

MFG318699

## **CADZilla Returns!!! – Fighting (and Winning) with Large Assemblies in Inventor**

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### **Learning Objectives**

- Learn how to minimize the impact of large files
- Learn how to work effectively with large assemblies
- Learn best practices for large assemblies
- Learn the Differences between Level of Detail and Design Views and when to use them.

### **Description**

Do you work with large or very large assemblies? Do you have long load times of the model and drawing files? The speaker’s department deals with massively large assemblies that require them to use most, if not all, of the subject matter this class will cover. This class will build on and update the speaker’s Autodesk University 2015 & 2018 classes: "CADZilla—Taming Large Assemblies." and “CADZilla Jr – Effectively Building and Detailing Large Assemblies in Inventor”. We will visit best practices, tools, and tips for dealing with large assemblies, from how to save, activate, and detail these monsters.

### **Speaker**

Jim has almost 40 years in the mechanical design field covering industries of automotive, power generation, industrial machinery, consumer goods, avionics, and hazardous waste removal. Jim has been using Inventor software since Release 3 and was the first commercial implementation of Vault Workgroup software. Awarded Autodesk, Inc.'s, Expert Elite status since 2014, an Autodesk Inventor Certified Professional and an Autodesk Certified Instructor. Jim has also been a speaker at Autodesk University 2015, 2016, 2017, 2018 & 2019 as well as a Speaker Mentor for AU2017, AU2018 & AU2019.

## Section #1 – Hardware & Application Settings – Minimize the Impact

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, poor user practices, or even 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 in this class. 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 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*

Rumor has it that Inventor doesn't use multi-core processors, this is total bunk. Granted, Inventor does still use a single core for most of its lesser intensive 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.

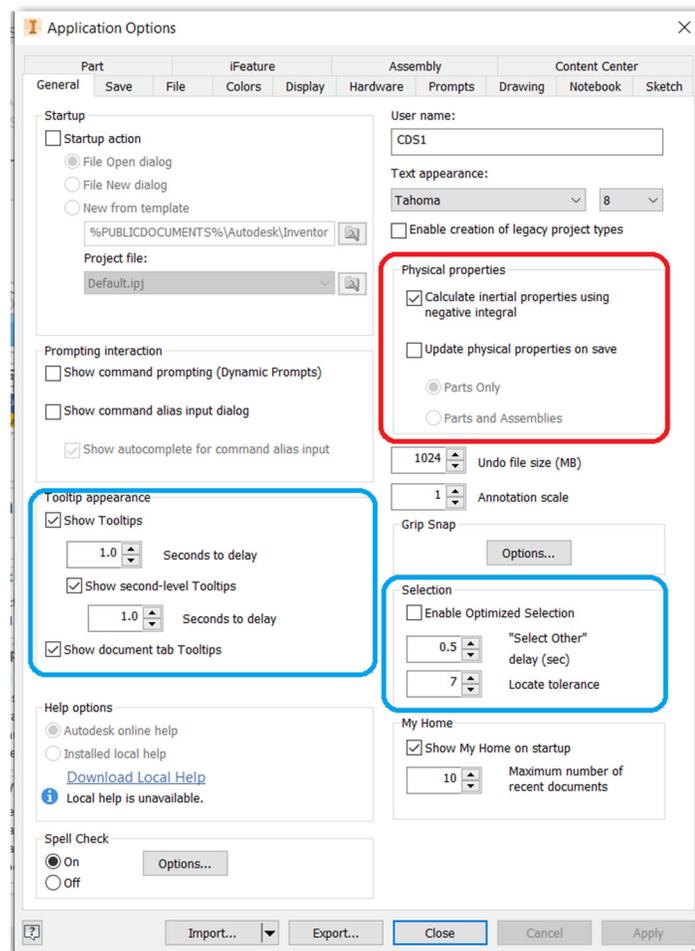
**Bonus: Inventor 2020 release allows you to use a 2<sup>nd</sup> monitor for the application.**

## Settings – Application Options

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.

