

## Exercise: Project Setup

You must first download and install the following dataset to perform this exercise.

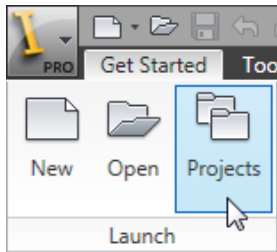
In this first section of the test drive, you learn how to set up a project in Inventor®.

The data files for each exercise are located in the folder where you installed the training materials. To access these data files for the exercises, you must make the Test Drive 2012 project the active project in Autodesk® Inventor®.

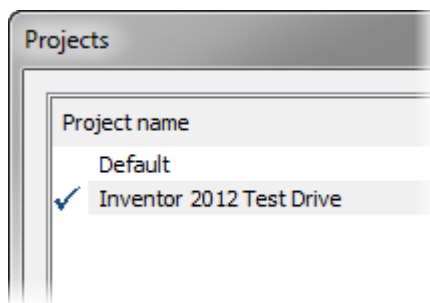
1. Start Autodesk Inventor 2012 by double-clicking the shortcut icon on your desktop.



2. On the ribbon, Get Started tab, Launch panel, click Projects.



3. If a Test Drive 2012 project does not appear in the Projects window, click Browse at the bottom of the dialog box, navigate to the folder where you installed the training materials, and then double-click the *Inventor 2012 Test Drive.ipj* file.
4. In the Projects window, verify that Inventor 2012 Test Drive has a checkmark next to it. If it does not, double-click it in order to make it the active project.



5. Click Done at the bottom of the Projects window.

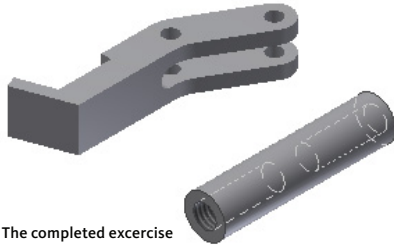


By default, the exercises are installed in the *C:\Autodesk Learning\Inventor 2012\Test Drive* folder.

## Exercise: Part Modeling

In this part of the test drive, you work with Autodesk Inventor and Inventor Fusion to create two different parts. During the creation of the parts you:

- Open an assembly file and review an engineer's note for the completion of the design.
- Start a new part and import an AutoCAD drawing to use as the basis of the Claw.
- Start a new part and model the link rod.
- Modify the Claw part in Inventor Fusion.
- Use Change Manager to apply the changes made in Inventor Fusion to the Inventor part file.



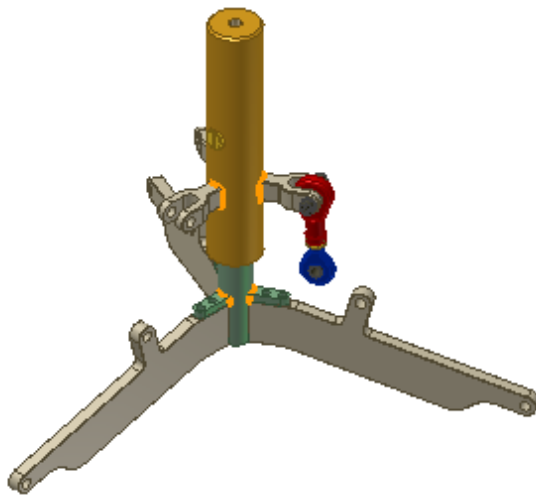
The completed exercise



### Review the Assembly and Engineers Notebook

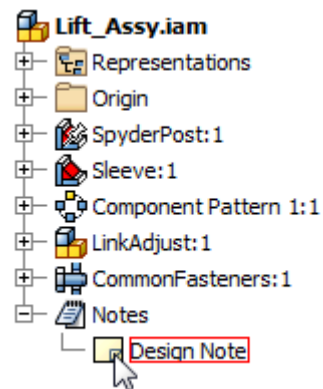
In this section of the test drive, you open and review a partially completed assembly and a note in the Engineers Notebook.

1. Open *Lift\_Assy.iam*. The partially completed assembly displays as shown.

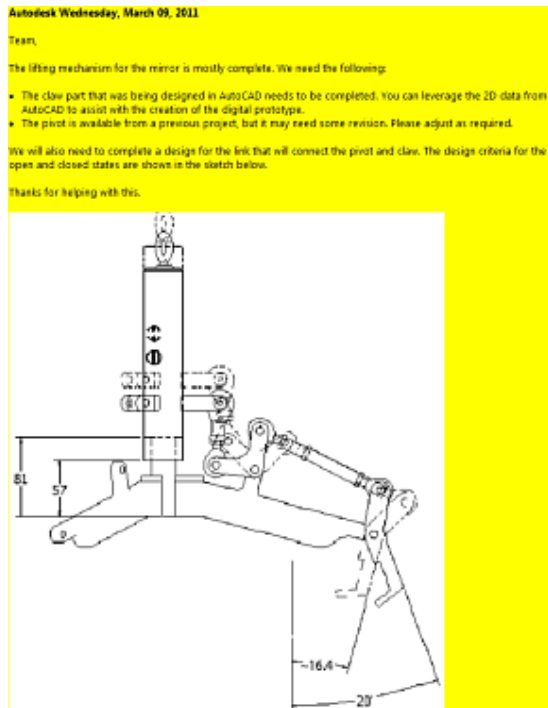


2. Use the Orbit (1) and Zoom (2) tools to view the assembly. When finished return to the Home View (3).

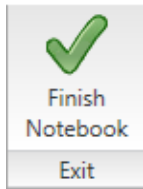
3. The assembly includes a design note created in the Engineers Notebook included in Autodesk Inventor. To view the note, in the browser under Notes, double-click Design Note.



4. Review the note. This note indicates what needs to be completed in the design. During this test drive, you will create and modify parts and subassemblies to complete this digital prototype.



5. On the Engineer's Notebook tab, Exit panel, click Finish Notebook.



## Import Data from a DWG File

In this section of the test drive, you begin creating a component by importing data from a DWG™ file.

1. On the Quick Access toolbar, click New.



2. In the New File dialog box:

- Click the Metric tab.
- Double-click the *Standard (mm).ipt* template.



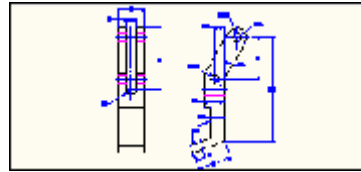
Standard (mm).ipt

A new part file is created with the sketch environment active.

3. To begin to insert AutoCAD data:

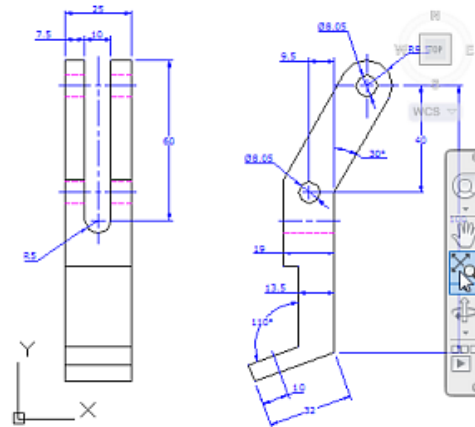
- On the Sketch tab, Insert panel, click ACAD.

- In the Open dialog box, click once on *Claw-2D.dwg*. Notice that the drawing is previewed in the lower left corner of the dialog box as shown.
- Click Open. The DWG Import Wizard is activated.

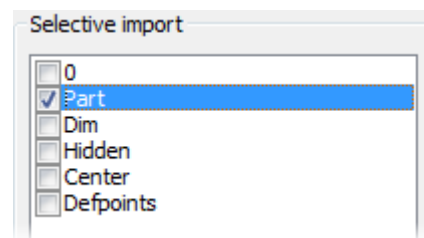


4. To preview the data contained in the AutoCAD file:

- To change the background color of the Import Preview, in the top right corner of the dialog box, select the white button.
- To pan the preview, position the cursor over the preview and then click and hold down the wheel-mouse while dragging to a new position.
- Use the wheel-mouse to dynamically zoom in and out of the preview.
- On the Navigation Bar, click Zoom Extents.

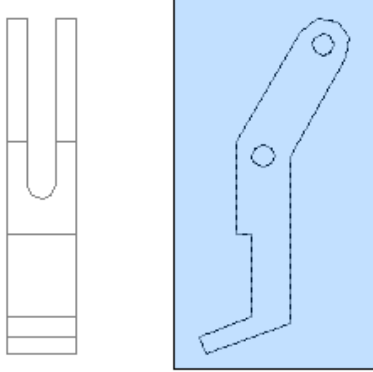


5. To select the layers to import, in the Selective Import list, clear all of the check boxes except for the Part check box as shown. Notice the preview displays only the drawing geometry that resides on the selected layer.



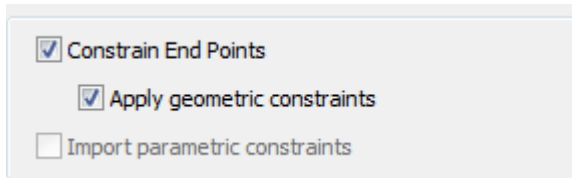
6. To select the geometry to import:

- In the Selection area, clear the All check box.
- In the preview window, click and drag a selection box that contains the objects in the right profile as shown.
- Click Next.

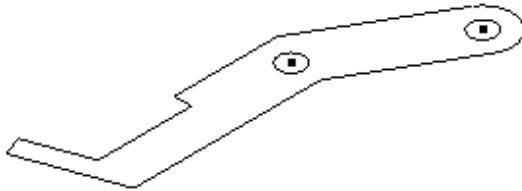


7. On the Import Destination Options page of the wizard:

- Click the Constrain End Points and Apply geometric constraints check boxes as shown.
- Click Finish

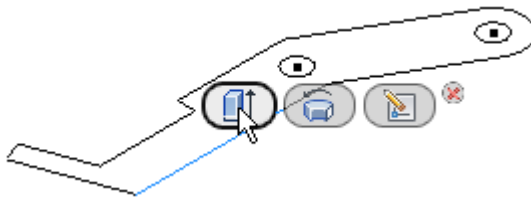


8. In canvas, right-click. On the marking menu, click Finish 2D Sketch.

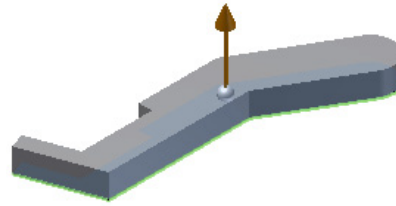


9. To begin creating the solid part from the sketch, in canvas:

- Select a segment of the sketch.
- On the in-canvas mini toolbar, click Extrude.

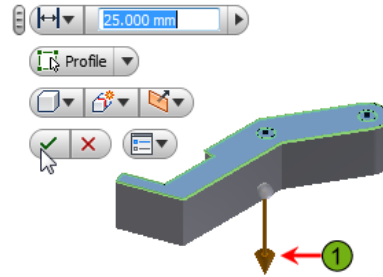


10. Select the area inside the outer boundary and the two holes defined by the sketch objects.



11. For the extrusion distance and direction, in canvas:

- Click and drag the arrow handle (1) down until it has an extrusion distance of 25 mm. If you want, after dragging the arrow handle down, you can directly enter the required distance in the field.
- Click OK.



12. To save the file:

- On the Quick Access toolbar, click Save.
- In the Save As dialog box, File Name field enter Claw.
- Click Save.



13. Close *Claw.ipt*.

## Create the Link Rod Component

In this section of the test drive, you create the link rod component for the assembly from scratch in a new part file.

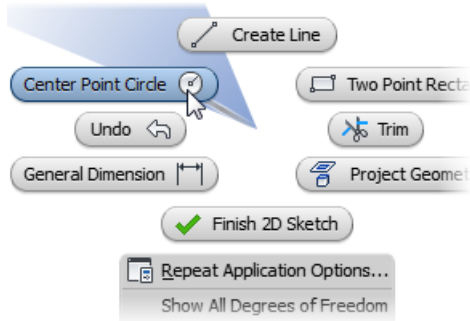
1. Start a new part file. To do this, do the following:

- On the Quick Access toolbar, click New.
- In the New File dialog box, Metric tab, double-click the *Standard (mm).ipt* template.

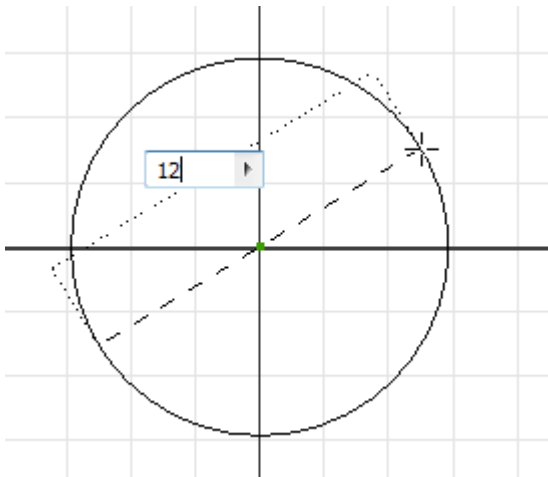


Standard (mm).ipt

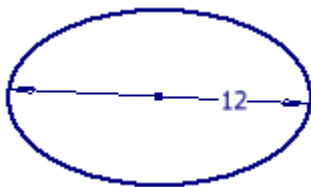
2. To begin to sketch and size the base circle shape for the link rod, in canvas, right-click. In the marking menu, click Center Point Circle.



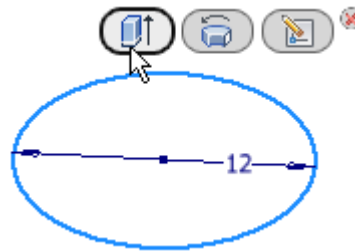
3. Draw the circle by doing the following:
  - For the center of the circle, click the point at the origin of the sketch.
  - In the dimension field, enter 12. Make sure you press ENTER after typing in the value.



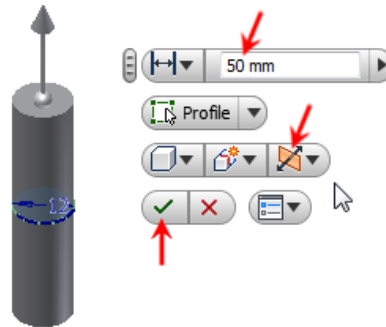
4. Finish editing the sketch. To do this, in canvas, right-click. On the marking menu, click Finish 2D Sketch.



5. To begin to create a cylinder from this circle, in canvas:
  - Select the circle.
  - On the mini-toolbar, click Extrude.

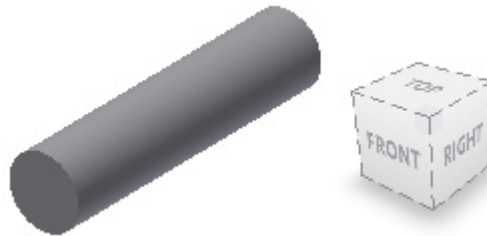


6. In the Extrude mini-toolbar:
  - In the Distance field, enter 50 mm.
  - In the direction list, select Symmetric.
  - Click OK. The feature is now created like the preview showed.



Note: The value of 50 mm will be modified later in the test drive. The actual length for this part will be determined based on other design criteria of the assembly. You will use Autodesk Inventor's adaptive capability to establish the length and thus the value for the dimension.

7. To change the viewing angle of the part:
  - On the Navigation Bar, click Free Orbit. Rotate your view to match as shown.
  - In canvas, right-click. Click Done.

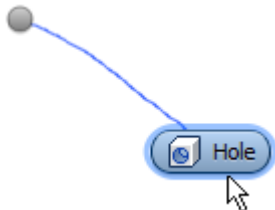


Note: Note: You can hold down the F4 key or SHIFT + wheel-mouse to activate the Free Orbit tool at any time - even while using another tool. Also, while the tool is active, you can click in canvas to have that point be the center point of the orbit.

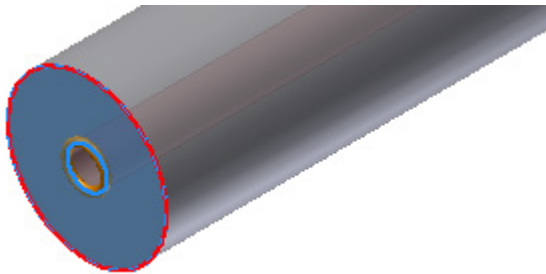
8. To redefine the Home View:
  - Click the corner of the View Cube as shown.
  - Right-click the View Cube, in the menu click Set Current View as Home > Fit to View.



9. Start the Hole tool using a gesture behavior. To do this, in canvas, right-click drag and release in a direction that is down and to the right of the initial cursor position as shown.

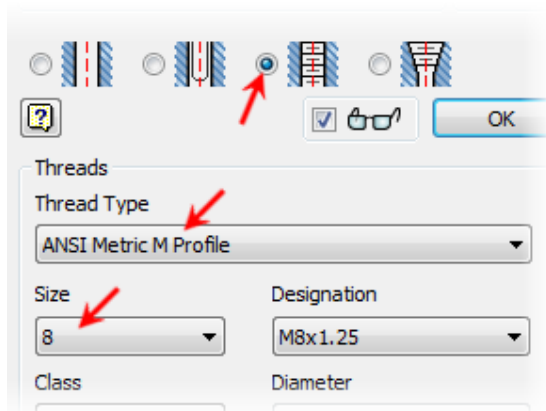


10. To position the hole:
- In the Hole dialog box, Placement list, select Concentric.
  - In canvas, select the end face of the cylinder.
  - Select the circular edge of the cylinder.



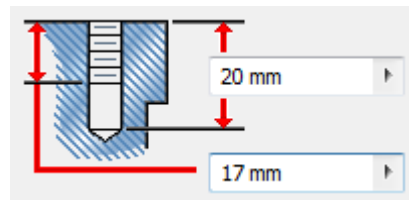
11. For the hole type and size, in the Hole dialog box:

- Select Tapped Hole.
- In the Thread Type list, select ANSI Metric M Profile.
- In the Size list, select 8.



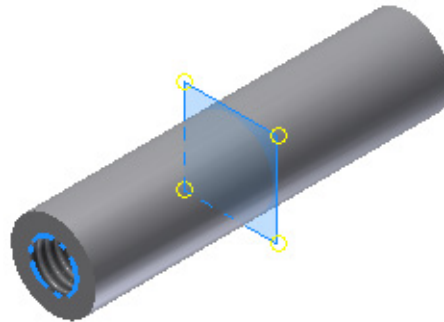
12. To specify the hole and thread depth and create the hole, in the Hole dialog box:

- In the Threads area, clear the Full Depth check box.
- In the Termination list, select Distance.
- In the hole depth field, enter 20 mm.
- In the thread depth field, enter 17 mm.
- Click OK.



13. To mirror the hole to the other end of the link rod:

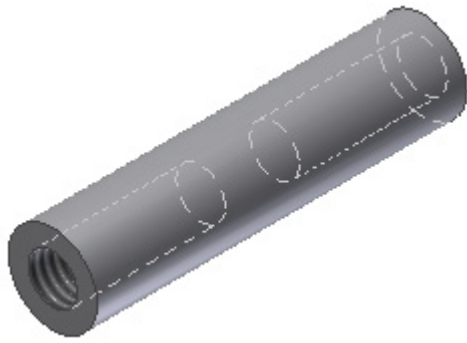
- On the Model tab, Pattern panel, click Mirror.
- In canvas, select the hole feature.
- In the Mirror dialog box, click the Mirror Plane select button.
- In the browser under Origin, select XY Plane. The preview displays as shown.
- Click OK.



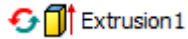
14. Orbit the view to see the mirrored feature. To do this, do the following:

- Press and hold SHIFT + wheel-mouse while moving the mouse.
- Release the key and button when done.
- Press F6 to return to the Home View.

15. To change the display so you can view all the features of the final part, on the View tab, Appearance panel, in the Visual Style list, select Shaded With Hidden Edges. The part displays as shown.



16. To set the extrusion as adaptive so its length can be dynamically changed at a later time, in the browser, right-click Extrusion1. Click Adaptive.



The adaptive symbol displays next to the feature in the browser. Later in the test drive you use this unique capability of Autodesk Inventor to have the link rod adapt (change length) to match other components in the assembly.

17. To save the file:

- On the Quick Access toolbar, click Save.
- In the Save As dialog box, File Name field, enter **LinkRod**.



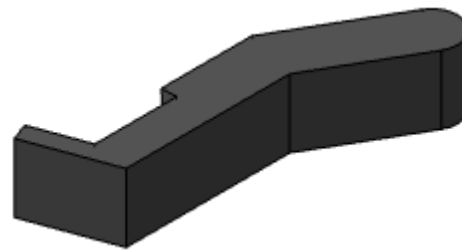
## Modify Parts in Fusion

In this section of the test drive, you open the Claw part and modify it in Inventor Fusion. This enables you to see the use of Fusion as a tool for creating and modifying digital prototypes. Fusion is available for someone who may or may not have Autodesk Inventor.

1. Start Inventor Fusion 2012 by double-clicking the Inventor Fusion 2012 shortcut icon on your desktop.

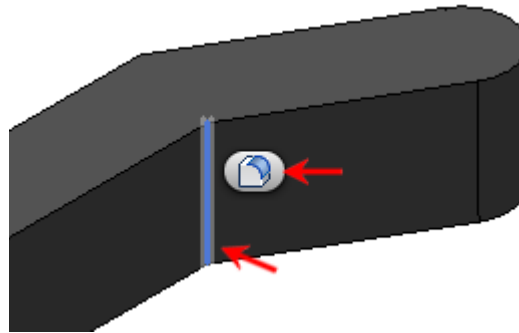


2. Open the part Claw.ipt that you created and saved in Inventor.

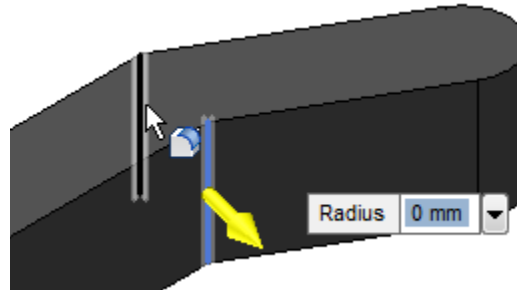


3. To begin to add fillets to the inside edge of the part, in canvas:

- Select the identified edge.
- Click Fillet.



4. Select the outside edge as shown to fillet both edges at the same time and with the same radius.

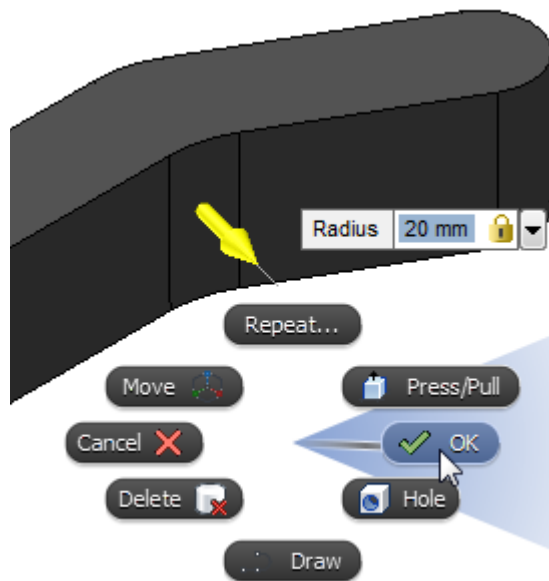


5. In the Adaptive Snap Bar, Snap Value edit box located in the lower right corner of the canvas, enter 2.

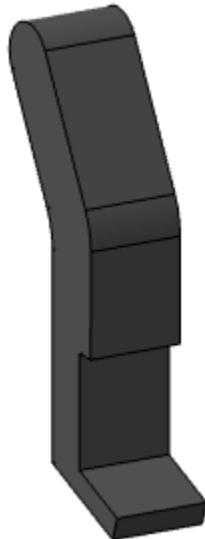


6. Set the fillet size by doing the following:

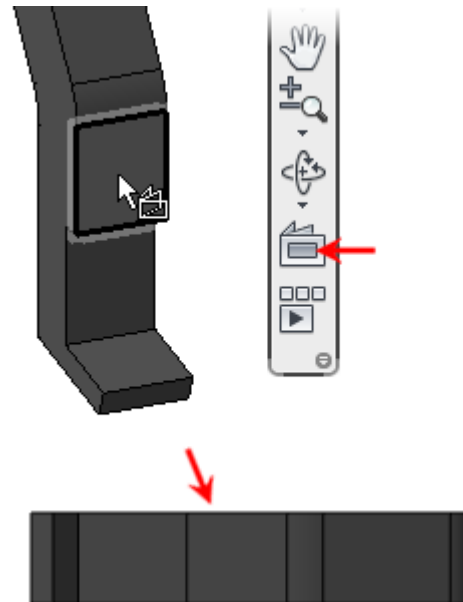
- Click and drag the fillet arrow manipulator to dynamically set the radius of the fillet.
- In the Radius field, enter 20.
- In canvas, right-click. Click OK.



7. On the Navigation Bar, click Orbit. Orbit the model to set the display approximately as shown.

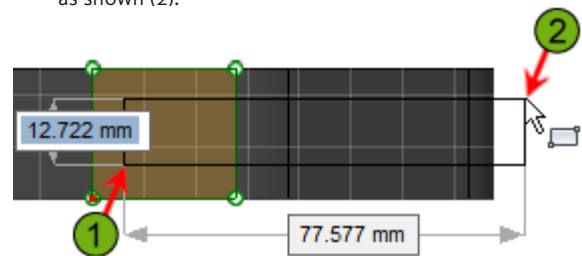


8. To look straight at one of the faces on the part:
- On the Navigation Bar, click Look At.
  - Select the face as shown.
  - The results of the Look At are shown at the bottom of the following illustration.

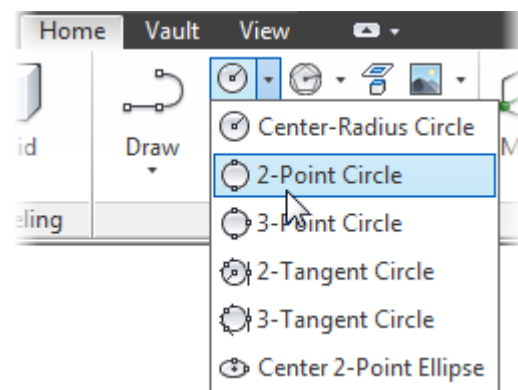


9. To begin sketching the shape for a slot through the part:

- On the Home tab, Sketch panel, click 2-Point Rectangle.
- For the first corner, click on the previously selected face at location (1).
- For the second corner, click a location outside the part as shown (2).

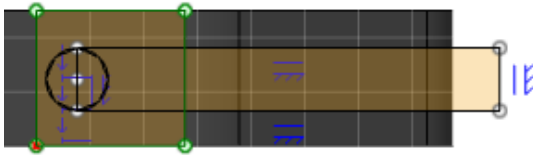


10. On the Sketch panel, click to expand the list for creating circles. Click 2-Point Circle.



11. In canvas, click the two endpoints of the rectangle to create the circle as shown. Press ESC to end the use of the tool.





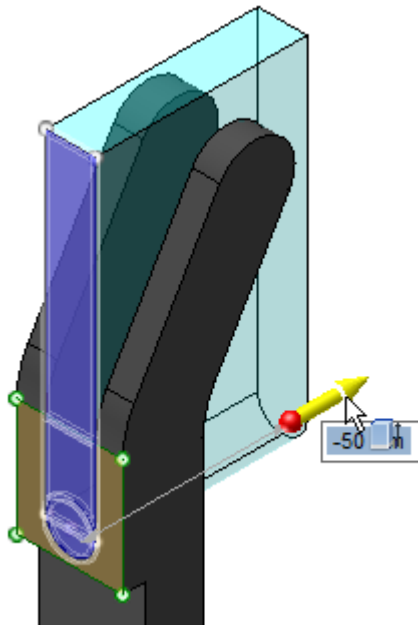
12. Set the viewing direction to preview the cut, on the View Cube:

- Click the arrow to rotate the display counterclockwise.
- Click the top right corner of the View Cube.

