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PD21883

Beyond Assembly Basics – Advanced Assembly Modeling with Inventor

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Learning Objectives

- Settings & Approach - Work more effectively in the assembly environment
- Simplify – Protecting your IP & Reducing File Size
- Advanced Features - Weldments and Frame Generator
- Free Stuff – Tips & Tricks, Best Practices

Description

Advanced Assembly Modeling – You know the basics, now let's look at some of the more advanced features of Assembly Modeling. We'll cover some tips and tricks, settings to help minimize file size of your assemblies, how to create assembly features such as weld preps and beads, creating frames and level of details

Your AU Expert(s)

Jim began his career as a co-op draftsman for a local manufacturer after winning numerous state scholastic awards for designs ranging from mechanical to architectural while still in high school. Now with more than 35 yrs. in the Mechanical Design field covering industries of Automotive, Power Generation, Industrial Design, Consumer Goods, Avionics and Nuclear Waste removal, Jim has worked his way through the ranks from Drafter to Designer to Applications Expert and is now the Engineering Manager at Versa-Gard, LLC. He has been on Autodesk's Inventor since Release 3 and as a CAD Administrator involved in Autodesk's Early Adapter program, managed the first commercial implementation of Autodesk's Vault WorkGroup. Jim was awarded Autodesk's Expert Elite status in 2014 and is an Autodesk Inventor Certified Professional and an Autodesk Certified Instructor as well. Jim was also a speaker of two classes at AU2015.

Section #1A – Settings – Working Effectively in Assemblies

Users that work with large, complicated part or assembly files waste vast amounts of time waiting for these files to generate and build upon activation. The causes for this can be anything from subpar hardware to default application settings that don't support your needs, to poor file maintenance practice. Making a few changes in the software settings and workflow practices can greatly reduce this wasted time and make the user more productive.

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. Inventor uses Direct3D as opposed to OpenGL like it did prior to the release of Inventor 2012. This has opened up the market for your selections to gaming cards instead of only CAD rated cards.

Memory

One key word is RAM, RAM, RAM…get as much RAM as you can afford. Once Inventor's process has exhausted all the memory of the graphics card, it then relies on your system's RAM to crunch all those mathematical equations that are needed to generate the surfaces and edges and textures, etc.

Processors

Inventor is still a single-thread application as of the 2017 release; it does not use multiple cores for its more typical processes. However, Inventor will utilize multiple cores for the more intense processes such as Task Scheduler, Ray Tracing and Stress Analysis.

Monitors

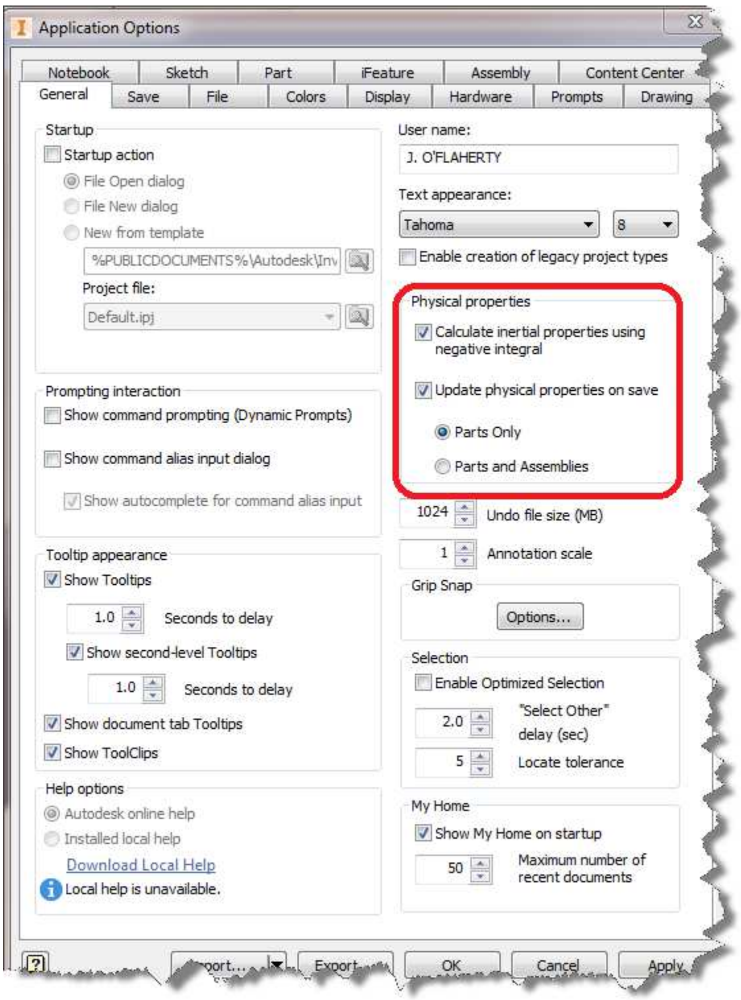
OK, a monitor won't help with load times, but having multiple monitors can really help you navigate between and around the applications. Typically two monitors will do the trick, have at least one that is a 16:9 ratio to use for CAD, use the other for MS applications. Be sure to have a graphics card dedicated to each monitor as opposed to one card split between the two. Why? You ask? If you have a 4 gig card and two monitors, the memory is divided equally to each, and thus the CAD monitor is now only using 2 gigs.

