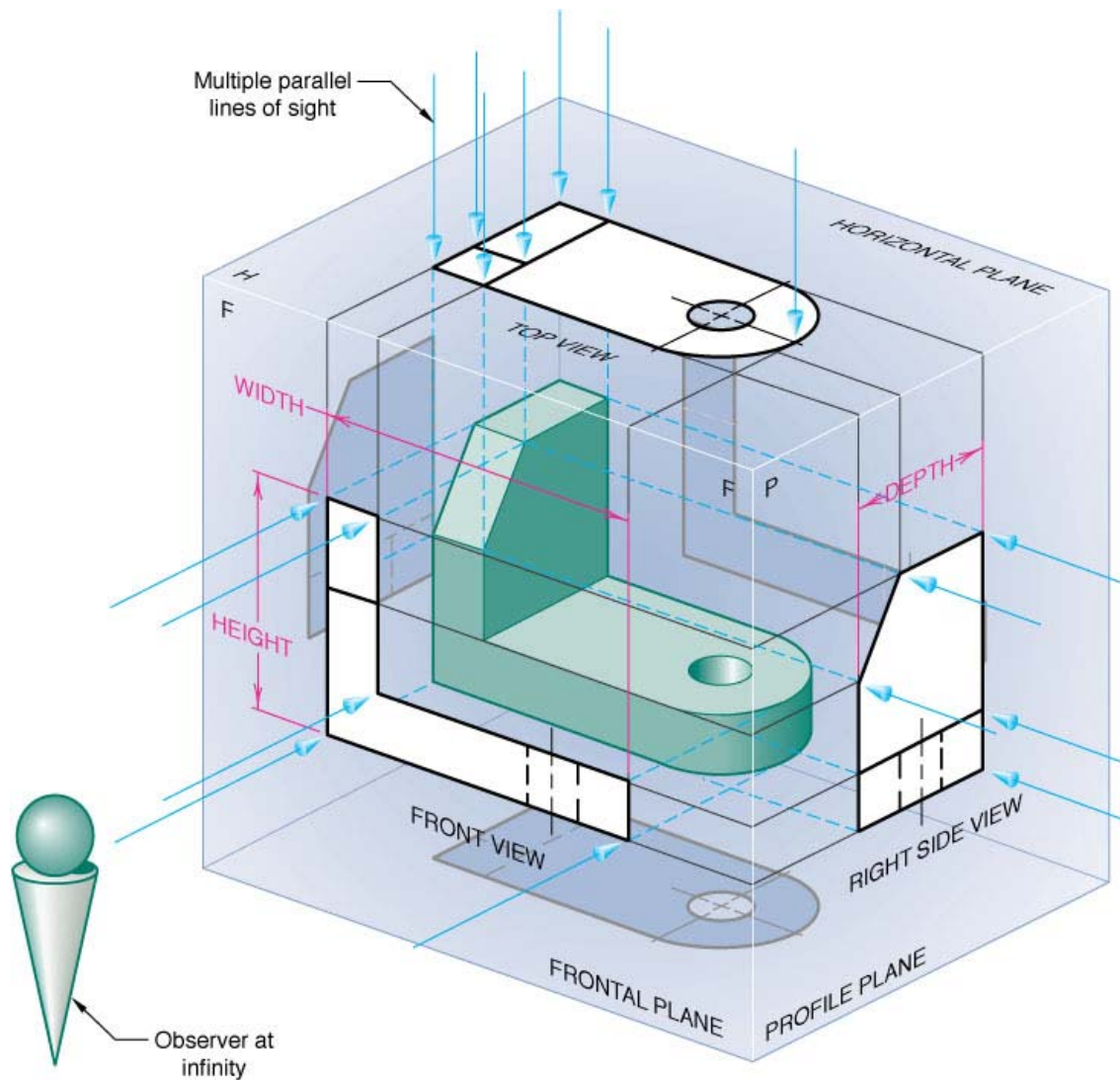


Leveraging 2D Data in 3D Modeling

As more and more companies switch to 3D modeling the question of utilizing existing 2D data and referenced 2D data from collaborative sources frequently arises. Users go through the gamut of questions including but not limited to:

- How valid is my 2D data?
- What are the benefits of migrating 2D geometry into 3D?
- How do I quickly create 3D models of 2D data?
- How much data can I reuse?
- How much data will need to be recreated?

The purpose of this white paper is to address some of these questions, their proposed solutions, and the most effective way to use Autodesk Inventor to aid in the process.



Benefits of 2D to 3D Migration

There are many benefits of 2D to 3D migration. On average users experience a 45% faster design cycle with 3D over 2D geometry. Some of these benefits reside in standard functionality of Autodesk Inventor including:

- Automatic Sheet Metal unfolding/refolding
- Automatic Bill of Materials
- Associative File structures to update changes automatically
- Physical information of components
- Assembly kinematics and FEA
- Plastic Part Design tools
- And many, many more



Multi-discipline collaboration becomes increasingly easier with 3D geometry than with 2D technical prints as well. Companies see benefits here residing in Inventor's overall graphical nature to share designs easily through DWFs, Studio Renderings, native and non-native file exports, and animations.

