AUTODESK UNIVERSITY

PD19377

The Interoperable Inventor

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Learning Objectives

- Learn how to import 2D DWG data to build a new model
- Learn how to use Inventor AnyCAD to reference third-party models
- Learn how to import wire data from Alias
- Learn how to export information for various downstream applications
- Learn how to utilize the Suite tools to utilize other Autodesk software
- Learn how to collaborate with others using Autodesk A360
- Learn how to build your designs with HSM CAM and 3D Printing

Description

Autodesk Inventor acts as a medium to collect, use, reuse, and exchange data of different types and from many sources. In this class, we will explore the different types of data supported by Inventor software for both import and export.

Mike Thomas – Technical Services Manager, Prairie Machine & Parts Mfg.

Mike is the technical services manager for Prairie Machine & Parts Mfg. (PMP), and he is responsible for overseeing the company's technical operations and strategic technical growth. His primary duties include the ongoing support of critical computer systems and programs and the interactions between departments, and he is also responsible for providing effective systems and technology, and working with the general manager on the development and implementation of a cohesive strategic plan for the technical growth of the company. Mike is an active contributor to Design & Motion (www.designandmotion.net).

Kevin Ellingson

Kevin is a graduate of both the North Dakota State College of Science (Mechanical Design and Drafting) and Minnesota State University (B.B Operations Management) and has intimate knowledge of machine fabrication, design documentation, and CNC programming. Kevin is currently the owner and main operator of Mechanical Advantage, bringing his years of industry experience across multiple products and disciplines.

The Interoperable Inventor

Interoperability (noun: in-ter-op-er-a-bil-i-ty) is the ability of different information technology systems and software applications to communicate, exchange data, and use the information that has been exchanged.

It is 2016 and the days of working in a silo are long past. We need to interact with others and other software, both internally and externally to our organizations. We need to do this efficiently, but also smartly, as we have deadlines to meet and projects to complete.

Autodesk Inventor provides workflows to collect, use, reuse, and exchange data. Piggybacking on other products contained within Product Design Suite, Inventor can import and export 2D DWG, associative reference 3rd party 3D models, generate PDF, exchange (to-and-from) with Revit, collaborate within the A360 cloud, and share bill of material information with downstream applications.

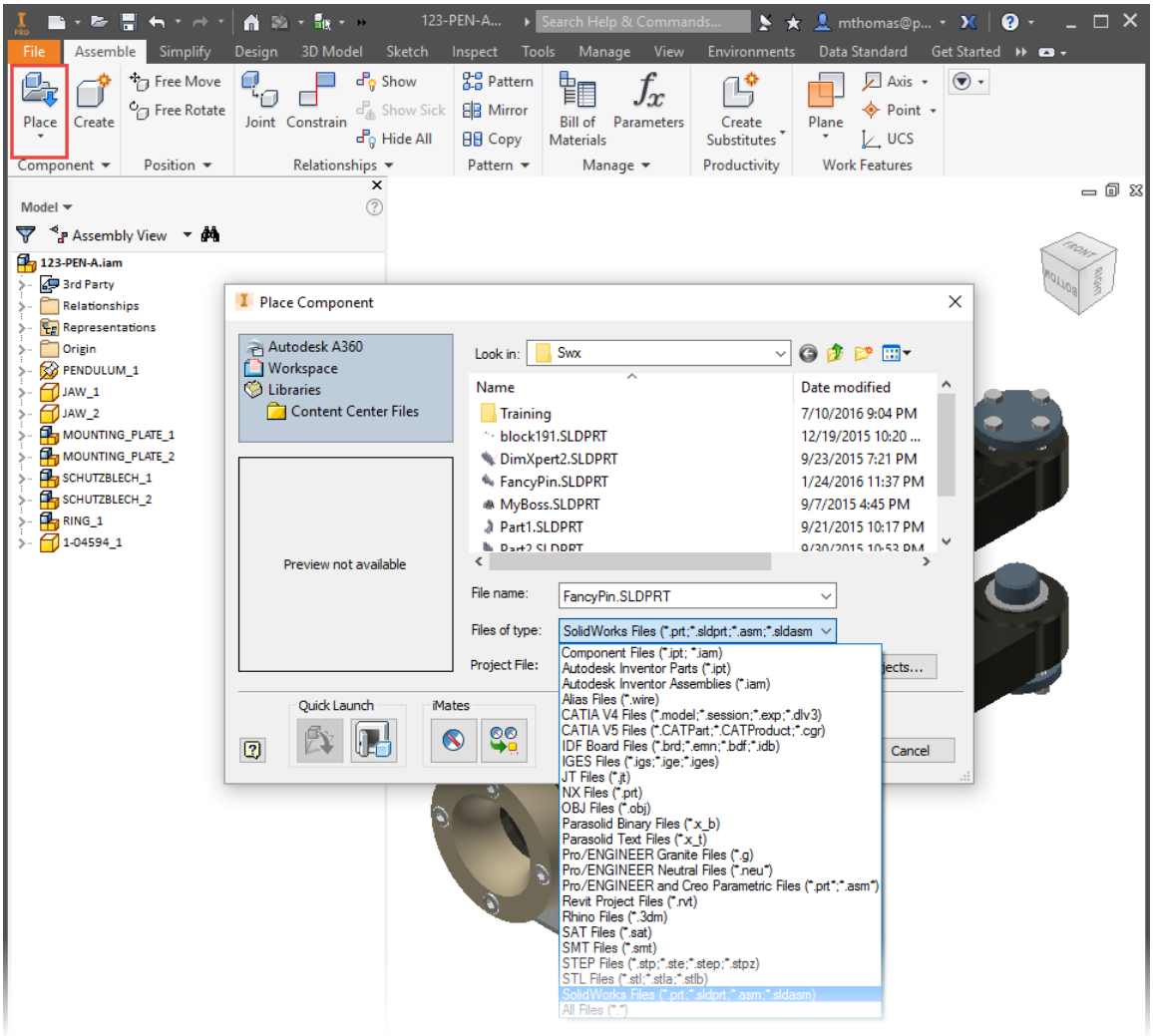
AnyCAD

Inventor can import 23+ different file types, but if you are using this data as a reference and have no intention of making changes, why go through the (sometimes painful) process of converting it into the Inventor format?

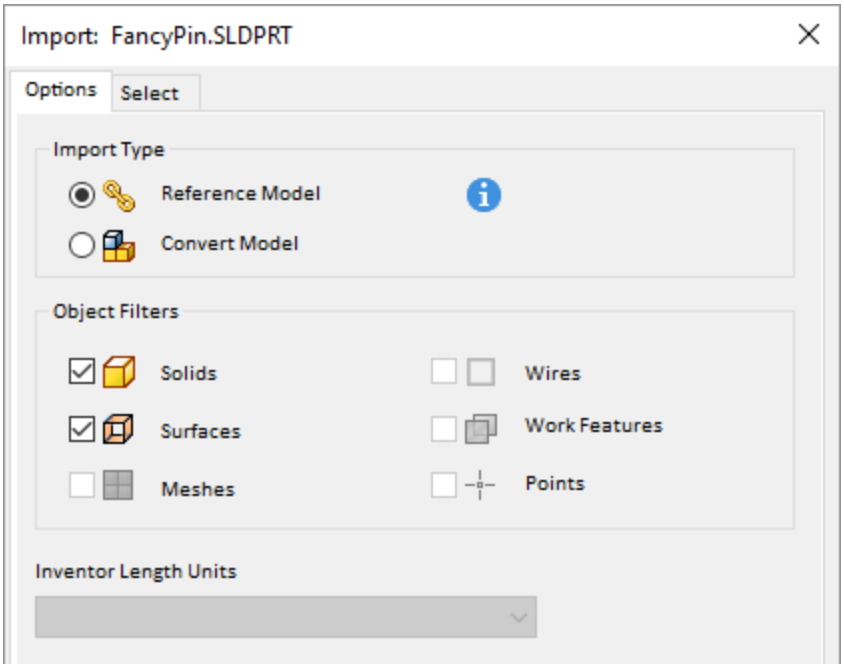
With the Inventor AnyCAD functionality, you can natively import AND maintain associativity to the attached data. With AnyCAD you open any of the supported file types, either parts or assemblies, straight into an Inventor part or assembly.

With a model attached as a reference, Inventor 2017 provides support for meta-data overrides via iProperties. What this means is that if you are not happy with the metadata of the associated CAD model you can tweak it within Inventor once it's linked. You even get to read the BOM of the source files via Inventor's Bill of Materials.

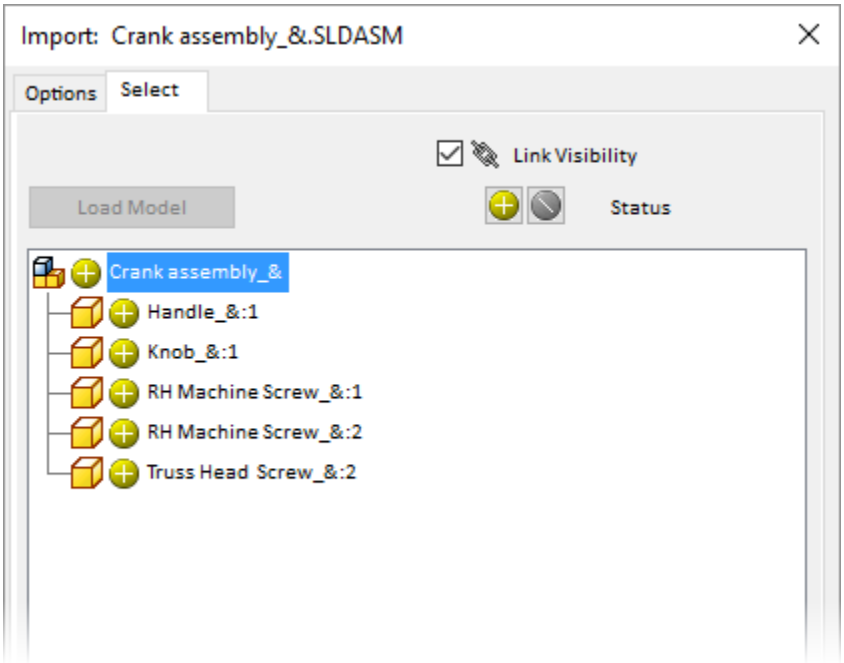
Use Place Component to select the component and insert it into the assembly. Use the Files of Type drop-down to pick the file type.

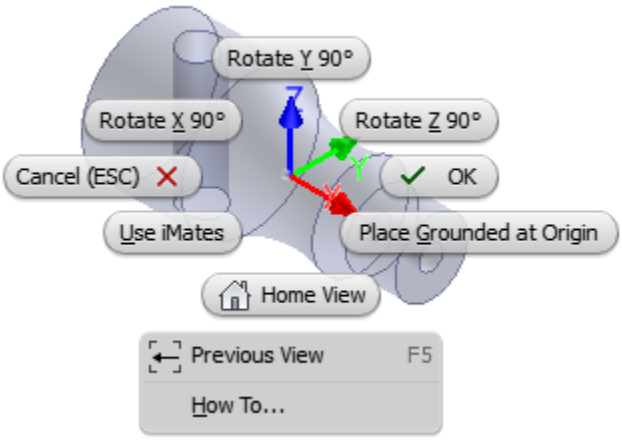


When the Options dialog appears, set the Import Type to Reference Model. Use the Object Filters to choose the type of data to import.



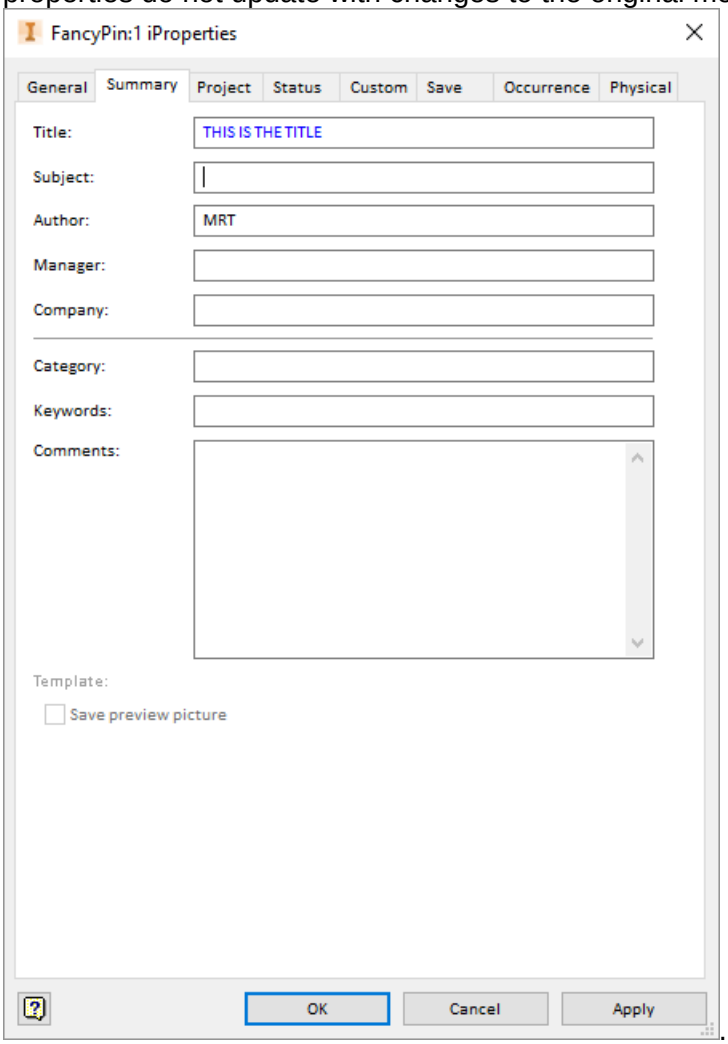
To manage the geometry imported activate the Select tab. After picking Load Model, the components contained within the assembly appear (this only works with assemblies and multi-body parts). Using the tree, you choose the components to include or exclude during the import. Alias models provide an additional option to imports objects as individual surfaces.





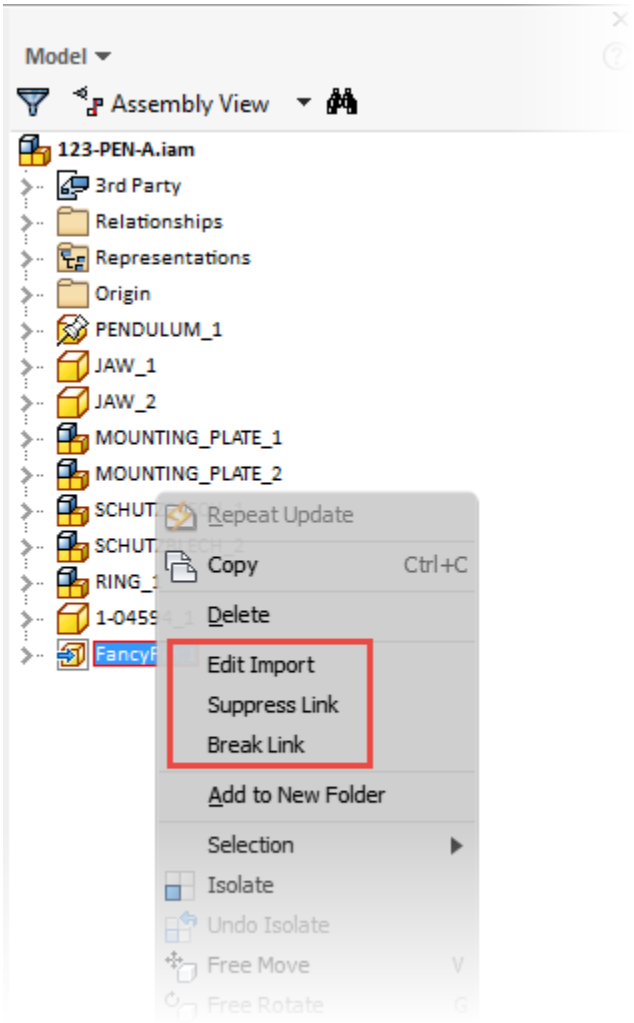
During placement Inventor provides all the same options as with placing native Inventor models.

Use iProperties or the Bill of Materials dialog to override the properties (meta data). Adjusted properties are shown in blue so you know which properties are overridden. Overridden properties do not update with changes to the original model.



When the referenced model changes Inventor shows a red lightning bolt in the browser, highlighting the component requiring updating. Selecting Local update from the main toolbar updates the assembly including these referenced components.

To work with the link, right click the component within the browser.



Use Break Link to sever the link to the model, meaning changes to the original model will never update within Inventor.

Suppress the Link if you only want to temporarily prevent updates to the referenced model. Unsuppressing restores the link, meaning the model will update again with changes.

Use Edit Import to access the Import dialog, to make changes to the import settings.

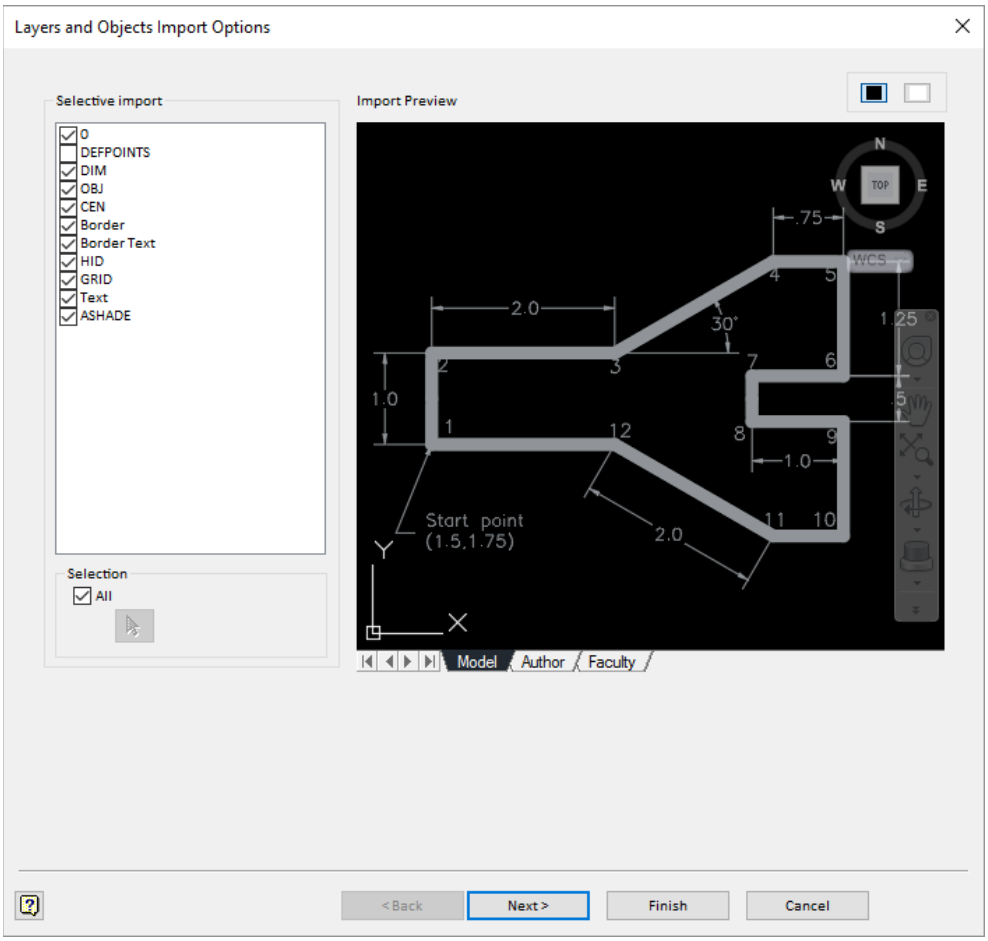
DWG (& DXF)

2D is still alive and well, and still used for collaboration and data exchange. For example, want to get a profile cut by a laser, waterjet, or plasma? More than likely you're sending a DXF.