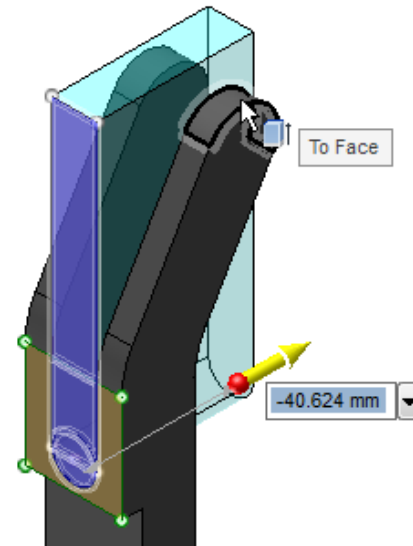


13. To begin extruding sketch areas to cut the part:

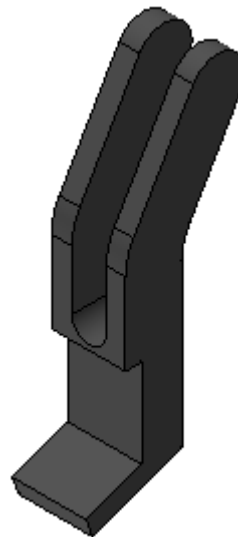
- On the Home tab, Solid panel, click Extrude.
- Select the four sketch areas.
- Click and drag the arrow manipulator back so the preview displays past the part.



14. To set the depth of the cut to match the tangent face of the part, select the face as shown.



15. On the Extrude tab, Extrude panel, click OK.



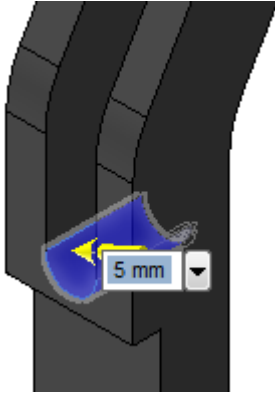
16. To begin adjusting the size of the rounded slot, in canvas:

- Select the arc face of the slot.
- Click Press/Pull.



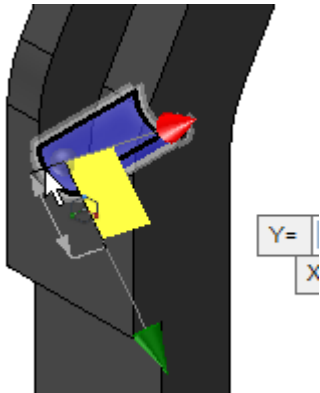
17. Drag the face to the required size. To do this, do the following:

- Click and drag the arrow manipulator, notice it increments every 2 mm.
- In the Adaptive Snap Bar, Snap Value edit box, enter 1.
- Drag the arrow handle again until it is at 5 mm.



**18.** To move the bottom of the slot:

- In canvas, right-click. Click Move.
- Select the arc face of the slot.
- Click and drag the triad origin sphere up the part.
- Release the mouse button when the triad and feature is located as shown.
- Right-click. Click OK.

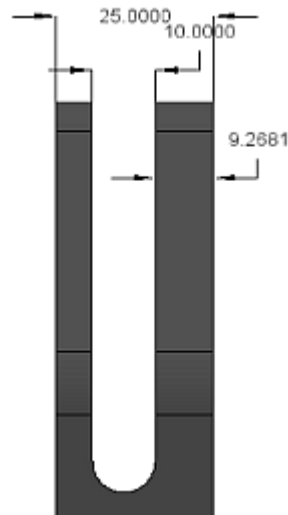


**19.** On the View Cube, click Left. The part displays as shown.



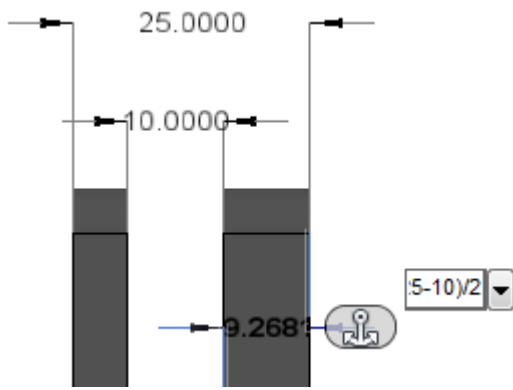
**20.** Begin to add dimensions to the 3D part to constrain its size.

- On the Home tab, Constraint & Dimension panel, click Dimension.
- Add dimensions to the outside of the part and slot as shown. Select the edges of the part along the straight edges where the 9.2681 dimension arrows point in the following illustration.
- Right-click. Click OK.

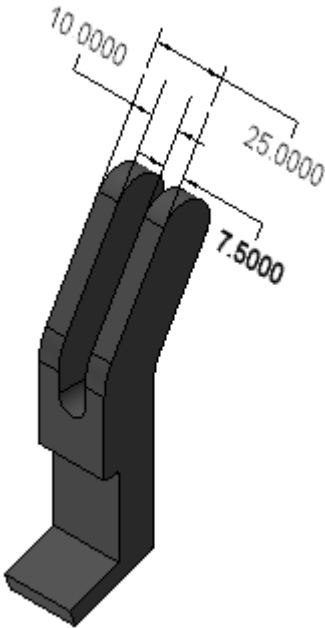


**21.** To set the material thickness value:

- Double-click the dimension for the material thickness.
- Press DELETE.
- In the dimension field, enter the equation  $(25-10)/2$ .

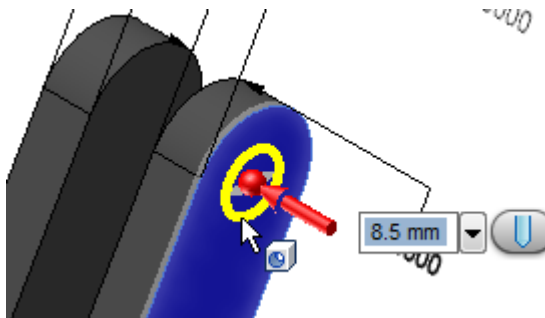


22. On the View Cube, click the upper right corner of the Left view. The viewing direction displays as shown. Note: The model dimensions were moved away from the part for clarity for your review.



23. To begin to add a hole concentric to the top rounded edge:

- In canvas, right-click. Click Hole.
- Select the side face of the part.
- Select the arc of the top rounded edge.
- Click and drag the Hole Diameter Manipulator until the diameter is 8.5 mm.

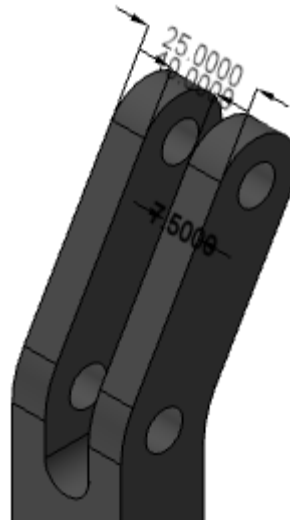


24. For the depth of the hole:

- Click and drag the arrow manipulator so the preview extends past the part.
- Click the face on the opposite side of the part.
- Right-click. Click OK.



25. Add another 8.5 mm hole that is located at the bend area of the part as shown.



26. Save your changes to *Claw.dwg*. To do this, do the following:

- On the Quick Access toolbar, click Save.
- In the Save As dialog box, click Save.

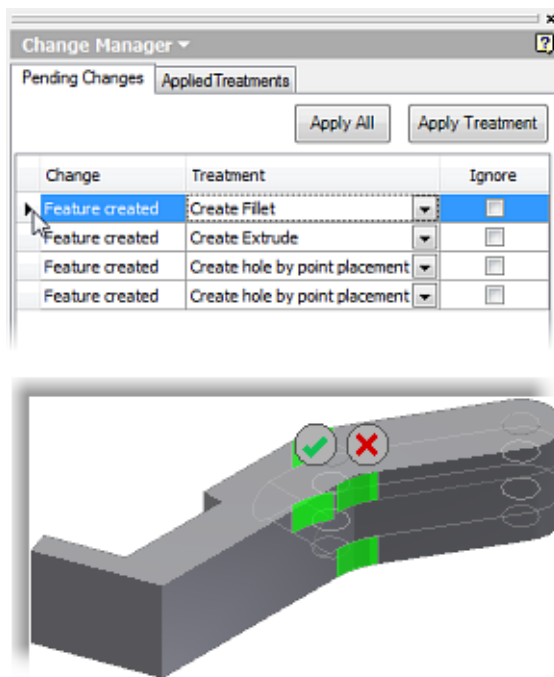
27. Close *Claw.dwg*.

## Fusion Modifications in Inventor

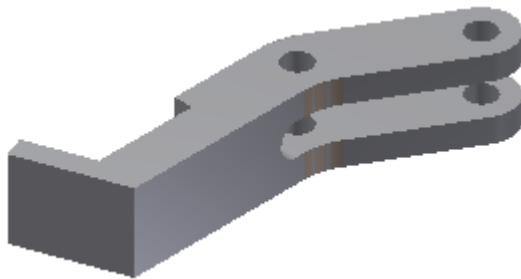
In this section of the test drive, you use Change Manager to apply the changes made in Inventor Fusion to the Inventor part file.

1. Switch to Autodesk Inventor.
2. On the Quick Access toolbar, click Open.
3. Select and open *Claw.dwg*. Make sure you select the DWG file and not the IPT file. The changes you made were saved in the DWG file.

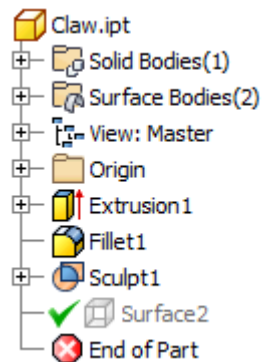
4. In the Change Manager palette, select each listed entry to have a preview of the change display in canvas.



5. In the Change Manager palette, Pending Changes tab, click Apply All.



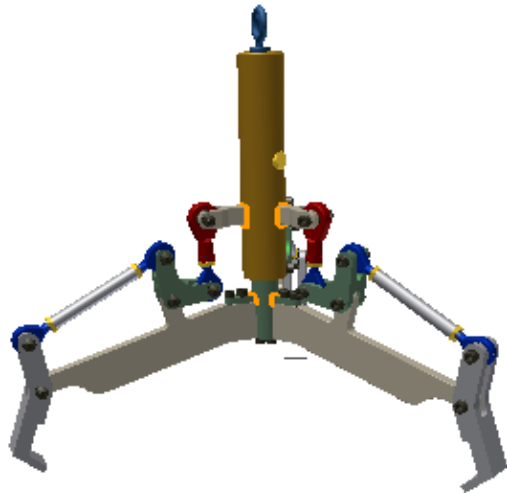
6. On the Fusion Change Manager tab, Exit panel, click Finish Change Manager.
7. In the browser, right-click Surface2. Click Visibility.



8. Save *Claw.ipt*.

## Exercise: Assembly Modeling

In this part of the test drive, you place and constrain components in the assembly. You use an Inventor® part and complete the mechanism by adapting the length of a link rod to match your design criteria. Finally, you drive an assembly constraint to examine the mechanism movement.



The completed exercise

### Add and Position Parts in an Assembly

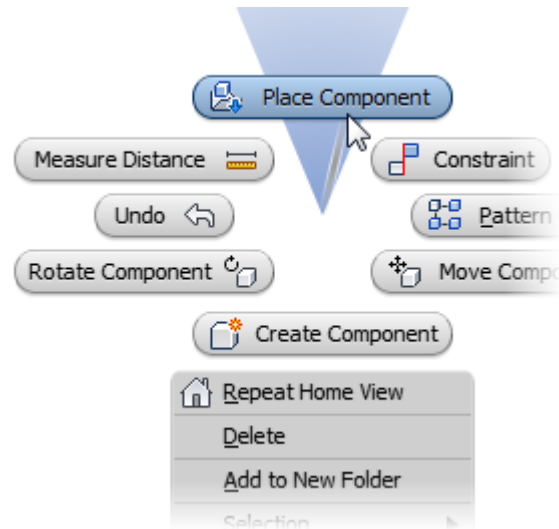
In this section of the test drive, you add the claw and lift ring parts to the assembly and add assembly constraints to position the parts.

1. Activate the lift assembly by selecting the *Lift\_Assy.iam* tab at the bottom of the canvas.

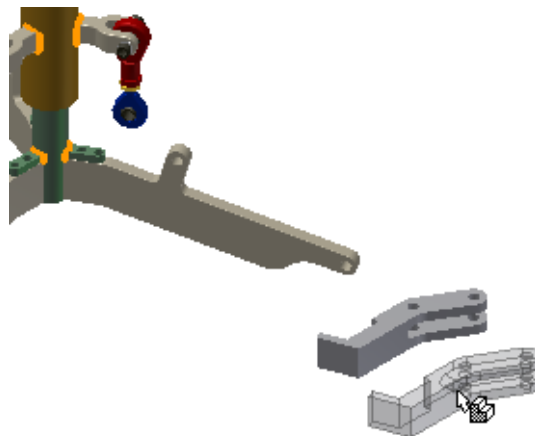


2. To begin placing the claw part in to the assembly:

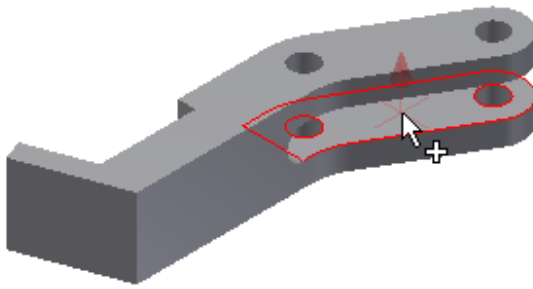
- In canvas, right-click. On the marking menu, click Place Component.
- In the Place Component dialog box, double-click *Claw.ipt*.



3. In canvas, click once near the end of the arm on the right side of the assembly. This adds one occurrence of the part.

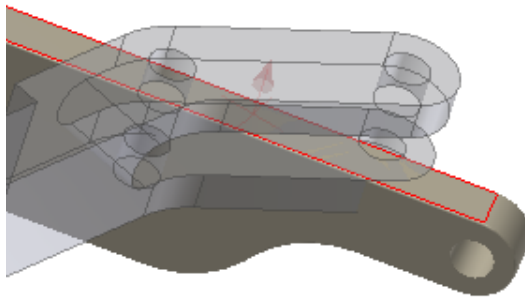


4. To begin to assemble the placed part without ending the current tool or starting a new tool, click the inside face of the claw as shown. Tip: Zoom in on the claw part and the end of the adjacent arm.

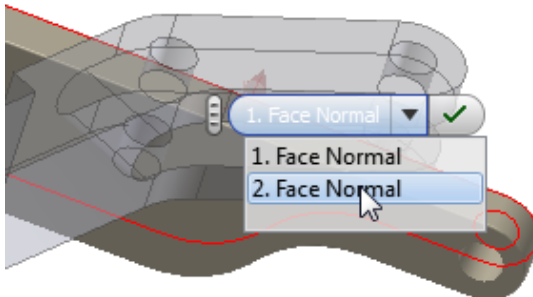


Note: You use assembly constraints to position and orient components in the assembly. An unconstrained component has three translational and three rotational degrees of freedom. Assembly constraints remove degrees of freedom and enable the component to be assembled according to design criteria.

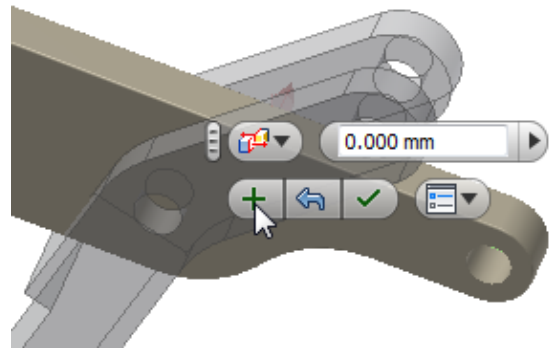
5. Position the cursor over the top face of the Spyder Arm as shown.



6. To select the opposite face of the SpyderArm, after the Select Other list displays, select 2. Face Normal from the list.

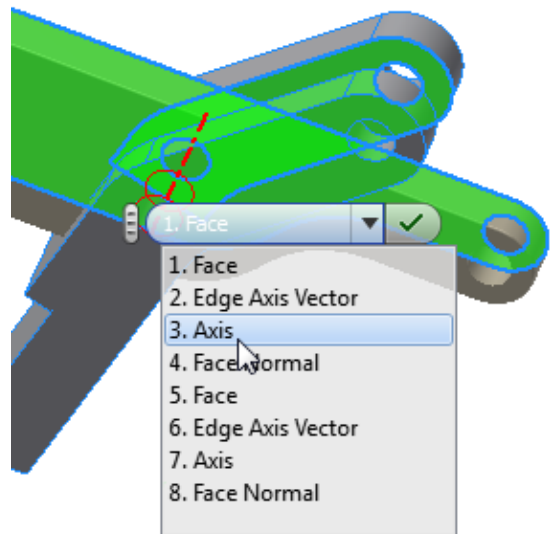


7. Click Apply. If the Apply button is not displayed, move the cursor close to the mini-toolbar.



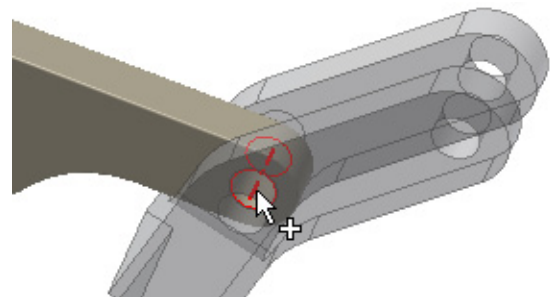
8. To select the axis of the hole in the claw:

- Position the cursor over the cylindrical wall of the hole in the claw.
- In the Select Other list, select the axis as shown.



9. To specify what the axis on the claw constrains to:

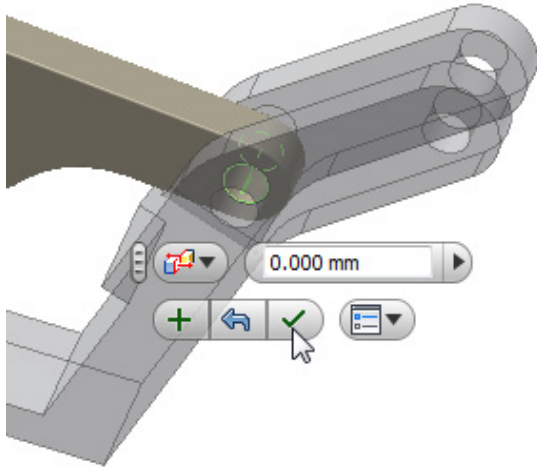
- Position the cursor over the hole in the arm.
- Click the hole when the axis preview displays.



10. Complete the process of adding and constraining the claw part by doing the following:

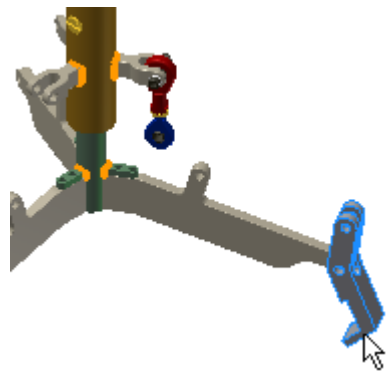
- Click OK.

- Right-click. Click Done.



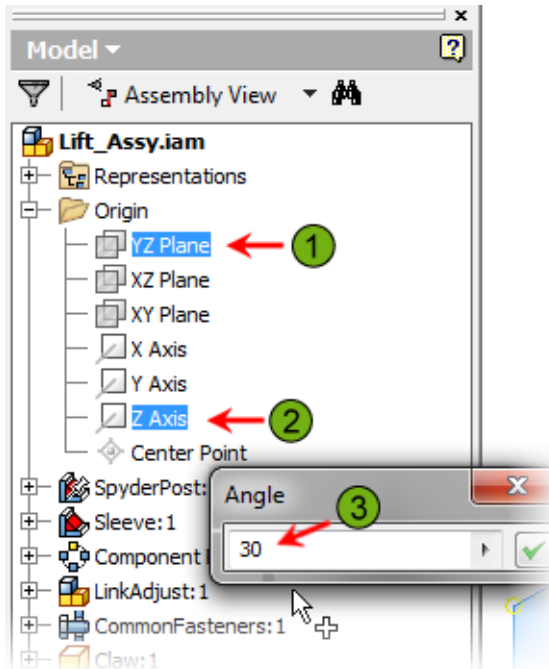
11. In Autodesk Inventor, you can drag components while respecting all assembly constraints. This is a good technique to determine the remaining degrees of freedom on the component. Click and drag the claw part. The only degree of freedom is rotation about the two axes you just constrained.

The engineer's note you reviewed earlier has information on the initial angle of the arm relative to a vertical plane. There is no readily available vertical plane in the assembly that is aligned with the faces on the claw part. Work features can also be created at the assembly level. You create them in a similar fashion as part level work features, by selecting the appropriate geometry.

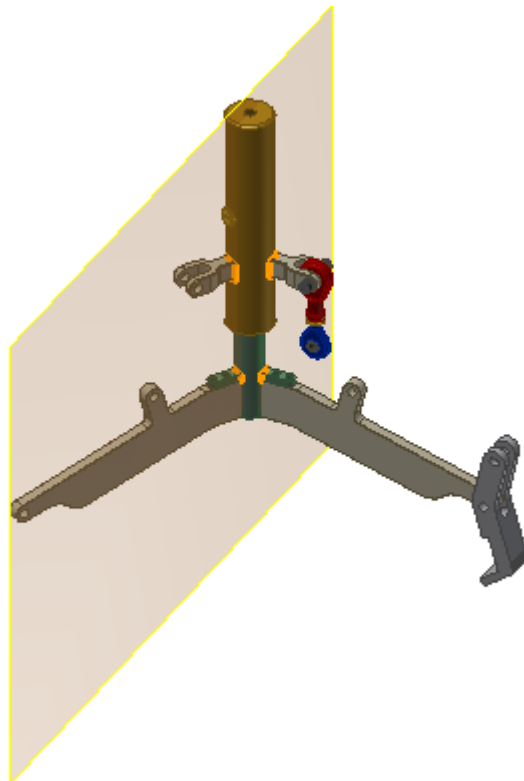


12. To add an assembly level work feature:
- On the Model tab, Work Features panel, click Plane.
  - In the browser, expand the Origin folder of the top level assembly.
  - Click YZ Plane.
  - Click Z Axis.
  - In the Angle edit box, enter 30.

- Click OK.

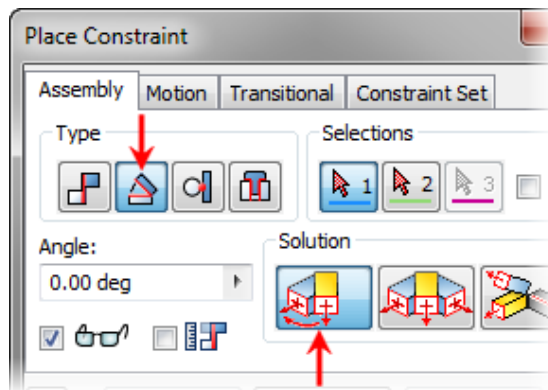


13. Review the results. The previous selections resulted in a work plane at an angle to the selected plane, through the selected axis. The work plane passes through the Z-axis and is at an angle of 30 degrees to the origin YZ plane. More importantly, it is a plane we can use to place the angle constraint described in the note.



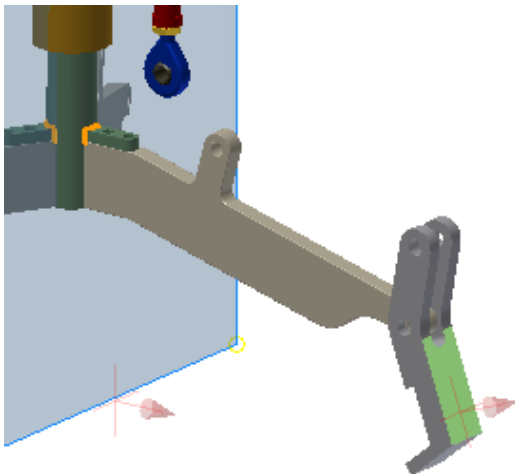
14. To begin to constrain the angle of the claw relative to the work plane:

- In canvas, right-click. On the marking menu, click Constrain.
- In the Place Constraint dialog box, Type area, click Angle.
- In the Solution area, click Directed Angle.

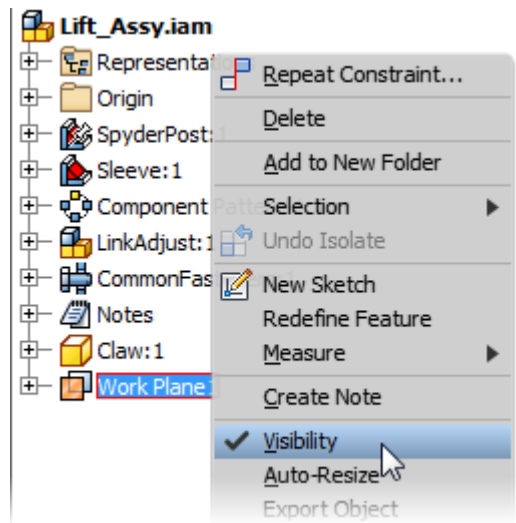


15. To place the constraint:

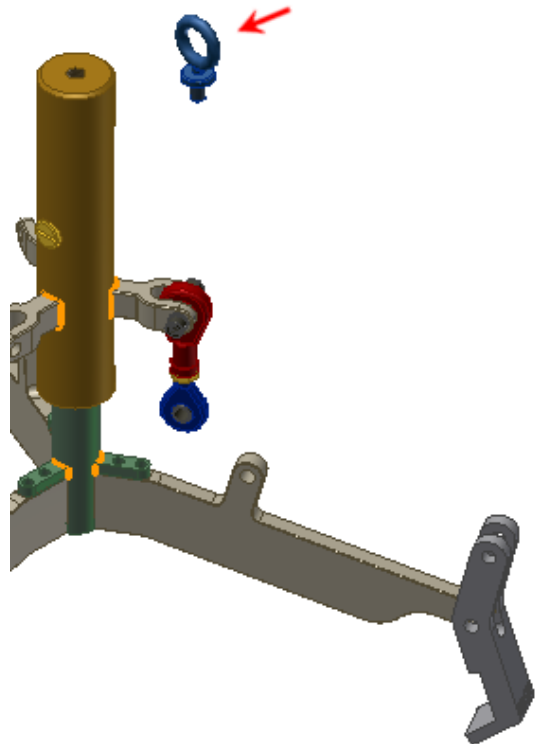
- Select the newly created work plane.
- Select the outside face of the claw.
- In the Place Constraint dialog box, Angle field, enter 20.
- With the preview displaying as shown, click
- OK to complete the constraint.



16. To turn off the visibility of the work plane, in the browser, right-click Work Plane1. Click Visibility (clear the check mark).



