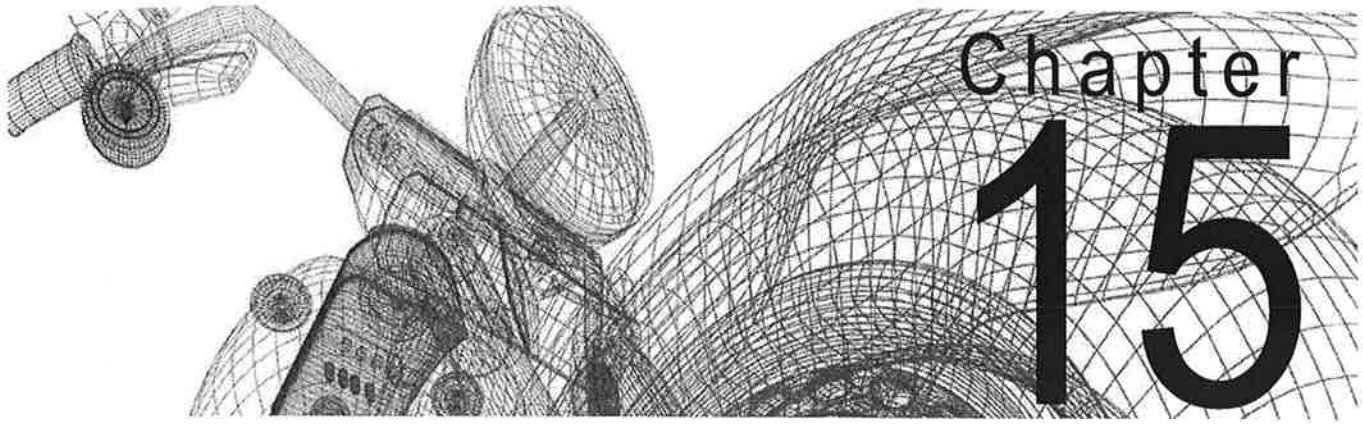

Chapter

15

Component Design Tools

Once you have learned how to insert and join components in a design, there are additional tools that can be used to further control a component's range of motion. These tools enable you to further limit motion to that which is permitted with the joint itself. Additional tools (such as interference analysis tools, Motion Linking, and Motion Studies) can also be incorporated into a design to simulate and test its motion.

Learning Objectives in this Chapter

- Fully constrain components relative to one another using a rigid group.
- Conduct an interference analysis between components in a design.
- Assign limits to a joint to further control its range of motion.
- Define the location of a component within its range of motion.
- Incorporate Contact Analysis into a design to limit a component's range of motion.
- Establish relationships between components that move relative to one another using the Motion Linking tool.
- Conduct a motion study.

15.1 Rigid Groups

The use of rigid groups in a multi-component design enables you to fix multiple components together so that they do not move relative to one another. Rigid groups are ideal in the following situations:

- When no mechanical movement exists between components, using rigid groups avoids having to add multiple rigid joints to remove all degrees of freedom.

Timeline icon: 

- When inserting imported assembly geometry from another CAD format, rigid groups can quickly constrain all of the components that do not move relative to one another.

The Spindle design shown in Figure 15–1 was imported from Autodesk® Inventor®. The four part models represent a single subassembly that moves all as one, so a rigid group was created.

A single joint can be added between a rigid group and other components in the assembly to fully define its placement and motion requirements.

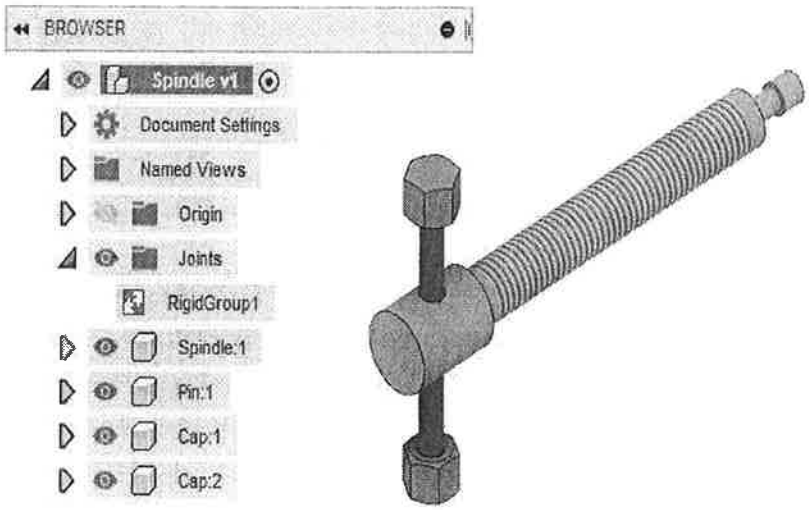

Figure 15–1

*Use the **Include Child Components** option to quickly select the children of selected components so that they do not have to be selected manually.*

- To create a rigid group, expand the ASSEMBLE panel and select  (Rigid Group). Using the RIGID GROUP palette, select the components to add to the group in the BROWSER or in the graphics window, and then click **OK**.

- Rigid groups are stored in the **Joints** folder in the BROWSER and are identified with  in the Timeline.

To edit a rigid group, right-click on the group you want to edit in the **Joints** folder or on the Timeline and select **Edit Rigid Group**.

15.2 Interference Detection

Interference can occur when you are using either the distributed design or multi-body design approach.

In a design that has multiple components, it is possible to position a component so that two components occupy the same space at the same time. To identify if components overlap, you can analyze the model for interference. To conduct an interference analysis, in the INSPECT panel, click

(Interference).

Interference analyses are conducted between selected components. Components can be selected either in the graphics window or in the BROWSER. To run an interference analysis on all components, select all of them in the graphics window. Once analyzed, the results are displayed in the Interference Results dialog box and highlighted areas in the model indicate where interference is occurring, as shown in Figure 15–2.

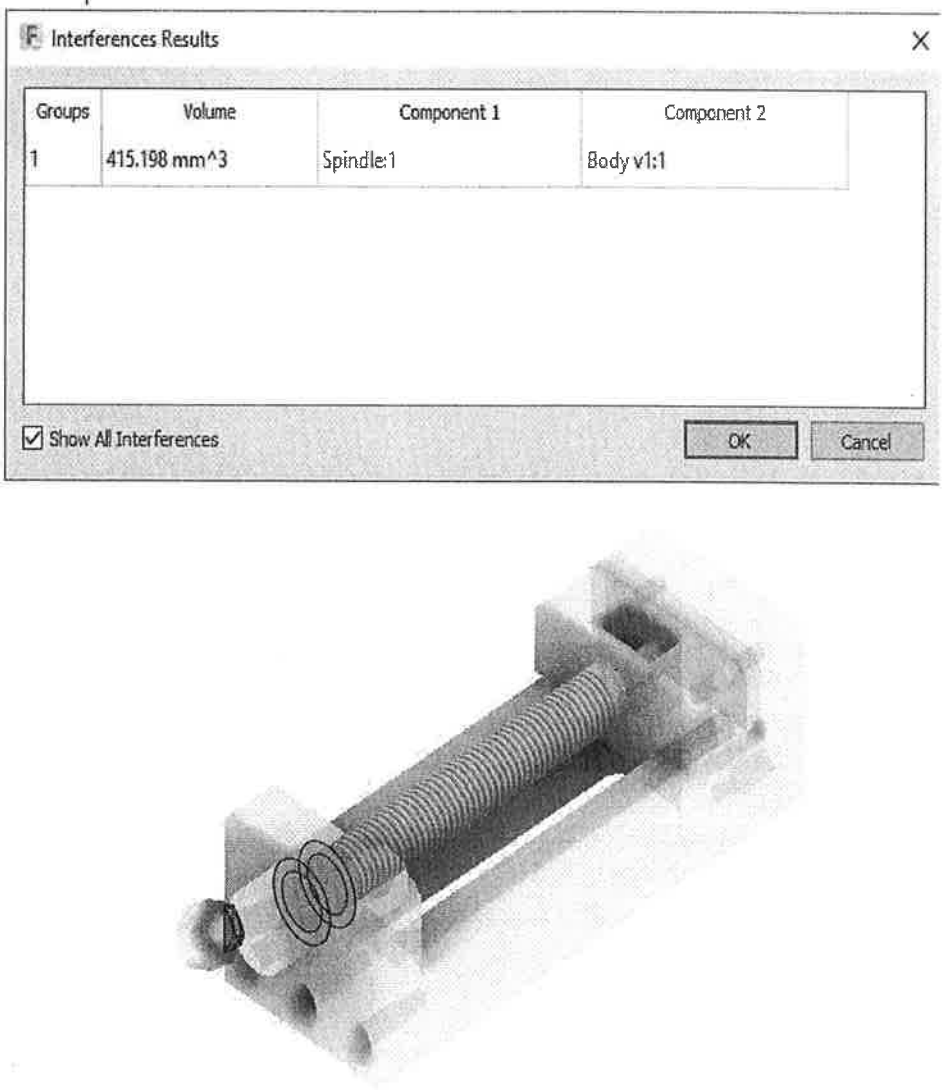

Figure 15–2

15.3 Miscellaneous Joint Tools

Joint Limits

To further control the motion that is permitted by a joint, you can define a specific range of motion to make the part more realistic and to limit the motion prior to when components come in contact. In the example shown in Figure 15–3, the cylinder tube was constrained to a pin using a revolute joint type, which defaults to permitting 360° of rotation. If this was permitted, interference would occur with other geometry.