Import DWG to use in Sketching

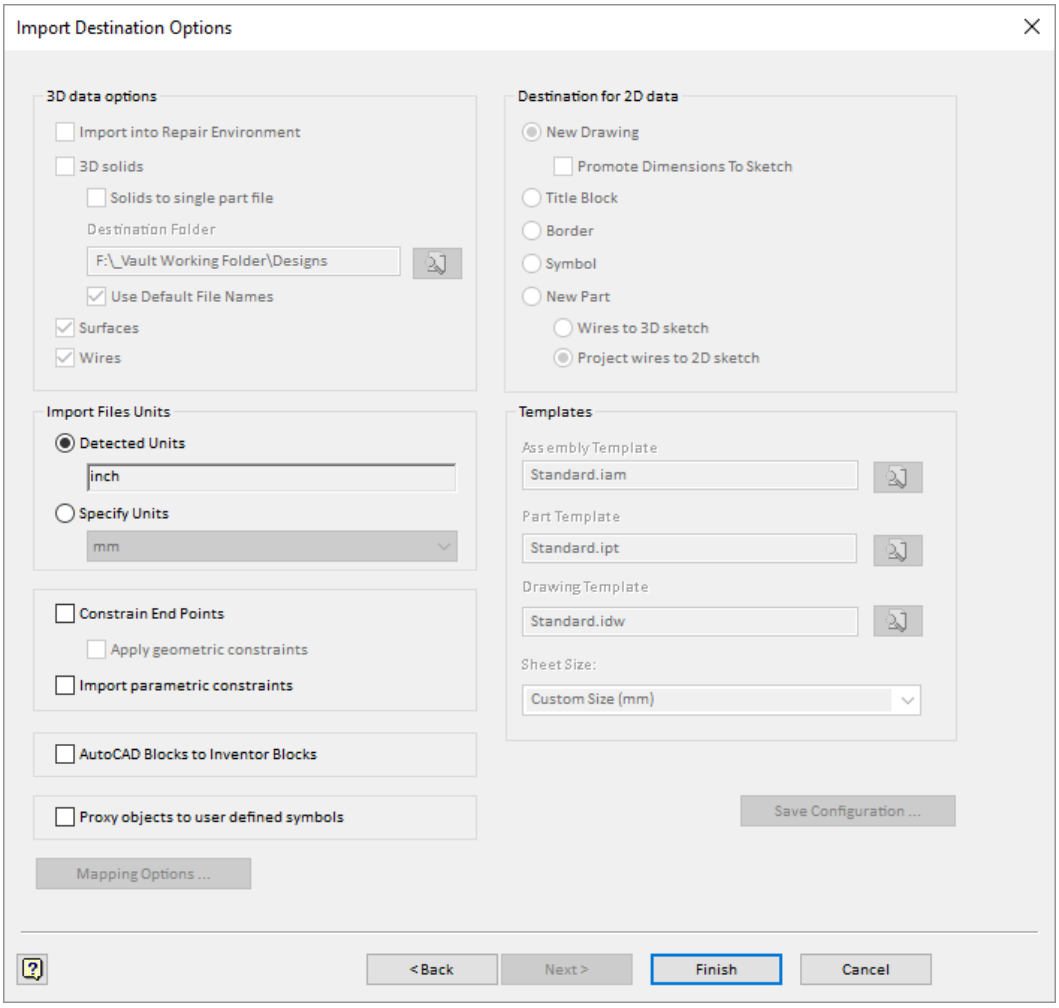
With a sketch active, select Insert AutoCAD File. Browse and select the DWG you wish to import.

The first step is selecting the geometry to import. Do this by selecting the layers that contain the geometry. If required, disable the All option and select the desired objects in the Import Preview window.



Click Next to move to the next step.

To override the detected units, choose Specify Units and select the desired units.



Four options are available for managing the imported geometry.

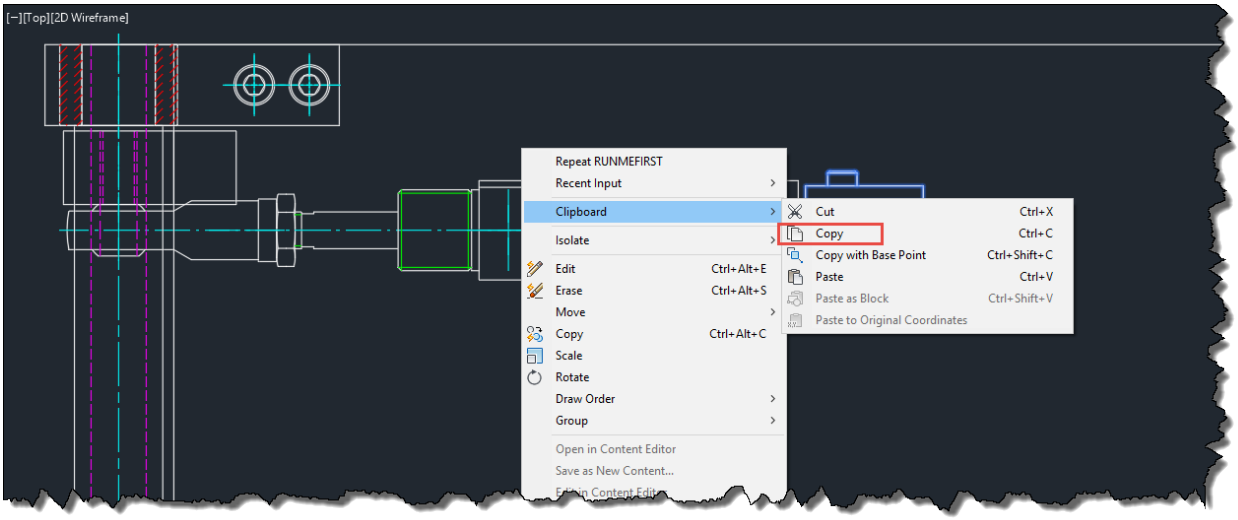
- Constrain End Points – applies coincident constraints to all connected points. When enabled additional enable Apply geometric constraints to apply constraints like horizontal, vertical, and others.
- Import parametric constraints – if parametric constraints are applied within AutoCAD, this feature transfers them to Inventor constraints.
- AutoCAD Blocks to Inventor Blocks – the AutoCAD blocks are converted to Inventor blocks opposed to being exploded into their base elements
- Proxy objects to user defined symbols – converts proxy objects (the objects AutoCAD couldn't recognize) into symbols in Inventor. If disabled proxy objects are ignored during import.

Click Finish to import the geometry.

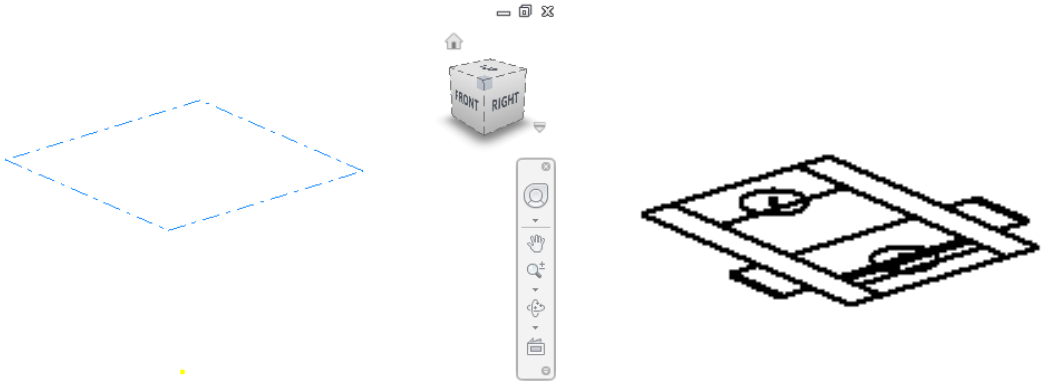
Copy & Paste to Insert AutoCAD Geometry

An alternative to using the import sketch wizard is straight-up-copy-and-paste…. Much faster!

Within AutoCAD, Copy (clipboard) the 2D objects you want to use in the Inventor Sketch.



Within Inventor, with a sketch active, paste the geometry. Pick a point to position the insert geometry



Quick tip: Use Auto Dimension with the Dimensions option disabled to quickly apply constraints.

DWG Underlays

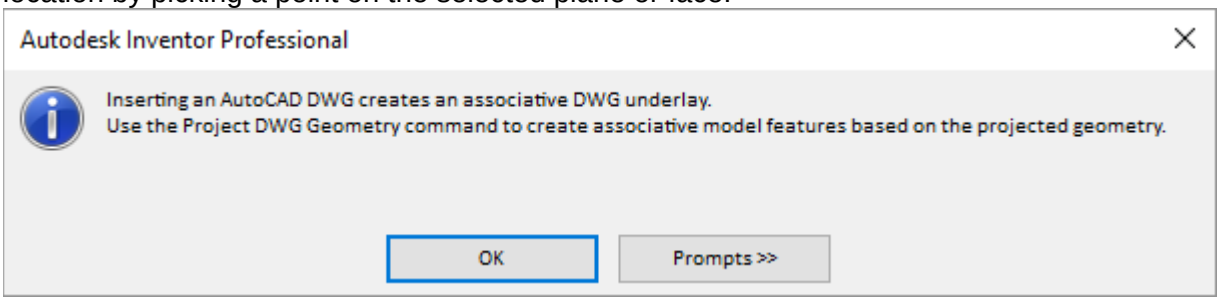
Like AnyCAD, Inventor can insert an AutoCAD drawing as an Underlay. This means the drawing is not converted, but associatively linked. It is like an AutoCAD XREF.

With Underlays, you can…

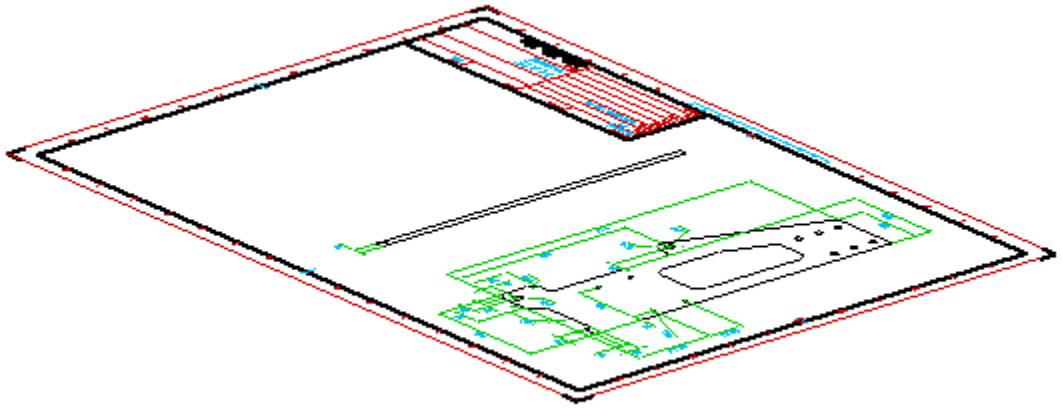
- add assembly relationships to the underlay geometry, meaning it can be used to position Inventor components.
- utilize the Underlay geometry in a new sketch via the Project DWG Geometry feature. The projected geometry remains associative to the underlay, meaning it updates as the AutoCAD drawing changes.
- Use the Underlay to create modeling features.

To insert the DWG as an underlay (into a part file) select Import from Manage > Insert panel. Change the files of type to DWG, browse for the DWG, select it, and click Open.

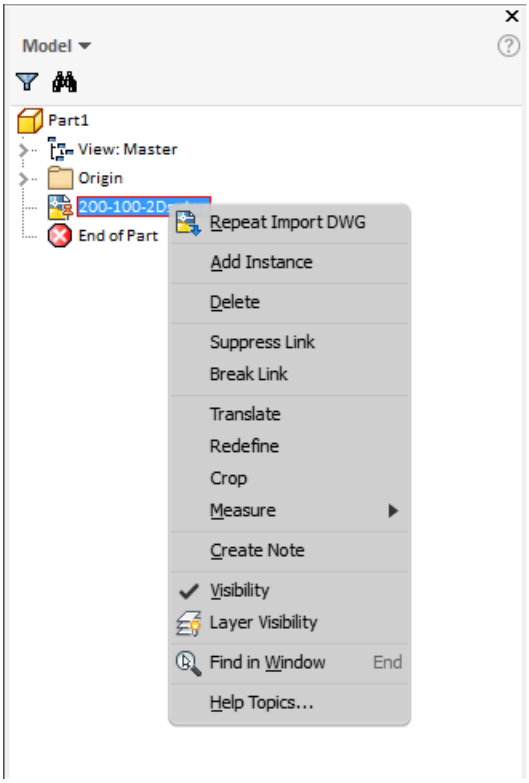
Within the graphics area select the plane or face to place the DW, then specify the origin location by picking a point on the selected plane or face.



The DWG is placed mapping the XY(+z) and origin point of the DWG to the selected plane and the specified point.



Right-clicking the Underlay within the browser provides features for working with the Underlay:



Translate to adjust the position.

Delete to remove the Underlay

Suppress Link to prevent updates. Unsuppress the Link to restore updates.

Break Link to sever the link to the DWG, meaning changes to the original DWG will never update the Underlay within Inventor.

Redefine to change the plane the Underlay is residing.

Crop to select a portion of the Underlay to display within the model.

Visibility to turn on or off the visibility of the Underlay.

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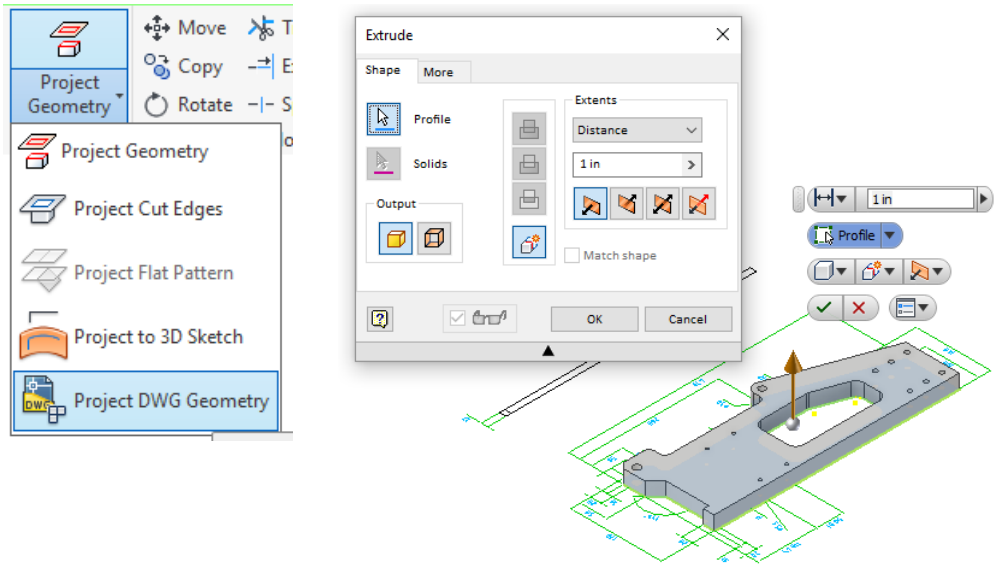
Layer Visibility to manage the layer visibility of the Underlay.

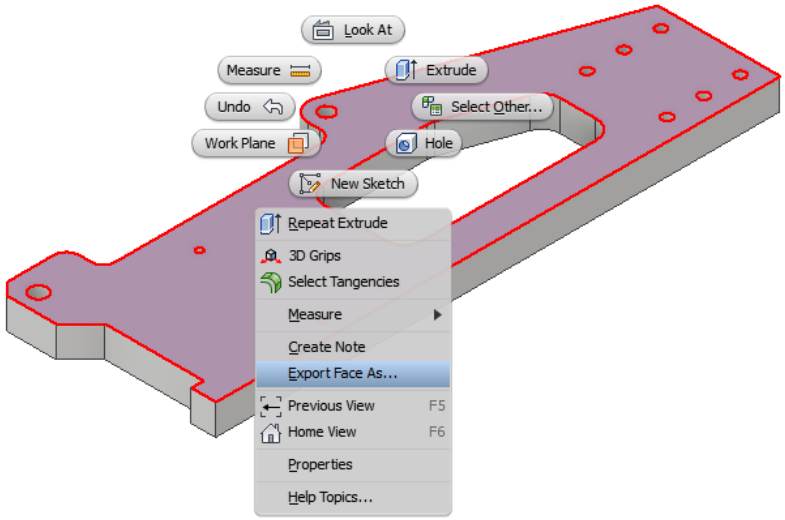
To use the Underlay's geometry, you need to first define a new 2D sketch and use Project DWG geometry. This feature associatively copies geometry from the Underlay into the new sketch, making it available to build the 3D model



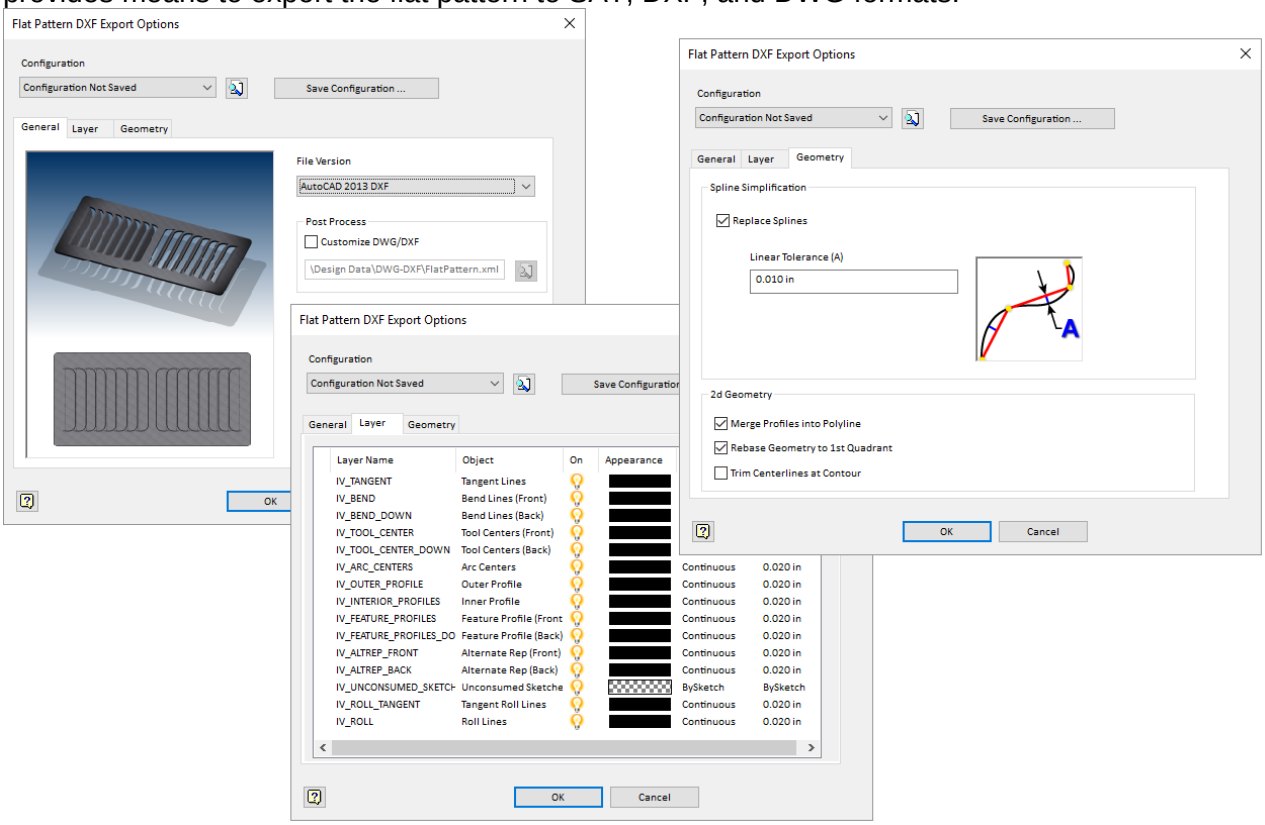
Changes to the original AutoCAD drawing propagate into the Inventor model, updating the Underlay, updating the projected geometry, and updating features built from the geometry.

