MFG502370

A Link in Chain Explains Infinite Chain-Inventor Link in AutoCAD Mechanical

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Learning Objectives

- Learn how to link the Inventor assembly (.iam) and part (.ipt) files.
- Learn how to create, edit Inventor linked drawing views, and annotate them.
- Learn how to update an Inventor linked model and get to know about commands and system variables.
- Learn how annotation data is exchanged between Inventor and AutoCAD Mechanical

Description

More connection speed, more reaction time—interoperability plays a vital role in our daily design collaborations that help with flexible production lines and reduced integration costs. Many industrial machinery and manufacturing industries have realized the benefits of using both 2D and 3D CAD software together—and the list of reasons is long, including better collaboration and increased efficiency, to name a few. In those cases, it's important that your data remains fully associative. In all such scenarios, AutoCAD Mechanical software provides a feature called Inventor Link that enables you to link Inventor assembly (.iam) and part (.ipt) documents to a drawing file and display the 3D model in model space. Then, you can generate 2D drawings of the 3D model in paper space and annotate them. This class aims to show how you can use Inventor Link in AutoCAD Mechanical for increased efficiency, reduced time, and better collaboration.

Speakers:

Dheen Abdul Azeez is a Technical Support Specialist for AutoCAD products in Customer Success Organization. He handles Technical support for AutoCAD products thus by resolving customer major issues within the software. He did his Postgraduate in Central Institute of Tool Design and has worked as an Application Engineer for Autodesk Product Design and Manufacturing Collection.

Sridhar Subramani has more than 20 years of CAD experience, is a Senior Product Owner for AutoCAD Toolsets at Autodesk, Inc. He is a frequent presenter at Autodesk University for last 8 years. He engages with customer and users to understand the product pain points and needs. Prioritizes the wish list items through qualitative & quantitative research and builds the roadmap for future releases. He has also been actively involved in newsgroups of AutoCAD Mechanical software and AutoCAD Architecture software and resolved over 1500 issues reported by customers. He conducts in-house training in AutoCAD and AutoCAD Mechanical software. He has written several technical solutions that are published on the Autodesk support website

Vinod Kumar Balasubramanian is our Principal Specialist for AutoCAD & Toolsets in Customer Success Organization. He handles Strategic Prioritization of escalations through Customer First approach, drives adoption through Stake holder management, provides Insights to Engineering on Product Road Map and Feature improvements. His key responsibility is to remove customer barriers and enhance Product Utilization through qualitative & quantitative analysis of customer issues. His overall focus is on improving Product & Customer Experience for all our Customers using AutoCAD Family Products.

Table of Contents

Introduction

This case study is intended to let the AutoCAD users to know about "Inventor link" feature and its capability when linked with an Inventor .ipt or an .iam file. It is meant to focus on the basics of Inventor link feature and overview of how this feature works and helps users to leverage Inventor models in AutoCAD Mechanical.

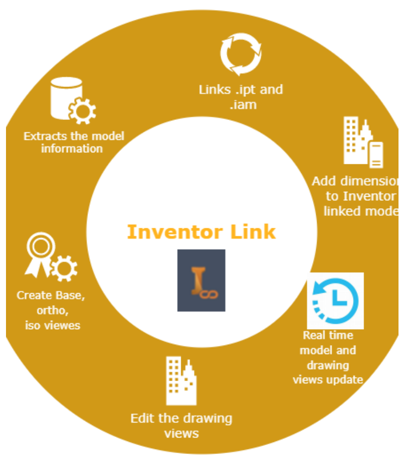
Inventor Link:

Inventor Link enables you to link Autodesk Inventor assembly (.iam) and part (.ipt) documents to a .dwg file and display the 3D model in model space. After that you can generate 2D drawings of the 3D model in paper space and annotate them.

The link between the .dwg file and the part file or assembly file is fully associative, i.e, Whenever you modify the part file or assembly file (in Autodesk Inventor), AutoCAD Mechanical toolset highlights the changed file. You can update the .dwg file by right-clicking the highlighted file in the browser and choosing the Update menu option and also you cannot modify the part file or assembly file from AutoCAD Mechanical toolset but only in Inventor.

Capabilities of Inventor Link

Inventor Link is available only on 64-bit systems, and it is not installed by default. There a many things Inventor Link is capable of such as real time update between Inventor model and the linked model, creating drawings views, editing the views, adding dimensions, using iProperties in part lists etc.



Significant Capabilities of Inventor link

How to install Inventor Link if not installed by default?

Inventor link is not installed by default. However, you can install Inventor link after you install AutoCAD Mechanical toolset.

1. Close all instances of AutoCAD Mechanical toolset.
2. From the Windows Control Panel select Programs > Uninstall a Program.
3. From the Programs and Features dialog box, select AutoCAD Mechanical 20XX and click the Uninstall/Change button.

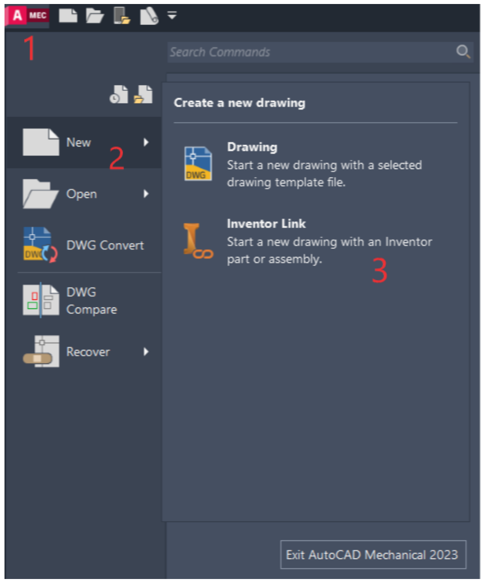
4. On the AutoCAD Mechanical toolset Installation Wizard, click Add or Remove Features.
5. Click Next until you arrive at the page containing the features to install.
6. Select Autodesk Inventor Link .
7. Click Update.

Now we are done with installing Inventor link, let us jump into completing our objectives for the same.

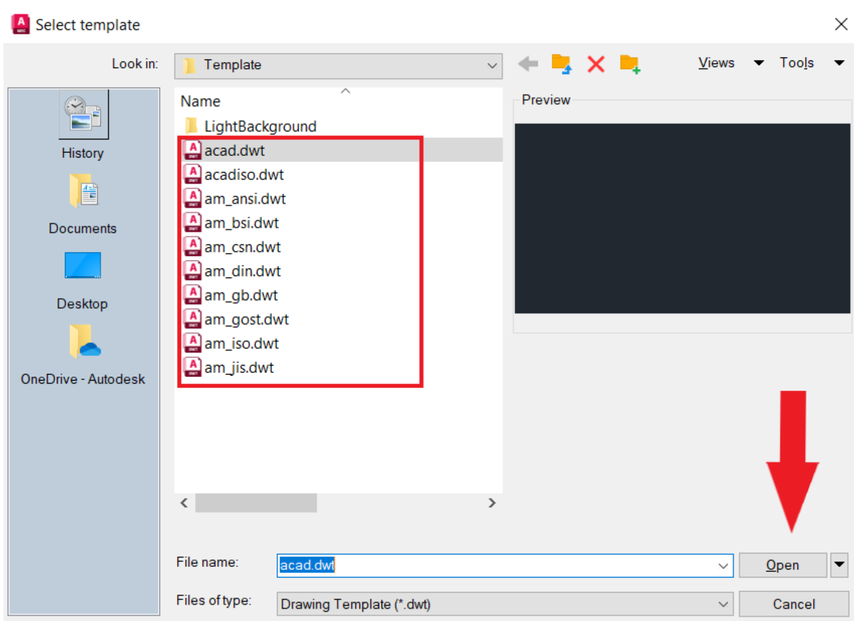
To link Inventor part(.ipt) or assembly(.iam) with AutoCAD Mechanical.

Link a part(.ipt) file with AutoCAD Mechanical.

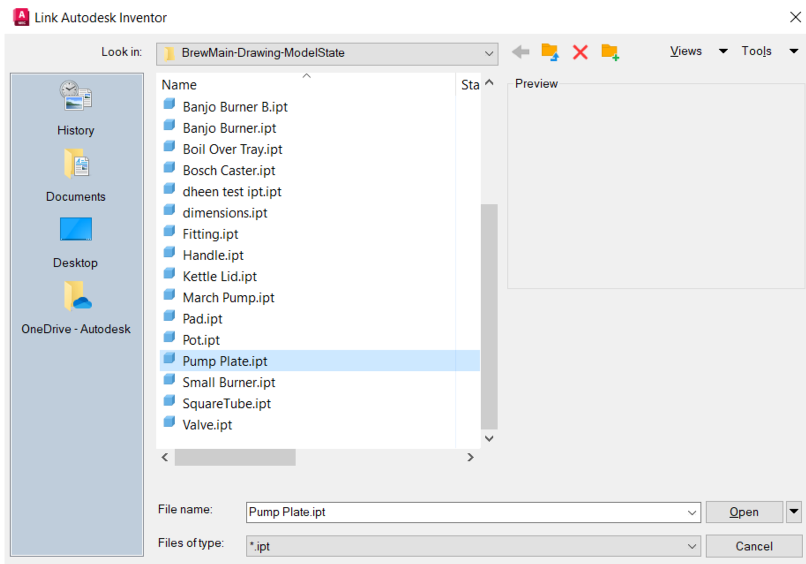
1. Click Application menu > New > Inventor Link.



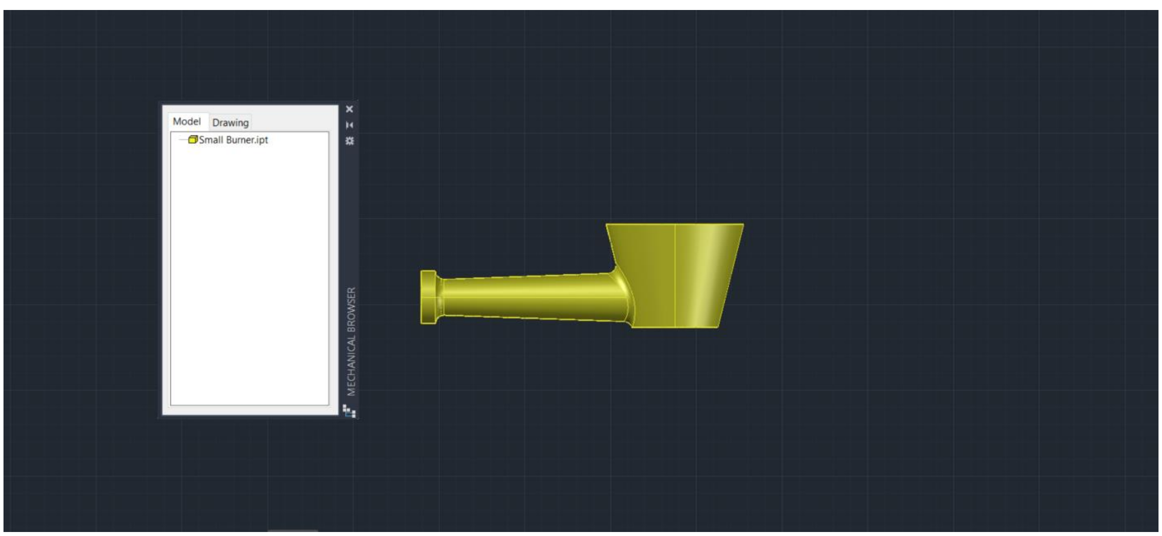
2. In the Select template dialog box, select a template.



