It's All in the (Level of) Details: Simplification Strategies for Autodesk® Inventor®

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Keep it simple. Easy to say, harder to do. In the exacting world of manufacturing, complete detail in a model is an absolute. However, details come with a price—performance, protection of intellectual property, and more. Can a mechanical designer have it all? This class shows you how to get all the detail you need and all the detail you don't. Starting with the concept of visual fidelity versus visual identity, we demonstrate the tools attendees need to successfully create a simplification strategy for detailed Inventor software models with minimum investment and maximum return. From advanced techniques for level of detail (LOD) use to incorporating construction geometry in your Content Center files to easily create shrink-wrapped models, this class gives you advanced tools to manage your models. If you deal with massive-sized assemblies, create BIM-ready content, or need to control intellectual property through managing the details you disclose, this class is for you.

Learning Objectives

At the end of this class, you will be able to:

- Use the principals of visual fidelity and visual identity to create a simplification strategy
- Use construction geometry in your simplification process
- Use advanced Level of Detail techniques to enable your simplification strategy
- Incorporate your simplification strategy directly into Inventor Library and Content Center files for reuse

About the Speaker

Matt is a Senior Consultant with Autodesk Global Services, where he develops custom training and workflow solutions for Autodesk® customers. Matt worked for several years as both a Product Designer and Product Architect for Autodesk® Inventor®; and has worked for the last several years helping customers implement the best possible processes around Autodesk Solutions for Digital Prototyping, Data Management, and BIM Ready Content. Matt has over 15 years of design experience in CAD, Mechanical Design, Systems Design, Simulation, Visualization, and Manufacturing.

Terms Used in this Handout

Term	Definition
LOD	Level of Detail: A representation created in Inventor Assemblies were the purpose is to manage capacity (i.e. memory use) of Large assembly models.
View Rep	View Representation: A representation created in Inventor parts or Assemblies that manages color, visibility, and even camera angle in the graphics.
Substitute or Substitute Part	An Inventor Part file that is created as a "stand-in" for the components of an assembly. When a Substitute LOD is created and activated, the child components of the assembly are removed from memory and are replaced by the associated Substitute without effecting the Bill of Materials and maintaining (typically) the constraints.

Tag-It Examples

The following shows step-by-step examples on how to tag files for easy selection.

Standard Files

For "standard" Inventor files (i.e. files that are not iParts or iAssemblies) you can add tag information to multiple files at once using the Inventor Bill of Materials editor.

1. Create a new Inventor assembly.
2. Add files to be tagged to the assembly, either using the Place Component command or by selecting in Windows Explorer and drag-and-drop into the Inventor assembly window.
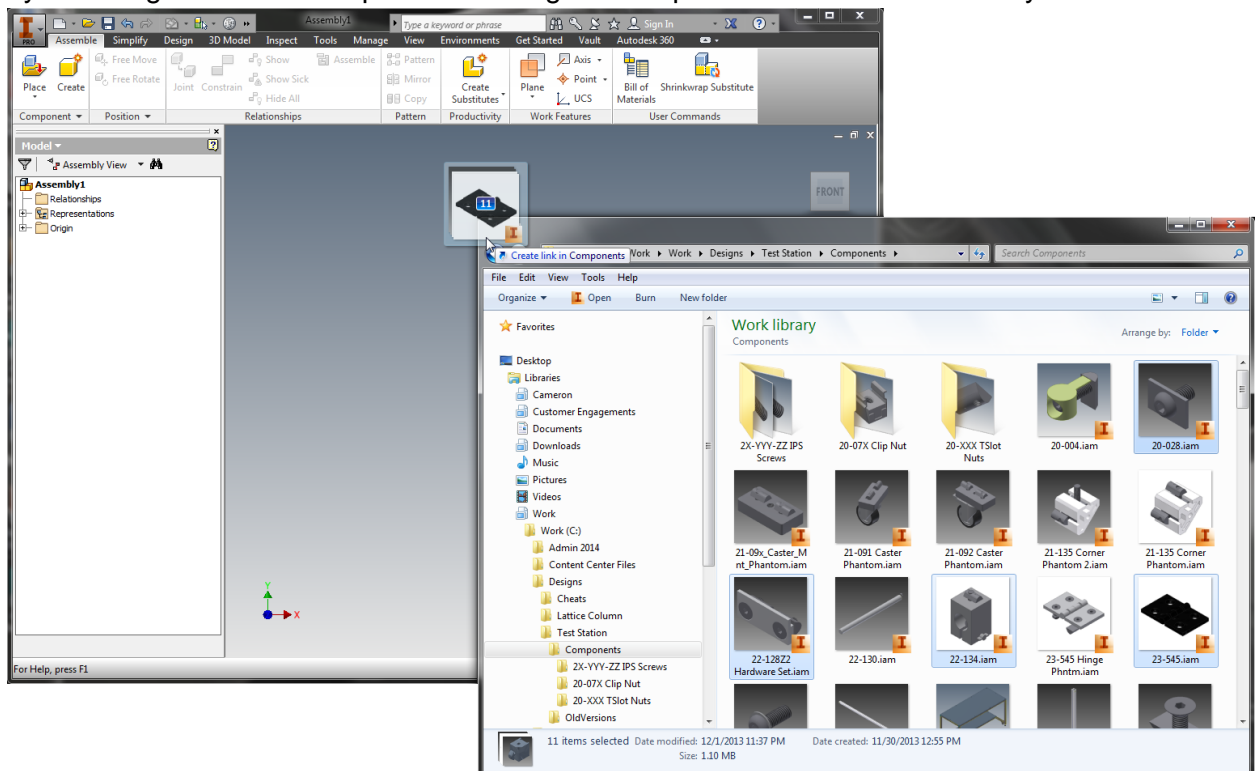
3. Once added, start the Bill of Materials command.
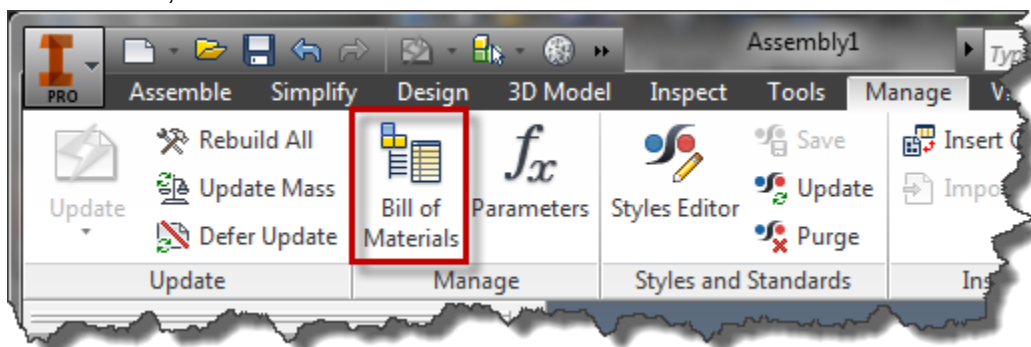
4. Add the Category iProperty (or another built-in iProperty of your choosing) to the editor using the Choose Column command.
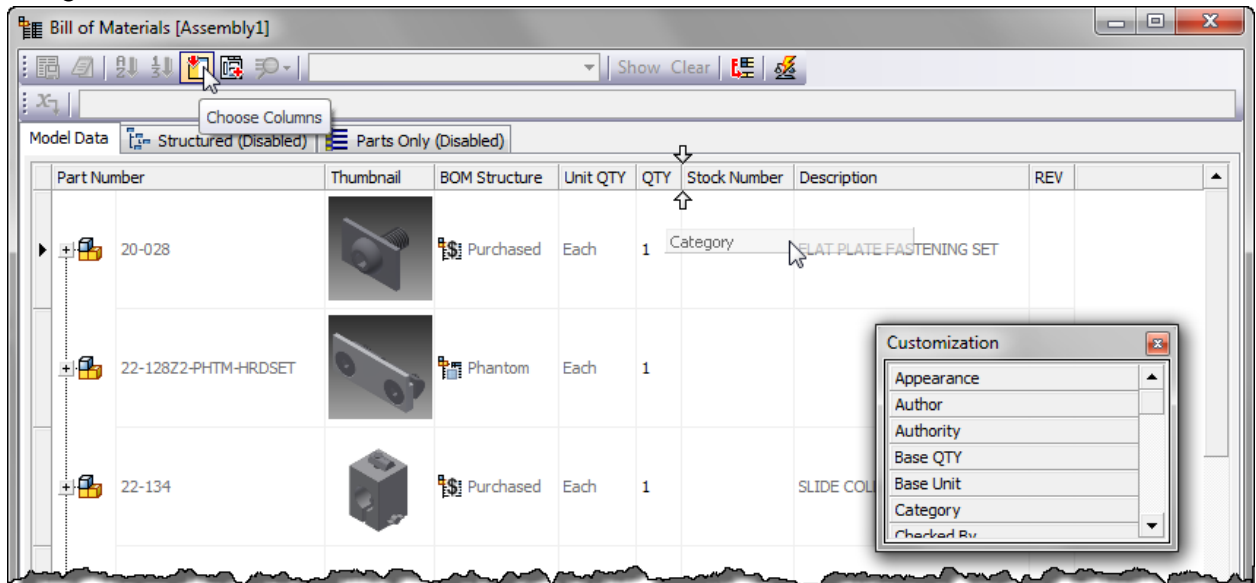
5. You can now add the values to the property as needed. Use Excel like operations for copy/paste, filling multiple cells, etc.
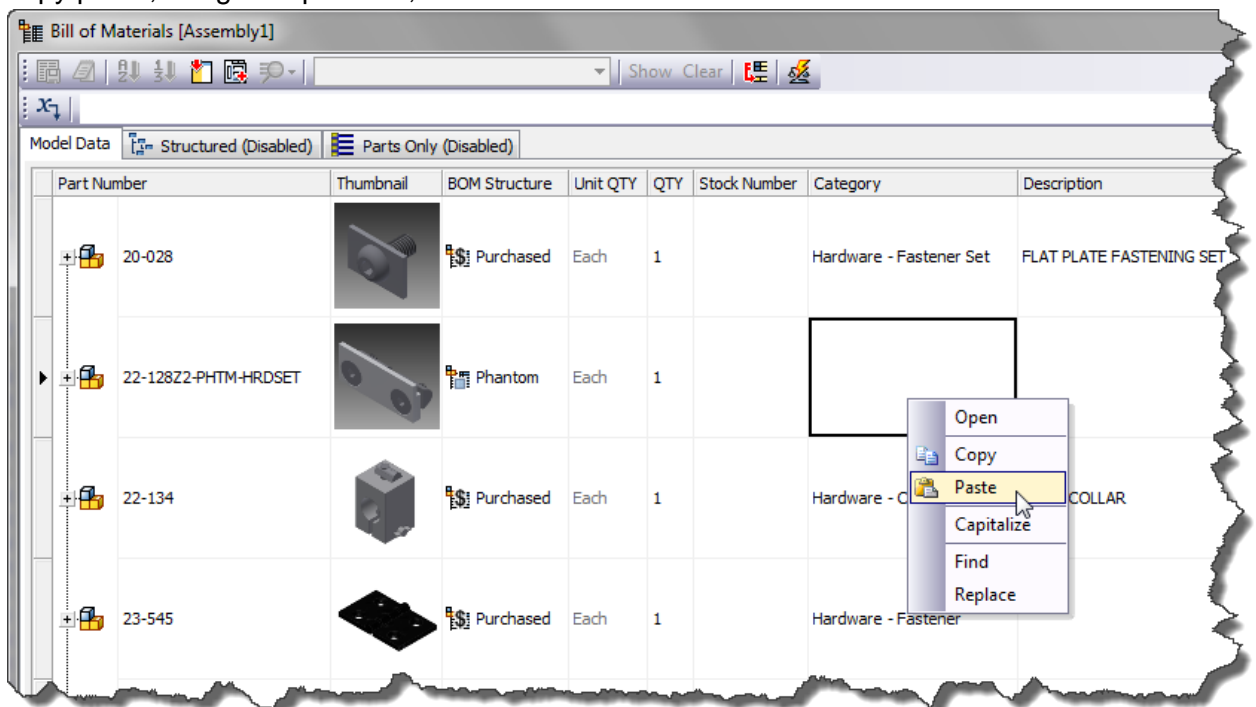
Tagging iParts & iAssemblies

To tag iParts or iAssemblies, it must be added in the Factory and included in the member files.

1. Open the iPart or iAssembly Factory file.

2. Edit the iPart or iAssembly table
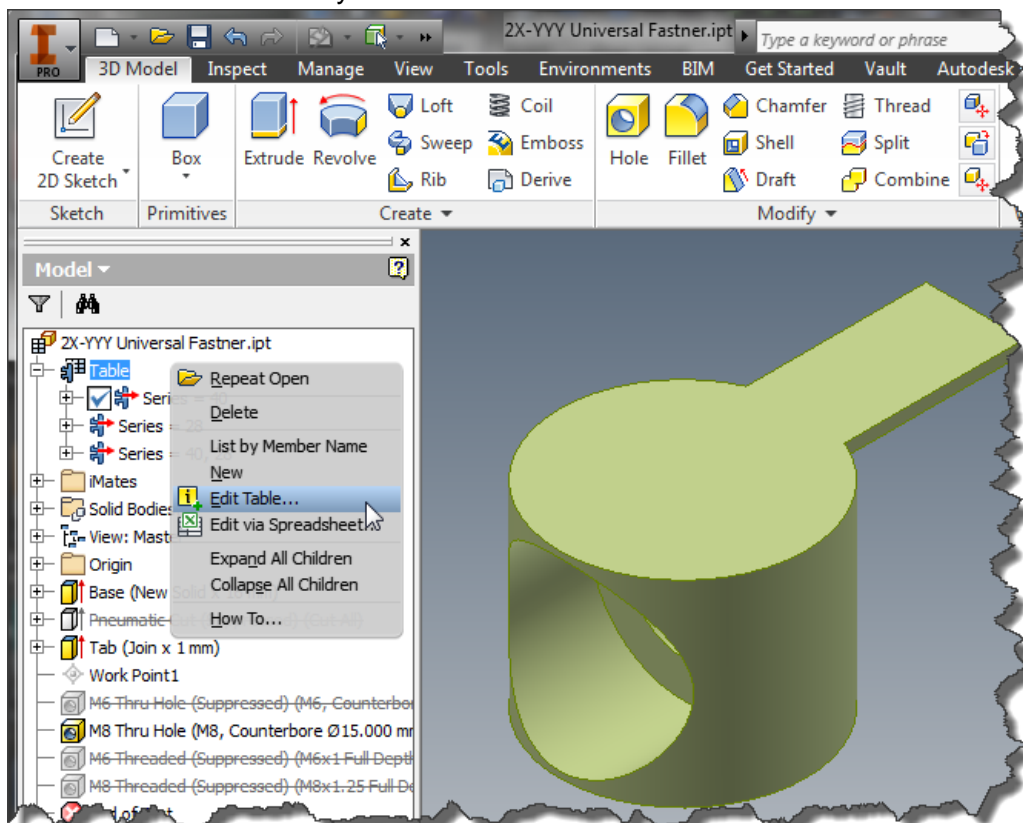
3. On the Properties tab, add the Category iProperty (or another built-in iProperty of your choice) to the iPart/iAssembly table. Fill in the values for each member that you want them to have.
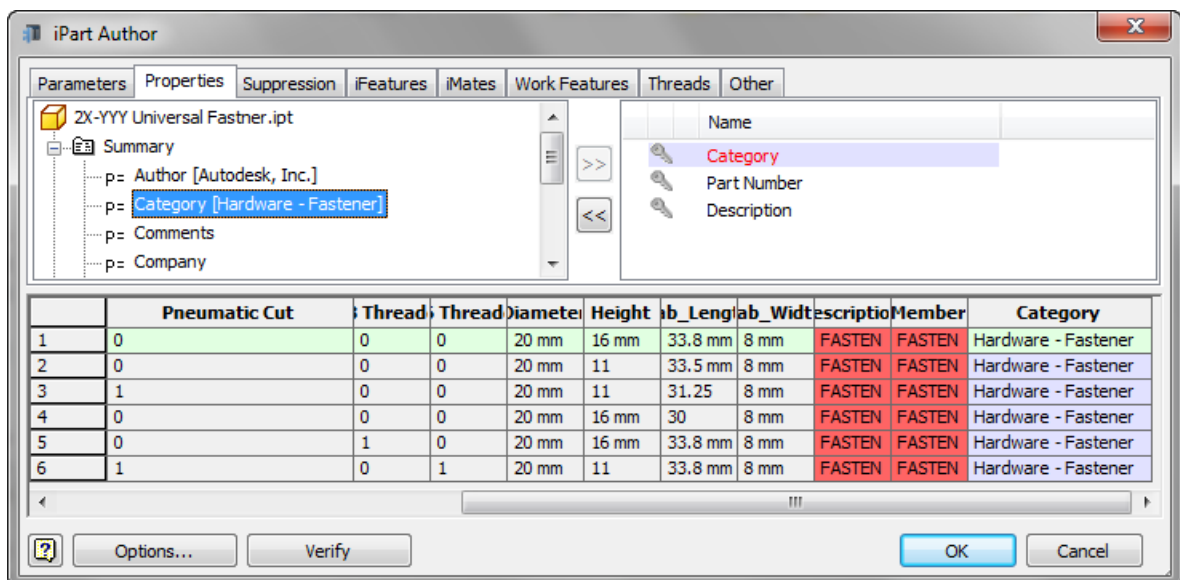
Content Center Files

While Content Center files can be easily suppressed using the All Content Center Suppressed LOD, there may be other times when you want to be more selective. You can tag content center components in a similar manner.

> Note: Editing Content Center families requires the Content Center Editor role if your content center is based on the Autodesk Data Management Server (ADMS).
> You will also need a read/write content center library to copy Content Center families from the Inventor Out of the Box libraries in order to edit them. This example assumes the need.

1. Start the Content Center Editor (log into Vault if necessary)
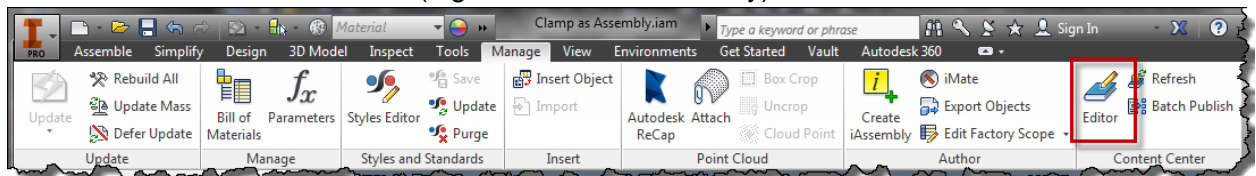
2. Change the Library View to the library you want to copy the family from.
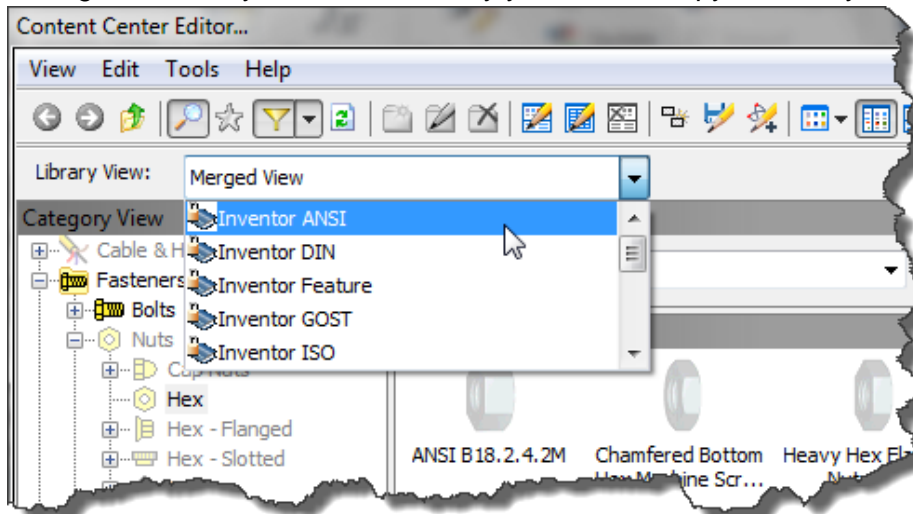
3. Select the families you wish to copy, right click, and select Copy To>{Read/Write Library} and select the library you want to copy the families to for editing
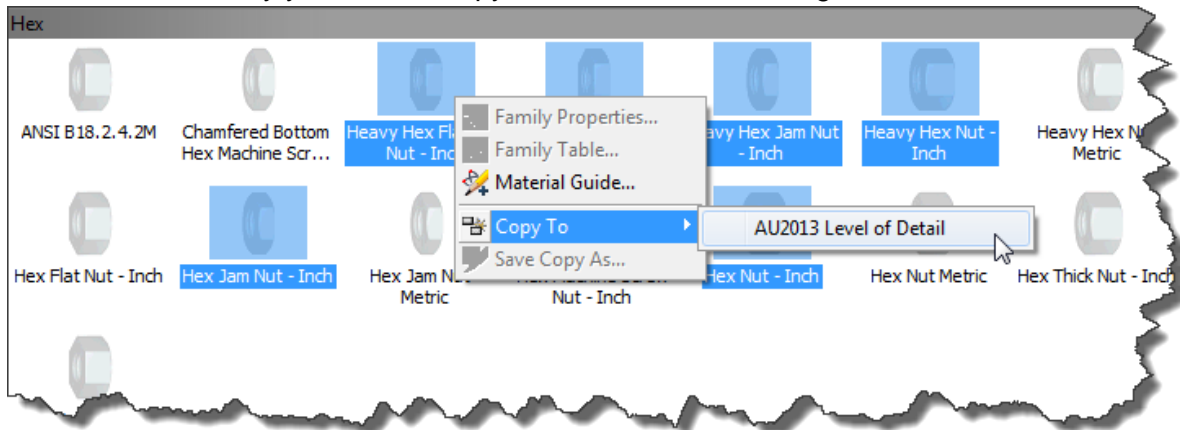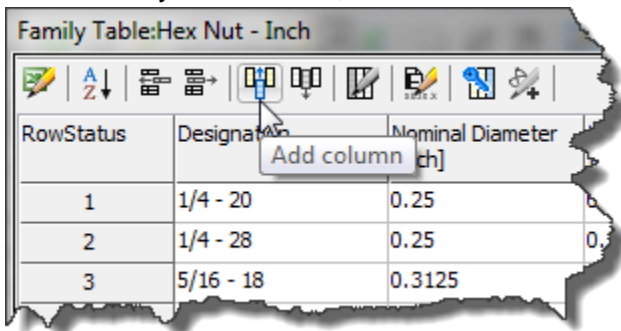

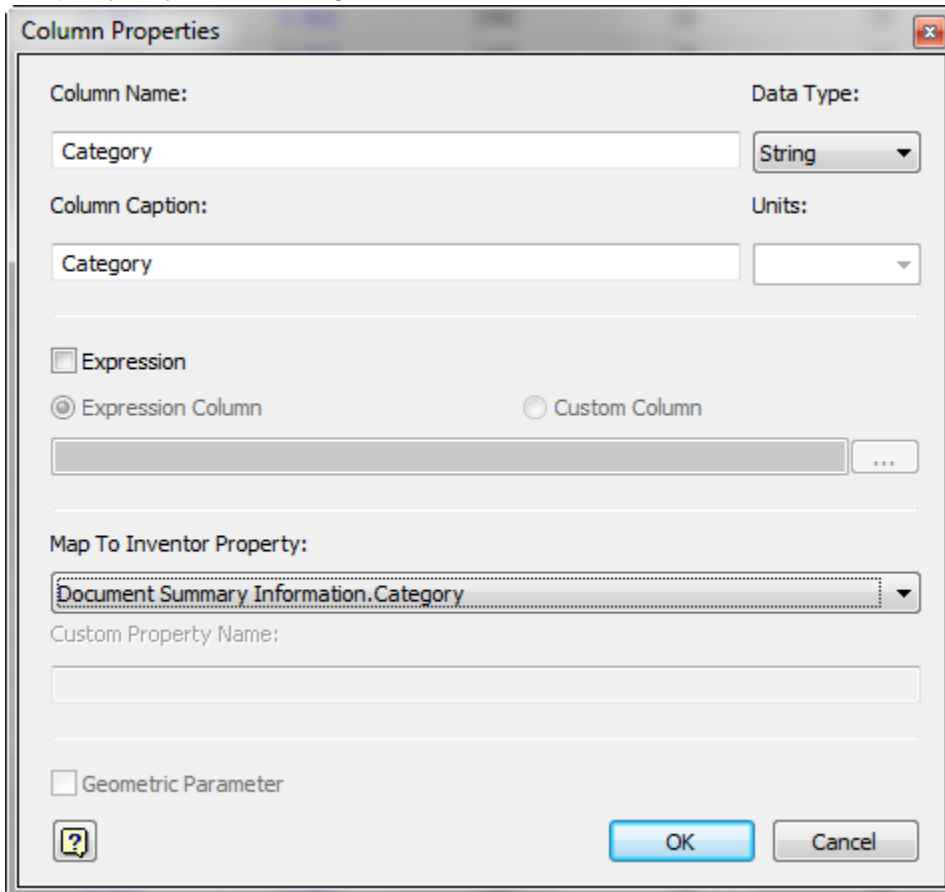4. Change the Library View to your Read/Write library
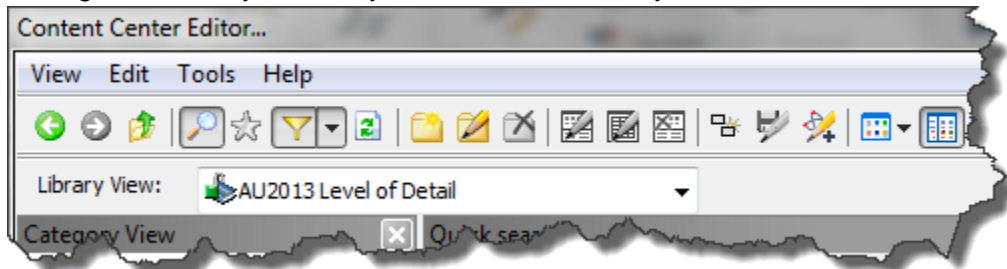
5. Select a family, right click and select Family Table…
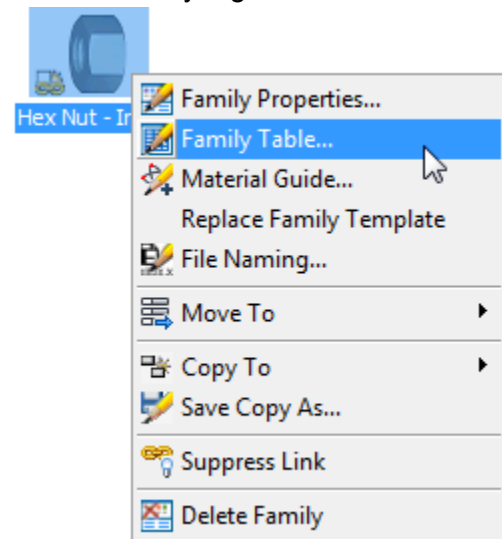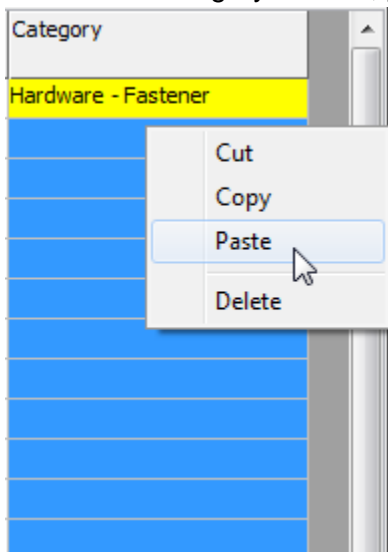
6. In the Family Table editor, select Add Column

RowStatus	Designation	Nominal Diameter [Inch]	
1	1/4 - 20	0.25	
2	1/4 - 28	0.25	0.
3	5/16 - 18	0.3125	

7. In the Column Properties dialog, setup the Category property (or any other built-in iProperty or your choosing) as shown below. Select OK to add the Column.

