

IM320977

AnyCAD and the Exchangeability of Inventor

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Learning Objectives

- Learn how to use Inventor AnyCAD to reference third-party models
- Learn how to use 2D DWG data to build a new model
- Learn how to share information back and forth between Inventor and Fusion 360
- Learn how the BOM can travel downstream

Description

We've all been there. Multiple sources of information, various software systems, and everyone needing to efficiently communicate to complete the project. AnyCAD software provides the medium to use, reuse, and exchange data of various types and from different sources. In this class, we'll explore the different types of data supported by Inventor software and the applications included in the Product Design & Manufacturing Collection.

Speaker

I have been using AutoCAD since r13, cut my solid modeling teeth on Mechanical Desktop, and have been using Inventor since before it was known as Inventor. Data Management has always been a big part of my professional life, with Autodesk Vault.

AnyCAD, AnyTime, AnyWhere, Your Way

AnyCAD is the name given by Autodesk to Inventor's ability to work with and exchange data from a variety of sources. Inventor natively imports and maintains associativity with all major CAD file formats. And not only with 3rd party formats, Inventor *talks* with AutoCAD, Revit, Fusion 360, and other products within the Autodesk family.

Why is this important? Here's Scott Moyses from designandmotion.net...

"There are plenty multi-CAD workplaces knocking about, whether that's because of mergers, legacy decisions in different departments or a heavy reliance on contract design staff. But this is also a big deal for the CAM users out there, those guys are receiving various data formats from all their clients every day... and they are also subject to in process change just like everyone else is."

With **AnyCAD** you do not just import the data, you associatively link it. So, opposed to a static component sitting in your design, you have a component that updates with changes to the original native file.

A different division uses Solidworks? No problem – associatively use it in your assemblies

Your customer uses Solid Edge? No problem – export to the Solid Edge format

The vendor is supplying data in 2D AutoCAD? No problem – underlay it into a sketch and build the 3D model

Contractor needs your model for their Revit BIM? No problem – exchange the data smartly

AnyCAD allows you to effectively leverage non-native files and is built into Inventor.

Holler, If You Hear Me!

Non-Native Data

At the heart of AnyCAD is the ability to import non-native files. The type of file determines the available options.

What does this include?

- 3rd party files from CATIA, NX, Pro-E/Creo, Rhino, Solid Edge, and Solidworks
- Neutral formats IGES, JT, Parasolid, OBJ, SAT, STL, and STEP
- IDF
- Autodesk Alias

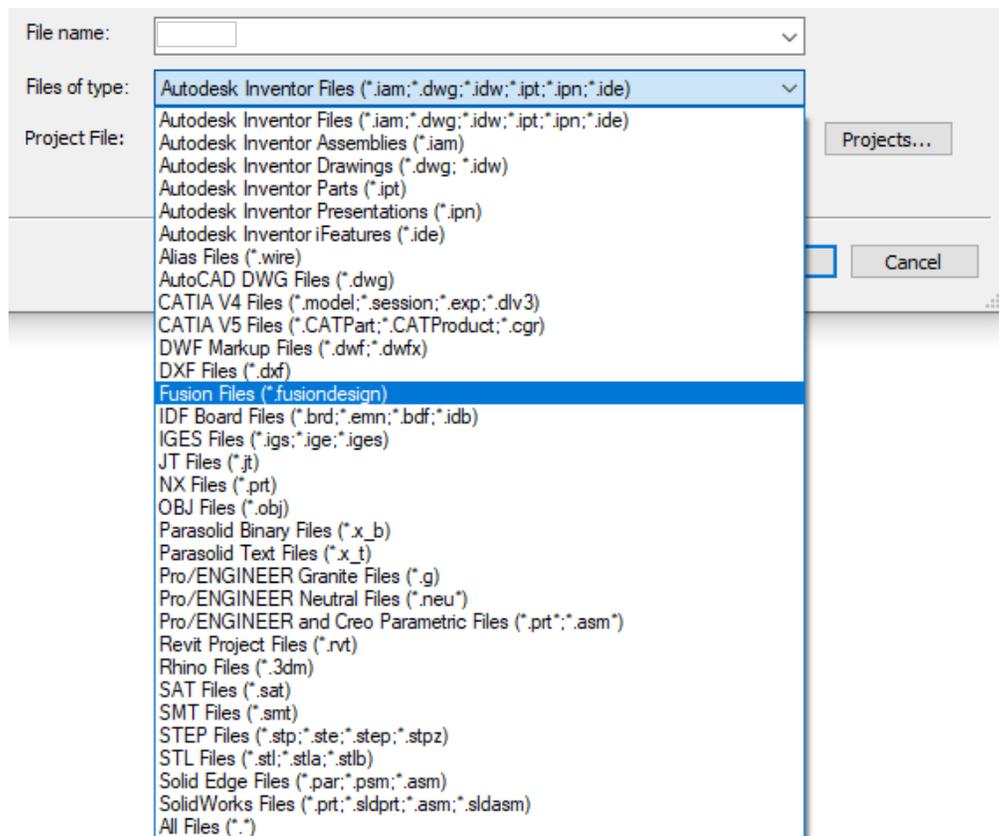


Figure 1: Inventor Open Files of Type

Convert vs Reference

Utilizing **AnyCAD**, you decide to either import the 3rd party model - converting it to Inventor parts and assemblies or use a Reference Model to establish a link to the 3rd party model. With the reference, as the 3rd party model changes, the changes reflect within your Inventor model.

A **Referenced Model** behaves as an imported model. You can apply assembly constraints, include it in weldment operations (Welding, Machining, etc), and assign iProperties. The referenced model has Mass Properties, including mass and volume. The model appears in the Bill of Materials and is checked-into Vault.

You can open or import model files from other CAD systems. You can also place part and assembly files as components into new or existing Inventor assemblies.

How do I decide which option to use?

- Does the model represent a purchased part that we are not modifying? **IMPORT...** insert it and forget it. Consider them as content center components.
- Is the model from the customer or supplier and we are anticipating changes? **REFERENCE MODEL...** why rebuild/redo when you can just have it update with changes?
- Do we need to modify the model? **IMPORT**

When opening or placing a 3rd party CAD file the following dialog appears.

- **Reference Model:** maintains the link to the selected file enabling the model to update as the model changes
- **Convert Model:** converts the model into Inventor parts and assemblies, the resultant model is not linked to the original

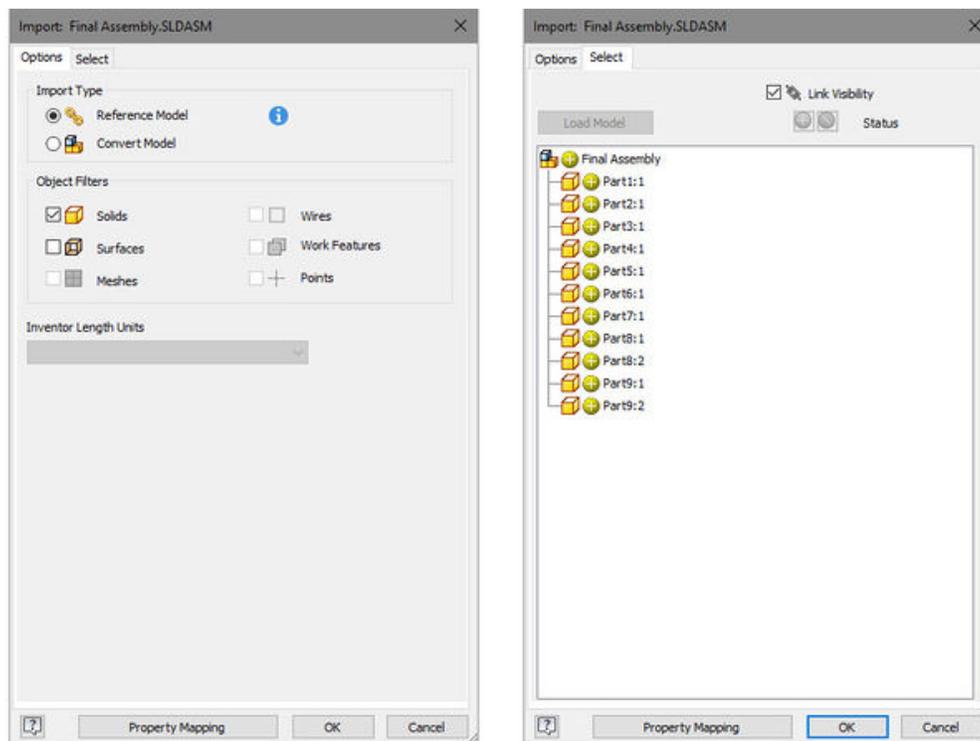


Figure 2: Import Dialog

Depending on the type of file, the **Reference** option may be disabled.

