Beyond Assembly Basics – Advanced Assembly Modeling with Inventor

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Who am I?

- James “Jim” O’Flaherty
- Engineering Manager at Versa-Gard, LLC
- Started career in 1980 using a drafting board and slide rule
- Began on Autodesk Inventor @ Release 3
- Autodesk Certified Inventor Professional
- Autodesk Certified Inventor Instructor
- Autodesk Expert Elite – Awarded in 2014
- Other CAD software? (those shall remain nameless)

Class summary

So now that you've been working in Inventor and you know the basics of Assembly modeling, let's look at some of the more advanced features and practices for assemblies.

Key learning objectives

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

- Use some of the more advanced features for modeling assemblies
- Minimize the impact of file size for large assemblies
- Create Weldments and other assembly level only features
- Work more effectively in the assembly environment

Section #1A – Settings - Working Effectively in Assemblies

Hardware

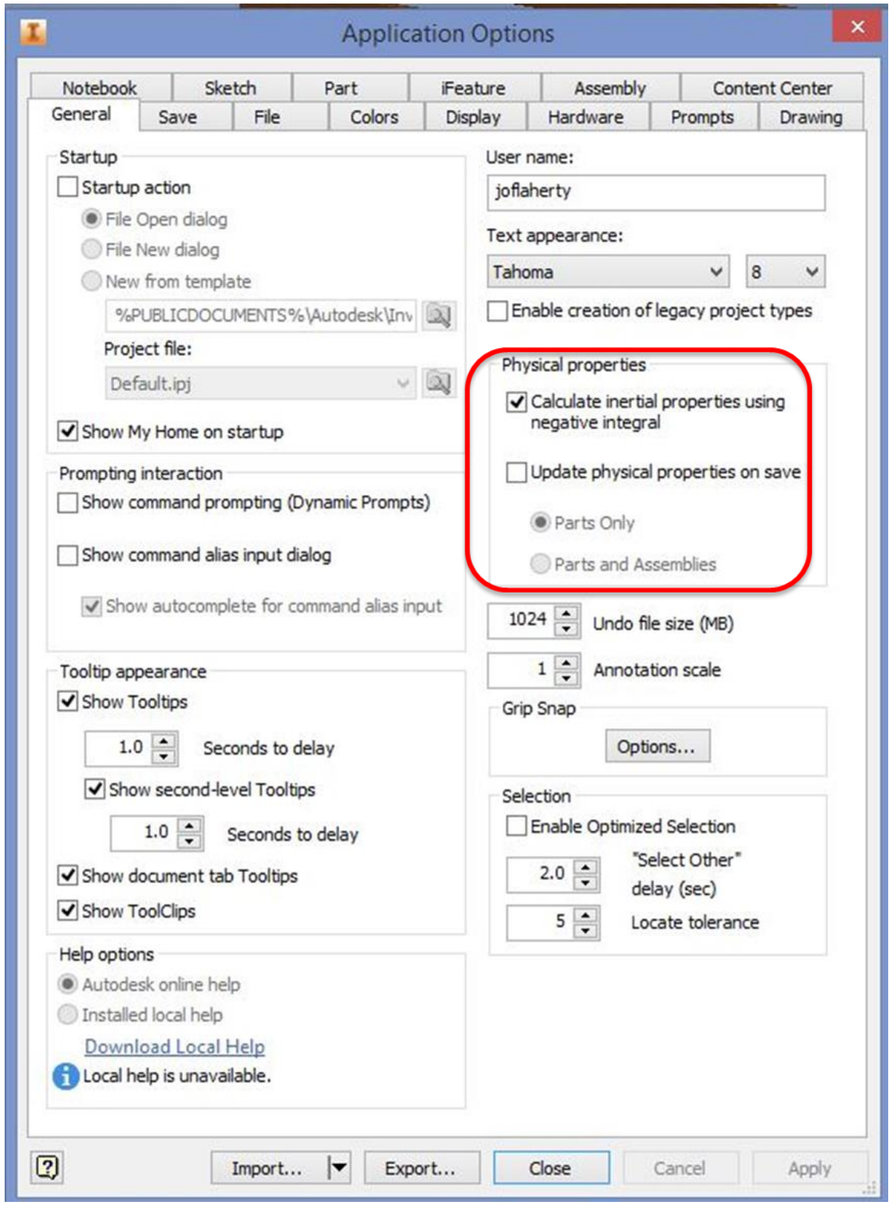
- The hardware you purchase or have to work with is the foundation for what we are about to accomplish here. The settings and practices we will discuss can only go so far with the hardware you have.
- Graphics cards - You want as much on-board memory as possible.
- One key word is RAM, RAM, RAM…get as much RAM as you can afford.
- Processors - Inventor is still a single-thread application as of the 2017 release, so it does not take advantage of multiple cores for its more typical processes.
- Monitors

Settings

Selecting these options and incorporating the practices we'll list here will make you more efficient in your everyday CAD work.

Settings

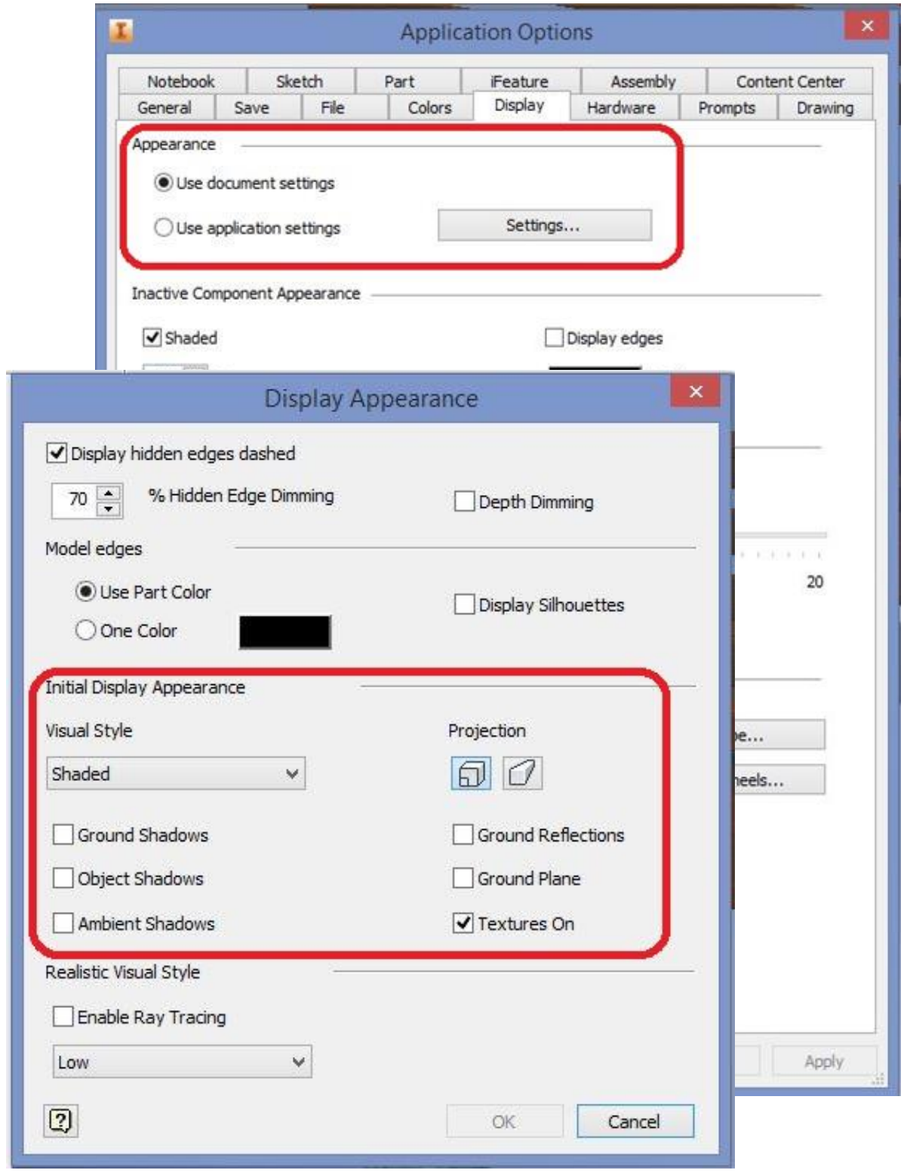
- General Tab:
  - ***Physical Properties***



