set the extruded *Distance* value to **2.00 in**.

7. Set the *Taper Angle* to **-2.0 deg**. The taper angle can be used in place of a separate draft feature on extruded geometry.

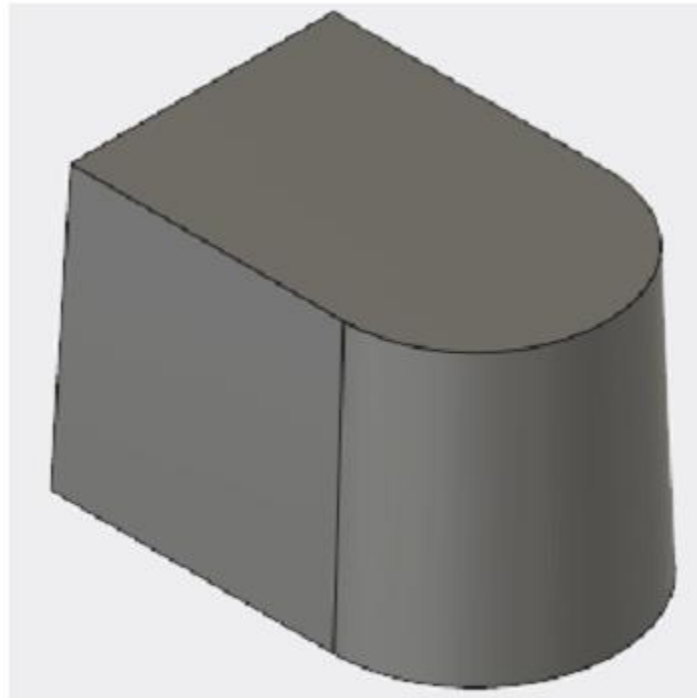8. Complete the feature. The part should display as shown in Figure 9–23.

Figure 9–23

Task 2 - Create a sketch that will be used to split a face.

1. Create a new sketch on the top face of the model (the face that contains an arced edge). Sketch, constrain, and dimension the profile for the split as shown in Figure 9–24.
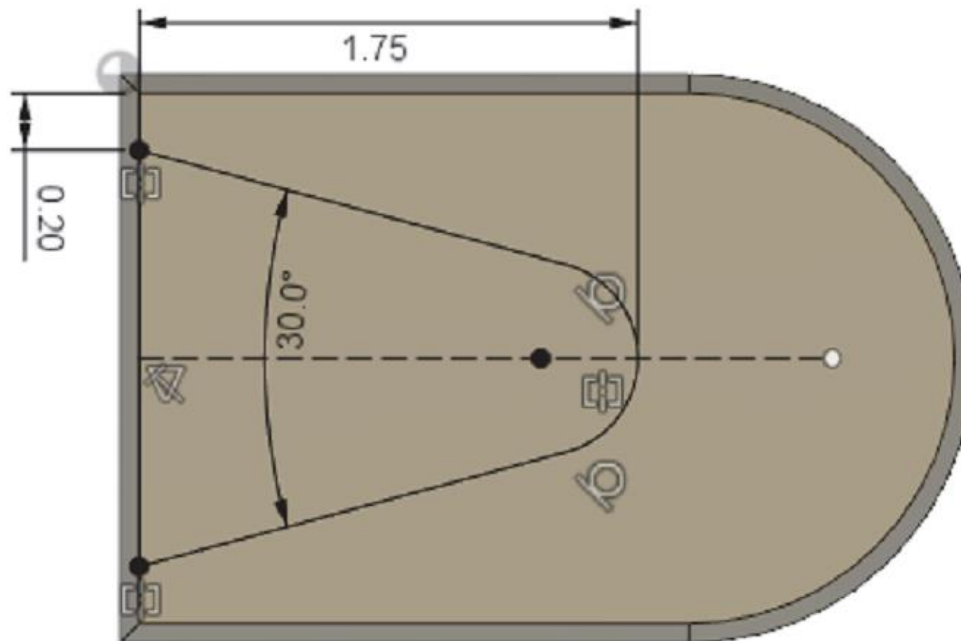

Figure 9–24

2. Stop the sketch.

Task 3 - Create the split.

