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

Speakers:

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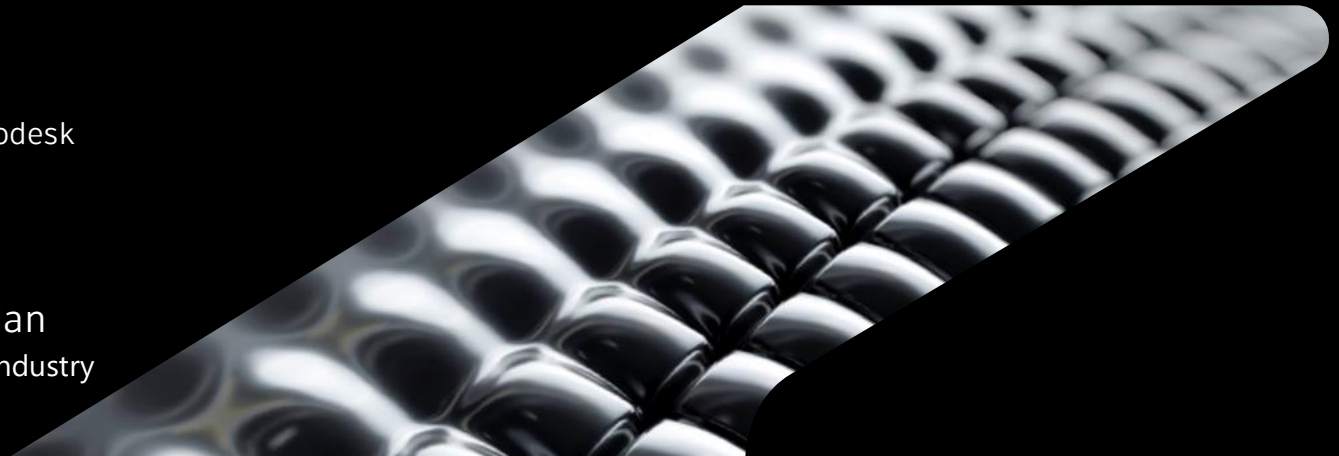

About the Speakers

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

About the 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.

About the Speakers

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.

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Summary

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.

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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

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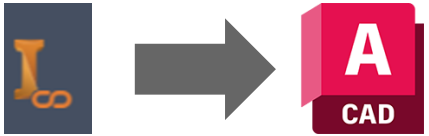
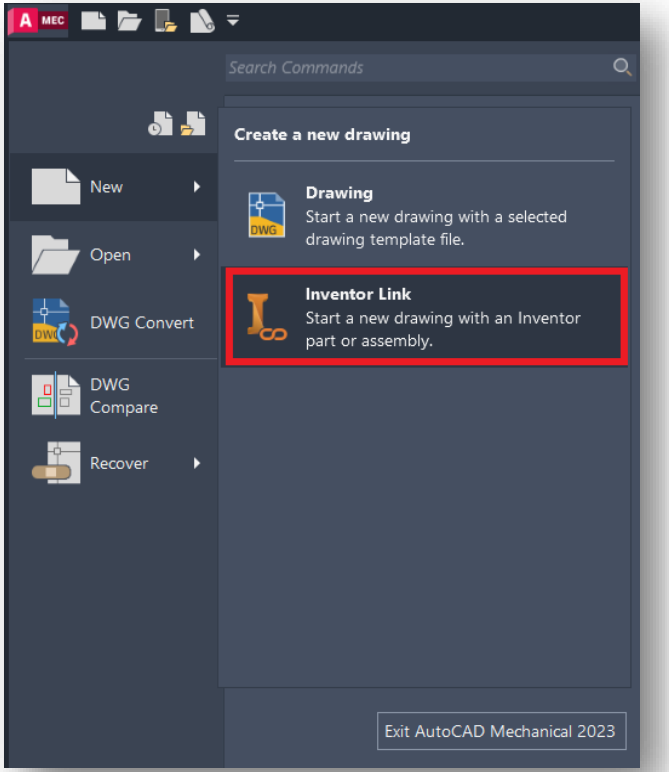
Learning Objective: 1