Settings – Application

Aside from the hardware you either purchase or have to work with currently, making some of the following Setting changes will make you more efficient in your everyday CAD work. Keep in mind, Application Settings are just that, settings you are making to the application, aka, Inventor. Users seem to get confused between Application Settings and Document Settings

General Tab – Physical Properties

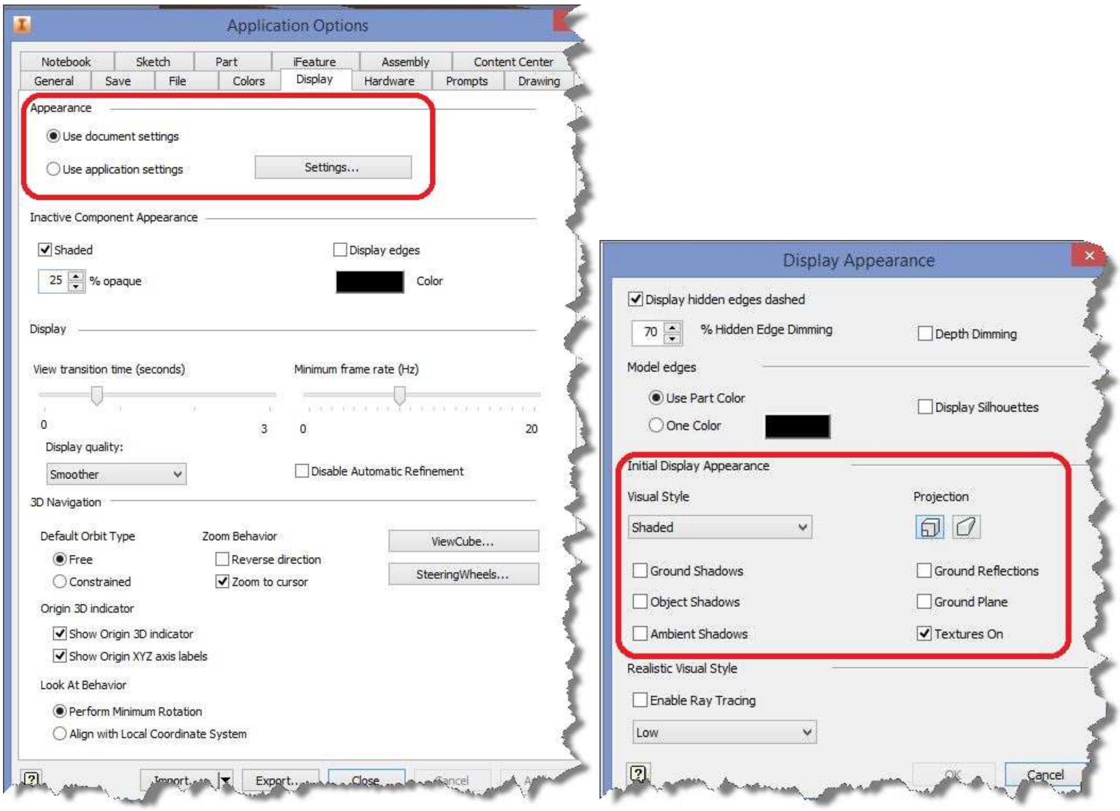
This setting saves/updates physical properties of the file each time you save the file. Setting this to “Parts Only” will make sure all your parts are up-to-date without forcing you to wait while a large assembly has its properties updated.



Display Tab - Appearance

The **Settings** button opens a pop-up menu. In this menu you can select/de-select the **Projection** of the part of either **Orthographic** or **Perspective** *(these same choices can be set in the View tab of the ribbon menu).* You can also set options such as **Ground** or **Object Shadows**, **Reflections**, **Planes**, and **Ray Tracing**, etc.

Note: Selecting these to "On" will actually slow the system down, the opposite of what we are attempting to accomplish here.



Display Tab – View Transition Time/Display Quality

These determine the amount of time your views will take to transition from their current state to the state requested in the command, such as Zoom All. Setting this to "zero" makes that transition instantly, setting it to its maximum of 3 seconds, you'll see it animate to the requested view state. Setting the Quality to "Rough" will speed up the regeneration process by simplifying the details of the file.

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

Unlike Inventor's competitors, you, the user, are allowed to build your assembly as you need it or as you see fit. You have the options to create the parts within the context of the assembly file, create the parts on their own, then insert them into the assembly file, or a combination of the two. No one knows your product better than you, so why be forced to create your models in the manner some programmer feels you should? But I digress. Each of the processes have their own Pros & Cons, knowing these will allow you to decide which approach is best for….YOU.

Top-Down Approach

This process is where you create all your individual part models within the Assembly file. The Top-Down process creates parts that are Adaptive, meaning they are directly tied into one another based on the references you chose to build them from. For example, you select a machined face of a casting as a reference to build the mating Gasket for that face. As the face changes (profile, location, etc.) so does the Gasket.