Use the **Object Filters** to specify the type of geometry to import.

- **Solids** imports solid bodies and water-tight stitched shells as individual solid bodies
- **Surfaces** imports surface bodies
- **Meshes** imports meshes. Mesh data is for visualization purposes only
- **Wires** import wires
- **Work Geometry** imports the desired work geometry

Use the **Inventor Length Units** to specify the type of Inventor length units for the imported geometry and parameter values. The unit value selected only changes the length units for the new document.

Select **Reduced Memory Mode** only if you are translating a large data set and expect additional memory will be required to complete the operation. This setting increases memory capacity and decreases performance. **Reduced Memory Mode** minimizes memory consumption by saving each component to disk during the import process.

When converting the model use the **Assembly & Part Options** to specify how to import solids into the file. Assembly Options:

- **Assembly** preserves the source structure
- **Multi-body** part imports an assembly as multiple solid bodies into a single part
- **Composite Part** creates composites from the imported levels, layer, or groups.

Part Options:

- **Composite** imports the assembly as a single composite feature in the part environment
- **Individual** imports the assembly as a single composite feature in the part environment
- **Stitch** (IGES and STEP files only) stitches several edge-matched surfaces or faces together

The **Update icon** within the **Quick Access toolbar** highlights and the lightning bolt icon appears within the browser indicating the original imported file has changed since it was initially referenced into Inventor. Select the **Update icon** or *right-click* on the item within the browser to load the revised version and incorporate the changes.

Use the **Select** tab to select the file or bodies to import/reference. If you see a yellow + Inventor includes the selected object. Use the grey \ to exclude the item from import.

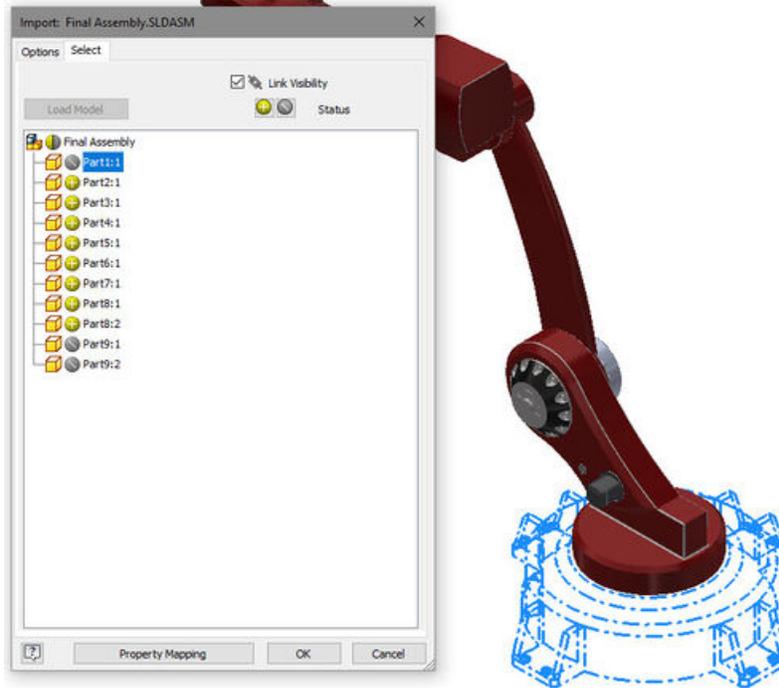


Figure 3: Select tab

Use **Property Mapping** to map properties from CATIA, Pro/Engineer and Creo parametric, NX, STEP, Solid Edge, and Solidworks to standard Inventor properties.

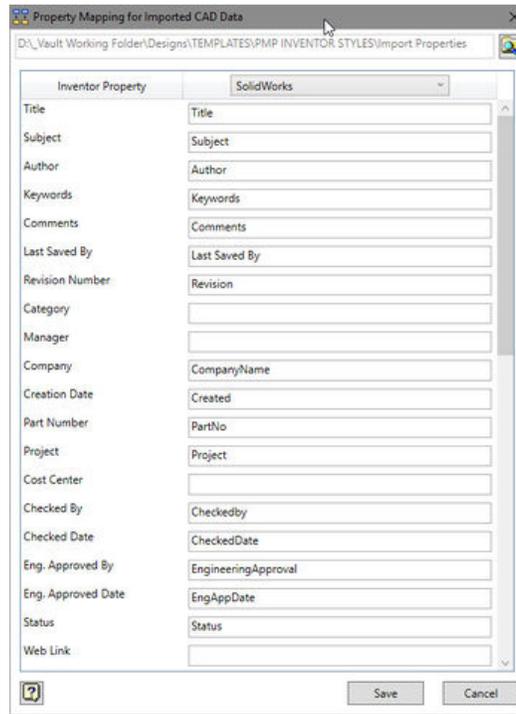


Figure 4: Property Mapping

Inventor from the Future

AnyCAD allows you to work with versions of Inventor files that are later than the version you are working in.

For example, you are working with Inventor 2019 and your customer sends you a 2020 file. You can place this component into your 2019 assembly.

If you happen to migrate to a newer version of Inventor, the future version references are migrated to standard Inventor place relationships.

Mesh Data

A **3D Mesh** is a 3D model built from a series of polygons. **Mesh elements** are typically triangles or quadrangles and allow for faster processing than other techniques. Systems like 3DS Max, Maya, and Blender 3D generate meshes as do processes generating 3D models from point clouds. In addition, the neutral formats STL and OBJ typically contain mesh information.

Note, the exported resolution of the geometry manages the accuracy, display, and interpretation of the data by annotations.

Inventor does not allow for STL or OBJ to be inserted as a reference, however, use mesh geometry as spatial references for designing components. Within Inventor use the mesh geometry to:

- add work features
- Inspect (measure)
- create surfaces and faces
- create 2D sketches and project mesh edges/vertices into 2D sketches
- constrain geometry (Assemble, Joint, and Constrain)

Fit Mesh Face creates surfaces from selected mesh facets. You can specify the surface/face type.

DWG Underlay

You can create 3D Inventor parts associative to source 2D geometry created in AutoCAD. The 3D part created in Inventor associatively updates when the 2D geometry changes in AutoCAD.

Use the **Import** command (3D Model tab) to insert an AutoCAD DWG file as an **associative DWG underlay**. When updating the original DWG file in AutoCAD, the corresponding Inventor files update with changes to the geometry size or position.

After selecting the drawing, you select a plane, planar face, sketch, or sketch curve to set the plane for sketch orientation. Finish the insertion by selecting the desired insertion point. Inventor pins (grounded) the underlay to the selected location.