Settings (Con't)

- Display Tab:
  - ***Appearance -*** Selecting the "Settings" button brings up the pop-up dialogue box to:
    - Set the Projection styles
    - Set Visual Styles
    - Shadows
    - Reflections
    - Ground Planes
    - Textures
    - Adjust hidden Edges
    - Adjust Model Edges

Note: Selecting some of these will actually defeat the purpose of what we are trying to accomplish here



Section #1B – Approach - How should you build your Assembly?

Building your Assembly

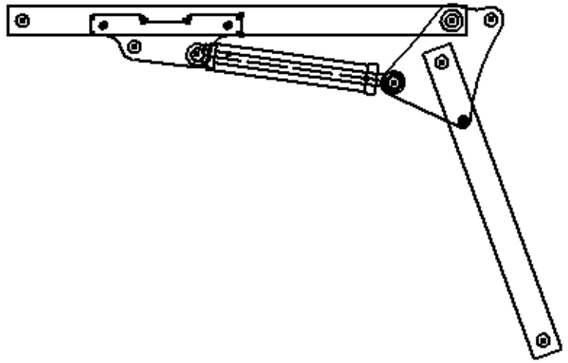
- ***Top-Down or Bottom-Up?***

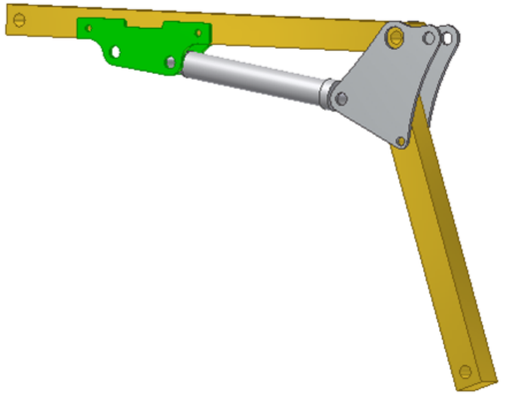
Inventor allows you to build your assemblies in either direction. Each has their Pros & Cons. The question is, which one is better for you? Let's look at both…

  - ***Top-Down***
  - ***Bottom-Up***
  - **Bonus: Middle-Out**

Building your Assembly

- ***The Top-Down Process (con't)***
  - Another process is called: Layout design.





  - Yet another approach in this fashion is Skeletal Modeling

Building your Assembly

- ***The Bottom-Up Process***
  - The Bottom-Up process, is where you build your individual parts and or sub assemblies and then insert them into the assembly file

Section #2 – Simplify an Assembly

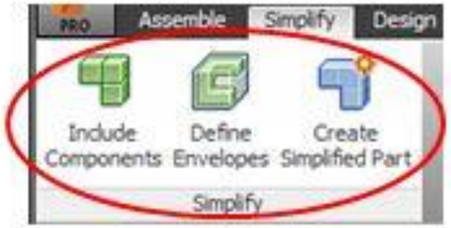
Simplifying your Assembly

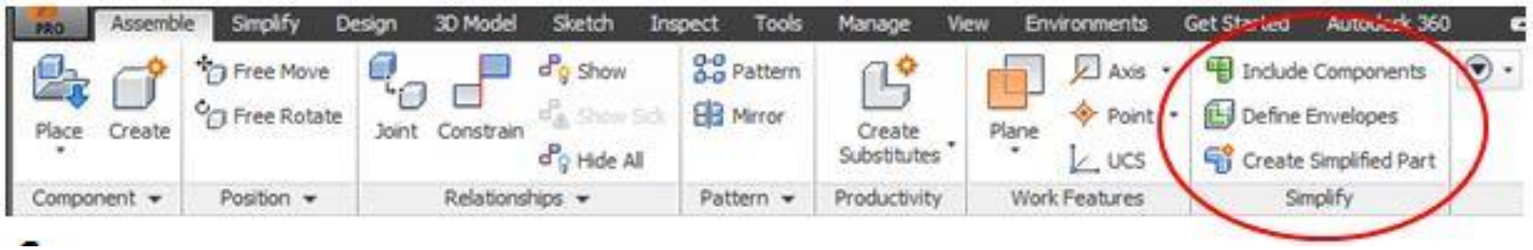
- ***Shrinkwrap***
  - Creates a simplified derived part of the original component
  - Shrinkwrap fills in the internals of your parts/assemblies, creating a solid interior, voiding out any sensitive or protected information from being seen.

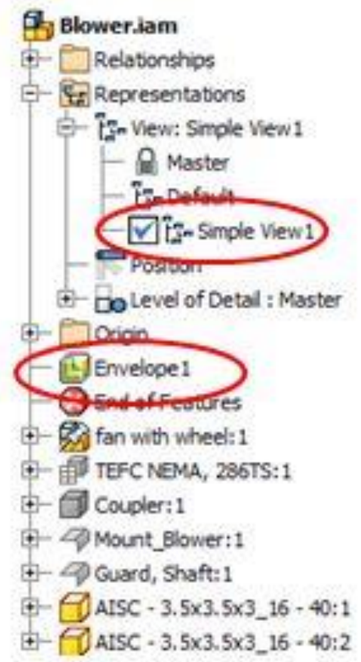
Shrinkwrap model sectioned to show solid interior.

Simplifying your Assembly

- ***The Simplify tools***
  - Besides Shrinkwrap, you also have a few tools that are provided specifically for assembly simplification purposes.

