"Fusion 360" Lab

Lecture 5

Lebanese University - Faculty of Engineering – Branch 3

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Dr. Ali HARKOUS

Contents

11.1. Sweeps

You can use sweep features to create specific geometry that cannot be created using standard extrusions. It enables you to sketch a cross-section and sweep it along a defined path.

Learning Objectives in this Chapter

- Create swept geometry using appropriate path and profile entities.
- Edit a Sweep feature.

11.1. Sweeps

A Sweep feature creates geometry that is defined by sweeping a profile along a path. You can use it to add or remove geometry from a design. Sweeps are useful for features that have a uniform shape, but an irregular path.

- Figure 11–1 shows an image of a paper-clip, where a circular profile was swept along a curved path. The path reference is an open path.
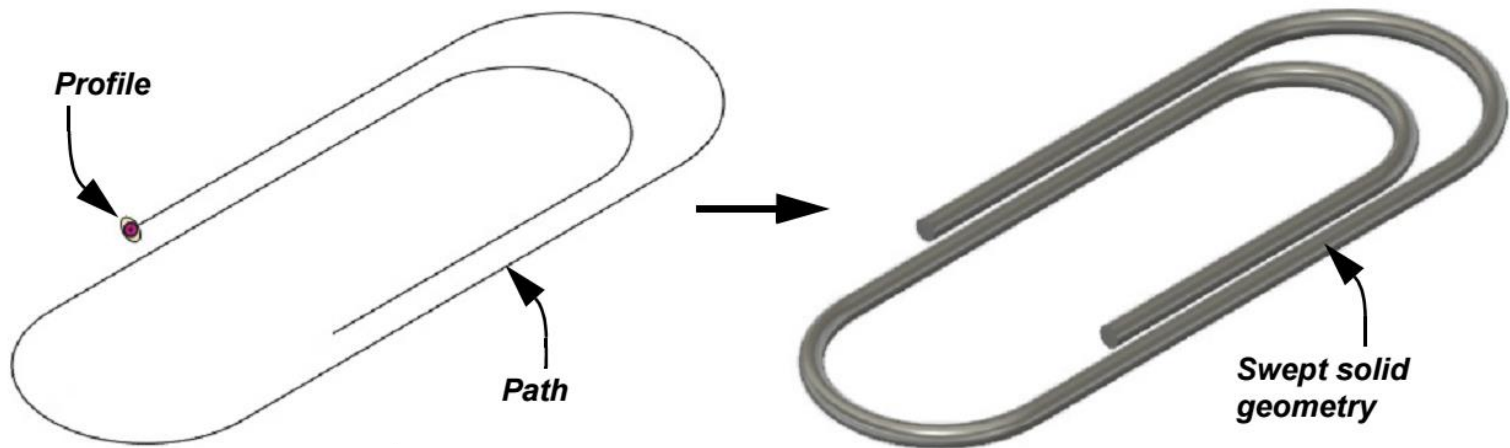

Figure 11–1

11.1. Sweeps

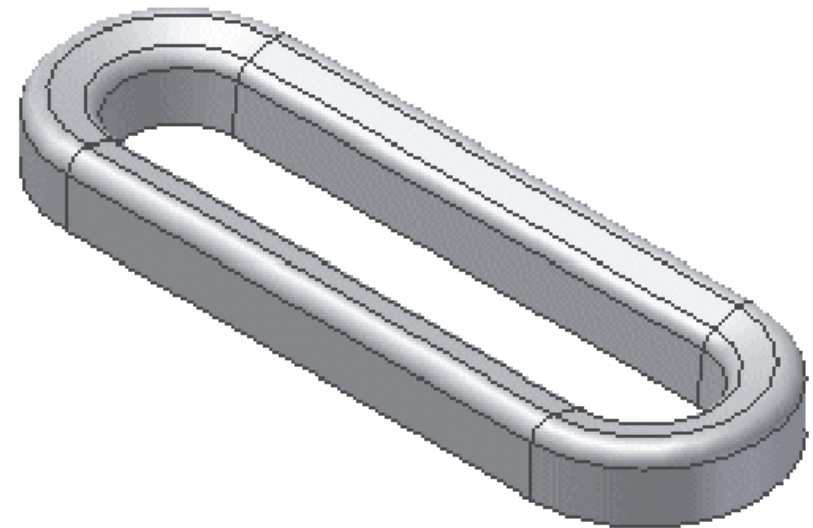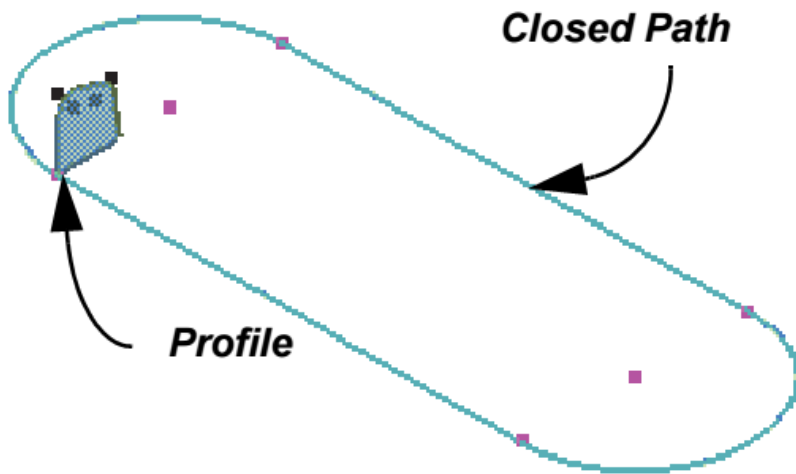- Figure 11–2 shows an image of swept geometry that uses a closed path.

Closed Path

Profile

Figure 11–2

11.1. Sweeps

Use the following general steps to create a single path sweep:

1. Sketch the profile and path entities, similar to that shown in Figure 11–3. They must exist prior to starting the sweep command.
2. Start the creation of the sweep using either of the following methods. The SWEEP palette displays as shown in Figure 11–3.