Autodesk Design Review also grants external users outside the CAD design environment the ability to measure, markup, print and track changes of the design in both 2D and 3D.

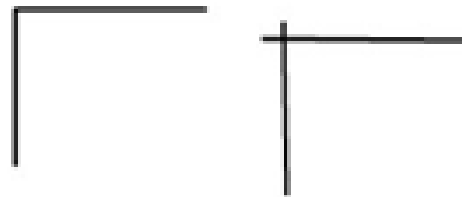
Autodesk®
 Design Review 2010



Evaluate the Quality of Existing 2D

The quality of existing 2D data should always be considered before moving it to 3D. AutoCAD has been around since the early 80's and a tremendous amount of data could exist in this format for existing companies. Over that period of time that 2D data crossed many engineers and designers throughout the products lifecycle. It is not uncommon for a wayward user under the pressure of a deadline to "fudge" a dimension or a revision change. This could be due to the project's deadline, improper back up of data, or lack of user knowledge/skill.

Some of the ideal conditions for the migration of 2D data are that the geometry is correctly placed on corresponding layers, correct 1 to 1 scaling, and valid multi-view projections. Other considerations are that end points meet correctly (improper use of Object Snaps); geometry that overlaps incorrectly, and varying elevations of lines.

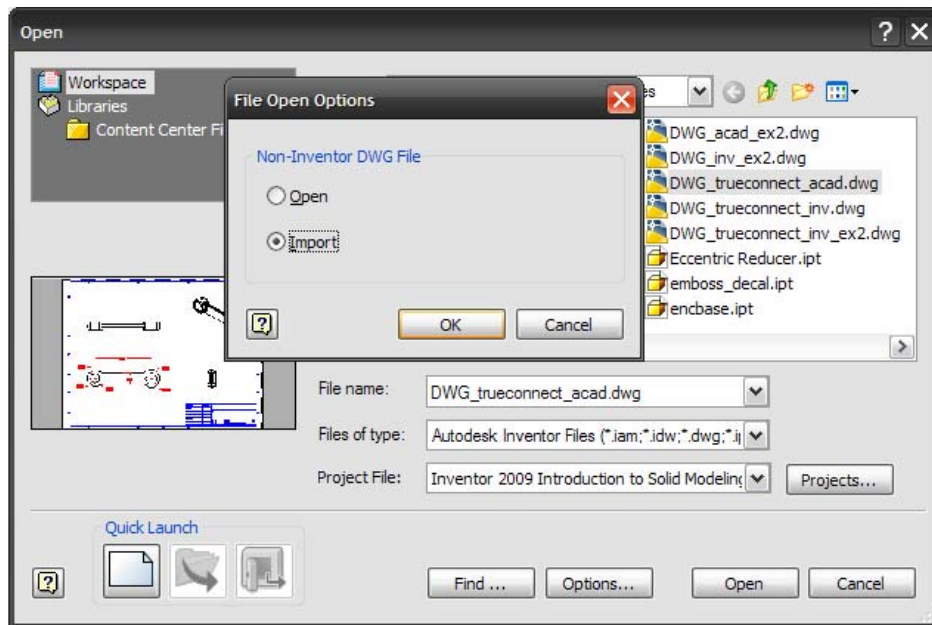
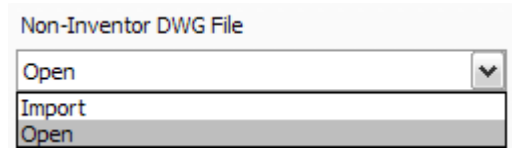


Ensuring that these guidelines meet your 2D data will make the process of importing the data into Inventor much easier. If not done correctly up front the geometry requires extra attention once it reaches a parametric sketch. It is often a recommended approach to take a sampling of your 2D data and examine it in this manner before making your final decision about the reuse and migration of your existing data. Following these guidelines will aid in future collaboration of 2D data from vendors and contractors as well.

Importing the Data

Once the 2D data is validated for import into Inventor we can begin the migration process. Inventor has two methods for working with AutoCAD data when chosen in the Open dialog box. The user has the option to choose which method with the Options button on the Open screen. There is a faster way through all of this but here is the way with the most available options first.