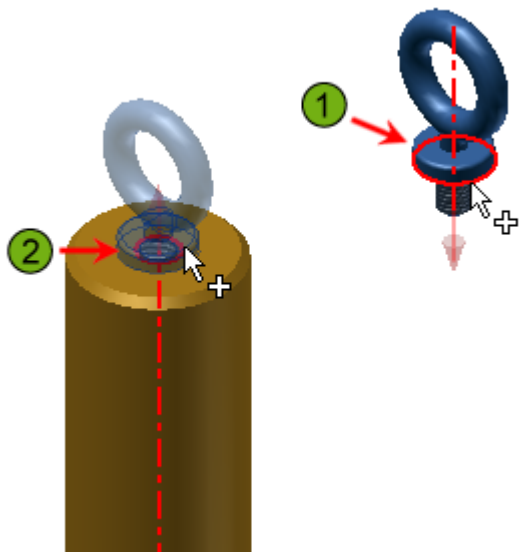
17. Start the Place Component tool again. Place a single occurrence of the LiftRing.ipt part near the top of the assembly as shown.



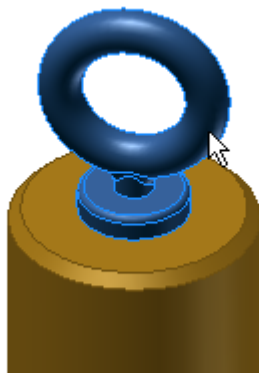
18. To constrain LiftRing:

- Position the cursor over the base of LiftRing as shown (1).
- Click when the circle and arrow are displayed.
- Position the cursor over the top of the cylinder (2).
- Click when the arrow displays.
- On the mini-toolbar, click OK.
- Right-click. Click Done.

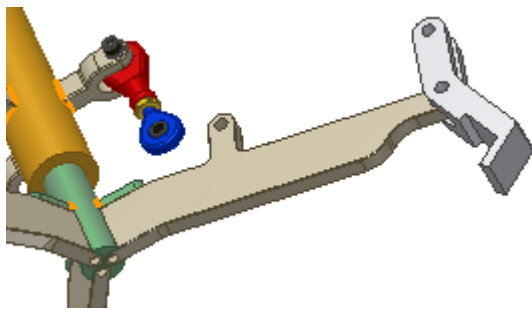




19. The lift ring part is constrained to the assembly. Click and drag the lift ring to the right or left. Notice that it can rotate but it cannot move left or right. It can only move up and down, which also moves the parts it is constrained to.



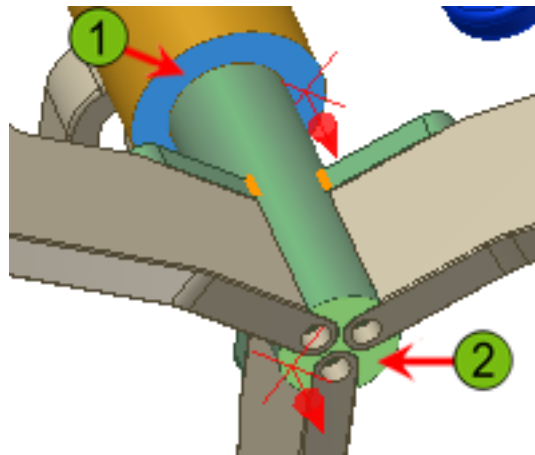
20. Rotate your view so the bottom of the fixture is visible.



21. To set the position of the sleeve on the post:

- On the Assemble tab, Position panel, click Assemble.
- Click the planar face on the underside of the sleeve (1).
- Click the face on the bottom of the post (2).
- On the mini-toolbar, for offset distance, enter 57.

- Click OK.

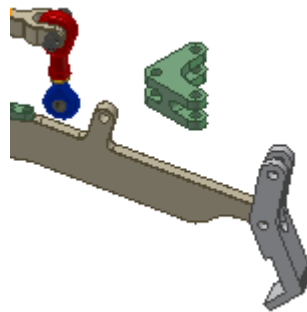


22. To return to the Home view, on the ViewCube, click Home.



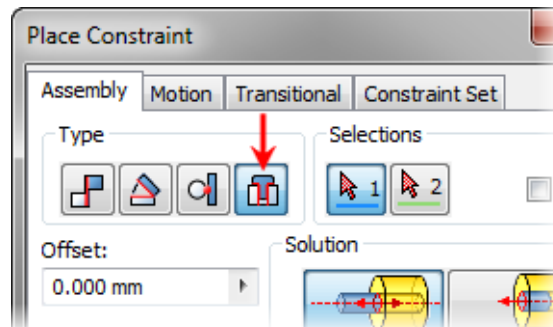
23. To add a pivot part to the assembly:

- Using Place Component, place the *Pivot.ipt* in the assembly near the raised support on the arm.
- Right-click. Click Done.



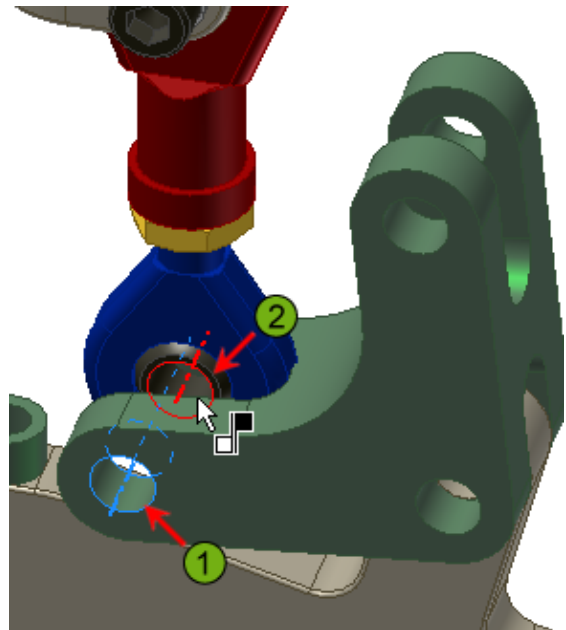
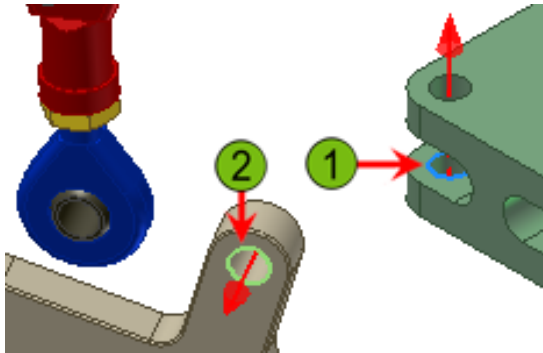
24. To begin to constrain the pivot:

- In canvas, right-click. On the marking menu, click Constrain.
- In the Place Constraint dialog box, under Type, click Insert.

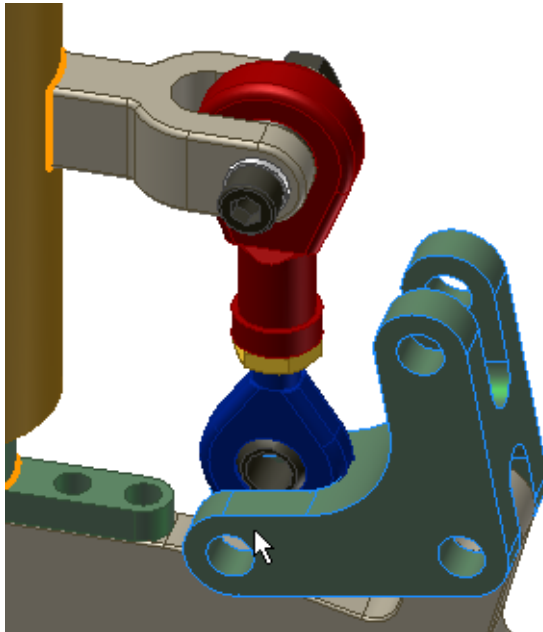


**25.** To assign an Insert constraint:

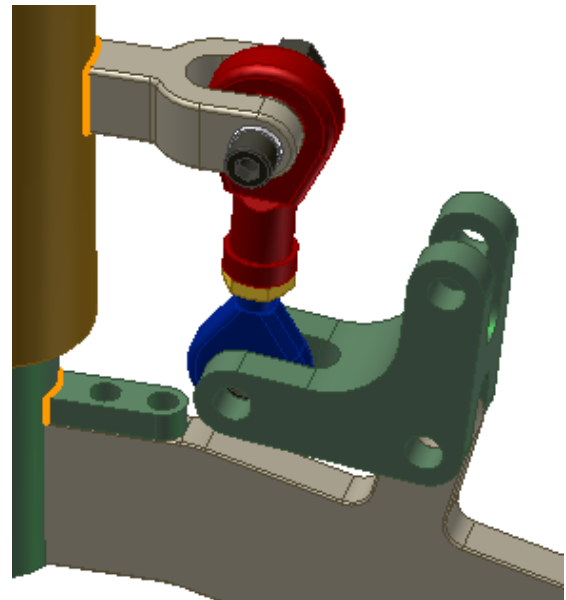
- In canvas, select the circular edge of the middle hole on the inside surface of the pivot (1).
- Select the edge of the hole on the Spyder Arm (2).
- In the Place Constraint dialog box, click OK.



**26.** Drag the pivot so it is more closely aligned to the rod end attached to the sleeve.



**28.** The components are constrained.



**27.** To add a mate constraint to the Pivot and rod end:

- Start the Constrain tool.
- Select the axis of the Pivot (1) as shown.
- Select the axis of the rod end (2).
- Click OK.

**29.** Save the file.

## Assemble Parts to Create a Subassembly

Next you assemble the link rod and two rod ends to create a subassembly. It is good practice to build subassemblies to match the subassemblies found in the physical model.

1. To start a new file:
  - On the Quick Access bar, click New.
  - On the Metric tab, double-click the *Standard (mm).iam* template.

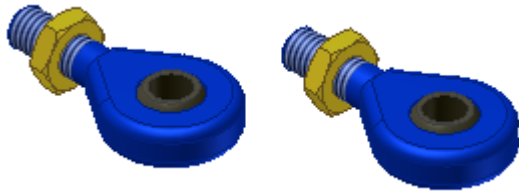


Standard (mm).iam

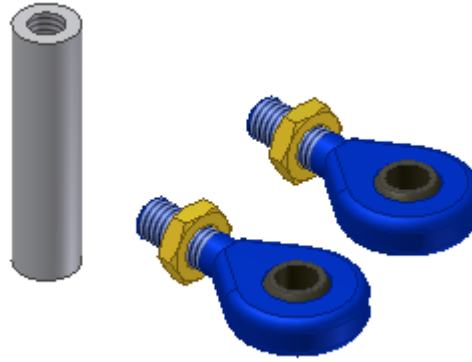
2. To save the file:
  - On the Quick Access toolbar, click Save.
  - For name, enter: Adjust\_Rod.
  - Click Save.



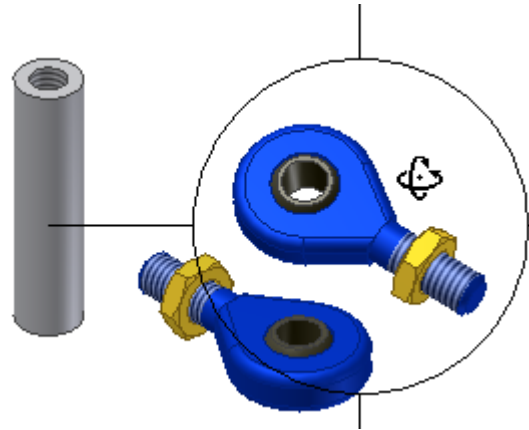
3. To place components in the assembly:
  - In canvas, right-click. Click Place Component.
  - In the Subassemblies folder, select *RodEnd.iam*.
  - Click Open. An occurrence of the rod end assembly is aligned to, and grounded at the coordinate system origin.
  - Click to the right of the grounded occurrence to place a second occurrence.
  - Right-click. Click Done.



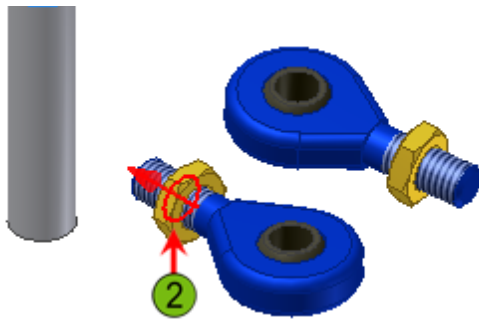
4. To place the link rod created at the beginning of the test drive:
  - Start the Place Component tool.
  - In the Place Component dialog box, shortcut list, click Workspace.
  - Select *LinkRod.ipt*.
  - Click Open.
  - Click in canvas to place an occurrence of the link rod.
  - Right-click. Click Done.



5. It can be helpful to preposition components prior to adding assembly constraints. To rotate a component in canvas:
  - Select the second rod end that you added to the assembly.
  - In canvas, right-click. Click Rotate Component.
  - Click and drag inside the circle to reorient the rod end roughly as shown.
  - Right-click. Click Done.

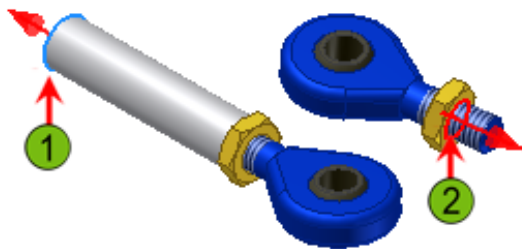


6. To constrain the link rod to the two rod ends:
  - In canvas, right-click. Click Constraint.
  - In the Place Constraint dialog box, under Type, click Insert.
  - Select a circular edge on the link rod.
  - Select the circular edge of a rod end jam nut (2) as shown.
  - Click Apply.



7. To add another insert constraint:

- Select the circular edge on the other end of the link rod (1).
- Select the circular edge of the other jam nut (2) as shown in the following illustration.
- Click OK.



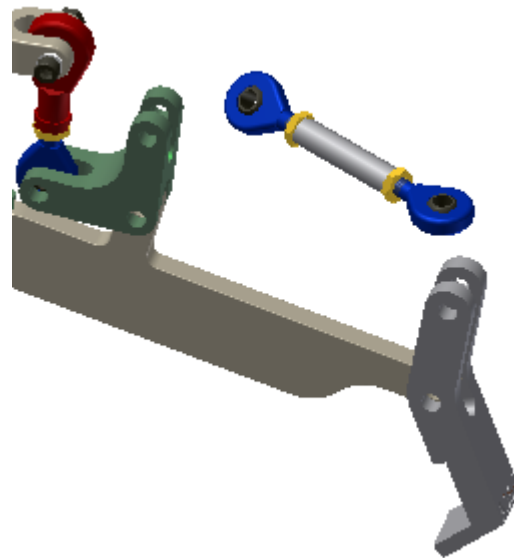
8. To complete the assembly:

- On the Quick Access toolbar, click Save.
- Close the Adjust\_Rod assembly. If prompted, save changes.
- Close the LinkRod part. Save changes.
- Close the Claw part. Save changes.

9. Verify that *Lift\_Assy.iam* is the only open file.

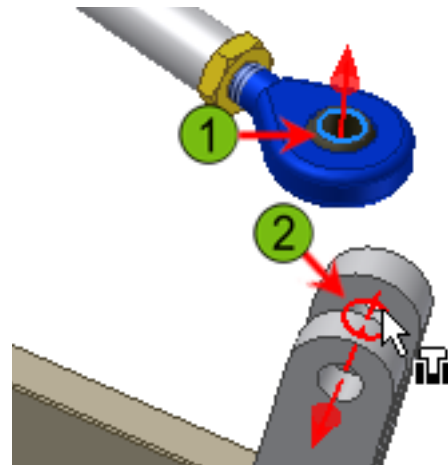
10. To place the link rod assembly:

- Start the Place Component tool.
- Double-click *Adjust\_Rod.iam*.
- Click in canvas to place an occurrence of the AdjustRod assembly.
- Right-click. Click Done.

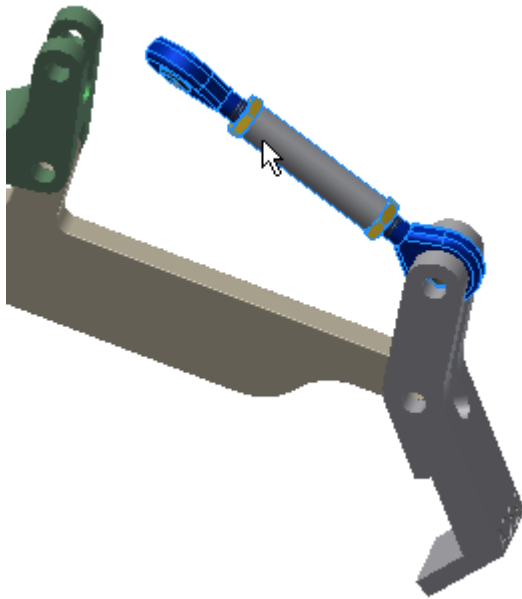


11. To constrain the link rod assembly to the claw:

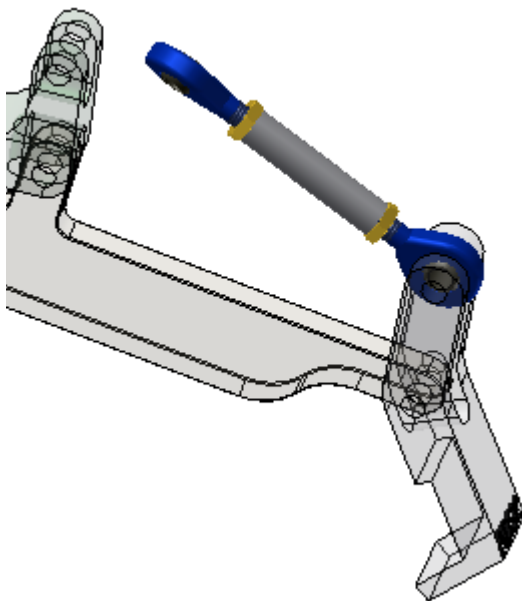
- In canvas, right-click. Click Constraint.
- Under Type, click Insert.
- Select the circular edge of a hole on the rod end (1).
- Select the edge of a hole on the slot in the claw (2).
- Click OK.



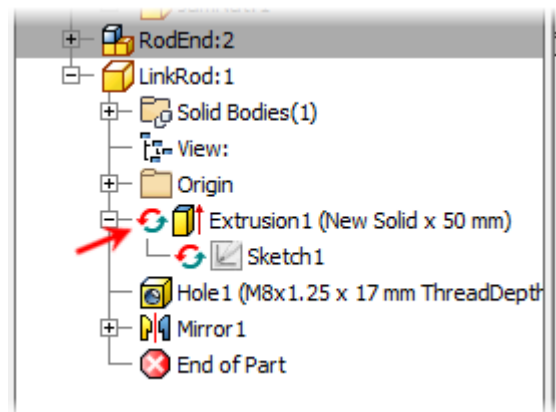
12. Drag the free end of the rod assembly to match the following illustration.



13. In the browser, double-click Adjust\_Rod:1. All components other than the adjustor subassembly are dimmed in canvas and grayed out in the browser. You are now editing the subassembly in-place in the top-level assembly.



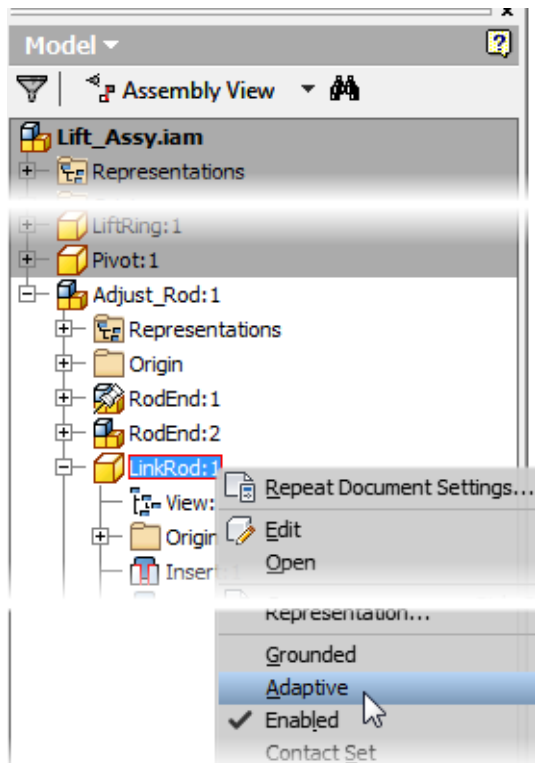
14. To activate the adaptivity of the link rod to enable it to change length and connect to the pivot:
- In the browser, double-click LinkRod:1. The part is now available to be edited in-place.
  - Ensure that Extrusion1 has the adaptive symbol next to it.
  - If Extrusion1 is not marked as adaptive, right-click it and click Adaptive.



15. To return to the Adjust\_Rod subassembly, on the Model tab, Return panel, click Return.

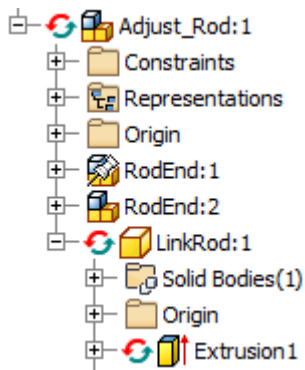
16. In the browser, right-click LinkRod:1 and click Adaptive.

The part is now adaptive within the subassembly. The length of the extrusion can change to match assembly constraints.



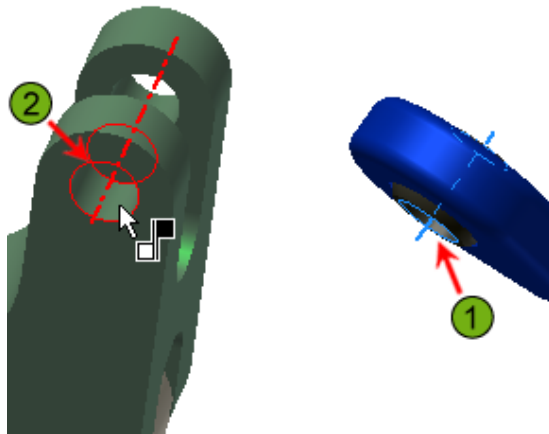
17. To return to the Lift\_Assy assembly, on the Return panel, click Return.

18. In the browser, right-click Adjust\_Rod:1. Click Adaptive. By setting the subassembly adaptive, the adaptive features in the subassembly components can respond to assembly constraints placed at the top-level assembly.



19. To adapt the length of the rod:

- Start the Constraint tool.
- Select the axis (not the circular edge center point) through the free rod end (1).
- Select the axis through the open hole in the pivot part (2).
- Notice that the preview of the LinkRod shows that its length has adapted to fulfill the axis/axis constraint.
- Click OK.

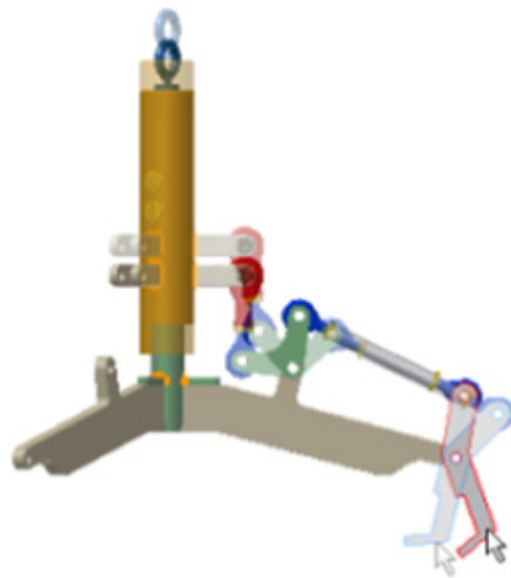


The rod ends in the subassembly retain their relationships to the link rod part.

20. To perform a constrained drag:

- In the browser under Sleeve:1, right-click Flush2 (57.000 mm). Click Suppress.
- In the browser under Claw:1, right-click Angle3 (20.00 deg). Click Suppress.
- In canvas, click and drag the sleeve, pivot, or claw.

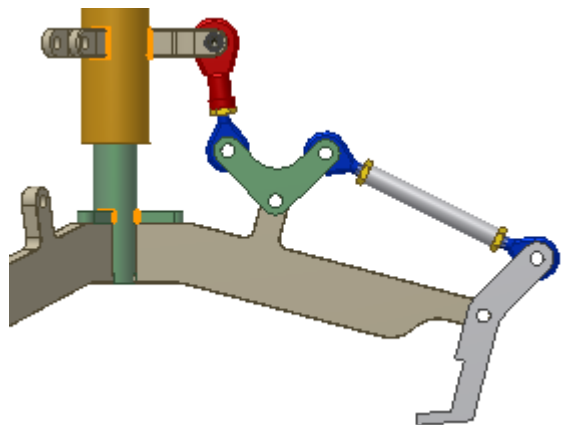
The mechanism operates while satisfying all placed assembly constraints.



Note: If the assembly flips to an unusable position, click Undo on the Quick Access Bar.

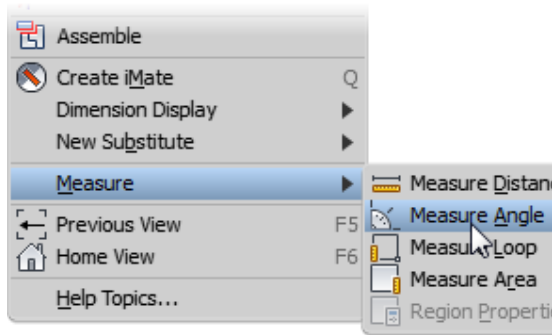
21. Now that the mechanism is complete, you check the closed state of the assembly against the desired angle (~16.4):

- In the browser, under Sleeve:1, right-click Flush2 (57.000 mm). Click Suppress. This removes the suppressed state of the constraint.
- Click in the background of the browser, and then click the same constraint.
- In the value field at the bottom of the browser, enter **81**. Make sure you press ENTER after entering the value. The position of the lift is now in the closed position.



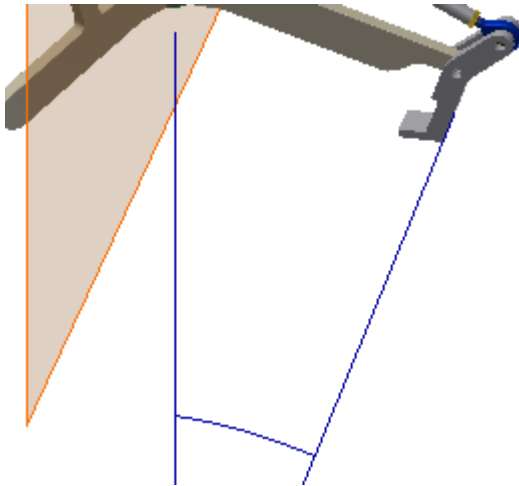
22. In the browser, right-click Work Plane1. Click Visibility. The work plane is now visible.

23. To access the measure angle tool, In canvas, right-click. On the shortcut menu, click Measure > Measure Angle.



24. To measure the current angle between the claw and the work plane:

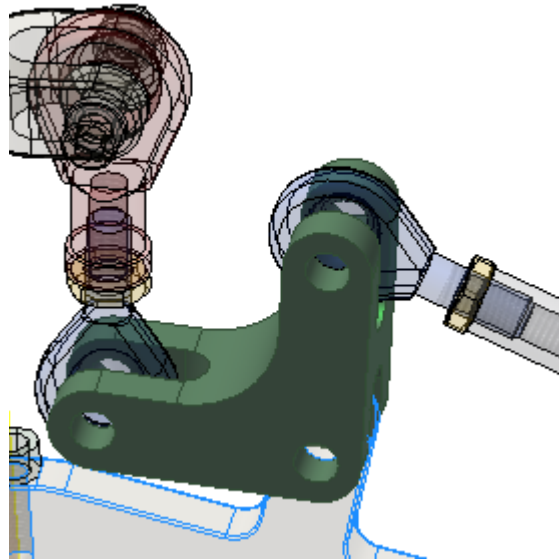
- In canvas, select the work plane.
- Select the back face of the claw.
- In the Measure Angle dialog box, review the value. The angle reported is not within the acceptable range from the engineer's note reviewed earlier in the test drive.