- In the CREATE panel, click (Sweep).
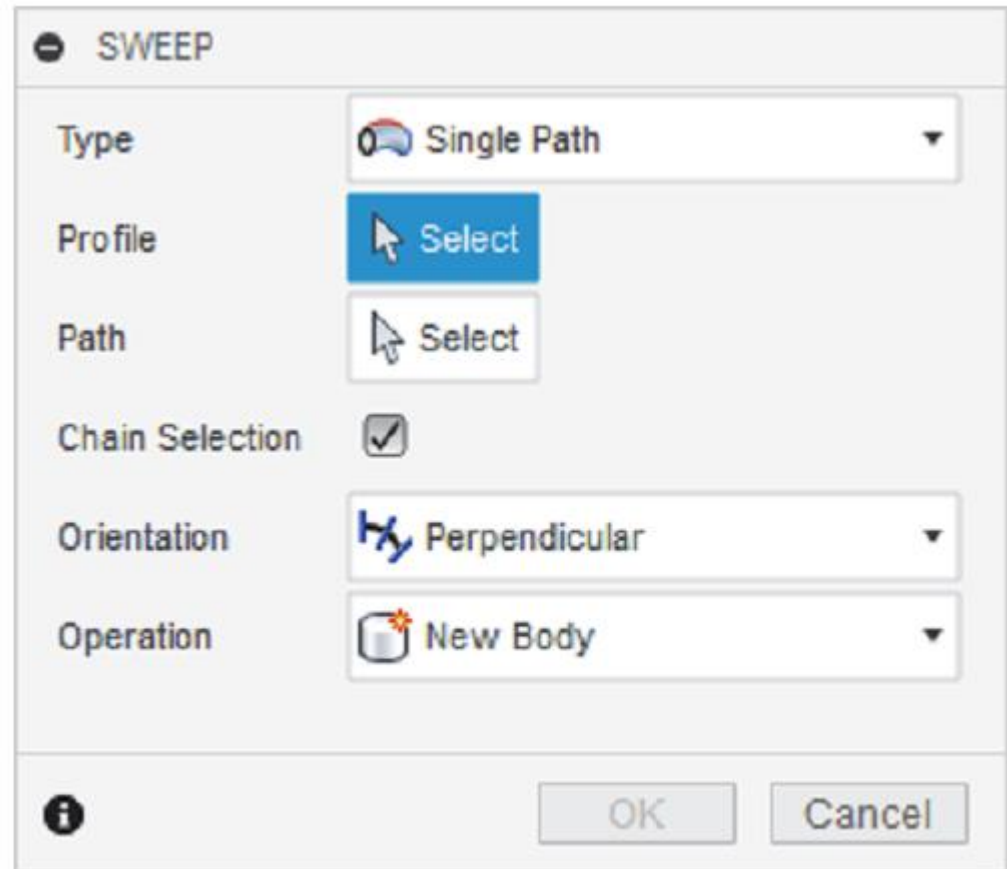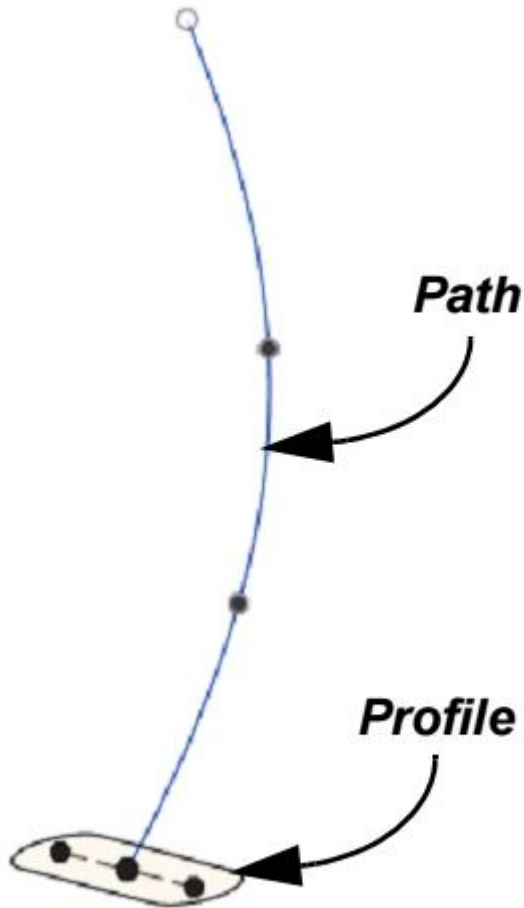- In the graphics window, right-click and select **Create>Sweep**.

11.1. Sweeps

Figure 11–3

11.1. Sweeps

3. Ensure that the *Type* is set to **Single Path**.
4. Select the *Profile* and *Path* references in the design. The profile selection is immediately active. To select the path, activate its selection field prior to selection. Once both references are selected, a preview of the geometry is displayed in the graphics window.

   - The sweep's profile must be a closed loop sketch.

   - The sweep's path can be an open or a closed sketch and can be either 2D or 3D.

   - The start point of a sweep should be located at the intersection of the profile plane and path. The profile geometry does not need to physically intersect the path.

11.1. Sweeps

5. Refine the shape of the sweep using the additional options.

   - The profile can be swept a set distance along the path using a *Distance* value, as shown in Figure 11–4. The value is defined as a proportion of the overall path. Alternatively, you can drag the manipulator arrow to define the value, as required.
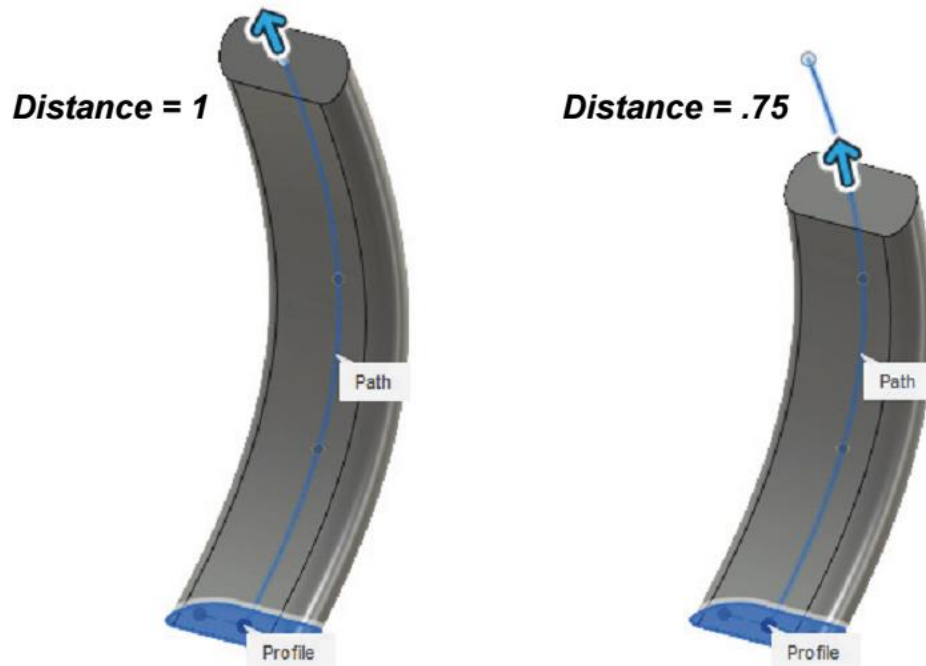

Figure 11–4

The **Path + Guide Rail** and **Path + Guide Surface** are advanced sweep types that provide for additional control over the path of the profile. For more information on these advanced types, refer to the Help documentation.

11.1. Sweeps

- *Taper Angle* and *Twist Angle* values can be set to vary how the profile is swept along the path. Taper varies the profile size along the path, while twist rotates the profile.