Learn how to link the Inventor assembly (.iam) and part (.ipt) files

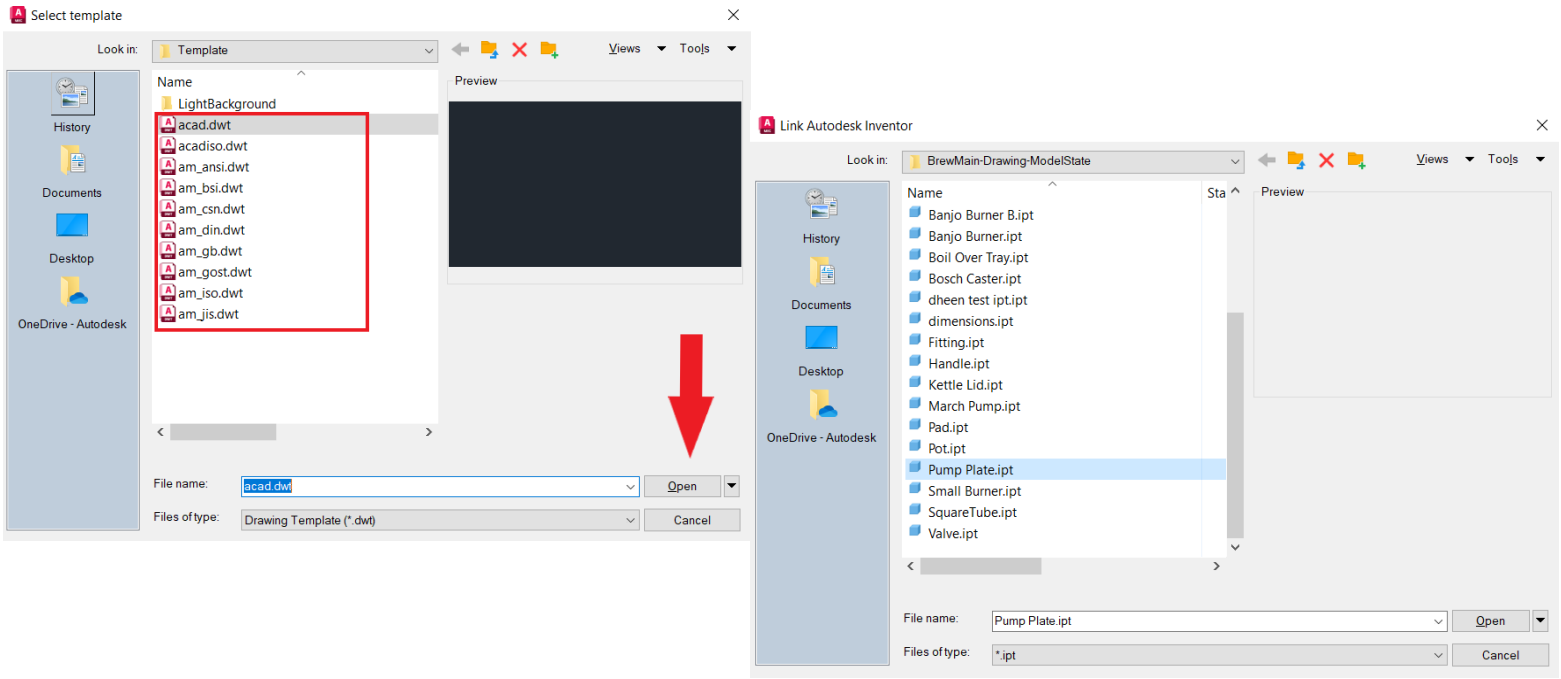
Using Inventor link for Inventor assembly (.iam) and part (.ipt) files



- Access the Inventor link through the start menu in AutoCAD Mechanical



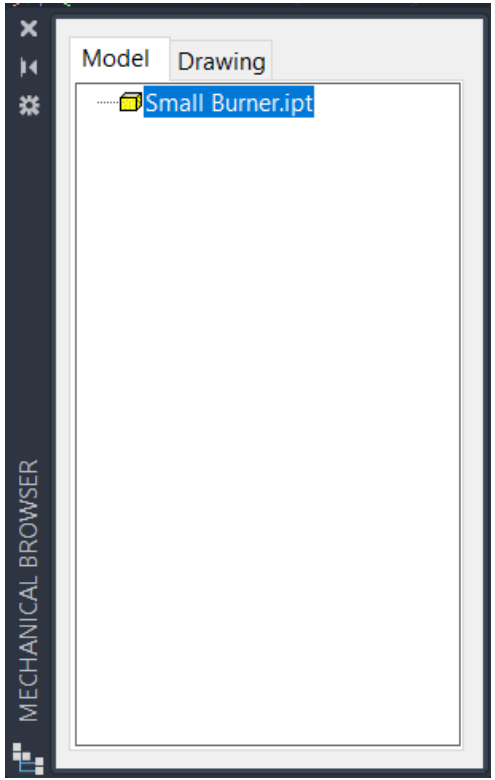
Linking files between Inventor & AutoCAD Mechanical

