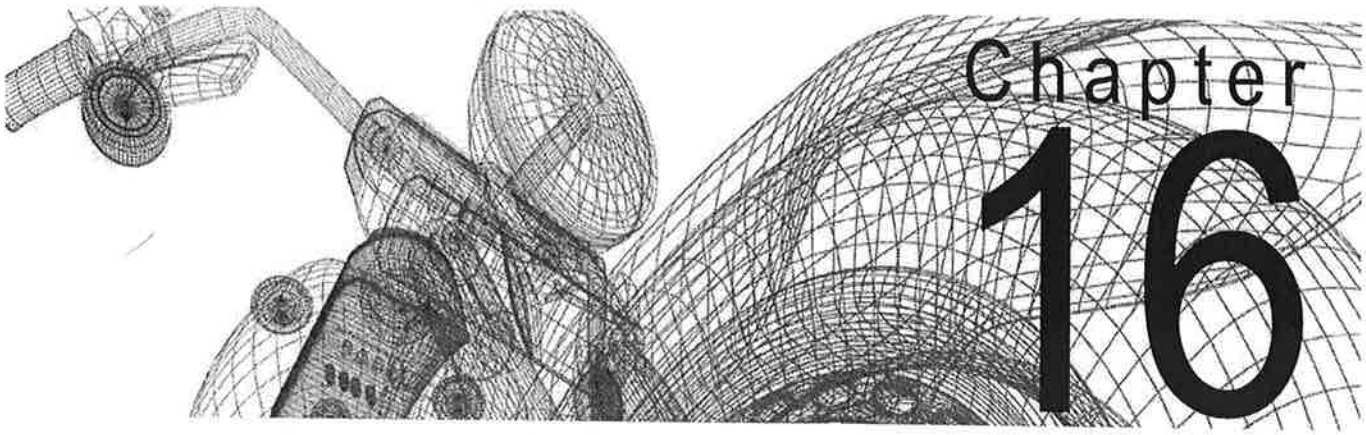


Chapter 16

Multi-Body Design

The multi-body design method enables you to create components that form an assembled product in one design file. A multi-body design starts with a single body and additional bodies are added to represent the overall design. To complete the design, these bodies are ultimately converted to components to which joint connections can be added to constrain them. This chapter discusses the multi-body design method and outlines the tools that can be used to improve your efficiency in creating designs.

Learning Objectives in this Chapter

- Design multiple bodies in a single design file.
- Create components in a single design file.
- Duplicate components in a design file.
- Add as-built joints between components in a design.

16.1 Multi-Body Design

The process of creating multiple bodies in the context of a single design file is known as multi-body design. This is one of the methods that is available in the Autodesk® Fusion 360® software to create a product that is comprised of multiple assembled models. This design approach enables you to create all or some of your assembled design as separate bodies within one design file. The bodies are then converted to components while still remaining within the one design file.

The advantages of creating a design using multi-bodies include the following:

- The entire design resides in a single file, and bodies are later extracted into components that remain in the same design. No top-level or subassembly design files are required.
- A complex part file can be better organized using separate bodies with respect to their function or position in the model.
- You can set up relationships between bodies.You can control the visibility of bodies as a group, rather than at the individual feature level.

All design files contain a **Bodies** folder in the BROWSER. This folder lists all of the bodies in the design. The design shown in Figure 16–1 is a folding easel. It was designed using a combination of the multi-body and distributed design methods. All of the wooden components in the easel were created in the context of the single design using bodies, and all of the hardware were inserted. Joints can be added to assign the required degrees of freedom that enable the easel to collapse and expand.

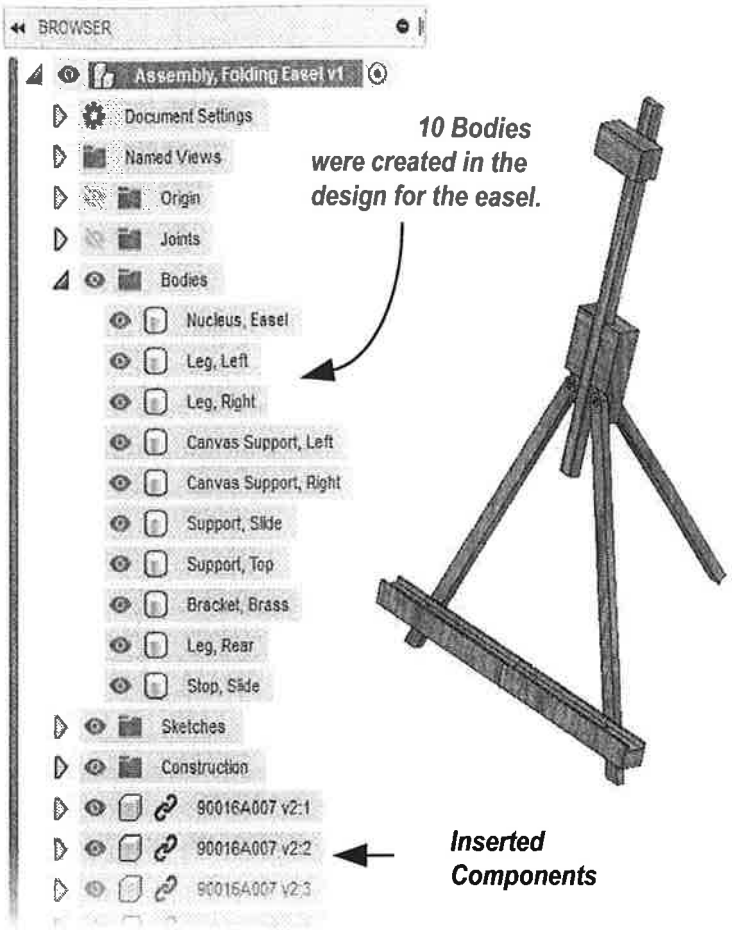

BROWSER
Assembly, Folding Easel v1
Document Settings
Named Views
Origin
Joints
Bodies
Nucleus, Easel
Leg, Left
Leg, Right
Canvas Support, Left
Canvas Support, Right
Support, Slide
Support, Top
Bracket, Brass
Leg, Rear
Stop, Side
Sketches
Construction
90016A007 v2:1
90016A007 v2:2
90016A007 v2:3
10 Bodies were created in the design for the easel.
Inserted Components

Figure 16–1

16.2 Multi-Body Design Tools

In single body design, the first solid feature in the design is added as Body1, and additional features are joined or cut away from this body as the geometry is created. You create a new body using the standard modeling tools (e.g., Extrude, Revolve, Loft, etc.).

When using a multi-body design method, you must use the *Operation* drop-down list to explicitly create a new feature as a separate body. Figure 16–2 shows a new extruded feature being added with the **New Body** *Operation* selected to create the feature as a new body.

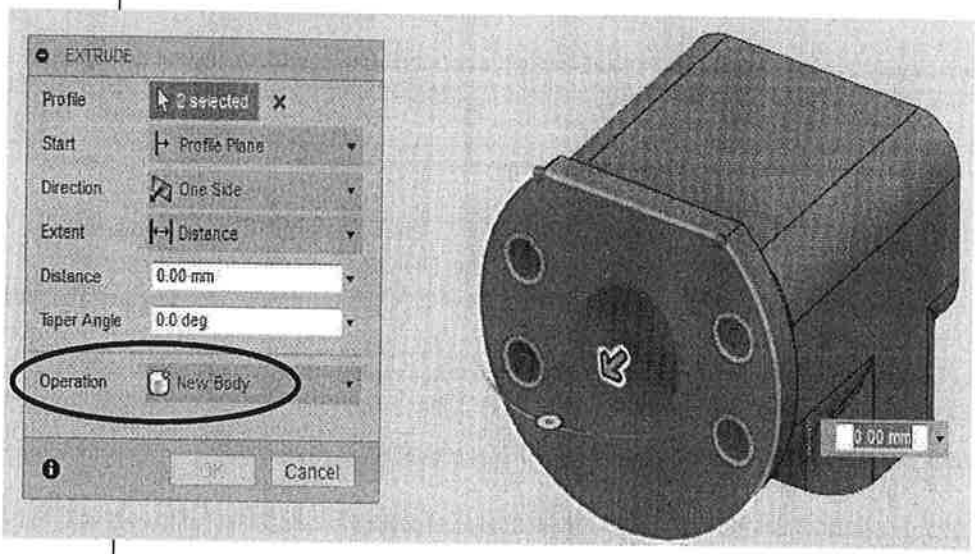


Figure 16–2

In addition to defining a new body based on the Operation definition in a feature, you can also create and manipulate bodies using the following tools:

The feature used to create a body still controls the geometry's size, even if the geometry is split in two.

- Use the (Split Body) tool on the MODIFY panel to split an existing body into two bodies. This is done by selecting a existing body to split (*Body to Split*) and a reference that will be used to split the selected body (*Splitting Tool*). The splitting tool can be a profile, face, or plane that divides the body. You can also select multiple non-intersecting bodies as the splitting tool. Figure 16–3 shows an example where a design change was required and geometry that was previously created as one body needed to be split. The splitting tool in this example was a construction plane.

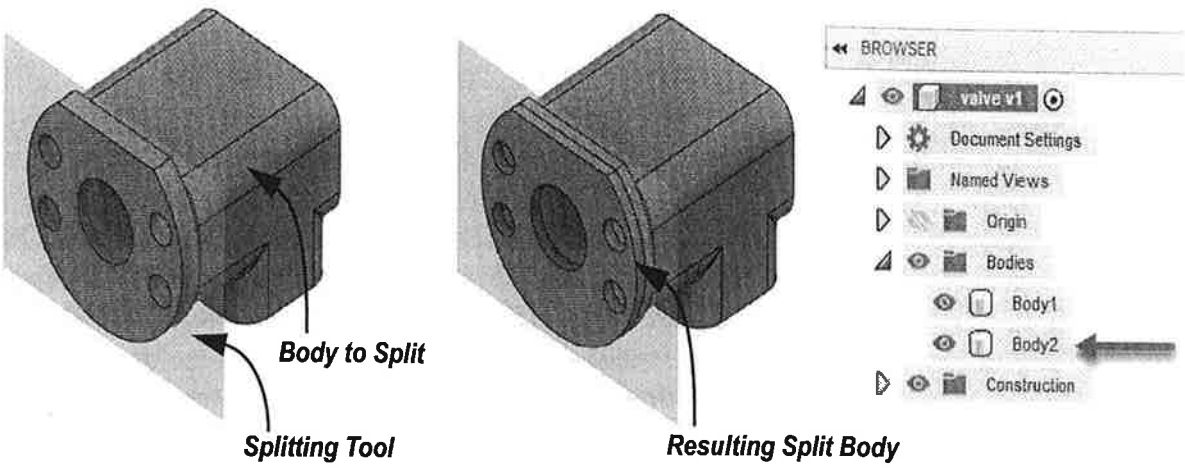


Figure 16–3

- The (Combine) tool on the MODIFY panel enables you to merge existing bodies in the design. This is done by selecting the body to which another body will be added (*Target Body*) and then selecting the body that will be added to the target (*Tool Bodies*). Multiple bodies can be combined at one time using this tool. Once combined, the bodies can be joined, cut, or result in the intersection of overlaying geometry using the *Operation* drop-down list. Figure 16–4 shows an example where a design change was required and geometry that was previously created as two bodies are combined into a single body.

