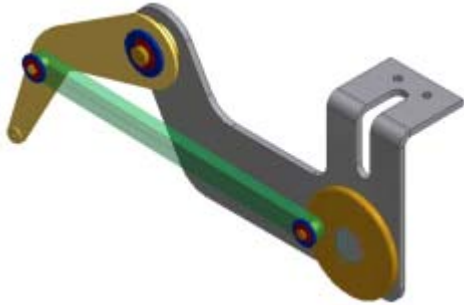


Quick Start for Autodesk Inventor

Autodesk Inventor® Professional is a 3D mechanical design tool with powerful solid modeling capabilities and an intuitive interface.

In this lesson, you use a typical workflow to create and document a small assembly.



Exercises

- Change the Active Project
- Create a New Part
- Assemble the Model
- Animate the Assembly
- Review a Presentation File
- Document the Assembly

Exercise - Change the Active Project

Engineers work on one or more projects at a time. To accommodate this, Autodesk Inventor uses project files to manage file locations.

Before starting the exercises, complete these steps:

Change the Active Project

1. Start Autodesk Inventor.

Note: If you are already in Autodesk Inventor, close all files.

2. On the Standard toolbar, click File menu > Projects.

3. Click Browse, and then navigate to the folder containing the exercise files.

4. Select *Autodesk Design Academy.ipj*, and click Open.

5. Click Done to close the dialog box.

The project file manages the location of existing files and the files you create.


Exercise 1 - Create a New Part

Create a new part file and sketch of a connecting link. You use the line, circle, dimension, and extrude tools.



Create the New Part

1. Start Autodesk Inventor. *Autodesk Design Academy.ipj* should be the active project.
2. On the Standard toolbar, click File menu > New.
3. In the New File dialog box, English tab, double-click *Standard (in).ipt*.

4.  On the 2D Sketch panel bar, click Line.
 - Click in the graphics window to start the line.
 - Move the cursor to the right, and then click again to set the endpoint of the line.
 - To create an arc, click and hold the end of the line, and then drag to preview the arc. When a vertical dotted line is displayed, release the mouse button to end the arc.
 - Move the cursor to the left until a vertical dotted line is displayed, and then click to create a line the same length as the bottom line.
 - Drag off the point to create an arc and close the profile.





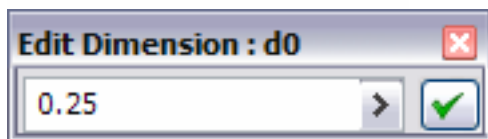
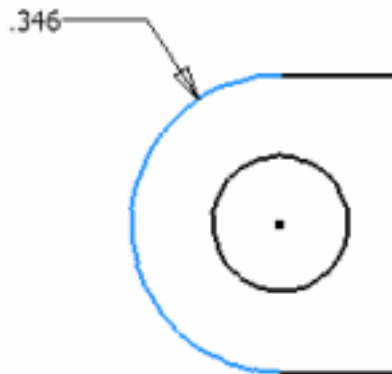
5. On the 2D Sketch panel bar, click Center Point Circle.

- On the left of the sketch, move the cursor over the center of the left arc. When a green dot displays, click to place the center point.
- Move the cursor to preview the circle radius, and then click to set.
- On the right of the sketch, click the center point, and then create another circle.



6. On the 2D Sketch panel bar, click General Dimension.

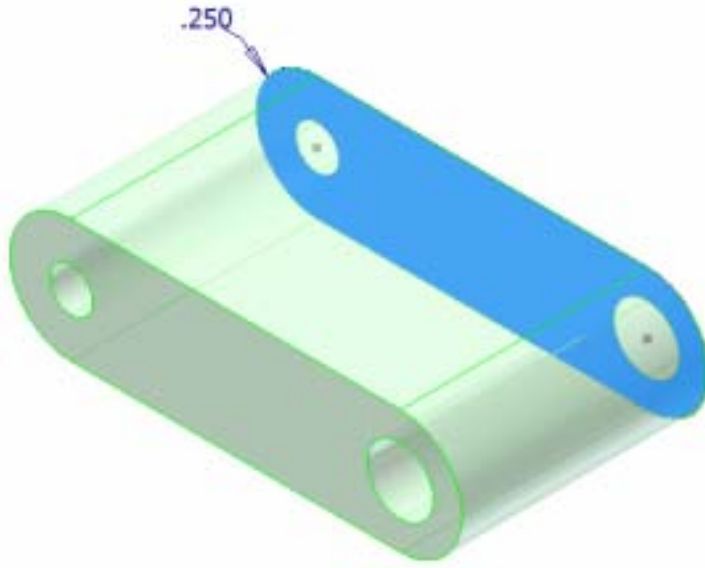
- On the left of the sketch, click the arc, and then drag to display the dimension.
- Click to place the dimension.
- Click the dimension value to edit it.
- In the Edit Dimension box, enter **0.25**, and then click the check mark.



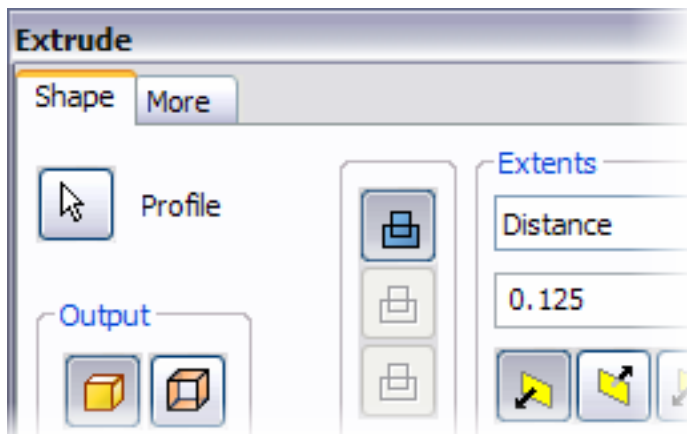
7. In the graphics window right-click, and select Isometric View.

