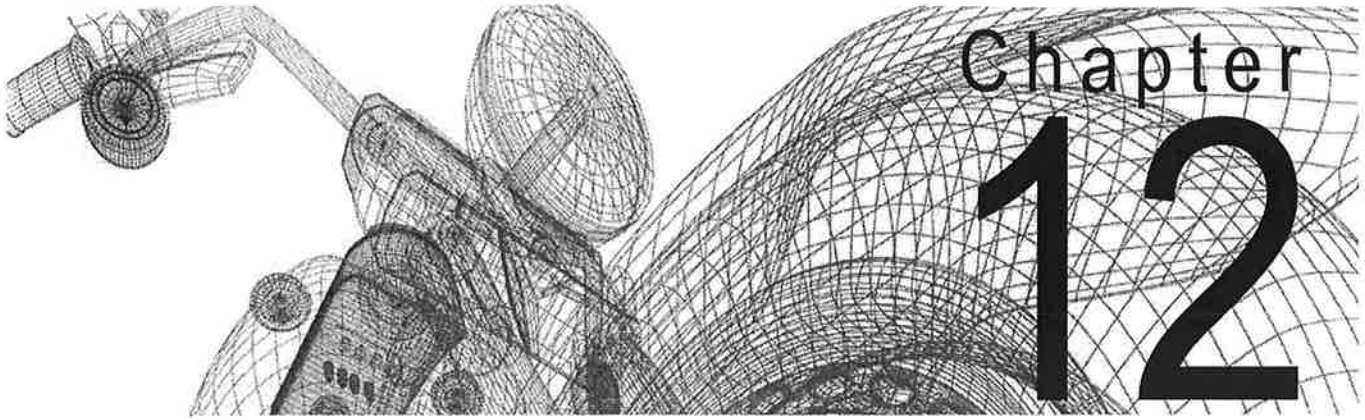

Chapter 12

Loft Features

A loft is a feature that can be used to create complex geometry that blends multiple profiles. Additional control of the resulting shape can be gained by assigning references for the geometry to follow between the profiles, defining end conditions, and manipulating the control points on each section.

Learning Objectives in this Chapter

- Create a loft feature using appropriate profile and reference entities.
- Control the shape and weight of how lofted geometry transitions from adjacent solid geometry.
- Manipulate how the control points on a section maps to the control points on adjacent sections.

12.1 Lofts

A loft feature enables you to create advanced geometry by blending multiple profiles. You can use a loft to either add or remove geometry in the design.

Figure 12–1 shows a loft that blends three sketched profiles and uses rails to control the shape of the geometry between profiles.

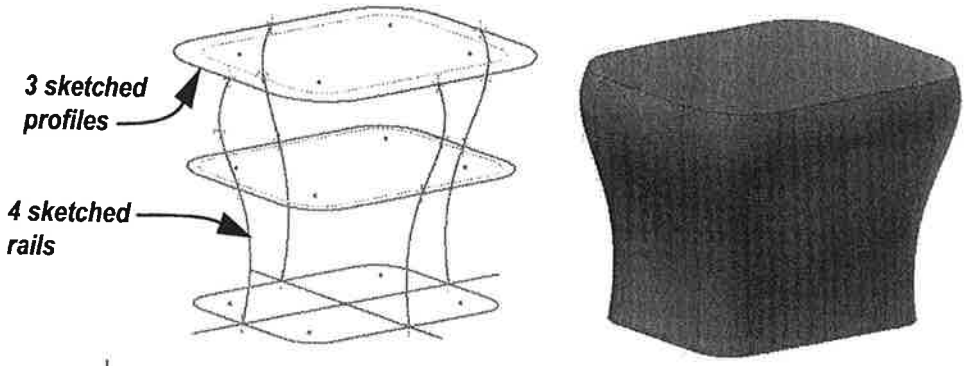

Figure 12–1

While rails must intersect each profile, a centerline is not required to intersect.

Figure 12–2 shows a loft that blends four sketched sections and uses a centerline reference to ensure that the profiles remain normal to a centerline reference as it blends.

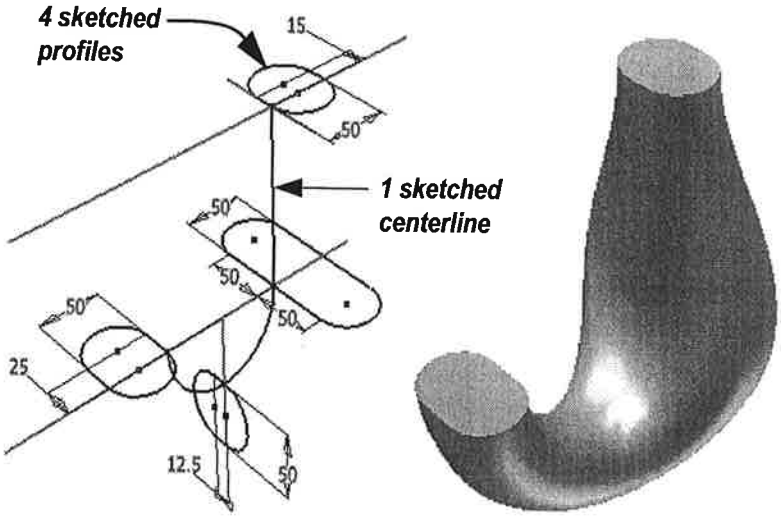

Figure 12–2

Timeline icon:

Use the following general steps to create a loft:

1. Sketch the profiles and any rails that will be used in the creation of the loft, similar to that shown in Figure 12–3. The profiles must exist prior to starting the Loft command.
2. Start the creation of the loft by clicking (Loft) in the CREATE panel.