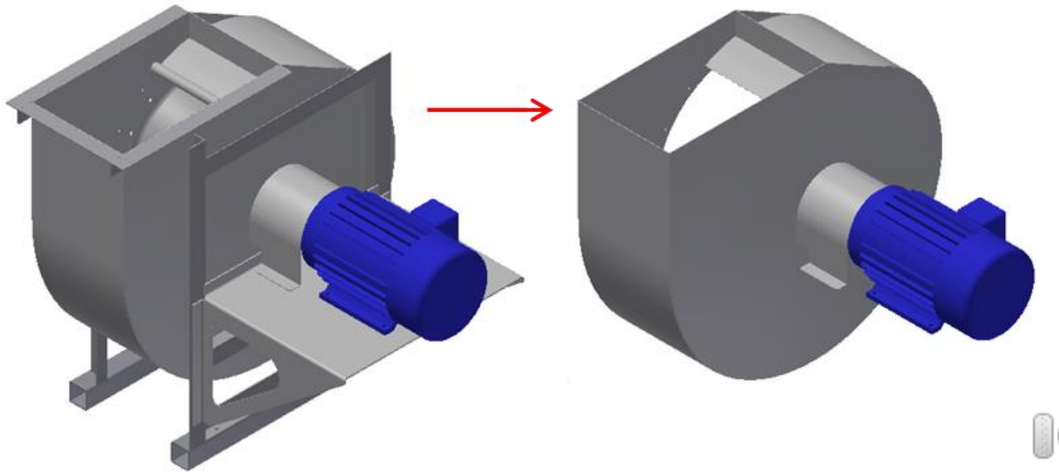


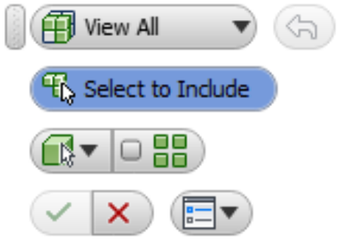


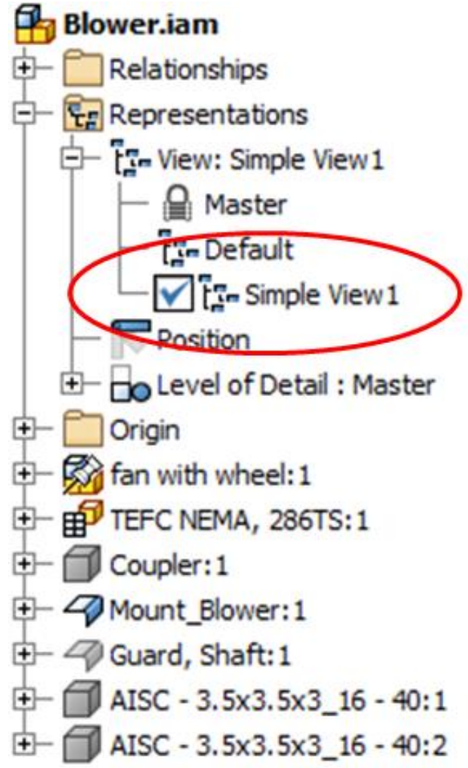


Simplifying your Assembly

- ***The Include Components tool:***

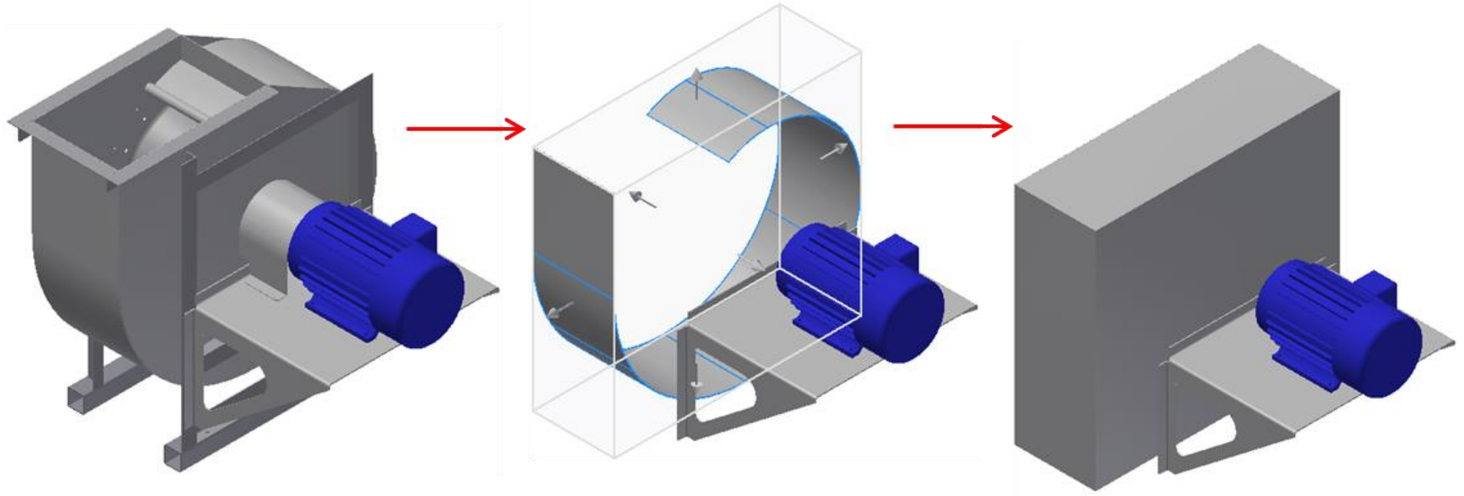


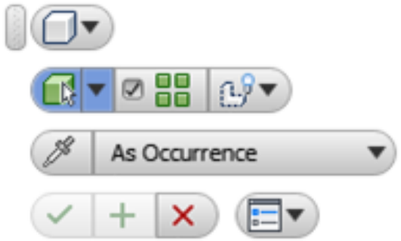


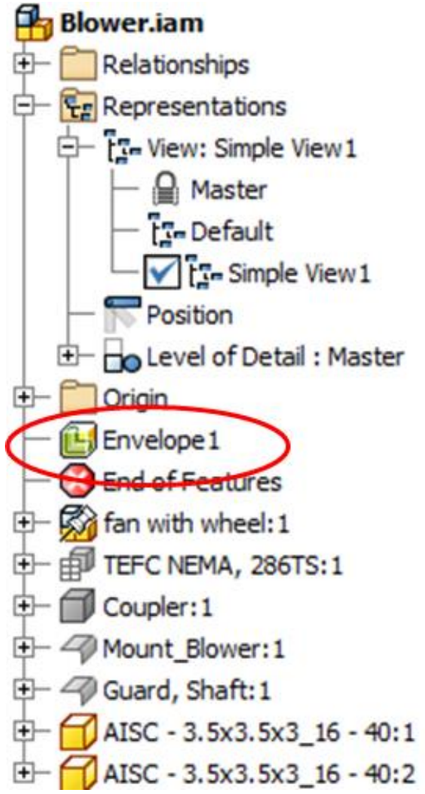


Simplifying your Assembly

- ***The Define Envelopes tool:***

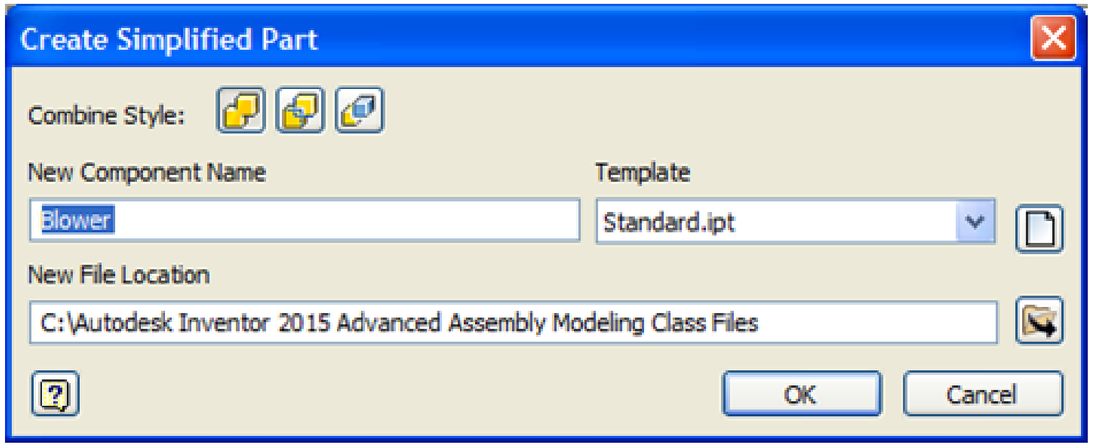


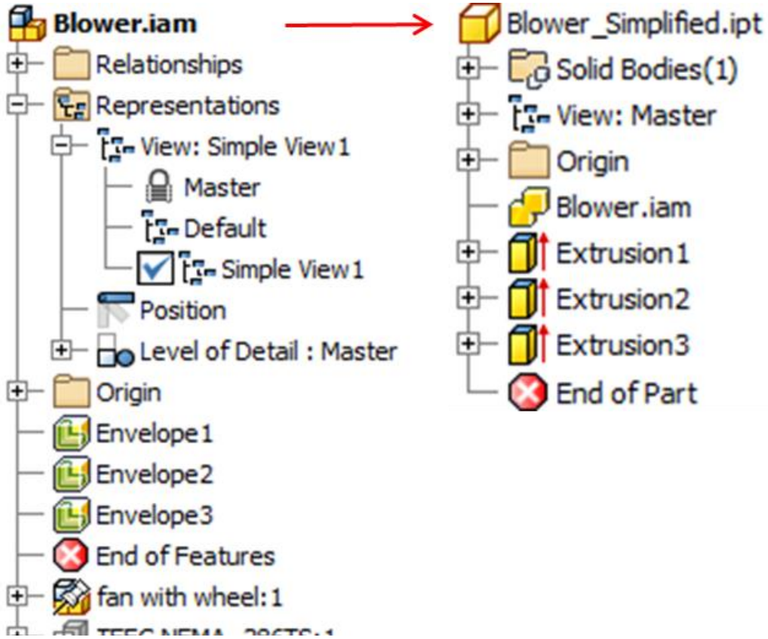