Column Properties

Column Name: Data Type:

Category   String

Column Caption: Units:

Category

☐ Expression

◉ Expression Column   ○ Custom Column

Map To Inventor Property:

Document Summary Information.Category

Custom Property Name:

☐ Geometric Parameter

[?]   OK   Cancel

8. In the new Category column, you can edit the values as desired.

9. Select OK to commit the edits to the Family.
10. If you have already used the content in Assemblies, you can update them to have the new tags using the Refresh Standard Components command. Simply open an assembly with the component(s) and run the command. You can also use Task Scheduler to update multiple assemblies as a batch task
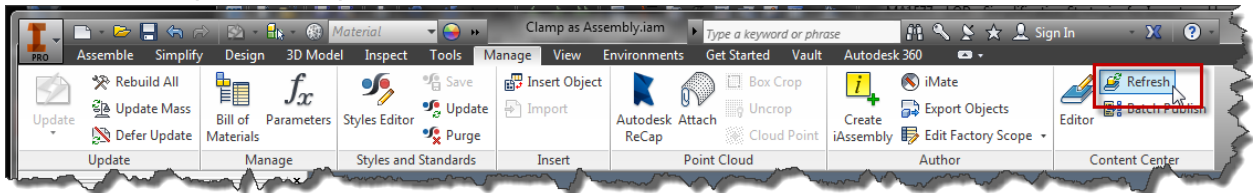
Creating & Using Searches

You can create and reuse searches for your tags as outlined below.

1. Create an Assembly or open an Assembly with components that are tagged (so you can test the searches).

2. Change the selection priority from Select Component Priority to Select Part Priority
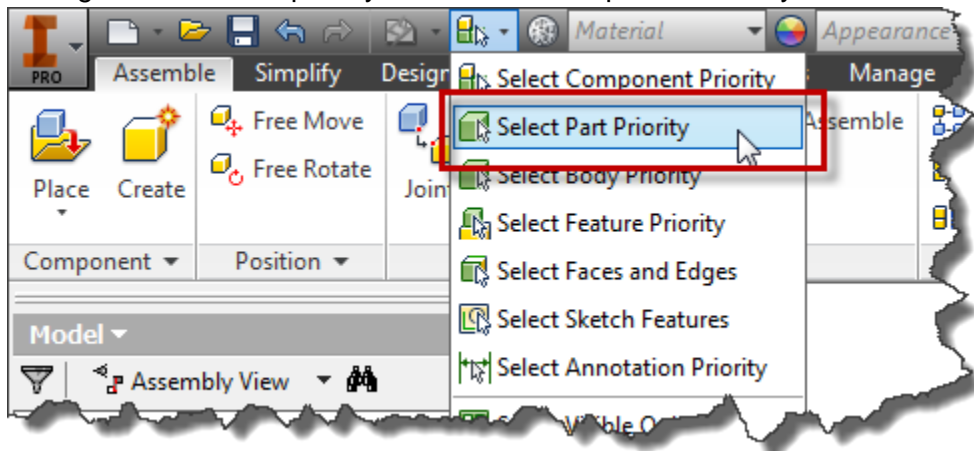
Note: Changing the select priority is important to insure that the Find Components command is able to select parts that are child components of sub-assemblies.

3. Select the Find Components command from the Tools tab or in the Model Browser.
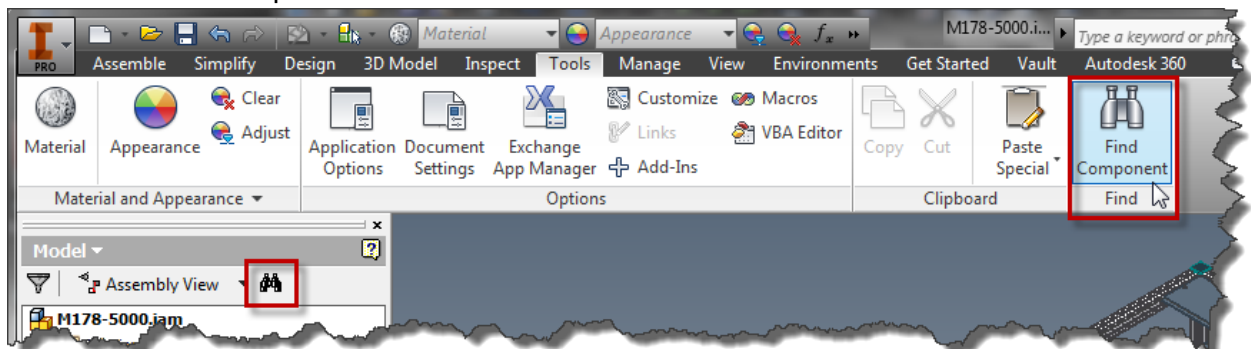
4. In the Find Components dialog, create a new line for the Category iProperty (or your selected built-in iProperty), set the Condition to includes, and set the Value. Select Add to List.
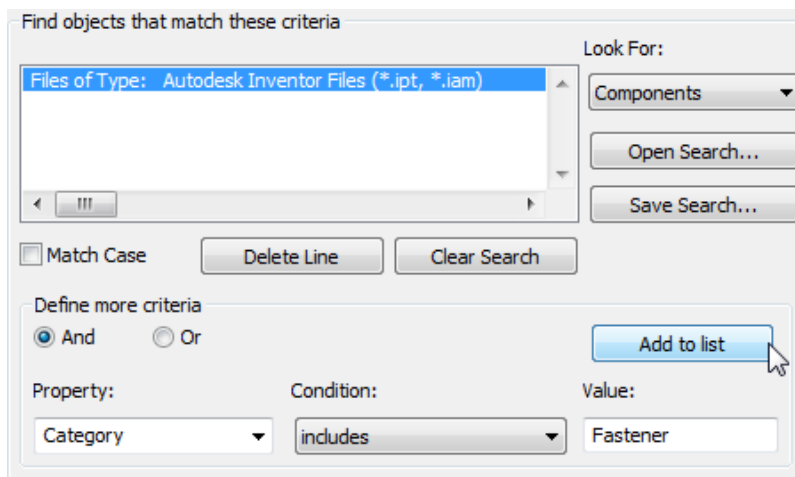
5. Select Save Search… and give the search a meaningful name. Select OK to save.
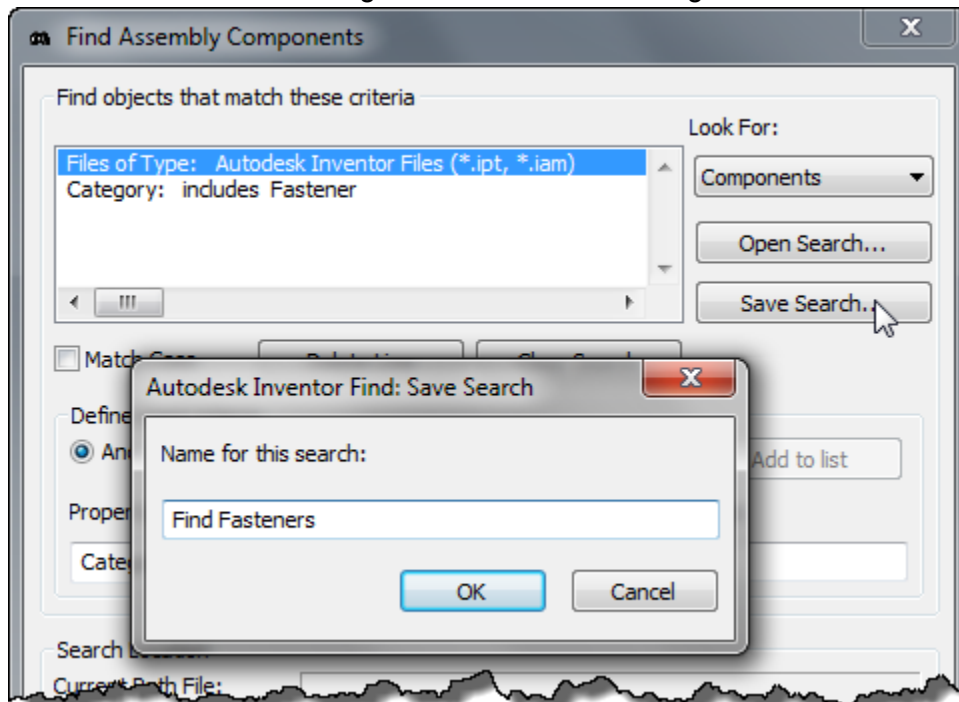
6. You can now select the Find Now button to find those tagged components and select them.
7. Use Open Search… to select a saved search to reuse.

Creating an Easy Button for Substitutes

This section walks through creating construction surfaces in iParts and Content Center family parts to make simplification easier.

Note: This model in this section is the 40-Series Profile.ipt part file in the example models zip file. This is a "custom" iPart, meaning that each placement creates an editable member part. For the Content Center example, the 40-Series Profile.ipt file was published as Frame Generator member without the surface first to setup a similar example to follow.

Build the Easy Button: General Setup
1. Open the file you want to add the construction surfaces to.

2. Create a sketch profile that will create the overall shape you want. Make sure it is parametric so it will adjust as you expect.
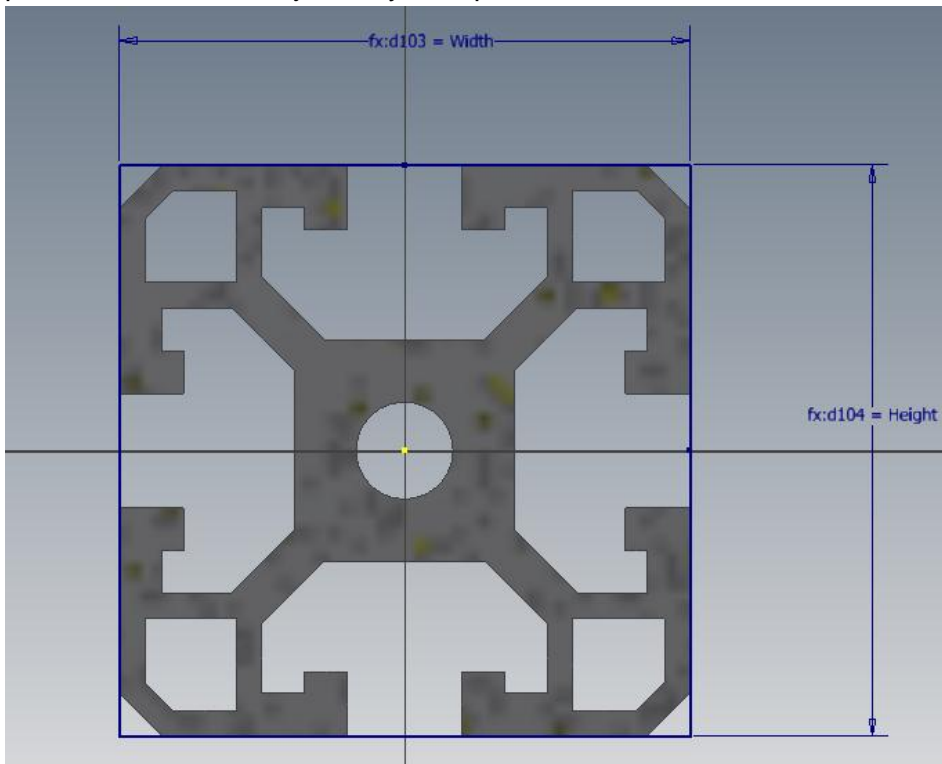
3. Create an extruded surface that encompasses the part. The surface will be "uncapped" at the ends. Just like the sketch, make sure it is parametric so it adjusts and encompasses the part
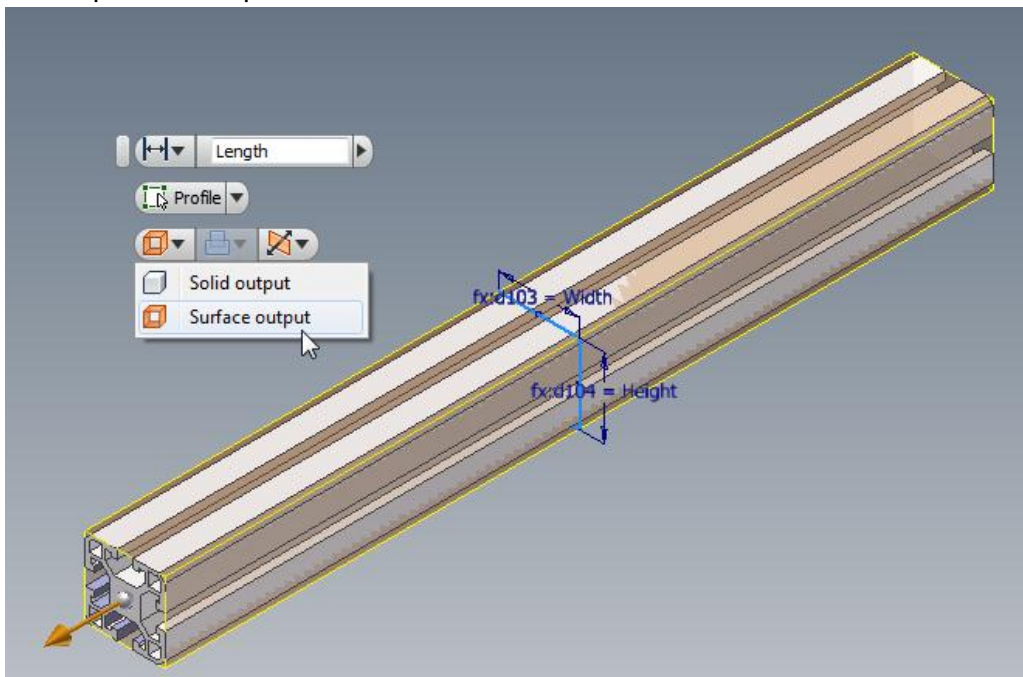
4. Using the Boundary Patch command, create caps on the ends of the surface so that it can be completely "water tight".
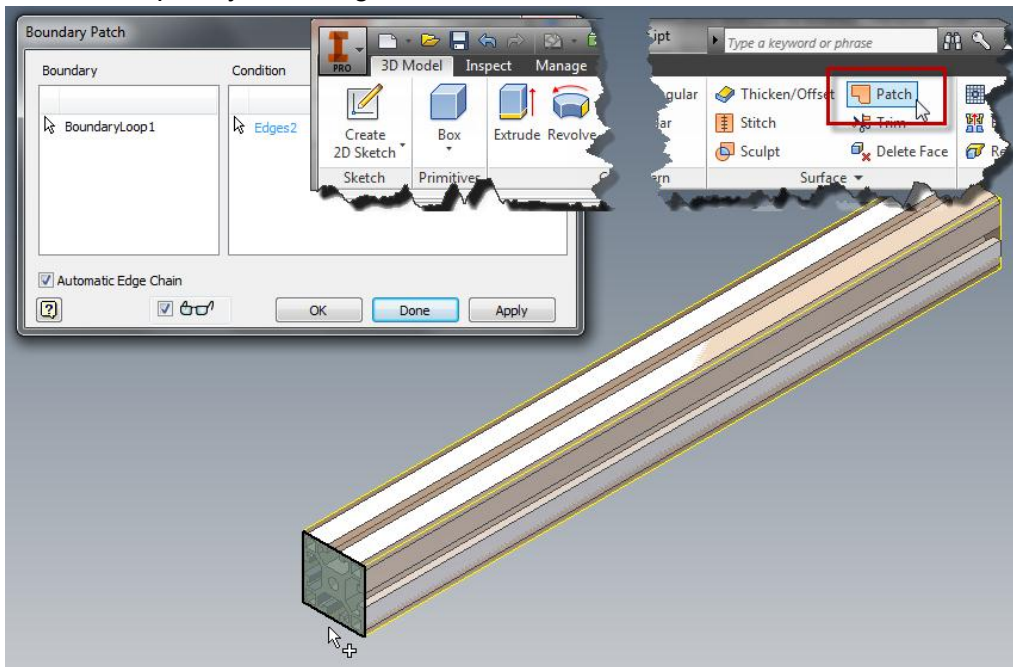
5. Now use the Stitch command to stitch the surfaces together to create a closed, "water tight" surface. Be sure to enable the Maintain as Surface option in the Stitch dialog to make sure only a surface is created, otherwise the stitch will create a solid.
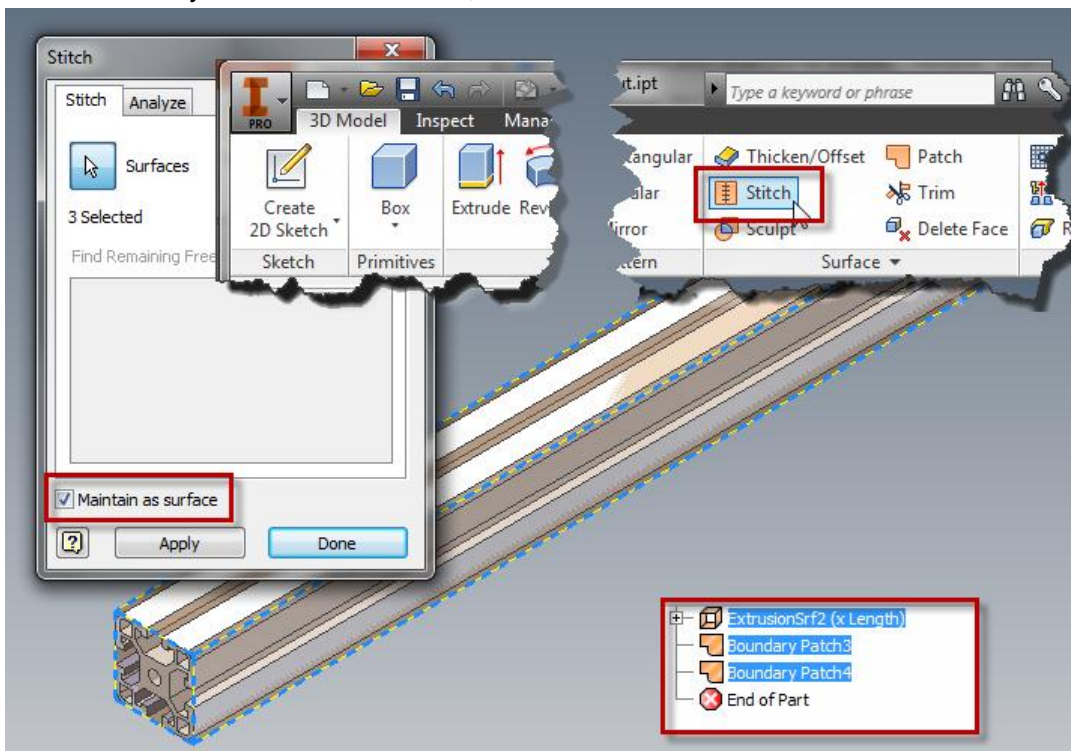
6. When finished create a Part View Rep that turns the construction surface(s) visibility on and one that turns it off to ease the use of the part afterwards. This can make them easier to use with the Derive functionality.
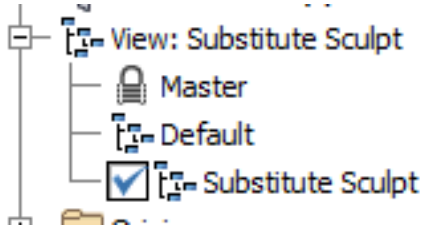
Building the Easy Button: iPart Specific Steps

iPart Member files do not directly support construction surfaces. You cannot include them in the member row. However you can get them into the member files indirectly using iMates.

1. Once the construction surface creation is finished (see Build the Easy Button: General Setup), start the iMate command. Create an iMate and make sure it is created on a face of the Construction Surface. The type of iMate does not matter.
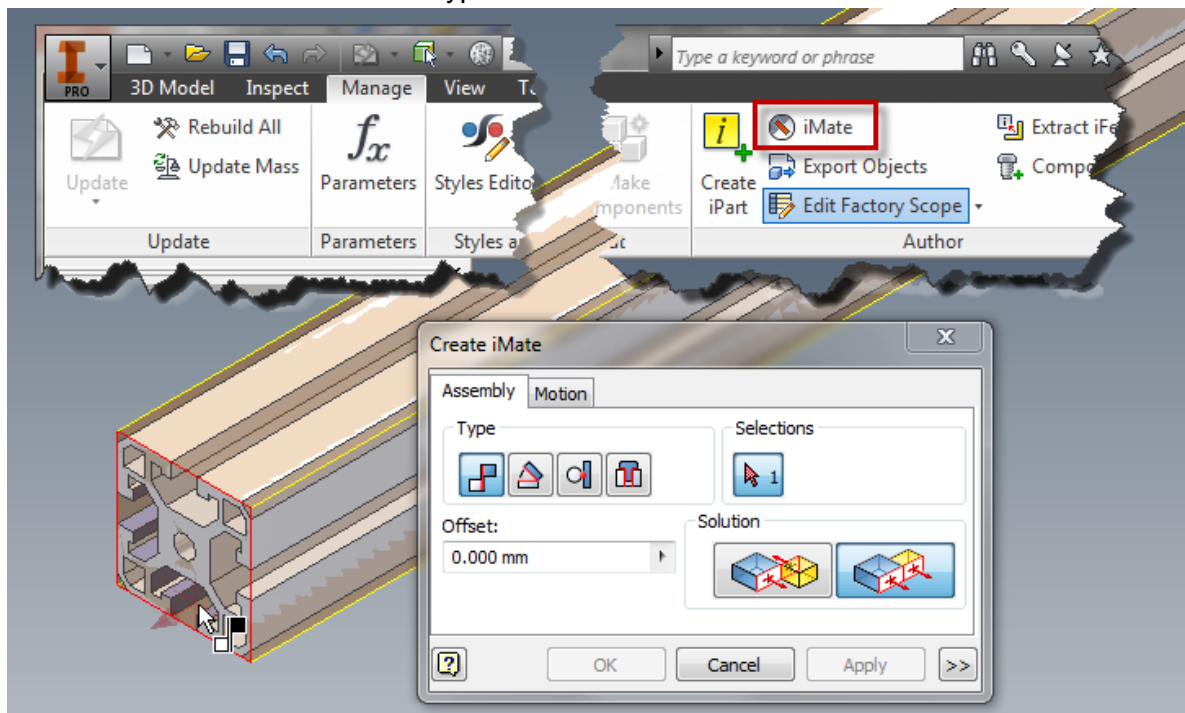
2. Edit the iPart Table. Add the iMate to the table.
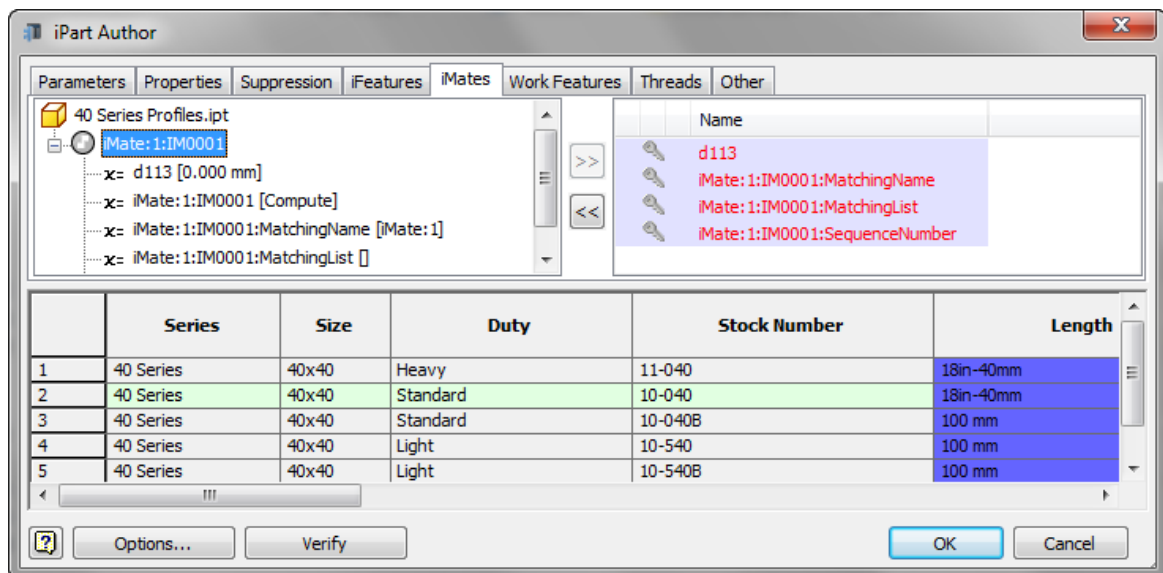
3. Once added save the iPart Factory. If members have already been placed from the factory, the members will have an updated pending the next time they are opened. When you update them the surfaces should be added to the member.

Note: Since iParts do not officially support surfaces, there are times when this method may not work. There is no guarantee that it will work in future versions of Inventor.

iPart members do not support View Reps from the Factory file, so adding them is not necessary.

Building the Easy Button: Content Center Families

You can add construction geometry to Content Center families by editing the family template. Instructions for this are outlined below.

Setup Family to Edit

If you are adding construction surfaces to an Out of the Box content center family, be sure to add it to a Read/Write library as described in Tag-It Examples: Content Center Files above.