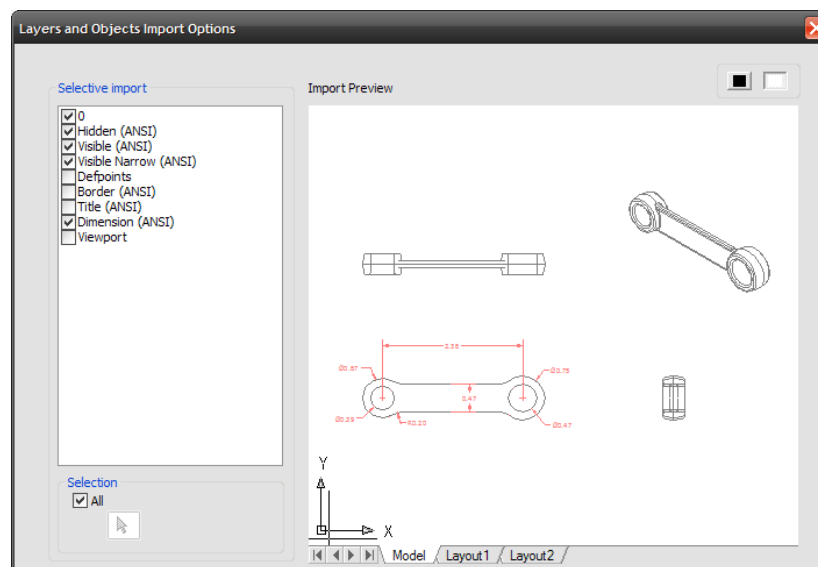
The **Open** option will use the DWG TrueConnect technology to open the AutoCAD drawing for viewing, measuring, copying, and printing of the data. **Import** will begin a wizard to walk a user through many ways information can be imported in.



Note: The default method depends on the Application Options on the Drawing Tab for “Non-Inventor DWG File”.

Using the Import selection will allow Inventor to automatically interpret the data as either AutoCAD files, Mechanical Desktop files, or DXF files. The next part involves interrogating what specifically will be brought into Inventor. The next screen allows selective import to disable mapping of individual layers.

Remove any layer that should not be imported into a sketch, drawing, title block, border or symbol that is not desirable for the intent of the import



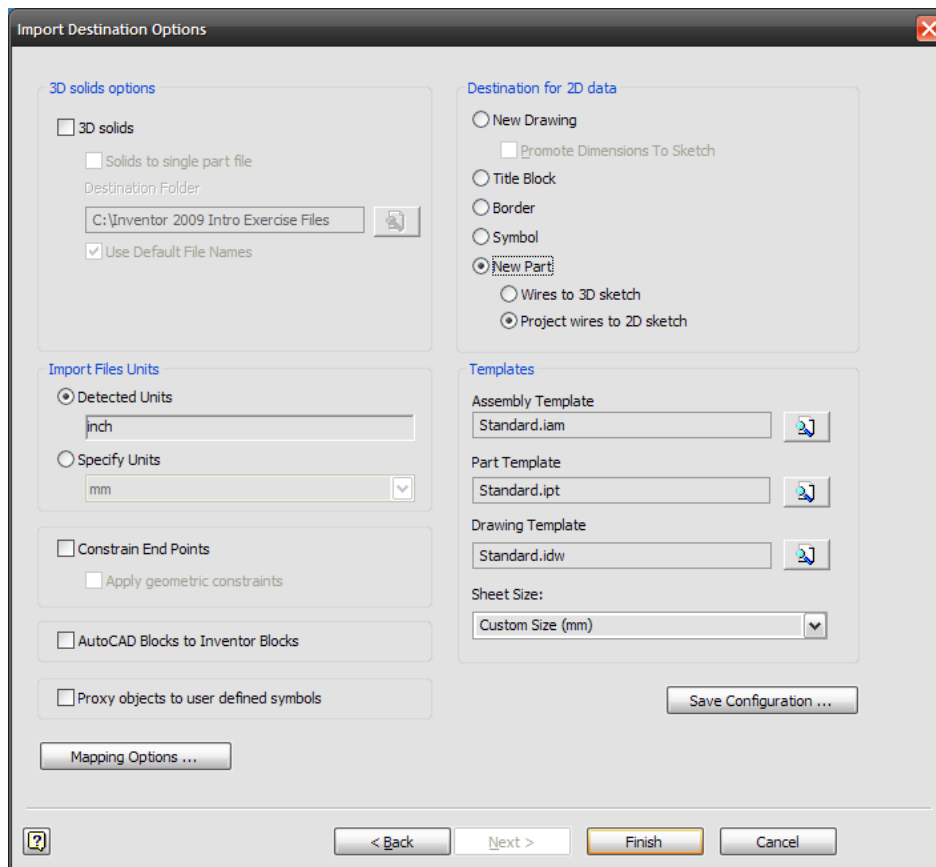
On the next screen many options arise. There are options for 3D solids if the originating file contained 3D AutoCAD solids, varying destinations for 2D geometry, different template selection depending on the geometry, unit control, sketch settings, and mapping options. Different destinations allow for the different options to become available based on the intent.

If the intent is to create a new Inventor part file the New Part radio button should be selected and usually the Project wires to 2D sketch sub selection. Beyond that there are a few other important selections for the New Part option.