   The LOFT palette opens as shown in Figure 12–3.

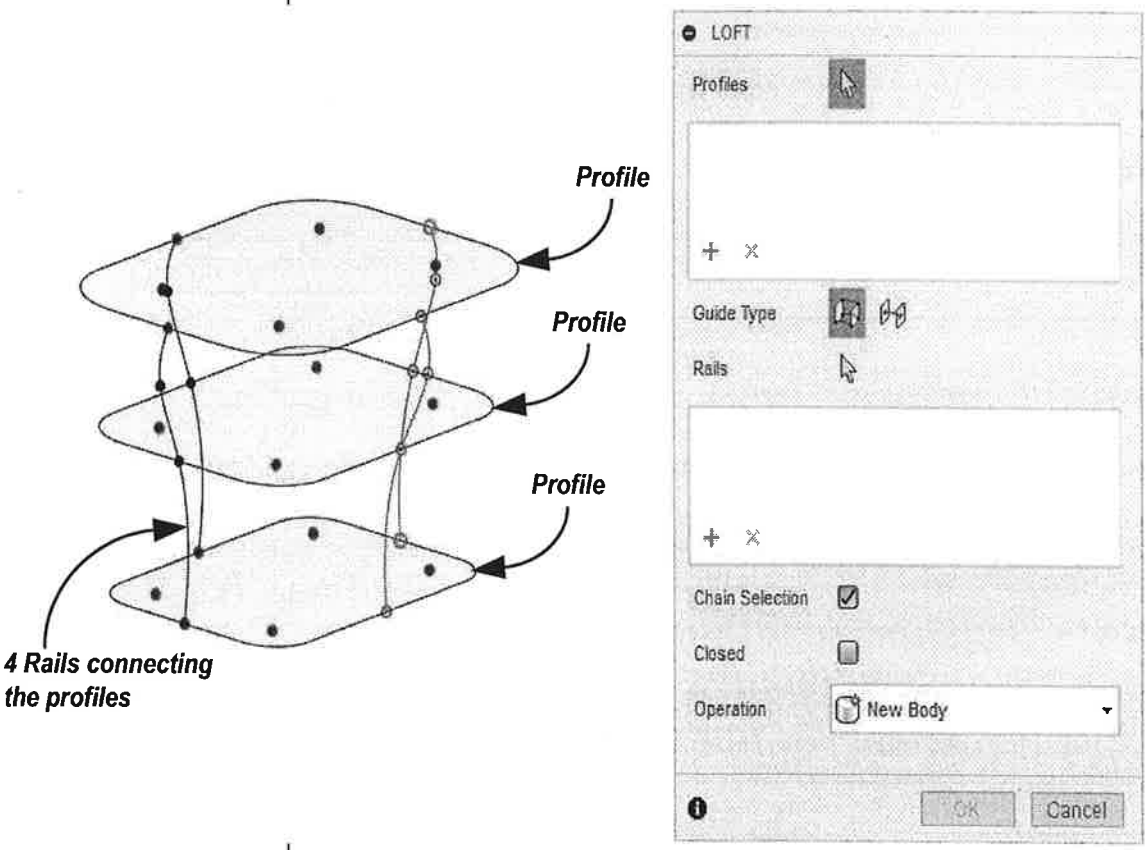

Figure 12–3

Profiles are added in the order that they are selected. To change the order, in the graphics window, select the Profile# symbol and select a new number.

3. Select the *Profile* references in the design. The profile selection is immediately active. Once at least two profile references are selected, a preview of the geometry is displayed in the graphics window, as shown in Figure 12–4.
   - The profiles must be a closed loop sketch, a planar face, or a point.

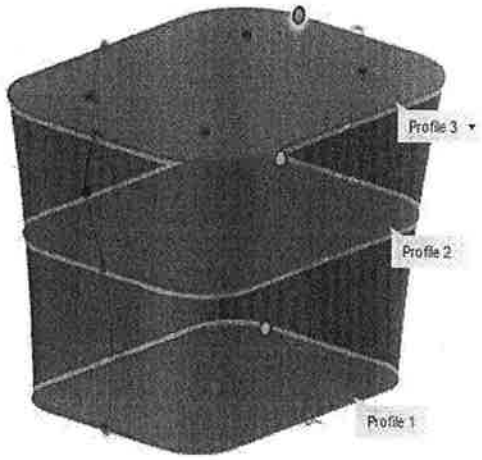

Figure 12–4

4. (Optional) Select the *Guide Type* and the required reference entities. Once the type is selected, select the sketched geometry in the model that you want to use as the rail(s) or centerline. Rails and centerlines are not required to create lofted geometry.

You can select multiple rails, but only one centerline can be selected.

   - Select (Rails) to control the path that the geometry takes as it blends between the selected profiles, similar to that shown in Figure 12–5.