Pros/Cons

Since you are referencing features of other parts in the assembly, changes propagate to all affected components automatically, a great time saver and most likely what you want the design to do. But at the same time, you need to be aware of what components these updates will change. You may not want such to happen due to part numbering, etc. Consider the example above of the Gasket. Let's say the Gasket is set by its dimensions, if you need a bigger or smaller gasket, you need a new part and thus part number.

This approach, as you can imagine, takes a lot of forethought, planning and knowledge of your product and how it may change

Bottom-Up Approach

This process is where you first create your part files independently of the assembly file and then they are brought in and assembled within the assembly environment. No design criteria is established between the parts. Best practice for this would be to try and mimic the order of the actual assembly process in the shop. Think along the lines of an assembly line. Insert the base or foundational part/sub-assembly, then build upwards and outwards. Always constrain your base (first inserted) part to the assembly file's default origins. These can never be modified nor deleted, giving you a solid base to build on.

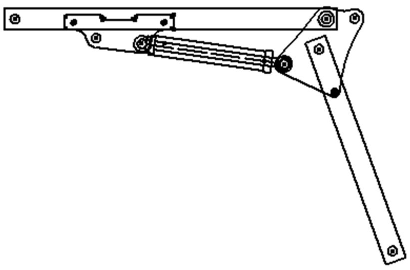
Pros/Cons

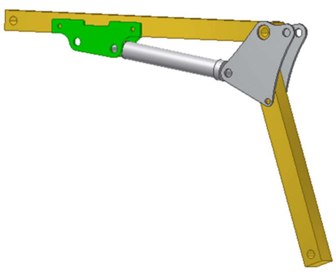
This approach, in my opinion, is probably the most common and frankly it's the one I find myself using most often, again, depending on what I'm designing. If you are working where numerous others are creating individual parts and all working concurrently on the same project, this will be the approach you'll be more apt to using. You'll have to build in any possible design criteria in a more manual form by way of constraints or joints, etc., as stated above, no design criteria is established between parts for automatic updating, etc.

Middle-Out Approach
This process is where you combine both the Top-Down and the Bottom-Up approach. This is where you are creating parts within the assembly but also need to add existing parts to that assembly.

Pros/Cons
Probably the best of both worlds, reference the parts you want to establish design criteria into and constrain the parts you don't want such for. Consider the Gasket example once again, you know this type of part is driven by its size requirements, thus create the gasket outside of the assembly model and insert it accordingly with assembly constraints. When or if a redesign to the forging happens, you simply swap out one gasket for the other.

Top-Down Approach – Extra #1
Getting back to the Top-Down Approach, there are a few different processes to this approach other than the one stated above. For example, *Layout Design* is where you sketch up the basic outlines and features of each component in 2D. This makes the sketch the main entity for your overall dimensions, spatial locations and the basic shape of the components. You then use these sketched outlines to generate your 3D models, in turn, making them associative/adaptive





Top-Down Approach – Extra #2
Another process of Top-down is a more generic or stripped down version of the *Layout Design* process, *Skeletal Modeling*. Here, you sketch out the basic outline of your components or the assembly's envelope in 2D and use it to constrain your components. As the Skeletal sketch changes, so do your associated parts.

Section #2 – Simplify an Assembly

There may be any number of reasons you want or need to simplify your part or assembly models. Some of the more common reasons are to protect company Intellectual Property (IP) or to minimize file size for emailing/sharing, or saving disk space or faster load times…the list goes on.

Shrinkwrap

Shrinkwrap has been an Inventor offering for some time now. Shrinkwarp basically takes your assembly model and fills in all voids, leaving a solid mass thus protecting any internal IP. A key issue for parts you send outside for quotes, reference, etc. Shrinkwrap can also be used on very large assemblies to minimize file size (by virtue of removing the unnecessary details).

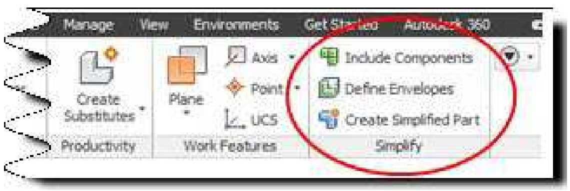
You can also Shrinkwrap parts and sub-assemblies and use them as "placeholders" in your larger or upper level assemblies.

Shrinkwrap can be used on in-coming supplier files. Retain only the critical features that you'll need to work with or around and "fill in" the non-critical features, minimizing file size and in turn, increasing performance, especially in large assemblies.

Shrinkwrap model sectioned to show solid interior.

