

Product Design Suite 2013

Documentation and Data Management

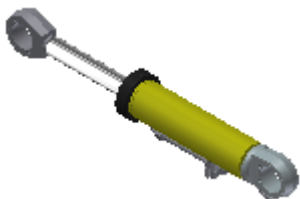
Exercise: Model Based Documentation

In this exercise, you create drawing documentation in AutoCAD Mechanical of an Autodesk Inventor assembly. After creating the views and adding dimensions, you then modify the assembly and a part and update the drawing.

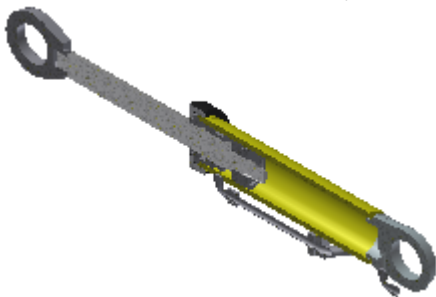
To complete the steps in this hands-on test drive you will need Autodesk® Product Design Suite 2013. If you are not currently a user, [click here](#) to download your free, 30-day trial of Autodesk® Product Design Suite Ultimate 2013.

For this exercise, make sure that *PDS2013-Documentation.ipj* is set as the active project in Inventor prior to starting the steps.

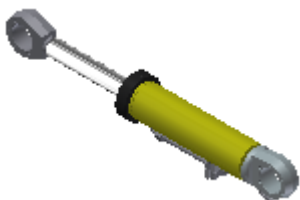
1. In Autodesk Inventor, open *_Cylinder PK46.14.01.000.iam*.



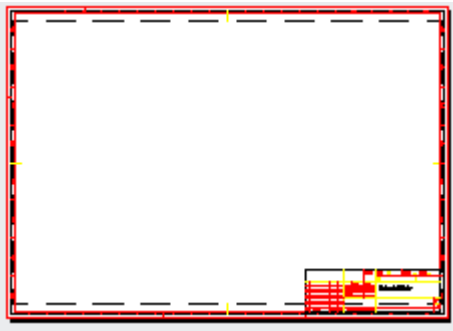
2. In canvas, orbit the assembly to review the visible geometry in this design.
3. To review the internal aspects of this design:
 - On the View tab, Appearance panel, click Quarter Section View > Half Section View.
 - In the browser, under Origin, click XY Plane.
 - On the in-canvas toolbar, click Continue.



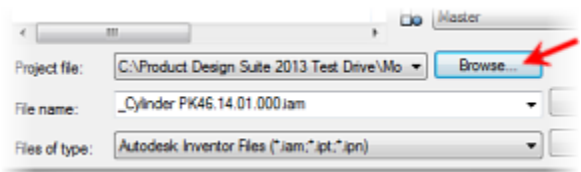
4. On the View tab, Appearance panel, click Half Section View > End Section View.



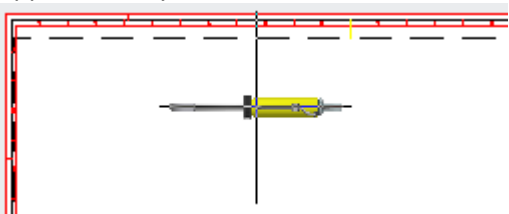
5. In AutoCAD Mechanical, open *PK46.14.001.000.dwg*.



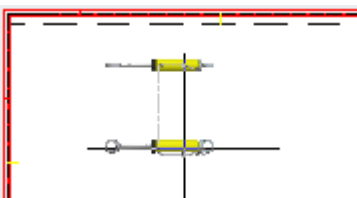
6. To begin to create a base view of the Inventor assembly in the AutoCAD Mechanical drawing file, on the Layout tab, Create View panel, click Base > From Inventor.
7. In the Select File dialog box:
- To the right of the Project File list, click Browse.
 - Navigate to and open *PDS2013-Documentation.ipj*.
 - Select and open *_Cylinder PK46.14.01.000.iam*.



8. For the location of the base view, click in canvas approximately as shown.



9. On the shortcut menu, click Exit.
10. To specify the location of a projected view from this base view, click in canvas approximately as shown.