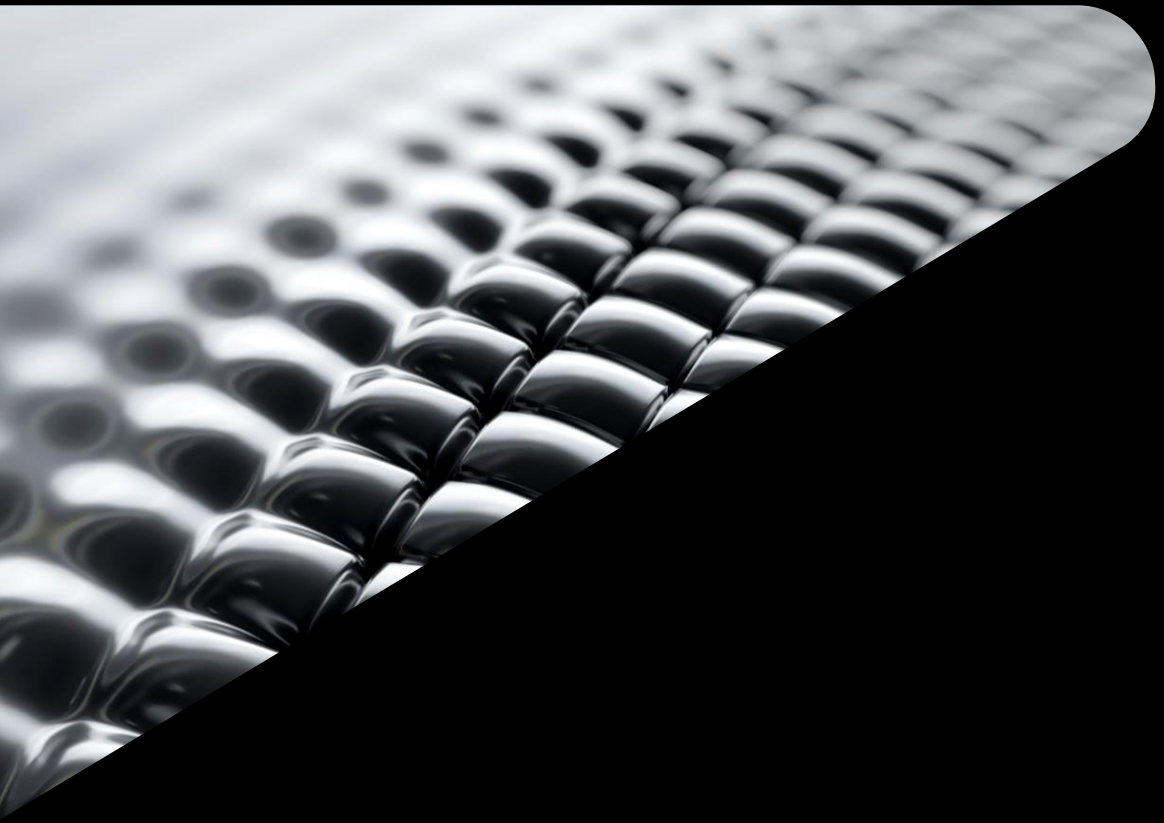


- Choose the template and respective .ipt or .iam file, link is established between Inventor and AutoCAD Mechanical.

Understanding Inventor Part file in AutoCAD Mechanical

- Mechanical Browser shows part & assembly hierarchy in AutoCAD Mechanical
- Access through Command line : **AMBROWSER**





