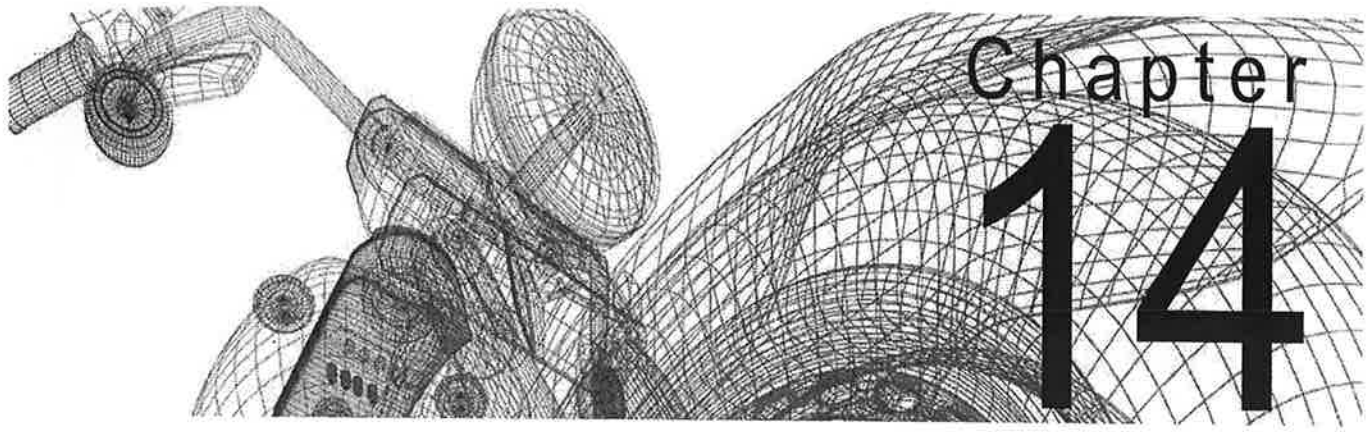


Chapter
14

Distributed Design

A design can consist of multiple components that communicate how an assembled product is designed. In the Autodesk® Fusion 360® software, there are two methods that can be used to design a product with multiple components: components can be combined into a single design, or all components can be created within the context of a design. This chapter discusses the Distributed design method, where components are inserted and joined with one another in a single design.

Learning Objectives in this Chapter

- Describe the methods used to create an assembly in the Autodesk Fusion 360 software.
- Insert components into a design.
- Create joint origins in a model for use in assigning joint references.
- Use the Joint command to connect components in a design while maintaining the defined degrees of freedom.
- Edit a joint connection so that the joint type, references, or its values can be changed.

14.1 Assembly Design Methods

There are two methods that can be used to create a multi-component (assembly) design in the Autodesk Fusion 360 software. These are: Distributed Design and Multi-Body Design.

Components can also be imported from other CAD products or the McMaster-Carr content library.

- **Distributed Design:** Using this method, components are created in separate design files first, and then they are independently added into one design to create the assembled product. Each component is stored separately and is linked into the top-level design. Any changes made in the individual component can be updated in the assembled design.

- **Multi-Body Design:** Using this method, multiple individual bodies are created within the context of a design file. These bodies are then converted to components. Each component represents a separate model in the assembled product. This approach is also known as top-down design.

In either method, components represent all of the unique parts in the multi-component design. The process of creating the components varies depending on the design method used.

Inserting Content from the McMaster-Carr Content Library

McMaster-Carr (www.mcmaster.com) provides many standard components that can be used in Autodesk Fusion 360 designs. These components represent purchasable components that are commonly used in a design (e.g., fasteners, screws, etc.). Models can be downloaded in multiple CAD formats for use in the Autodesk Fusion 360 software. These components can be used for either design method.

- To add a component directly into a design, expand the INSERT panel and click  (Insert McMaster-Carr Component). The component is stored in the design. If the component is needed for another design, you would have to insert it again.

- To save the component as a unique file, download the file from the McMaster-Carr site to your local system. You can then upload the file to your Autodesk 360 account through the Data Panel, at which point it is available for insertion into a design.

14.2 Distributed Design

The distributed design method uses previously created components and designs which are then inserted into another design file. Once in the design, the components are constrained to one another using Joints.

Use the following steps to insert components into a design file:

If the assembly model's design file was newly created, it must be saved before you can insert components.

1. Create a new design and save the file.
2. Open the Data Panel and locate the component to be inserted as the first component in the design.
3. Insert the first component into the design file.
4. Use the MOVE/COPY palette to locate the component in the assembly.
5. Continue to insert components to complete the design.

A combination of distributed design and multi-body design can be incorporated into designing a multi-component model. This is convenient if some components are not individually created and it might be easier to create them in the context of the multi-component design to ensure fit and function.

Inserting Components

To insert a component into a design, in the Data Panel, right-click on the component's thumbnail image and select **Insert into Current Design**, as shown in Figure 14–1.

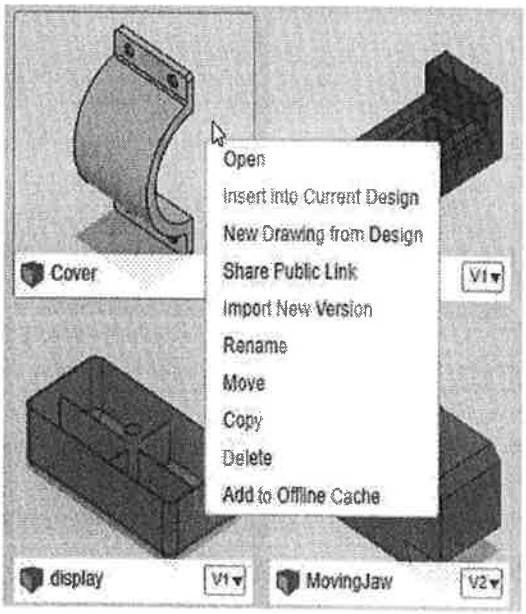


Figure 14–1

Timeline icon:

- Once inserted, you can reposition the component in the file using the MOVE/COPY palette or using the handles on the triad. Click **OK** to place the component.
- Inserted components are listed in the BROWSER. They are identified using (as shown in Figure 14–2), which indicates that there is a link to the source component file.
- To break the link to a component, right-click on the component name(s) in the BROWSER and select **Break Link**. Once broken, you can edit the component in the assembly design file, but the link cannot be restored.

To create subassembly components, insert the components together into one design file and then use that file in a top-level design.

- Inserted components can be either a single component design file or a subassembly where multiple components have been previously combined. In Figure 14–2, the **Body** and **FixedJaw** components are single component design files ( ), whereas the **MovingJaw** component is a subassembly ( ).

Figure 14–2

- Inserted components are read-only in the context of the top-level design. To make a change to geometry in the model, you must return to the source design file.
- If a change is made to a source component, the BROWSER icon updates to indicate that the component is out-of-date ( ). To update an out-of-date component, right-click on the component name and select **Get Latest**. If multiple components are out-of-date, right-click on the design's name and select **Get All Latest**.

MOVE/COPY Palette

The MOVE/COPY palette displays as soon as a component is inserted into a design file. The palette and the accompanying triad (shown in Figure 14–3) can be used to position the component.

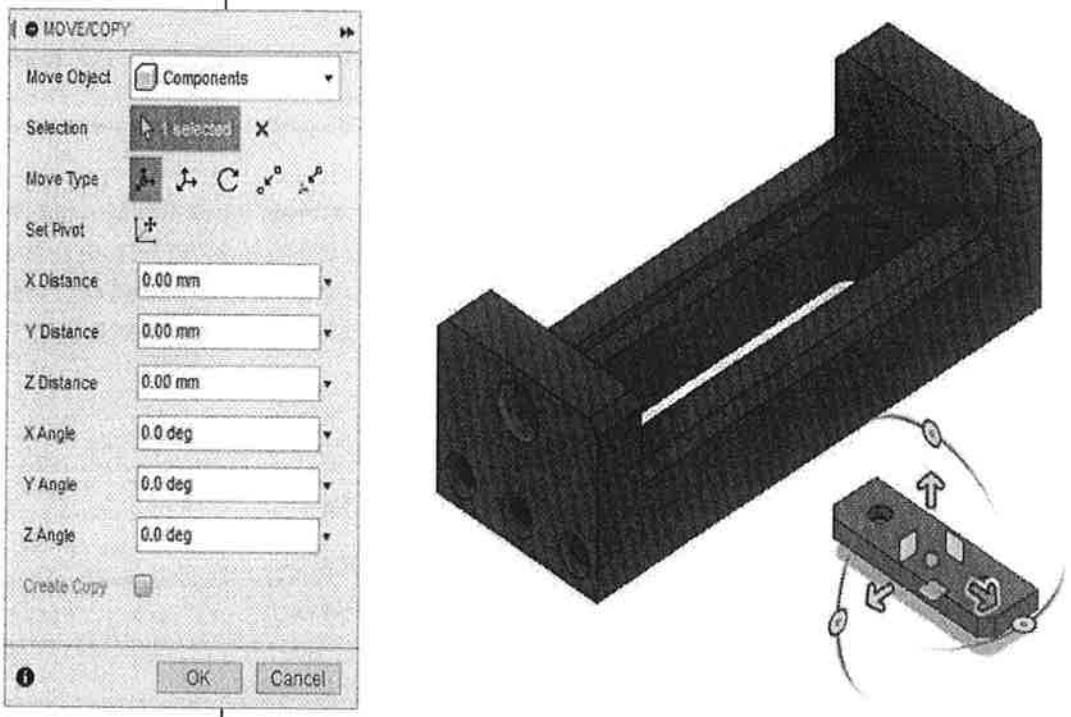


Figure 14–3

*Maintain the **Components** option in the Move Object area to move the newly inserted component. If another option is select the component will be placed automatically at its current location.*

To define the component's position, you can use any of the following methods:

- Drag the handles on the triad to locate the component.
  - Use the arrowhead icon to move linearly in the X, Y, or Z directions.
  - Use the plane icon to move in the XY, XZ, or YZ planes.
  - Use the circular icons to rotate about the X, Y, or Z axes.
- Enter translational or rotational values in the MOVE/COPY palette.
- Activate a handle on the triad and enter a value in the entry field that displays.

Once the position is defined, click **OK** in the MOVE/COPY palette to position the component in the assembly. Note that this does not define how the components are constrained relative to one another, which is done using joints.

Tips for Working with the MOVE/COPY Palette

Consider using the (Point to Point) Move Type option to align specific entities in the components. You can also use the (Set Pivot) option to define a new location for the transform triad on the component.

Grounding Components

Timeline icon:

You can ground components in an assembly to ensure that they form a stable base for the design and do not move relative to other components. To ground a component, in the BROWSER, right-click on the component's name and select **Ground**, as shown in Figure 14–4.

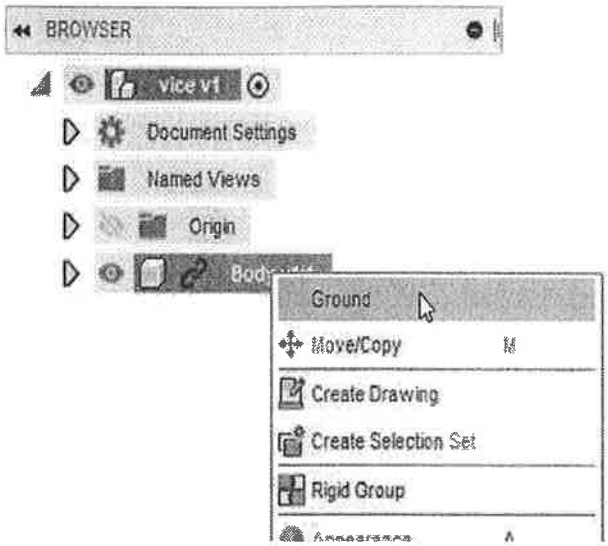


Figure 14–4

- Grounded components are identified in the BROWSER with .
- There can be a single grounded component or multiple grounded components in an assembly. Generally, the most prominent component in the design should be grounded.