1. In the MODIFY panel, click ![Split Face icon] (Split Face). The SPLIT FACE palette opens as shown in Figure 9–25.
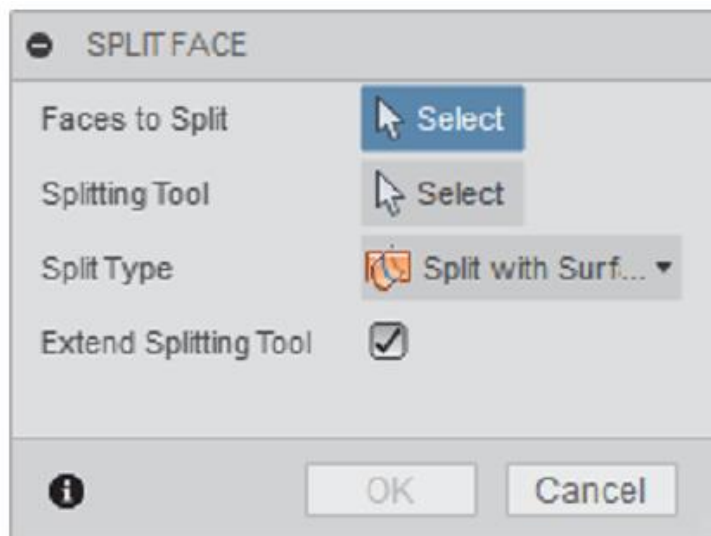

Figure 9–25

Practice 2

2. Select the face that you sketched on as the face to split.

3. Select the *Splitting Tool* field to activate it in the palette and select the sketch, as shown in Figure 9–26.

4. Clear the **Extend Splitting Tool** checkbox, if selected. A temporary surface is displayed to show the extension of the splitting tool and indicates how the selected face will be split.
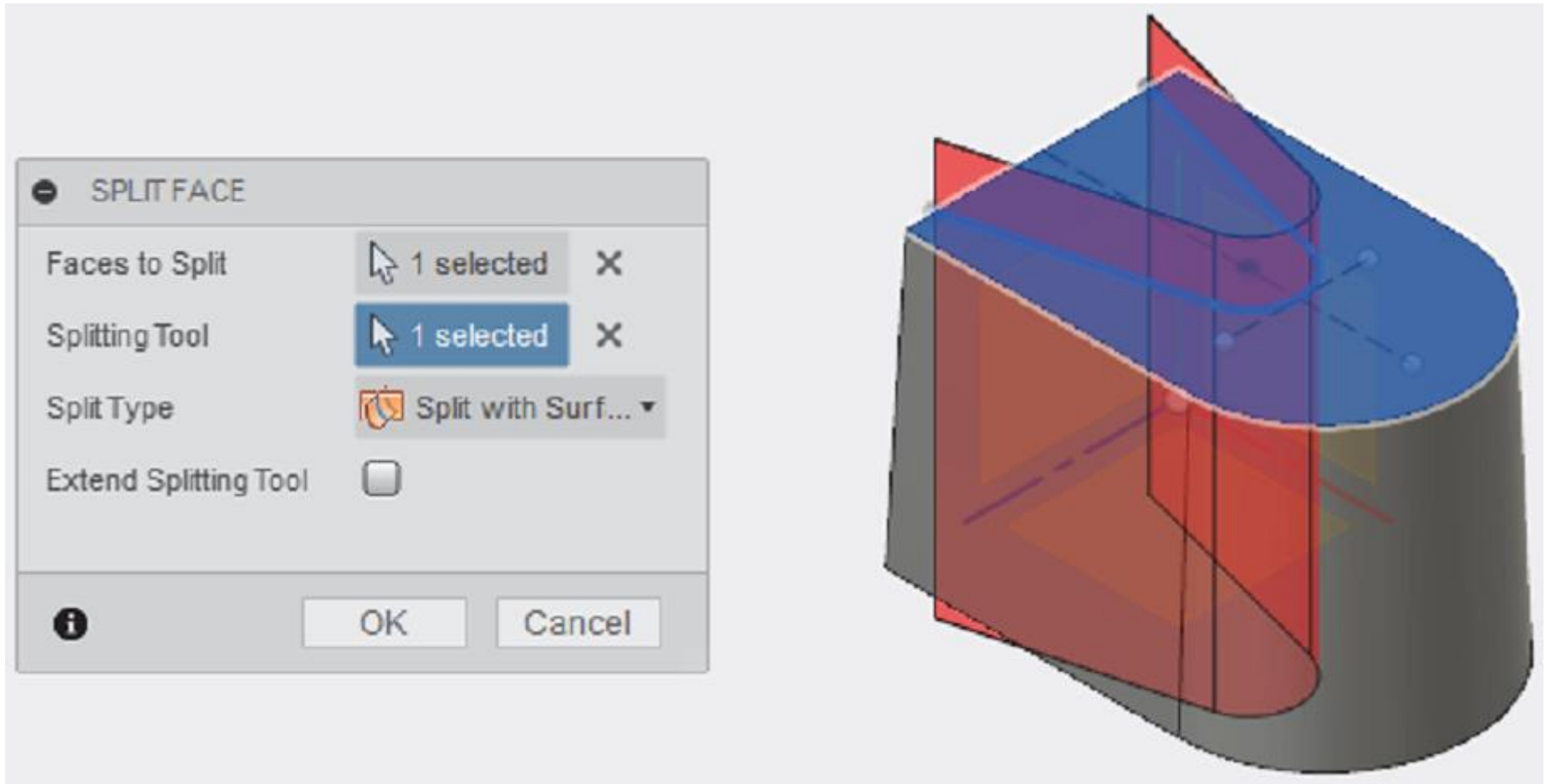
Figure 9–26