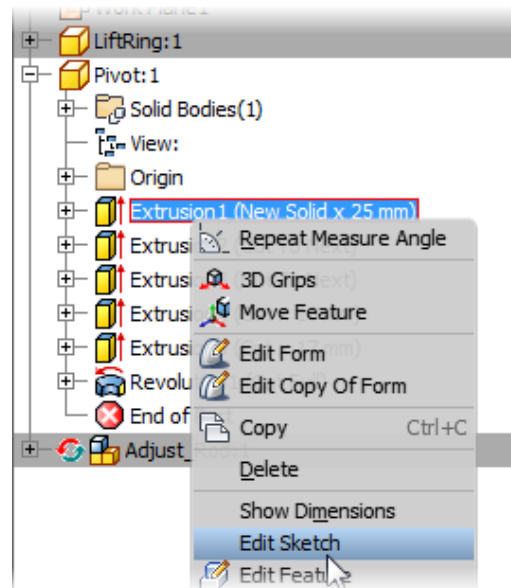
Note: Because we did not set an exact position for the pivot hole in the claw part, your angle value will be different.

25. Right-click in canvas. Click Done. After careful consideration you determine that if the pivot arms are shortened, the claw rotates further for the same movement of the sleeve. However, the 20 initial angle must be retained, so an iterative solution seems in order.
26. Next, you reset the constraints to their initial conditions so that all modifications to the pivot are made at this lift position.
- In the browser, under Sleeve:1, click Flush2 (81.000 mm).
  - Enter **57** as the new offset.
  - Under Claw, unsuppress the Angle (20 deg) constraint.

27. In canvas, double-click the Pivot part. The pivot is isolated in the assembly, and the part environment is activated.



28. To access the initial sketch, in the browser under Pivot:1, right-click Extrusion1. Click Edit Sketch.



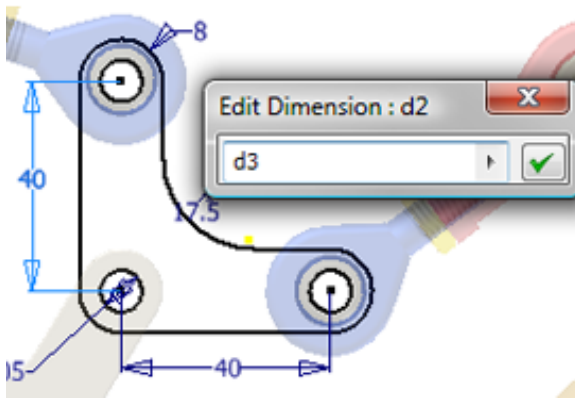
29. To modify the part and create an equation between dimensions:

- Double-click the 40 mm vertical dimension.
- Ensure that all the text in the dimension edit box is selected.
- Select the horizontal 40 mm dimension.
- In the Edit Dimension dialog box, click the green check mark.

The 40 mm dimension is replaced with the name of the selected dimension (d3). The vertical dimension (d2) will now update to



match any change in the horizontal dimension ( $d_3=d_2$ ).



30. Next, you edit the horizontal arm length dimension to change the size of the pivot. To edit the arm length:

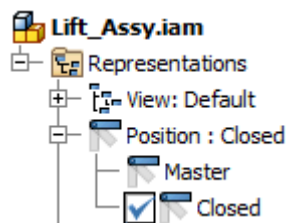
- Double-click the 40 mm horizontal dimension.
- Enter **39** mm. Click the green checkmark.
- Click in canvas to clear the selection.
- Right-click. Click Finish 2D Sketch.
- Right-click. Click Finish Edit.

Both 40 mm dimensions update to 39 mm.

The sketch and part update, and the assembly environment is reactivated. The length of the LinkRod part adapts to meet the new length of the pivot arm.

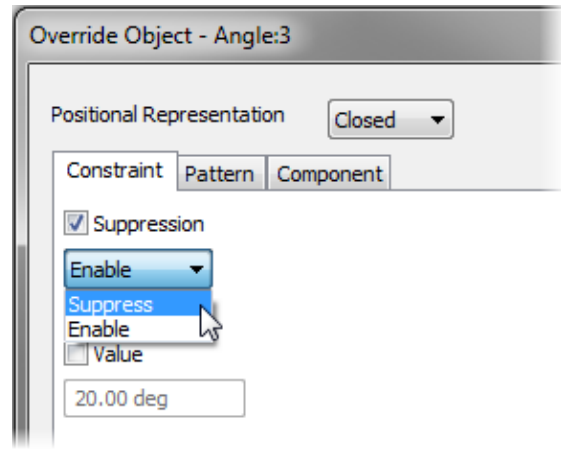
31. To check the angle in the lift closed position, you could suppress the claw angle constraint, and then edit the sleeve offset to the closed position. Since you may need to repeat this process to arrive at a solution, you set up a positional representation of the lift in the closed position. In the browser:

- Under Lift\_Assy.iam, expand Representations.
- Right-click Position. Click New.
- Expand the Position: Position1 node.
- Click the Position1 node, and then slowly click it again.
- To rename the node, enter: Closed.



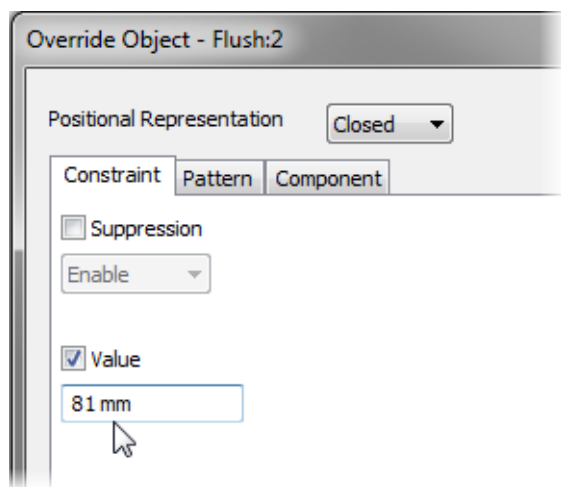
32. Begin to apply a constraint override for this positional representation by doing the following:

- In the browser under Claw:1, right-click Angle:3 (20.00 deg). Click Override.
- In the Override Object dialog box, Constraint tab, select the Suppression check box.
- From the list, select Suppress.
- Click OK.



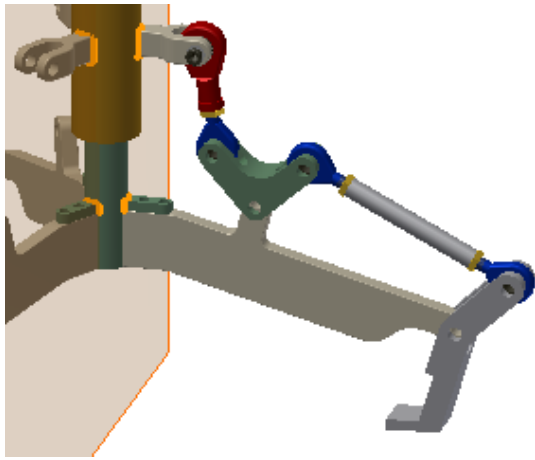
33. Place an additional constraint override for the positional representation. To do this, do the following:

- In the browser under Sleeve:1, right-click Flush:2 (57.00 mm). Click Override.
- In the Override Object dialog box, Constraint tab, select the Value check box.
- In the value field, enter 81 mm.
- Click OK.

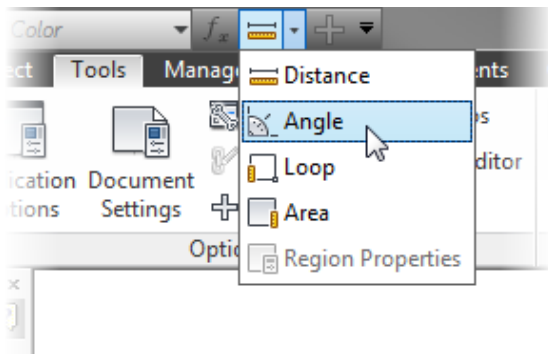


34. The assembly updates to match the overridden constraints.

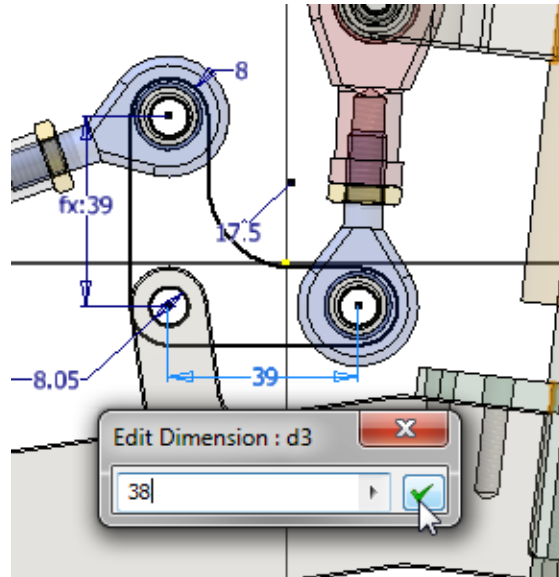




35. On the Quick Access toolbar, Distance flyout menu, click Angle.



36. To check the results of the pivot edit:
- In canvas, select the back face on the claw.
  - In canvas or in the browser, select Work Plane 1.
  - In the Measure Angle dialog box, review the value.
  - Right-click. Click Done.
37. In the browser, under Representations, double-click Master.  
The assembly returns to the open position and the claw angle is again fixed at 20.  
Note: Modeling operations can only be performed when the Master positional representation is active.
38. To edit the pivot part:
- Double-click the Pivot part to edit it inplace.
  - Edit the sketch under Extrusion1.
  - Edit the 39 mm horizontal dimension and enter 38 mm as the new value.
  - Right-click. Click Finish 2D Sketch.
  - Right-click. Click Finish Edit.



39. To check the results of the pivot edit:
- In the browser, under Representations, double-click Closed.
  - Click in canvas to clear any selections.
  - Start the Measure Angle tool.
  - Select the outside face of the Claw.
  - Select Work Plane1.
  - In the Measure Angle dialog box, review the value.
  - Right-click. Click Done.
40. In the browser, under Representations, double-click Master.
41. Save the file.

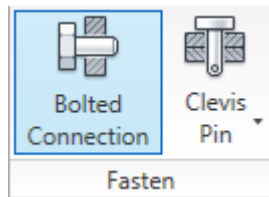
## Place Bolted Connections

Next, you will place bolted connections into the assembly.

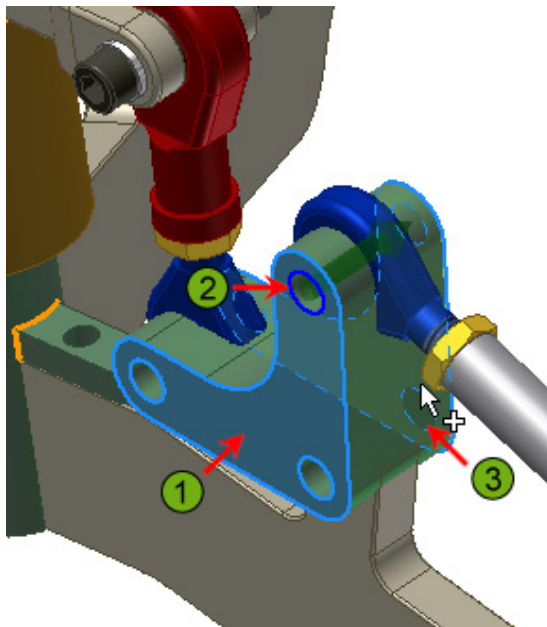
- Before continuing, clean up some details after the link rod modeling session, in the browser:
  - Under Claw:1, suppress Angle:3 (20.00 deg).
  - Right-click Adjust\_Rod:1. Click Adaptive. The adaptive setting for the subassembly is now off.
  - Right-click Work Plane1. Click Visibility. The work plane is now hidden.

Note: You just turned off adaptivity at the assembly level after completing the adaptive modeling changes. You can easily toggle adaptivity on if further design changes are required.

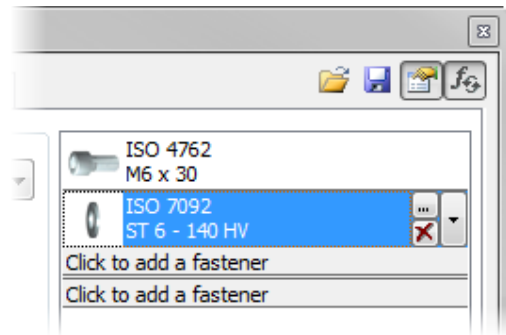
- On the Design tab, Fasten panel, click Bolted Connection.



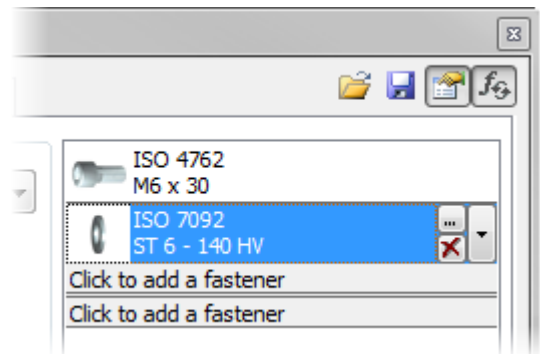
3. In the Bolted Connection Component Generator dialog box:
  - In the Placement list, select Concentric.
  - For Start Plane, select the flat face of the Pivot (1).
  - For the circular reference, select a circular edge from a hole in the Pivot (2).
  - Select the back face of the Pivot for the termination plane (3). Note: Autodesk Inventor understands that the termination plane will be parallel to the start plane. Therefore, the termination plane can be selected without rotating the model or using the Select Other tool.



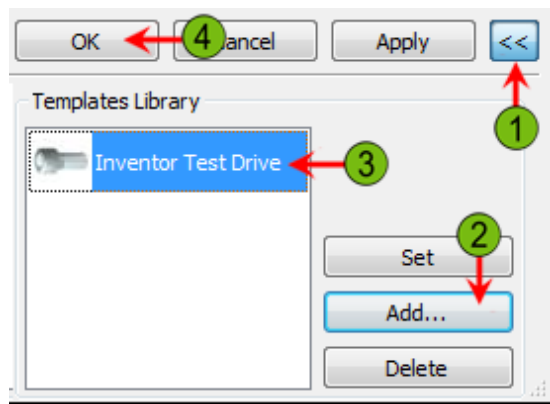
4. To add fasteners to be placed into the assembly, in the Bolted Connection Component Generator dialog box:
  - Click the bar titled Click to Add a Fastener.
  - In the Category list, select Socket Head Bolts.
  - Select ISO 4762.
  - Click the bar titled Click to Add a Fastener just below the ISO fastener in the list.
  - Select ISO 7092. The results displays as shown.



5. To add fasteners to the termination side of the fastener assembly:
  - Below the two line divider, click the bar titled Click to Add a Fastener.
  - Select ISO 7092.
  - Below the second washer, click the bar titled Click to Add a Fastener.
  - Select ISO 4032. The results display as shown.

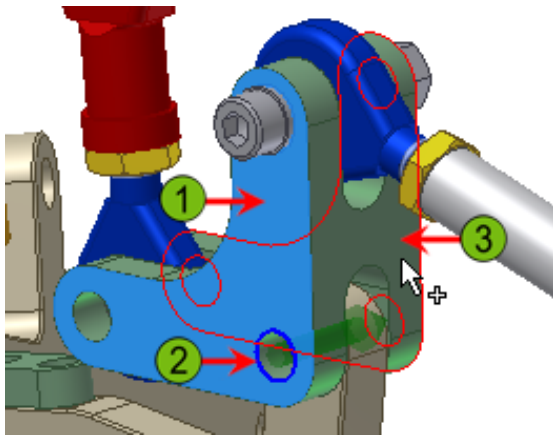


6. This bolted connection will be reused, so a template will be created that can be leveraged. To create a template:
  - Expand the Bolted Connection Component Generator dialog box (1).
  - Click Add (2).
  - Enter Inventor Test Drive in the Template Description dialog box to add it to the template library (3).
  - Click OK in the Template Description dialog box.
  - Click OK in the Bolted Connection Component Generator dialog box (4).
  - In the File Naming dialog box, click OK.

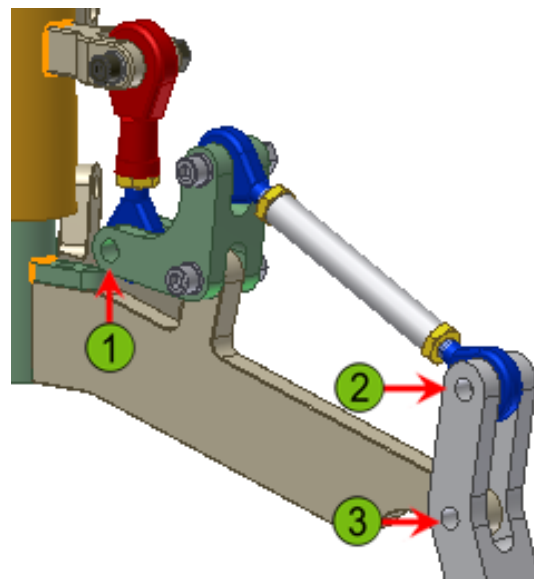


7. To add another bolted connection:

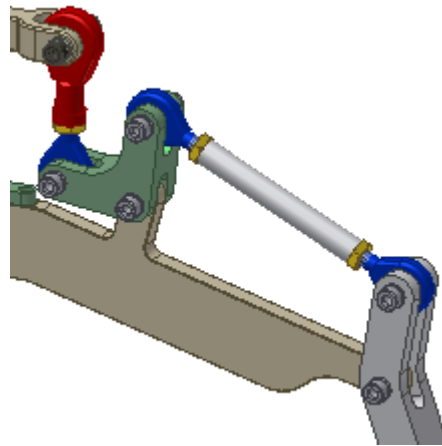
- On the Design tab, Fasten panel, click Bolted Connection.
- Under Placement, select Concentric from the list.
- For Start Plane, select the flat face of the Pivot for the start plane (1).
- Select a circular edge from a hole in the Pivot for the circular reference (2).
- Select the back face of the Pivot for the termination plane (3).
- In the Templates Library list, select Inventor Test Drive.
- Click Set.
- Click OK.
- In the File Naming dialog box, click OK.



8. Repeat the previous process to place fasteners in the three remaining locations.



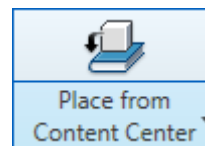
9. Save the file.



## Use a Library of Standard Components

Autodesk Inventor includes a comprehensive library of standard components that can be used. The subassembly CommonFasteners.iam that was created with the Bolted Connection tool consists of a number of components from the content library.

1. On the Assemble tab, Component panel, click Place from Content Center.

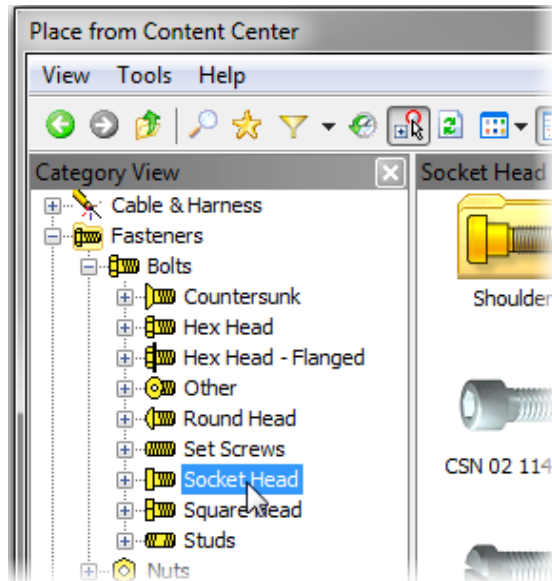


Note: You may need to select the down arrow next to the Place tool to access Place from Content Center.

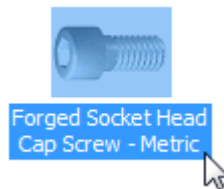
2. To select a fastener, in the Place from Content Center dialog box, under Category View:

- Expand Fasteners.
- Expand Bolts.

- Select Socket Head.



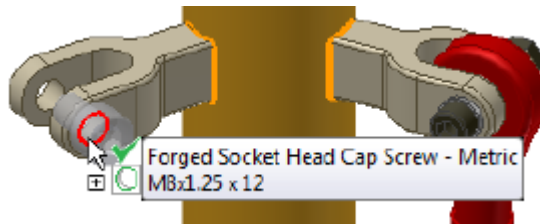
3. Double-click Forged Socket Head Cap Screw - Metric in the preview pane of the Content Center.



4. The Content Center is made up of parts that will automatically size when positioned on holes in your design. When a specific size has not been calculated, the fastener is attached to your cursor and displays an icon with a red question mark.



5. Position the cursor (do not click) over a hole on one of the sleeve boss components.



A green checkmark is added to the icon next to the component and the tooltip displays the fastener size based on the hole size.

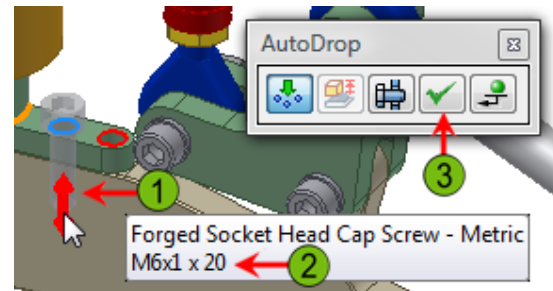
6. To place the component in the assembly:

- Position the cursor over a hole in the bracket top component.
- Click once when the green check mark is displayed and the tooltip displays the size of the component.

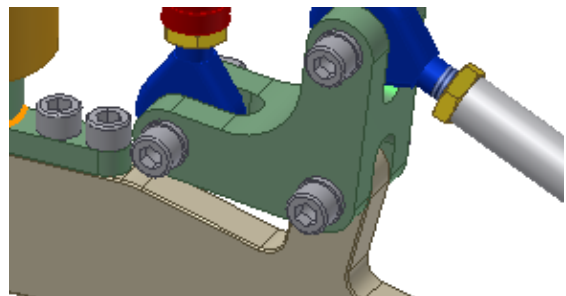


7. In canvas:

- Click and drag the red arrow handle (1) until the tooltip displays a component length of 20 (2).
- In the AutoDrop dialog box, click Apply (3).

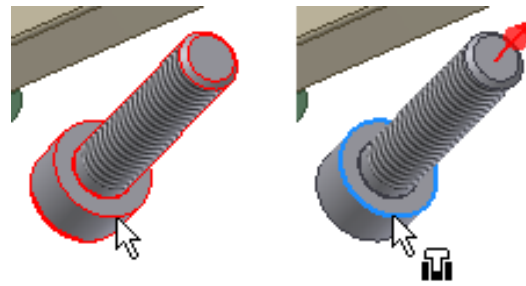
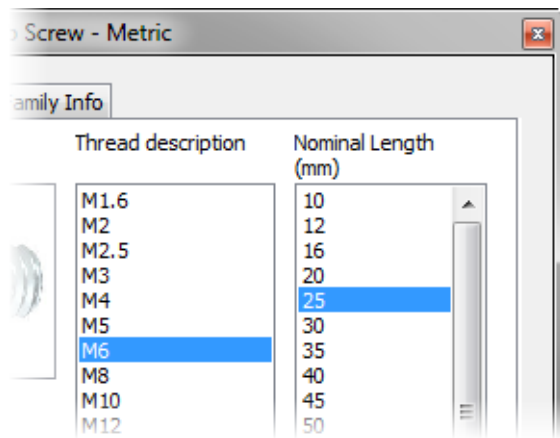


8. The component is placed in the assembly and because it was a patterned hole, a fastener is placed in both occurrences of the patterned feature.



9. To begin to place a third occurrence of the fastener:

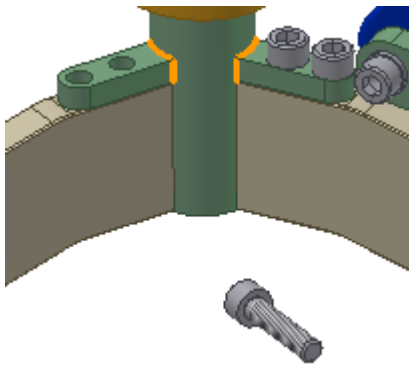
- In canvas, right-click and select Change Size in an open area.
- In the Forged Socket Head Cap Screw - Metric dialog box, Thread Description list, select M6.
- In the Nominal Length list, select 25.
- Click OK.



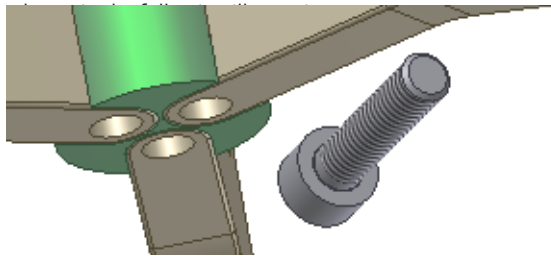
Note: The initial selection determines the type of constraint that will be placed. The default assembly constraint while dragging a circular edge is Insert. If your initial selection is the cylindrical surface of the fastener, the default constraint type is Tangent. Press the I key to change to an Insert constraint.

10. Place an occurrence of the bolt. Do the following:

- Click near the bottom of the lift assembly.
- In canvas, right-click. Click Done.



11. You can place assembly constraints without using the Place Constraint dialog box. You use ALT+Drag to place assembly constraints using part geometry and iMate symbols (iMates are not covered in this test drive). Orient your view so you can see the bottom of the sleeve as



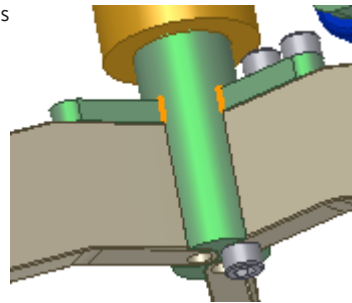
12. To constrain a component using the ALT+Drag method:

- Hold down the ALT key and click and drag on the circular edge under the head of the socket head cap screw.

13. To apply the constraint:

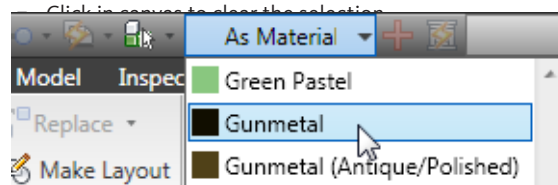
- Drag over the hole in the spyder arm and post.
- Release the mouse button when the hole edge is highlighted.

The third fastener is constrained to the underside of the pos



14. To change the color of the fastener in the assembly:

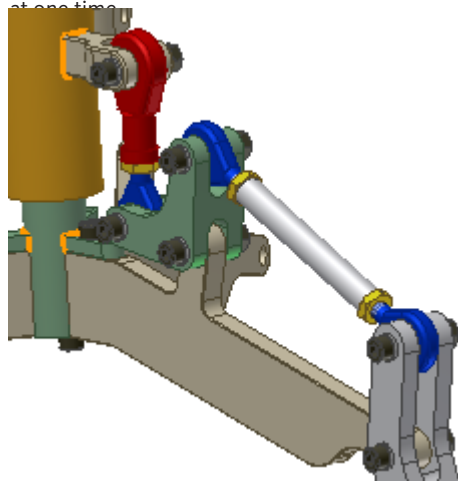
- In the browser, select the first Forged Socket Head Cap Screw.
- Hold down the SHIFT key and select the last Forged Socket Head Cap Screw. All three occurrences of the fastener are selected.
- On the Quick Access toolbar, Color Override list, select Gunmetal.