3. In the Link Autodesk Inventor File dialog box,
    a) Navigate to the part file location
    b) Select the part(.ipt) file you wish to link.
    c) And, Click Open.

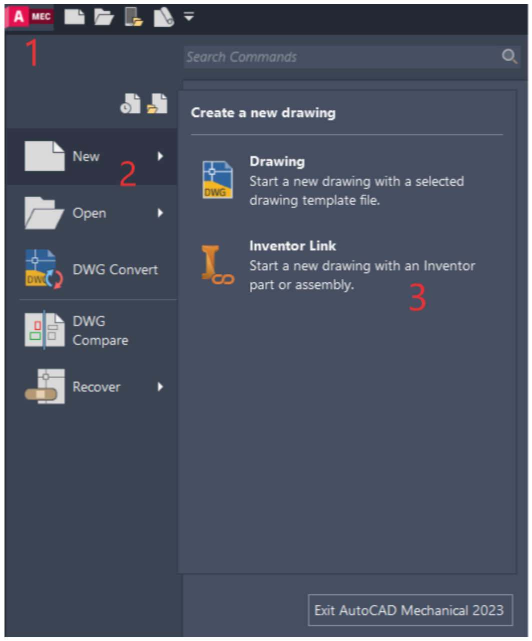


4. A new drawing file containing a 3D model is created as show below.

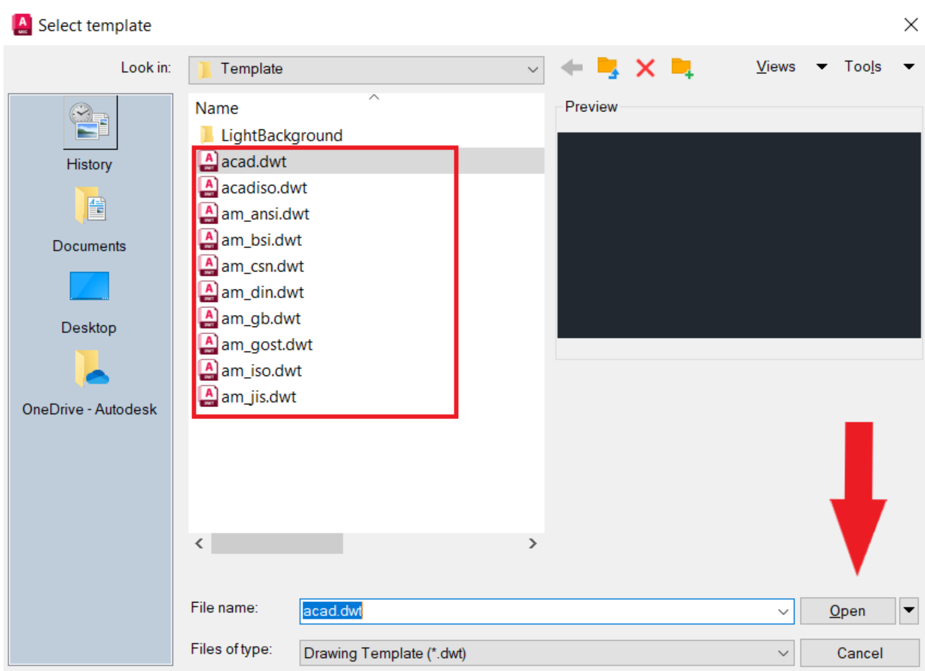


Link an assembly (.iam) file with AutoCAD Mechanical.

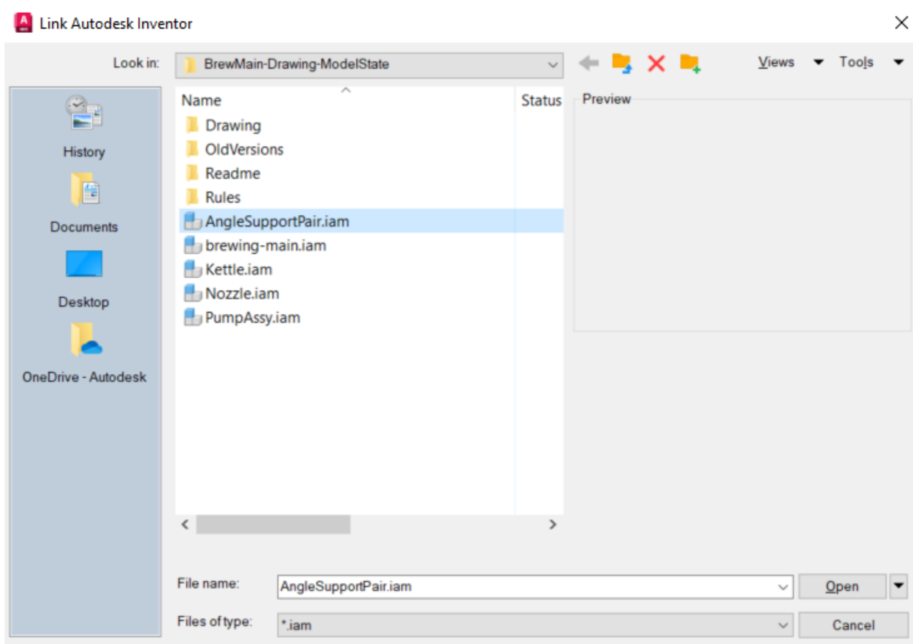
1. Click Application menu > New > Inventor Link.



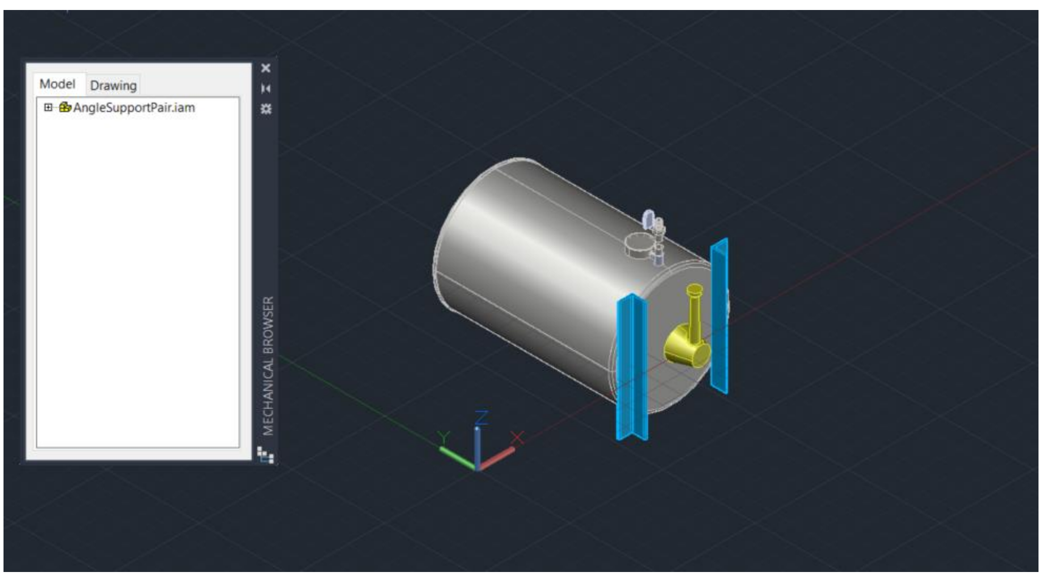
2. In the Select template dialog box, select a template.



3. In the Link Autodesk Inventor File dialog box,
    a. Navigate to the part file location
    b. Select the assemby(.iam) file you wish to link.
    c. And, Click Open.

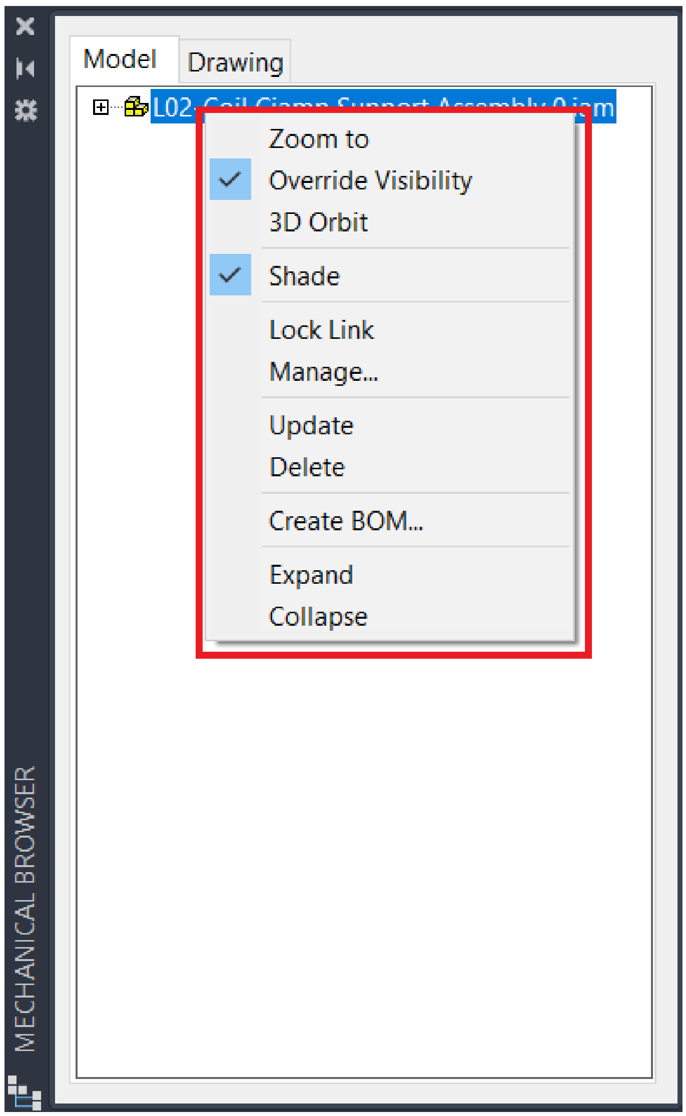


4. A new drawing file containing a 3D model is created as show below.



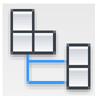
Manage your Inevntor linked model through Mechanical Browser.

By default, when you link an Inventor part or assembly with AutoCAD Mechanical, the Mechanical Browser is turned off. Using the Mechanical Browser, you can manage your inventor linked model such as updating the model, delete, create BOM, Zoom, toggling visibility, switch to drawing tab, create views etc., as shown below.



AMBROWSER (Mechanical Browser)

Displays or hides the mechanical browser



Summary

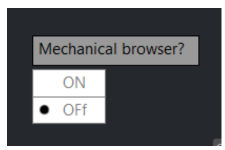
The Mechanical Browser shows the hierarchical organization of parts and assemblies so you can quickly identify mechanical structure objects as shown below. The right-click menu enables you to perform context sensitive operations on each object.

List of Prompts

The following prompts are displayed.