Constrain End Points – Makes touching endpoints constrained with a Coincident constraint (recommended)

Apply geometric constraints – Applies Inventor defined constraints to remaining geometry (caution)

Mapping Options – Force layers to map to an individual sketch or create multiple sketches (1 for each layer)

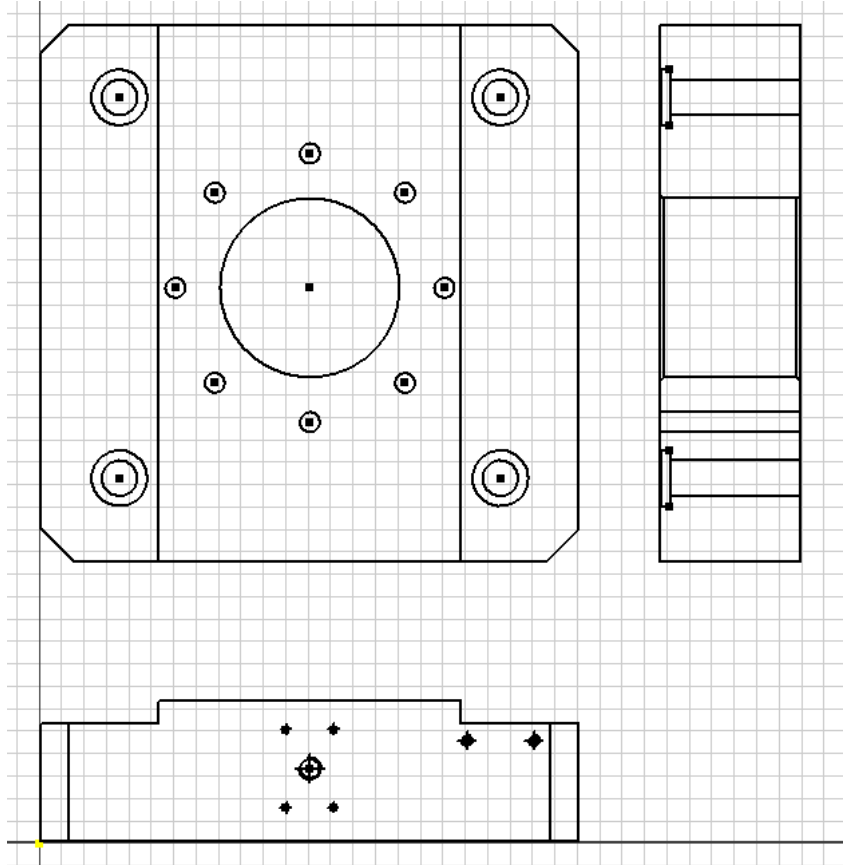


Using the “Constrain End Points” is recommended so that all the end points work like a normal sketch. The rest of the geometry could have been constrained with the “Apply geometric constraints” but this sometimes overcomplicates the intelligence of the sketch and makes it difficult to modify.

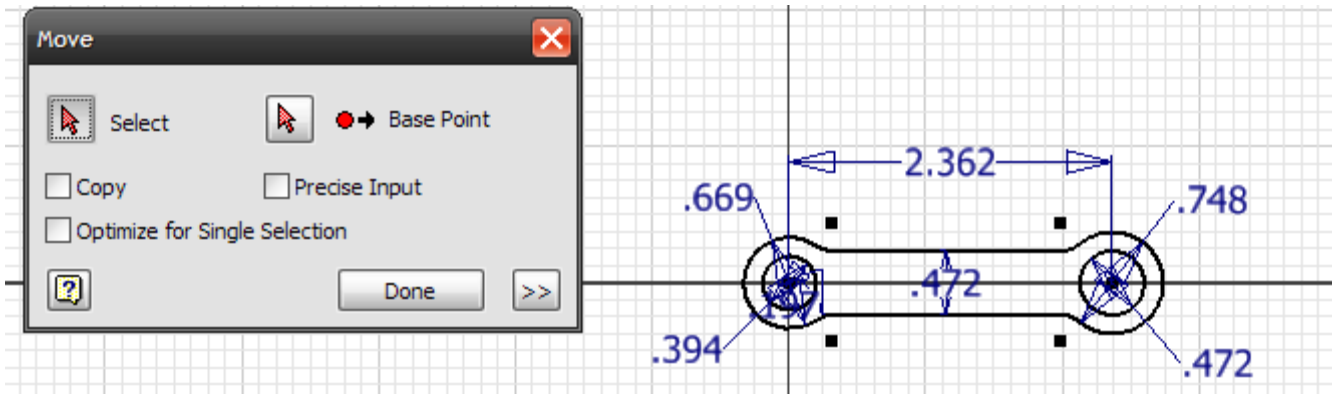
Note: Not using the Apply geometric constraints retains your geometry as flexible unconstrained entities, but if used creates a complex series of constraints. Consider window selecting the geometry and applying Fix constraints (🔒) to Fix all endpoints essentially locking the profile in place. This is a valid method since the geometry is already validated for size and shape.

The sketch imported in the figure to the right will be a master sketch to copy geometry from inside the model to multiple sketch planes on multiple work planes in order to accommodate the modeling process.