Export to 2D DWG & DXF

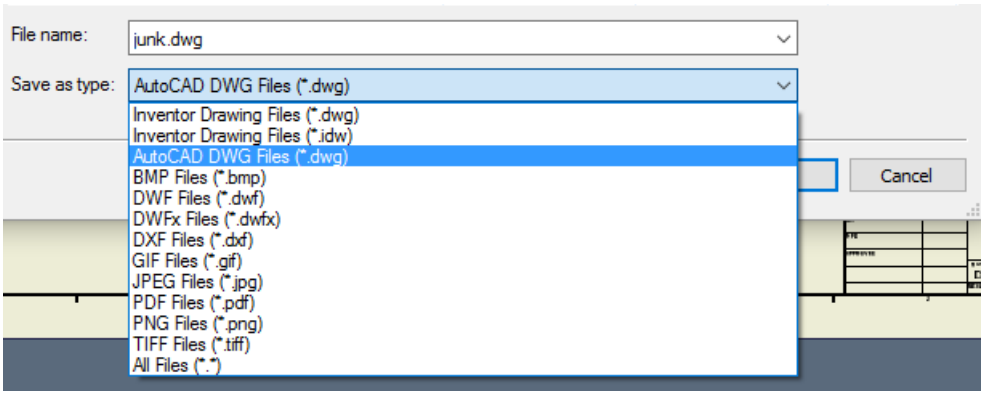
From any planar face in a 3D model, right-click, and from the menu select Export Face As to export the selected faces edges to a DWG or DXF.



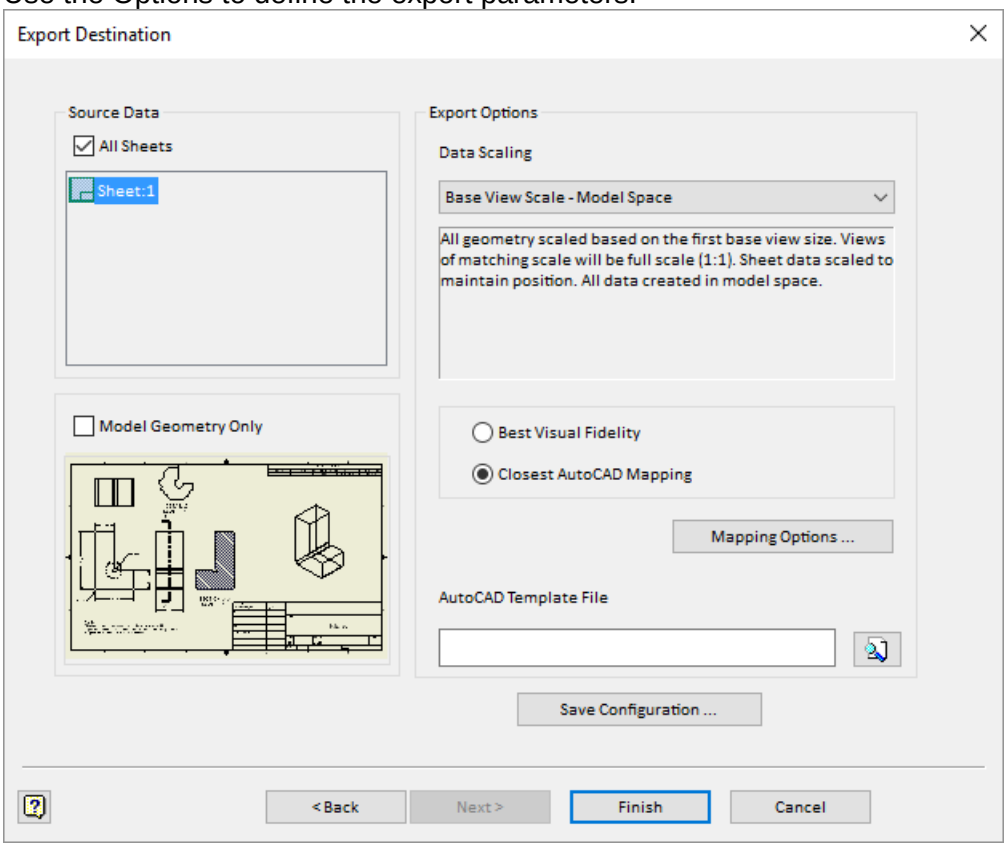
Very similar, when right-clicking on the Flat Pattern within the browser selecting Save Copy As provides means to export the flat pattern to SAT, DXF, and DWG formats.



From a drawing (IDW or DWG), use Save Copy As to export the drawing to DWG or DXF. The drawing views are written as lines, arcs, circles, and other AutoCAD objects.

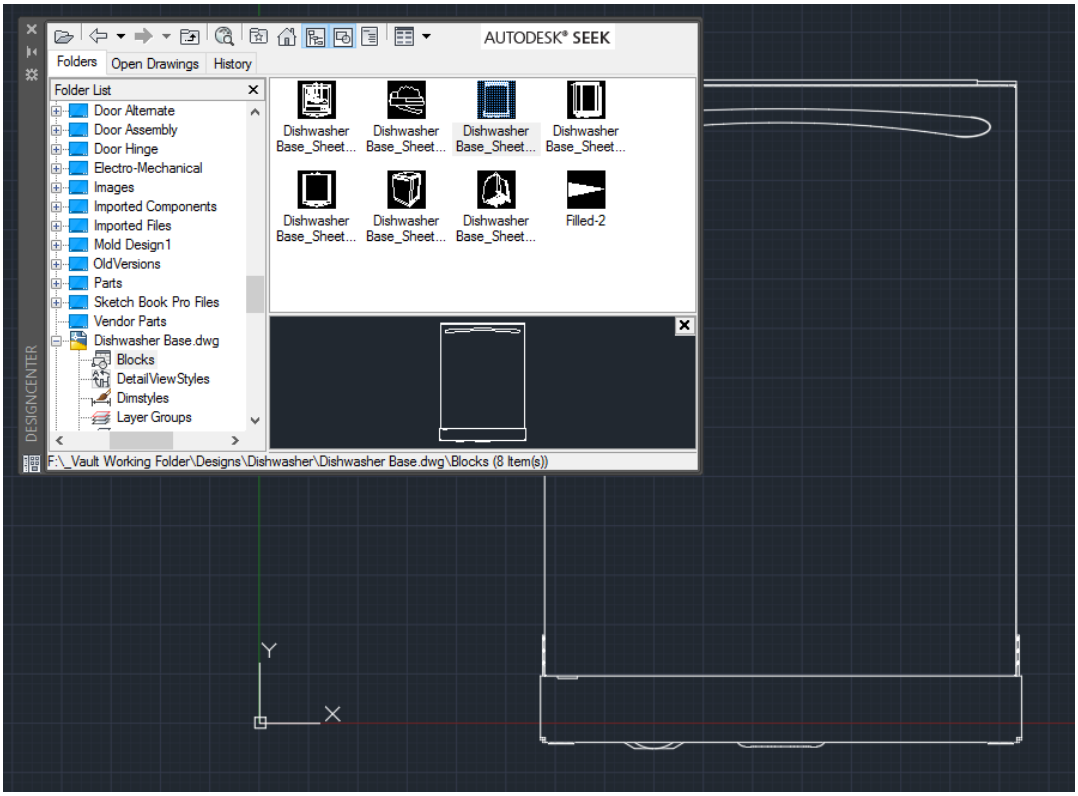


Use the Options to define the export parameters.



Inventor Views as AutoCAD Blocks

Using the AutoCAD Design Centre, you can insert Inventor drawing views as blocks. Every drawing view created in Inventor is available for insertion as a block.

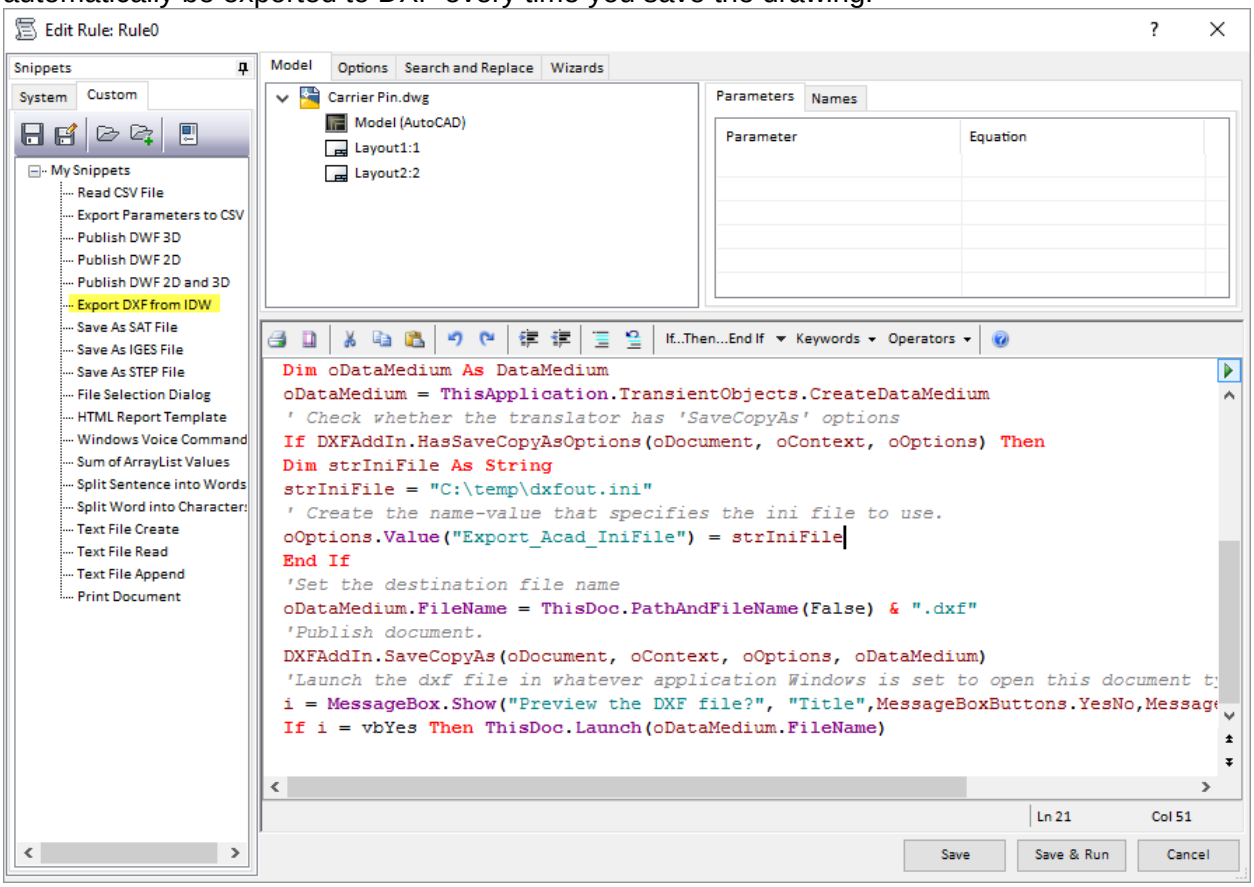


The Inventor drawing is not required to work with the blocks, meaning the AutoCAD drawing can be sent to anyone, without needing to include the Inventor drawing.

If the Inventor drawing updates, use Redefine to update the block within the AutoCAD drawing.

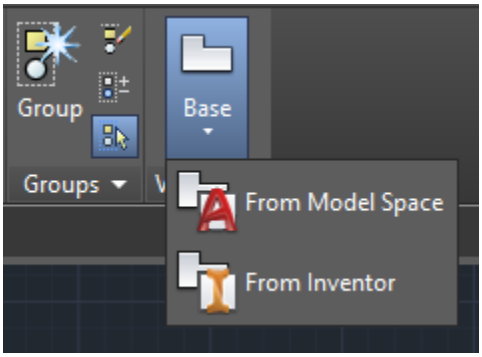
iLogic

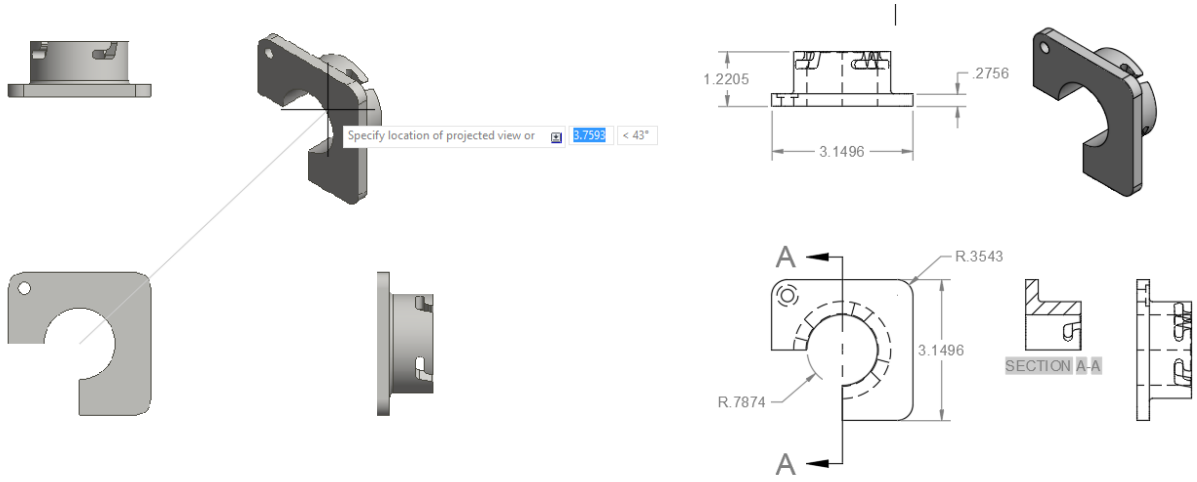
An iLogic snippet “Export DXF from IDW” provides the ability to export the active drawing to DXF. Attaching the rule to the Before Save Document event means the drawing will automatically be exported to DXF every time you save the drawing.



Use AutoCAD to create drawing views of Inventor models

Using AutoCAD, you can generate drawing views of Inventor models and then use AutoCAD's annotation features to detail the model.





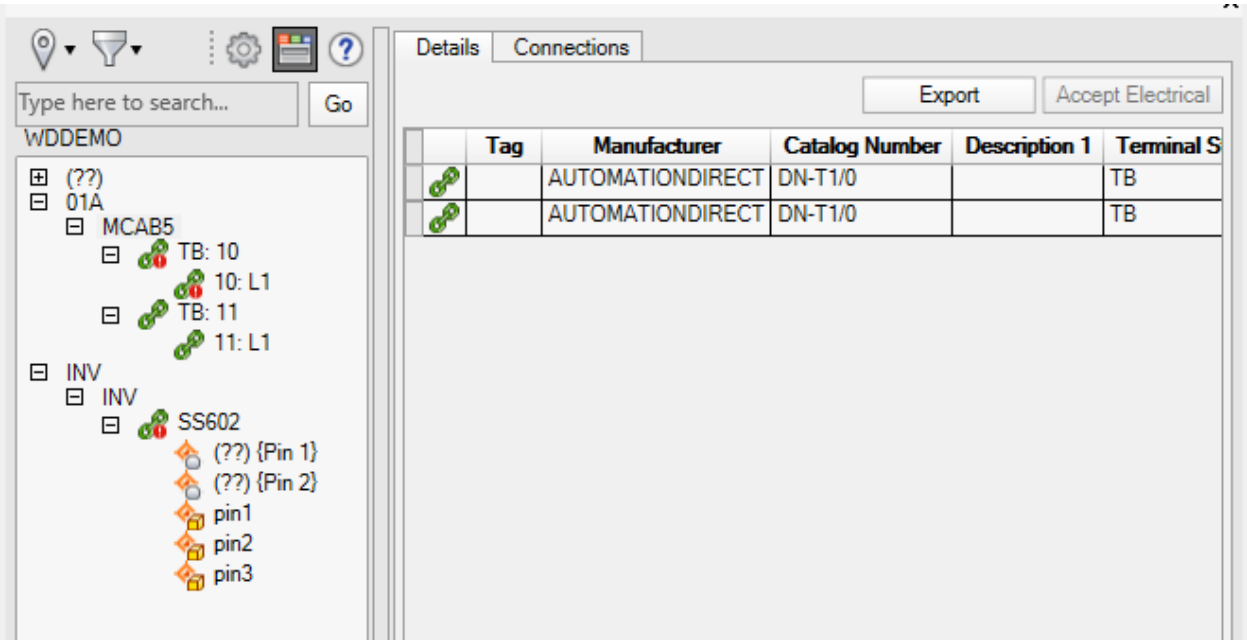
Changes to the Inventor model automatically propagate and update the AutoCAD drawing views.

AutoCAD Electrical Integration

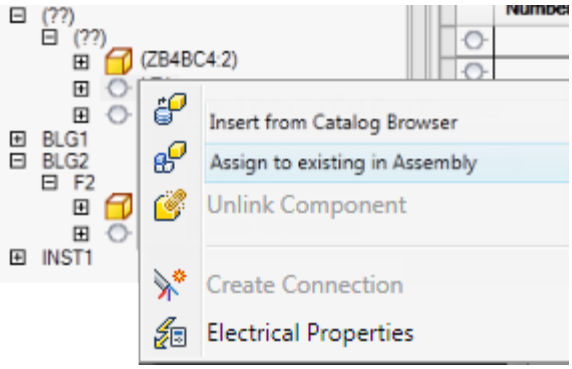
Although the Product Design Suite and Product Design Collection includes AutoCAD, the real benefit is the access to AutoCAD Mechanical and AutoCAD Electrical.

Based on AutoCAD, AutoCAD Electrical (ACE) automates common design tasks and facilitates drafting productivity with electrical engineering features such as symbol libraries, bill of materials (BOM) reporting, and PLC I/O design. With AutoCAD Electrical, you develop electrical control schematics, panel layouts, and PLCs quickly.

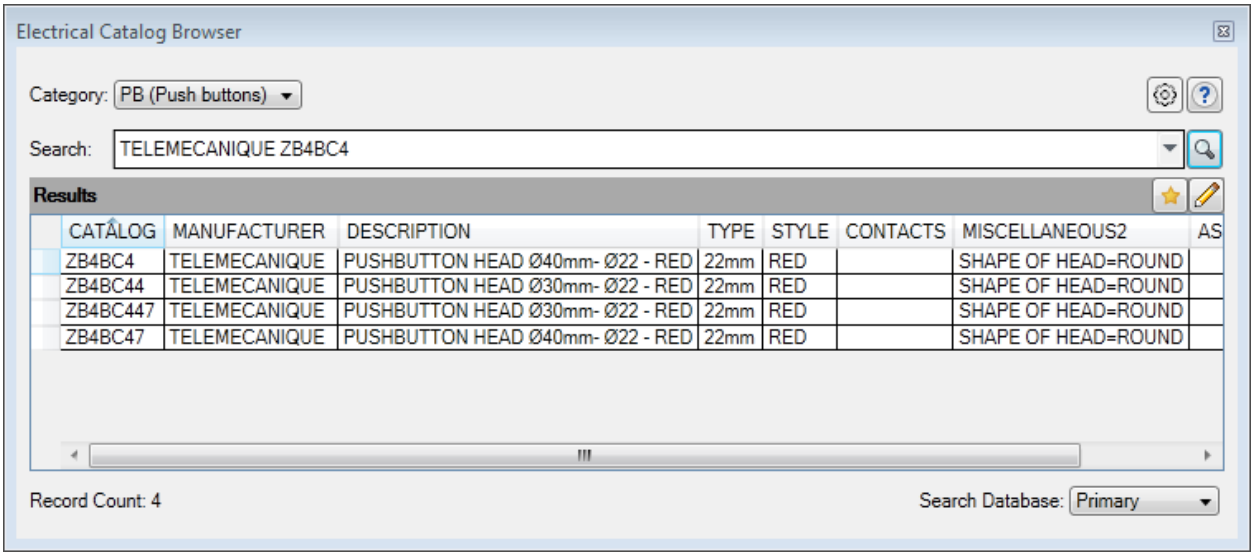
By creating an Electromechanical project, you build a link between the AutoCAD Electrical project and the Inventor project. Design changes made in one product, update in the other. With the Electromechanical project established, installation and location information from the AutoCAD Electrical schematic becomes available within Inventor.