8. In the graphics window right-click, and click Done.

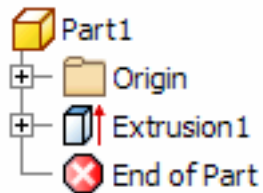
9. In the graphics window right-click, and click Create Feature > Extrude.
 - Select the profile to extrude. The holes should not be selected.



- In the Extrude dialog box, for Distance, enter **0.125**.

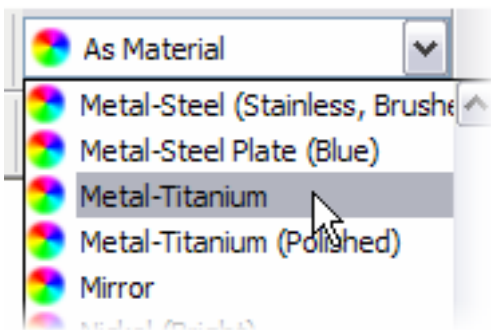


- Click OK. The extrusion is created and an entry is added to the browser.



10. On the Standard toolbar, select Metal-Titanium from the material list.

Note: The list may be hidden behind the exercise window.



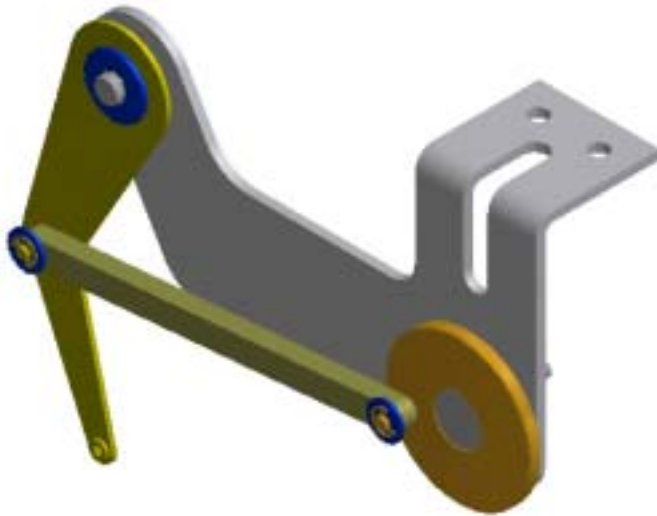
11.  On the Standard toolbar, click Save.

- In File Name, enter **Link**.
- Click Save.
- Close the file.



You created a new part file, sketch, and extrusion of a connecting link.

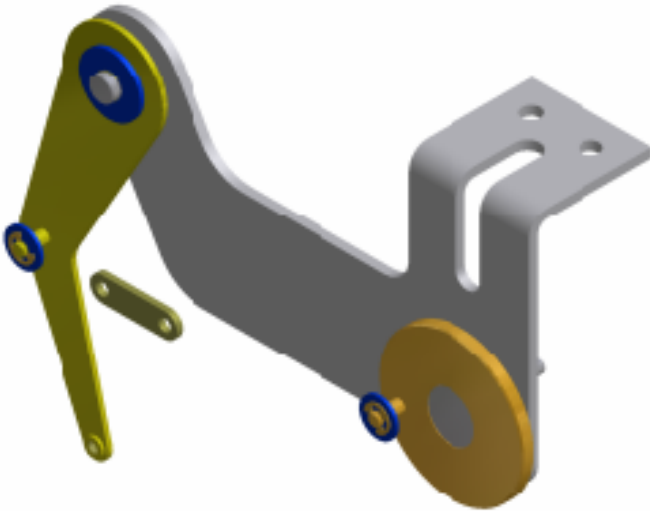
Exercise 2 - Assemble the Model


Open an existing assembly file and place the link in the assembly. You add assembly constraints to complete the linkage mechanism.



Assemble the Model

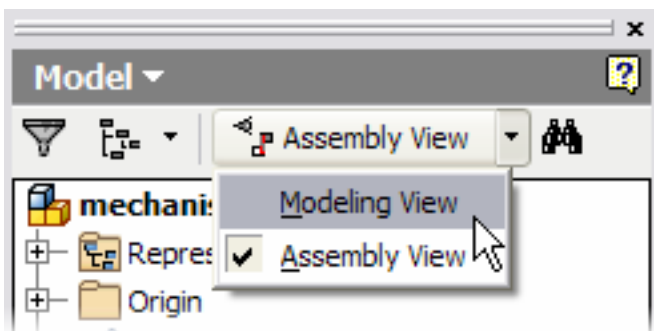
-  1. Open *mechanism.iam*.
-  2. On the Assembly panel bar, click Place Component.
 - Select *Link1.ipt*, and click Open. The link is displayed on the cursor.
 - Move the link to the left side of the assembly, and then click to place an occurrence of the link.
 - Right-click in the graphics window, and select Done.