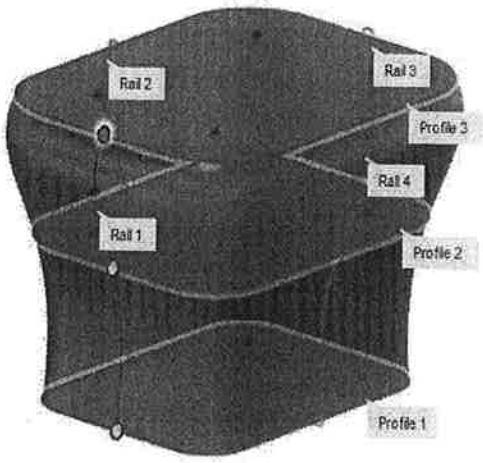

Figure 12–5

   - Select (Centerline) to ensure that the profile remains normal to a centerline reference as it blends between the profiles.

5. (Optional) Refine how the geometry blends between profiles. You can do this using end conditions and control points.
   - End conditions are assigned at the start and end profiles (to control tangency into existing geometry), and at the end point, as shown in Figure 12–6. To define an end condition, select the option for the profile.

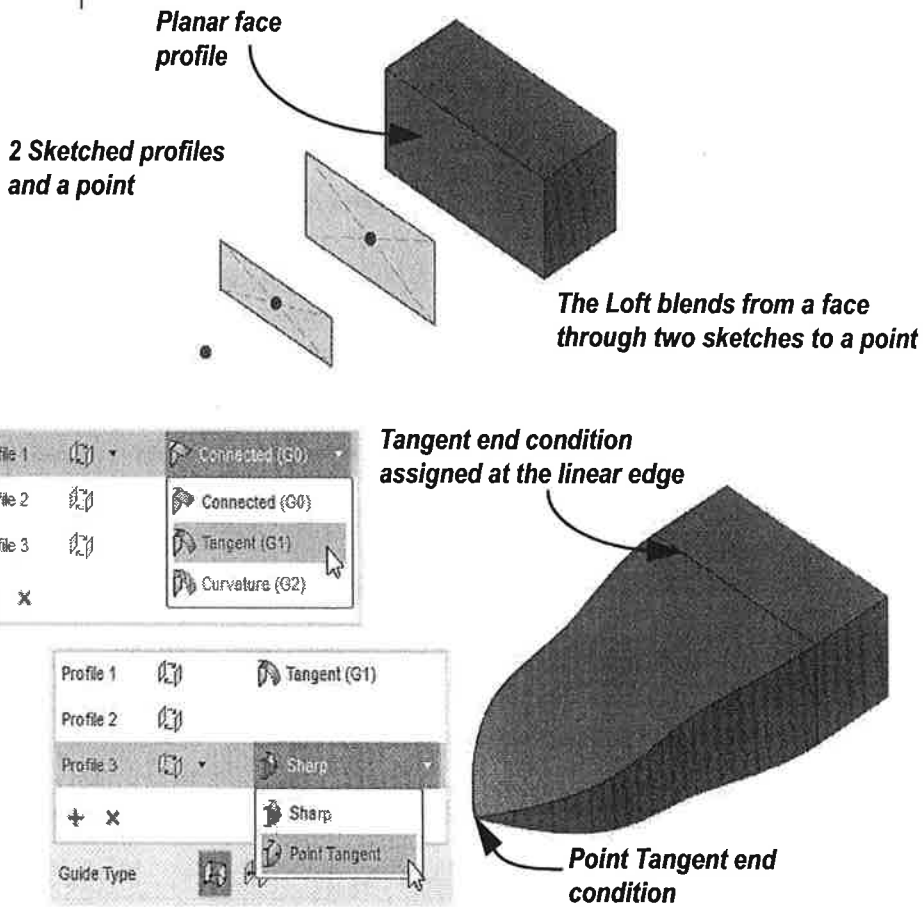

Figure 12–6

   - Control points (white vertices) control how sections blend. Control points can be selected and moved to change loft geometry, as shown in Figure 12–7.