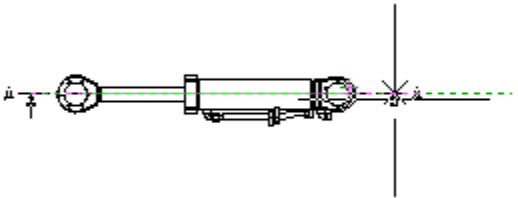
11. Right-click. Click Exit.



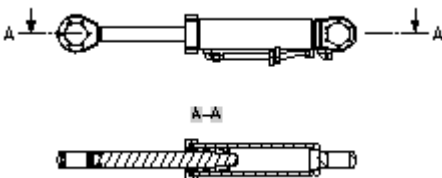
12. To begin to create a full section of the assembly relative to the existing projected view, on the Create View panel, click Section.
13. When prompted to select the parent view, in canvas, select the projected view.



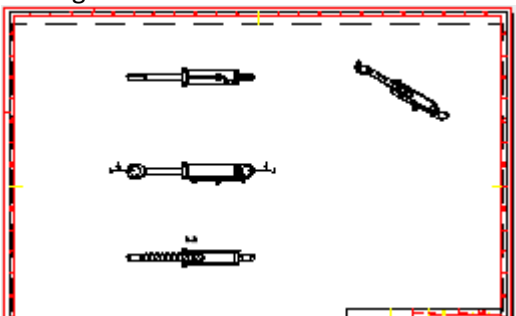
14. For the location of the section view's cutting plane line:
 - When prompted to specify the start point, click a point to the left of the view by tracking to the left relative to the circle center.
 - When prompted for the next point, click a point directly to the right past the view.
 - Right-click. Click Enter.



15. For the location of the section view, click a point directly below the base view. Click Exit.

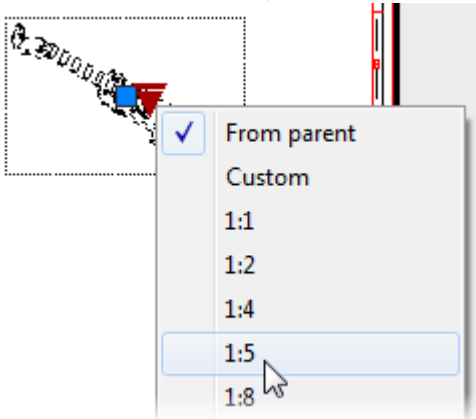


16. To create a projected pictorial view from the section view:
 - On the Create View panel, click Projected.
 - In canvas, select the section view.
 - Click a point in the upper right corner of the sheet.
 - Right-click. Click Enter.



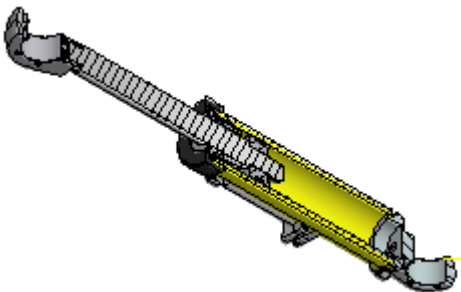
17. To increase the drawing size of the pictorial view:

- Select the view to activate its grips.
- Click the down arrow grip.
- In the list of scales, select 1:5.



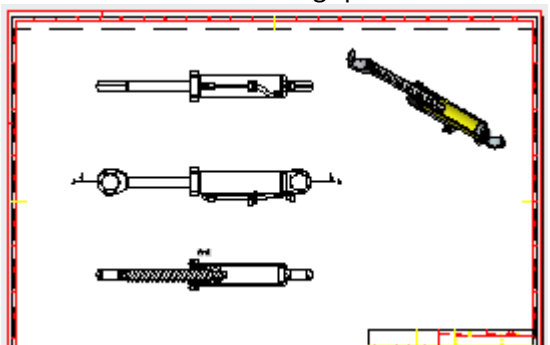
18. To change the pictorial view so it displays shaded:

- Double-click the pictorial view. Click Hidden Lines.
- Click Shaded with Visible Lines.
- Click Exit.