Limits cannot be specified for where the degree of freedom does not exist in the model.

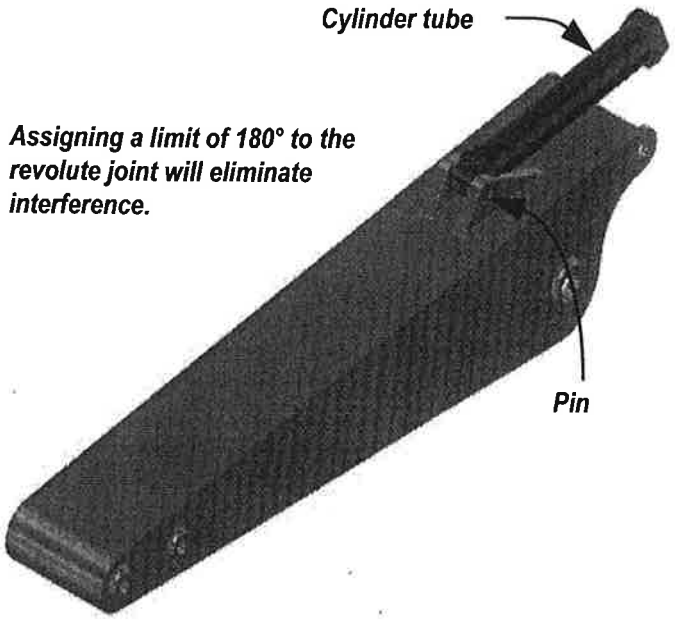

Figure 15–3

To assign limits, in the **Joints** folder in the BROWSER or in the Timeline, right-click on a joint's name or icon and select **Edit Joint Limits**. Alternatively, in the BROWSER, hover the cursor over the joint name and select (Edit Joint Limits). The EDIT JOINT LIMITS palette opens as shown in Figure 15–4.

You can also locate a joint in the BROWSER and Timeline by hovering the cursor over the joint icon in the graphics window until it highlights.

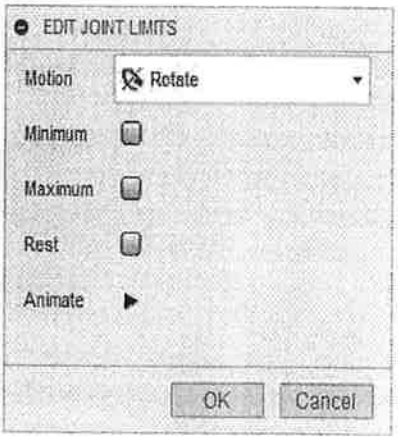

Figure 15–4

- Depending on the permitted range of motion for the joint that is being edited, the options available in the *Motion* drop-down list varies. For joints that have two or more degrees of freedom (DOF), select the joint that is to have limits set.
- To limit the motion in both directions for the set DOF, enable the **Minimum** and **Maximum** options. The Autodesk® Fusion 360® software estimates these values based on when the components will come in contact. If these values are not satisfactory, enter new values.
- The values for the *Minimum* and *Maximum* fields can be entered manually, or you can drag the joint glyph in its range of motion to define the limits.
- The **Rest** option can be used to define a home position for the components referenced during the joint's reference assignment. Its values must be between the maximum and minimum values.
- Once the limits are set, click  in the *Animate* area to play the animation and verify the limits.

Drive Joints

When testing the joint motion for a component, you can drag the component through its range of motion. To stop the motion at a specific point in its range, in the ASSEMBLY panel, use the **Drive Joints** tool. This tool enables you to enter a specific value for a selected joint, and moves the component to this position. This tool is ideal for positioning components for interference checking.

Contact Sets

One way to define the range of motion for a component is to assign joint limits. Another is to use the **Contact Sets** tool. When active, this tool analyzes for interference as you drag a component through its range of motion. Once contact is determined, motion is stopped. This method can be efficient as you do not have to manually calculate the required values used to set a joint limit, but can require significant computer processing power.

- You can use the Contact Sets tool to automatically analyze the interference between all of the components, or you can select specific components to define contact sets for analysis.
- It is recommended that contact analysis should be disabled unless you are testing movement.

Motion Linking

On the ASSEMBLE panel, the **Motion Link** tool enables you to set a relationship between the movement of two components. You can do this by relating the degrees of freedom of the joints that were used to constrain the components. When dragged, the components that have motion links established move based on the defined relationship. This can be used to simulate and study the motion in a design.

Motion Studies

In the Autodesk Fusion 360 software, you can create a motion study that enables you to simulate the motion in a design without dragging the components. A motion study runs over 100 steps where you can set values for the joints being studied to show how the design should move. Multiple joints can be added into a study and the position of each joint can be customized. When the motion study is played, the entire design is displayed in motion at once, enabling you to ensure that it has been set up correctly and is working as expected.

Practice 15a

Incorporating Motion Between Components in a Design

Practice Objectives

- Insert components in an design.
- Select and assign joints that capture the motion of the component in the overall design.
- Assign joint limits and contact sets to set the range of motion for components in the design.
- Conduct a motion study on the joints in the design.
- Add a motion link condition to joints in the design.

In this practice, you will create a new design and insert components, as shown in Figure 15–5. To complete this design, you will create a rigid group, add joints, assign limits to a joint, and create a contact set to accurately define how the design moves. You will then create and conduct a motion study analysis to study the movement and add a motion link that relates the slider to the revolute joints.

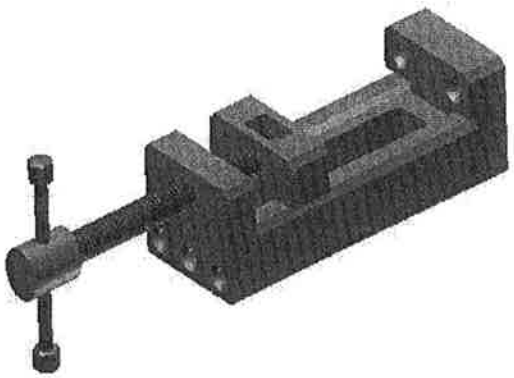

Figure 15–5

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

In this task, you will prepare to create a design by loading all of the models that will be used into your Autodesk Fusion 360 project. All of the models are imported geometry from the Autodesk Inventor software.

1. In the Application Bar at the top of the interface, click (Show Data Panel). Verify that the active project is **Autodesk Fusion 360 Practice Files**. If it is not, return to the top-level folder for this project before continuing by selecting in the Data Panel.
2. At the top of the Data Panel, click **New Folder**. Type **Vise_Assembly** as the name of the new folder.
3. Double-click on the *Vise_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 computer and select the four files in the *Vise_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 15–6.

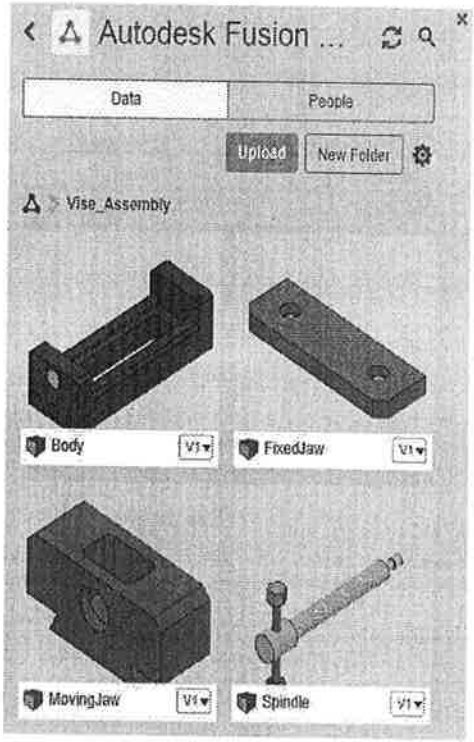

Figure 15–6

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 and then ground the component.

1. In the **File** menu, create a new design using the **New Design** option.
2. Save the new design as **Vise_Assembly**. The components cannot be inserted into this design until it is saved. Ensure that the *Location* drop-down list is set to **Autodesk Fusion 360 Practice Files>Vise_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 **Body** and select **Insert into Current Design**, as shown in Figure 15–7.