"Extract" a Template to edit

1. Create a new assembly file
2. Select Place from Content Center.

3. Select the family you want to add and select OK from the Place from Content Center dialog. Be sure that Autodrop is turned off.
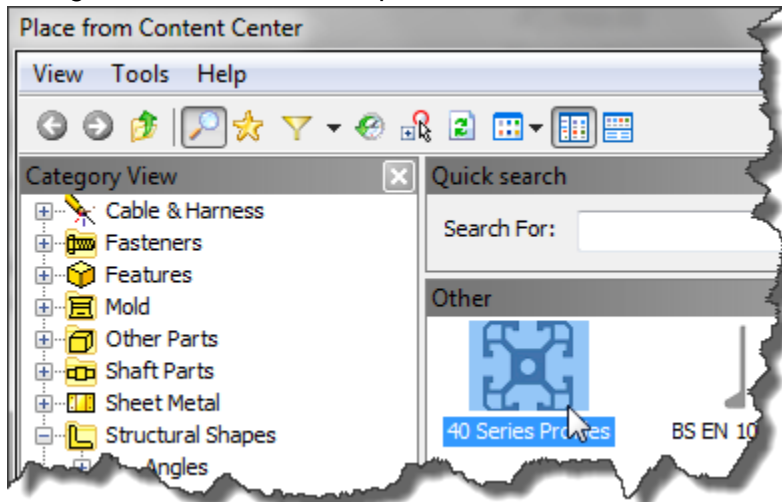
4. In the placement dialog, select a member to place and be sure that As Custom is selected to place the member so it can be edited. Select OK.
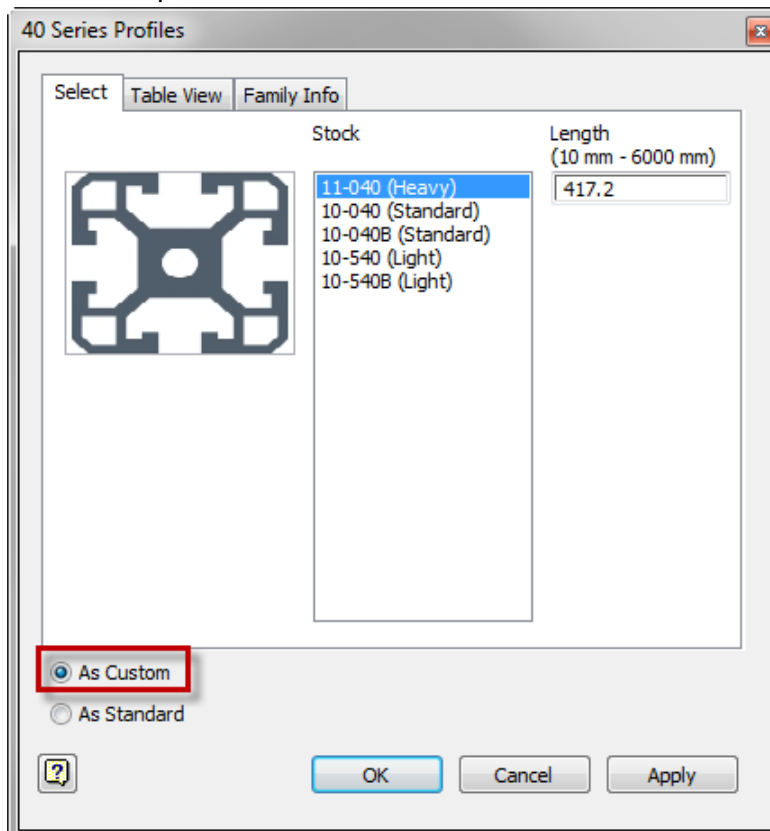
5. Give the file a meaningful name in the Save As dialog and select OK
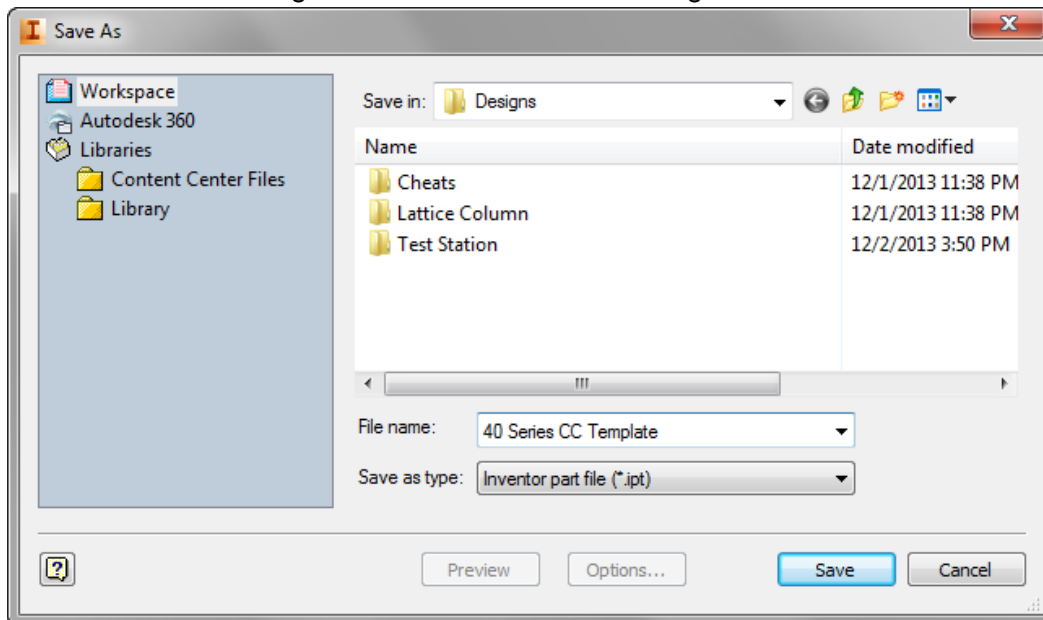
6. Place the new file in the assembly, right click, and select OK. You can now open the "template" file to edit it and add the construction surfaces as outlined in Build the Easy Button: General Setup.

Note: It is recommended that you create the View Reps described in the Build the Easy Button: General Setup and set the one that hides the surfaces as the active View Rep.

Update Content Center File Template
Once the template created above has been edited, use the following steps to update the template in the Content Center family in Content Center.

1. Start the Content Center Editor (Log into Vault if necessary)
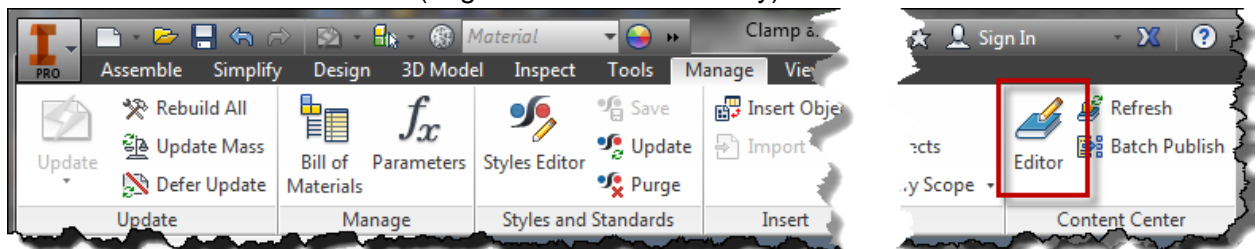
2. Switch the Library View to the Read/Write library that contains the family to update. Select that family
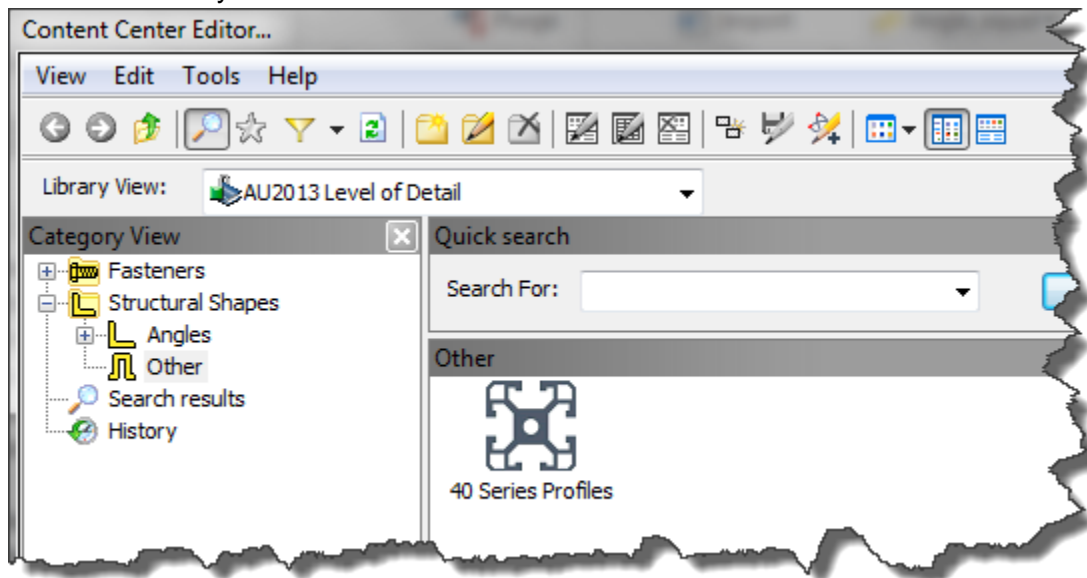
3. Right click on the family and select Replace Family Template
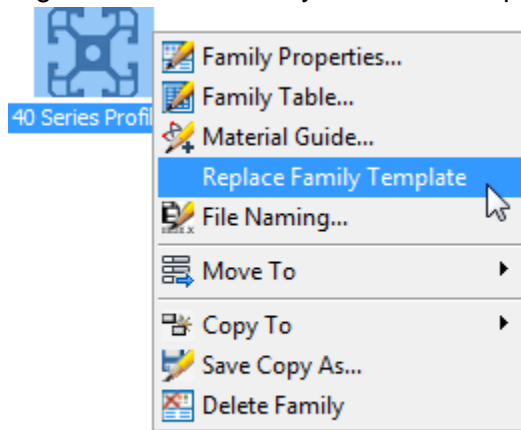
4. Select the modified template file in the Open dialog and select OK
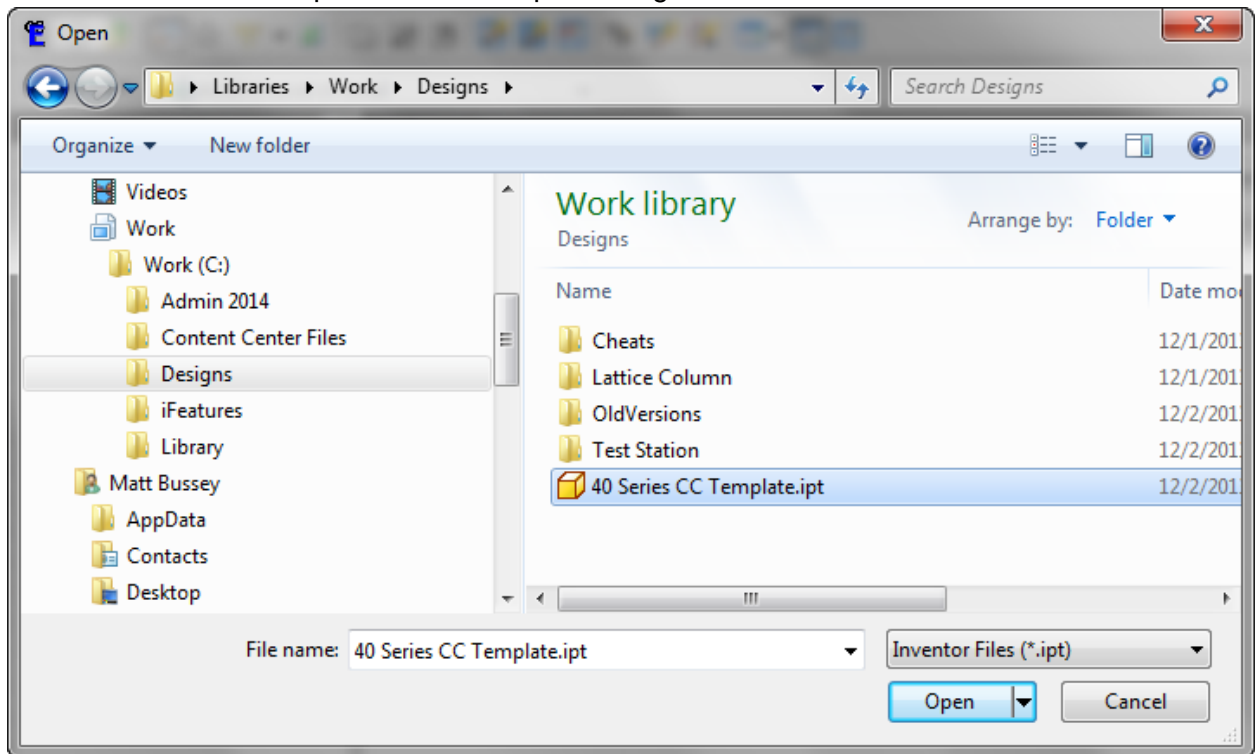
5. Select OK from the Publish Success dialog. The family should now be updated. Be sure to test it thoroughly to ensure it updates properly. The surfaces should now be available to use.

> Note: If you add construction surfaces to Frame Generator families or other families that typically create custom members, the new surfaces will only be available in newly created components from the family, not in ones that were created before the update.

Using the Construction Surfaces in a Substitute

> Note: this example uses the M178-2000.iam assembly found in the example files.

Once you have created the construction surfaces in your library and Content Center parts, you can apply them when creating Substitute Levels of Detail in an assembly.

1. Open an assembly with components containing the components with Construction Surfaces, or create one with components that have construction surfaces.
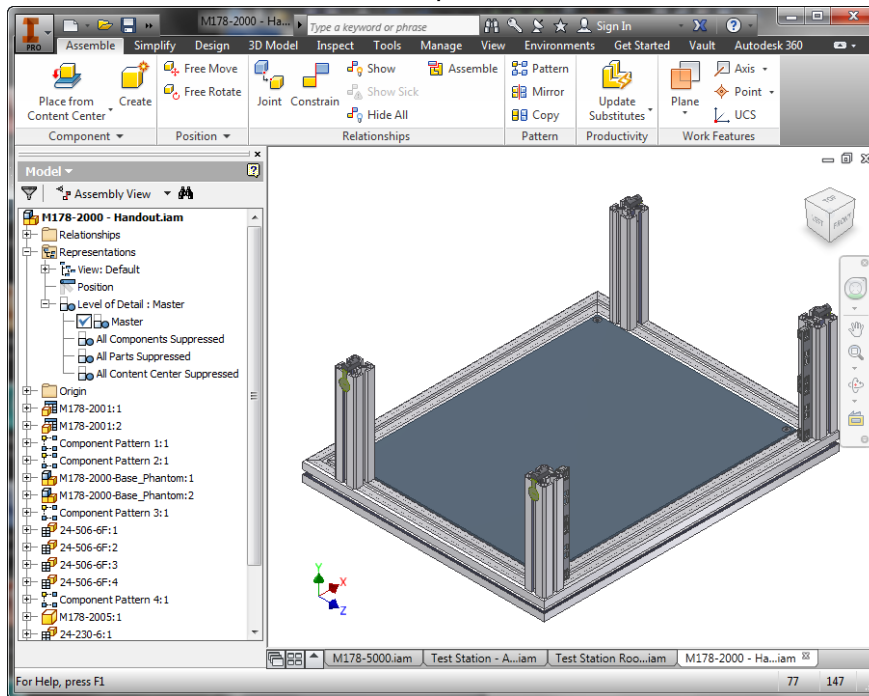
2. Create a View Rep that makes the construction surfaces you want to use visible and give in a meaningful name, such as "Substitute Sculpt".
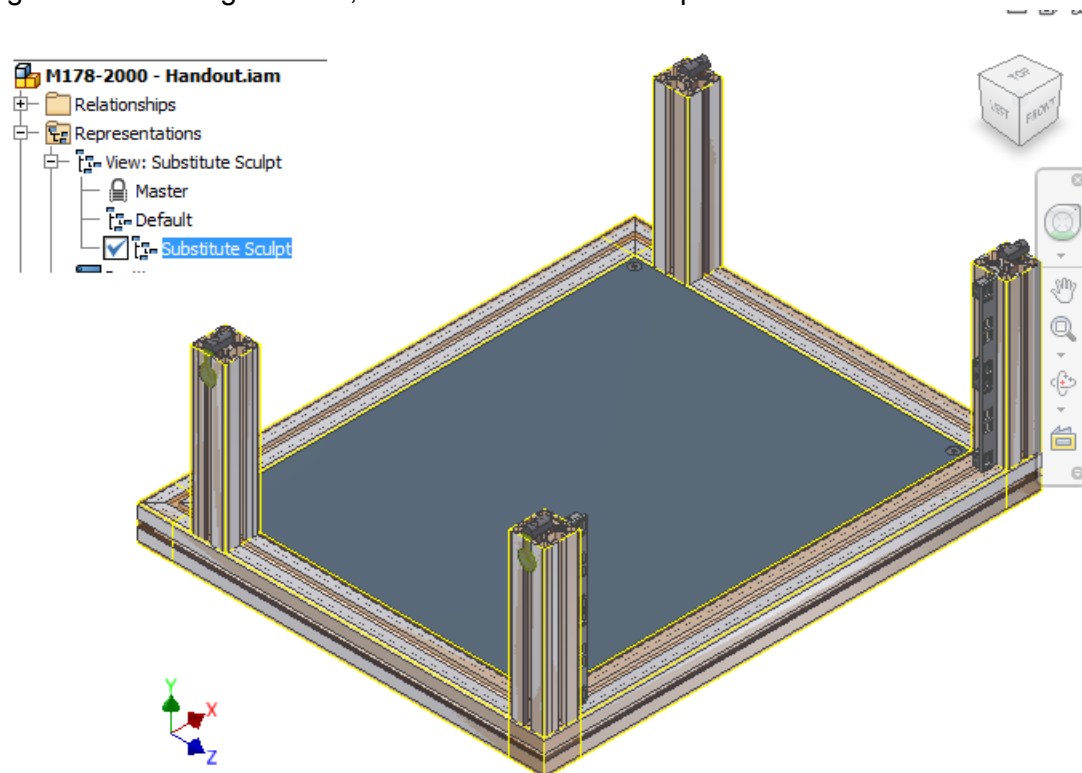
3. (Optional) Activate the All Content Center Suppressed LOD to remove all the content center standard parts from the assembly.
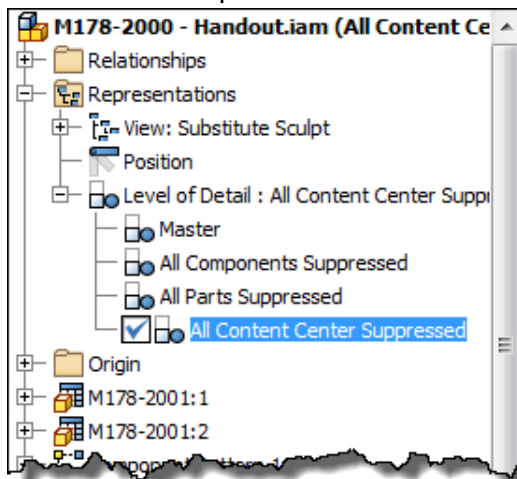
4. (Optional) Create a new Standard Level of Detail by right clicking on the Level of Detail browser node. Name it No Hardware. Use the methods in Tag-It Examples to find and suppress any hardware you do not want to carry over into the substitute. Once done save the file.
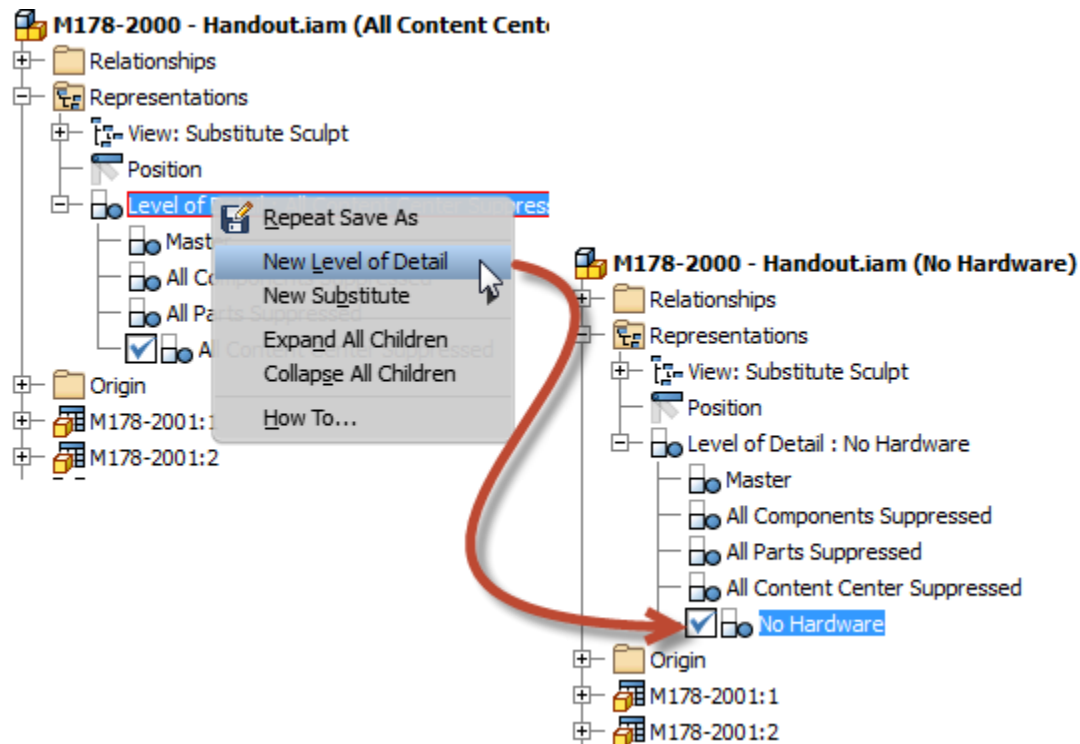
5. With the No Hardware or another Standard Level of Detail active, and the Substitute Sculpt View Representation active (to make sure the construction surfaces are visible, right click on the Level of Detail browser node and create a New Substitute > Derive Assembly.
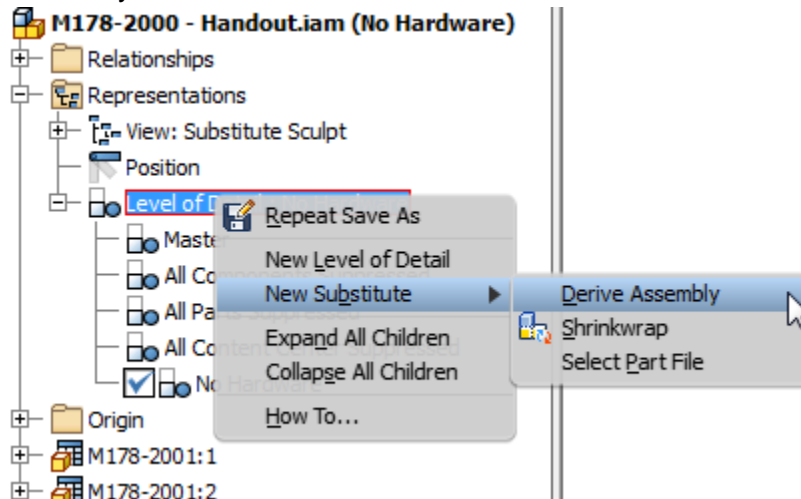
6. In the New Derived Substitute Part dialog, give the new substitute part a meaningful name and location and select OK.
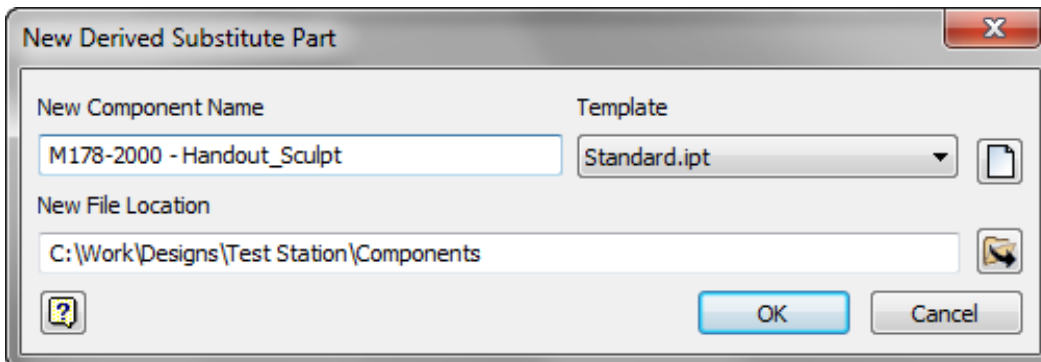
7. In the Derive Assembly dialog, make sure the Derive Style is set to Solid on the Bodies tab. On the Representations tab make sure the View Representation is set to the one that make the construction surfaces visible, and that the Level of Detail is set to the one you wish (like No Hardware). On the Options tab turn on Hole patching and any other "Shrinkwrap" options you wish. Select OK.
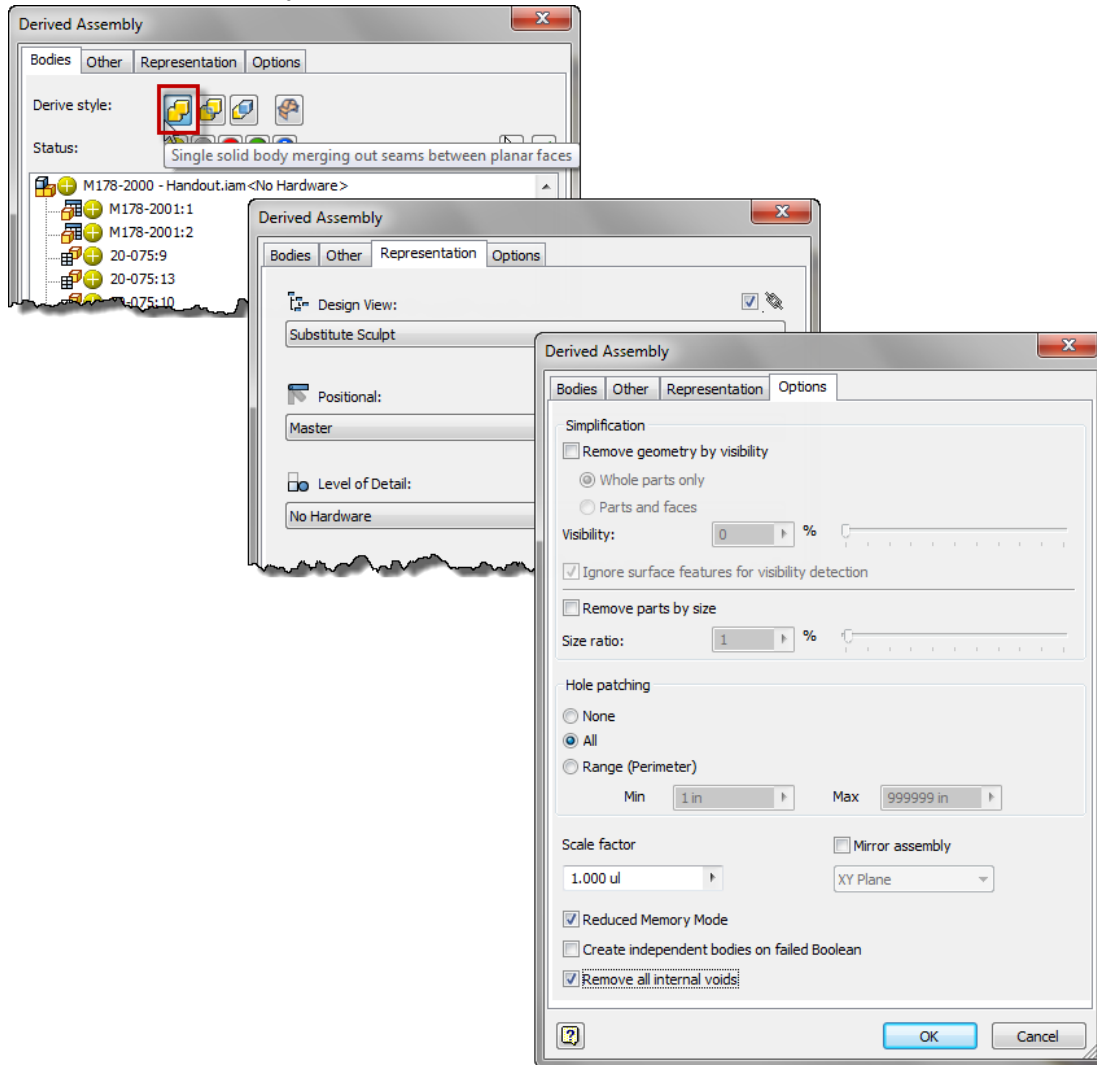
8. Save the assembly once the substitute is created and active. Then right click on the Substitute part and select Edit.
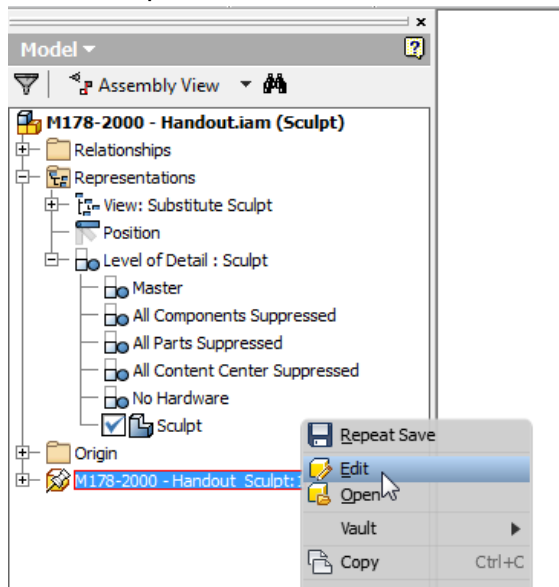
9. Start the Sculpt command. In the browser, multi-select the surfaces derived from the assembly, then select OK in the Sculpt dialog.
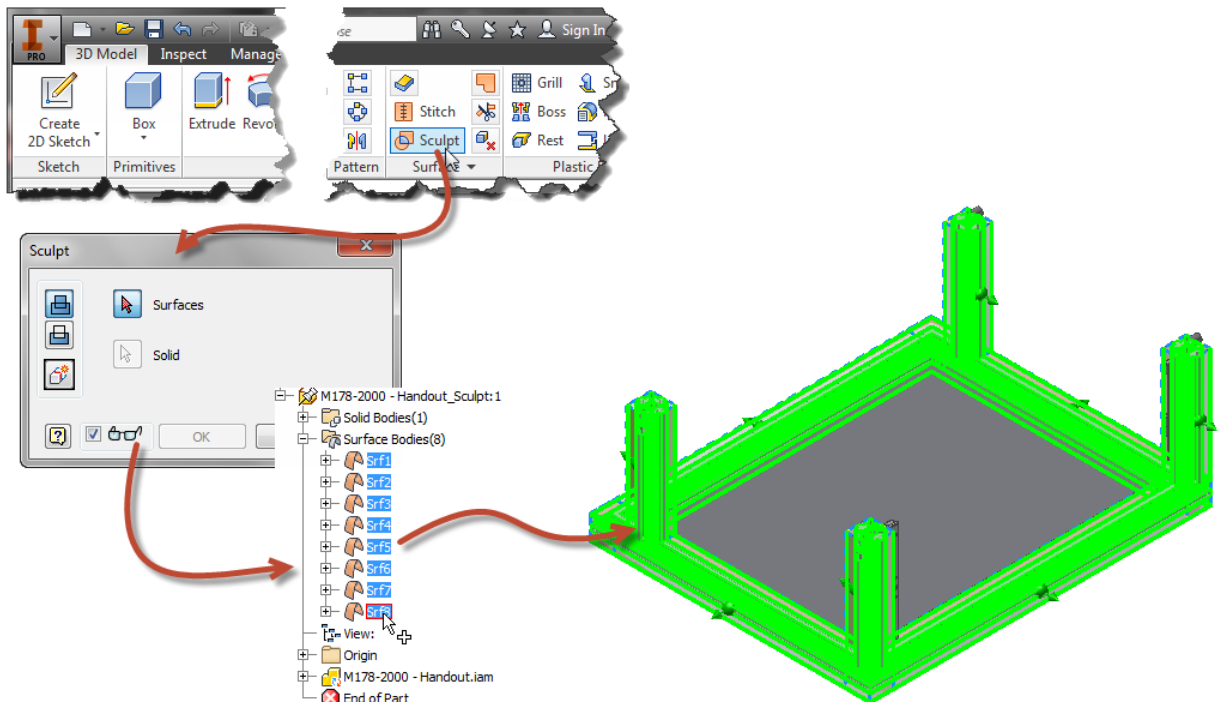
10. Save the file, return to the assembly and save the assembly.
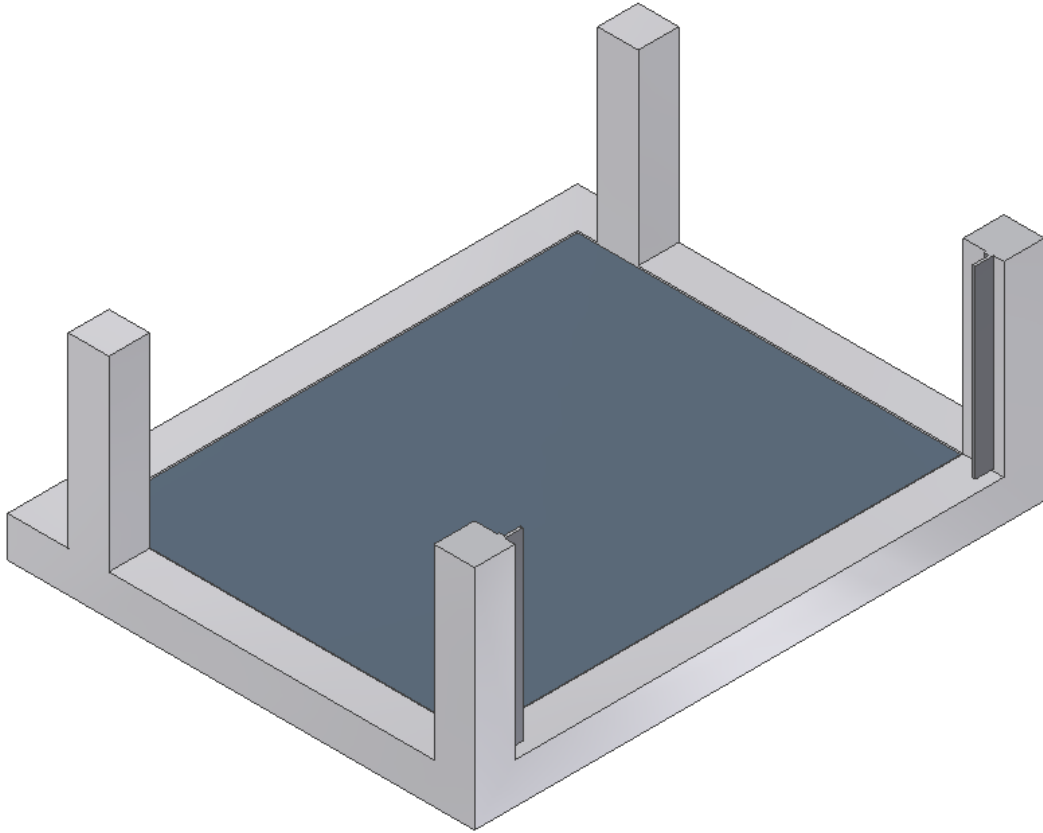
"Cheating" Examples