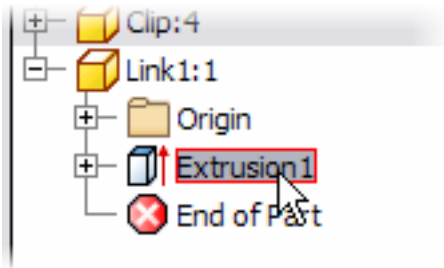
3.  On the Standard toolbar, click Zoom Window, and then define a window around the link and the lower pin.



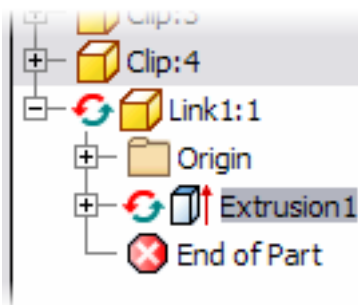
4. In the browser, Modeling View should be active. If not, click the arrow beside Assembly View, and then select Modeling View.



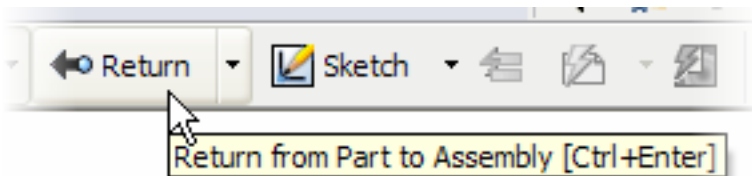
5. In the browser, right-click Link1:1, and select Edit.
 - Right-click Extrusion1, and select Adaptive.



The adaptive symbol is added to the link and extrusion entries in the browser.



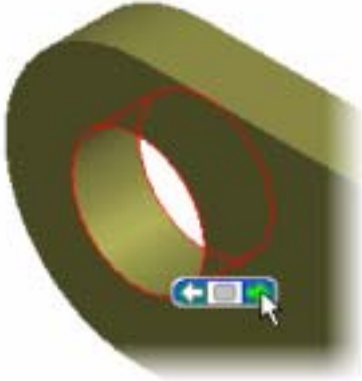
6. On the Standard toolbar, click Return from Part to Assembly.





7. On the Assembly panel bar, click Place Constraint.

- Select the inside cylindrical surface of the link. Make sure that the centerline is not displayed.



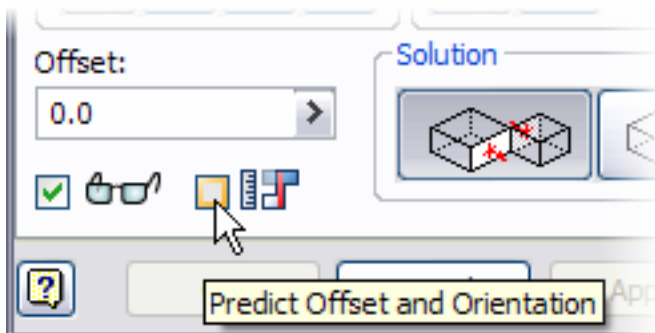
Note: If the inside surface is not displayed, hover the cursor over the part until the Select Other tool is displayed. Select one of the arrows to cycle through the options, and then click the center green square when the inside surface is highlighted.

8. Select the outside surface of the pin, and click Apply. The link moves onto the pin and the hole diameter adapts to the correct size.

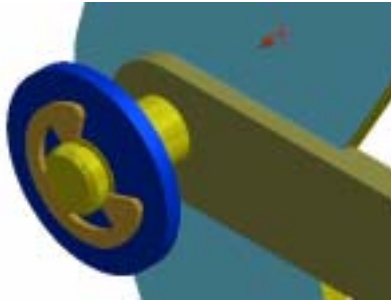


9. The face of the link has to be constrained to the arm.

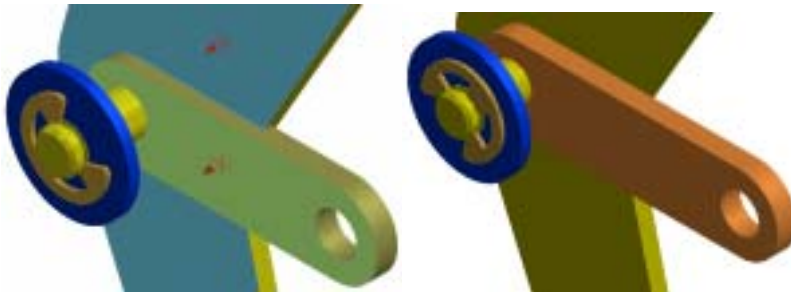
- Make sure there is no check mark in Predict Offset and Orientation.





- Select the front face of the arm.