5. Ensure that the default *Split Type* of **Split with Surface** is selected.

6. Click **OK** to split the face.

7. Hover the cursor over the split faces. They highlight individually (below the curve, and above the curve), as shown in Figure 9–27.
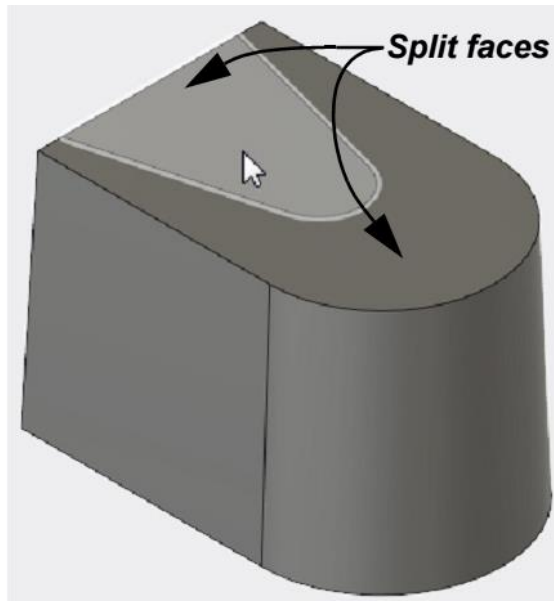

Figure 9–27

Practice 2

Task 4 - Create draft faces.