This section shows some techniques to keep in mind that allow you to simplify models in different ways.

Appearances

Designers of video games and special effects have used texture mapping to create realistic visuals for a long time. You can use appearances in a similar manner to create components that are realistic but simple. For example: Fencing, perforated flooring for mezzanines, etc. Inventor comes with appearances that already have cutouts, a bitmap that sets transparent and opaque areas on a surface where the appearance is applied. For example, the Fence Fabric in the Inventor Material Library can be used to do simple chain link fences.
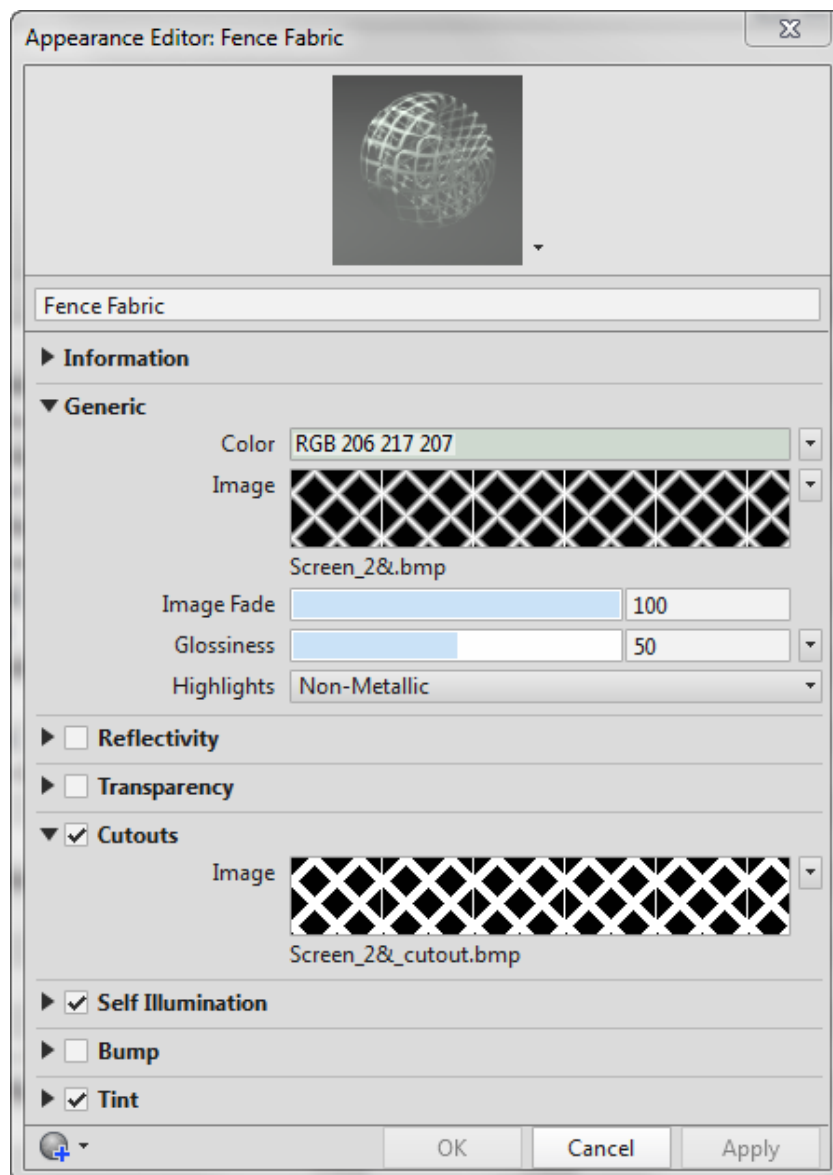
If you wish to create your own patterns, you can do so by creating bitmaps to use as a texture and a cutout (one bitmap each). Be sure to create them the same size (i.e. make both 49x49 pixels, for example) and layout the patterns similarly. For the Cutout bitmap, it should be only Black and White, where the Black pixels are the transparent areas and the white pixels are the opaque areas. Example textures and bitmaps can be found in "C:\Users\Public\Documents\Autodesk\Inventor 2014\Textures\surfaces".

Note: To get a realistic mass, you can edit the iProperties of the part or assembly and input a density or mass directly. This may not always give you want you want in terms of Center of Gravity so be aware of this.

Appearances with transparency do not translate into drawing views with "hidden lines". They will only look "correct" in rendered drawing views.

Make an Assembly from a Library Part

Making a purchased item as a single part file is quite common, and there is nothing wrong with that approach. However it does not always lend itself to easily creating simplifications/levels of detail.
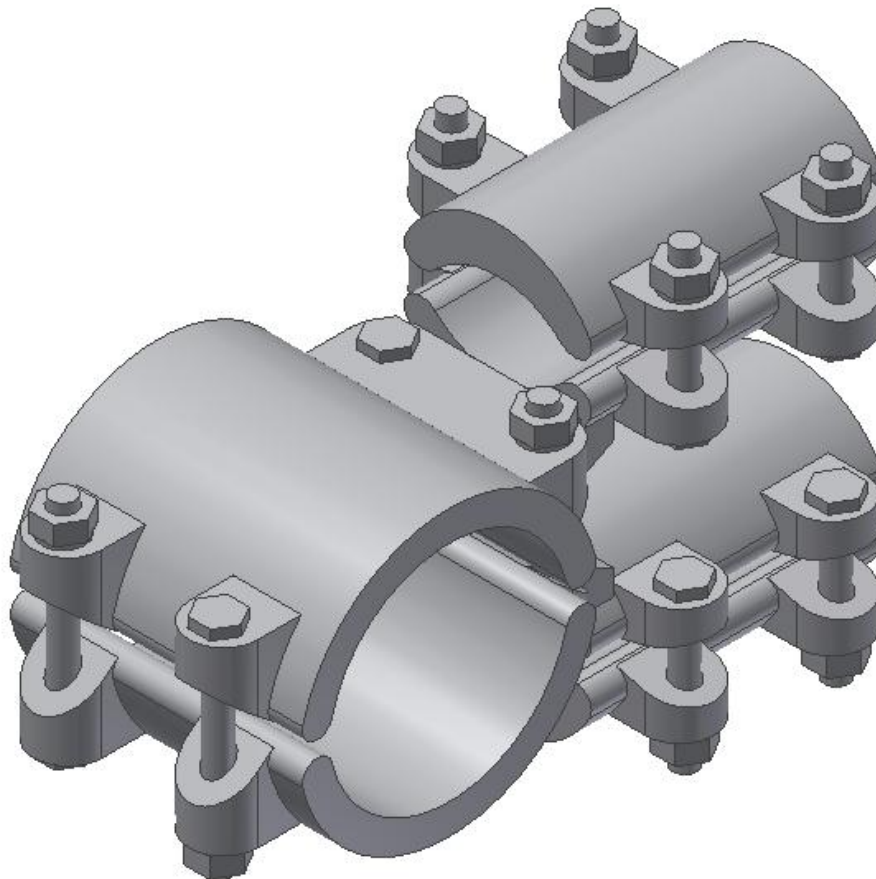
In this example a clamp is modeled as a single part. However you would like to modify it and make an assembly that can have LOD's that will work with other library and reusable components. How hard it is to remodel will depend on how it was modeled to begin with, but the first approach to take before remodeling from scratch is to make the part multi-body. The plan is to convert it to a multi-body part, derive it into an assembly with a part derived from each body, then setup LOD's in the assembly. The LOD's created for this example model are outlined in the next section: Creating Reusable LOD's.

Making the Example Part a Multi-Body Part

Note: the example file Clamp as Single-Body.ipt shows the model in its single body state. The example file "Clamp as Multi-Body.ipt"

Usually you will not have a model as clean as the example model, but this one is setup to show you some of the issues could run into trying to convert a single body part into a multi-body part.

Typically you will want to try to reorganize features into groups that represent the "components" you want to have in the new assembly. In the example part this is already done and the features have been named to help match them. In this example the "components" will be:

- Upper Clamp Top
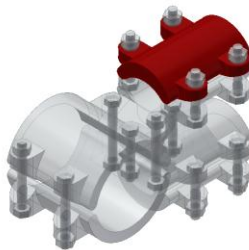
- Lower Clamp Top
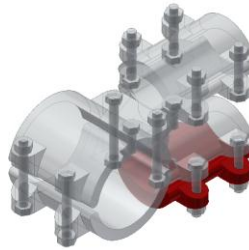
- Bolts
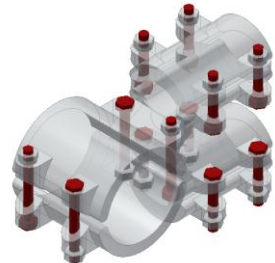
- Upper Clamp Bottom
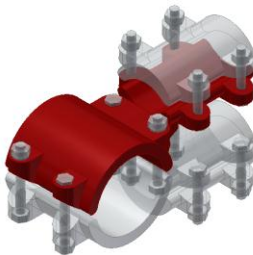
- Lower Clamp Bottom
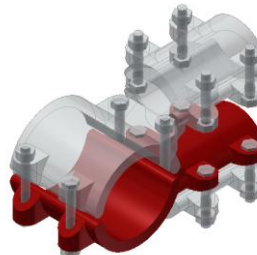
- Washers
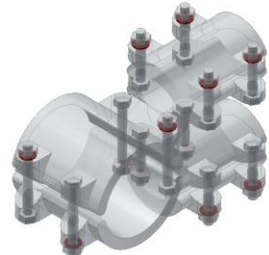
- Nuts
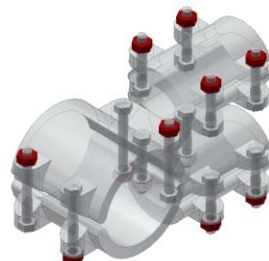
Note that the "hardware" does not need to be separated into discrete components. Since they are not purchased separately but as part of the clamp, we only need to separate them enough to suppress them as groups.

Note: Sometimes it is easier to move the End-of-Part marker up to the first feature and move it own feature by feature during this process. The model is setup so you do not have to do this, but consider it if trying to duplicate this process on your own models.

The process for converting the example model is outlined below:

1. Open the Clamp as Single-Body.ipt file
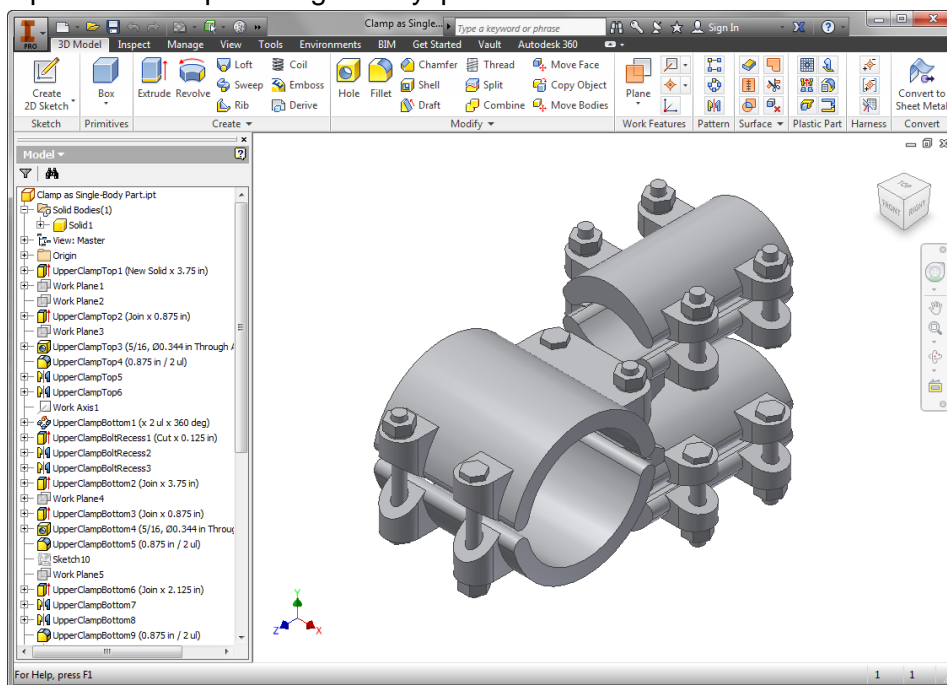
2. The first body is the Upper Clamp Top, which is created by the first few features. Where the next body is going to begin at the UpperClampBottom1 pattern feature.
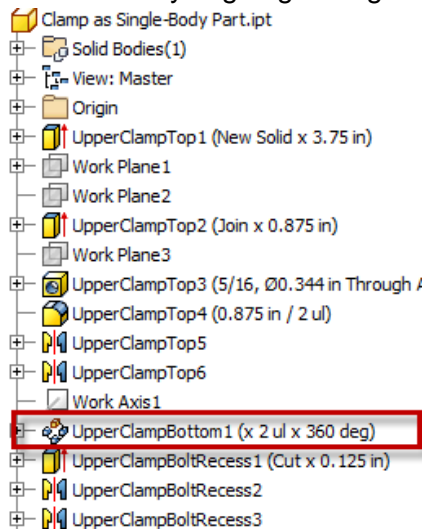
3. Select the UpperClampBottom1 pattern feature and edit the feature. In the dialog select the Create new bodies option and select OK. This will create a new Solid which can be found in the Solid Bodies folder in the model browser.
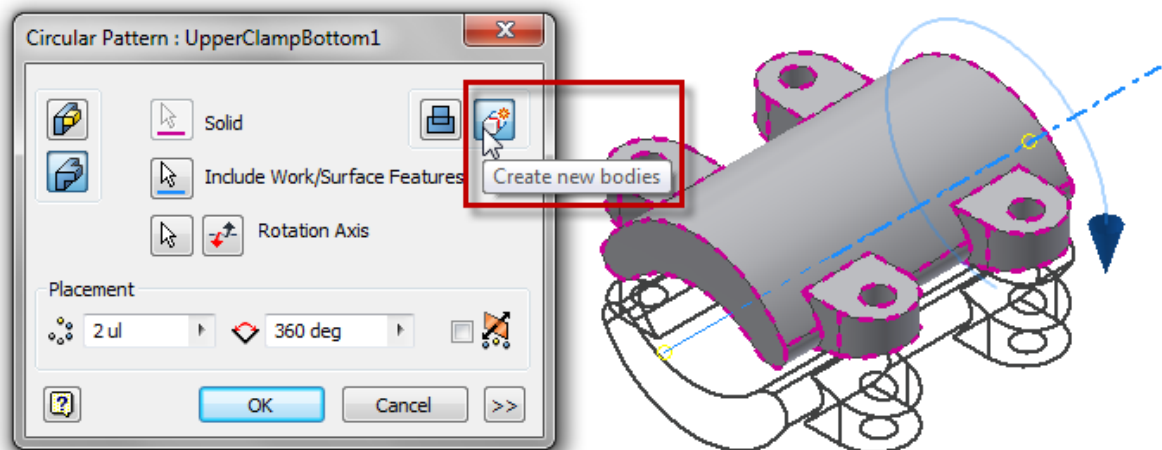
When an error dialog comes up after any of these steps, Accept the error. These errors will be fixed as you move though the steps of the model.
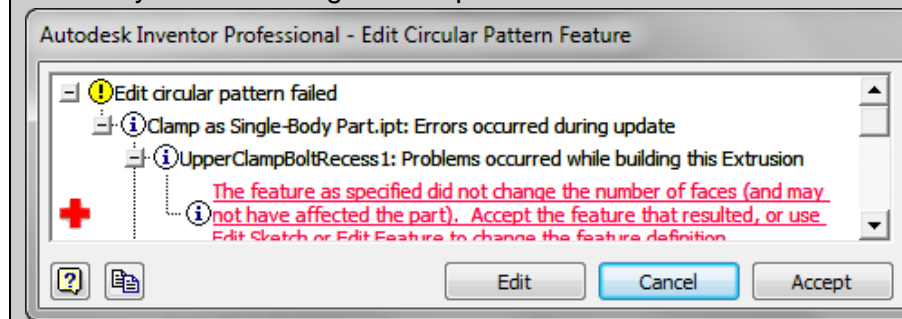
4. Rename the new solid that was create Solid2.
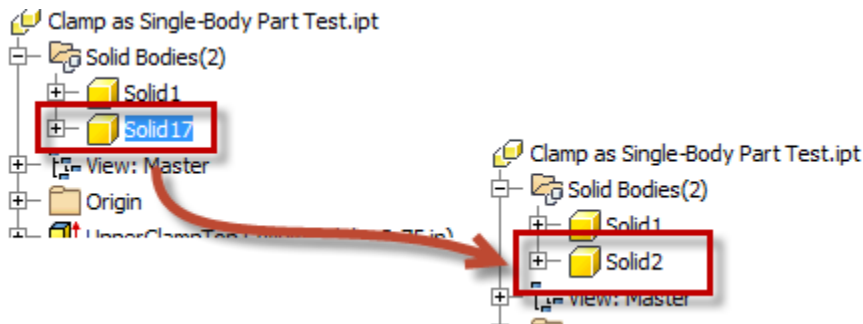
5. The next feature, the UpperClampBoltRecess1 extruded cut, has an issue because it now lies inside of a new Body. You will have to now have it target the new body that was created. Edit UpperClampBoltRecess1, select the Solids button in the dialog, and using the CTRL key unselect the Solid1 body in the browser and select the Solid2 body so only it is selected, then select OK.
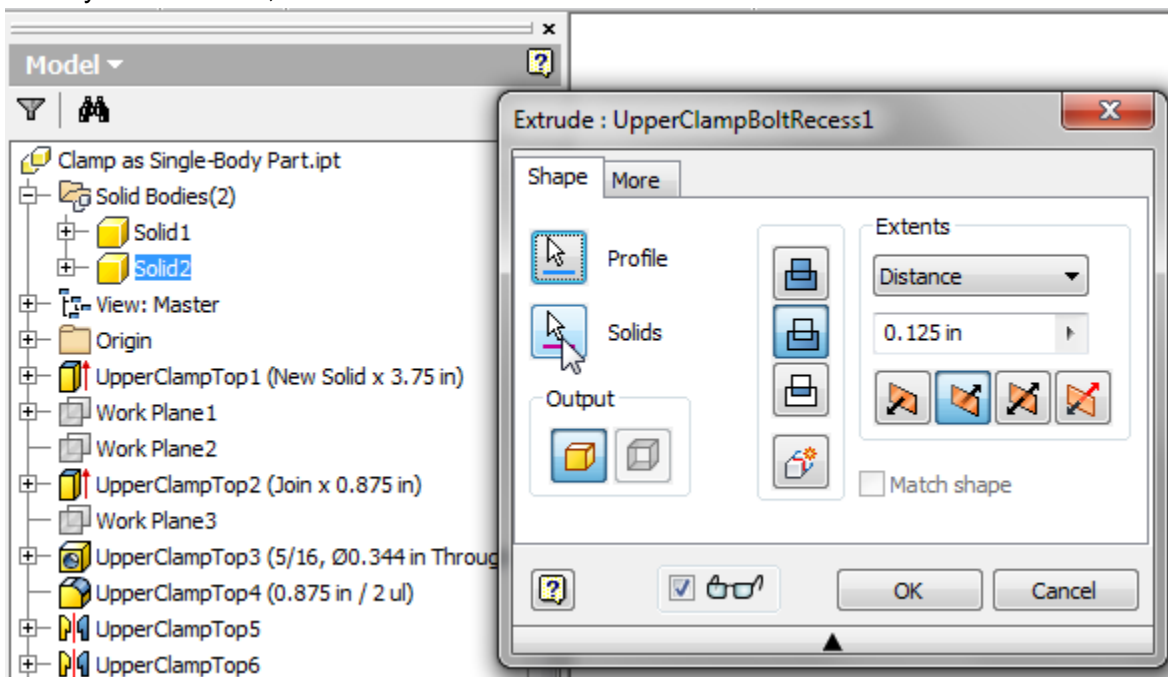
6. Next fix the UpperClampBoltRecess2 mirror feature by editing it, selecting Body in the dialog, and selecting the Solid2 body in the browser. Select OK to commit. Repeat this process for the UpperClampBoltRecess3 feature.
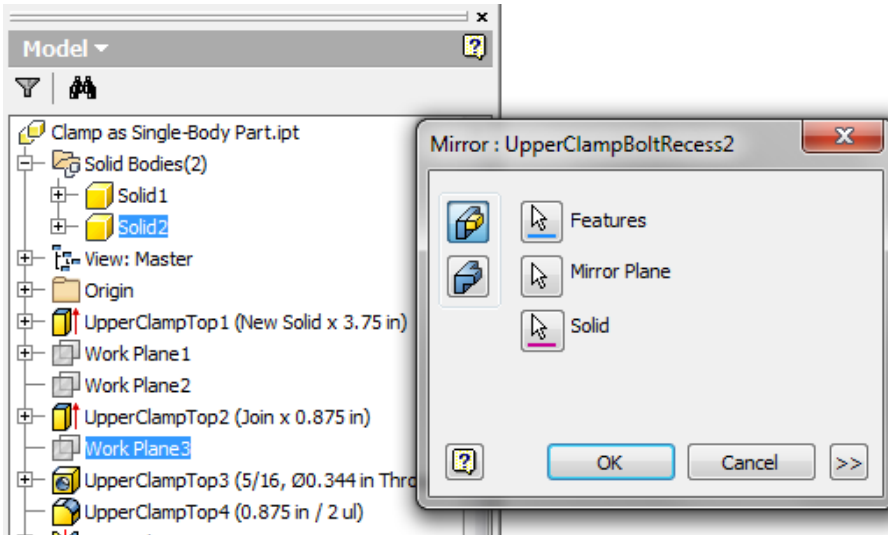
7. Next we will need to change several features for the Upper Clamp Bottom we want to create. Starting with the UpperClampBottom2 Extrude feature all the way to the UpperClampBottom11 mirror feature, edit each feature in the tree except sketch and work features and change them from joining Solid1 to joining Solid2.
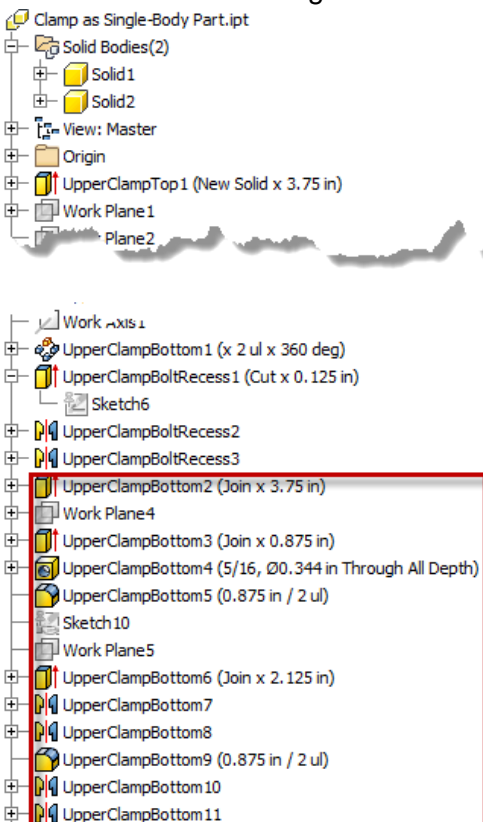
8. Sometimes when creating multiple bodies from a single body such as this, one of the "casualties" will be sketch coordinates and projected geometry in sketches. This happens because the faces and edges that these are based on are suddenly not part of the original "body" that the sketch was looking at. This is the case with Sketch11, which is used to create the UpperClampBoltRecess4 feature which can be seen by right clicking on Sketch11 and selecting Recover.
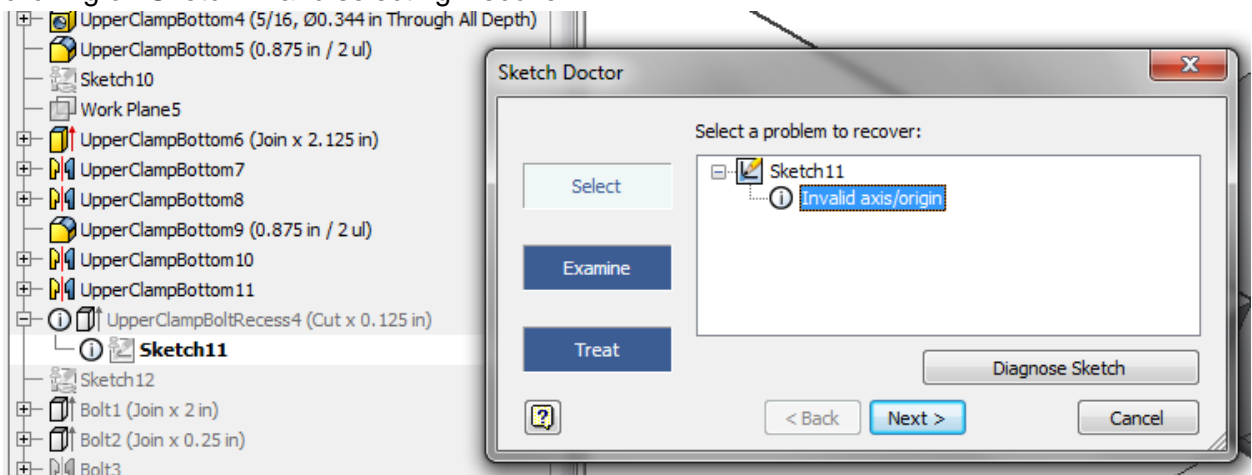
9. To fix the sketch coordinates, do the following steps:
    a. Right click on Sketch11 and select Edit Coordinate System
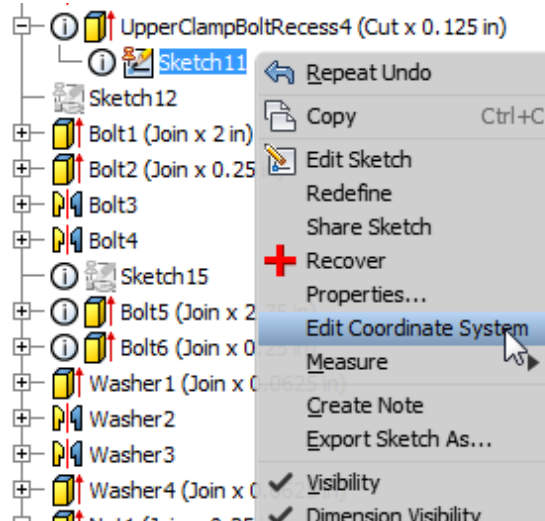
b. Select the "Origin" of the coordinate widget in the graphics
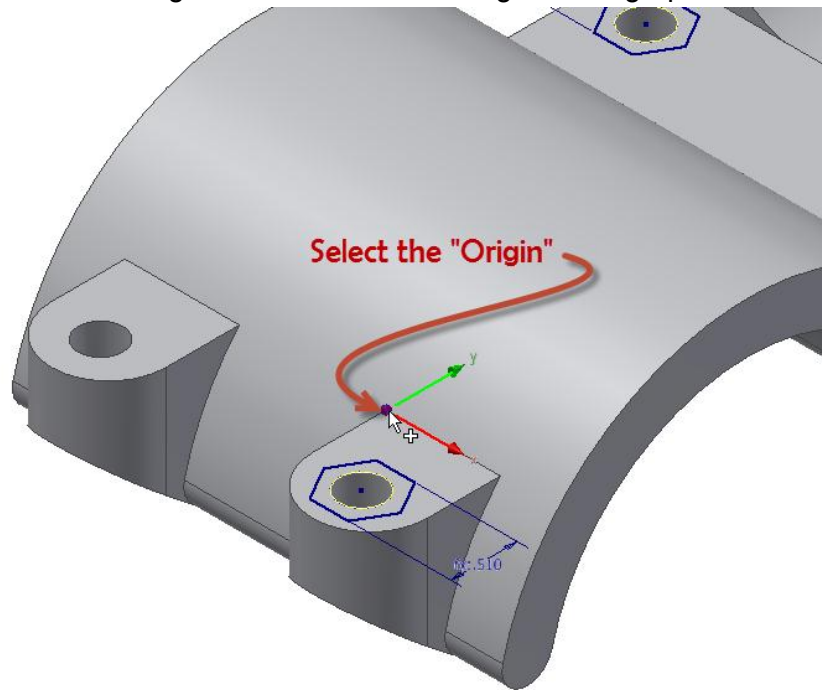
c. Select the exact same point again on the part geometry. It is important to place the origin back at the same point, otherwise sketch geometry can move in space.
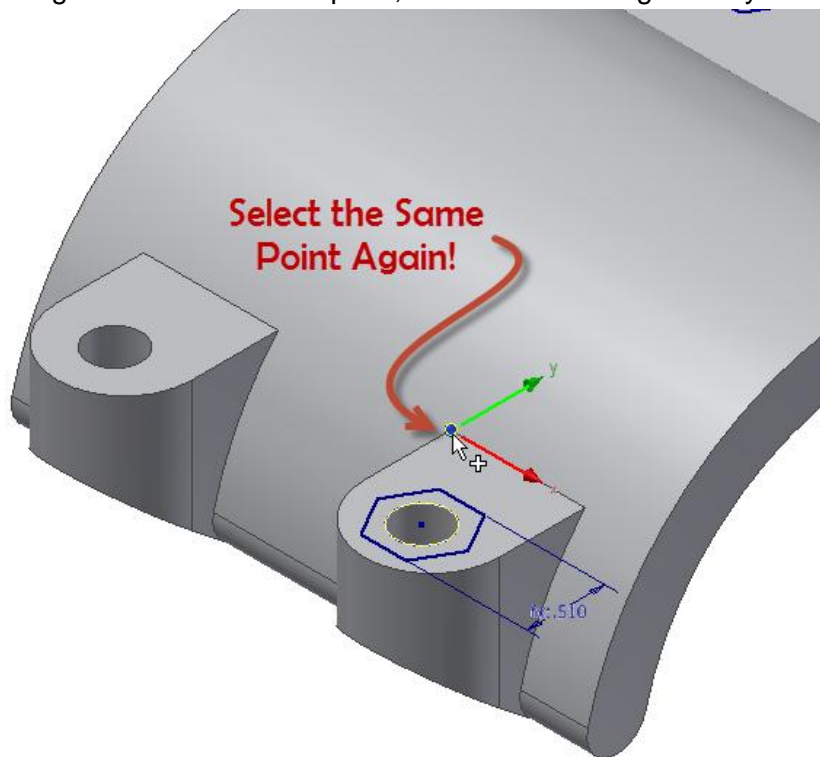
d. Right Click and select OK to finish the coordinate system edit
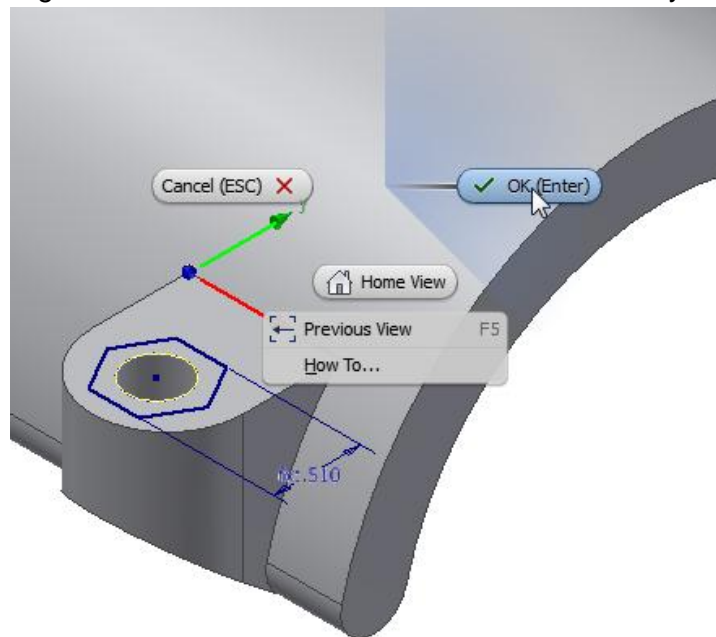
10. Now it is time to separate out the bolts into their own body. Edit the Bolt1 extrude feature and change it from a Join to a New Solid, then select OK. Rename the solid created to be Solid3.
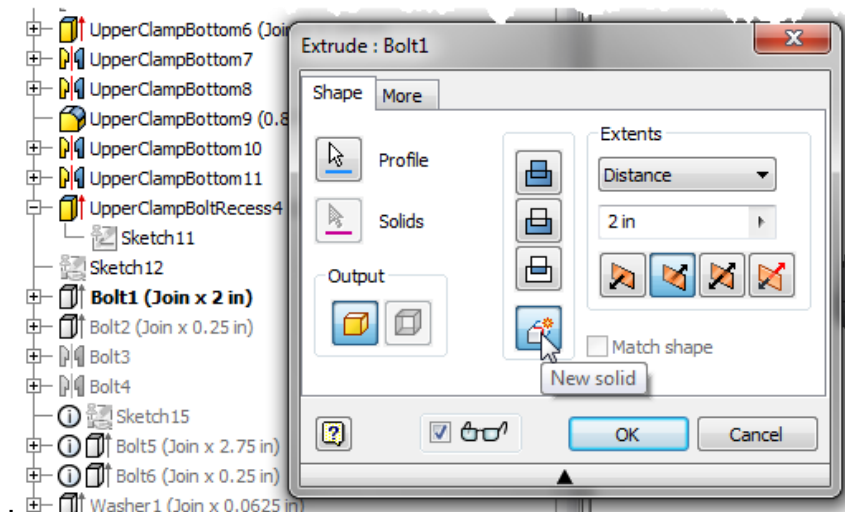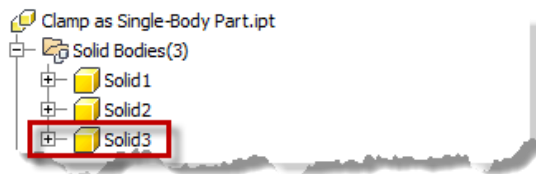
11. Now edit the Bolt2 thru Bolt6 features and change them so they are part of Solid3 instead of Solid1, just like in steps 4 through 6. Sketch15 will have the same problem as Sketch11, so repeat that process of editing the Coordinate System.