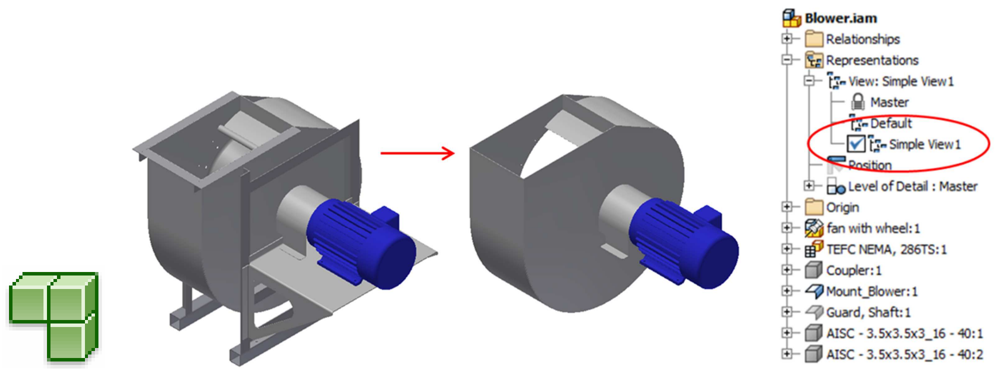
Simplify Tools

Besides Shrinkwrap, you also have a few tools that are provided specifically for assembly simplification purposes. These are found on the Simplify tab of the ribbon menu or on the Assemble tab in the Simplify menu.



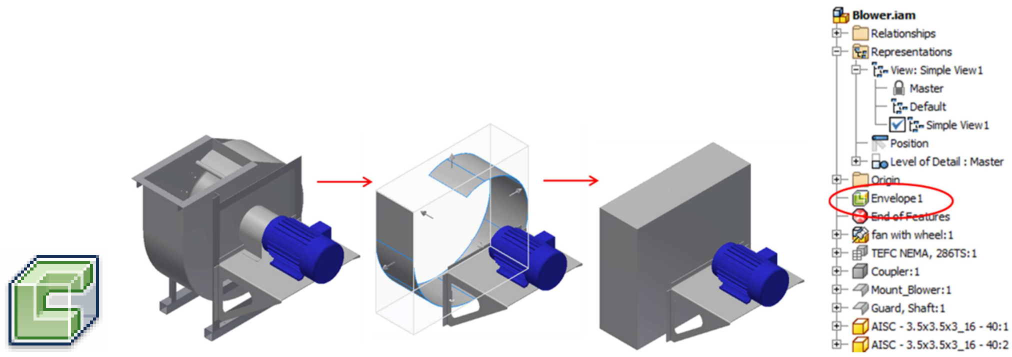
Include Components

The *Include Components* tool allows you to create a simplified version of an assembly that includes or excludes selected components. This creates a “view” under the Representations folder in the assembly browser.



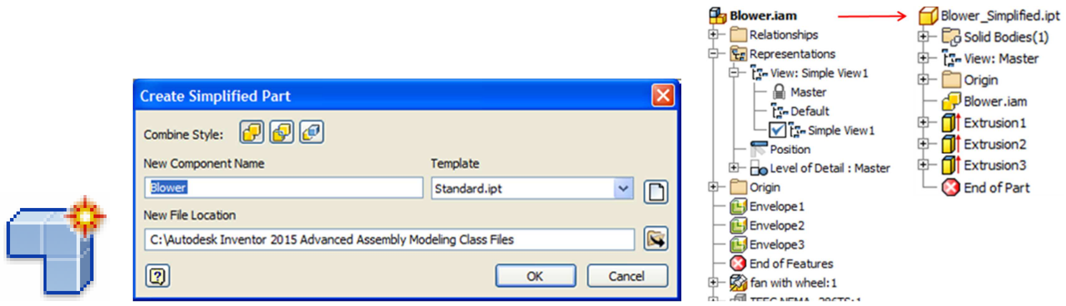
Define Envelopes

The *Define Envelopes* tool allows you to replace the selected component with a bounding box or cylinder that replaces the component. This command creates a “Envelope” feature to the assembly browser.



Create Simplified Part

The *Create Simplified Part* tool allows you to create a new, simplified component. This new component incorporates the active simplified view, envelopes and any component's visibility that has been disabled. This command creates a new part (.ipt) file of the assembly.



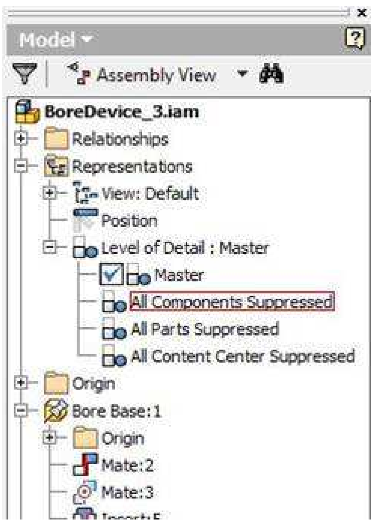
Level of Detail

Opening a large assembly can be a very time consuming task, especially if you have numerous assemblies to open. This doesn't need to be the case. Like using the part file's End of Part (EOP) marker, which reduces the file size to a minimum, utilizing the *All Parts Suppressed* level of detail will accomplish the same results on an assembly.