1. In the MODIFY panel, click [icon] (Draft). The DRAFT palette opens.

2. First you need to select the neutral plane from where the draft angle will be measured. Select the YZ Origin plane.

3. Clear the **Tangent Chain** checkbox and select the portion of the split face highlighted in Figure 9–28.
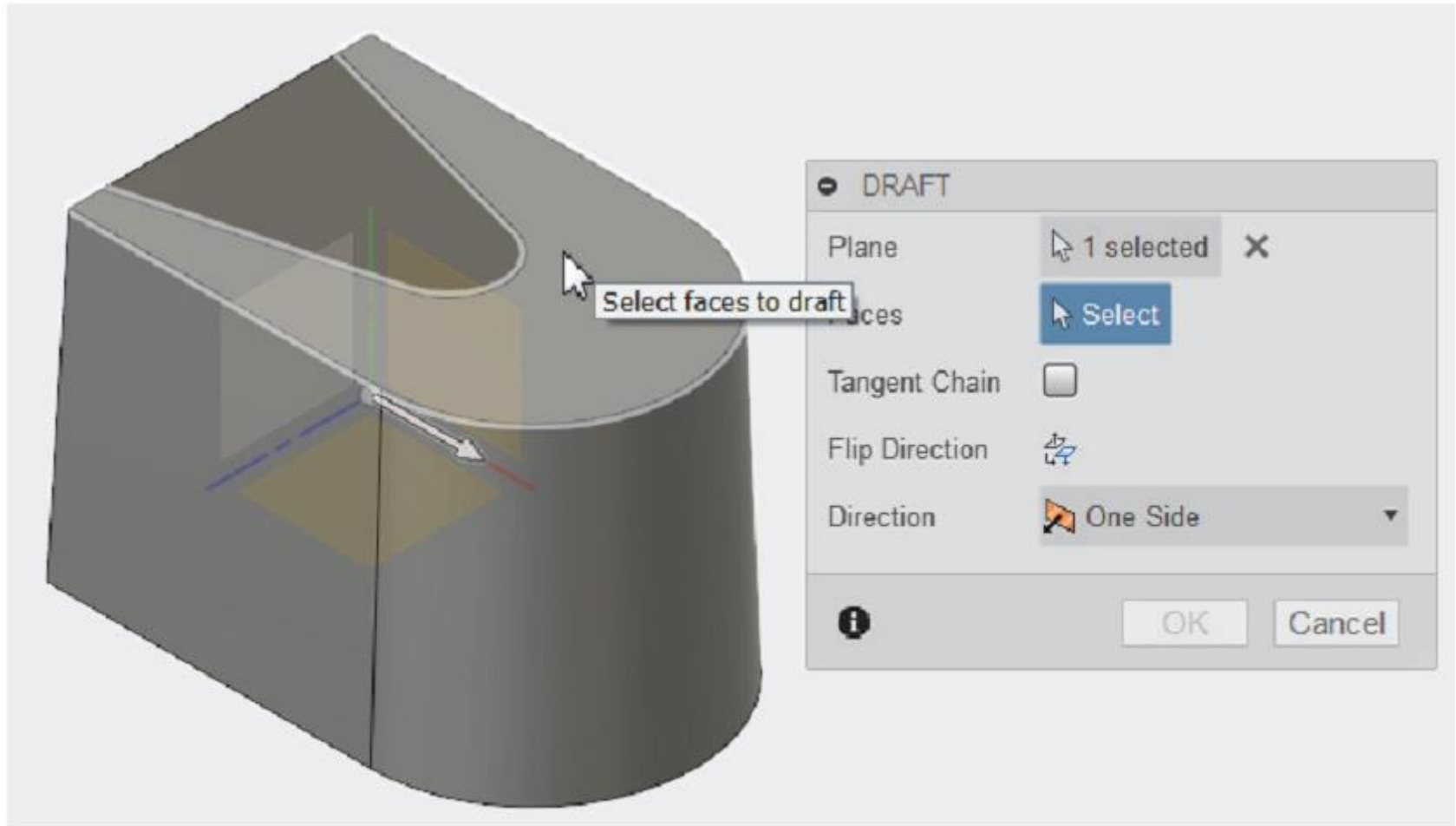
Practice 2

Figure 9–28

4. Set the angle to **10** degrees. If required, flip the direction.

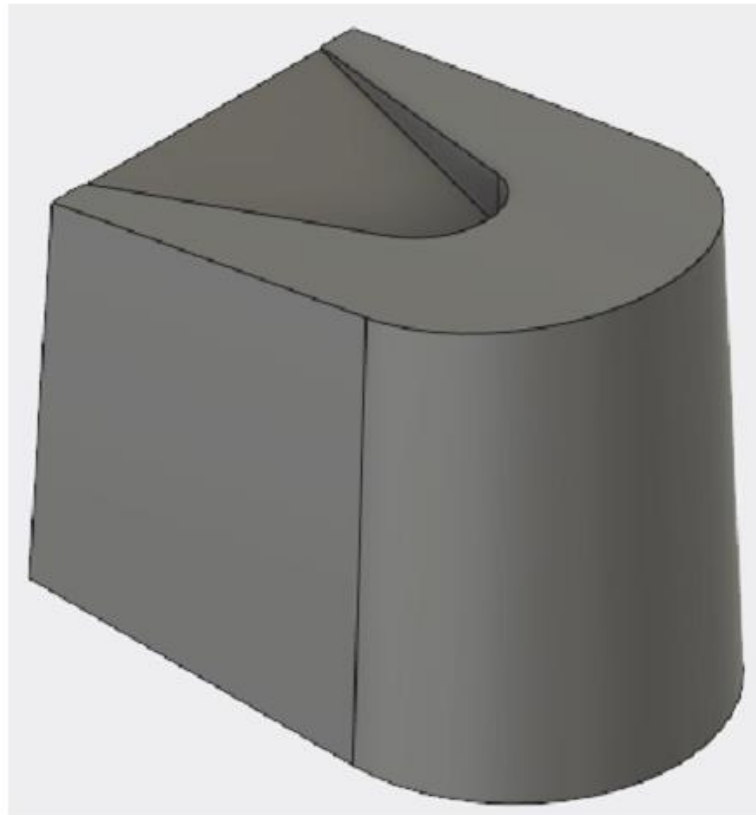5. Click **OK** to draft the face. The design displays as shown in Figure 9–29.