For Sketch15, the Origin Point is here
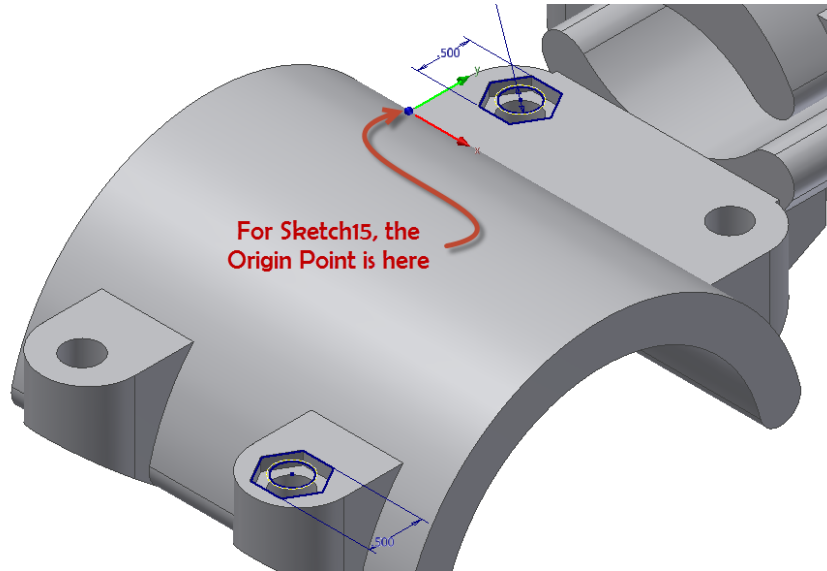
12. Repeat the process for the Washer features by changing the Washer1 extrude feature to create a New solid and rename the new Solid4, then edit Washer2 thru Washer4 to join Solid4. Repeat the process for the Nut features changing Nut1 to create a New Solid, rename the solid to Solid5 and edit Nut2 thru Nut4 to join Solid5.

Washer 1 (New Solid x 0.0625 in)
Washer 2
Washer 3
Washer 4 (Join x 0.0625 in)
Nut1 (New Solid x 0.25 in)
Nut2
Nut3
Nut4 (Join x 0.25 in)
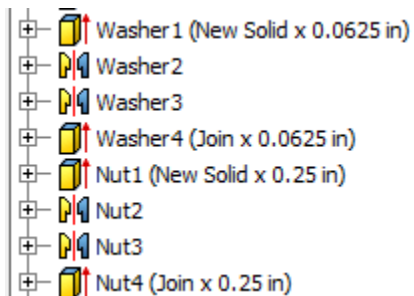
13. Next edit Circular Pattern7 and change it from a Join to Create new bodies and select OK. This will create Solid6, which creates the Lower Clamp Top as it is mirroring Solid1 only (which is Upper Clamp Top). (Optional) Change the name of Circular Pattern7 to LowerClampTop1 if desired. Rename the Solid to be Solid6.
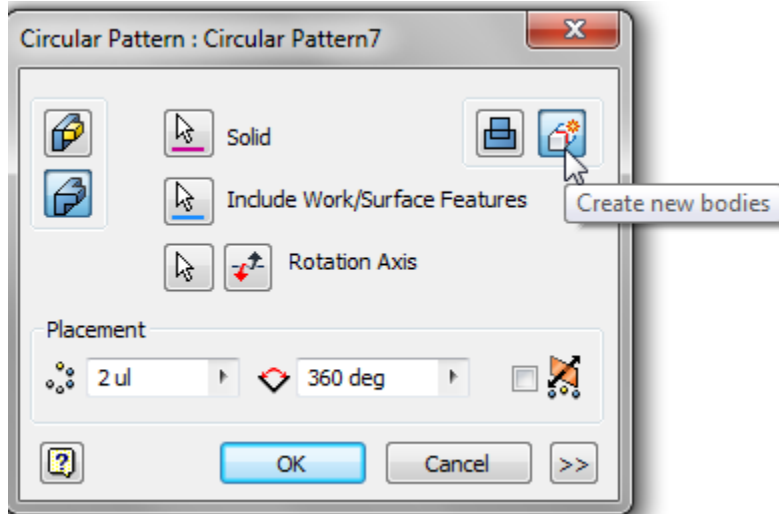
14. To create the Lower Clamp Bottom component, create a new Circular Pattern. Change the pattern to Pattern a Solid, set to create new bodies, select Solid2 as the body to pattern, the Z Axis as the Rotation Axis, and the number of occurrences to 2, then select OK. Rename the solid created to be Solid7.
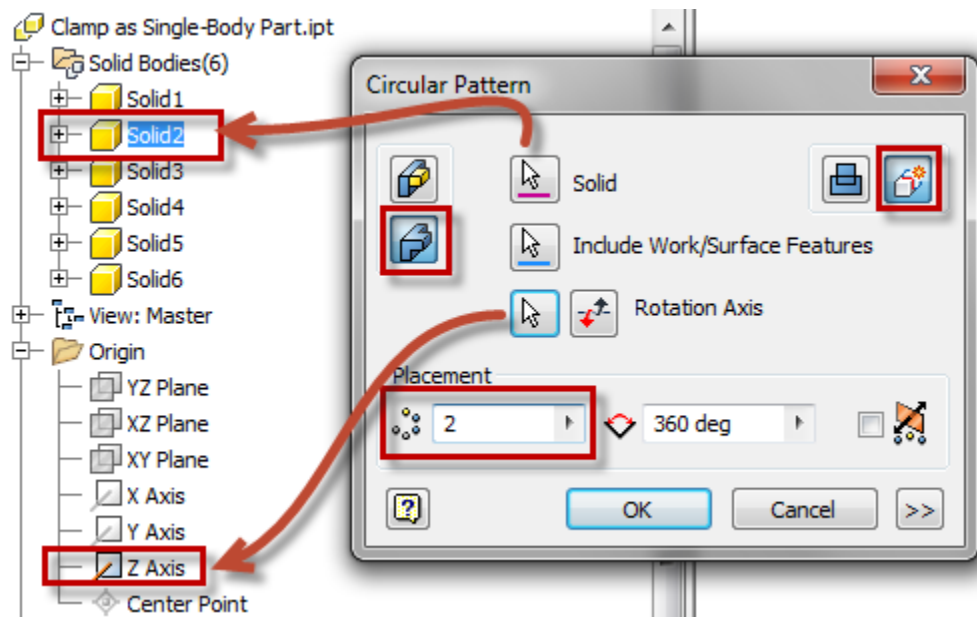
15. The Bolts, Washers, and Nuts also need to add a circular pattern to finish the model. For the Bolts, create a new Circular Pattern, change the pattern to Pattern a Solid, select Solid3, select Z Axis as the Rotation Axis and set the occurrences to 2. Keep the pattern as a Join so the pattern will be part of Solid3, then select OK. This will not create a new solid, which is what is desired.
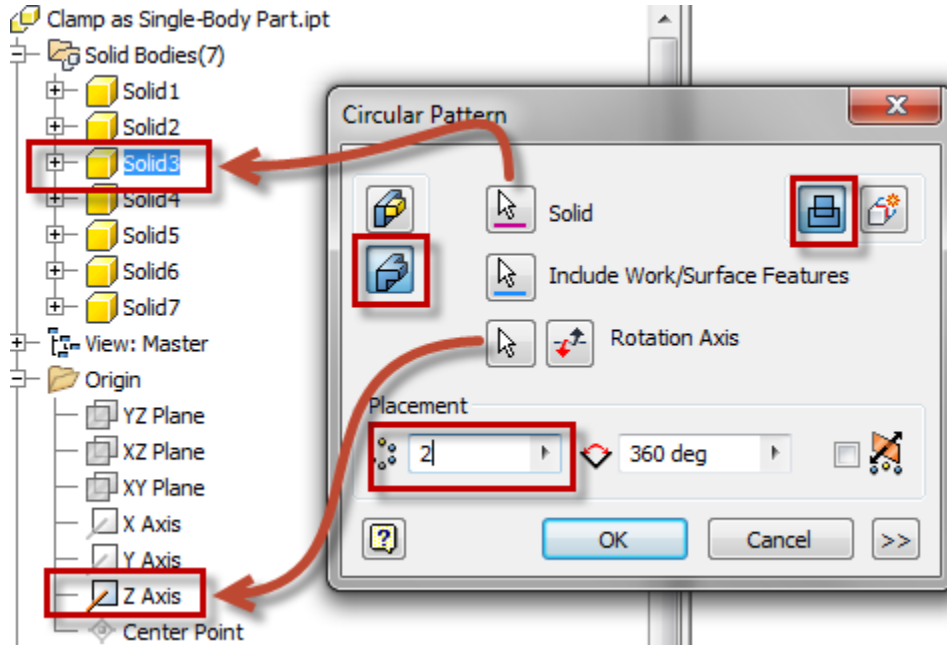
16. Repeat step 14 for both Washers and Nuts, selecting Solid4 for the Washers and Solid5 for the Nuts. This will complete the edits to the model, leaving seven bodies that align with the components desired for creating LOD's. Save the part.
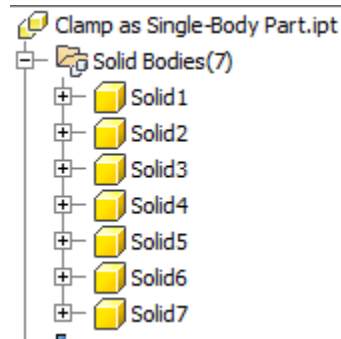
17. Rename the bodies as follows:
    a. Solid1 -> Upper Clamp Top
    b. Solid2 -> Upper Clamp Bottom
    c. Solid3 -> Bolts
    d. Solid4 -> Washers
    e. Solid5 -> Nuts
    f. Solid6 -> Lower Clamp Top
    g. Solid7 -> Lower Clamp Bottom

Note: there are 3 Circular patterns in the example model below the End of Part markers that are the patterns created in steps 14 and 15.

Creating the Clamp Assembly

Creating the Clamp Assembly is easy once the Clamp is a multi-body part.

Note: this example uses the Clamp as Multi-Body Part.ipt example file.

1. Open the Clamp as Multi-Body Part.ipt example file.
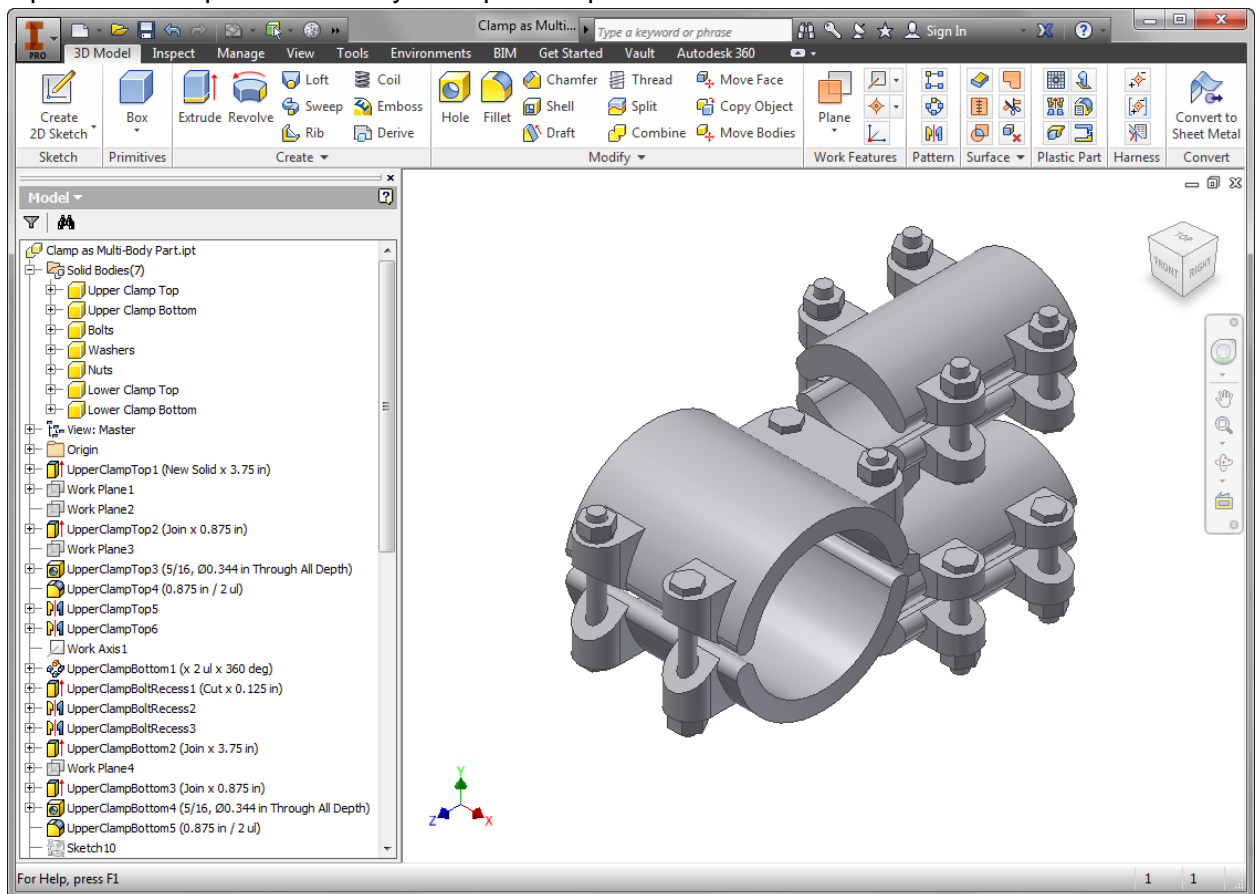
2. On the Manage tab, select Make Components from the Layout group.
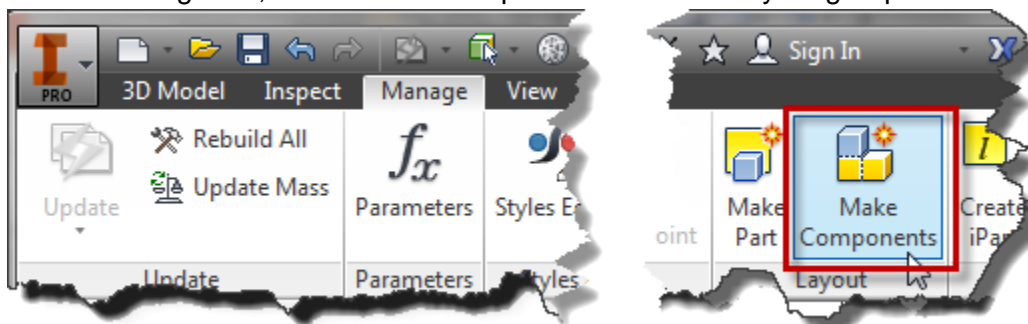
3. In the Make Components: Selection dialog, select the bodies in the browser. Change the Target assembly name to Clamp as Multi-Body Assembly.iam. (Optional) choose a Target assembly location other than the default. Select Next>>
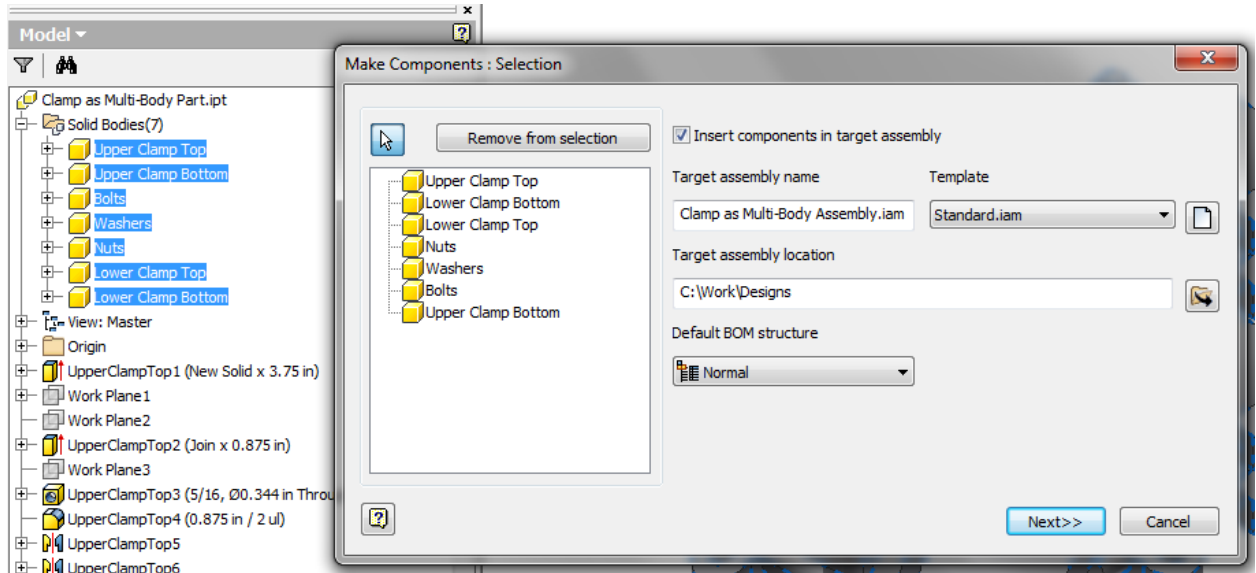
4. In the Make Components: Bodies dialog, give the components meaningful names and locations as desired, then select OK. This will now create the library "Assembly" that can have LOD's in it for reuse.
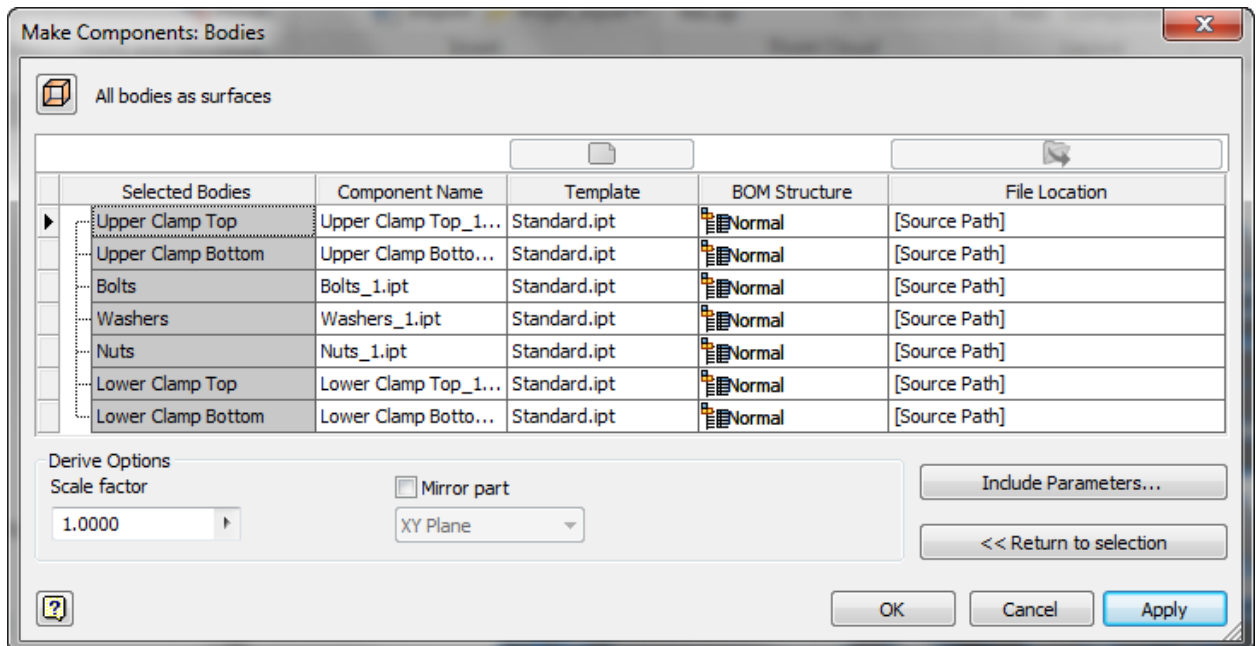
Make your own Substitute Part

Sometimes the shrinkwrap and derive functions will not give you exactly what you want in terms of simplified geometry. In these cases, you have the option to create your own part from scratch and use it as a substitute part, or create a Shrinkwrap or Derived Assembly substitute and modify it.