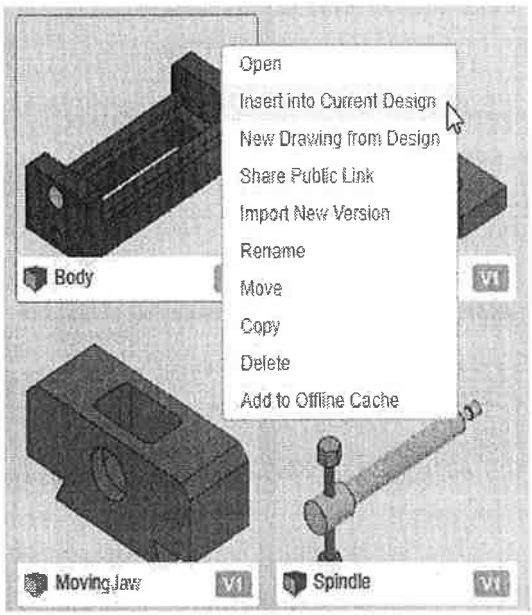

Figure 15–7

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

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

5. In the BROWSER, right-click on the **Body v1:1** component and select **Ground**, as shown in Figure 15–8. This sets the component as the grounded component in the assembly. This component should be grounded because it forms the foundation of the design and does not physically move relative to the other components in the design.

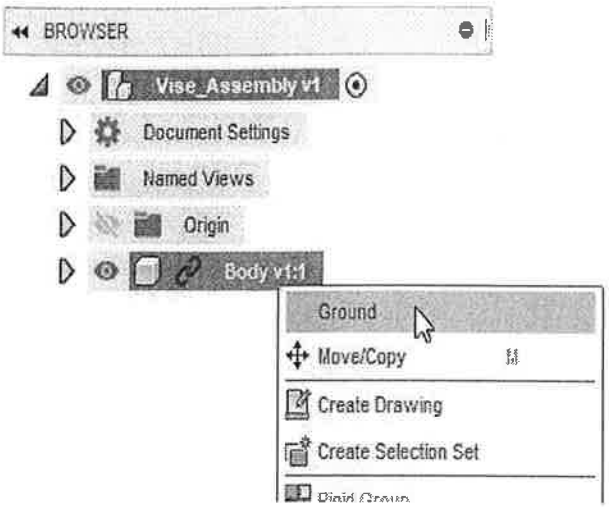

Figure 15–8

Components that are inserted into a design maintain the link to the original component. The link can be broken in the BROWSER by right-clicking on the filename and selecting ***Break Link****.*

6. In the BROWSER, note that the Body 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 15–9.

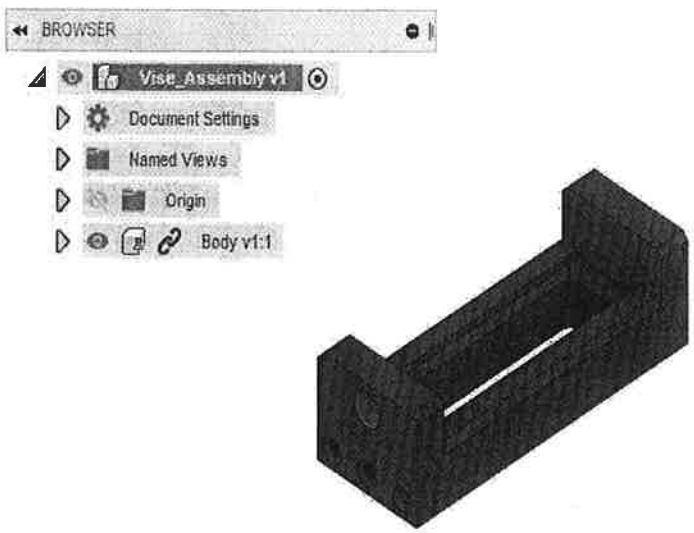

Figure 15–9

Task 3 - Add and join the FixedJaw component.

In this task, you will add and join the FixedJaw component into the design. The final placement of the component is shown in Figure 15–10.

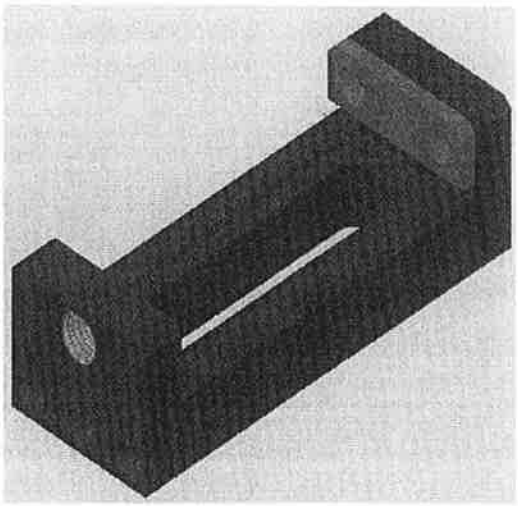

Figure 15–10

1. In the Data Panel, right-click on **FixedJaw** and select **Insert into Current Design**.

Moving components during an Insert operation can help you locate the component and more easily select references during joint creation.

2. Using the Move triad, position the component similar to that shown in Figure 15–11. This places the component in a better location for assigning references during joint assignment. Click **OK**.

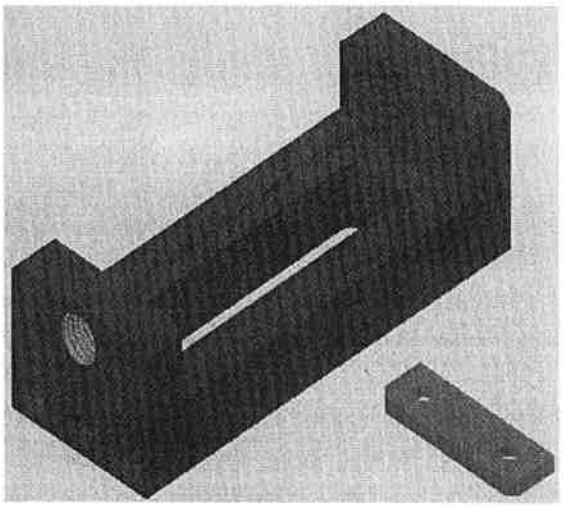

Figure 15–11

3. In the ASSEMBLE panel, click (Joint). The JOINT palette opens.

4. In the *Type* drop-down list, select **Rigid** (if not already selected by default). The FixedJaw component cannot move in the assembly, so the Rigid joint type is assigned.

5. The *Component1* field is active by default. Hover the cursor over the circular edge of the hole in the FixedJaw component, as shown in Figure 15–12. Once the center point of the edge is active, click on the center point to select it as the joint origin.

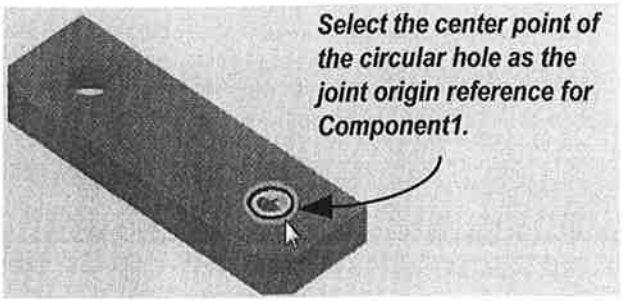

Figure 15–12

6. Hover the cursor over the circular edge of the hole in the Body component, as shown in Figure 15–13. Once the center point of the edge is active, click on the center point to select it as the joint origin.

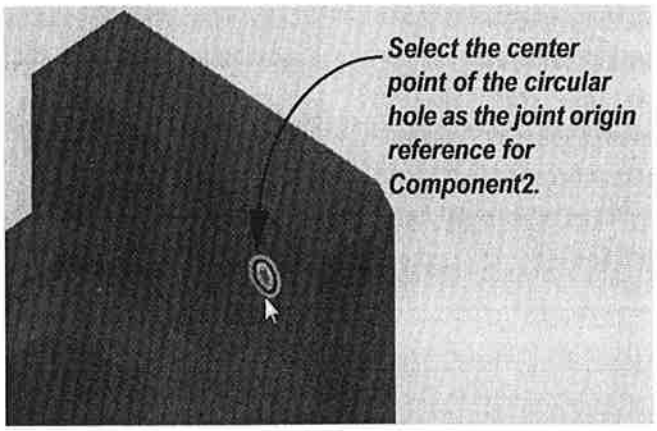

Figure 15–13

7. The FixedJaw component moves into position and a short animation of its motion is played. Because this is a rigid connection, the animated motion only shakes slightly, indicating that it has no degrees of freedom remaining.

8. Click **OK** in the JOINT palette to complete the creation of the rigid joint.

Task 4 - Add another component and add a joint to locate it in the design.

In this task, you will insert the MovingJaw component in the design and assign the Slider joint type. Once inserted and placed, the design should display as shown in Figure 15–14.