15. Optionally, use the previous process to change the color of the components that were placed using the Bolted Connection tool.

Tip: You can use the CTRL key to select multiple components in the browser or graphics window and then select a material using the Color

Override list to change the color of multiple components at one time.



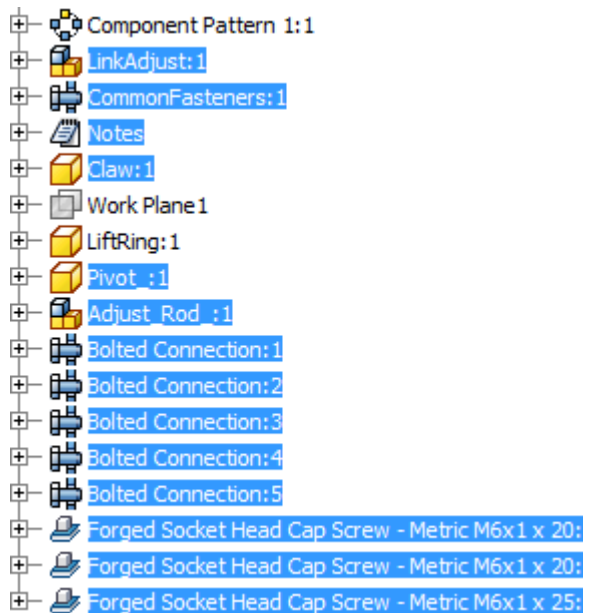
16. Save the file.

## Create an Assembly Pattern and Simulate the Mechanism

To complete the assembly, you create a pattern of assembly components based on the pattern of arms on the spyder.

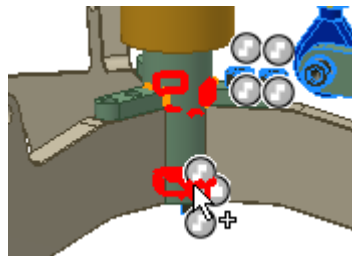
1. To select components to pattern, in the browser:

- Click LinkAdjust:1.
- Hold down the SHIFT key and then click the last component in the browser. All components between LinkAdjust and the last component in the browser are selected.
- Hold down the CTRL key and select LiftRing:1 to remove it from the selection.
- Hold down the CTRL key and select Work Plane1 to remove it from the selection.

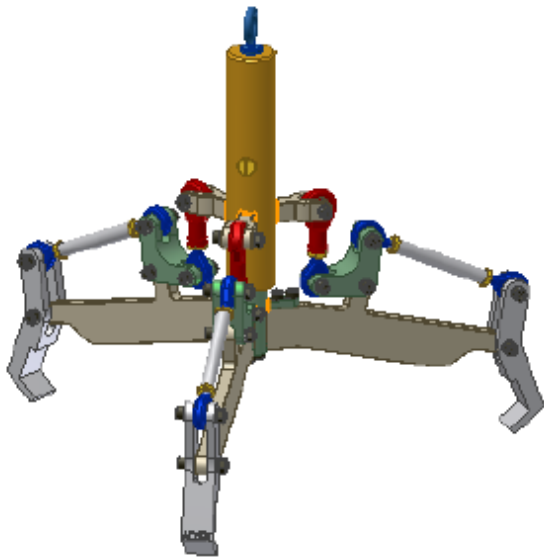


2. To complete the pattern:

- On the Assemble tab, Component panel, click Pattern.
- In the Pattern Component dialog box, Feature Pattern Select area, click the Associated Feature Pattern select button.
- Position the cursor over the bottom of the center post. Click when the pattern of slots is highlighted.

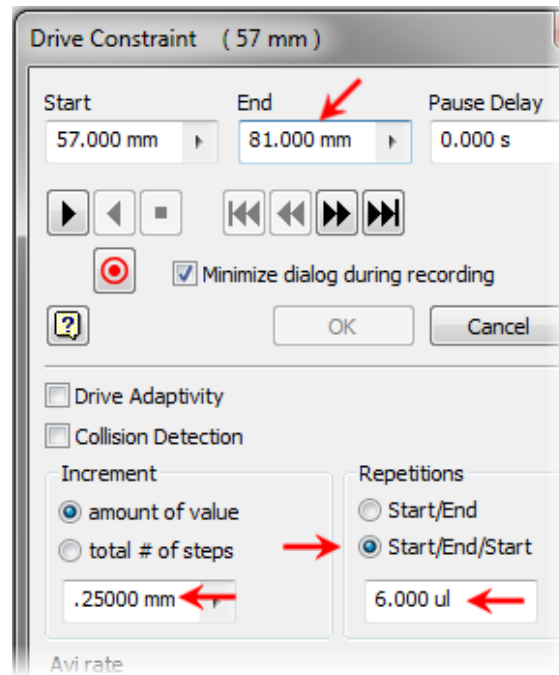


3. Click OK to create the component pattern.

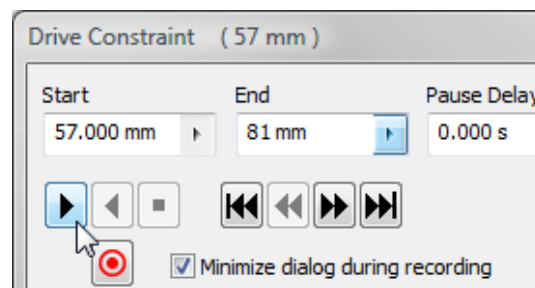


4. Finally, you drive one of the assembly constraints to view a simulation of the lifting mechanism.  
To access the Drive Constraint dialog box:
  - In the browser, expand Sleeve:1.
  - Right-click Flush2 (57.000 mm). Click Drive Constraint.

5. To set the Drive Constraint parameters:
  - For End, enter 81 mm.
  - Click the More (>>) button to expand the dialog box.
  - For the Increment value, enter .25.
  - In the Repetitions area, click Start/End/ Start.
  - For the Repetition value, enter 6.



6. To complete the drive constraint:
  - Click the Less (<<) button to reduce the dialog box size.
  - Click the Forward button. The mechanism moves through its range three times.



7. In the Drive Constraint dialog box, click Cancel.
8. Save the file.



## Exercise: Advanced Visualization

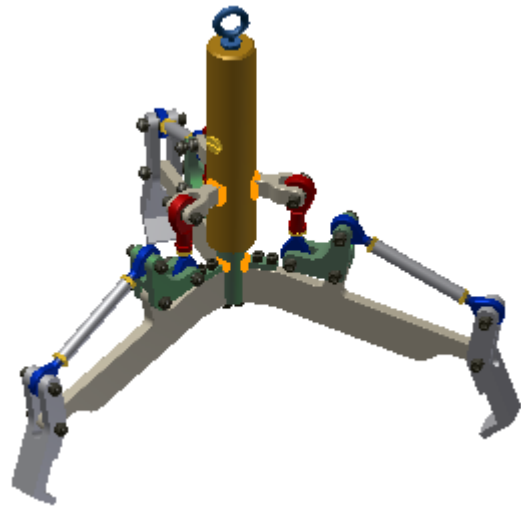
Now that you have finished the assembly, you can create a near photo-realistic rendering. Autodesk Inventor provides state-of-the-art tools that design engineers need to quickly create renderings.



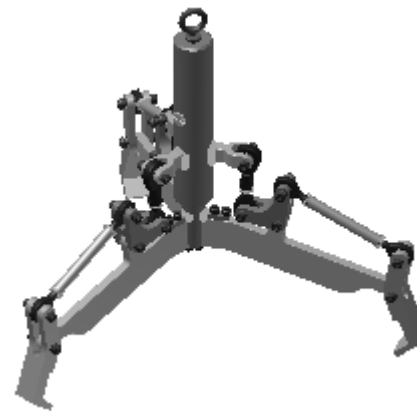
The completed exercise

### Create a Photo-Realistic Rendering

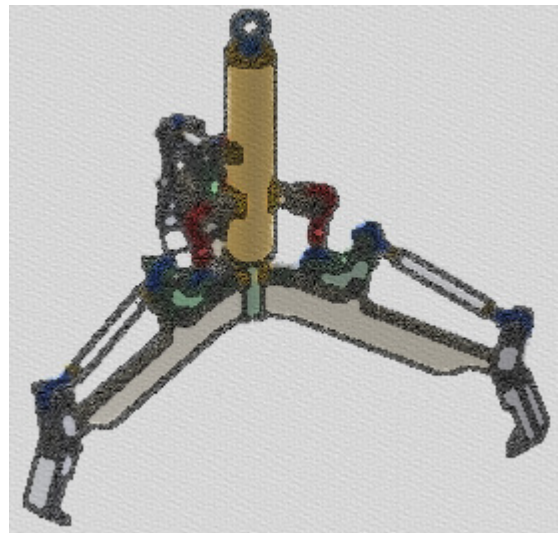
1. Continue working with *Lift\_Assy.iam* that you saved in the Assembly Modeling exercise. If you did not complete that exercise or save that file, open *.\Completed Files\Lift\_Assy.iam*.



2. On the View tab, Appearance panel, Visual Style list, select Monochrome.

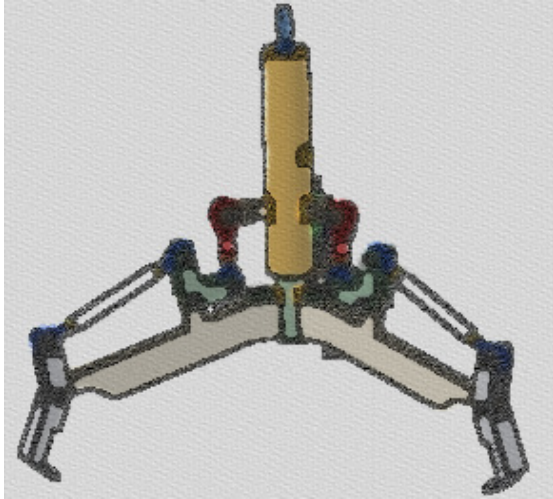


3. In the Visual Style list, select Watercolor.

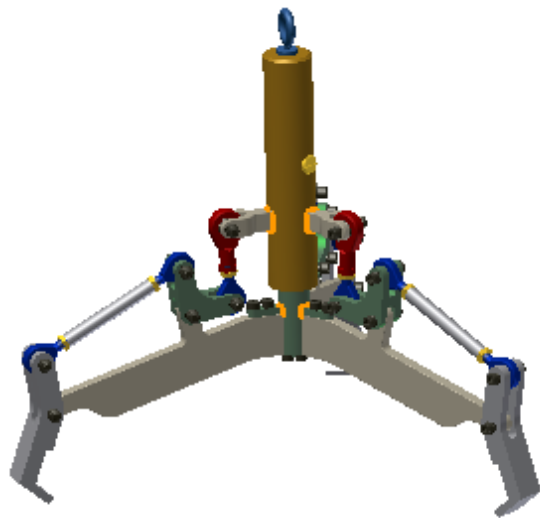




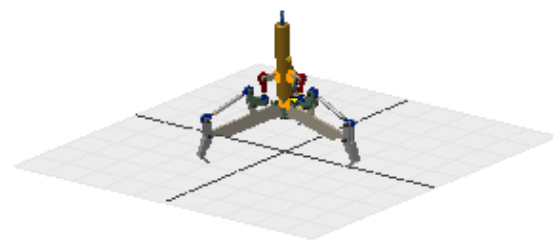
4. Orbit the model to view it from a direction approximately as shown. Notice how the visual style is dynamic as you orbit the model.



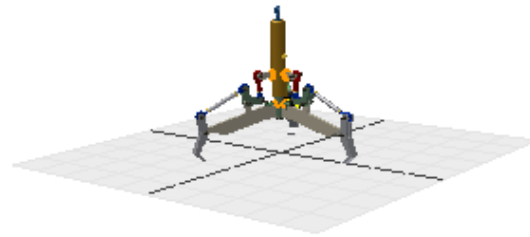
5. In the Visual Style list, select Shaded.



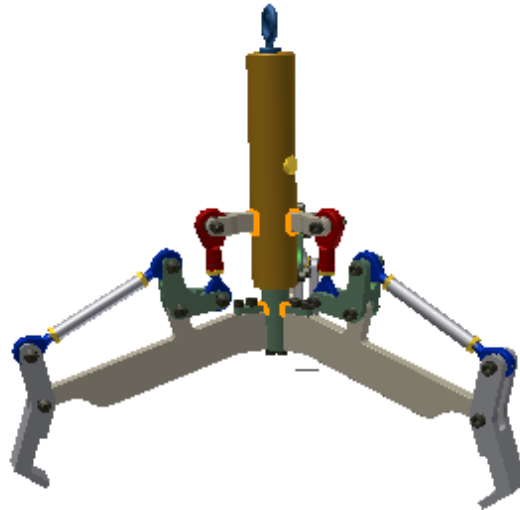
6. On the Appearance panel, click Ground Plane. Notice how the opposite edges of the ground plane display parallel to each other.



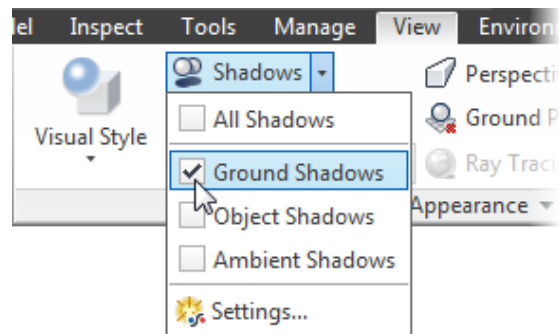
7. On the Appearance pane, Orthographic list, select Perspective. Notice how the opposite edges of the ground plane appear to converge.



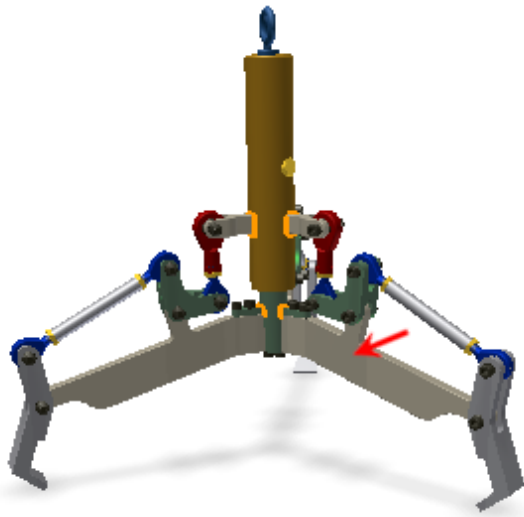
8. On the Appearance panel, click Ground Plane. The display of the ground plane is now toggled off.



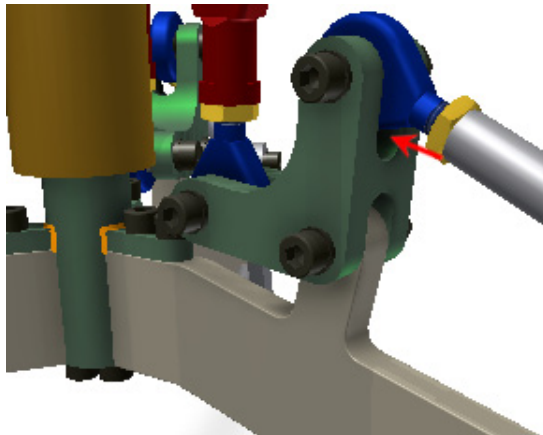
9. Begin to add one shadow setting at a time and review their impact. On the Appearance panel, Shadows list, select the Ground Shadows check box.



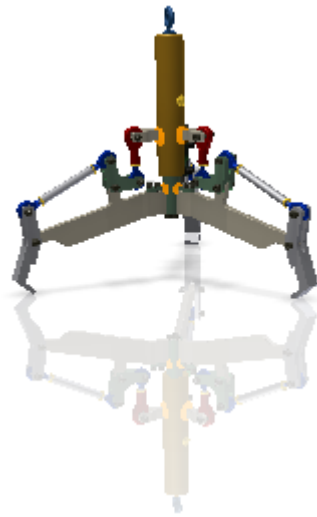
10. In the Shadows list, select the Object Shadows check box. The model now displays as shown. The ground shadow continues to display and object shadows are cast as indicated.



- 11.** In the Shadows list, select the Ambient Shadows check box. Notice the darkening of the display of the models in areas where ambient light would not be as bright.  
Tip: Toggle this setting on and off to make it easier to see where the ambient shadows are being applied.



- 12.** On the Appearance panel, click Reflections. A reflection of the design appears reflected off the ground plane.

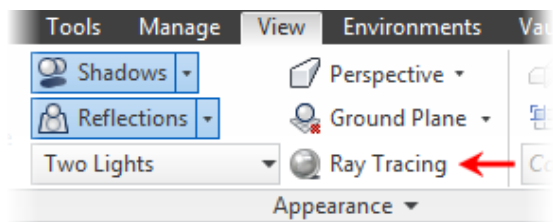


- 13.** To have the digital prototype display even more realistic, in the Visual Style list, select Realistic. By default this style uses Ray Tracing to calculate the display for material, shadows, and lighting. Notice the shadow casting from one part to another is even more pronounced.

Note: The speed of calculation depends on your design and your computer. The quality of the material is impacted by the material settings and lighting style.



- 14.** To continue to use the Realistic visual style but without Ray Tracing, on the Appearance panel, click Ray Tracing.

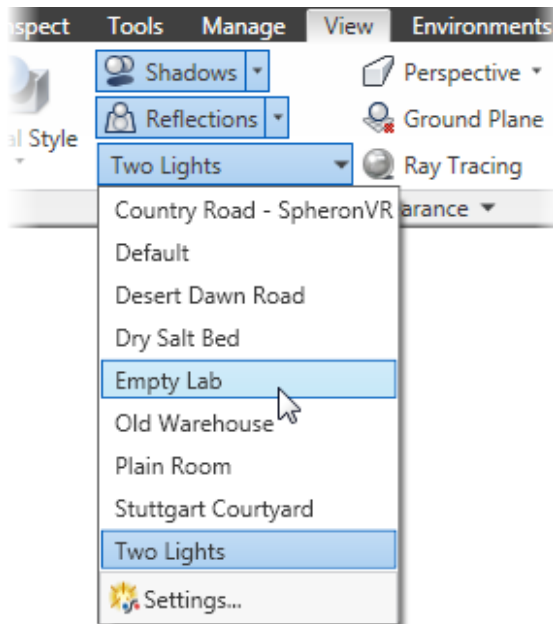


15. To export the current rendering display of the design to a neutral image file:

- On the Application menu, Export fly out, click Image.
- In the Save Copy As dialog box, Save As Type list, select PNG Files (\*.png).
- Review the file name and location where the file will be saved.
- Click Save.

Tip: In Application Options toggle off the display of the Origin 3D Indicator so it does not display in the exported image.

16. To view the design with a different lighting style, on the Appearance panel, lighting style list, select Empty Lab.



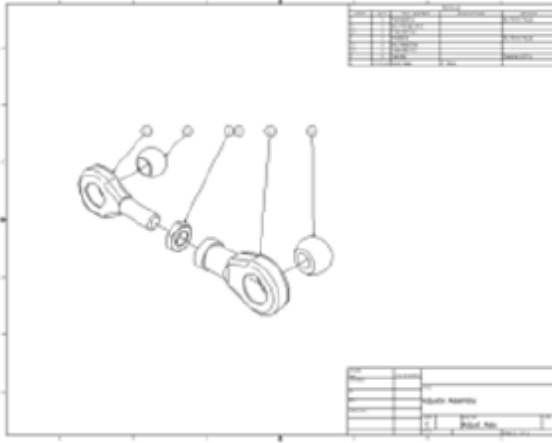
17. Zoom out and orbit the display to review the 3D environment around the digital prototype and the light source for lighting the design.



18. Save and close the assembly file.

## Exercise: Drawings

In this part of the test drive you review the styles associated with a drawing, and add new text and dimension styles to your drawing. You then add drawing views of a subassembly and one of the parts. Finally, you annotate the views to document your design.



The completed exercise

### Create a New Drawing and Define Styles

Next, you create a new drawing and review the styles that control the appearance of drawing annotations. You add a new text style and dimension style to the drawing.

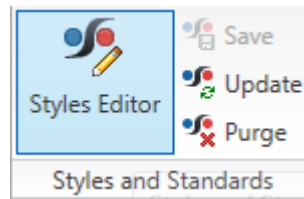
1. To start a new file:
  - On the Quick Access bar, click New.
  - On the Metric tab, double-click the *ANSI (mm).dwg* template.



ANSI (mm).dwg

2. To save the file:
  - On the Quick Access toolbar, click Save.
  - Enter a name of Adjust\_Assy.
  - Click Save.
3. Styles control the appearance of annotations in the drawing. Styles are stored in external libraries to enable sharing of standards on a project or company basis. Style setup and management is beyond the scope of this test drive but you will add new styles to the current drawing. These new styles could then be exported to a library where other users could import and use them in their drawings.

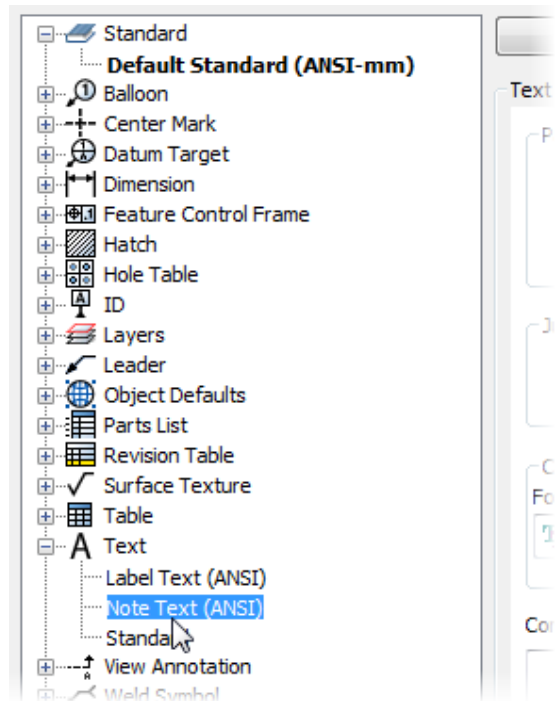
- On the Manage tab, Styles and Standards panel, click Styles Editor.



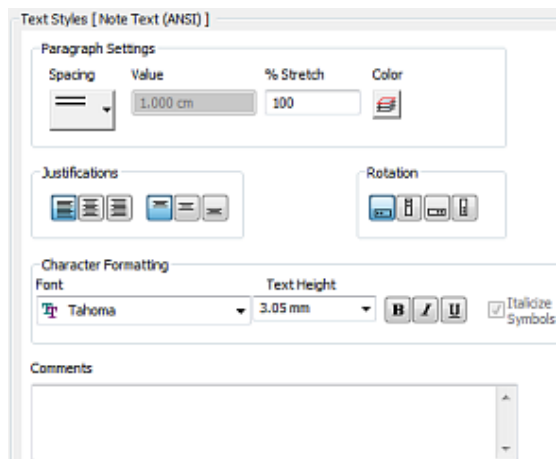
The default Drafting Standard style is ANSI (mm). Think of this as the top-level style in a hierarchy. The other styles such as text and dimension styles are child styles of the Standard style. Styles control text and dimension settings, in a similar fashion to AutoCAD® styles.

4. In the Style and Standard Editor dialog box:

- In the styles list, expand Text.
- Click Note Text (ANSI).

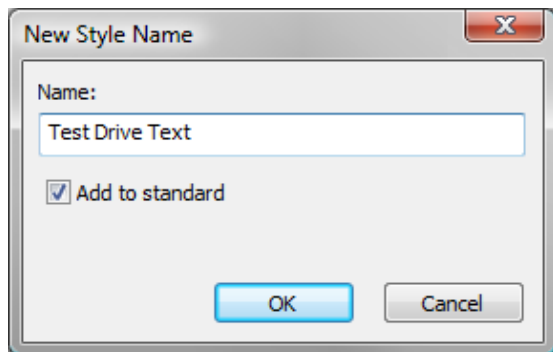


5. The settings for this style are displayed in the Editor.



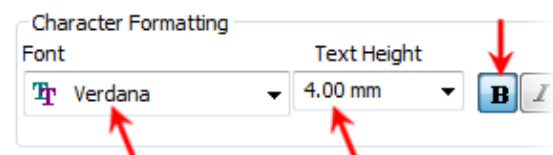
6. To create a new text style:

- At the top of the Style and Standard Editor dialog box, click New.
- In the New Local Style dialog box, Name field, enter **Test Drive Text**.
- Click OK.



7. The new text style is created, selected, and added to the list of styles. To edit the values for the new text style:

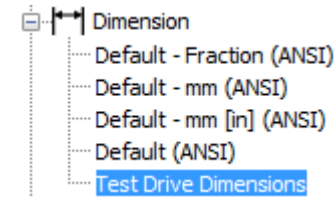
- In the Character Formatting area, Font list, select Verdana.
- For Text Height, enter **4 mm**.
- Click Bold.
- Click Save.



8. Each Standard Style includes pre-defined dimension styles. These styles can be edited or used as a foundation for creating new dimension styles. To add a dimension style:

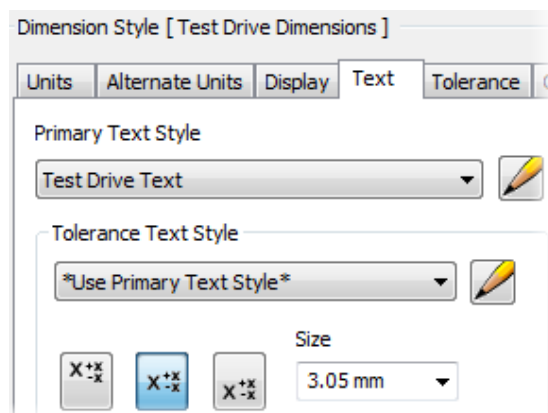
- In the styles list, expand Dimension.
- Click Default - mm (ANSI).

- Click New.
- In the New Local Style dialog box, for Name, enter Test Drive Dimensions.
- Click OK.



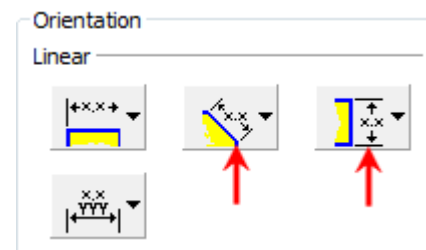
9. To set Test Drive Text as the primary text style for this new dimension style:

- In the styles list, verify that Test Drive Dimensions is selected.
- In the Dimension Style area, click the Text tab.
- In the Primary Text Style list, select Test Drive Text.



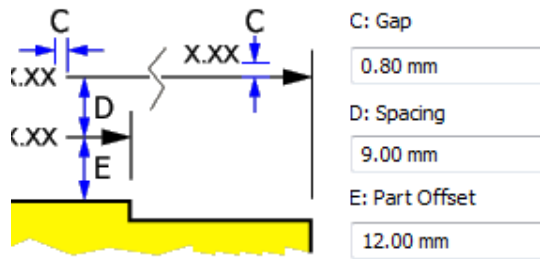
10. To set the text orientation, on the Text tab, in the Orientation area under Linear:

- In the Aligned dimension list, select Inline - Horizontal.
- In the Vertical dimension list, select Inline - Horizontal.