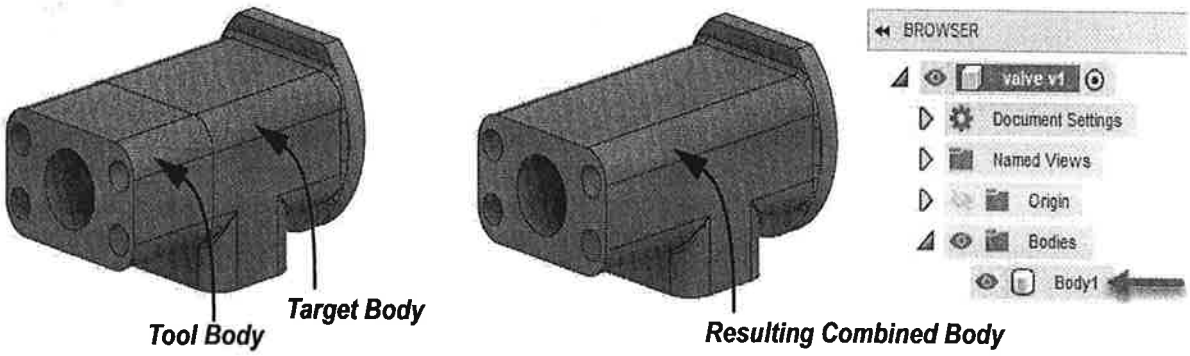
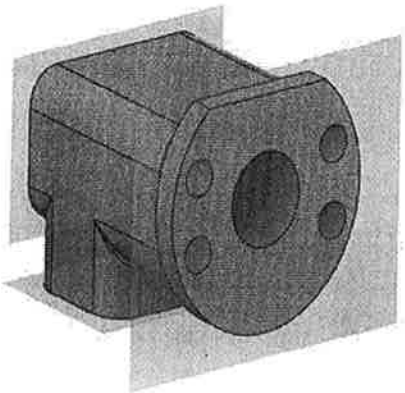
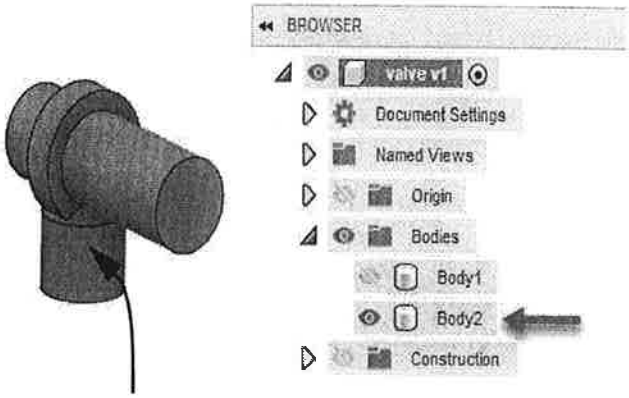


Figure 16–4

- The (Boundary Fill) tool on the CREATE panel enables you to create, join, or remove volumes in a design. This can be done in either the Solid or Surface environments by selecting reference geometry (surfaces, solids, or workplanes) that define single or multiple closed volumes (boundaries). The selected geometry represent the *Tools* and the resulting closed volume(s) are the *Cells*. To complete the definition of the Boundary Fill, you must select which *Cells* to keep by selecting checkboxes on the model. Checkboxes are available for each fully enclosed volume that is generated. Each new cell can be created as a new body, joined to an existing body, or removed from an existing body. Consider using the Boundary Fill tool as an alternative to using the Patch and Stitch options together to create the same geometry. An additional benefit of this tool is that the source surfaces remain in the model, whereas with patching and stitching they are merged and removed. Figure 16–5 shows an example where the Boundary Fill option was used to determine the internal volume of the design. The Properties option can be used on Body2 to calculate its volume.



The 3 planes and Body1 are the selected Tools.



Resulting Boundary Fill (Body2). Body 1 has been turned off for clarity in this image.

Figure 16–5

16.3 Components

Components are used in a design to represent the parts used to form the assembled design. A component can consist of a single body or multiple bodies of features that define the required geometry. Components are either inserted from existing designs, or are created within the context of a multi-body design. Components that are created in the context of the design and that reference other components update when changes are made, enabling you to build design intent into your design.

Creating Components

Components can be created in a design in a number of different ways:

- During geometry creation, in the *Operation* drop-down list, select **New Component**. The component is added to the BROWSER.

Timeline icon: 

- In an existing design file, in the **Bodies** folder, right-click on an existing body and select **Create Components from Bodies**.

Renaming components in the BROWSER can help you to identify them in the design.

- In the CREATE panel, click  (New Component). Using the NEW COMPONENT palette, you can either create a new empty component, or convert an existing body into a new component. Both options are available when you select the tool in the CREATE panel.

- In the BROWSER, right-click on the top-level design file's name and select **New Component**. This creates an empty component in the design. To create a subassembly type structure in a component, right-click on a component name and select **New Component**. Alternatively, you can drag and drop components to create a subassembly structure.

Any new geometry or components that are added to a design are added to the active level. By default, the top-level of the design is active. To activate another component, in the BROWSER, hover the cursor over the component and select the circle shown in Figure 16–6. The active component is displayed and all other components are faded (disabled). Any new geometry or components are added to the active component.

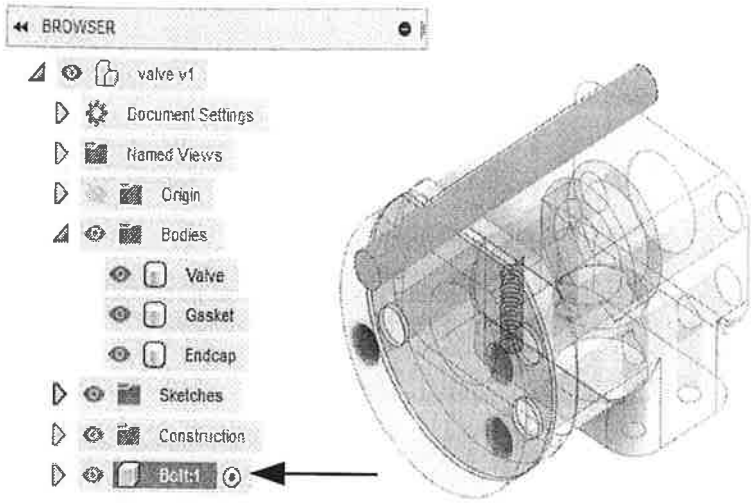


Figure 16–6

To reactivate the top-level design, in the BROWSER, hover the cursor over the design's name and select the circle adjacent to it, as shown in Figure 16–7.

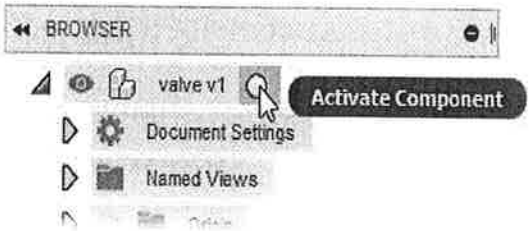


Figure 16–7

Using the BROWSER

Understanding how to use the BROWSER and its options can help you when working with components in a top-level design. There are several tools that can improve selection and visibility in a design.

- The **Show/Hide** and **Selectable/Unselectable** tools enable you to control the visibility and selectability of components. This can make component selection much easier in a complex design.
- Selection sets can be created in the BROWSER to further help with efficient component selection.

- The **Isolate** tool clears the display of all except the selected component.
- Opacity can be set for components to enable you to see through the component. Note that this is only a visual setting: components that have an opacity value assigned to them function as a normal component.

The component's properties are accessed in the BROWSER by right-clicking on the component and selecting **Properties**. This provides you with details on the area, density, and mass of the design based on its assigned material.

Reusing Components

Components can be easily reused in a design to prevent you from needing to create the same geometry multiple times. Reusing components ensures that changes made to the source component is reflected in all of the copies. To reuse components, consider using either a pattern or the copy and paste functionality.

- On the CREATE panel, the **Pattern** tool enables you to pattern selected components in a rectangular or circular shape or along a path. This tool functions similar to those used for geometry patterns.
- On the CREATE panel, the **Mirror** tool enables you to mirror a selected component about a mirror plane, which duplicates a component in a design.

*The **Save Copy As** tool can be used to create new components. It does not maintain associativity with the original design.*

- Components can be copied and pasted in a design. To copy, in the BROWSER, right-click on the component name and select **Copy**. To paste, right-click in the graphics window and select **Paste**. Once pasted, the component displays and the MOVE/COPY palette opens. Use the triad to locate the new component in the design.

Any duplicated component is added to the BROWSER as a new component. It uses the same name as its parent with an instance number added to make it unique. Joints must be added to each duplicated instance to constrain them in the design.

Positioning Components

Using the multi-body design method, components are generally created in their required locations. Even though the components are created relative to one another, they are not automatically constrained in the design. Consider the following:

- Use the **Ground** tool to lock a component in space. The grounded component in the design should be the component that forms the foundation of the design and does not move.
- Components can be moved by selecting and dragging them in the graphics window.
  - To move components using exact translation and rotation values, use the **Move/Copy** tool.
  - Moving does not establish a relationship between components, it only helps to locate them relative to one another.
  - In the POSITION panel, click  (Capture Position) to lock a component in position as a Timeline item.
- Use the **Align** tool to align components relative to one another to help position them in the design. Similar to moving components, aligning does not establish a relationship between components, it only helps to locate them relative to one another.