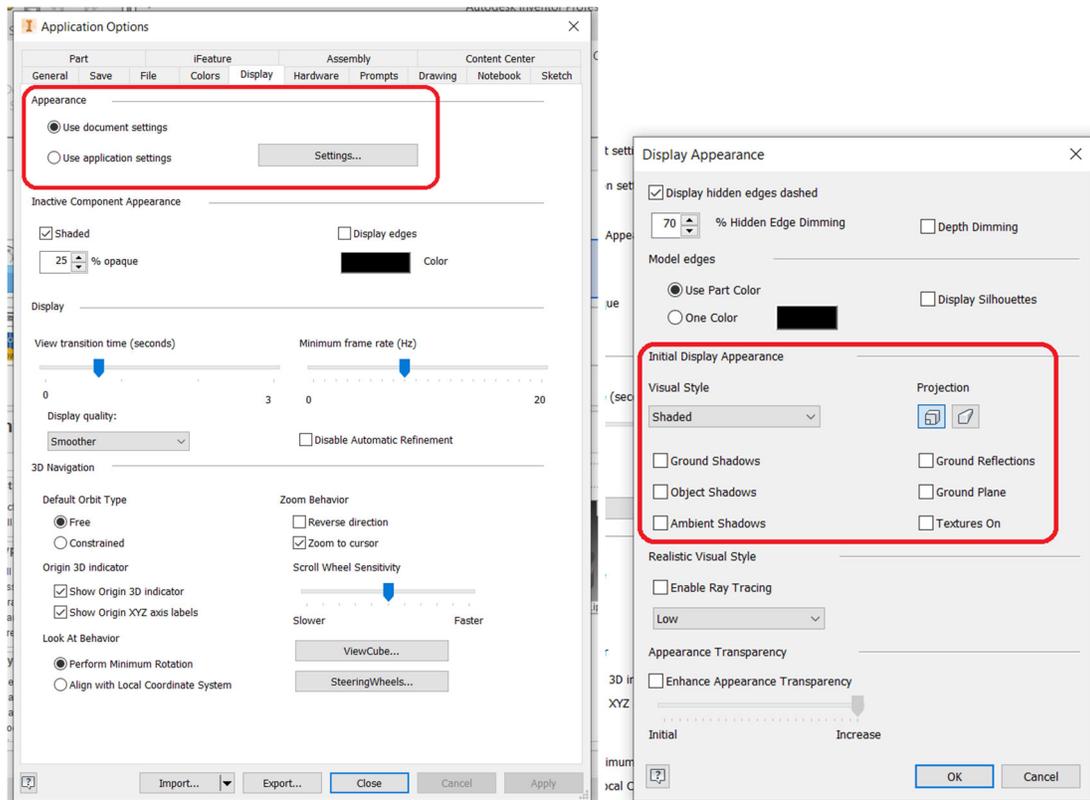


*Bonus: Changing the Tooltip & Selection delays won't speed up load times, but they'll speed up your working time within the files.*

### 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 will actually slow load times down, the opposite of what we are attempting to accomplish here.*