Figure 9–29

Task 5 - Create fillets using Press Pull.

1. Select the two edges highlighted in Figure 9–30 and press <Q> to activate the **Press Pull** tool. Because edges have been pre-selected, the **Press Pull** tool recognizes that the intent is to create fillets.
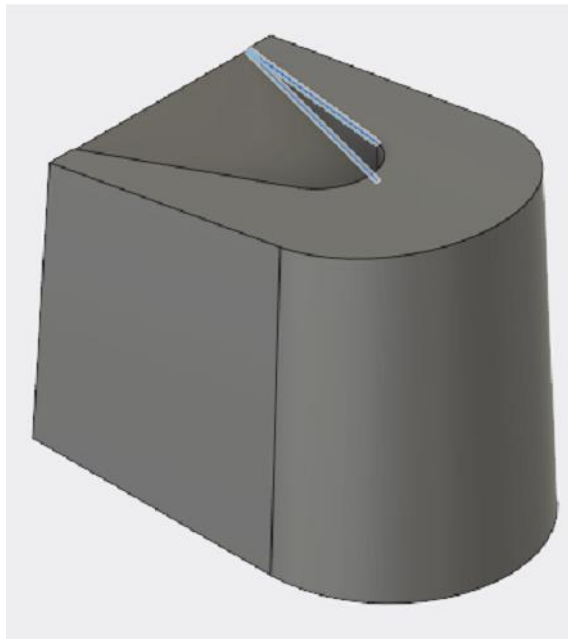

Figure 9–30

2. For the radius, enter **0.0625 in**. Ensure that the **Tangent Chain** option is selected and complete the feature.

3. Create a fillet with a radius of **0.125** in on the edge shown in Figure 9–31.
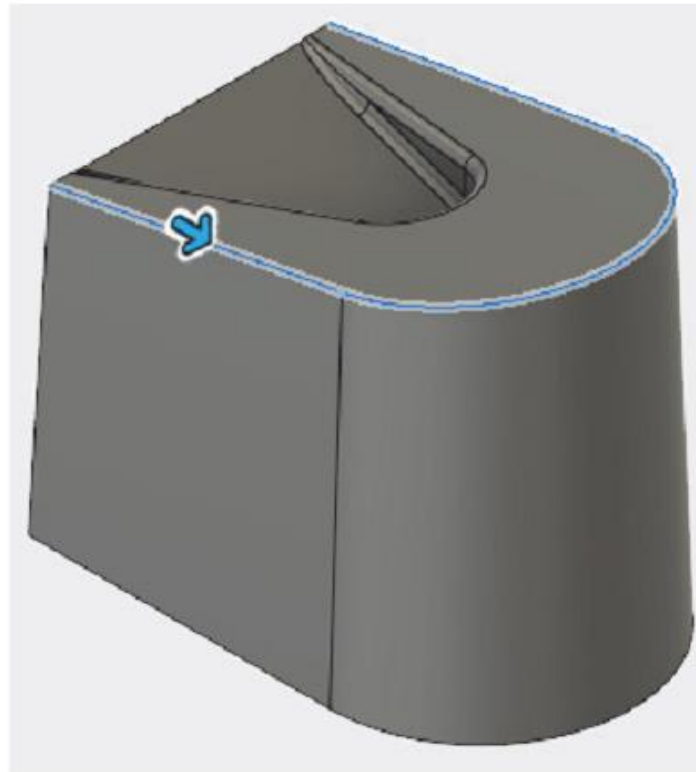