To fully constrain components in the design you must add joints. Once aligned, you can also use as-built joints.

16.4 As-Built Joints

When the **Joint** tool is used to constrain two components relative to one another, a joint origin must be selected on both components to fully define it. In a multi-body design, the bodies that are converted to components are already in position based on how they were created. Using the **As-built Joint** tool, joint origin selection is not required, and these references are assumed based on how the selected components are positioned. The only references that are required include the selection of the joint type, and a position reference if the joint permits motion.

Timeline icon:

The design shown in Figure 16–8 was created using the multi-body design method. The valve was created first as a single body. The gasket was created as a second body in the design using the face of the valve as its sketch plane and projected edges. Based on its construction, the Rigid as-built joint can be easily assigned by selecting the two components.

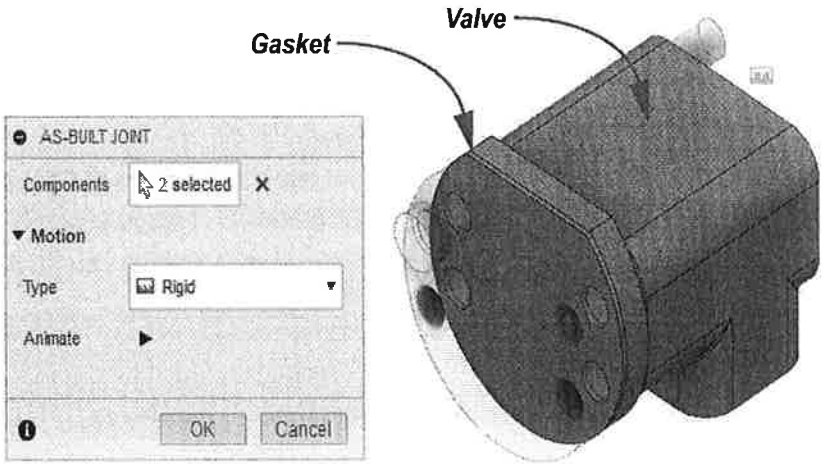


Figure 16–8

In Figure 16–9, the End Cap and Bolt are being constrained using the Revolute as-built joint. The connected faces were defined based on how the two components are positioned to one another. The revolution axis is defined by selecting the center point on the bolt.

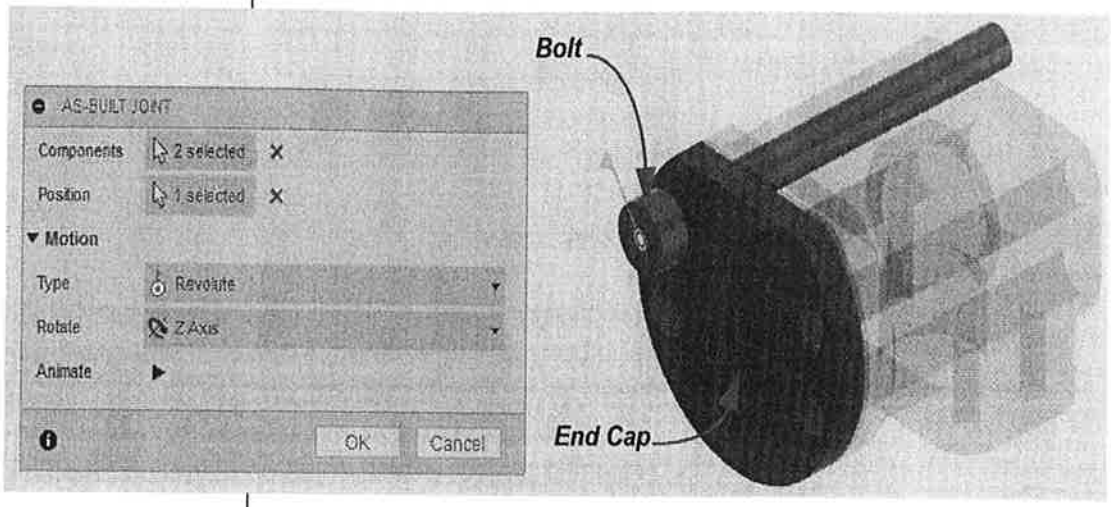


Figure 16–9

Hint: Using As-built Joints for Inserted Components

As-built joints typically work with components that are created directly in the design using the multi-body method. However, you can also use the **As-built Joint** option with inserted components. This is especially helpful if the component was inserted and moved into its exact position using the Move triad when it was placed.

Practice 16a

Multi-Body Design

Practice Objectives

- Use the multi-body design method to create multiple bodies in a design to represent the components of an assembly.
- Convert bodies to components in the design.
- Create as-built joints.

In this practice, you will begin by opening a design that contains a single body that was imported from another CAD software tool. Using references on this body, you will create subsequent bodies that will each represent components in an overall assembly design. To complete the practice, you will convert each body to components and add as-built joints to constrain the design. The completed design is shown in Figure 16–10.

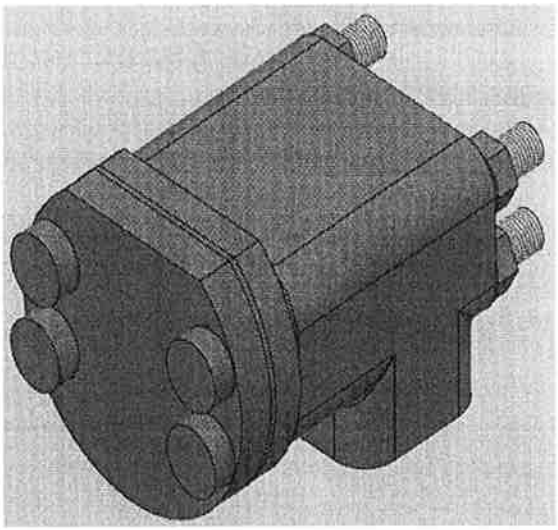


Figure 16–10

Task 1 - Open a design file.

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\MultiBody_Design* folder, select **valve.f3d**, and click **Open**.
3. Save the design as **Valve** in your *Autodesk Fusion 360 Practice Files* project.

Task 2 - Create additional bodies in the design.

In this task, you will create a new body in the design using references on the existing geometry.

1. Ensure that the Timeline is displayed at the bottom of the graphics window. If it is not, at the top of the BROWSER, right-click on the valve name and click **Capture Design History**.
2. In the BROWSER, expand the **Bodies** folder and note that there is currently a single body in the design called **Body1**.
3. Rename the **Body1** body to **Valve**.
4. If not already set, change the *Visual Style* display to **Shaded with Visible Edges Only**. This helps you distinguish the bodies in the design.
5. Create a new sketch on the face shown in Figure 16–11.

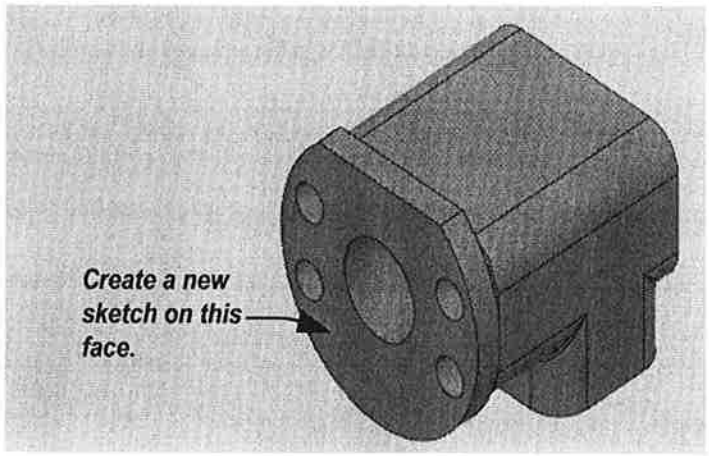


Figure 16–11

6. Complete the sketch. All of the edges that lie on the sketch plane define the sketch.

7. Begin the creation of a new Extrude feature. Select profiles from the sketch so that the section being extruded is as shown in Figure 16–12.

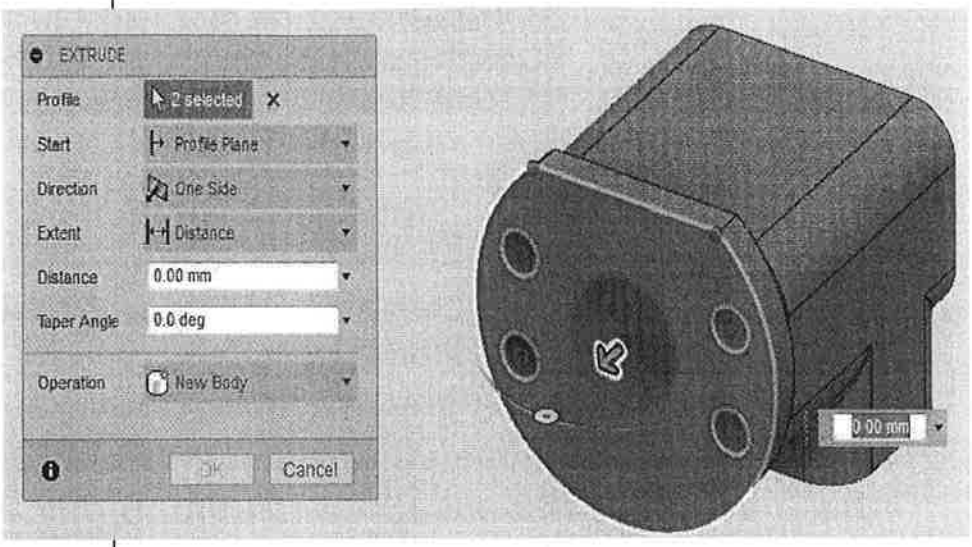


Figure 16–12

8. Drag the depth value to **20 mm**, or enter this value in the *Distance* field.

9. In the *Operation* drop-down list, note how **Join** is automatically set as the default value. Using this option would add the feature to the Valve body.

You can edit features to change the Operation if you do not select it correctly initially.