Note: this example is uses the Tower Peak Assy.iam example file. The finished substitute part that is created/modified is the Tower Peak Assy_Envelope.ipt file, which is associated with the Envelope LOD in the assembly.

In this example, an "envelope" for the Tower Peak of an electrical substation structure is desired. However, creating the desired geometry using the derived assembly Envelop option only gives overall bounding boxes and not a "clean" envelope.
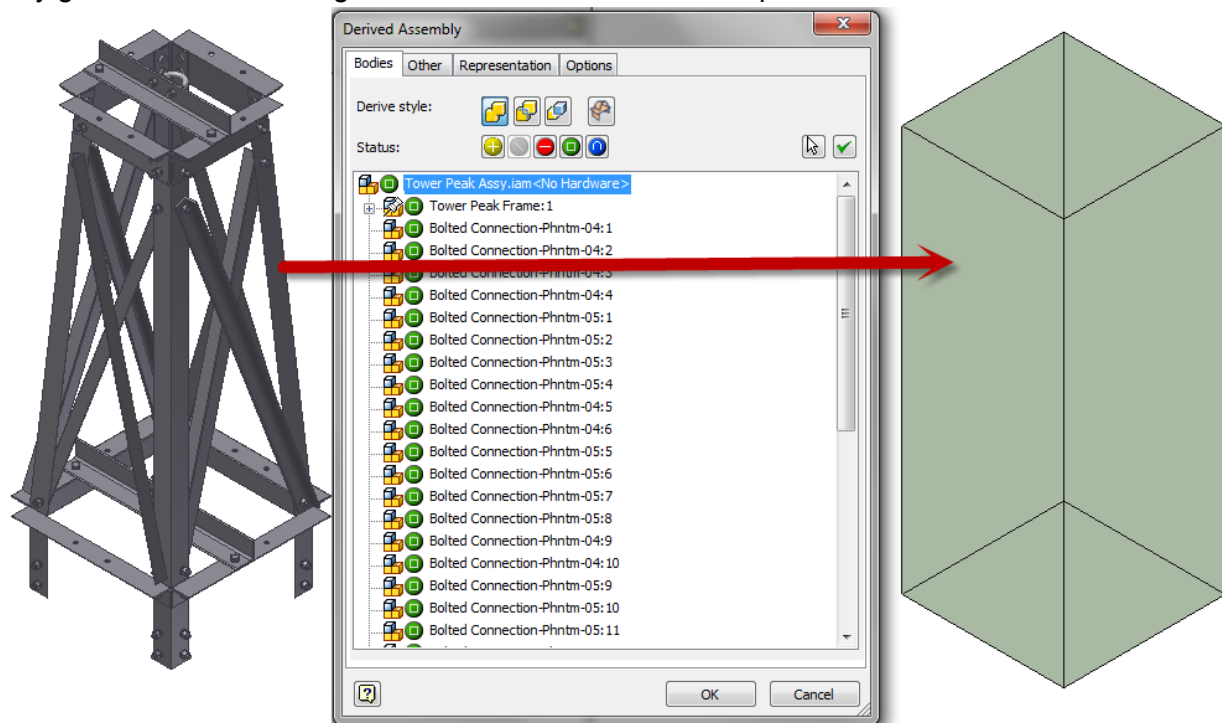
To create an envelope that is more of a shape that follows the "silhouette" of the tower peak, one is created manually.

1. Open the Tower Peak Assy.iam file.

2. Activate the No Hardware LOD that is already setup to remove all Content Center and other hardware from the model.
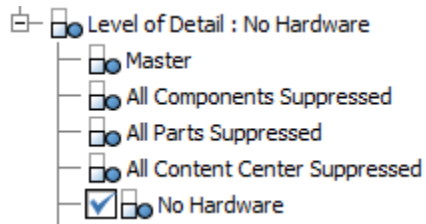
3. Right click on the Level of Detail node and select New Substitute > Shrinkwrap
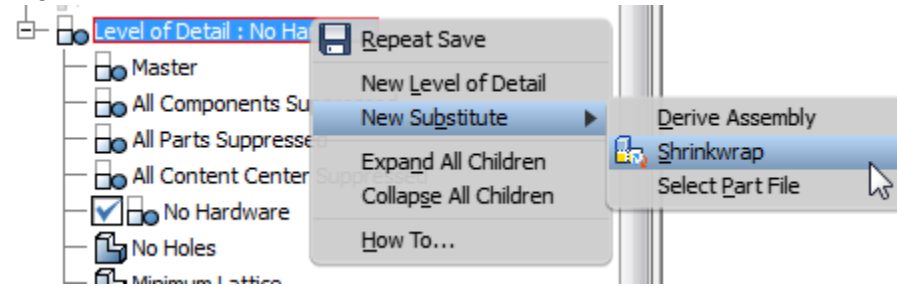
4. Give the Substitute Part a meaningful name and select OK to continue
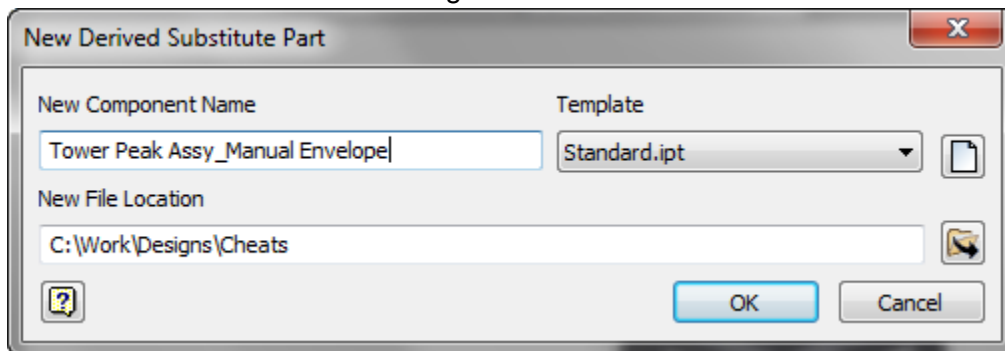
5. In the Assembly Shrinkwrap Options dialog, change the Style to Solid of some type (like Single Solid body with no seams.) Turn on Hole Patching to patch all holes and select OK to create the substitute. Once created save the assembly and rename the LOD to something meaningful such as Manual Envelop.
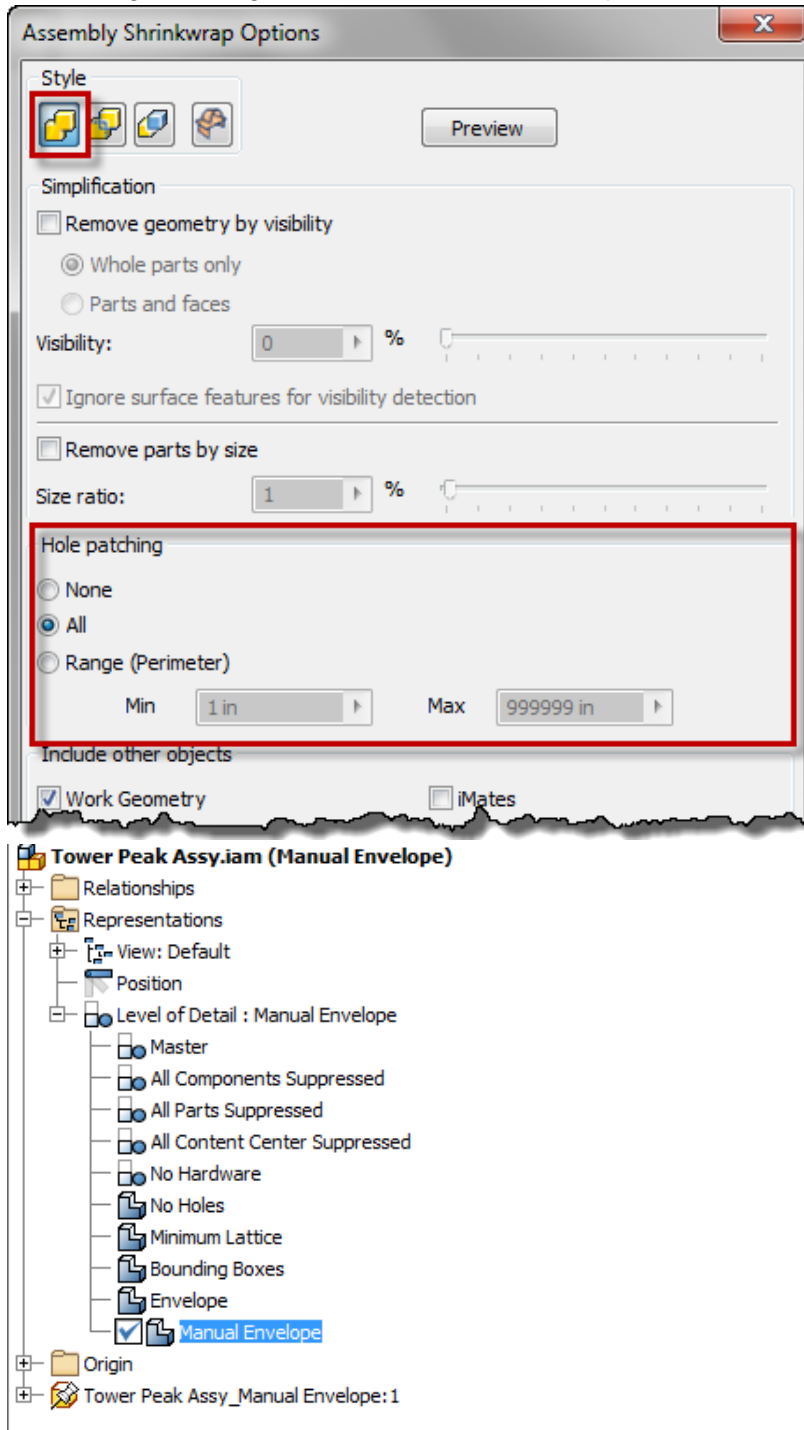
6. Edit the Tower Peak Assy_Manual Envelope.ipt file in place.

7. Create a new sketch on the XZ plane. Project the origin point into the sketch, and create a rectangle that is centered on the projected point and has equal sides of 30 inches.
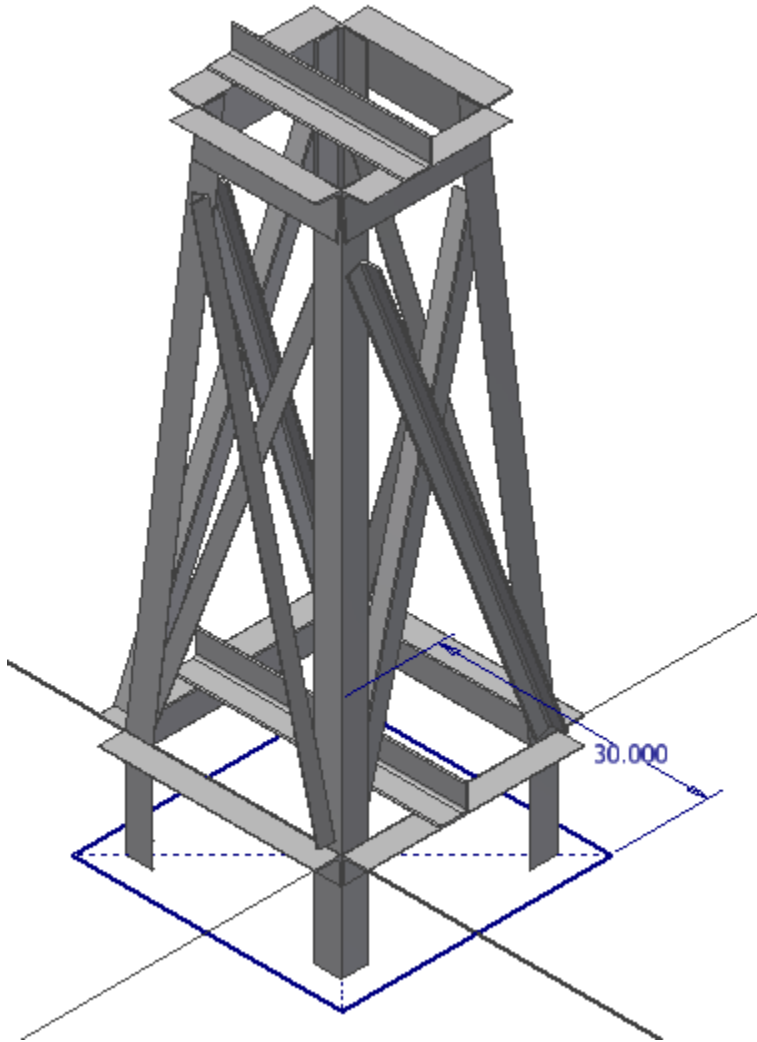
8. Extrude the sketched rectangle "To" the vertex at the top of the angle iron profile as shown, this will create the base of the envelop.
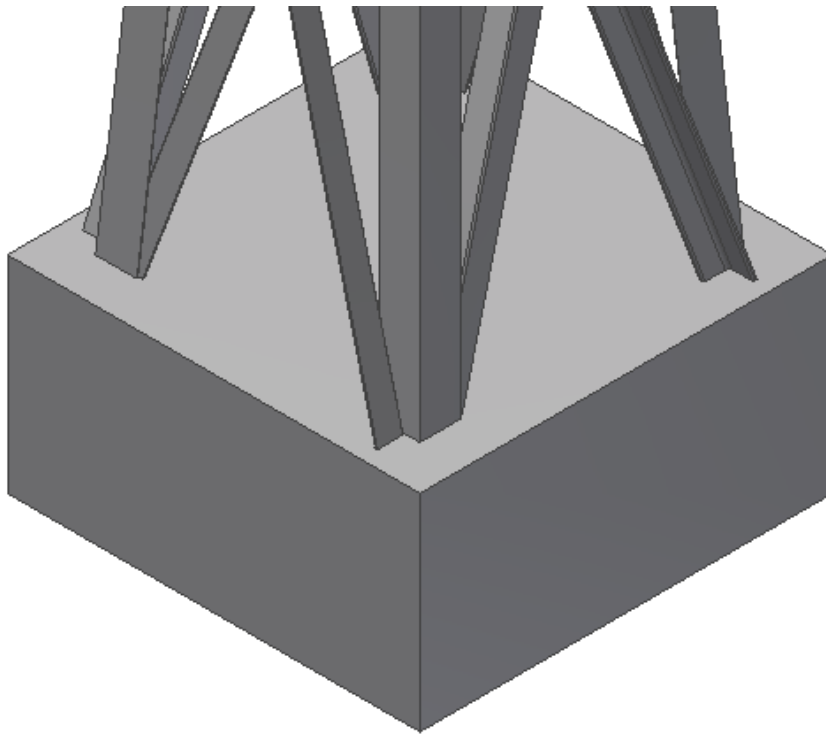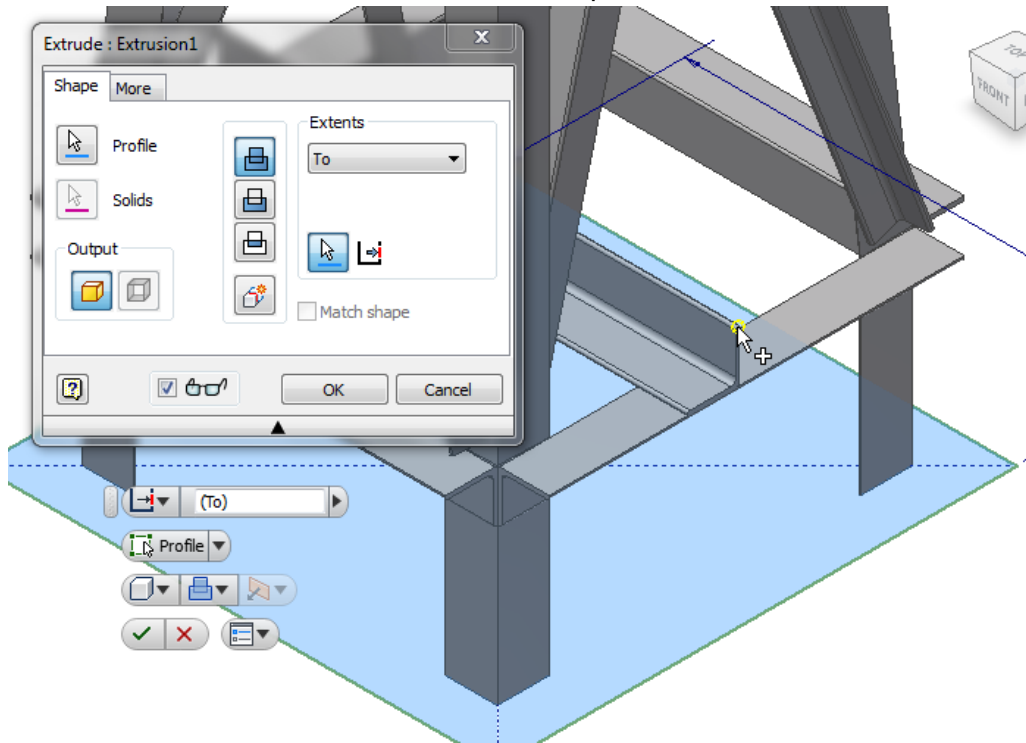
9. Next create a Work Plane on the top of the tower peak by selecting the top edges of the structural profiles as shown. Note that these profiles are at a slight angle, and since there is a small fillet along these edges be careful to select the top most edges.
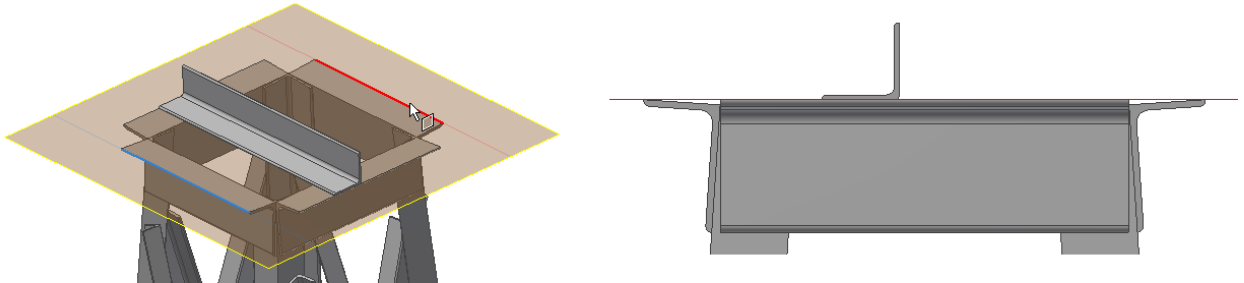
10. Create a sketch on the new Work Plane, project the origin point, and center a rectangle on the projected point that is 22 inches on each side.

22.000
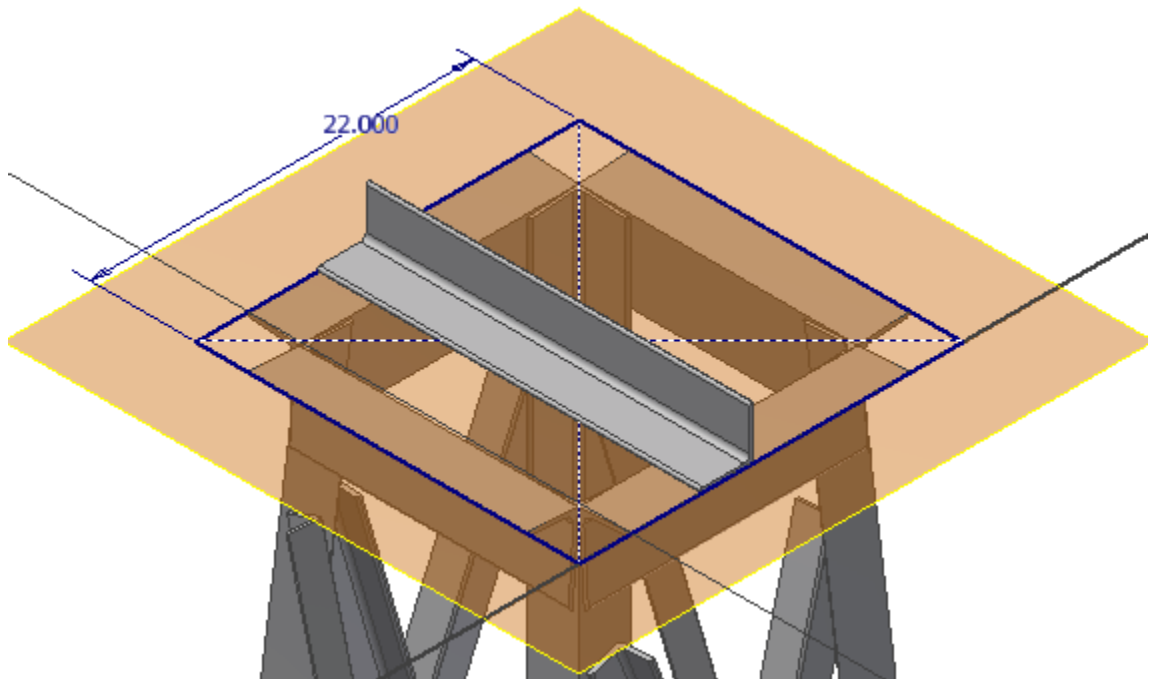
11. Create a simple loft using the edges of the top of the base extrusion and the new sketch.
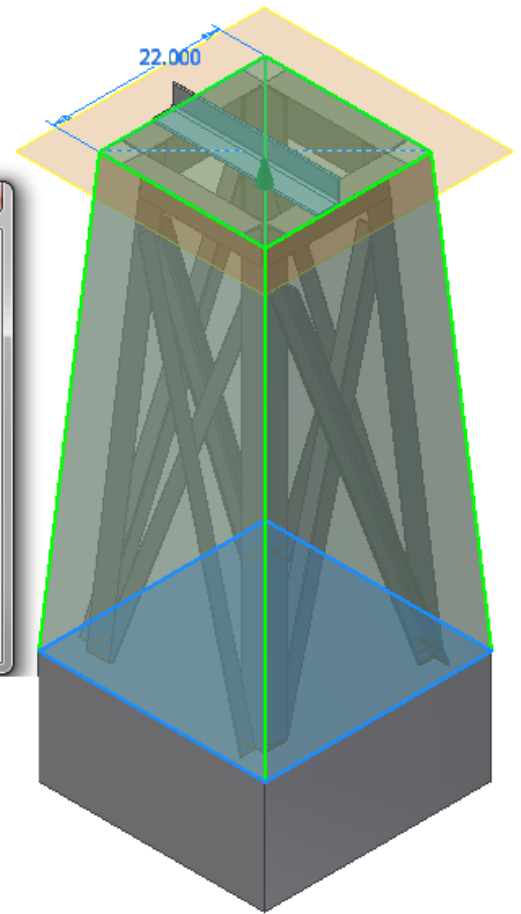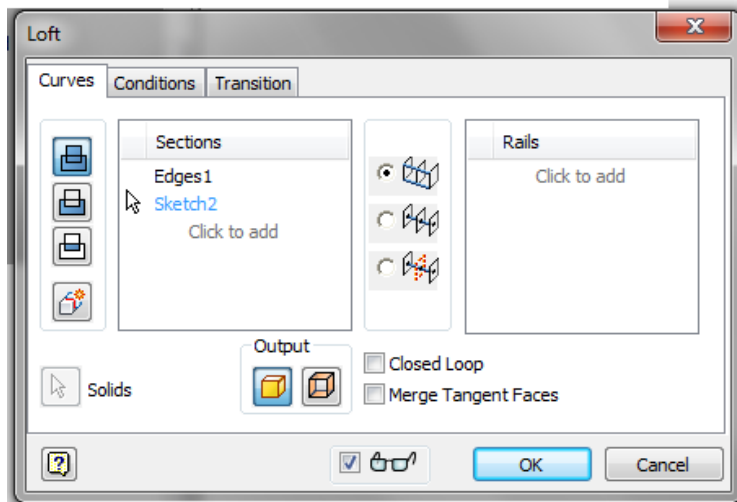
12. Create a sketch that encompasses the profile of the top angle, then extrude the sketch to the end of the top profile to encompass it.
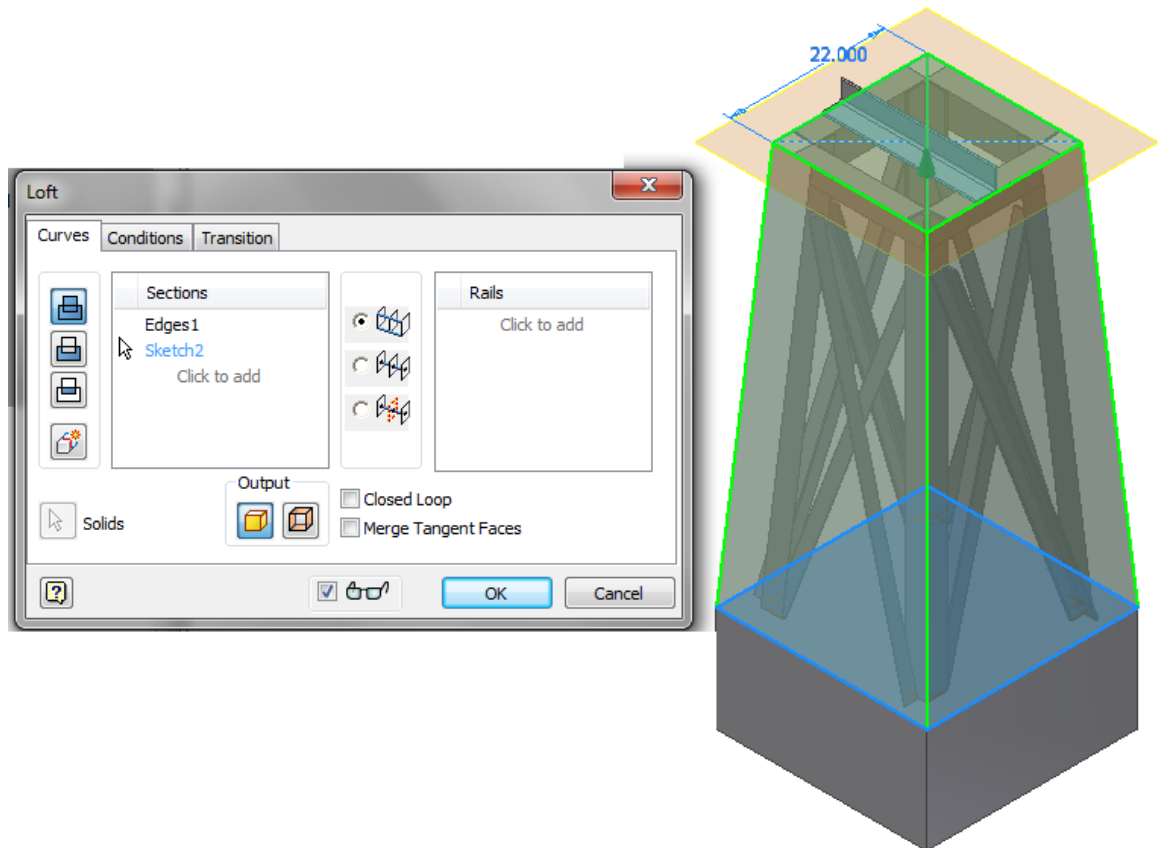
13. Save the modified substitute part, return to the assembly and save the assembly.
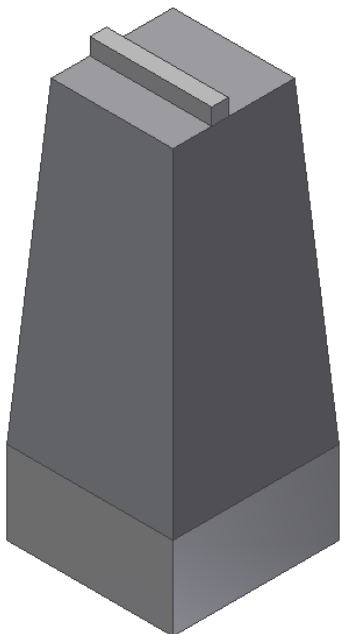
Creating Reusable LOD's
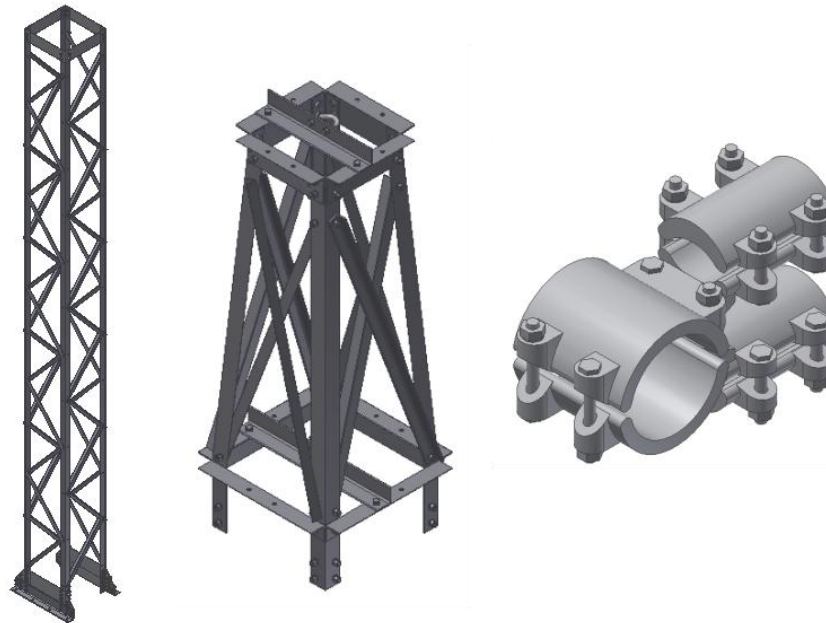
Note: This example uses the Lattice Column.iam, Tower Peak Assy.iam, and Clamp as Assembly.iam files