Figure 9–31

4. Create a fillet with a radius of **0.125 in** on the edge shown in Figure 9–32. The part should display as shown in Figure 9–33.
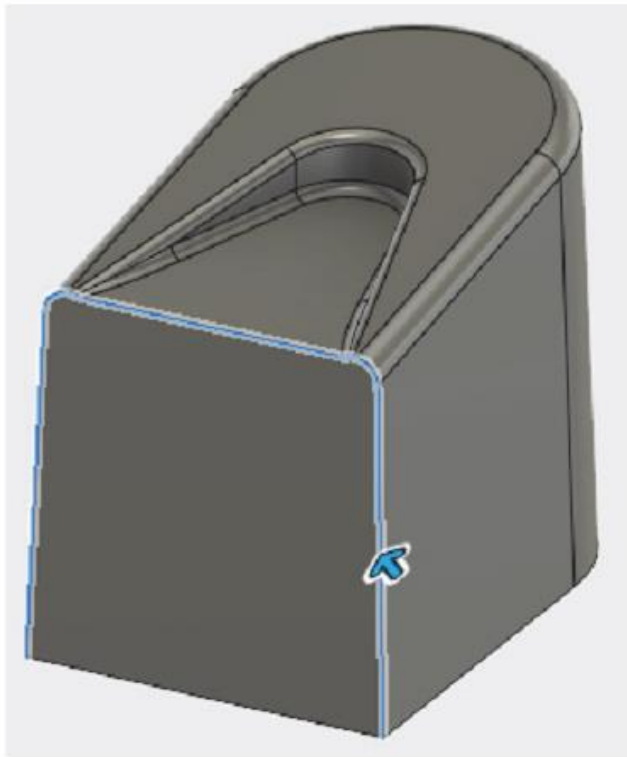

Figure 9–32

Figure 9–33

Task 6 - Create a hole, a shell, and a thread.

1. Orient the part as shown in Figure 9–34 and select the indicated face to place a hole. Position the hole on the face so that it snaps to the center of the circular edge.
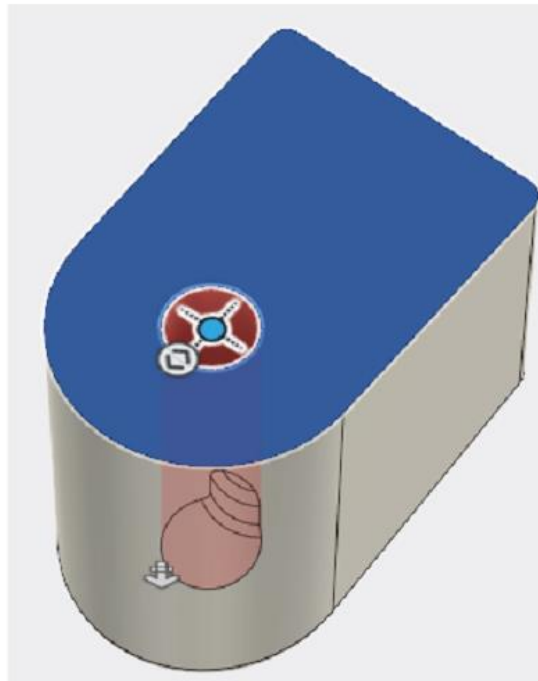

Figure 9–34

2. Set the *Diameter* value to **0.25 in** and the *Extents* option to **All**. Complete the feature. The design displays as shown in Figure 9–35.
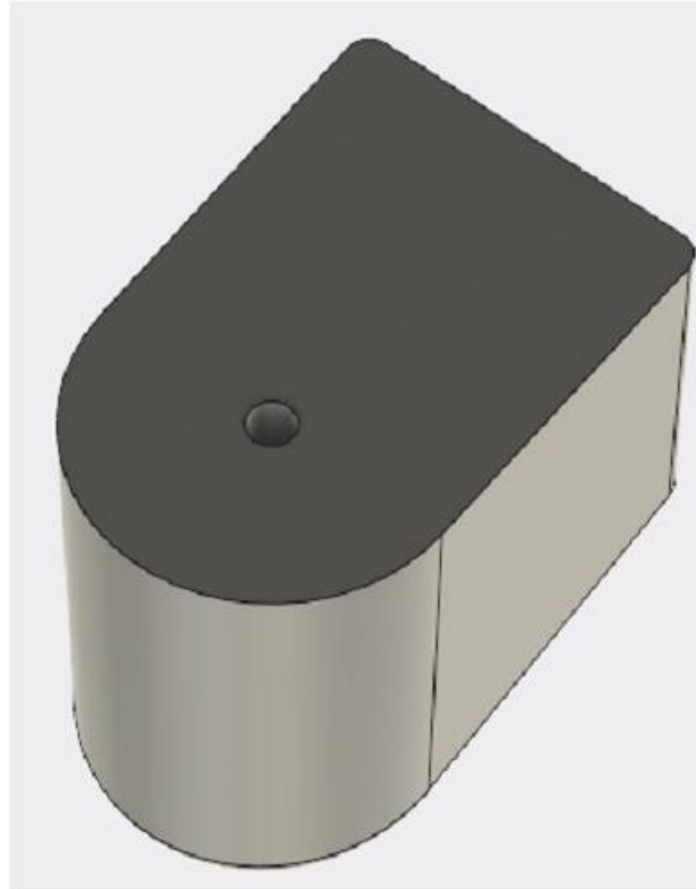