10. In the *Operation* drop-down list, select **New Body**. This creates the new feature as its own body.

11. Click **OK**.

12. Rename the newly created body to **EndCap**. The BROWSER and design display as shown in Figure 16–13.

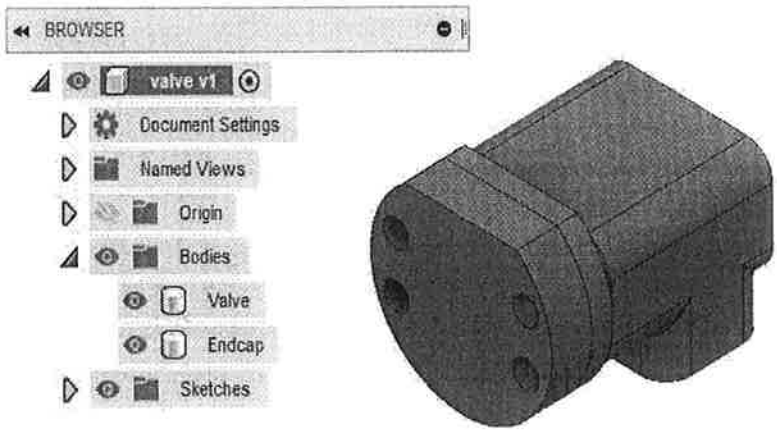


Figure 16–13

Task 3 - Split an existing body.

Prior to creating the EndCap, a gasket should have been created. This can be done using a number of different techniques. In this task, you will split the EndCap body to create a new body that represents the gasket.

1. Toggle off the display of the EndCap body by selecting (**Show/Hide**) adjacent to its name in the browser.
2. Create a construction plane that is offset 2 mm from the face shown in Figure 16–14.

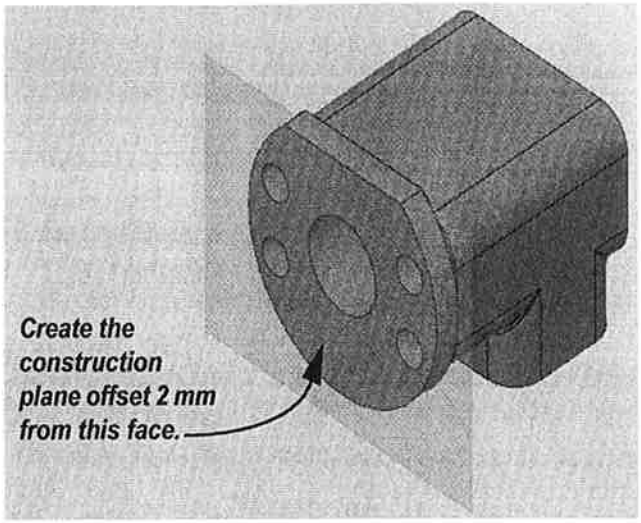


Figure 16–14

3. In the Timeline, drag the newly created construction plane's icon so that it is listed before the extruded feature that was created to form the EndCap body.
4. Toggle on the display of the EndCap body.
5. In the MODIFY panel, click (Split Body). The SPLIT BODY palette displays as shown in Figure 16–15.

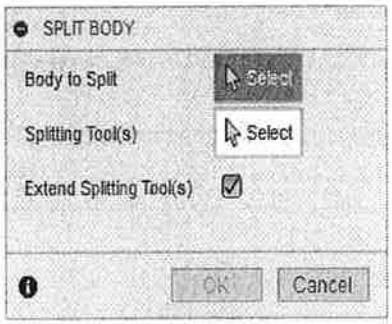


Figure 16–15

6. By default, the *Body to Split* field is active. Select **EndCap** as the body to be split.
7. Select the *Splitting Tool* field to activate it and then select the construction plane that was just created.
8. Click **OK** to split the body.
9. Toggle off the display of the offset construction plane.
10. In the BROWSER, rename the bodies so that the body between the Valve and End Cap bodies is called **Gasket**, as shown in Figure 16–16.

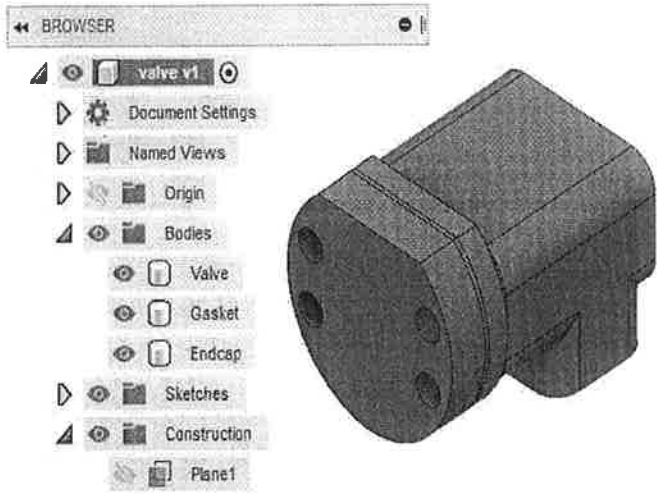


Figure 16–16

11. In the Timeline, edit the Extrude feature. Note that this feature is still 20 mm. Change this to **14 mm**. This value defines the combined depth of the EndCap and Gasket bodies. Because the split plane's location does not change, the Gasket's depth stays at 2 mm, while the EndCap's depth changes to 12 mm.

Task 4 - Create a fastener for the EndCap.

In this task, you will create a bolt for the design. The bolt could be created as a separate body (similar to how the EndCap and Gasket were created), but in this task, you will create the bolt as its own component. This is an alternate approach that you can use if you know that each body is its own component and means that you do not have to convert the body to a component later.

1. Create a new sketch on the face shown in Figure 16–17.

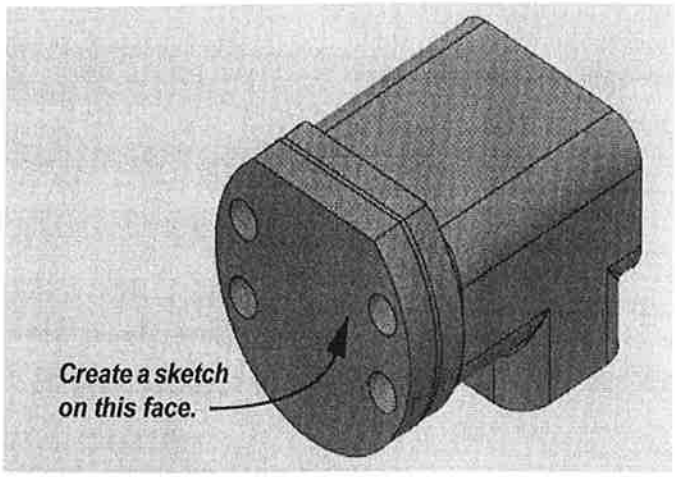


Figure 16–17

2. Complete the sketch.
3. Create the extruded geometry shown in Figure 16–18. For the *Distance* value, enter **-130.00 mm**.
4. In the *Operation* drop-down list, select **New Component**, as shown in Figure 16–18. This is an alternative to creating the body and then converting it to a component.

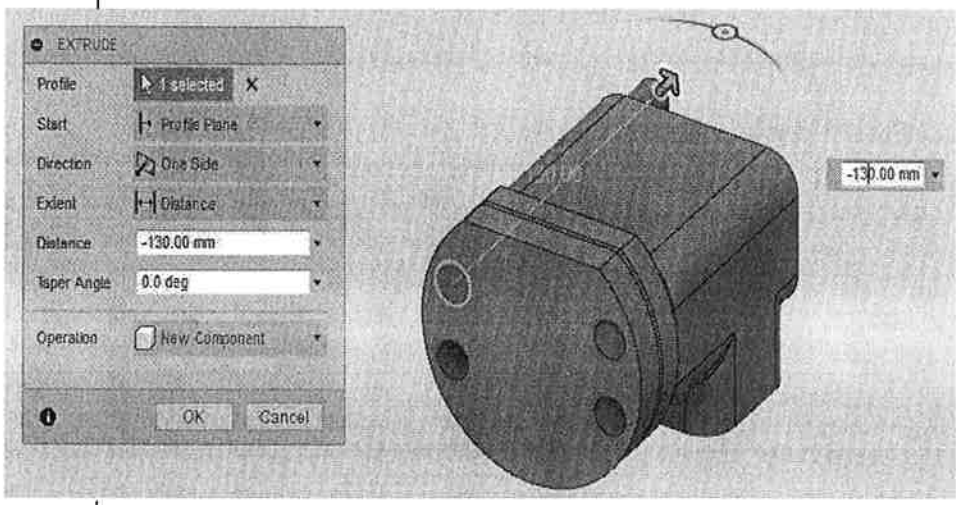


Figure 16–18

5. Click **OK** to create the new component.
6. The new component is added to the bottom of the BROWSER as **Component1**. Rename it to **Bolt**.

7. In the BROWSER, hover the cursor over the **Bolt:1** component and select the circle, as shown in Figure 16–19. All other components are disabled and the shaft of the Bolt is activated.