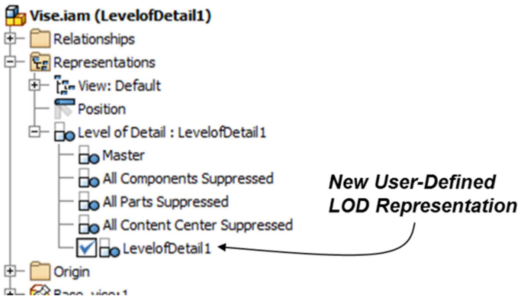
Process:

In the Assembly browser, expand the *Representations* folder and the expand the *Level of Detail* node, the default LOD's are:

- Master – No parts are suppressed
- All Components Suppressed – Everything is suppressed
- All Parts Suppressed – All parts at all levels are suppressed
- All Content Center Suppressed – All Content Center parts are suppressed

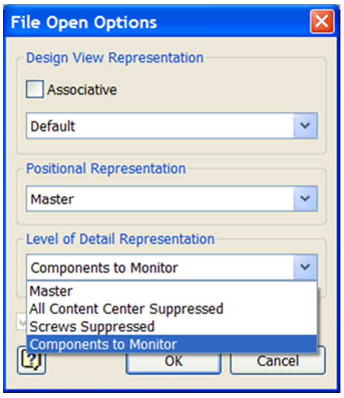


Where these default LOD's are close to "all or nothing", you can also create your own LOD which specifies what level of parts you have active upon opening the file. You are not limited to the number of LOD's you create.



You are able to specify which LOD you open at the file open state, not just while you're in the file itself.

In the Open File dialogue box, select the "options…" button, in the "File Open Options" dialogue box, select the desired LOD to open in the "Level of Detail Representation" pull-down menu.



Considerations:

If you are going to use the LOD process for file size and loading benefits, there are some issues or drawbacks to keep in mind. You'll need to consider if the Pros outweigh the Cons.

One of the most common issues that'll you'll see right away in the thumbnail icon. If you scan/search for your files by looking at this icon image, the LOD process will hinder/eliminate that ability, depending on which LOD was active when the file was last saved.

When in the process of inserting an assembly with the LOD process such as All Parts Suppressed into another assembly file, you'll have to be sure to set the Options to Master LOD prior to insertion, or activate the Master LOD immediately after insertion – otherwise, you won't see the assembly components you just inserted.

Section #3A – Advanced Features - Weldments

A Weldment is a "specialized" assembly file in Inventor much in the way a Sheet Metal file is a "specialized" part file. Using the Weldment template provides you with weld specific tools to create your weldment file.

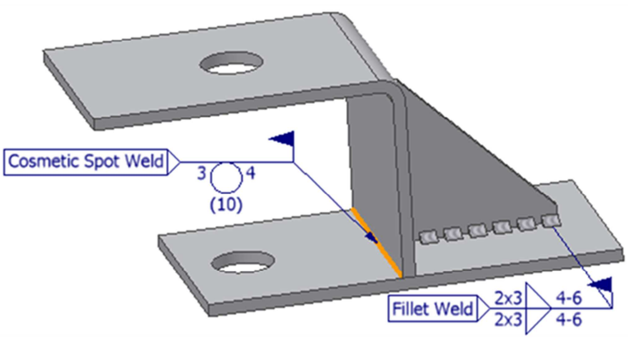
Weldment Steps:

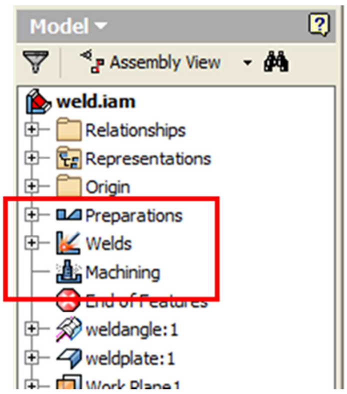
There are three main categories

- **Preparations** – These tools are what you use to create weld preps or material removal on the parts prior to welding them together
- **Welds** – Tools used to depict the joining of the parts of the welding process. This is where you'll see the weld beads added to the assembly model.
- **Machining** – Tools that allow you to depict any machining processes done post weld

Inventor also allows you the option to convert an existing assembly to a weldment much in the way you can convert a part into a sheet metal part. Use the *Convert to Weldment* icon found on the Assemble tab on the Convert panel. 

Welds are features of the assembly or weldment file, thus they add material to the assembly file (with the exception of *Cosmetic*) and are edited as a feature within the assembly file. Weld symbols can also be shown in the weldment assembly model as shown here:

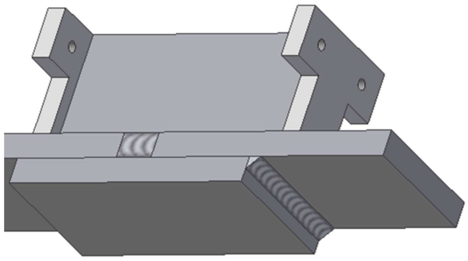




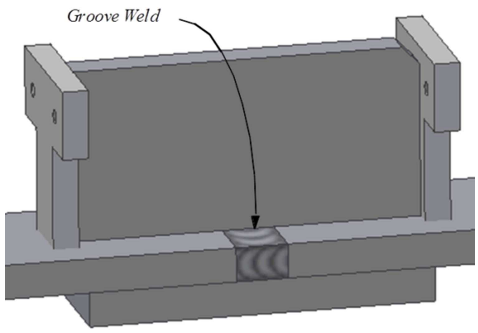
Weldment Types:

There are three main types of welds:

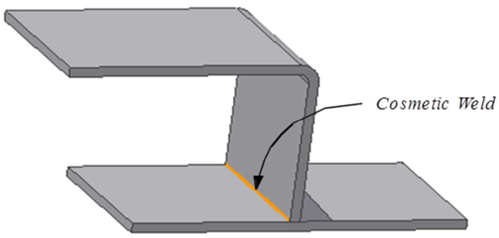
- **Fillet** – Adds material to the model, used where interference checking or Mass Properties are important.