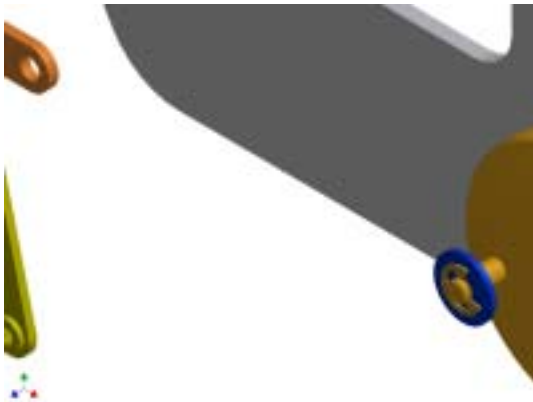


10. Select the front face of the link, click Apply, and then click Cancel.



11.  On the Standard toolbar, click Zoom All.

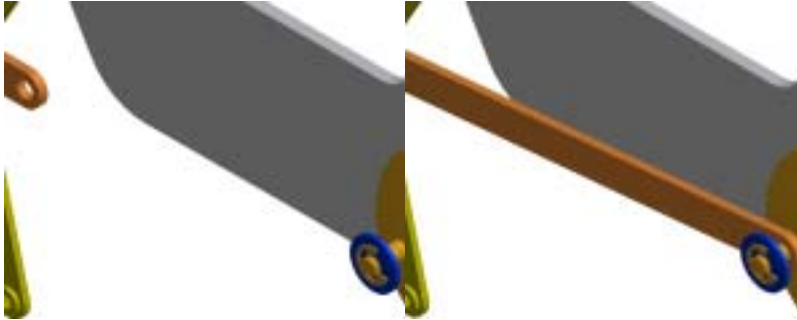
12.  On the Standard toolbar, click Zoom Window, and then define a window around the right end of the link and the pin on the eccentric.



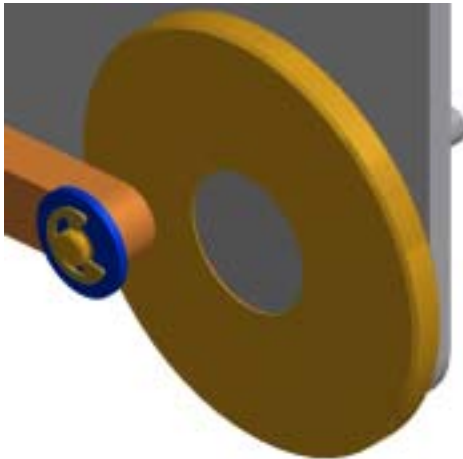


13. On the Assembly panel bar, click Place Constraint.

- Select the inside cylindrical surface of the link.
- Select the outside surface of the pin, and click Apply.
- Click Cancel.



14. Using the workflow learned in this lesson, place an assembly constraint between the front face of the link and the back face of the washer.



15. In the graphics window, right-click, and then select Isometric View.

16. Close the file without saving changes.

You placed a component in an assembly and placed assembly constraints to locate and size the component.

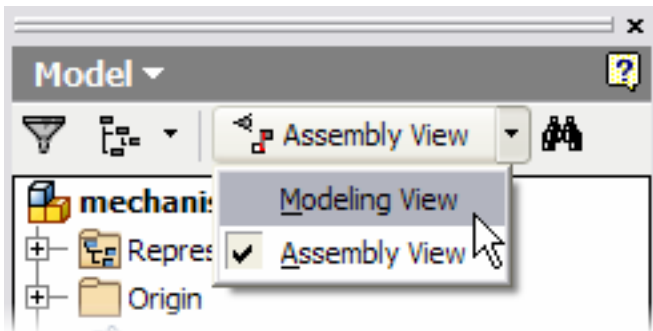
Exercise 3 - Animate the Assembly

Open an existing assembly file, and then animate the assembly by driving assembly constraints.

Animate the Assembly

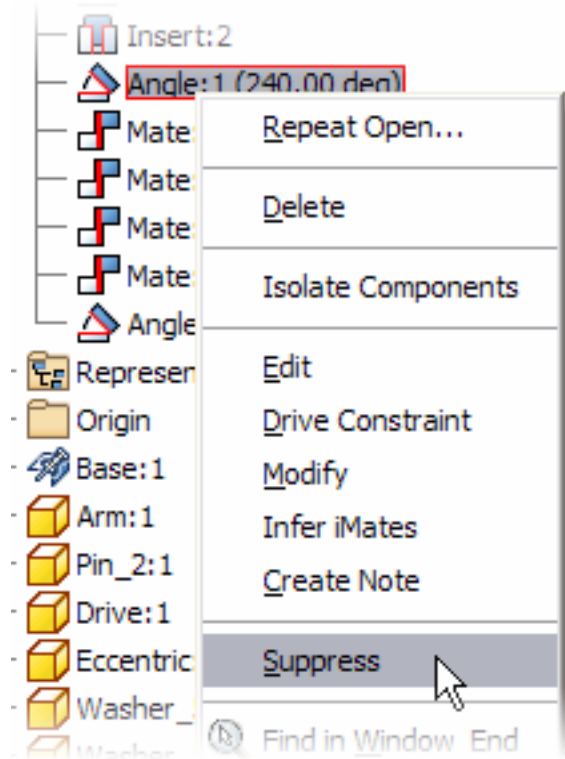
1.  Open *mechanism1.iam*.

2. On the browser, Modeling View should be active. If not, click the arrow beside Assembly View, and select Modeling View.

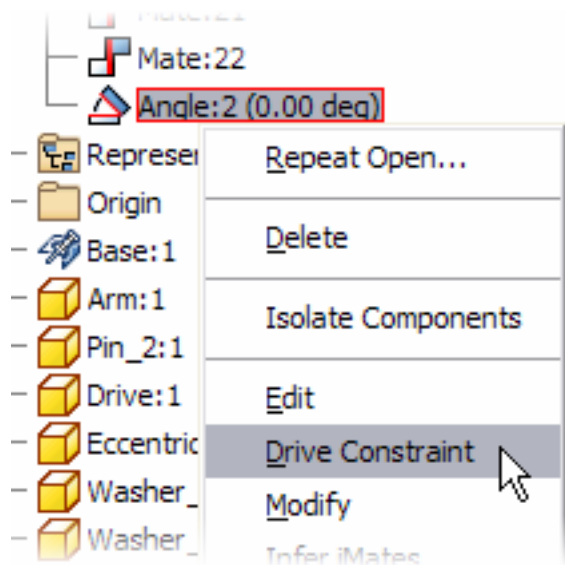


3. In the browser, click the plus (+) symbol next to Constraints to display all the assembly constraints.



4. Right-click the Angle:1 (240.00 deg) constraint, and select Suppress.



5. Right-click the Angle:2 (0.00 deg) constraint, and select Drive Constraint.



6. In the Drive Constraint dialog box:

- For End value, enter **360**.
-  Click More to expand the dialog box.
- For Increment value, enter **5**.
- For Repetitions value, enter **3**.
-  Click Forward to start the animation.