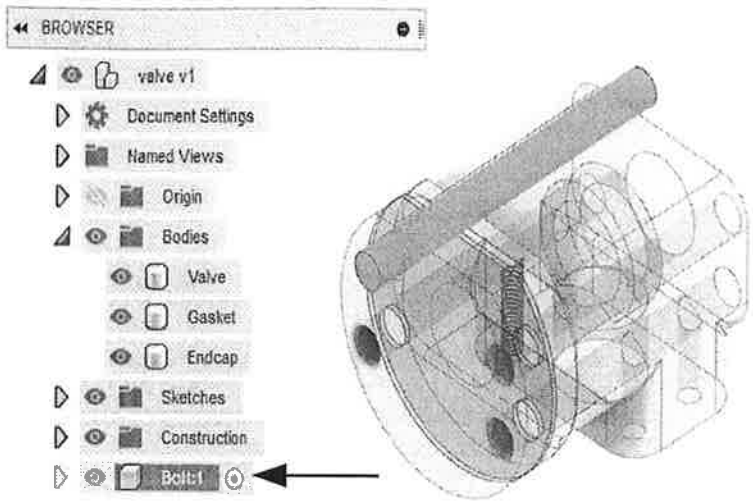


Figure 16–19

8. Using standard geometry creation tools, create the head of the bolt, similar to that shown in Figure 16–20.

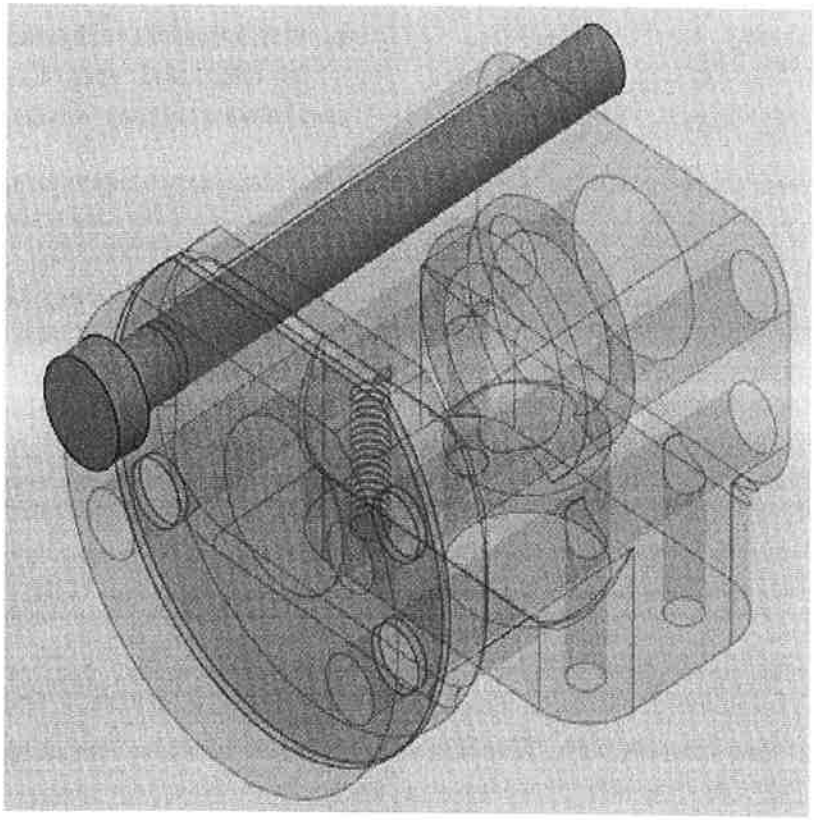


Figure 16–20

9. At the top of the BROWSER, hover the cursor over the top node in the design (**valve**). Select the circle adjacent to its name to reactivate the top-level of the design, as shown in Figure 16–21.

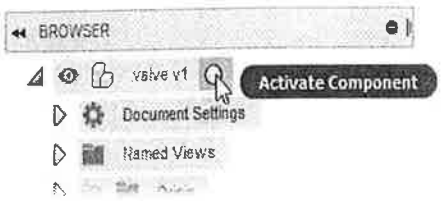


Figure 16–21

10. Save the design.

Task 5 - Convert the bodies to components.

In this task, you will convert each of the three bodies that remain in the design into components. Once converted, you will rename these components.

1. In the **Bodies** folder at the top-level of the design, select **Valve**. Note the geometry that is highlighted in the design.
2. In the **Bodies** folder at the top-level of the design, right-click on **Valve** and select **Create Component from Bodies**, as shown in Figure 16–22.

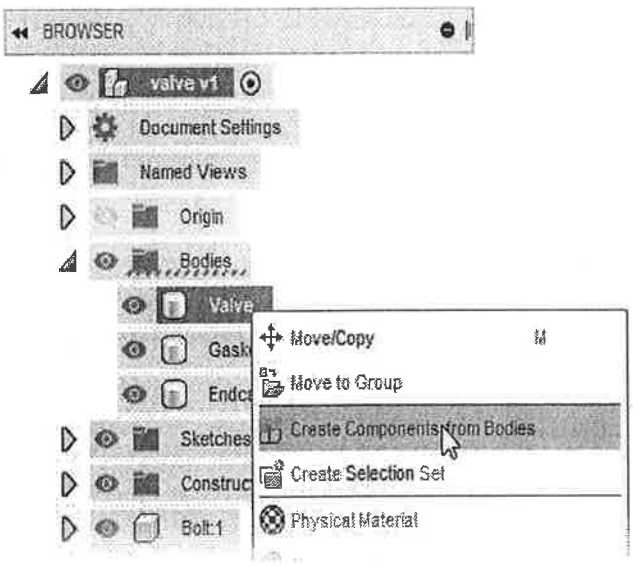


Figure 16–22

3. The **Valve** body is removed from the **Bodies** folder and is listed as a new component named **Valve**. It is identified in the BROWSER using  .

4. Select the component and note how it is the same geometry as the Valve body.

5. Repeat Step 2 for both the **Gasket** and **EndCap** bodies to convert them to components. Once completed, the BROWSER should display as shown in Figure 16–23.

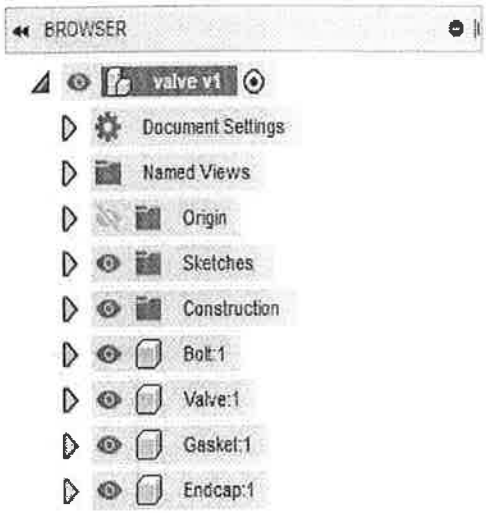


Figure 16–23

6. At the top of the BROWSER, note that  is displayed adjacent to the name of the valve design. This indicates that it consists of multiple components (representing an assembly).

7. Save the design.

Task 6 - Define the as-built joints in the design.

The bodies that were used to create the components in this design were created by referencing existing faces. Because of this design approach, the components are all in their required locations. Using the **As-built Joint** option enables you to define the joint based on their current location, eliminating the need to select joint origins.

1. In the BROWSER, right-click on the **Valve** component and select **Ground**. This sets the valve as the grounded component in the design.

2. In the ASSEMBLE panel, click (As-built Joint). The AS-BUILT JOINT palette opens, as shown in Figure 16–24.

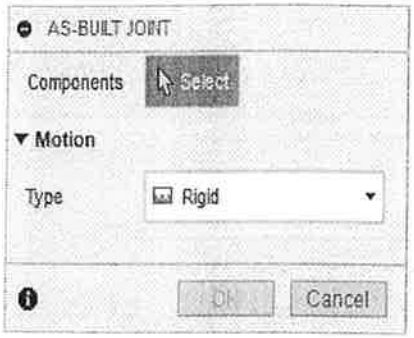


Figure 16–24

3. In the AS-BUILT JOINT palette, in the *Type* drop-down list, select **Rigid**.

4. In the BROWSER or in the graphics window, select the Valve and Gasket components, as shown in Figure 16–25. A shaking animation displays indicating that the two components are now rigidly connected.

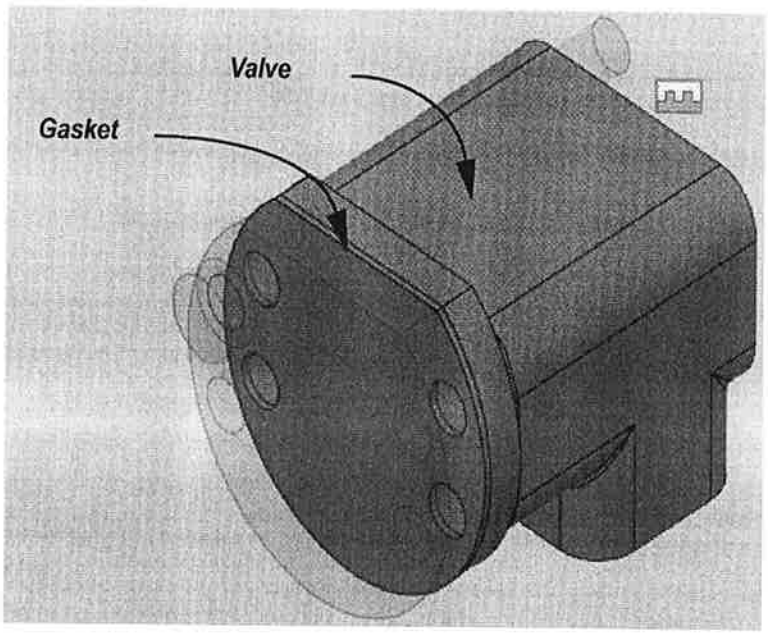


Figure 16–25

5. Click **OK**. Note that the joint is now listed in the BROWSER in the **Joints** folder.

6. Rename the joint as **Valve/Gasket**. This enables you to easily identify the joint.

7. In the ASSEMBLE panel, click (As-built Joint) again.

8. In the AS-BUILT JOINT palette, in the *Type* drop-down list, ensure that **Rigid** is still selected.

9. Select the Gasket and EndCap components, as shown in Figure 16–26. A shaking animation is displayed, indicating that the two components are now rigidly connected.

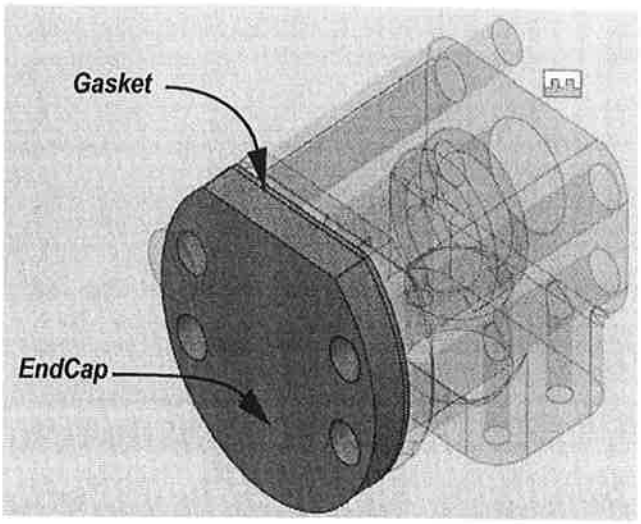


Figure 16–26

10. Click **OK**.

11. Rename the joint as **Gasket/EndCap**.

12. Drag the Bolt component and note that it can still move in the design. Undo the move.

In general, fasteners are joined using rigid joints to represent its final assembled DOF.