Start with one view creating features off the shape. Create a new sketch on another face and copy geometry from the master to the new sketch and so on until all the views are in the correct projection plane. This process is a tried and true method. So much that Autodesk created the 2D to 3D Utility; an add in to automate this process.



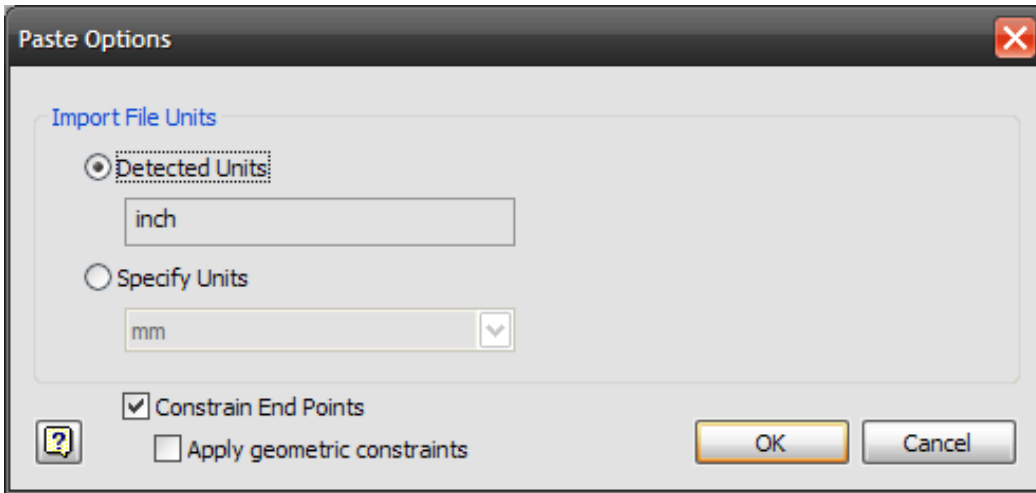
Anchoring the design with the first view is important to maintain strong modeling and assembly downstream. In this example the center of the left hole is placed on the origin of the model with a Move command. This allows for constraining of the Origin Axis inside the Assembly environment which is a very stable way to constrain.



This Import method allows the user the most control over what geometry comes in and where it goes, but may not be the best workflow for bringing data as it is validated for 3D migration. A faster method with less control for valid geometry is available to hasten the process.

The Quick method:

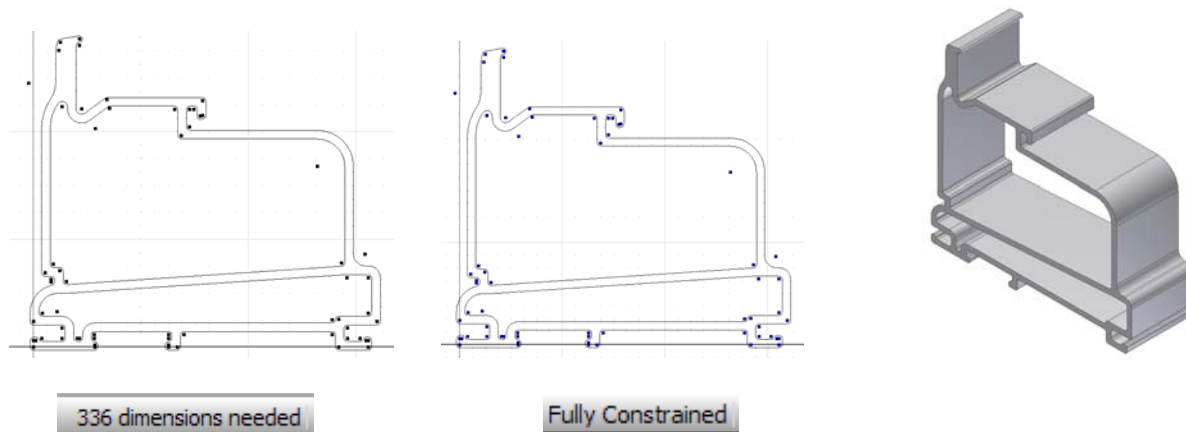
1. Open the Drawing in Inventor or AutoCAD
2. Turn off Layers not needed for copying (Annotate Panel; Format Tab in IV2010)
3. Select the geometry you wish to copy into a sketch
 - a. Window Selection in Inventor (use Shift for multiple selection windows)
 - b. Window Selection or QSELECT in AutoCAD
4. Right click and select Copy
5. Go to new Inventor file
6. Right click and select Paste
7. Before clicking to place the geometry right click again for Paste Options



8. Click on screen to place the geometry
9. Anchor the Sketch to the Origin geometry or corresponding geometry using the Move command
10. Continue copying and pasting to new sketch/work planes for projected views

Tip: If your 2D migration data is only a simple front profile to be extruded...

1. Follow steps 1-9 above
2. Window select the geometry
3. Click the Fix constraint (🔒) to Fix all endpoints essentially locking the profile in place

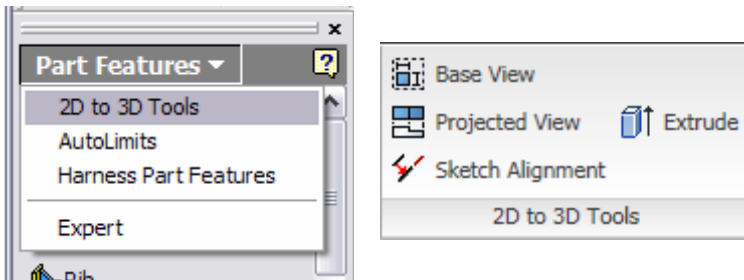


2D to 3D tool

The Autodesk 2D to 3D Technology Preview for Autodesk Inventor arrived on the Autodesk Labs site during the Inventor 11 release cycle and has been an invaluable tool ever since. The utility essentially takes a single imported 2D drawing that was placed into a sketch and builds the required work planes and sketches in order to accommodate quick 2D to 3D migration.

2D to 3D Tool for Inventor

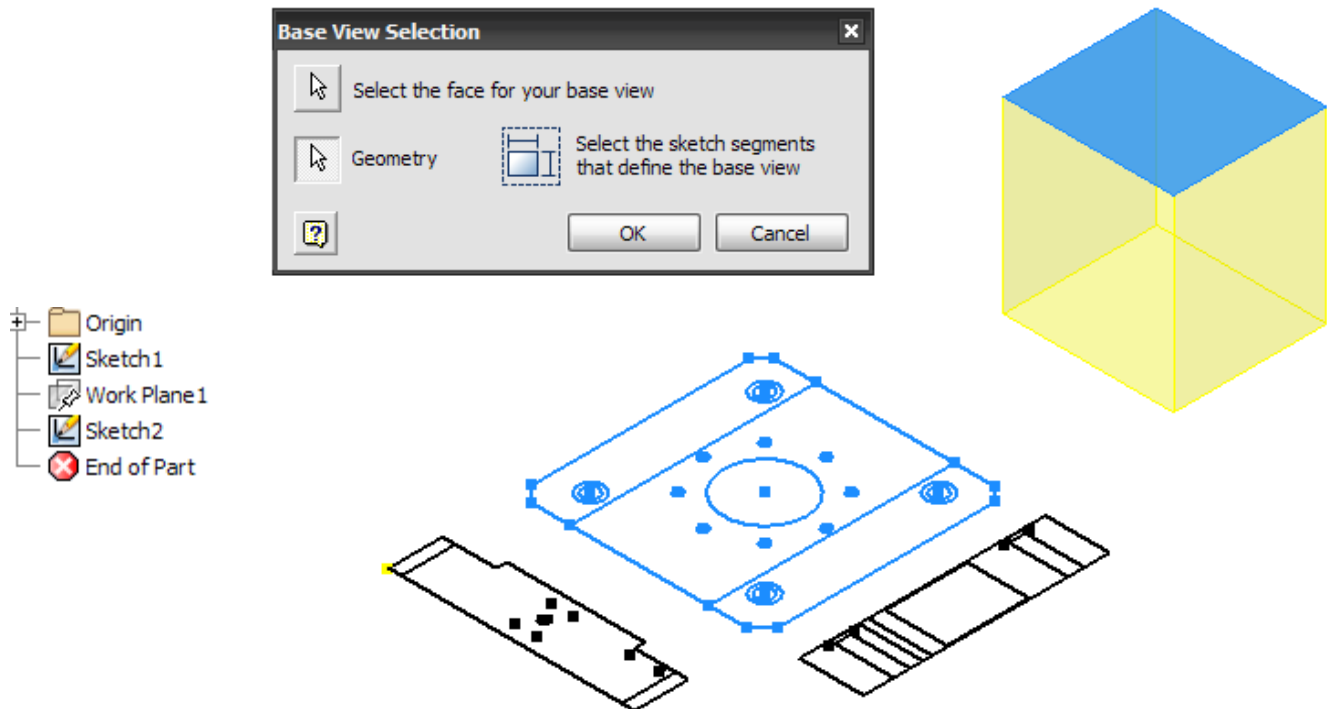
To correctly utilize the tool follow the same aforementioned guidelines including correct multi-view projection in the AutoCAD file, mapping all layers to a single layer and constraining all endpoints.



In order to gain access to the 2D to 3D Tools, the part file must be empty and contain only one initial sketch. The tools are found in the main part mode under the pull down for Part Features in R11, R2008 and R2009; and on the Add-Ins Panel in 2010. (At time of authoring the 2010 code has not been fully updated)

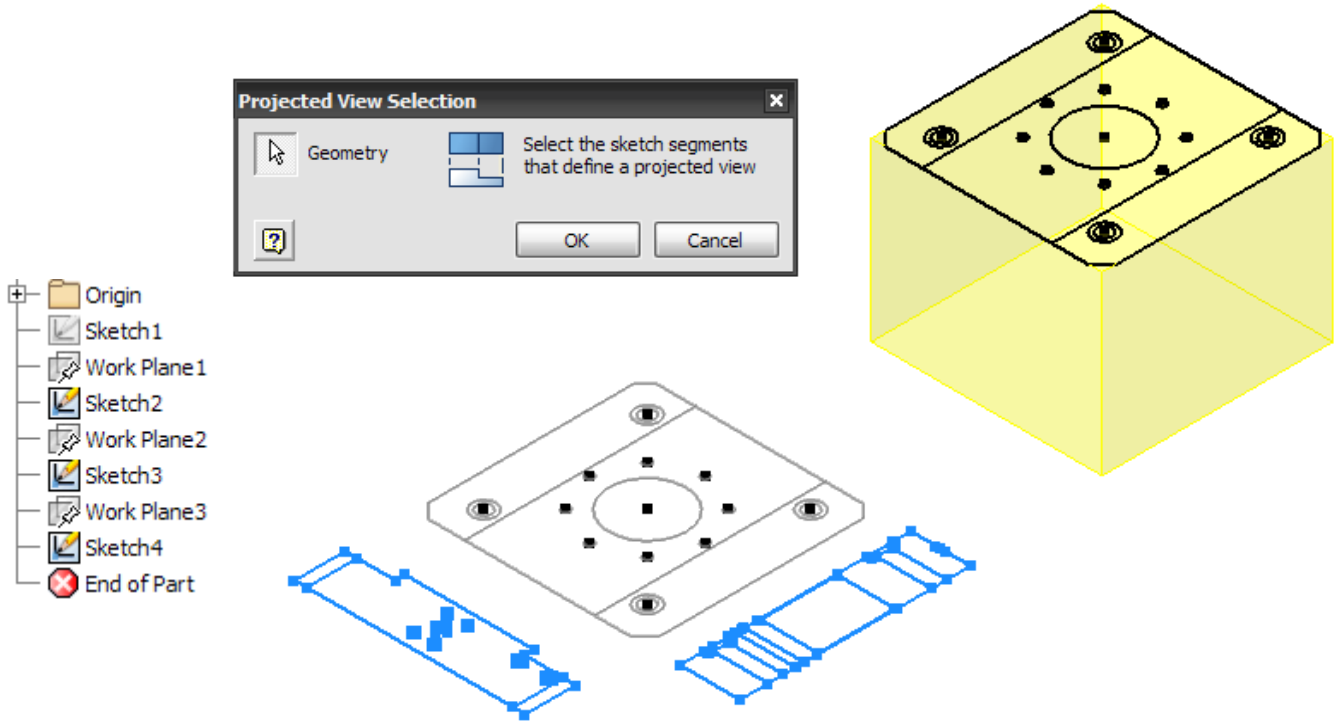
Step 1: Base View

Starting this command orients the sketch in an isometric view and produces a reference cube to map views to. Select a face of the cube and then select a logical geometry set by use of Window Selection. Click ok will remove the geometry from the sketch and map it to the face as well as creating a new sketch and work plane.



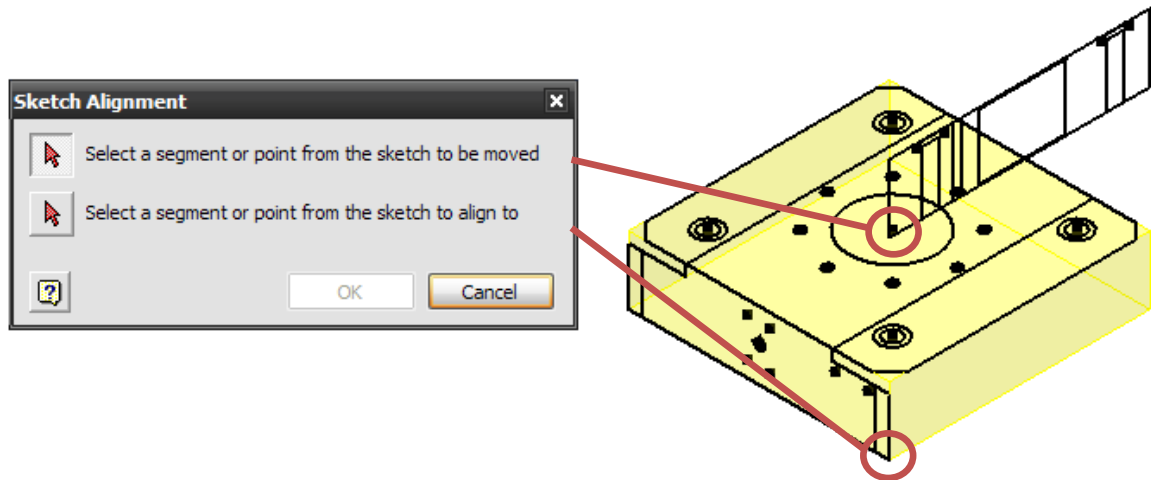
Step 2: Projected View

Multiple Projected Views can be selected all in one command. Placement of the projected views is in relation to the original base view. Additional sketches and work planes are created once again.



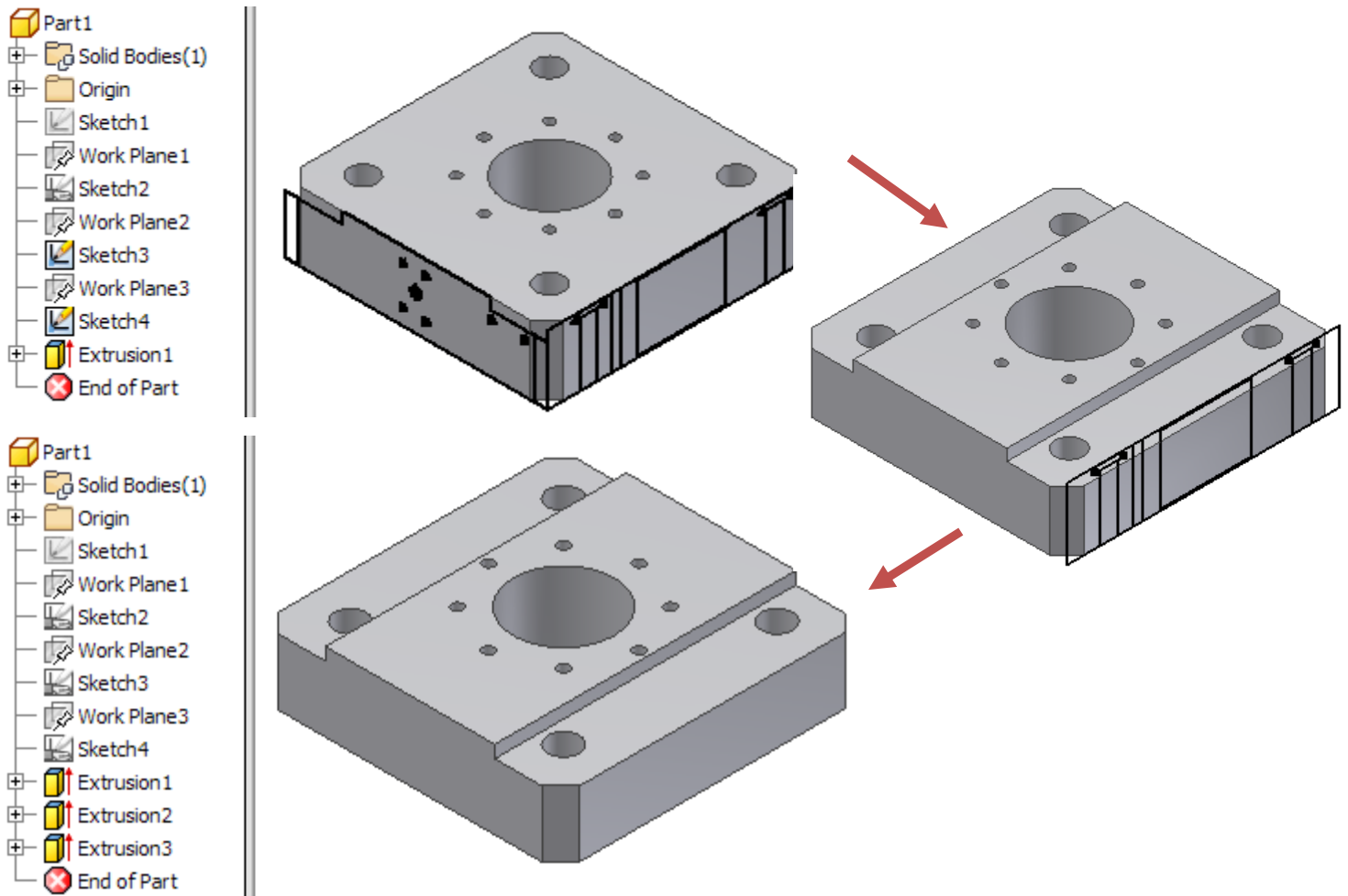
Step 3: Sketch Alignment (Optional)

Sometimes it becomes necessary to adjust the alignment of the sketch geometry to match view projections correctly. Select a segment or a point from the incorrect placement and then click a corresponding segment or point to align it to on the corresponding projection



Step 4: Extrude

This is the same Extrude tool as in Part mode. In order to recreate the geometry a series of Extrusions with Join, Cut, and Intersect is required.



Remaining features afterwards can then be applied if they did not translate well or were overlooked during the extrusion process. At this point the geometry that was created did not require any dimensions, manual sketches, or user work planes to reach the 3D model. On average this halves the time for migration work.

The downside to the 2D to 3D tool is that it does not anchor the geometry to the model's origin. In Inventor 2010 this can be rectified with the **Move Bodies** command.

Enable the Measure command and obtain an X, Y, and Z offset from the Origin to the point desired to be placed at the Origin. Input the opposites of these values into the Move Bodies dialog box. For 2009 and older there is no easy way to re-establish the origin without using some advanced methods.