Demo

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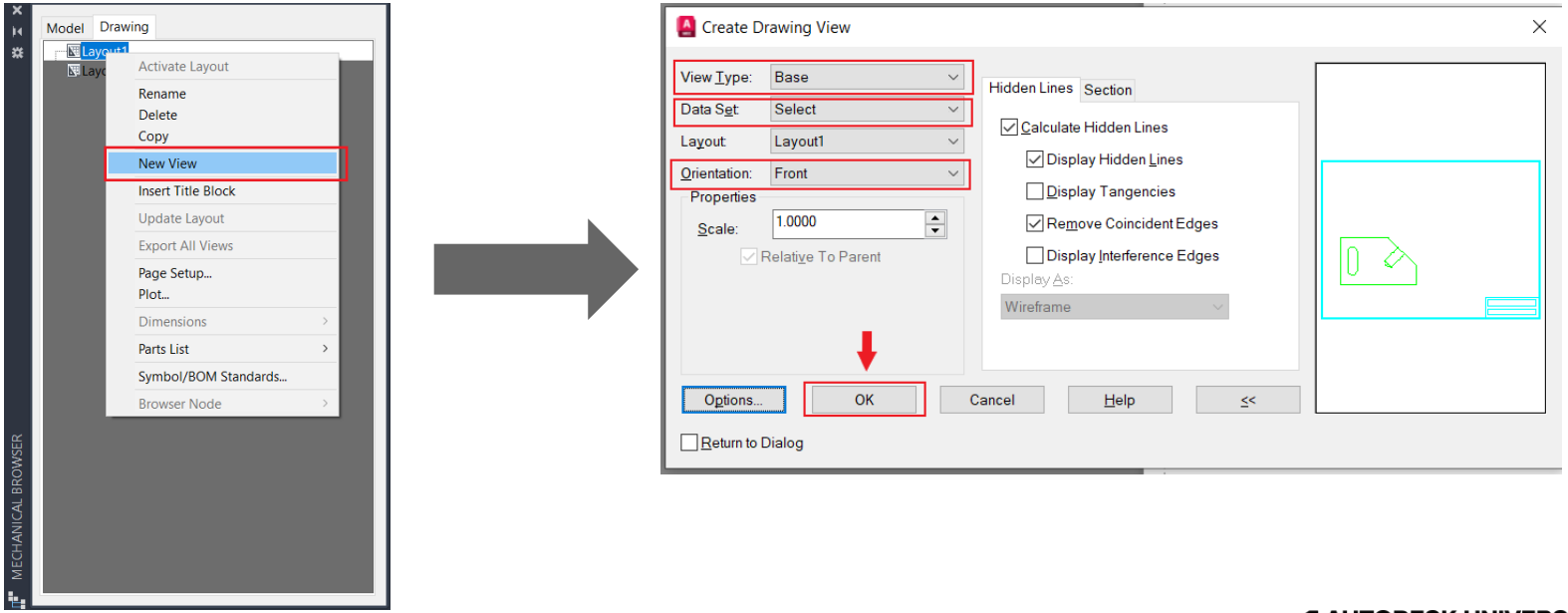
Learning Objective: 2

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

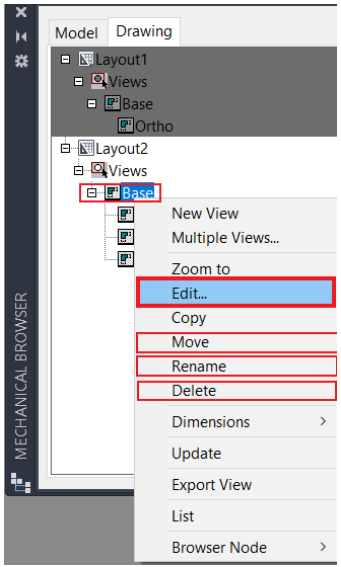
Generating drawing views from Inventor linked Part & Assemblies

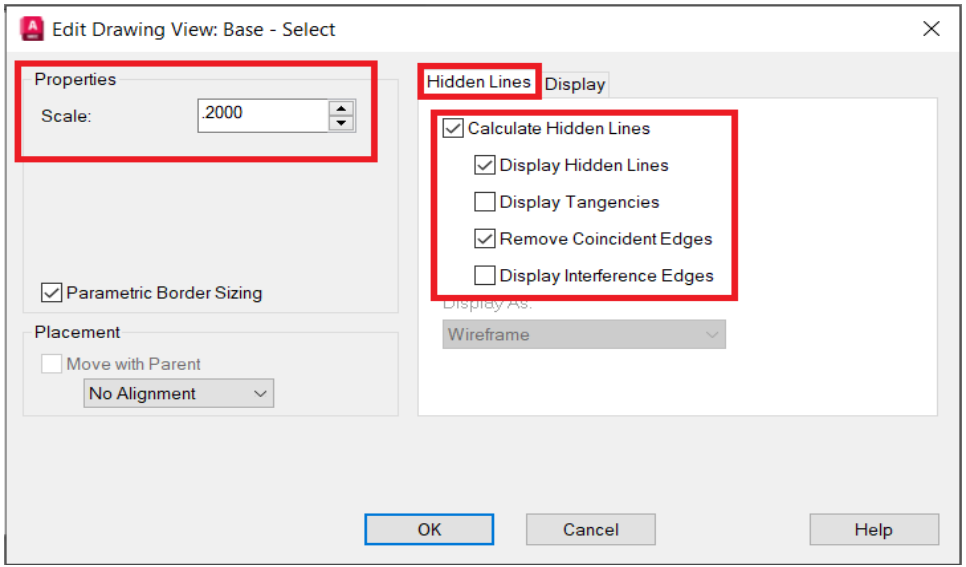
- Create a new view using **AMDWGVIEW** or by doing a right-click on the layout in Mechanical Browser.