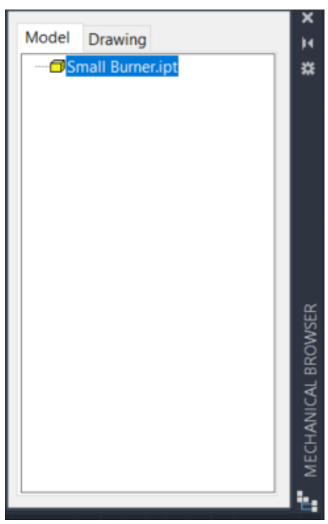
On

Displays the mechanical browser.



Off

Hides the mechanical browser.



Learn how to create, edit Inventor linked drawing views, and annotate them

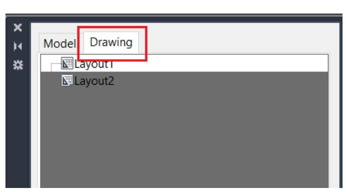
With your Inventor model linked to AutoCAD Mechanical, you can create drawing views from the linked model and can annotate them. You can also edit the Inventor linked drawing views any time. Let us dive deep into creating, editing drawings views.

About Creating Base, Orthographic, and Isometric Views

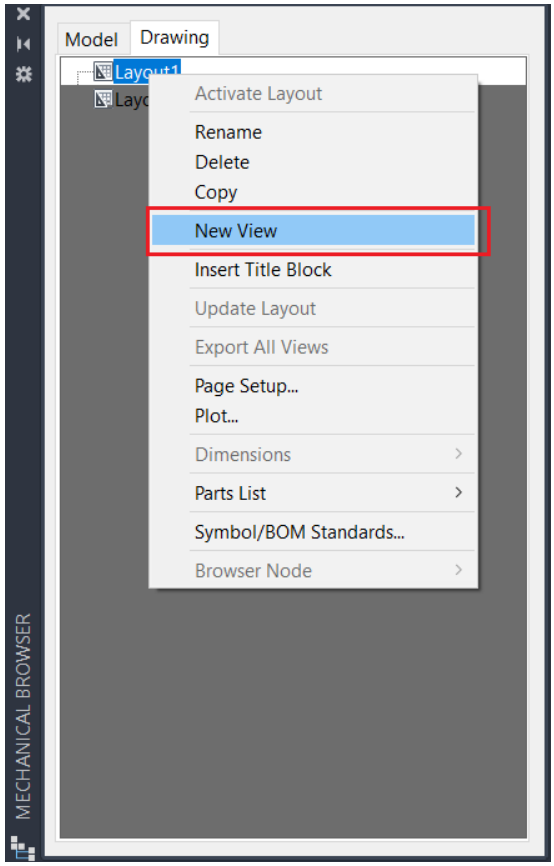
- You project drawing views from Inventor linked models to create drawings.
- You can create traditional 2D orthographic, isometric, auxiliary, section, and detail views of parts and assemblies, as well as exploded views. The program creates these views complete with dimensions derived from the models. Because the program derives the views from the models, it updates them automatically whenever you update the model.
- You can create a variety of drawing view types for a part or an assembly, but you must create a base view first.
- Every layout must have a base view so that dependent views can refer to it. In defining the base view, specify the orientation of the view and then position it on the layout.
- After you create the views, you can hide or unhide parametric dimensions and add annotations, reference dimensions, mechanical symbols, balloons, BOMs, and parts lists title blocks.

To Create a Base View from an Inventor Linked Model

1. In the Mechanical Browser, choose the Drawing tab.



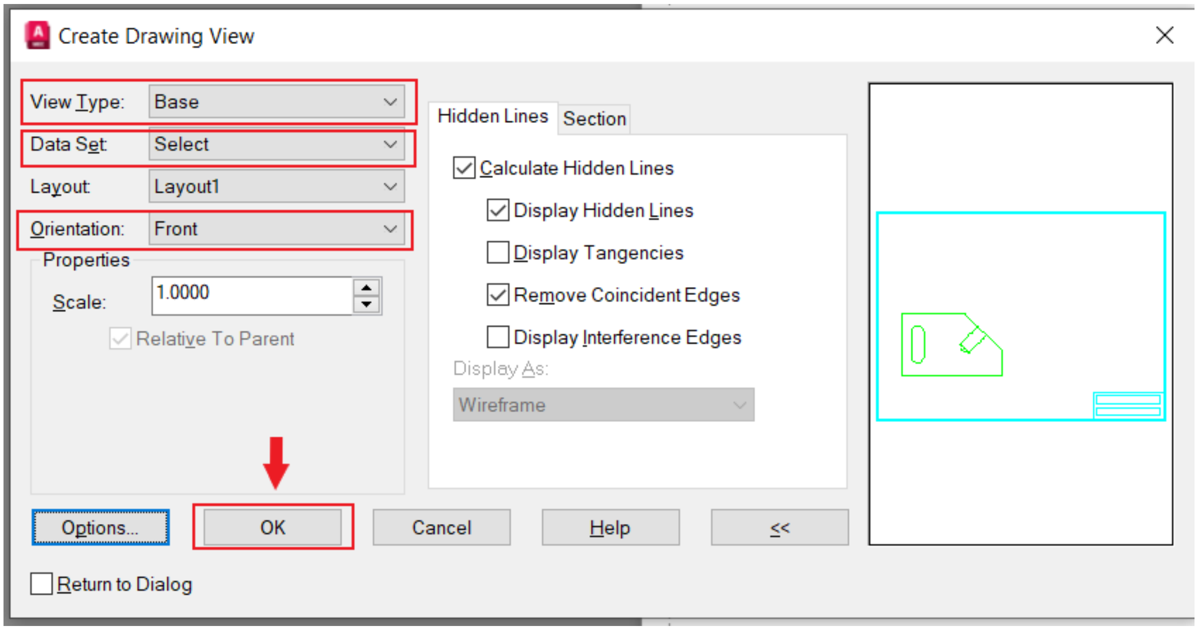
2. Double-click a Layout to make it the active layout.
3. Right-click the Layout icon, and then choose New View



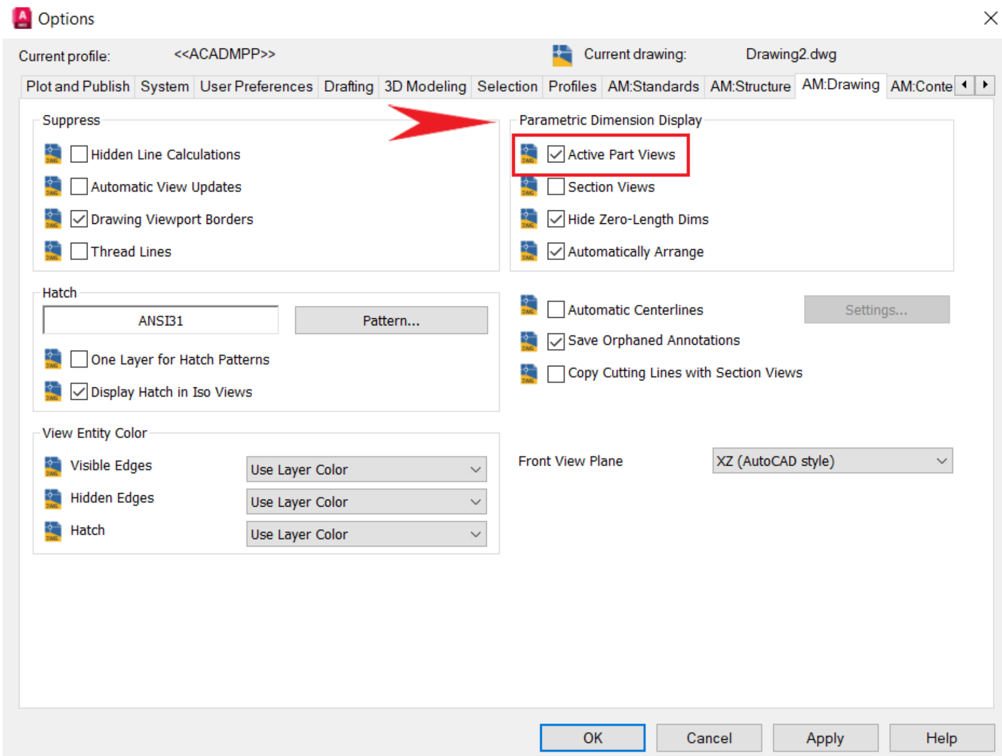
4. In the Create Drawing View dialog box,

- View Type: Base
- Data Set: Select
- Orientation: Front

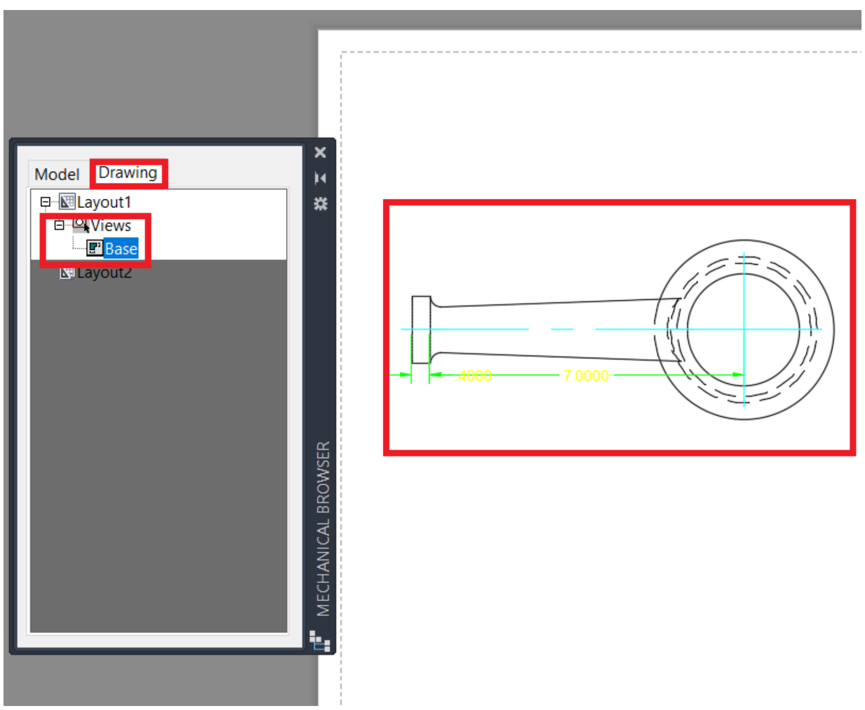
- Choose OK.



(**Note:** if you wish to generate parametric dimensions for this base view, click options and verify that the Active Part Views option in the Parametric Dimensions Display section is selected) as shown below:

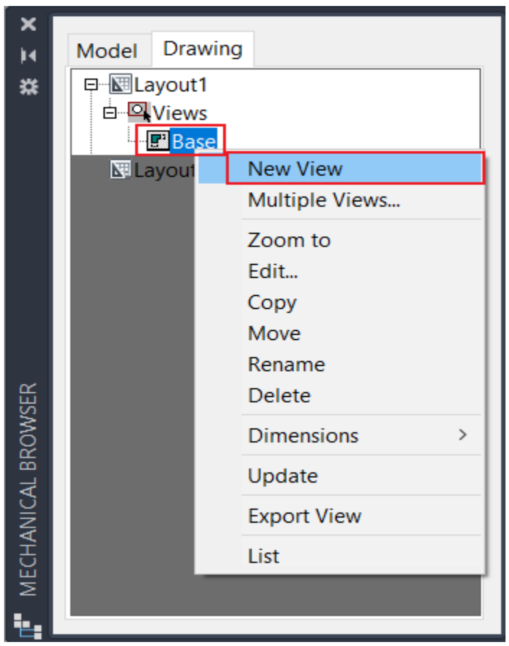


5. In the upper-left part of the drawing area, click a location for the base view, and then press ENTER and now you can see the base view for the linked model is created along with the name of the view in the Mechanical Brower hierarchy under drawing tab.

