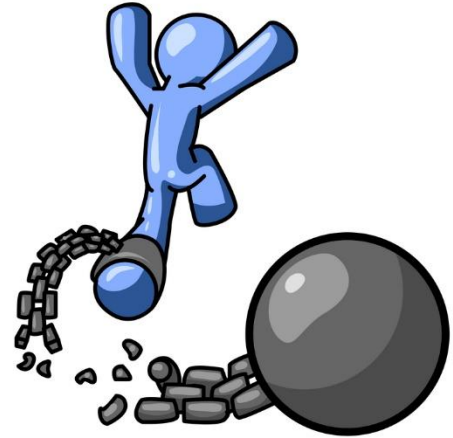


Creating and Removing OLE Links in Inventor Files

Autodesk Inventor uses internal hyperlinks embedded into assembly, drawing, and presentation files in order to ensure proper updating of files as the modeling and subassemblies change. This is similar to OLE (Object Linking and Embedding) in that when the referenced file changes the corresponding document updates accordingly.

Inventor will create OLE links in certain scenarios such as when images are inserted for use in Decals or for when an image map is used in a texture or appearance. OLE links are also created when running the default settings for Inventor's Finite Element Analysis module located in the Professional version of the software.



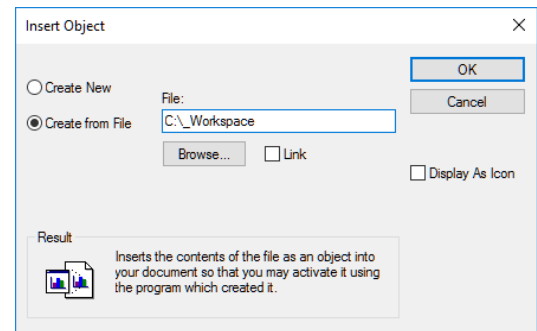
This whitepaper will discuss these OLE links, their overall ramifications to the design method, and the removal and prevention of the create accidental links that cause havoc to file maintenance.

Inserting Objects

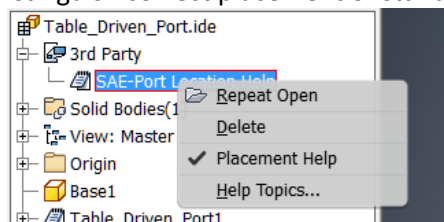
You can link or embed external files (such as spreadsheets or word processing documents) in an Autodesk Inventor model or drawing. When you insert an object, a 3rd Party folder is added as the first folder in the browser. All linked or embedded files are found under the 3rd Party folder.

1. On the ribbon, click Manage tab ► Insert panel ► Insert Object .
2. In the Insert Object dialog box, choose the options to achieve the desired result:

- To create a file, select the object type and select Create New. The application for the object type opens so that you can create the object. The new object is embedded in the model or drawing and does not exist as a separate file.
- To embed the content of an existing file, select Create from File then click Browse to find the file. After the file is embedded, you can make changes that do not affect the source file.
- To link to an existing file, select Create from File, select Link, and then click Browse to find the file. After the file is added to the Autodesk Inventor file, any changes you make also change the source file. Save the edits to show the updates in Autodesk Inventor.