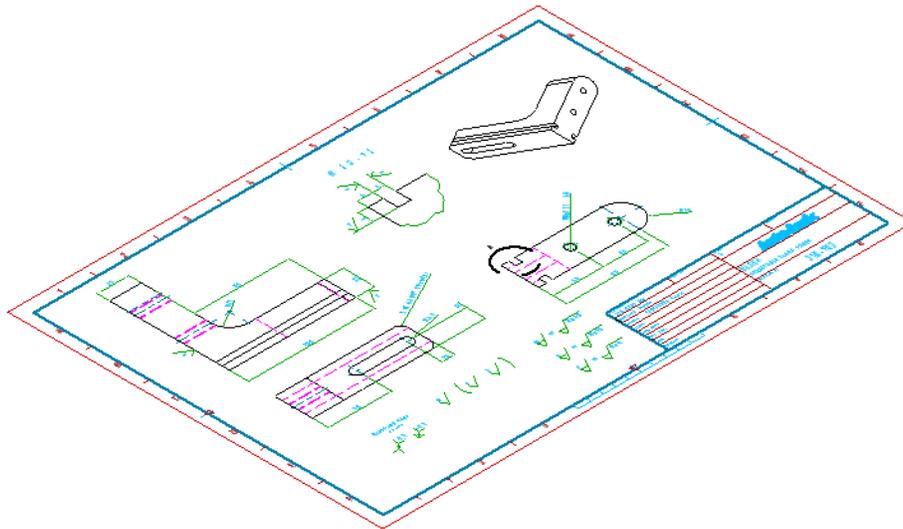


Figure 5: DWG Underlay

In an assembly file, you can use the **Joint**, **Constrain**, and **Assemble** to create relationships between the DWG underlay geometry and a part.

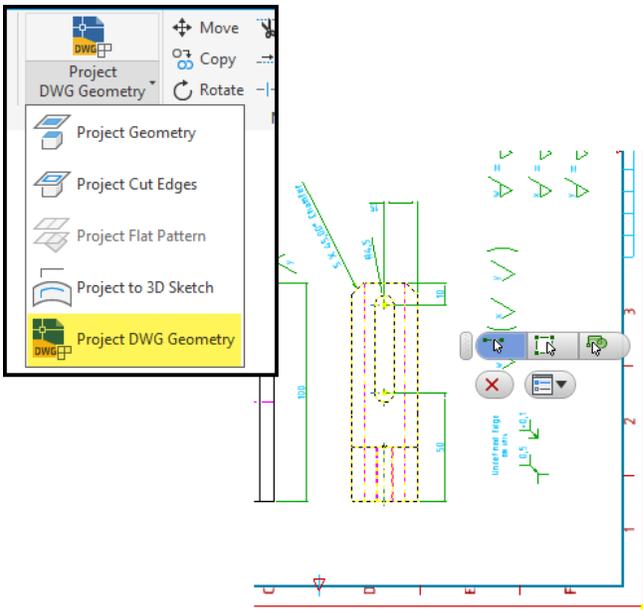


Figure 6: Project DWG Geometry

To utilize the underlay to generate 3D geometry create a sketch and use the **Project DWG Geometry** to copy the geometry into the sketch. Add additional geometry as required. Then utilize Inventor's modeling features to build the 3D model.

Use **Project DWG Geometry** to project single geometry elements, polylines, open or closed loops, and DWG blocks. Then you can use the projected sketch geometry to create associative model features.

In an assembly file, you can project DWG geometry in another part that is also open for edit.

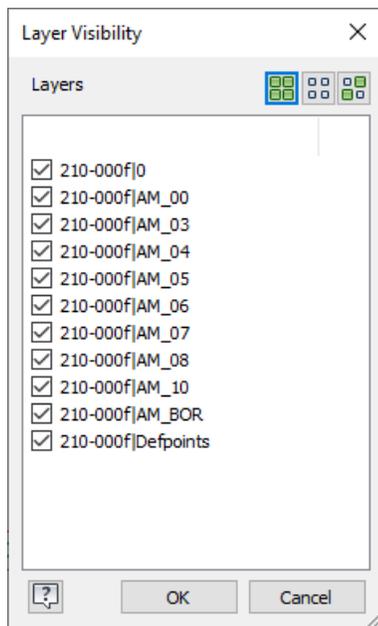


Figure 7: Layer Visibility

Right-click the AutoCAD files in the browser to open the **Layer Visibility dialog**. You can select the individual layers you want to display or control the layer visibility for the whole DWG.

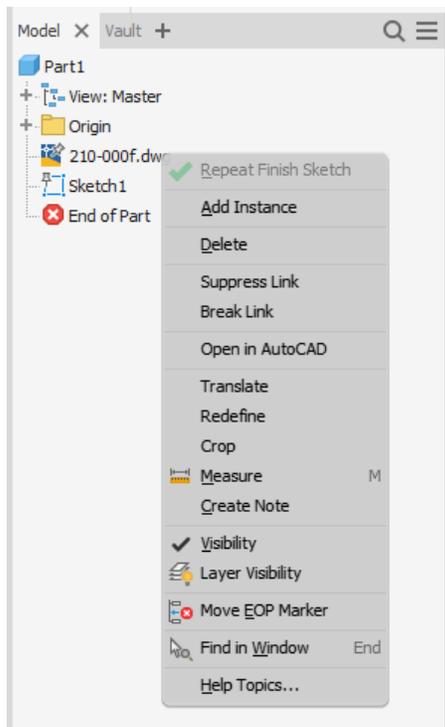


Figure 8: Underlay Right-click Menu

Use the browser right-click menu to access features for working with the underlay:

- **Add Instance** – inserts another instance of the DWG
- **Suppress Link** – temporarily turn off DWG changes updating the Inventor model
- **Break Link** – severs the link to the DWG, meaning changes to the drawing do not reflect within Inventor
- **Open in AutoCAD** – launch AutoCAD and open the drawing
- **Translate** - reposition the DWG data in the part file by using Triad, either moving or rotating with a specified value.
- **Redefine** – change the plane and insertion point for the reference
- **Crop** – initiate crop to rectangularly select a portion of the underlay to keep
- **Visibility** – toggles the visibility of the underlay on and off

Inventor Drawings (DWG) in AutoCAD

Inventor provides two formats for detailing your designs as drawings, IDW and DWG.

The advantage of using DWG is you can open Inventor DWGs natively with AutoCAD (and TrueView) without any translation or exporting. Within AutoCAD you cannot change geometry, but you can annotate and detail the Inventor views. You can also add geometry, which will be visible when the drawing is opened in Inventor.

Inventor DWGs include the ability to use DWG blocks, whether created within AutoCAD or Inventor.

As a bonus, every drawing view is available within the Design Center for insertion as a block. If the Inventor view changes, use Redefine to update the block within your AutoCAD drawing.