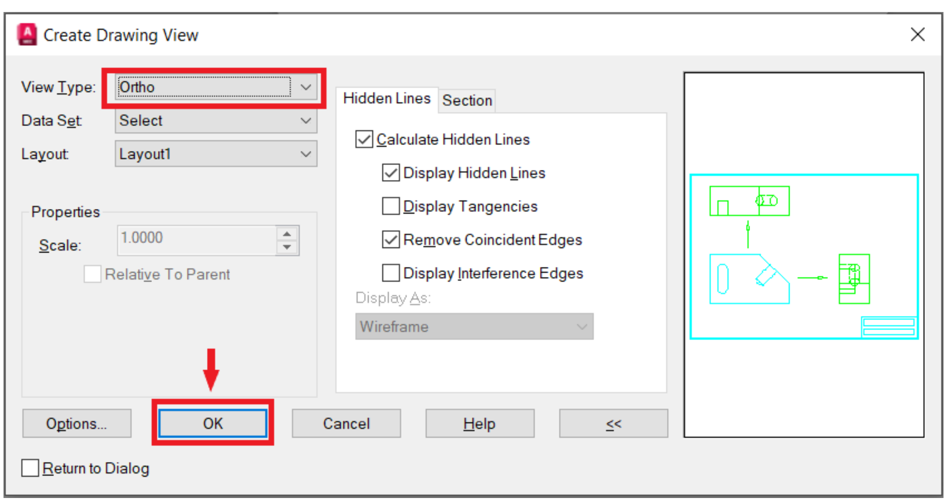


To Create an Orthographic View from an Inventor Linked Model

1. In the Browser, right-click the Base view icon, and then choose New View.

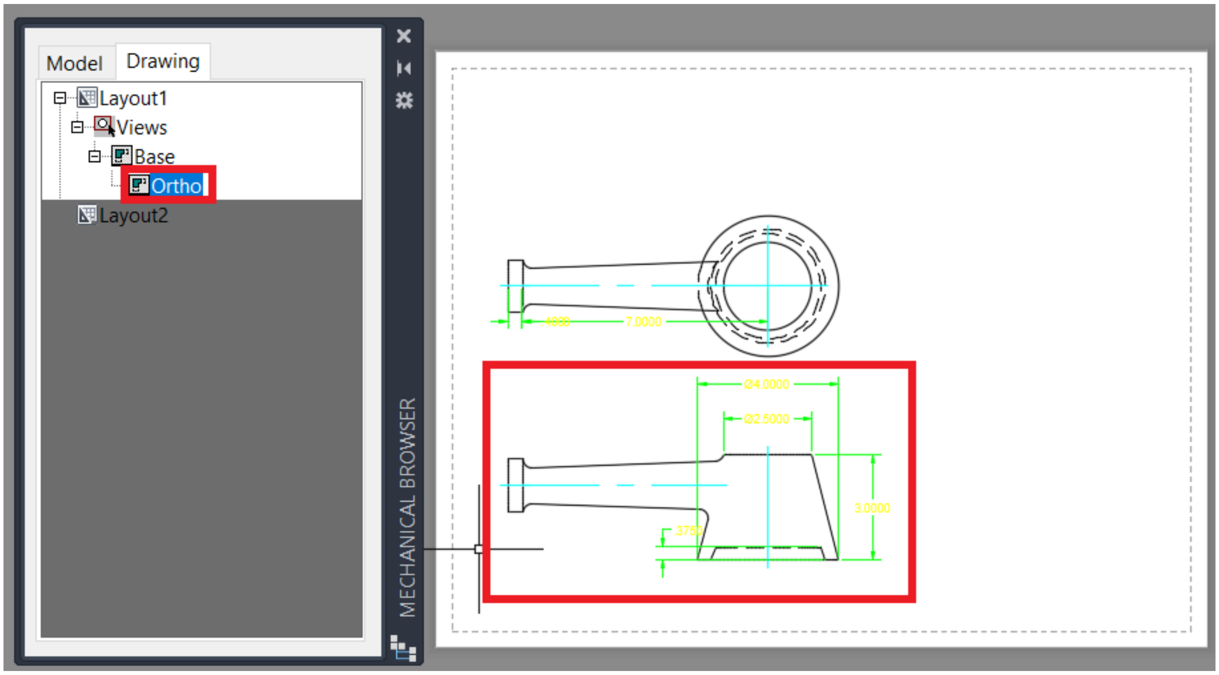


2. In the Create Drawing View dialog box, select the view type Ortho, and choose OK



3. Select a location for the

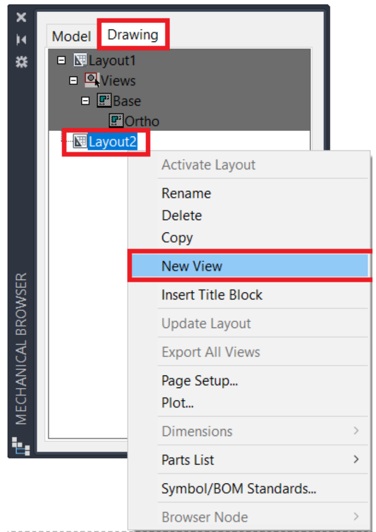
orthographic view directly below the base view, and then press ENTER and and now you can see the ortho view for the linked model is created along with the name of the view in the Mechanical Brower hierarchy under drawing tab



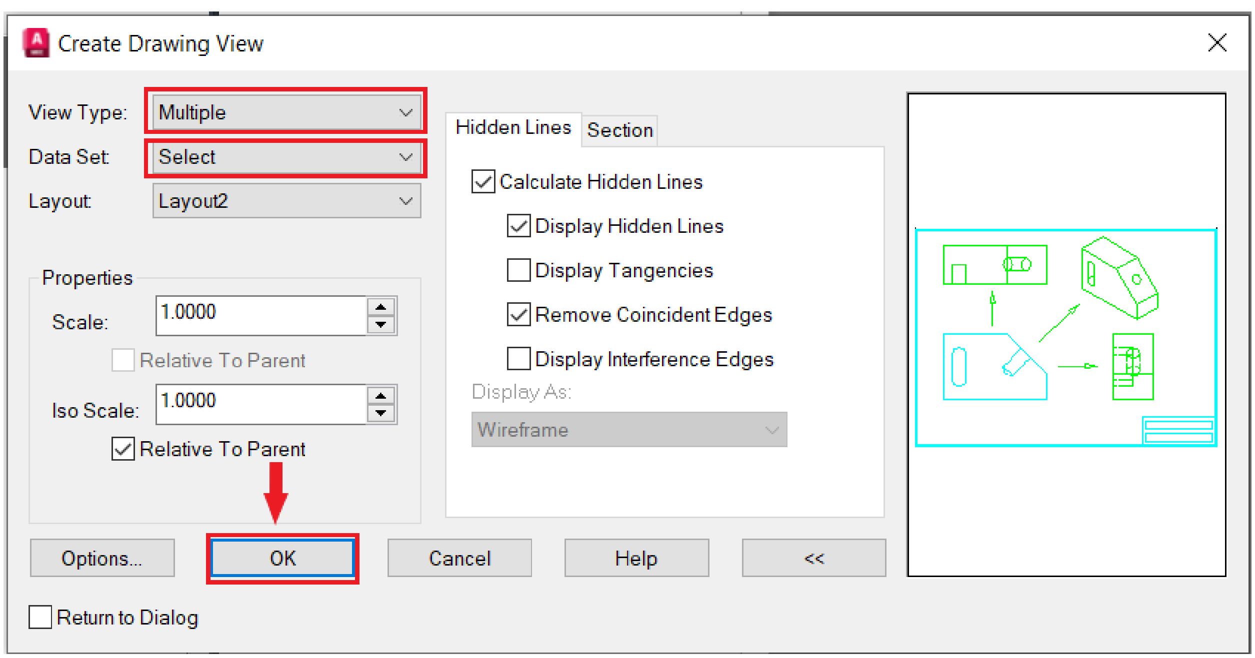
To Create Multiple Drawing Views

You can create multiple drawing views from Inventor linked models, within the same command session.

1. In the Browser, choose the Drawing tab.
2. Double-click Layout2 to make it the active layout.
3. Right-click the Layout icon, and then choose New View.

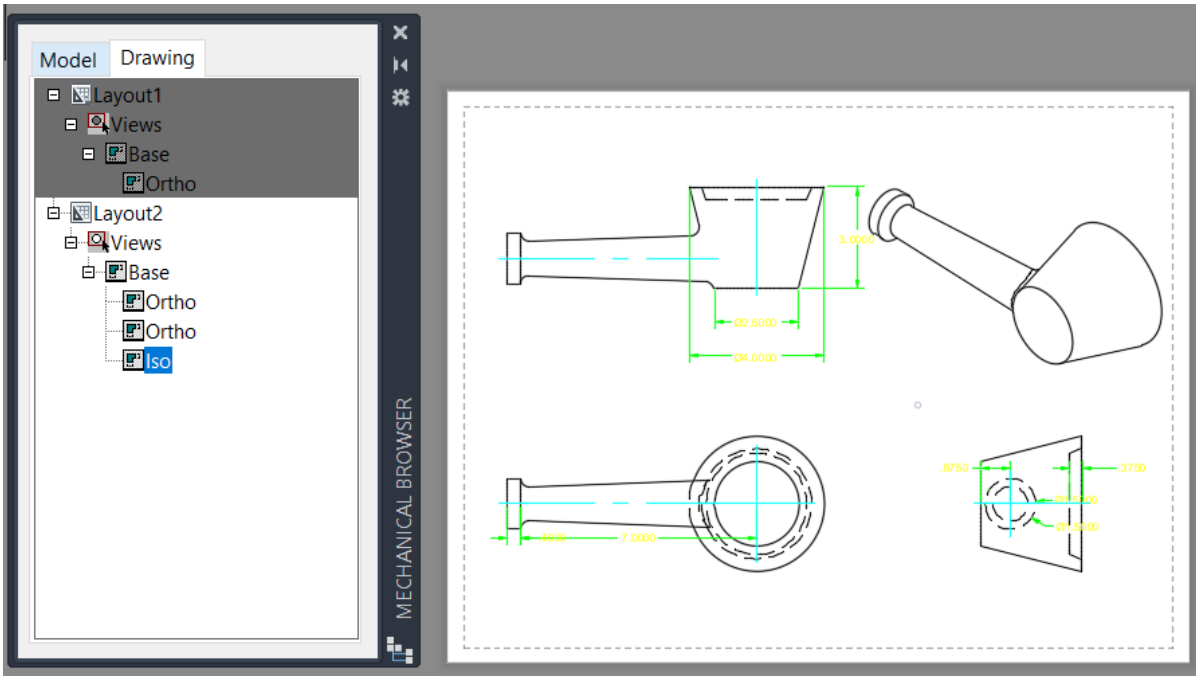


4. In the Create Drawing View dialog box,
    - View Type: Multiple
    - Data Set: Select
    - Choose OK. As shown below.