For substation design, there are many reusable elements that build up a full station. These elements need to have compatible LOD's to use in the upper level assemblies they are used in. In this example the Lattice Column, Tower Peak, and Clamp will have the following LOD's created:

- **No Hardware**: A standard LOD that suppresses all small hardware such as nuts, bolts, washers, etc.
- **No Holes:** A simple Substitute Shrinkwrap LOD that takes the No Hardware LOD and patches all the holes and fills all the voids in it.
- **Minimum Lattice:** This LOD, specific to the industry, removes the back and side lattice elements for use in creating side views of the lattice structures, where the lattice in the "back" of the view does not need to be seen.
- **Bounding Boxes:** An alternative to the Minimum Lattice LOD for use in "long shots" and isometric views in drawings
- **Envelope:** Relatively simple bounding box of each major element for use in views of the full assembly where individual elements will not be seen, but the silhouette of the subassemblies are important.

Creating each of these LOD's in each of the assemblies are described below.

Creating the No Hardware LOD's

Lattice Column & Tower Peak
For the Lattice Column & Tower Peak assemblies, create the No Hardware LOD as follows:

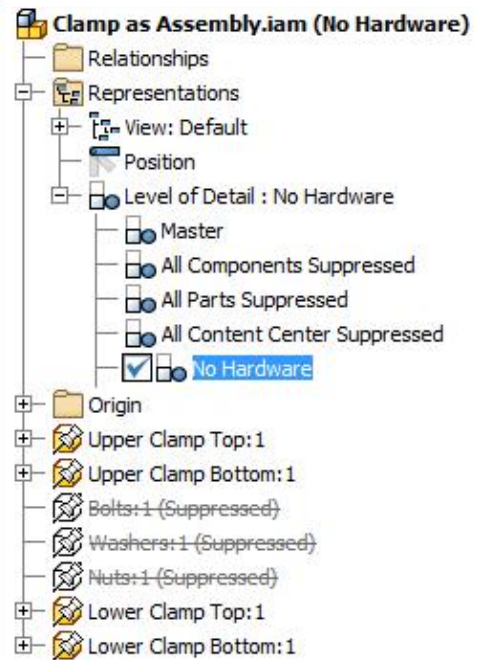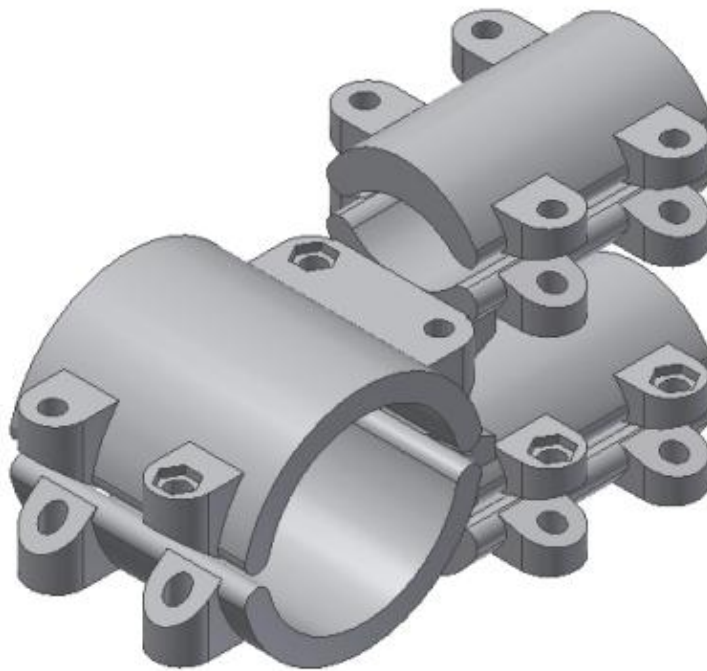1. Activate the All Content Center Suppressed LOD
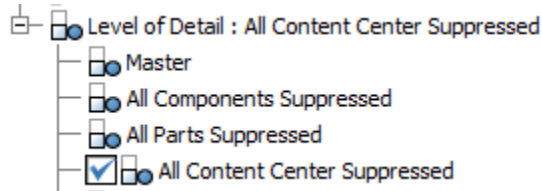
2. Create a new Standard LOD and name it No Hardware
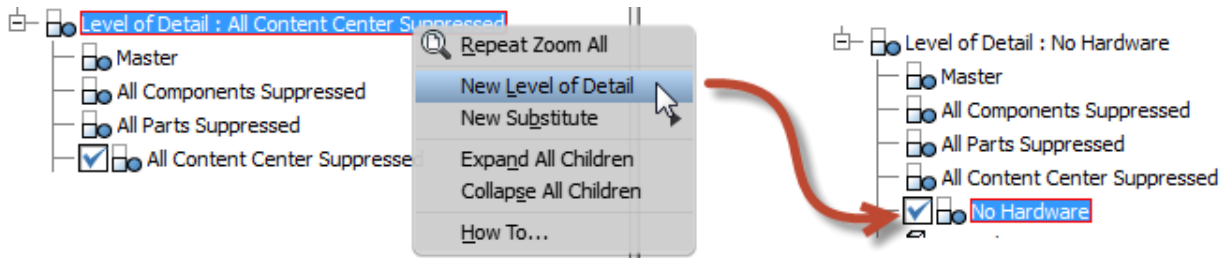
3. Using techniques such as Tag-It or the Assembly selection tools, remove anything considered Hardware that you do not want. Save the assembly when finished.
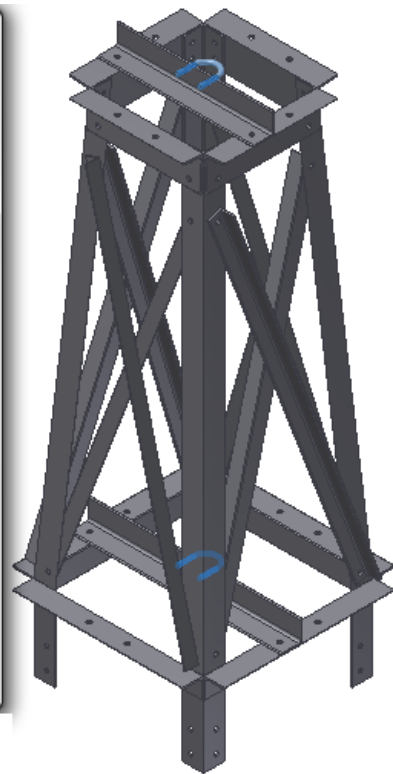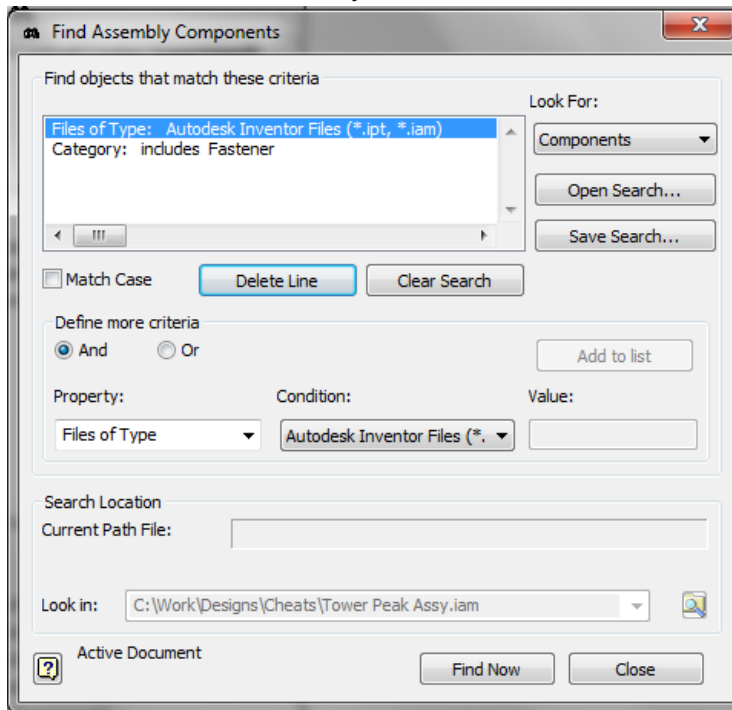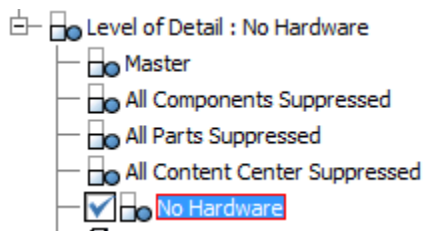
Clamp Assembly

For the Clamp Assembly, since the "hardware" created are derived parts, simply create a No Hardware named Standard LOD and suppress the parts representing the hardware (Bolts.ipt, Washers.ipt, and Nuts.ipt).

Creating the No Holes LOD

Lattice Column & Tower Peak

1. Set the active LOD to be the No Hardware LOD.

2. Create a New Substitute > Shrinkwrap. Name the Shrinkwrap part "{Assy Name}_No Holes.ipt".
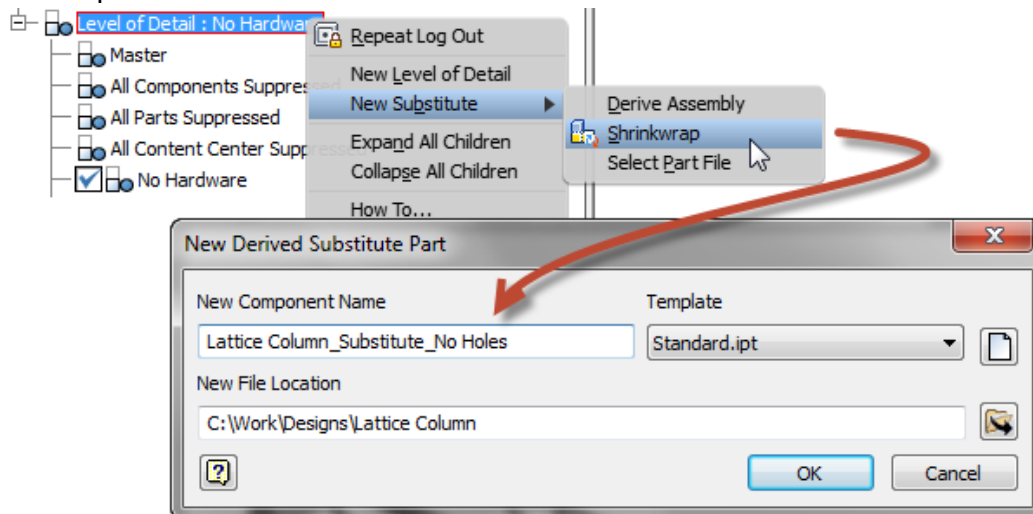
3. Set the shrinkwrap options to create a solid part, patch all holes, and fill all internal voids.
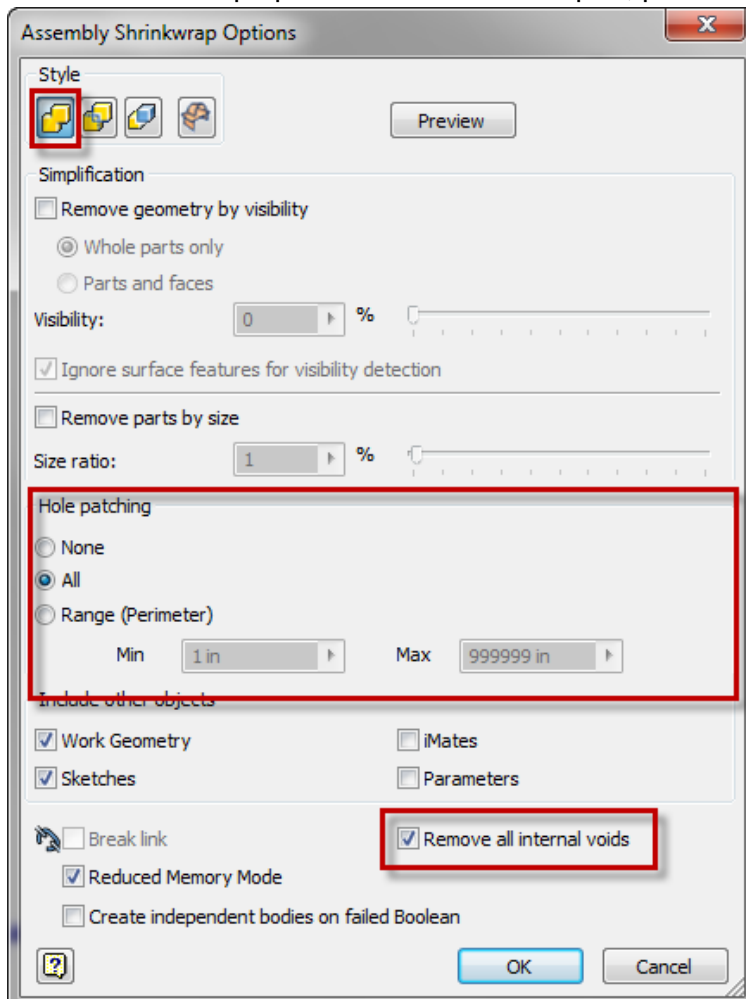
4. Once the substitute is created rename the LOD to be No Holes. Save the file and the assembly.
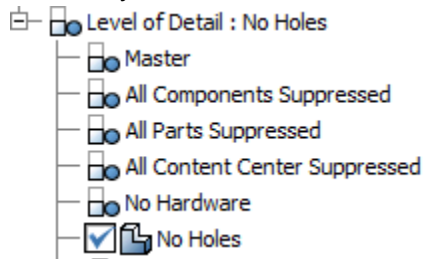
Clamp Assembly

For the Clamp Assembly No Holes Substitute, follow the same steps as for the Lattice Column & Tower Peak. Once the LOD and Substitute are created, the substitute part will require some additional editing since some of the holes cannot be patched by the shrinkwrap.
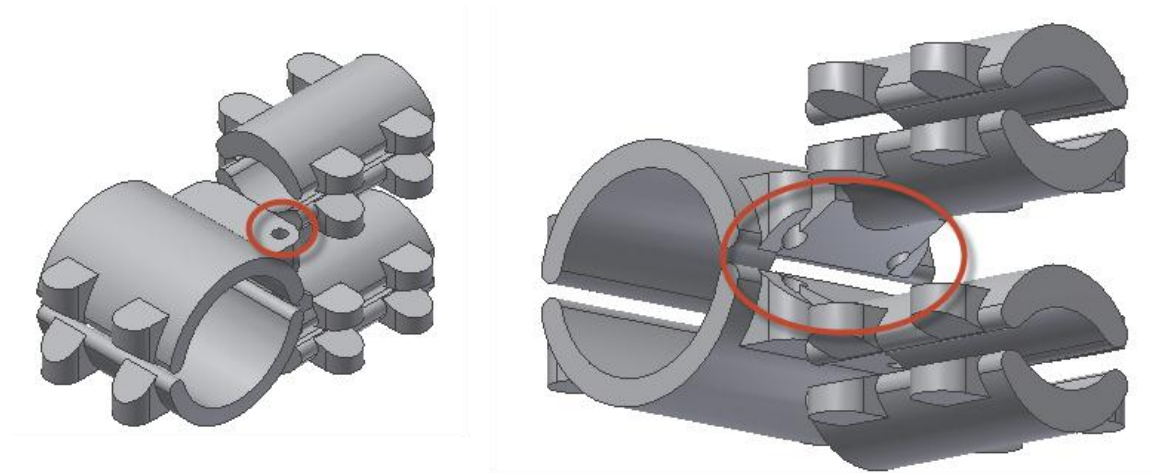
1. Edit the Clamp Assembly No Holes substitute part
2. Create a sketch on the top face as shown and project the outline loop
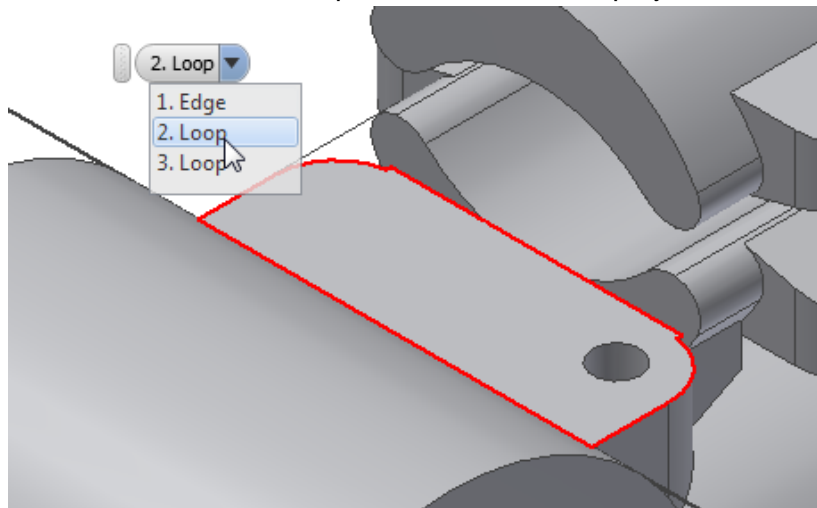
3. Project the bottom edge of the wedge feature into the Sketch as well
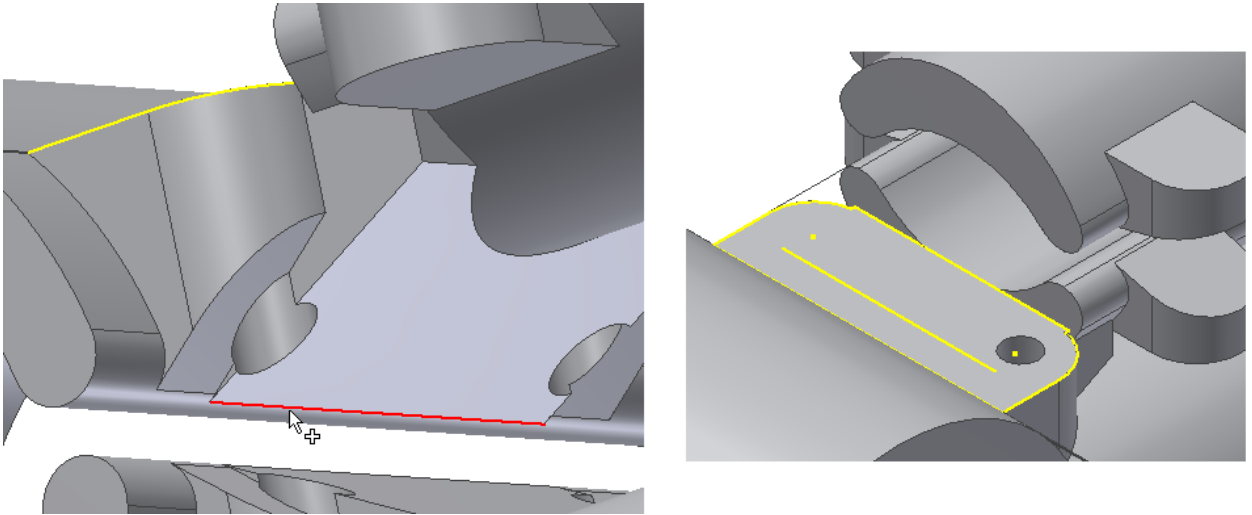
4. Close the profile as shown
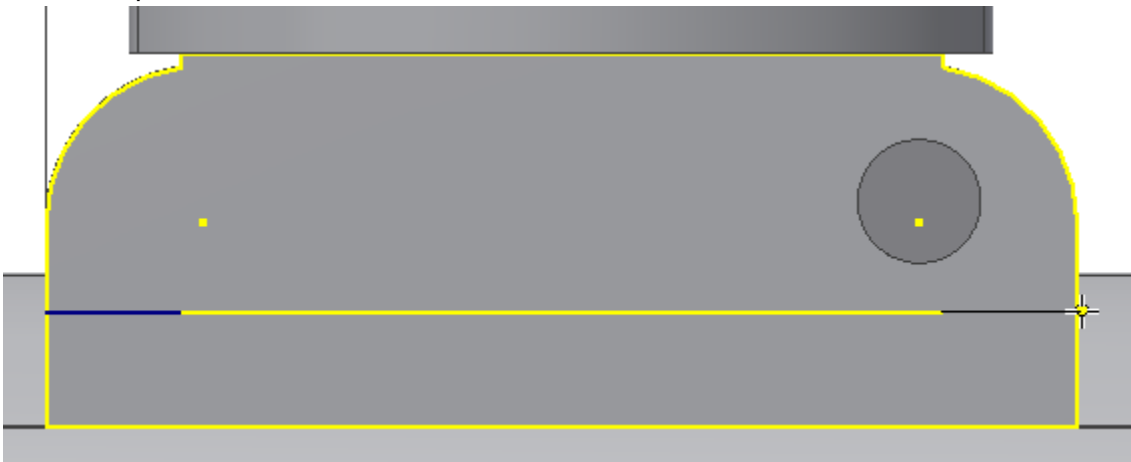
5. Extrude the profile to the lower part of the wedge
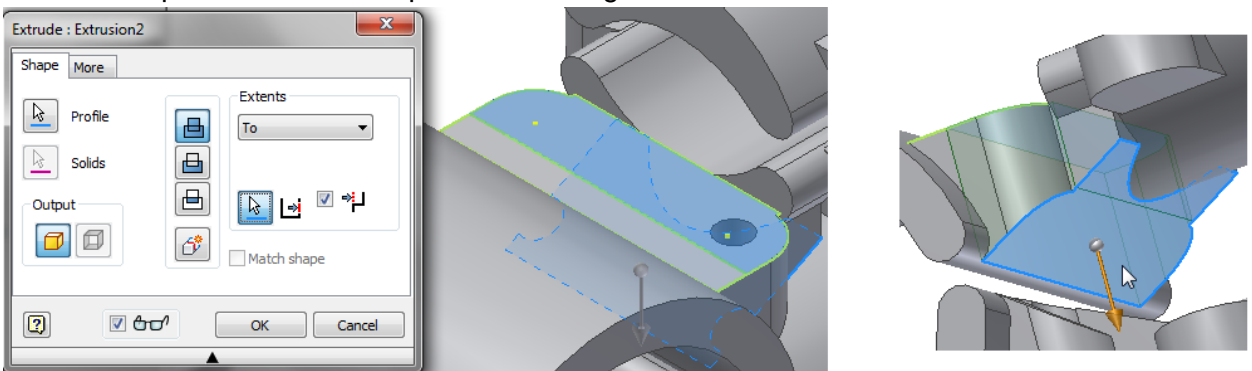
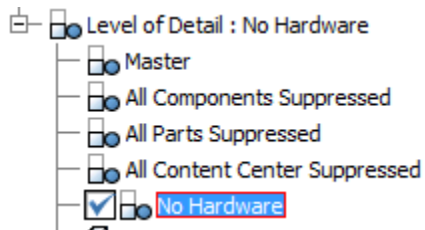6. Mirror the extrude across the XZ plane. Save the file and the assembly.
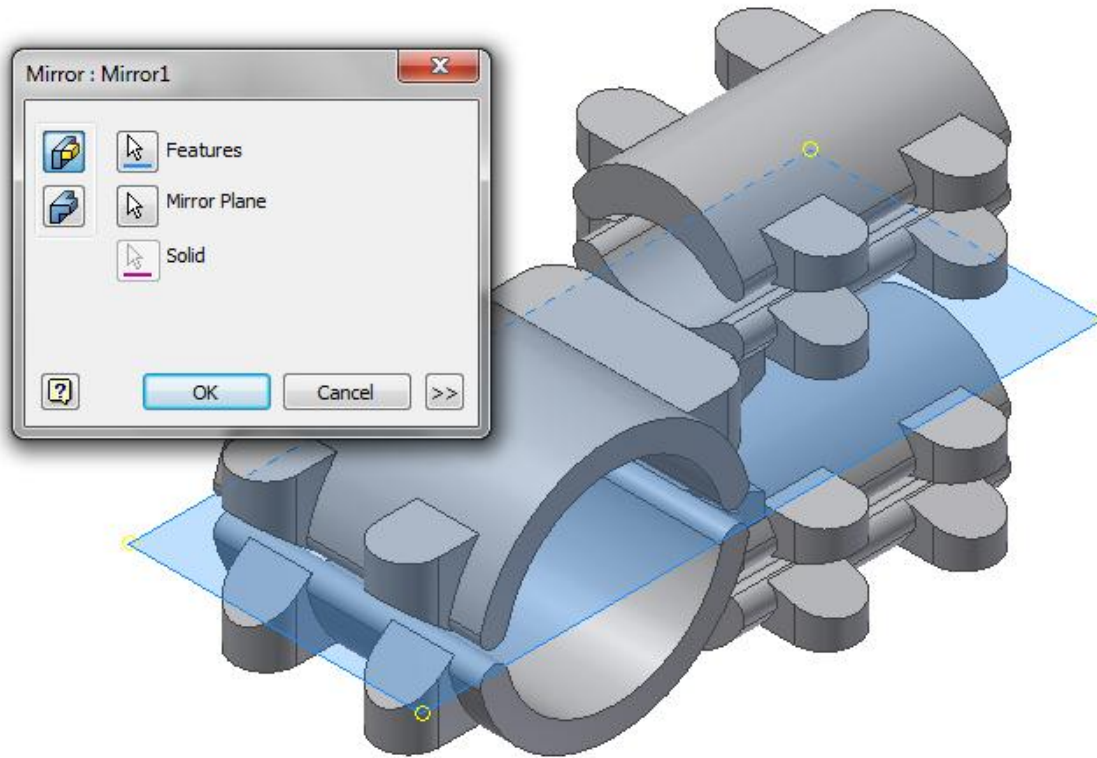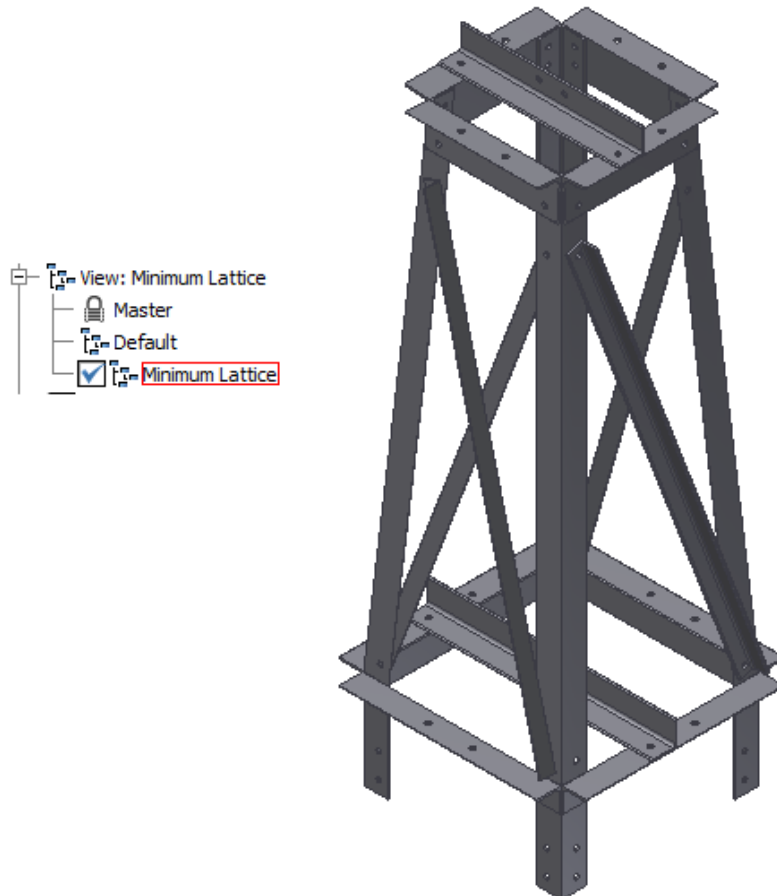
Create the Minimum Lattice LOD
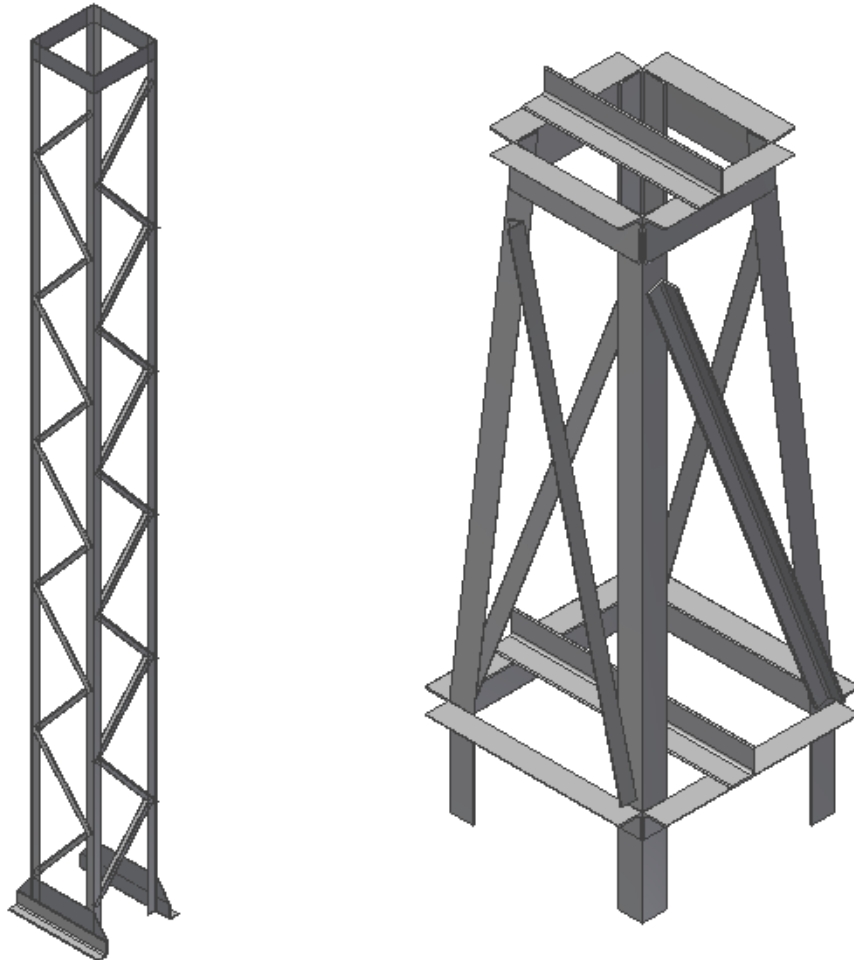
The minimum lattice LOD removes the lattice from the Back and one side of the lattice structures.

**Lattice Column & Tower Peak**
1. Activate the No Hardware LOD

2. Create a View Representation called Minimum Lattice and turn off the visibility of the lattice components on one side and the back.


3. Create a new shrinkwrap substitute level of detail with the Minimum Lattice view rep active. Patch all holes and fill voids the same as the No Holes LOD. Save the file and the assembly.
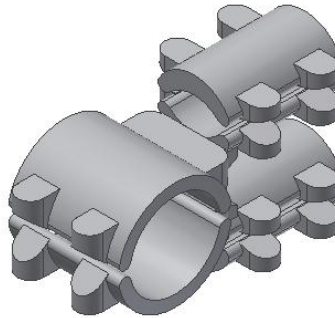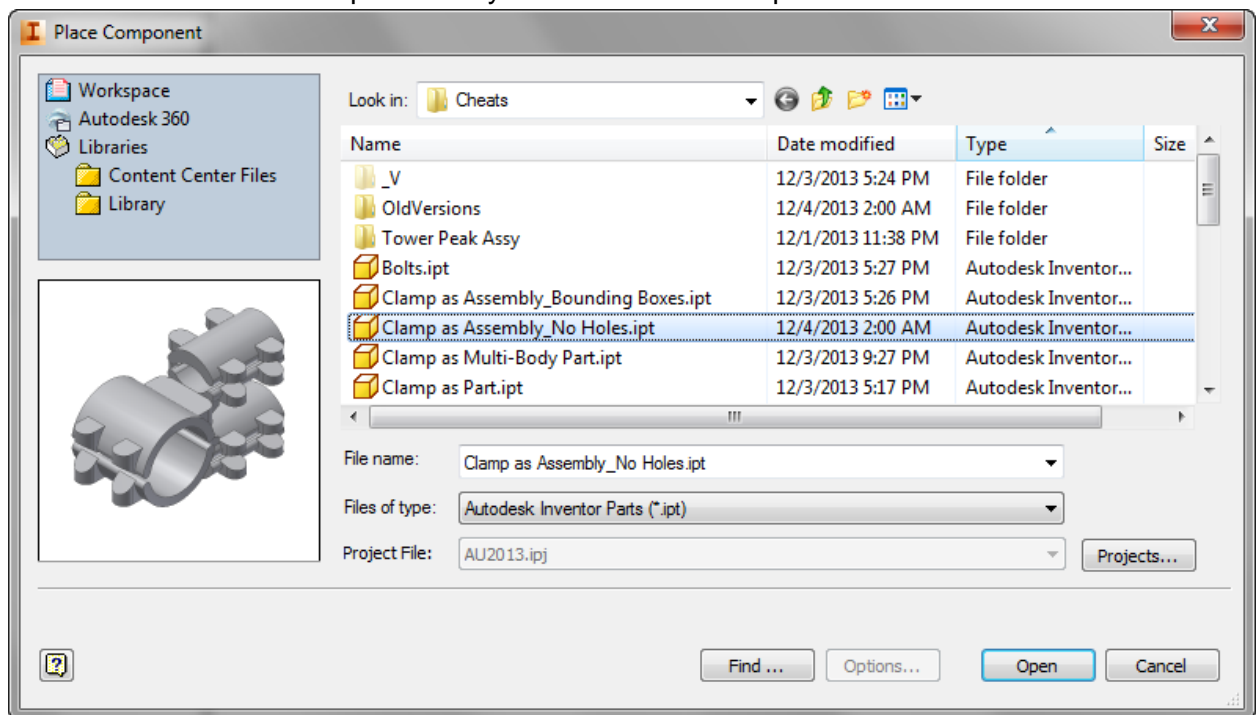
Clamp Assembly

With the Clamp Assembly, since there is no lattice so an alternative for that level of detail is needed. In this case the best answer is to reuse the No Holes LOD.

1. Right click on the Level of Detail node and choose New Substitute > Select Part File
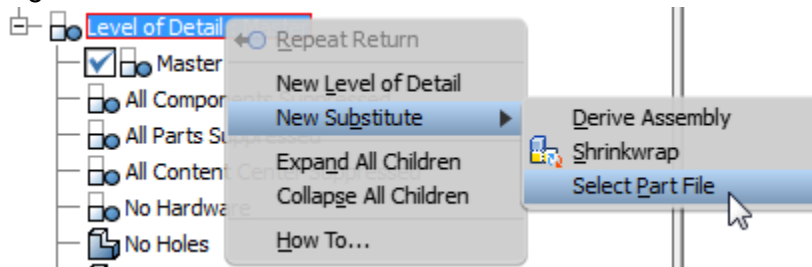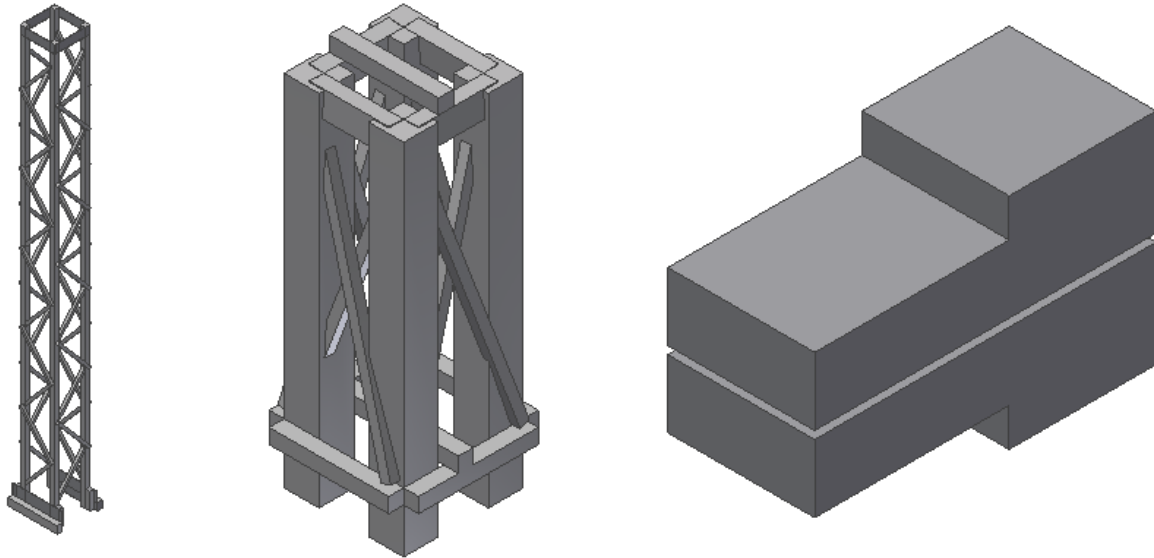

2. Fine and choose the Clamp Assembly No Holes substitute part file and choose it.
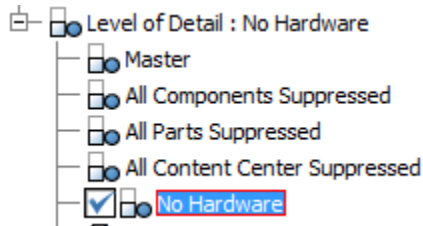

3. Name the substitute Minimum Lattice and save the assembly.

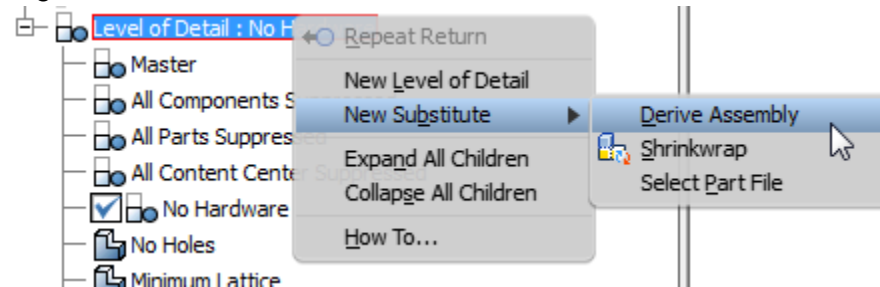## Creating the Bounding Boxes LOD

Creating the Bounding Boxes LOD is the same for all of the assemblies.
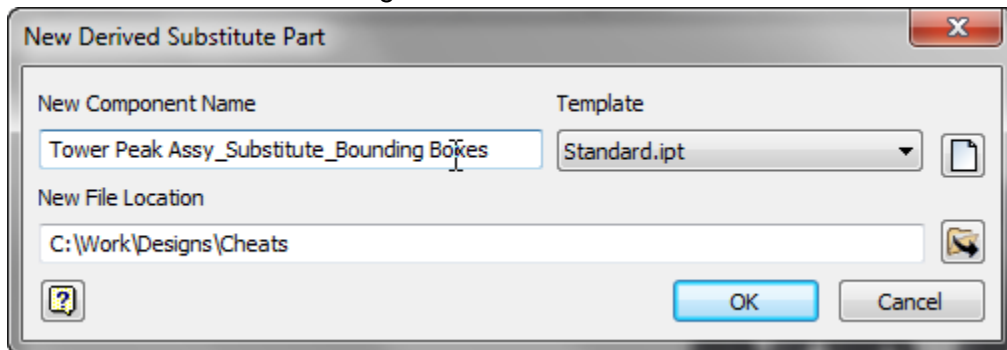
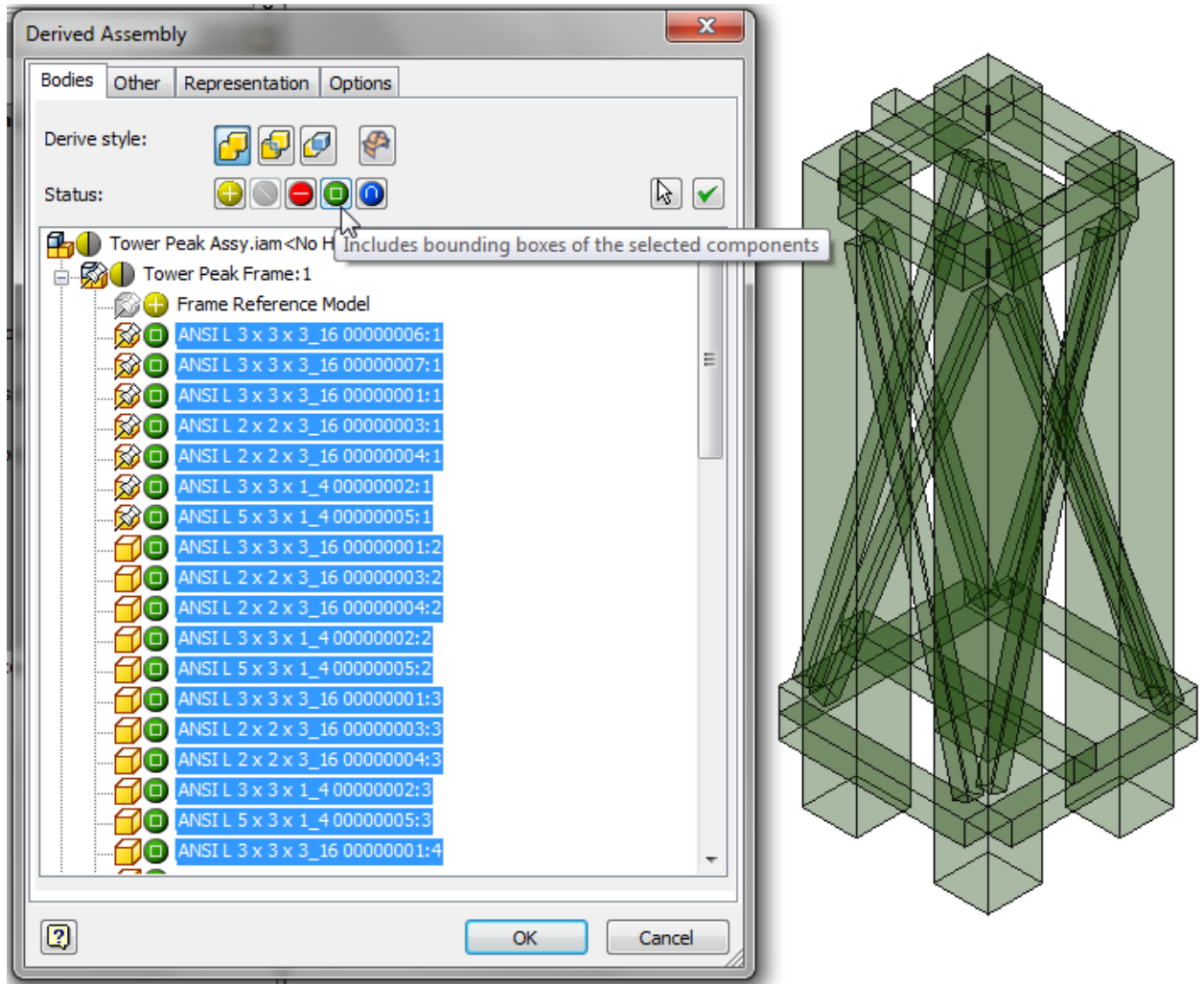1. Activate the No Hardware LOD

2. Right click the Level of Detail node and select New Substitute> Derive Assembly

3. Give the substitute a meaningful name and select OK

4. In the Derive Assembly dialog, select all of the visually significant components and select the Include bounding boxes status. Select OK.
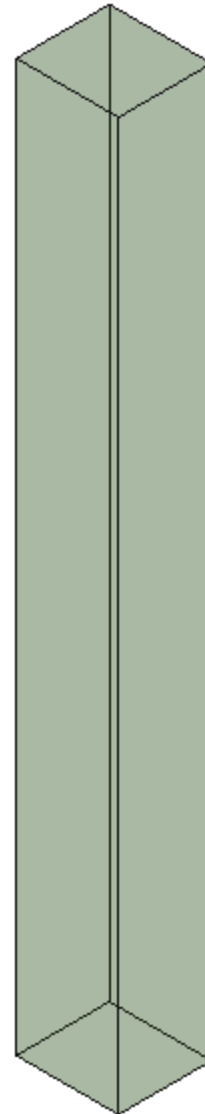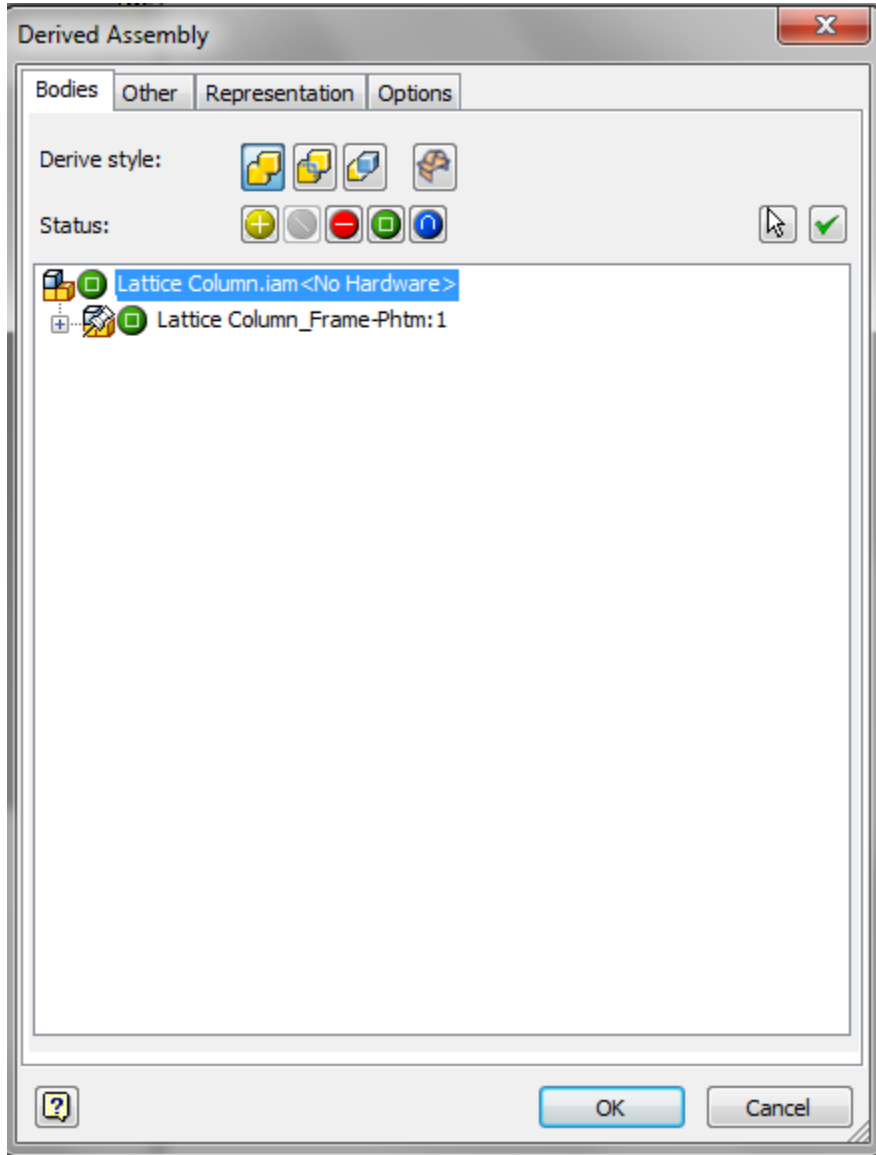

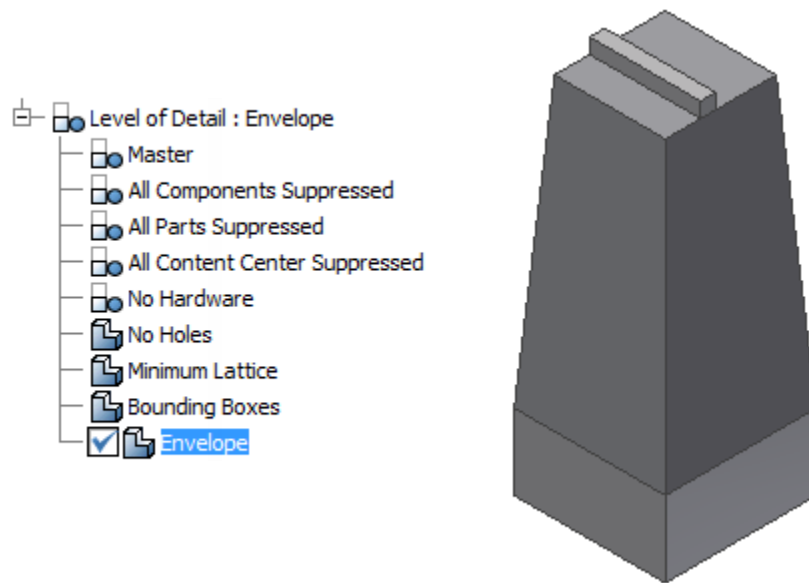5. Rename the LOD to Bounding Boxes and save the assembly.

Creating the Envelope LOD's

Lattice Column

For the Lattice Column, follow the same instructions as for the Bounding Boxes LOD (see Creating the Bounding Boxes LOD), except in the Derive Assembly dialog select the top node and choose the Include bounding boxes status.
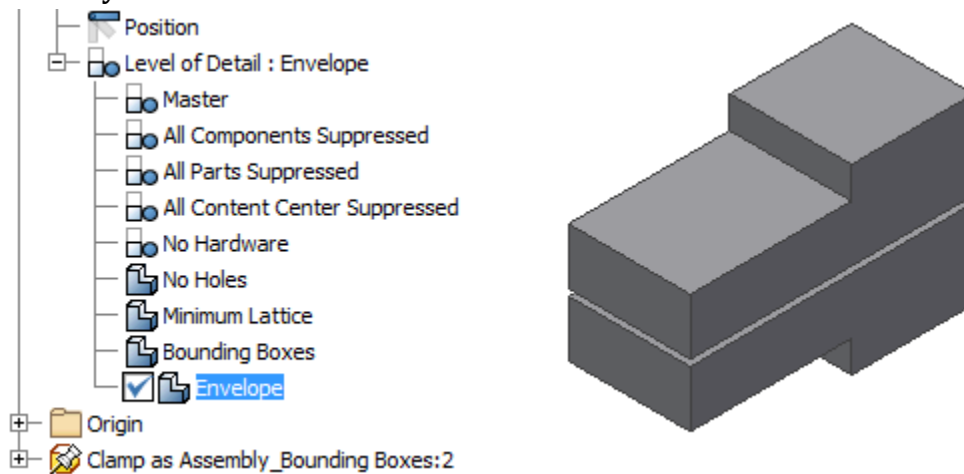
Tower Peak

For the Tower Peak assembly, create the envelope as described in "Cheating" Examples Make your own Substitute Part section.

Clamp Assembly

For the Clamp Assembly, create the Envelop LOD in the same way that the Minimum Lattice LOD was created for the Clamp Assembly, except select the Bounding Boxes substitute part instead.