The AutoCAD DWG Launcher understands the differences between Inventor DWG and AutoCAD DWG and opens the appropriate application (when opening drawings from Windows Explorer)

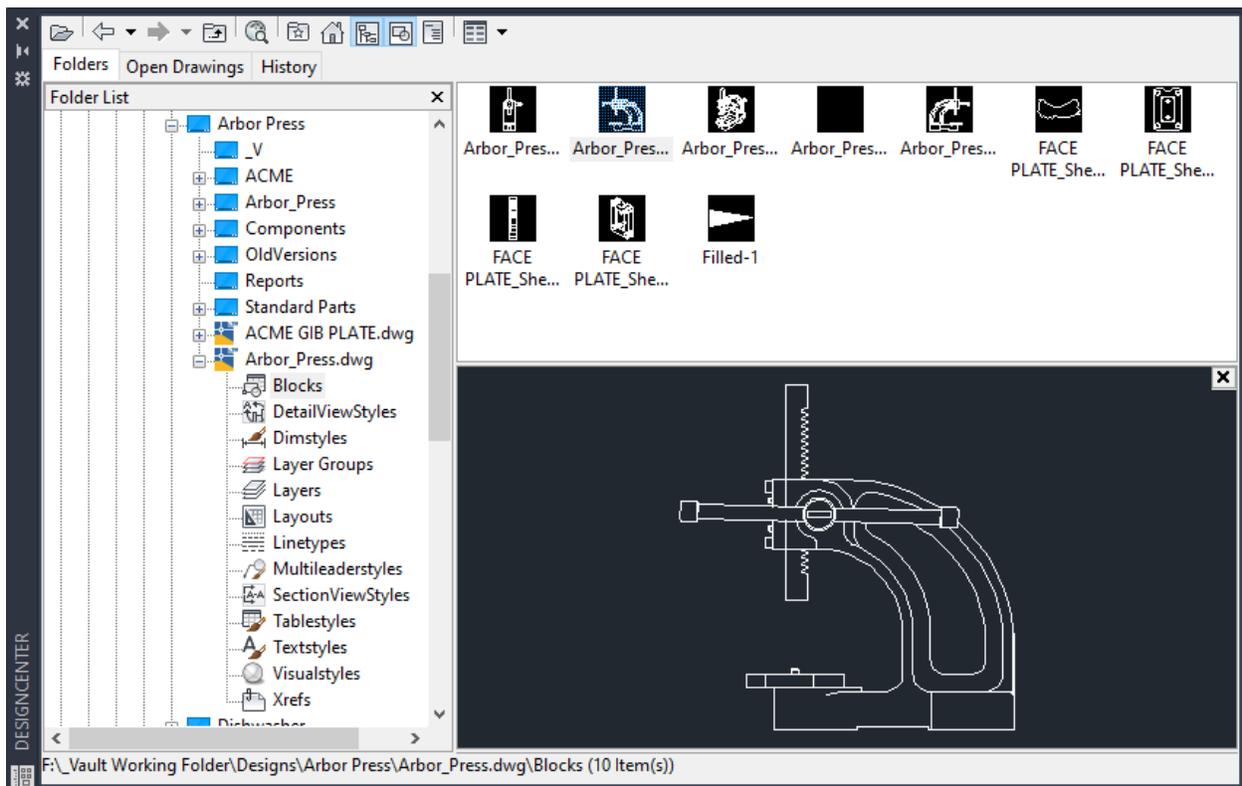


Figure 9: Inventor Views in the Design Center

IDF

The build-in **IDF Translator** transfers PCA data from your PCB layouts into Inventor (Professional). The IDF acts as the medium to transfer the designer's board layout (with components) to the mechanical design in Inventor. The IDF format consists of board/panel files as well as library data:

With Inventor, either **open** the IDF as a new assembly or part document, or **place** the component in an existing assembly. Within the dialog, use the Files of type drop-down to select IDF. Note this is an import, not an associative reference link.

Within the Import IDF Options dialog, view the summary data and then select the type of document to create:

- When imported as an assembly document, the board components translate into multiple parts contained within a new assembly.
- When imported as a part document, the board components translate into the sketches and extrusions comprising the part model hierarchy.

The advantage to importing IDF data as a single part is that it simplifies file management by reducing the number of files.

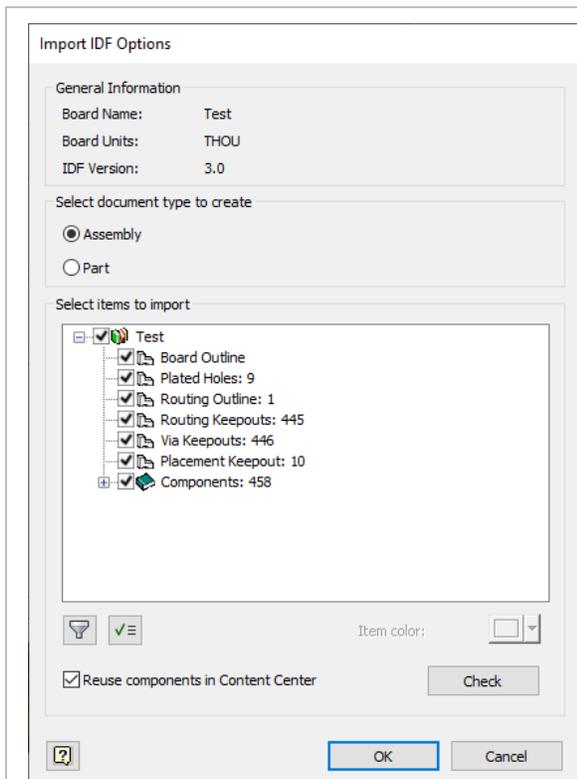


Figure 10: Import IDF Options

Select the items to import, using the checkboxes.

Each component is assigned a colour. You can adjust the colour of the entire board, other outlines, all components, or individual components as needed. Select the item and click the Item colour arrow to select a different colour from the palette.

Use the **Filter Settings** to filter:

- **Nonplated Holes** – filter by diameter
- **Plated Holes** – filter by diameter
- **Components** (resistors, chips, etc.) – filter by height
- **Outlines** – filter by type (Board cutouts and Keep-out areas)

By default, if you have published the IDF components to the Content Center library, Inventor will reuse these parts from the library instead of creating new parts

To identify any components that are not already in the Content Center library select **Check**

Export

Inventor supports many file types for export. Use **Save Copy As** or **Export** to select the desired **Save as type**, the name, and the location.

The difference between **Save Copy As** and **Export** is that *Save Copy As* provides all the options for the active file type, whereas *Export* filters the list based on the selected category.

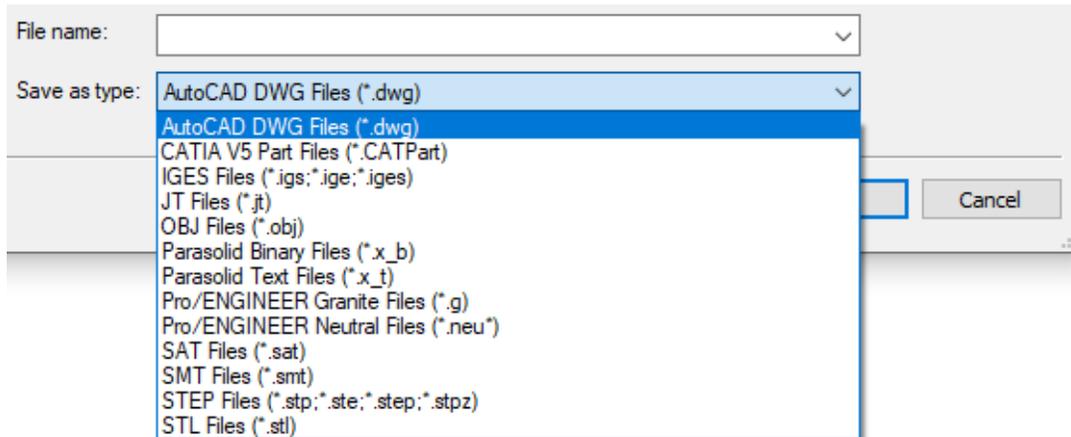


Figure 11: Save as type

Exporting from a **part** (IPT) or **assembly** (IAM) includes saving to an image (BMP, JPEG, PNG, TIFF), 3D PDF, PDF, DWG, DWF, and CAD Format (Neutral and 3rd party formats).