Simplifying your Assembly

- ***The Create Simplified Part tool:***

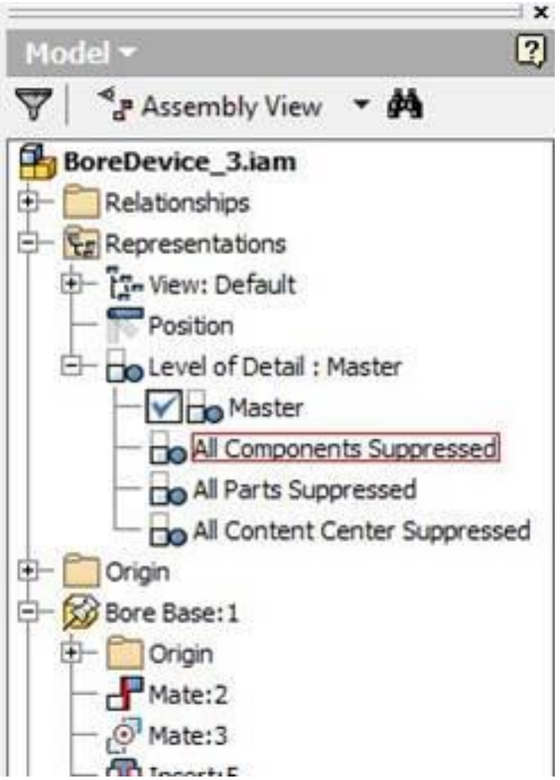




Level of Detail

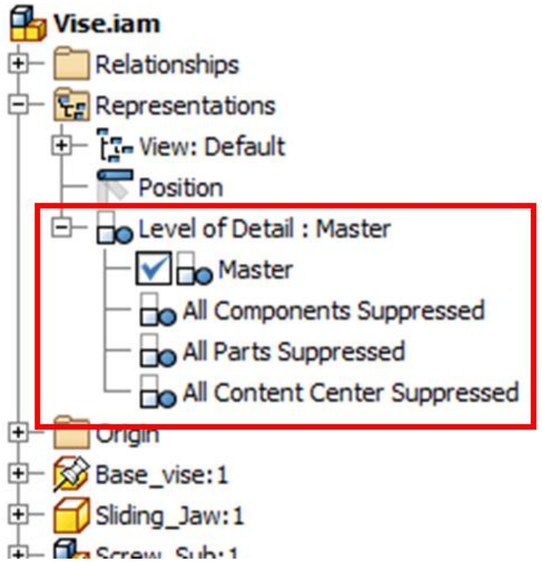
- ***Level of Details***

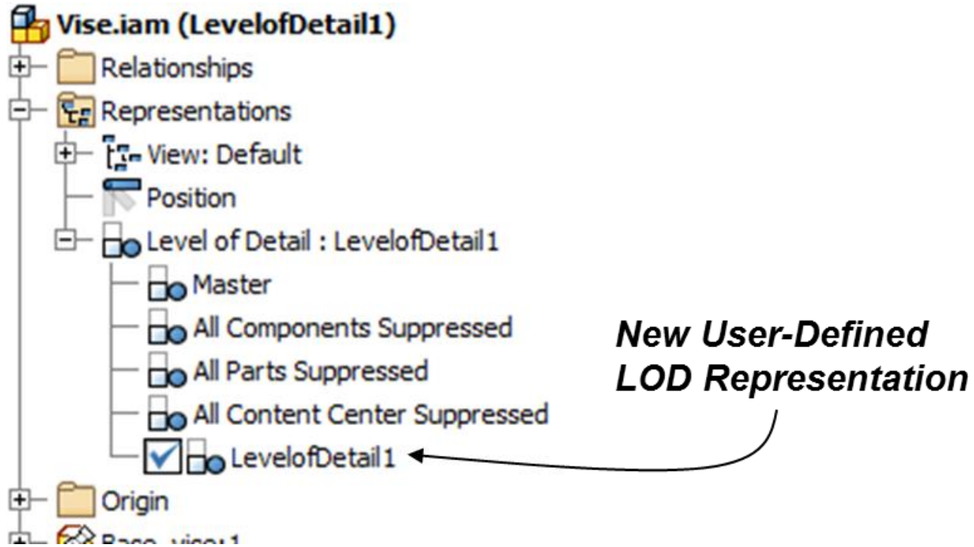
- In the Assembly browser expand the *Representations* folder and then expand the Level of Detail node, the default LODs are:
  - Master – No parts are suppressed
  - All Components Suppressed
  - All Parts Suppressed
  - All Content Center Suppressed