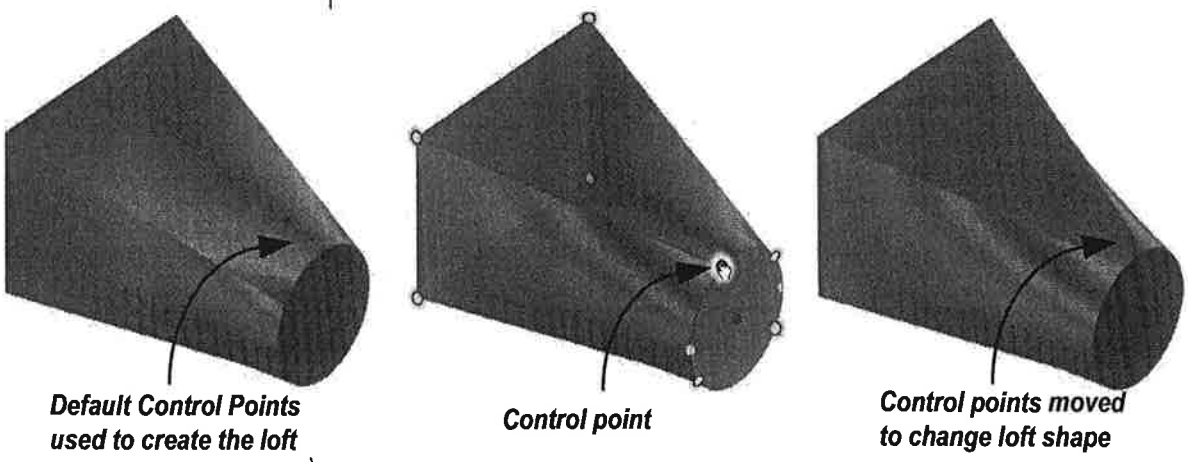

Figure 12–7

*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.*

6. Enable the **Closed** option to connect the first and last profiles to create a closed loft.
7. 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 loft features, the (Join), (Cut), and options can be used to define how the new geometry is combined with existing geometry. Join adds material and Cut removes material.
   - When creating secondary loft features, the (Intersect) option creates geometry from the shared volume of the new and existing features and removes material outside of the shared volume.
   - The (New Component) option creates the feature as a new component in the design. This is used when creating a multi-component assembly design.
8. Complete the feature.

Practice 12a

Creating Rail Lofts

Practice Objectives

- Create lofted geometry using appropriate profile and rail entities.
- Edit a loft feature.

In this practice, you will create the model shown in Figure 12–8 using the loft feature. You will use rails to help control the overall shape between the sections that are being blended together.

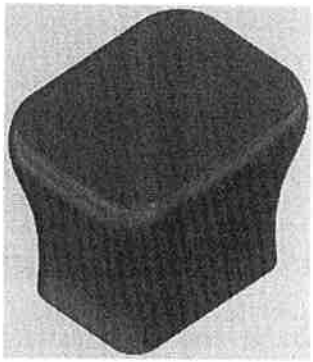

Figure 12–8

Task 1 - Open a part file and create a loft.

In this task, you will open an existing model and create a solid loft feature using sections and rails that have been provided.

1. Click  (File)>**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 **Rail_Loft.f3d**, and click **Open**. The sections and rails required for the loft feature are provided, as shown in Figure 12–9.

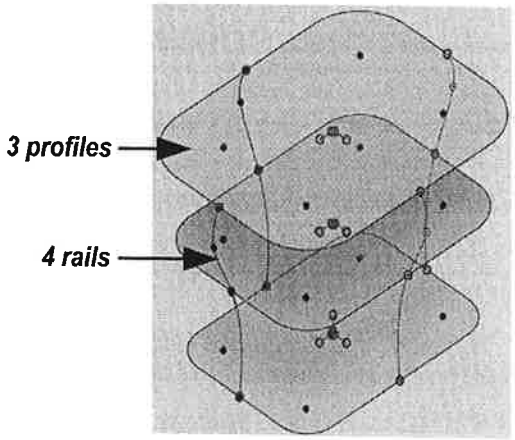

Figure 12–9

3. In the *SOLID* tab>CREATE panel, click (Loft). The LOFT palette opens as shown in Figure 12–10.

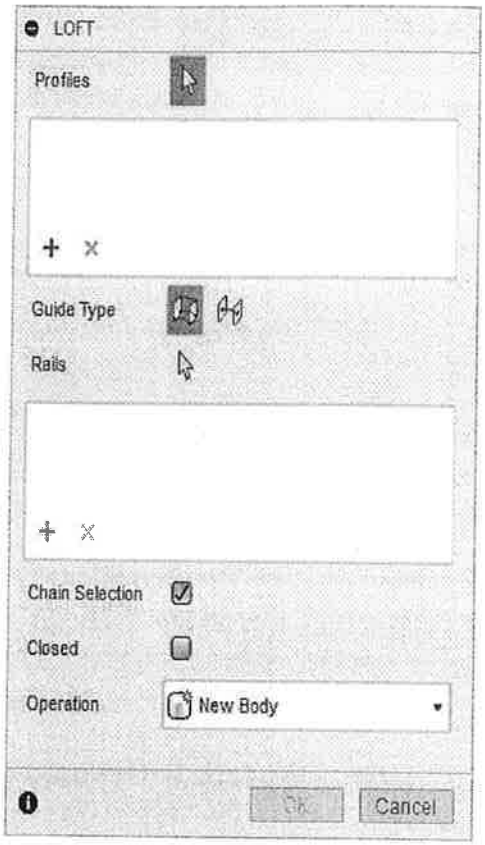

Figure 12–10

4. Ensure that the **Chain Selection** option is enabled. This ensures that all of the entities in a closed loop are selected at once during selection.

The order in which you select the profiles defines how they are lofted together.

5. Ensure that the *Profiles* area is active. In the graphics window, select the three profiles from bottom to top (**Profile1**, **Profile2**, and **Profile3**).

6. Note that because this is the first solid geometry in the model, the *Operation* option defaults to **New Body**.

7. Click **OK** to create the loft.

8. In the BROWSER, expand the **Sketches** folder and note that the three profile sketches are automatically toggled off once they are used. The model displays as shown in Figure 12–11.

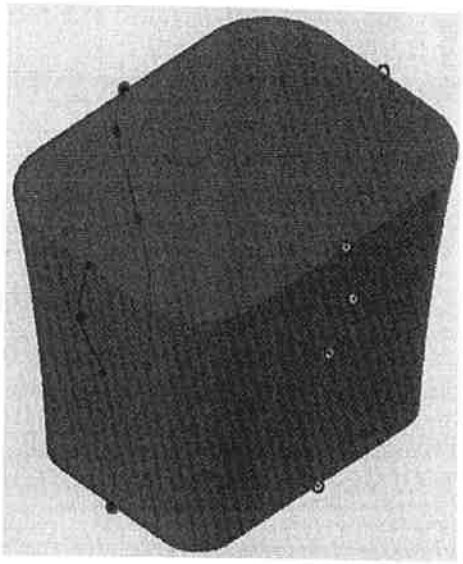

Figure 12–11

Task 2 - Add rails to the loft.