13. Using the **As-built Joint** command, connect the Bolt and the EndCap components using a rigid joint. Rename the joint to **Bolt/EndCap**.

Task 7 - Copy the Bolt component in the design.

In this task, you will duplicate the Bolt to the three other holes. As in most designs, this can be accomplished a number of ways. You can copy and paste the component and use joints to move them into position, or you can pattern the components. In this task, you will use both methods.

1. In the BROWSER, right-click on **Bolt** and select **Copy**.

2. In the Graphics Window, right-click and select **Paste**. Drag the new component away from the design and click **OK**.

3. Use the **Joint** command to constrain the new Bolt component to the hole shown in Figure 16–27 using appropriate joint origin references.

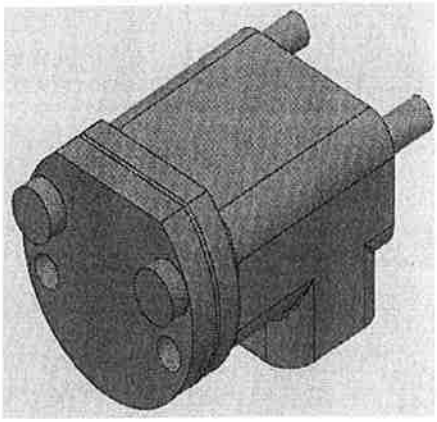


Figure 16–27

As an alternative to copying and pasting the bolt again, you will now use the Pattern tool.

Hint: Consider turning off the display of the existing Bolt components when measuring to ensure that you select the center of the holes.

4. Using the **Measure** tool, measure the distance between the top row of holes and the bottom. Ensure that this distance is 25.4 mm. This distance is required for patterning the bolts.
5. In the CREATE panel, click **Pattern>** (Rectangular Pattern).
6. In the RECTANGULAR PATTERN palette, in the *Pattern Type* drop-down list, select **Components** and select the two Bolt components in the design, as shown in Figure 16–28.
7. Select the *Directions* field to activate it. Select the top linear edge on the EndCap to define the direction reference for the pattern, as shown in Figure 16–28.
8. Select the patterning direction arrow that is perpendicular to the selected edge, as shown in Figure 16–28.

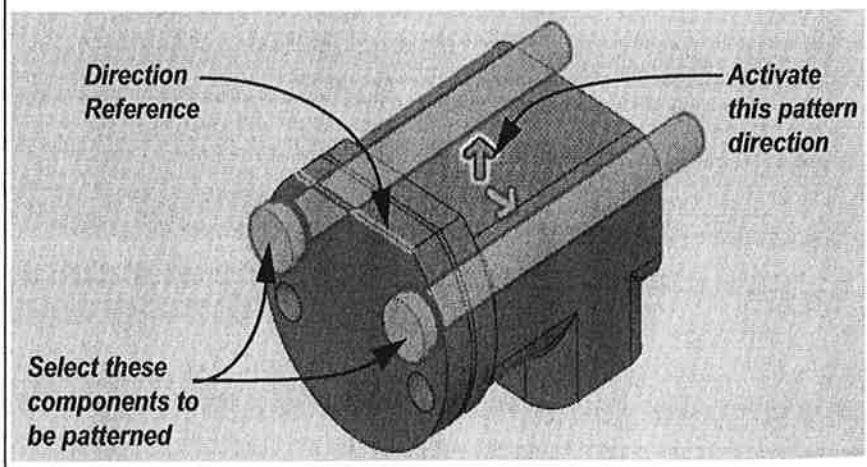


Figure 16–28

9. For the number of instances to pattern, enter **2** in the entry field that displays.
10. For the *Distance* value, enter **-25.4 mm** in the entry field that displays.
11. Click **OK** to complete the pattern.
12. Use the **As-built Joint** tool to rigidly constrain the remaining bolts.
13. Save the design.
14. Close the file.

As an alternative, you could also use the pattern tool to create the bolt that was initially copied. You would do this by patterning in two directions.

Task 8 - (Optional) Complete the design.

If time permits, complete the design as follows:

- Use the multi-body design approach to create the washer and nut components shown in Figure 16–29.
- Fully constrain any newly created components in the design to remove all degrees of freedom.
- Add a thread to the Bolt component, as shown in Figure 16–29.

To help identify the components in the design, the system can assign default colors to each component. In the INSPECT panel, click  *(Component Color Cycling Toggle). This temporarily assigns colors to the design until the option is cleared.*

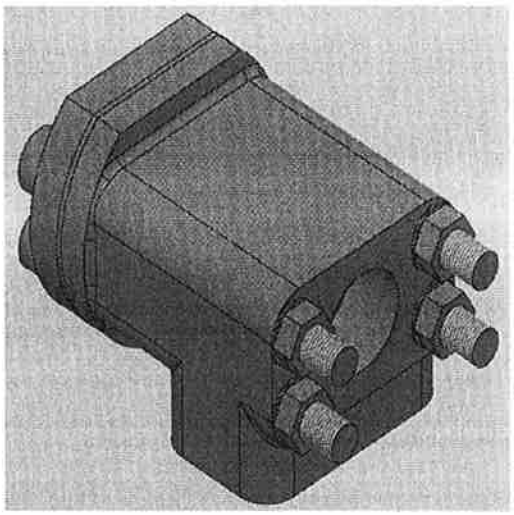


Figure 16–29

Practice 16b

Working with Multi-Bodies to Create an Assembled Design

Practice Objectives

- Upload a .F3Z file to your Autodesk account.
- Convert bodies to components in the design.
- Create as-built joints.
- Enable contact analysis and create contact sets between components.
- Add a motion link condition to joints in the design.

In this practice, you will work in a design that has been created for you. To complete this design you will convert bodies to components and set up as-built joints between the components to create a folding easel design. Using the tools that are available on the ASSEMBLE panel, you will also set up a motion link and contact sets to test the easel's range of motion. The completed design is shown in Figure 16–30 in its expanded and collapsed state.

Fasteners have been added into the design and rigid joints have already been set up for you.

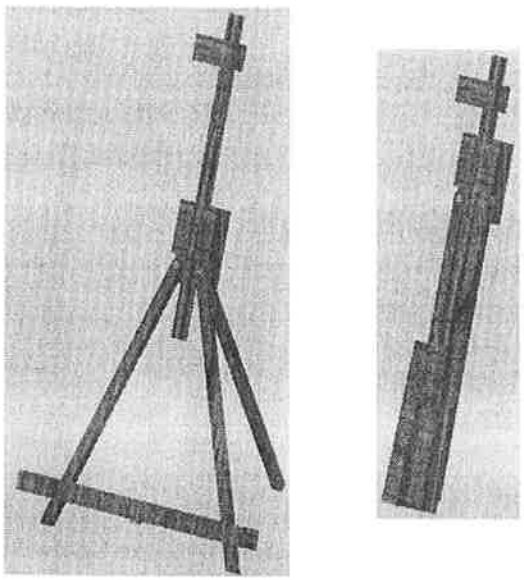


Figure 16–30

Task 1 - Upload a design file to your project.

In this task, you will upload a design into your Autodesk Fusion 360 project. The design file that is provided was designed using the multi-body design method. Additionally, fasteners pulled from the McMaster-Carr Content Library have been inserted.

1. Open the Data Panel and ensure that the active project is *Autodesk Fusion 360 Practice Files*.

*A *.F3Z file is a compressed, archived file. It enables you to export a design file and any referenced components to share with other users. Once imported into your account, the files are automatically extracted.*

2. Create a new folder in the project called **Folding Easel** and then open the folder.
3. At the top of the Data Panel, click **Upload**.
4. In the Upload dialog box, click **Select Files**. Navigate to the practice files folder on your local drive and in *Folding_Easel* folder, select **Folding Easel.f3z**. Click **Open**.
5. Click **Upload** to begin the upload. The Job Status window shows the progress of the upload. All of the components that are in the .F3Z file are shown in the list.
6. Once the files have been uploaded, click **Close** in the Job Status window. The files should display as shown in Figure 16–31. Note that you might need to refresh the display.