14.3 Joint Origins

To apply a joint, you must select reference geometry on both components. The reference geometry that you select on each component is called a *Joint Origin*. A single joint origin on each component is required to fully define a joint. To join the components, the two selected joint origins are aligned to one another.

Selecting Joint Origins

Joint origins can be placed at the time of joint creation. This is done by selecting default snap-points on entities in the model. As you hover the cursor over a component face, edge, or cylindrical or spherical geometry, highlighted snap-points display with the active glyph ( ) attached to the snap-point closest to the cursor. This glyph represents the joint origin and is the point to which the joint is assigned once it is selected. The location of the cursor on the entity controls which glyph is active. If the required glyph is not active, move the cursor closer to another snap-point to activate it.

Face Snap-Points

For planar faces, the snap-points that define the joint origin can be located at one of three locations, as shown in Figure 14–5:

- Center point of the face
- Midpoint of an edge on the face
- Endpoint of an edge on the face

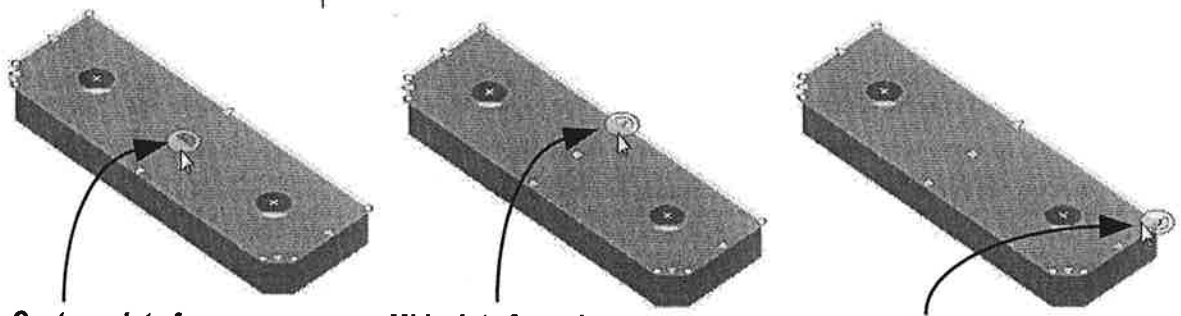


Figure 14–5

Hint: Joint Origin Selection

As you move the cursor on a model the reference entity might change. To lock a reference entity, ensure that you are hovered over the required entity, press and hold <Ctrl> and continue to move the cursor to locate the required joint origin on that single entity.

Edge Snap-Points

For an edge, the snap-points that can define the joint origin can be located at one of two locations, as shown in Figure 14–6.

- Midpoint of the edge
- Ends of the edge

For a circular edge or face, the only possible snap-point is at the center.

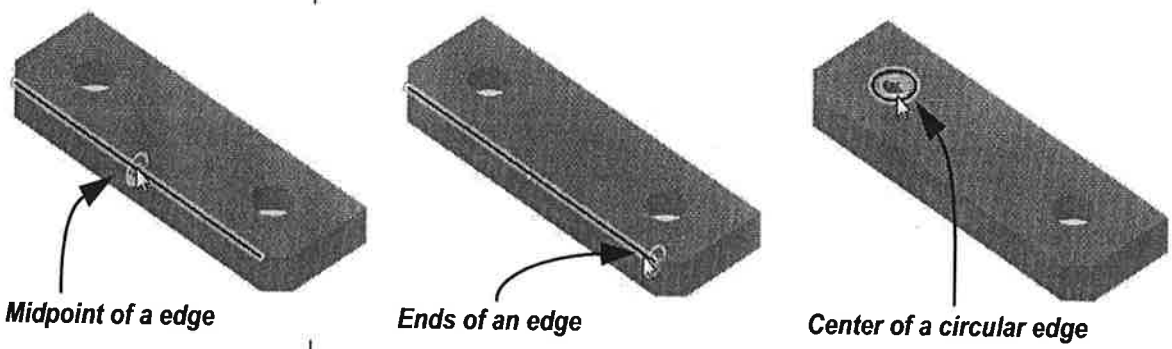
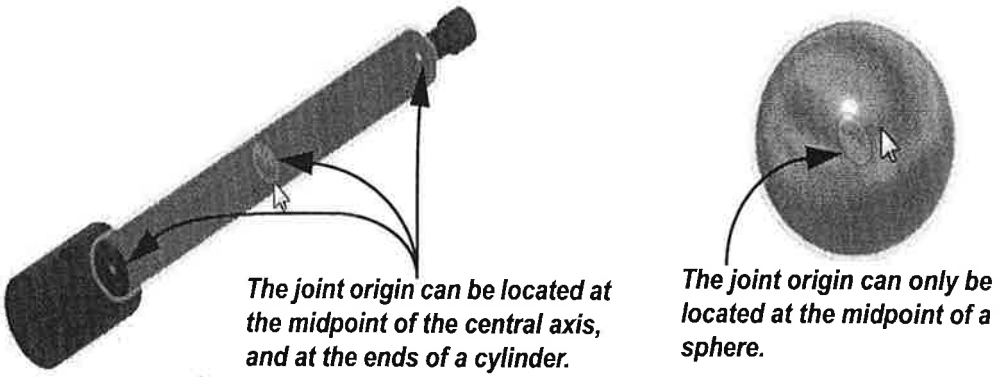


Figure 14–6

Cylindrical and Spherical Geometry Snap-Points

- For cylindrical geometry, a snap-point exists at the midpoint of the central axis and at each end.
- For spherical geometry, the only available snap-point is at the center of the geometry.

The snap-points for cylindrical and spherical geometry are shown in Figure 14–7.



The joint origin can be located at the midpoint of the central axis, and at the ends of a cylinder.

The joint origin can only be located at the midpoint of a sphere.

Figure 14–7

Creating Joint Origins

Timeline icon:

Joint origins can be manually created in a component for later use when a joint is being assigned. To create a joint origin, expand the ASSEMBLE panel and click (Joint Origin). A joint origin can be created using the JOINT ORIGIN palette, in one of two ways: (Simple) or (Between two Faces), as shown in Figure 14–8.

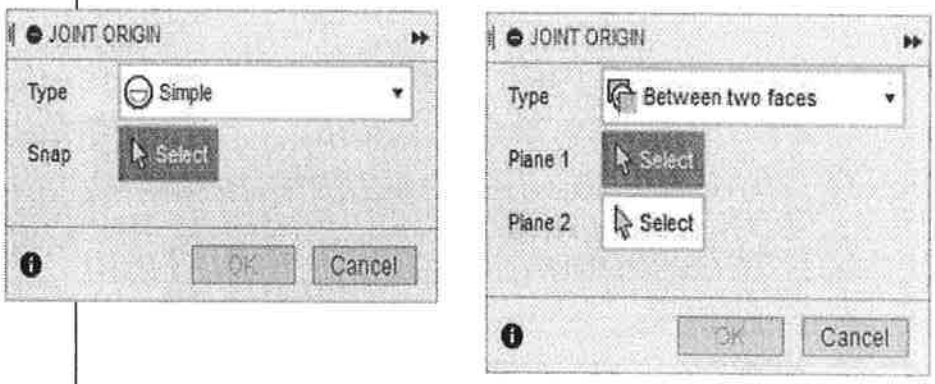


Figure 14–8

Simple

Simple joint origins reference the snap-points that are available on faces, edges, and spherical or cylindrical geometry. The benefit of creating a simple joint origin is that the Move triad is available once the snap-point is selected, enabling you to move the joint origin to a more exact location that does not lie directly on a snap-point.

- Figure 14–9 shows how a joint origin was created and then offset linearly 20 mm from the midpoint of cylindrical geometry. You would not be able to create this joint origin by selecting a reference during joint assignment, so it had to be created independently.

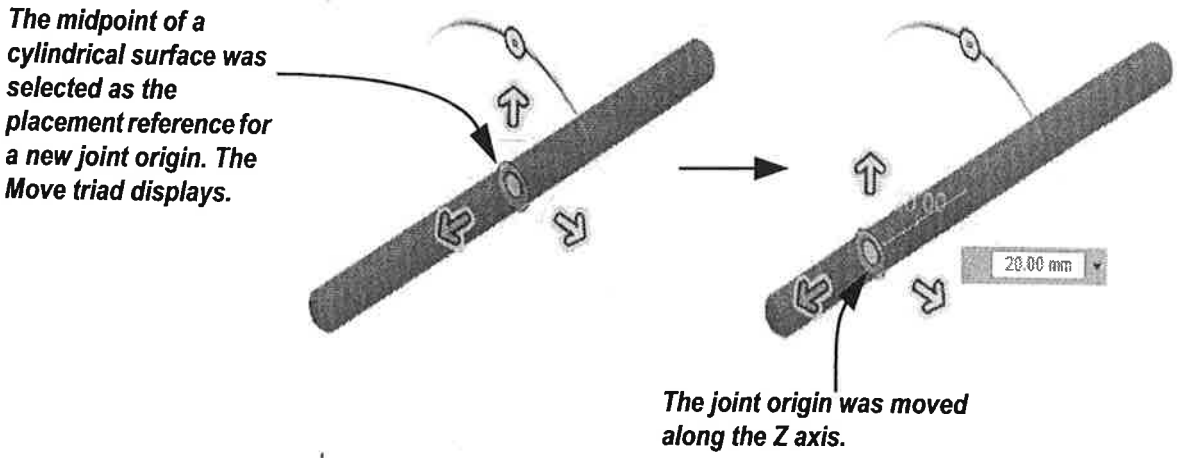


Figure 14–9

- Figure 14–10 shows how a joint origin was created on a construction point. The geometry of the model did not provide the required snap-point, so construction entities were created.