To associate the Inventor component to the AutoCAD Electrical component right-click on it within the Electromechanical browser and select Assign to Existing in Assembly. In the graphics window (or browser) select the Inventor component.



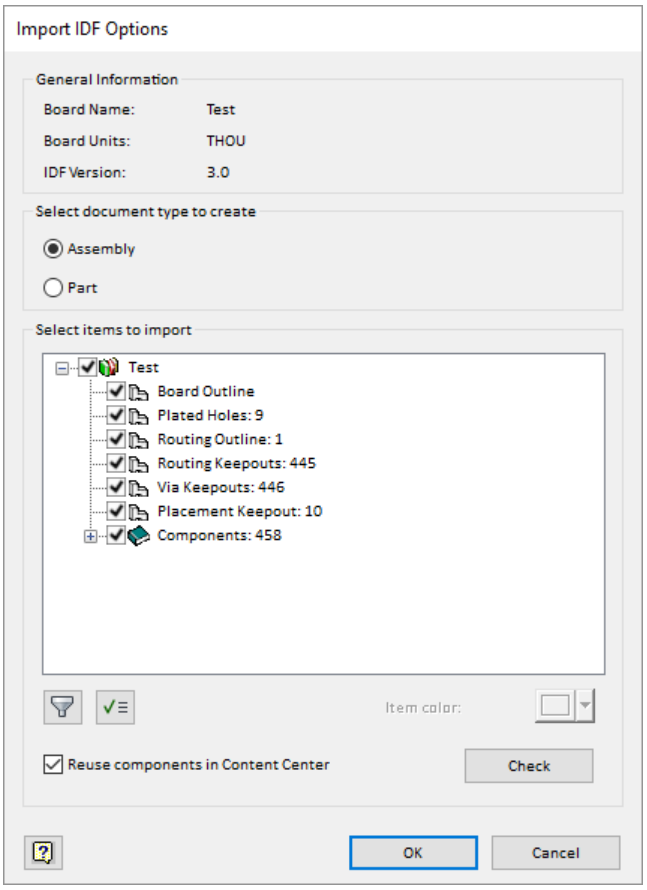
The Electromechanical toolset also provides a shared component library, making for a singular component to manage.



Cable and wire connections can be defined from Inventor or AutoCAD Electrical. Use Create Connection to create wires and cables connections automatically in Inventor.

IDF

The Intermediate Data Format (IDF) transfers PCB board layout information into the mechanical design software (Inventor). The IDF transfer can be in two parts, the board/panel file (BRD, EMN, BDF, IDB) and a library file containing board property information. Inventor will import Outlines (board and panels), Keepouts, Group Areas, Drilled Holes, and Components (connectors, switches, etc)



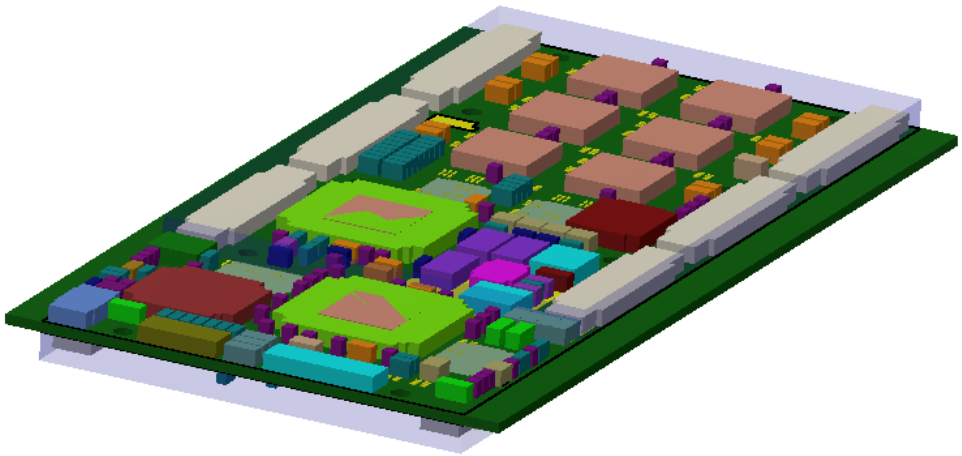
The IDF can be imported as a multi-solid part or as an assembly. As an assembly, each component is imported and the holes created.

Each component is assigned a color for differentiation between the other components. You can change the color of the entire board or individual components.

The Filter Settings provide a method to filter the import based on Nonplated Holes, Plated Holes, Components, and Outlines.

Inventor will automatically substitute imported components with the ones existing in your Content Center library. Use the Check tool to identify components that are not already in your Content Center library.

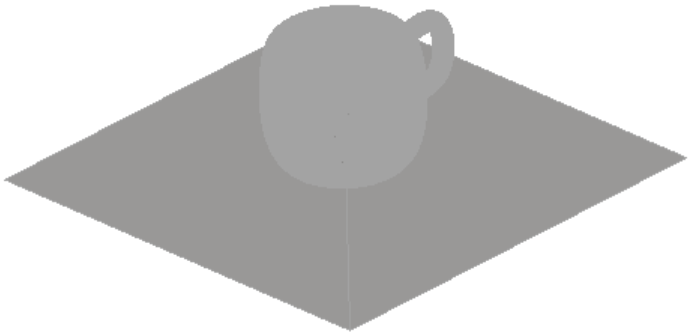
The IDF Component command is used to publish IDF parts to the Content Center Library.



Mesh Models

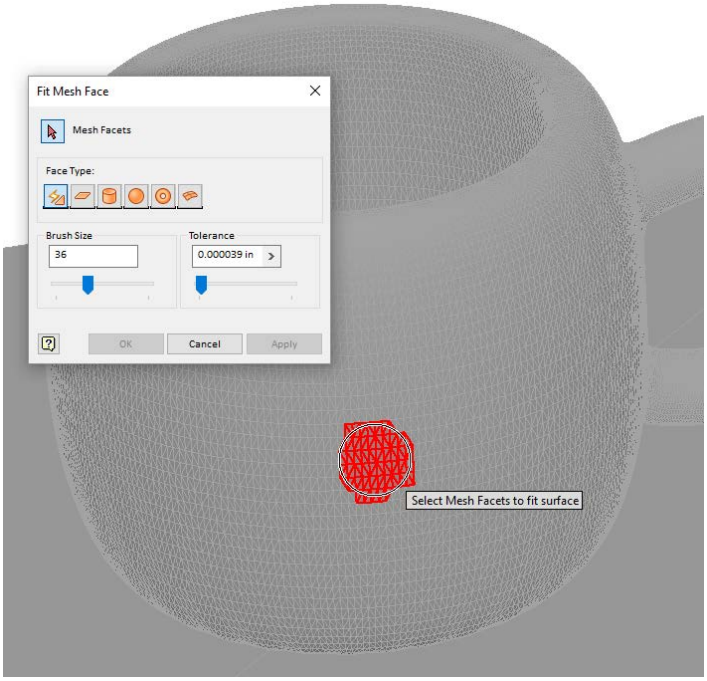
A mesh model is a basic form of 3D modeling which is also very common. A mesh is a collection of points, edges, and flat surface faces (sometimes referred to as polygons). Because of the high number of faces, it can be quite tedious to work with meshes.

Autodesk Inventor can import mesh OBJ and STL mesh models. This includes both parts and assemblies. The Open dialog provides both file types as a Files of type option.



Within an assembly relationships can be established between the mesh model and other components.

With the Mesh imported into a part, use the Fit Mesh Face surface feature to extract surfaces from the mesh facets. With Fit Mesh Face, you let Inventor determine the surface type (Automatic) or you specify the surface type.

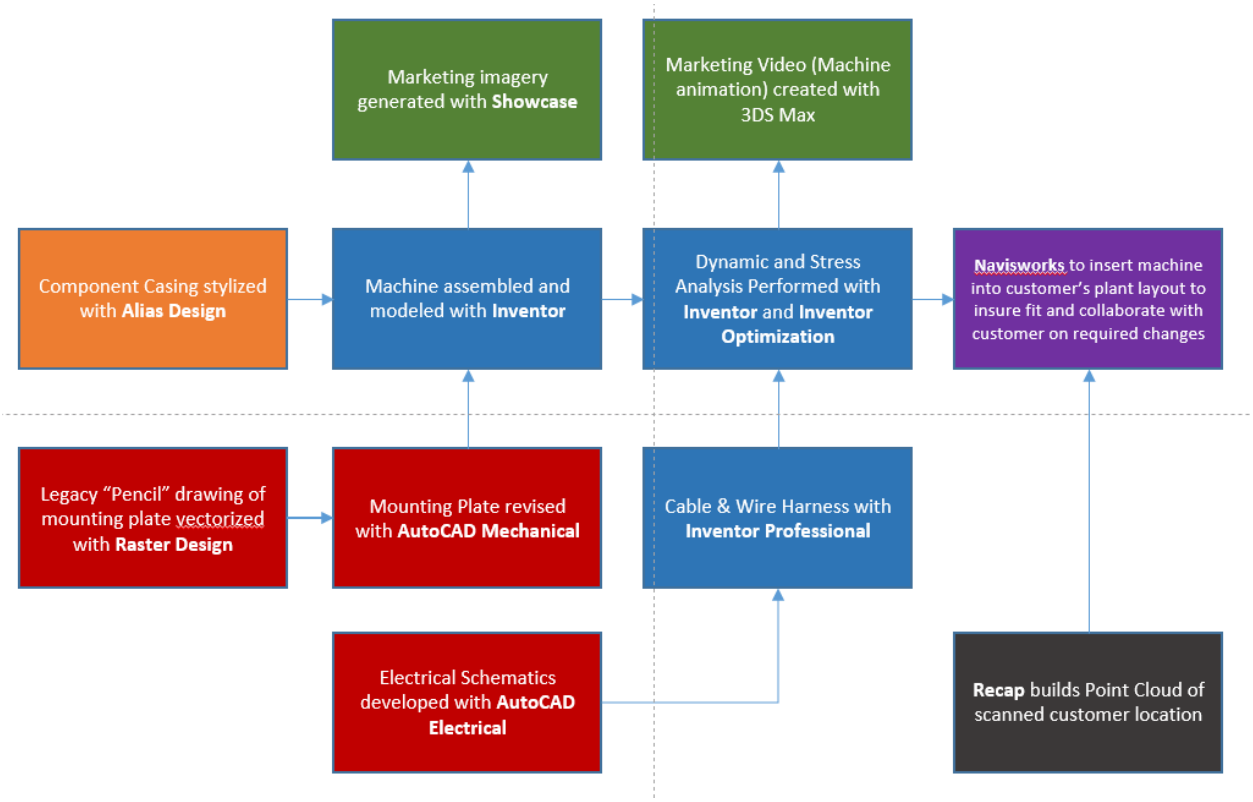


After adjusting the tolerance and brush size, pick the desired faces.

Suite Workflows

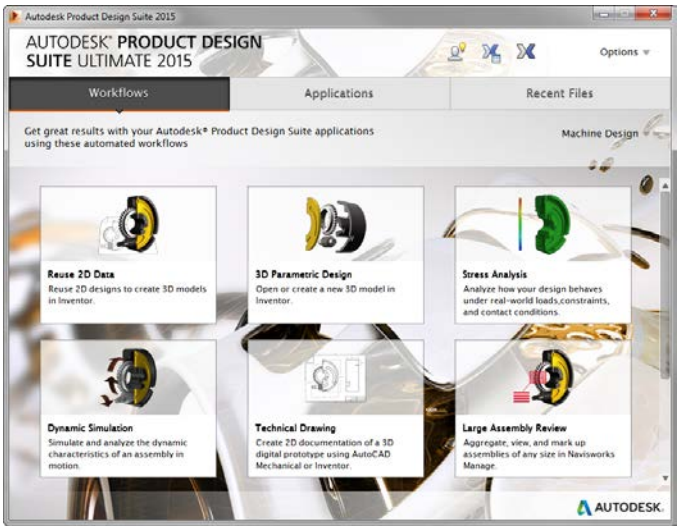
Many of the Product Design Suite Applications have embedded workflows and features to transition data from one application to another. In many instances, the data is associative so that as it changes in one application it associatively updates in the other.

Example Workflow



Autodesk Product Design Suite Launcher

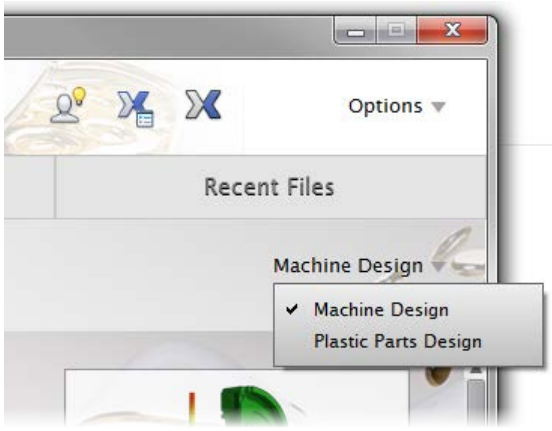
After installing PDS, regardless of the combination of software installed, the PDS Launcher is installed. Autodesk designed this tool to be the one-stop place for everything PDS. This includes workflows, a listing of installed applications, a list of recent files, and access to the Autodesk Exchange app store.



PDS LAUNCHER WORKFLOW OPTIONS

More than simply launching the software it steps you through options in selecting templates, creating files, and selecting files so that you are setup and ready to go quicker.

For example, by selecting Reuse 2D Data you are prompted to first select an AutoCAD drawing and interactively select the layers to import. A new Inventor part is created with the selected 2D geometry inserted as a new sketch.
Two “sets” of Workflows are available: Plastic Part & Machine Design that is toggled from a drop-down in the upper right area of the dialog.



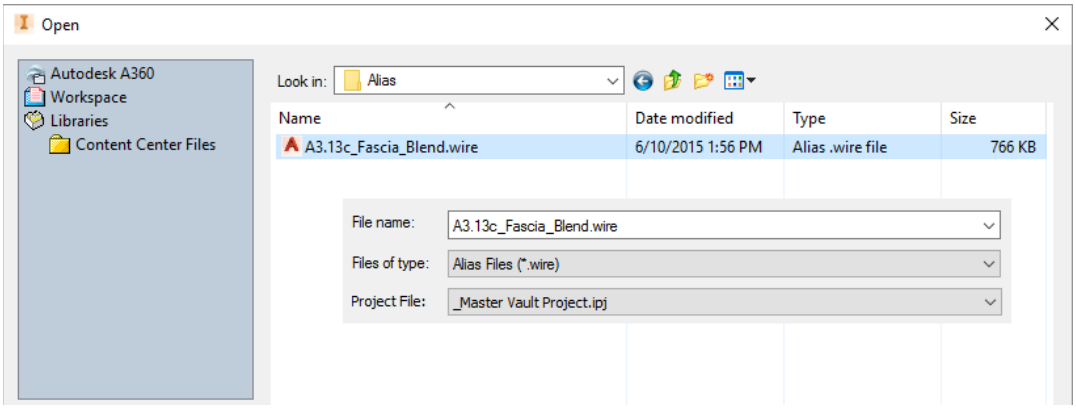
TOGGLING THE WORKFLOW TYPES

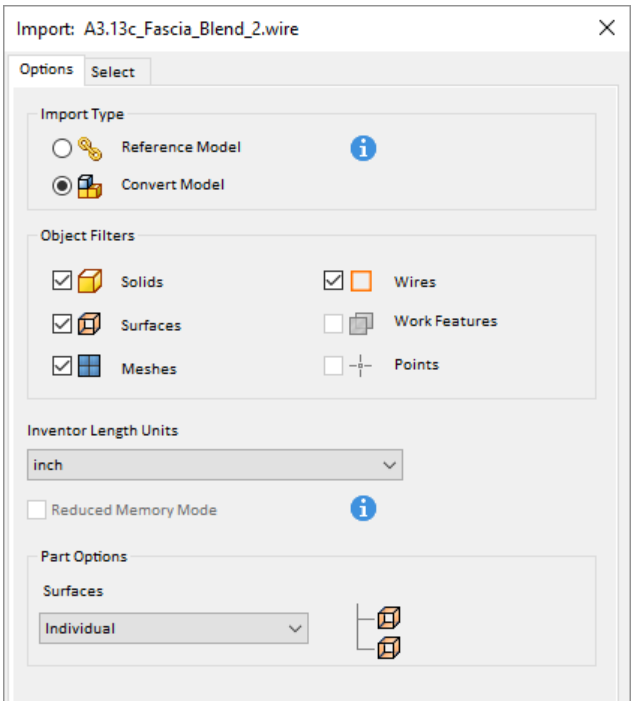
Alias Design

"Alias industrial design software powers your creative process with a range of sketching, modeling, surfacing, and visualization tools. Create compelling surface designs with Digital Prototyping tools tailored for industrial design, automotive design, and product design. Get advanced capabilities, from initial concept sketch to development of production-ready Class-A surfaces."

Alias Design provides powerful "technical" surfacing capabilities to digitally create your concept model or design model in an environment with real-time design visualization. Start with a sketched illustration, which moves into a conceptual surface model before being rounded-out to become the final design model.

Inventor can open Alias Files and utilizes the AnyCAD functionality. The Alias data can be associatively linked so that changes will update within Inventor, or translated into Inventor geometry.





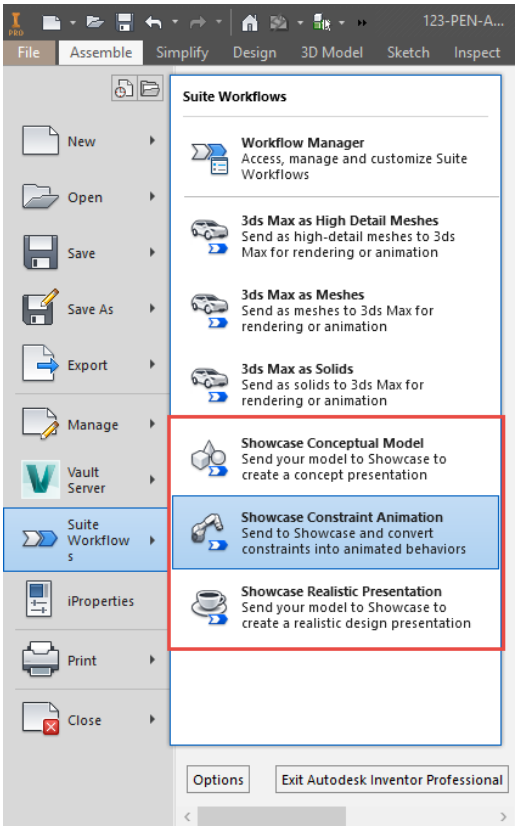
Showcase

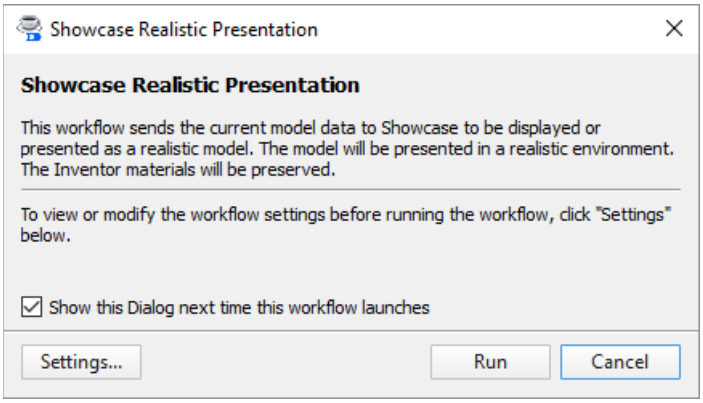
"Showcase® 3D visualization and 3D presentation software provides easy-to-use presentation and design exploration tools for architects, designers, engineers, and marketing professionals. Quickly transform 3D CAD models into interactive walk-throughs and presentations, so that you can evaluate aesthetic and design alternatives in real time with peers, constituents, and customers."

AUTODESK SHOWCASE (IMAGES COURTEOUS OF AUTODESK.COM)

Showcase's advanced rendering engine provides a system to create "stunning" realistic views of your models, which can be manipulated real-time. As it has no geometry creation tools of its own it is intended to take your AutoCAD and / or Inventor 3D models to create rendered 3D scenes.

Inventor models can be imported directly into Showcase via the Open command. From within Inventor use one of the three provided Suite Workflows to push the model into Showcase with preconfigured settings.

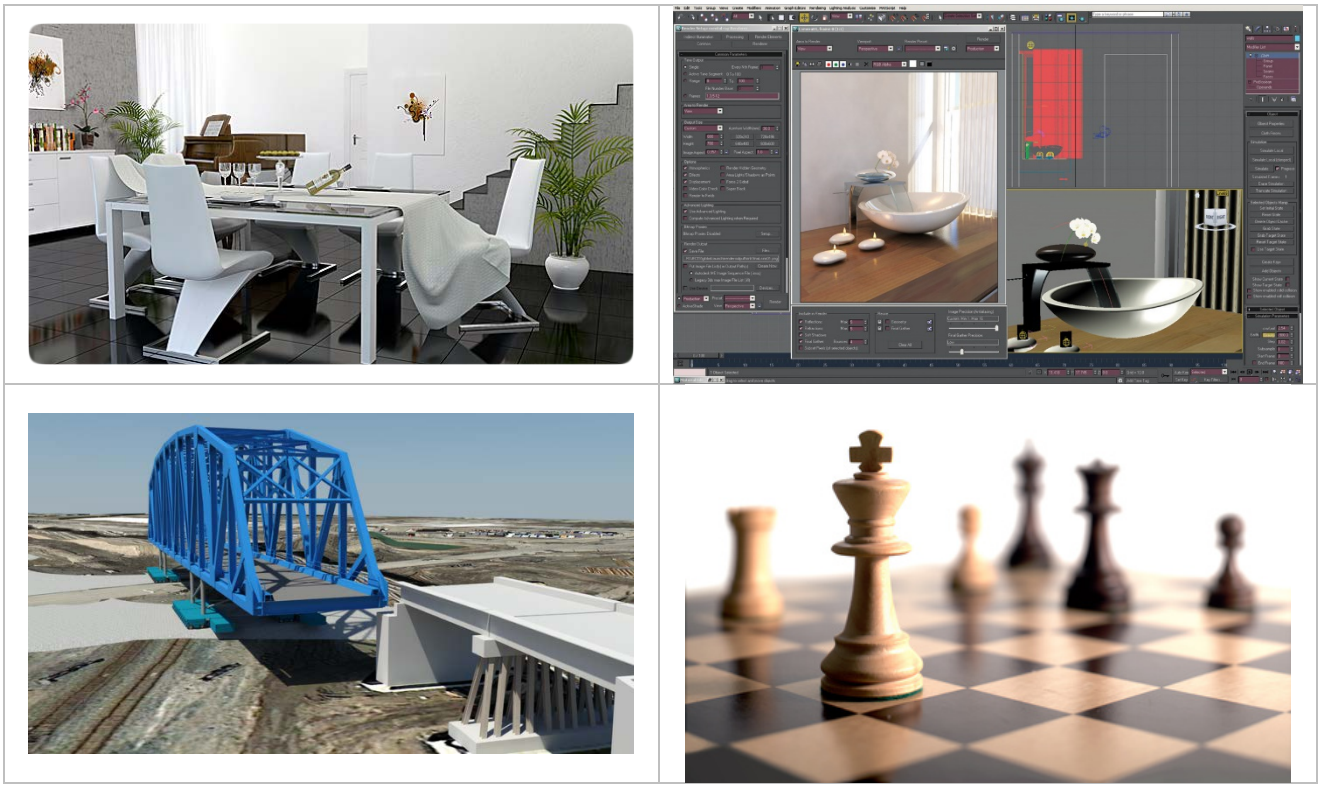



Showcase Realistic Presentation
Showcase Realistic Presentation
This workflow sends the current model data to Showcase to be displayed or presented as a realistic model. The model will be presented in a realistic environment. The Inventor materials will be preserved.
To view or modify the workflow settings before running the workflow, click "Settings" below.
Show this Dialog next time this workflow launches
Settings...
Run
Cancel

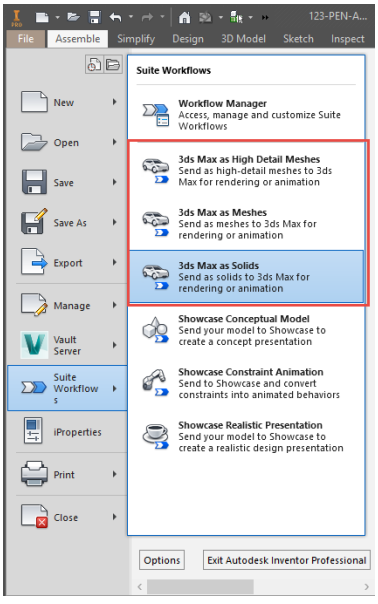
PIVOT
Visual Styles
D
Both Shadows
R
Ray Tracing
Shaded Textu
Library

3DS Max

3DS Max is one of the well-known and well-respected programs for creating 3D animations, models, games, and images. With 3DS Max you can add dynamics and effects to your models to create highly realistic 3D Renderings and animations. In conjunction with the general animation, tools 3DS Max includes animated deformers and character animation and rigging tools to quickly build up real-world actions on your models.



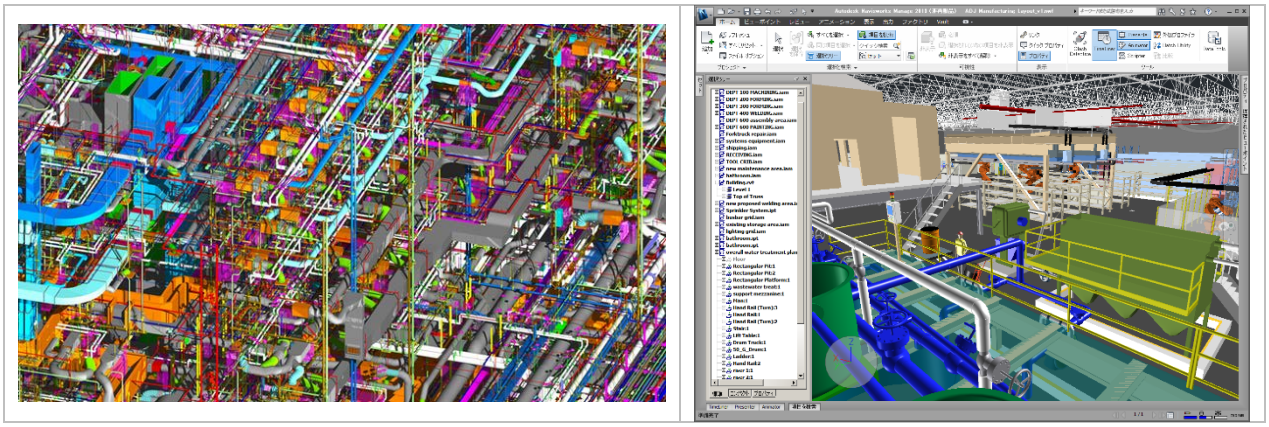
3DS MAX (IMAGES COURTEOUS OF AUTODESK.COM)



3DS Max supports the import of Inventor files natively, meaning no exporting required from Inventor. Additionally, Inventor provides three Suite workflows to push your active Inventor model into 3DS Max with preconfigured settings.

Navisworks

Although primarily used in the construction industry Navisworks provides tools to manage large projects and models from various data sources. With Navisworks you can combine the 3D models to navigate, collaborate, markup, and perform collision style interference checks. The purpose of Navisworks is to provide an environment for all project stakeholders to review the models and data to plan projects before they start.



NAVISWORKS (IMAGES COURTEOUS OF AUTODESK.COM)

Why use Navisworks?

- Coordination - clash detection and interference checking
- Model Review - insert multi-format models, measure, markup, add comments, and publish to distributable formats
- Model simulation & analysis - integrated 2D & 3D quantification, 5D project scheduling, Photorealistic model rendering, and object animation.
- Project Viewing: Cloud rendering, Reality capture, real-time navigation

Navisworks provides the opportunity to “pull” data from a wide range of formats into one document. This includes Inventor files in both IAM and IPT formats.

Point Clouds

A Point Cloud is a collection of vertices, each point defined by a X,Y,Z coordinate. Point Clouds typically represent points on the external surface on an object, acquired by a 3D scanning device. Point Clouds can contain thousands if not millions of points to represent the object correctly.

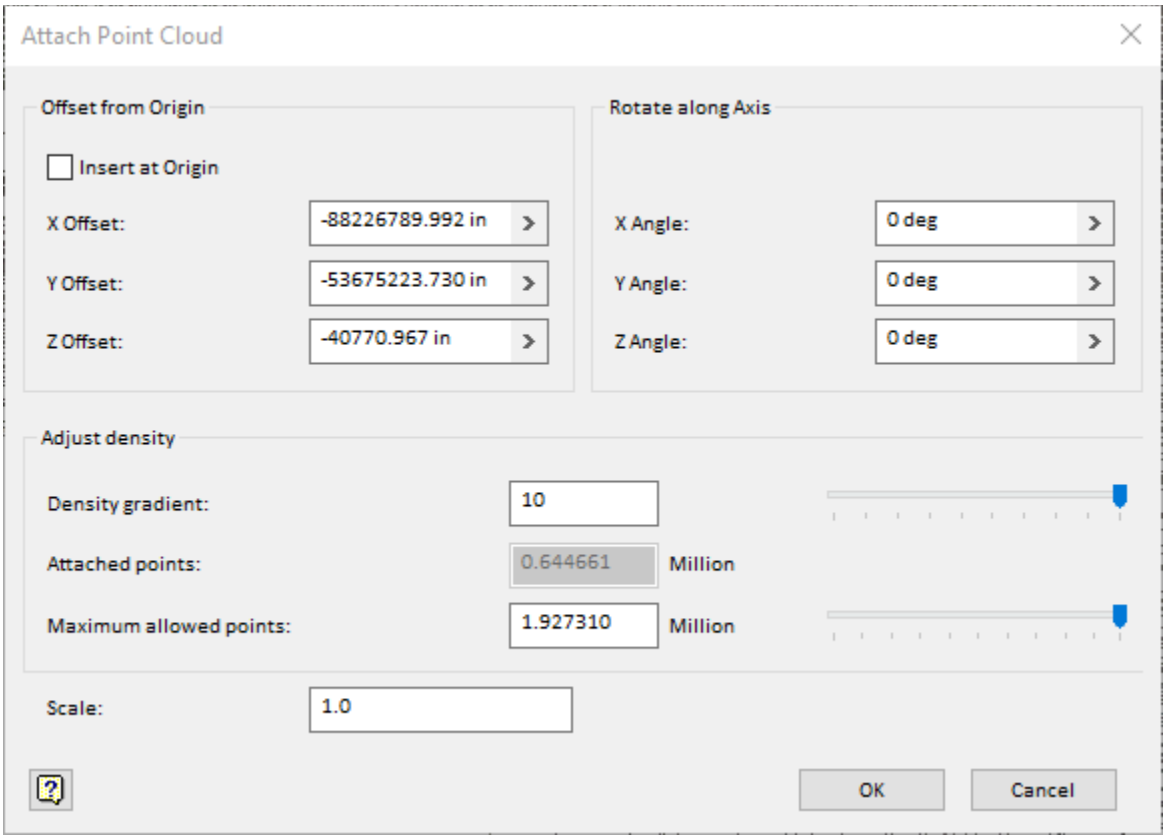
Before attaching a point cloud into Inventor, it must be first indexed using Autodesk Recap.

Autodesk Recap is a desktop application (compared to the cloud-based ReCap 360) for indexing, editing, and visualizing 3D data from laser scans. It is used as the intermediate between the scanned data and other Autodesk products, like AutoCAD and Inventor. Recap is included within the Product Suites (except for the now retired AutoCAD Design Suite), as well as the Industry Collections.

Recap supports many types of scanned information; Inventor supports just ReCap files.

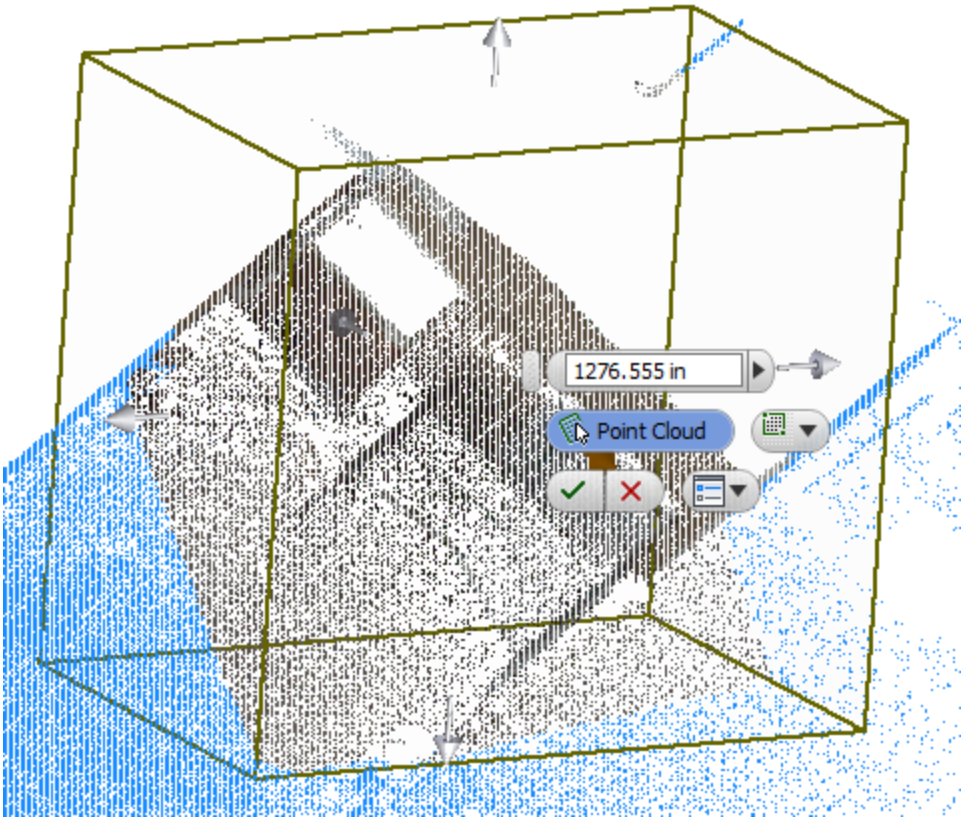
To attach a point cloud in Inventor:

1. Attach the point cloud (Manage > Point Cloud panel > Attach)
2. Pick a point in the graphics window to locate the point cloud
3. Use the Attach Point Cloud dialog to insert the point cloud at the origin or to specify its location with x,y,z offset coordinates.
4. Additionally, adjust the X, Y, Z rotation, if you want to rotate the cloud about one or more axis.
5. Use the Scale value to increase or decrease the size.
6. As necessary adjust the density via the Density gradient and maximum allowed points.



At any time, you can revisit these options by invoking Edit Attached Data, available by right-clicking on the point cloud within the browser.

To remove a portion of the point cloud use the Box Crop feature. The default Keep Inside keeps the inside portion of the bounding box, whereas Keep Outside removes the inner portion.



Additional point cloud tools:

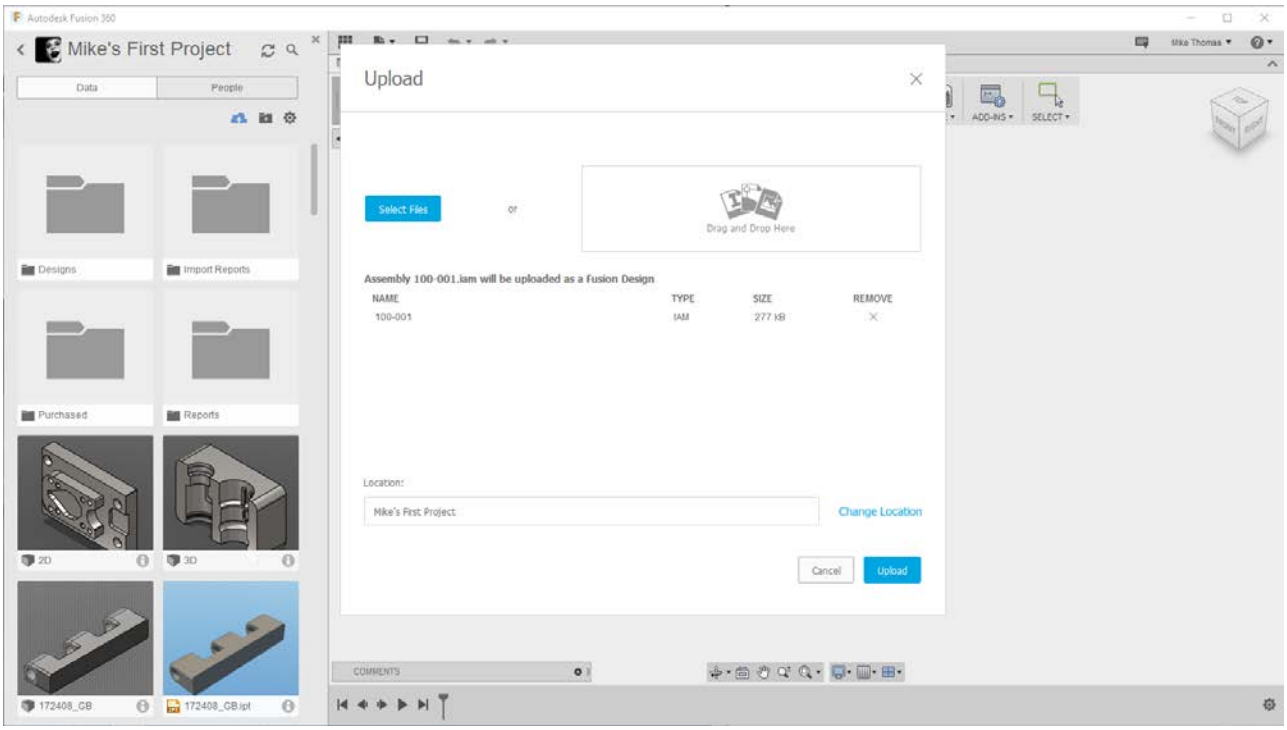
- Cloud Point creates a work point coincident with a selected point within the point cloud
- Cloud Plane extracts a plane from a set of cloud points. Hold SHFIT to switch to another point cloud plane during creation
- The Point Cloud Navigator is used to manage the visibility of the point clouds contained within the model

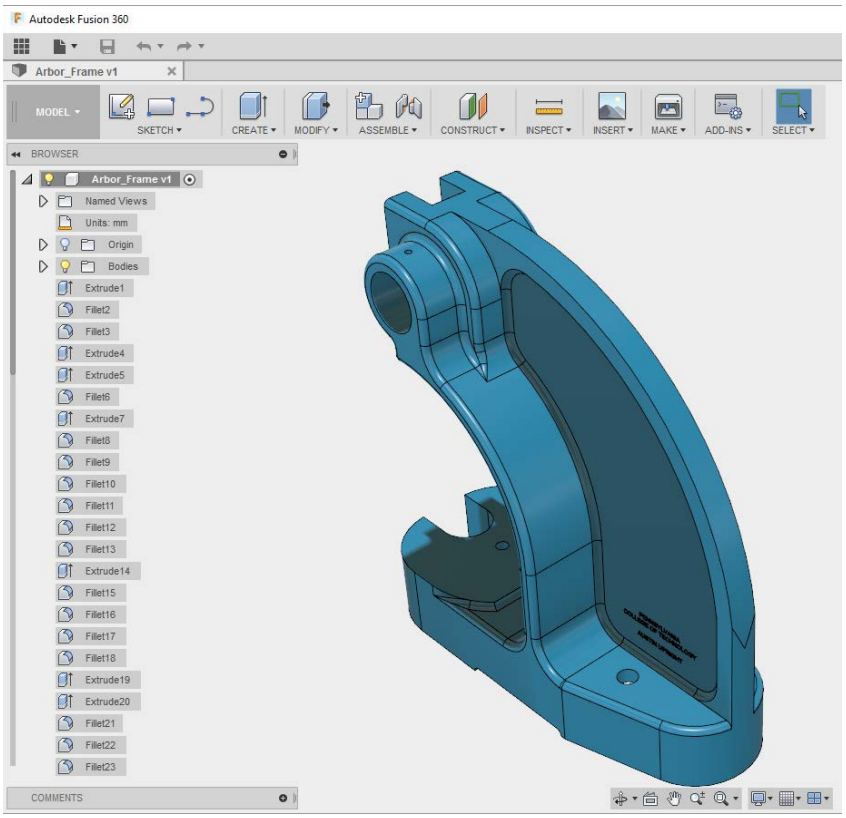
Fusion 360

Autodesk Fusion 360 is a cloud-based application providing CAD, CAM, and CAE features within a single platform.

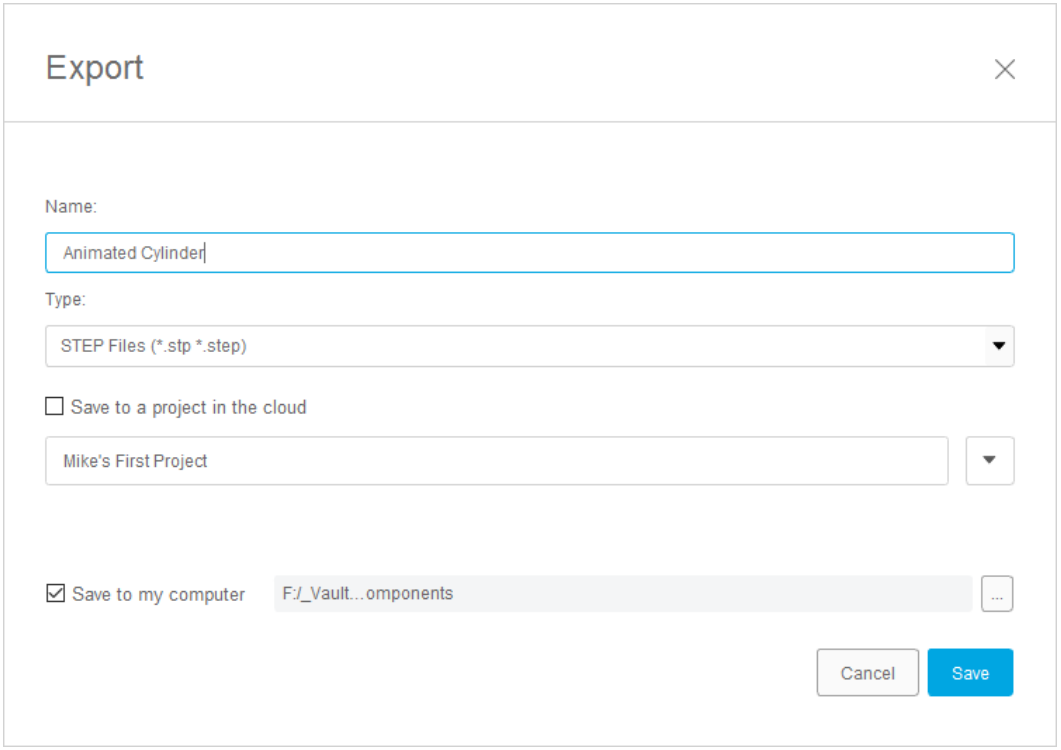
As Fusion 360 is based on Autodesk A360 it supports working with many types of files. Inventor files are unique as Fusion will translate Inventor features into quasi-Fusion features. Once translated, you can edit and make changes to the model within Fusion.

Using the Fusion data panel, you upload the Inventor model. This loads the model to your A360 account, making it available for Fusion 360.





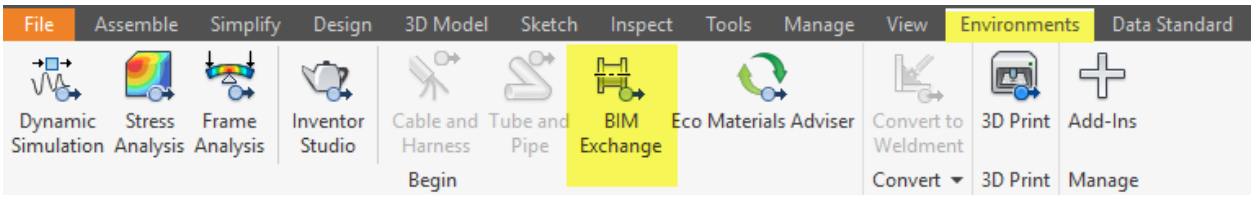
To get the Fusion data into Inventor, export your model to the STEP format.

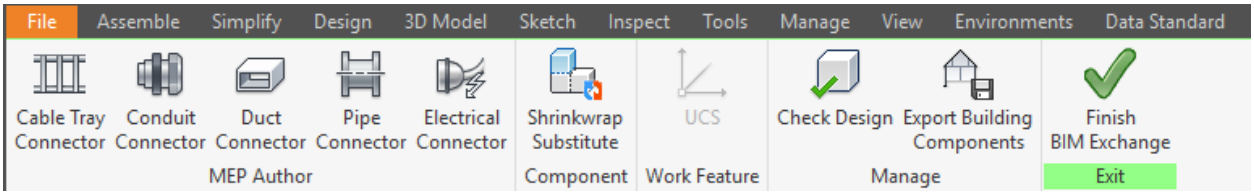


BIM Exchange

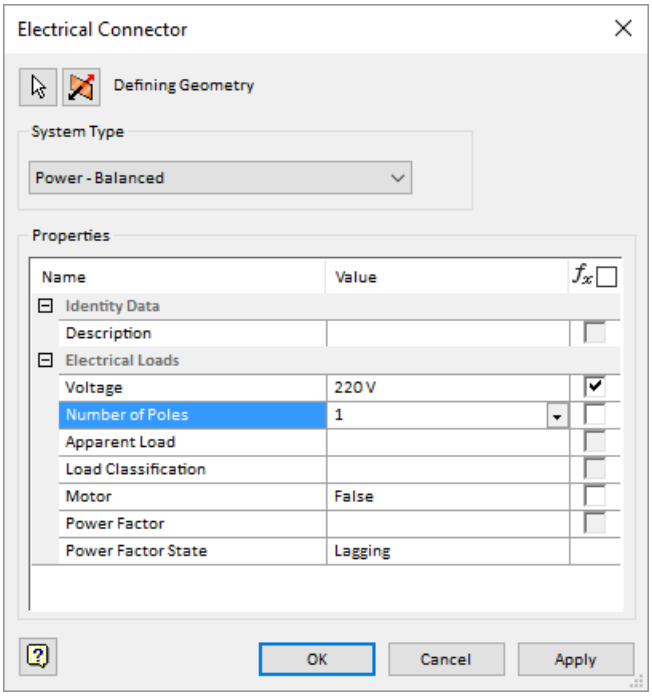
The Inventor BIM Exchange tools provide a method to take an Inventor mode, simplify it, and add Revit system connectors, meta data, and other properties. After verifying the data you export the components as a Revit Family File (RFA) for using in Revit.

To get started activate the BIM Exchange environment.

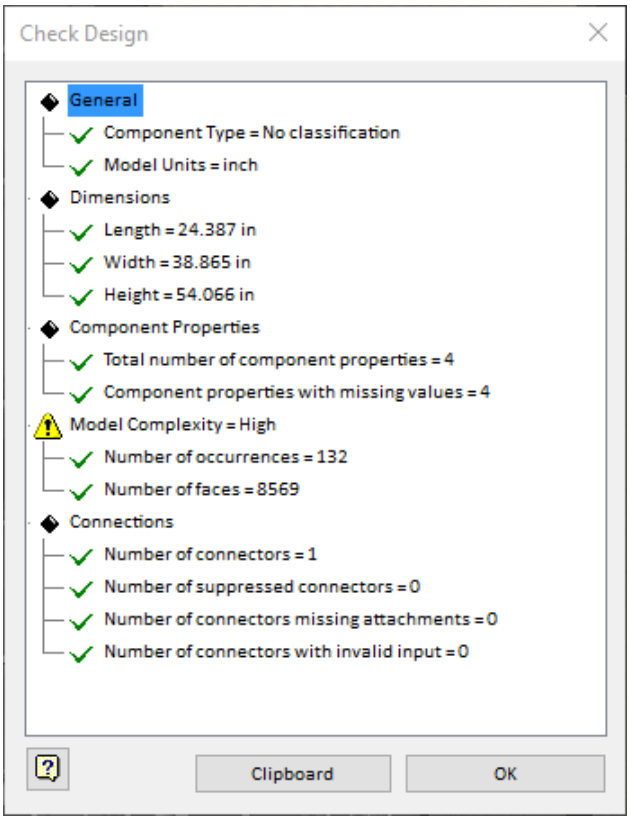




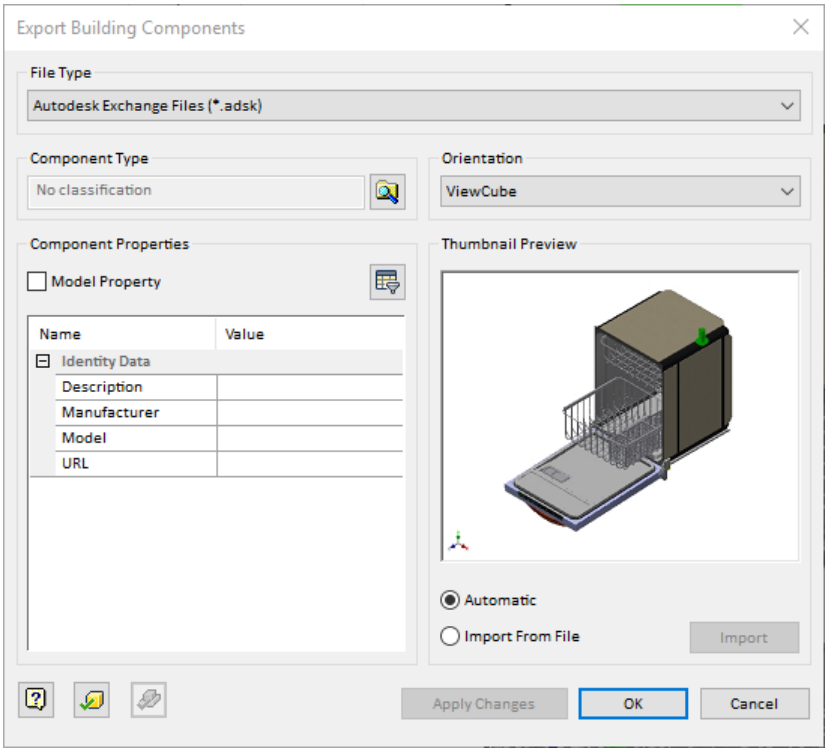
Start with the MEP Author panel, selecting the type of connector required for the active component. For the connector specify the defining geometry and set the desired properties (meta data).



With the connections defined, the geometry simplified (using Level of Details) Check the Design before exporting it. Think of this as a double-check, verifying the model will import into Revit correctly.



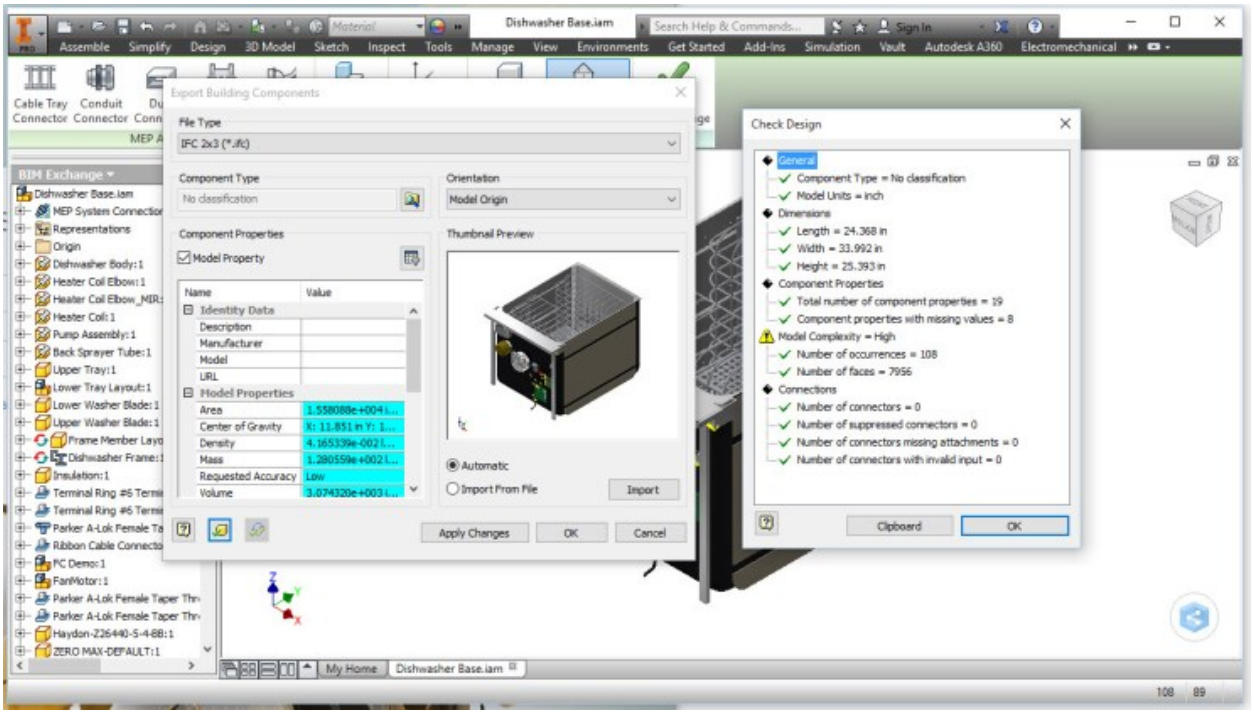
With the model verified, use Export Building Components to export the model to either the Autodesk Exchange File (ADSK) or Industry Foundation Classes (IFC) format.



IFC

Industry Foundation Classes (IFC) is an initiative originally started by Autodesk, but developed and maintained by the non-profit organization buildingSMART. Think of IFC as the DXF or STEP of the architecture, engineering, and construction (AEC) industry. It is used as an open source, platform-neutral format, for the exchange of data between systems.

Why export IFC from Inventor? to share 3D mechanical designs with building and construction applications. Inventor “leverages buildingSMART-certified IFC export capabilities in Revit to provide this capability.”



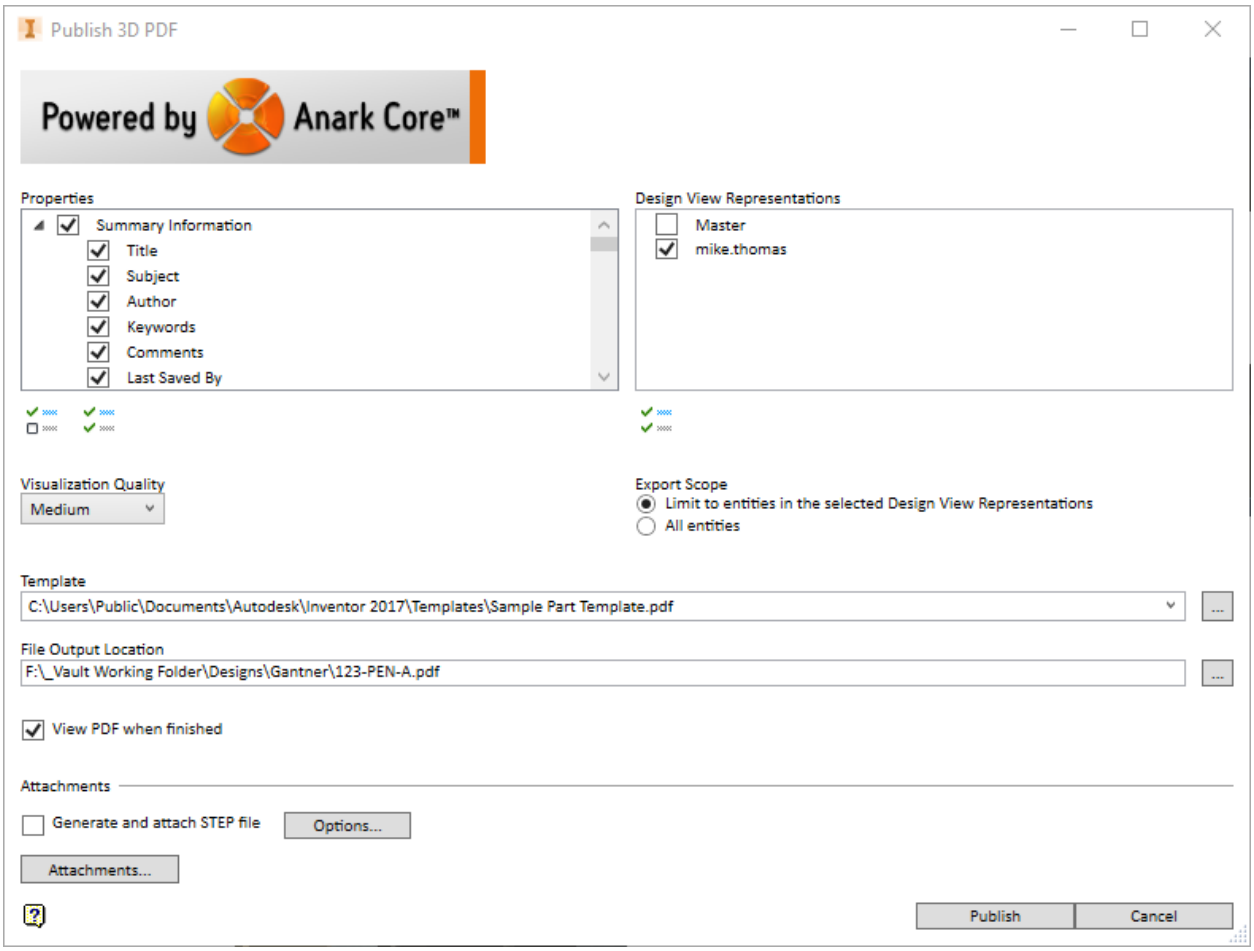
To export a part or an assembly select IFC from the Export Building Components command, located within the BIM Exchange environment. Both the geometry and material properties are written to the IFC.

PDF

Inventor 2017 includes the ability to export your models to 3D PDF. This is powered by Anark Core, meaning it is not Autodesk technology but third-party powered. What's also important is this same technology has been licensed for Vault as well.

"Adding Anark to our partner program will confirm our commitment to providing access to the best 3D MBE technology in the shortest amount of time possible" – Buzz Kross, Senior VP of Manufacturing for Autodesk.

File > Export > 3D PDF



From the Publish 3D PDF dialog select:

- the properties to be included with the PDF
- the desired Design View reps and the Export Scope (just the visible entities in the select Design View reps, or All entities)
- the Template and the output location
- the attachment option, as in do you want STEP exports generated for each component and attached to the PDF

Connected Design

[Portions of this are extracted from the Design & Motion article Autodesk Inventor 2016 R3]

How do you share and review your Inventor designs with others? Especially those co-workers, clients, or suppliers who are not located in your office?

You could send an email, but what do you attach? You probably don't want to send the Inventor model, either because of security or protecting your intellectual property (IP) …. or just because of the large number of files you would need to send. Images are static. PDFs can contain 3D models, but Inventor doesn't create 3D PDFs natively. DWF is great, but it's basically a dead format and the recipient needs Design Review. On top of this, people typically have a limit on the size of email they can receive… and in this day-and-age of cyber security, attachments are closely monitored…. and regularly blocked!

Alternatives to email? FTP sucks, so what about Dropbox, Box, or Google Drive? Potential problems with logins, getting access, and then downloading the files. Autodesk Buzzsaw is great, but it's not really built for short-term, one-time access. You could always GotoMeeting or something similar, but again you have problems with logins and these are not always great for the transmission of 3D models.

This is where Connected Design shines: "Inventor Connected Design enables fast, easy 3D design reviews with your team and other project collaborators anywhere, on any device."