Modifying Inventor linked Drawing Views

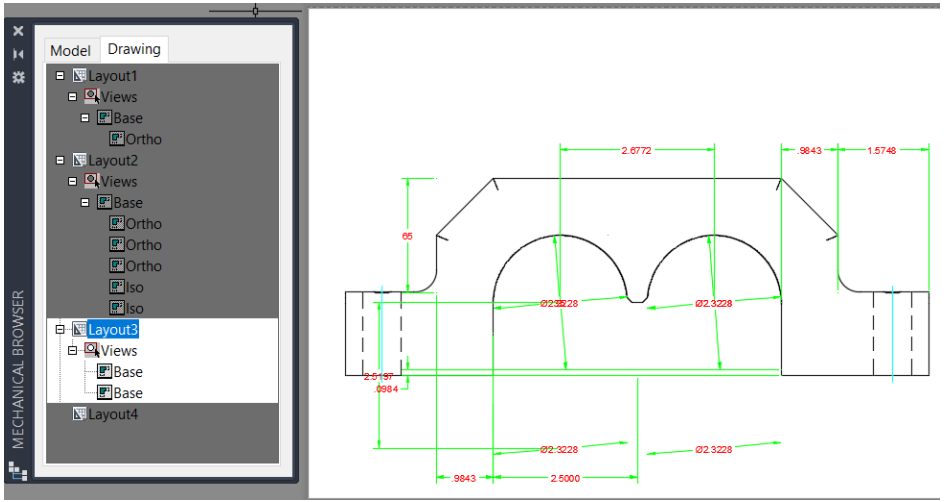
- Modify drawing views using **AMEDITVIEW** or by doing a right-click on the view and Edit.