- **Groove** – Adds material to the model, used where interference checking or Mass Properties are important.



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



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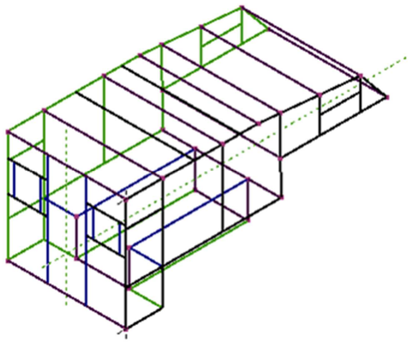
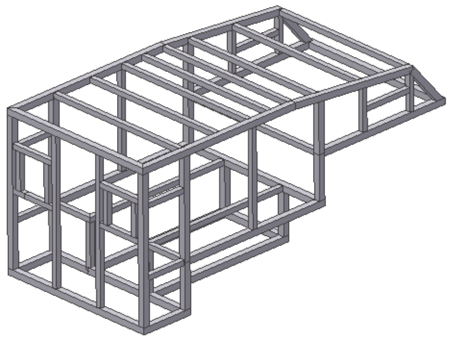
Section #3B – Advanced Features – Frame Generator

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

Frame Generator General Steps:

1. Create a skeletal model by either 2D or 3D sketching
2. Insert the skeletal model into an assembly file
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 – Free Stuff – Tips & Tricks (Best practices)

All too often, the moment users think about applying best practices is after they’ve been impacted in some way while working in a large file. Setting these options beforehand will make your files much easier to open and work with, along with making them more stable and less prone to data corruption. Convincing everyone else in your department to follow these same settings and practices will vastly improve everyone’s experience

Making small changes at the Part level, making the model more stable and keeping the file as small as possible, lends itself to more stable and smaller assembly files. This approach becomes very relevant when you have to create a large assembly.

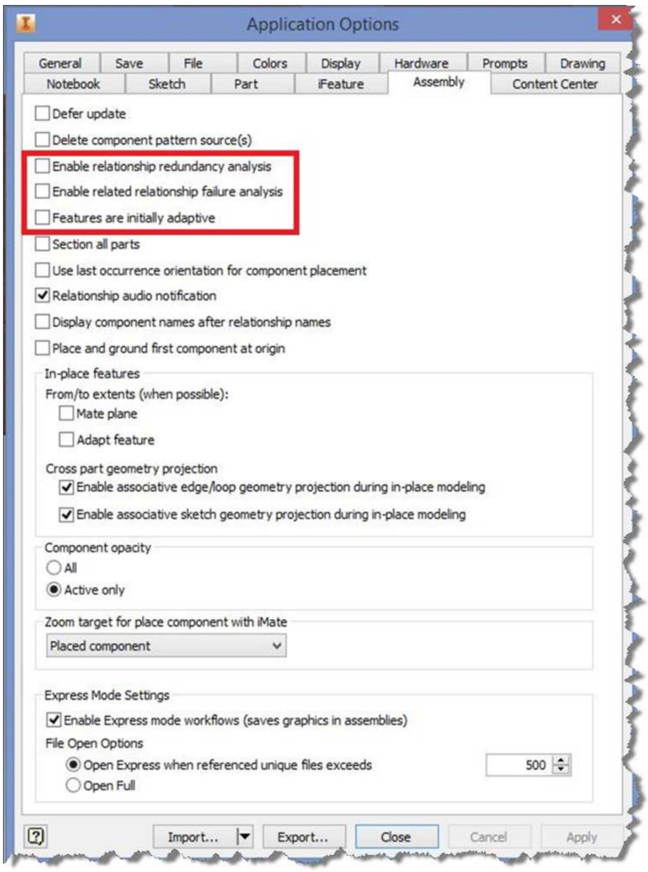
- ***Shutting Down*** - How often do you shut down, not only the application, but your workstation? I’ve seen numerous people who, at the end of the work day or even the week, simply get up and walk away and go home. Aside from the obvious security concerns of leaving your workstation unattended and open for anyone, shutting down the workstation or at the very least, the application allows the system to release “leaked” memory. There are numerous subroutines and drivers that grab this memory but do not release it once finished, shutting down will clear this reserved memory.

- ***Assembly Files***

You can also speed up load times for assemblies by limiting or minimizing the number of constraints you use. Build upper level assemblies by means of sub-assemblies. This eliminates or minimizes the number of redundant calculations Inventor has to solve and complete for that upper level assembly.