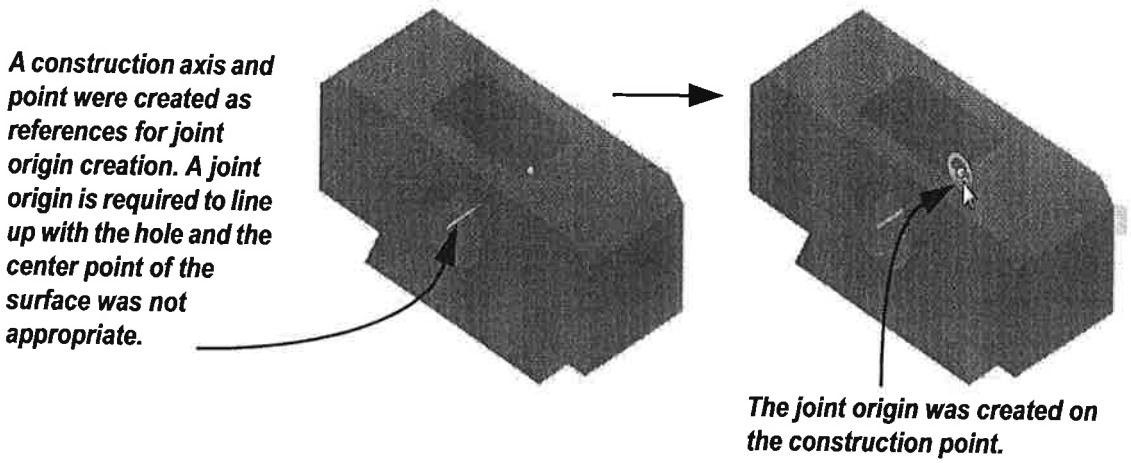
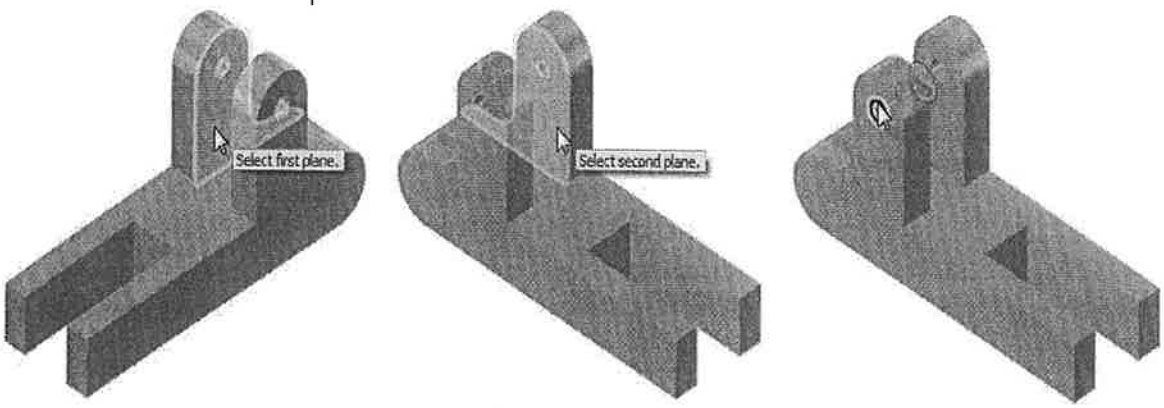


Figure 14–10

Between Two Faces

The **Between two faces** joint origin type enables you to create a joint origin so that it is placed on a mid-plane between two selected faces. The joint origin is located on the plane based on the selection of a snap reference. The snap reference can be a linear axis, edge, or snap-points on other entities. Similar to a Simple joint origin, the Move triad displays once the references are defined.

- In the assembly shown in Figure 14–11, a snap-point does not exist that permits the assembly of a pin. It needs to be constrained along the central axis at a mid-plane between the two faces. A joint origin was created between two faces and aligned to the axis of a hole.



Select the first plane. *Select a parallel plane as the second reference.* *Hover over the cylindrical face of the hole to locate the joint origin.*

Figure 14–11

Hint: Joint Origin Creation

Joint origins are stored within the component that was referenced during placement.

- In the case of a multi-component design created using the distributed design method, the joint origin can only be created at the lowest component level where you have write access to the file.
- For designs that use a file from the McMaster-Carr Content Library, joint origins can be created in the top-level assembly because the component is stored in the top-level assembly and you have write access.
- Using the multi-body design method, joint origins can be created in the top-level assembly because the components were created in the context of the top-level design and you have write access to each component. Additionally you can select references that exist on different components.

14.4 Assigning Joints

In the Autodesk Fusion 360 software, a joint defines the mechanical movement of a component in relation to another component. Joints are assigned between two components by selecting joint origins on the components.

There are various joint types available. Each joint type defines the degrees of freedom that remain in the model once it is assigned. For example, a rigid joint removes all degrees of freedom between components, while a cylindrical joint permits translation and rotation along and around a linear reference.

Timeline icon:

To add a joint between components, complete the following:

1. In the ASSEMBLE panel, click (Joint). The JOINT palette opens, as shown in Figure 14–12.

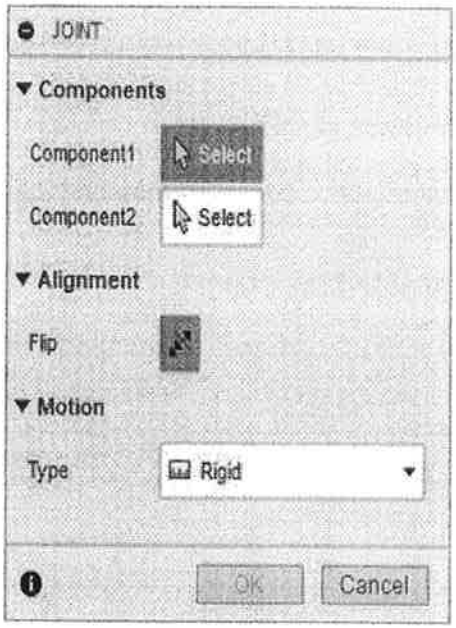


Figure 14–12

A Between Two Faces joint origin can be created during reference selection by right-clicking in the graphics window and selecting ***Between two faces****.*

2. Select a joint origin on the first component. The first component you select should be the component to be moved into position once the joint is complete.
3. Select a joint origin on the second component.
4. If required, flip the alignment of the joint origins in the *Alignment* area by clicking (Flip). Additional options enable you to set an offset and rotational values between joint origins.

5. In the *Type* drop-down list, select the joint type. Additional options might display depending on the type of joint selected, enabling you to set additional options (e.g., to specify the permitted axis for translation or rotation, etc.).
6. To complete the joint, click **OK.**

Joint Types

Each joint type permits a specific number of degrees of freedom (DOF). The degrees of freedom that remain define the type of mechanical movement between the two components that is required in the design.

By default, the previously assigned Joint Type is set in the palette. To change the type, expand the drop-down list and select a new option.

The available joint types and their degrees of freedom are as follows:

Joint Type	Icon	Remaining DOF
Rigid		0 Translational, 0 Rotational
Revolute		0 Translational, 1 Rotational
Slider		1 Translational, 0 Rotational
Cylindrical		1 Translational, 1 Rotational
Pin-slot		1 Translational, 1 Rotational
Planar		2 Translational, 1 Rotational
Ball		0 Translational, 3 Rotational

To fully define a joint type, a joint origin reference is required on two components. Based on the selected references, the components are joined to one another and the joint type defines the permitted movement between the components. The following examples explain how components were joined using the various joint types.

Rigid

The **Rigid** joint type removes all of the degrees of freedom, preventing any relative motion between the two components.

An example of a rigid joint is shown in Figure 14–13.

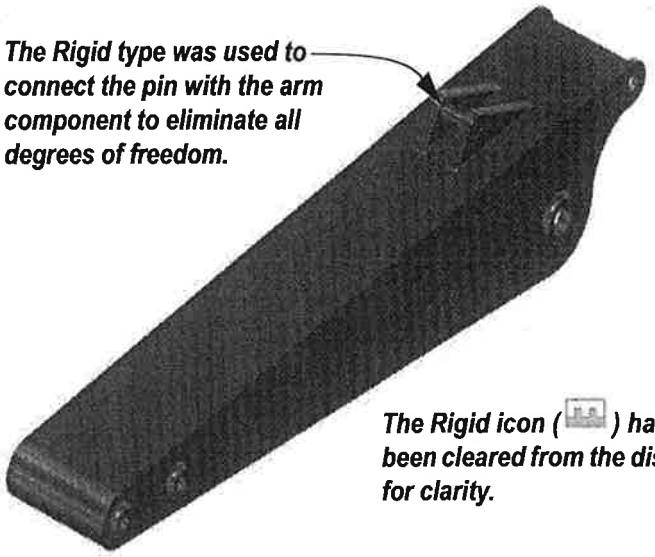


Figure 14–13

Revolute

The **Revolute** joint type permits for rotation about an axis. This joint type removes five degrees of freedom. Once the component references are selected, you can select the axis of revolution in the *Motion* area (if the default selection is incorrect).

An example of a revolute joint is shown in Figure 14–14.

A custom axis can be defined if the X, Y, or Z axis do not permit the required motion.

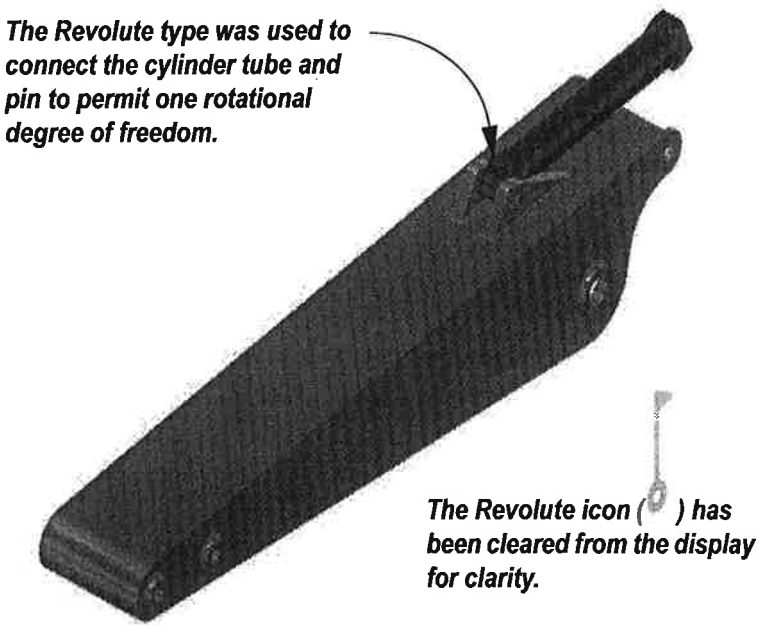


Figure 14–14

Slider

The **Slider** joint type permits translational movement along an axis. This joint type removes five degrees of freedom. Once the component references are selected, you can select the linear axis in the *Motion* area (if the default selection is incorrect).

An example of a slider joint is shown in Figure 14–15.

A custom axis can be defined if the X, Y, or Z axis do not permit the required motion.

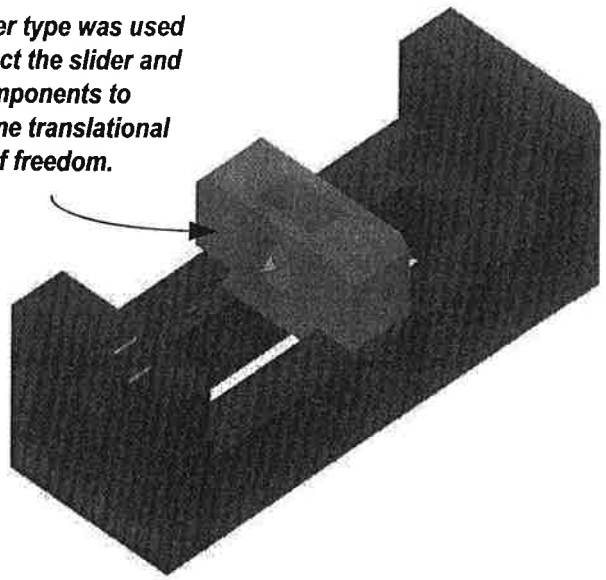


Figure 14–15

Planar

The **Planar** joint type enables a component to move in a plane as shown in Figure 14–16. Two translational and one rotational degrees of freedom remain. Once the component references are selected, you can select the linear and normal axes in the *Motion* area (if the default selections are incorrect).

An example of a planar joint is shown in Figure 14–16.

A custom axis can be defined if the X, Y, or Z axis do not permit the required motion.