- A sweep's orientation can alter the shape of the final geometry. A **Perpendicular** orientation ensures that the profile remains perpendicular to the path, whereas a **Parallel** orientation keeps the profile parallel to the profile's sketch plane, as shown in Figure 11–5.
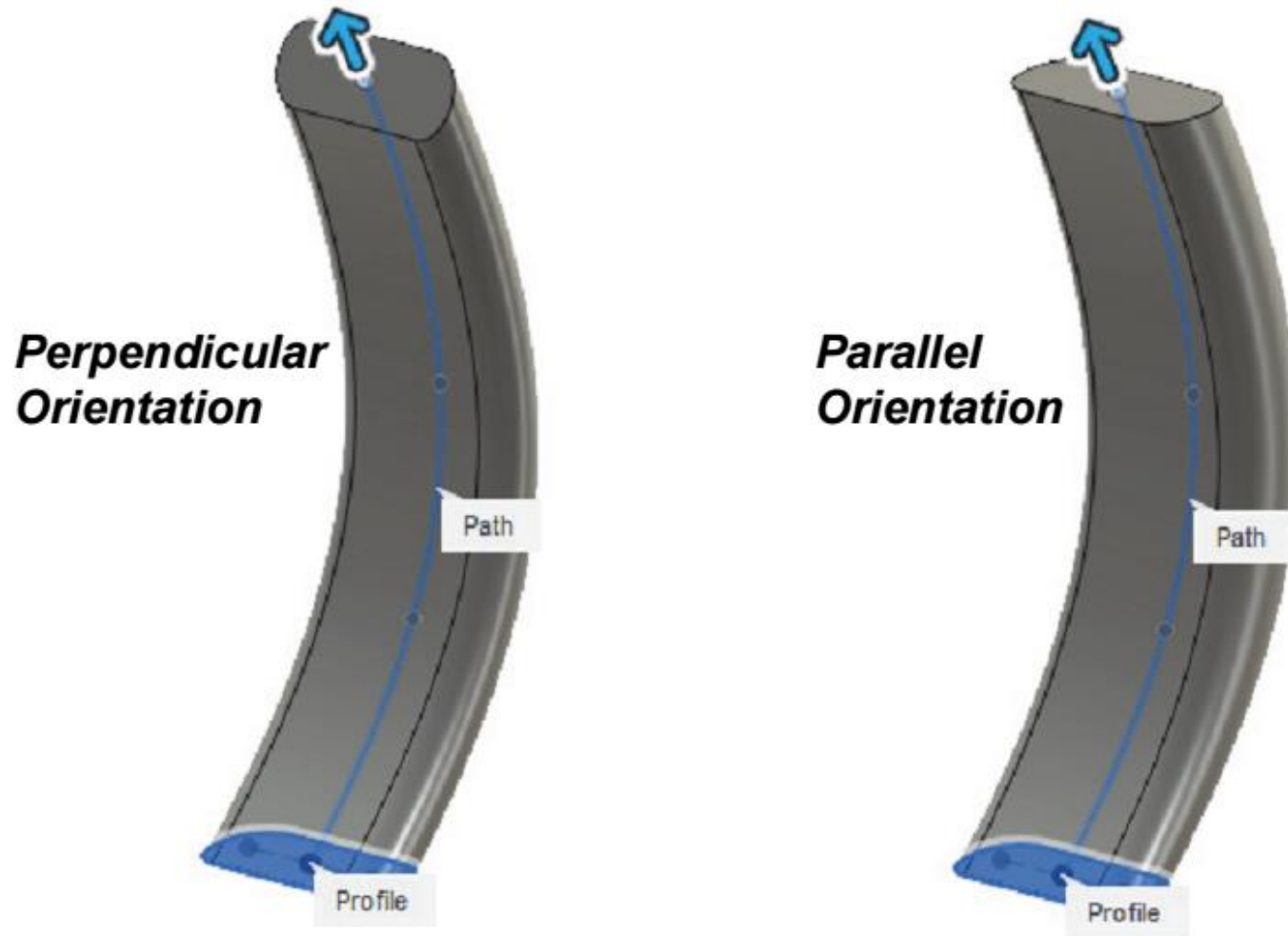
11.1. Sweeps

Perpendicular Orientation

Path

Profile

Parallel Orientation

Path

Profile

Figure 11–5

11.1. Sweeps

6. Select the *Operation* required to create the geometry.

    *   The (New Body) option creates the feature as a new body in the design. The first solid feature in the design defaults to a new body.

    *   When creating secondary swept features, (Join), (Cut), and options can be used to define how the new geometry is combined with existing geometry. Join will add material and Cut removes material (Cut).

    *   When creating secondary swept features, the (Intersect) option creates geometry from the shared volume of the new and existing features and removes material outside of the shared volume.

11.1. Sweeps

- The ▱ (New Component) option creates the feature as a new component in the design. This is used when creating a multi-component assembly design.

7. Complete the feature.

> *Individual bodies can be manipulated independently in the design and combined with other bodies to create a single solid. This is discussed in the Multi-Body Design content. Bodies are listed in the BROWSER in the **Bodies** folder.*

Command Summary

Button	Command	Location
	Sweep	• **Ribbon:** *Model* Workspace>CREATE panel • **Context Menu**: Right-click in the graphics window and select **Create**.

Chap 11
Practice

Creating Swept Geometry I

Practice Objectives

- Create swept geometry using appropriate path and profile entities.
- Edit a sweep feature.

In this practice, you will create two sweep features. For both of these features, you will sketch their profile entities. However, the path for one will be sketched and the other will be selected. The model will display similar to that shown in Figure 11–6.
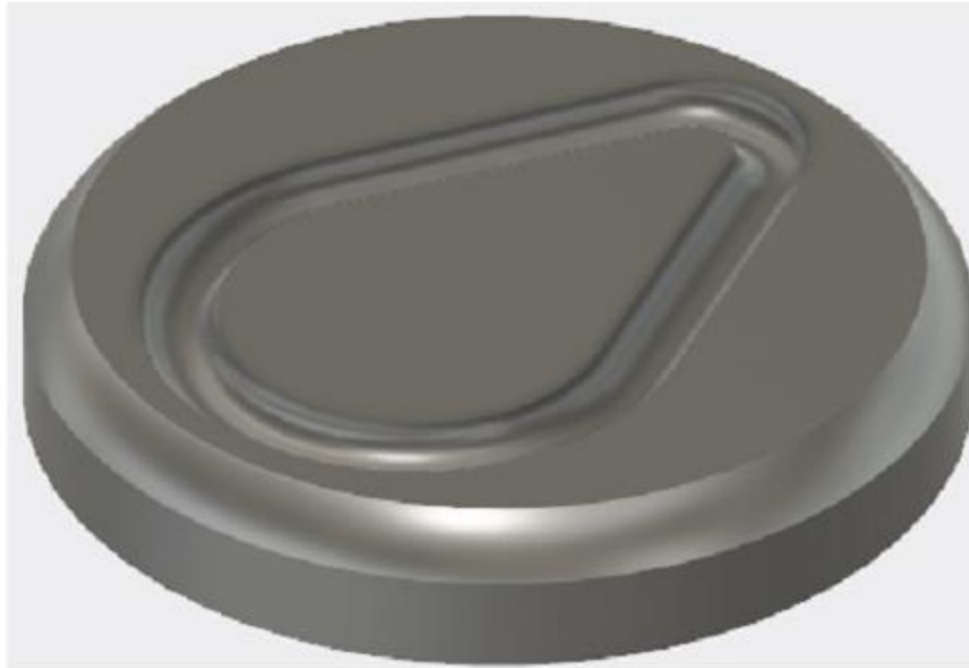
Figure 11–6

Practice 1

Task 1 - Create a new part file with a cylindrical base feature.

1. Create a new design using the **Inch** as the *Active Units*.

2. Use the **Cylinder** quick shape to create a base shape on the XZ plane. Create the geometry with a diameter of **6 in** and depth of **1 in**.

Task 2 - Create the path and profile for the sweep.