Level of Detail

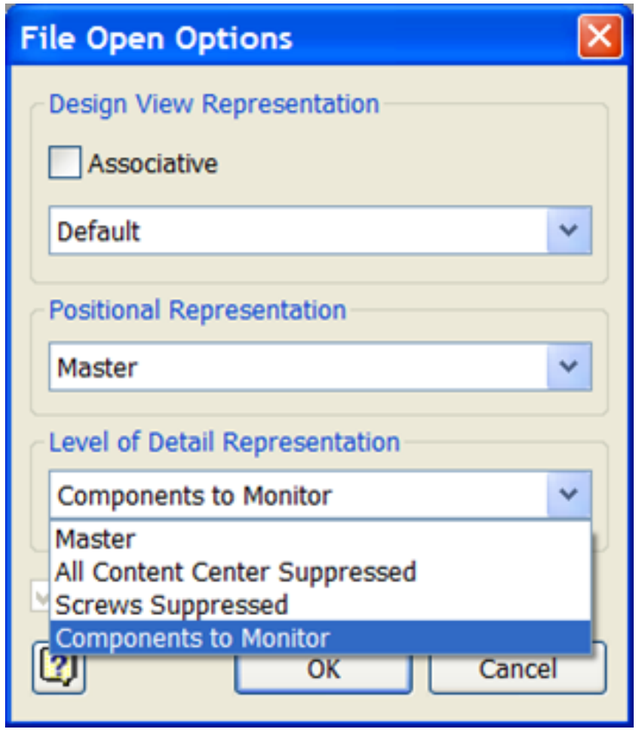
- ***Level of Details (Con't)***
  - Select either "*All Components Suppressed*" or "*All Parts Suppressed*" depending on your needs and save the assembly, check the file size.
  - You can also create your own LOD which specifies what level of parts you have active upon opening the file. You are not limited to the number nor the names of your LODs.





Level of Detail

- ***Level of Details (Con't)***
  - This can also be done at the open file dialog box if you rather operate that way



Level of Detail

- ***Level of Details (Con't)***
  - Some common issues
    - Thumbnail (Preview)
    - Insertion of blanked LOD assembly into upper level assembly
    - Benefits – Do they outweigh the drawbacks?

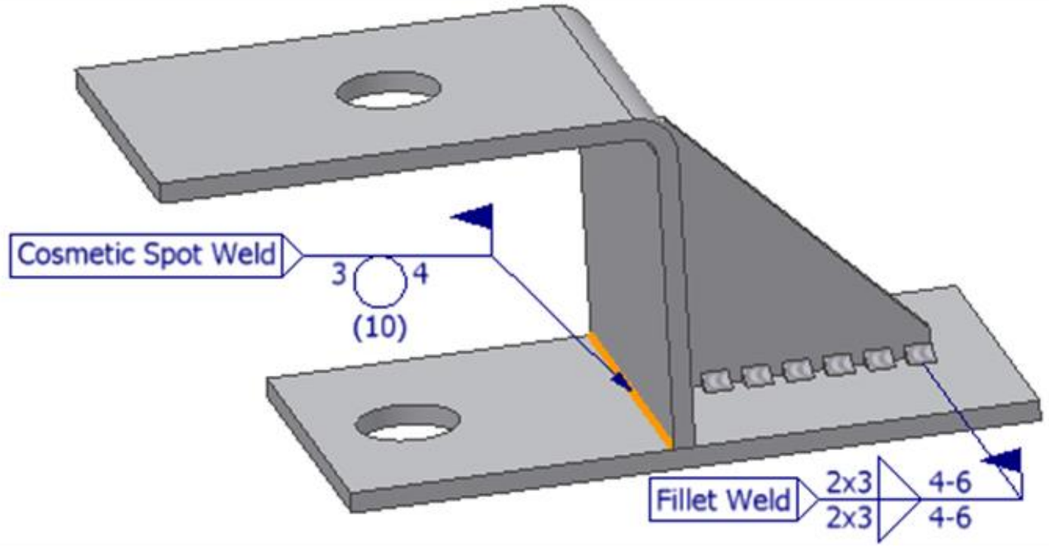
Section #3A – Advanced Features - Weldments

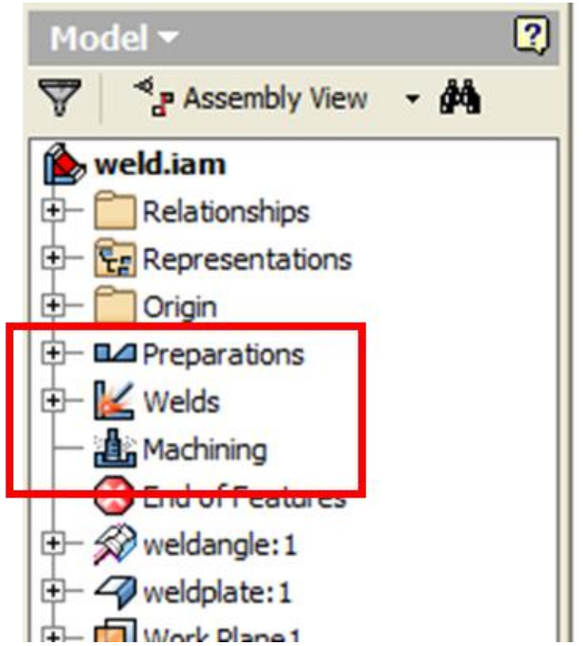
Working with Weldments

- A weldment in Inventor is a “specialized” assembly file much in the way that a Sheet Metal file is a “specialized” part file. Using the Weldment template provides you with weld specific tools to create your weldment file.
  - There are three main categories
    - **Preparations**
    - **Welds**
    - **Machining**
  - You can also convert an existing assembly to a weldment much in the way you can convert a part to a sheet metal part
  - To convert to a weldment assembly: Click  (Convert to Weldment) in *Assemble* tab>Convert panel