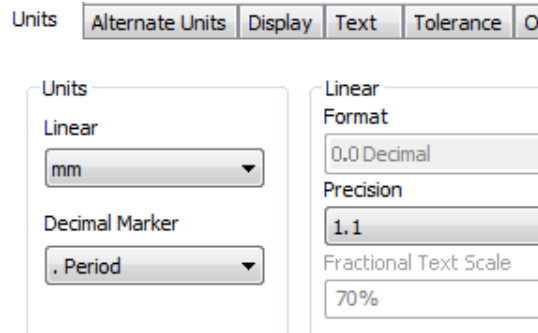
11. To control dimension and extension line settings:

- Click the Display tab.
- For D:Spacing, enter **9.00 mm**.
- For E:Part Offset, enter **12.00 mm**.



12. To modify the units:

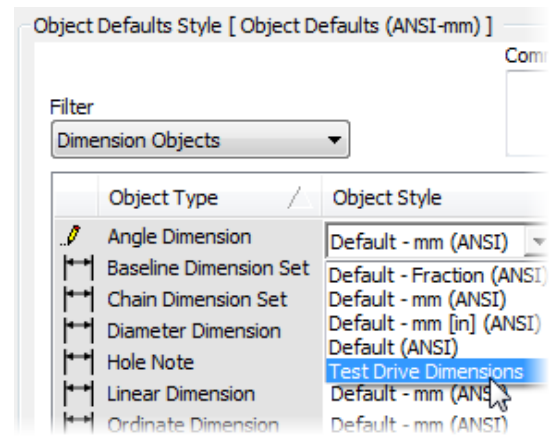
- Click the Units tab.
- In the Linear area, Precision list, select 1.1.
- Click Save.



13. Examine the information on the other tabs. The Notes and Leaders tab provides tools to create custom hole notes. Hole parameters from the model are included in the hole note, providing an associative link between the model and the drawing hole note.

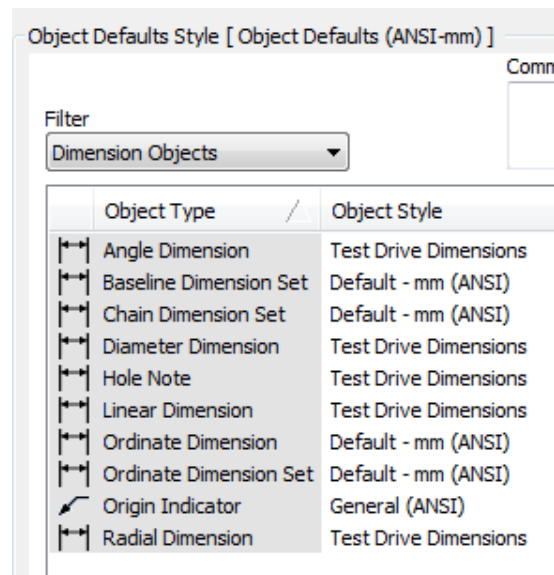
14. Styles are mapped to annotations through the Object Defaults style. Next, you change the default style for linear, radial, diameter, and angle dimensions for the new dimension style.

- In the styles list, expand Object Defaults.
- Click Object Defaults (ANSI-mm).
- Under Object Defaults Style, for Filter, select Dimension Objects.
- For Angle Dimension, in the Object Style column, select Test Drive Dimensions.



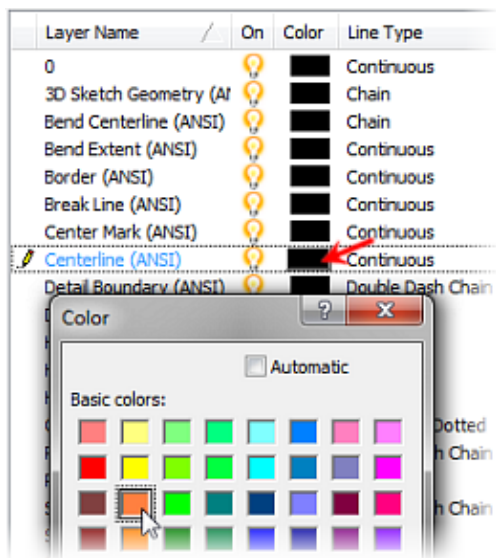
15. Continue to assign styles to the object default style:

- Select Test Drive Dimensions as the default
- Object Style for:
  - ☐ Diameter Dimensions
  - ☐ Hole Note
  - ☐ Linear Dimensions
  - ☐ Radial Dimensions.
- Click Save.



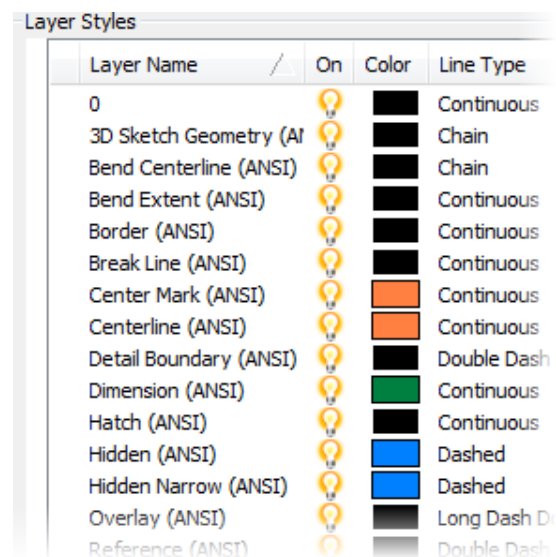
16. To edit Layer settings:

- In the Styles List, expand Layers.
- Click any Layer in the list.
- For Centerline, click the color swatch in the Color Column.
- In the Color dialog box, click Orange.
- Click OK.



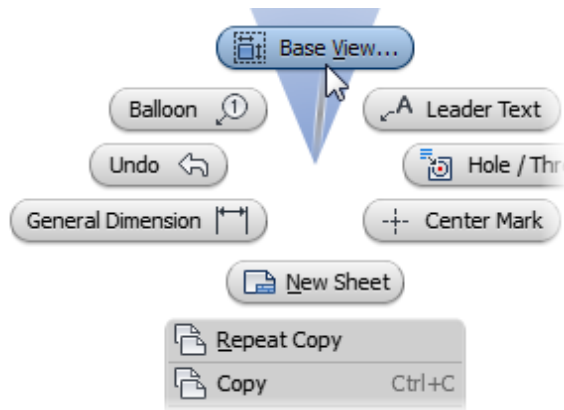
17. Repeat for the following layers:

- For Center Mark, click Orange.
- For Dimension, select one of the dark Green.
- For Hidden, click Blue.
- For Hidden Narrow, click Blue.
- Click Save.
- Click Done.



18. Next, you place an exploded view from a presentation file. You then create a parts list based on this view, and balloon the components in the view.

- In canvas, right-click. On the marking menu, click Base View.

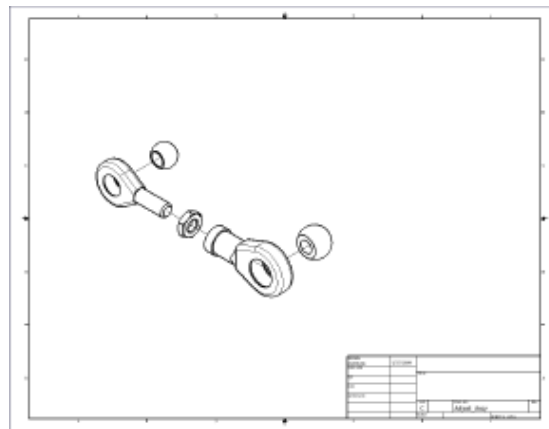


19. To select a file:

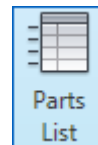
- To the right of the File list, click Open an Existing File.
- In the Open dialog box, double-click *LinkAdjust\_For\_Drawing.ipn*.

20. To place the exploded view on the drawing sheet:

- In the Drawing View dialog box, View/Scale Label area, in the Scale list, select 2:1.
- In the Orientation list, select Iso Top Right.
- Click on the sheet to place the drawing view of the exploded assembly.
- Right-click. Click Done.



21. On the Annotate tab, Table panel, click Parts List.

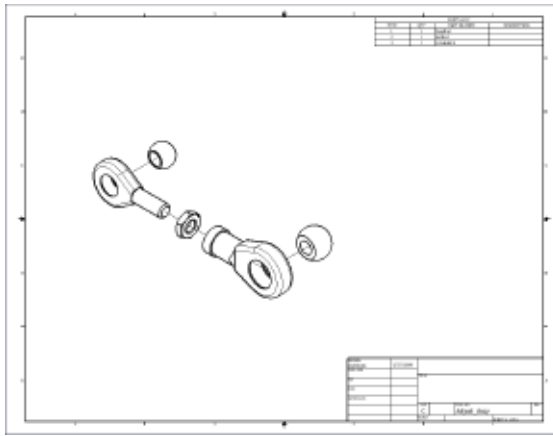


22. To create a parts list based on the exploded assembly view:

- Click inside the exploded view boundary.
- In the Parts List dialog box, in the BOM
- Settings and Properties area, Level list, select All Levels.
- Click OK.



- Click the top right corner of the drawing sheet border to place the default parts list.



23. In the browser or canvas, right-click the Parts List. Click Edit Parts List.

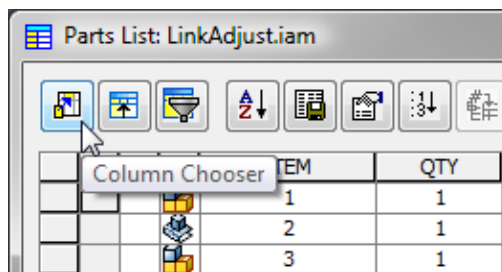
From the Parts List dialog box, you can:

- Add and remove columns that link to model property values.
- Expand and contract subassemblies to display a hierarchical parts list.
- Sort the parts list by column.
- Add user-defined custom rows for assembly components such as fluids.
- Modify the layout of columns in the parts list.
- Format column values.

PARTS LIST				
ITEM	QTY	PART NUMBER	DESCRIPTION	
1	1	MaleEnd		
2	1	JamNut		
3	1	FemaleEnd		

24. To add another column to the parts list:

- In the Parts List dialog box, click the Column Chooser tool.
- In the Parts List Column Chooser dialog box, under Available Properties, select Vendor.
- Click Add.
- Click OK.



Note: The entries in each column reflect the current value of the associated model property. You can override any cell with black text. Cells with red text cannot be changed.

25. To edit the width of the columns:

- In the Parts List dialog box, right-click the Part Number column header.
- Click Column Width.
- For Column Width, enter **45**. Click OK.
- Change the width of the Description column to 60.
- Change the width of the Vendor column to 45.

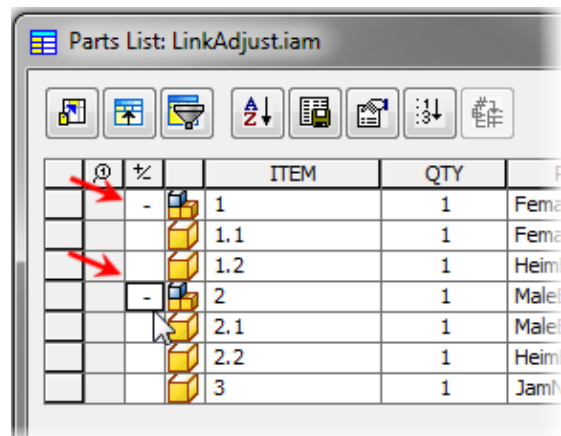
ITEM	QTY	PART NUMBER	DESCRIPTION	VENDOR
1	1	FemaleEnd		Test Drive Parts House
2	1	MaleEnd		Test Drive Parts House
3	1	JamNut		Fastener's R Us

26. To edit the justification of the values in a column:

- Right-click the Item column header. Click Format Column.
- In the Format column dialog box, in the Justification area, for Value, click Align Left.
- Click OK.

ITEM	QTY	PAR
1	1	FemaleE
2	1	MaleEnd
3	1	JamNut

27. To display subassembly hierarchies, click the plus signs in the far left of each of the two rows representing the MaleEnd and FemaleEnd subassemblies.



28. To update the Parts List in the drawing, click OK.

PARTS LIST				
ITEM	QTY	PART NUMBER	DESCRIPTION	VENDOR
1	1	FemaleEnd		Test Drive Parts House
1.1	1	FemaleEnd		Test Drive Parts House
1.2	1	FemaleEnd		Test Drive Parts House
2	1	MaleEnd		Test Drive Parts House
2.1	1	MaleEnd		Test Drive Parts House
2.2	1	MaleEnd		Test Drive Parts House
3	1	JamNut		Fastener's R Us

29. To add rows to the parts list to account for non-modeled components:

- Edit the parts list.

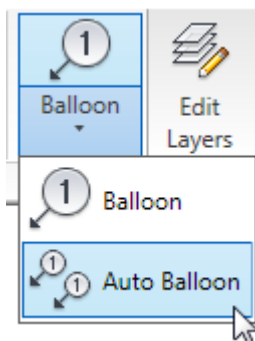


- Right-click the row header (arrow) next to the last item in the parts list. Click Insert Custom Part.
- For QTY, enter **0.1 ml.**
- In the Part Number cell for the new item 4, enter Anti-Seize.
- In the Description cell, enter XT Slick .
- Click OK.

	⊕	⊖		ITEM	QTY	PART NUMBER	DESCRIPTION
				1	1	FemaleEnd	
				1.1	1	FemaleEnd	
				1.2	1	HeimBall12	
				2	1	MaleEnd	
				2.1	1	MaleEnd	
				2.2	1	HeimBall10	
				3	1	JamNut	
				4	0.1	Anti-Seize	XT Slick

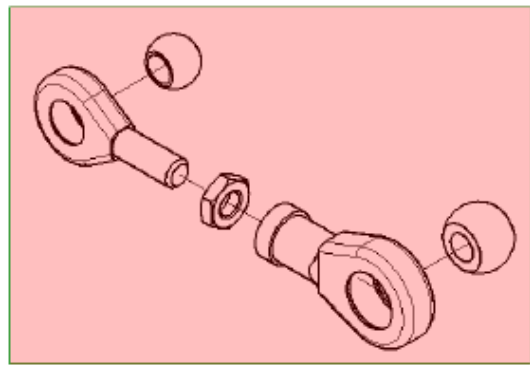
**30.** To complete the assembly drawing, you add balloons corresponding to the item numbers in the parts list. Autodesk Inventor includes both a single balloon tool, and an Auto Balloon tool to generate balloons for each component in the assembly with a matching visible line in the parts list.

- On the Annotate tab, Table panel, click Auto Balloon. Note: The Auto Balloon tool is accessed by clicking the arrow below the Balloon tool.



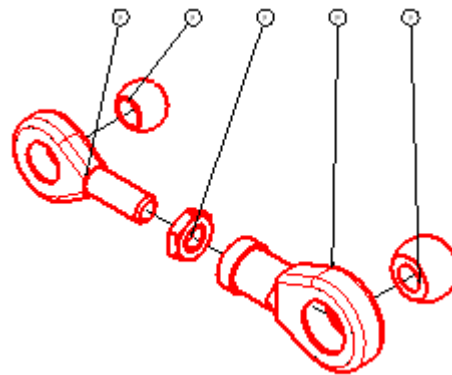
**31.** To select components to balloon:

- Click inside the exploded drawing view.
- Drag a window around all the parts in the drawing view.

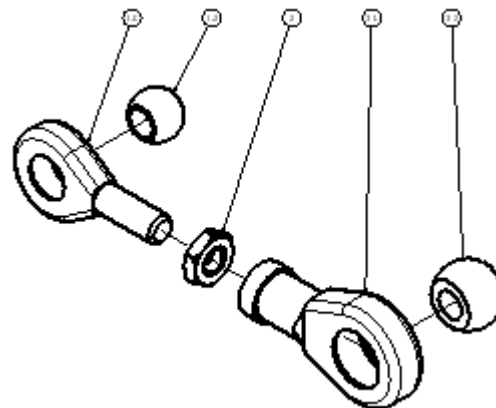


**32.** In the Auto Balloon dialog box:

- In the Placement area, click Horizontal.
- For Offset Spacing, enter **30.00 mm.**
- Click Select Placement.
- Click above the drawing view to preview the balloon placement.
- Click OK.



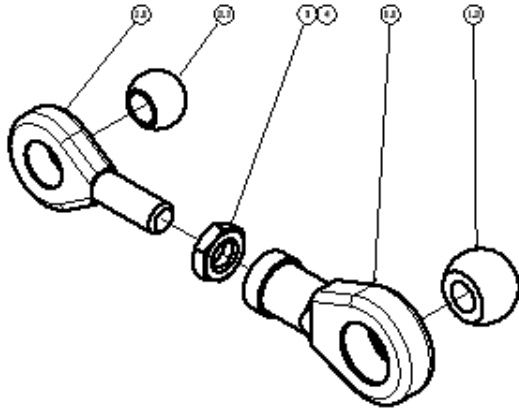
**33.** Drag the balloon arrows to more appropriate edges on the parts.



**34.** To add a balloon for the anti-seize compound:

- Right-click the balloon for the JamNut. Click Attach Balloon from List.
- Select the check box for the Item 4 Anti- Seize custom part.
- Click OK.

- Without clicking, move the cursor around the existing balloon until the new balloon is previewed at the 3 o'clock position next to the existing balloon.
- Click to place the custom part balloon.



35. The default title block on the sheet contains placeholders for properties associated with the drawing. You can modify the standard title blocks or create your own from scratch. You can add placeholders for model properties in place of, or in addition to the referenced drawing properties.

- At the top of the browser, right-click Adjust\_Assy. Click iProperties.
- On the Summary tab, in the Title field, enter Adjustor Assembly.
- Click OK.

The title block reflects the change.

DRAWN	5/27/2009	
CHECKED		
QA		TITLE
ENG		Adjustor Assembly
APPROVED		
	SIZE	ENG. NO.
	C	Adjust Assy
	SCALE	SHEET 1 OF 1

36. Save the file.

## Create Views on Multiple Sheets

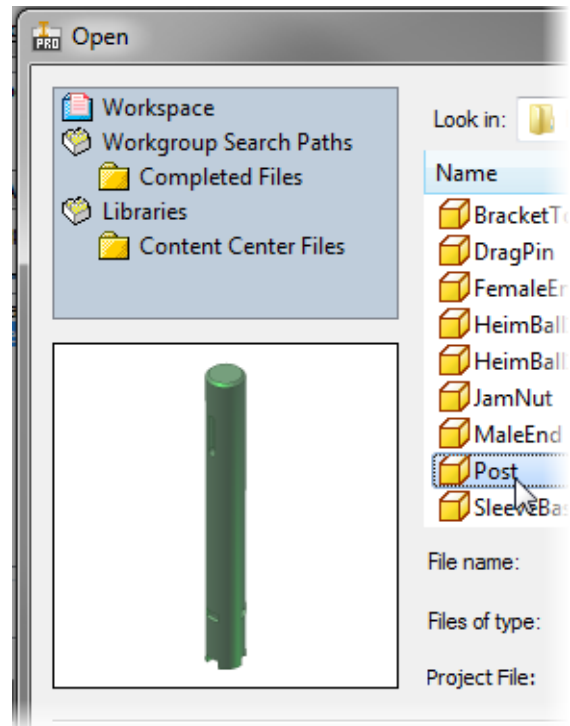
You can create separate drawing files for each assembly component, or add additional sheets to the existing drawing, and detail components within the same drawing document.

- To create a blank sheet:
  - Right-click in the background of the browser.
  - Click New Sheet.

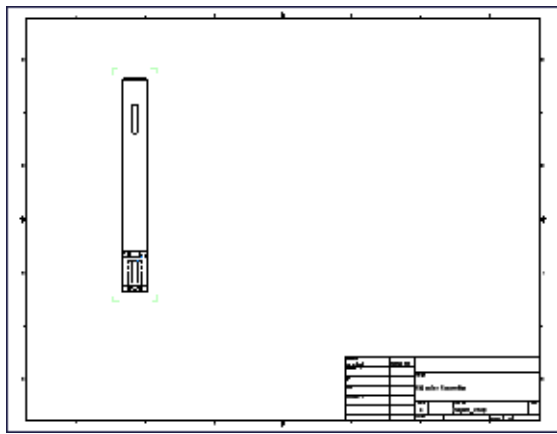
A blank sheet is created with the same sheet size, border, and title block as the first sheet. When creating a new sheet, it automatically becomes the active sheet.

Tip: Double-click a sheet in the browser to activate it.

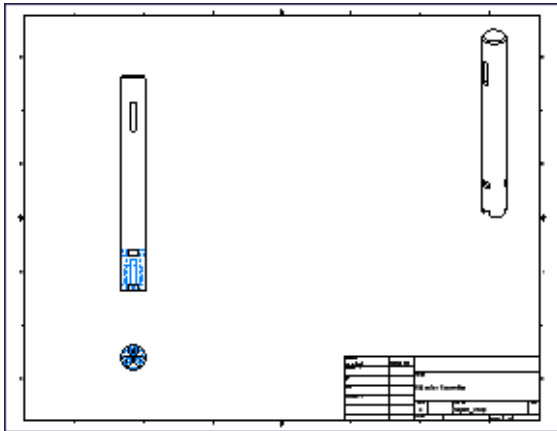
- To select a source part for a drawing view:
  - On the Place Views tab, Create panel, click Base.
  - To the right of the File list, click Open an Existing File.
  - In the Open dialog box, For Files of type: select Inventor Files (\*.ipt; \*.iam; \*.ipn).
  - Navigate to the ...|Workspace|Parts folder and select *Post.ipt*.
  - Click Open.



- To place a base view of a part:
  - In the Scale list, select 1:1.
  - in the Orientation list, select Bottom.
  - Click on the left side of the sheet to place the view.



4. To add projected views from the base view:
  - Move the cursor below the first view and click to place the view.
  - Move the cursor to the right and above the original view. An isometric projection of the original view is previewed.
  - Click to place the view.
  - Right-click. Click Create.
  - After the views are created, click the border of the isometric view and drag it to the topright corner of the sheet.

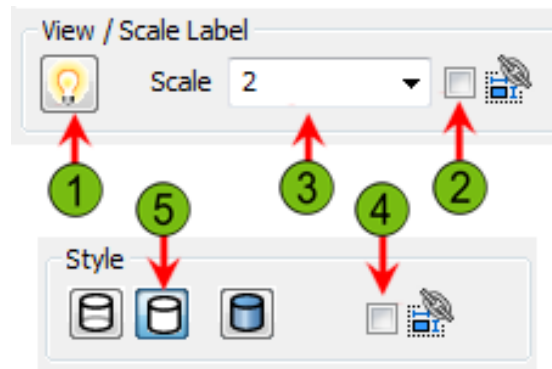


5. You can specify that a view shows hidden lines, has hidden lines removed, or is a shaded view. To modify a view to be a shaded view:
  - Right-click inside the isometric view border, but not on a line or other entity inside the view.
  - Click Edit View.
  - In the Style area, click Shaded (blue icon).
  - Click OK.

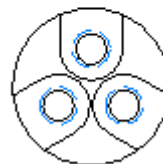
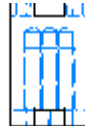
The isometric view changes to a shaded view of the post.



6. To modify the view scale:
  - Right-click inside the projected view of the end of the post. Click Edit View.
  - In the View/Scale area, click Toggle Label Visibility (1).
  - Clear the Scale from Base check box (2).
  - In the Scale field, enter 2 (3).
  - In the View Identifier field, enter BOTTOM VIEW.
  - In the Style area, clear the Style from Base check box (4).
  - Click Hidden Line Removed (5).

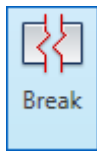


7. In the Drawing View dialog box, click OK.



BOTTOM VIEW  
SCALE 2

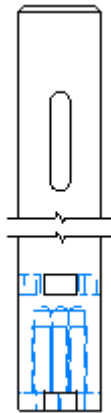
8. The center section of the post has no detail so you create a broken view to obtain some additional sheet real estate.
  - On the Place Views tab, Modify panel, click Break.



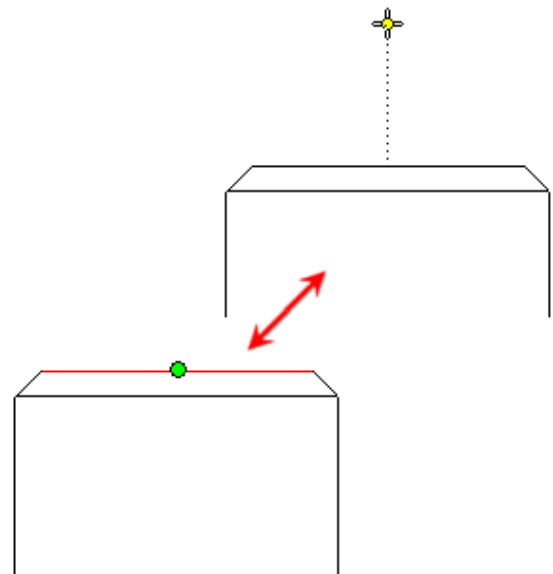
9. To create the broken view:

- Click the base view (first created view) of the post.
- Click just below the keyway in the base view.
- Click just above the slots in the base view.

The view is changed to a broken view. You can edit the break through a context menu, or by dragging the break lines.



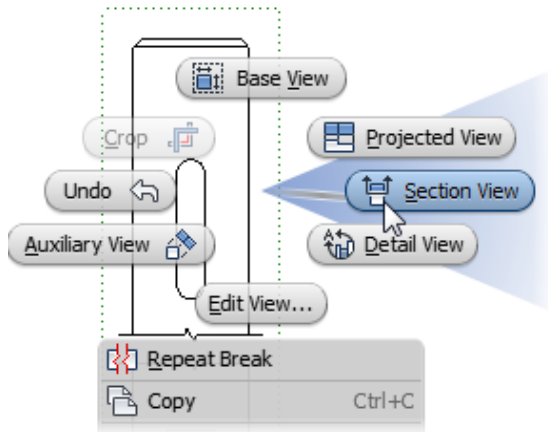
- Position the cursor over the upper horizontal edge of the post in the base view. Pause when a green dot indicates the midpoint of the line. Do not click.
- Move the cursor above the edge. A dotted line implies alignment with the edge midpoint.
- When the cursor is positioned above the midpoint as shown, click to set the first point.



10. To obtain information on the key seat and mounting slot/hole at the base of the post, you create a section view from the base view.

To access the Section View tool:

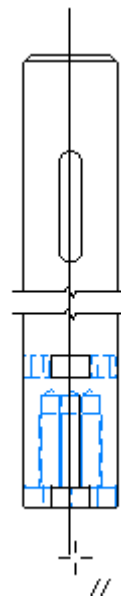
- Right-click in a blank area inside the base view.
- On the marking menu, click Section View.



11. To create the section view:

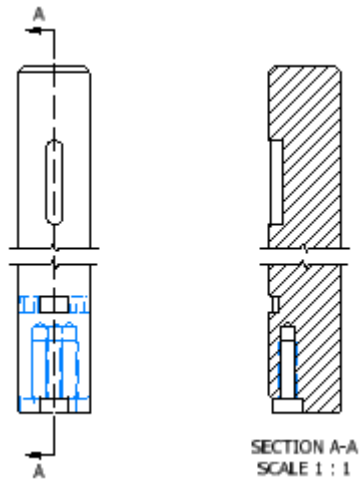
12. To continue placing the section view:

- Move the cursor below the part. Ensure that a perpendicular or parallel constraint symbol is visible. This indicates the section line is vertical.
- Click outside the bottom of the view.



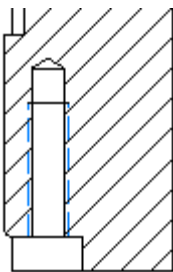
13. To complete the section view:

- Right-click. Click Continue.
- Move the cursor to the right. A projected view is previewed.
- Click to place the section view.