1. Sketch on the flat top surface of the cylinder to create the path for the sweep. Draw and constrain the section as shown in Figure 11–7. Complete the sketch.
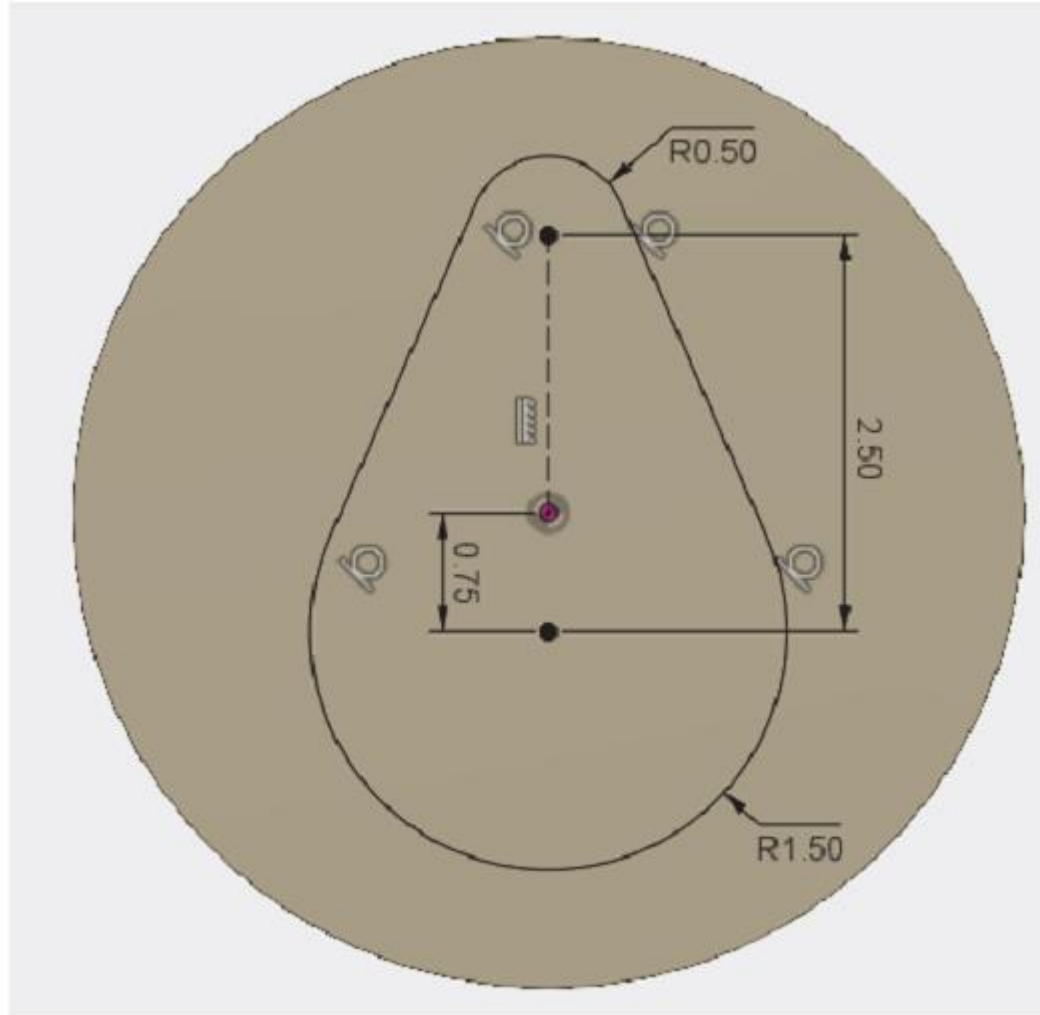
Figure 11–7

To sketch a line that is tangent to two circles, start the **Line** command. Hover the cursor over one of the two circles. Click and hold the left mouse button, and then drag the cursor to the next circle until the tangent constraint symbol displays. Release the left mouse button.

2. In the BROWSER, rename the new sketch as **Path**.

3. Sketch a circle on the YZ workplane, as shown in Figure 11–8. This will be the profile for the sweep.
   - When sketching the circle, constrain it to a point where the previously created path intersects the YZ workplane. (**Hint:** To constrain the circle, use Project Geometry to project entities in the path onto the sketch plane.)
   - Complete the sketch.

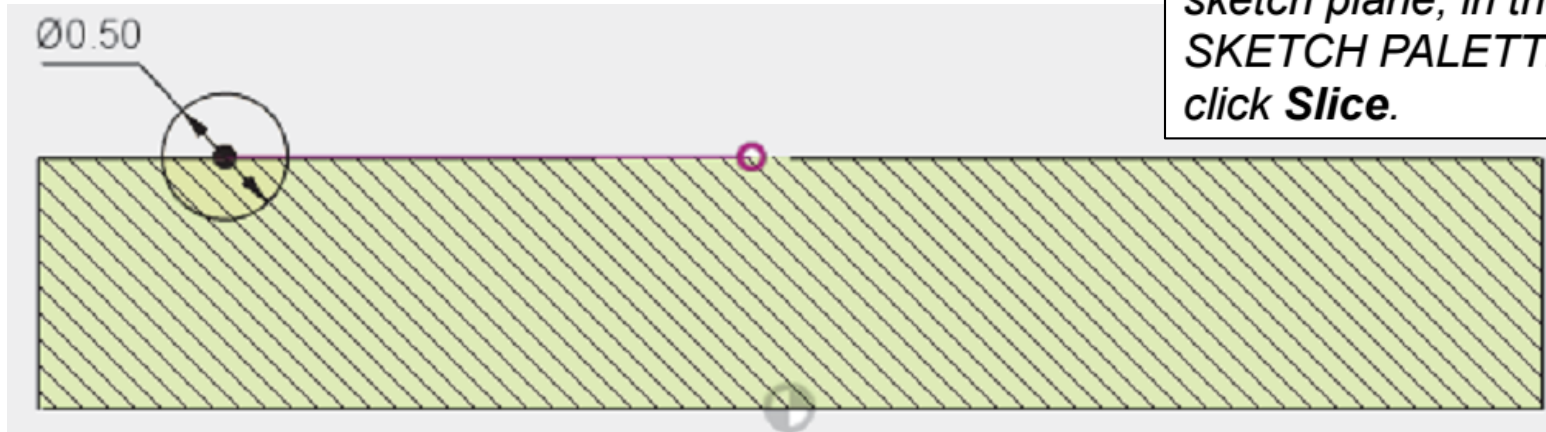*To display a slice of the sketch plane, in the SKETCH PALETTE, click **Slice**.*

Figure 11–8

4. In the BROWSER, rename the sketch as **Profile**.

Task 3 - Create the sweep.

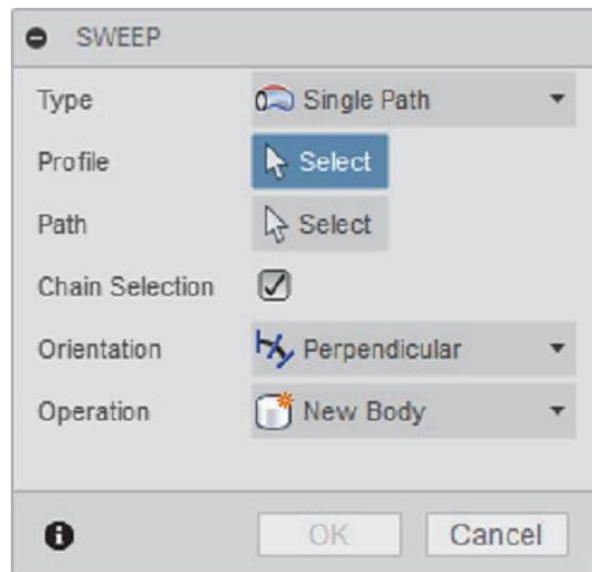1. In the CREATE panel, click (Sweep). The SWEEP palette opens, as shown in Figure 11–9.