Working with Weldments

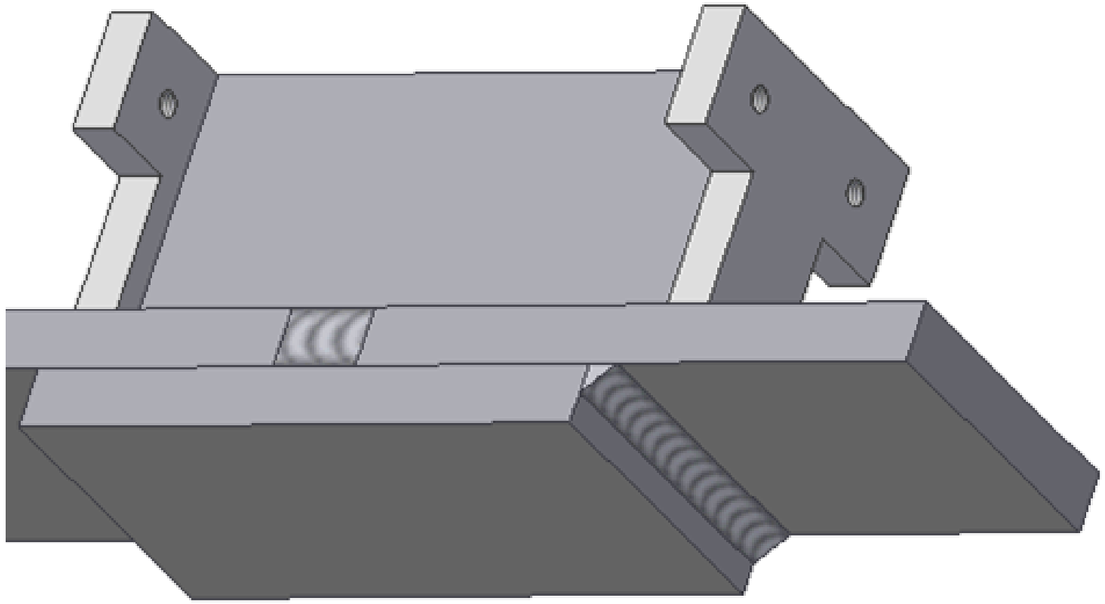
- Welds are features of the weldment assembly
- Weld symbols can also be shown in the weldment assembly as shown here:



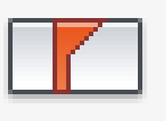


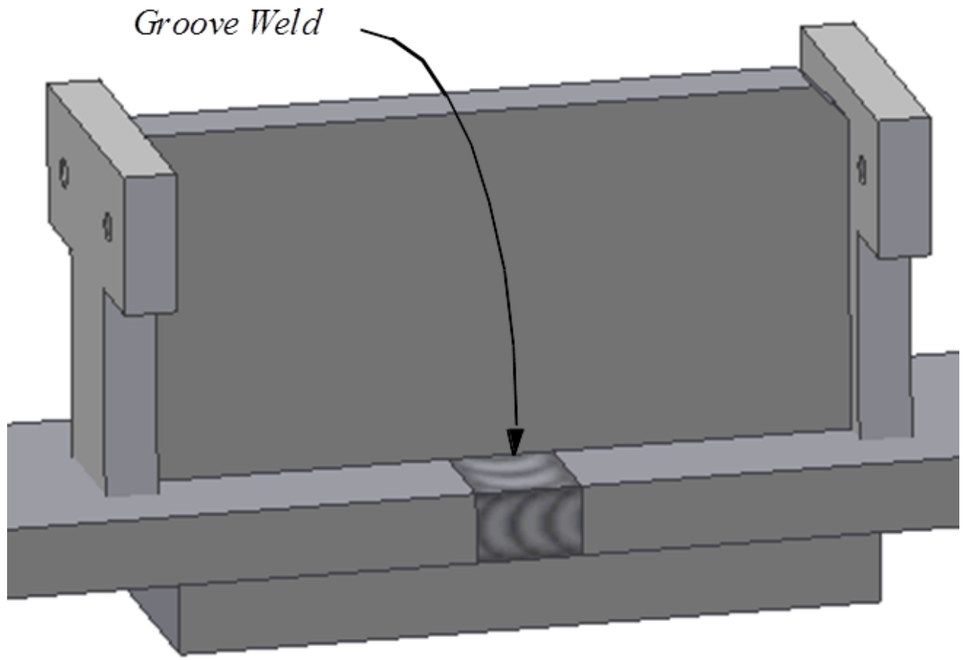
Working with Weldments

- **Welds -** There are three main types of welds:
  -  **Fillet** – Add material to model, are used where interference checking or Mass Properties are important



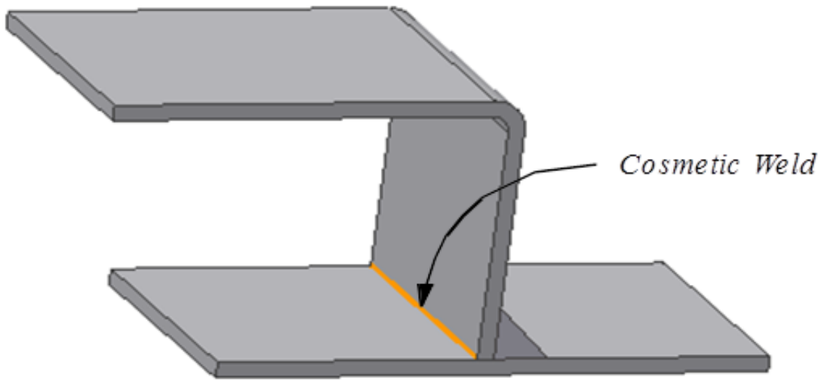
Working with Weldments

- **Welds**
  -  **Groove** – Add material to model, are used where interference checking or Mass Properties are important



Working with Weldments

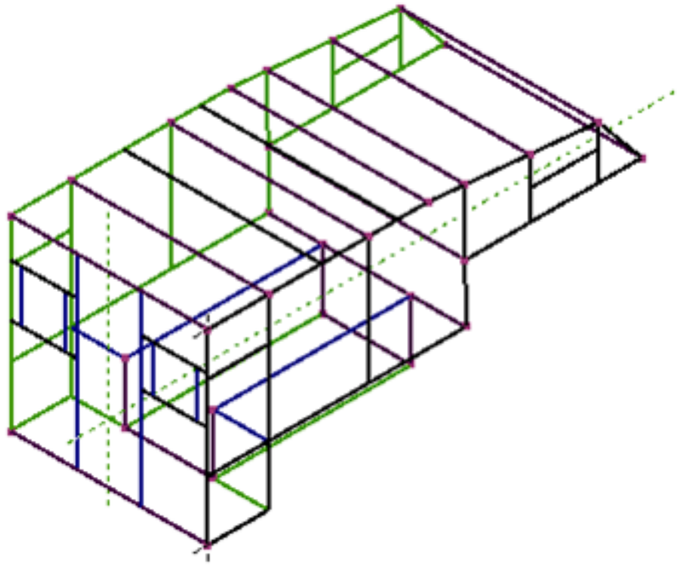
- **Welds**
  -  **Cosmetic** – Display as highlighted edges, not a bead in the model. They also do not reflect in Mass Properties of the file nor are they reflected in interference calculations unless the "**Include Cosmetic Welds**" property is set.