Exporting from a **drawing** (DWG or IDW) includes options for images (BMP, JPEG, PNG, GIF, TIFF), PDF, DWG, DWF, and DXF

The Tower of Power, Too Sweet to be Sour

AutoCAD Documentation

Using AutoCAD's layout tools, you can generate base views, projected views, section views, and detail views of existing solid and surface models, **as well as models created with Autodesk Inventor!**

The views remain associative to the originating objects, meaning changes to the models propagate to the drawing views as well.

The Base view, as the name suggests, sets the base, or initial, view of the model within the layout. This view becomes the base which you create other views from. To build a base view, use the File option to select an Inventor file.

With the views created in AutoCAD the full suite of dimensioning and annotation tools are available for detailing the drawing.

AutoCAD Mechanical Inventor Link

Within AutoCAD Mechanical use the Inventor Link feature to associatively link an Inventor model to your AutoCAD Mechanical drawing as a 3D model within model space. With the link established generate 2D drawings and use AutoCAD Mechanical's dimensioning and annotation tools to detail the Inventor model.

The link extracts parametric dimensions, iProperties, and thread/hole information:

- The parametric dimensions are available for display on the drawing views
- iProperties are available to populate the bill of materials / parts list
- Use AMNOTE to intelligently annotate the thread and hole features contained in the model

Images (Decals and Sketches)

Inventor's **Decal** feature using an image and wraps it to selected model faces.

From within a sketch, insert the image. Inventor supports BMP, JPG, and PNG. By using the Link option, changes to the image reflect within your Inventor model.

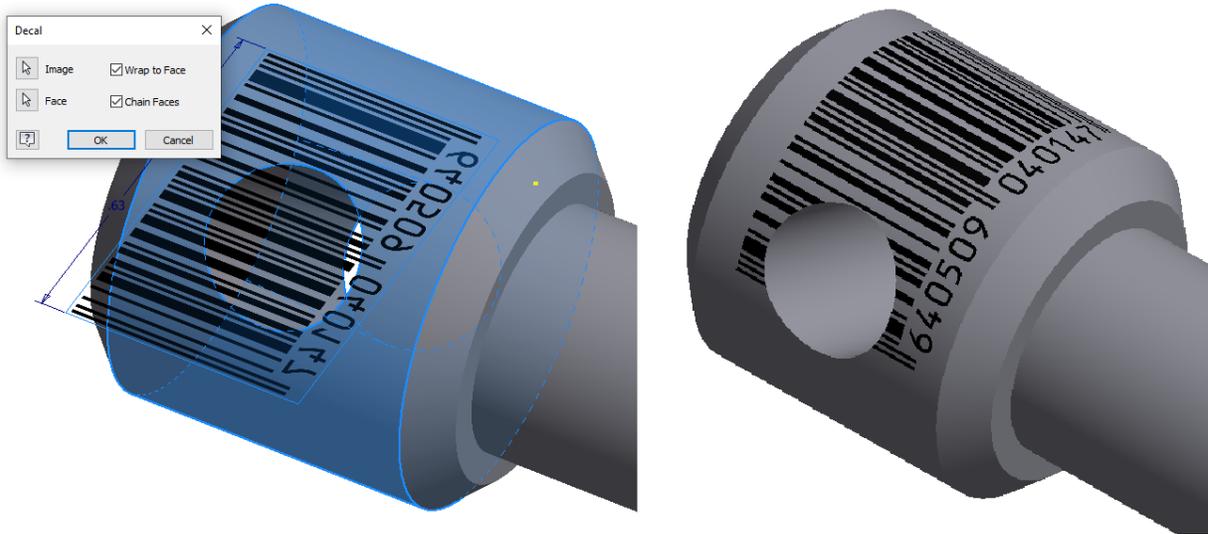


Figure 12: Decal an Image

You can position the image and manipulate its size using sketch dimensions and constraints.

Point Clouds

As a large collection of points, **Point Clouds** represent existing structures or the surfaces of objects. Each point is defined by an X, Y, Z coordinate and are acquired by a 3D scanning device. **Point clouds** will typically contain thousands, if not millions of points to represent the object correctly.

Point clouds are derived from the raw data obtained by 3D scanning an object. Inventor does provide tools for processing raw data. Therefore, before attaching a point cloud, process and index the raw data using **Autodesk Recap**. Many of the Autodesk Industry Collections include **Recap Pro**.

Recap converts the raw point into scan files (RCS files) or project files (RCP files) that reference multiple RCS files. Additionally, Recap segments the point data into groups.

Inventor allows for the attachment of point clouds. Use the point cloud as a guideline, providing real-world context. Using **Attach Point Cloud** browse for and select a point cloud scan (RCS) or project file (RCP). Pick a point within the graphics window to place the point cloud.

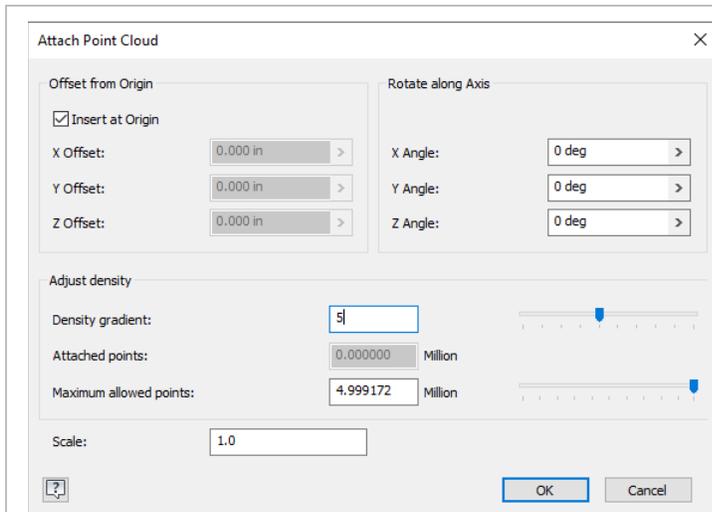


Figure 13: Attach Point Cloud

Use the **Insert at Origin** option to locate the cloud at 0,0,0 or use the **X, Y, Z offset** values to move the cloud to a point relative to the origin.

Use the options to adjust the scale and rotation.

Use the **Density gradient** to adjust the point cloud's density, increasing or decreasing the number of points displayed and available within Inventor.

With the point cloud attached, a node appears within the browser. *Right-click* on the node and select **Edit Attached Data** to relaunch the Attach Point Cloud dialog and adjust the inserted data.

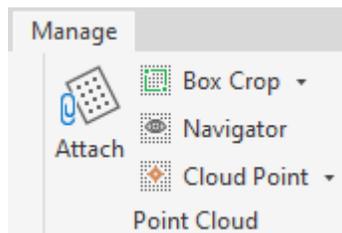


Figure 14: Point Cloud Panel

Box Crop removes a selected portion of the point cloud. Use **Uncrop** to restore the point cloud to its uncropped state.

Use the **Navigator** to manage the visibility of point clouds and their groups.

Cloud Point creates a work point coincident with a selected point cloud vertex.

Cloud Plane extracts (creates) a work plane from a series of selected cloud vertices.

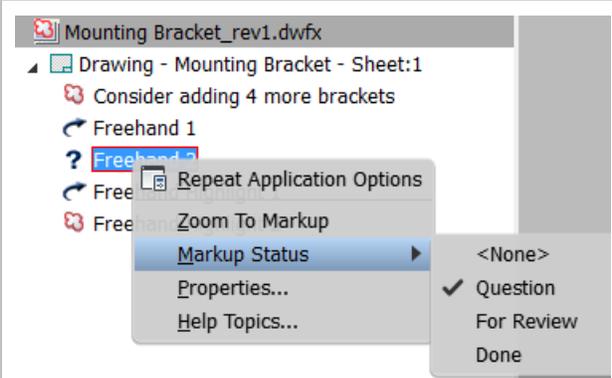
To 3DS Max

3DS Max imports Inventor parts (IPT) and assemblies (IAM) natively, without the need to use an intermediate file. The import is a conversion, meaning with changes to the model within Inventor means a new import into 3DS Max.