In this task, you will edit the loft feature and add rails. Rails help control the shape of a loft between sections. When creating rails, remember that they must intersect all of the sections in the loft.

1. Edit the loft feature that you just created.
2. In the EDIT FEATURE palette, in the *Guide Type* area, click (Rails) and select the four rails, as shown in Figure 12–12. Note that the shape of the loft updates as you select the rails.

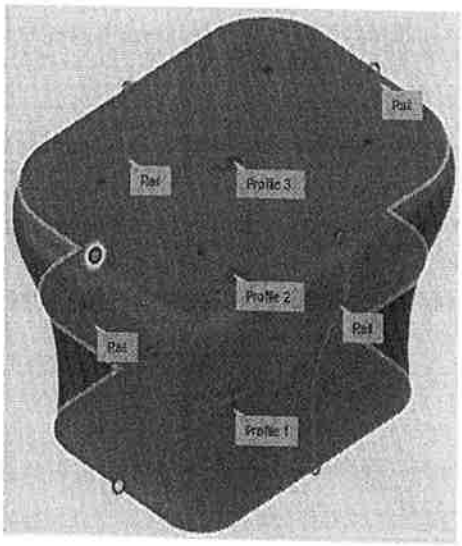

Figure 12–12

3. Click **OK** to update the loft and close the palette.

4. Note that the display of the two rail sketches are automatically turned off once they are used in the Loft feature. The model displays as shown in Figure 12–13.

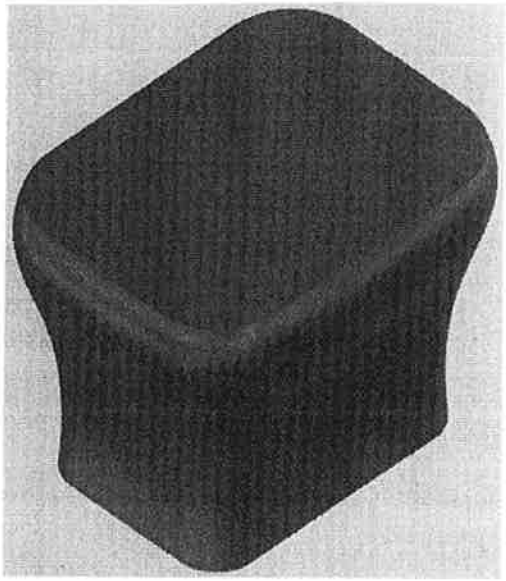

Figure 12–13

5. Save the design with the name **Rail_Loft** to your *Autodesk Fusion 360 Practice Files* project.

6. Close the file.

Practice 12b

Creating Centerline Lofts I

Practice Objective

- Create a loft feature using appropriate profile and centerline entities to drive geometry based on a selected centerline.

In this practice, you will create the loft shown in Figure 12–14. The geometry is generated by selecting two profiles and a single centerline reference.

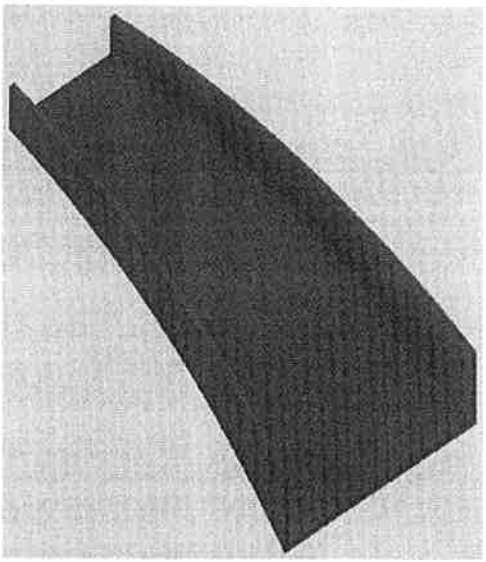

Figure 12–14

Task 1 - Open a part file and create a loft.

In this task, you will open an existing model and create a center line loft using the sketches provided.

1. Click (File)>**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 **CenterLine_loft1.f3d**, and click **Open**. The loft's center line and sections have been created for you.
3. In the CREATE panel, click (Loft). The LOFT palette opens.

4. By default, the *Profiles* area is active. In the graphics window, select the two profiles (**Profile1** and **Profile2**). The loft is generated by blending between the two profile sketches, as shown in Figure 12–15.

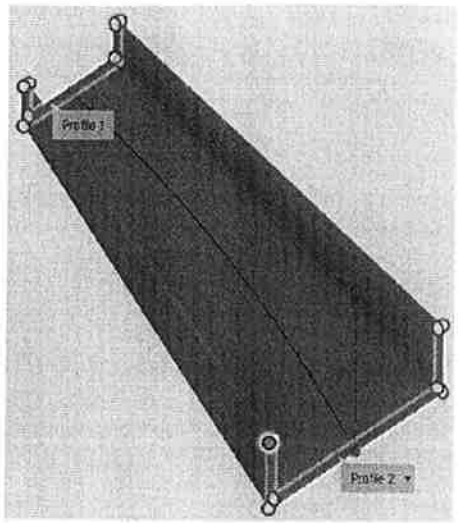

Figure 12–15

5. Click (Centerline) to create a Centerline Loft.
6. Select the **Centerline1** sketched arc.
7. This is the first solid geometry in the model, so the *Operation* option defaults to **New Body**.
8. Click **OK** to complete the loft. The model displays as shown in Figure 12–16.