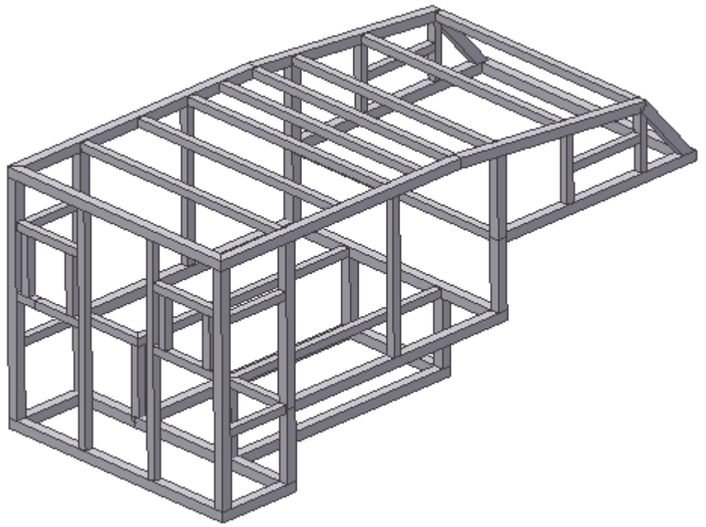


Section #3B – Advanced Features - Frame Generator

Frame Generator

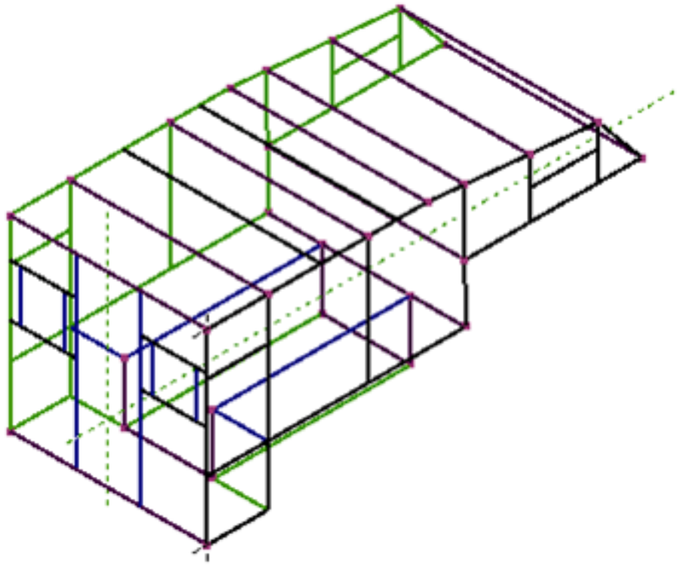
- Frame Generator allows you to quickly and easily create structural frames, using a skeletal wireframe part to define the location of the frame members
- Frame Generator also offers tools to create and adjust the required end joints





Frame Generator

- General Steps:
  1. Create a skeletal model by either 2D or 3D sketching
  2. Insert the skeletal model into an assembly
  3. Insert the frame members  (Design tab > Frame panel)
  4. Select the entities of the sketch to place the frame member
  5. Select the “end treatments” for your frame members



Section #4 – Tips & Tricks

Tips & Tricks (Best Practices)

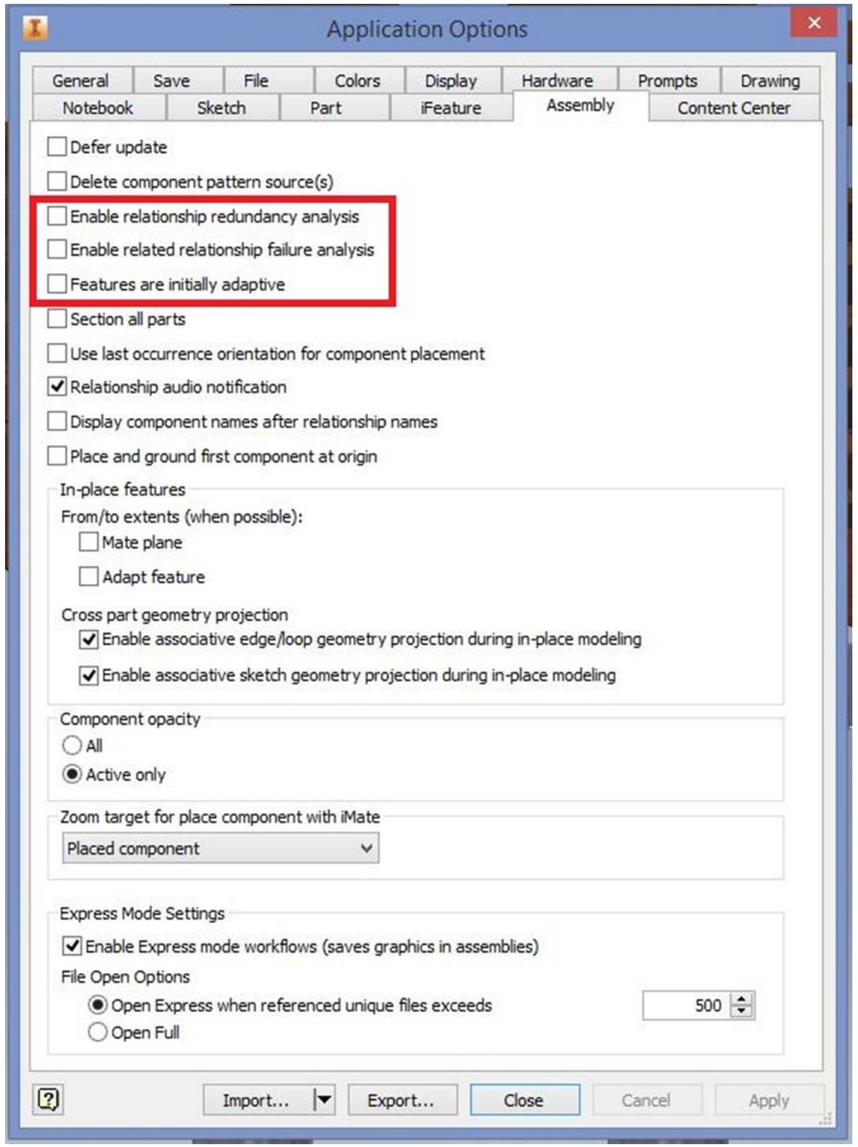
- ***Shutting Down*** - How often do you shut down not only the application, but your workstation?

Tips & Tricks (Con’t)

- ***Assembly Files***

When working in larger assemblies, open the ***Application Options*** dialog box and be sure to deselect the following options:

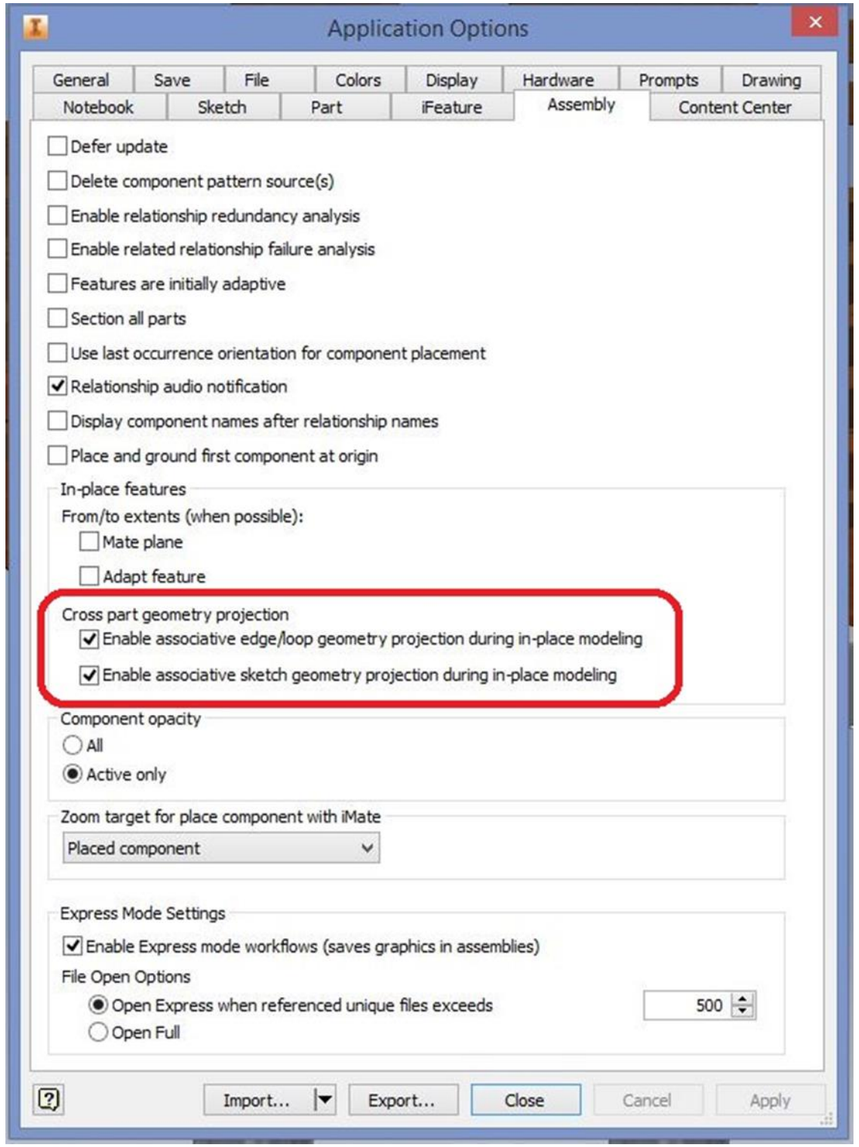
- *Enable Relationship Redundancy Analysis*
- *Enable Related Relationship Failure Analysis*
- *Features Are Initially Adaptive*



Tips & Tricks (Con’t)

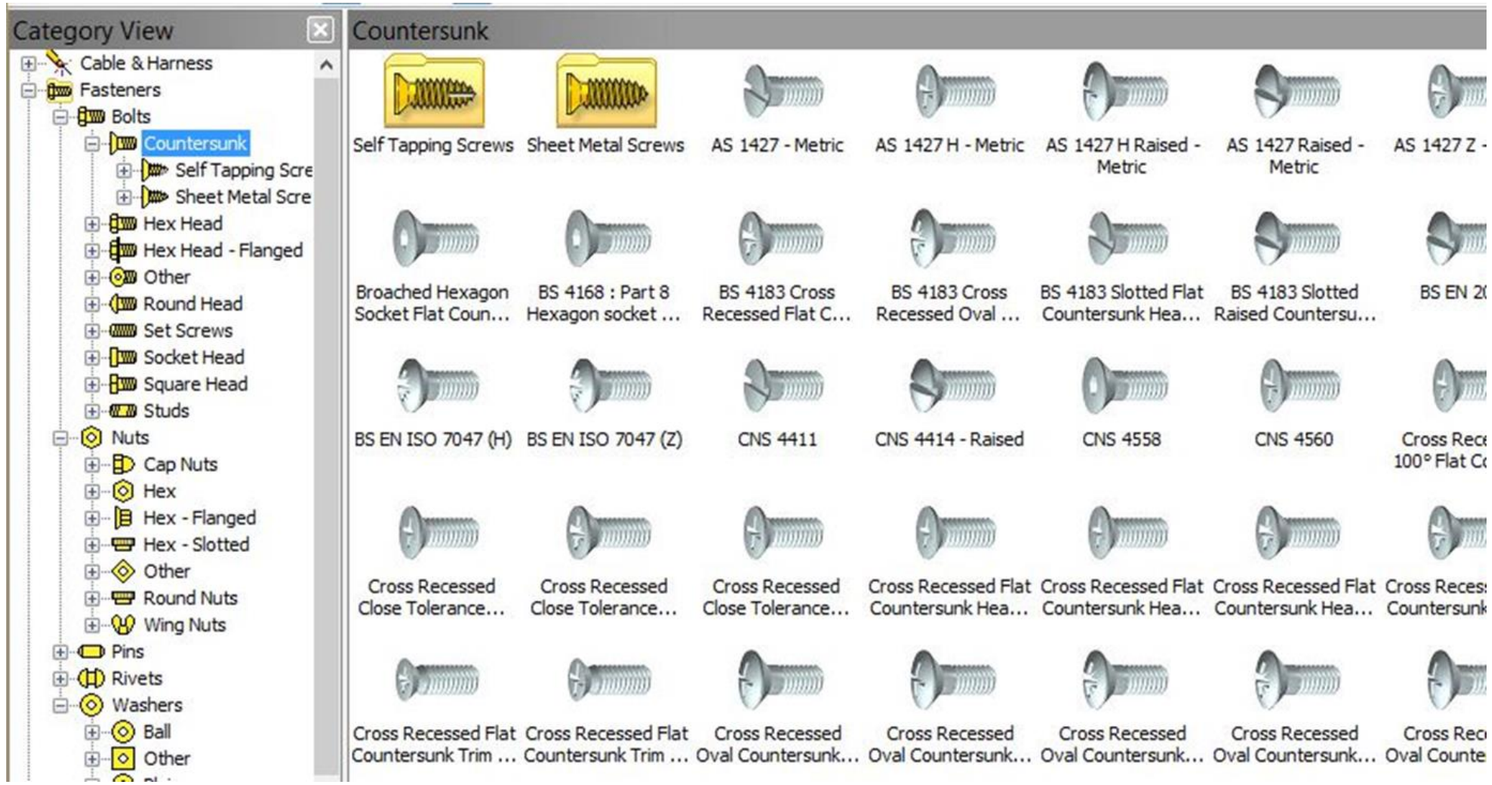
- ***Adaptivity***

- You can choose to disable this option in the **Application Options** dialog box, in the *In-Place Features* heading, de-select the check box for *Enable Associative Edge/Loop Geometry Projection During In-Place Modeling*.



Tips & Tricks (Con’t)

- ***Standard Parts***



Tips & Tricks (Con’t)

- ***Constraints***
  - Make constraints your friends
  - Always consider constraining to the default origin planes/axis/point

Tips & Tricks (Con’t)

- ***Defrag Your Files!***
  - How often do you “defrag” your CAD workstation? Once a day, week, month, year….ever? How about doing a “defrag” on your home PC? Ever consider “defraging” your part files? …what? You can do that?

Tips & Tricks (Con’t)

- Want to verify the Inventor application is still running when it seems to be “stuck”?

Access the ***Memory Probe Tool***. It is found in the ***Inventor/Bin*** folder as ***memprobe.exe***

Conclusion

- Use the tools and practices for the process you want to complete
- If you are noticing a lot of lag time when dealing with large or complex files, first adopt whatever settings described here that apply to your situation
- Use best practices and develop better modeling habits

Beyond the Basics

- ***Questions?***
- ***Comments?***
- *Be sure to fill out the class survey*

Thank You for coming!