14. Next, you create a detail view of the slot/hole in the section view.

- Zoom in to the bottom of the section view.

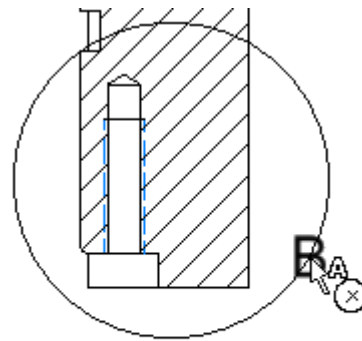


## SECTION A-A SCALE 1 : 1

15. Right-click in a blank area inside the section view. On the marking menu, click Detail View.

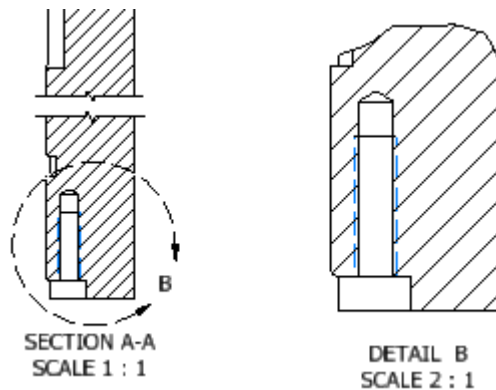
16. To create the detail view:

- Click near the intersection of the post centerline and a horizontal line approximately halfway along the length of the hole. This defines the center of the detail view circle.
- Move the cursor outside the post, and click when the circle encompasses the lower portion of the post.



## SECTION A-A SCALE 1 : 1

17. A preview of the detail view is attached to the cursor. To place the detail view, click to the right of the section view to place the detail view.

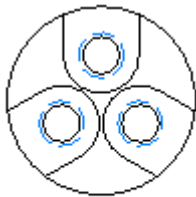


18. Save the file.

### Add Annotations to the Drawing

Autodesk Inventor drawing annotations take advantage of the information contained in the model. Dimensions, notes, centerlines, and so on are all associative to the model geometry. Changes to the model are automatically reflected in the drawing views and annotations. In this section of the test drive you add a variety of common drawing annotations to the drawing views.

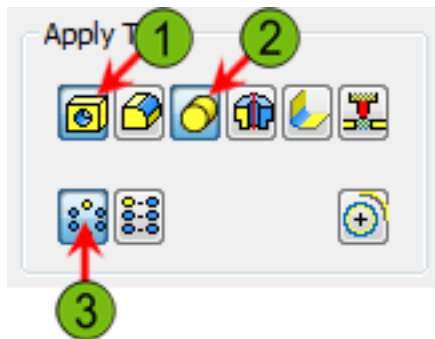
1. You can place centerlines on arcs, circles and on cylinders shown in elevation. You use a number of different centerline annotation tools in the different views.
  - Zoom in on the end view of the post.



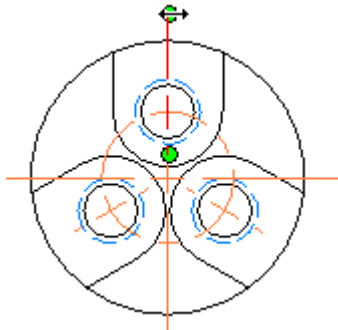
**BOTTOM VIEW  
SCALE 2**

2. To automatically place centerlines:

- Right-click the end view of the post. On the shortcut menu, click Automated Centerlines.
- In the Automated Centerlines dialog box, Apply To area, verify that Hole Features (1), Cylindrical Features (2), and Circular Patterned Features (3) are selected.
- Click OK.

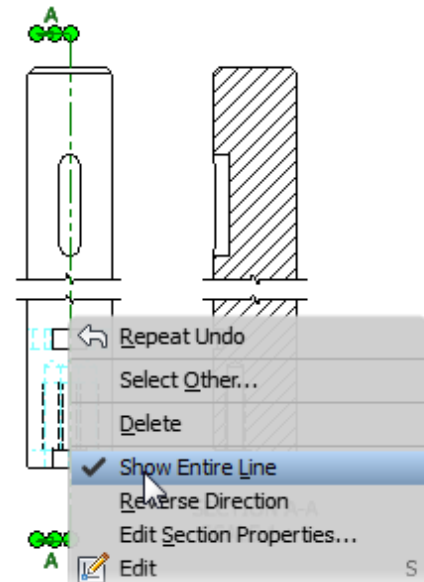


3. Drag the endpoints of the center mark on the large circle to match the following illustration.



4. To modify the display of the section line so it is less intrusive to the view:

- Zoom in on the base view.
- Right-click the section line. Click Show Entire Line to clear the check mark.

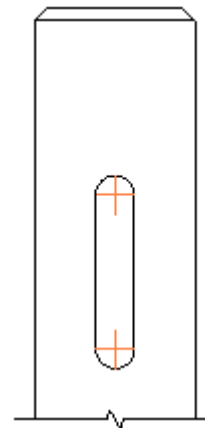


5. On the Annotate tab, Symbols panel, click Center Mark.



6. To add center marks to individual arcs and circles:

- Right-click on the sheet. Click Extension Lines to clear the check mark. The placed center marks will be a single cross at the arc/circle center.
- Click the arc at the top of the key seat.
- Click the arc at the bottom of the key seat.
- Right-click. Click Done.



7. To create a centerline on the section view of the cylinder, the Centerline Bisector tool is used.

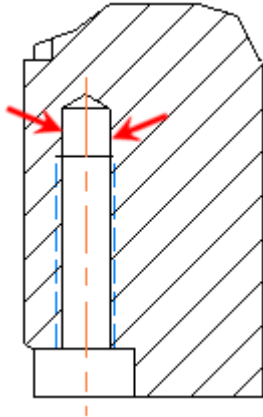
- Zoom in on the Detail B view.

- On the Annotate tab, Symbols panel, click Centerline Bisector.



8. To place the centerline bisector:

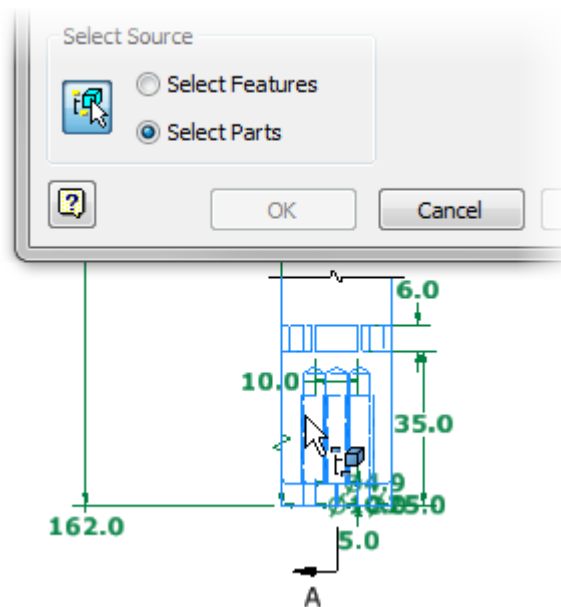
- Click the vertical edge on the left side of the hole.
- Click the vertical edge on the right side of the hole.
- Right-click. Click Done.
- If desired, drag the endpoints of the centerline past the extents of the hole.



9. You can import model sketch dimensions into drawing views and also add dimensions to the views with the drawing annotation tools. The placed dimensions are associative to the model. You can place baseline and ordinate dimensions in addition to using the general dimension tool. You can select a dimension style prior to placing dimensions, or change the dimension style of selected dimensions after creation.

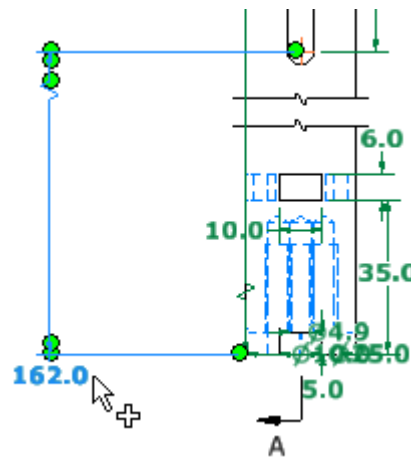
To retrieve model sketch dimensions:

- Zoom in to the base view.
- Right-click the base view. In the shortcut menu, click Retrieve Dimensions.
- In the Retrieve Dimensions dialog box, Select Source area, click Select Parts.
- In canvas, select the model in the base view.

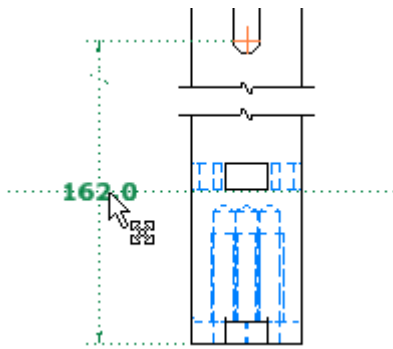


10. To select specific model sketch dimensions:

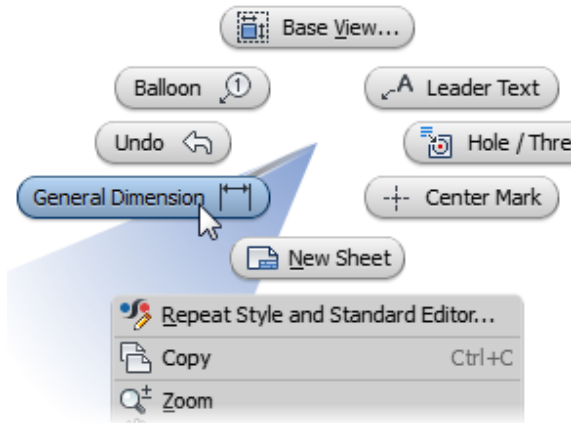
- In the Retrieve Dimensions dialog box, click Select Dimensions.
- In canvas, select the broken 162.0 dimension.
- Click OK.



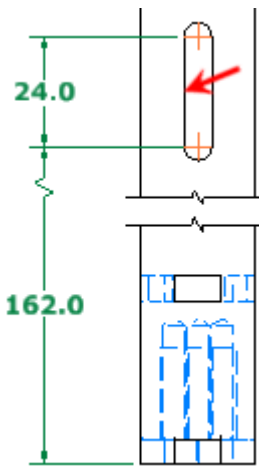
11. Click and drag the dimension until it is positioned with the centerline and the dimension displays as a dotted line. Note: The dotted lines indicate alignment with the centerline and the dimension offset you specified in the Test Drive Dimensions dimension style.



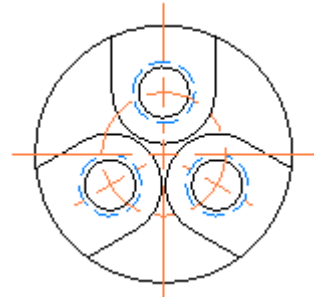
12. To activate the general dimension tool, in canvas, right-click. In the marking menu, click General Dimension.



13. To place a dimension on the key seat and align it to the first dimension:
- Click the left edge of the key seat (click the line, not a vertex).
  - Rub the cursor over the first dimension (do not click) until a yellow dot is displayed.
  - Click to place the dimension text midway along the length of the key seat, and aligned with the first dimension. The yellow dot implies alignment with the 'scrubbed' dimension.
  - If the Edit Dimension dialog box displays, click OK.
  - Right-click. Click Done.

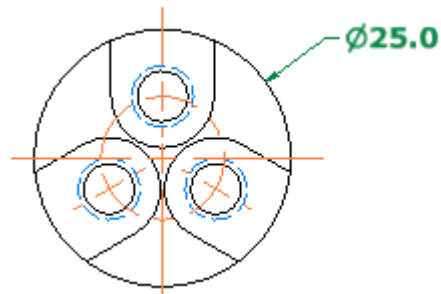


14. Zoom in on the end view of the post.



15. To add a diameter dimension you leverage the dimensions created during the modeling process, similar to placing the broken dimension on the base view:

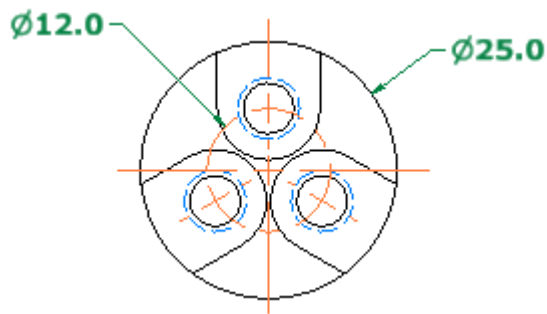
- Right-click the view of the end of the post. On the shortcut menu, click Retrieve Dimensions.
- In the Retrieve Dimensions dialog box, click Select Parts.
- Click the part in the drawing view of the post end. A preview of all available dimensions is displayed. Alternately, you could select specific features to preview available dimensions using the Select Features option.
- In the Retrieve Dimensions dialog box, click Select Dimensions.
- Select the diameter 25 dimension.
- Click OK.



16. To dimension the bolt hole circle.

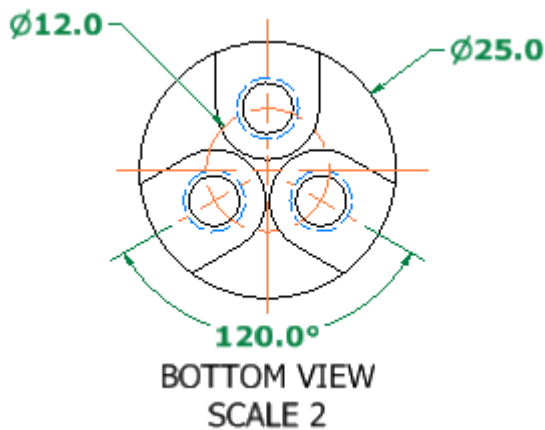


- Start the General Dimension tool.
- Select the bolt hole circle.
- Click to the upper left of the view to place the dimension.
- If the Edit Dimension dialog box displays, click OK.



17. To add an angle dimension to the hole pattern:

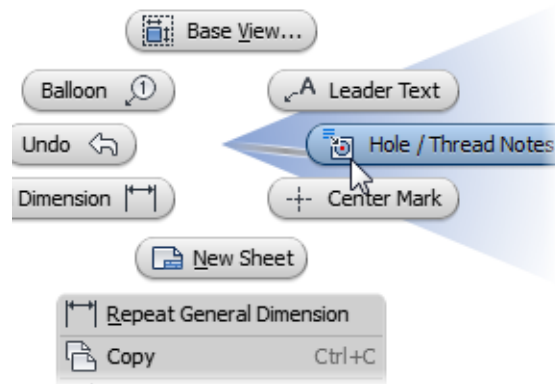
- Select the centerline through the lower left hole.
- Select the centerline through the lower right hole.
- Move below the part and click to place the angular dimension.
- If the Edit Dimension dialog box displays, click OK.
- Right-click. Click Done.
- Click and drag the view label to provide room for the dimension.



18. The Hole and Thread Note tool captures data from the model. You can create custom hole note templates in a dimension style for drilled, tapped, counterbored and countersunk holes.

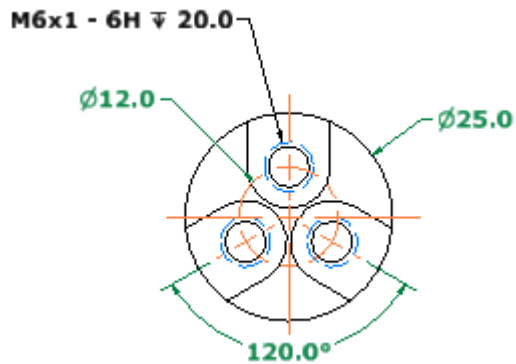
To access the Hole and Thread Note tool:

- Right-click in a blank area of the drawing. (Not in a view).
- Click Hole/Thread Notes.



19. To place the hole note:

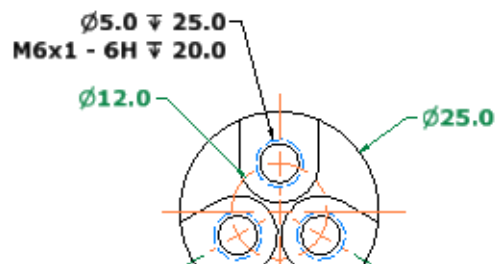
- Select the edge of the upper hole in the same view.
- Click to place the hole note as shown.
- Right-click. Click Done.



20. You can also edit individual dimensions and hole notes to add text or other information. To edit the hole note:

- Right-click the hole note text. Click Edit Hole Note.
- In the Edit Hole Note dialog box, Options area, select the Tap Drill check box.
- Click OK.

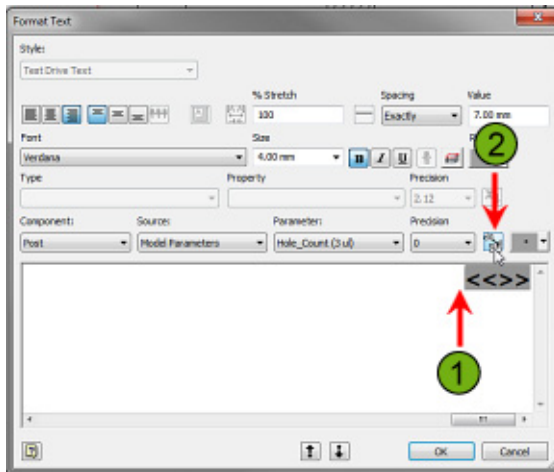
The hole note now shows the tap drill information.



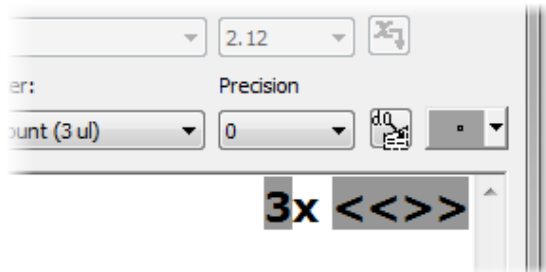
21. You can customize the text for any dimension or hole note. To edit the hole note text:

- Right-click the hole note text. Click Text.

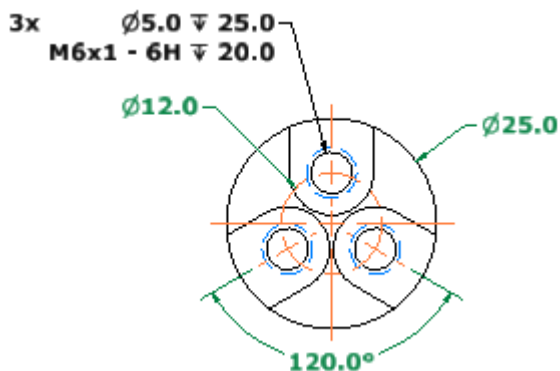
- In the Format Text dialog box, Component list, select Post.
- In the Parameter list, select Hole\_Count (3.00 ul). In the Precision list, select 0.
- Click to position the cursor at the beginning of the hole note placeholder (1).
- Click Add Parameter (2).



22. Between the 3 and the placeholder, enter X and a space.

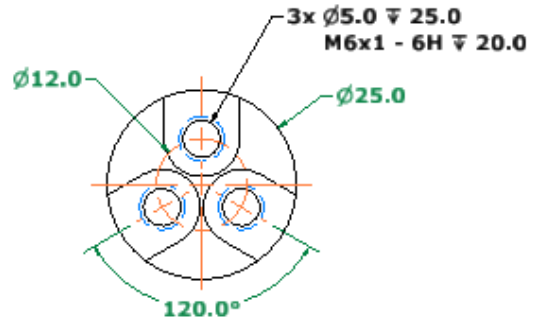


23. To complete the hole note, click OK. The additional text is not aligned with the hole note.



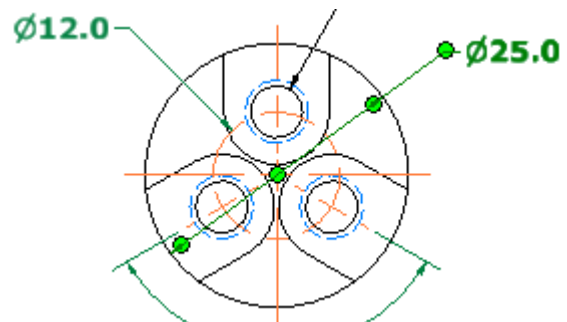
24. To reposition the text on the sheet:

- Click on the hole note and drag it to the right. The text justifies to the left when placed on the other side of the leader.



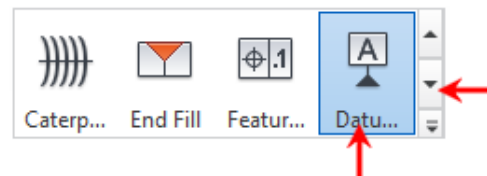
25. To modify a dimension:

- Right-click the 25.0 diameter dimension. On the shortcut menu, click Options > Single Dimension Line.
- Drag the green dot at the horizontal jog of the dimension line to match the following illustration.



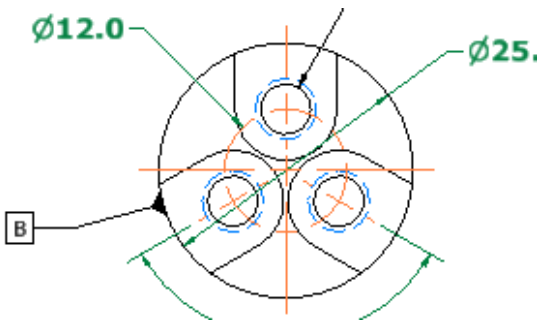
26. To access the Datum Identifier Symbol tool:

- On the Annotate tab, Symbols panel, click the down arrow to scroll through symbols.
- Click the Datum Identifier Symbol tool.



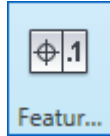
27. To place the symbol:

- Click a point on the outer diameter of the post.
- Move the cursor outside the post and click to define the leader endpoint.
- Right-click. Click Continue.
- In the Format Text dialog box, for the text, enter B.
- Click OK.
- Right-click. Click Done.



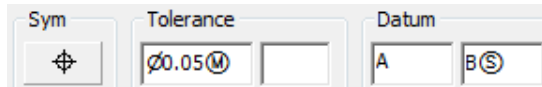
28. Next, you add a feature control frame to the hole note.

- On the Annotate tab, Symbols panel, click Feature Control Frame.



29. To place the feature control frame:

- Click a location below the hole note text.
- Right-click. Click Continue.
- In the Feature Control Frame dialog box, enter the information as shown in the following illustration. Use the GDT symbols displayed near the bottom of the dialog box.



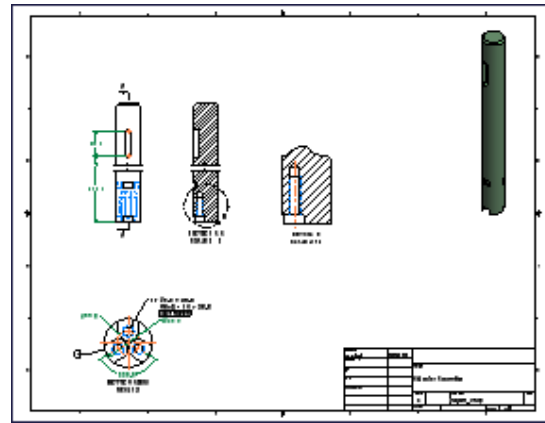
30. To place the feature control frame:

- Click OK to place the feature control frame.
- Right-click. Click Done.

**3X Ø5.0 ∇ 25.0**  
**M6x1 - 6H ∇ 20.0**  
 [Feature Control Frame: Ø0.05M A B S]

31. To complete the drawing:

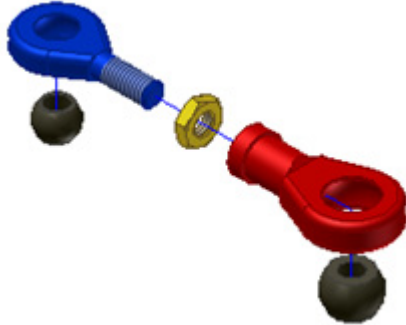
- On the Navigation Bar, click Zoom All.
- Save the file.



32. Close all open files.

## Exercise: Presentation Files

In this part of the test drive you create an exploded version of the assembly model using the Presentation tools in Autodesk Inventor. You will add tweaks to assembly components in order to create the exploded view, view animations of the component tweaks, and use the Sequence View to review tweak motion.

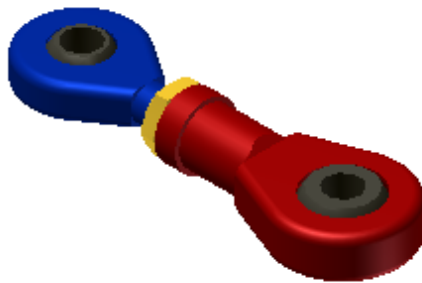


The completed exercise

### Create an Exploded Version of the Assembly

In this section of the test drive, you create a presentation file with components exploded from their assembled position.

1. Open *LinkAdjust.iam*.



2. Use the Orbit and Zoom tools to examine the assembly.
3. To start a new file:
  - On the Quick Access toolbar, click New.
  - On the Metric tab, double-click the *Standard (mm).ipn* template.



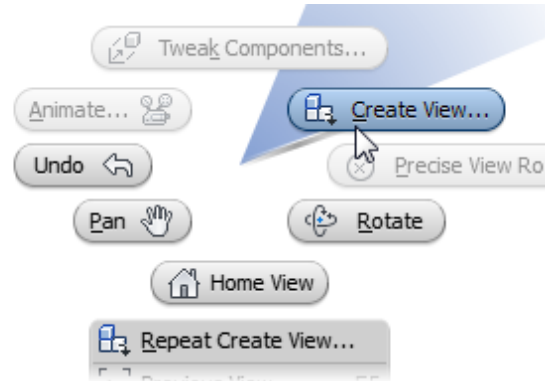
Standard (mm).ipn

4. To save the file:
  - On the Quick Access toolbar, click Save.

- Enter a name of *LinkAdjust\_Presentation*.
- Click Save.

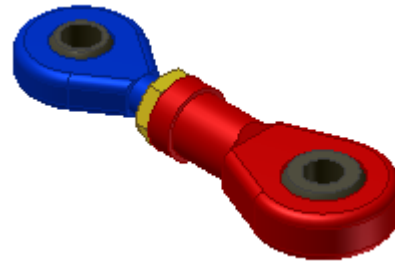


5. In canvas, right-click . Click Create View.

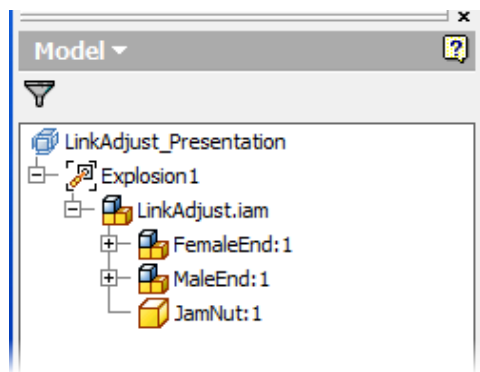


6. In the Select Assembly dialog box:
  - Verify that the *LinkAdjust.iam* file is selected.
  - Verify that Manual is selected.
  - Click OK.

Note: The *LinkAdjust* assembly file is automatically selected if it is the only open assembly file in Autodesk Inventor. If it is not selected, click the Open an Existing File button next to the File field and choose the file. The assembly view is placed in its assembled state in the presentation file.