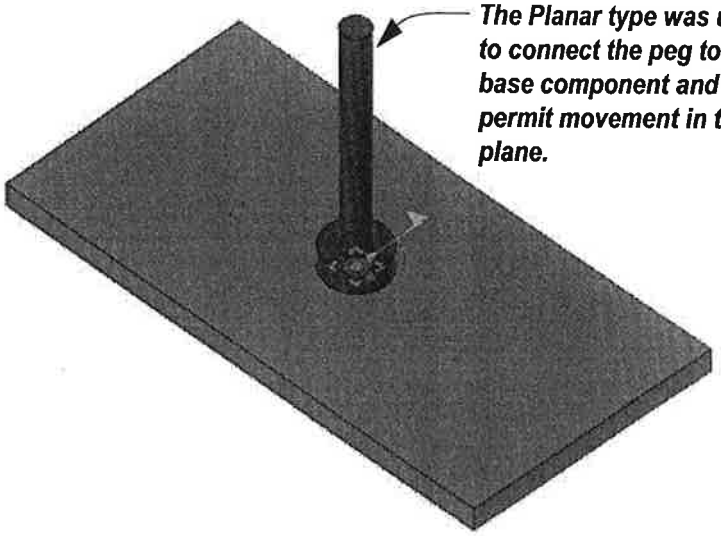


Figure 14–16

Cylindrical

The **Cylindrical** joint type enables a component to translate and rotate about a specific axis leaving two degrees of freedom available. Once the component references are selected, you can select the axis in the *Motion* area (if the default selection is incorrect).

An example of a cylindrical joint is shown in Figure 14–17.

A custom axis can be defined if the X, Y, or Z axis do not permit the required motion.

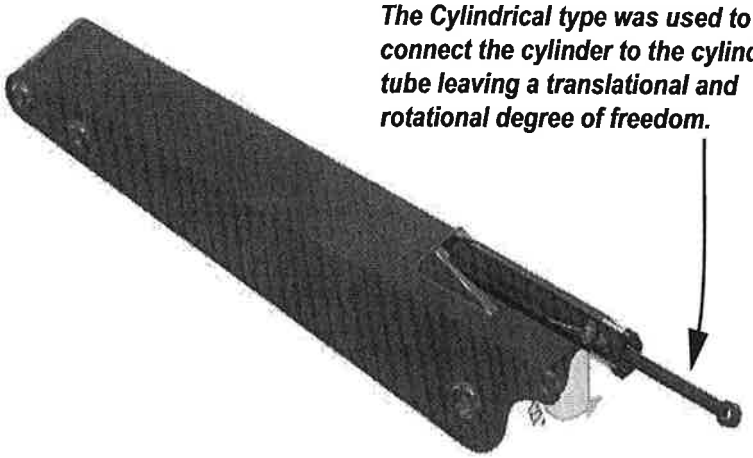


Figure 14–17

Pin-slot

The **Pin-slot** joint type enables a component to move in a slot. One translational and one rotational degree of freedom remain with the component. Once the component references are selected, you can select the linear and rotational axes in the *Motion* area to slide and rotate about (if the default selections are incorrect).

A custom axis can be defined if the X, Y, or Z axis do not permit the required motion.

An example of a pin-slot joint is shown in Figure 14–18.

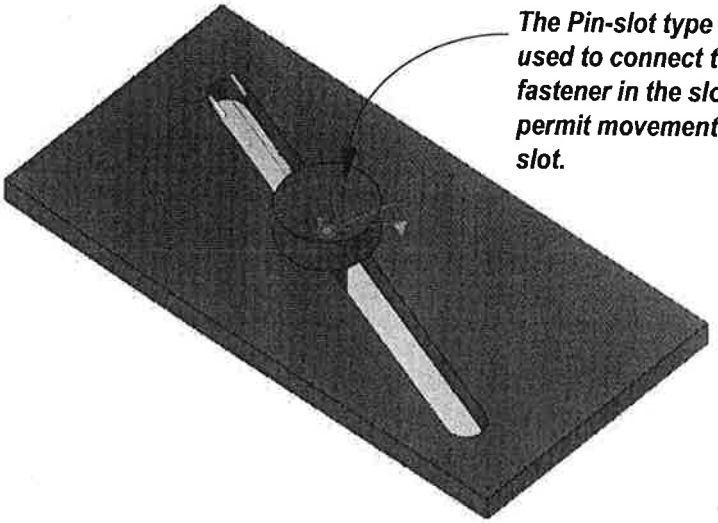


Figure 14–18

Ball

The Ball joint type enables a component to rotate in any direction about the origin joint, as shown in Figure 14–19. All translational degrees of freedom are removed. Once the component references are selected, you can select the pitch and yaw axes in the *Motion* area (if the default selections are incorrect).

An example of a ball joint is shown in Figure 14–19.

A custom axis can be defined if the X, Y, or Z axis do not permit the required motion.

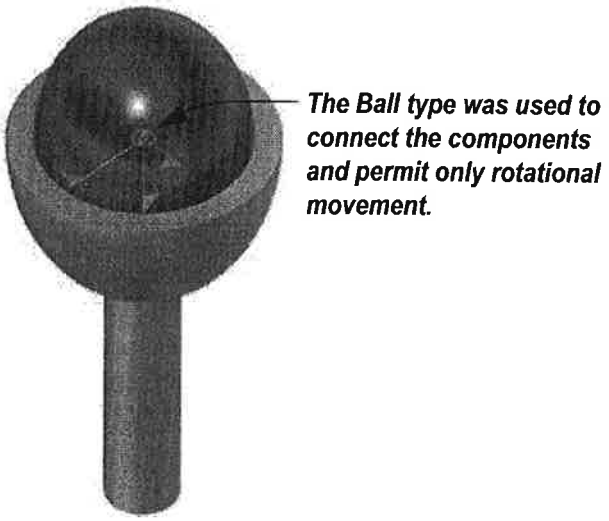


Figure 14–19

Hint: Working with Joints

- The default naming structure for a joint is its type, followed by a number that indicates the number of joints that have been assigned. To rename a joint, select the joint name twice in the *Joints* folder in the BROWSER and enter a descriptive name.

- To change the joint type or its references, right-click on the joint name in the BROWSER or in the Timeline and select **Edit Joint**. The EDIT JOINT palette opens and you can change any of original settings.

To hide the display of the Joint icon in the graphics window, locate the joint name in the *Joints* folder in the BROWSER and select  (Show/Hide) so that it changes to  . Select it again to toggle the icon back to  and show the joint. To hide all Joint icons at once, toggle the  adjacent to the *Joints* folder.

- If a component is able to move, you can lock the joint to temporarily disable its permitted degrees of freedom. This might help you to create additional bodies or test or add other Joints. To lock a joint, locate the joint in the BROWSER or Timeline, right-click the joint name, and select **Lock**. To unlock a joint, right-click on the joint name and select **Unlock**.

- Joints can be temporarily suppressed by right-clicking on the joint name in the BROWSER and selecting **Suppress**. Suppressed joints are displayed in dark gray in the BROWSER to easily identify them.

- Once a joint is assigned, you can animate its available motion (including any set limits) in the BROWSER by right-clicking on the joint name and selecting **Animate Joint**. To stop the animation, press <Esc>.

- To return a component to its home location, in the BROWSER, right-click on the joint's name and select **Go to Home position**. The Home position is the location where you initially placed the component when you added it to the design.

Practice 14a

Creating a Distributed Design

Practice Objectives

- Use the **Joint** command to fully connect components.
- Drag components to verify the movement of the design.

In this practice, you will create a new assembly and insert the components as shown in Figure 14–20. The fasteners that are used will be inserted from the McMaster-Carr library, and the remaining components will be inserted from files that have been provided. You will also use the Joint tool to join components relative to one another.

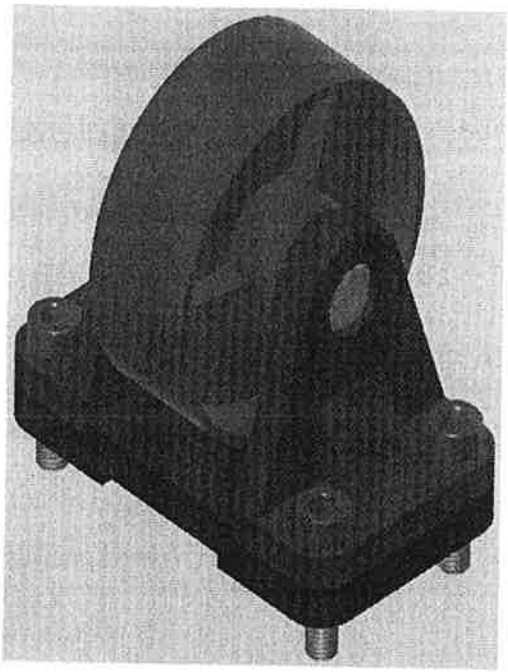


Figure 14–20

Task 1 - Upload the components that will be used in the assembly design.

In this task, you will begin the creation of an assembly design by loading all of the models that will be used into your Autodesk Fusion 360 project. This is an alternate method to opening the design from file and is a convenient method of uploading multiple files. The models that will be used each contain a single body and were created by importing geometry from the Autodesk® Inventor® software.

1. In the Application Bar at the top of the interface, click  (Show Data Panel). Ensure that **Autodesk Fusion 360 Practice Files** is the active project.
2. At the top of the Data Panel, click **New Folder**. Enter **Wheel_Assembly** as the name of the new folder.
3. Double-click on the *Wheel_Assembly* folder to open it.
4. At the top of the Data Panel, click **Upload**.
5. In the Upload dialog box, click **Select Files**. Navigate to the practice files folder on your local drive and select the four files in the *Wheel_Assembly* folder. Click **Open**.
6. Click **Upload** to begin the upload. The Job Status window displays the progress of the upload.
7. Once the files have been uploaded, click **Close** in the Job Status window. The files should display as shown in Figure 14–21.

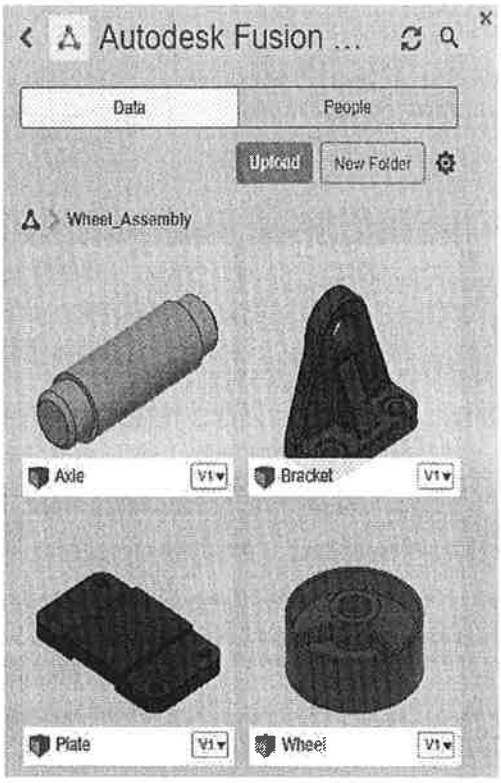


Figure 14–21

Task 2 - Create a new design and insert the first grounded component into the design.

In this task, you will insert the first component into a new design. This is one of the methods for creating a multi-component design (assembly). In a future task you will insert additional components and assign Joints to constrain the components to one another.

1. In the **File** menu, create a new design by clicking **New Design**.

2. The components cannot be inserted into the new design until it is saved. Save the new design as **Wheel_Assembly**. Ensure that the *Location* drop-down list is set to **Autodesk Fusion 360 Practice Files>Wheel_Assembly**. If not, expand the drop-down list and double-click on the project and folder to select them. Click **Save**.

3. In the Data Panel, right-click on **Plate** and select **Insert into Current Design**, as shown in Figure 14–22.

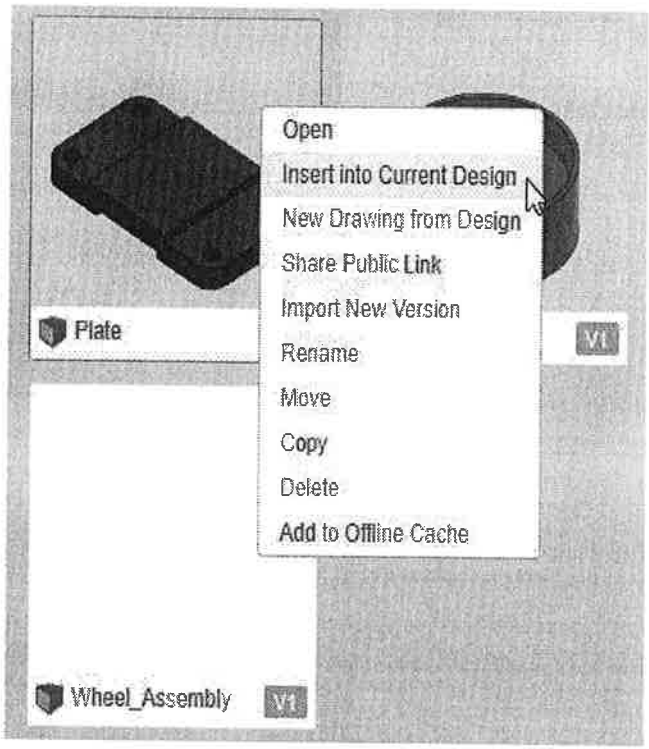


Figure 14–22

4. The component is inserted and the MOVE/COPY palette is immediately active. Enter **90.0 deg** in the *X Angle* field and click **OK**.

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

5. In the BROWSER, right-click on the **Plate v1:1** component and select **Ground**, as shown in Figure 14–23. This sets the plate component as the grounded component in the assembly. The **Plate v1:1** component is used as the grounded component because it is the foundation component in the design and does not move.

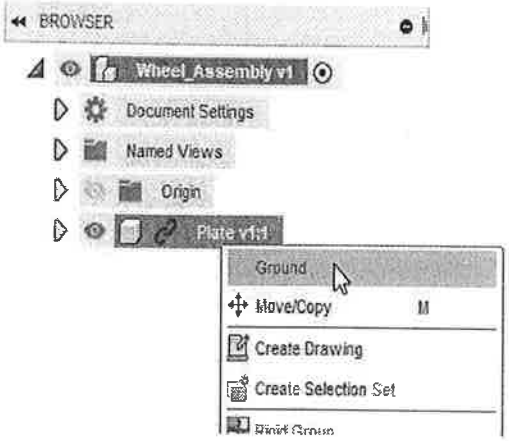


Figure 14–23

Inserted components maintain their link to the original component unless the link is explicitly broken. To break the link, right-click on the filename in the BROWSER and select ***Break Link****.*

6. In the BROWSER, note that the Plate component has been added with the  and  icons next to it.

   - The  icon indicates that the component is grounded.
   - The  icon indicates that the component has been inserted from a source file and a cross-reference has been created.

   The model displays as shown in Figure 14–24.

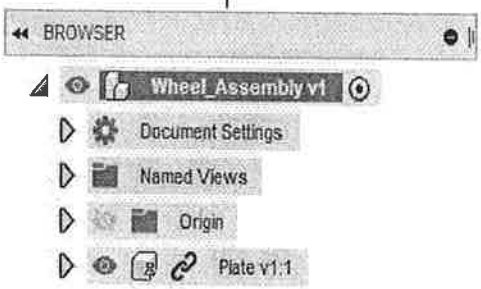


Figure 14–24

Task 3 - Add another part and add a joint to locate it in the assembly.

In this task, you will insert the **Bracket** component and assign a Rigid joint. Once inserted and placed, the assembly should display as shown in Figure 14–25.

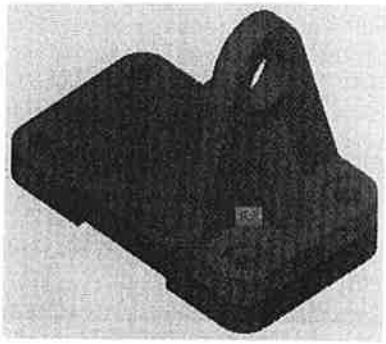


Figure 14–25

1. In the Data Panel, right-click on **Bracket** and select **Insert into Current Design**.
2. Using the Move triad that automatically displays, move the component similar to that shown in Figure 14–26 using both linear and rotational movements. This orientation makes reference selection easier when placing the joint. In the MOVE/COPY palette, click **OK** once the orientation is set.

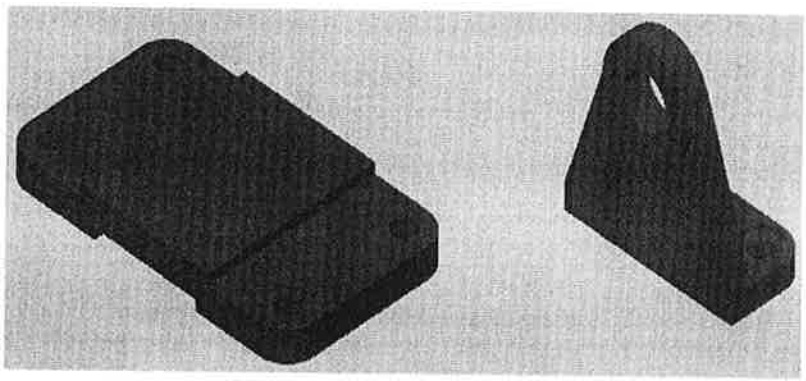


Figure 14–26

3. In the graphics window, select the bracket and drag it around the screen using the cursor. Note how the part moves freely. Now try and move the plate. This component will not move because it is grounded.
4. In the POSITION panel, click (Revert) to return the bracket component to the original location.
5. In the ASSEMBLE panel, click (Joint). The JOINT palette opens as shown in Figure 14–27.

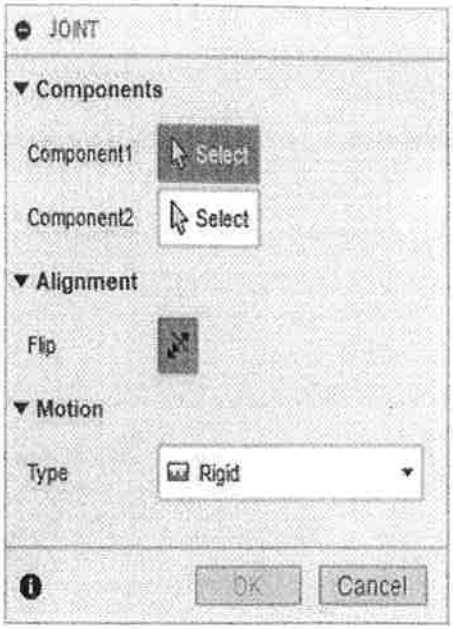


Figure 14–27

Once you have assigned a joint, that Type becomes the default until the software is restarted.

6. Note that **Rigid** is set as the default joint type in the *Motion* area.

7. You will join the two components so that all of the degrees of freedom are removed. Keep the default **Rigid** selection and note that the **Plate** component is dimmed, indicating that it cannot be selected as the first component reference because it is grounded.

8. The *Component1* field is active by default. Hover the cursor over the bottom edge of the Bracket component, as shown in Figure 14–28. Move the cursor and note that even with slight movements, the back face can become active. Ensure that the edge is active and hold <Ctrl>. This locks reference selection to the edge, while still enabling you to move the cursor to activate other snap-points on the edge. Once the midpoint of the edge is active as the joint origin, select it using the left mouse button.

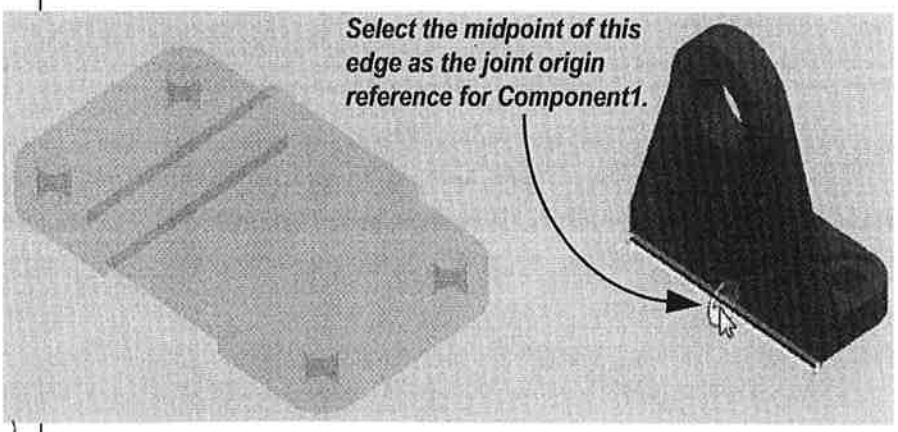


Figure 14–28

Consider holding <Ctrl> as you are locating the snap-point for the second reference.

9. **Plate** is no longer dimmed in the assembly and the *Component2* field is active. Hover the cursor over the edge shown in Figure 14–29. Once the midpoint of the edge is active as the joint origin, click to select it.

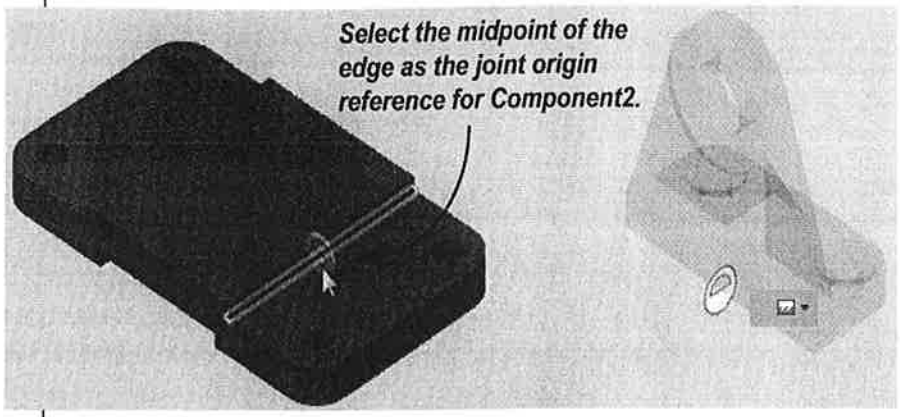


Figure 14–29

Reference Selection

While the edge on *Component2* was selected to locate the Bracket, the circular edges on each component could also be selected to ensure hole alignment.

The first component selected is the one that moves to the second component once both references are selected.

10. The **Bracket** component moves into position. A slight shaking animated motion is shown indicating the Rigid joint.
    - If the component is oriented incorrectly, review the component references to ensure that the midpoint of the edges were selected. Alternatively, click (Flip) to flip the component or enter an *Angle* value to rotate it into position. The final position should be as shown in Figure 14–30.
    - Note the linear and rotational handle that display on the model. These enable you to set an offset between the two joint origins or rotate about them. In this case, no further manipulations are required.

Angle and Offset values can be set in the JOINT palette, in the entry field in the graphics window, or by dragging the linear and rotational handles.

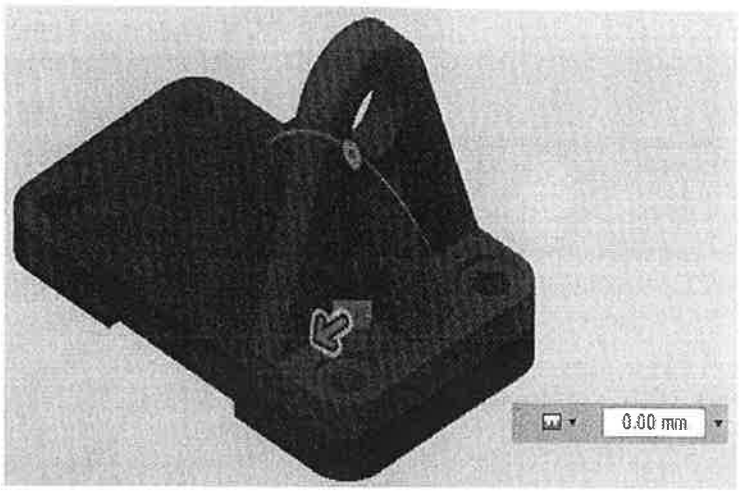


Figure 14–30

11. To replay the animation, in the *Motion* area, click  . Click  to stop the animation.

12. In the JOINT palette, click **OK** to complete the creation of the rigid joint. Figure 14–31 shows the joined components with the Rigid Joint icon () displayed.

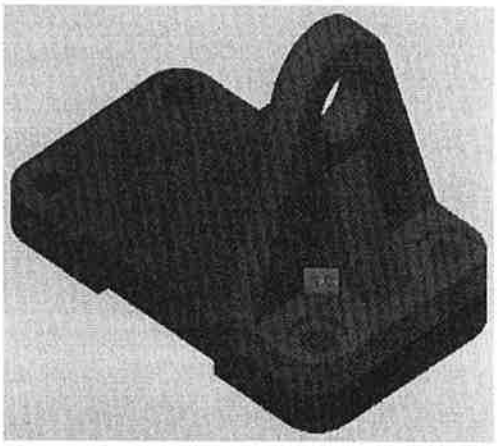


Figure 14–31

13. Select and try to drag the Bracket component. Note that you cannot drag the component.

Task 4 - Add a second instance of the Bracket component.

In this task, you will add and join another instance of the Bracket component into the assembly. The final placement of the component in the assembly is shown in Figure 14–32.

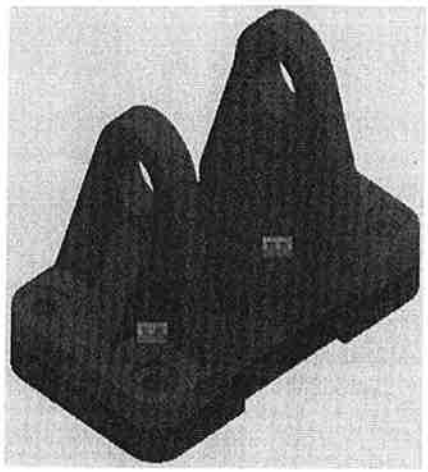


Figure 14–32

1. In the Data Panel, right-click on **Bracket** and select **Insert into Current Design**. Alternatively, you can:
   - Right-click on the existing **Bracket** component in the BROWSER and select **Copy**. Then, in the graphics window, right-click and select **Paste**.
   - Select and drag the component from the Data Panel into the graphics window.
2. Reorient the assembly and use the Move triad to arrange the components similar to that shown in Figure 14–33.

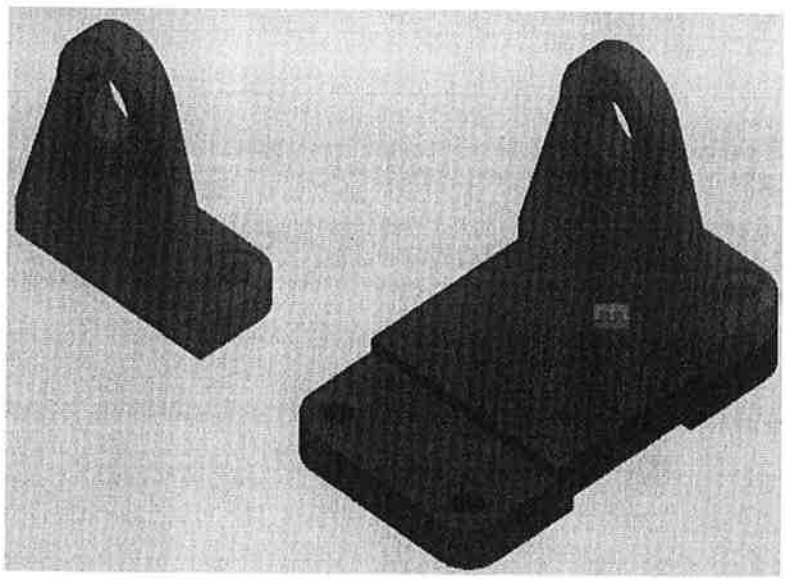


Figure 14–33

3. In the ASSEMBLE panel, click (Joint). The JOINT palette opens. Note that **Rigid** is still set as the default *Type* in the *Motion* area because that was the last joint type that was assigned.

4. The *Component1* field is active by default. Select the snap-point at the middle of the bottom edge on **Bracket v1:2**, as shown in Figure 14–34.

5. For the *Component2* reference, select the snap-point at the middle of the edge on the Plate, as shown in Figure 14–34.

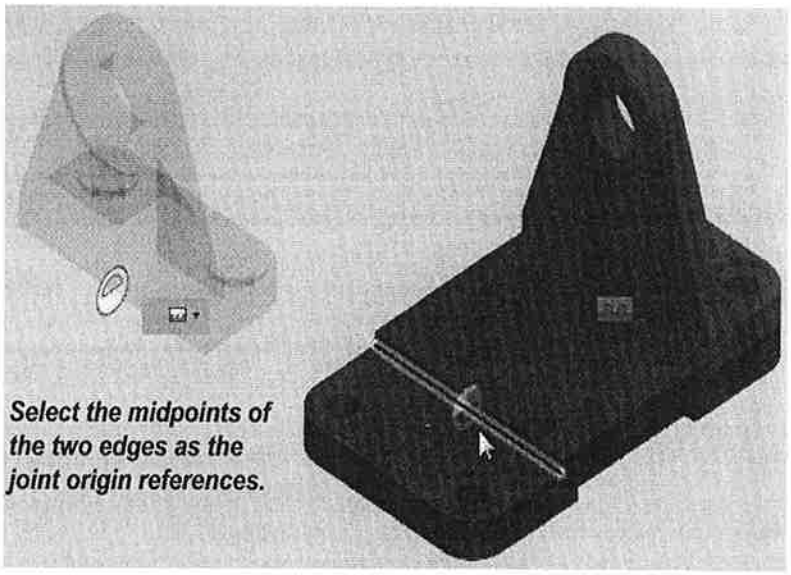


Figure 14–34

6. Based on the orientation of the joint origins, the components might not align correctly. Select (Flip) to flip the Bracket or enter a 180° rotation value to position the component correctly.

7. In the JOINT palette, click **OK** to complete the creation of the rigid joint, as shown in Figure 14–35. Note that two Rigid Joint icons display on the model, identifying how the components are joined.

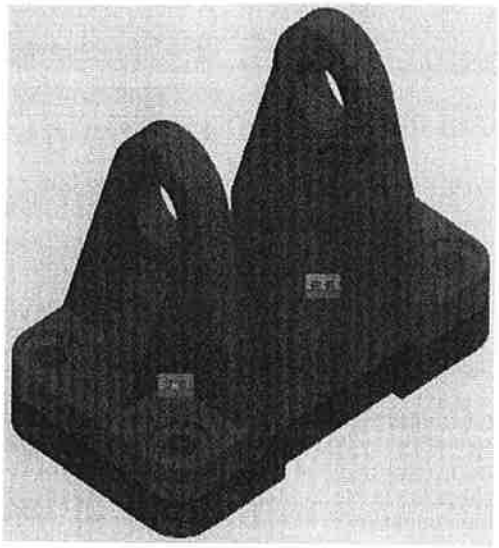


Figure 14–35

Task 5 - Insert and join the Axle component.

1. In the Data Panel, insert the **Axle** component into the assembly design using one of the techniques previously discussed.

2. Use the MOVE/COPY palette to move the component into a more convenient orientation, similar to that shown in Figure 14–36. Click **OK**.

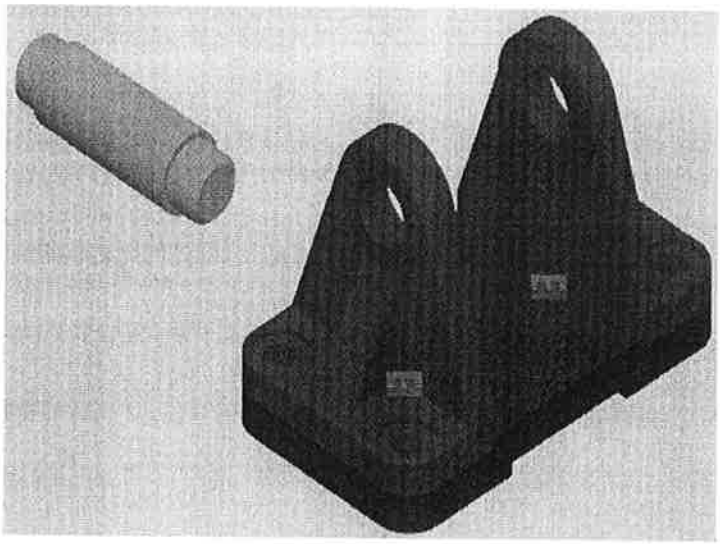


Figure 14–36

3. In the INSPECT panel, click (Measure). The MEASURE palette opens. Select the two circular faces on each of the **Bracket** components (shown in blue in Figure 14–37) to measure the distance between them. The value **57.00 mm** should display in the graphics window. This value is required to join the axle.

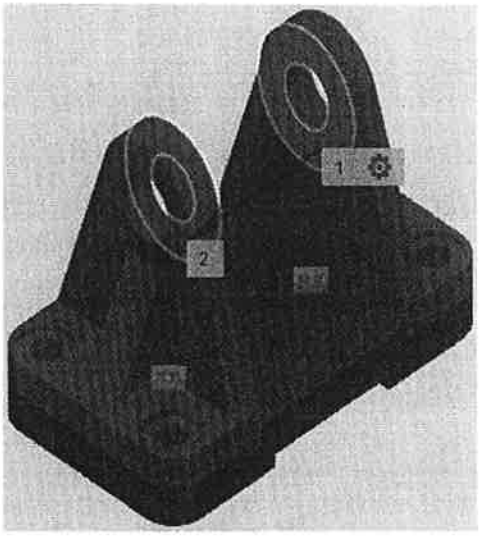


Figure 14–37

4. Initiate the creation of a new joint. In the *Type* drop-down list, ensure that the **Revolute** type is selected. The two components will be joined so that a single rotational degree of freedom remains.

5. For the *Component1* reference, select the snap-point at the middle of the central axis of the **Axle** component. The joint origin displays as shown in Figure 14–38.

6. For the *Component2* reference, select the snap-point at the end of the cylindrical face of **Bracket v1:1**, as shown in Figure 14–38.

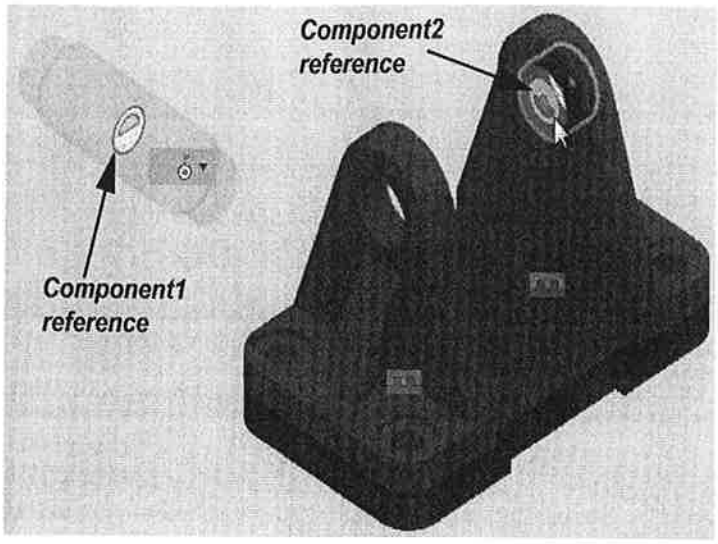


Figure 14–38

7. Once the components move into position, the **Axle** displays as shown in Figure 14–39.

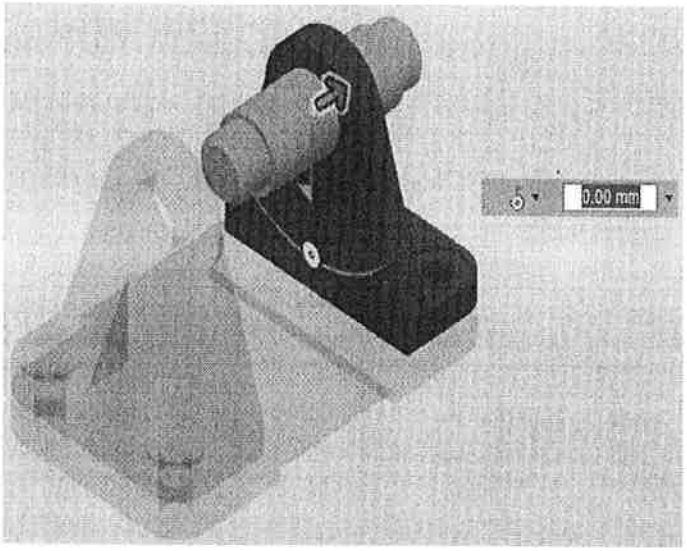


Figure 14–39

8. You must offset the two joint origins to position the **Axle** correctly. Enter **-28.5 mm** as the *Offset* value (i.e., half of the distance between the two brackets). With the arrow manipulator active (blue), you can enter the value in the entry field in the graphics window, or in the JOINT palette in the *Offset* Z field.

9. Note that the **Z Axis** is automatically selected as the *Rotate* reference for the joint. This is the correct motion, so selecting an alternate axis in the drop-down list is not required.

10. Click  in the *Motion* area to play the motion. Note that the Axle rotates fully. Click  to stop the animated motion.

11. Click **OK** to create the joint. Note that the Revolute icon is displayed in the graphics window, as shown in Figure 14–40.

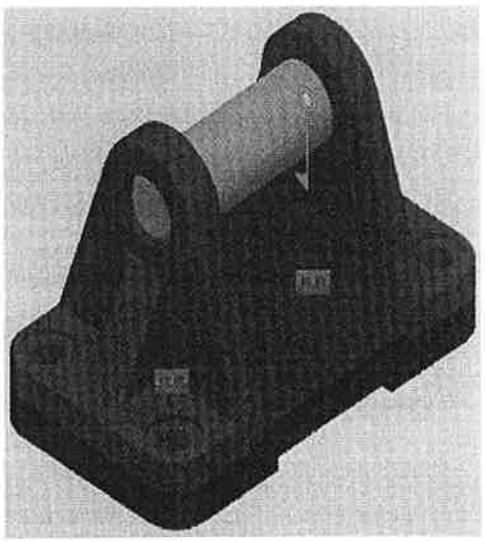


Figure 14–40

Task 6 - Review the joints in the design and rename them.

1. In the BROWSER, expand the *Joints* folder. Note that all three existing joints are listed.

2. The Timeline displays as shown in Figure 14–41. These icons represent the four inserted components, the three Joints, and when the ground constraint was assigned to the **Plate** component.

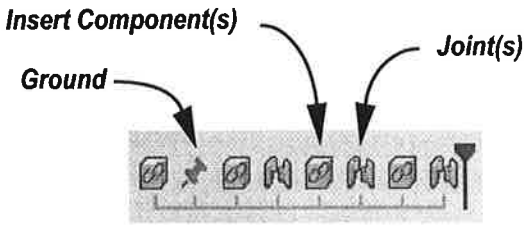


Figure 14–41

In general, selecting the constraints is easier in the Joints folder.

Renaming the joints helps identify them once multiple joints are added to the design.

3. In the Timeline, you can individually identify the joints by hovering over the Joint icons to reveal their names, or by selecting them to highlight the joint in the graphics window.

4. Click twice on the **Rev3** joint in the *Joints* folder to rename it. Enter **Axle** as the new name.

The icons that identify the joint type can be hidden in the graphics window, if required. In the *Joints* folder, select (Show/Hide) adjacent to the two rigid joints to hide them from the display. Figure 14–42 shows the BROWSER and model after renaming the axle joint and hiding the Rigid icons from the display.

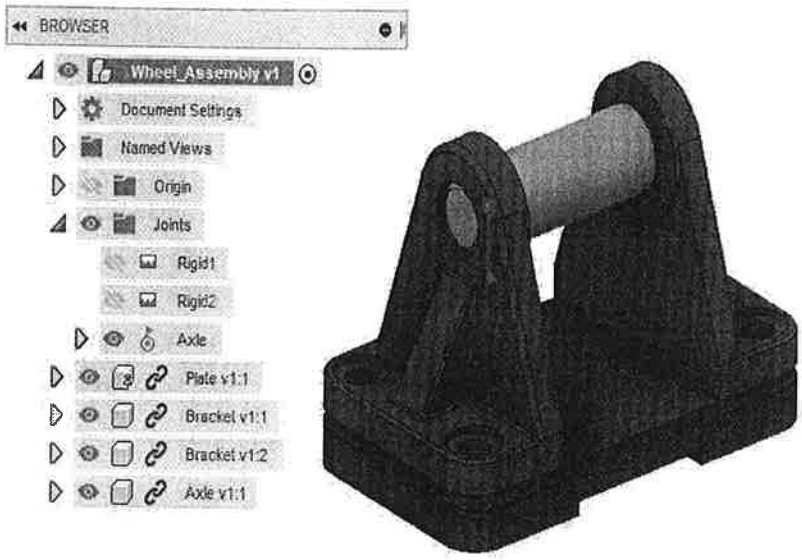


Figure 14–42

Task 7 - Modify an existing joint in the design.

As the wheel on the axle is required to rotate, and not the axle itself, edit the Axle joint to make it rigid.

1. In the BROWSER, right-click on the **Axle** joint and select **Edit Joint**.

2. In the *Type* drop-down list, select **Rigid** to prevent the component from moving. Click **OK**.

3. Hide the Rigid icon from the display.

Task 8 - Insert the Wheel component.

1. Insert the Wheel component and join it as shown in Figure 14–43.
   - Assign the joint that permits rotation about the **Axle**. No translational motion is required.
   - Remember to select the **Wheel** as the first reference so that it moves onto the **Axle** once reference selection is complete.
   - When selecting the reference on the **Wheel**, consider holding <Ctrl> to ensure that the correct snap-point is selected so that the midpoint on the **Axle** and the **Wheel** are selected.
   - Ensure that the *Offset* and *Angle* values are set to **0**.

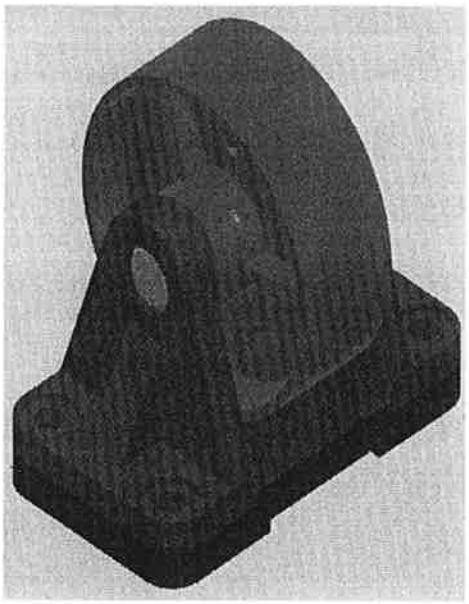


Figure 14–43

Task 9 - Insert the Screw components using a McMaster-Carr Component.

In this task, you will retrieve a component from the McMaster-Carr library and use its CAD model in the design.

1. Expand the INSERT panel and select (Insert McMaster-Carr Component).
2. If you are prompted that the use of this feature will connect you to an external website, click **OK**. This website contains standard library parts that can be purchased, or you can download a SOLIDWORKS or STEP file for use in your design.

*If you know the component number that is required, you can search for it. In this task, you will use the **91290A526** component.*

3. In the INSERT MCMASTER-CARR COMPONENT palette, a list of categories display. Select the **Fastening & Joining** category. On the right, select **Screws & Bolts**.

4. In the Screws list, select the **Socket Head Screws** category. The palette updates as shown in Figure 14–44.

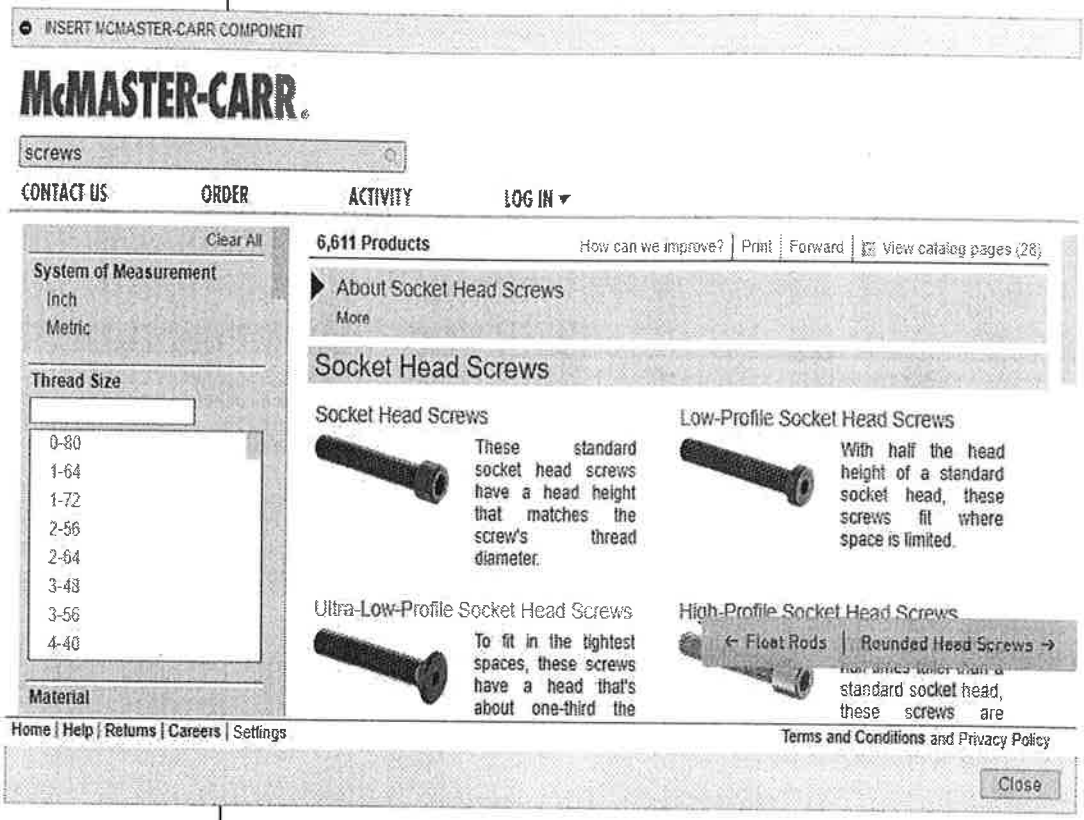


Figure 14–44

5. The left side of the palette enables you to refine the search to help locate a required component. Scroll down in the list to view the filtering options.

6. In the *Thread Size* list, select **M10**.

7. In the *Threading* area, select **Fully Threaded**.

8. In the *Length* list, select **35 mm**.

9. In the right-hand frame, select **Alloy Steel Socket Head Screws**.

You can continue to select additional criteria in the Narrow By list to refine the results that are displayed, as required.

10. In the list of components, scroll down and select the **91290A526** component in the list of cap screws that meet the criteria.

11. You can enter order details, if you are required to purchase the model. In this practice, you will only be downloading a copy of the CAD data for use in your design.

12. In the expanded *Order* area, click **Product Detail**.

13. Scroll down. In the list of CAD formats, select **3-D STEP** and then click **SAVE**.

14. The model is inserted into the design and the MOVE/COPY palette displays. Move the component so that you can more easily select references and assign a joint. Click **OK**.

15. Begin the creation of a new joint. When prompted that some components have been moved, click **Capture Position**. This ensures that the component remains where it is for component reference selection. This is done by adding a Component Move action to the Timeline.

16. In the JOINT palette, set the *Type* to **Rigid** to permit no degree of freedom (Rigid). Define the joint origins to locate the component as it is shown in Figure 14–45.

To ensure that the cap screw is inserted correctly, conduct a Section Analysis. Hint: create a plane to conduct this analysis.

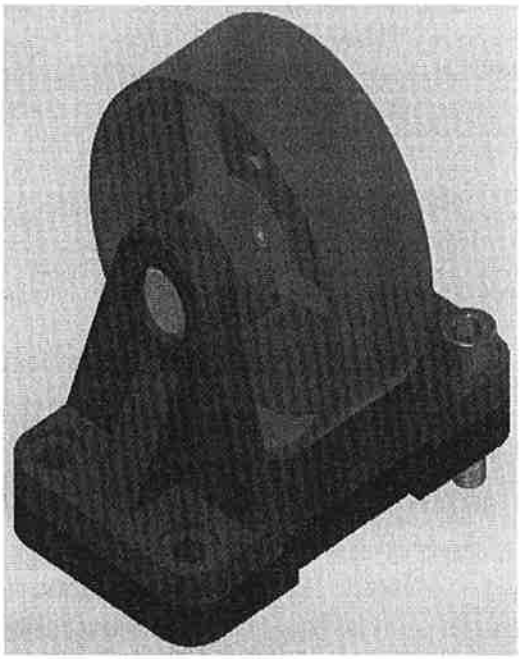


Figure 14–45

17. In the Timeline, right-click on (Capture Position) (the second to last icon) and select **Delete** to remove it. Now that the component is constrained, this action is not required.

18. In the BROWSER, right-click on the **91290A526:1** component and select **Copy**. In the graphics window, right-click and select **Paste**. Move the new component so that it does not overlay the source component.

19. Copy and paste two additional **91290A526** components.

20. Add Rigid joints so that the design displays as shown in Figure 14–46. All Rigid joint icons have been hidden from the display.

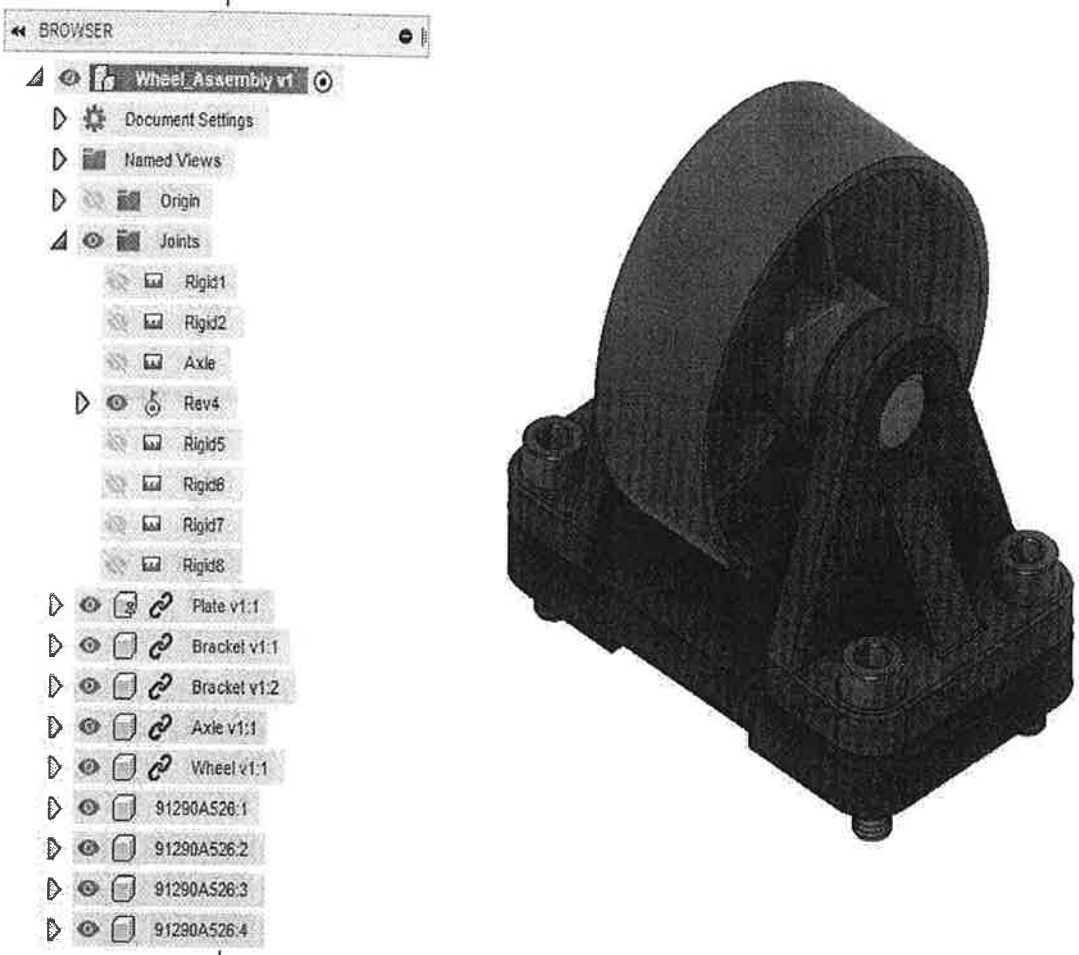


Figure 14–46

Task 10 - Make a design change and update the components.

In this task, you will make a design change to the **Bracket** component and review how the change affects the assembly.

1. Right-click on one of the **Bracket** components in the BROWSER and select **Open**. The model opens in its own window.

2. Use the **Press Pull** tool to modify the height and width of the rib (3 faces) so that it is similar to that shown in Figure 14–47.

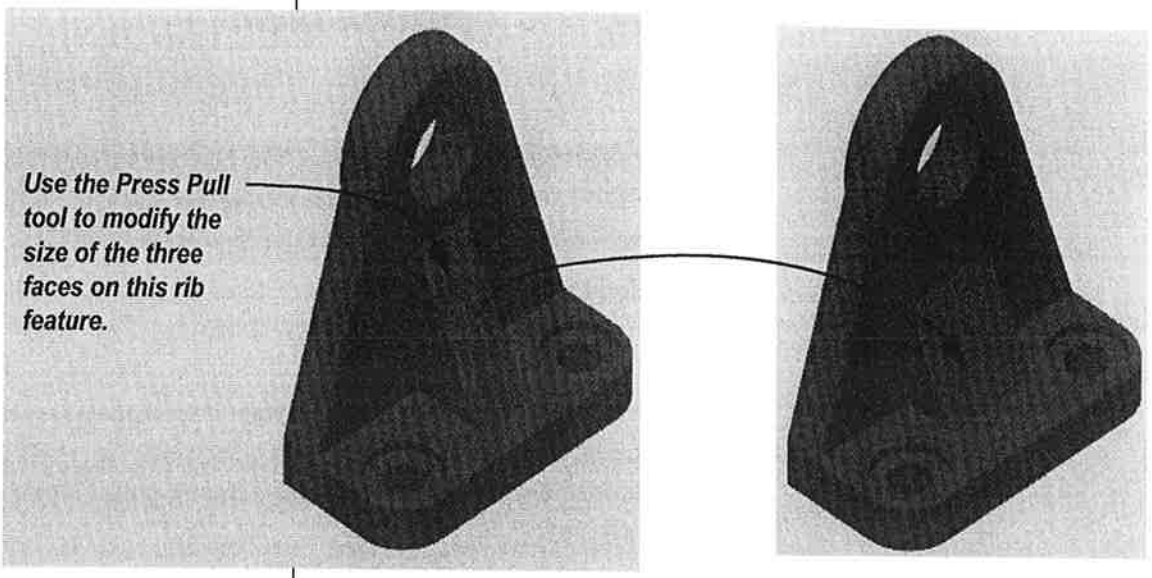


Figure 14–47

3. Save the design.
4. Return to the **Wheel_Assembly** window. Note that the BROWSER indicates that the components and the top-level assembly are out-of-date (⚠), as shown in Figure 14–48.

Figure 14–48

*If multiple components are displaying as out-of-date, right-click on the assembly name and select **Get All Latest** to update all of the components at once.*

5. Right-click on **Bracket v1** and select **Get Latest**. Note how the assembly updates to reflect the change to the **Bracket** component.
6. Save the design to your Autodesk Fusion 360 Practice Files project.
7. Close the file.

Command Summary

Button	Command	Location
N/A	**Animate Joint**	• **BROWSER:** *Context Menu with a joint name selected*
N/A	**Delete** (Joint)	• **BROWSER:** *Context Menu with a joint name selected* • **Timeline:** *Context Menu with a joint icon selected*
N/A	**Edit Joint**	• **BROWSER:** *Context Menu with a joint name selected* • **Timeline:** *Context Menu with a joint icon selected*
N/A	**Get All Latest**	• **BROWSER:** *Context Menu with a component name selected*
N/A	**Get Latest**	• **BROWSER:** *Context Menu with a component name selected*
N/A	**Go to Home Position**	• **BROWSER:** *Context Menu with a joint name selected*
N/A	**Ground**	• **BROWSER:** *Context Menu with a component name selected*
N/A	**Insert into Current Design**	• **Data Panel:** *Context Menu with a component's thumbnail image selected*
	**Joint**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> ASSEMBLE panel • **Context Menu**: Right-click in the graphics window and select **Assemble.**
	**Joint Origin**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab>ASSEMBLE panel • **Context Menu**: Right-click in the graphics window and select **Assemble.**