5. Select the location of the projected view.

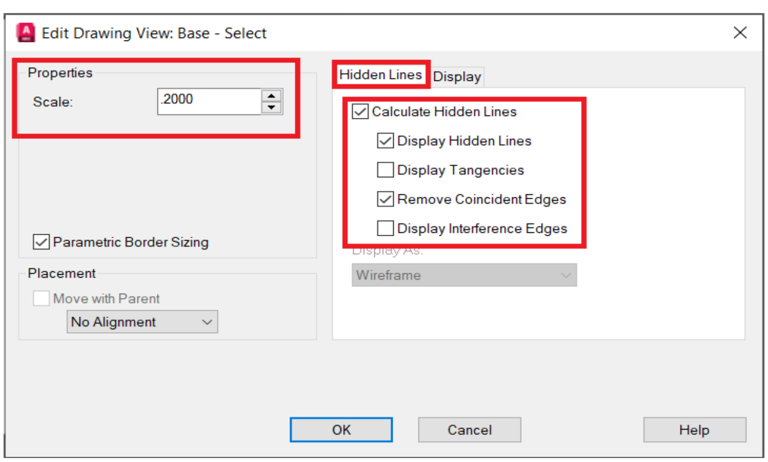
6. Continue placing as many views as desired and the rresecive view names are displayed in the browser hierarchy respectively as shown below.



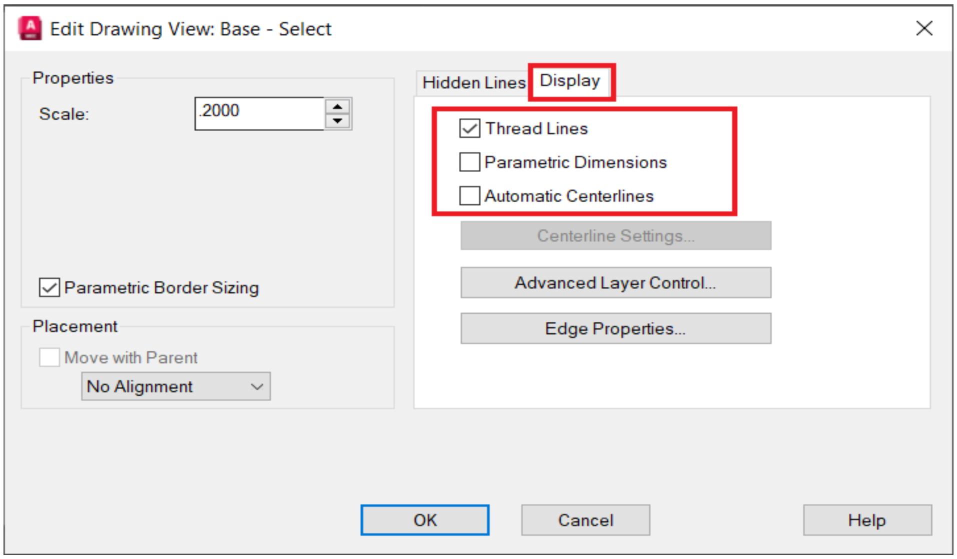
7. Press ENTER to exit the command.

About Editing Inventor Link Drawing Views

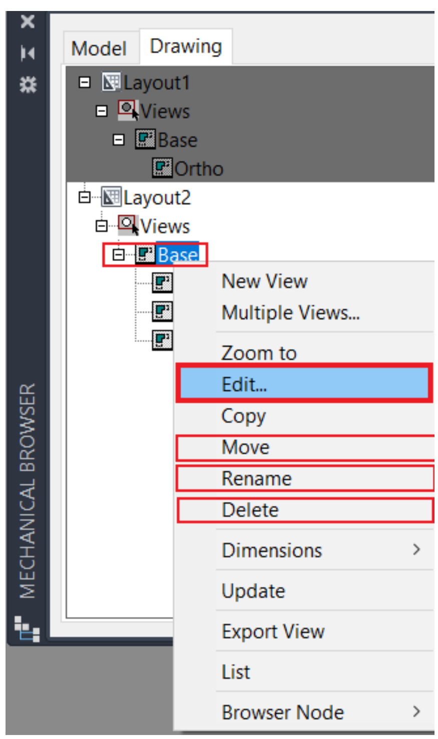
- You can edit drawing views using the Edit Drawing View dialog box and the Browser.



- The Edit Drawing View dialog box contains options for modifying part edges, changing hatch patterns, modifying hidden lines, and changing the display of annotations.



- You can use the Browser right-click menu to rename, edit, move, or delete drawing views



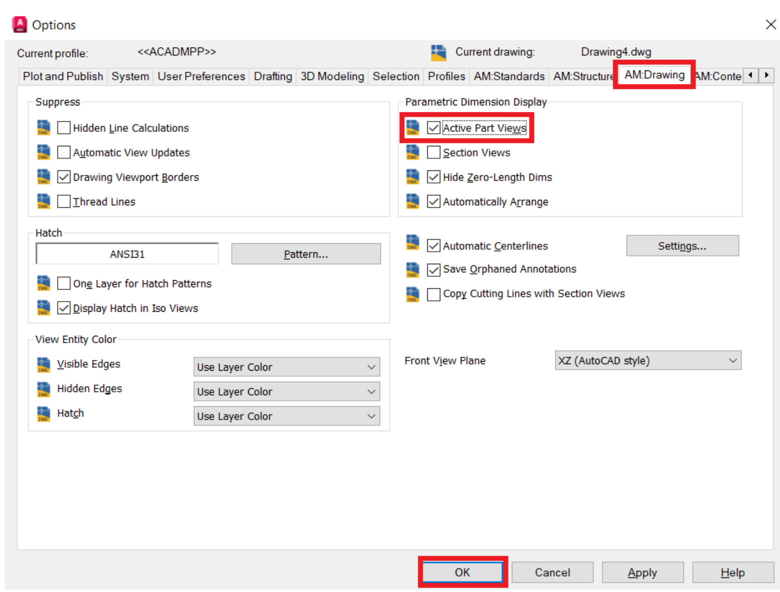
Adding Dimensions to Inventor Link Drawing Views

- Inventor Link exposes you to two types of dimensions; parametric dimensions and reference dimensions.
- A parametric dimension controls a parametric part feature. If you change the dimension, you change the part. You place a reference dimension on a part in Drawing mode.
- Unlike parametric dimensions, reference dimensions do not control parts but follow modifications to the part.
- Inventor Link gives you the ability to display parametric dimensions defined in Inventor and add reference dimensions in AutoCAD Mechanical toolset.

To Enable Parametric Dimensions to be Transferred at Drawing View Creation

When an Autodesk Inventor linked file is created, the link is able to read parametric dimensions from the part/assembly file. However, to display parametric dimensions on a drawing view, they must be transferred to paper space, typically, at the same time the drawing view is generated. You can however opt to add parametric dimensions later

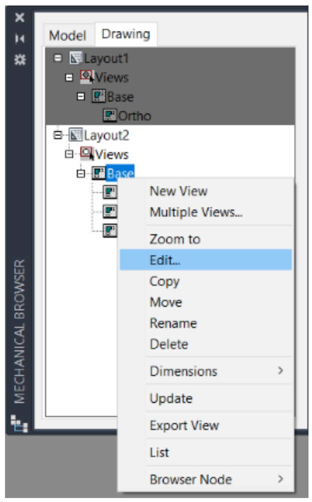
1. At the Command prompt, enter OPTIONS.
2. Click the AM:Drawing Tab
3. From the Parametric Dimension Display section, select the Active Part Views option.
4. Click OK. Whenever a drawing view is created, parametric dimensions will also be created.



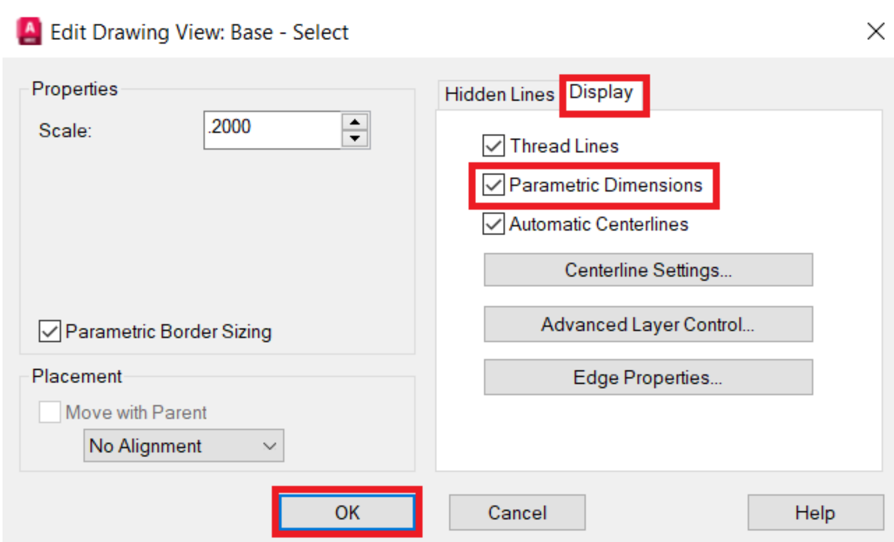
Although Inventor Link can read parametric dimensions from the part/assembly file, it is unable to display parametric dimensions on a drawing view unless you transfer the parametric dimension information from the model to the drawing views

To Transfer Parametric Dimensions After Creating Inventor Link Drawing Views

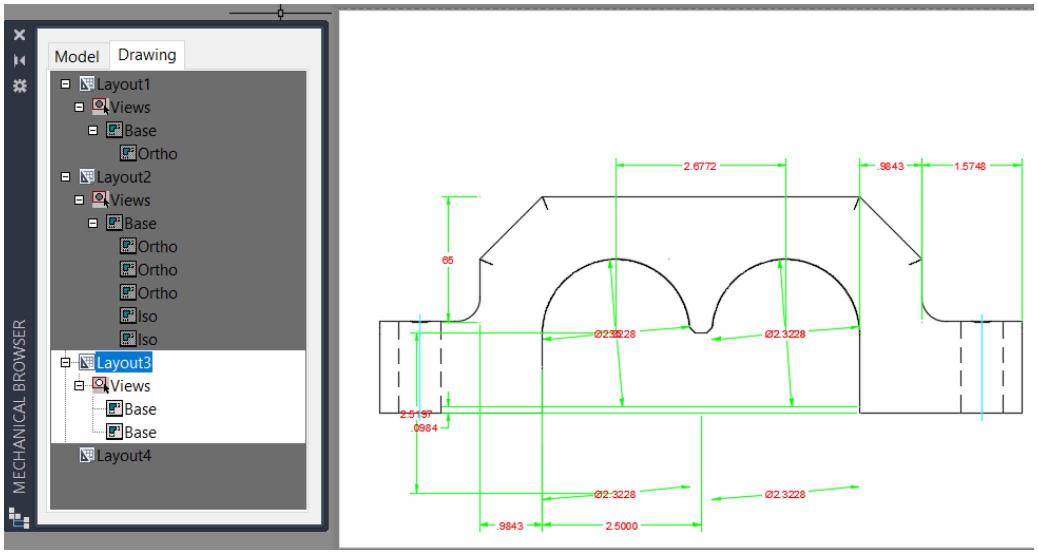
1. In the browser, right-click the view you want to generate dimensions for. A menu is displayed.