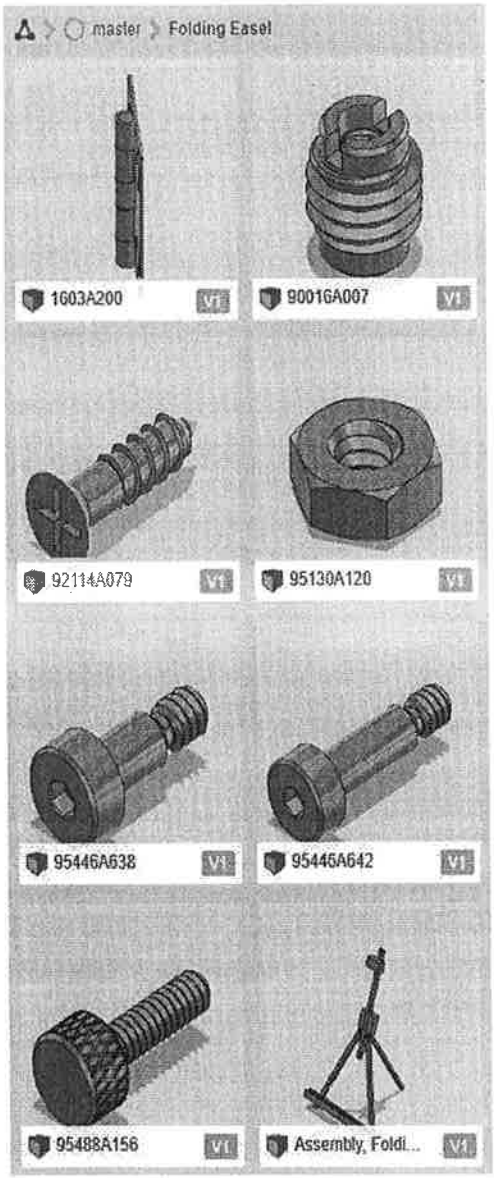


Figure 16–31

7. Open the **Assembly, Folding Easel** design. The BROWSER and design display as shown in Figure 16–32.

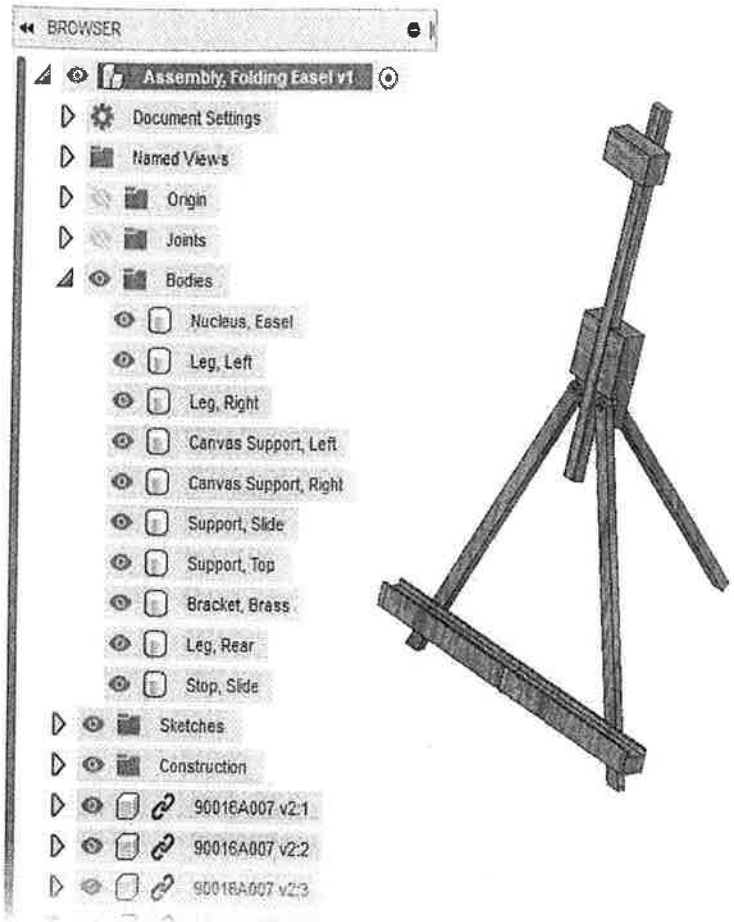


Figure 16–32

8. Complete the following in the BROWSER:
   - Select some of the components. Note how these all represent the fasteners that join the wooden pieces. None of the inserted components represent the geometry of the easel.
   - Hide all of the fasteners from the display. By hiding them from the display, it will ease reference selection during joint definition.
   - Expand the **Bodies** folder, if not already expanded. Note that there are 10 bodies in the folder. Select some of the bodies. They were all created in the context of this design and represent each component in the folding easel assembly.

By default, the naming convention for bodies in a design is Body1, Body2, etc. Each body was renamed in this design for clarity.

Task 2 - Convert the bodies to components.

In this task, you will convert each of the bodies that have been created into components.

1. In the **Bodies** folder, right-click on **Nucleus, Easel** and select **Create Component from Bodies**.
2. The **Nucleus, Easel** node is removed from the **Bodies** folder. Scroll to the bottom of the BROWSER. A new component named **Nucleus, Easel** has been added. It is identified with a  in the BROWSER.
3. In the **Bodies** folder, hold <Ctrl> and select the remaining nine bodies. Right-click and select **Create Component from Bodies**. Once completed, the bottom of the BROWSER should display as shown in Figure 16–33. The 10 bodies are now 10 separate components in the design.

Figure 16–33

Task 3 - Define rigid as-built joints in the design.

In this task, you will ground the Nucleus, Easel component, and then add a joint between the Nucleus, Easel and Bracket, Brass and the Support, Slide and Support, Top components. Both of these will be assigned using the Rigid joint type, as no motion is required between these sets of components.

A grounded part is fixed to a location in the assembly and is not dependent on other parts.

1. In the BROWSER, right-click on **Nucleus, Easel:1** and select **Ground** to ground this component.
2. In the BROWSER, expand the **Joints** folder. The rigid joints that are listed were added to constrain the fasteners in the assembly.

Note: In the upcoming images for joint creation, components that are not needed in the image have been suppressed. This was done to help you to easily identify what is being selected.

3. In the ASSEMBLE panel, click (As-built Joint). The AS-BUILT JOINT palette opens.
4. In the AS-BUILT JOINT palette, in the *Type* drop-down list, ensure that **Rigid** is selected.
5. In the BROWSER or in the graphics window, select the Nucleus, Easel and Bracket, Brass components, as shown in Figure 16–34. A shaking animation is displayed indicating that the two components are now rigidly connected.

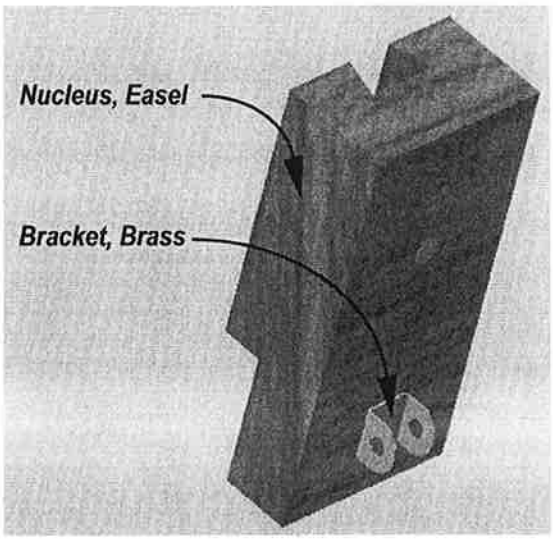


Figure 16–34

6. Click **OK**. Note that the joint is now listed in the BROWSER in the **Joints** folder.
7. Rename the last joint that was created as **Easel Nucleus/Brass Bracket**. This enables you to easily identify the joint.
8. Create an as-built Rigid joint between the Support, Slide and Support, Top components, as shown in Figure 16–35. Rename the joint as **Slide/Top Support**.

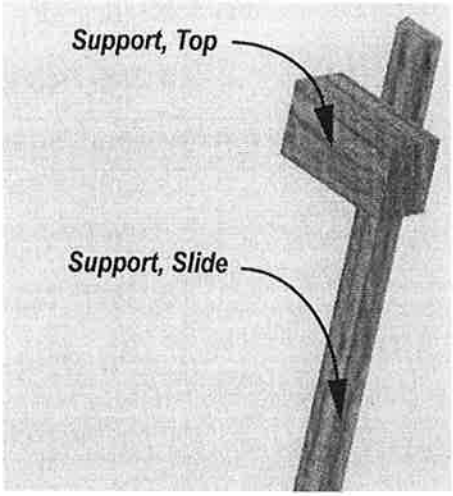


Figure 16–35

Task 4 - Define a Slider as-built joint in the design.

In this task, you will add a joint between the Support, Slide and Nucleus, Easel components that permits sliding motion. You will also assign limits on the range of motion for the Support, Slide relative to the grounded Nucleus, Easel component.

1. Create a Slider as-built joint between the Support, Slide and Nucleus, Easel components, as shown in Figure 16–36.
2. For the *Position* reference, select the face shown in Figure 16–36. This defines the sliding direction. Once the *Position* reference is selected, a sliding animation is displayed.

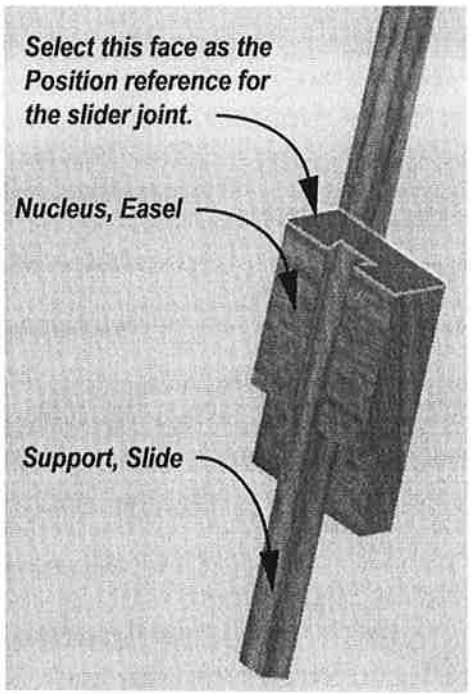


Figure 16–36

3. Click **OK**.
4. Rename the joint as **Easel/Support Slide**.
5. Drag the Support, Slide component and note how you can drag it through each end of the grounded Nucleus, Easel component.
6. Revert the component back to its initial location by selecting (Revert).
7. In the BROWSER, hover over the Easel/Support Slide joint and select (Edit Joint Limits), which displays adjacent to the joint name.
8. The EDIT JOINT LIMITS palette opens. It identifies that the only permitted motion of this joint is sliding, as shown in the *Motion* drop-down list in Figure 16–37.

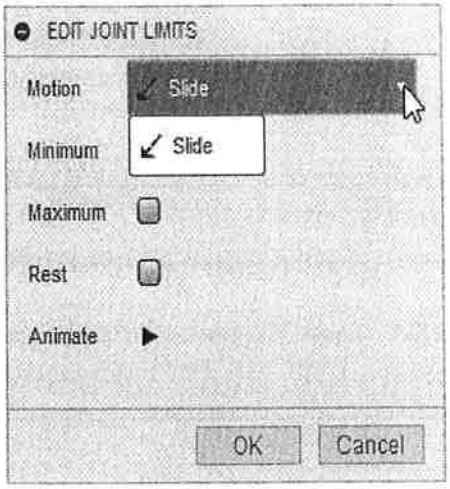


Figure 16–37

9. Select the **Minimum** and **Maximum** checkboxes to enable setting limits on the range of motion.
10. For the *Minimum* value, enter **-3.00 in**.
11. For the *Maximum* value, enter **4.00 in**.
12. Click in the *Animate* area to play the range of sliding motion. Click to stop the animated motion.
13. Click **OK** to complete the limits.
14. Revert the component's position back to where it was initially located.

15. Drag the Support, Slide component and note how the limits are assigned.

Task 5 - Complete the remaining as-built joints.

In this task, you will create the remaining joints in the design to constrain the design for its required motion.

1. Create a new Revolute as-built joint between the Bracket, Brass and Leg, Rear components, as shown in Figure 16–38. For now you will permit a full 360° rotation, even though there is interference. This will be controlled later in the practice.