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

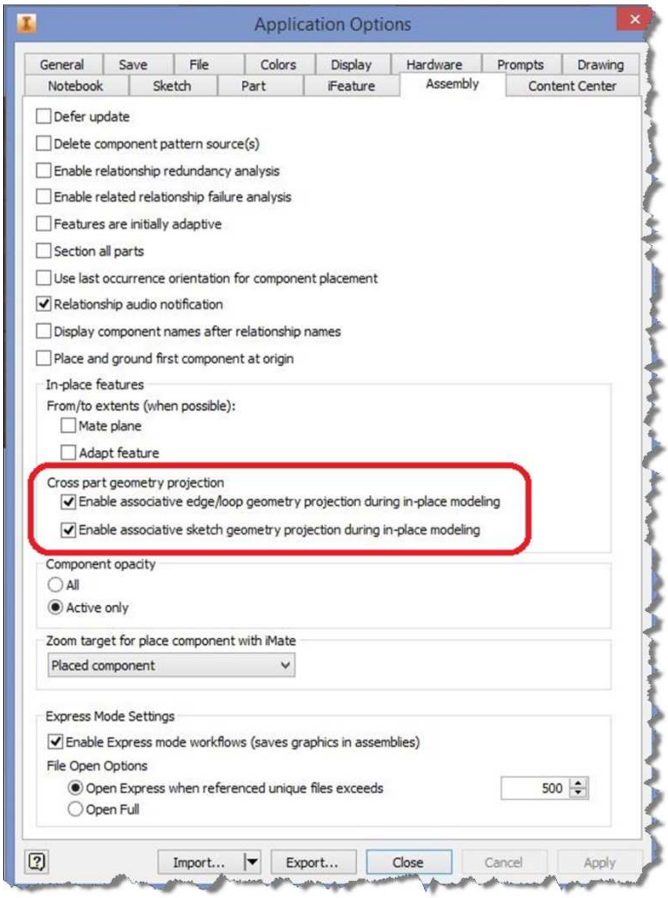
- *Enable Relationship Redundancy Analysis*
- *Enable Related Relationship Failure Analysis*
- *Features Are Initially Adaptive*



- *Adaptivity* – As described earlier, adaptive parts are created within the assembly when you reference an edge or feature of a mating part. This is a great feature to use on parts that you want to update when their mating part changes and can be very useful early in the design process when changes are happening quickly. But at the same time these adaptive parts can bring an assembly to its knees when used extensively.

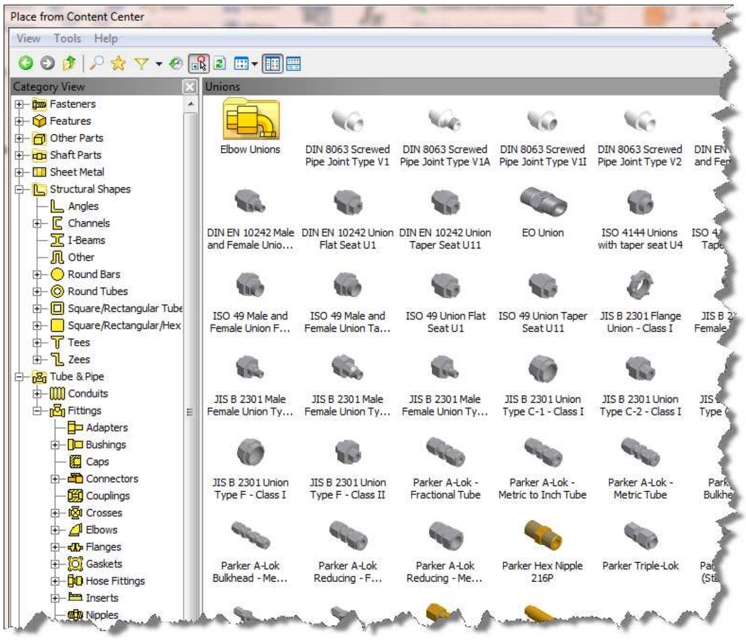
- You can choose to disable this option in the **Application Options** dialog box, in the *In-Place Features* heading, de-select the check box for *Enable Associative Edge/Loop Geometry Projection During In-Place Modeling*.

- Another option to disable this on the fly is to right click the part in the assembly browser and de-select "*Adaptive*" in the fly-out menu.



- ***Standard Parts***

  Use the standard parts in Content Center whenever possible, stop re-inventing the wheel. Content Center has 10's of thousands of industry standard parts, not just fasteners, use them where you can**.**