2. Click Edit. The Edit Drawing View dialog box is displayed.
3. Click the Display tab and select the Parametric Dimensions option.



4. Click OK. Parametric dimensions are generated and displayed on the drawing view.



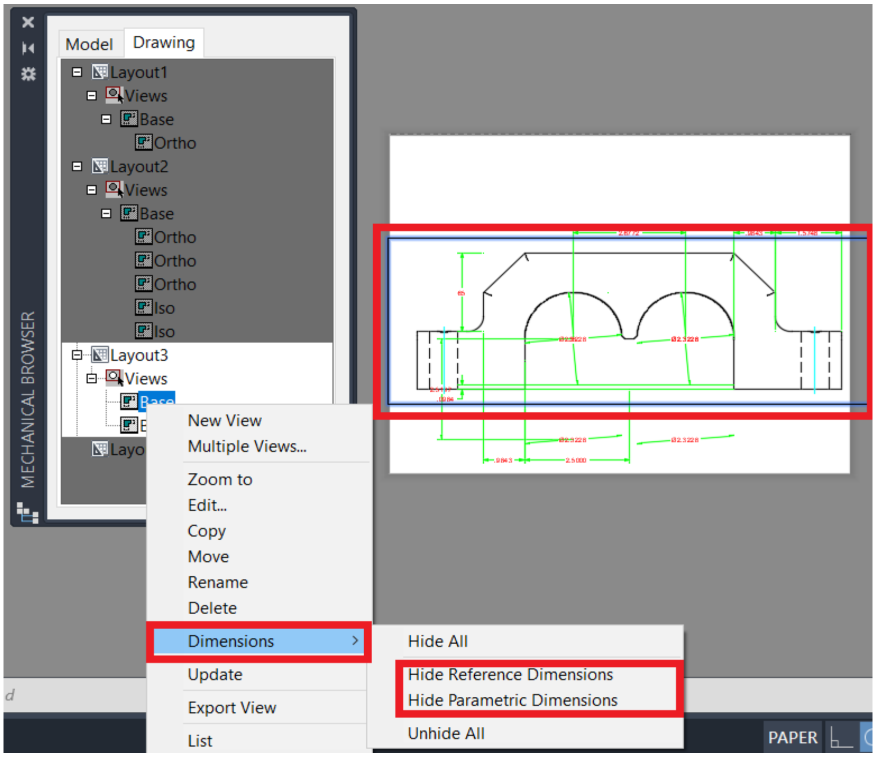
- However, in some cases (*for example a drawing view of an assembly that looks cluttered when all parametric dimensions appear*), transferring parametric dimensions is not desirable. In such cases, you can opt not to transfer parametric dimensions automatically, but to transfer parametric dimensions manually after you create the view.

- Once parametric dimensions appear, you can use the **AMPOWERDIM** or **AMAUTODIM** command to add more dimensions as required. Such dimensions act as reference dimensions.

- AutoCAD Mechanical toolset also provides the capability to hide dimensions by type.

To Temporarily Hide Parametric and Reference Dimensions on Inventor Link Drawing Views

1. In the browser, right-click the view you want to hide the dimensions of. A menu is displayed.
2. Click Dimensions > Hide Parametric/Parametric Dimensions. All parametric/reference dimensions are hidden as shown below.



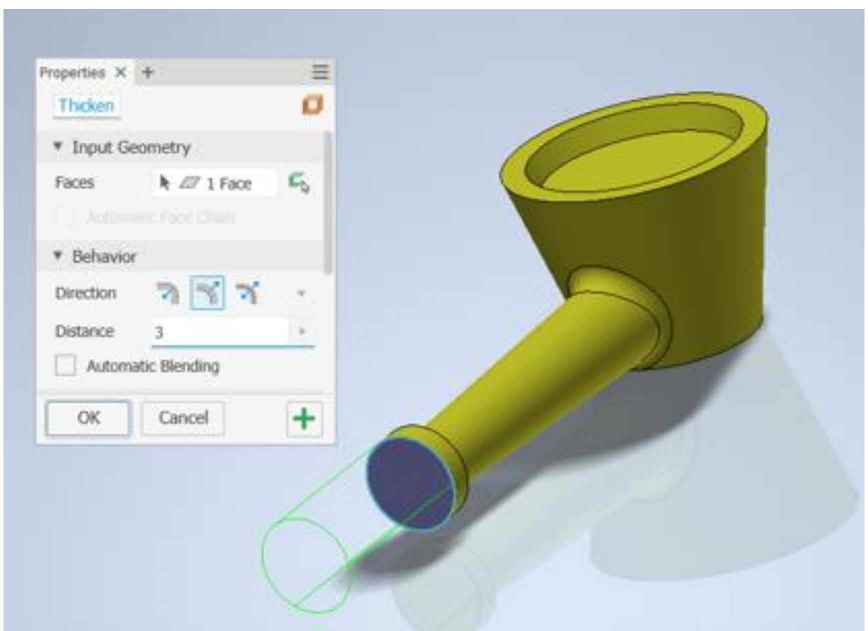
Model
Drawing
Layout1
Views
Base
Ortho
Layout2
Views
Base
Ortho
Ortho
Ortho
Iso
Iso
Layout3
Views
Base
MECHANICAL BROWSER
New View
Multiple Views...
Zoom to
Edit...
Copy
Move
Rename
Delete
Dimensions
Update
Export View
List
Hide All
Hide Reference Dimensions
Hide Parametric Dimensions
Unhide All
PAPER

Inventor Models That Are Modified at the Source (at Autodesk Inventor):

- When a part or assembly file to which you have linked changes, the program highlights the changed file in the browser.
- When you right-click the highlighted file and select Update from the menu, the program updates the model as well as all generated drawings views.
- If you have overridden parametric dimensions (for example, adding tolerance values to a parametric dimension), the program discards those overrides. None of the other dimensions are affected.
- As with parametric dimensions, if you override any of the BOM columns derived from iProperties, they are overridden.

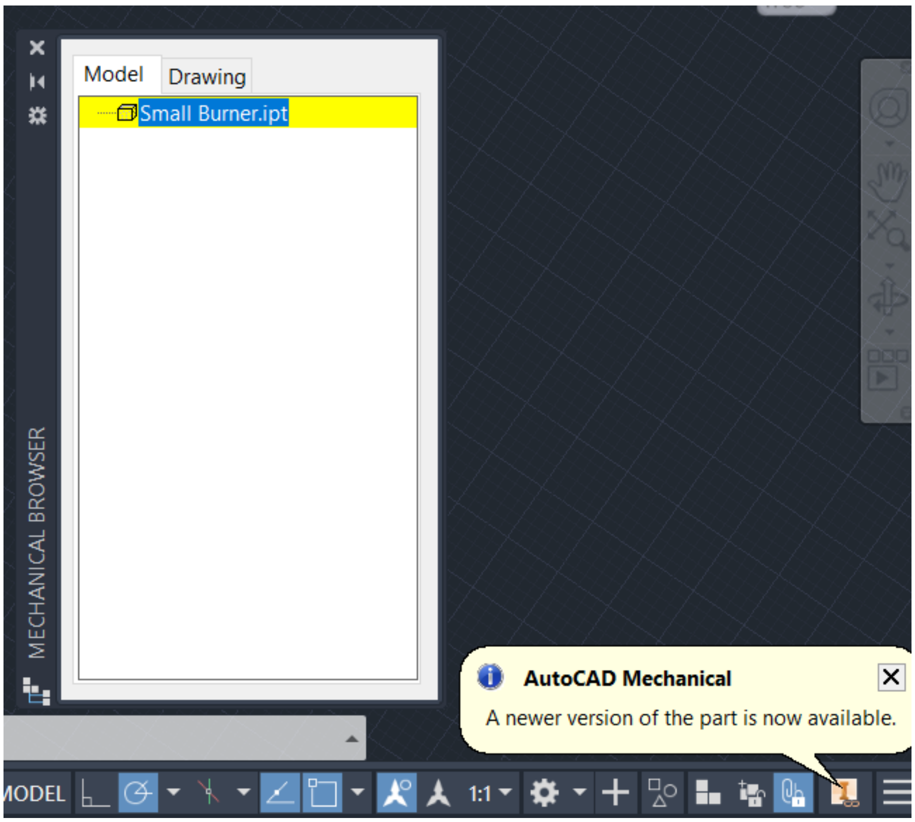
How To Update an Autodesk Inventor Linked Model

1. First you have to make necessary changes in an Inventor part or assembly in Autodesk Inventor.
2. For instance, add a fillet to any edges in the Inventor model as shown below:

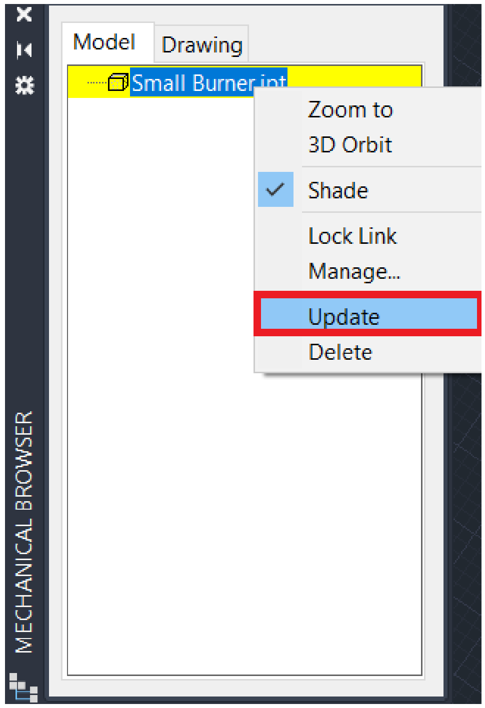


3. Click the Save icon and not necessarily close the Inventor application.

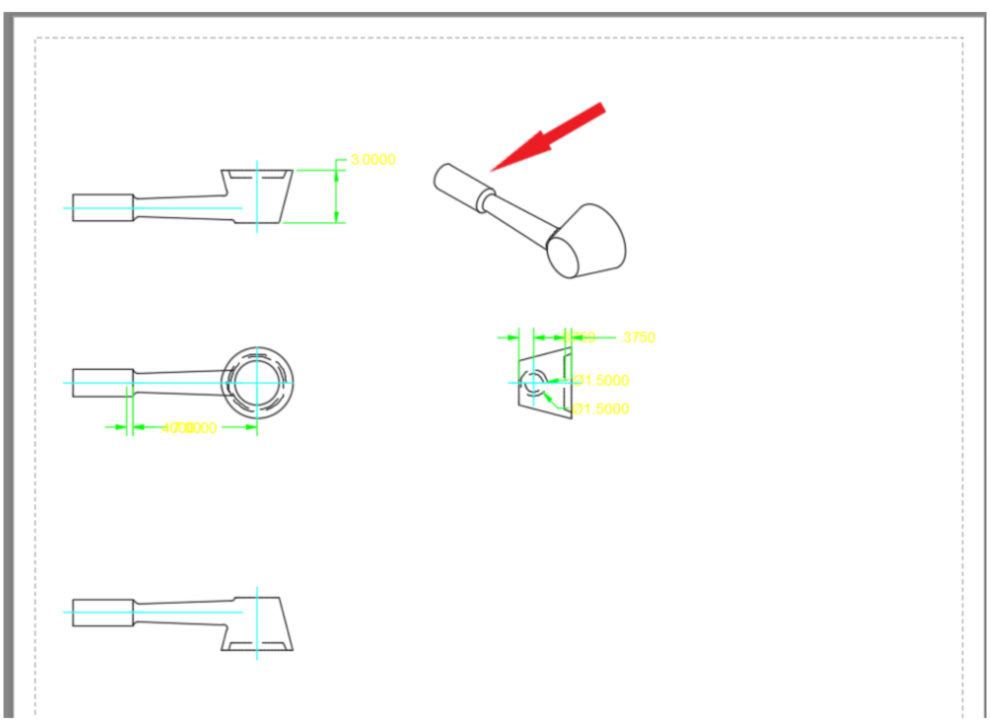
4. Now open AutoCAD Mechanical and you can see the part name is highlighted with yellow colour in the Mechanical browser and a message saying "A newer version of the part is now available" pops up at the right bottom corner as showsn below:



5. In the Model tab of the browser, right click the name of the file. A menu is displayed.



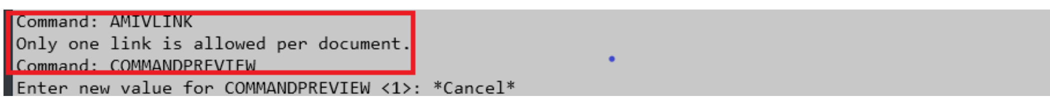
6. Select Update. The 3D model and all views in paper space are updated as shown below:



Commands and system variables for Working With Inventor Link

AMIVLINK

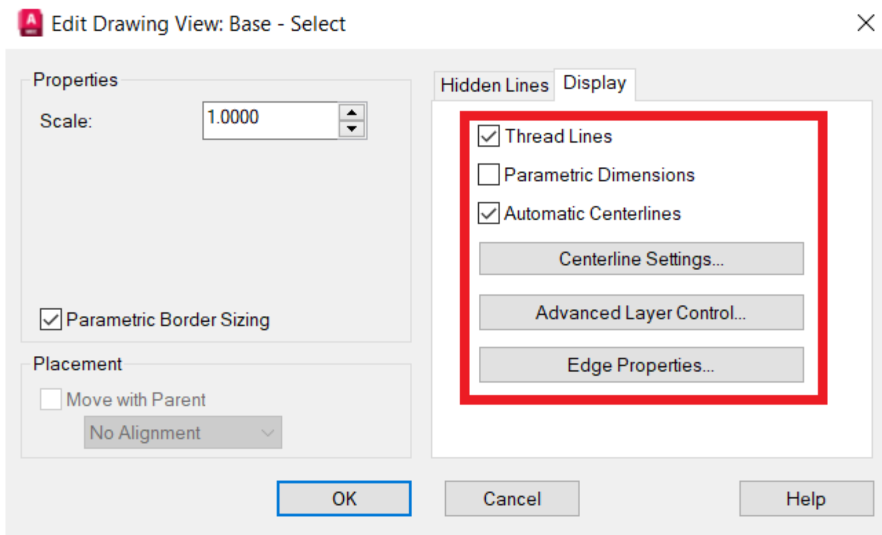
- Creates a link between a drawing file and an Autodesk Inventor assembly (.iam) or part (.ipt) document.
- You can use this command only on an Autodesk Inventor linked dwg file that has had the link to the Inventor document deleted.
- Note: The program allows only one link per document.
- If you try to use this command on an already linked model, then the following message will show in the command preview.

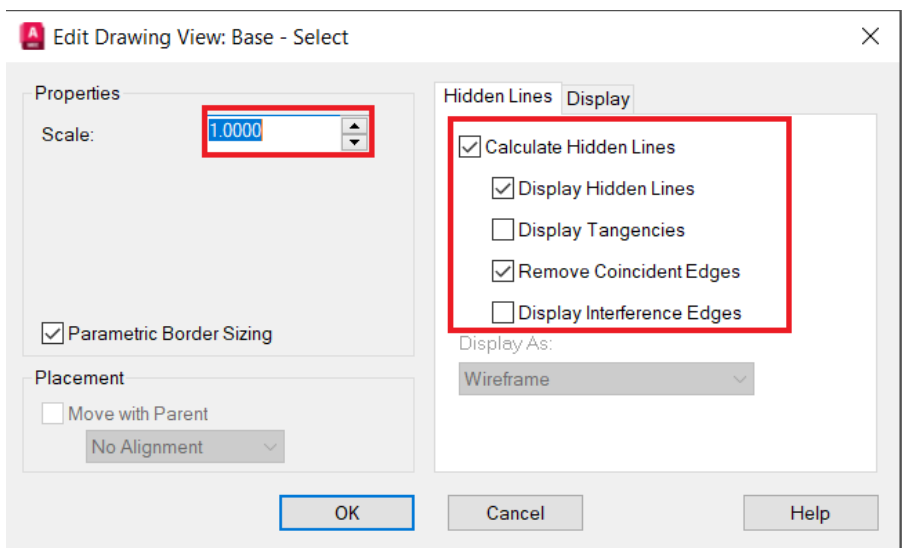


- If the Inventor link is deleted from the browser, then you can use the command AMIVLINK to create a link again.

AMEDITVIEW

- Modifies the attributes of an Inventor link drawing view
- You can modify the scale, associativity, and hidden line display of a drawing view, as well as controls visibility of centerlines and viewport borders and layers.





AMIVUPDATE

- Use AMIVUPDATE to reread the Autodesk Inventor part or assembly files and refresh the 3D model in model space and all views in paper space.

- When a drawing file that is linked to an Autodesk Inventor part or assembly is out of synchronization, the browser highlights the affected part, assembly view nodes, in yellow.
- Additionally, a bubble window in the status bar may appear, notifying you that a newer version is available.

System Variable:

AMREUSEDIM

Controls whether Inventor link drawing views are created with the ability to display parametric dimensions.

0 and 2 – Off

1 and 3 – On

Learn how annotation data is exchanged between Inventor and AutoCAD Mechanical

The link also extracts the following details from the part/assembly file:

- Parametric dimensions
- iProperties (inclusive of custom iProperties)
- Thread and Hole annotation information

Parametric Dimensions

The Autodesk Inventor Link automatically reads the following parametric dimensions and makes them available for display on drawing views.

Sketch dimensions:

- Two-point distance
- Two-line angle
- Three-point angle
- Diameter
- Radius
- Ellipse radius
- Offset
- Tangent Distance

Feature dimensions: (dimensions created based on the existing features of the model)

- Extrusion
- Revolution

If the part or assembly file does not contain nominal dimensions, the program does not import parametric dimensions. A message appears when you link to the part or assembly file.

Since the program obtains parametric dimensions from the part/assembly file, you cannot modify them. You can, however, change (override) tolerance values. If you change the file in Autodesk Inventor and then update the linked drawing, the program discards all overrides.

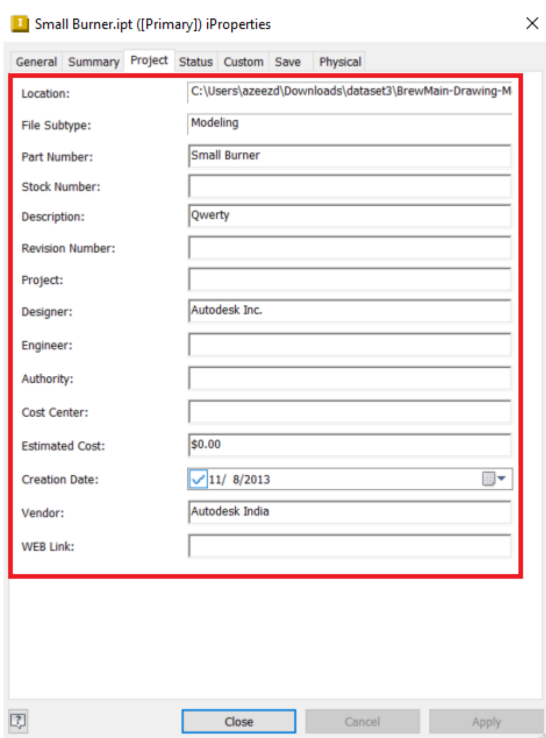
iProperties

iProperties read from the part/assembly file are available to be added as columns of the Bill of Materials (BOM). You can use them for parts lists.

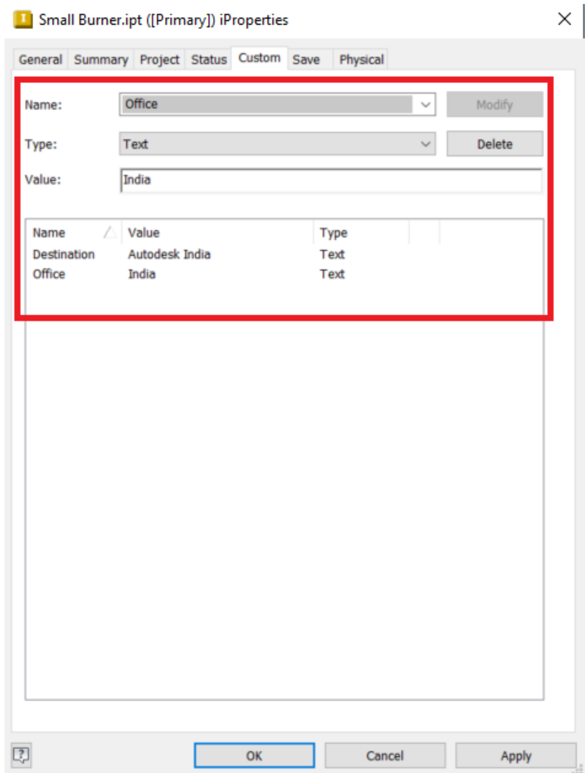
If you override the BOM properties from AutoCAD Mechanical toolset, the program discards the overrides when you update the drawing file.