8. Close the file without saving changes.



You opened an existing assembly file, and then animated the assembly by driving assembly constraints.

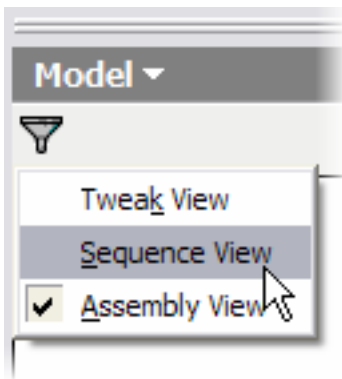
Exercise 4 - Review a Presentation File

Open an existing presentation file, and then review an animation of the file.

You can create exploded isometric views by adding tweaks to each part in the assembly. In this exercise, the tweaks are complete.

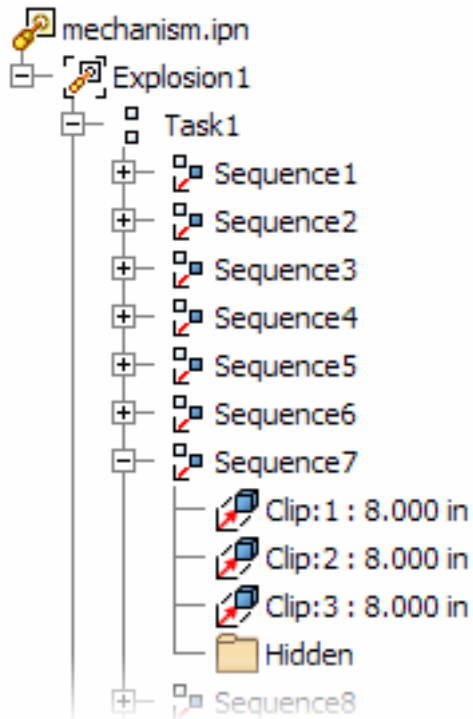
Review the Presentation File


-  Open *mechanism.ipn*.
-  On the browser, click Browser Filters > Sequence View.





- On the browser, expand Explosion1, and then Task1 to display the sequences.

4. Expand Sequence7. This sequence contains three of the clips in the assembly. The values are the tweak distances.



5.  On the Presentation panel bar, click the Animate tool.

-  In the Animation dialog box, click More.
-  Click Play Forward. As each sequence plays, the sequence is highlighted in the dialog box.
- When the animation is complete, close the dialog box.

6. When the exploded assembly is complete, you can create an AVI. For the purpose of this exercise, a sample animation file is provided for your review.



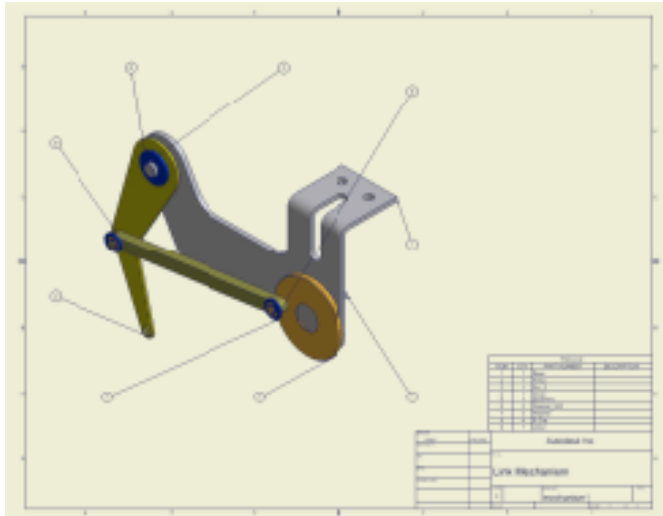
Show me an animation of this presentation file.

7. Close the file without saving changes.

Exercise 5 - Document the Assembly



Create a new drawing file, and then document the assembly.


You create two sheets. On the first sheet, you create an isometric view of the assembly, balloon all parts, and then create a parts list.



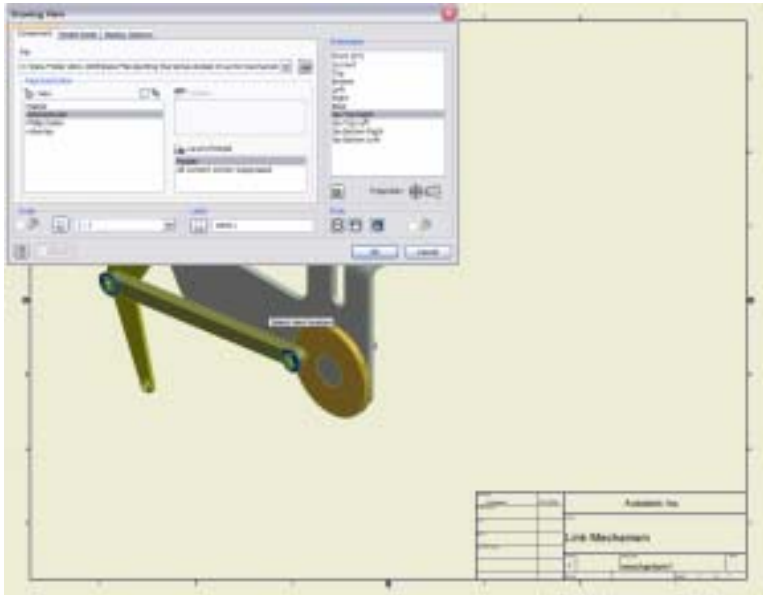
Show me an animation of this exercise.