- ***Defrag your Files***

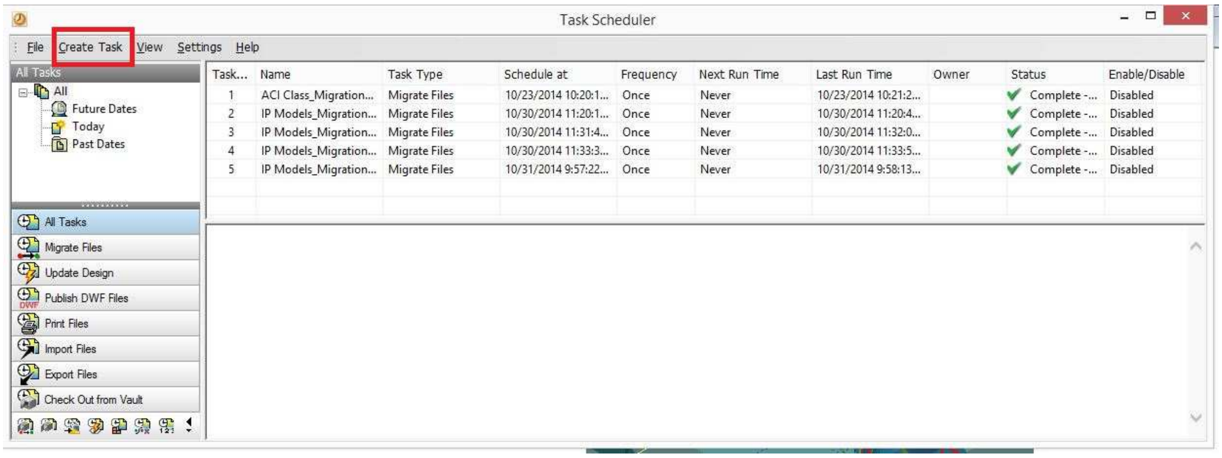
You know (or should know) you can defragment your HD, but what about your CAD files? Well, you can and should. When was the last time you ran the defrag process on your HD? For those of you who are not familiar with this process, running the defragmentation process on your HD scans and finds all the fragments of the files and applications scattered throughout the HD and organizes them into complete sections, thus preventing your system from having to search for each fragment in order to run the application or access the file. If you have run this process, you know how it can increase the speed of your PC, now think about the same deal, but for your CAD files.

As we all know our typical day of designing parts always involves someone, typically an Engineer who likes to change his mind more often than…well, you know. How many times have you made a change to your CAD files during a typical design process? How many features have you deleted? All these iterations end up being fragments of that part.

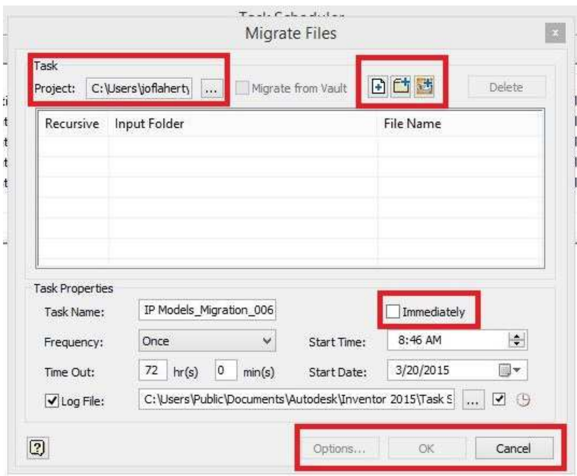
So, what is this magic elixir? Inventor's Task Scheduler application. Once you open the application, you'll notice a number of different uses it offers from Migrate, Publish, Print, Import, Export, etc., for this "trick", we will focus on the Migrate task. The Migrate task will take the file(s) and run it through the cleanup and compact options you select, and save the file.

How to use this application:

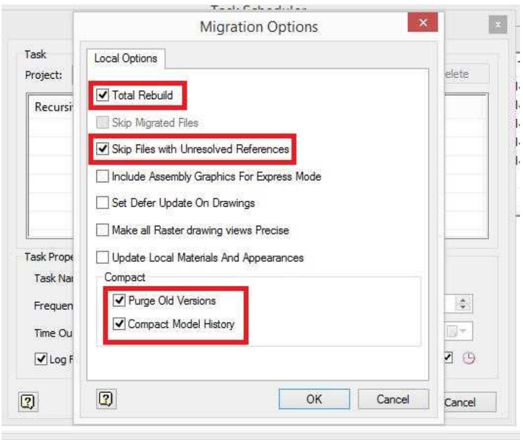
- Save and close all files open in Inventor
- Close down the Inventor application
- Open **Task Scheduler** (depending on the version of Windows you are using, it can be found under:
  - For MS Windows prior to Win 7 – Programs>Autodesk>Inventor 20xx
  - For MS Windows 7 and newer – it is found on the Apps page under the current Inventor release heading
  - I recommend saving it as a shortcut to your desktop
- In the **Task Scheduler** dialogue box
  - Select **Create Task** from the top menu
  - Select **Migrate Files** from the fly out menu



- In the **Migrate Files** menu
  - Select the **Project** (if not defaulted)
  - Select the **Add Files, Add Folder** or **Add Project** tab



- In the **Task Properties** section of the menu, set the **Date** and **Time** to run the process or select the **Immediately** box
- Select the **Options** button



- In the **Local Options** tab, select the check boxes of:
  - Total Rebuild
  - Skip Files with Unresolved References
  - Purge Old Versions
  - Compact Model History
- Select **OK**
- Select **OK** again

A progress bar will open showing the Percentage Complete, the Successful Files and the amount of Failed Files. Once complete, you'll be provided a link of the job log that will list any info or errors generated in the process.

- *Some information provided here was referenced from the Autodesk Inventor on-line Help files and my Imaginit Technologies White papers titled "Working with Large Files in Autodesk Inventor" & "How to Defrag your Files in Autodesk Inventor".*

Notes: