

Interoperability

2-B | 10:30 – 11:45 am

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AMA – Interoperability

In the context of Computer Aided Design software, interoperability references the ability of different programs to transfer information between each other. This can be geometry transferring between CAD systems, data exported for Computer Aided Manufacturing systems, or even documentation transferring from CAD to ERP Systems.

While there is much flexibility provided by available tools, the price paid for this flexibility is often choosing which option is best for a given operation.

Here we hope to help 'clear the smoke' a bit and help provide some clarity on both the options available, and how to make the best use of this information. We'll start from the beginning, importing other data.

Import

The Autodesk product line can import several types of files. When faced with the decision of which files to use it is helpful to know which file types can be exported from the source and which types can be imported. One of the biggest criteria to consider is what you intend to do with the model.

Examples for the use of the data would be:

- 1) Inclusion in a design tool (such as Inventor, Autodesk Max, AutoCAD, etc) for use in a design process. (examples: *.stp, *.dwg)
- 2) Opening a 2D geometry in order to create 3D geometry from it in Autodesk Inventor. (examples: *.dwg or *.dxf)
- 3) Generating a drawing from a part exported from a non Autodesk source. (example: *.sldasm, *.prt, *.step files, among others)

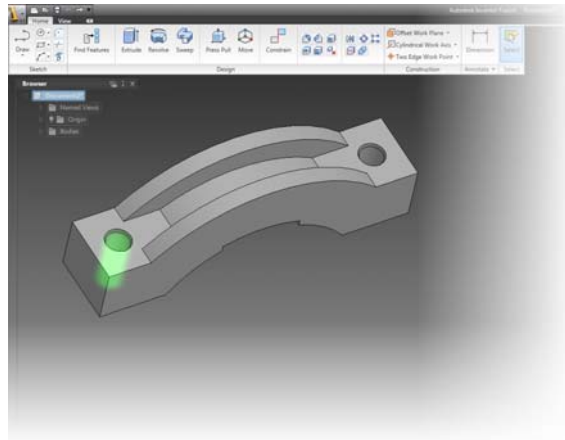
Any one of these options (and others) could play a role in what format the data is received in. In many cases, there may be more than one option available.

Preview into Inventor Fusion

The 'new kid on the block' Inventor Fusion allows 'direct manipulation' of solid formats. In brief, users can open a file format, make changes quickly, and return that file in a format usable in another software. While still in its infancy, the technology shows a great deal of potential in quick modifications and conceptualizations of geometry in multiple formats.

Currently Fusion has the ability to import, dwg solids, Inventor modeling files, dwg solids, STEP, and SAT files.

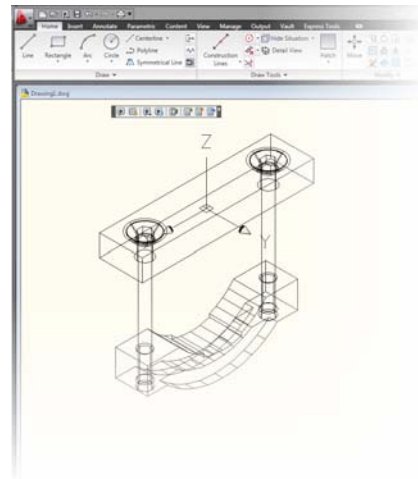
Step file imported into Inventor Fusion



Solids Editing

Once the data has been imported, what can you do with it? Formats imported in 2D will be editable, with geometry capable of being modified, and the ability to create new geometry. In AutoCAD, most of the solids will become AutoCAD solids. They can be manipulated as any AutoCAD solids, as if they were created in the native format.

Example of a Step file imported in AutoCAD Mechanical 2010



Inside of Inventor, most 3D formats will import in as a base solid (a solid without feature history), or as a surface (Mechanical Desktop files do bring in feature history in Inventor).

Model in Step format imported into Inventor



Models imported into Inventor may not have the detailed feature history of a model created in Inventor. However, you can still add features and cut away volume using the modeling tools. Materials can also be applied, components can be animated, and colors of faces can also be changed. So, while often referred to as a 'dumb' solid, the imported solid can still provide much useful information to the user.

Export

Once files are created, it's likely that files will need to be sent to other programs as well. As with importing, there are several formats available, and which file to choose from will rely on the destination of the file.

A few examples of export scenarios are:

- 1) Exporting a sheet metal flat pattern to a CNC machine such as water jet, laser mill, or punch press (*.dwg or *.dxf)
- 2) Exporting a model to create a rapid prototyping model from a 3D printer for early stage prototyping of form fit and function (*.stl).
- 3) Exporting a file for review by a user without an associated CAD program (*.dwf, *.dwfx, *.pdf)
- 4) Exporting Autodesk Inventor components in format for placement Autodesk Revit for integration into Building Information Models (*.adsk).

There are many scenarios that can be encountered, and one of the best things to do is communicate with the person or persons on the receiving end of the data. Find out what formats they can read, and send the files in one of those formats. When in doubt, try two or more formats if there's more than one possibility. You can then find out which format works best for your process, and proceed with that format from that point forward.

When exporting to another CAD system, the simplest place to start is the export format of the native program (Solidworks, Pro/Engineer, etc). Another tried and true method is to use a step file (which are still very effective).

Rule of thumb step for solids iges or surfaces

There are different standard formats available for each.

Specialized programs, such as CAM programs may be able to accept Autodesk files in their native format (such as *.dwg files for AutoCAD, or *.ipt or *.iam files for Inventor).

A specialized file format is *.stl, or stereolithography file. This type of file can be exported to rapid prototyping machines, such as 3D printers that can create prototypes of components for the purposes of form and fit.

AEC Exchange

Another specialized file format is the *.adsk file format. This format 'shrinkwraps' an Inventor file, creating a specialized, lightweight file that can be sent to Autodesk Revit. This creates a lightweight file that carries only the necessary information (such as connections and dimensions) into Autodesk Revit, without imposing details not required for the Revit model.

Features not required by the Revit model would include things such as small bends, holes, and other small features. While necessary to manufacture the part, they unnecessarily weigh down the Revit model. By removing these and leaving only critical information, Inventor data can be more efficiently reused in the Inventor models.