Connected Design leverages A360 to provide a platform to quickly share your Inventor models with others. The publishing process creates an extremely lightweight "viewable" called a Design Share. Others can view your Design Share on just about any device, including computers, smartphones, and tablets. Only what is visible on the screen is published, meaning you can control what others see.

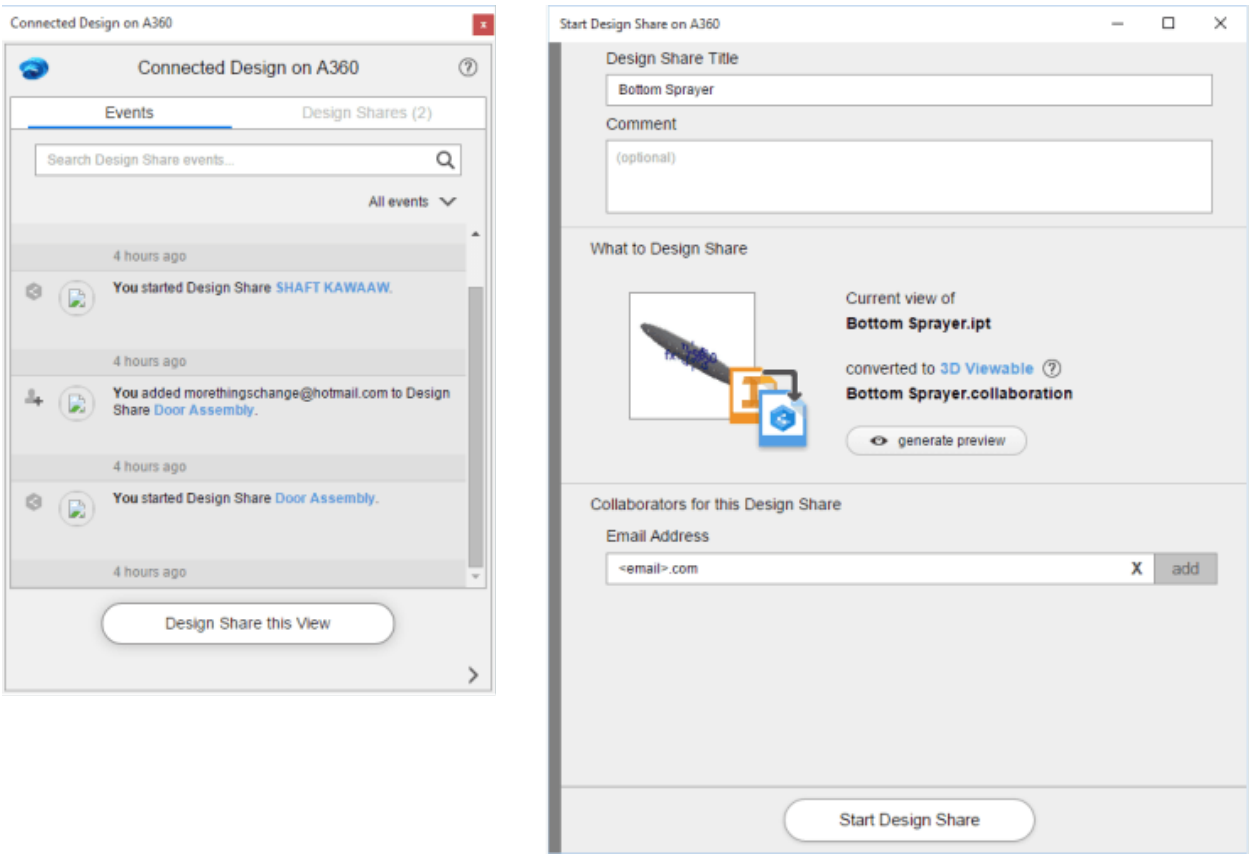
[Want to learn more about A360? Check out Kevin's blog on Mechanical Advantage]

The Design Share is created from the current view, so the first step should be to make invisible any components you do not want to share. You may also want to simplify the model. Basically, what you see is what you share.

The process starts by clicking the Design Share button located at the bottom right corner of the Inventor window. If you are not already logged into A360 you will be prompted to do so.

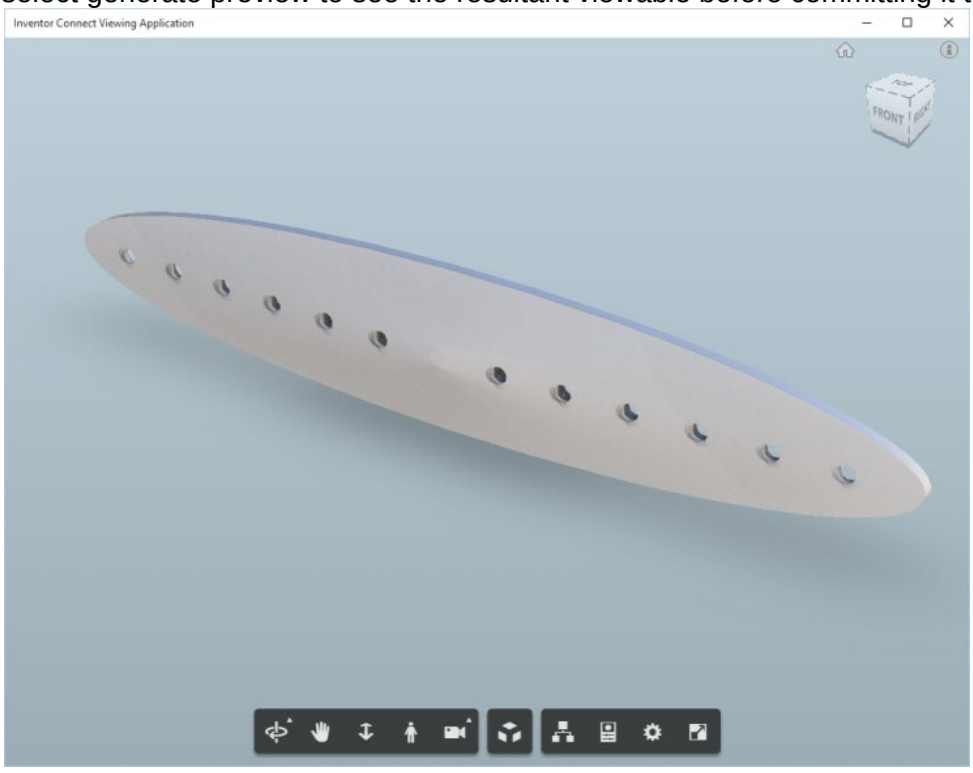
DESIGN SHARE BUTTON

This launches the Connected Design panel. Two tabs provide information about existing Design Shares. Pick Design Share this View on the bottom of the panel to create a new share. Specify the details including the title, a comment, and the email addresses of the required collaborators.



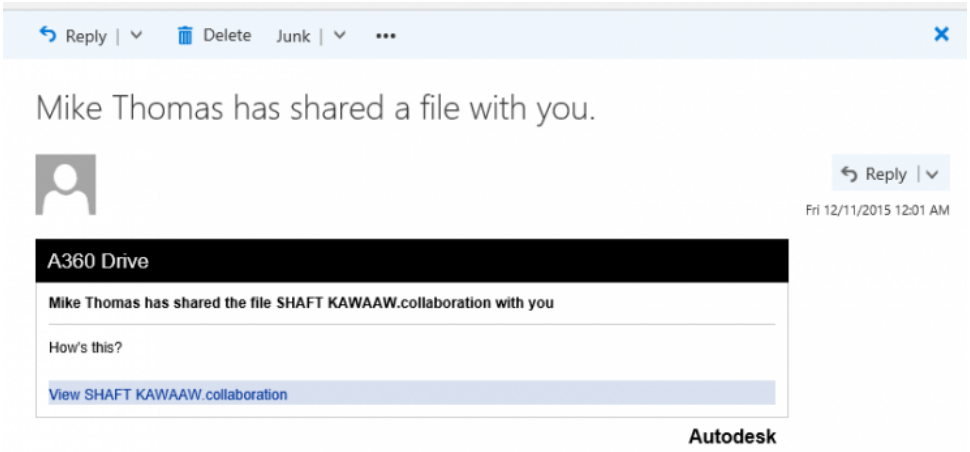
STARTING A DESIGN SHARE

Optionally select generate preview to see the resultant viewable before committing it to A360.



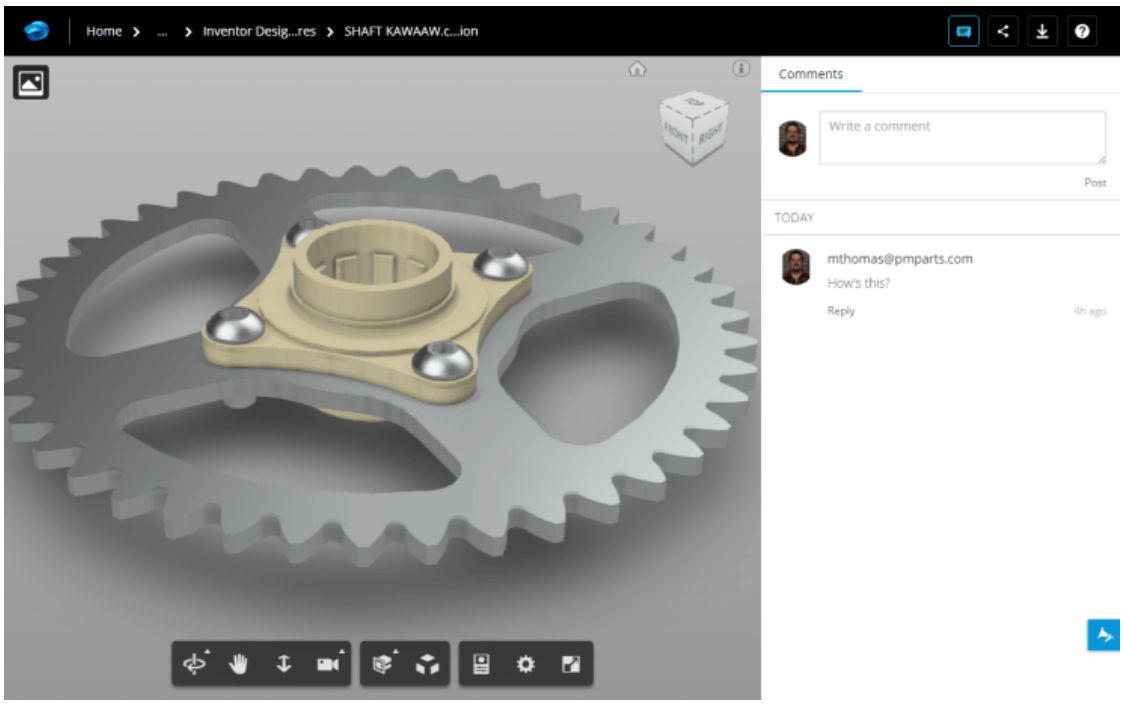
DESIGN SHARE

When satisfied select Start Design Share to publish the view to your A360 account. Your recipients will receive email similar to this.



SHARED FILE EMAIL

Notice the embedded link, meaning they can click and gain almost instant access to the file. The A360 viewable appears and functions just like the preview, but it does contain the option to add comments. The comments are visible to all collaborators with access to the Design Share.



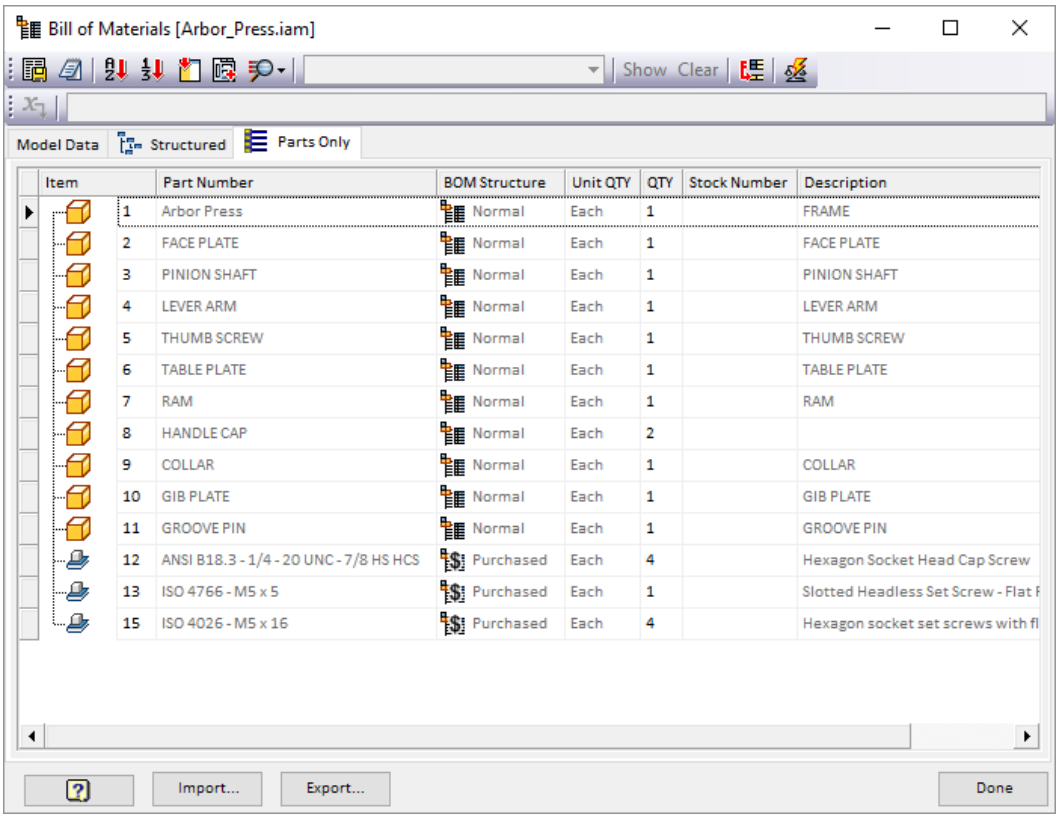
THE DESIGN SHARE

Important to note, currently the following is NOT published: The Feature tree, Part properties, & Component names

Bill of Materials

Inventor tracks the components within an assembly and the components meta data (iProperties). This data can be viewed and edited via the Bill of Materials (BOM).

The BOM can be presented as Structured (sub-assemblies and parts) and Parts Only.

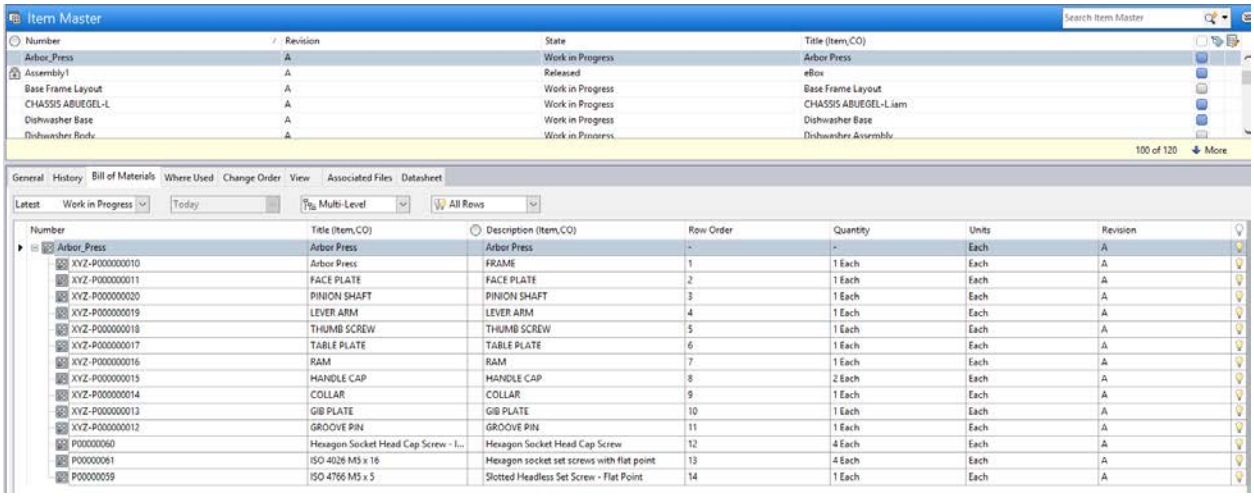


Item	Part Number	BOM Structure	Unit QTY	QTY	Stock Number	Description
1	Arbor Press	Normal	Each	1		FRAME
2	FACE PLATE	Normal	Each	1		FACE PLATE
3	PINION SHAFT	Normal	Each	1		PINION SHAFT
4	LEVER ARM	Normal	Each	1		LEVER ARM
5	THUMB SCREW	Normal	Each	1		THUMB SCREW
6	TABLE PLATE	Normal	Each	1		TABLE PLATE
7	RAM	Normal	Each	1		RAM
8	HANDLE CAP	Normal	Each	2		
9	COLLAR	Normal	Each	1		COLLAR
10	GIB PLATE	Normal	Each	1		GIB PLATE
11	GROOVE PIN	Normal	Each	1		GROOVE PIN
12	ANSI B18.3 - 1/4 - 20 UNC - 7/8 HS HCS	Purchased	Each	4		Hexagon Socket Head Cap Screw
13	ISO 4766 - M5 x 5	Purchased	Each	1		Slotted Headless Set Screw - Flat F
15	ISO 4026 - M5 x 16	Purchased	Each	4		Hexagon socket set screws with fl

The BOM dialog provides an export option to export the bill of material information for use in downstream applications.

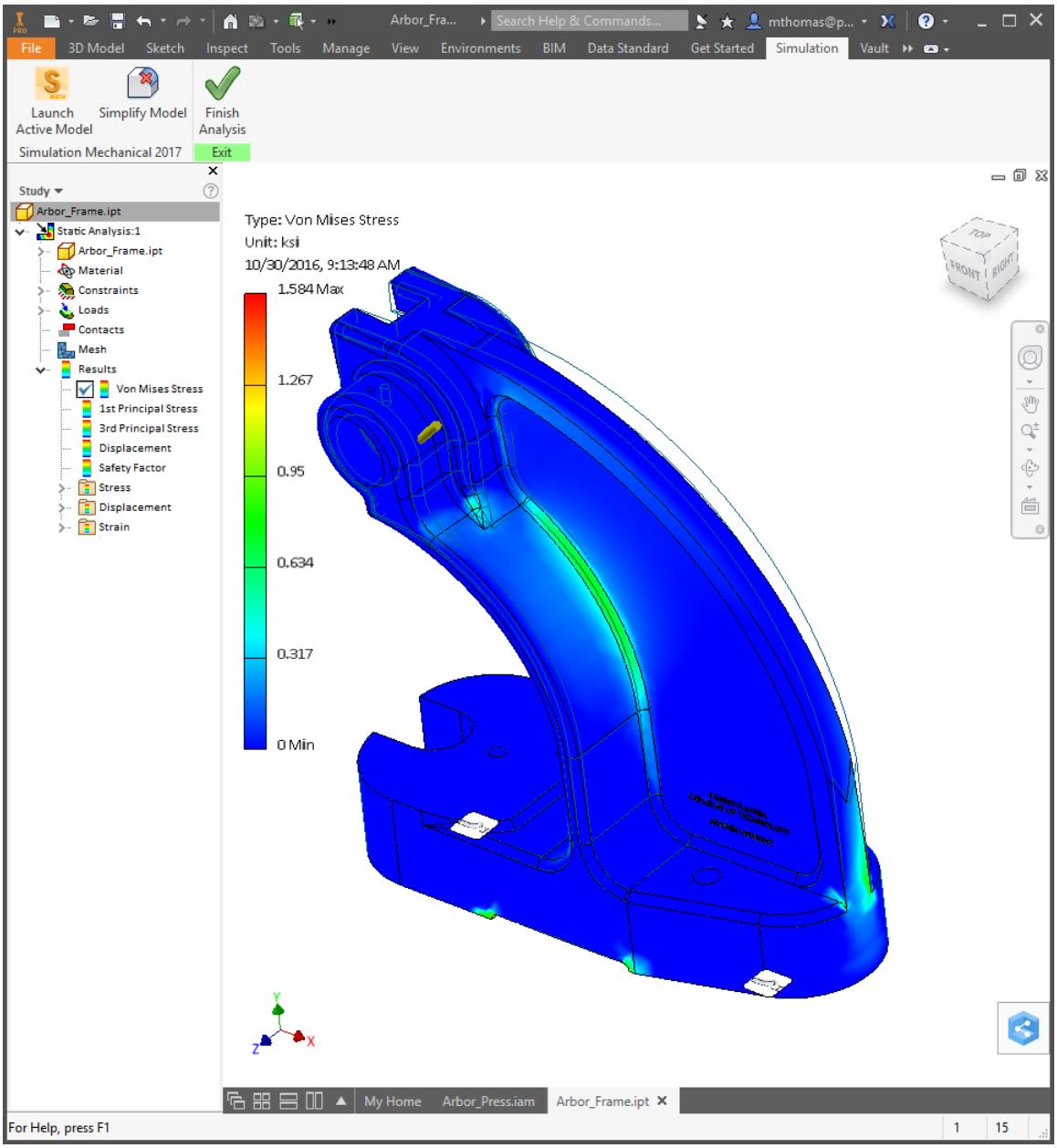
The BOM info can be exported to Excel spreadsheet, Access database, dBASE IV, Text (tab delimited, comma delimited), and Unicode.