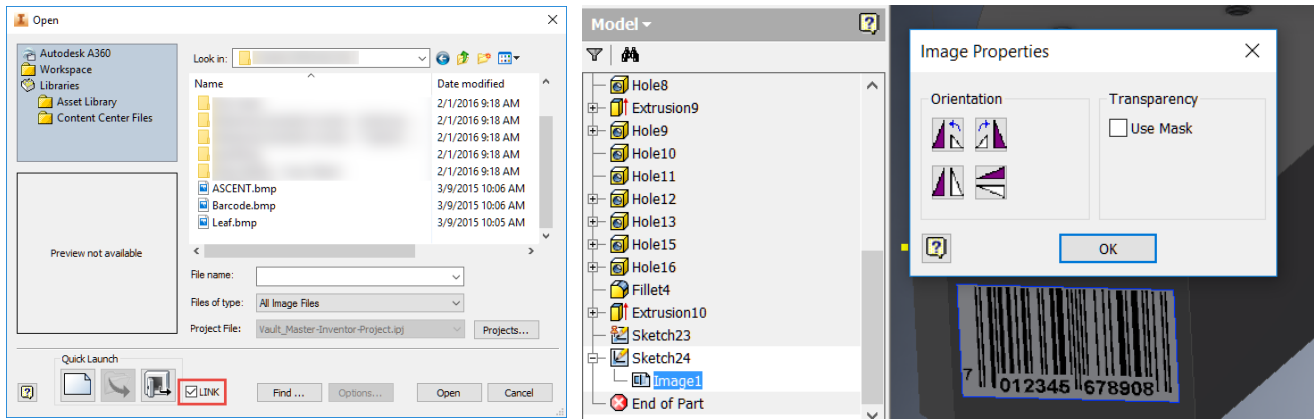


Common practice around Inserting Objects involves adding a specification sheet in PDF format or Excel spreadsheets used in iLogic programming or driven designs. These types of objects are also used in iFeature (.ide) placement help nodes to help teams distinguish correct placement of standard features.



Insert Image (2D Sketch)

Inserting images into Inventor 2D sketches is a common practice when designing decals for parts, title block definitions, or sketched symbols for drawings. Sketch Tab > Format panel > Image



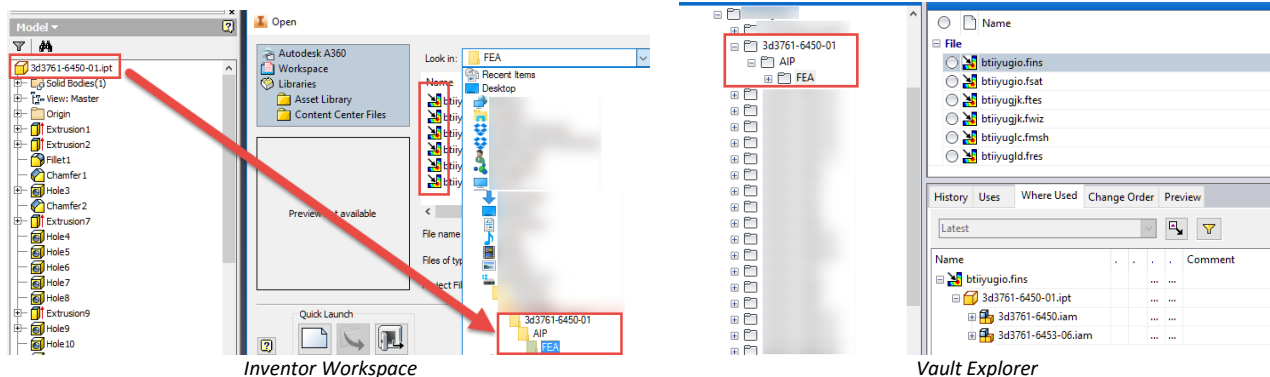
Perhaps the most crucial part of this step is to uncheck the Link box unless for some reason the design dictates that the outside image adjustment is needed to process across all files it is utilized in. Generally, this image is commonly lost during the movement of files or deletion of working directories. Autodesk Vault can handle the image link during a related files check-in to the Vault software, but in most scenarios the image is a onetime use image and therefore the Link is not needed.

Inventor Drawing title block definitions are usually the most common culprit of this Link scenario in reference to company logos. In regards to the Inventor drawing side of this discussion, the best practice is to actually sketch the company logo in vector form instead of using an image. This is ideal because when Inventor drawings are saved with embedded images as AutoCAD drawings the logo will recreate itself each time where the AutoCAD file is saved and linked images have issues when removed or located into incorrect folders. Both of these issues can cause headaches to the design team and downstream users.

NOTE: In Autodesk Inventor 2017 and up, there is a new Application Option to toggle off the Link box every time the Image command is chosen to avoid creating the links by mistake.

Preventing OLE Links with FEA Files

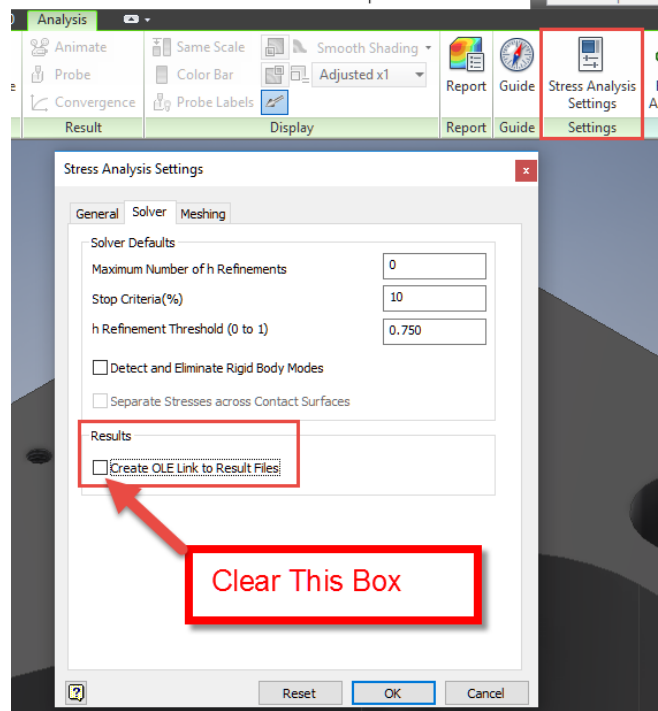
By default, Autodesk Inventor's templates have settings stored for all environments. Even though you may not consider modifying your base template for parts in a normal implementation, if you are going to use FEA files with or without Vault, it is a consideration to make. By default, Inventor parts and assemblies will create resultant files (mesh settings, that automatically add two things that complicates the design and file management process. The FEA links will create a folder structure as seen below and add the resultant files there. These would get added to Vault as well during a check in procedure and duplicate the folder structure there as well.



To curb the creation of these FEA resultant files, we can adjust some settings in the file we are working on with the FEA. Once in the Stress Analysis Environment, go to Stress Analysis Settings and clear the box “Create OLE Link to Resultant Files”.

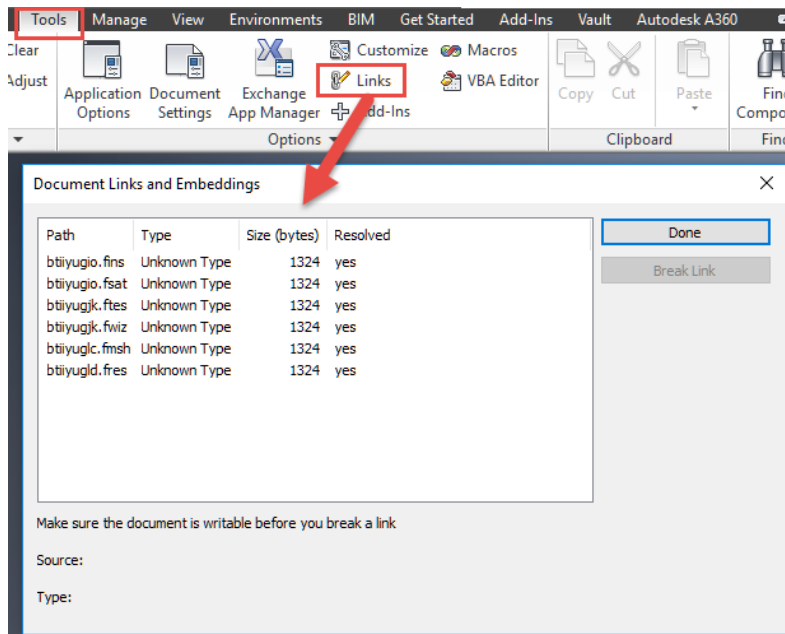
Performing this same procedure in Inventor part and assembly templates will also ensure this setting is turned off for all new files that are created from that point in time. The process is the same even though there is no geometry in the files. Start the Stress Analysis Environment, create a new Study and then adjust the same settings.

If Inventor has the Shape Generator tools from 2016 R2 or 2017, then this also creates resultant files and can be avoided by adjusting these settings in the standard templates as well.



Breaking OLE Links in Autodesk Inventor

In the event of any accidental or purposeful link creations, there may be a need to remove the image or other 3rd party document from the design file. To remove the external references from the modeling file, start the Links command found on the Tools Tab in Inventor. If no links are available, the command will be greyed out.



NOTE: In the event that during a Vault check in, there appears to be an image or other file that cannot check in and does not have an editable link in an Inventor drawing (.DWG), you may have to open the file in AutoCAD and detach the image using the XREF tools. Sometimes when organizations incorrectly implement Inventor by using an AutoCAD border as a starting point, it can create this issue.