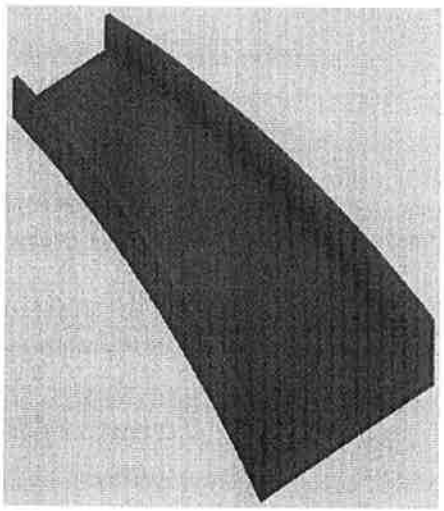

Figure 12–16

9. Save the design with the name **Centerline_Loft1** to your *Autodesk Fusion 360 Practice Files* project.
10. Close the file.

Practice 12c

Creating Centerline Lofts II

Practice Objectives

- Create a loft feature using appropriate reference entities.
- Control the shape and weight of how lofted geometry transitions from adjacent solid geometry.

In this practice, you will create a loft feature between two existing solid features. You will begin by creating a Centerline Loft and then edit this to customize the transition between the existing solid and lofted geometry, as shown in Figure 12–17.

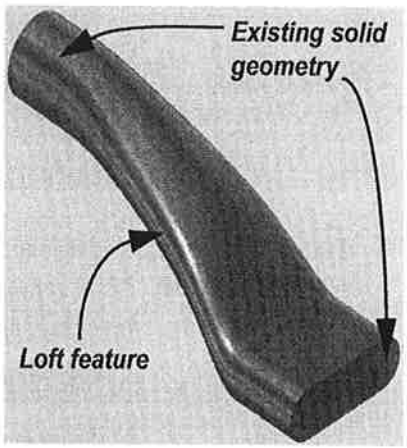

Figure 12–17

Task 1 - Open a part file and create a sketch that will be used as the centerline reference.

1. Click (File)>**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 **Centerline_Loft2.f3d**, and click **Open**.

3. The model displays as shown in Figure 12–18. You will eventually select existing faces as the profiles for the loft, but you must first create a curve that can be used as the centerline reference.

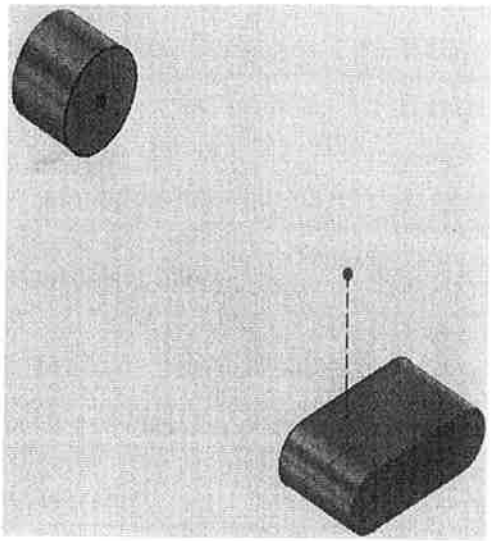

Figure 12–18

4. Start a new sketch on the XY plane. Draw a **3-Point Arc** in the sketch that connects the center points of both sections. Assign a radius of **240 mm**.

5. Complete the sketch. The model should display similar to that shown in Figure 12–19.

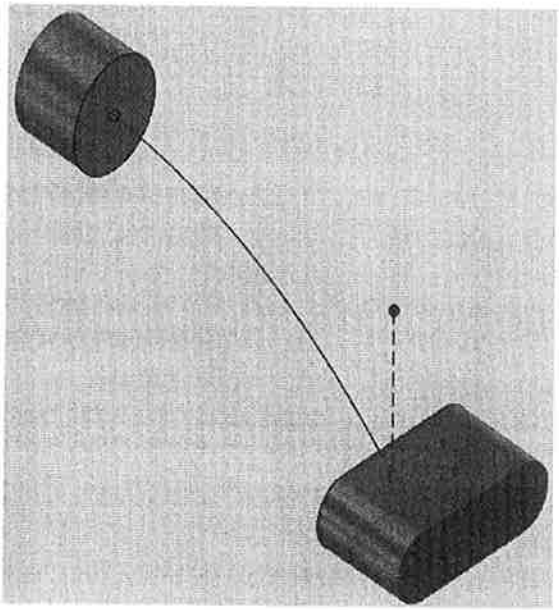

Figure 12–19

It is important that you select solid faces (as opposed to sketches), when necessary, to ensure that end conditions can be assigned to blend the new loft with existing geometry.

Task 2 - Create a centerline loft.