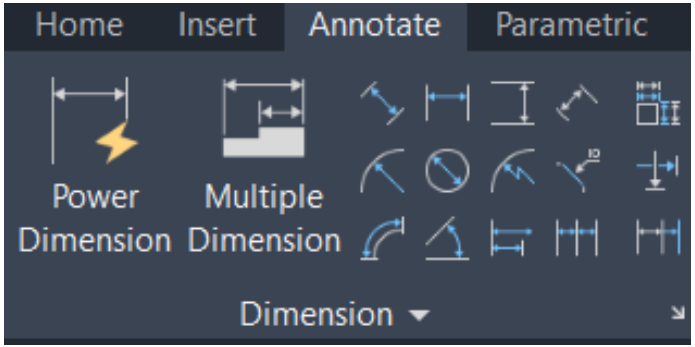


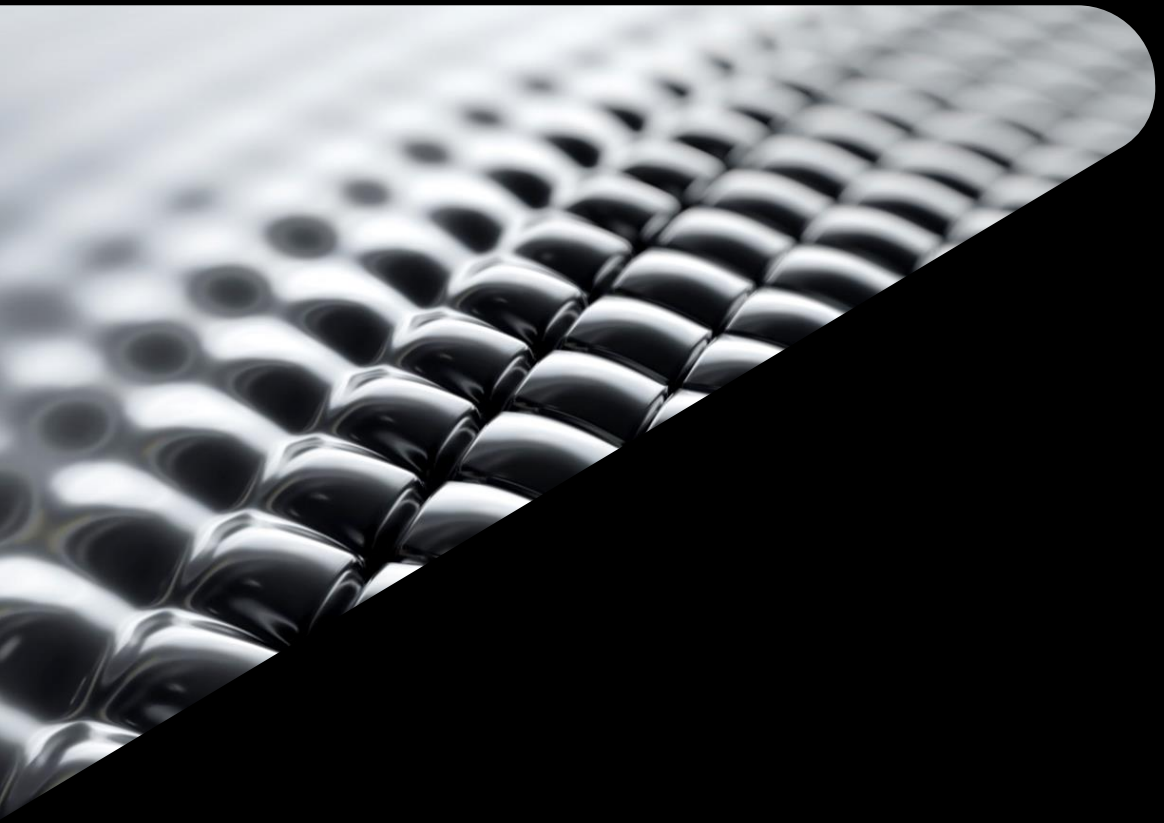


Adding dimensions to Inventor linked drawing views

- Assign reference dimensions to the drawing views using **AMPOWERDIM**
- Automate the process using **AMAUTODIM** command
- Both commands are in Ribbon under -> Annotate tab.







Demo

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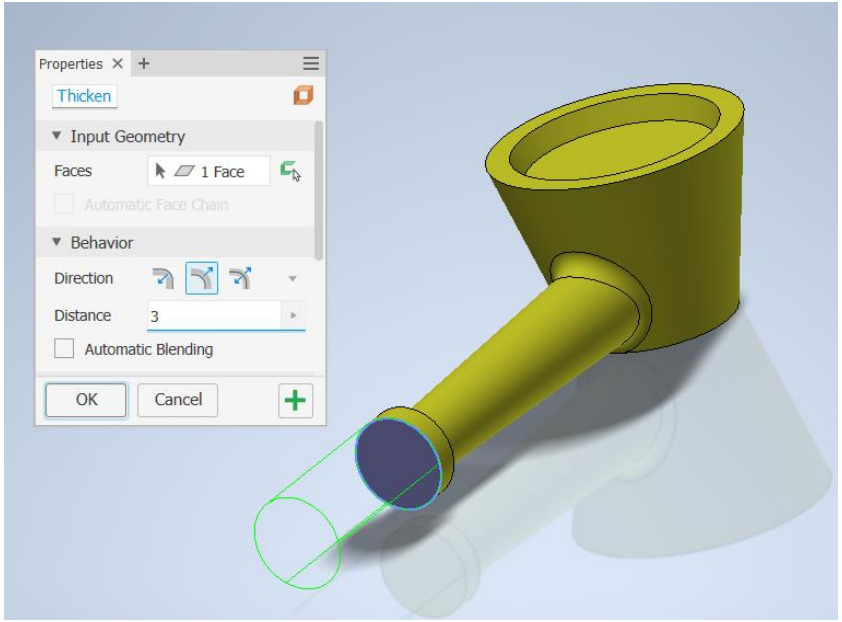
Learning Objective: 3

Learn how to update an Inventor linked model and get to know about commands and system variables on Inventor link

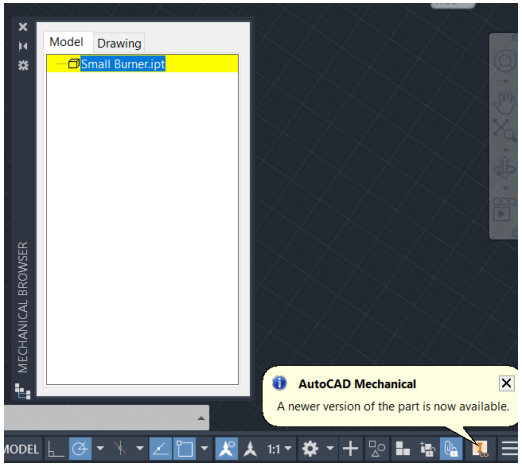
Modifying Inventor .ipt or .iam at source

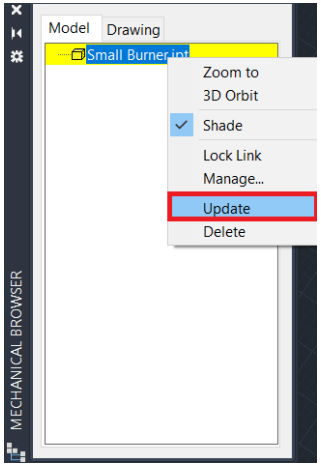
- Make any changes to Inventor Part (.ipt) or Assembly (.iam) to be updated in AutoCAD Mechanical.

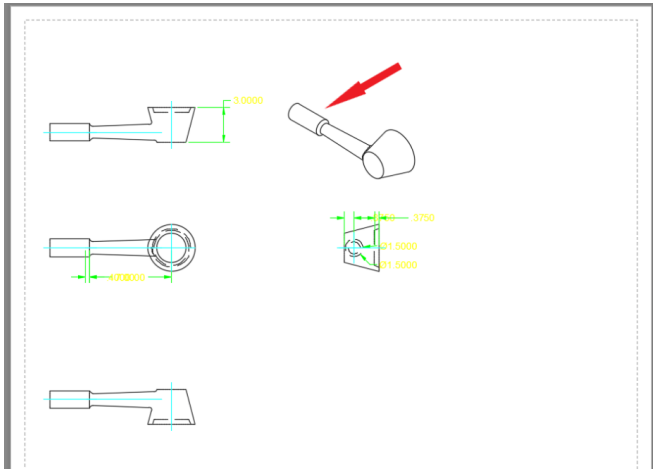


Model changes getting updated in AutoCAD Mechanical

- Changes to Inventor part being reflected in AutoCAD Mechanical.
- Changes displayed both in model space and drawing views in paper space.







Some System variables for working with Inventor link

AMIVLINK

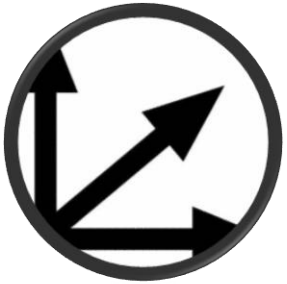
Creates a link between a drawing file and an Inventor assembly (.iam) or part (.ipt)

AMEDITVIEW

Modifies the attributes of an Inventor link drawing view

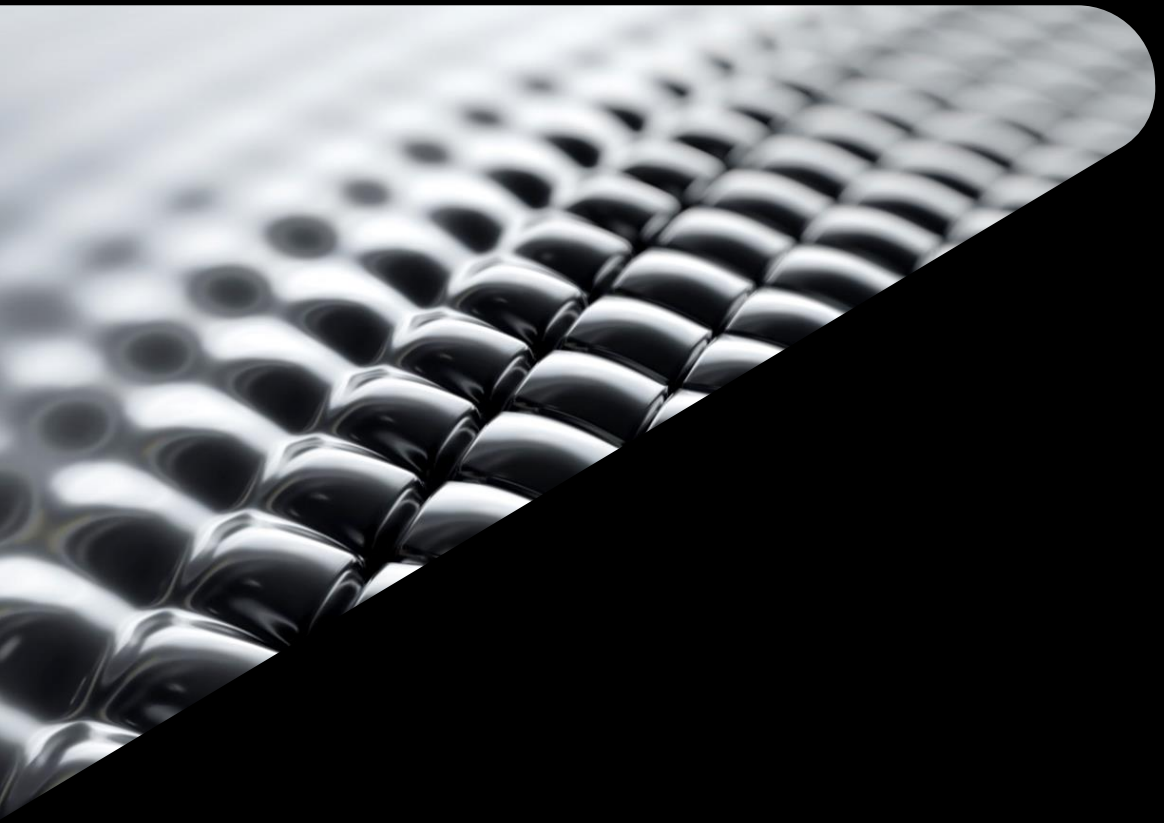
AMIVUPDATE

To reread the Autodesk Inventor part or assembly files and refresh the 3D model in model space and all views in paper space.



AMREUSEDIM

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



Demo

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Learning Objective: 4

Learn how annotation data is exchanged between Inventor and AutoCAD Mechanical

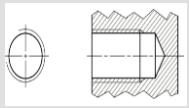
Data exchanged between AutoCAD Mechanical and Inventor through Inventor link

Parametric Dimension

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

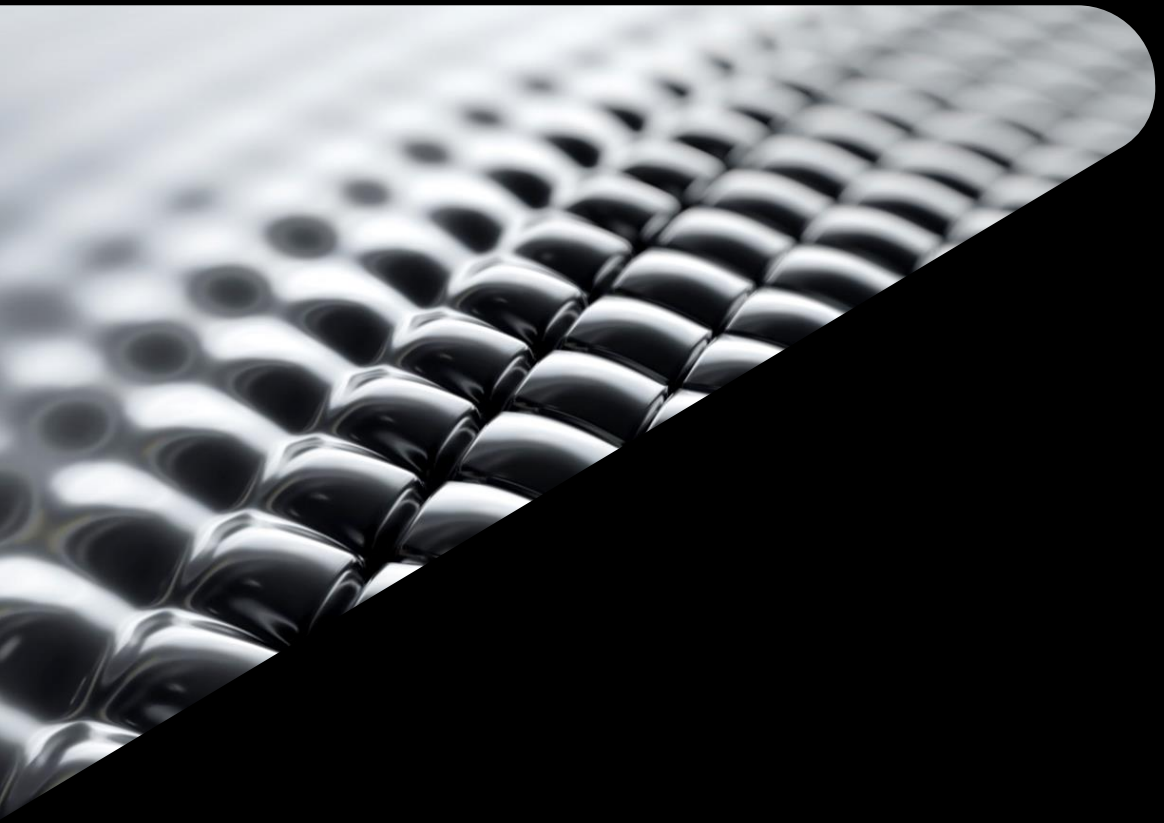
iProperties

- 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



Thread and Hole annotation information

- You can annotate thread and hole details extracted from the part/assembly with the AMNOTE command.



Demo

Additional resources

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