If the Inventor models utilize materials from the Shared Autodesk Material Library, the materials translate into 3DS Max.

DWF (Markup)

Within Inventor open a marked-up DWF/DWFX to overlay the markups over the 3D model.



The **Markups browser** displays the markup set in the tree view. You select the markups to view, see their status, and adjust the status.

To save the changes to the markups right-click the DWF root node and select **Save Markup History**.

To republish the DWF markup, right-click the DWF root node and select **Republish Markup Sheets**.

Figure 15: Markup Browser

Revit

Inventor-to-Revit

Typically, your Inventor models contain more information than required within the Revit BIM model. The *Inventor-to-Revit workflow* removes manufacturing or non-required geometry, creating a construction level of detail Revit Family (RFA). You want to remove intellectual property (IP), details not required by the consumer (i.e. Revit users), and non-required features that overly complicate the model or do not significantly contribute to the visual presentation.

If wanting to create a **non-configurable assembly**, start by using Inventor's shrinkwrap tools to remove unnecessary components and/or features. This combines the components into a simplified singular part.

To generate the RFA (non-configurable or configurable):

- use the **BIM Content Environment** to generate **MEP connectors**, allowing for smart placement of the published model within the BIM model.
- use the UCS tool to specify product placement positioning
- use the Author Building Component tool to set product metadata, specify the orientation, and set the Omniclass title and number

TIPS:

- Label your shrinkwrapped substitute BIM. If you decide to publish the BIM model with Configurator 360 the substitute must be called BIM
- Add a custom iProperty *RevitFamilyTypeName* and set the value to the desired family type name within Revit.

The BIM Content environment authors and publishes components so that you can use them with AutoCAD and Revit. Both these applications contain purpose-built building design and construction tools for documenting MEP information.

Publishes 2D and 3D design data as:

- As Revit geometry (RFA)
- As Industry Foundation Classes (.IFC)
- As Autodesk Exchange (ADSK)

What is OmniClass?

"Construction Classification System (known as OmniClass™ or OCCS) is a classification system for the construction industry. OmniClass is useful for many applications, from organizing library materials, product literature, and project information, to providing a classification structure for electronic databases."

For more about OmniClass, go to <http://www.omniclass.org>

Revit Interoperability

Within Inventor use **Place Component** to import **Revit Projects** (.rvt) and place the model within your assembly. The imported model adds mass and volume to your assembly and is useable for constraining other components.

Fusion 360

Fusion 360 is a cloud-based CAD/CAM tool developed for collaborative product development merging industrial design, mechanical engineering, and machine tool programming into one software solution. Many consider Fusion 360 Inventor's little brother, as Inventor is the older (more mature) application whereas Fusion 360 is the up-and-coming design application.

Fusion-to-Inventor

Export Fusion designs for use in Inventor by selecting **Export** (File menu) and choosing Autodesk Inventor as the export format. This generates assemblies (iam) and part (ipt). After the conversion there is no link between the original Inventor model and the new Fusion native model.

Inventor-to-Fusion

Using the **Data Panel** within Fusion 360 you can upload an Inventor model (IPT or IAM) and Fusion uses its cloud translation process to translate (convert) the design into the native Fusion 360 format.

After the conversion there is no link between the original Inventor model and the new Fusion native model.

To ensure you have all the required Inventor details utilize **Pack and Go** (File menu) to collect the assembly and all its children components into a singular location. Use this copied dataset to upload into Fusion 360.

The Inventor-Fusion Connection

From within Inventor, use AnyCAD to associatively reference Fusion 360 file. Like other data types, either place the file as a **Reference Model** or use the **Convert Model** option to convert the model into an Inventor component. When placing the Fusion 360 file as a **Reference Model**, changes to the Fusion 360 file automatically propagate to your Inventor assemblies.

The sharing of data between Inventor and Fusion 360 requires the [Desktop Connector](#). This enables sharing the data between your local system (Inventor) and the cloud (Fusion 360).

In addition to the **Desktop Connector**, you require a **Fusion Team** account. Fusion Team provides a central workspace for sharing data.

Fusion >>> Inventor

From within Fusion 360 share the design with the appropriate **Team Hub**. Within Inventor, use Open or Place Component and browse to the **Fusion 360 Drive**. Select the Fusion 360 design (.fusiondesign) to open it or place it within Inventor.

Inventor >>> Fusion

To associatively use Inventor data within Fusion start by saving your Inventor model (and its children) via the **Desktop Connector** to your Fusion Team Hub.

From within Fusion 360, you can then open the Inventor model and the file is translated and linked. You will not have access to the original Inventor features (extrudes, revolves, etc.) but do have the ability to direct edit the geometry. You can place it into your Fusion assemblies and constrain other components to it.

And That's the Bottom Line

Shared Views

Utilize **Shared Views** to publish a visual representation of your model to the cloud and allow others to review and comment on your design. You are not sharing the model itself, but a viewable representation, protecting your model. This **Shared View** technology is common across the Autodesk product line and functions the same regardless the application used.

Inventor generates a shareable link, which you share with others to provide them access. The collaborator, the one you are sharing the view with, does not require Inventor or another CAD application as they view it within a browser using the [Autodesk Viewer](#). The **Shared View** is accessible on any web-enabled device, including desktops, tablets, or mobile device.

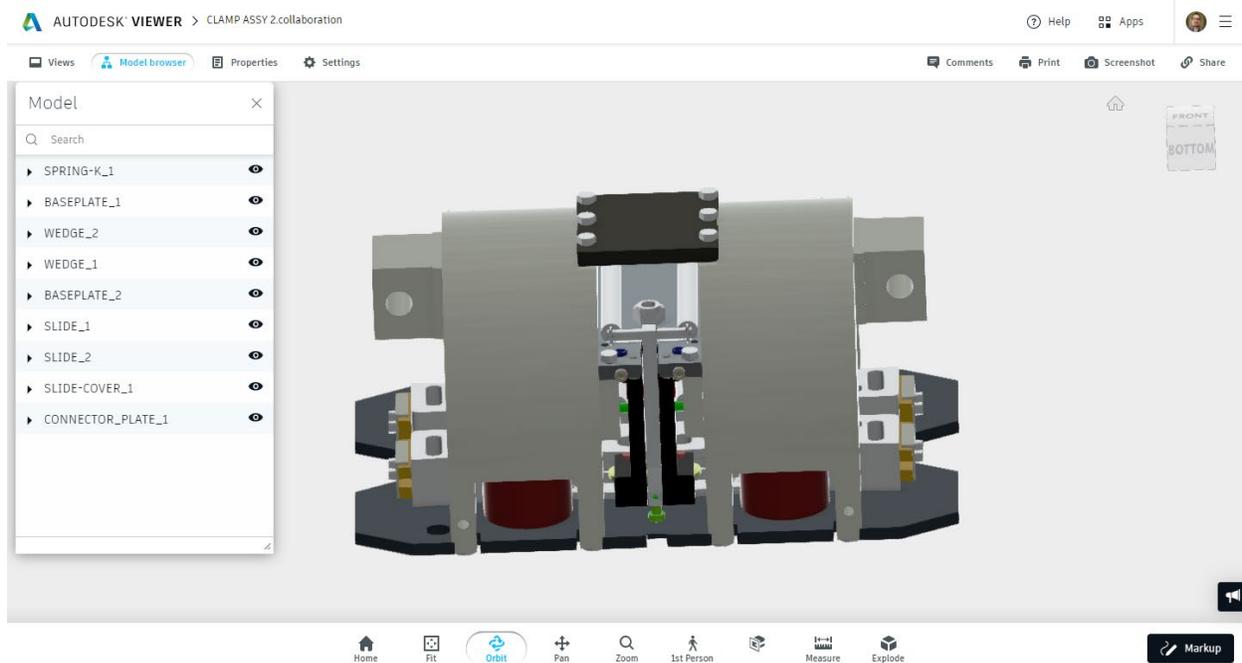


Figure 16: Autodesk Viewer

While accessing **Shared Views**, the collaborator can view, review, measure, comment, and markup on the drawing. The comments and markups added to the view are accessible within AutoCAD. The shared view automatically expires after 30 days, by default.