1. In the BROWSER, hide **Sketch1** and **Sketch2**. These were used to create the existing solid geometry. By clearing their display, you ensure that when selecting the profiles for the loft that you will be selecting solid faces and not a sketch.

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

3. The *Profiles* area is active by default. Select the two faces shown in Figure 12–20.

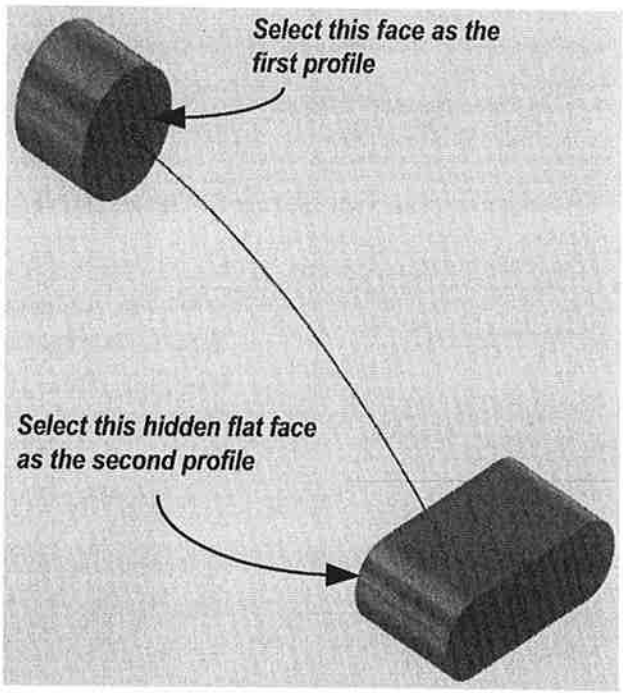

Figure 12–20

4. The new loft geometry is previewed in the graphics window. On the ViewCube, select **FRONT** to reorient the model, as shown in Figure 12–21.

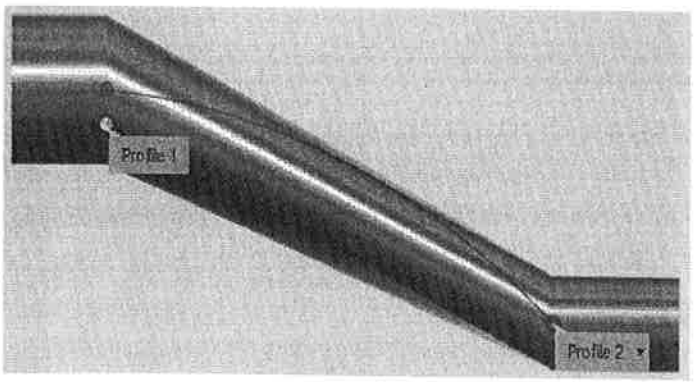

Figure 12–21

5. For the *Guide Type*, select (Centerline).

6. In the *Centerline* area, select the new sketched curve as the centerline reference. The loft geometry updates as shown in Figure 12–22. Note how the centerline controls how the geometry is shaped between the faces.

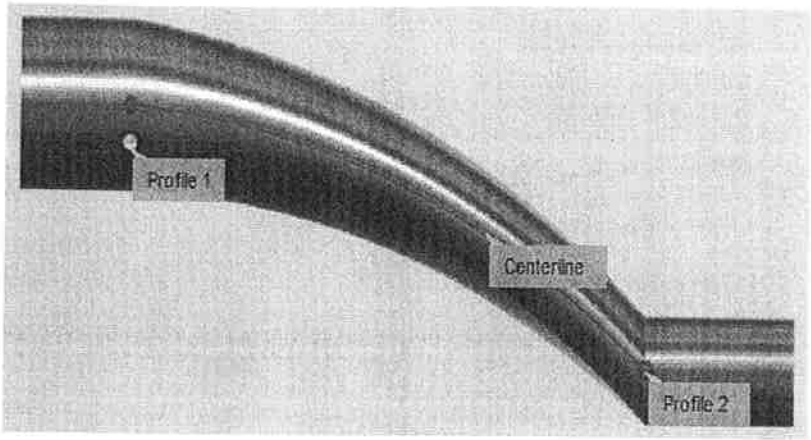

Figure 12–22

7. Click **OK**. The model displays as shown in Figure 12–23. Note how edges exist where the loft joins with the two selected faces. The Loft does not have continuity into the solid geometry.

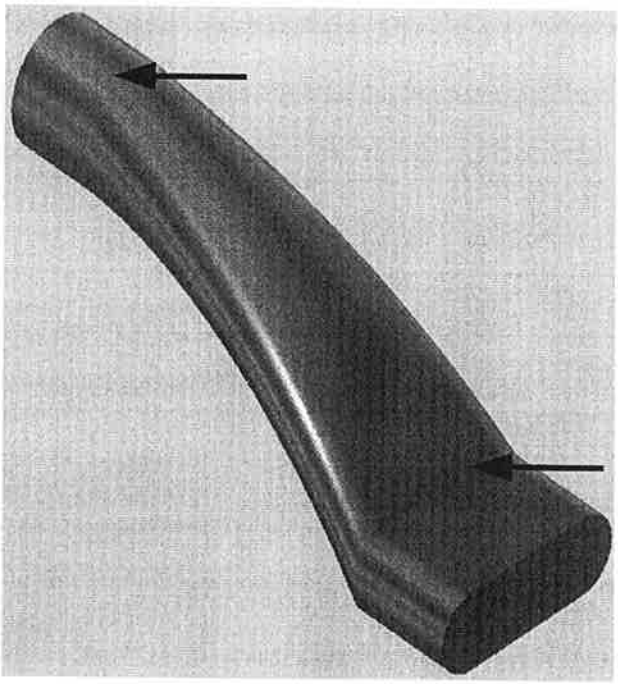

Figure 12–23

Task 3 - Customize the end conditions for the loft.