How to make iProperties make visible in AUtoCAD Mechanical BOM/part list

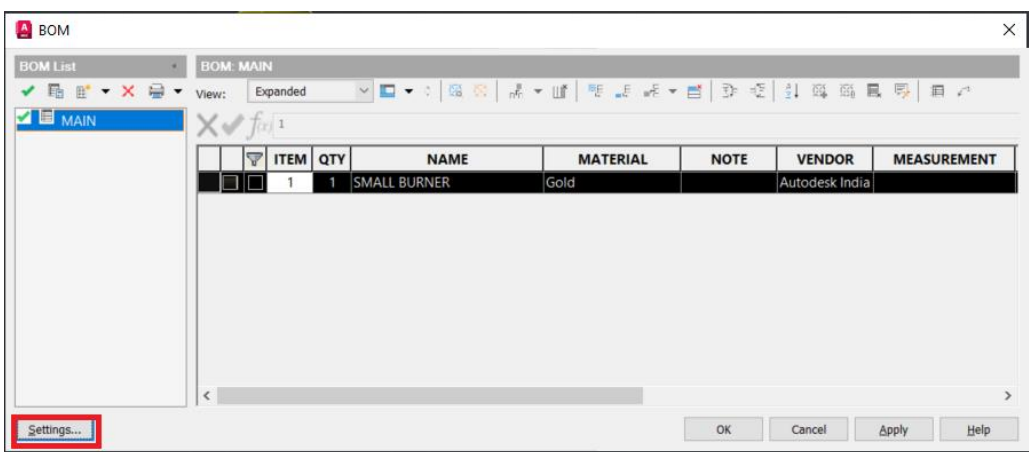
1. Open the part/assembly file in Inventor and go to the iProperties
2. Under the Project tab, give values for the properties such as Designer, Vendor, Revision number etc.,



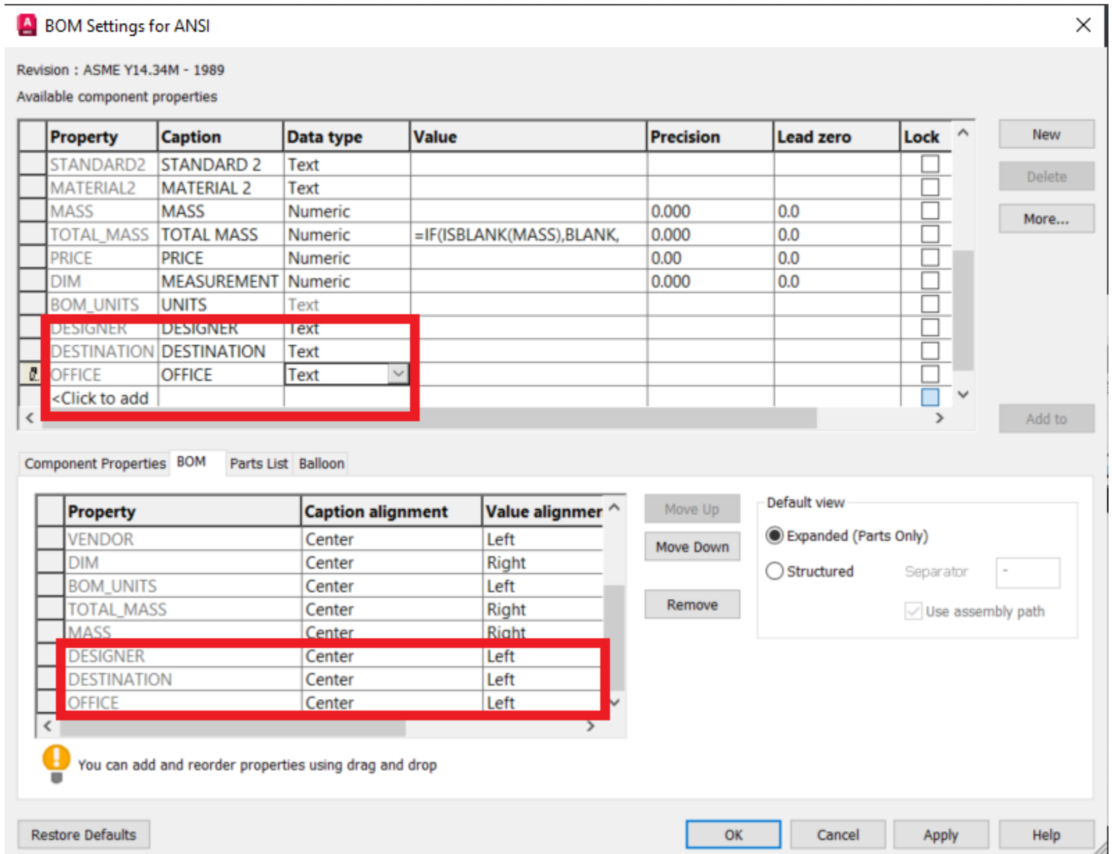
3. Now switch to Custom tab and add a custom iProperty and its value in it.



4. Click Apply and close.
5. Save the Inevntor file.
6. Now open the BOM settings in AutoCAD Mecanical through Annotate tab or by using AMBOM command.

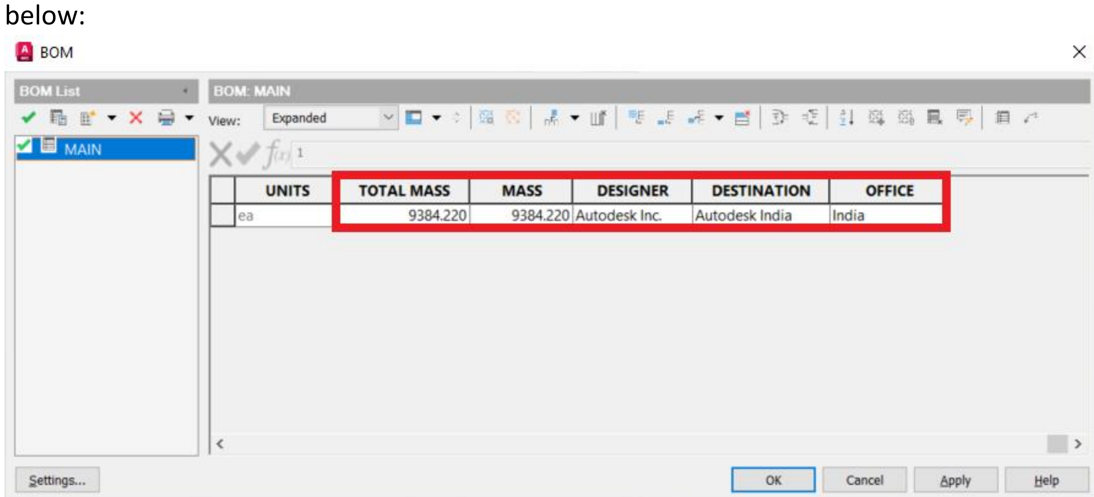


7. In the BOM settings, add the default and custom iProperty names to the list.



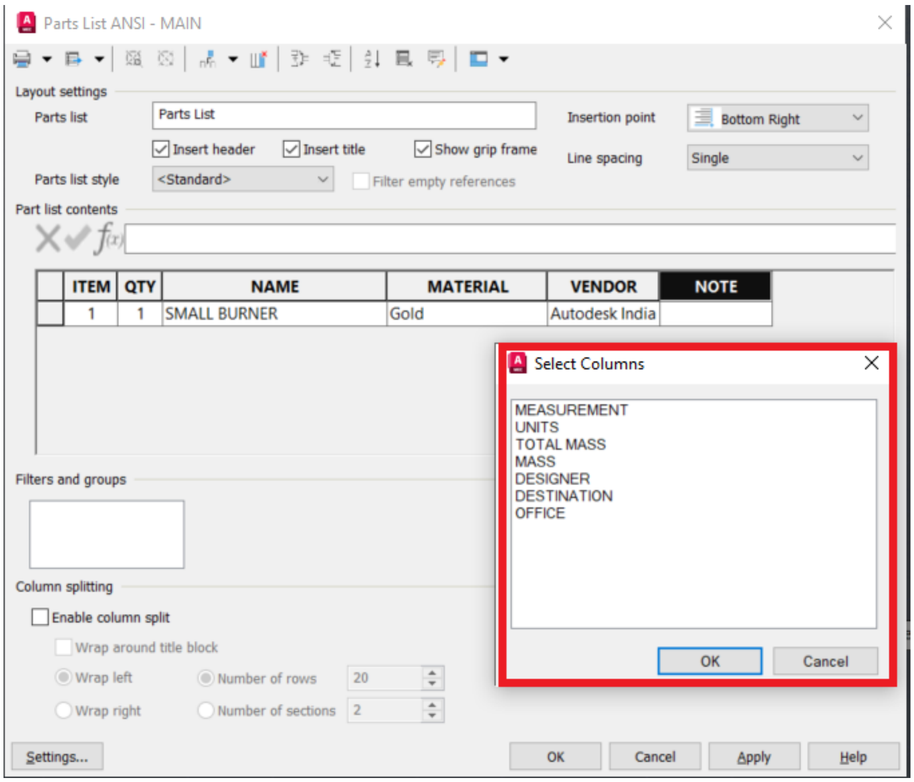
NOTE: while adding the names the property and the caption value is case sensitive and it should be caps to fetch the iProperty values.

8. After adding the properties, click Apply and OK.
9. Now you will be able to see the iProperties and it values in the BOM lists like shown below:

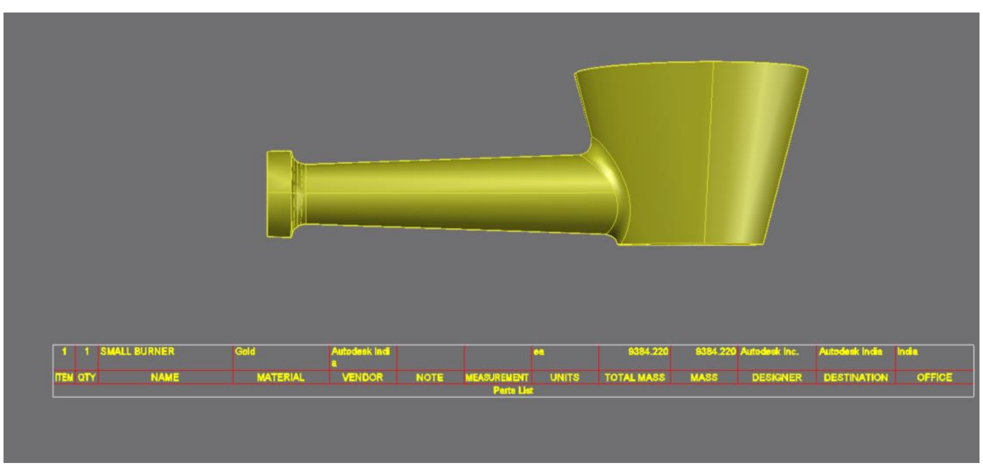


10. Now insert the BOM into you rdrawing using the Part list under Annotate tab or using AMPARTLIST command.

NOTE: Add new columns in the part list dalog box to accommodate the new iPorperties.

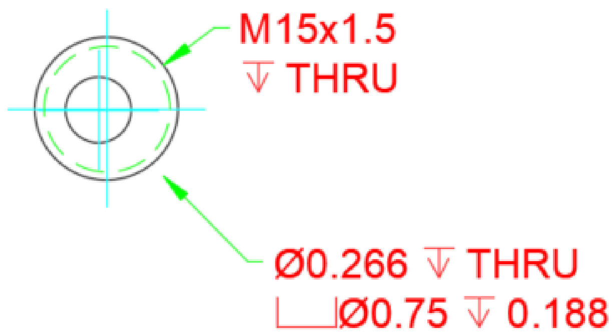


Click Apply and OK to generate the BOM.



Thread and Hole Annotations

You can annotate thread and hole details extracted from the part/assembly with the AMNOTE command like below:



AMNOTE recognizes a thread as:

- an external thread, if the picked face belongs to an extrusion add.
- a whole thread, if the picked face belongs to an extrusion cut.

Additional resources

- About Inventor Link
- Autodesk Inventor Link Error Message Reference
- Commands for Working With Inventor Link
- About Using iProperties in Parts Lists