Prior to creating a **Shared View**, you must first log into your Autodesk account. If the feature is not available, your Subscription Administrator may have disabled it.

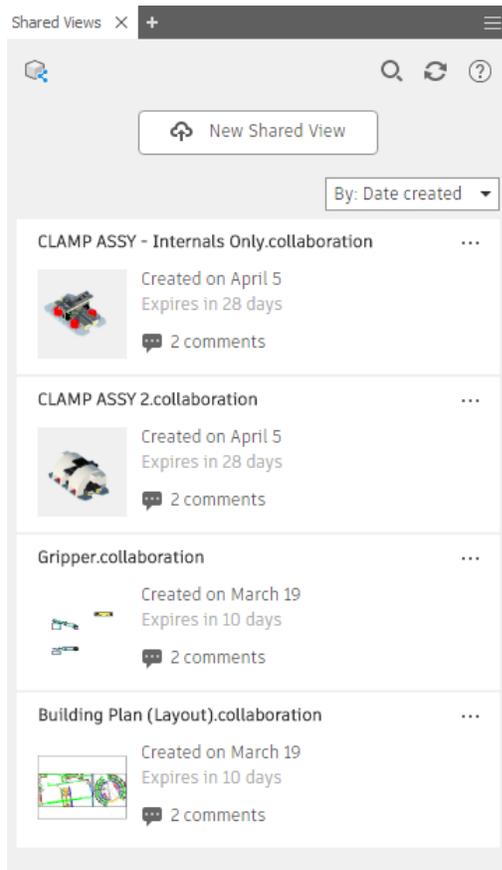


Figure 17: Shared Views Browser

Initiate the process from the **Share section** of the **File Menu** or from the **Collaborate Ribbon tab**. This is available for parts (IPT), assemblies (IAM), and drawings (IDW/DWG). This launches the **Shared Views browser**.

The **Shared Views browser** lists all previously shared views and allows you to create new shared views.

Within the **Shared View browser**, select **New Shared View** to generate a new view. In the **Share View dialog**, specify the name of the view. Within the options specify to hide Component Names and Part Properties.

Use the **Refresh** button to retrieve the latest comments and other information from the cloud.

Every view display as a thumbnail with information including the title, the date created, its expiration date, and the number of attached comments.

Use the options menu (...) on each view to perform various operations including:

- **View in browser** – launches the view in the Autodesk Viewer
- **Copy Link** – copies the link to the clipboard, allowing you to share with others
- **Extend** – resets the expiration date for another 30-days
- **Delete** – deletes the view, meaning shared links no longer works

PDF

PDF is the de-facto standard for the exchange of information, traditionally 2D but recently 3D models as well.

Inventor supports the export of parts, assemblies, and drawings to PDF (2D). When exporting the 3D model to PDF it captures the current view and component/feature visibility and provides no options for defining other export parameters.

When exporting to PDF from a drawing (DWF or IDW) use the Options to define the print range (sheets to include), the vector resolution, and settings to export all objects as black and remove object line weights.

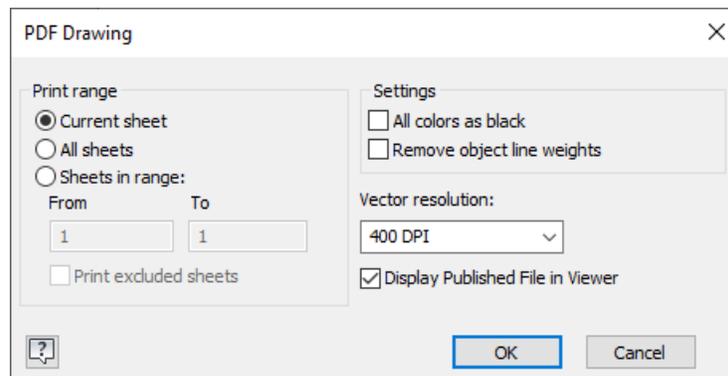


Figure 18: PDF Drawing Options

Prior to publishing the model to a 3D PDF create **Design View Representations** to effectively communicate the information. You should manage component/work feature visibility and the display of 3D annotations with the View Reps.

You will find **3D PDF** in the **Export section** of the **File menu**.

Within the **Publish 3D PDF dialog** specify:

- the **iProperties** to include
- the **Design View Representations** for inclusion
- the **Visualization Quality** – the higher the resolution the more details but also the larger the file size
- the **Export Scope** – either all components (regardless their visibility) or just the visible components within the View Rep.
- the **Template**
- the **File Output Location**

Use the **Attachments options** to generate and include a STEP file with the PDF, useful in providing a downloadable neutral format file with the PDF.