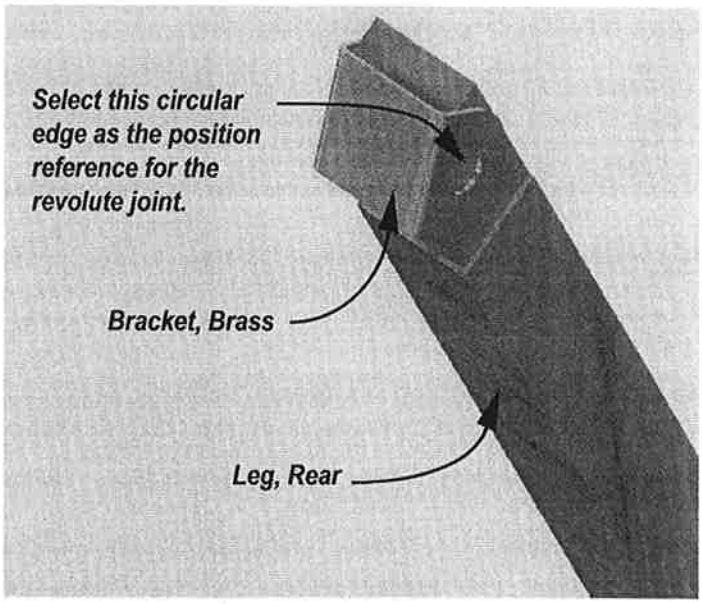


Figure 16–38

2. Rename the Revolute joint as **Bracket/Rear Leg**.

3. Create additional Revolute as-built joints between the following components and rename the created joint as shown below and in Figure 16–39. Ensure that a circular reference that defines the axis of rotation is selected as the *Position* reference.
   - Leg, Left and Nucleus, Easel components (rename the joint as **Left Leg**)
   - Leg, Right and Nucleus, Easel components (rename the joint as **Right Leg**)
   - Canvas Support, Left and Leg, Left components (rename the joint as **Left Canvas Support**)
   - Canvas Support, Right and Leg, Right components (rename the joint as **Right Canvas Support**)

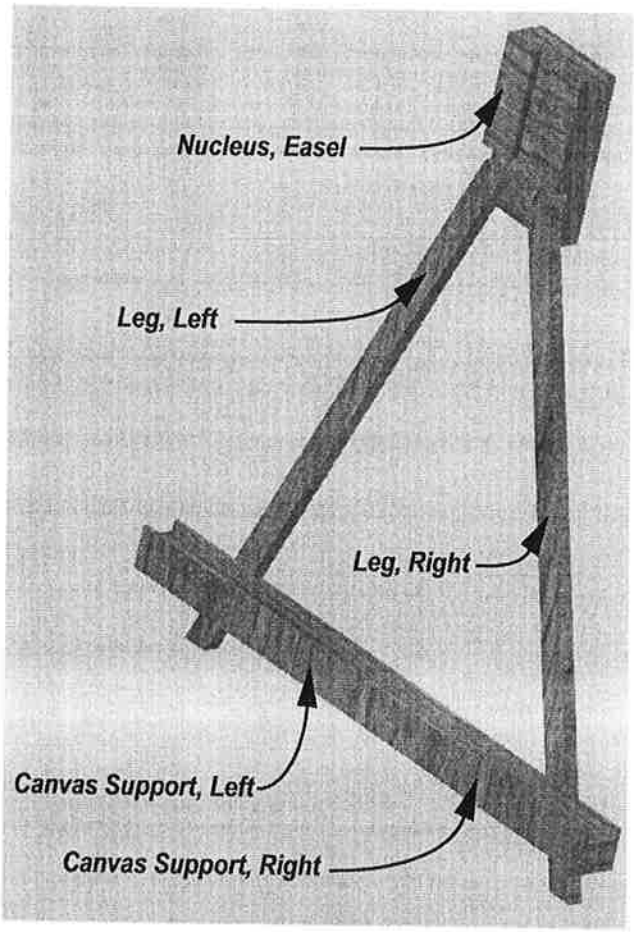


Figure 16–39

4. In the BROWSER, expand the **1603A200** node. This component contains sub-components that have been renamed for you. If the component is hidden, toggle on its visibility.

Joints were created for you in the hinge component that permit the rotational motion of a hinge model.

5. Create additional Rigid as-built joints between the following components and rename the created joint as shown in brackets.
   - Leaf, Right and Canvas Support, Left (rename the joint as **Left Canvas Support Hinge**)
   - Leaf, Left and Canvas Support, Right (rename the joint as **Right Canvas Support Hinge**)
6. Save the design

Task 6 - Limit the motion of the easel using contact sets.

In this task, you will create a contact set to limit the motion of the design.

1. If you cleared any of the components from the display, toggle their display back on so that all components are being displayed.
2. Drag the left or right Canvas Support components to test the motion of the design. Note that the Leg, Left and Leg, Right can swing and interfere with the Support, Slide component. The Leg, Rear can also rotate through 360°.
3. In the POSITION panel, click  (Revert) to return the design to its default position.
4. In the ASSEMBLE panel, click  (Enable Contact Sets).
5. There are no sets currently set up in the model. In the ASSEMBLE panel, click  (New Contact Set). The NEW CONTACT SET palette opens, enabling you to define a new set.
6. In the graphics window, select the Leg, Rear, Bracket, Brass and Nucleus easel components. Click **OK**. Test that the range of motion is now as expected.
7. Create a second contact set that consists of the Leg, Left, Leg, Right, and Support, Slide components. Test the range of motion.
8. Return the design to its default position and save.

Task 7 - Create a relationship between the joints on the right and left legs.

1. In the ASSEMBLE panel, click (Motion Link). The MOTION LINK palette opens.
2. In the BROWSER, select the Left Canvas Support and Right Canvas Support joints.
3. For the *Left Canvas Support,* enter **90 deg**. For the *Right Canvas Support,* enter **-90 deg**.
4. Click **OK** and test the motion by dragging the components to their expanded and collapsed positions, as shown in Figure 16–40.

Note the animated preview of the Motion Link might not display exactly as expected.

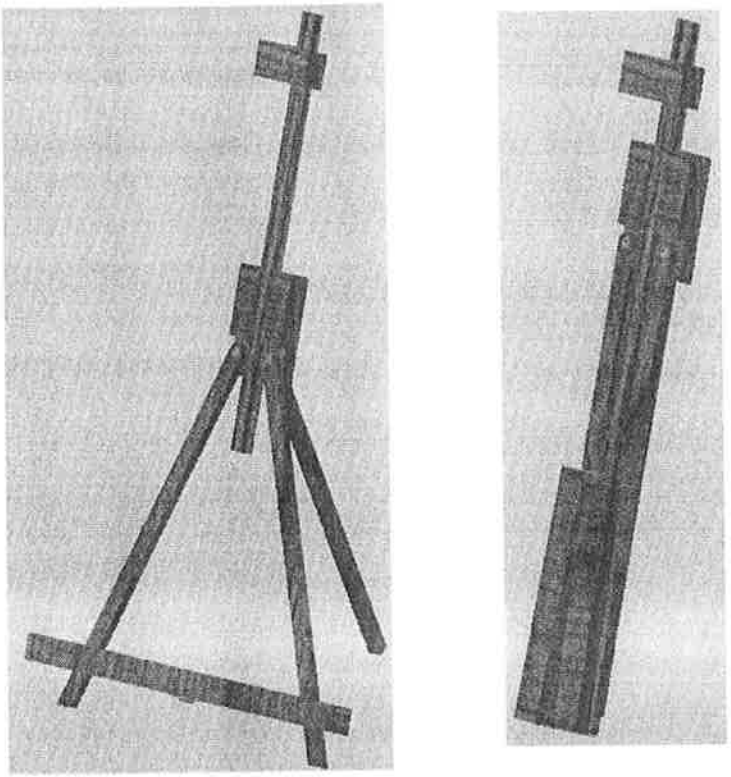


Figure 16–40

5. Save the design and close the file.

To review a completed version of this design, upload the **Folding Easel Final.f3z** file from the *Folding_Easel* folder of your practice files and open **Folding Easel Final** in the Autodesk Fusion 360 software.

Command Summary

Button	Command	Location
	**Align**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> MODIFY panel • **Context Menu**: Right-click in the graphics window and select **Modify**.
	**As-built Joint**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> ASSEMBLE panel • **Context Menu**: Right-click in the graphics window and select **Assemble**.
	**Circular Pattern (feature)**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> CREATE panel • **Context Menu**: Right-click in the graphics window and select **Create**.
	**Combine**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> MODIFY panel • **Context Menu**: Right-click in the graphics window and select **Modify**.
N/A	**Copy**	• **Context Menu:** *With one or more bodies or components selected in the BROWSER*
N/A	**Create Components from Bodies**	• **Context Menu:** *With a body selected in the* ***Bodies*** *folder of the BROWSER*
N/A	**Ground**	• **Context Menu:** *With one or more components selected in the BROWSER*
N/A	**Isolate**	• **Context Menu:** *With one or more components selected in the BROWSER*
	**Move/Copy**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> MODIFY panel • **Context Menu**: Right-click in the graphics window and select **Modify**.
	**New Component**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> CREATE panel • **Context Menu**: Right-click in the graphics window and select **Create**. • **Context Menu:** *With the design's name selected or a component selected in the BROWSER*
N/A	**Opacity Control**	• **Context Menu:** *With one or more components selected in the BROWSER*
N/A	**Paste**	• **Context Menu:** In the graphics window
	**Pattern on Path (feature)**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> CREATE panel • **Context Menu**: Right-click in the graphics window and select **Create**.

N/A	**Properties**	• **Context Menu:** *With one or more components selected in the BROWSER*
	**Rectangular Pattern (feature)**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> CREATE panel • **Context Menu**: Right-click in the graphics window and select **Create**.
N/A	**Selectable/ Unselectable**	• **Context Menu:** *With one or more components selected in the BROWSER*
N/A	**Show All Components**	• **Context Menu:** *With one or more components selected in the BROWSER*
N/A	**Show/Hide**	• **Context Menu:** *With one or more components selected in the BROWSER*
	**Split Body**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> MODIFY panel • **Context Menu**: Right-click in the graphics window and select **Modify**.