Document the Assembly

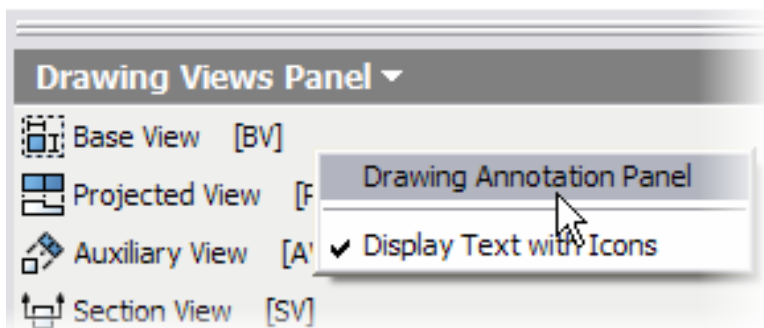
-  1. Open *mechanism1.idw*.
-  2. On the Drawing Views panel, click Base View.


3.  In the Drawing View dialog box, click Explore Directories.

- From the Files of Type list, select Assembly Files (*.iam).
- Select *mechanism2.iam*, and click Open.
- In Orientation, select Iso Top Right.
- In Style, click Shaded.
- Click in the left center of the graphics window to place the view.



4. Right-click on the Drawing Views Panel, and then select Drawing Annotation Panel.



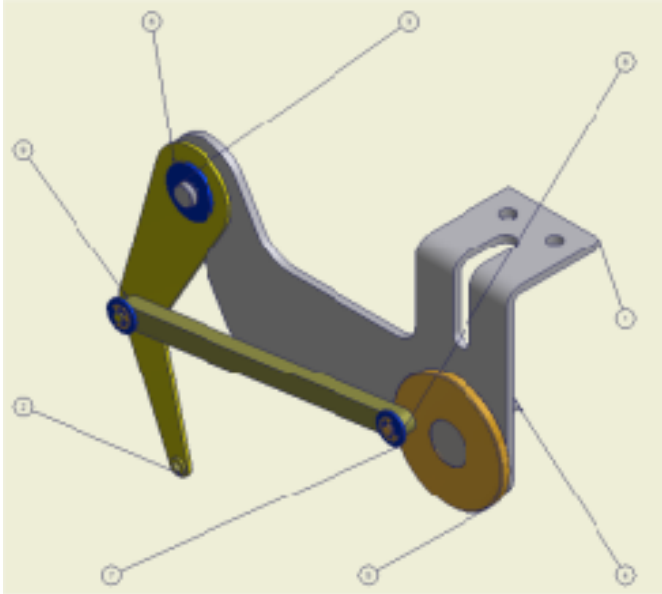
5.  On the Drawing Annotation Panel, click the down arrow next to Balloon > Auto Balloon.

- In the graphics window, select the isometric view.

Note: A red dotted line should be visible around the view.

- Right-click on the assembly, and select Select All.

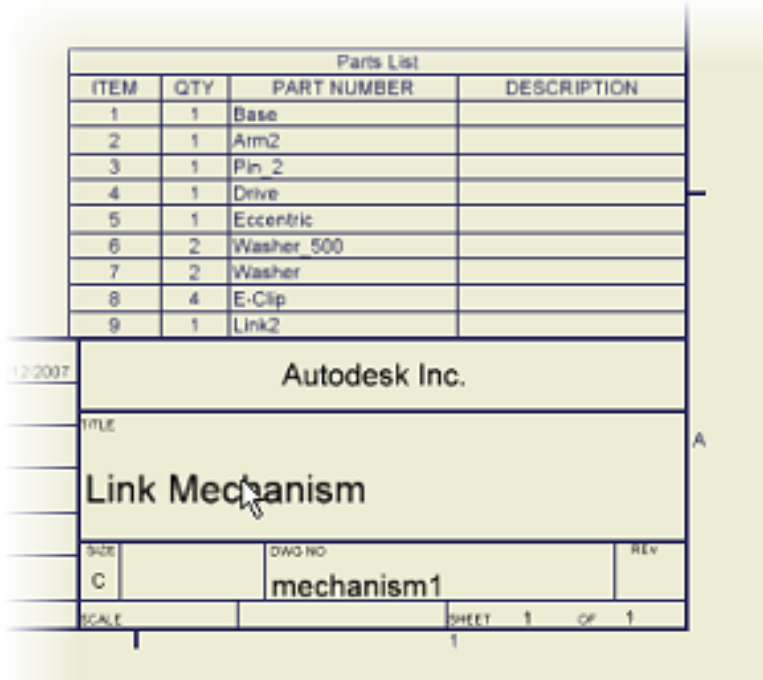
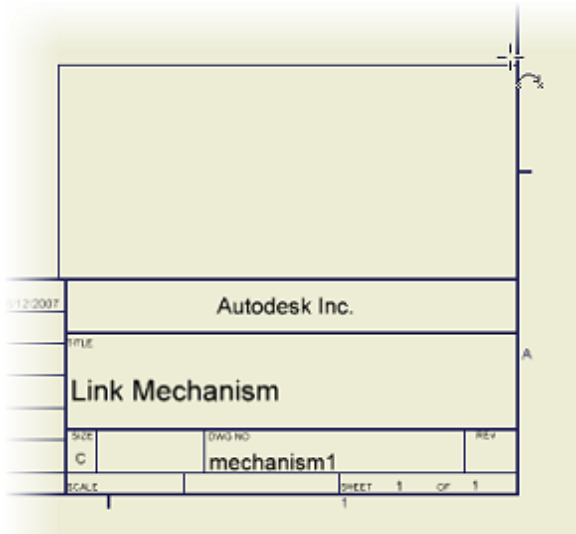
- Right-click again, and select Continue. A preview of the balloons is displayed.
- In Placement, select Around.
- Click in the graphics window.
- Click OK. Balloons are added to each part.