19. To change the drawing size of the base view:

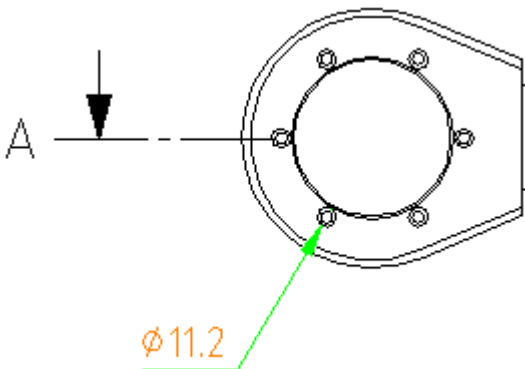
- Select the first view that you created to activate its grips.
- Click the down arrow grip.
- In the list of scales, select 1:5.
- Press ESC to clear the grips.



20. To begin to add dimensions to a view, on the Annotate tab, Dimension panel, click Power Dimension.

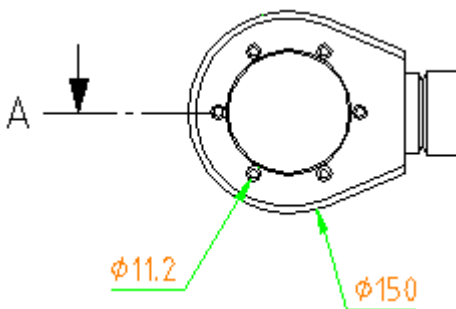
21. To create the first dimension:

- Press SPACEBAR.
- Select the edge of the hole.
- Click to place the dimension.
- Press ENTER.



22. To add a second diameter dimension:

- Right-click. Click Radial.
- Click Diameter.
- Select the outer arc edge.
- Click to place the dimension.

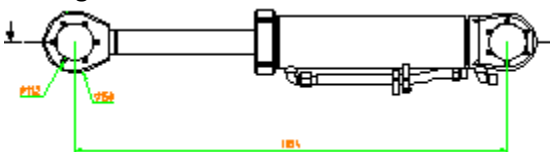


23. To switch from adding a diameter dimension to a linear dimension:

- Right-click. Click Exit.
- Click Exit.

24. To add a linear dimension between the piston and base pin centers:

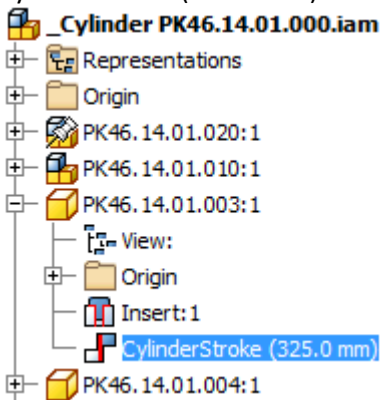
- Object snap to the center of the left circle.
- Object snap to the center of the right circle.
- Click to place the dimension.
- Right-click. Click Enter
- Right-click.



25. Save and close *PK46.14.001.000.dwg*.

26. Switch to Inventor and the file *_Cylinder PK46.14.01.000.iam*.

27. In the browser, under PK46.14.01.003, select CylinderStroke (325.0 mm).



28. To change the position of the piston:

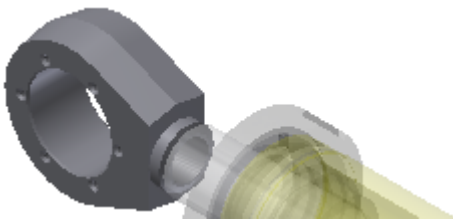
- In the value field at the bottom of the browser, enter **50**.
- Click in an open area of the canvas to clear the selection of the assembly constraint.



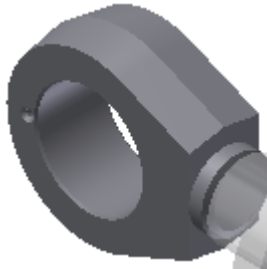
29. In canvas, double-click the eye on the end of the piston. The piston subassembly is now active for in-place editing.



30. In canvas, double-click the eye part again. This part is now active for in-place editing.



31. In the browser, click and drag End of Part above the Circular Pattern 1 feature. The part now displays as shown.



32. On the 3D Model tab, Exit panel, click Return > Return to Top.

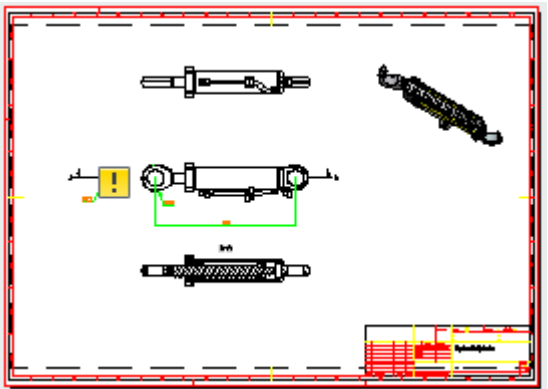
33. On the Quick Access toolbar, click Save.

34. In the Save dialog box:

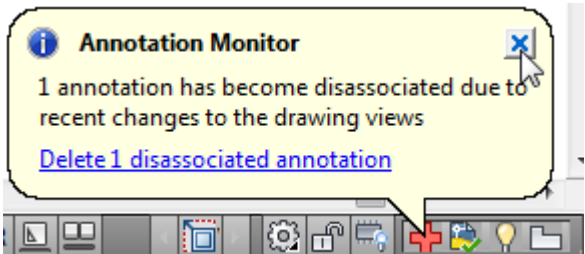
- Click Yes to All.
- Click OK.

35. Switch to AutoCAD Mechanical.

36. Open *PK46.14.001.000.dwg*. The views automatically updated because the Auto Update setting is selected in the drawing. This setting can be found on the Layout tab, Update panel.

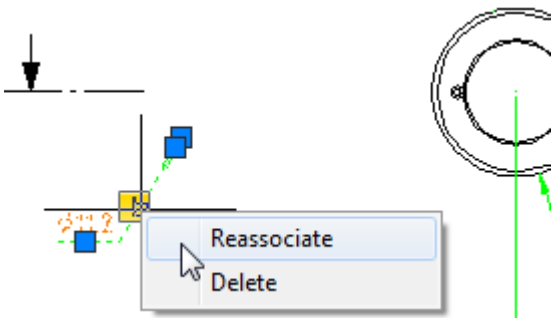


37. In the Annotation Monitor balloon, click Close.

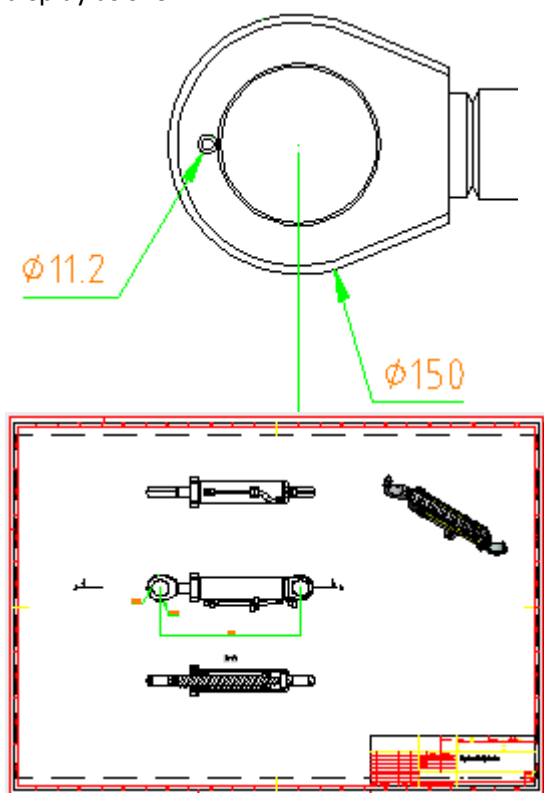


38. To begin to reassociate the diameter dimension that had been added to a hole that is no longer there to the hole that remains:

- In canvas, click the information glyph.
- Click Reassociate.



39. Select the circular edge of the hole. The results display as shown.



40. Close all files. Do not save changes.