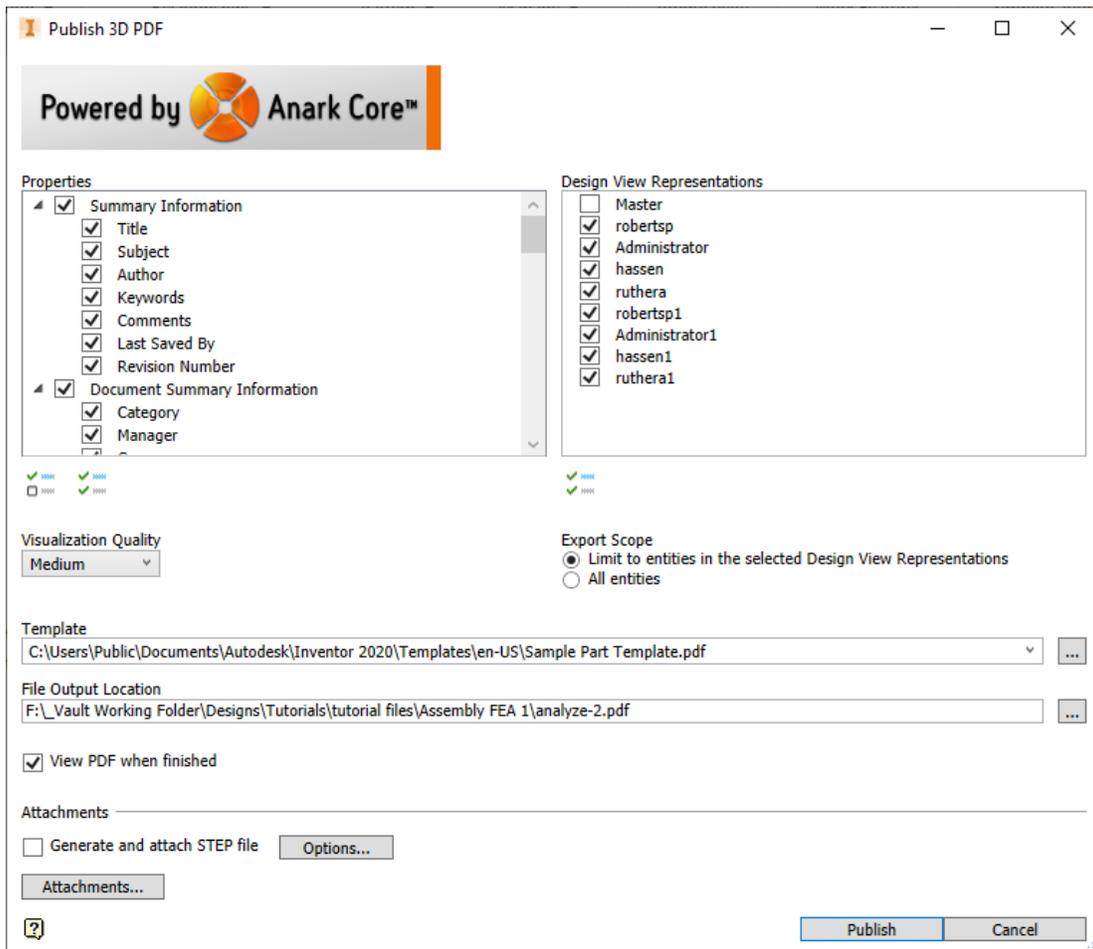


Figure 19: Publish 3D PDF

DWG/DXF

Use the DWG export feature to generate AutoCAD drawings, either containing 3D models (from parts or assemblies) or 2D geometry (from DWG or IDW).

When exporting to AutoCAD DWG from a part or assembly the options allow you to specify the type of information to include (solids, surfaces, and sketches) the DWG version.

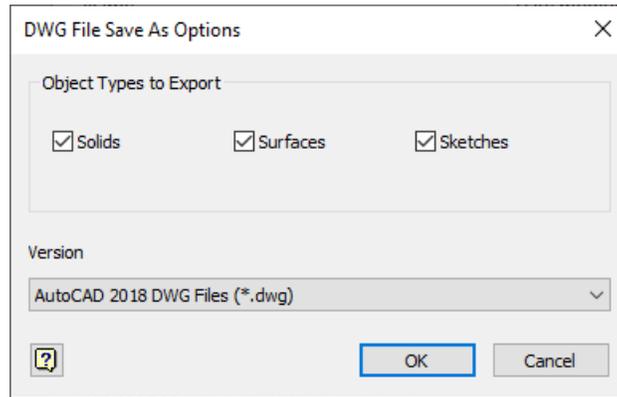


Figure 20: DWG Save As Options

When exporting from a drawing (DWG or IDW) you create an AutoCAD drawing (DWG) or a DXF. Use the Options to set the AutoCAD file version, the sheets to include, the data scaling, the visual fidelity, and an optional AutoCAD Template.

Note, DXF is only available from the Save Copy As dialog.

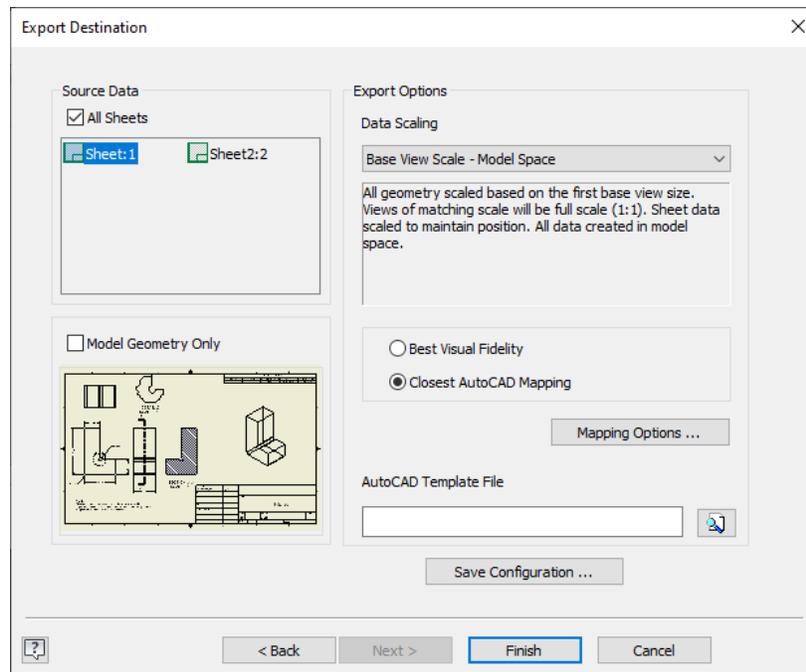


Figure 21: DWG Export Options

The exported DWG or DXF is not associative with the Inventor drawing, meaning subsequent changes within Inventor will NOT be applied to the AutoCAD drawing.

DWF/DWFX

Design Web Format (DWF) is a compressed, secure format created by Autodesk for the intention of sharing data with others. Publish drawings, parts, weldments, sheet metal parts, mesh geometry, and presentations to the DWF(x) format.

DWFX is the newest version. As it is based on the *XML Paper Specification (XPS)* it can be viewed and printed with the **Microsoft XPS Viewer**, embedded within Windows.

DWF and DWFX are highly compressed file formats more suitable for distributing data for review. In addition, file management for exchange is easier as opposed to worrying about multiple component files, the assembly, the drawing, and other information you create a singular DWF.

Reviewers do not require Inventor or another CAD application as they can use the free [Design Review](#). This program includes QVP (query, view, and print) tools in the form of mark ups, measuring, printing, and the ability to track changes.

From the Export section (File menu) select **Export to DWF**. The publish options include:

- **Express** publishes the current state of the file
- **Complete** publishes everything
- Select **Custom** to specify the content to include

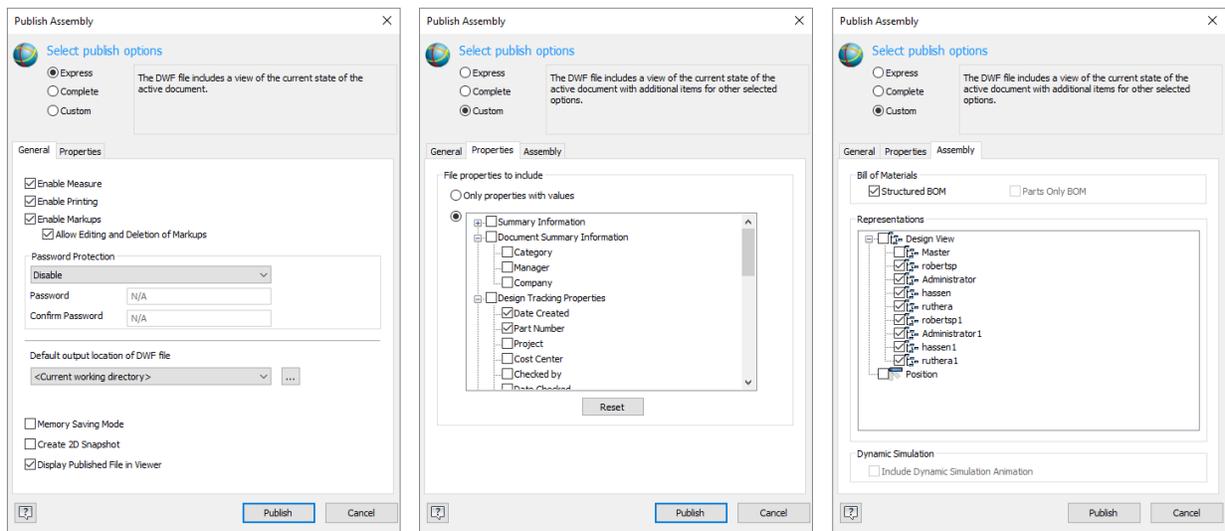


Figure 22: Publish DWF

Within the **General tab** specify the information to include. Use the **Properties tab** to select the iProperty information to publish. With the Custom publish option enabled, use the **Custom tab** to select additional information to include.