6. On the Drawing Annotation Panel, click Parts List.

- In the graphics window, select the isometric view.
- Click OK.
- Move the preview of the parts list above the title block, and click a point to locate the parts list.



You opened an existing drawing, created a base view, and added balloons and a parts list.

Create a New Sheet

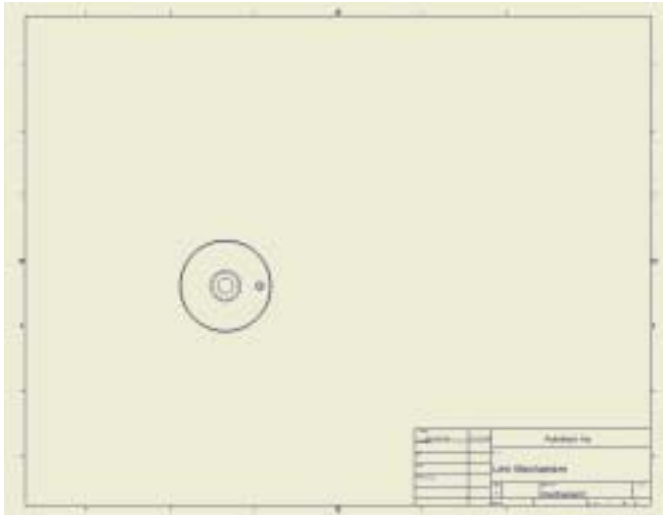
1. Right-click on Drawing Annotation panel, and select Drawing Views Panel.

2.  On the Drawing Views Panel, click New Sheet.

3.  On the Drawing Views Panel, click Base View.

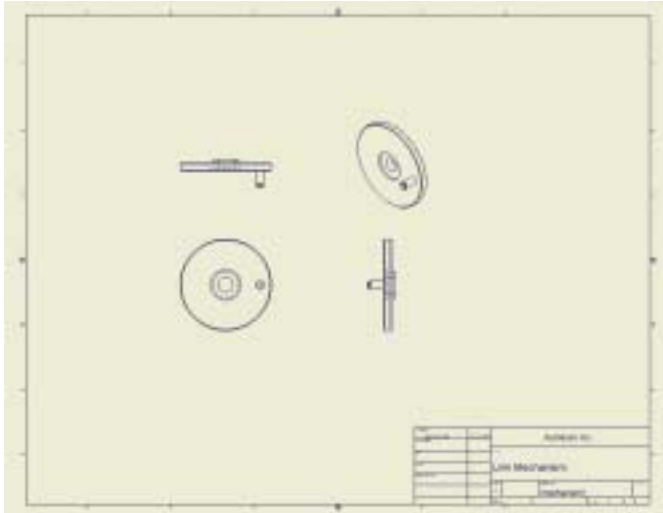
4.  In the Drawing View dialog box, click Explore Directories.

- In the Files of Type list, select Part Files (*.ipt).
- Select *Eccentric.ipt*, and click Open.
- In Orientation, select Back.
- Click in the left center of the graphics window to locate the view.



5.  On the Drawing Views Panel, click Projected View.

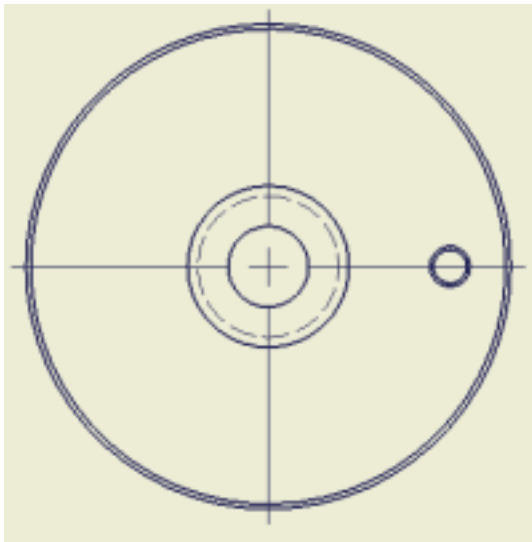
- In the graphics window, select the base view.
- Move the cursor vertically to a point above the base view, and then click to place the view.
- Move the cursor horizontally to the right of the base view, and then click to place the view.
- Move the cursor diagonally to a point above and to the right of the base view, and then click to place the isometric view.
- Right-click in the graphics window, and select Create.



6. Zoom in to the front view.
7. Right-click on the panel, and select Drawing Annotation Panel.



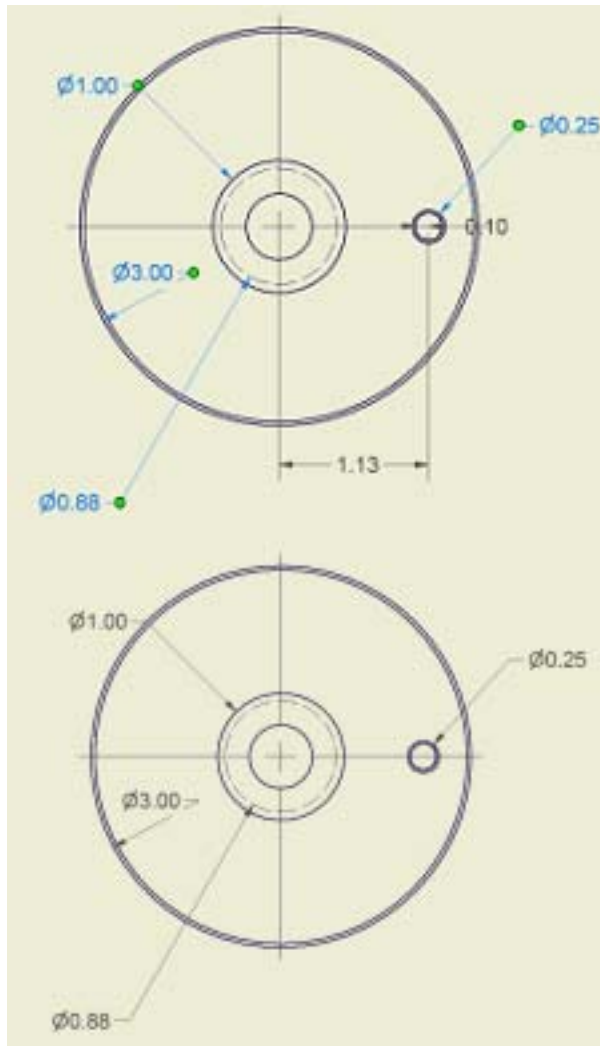
8. On the Drawing Annotation panel, click Center Mark.
9. Click the outside circle.



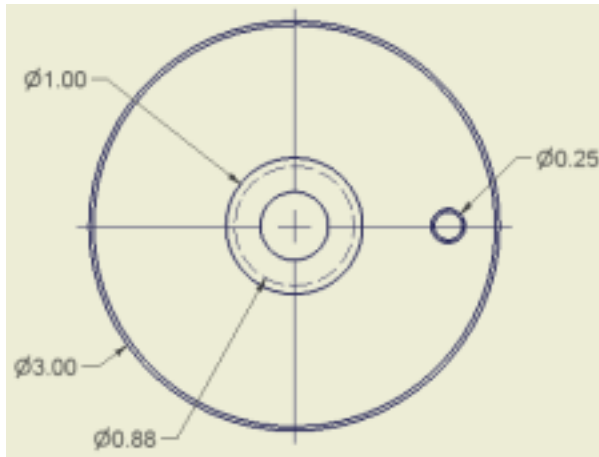


10. On the Drawing Annotation Panel, click Retrieve Dimensions.

- Select the front view.
- Click Select Dimensions.
- Select the four dimensions as shown, and then click OK.



11. Click on a dimension, and drag the green dot to reposition the dimension.



12. Add centerlines and retrieve the required model dimensions for the top and right side views. If necessary, reposition or delete dimensions.

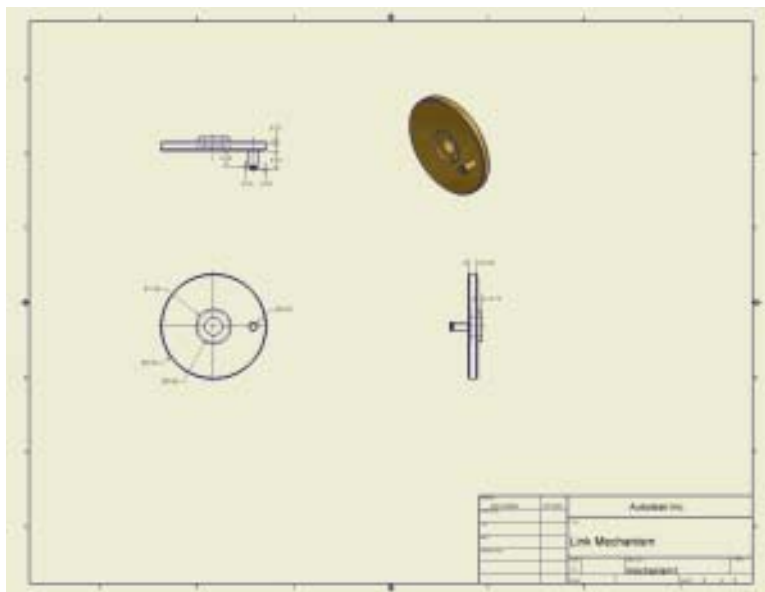
Note: To add centerlines between parallel lines, click the arrow next to the active center marking button, and then select Centerline Bisector from the pop-up menu.

13. Zoom into the isometric view.

- Right-click the isometric view, and then click Edit View.
- In Style, click Shaded, and click OK.



- On the Standard toolbar, click Zoom All.



14. Close the file without saving changes.

Note: You can use this workflow to create a complete set of drawings. Create a new sheet for each part in the assembly.

You added a new sheet, inserted drawing resources, and created orthographic views of one part.