When using Vault Professional the BOM is automatically transferred to the associated Vault Items. The Vault BOM information can be massaged to provide the desired view. Vault provides an alternate method of exporting the information for use in downstream applications. The advantage is Vault will revision track the information and offer workflows to prevent the sharing of data until it is approved.

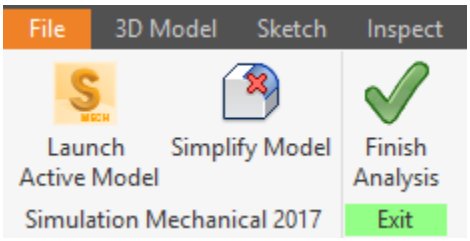


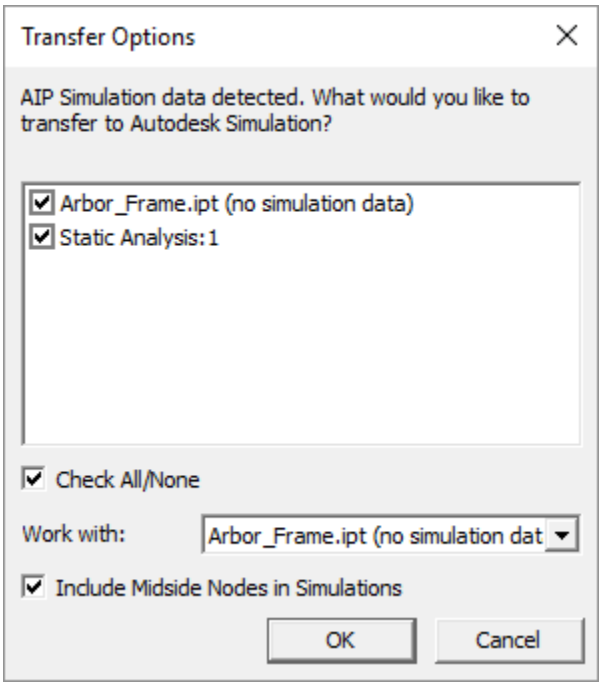
FEA

Inventor Professional provides in product stress analysis. With Inventor, you can run quick checks on parts or perform in-depth analysis of the entire assembly at any stage. Testing throughout the design process helps you iterate and improve your product's performance early in the process. Materials carry over from the model, and within the FEA environment you apply the loads. Inventor will automatically detect contacts between parts.



When you need more from your analysis you can “push” your model into Autodesk Simulation Mechanical. Autodesk Simulation Mechanical connects with Inventor, transfers the model and FEA data, and saves it to the Simulation Mechanical. fem file



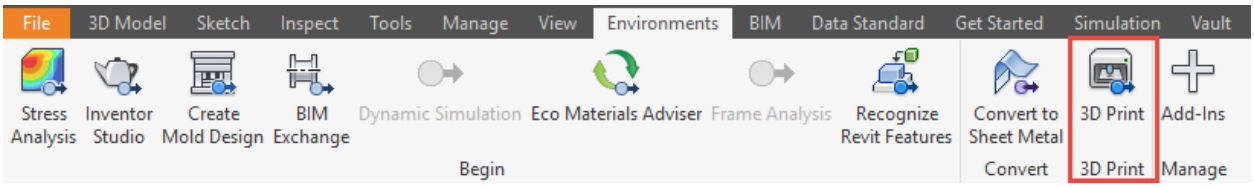


By initiating the transfer from within Inventor (push) additionally creates an associative link between the original Inventor model and the Simulation Mechanical analysis model. When the Inventor model changes the analysis model within Simulation Mechanical updates as well.

3D Printing

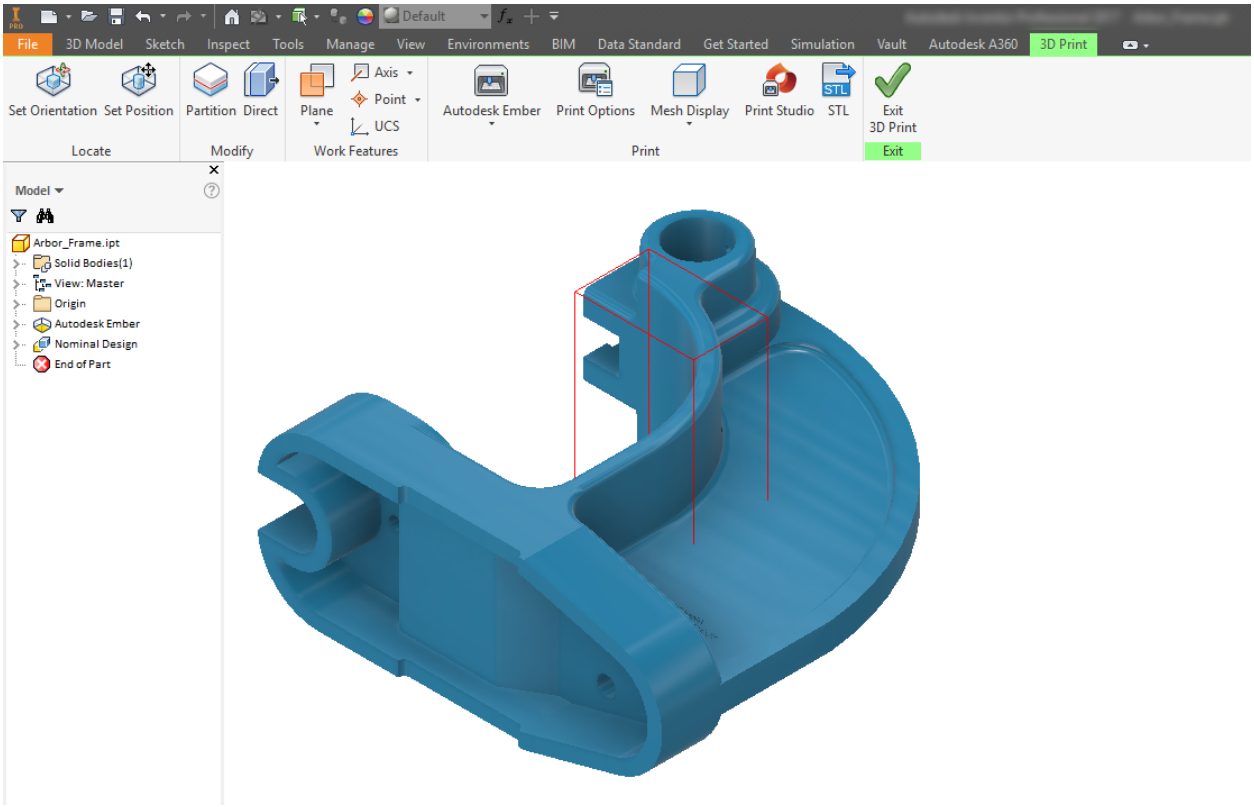
Use the Inventor 3D Print Environment to prep your part model for additive manufacturing. You can print models whole or in-part based on the selected 3D printer. To print an assembly model, you need to first save the assembly to STL.

The 3D Print Environment is accessed from the Environments tab

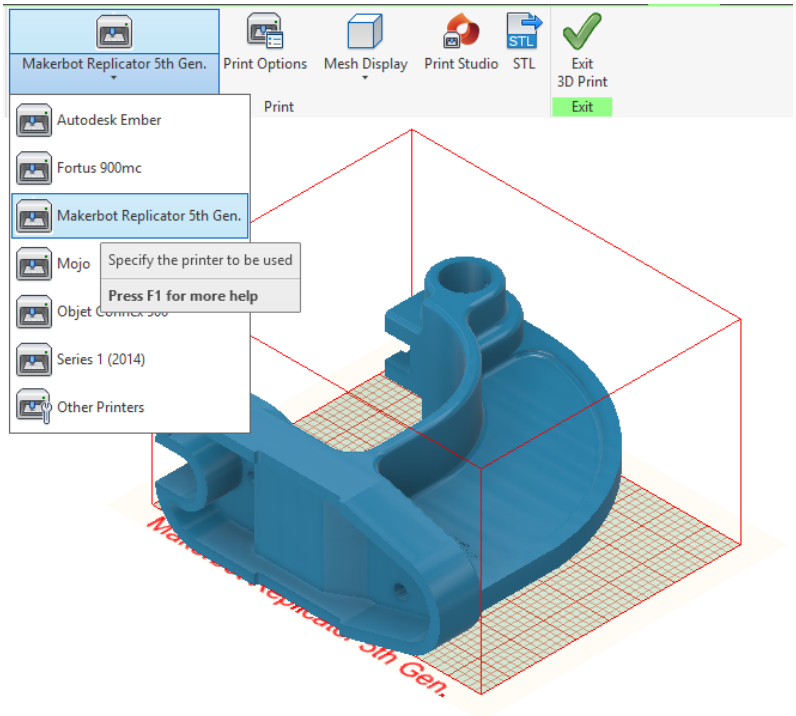


Only visible features are printed, meaning you can suppress features you do not want printed prior to launching the 3D Print feature.

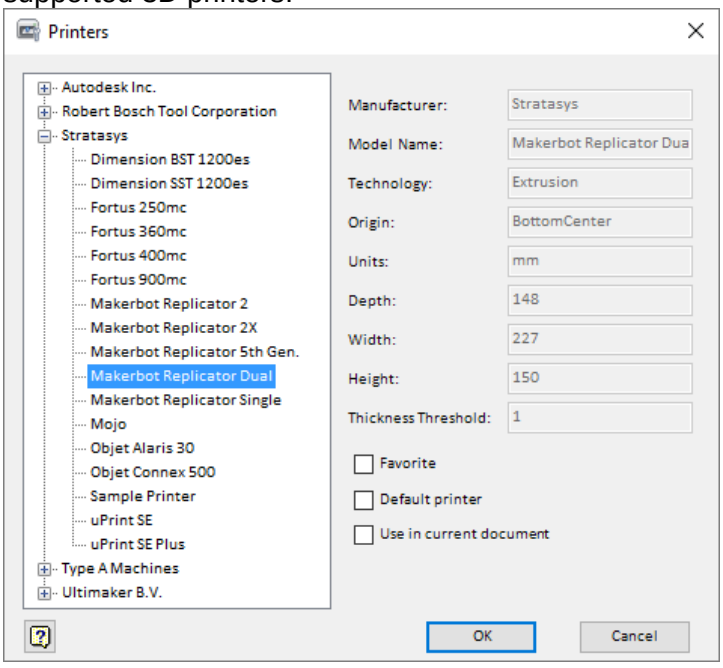
Most of the standard part modeling features are available while in the 3D Print environment. Adjustments to the model only apply to the 3D Print and do not affect the original model.



Use the printer feature to select the desired 3D printer. The printer build space updates with the selected printer's information.

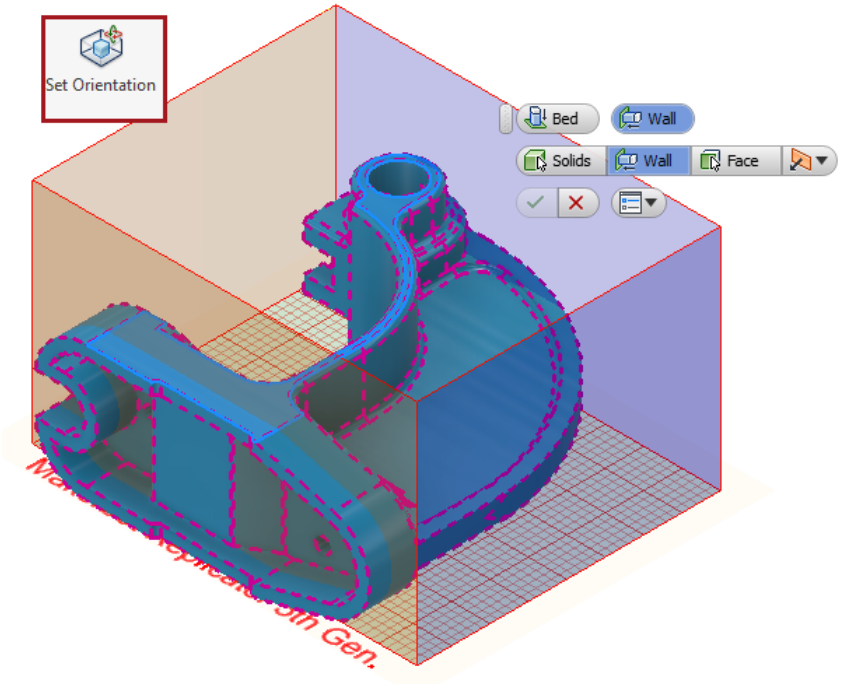


From the list select Other Printers to launch the Printers dialog box. This dialog lists all supported 3D printers.

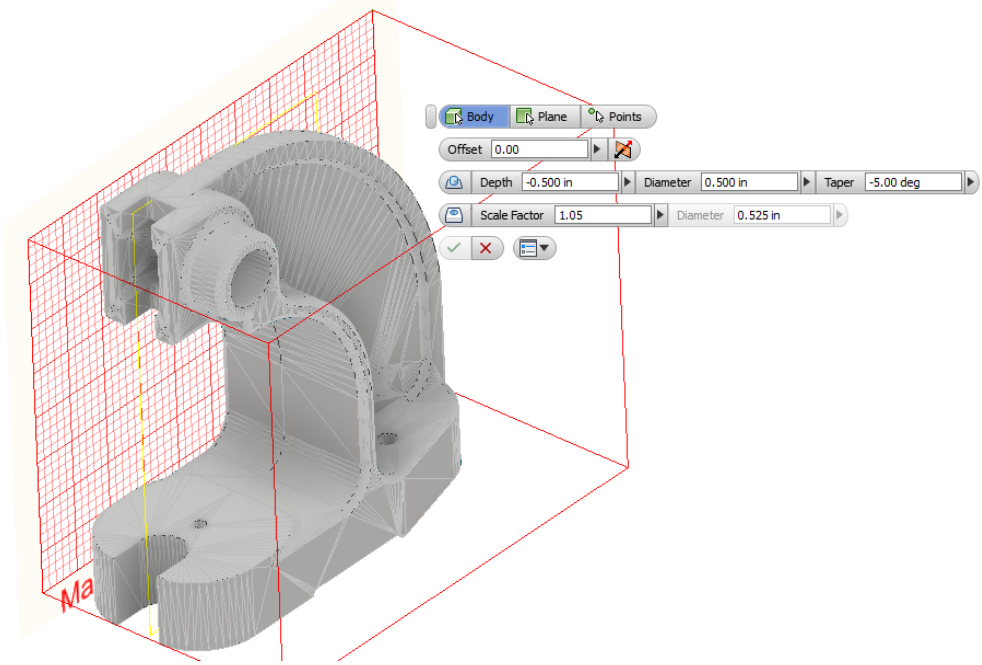


Tag the printer as a Favorite for it to appear on the drop-down menu. Set the printer as default to have the printer as the first item on the list and the item that appears in the ribbon (menu collapsed). Select Use in current document to make it the active printer and load its settings into the workspace.

By default, the Z-axis points upward. Adjust the orientation using the Set Orientation feature. Position the component relative to the build space walls using Set Position.



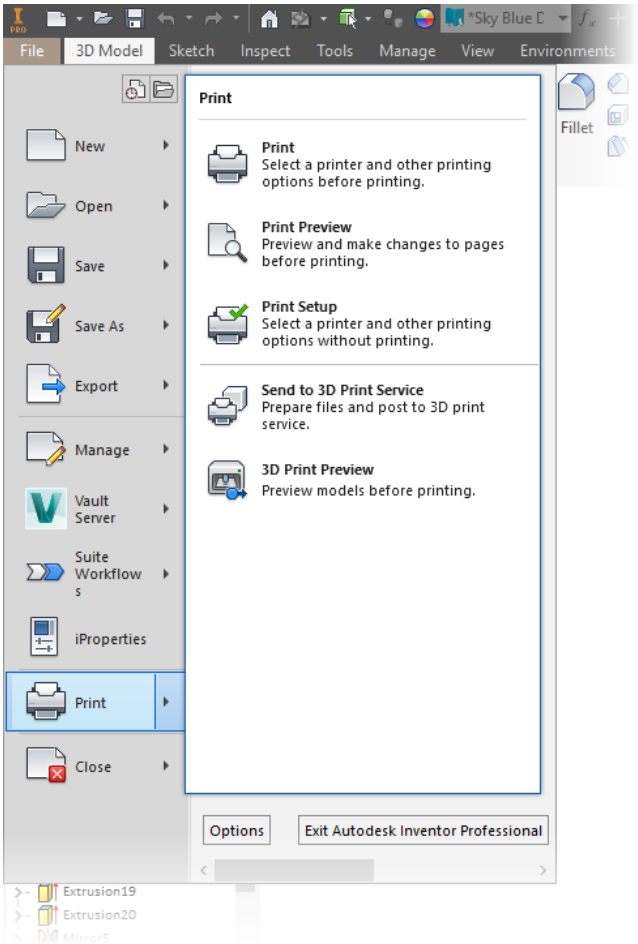
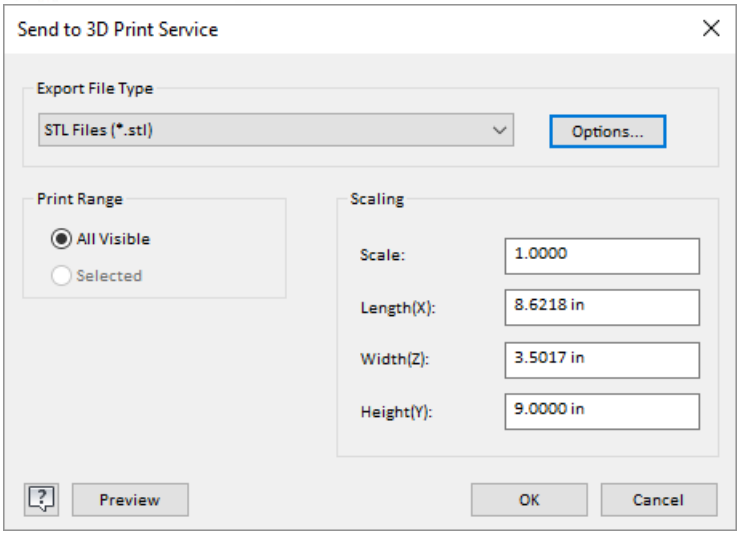
To print a component that is larger than the physical build space you can partition it into multiple bodes using the Partition feature.



A partition is composed of various features. When modification to the print model geometry goes beyond the Partition command capability, you can explode the Partition and work with the individual features. Explode decomposes the partition and exposes the features made it up.

For additional tools for working with the model prior to 3D printing use Autodesk's Print Studio. This application heals, prepares, and prints 3D models optimized for each supported 3D printer.

3D Printing Service



If you do not have access to a 3D printer, Inventor provides tools to prep and send your model to the 3D Print Service