1. Edit the loft feature.
2. By default, the end condition between a new loft and selected geometry is set so that no end condition is applied (**G0**). In the *Profiles* area, expand the end condition drop-down list for *Profile1* and select **Tangent (G1)**, as shown in Figure 12–24.

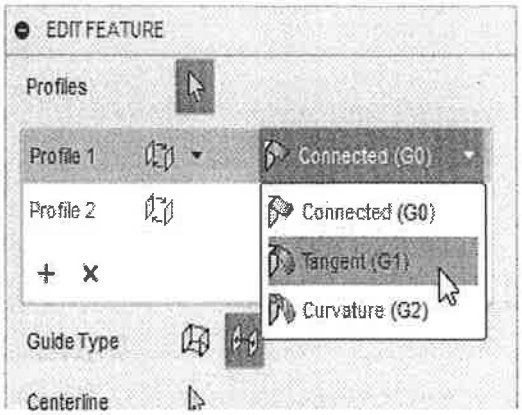

Figure 12–24

3. In the *Profiles* area, select **Profile 2**.
4. Change the end condition for *Profile 2* to **Tangent (G1)**. The loft geometry updates as shown in Figure 12–25. Note how the geometry transitions at the two selected faces to remain tangent to the existing solid geometry.

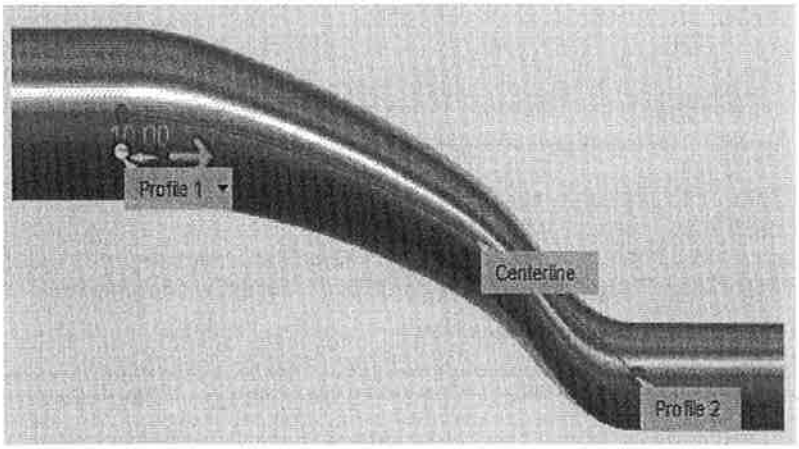

Figure 12–25

5. The *Tangency Weight* value for **Tangent** and **Smooth** end conditions can be edited to control how far the tangency/ continuity is taken to establish the condition. By default, the value for a selected profile is **1**. To change the value, ensure that the profile is selected in the *Profiles* area and enter a value. Note the effect that this value has on the loft geometry when changed to **2** or **0.5** for the two profiles.

6. Return the *Tangency Weight* value for Profile 1 to **1**, and for Profile 2 to **0.5**.

7. Click **OK**. The model displays as shown in Figure 12–26.

The number of vertices in each face are equal. Control points are not available to customize the shape of the loft.

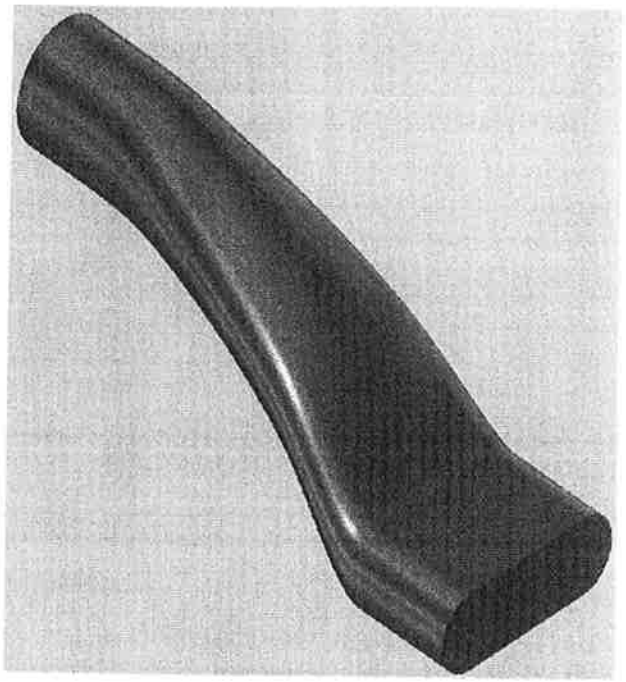

Figure 12–26

8. Save the design with the name **Centerline_Loft2** to your *Autodesk Fusion 360 Practice Files* project.

9. Close the file.

Command Summary

Button	Command	Location
	**Loft**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab>CREATE panel