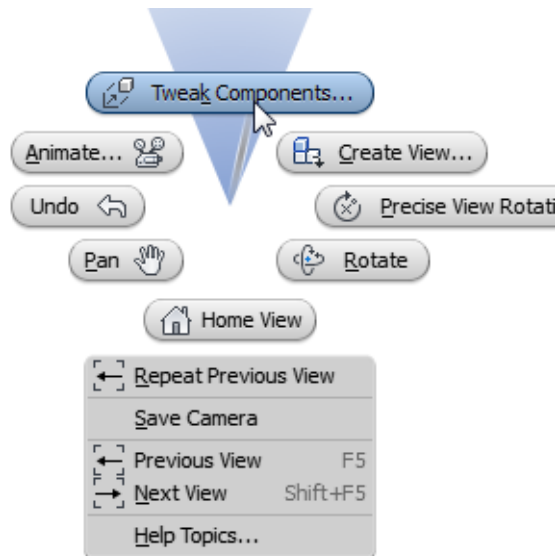
7. To expand the display of the assembly components:
  - Click the plus sign in front of *Explosion1*.
  - Click the plus sign in front of *LinkAdjust.iam* to expand its display.



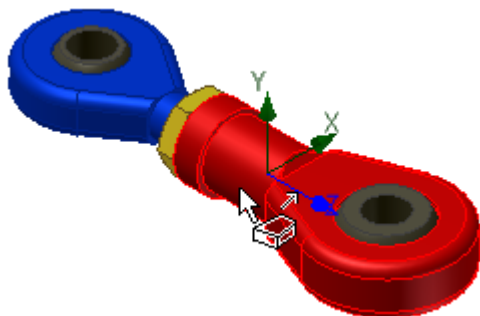
8. Next, you add tweaks to the presentation file that will move the assembled components into an unassembled state.

To access the Tweak Components tool:

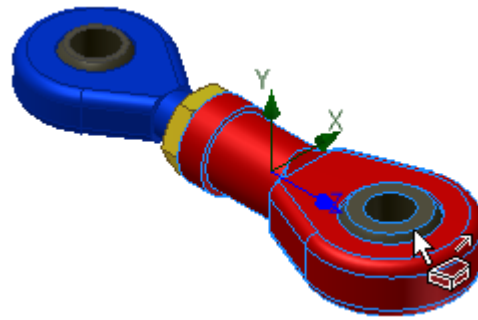
- In canvas, right-click. Click Tweak Components.



9. To specify the tweak direction, select the shaft of the FemaleEnd subassembly where indicated in the following illustration. You know you selected the right geometry on the part when the triad displays as shown.



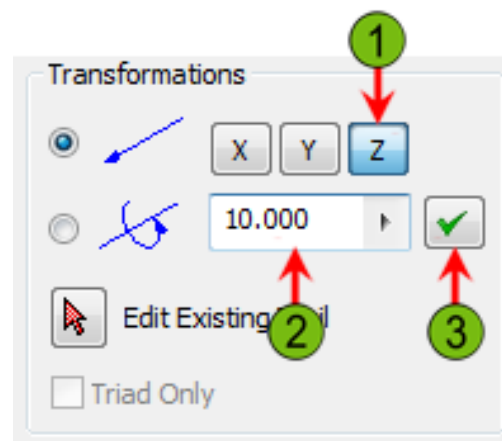
10. For the components to tweak, select the FemaleEnd and HeimBall components.



Tip: You can select components in canvas or in the browser.

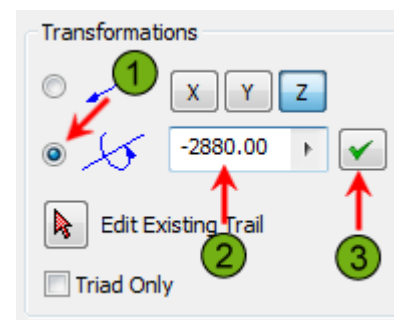
11. To move (tweak) the selected components:

- Verify that the Z button is selected in the dialog box (1).
- In the Tweak Distance field, enter 10 (2).
- Click Apply (3).



12. To create a rotation tweak with the selected components:

- Click the Rotation transformation type (1).
- Enter -2880 in the Tweak Distance field (2).
- Click Apply to create the tweak (3).

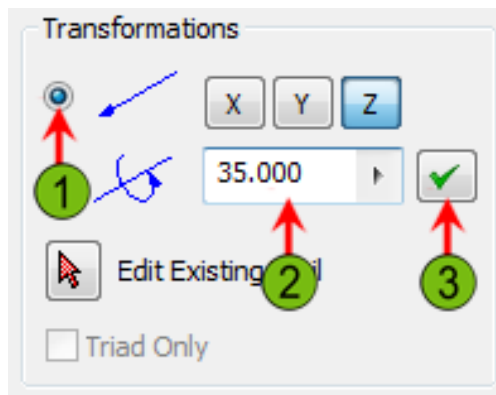


Note: The components will not appear as though anything has changed. The -2880 degree increment rotates the components exactly eight times. Later in the test drive, you will group this tweak with the first one so that it appears that the female end is being rotated into place.

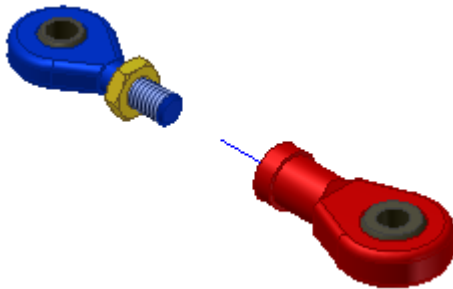
13. To add an additional tweak:

- Click the Translation transformation type (1).
- In the Tweak Distance field, enter 35 (2).

- Click Apply (3).
- Click Clear.

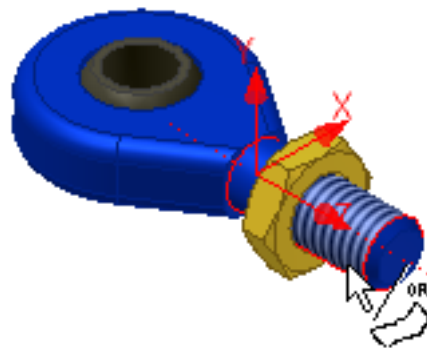


14. The position of the FemaleEnd subassembly should display as shown in the following illustration.



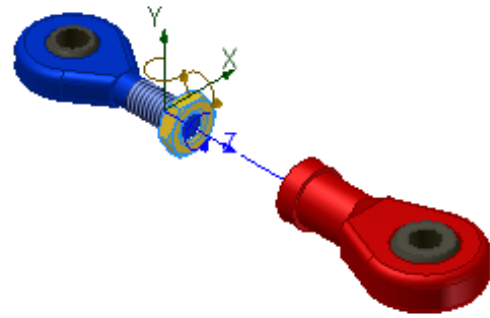
15. To add tweaks to the JamNut component:

- In the Tweak Component dialog box, verify that Direction is selected.
- Select the shaft of the MaleEnd subassembly as shown.



16. To add a tweak:

- Select the JamNut component to select it.
- Verify that the Z button is selected in the dialog box.
- In the Tweak Distance field, enter 12.
- Click Apply.



17. To apply a rotational tweak:

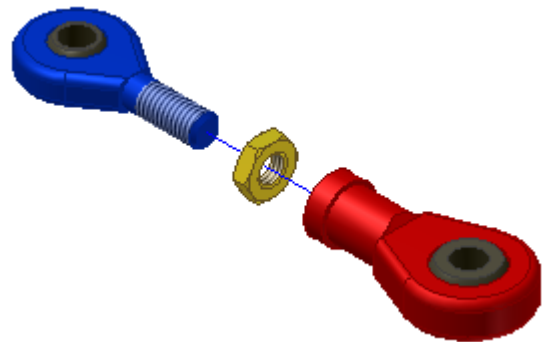
- Click the Rotation transformation type.
- Enter -288.0 in the Tweak Angle field.
- Click Apply.

Note: The component will not look like it has changed.

18. To add an additional tweak:

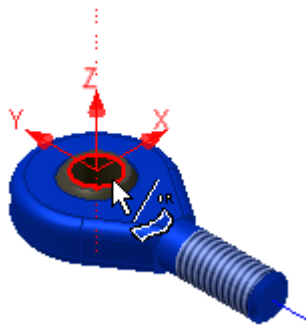
- Click the Translation transformation type.
- In the Tweak Distance field, enter 17.
- Click Apply.
- Click Clear.

The JamNut should be positioned approximately half-way between the Male and Female End subassemblies.



19. To add tweaks to the two Heim Ball components:

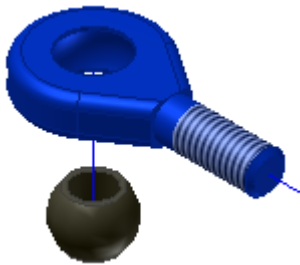
- In the Tweak Component dialog box, verify that Direction is selected.
- Position your cursor over the circular edge of the HeimBall component in the MaleEnd subassembly until the temporary Z axis aligns with the axis of the component.
- Click to accept the axis orientation.
- Click the HeimBall component to select it.
- Verify that the Z axis is selected.



**20.** In addition to entering precise values in the Tweak Distance edit box, you can also drag components in canvas to position them.

To tweak the HeimBall component:

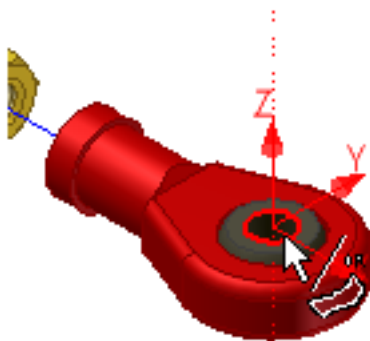
- Click and drag the component in canvas to move the HeimBall below the MaleEnd subassembly. The Tweak Distance field should read approximately -25.
- Click Apply.
- Click Clear.



Note: Once you have selected the component, to drag to location, the cursor does not need to be on the component.

**21.** To specify a direction for the next tweak:

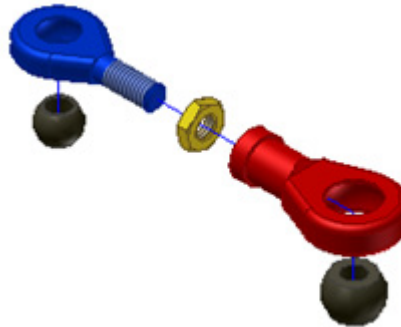
- Verify that Direction is selected.
- Position your cursor over the circular edge of the HeimBall component in the FemaleEnd subassembly until the temporary Z axis aligns with the axis of the component, as shown in the following illustration.
- Click to accept the axis orientation.



Tip: When selecting the direction vector or changing the vector that you want to move the component(s) along, you can select the X, Y, or Z axis from either the Tweak Component dialog box or select the axis from the tri-pod in canvas.

**22.** In the Tweak Component dialog box:

- Click the HeimBall part to select it.
- Verify that the Z axis is selected.
- Click and drag the component in canvas to move the HeimBall below the MaleEnd subassembly. The Tweak Distance field should read approximately -25.
- Click Apply to create the tweak.
- Click Close.

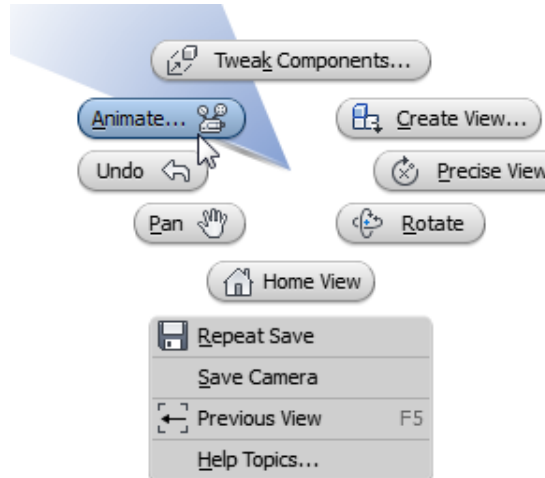


**23.** Click Save.

## Animate and Edit Presentations

In this section of the test drive, you animate the presentation, modify how the parts animate, and create a neutral video file of the animation.

**1.** Next, you will view and modify the animation. In canvas, right-click. Click Animate.



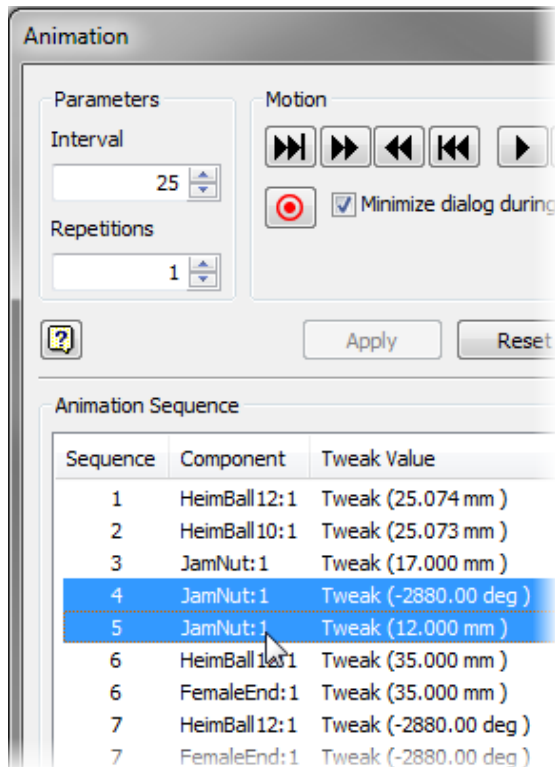
**2.** To preview the animation:

- In the Animation dialog box, Parameters area, Interval field, enter 10.
- Click Apply.
- Click the Play Forward button and watch the animation.
- Click Reset when the animation is finished.

Tip: The interval setting adjusts the number of steps that a component moves along a particular tweak. By specifying a lower number, the animation is played faster.

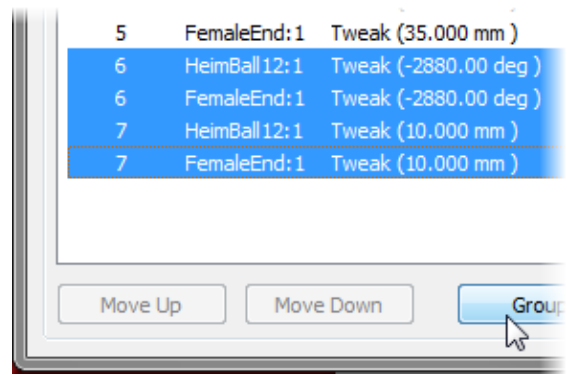
3. Notice that there are a few items in the animation that need to be changed to provide a better presentation. To group tweaks together:

- In the Animation dialog box, click the More (>>).
- Press CTRL and select Sequence numbers 4 and 5.
- Click Group.
- Click Apply.



4. To group additional sequences:

- Repeat the step above to group Sequence numbers 6 and 7.
- Click Apply in the Animation dialog box.



Note: Verify that you select all four items labeled either 6 or 7 in the dialog box.

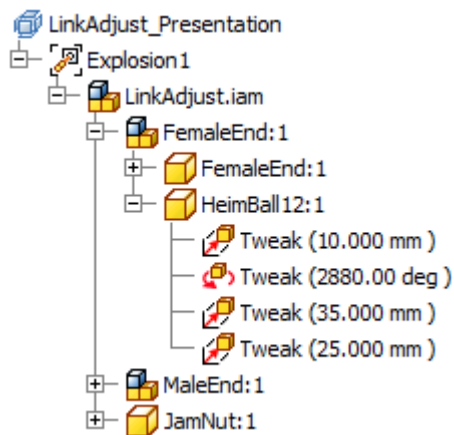
5. To play the animation:

- Click the more (<<) button to reduce the dialog box to its original size.
- Click the Play Forward.
- Click Cancel to close the Animation dialog box.

6. Next, you will modify some tweak values.

- Expand FemaleEnd:1 and HeimBall12:1 in the browser.

Note: The hierarchy displays tweaks nested below each component definition and their associated values.

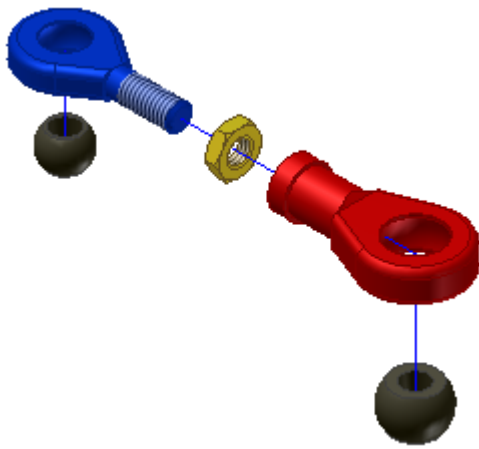


7. To modify the tweak value:

- In the browser, under HeimBall12:1, select the Tweak that is approximately 25 mm.
- An offset field is displayed at the bottom of the browser. In the offset field, enter 40.
- Press ENTER.

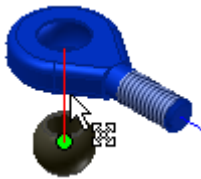
The HeimBall component moves to its new position.





8. To modify the tweak applied to the other HeimBall component:

- In canvas, click the trail that is associated with the vertical 25 mm tweak under HeimBall10:1.
- Click and drag the green dot to move the HeimBall so that it is approximately double the original distance.
- Click anywhere in canvas to deselect the trail.

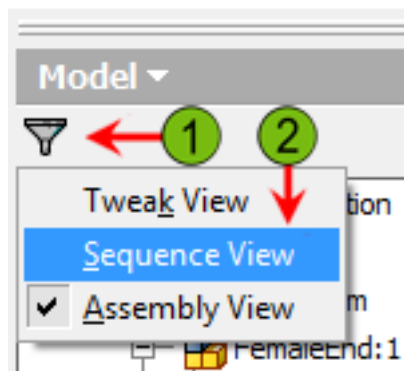


Tip: You can double-click a trail in canvas to access the Tweak Component dialog box.

9. Save the Presentation.

10. To complete the animation, you will add specific viewing angles and descriptions to the animation using functionality within a Sequence View.

- At the top of the browser, click the Browser Filters (1).
- Select Sequence View from the menu.

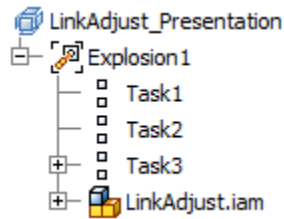


The browser displays Tasks and Sequences for the tweaks that have been created.

11. To create additional tasks:

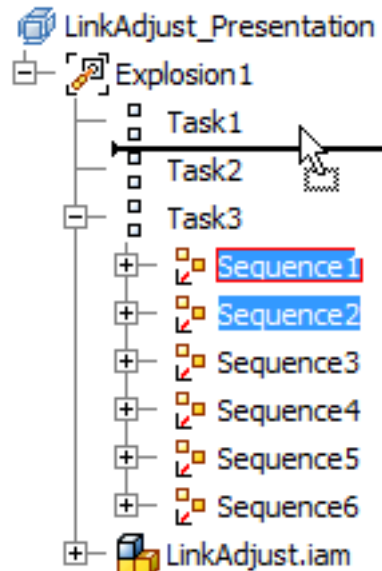
- In the browser, right-click Explosion1.

- Click Create Task.
- Repeat the process to create one more task in the browser.



12. To move sequences between tasks:

- Expand Task3 in the browser.
- Select Sequence1.
- Hold the CTRL key and select Sequence2.
- Drag and drop them below Task1.



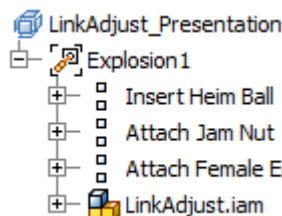
13. Repeat the drag and drop process to move the following sequences to the appropriate task:

- Sequences 3 and 4 to Task2.
- Sequences 5 and 6 should be in Task3.

Note: Adding tasks is not necessary, but is a way that you can organize your presentation into a specific set of assembly instructions.

14. To rename the tasks in the browser:

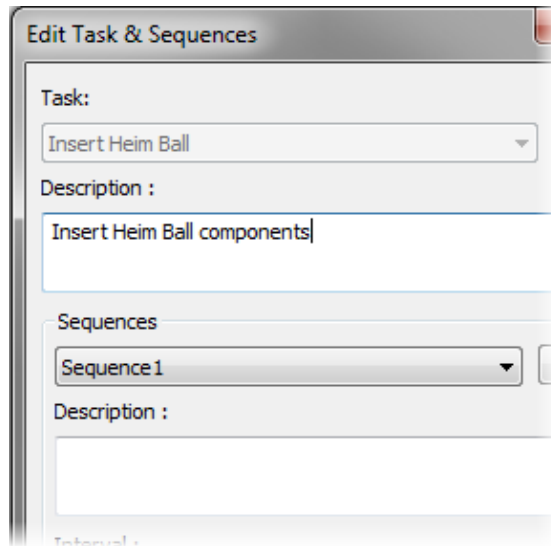
- Slowly double-click Task1 in the browser.
- Rename it to Insert Heim Ball.
- Rename Task2 to Attach Jam Nut.
- Rename Task3 to Attach Female End.



15. To edit a task or sequence:

- In the browser, right-click Insert Heim Ball. Click Edit.

- In the Edit Task & Sequences dialog box, under Tasks > Description, enter Insert Heim Ball components.
- Click Apply.



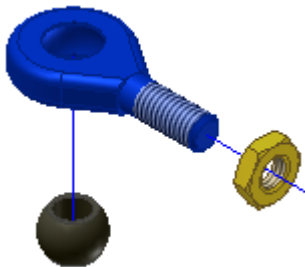
16. You can specify specific camera angles to be set before a sequence is animated. With the Edit Task & Sequences dialog box still open, and Insert Heim Ball selected for the Task and Sequence 1 selected from the Sequences list:

- In canvas, right-click. Click Home View.
- In the Edit Task & Sequences dialog box, click Set Camera.
- Click Apply.

Note: This will specify that the presentation will begin from the Home View.

17. To add additional camera angles and task descriptions:

- Under Tasks, select Attach Jam Nut.
- In the Task > Description field, enter Attach Jam Nut to Male End.
- In canvas, zoom in on the Jam Nut and Male End components.
- In the Edit Task & Sequences dialog box, click Set Camera.
- Click Apply.



18. To specify settings for the next task:

- From the Task list, select Attach Female End.
- In the Task > Description field, enter Attach Female End to Male End.
- Click Apply.
- Right-click in canvas. Click Home View.
- Click Set Camera.
- Click Apply.
- Click OK.

19. Click the Save tool.

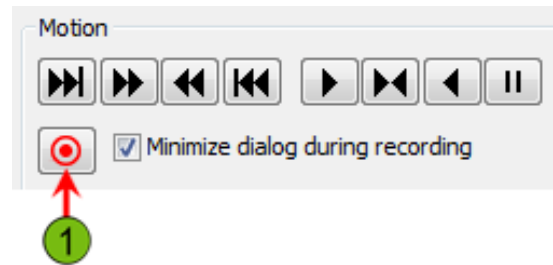
20. To animate the tweaks:

- In canvas, right click. Click Animate.
- Click the Play Forward button and watch the animation.
- Click Reset when the animation is finished.

Note: Notice the camera movements added to the presentation.

21. To create a WMV file from your presentation:

- Reduce the canvas to a size of your choice.
- Click the Record button in the Animation dialog box.



22. To save the animation:

- In the Save As dialog box, enter My\_LinkAdjust\_Presentation.
- For Save As Type, select WMV Files (\*.wmv).
- Click Save.
- In the ASF Export Properties dialog box, Network Bandwidth area, click BroadBand(250kbps)
- For Image Size, click Custom.
- Click OK.
- Click the Play Forward button.
- Click Reset when the Animation is complete and the dialog box is displayed.
- Click Cancel.

23. Save and close all files in Autodesk Inventor.

24. In Windows Explorer, navigate to the file location and double-click the created file to view the video.

## Conclusion

We hope you have enjoyed your 3D design experience. This test drive only scratches the surface of the full capabilities of Autodesk® Inventor®.

The Autodesk Inventor product line provides a comprehensive and flexible set of software for 3D mechanical design, simulation, tooling creation, and design communication that helps you cost effectively take advantage of a digital prototyping workflow to design, visualize and simulate your products digitally before they are ever built. Inventor allows engineers to integrate AutoCAD drawings and 3D data into a single digital model creating a virtual representation of the final product.

Move beyond 3D and develop a complete digital prototype of your product with Inventor.

For more details, please visit us online at [www.autodesk.com/inventor](http://www.autodesk.com/inventor).

- *The Autodesk Inventor Team*

Autodesk supports products through Autodesk Authorized Resellers, industry experts who offer consultation, training, support, and other related services to help meet your business needs. We encourage you to contact an Autodesk Authorized Reseller or your local Autodesk representative to learn how Autodesk Manufacturing Solutions can help you get to market faster and at a lower cost.

To find an Authorized Autodesk Reseller in your area, call 800-964-6432 (United States and Canada only) or visit our website at [www.autodesk.com/reseller](http://www.autodesk.com/reseller).

3DEC (design/logo), 3December, 3December.com, 3ds Max, ADI, Algor, Alias, Alias (swirl design/logo), AliasStudio, Alias|Wavefront (design/logo), ATC, AUGI, AutoCAD, AutoCAD Learning Assistance, AutoCAD LT, AutoCAD Simulator, AutoCAD SQL Extension, AutoCAD SQL Interface, Autodesk, Autodesk Envision, Autodesk Intent, Autodesk Inventor, Autodesk Map, Autodesk MapGuide, Autodesk Streamline, AutoLISP, AutoSnap, AutoSketch, AutoTrack, Backburner, Backdraft, Built with ObjectARX (logo), Burn, Buzzsaw, CAiCE, Can You Imagine, Character Studio, Cinestream, Civil 3D, Cleaner, Cleaner Central, ClearScale, Colour Warper, Combustion, Communication Specification, Constructware, Content Explorer, Create>what's>Next> (design/logo), Dancing Baby (image), DesignCenter, Design Doctor, Designer's Toolkit, DesignKids, DesignProf, DesignServer, DesignStudio, Design|Studio (design/logo), Design Web Format, Discreet, DWF, DWG, DWG (logo), DWG Extreme, DWG TrueConvert, DWG TrueView, DXF, Ecotect, Exposure, Extending the Design Team, Face Robot, FBX, Fempro, Filmbox, Fire, Flame, Flint, FMDesktop, Freewheel, Frost, GDX Driver, Gmax, Green Building Studio, Heads-up Design, Heidi, HumanIK, IDEA Server, i-drop, ImageModeler, iMOUT, Incinerator, Inferno, Inventor, Inventor LT, Kaydara, Kaydara (design/logo), Kynapse, Kynogon, LandXplorer, Lustre, MatchMover, Maya, Mechanical Desktop, Moldflow, Moonbox, MotionBuilder, Movimento, MPA, MPA (design/logo), Moldflow Plastics Advisers, MPI, Moldflow Plastics Insight, MPX, MPX (design/logo), Moldflow Plastics Xpert, Mudbox, Multi-Master Editing, NavisWorks, ObjectARX, ObjectDBX, Open Reality, Opticore, Opticore Opus, Pipeplus, PolarSnap, PortfolioWall, Powered with Autodesk Technology, Productstream, ProjectPoint, ProMaterials, RasterDWG, Reactor, RealDWG, Real-time Roto, REALVIZ, Recognize, Render Queue, Retimer, Reveal, Revit, Showcase, ShowMotion, SketchBook, Smoke, Softimage, Softimage|XSI (design/logo), Sparks, SteeringWheels, Stitcher, Stone, StudioTools, Topobase, Toxik, TrustedDWG, ViewCube, Visual, Visual Construction, Visual Drainage, Visual Landscape, Visual Survey, Visual Toolbox, Visual LISP, Voice Reality, Volo, Vtour, Wire, Wiretap, WiretapCentral, XSI, and XSI (design/logo) are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document. © 2011 Autodesk, Inc. All rights reserved.