Figure 11–9

2. Ensure that the *Type* option is set to **Single Path**.

3. In the graphics window, select the **Profile** sketch as the profile for the sweep.

4. In the SWEEP palette, click inside the *Path* field to activate it. In the graphics window, select the **Path** sketch as the path for the sweep. The geometry displays as shown in Figure 11–10.
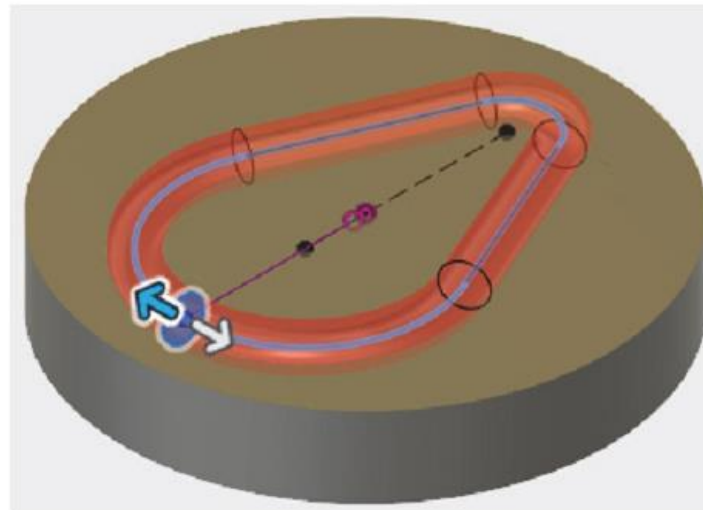

Figure 11–10

5. The two *Distance* fields define how far the profile extends along the path on both sides of the profile. In the first *Distance* field, enter **0.5**. Note how the profile extends for 50% of the entire path length. For the second *Distance* value, enter **0.25** and note how the preview displays.

6. Return the *Distance* fields to **1.0** and **0**, respectively.

Sweeps that use open sketches as the path only have one Distance field in the SWEEP palette.

7. In the *Orientation* field, ensure **Perpendicular** is selected so that the profile remains perpendicular to the path for the entire distance.

8. In the *Operation* field, select **Join** to add material to the solid model.

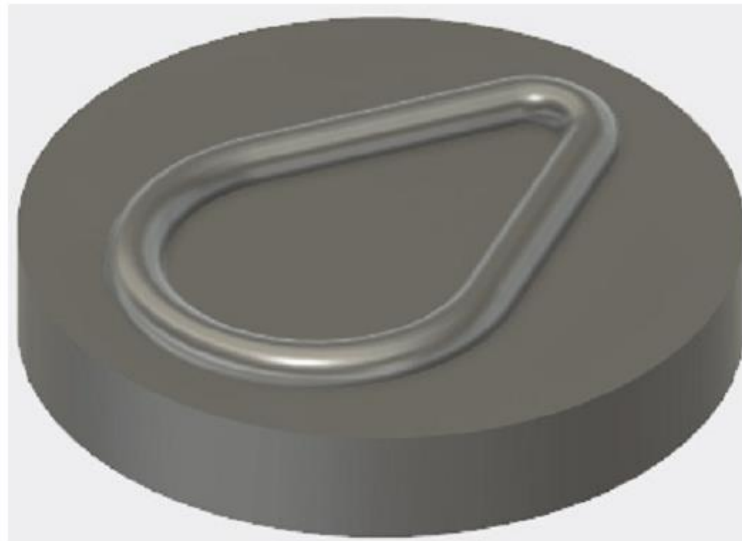9. Click **OK** to create the sweep. The model displays as shown in Figure 11–11.

Figure 11–11

Task 4 - Edit the sweep.

1. Edit the sweep to open the EDIT FEATURE palette.

2. In the *Operation* field, select **Cut** to change the sweep feature from a join to a cut. Expand the *Objects to Cut* area, which is automatically set as **Body1**, by default. This is because Body1 is the only solid body in the model. Complete the feature. The model displays as shown in Figure 11–12.
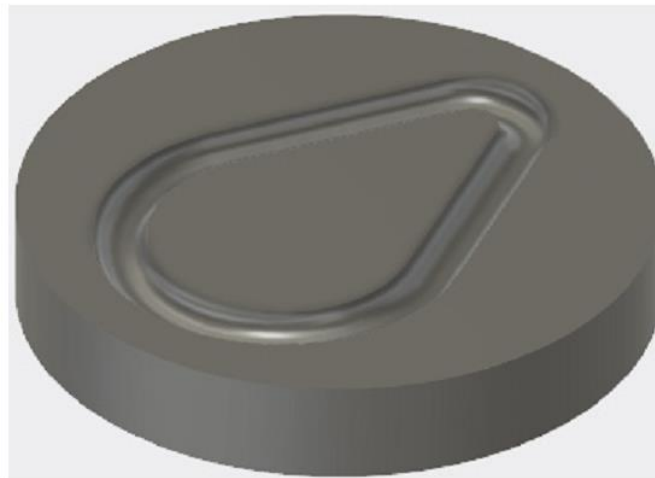

Figure 11–12

Task 5 - Create a new sweep feature that references an existing solid edge.

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1. Sketch the arc shown in Figure 11–13 on the YZ workplane. This will be the profile for the sweep. When sketching the section, consider the following:

   - Use the **Intersect** projection option to project the solid body at the intersection of the sketch plane. This provides the projected references on the sides because it is a cylindrical face and does not have an edge to reference.

   - Sketch the **3-Point** arc so that no tangency is assigned and coincident constraints exist. Dimension the arc as shown in Figure 11–13. Once you select the top edge it will be projected into the sketch.
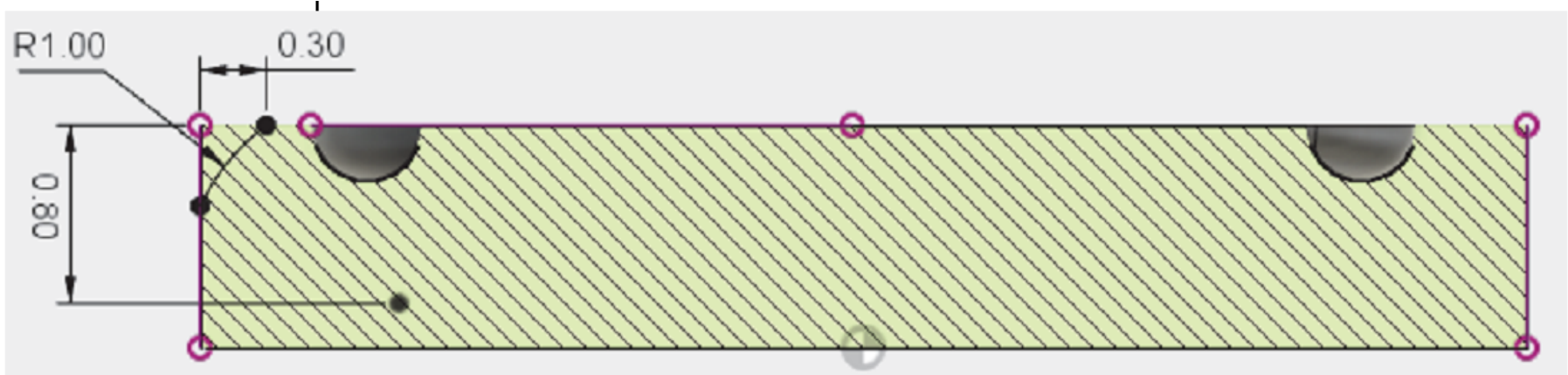
   - Finish the sketch.

Practice 1

Figure 11–13

2. In the Create panel, click (Sweep).

3. Ensure that the *Type* option is set to **Single Path**.

4. In the graphics window, select the arc sketch as the profile for the sweep.

5. In the SWEEP palette, click inside the *Path* field to activate it. Select the existing outside edge of the cylinder as the path for the sweep. The geometry displays as shown in Figure 11–14.

Figure 11–14
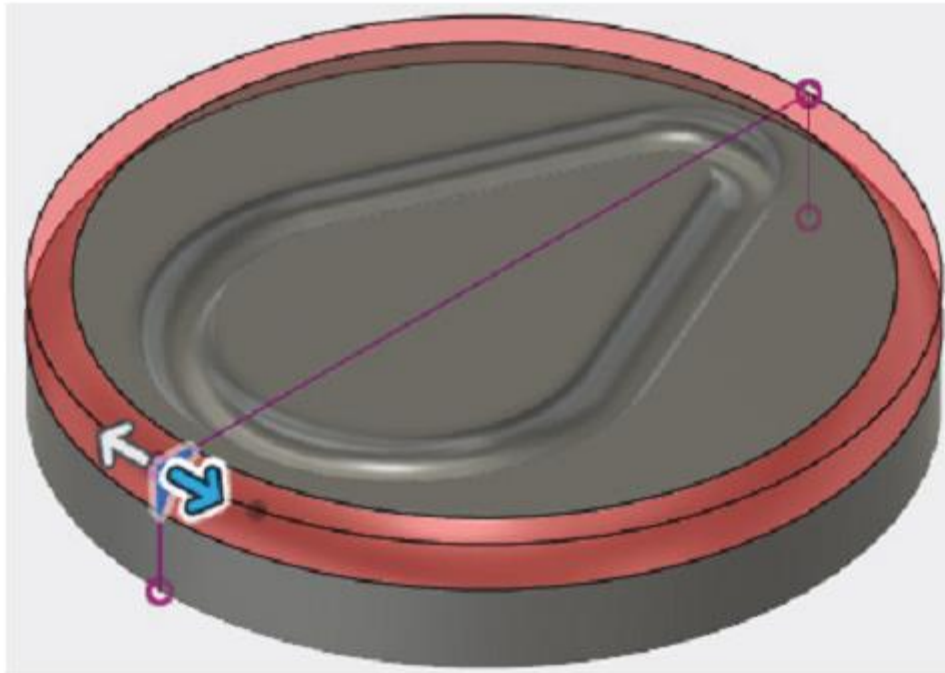
6. The sweep removes material from the entire length of the edge. Set the *Distance* fields as **1.0** and **0**, respectively.

7. In the *Orientation* field, ensure that **Perpendicular** is selected so that the profile remains perpendicular to the path for the entire distance.

8. In the *Operation* field, select **Cut** to remove material from the solid model.

9. Click **OK** to create the sweep. The model displays as shown in Figure 11–15.
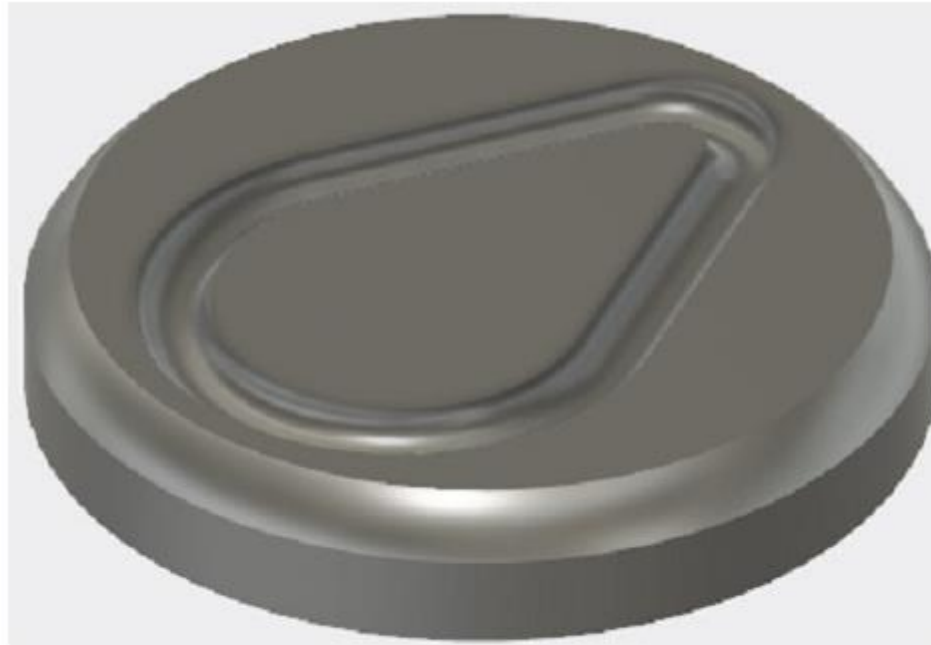
Figure 11–15

10. Save the design with the name **Sweep** to your *Autodesk Fusion 360 Practice Files* project.

11. Close the file.

Creating Swept Geometry II

Practice Objective

- Create swept geometry using appropriate path and profile entities.

In this practice you will create two sweep features. The first represents the handle of the dipstick, and the second represents the metal rod on the dipstick model, as shown in Figure 11–16.

Practice 2

Figure 11–16

Practice 2

Task 1 - Open a part 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 **Dipstick.f3d**, and click **Open**. The model displays as shown in Figure 11–17.
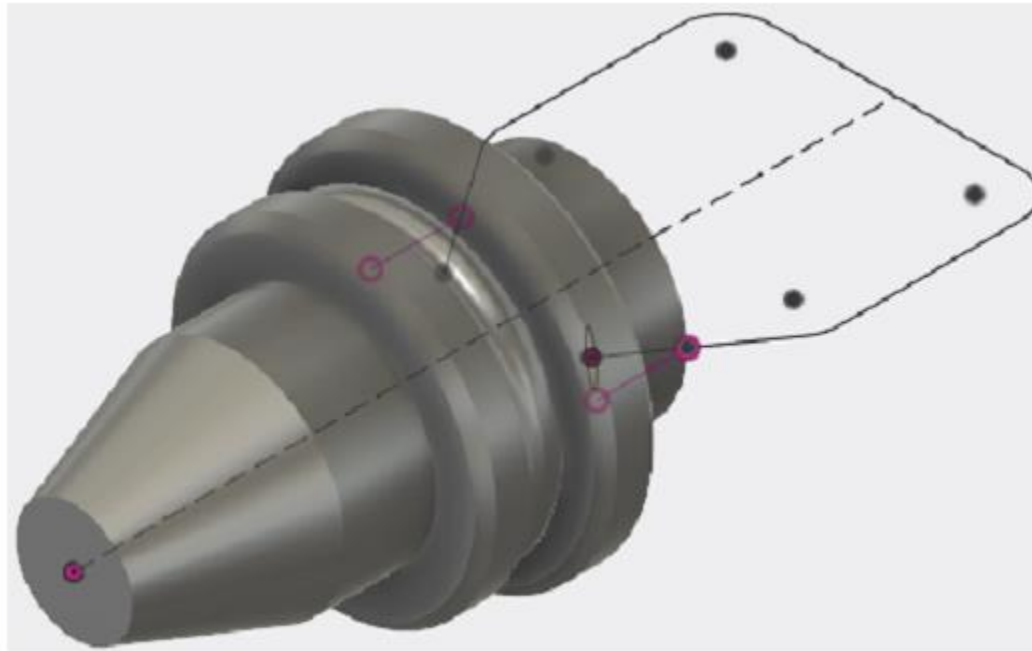
Figure 11–17

3. In the BROWSER, hover the cursor over the **Handle_Path** and **Handle_Profile** sketches to identify where they are in the model.

Practice 2

Task 2 - Create the sweep for the handle.

1. In the CREATE panel, click ⊕ (Sweep).

2. Ensure that the *Type* option is set to **Single Path**.

3. In the graphics window, select the **Handle_Profile** sketch as the profile for the sweep.

4. In the SWEEP palette, click inside the *Path* field to activate it. In the graphics window, select the **Handle_Path** sketch as the path for the sweep. The geometry displays as shown in Figure 11–18.
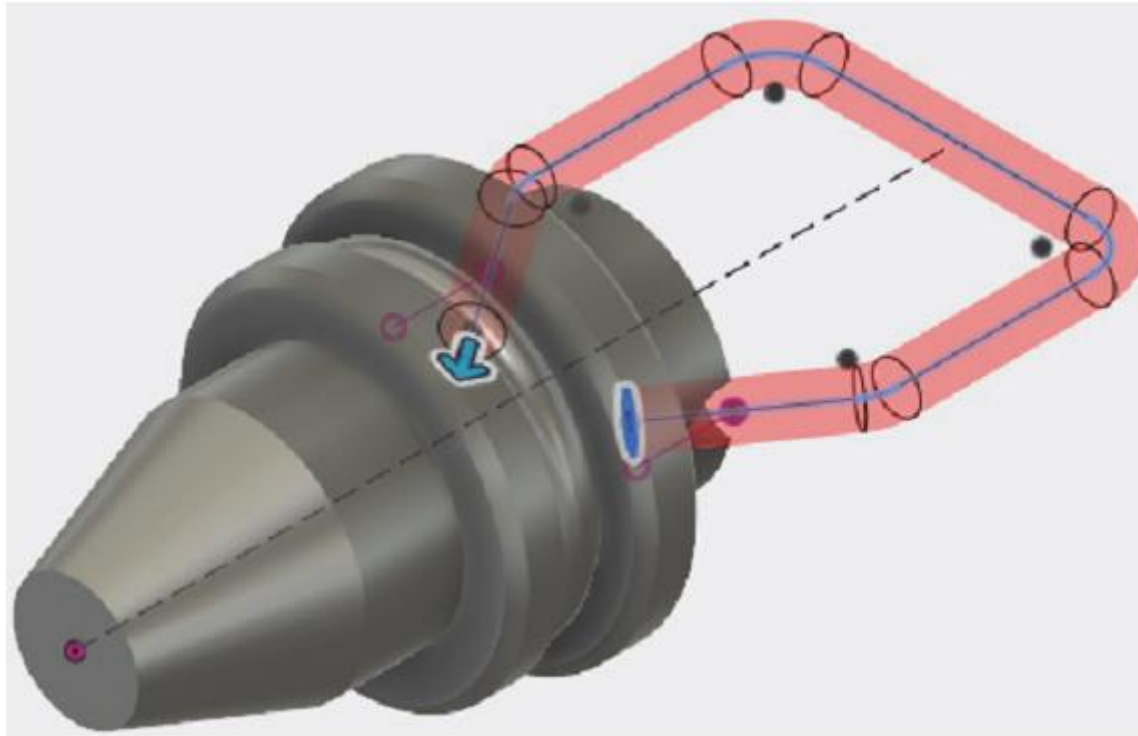
Figure 11–18

Practice 2

5. Note how there is only one *Distance* field because the path reference is an open profile and the profile starts at one end. Ensure that the *Distance* is set to **1.0**.

6. Set the *Orientation* option to **Perpendicular** and the *Operation* option to **Join** to add material to the solid model.

7. Click **OK** to create the sweep. The model displays as shown in Figure 11–19.

Figure 11–19

Task 3 - Create another sweep feature to represent the rod of the dipstick.

1. On the YZ workplane, sketch a sweep path similar to the one shown in Figure 11–20.

   • Sketch the line so that it snaps to the origin centerpoint to locate the start point of the sketch.

   • When sketching, consider clearing the SKETCH PALETTE > **Sketch Grid** option to avoid snapping to the grid lines as you are sketching.

   • Do not worry about the dimensions or its exact shape.

Practice 2

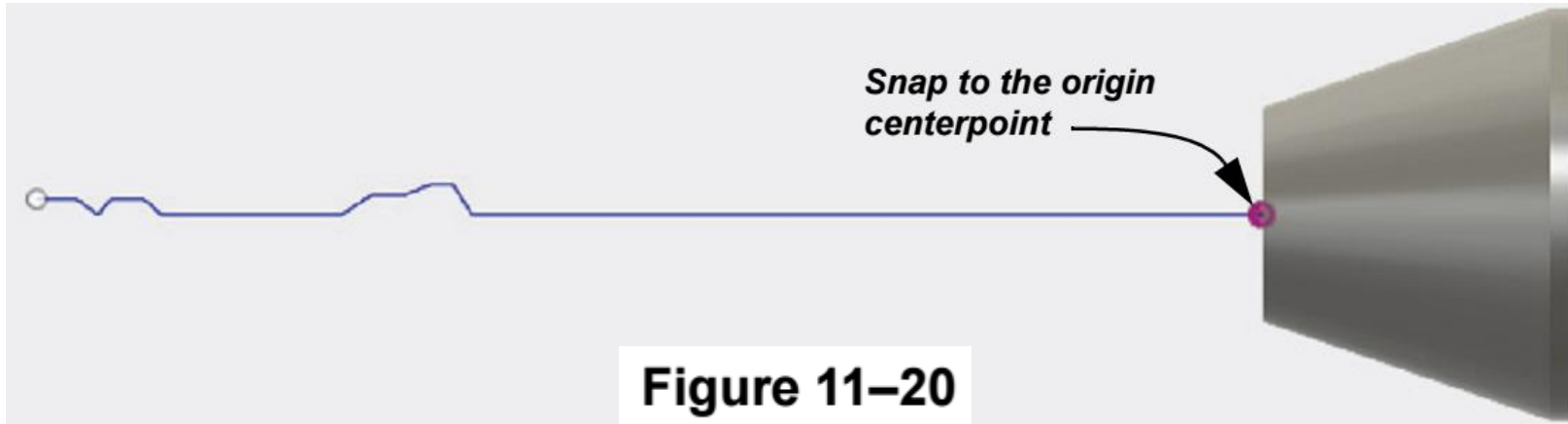**Snap to the origin centerpoint**

Figure 11–20

2. Sketch the sweep's rectangular profile on the bottom face of the revolved feature, similar to that shown in Figure 11–21.

- Project the origin centerpoint and use the **Center Rectangle** sketch option to center the rectangle on the centerpoint.
- Do not worry about the dimensions or its exact shape.
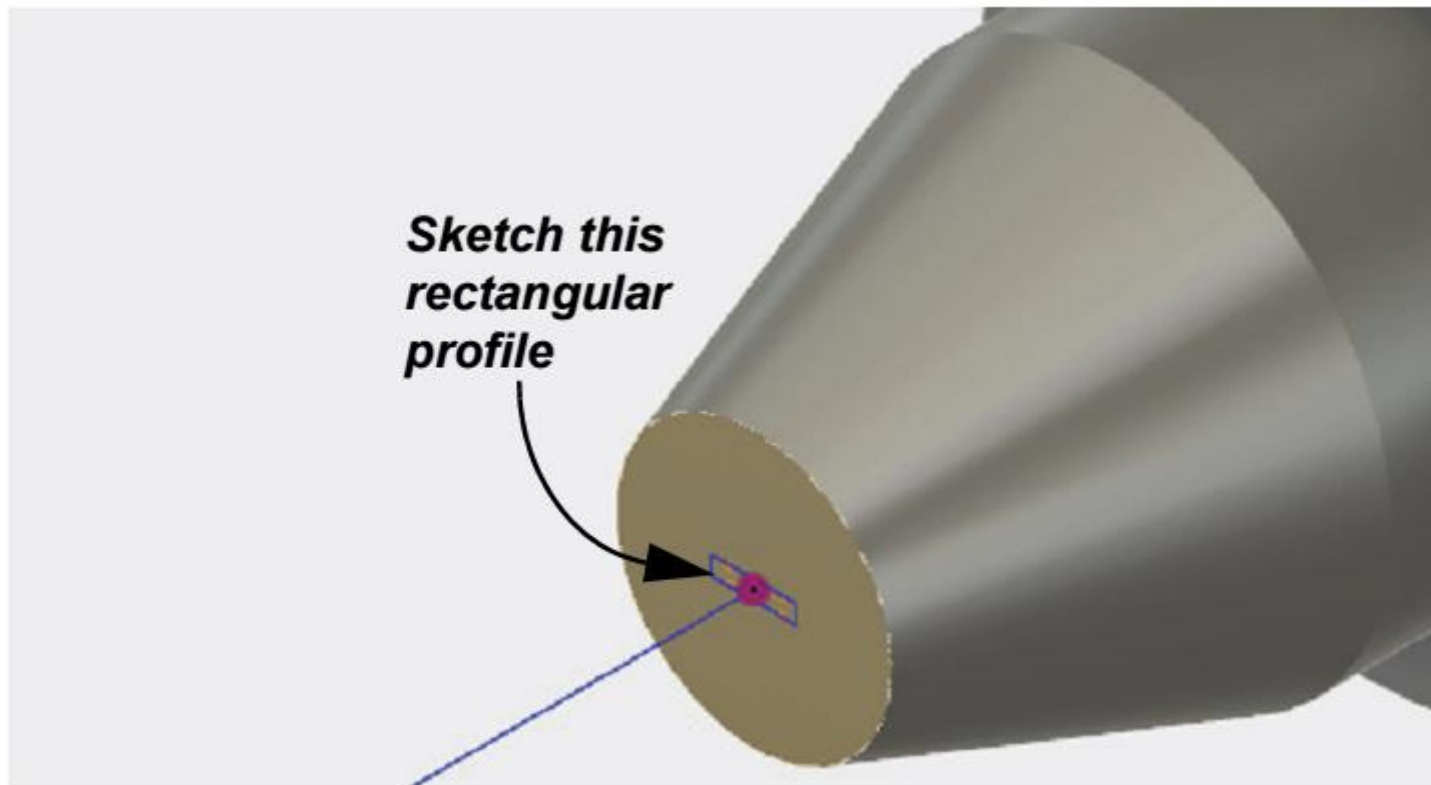
Practice 2

Sketch this rectangular profile

Figure 11–21

3. The profile and path are now defined. Use the **Sweep** command to create the geometry shown in Figure 11–22.

Figure 11–22

In this situation, you could have created the geometry as an extrude. However, the section to extrude would need to be a closed section. You can create the same geometry using either method. With more experience, you will establish modeling preferences.

4. Save the design with the name **Dipstick** to your *Autodesk Fusion 360 Practice Files* project.

5. Close the file.

Additional Swept Geometry (Optional)

Practice Objective

- Create swept geometry using appropriate path and profile entities.

In this practice you are provided some additional geometric shapes that can be created using a Sweep feature. Create each of these using appropriate path and profile geometry.

Task 1 - Create new parts using a single feature.

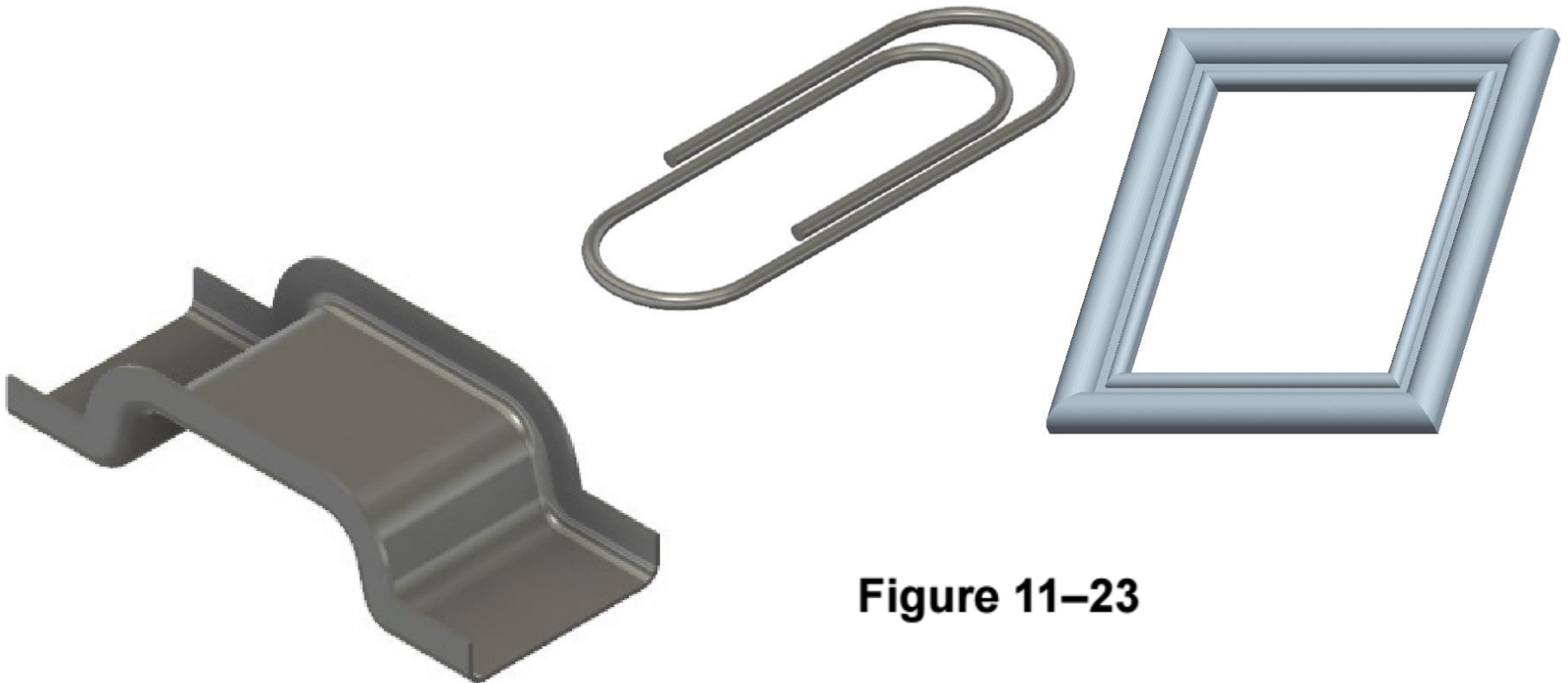1. Create the parts shown in Figure 11–23 using swept features.

Figure 11–23