Figure 9–35

3. Create a Shell that removes the bottom face (the same face as the hole's placement face). Set the *Inside Thickness* to **0.05 in**. The design displays as shown in Figure 9–36.
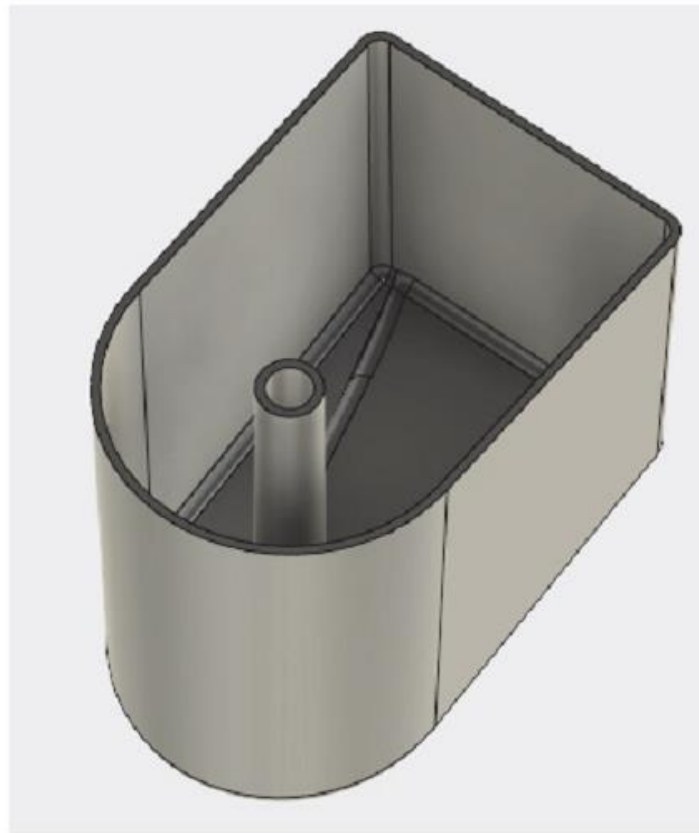

Figure 9–36

Practice 2

4. In the CREATE panel, click ⬚ (Thread).

5. Select the inner surface of the hole.

6. In the THREAD palette, enter the parameters shown in Figure 9–37. The completed thread should display as shown in Figure 9–38.
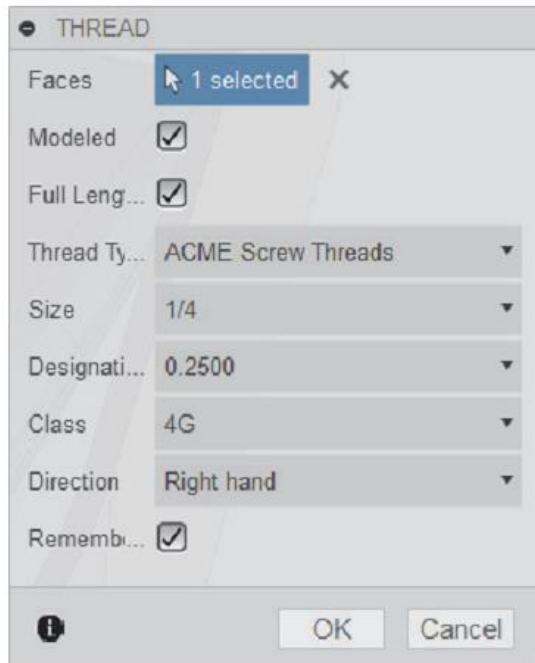
Practice 2

Figure 9–37

Figure 9–38

7. Save the design with the name **split_draft** to your *Autodesk Fusion 360 Practice Files* project.

8. Close the file.