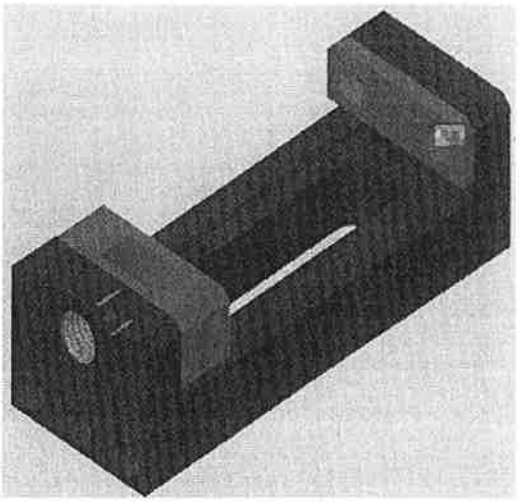

Figure 15–14

1. In the Data Panel, right-click on **MovingJaw** and select **Insert into Current Design**.
2. Using the Move triad, position the component similar to that shown in Figure 15–15. Click **OK**.

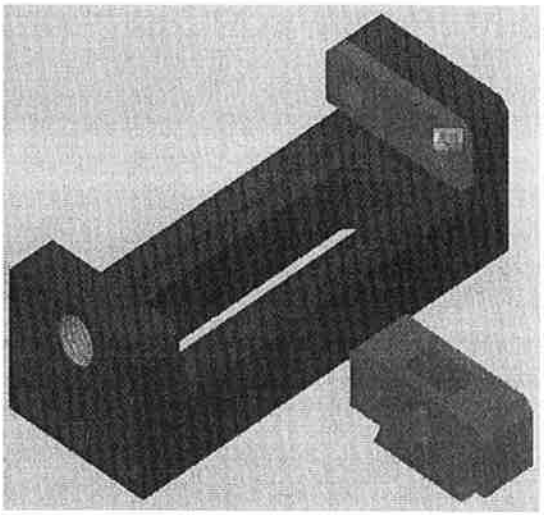

Figure 15–15

3. In the ASSEMBLE panel, click (Joint). The JOINT palette opens.

4. In the *Type* drop-down list, select **Slider**. This joint permits sliding so that the holes in the two components remain aligned.

5. The *Component1* field is active by default. Hover the cursor over the circular edge of the hole in the MovingJaw component, as shown in Figure 15–16. Once the center point of the edge is active, click on center point to select it as the joint origin.

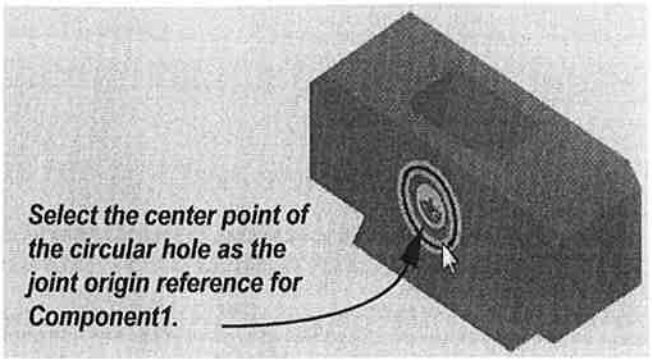

Figure 15–16

6. Rotate the design and hover the cursor over the circular edge of the hole in the Body component, as shown in Figure 15–17. Once the center point of the edge is active, click on the center point to select it as the joint origin.

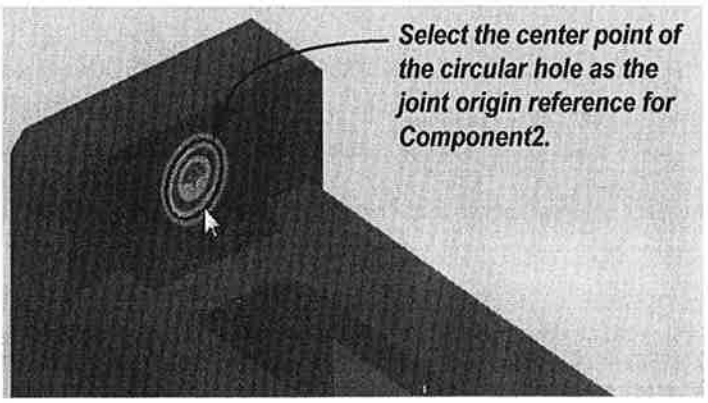

Figure 15–17

7. The MovingJaw component moves into position and a short animation of its motion is played. In the *Motion* area, ensure that the *Slide* option is set to **Z Axis** (if not, select it from the drop-down list). This ensures that sliding is permitted along the Z axis.

8. To replay the animation, in the *Motion* area, click . Click to stop the animated motion.

9. In the JOINT palette, click **OK** to complete the creation of the slider joint.

10. Select and drag the MovingJaw component. Note that you can drag the component through the Body component. In the next task, you will learn how to constrain the movement.

11. In the POSITION panel, click (Revert) to return the MovingJaw component to the original location it was placed when it was inserted.

Note that the Rigid and Slider joint icons display on the model, helping to identify how the components are joined.

*To toggle off the display of the Joint icons, in the BROWSER, select the icon associated with each joint in the **Joints** folder.*

Hovering over a joint name in the BROWSER highlights the joint in the Timeline.

Task 5 - Define the range of motion for the MovingJaw component.

In this task, you will learn how to assign limits to the motion of a component.

1. In the Timeline, right-click on the Slider joint that was added between the MovingJaw and Body components. Select **Edit Joint Limits**. Alternatively, in the BROWSER, you can expand the **Joints** folder, right-click on **Slider1**, and then select **Edit Joint Limits** or click (Edit Joint Limits) that displays adjacent to the joint name.

2. The EDIT JOINTS LIMITS palette opens. Note that the only permitted motion of this joint is sliding, as shown in the *Motion* drop-down list in Figure 15–18.

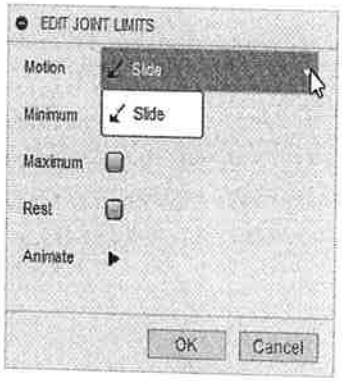

Figure 15–18

3. Select the **Minimum** and **Maximum** checkboxes to enable you to set limits on the range of motion.

4. In the *Minimum* drop-down list, select **Measure**. Measure the distance between the FixedJaw and MovingJaw components, as shown in Figure 15–19:
   1. On the MovingJaw component, select a reference on the face that the FixedJaw will touch.
   2. Select the face of the FixedJaw component that will touch the MovingJaw component.

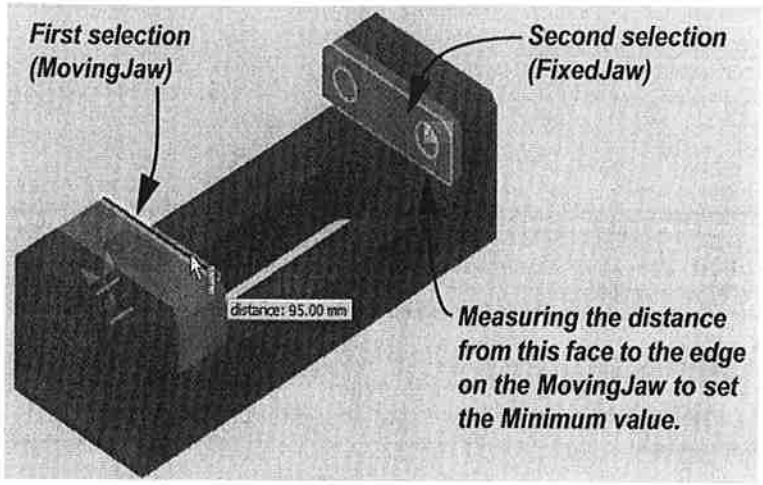

Figure 15–19

5. The *Minimum* field displays a value of approximately **95 mm**. Change the value to a negative value (i.e., approximately -95 mm).

6. Enter **0.00 mm** in the *Maximum* value field.

7. In the *Animate* area, click ▶ to play the range of sliding motion. Click ■ to stop the animated motion.

8. Click **OK** to complete the limits.

9. Drag the MovingJaw component and note that it no longer moves through the Body component.

Task 6 - Insert the Spindle component and create a rigid group.

In this task, you will add the Spindle component and consider its design intent in the overall design. The Spindle component contains four design models and would be considered a subassembly in CAD terms. This was imported from Autodesk Inventor as a single model. Its design intent is that it moves as one. To accomplish this, you will define a Rigid Body.

1. In the Data Panel, right-click on **Spindle** and select **Insert into Current Design**.
2. Move the component into a convenient location. Click **OK**.

When the Spindle was imported from the Autodesk Inventor software, the Autodesk Fusion 360 software retained the structure of the file. This is true for any imported CAD file (e.g. Inventor, SOLIDWORKS).

3. Note that  displays in the BROWSER, indicating that this component contains sub-components. Expand Spindle in the BROWSER to view the sub-components.
4. Select a component in the Spindle and note that it moves independently of the other components.
5. Click  (Undo) in the Quick Access toolbar to undo the move.

To constrain all of the components to move together, either three rigid joints could be added, or the components could be created as a rigid group. In this practice, you will use a rigid group.

6. In the ASSEMBLE panel, click  (Rigid Group). The RIGID GROUP palette opens as shown in Figure 15–20.

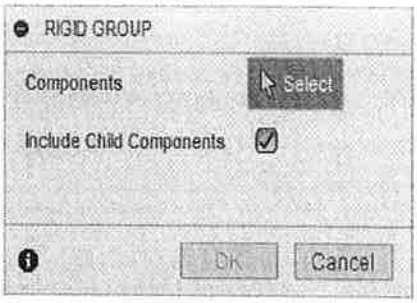

Figure 15–20

7. In the graphics window, select the four components that are in the Spindle design, as shown in Figure 15–21. The components highlight in blue.

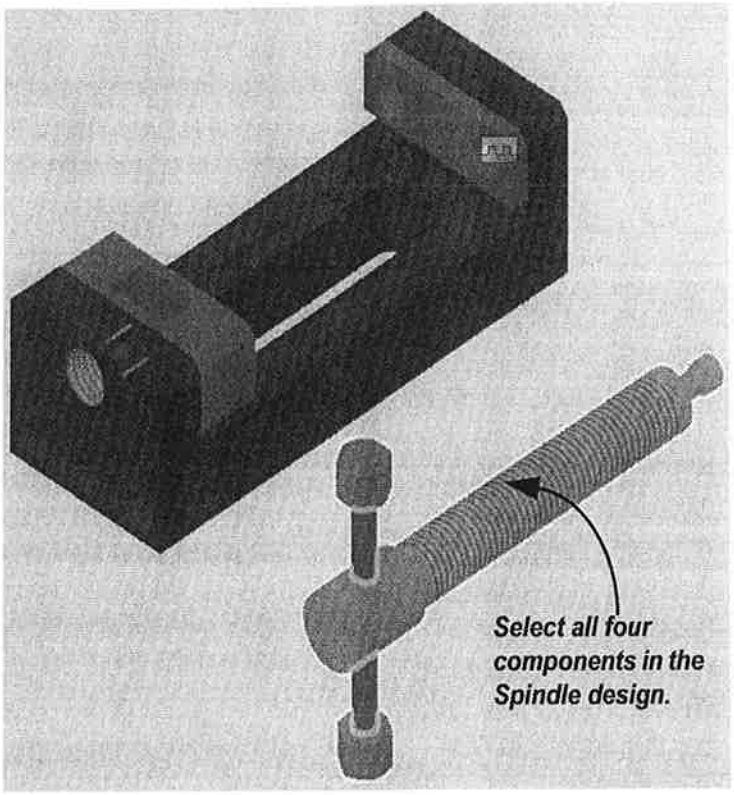

Figure 15–21

8. Click **OK** to create the rigid group. A **RigidGroup** node is added to the **Joints** folder in the BROWSER, as shown in Figure 15–22.

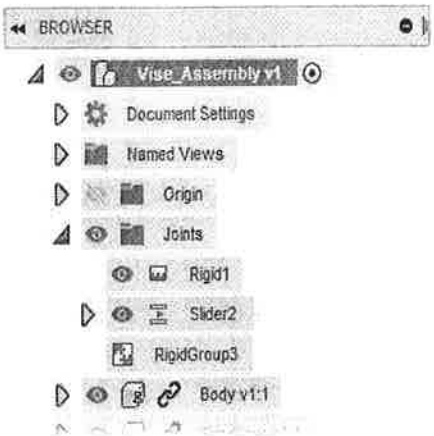

Figure 15–22

9. Rename the new group to **Spindle Rigid Group** to help you easily identify it in the BROWSER.

10. In the graphics window, try to select and drag any of the components in the Spindle. Note how the designs all move as one. At this point, only a single joint is needed to constrain the component to the rest of the design.

11. Return the components to their original position.

Task 7 - Add a joint between the Spindle and MovingJaw.

1. Drag the MovingJaw component into the middle of the Body component. This will help you to select the joint references for the Spindle.

2. In the ASSEMBLE panel, click (Joint).

3. When you are prompted that some of the components have been moved, click **Capture Position.** This ensures that the component remains where it is for component reference selection, and adds a (Position) action to the Timeline.

4. In the *Type* drop-down list, select **Revolute**. The Spindle component must rotate to move the MovingJaw component.

5. The *Component1* field is active by default. Hover the cursor over the circular edge at the end of the Spindle component, as shown in Figure 15–23. Once the center point of the edge is active, click on the center point to select it as the joint origin.

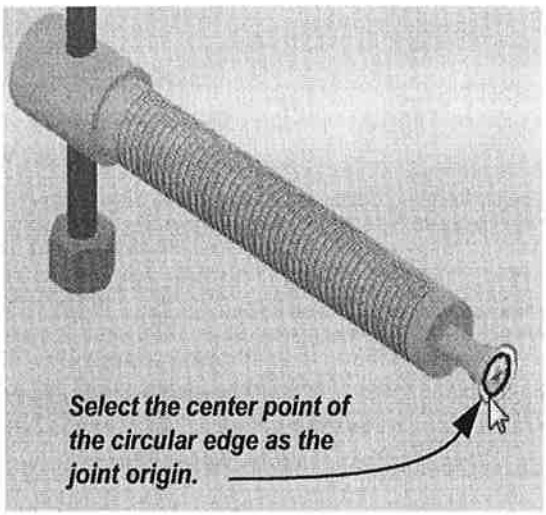

Figure 15–23

6. Hover the cursor over the face of the MovingJaw component that is shown in Figure 15–24. Once the center point of the face is active, click on the center point to select it as the joint origin.

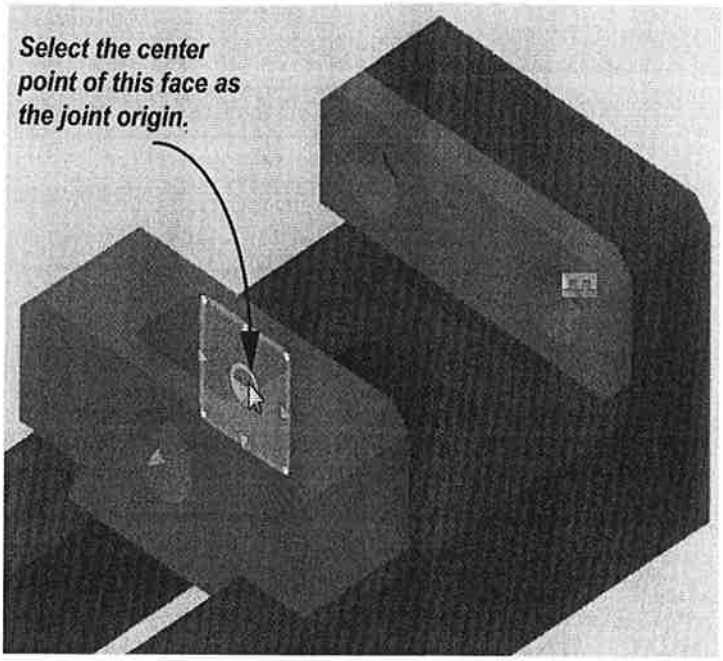

Figure 15–24

7. The spindle moves into position and a short animation of it rotating is displayed.

8. In the JOINT palette, click **OK** to complete the creation of the revolute joint. Note that all of the remaining components in the Spindle rigid group move into position to maintain the group.

9. Review the components. Note that there seems to be possible interference between the Spindle and MovingJaw components, as well as between the Spindle and the Body.

10. In the INSPECT panel, click (Interference). The INTERFERENCE palette opens.

11. Select the Spindle, MovingJaw, and Body components in the graphics window.

12. In the *Compute* area, click (Compute) to run the analysis. The Interference Results dialog box and the model display as shown in Figure 15–25, indicating that there is interference. The interference occurs because the center point on the face of the MovingJaw was not in line with the axis of the hole. A new joint origin must be created to place this component.

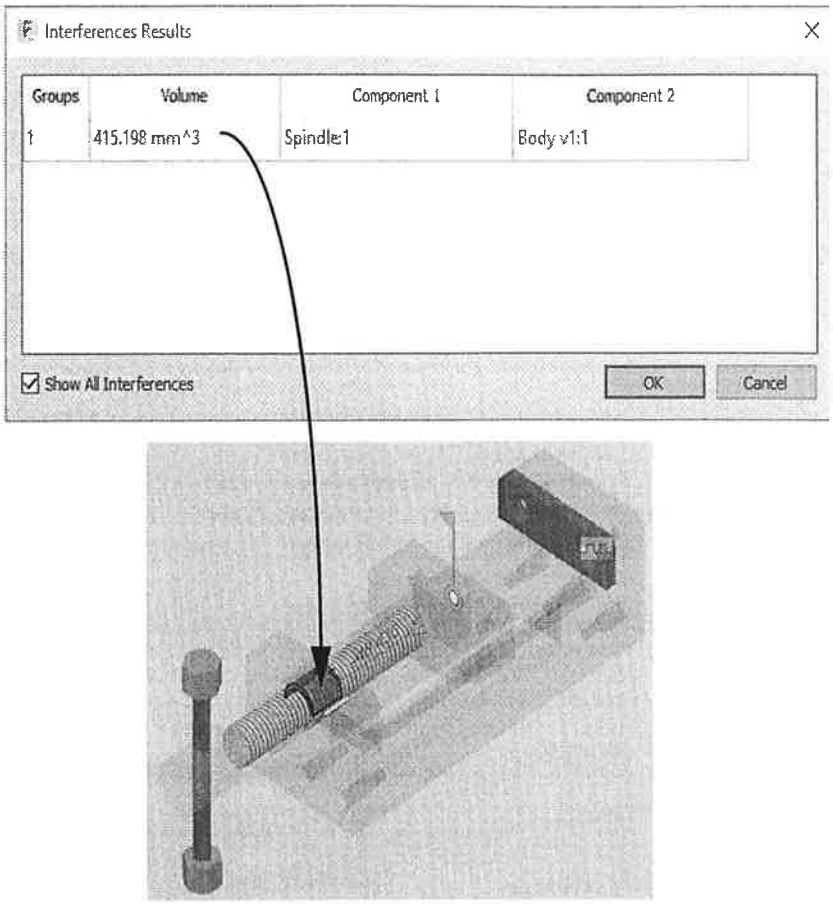

Figure 15–25

13. Click **OK** to close the Interference Results dialog box.

14. Right-click on the **Rev4** revolute joint that was created and select **Delete** to remove it.

Task 8 - Create a joint origin in the MovingJaw component.

In this task, you will create a new joint origin. Since the design was created using the Distributed Design method, joint origins must be created in each individual design file so that it can be saved with the file. Once created, you will use this joint origin to create the joint.

1. In the BROWSER, right-click on **MovingJaw** and select **Open**. This opens this design in its own window.

2. Switch to the *DESIGN* workspace, if it is not already active. Rotate the design so that it displays similar to that shown in Figure 15–26. This helps to ensure that the construction entities are easily created and selectable.

3. In the CONSTRUCT panel, click **Axis Through Cylinder/Cone/Torus** and select the cylindrical face at the front of the component, as shown in Figure 15–26.

4. In the CONSTRUCT panel, click **Point at Edge and Plane**. Select the axis that you just created and then select the planar face. The references are shown in Figure 15–26.

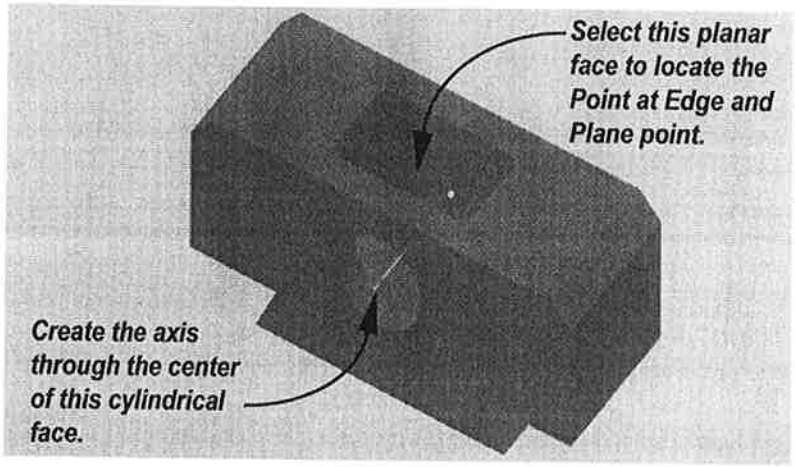

Figure 15–26

5. In the ASSEMBLE panel, click (Joint Origin).

6. In the JOINT ORIGIN palette, in the *Type* field, ensure that **Simple** is selected.

7. Rotate the model so that you can clearly see the point. Select the point that was just created on the face of the MovingJaw component.

8. Ensure that the new joint origin was created on the point and not on another vertex. Once confirmed, click **OK**.

9. In the BROWSER, hide the construction features by clicking (Show/Hide).

10. Save the component.

11. Return to the *Vise_Assembly* window. Note that the BROWSER indicates that a component and the top-level assembly are out-of-date ( ).

12. Right-click on **MovingJaw v1** and select **Get Latest**. The assembly updates to show the new joint origin.

13. Using the steps previously listed in Task 7, recreate the revolute joint between the Spindle and the MovingJaw components. Use the new joint origin as the *Component2* reference.

14. In the INSPECT panel, click (Interference). Select the Spindle, MovingJaw, and Body components to test for interference again. Click (Compute) to run the analysis. No interference is detected.

15. Drag and rotate the Spindle to verify the motion is correctly defined. Note how when the vise is closed, the Spindle interferes with the Body component.

Task 9 - Define a contact set between components.

As a alternative to explicitly calculating and setting up joint limits to define the full range of permitted motion for a component, you can also define the range of motion based on the physical contact of components in the design. This is done using contact sets.

1. Drag the Spindle to a position where there is no contact between it and the Body component.

2. In the ASSEMBLE panel, click (Enable Contact Sets). In the BROWSER, the **Contact: sets** node is added, as shown in Figure 15–27. This indicates that contact sets are being analyzed in the model.

*Enabling (Enable All Contact) verifies contact between all of the components in a design. This can be intensive on your computer's resources. In situations where there are only a few possible locations for interference, consider using the **Contact Sets** option and explicitly defining the components to be analyzed.*

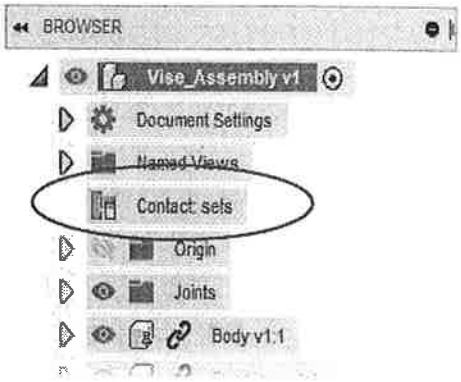

Figure 15–27

3. Currently there are no sets actually created in the model. In the ASSEMBLE panel, click (New Contact Set). The NEW CONTACT SET palette opens, enabling you to define a new set.

4. In the graphics window, select the Body and Spindle components, as shown in Figure 15–28.

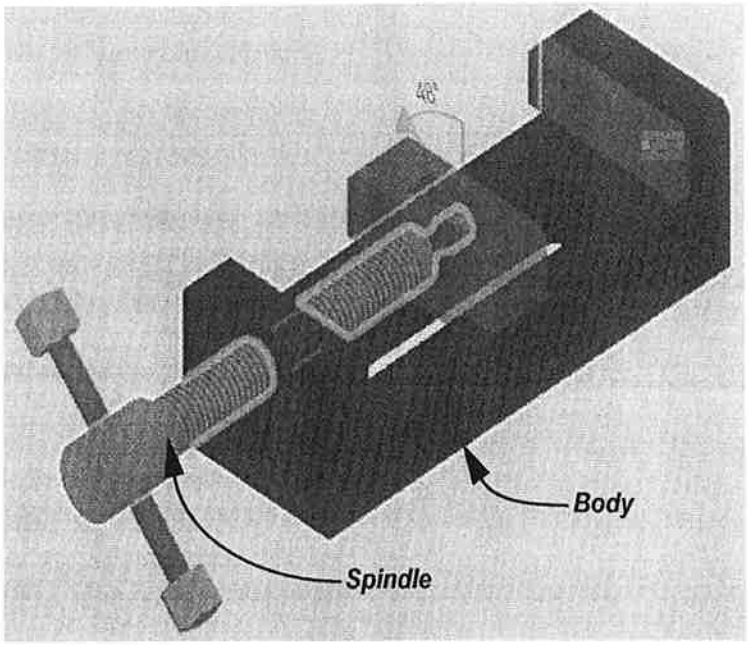

Figure 15–28

5. Click **OK** to create the contact set. It is added to the **Contact: sets** node in the BROWSER, as shown in Figure 15–29.

Figure 15–29

6. Rename the newly created contact set as **ContactSet: Body_Spindle** to easily identify it.

7. Drag and rotate the Spindle again and note how the vise does not fully close. When the Spindle interferes with the Body component, movement stops because contact and interference has been identified.

You can suppress a contact set to improve performance in the design when you are not concerned about testing motion.

8. Hover the cursor over the **ContactSet: Body_Spindle** set and click (Suppress contact set). Once suppressed, you can once again close the vice, even though there is interference.

9. Hover the cursor over the **ContactSet: Body_Spindle** set and click (Unsuppress contact set). The Spindle's movement now ends once it comes in contact with the Body component.

10. As an alternative to individually suppressing contact sets, you can also temporarily disable all contact analysis. Hover the cursor over the **Contact: sets** node. Note the icons that display, as shown in Figure 15–30.

Figure 15–30

*When hovering over the **Contact: sets** node in the BROWSER, you can switch from analyzing contact sets to analyzing all contact between components by selecting (All Bodies Contact).*

11. Click (No Contact) to disable the use of all contact sets in the model. Note that the **Contact: sets** node is completely removed from the BROWSER.

12. To enable contact sets again, it must be selected in the ASSEMBLE panel. Click (Enable Contact Sets). Once enabled, all previous contact sets are displayed.

Task 10 - Define a specific rotation value for a joint.

When dragging a component through its range of motion, you are unable to enter a specific value to stop the component at a specific point in its permitted range of motion. In this task, you will learn how to use the **Drive Joints** tool to enter a value and move the design to a specific location within its permitted range of motion.

1. Drag the MovingJaw along the Body component. Note that there is no entry field to set a specific location for the component.

2. In the ASSEMBLE panel, click  (Drive Joints). The DRIVE JOINTS palette opens.

3. In the BROWSER, expand the **Joints** folder and select the **Rev** joint.

4. The *Rotation* field is added to the DRIVE JOINTS palette. The default value is based on the current angular value in the model. Enter **45 deg** and note how the revolute joint angle automatically updates. This enables you to define a specific location for the handle.
   - You can also set a specific rotation value for the Slider2 joint to set the position the MovingJaw component.

5. In the DRIVE JOINTS palette, click **OK**.

6. Save the design.

Task 11 - Conduct a motion study.

Depending on the design, you might not be able to drag multiple components at one time to test their range of motion. Using the Motion Study tool, you can add specific joints to the study, assign values, and play the study to review its motion. In this task, you will create a motion study that includes the Slider and Revolute joints.

1. In the BROWSER, expand the **Joints** folder. Rename the **Slider2** joint as **MovingJaw/Sliding**.

2. Rename the **Rev5** joint as **Spindle Rotation**. Renaming the joints makes it easier to identify them when using them in the motion study.

Consider the order of selection if the motion you are simulating is sequential. In this case, add the joints that move first, then second, etc.

3. In the ASSEMBLE panel, click (Motion Study). The Motion Study dialog box opens.
4. In the BROWSER, select the **MovingJaw/Sliding** joint and then the **Spindle Rotation** joint.
5. Note how each joint is assigned a color in the panel on the right of the dialog box, as shown in Figure 15–31.

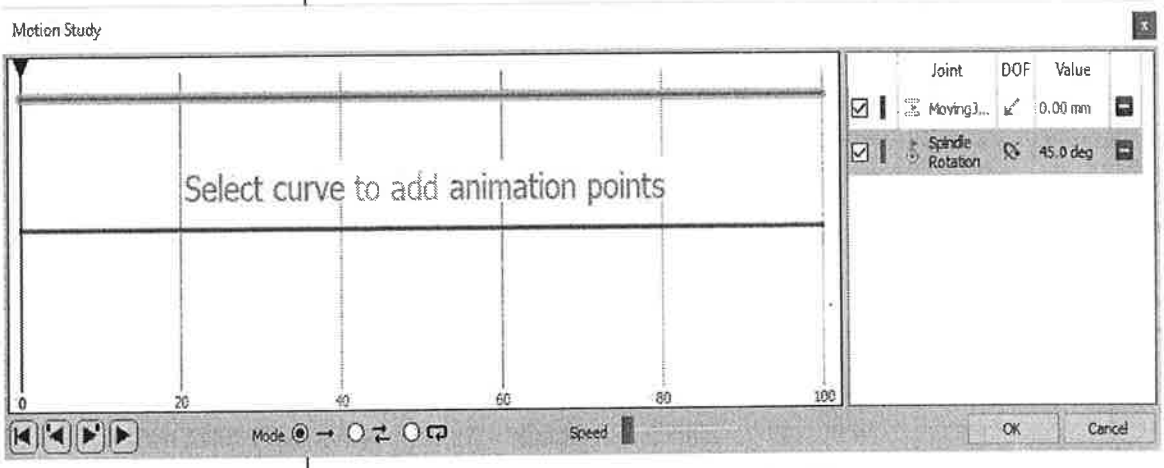

Figure 15–31

6. Hover the cursor on the blue curve (Spindle Rotation joint) and select a location near the beginning (Step 1) of the line to define its first position in the study.
7. Ensure that **1** is displayed in the *Step* field, entering it manually if required, as shown in Figure 15–32.
8. In the *Angle* field, enter **0.0 deg**, as shown in Figure 15–32.

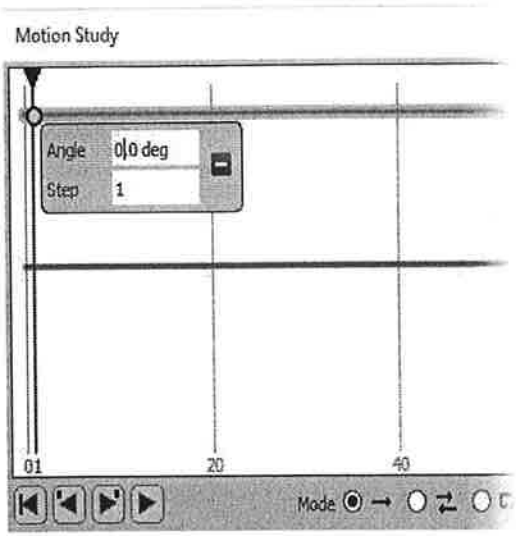

Figure 15–32

9. Press <Enter>.

10. Move further along the blue line and select near Step 20. Ensure that **20** is displayed in the *Step* field, entering it manually if required. In the *Angle* field, enter **360.0 deg**, as shown in Figure 15–33.

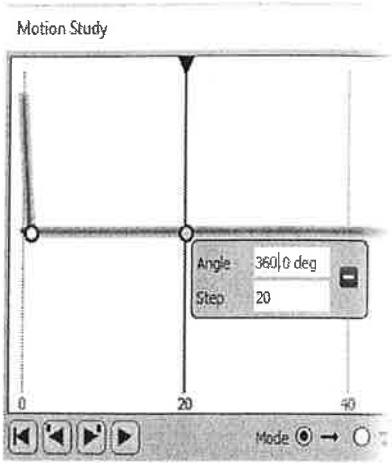

Figure 15–33

To delete a motion study point, click ▬ in the window that displays to define a point.

11. Continue to move along the blue line that defines the motion for the revolute joint and enter the following values:

Step	Angle
40	720 deg
60	1080 deg
80	1440 deg
100	1800 deg

Once you have entered all of the points for the motion study, the dialog box should display as shown in Figure 15–34.

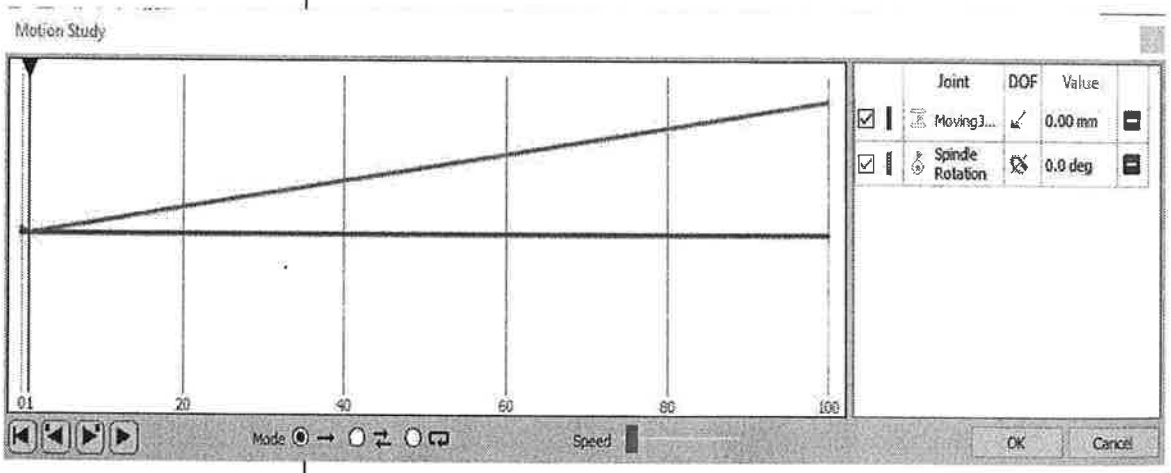

Figure 15–34

12. Hover the cursor on the red curve (MovingJaw/Sliding joint) and select a location near the beginning (Step 1) of the line to define its first position in the study.

13. Ensure that **1** is displayed in the *Step* field, entering it manually if required, as shown in Figure 15–35.

14. In the *Distance* field, enter **-55 mm**, as shown in Figure 15–35.

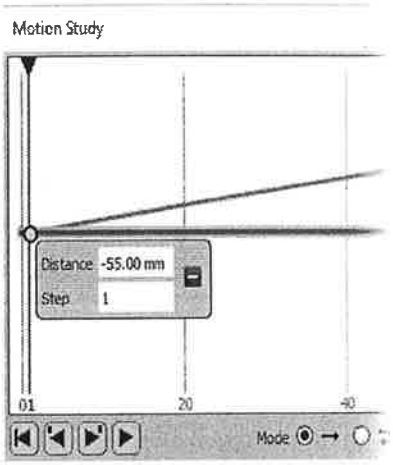

Figure 15–35

15. Press <Enter>.

16. Move further along the red line and select near Step 20. Ensure that **20** is displayed in the *Step* field, entering it manually if required. In the *Angle* field, enter **-45 mm**, as shown in Figure 15–36.

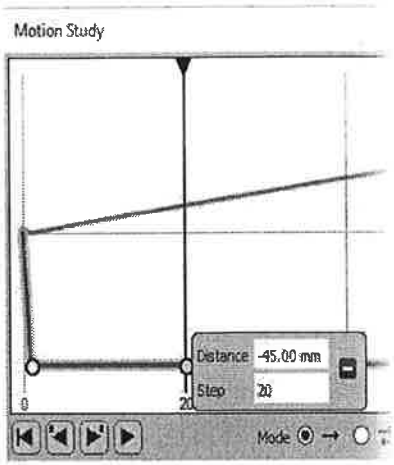

Figure 15–36

To delete a motion study point, click in the window that displays to define a point.

17. Continue to move along the red line that defines the motion for the slider joint and enter the following values.

Step	Angle
40	-35 mm
60	-25 mm
80	-15 mm
100	-5 mm

Once you have entered all of the points for the motion study, the dialog box should display similar to that shown in Figure 15–37. The dialog box might vary depending on the original position of the MovingJaw.

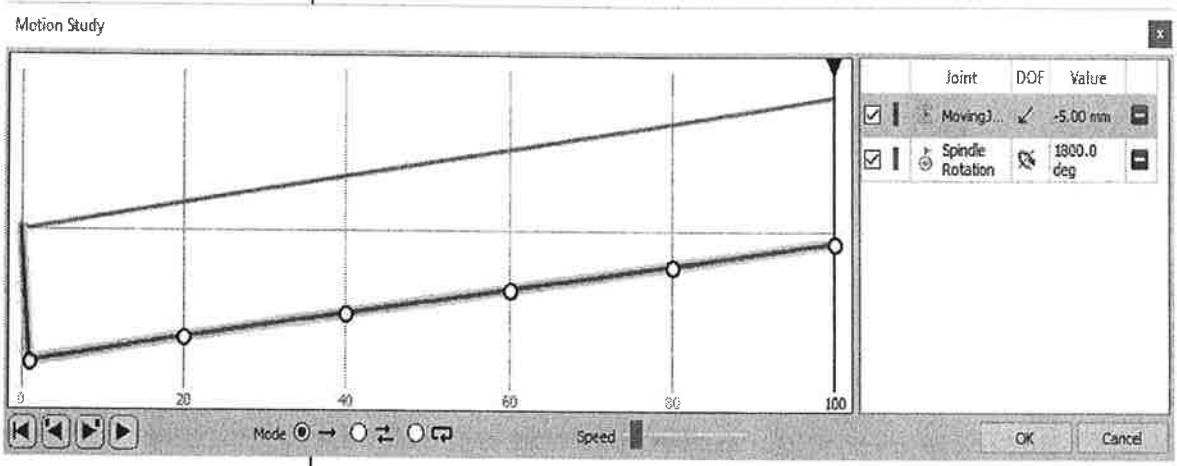

Figure 15–37

Consider using the Mode options at the bottom of the Motion Study dialog box to control the playback mode. It can be set to play forward, play forward and then backward, or loop.

18. To play the motion study, you can do either of the following:
    - Drag the scrub line that displays in the pane on the left along the 100 step line.
    - Use the playback controls at the bottom of the Motion Study dialog box. Click to rewind to the beginning of the study and click to play the study.
19. Drag the *Speed slider* to the left to slow the motion as much as possible.
20. Click **OK** to close the study.

21. Studies are added to the BROWSER in the **Motion Studies** folder, as shown in Figure 15–38.
    - To edit an existing study, in the BROWSER, right-click on the study and select **Edit**.
    - To create a new study, in the ASSEMBLE panel, click (Motion Study).

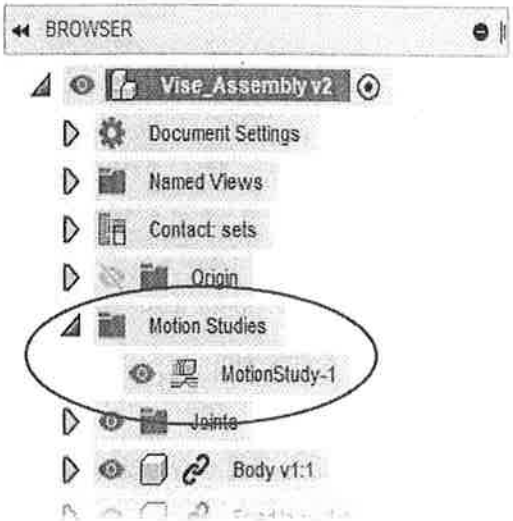

Figure 15–38

22. Save the design.

Task 12 - Create a motion link between the slider and revolute joints to simulate motion.

As an alternative to using the Motion Study tool to simulate motion, you can also use the Motion Link tool. In this task, you will create a relationship between the slider and the revolute joints that defines how far the MovingJaw will slide for every 360 degree rotation of the Spindle.

1. In the ASSEMBLE panel, click (Motion Link). The MOTION LINK palette opens.
2. In the BROWSER, select the slider (**MovingJaw/Sliding**) joint first and then select the revolute (**Spindle Rotation**) joint.

3. In the MOTION LINK palette, in the *Distance* field, enter **20.00 mm**, and in the *Angle* field, enter **360.0 deg**, as shown in Figure 15–39. This sets the motion so that for every 360° rotation, the MovingJaw moves 20 mm.

Figure 15–39

4. Click ▶ in the *Animate* area to play the range of sliding motion. Click ■ to stop the animated motion.
5. Click **OK**. Note that the new motion link is added in the BROWSER to the **Joints** folder.
6. Drag the MovingJaw and note how the motion link controls the movement of the design.
7. Save the design to the *Autodesk Fusion 360 Practice Files* project.
8. Close the file.

Command Summary

Button	Command	Location
	**Disable Contact**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> ASSEMBLE panel • **Context Menu**: Right-click in the graphics window and select **Assemble**. • **BROWSER:** Right-click the **Contact** node.
	**Drive Joints**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> ASSEMBLE panel • **Context Menu**: Right-click in the graphics window and select **Assemble**. • **BROWSER:** Right-click the *Joint* name or hover over the *Joint* name.
	**Edit Joint Limits**	• **BROWSER:** Right-click the *Joint* name or hover over the *Joint* name and select the icon. • **Timeline:** Right-click the *Joint* icon.
N/A	**Edit Rigid Group**	• **BROWSER:** Right-click the *Rigid Group* name. • **Timeline:** Right-click the *Rigid Group* icon.
	**Enable All Contact**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> ASSEMBLE panel • **Context Menu**: Right-click in the graphics window and select **Assemble**. • **BROWSER:** Right-click the **Contact** node and select icon when **Enable Contact Sets** is active.
	**Enable Contact Sets**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> ASSEMBLE panel • **Context Menu**: Right-click in the graphics window and select **Assemble**. • **BROWSER:** Right-click the **Contact** node and select icon when **Enable All Contact** is active.
	**Interference**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> INSPECT panel • **Context Menu**: Right-click in the graphics window and select **Inspect**.
	**Motion Link**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> ASSEMBLE panel • **Context Menu**: Right-click in the graphics window and select **Assemble**.
	**Motion Study**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> ASSEMBLE panel • **Context Menu**: Right-click in the graphics window and select **Assemble**.

	**New Contact Set**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> ASSEMBLE panel • **Context Menu**: Right-click in the graphics window and select **Assemble.** • **BROWSER:** Right-click the **Contact** node.
	**Rigid Group**	• **Toolbar:** *DESIGN* Workspace>*SOLID* tab> ASSEMBLE panel • **Context Menu**: Right-click in the graphics window and select **Assemble.**
	**Suppress Contact Set**	• **BROWSER:** Right-click the **Contact: sets** node.