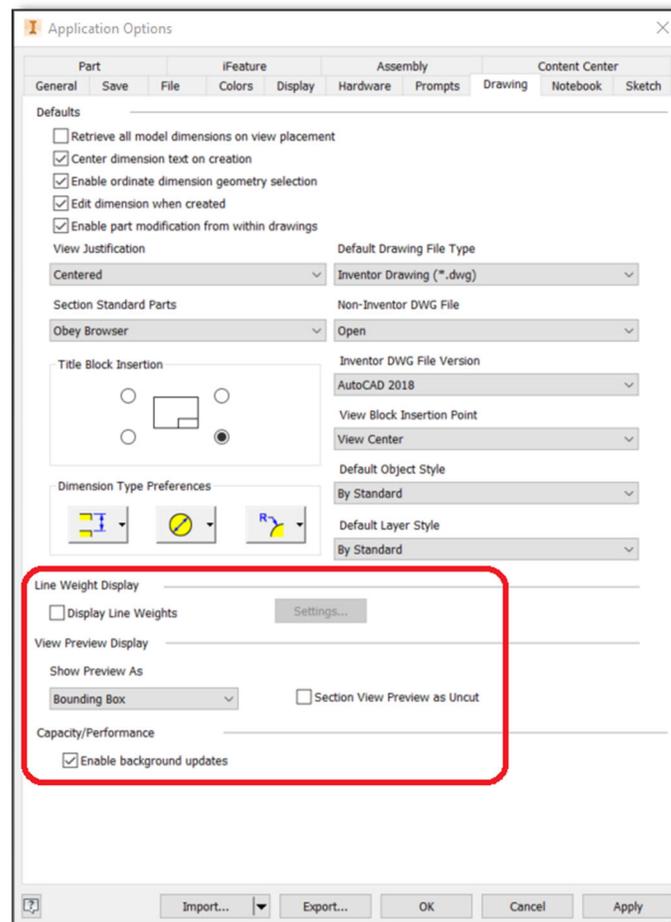
### Display Tab – View Transition Time/Display Quality

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.

**Drawing tab:**

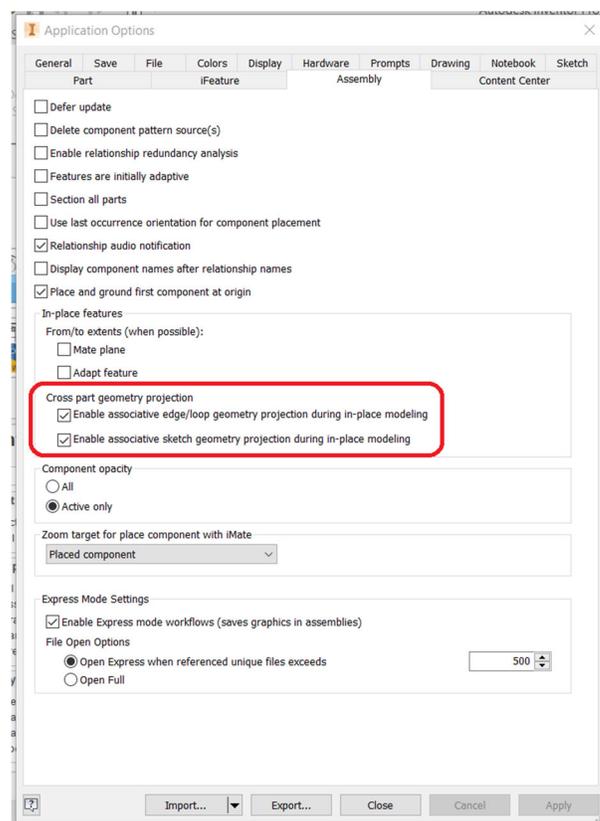
- Line Weight Display**  
 This setting enables/disables the display of line weights in your drawings. Deselect this box, it will only affect the lines within the drawing display, not when printing
- View Preview Display**  
 Set this to “Bounding Box”, this provides a rectangular boundary of the view and saves the application from having to recalculate the edges as you move the mouse to locate the view – a huge memory consumer during view placement and generation/update during drawing activation
- Capacity/Performance**  
 This sets the application to place raster views while the precise version of the view is being calculated in the background, allowing you to continue working. Very helpful when working in large assemblies

**Note:** You can save and/or close a drawing while the raster views are computing. The process will be stopped at that point but will continue when the drawing file is activated again.



## Adaptivity

- 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.



## Shutting Down

How often do you shut down not only the application, but your workstation? How many times have you seen people who, at the end of the work day or even the week, simply get up and walk away and go home...are you one of them? Shutting down the workstation 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 for at least 10 minutes will clear this reserved memory.

*Bonus: Want to free up even more memory? Disable MS’s Superfetch routine. This routine grabs holds memory so other applications like Spotify opens a little faster.*

## Colors and Finishes

Some colors and finishes can increase the file size as well. Adding a finish such as a reflective color will increase the file size the most. Colors that are “flat” increase the file size the least, consider using those where applicable.

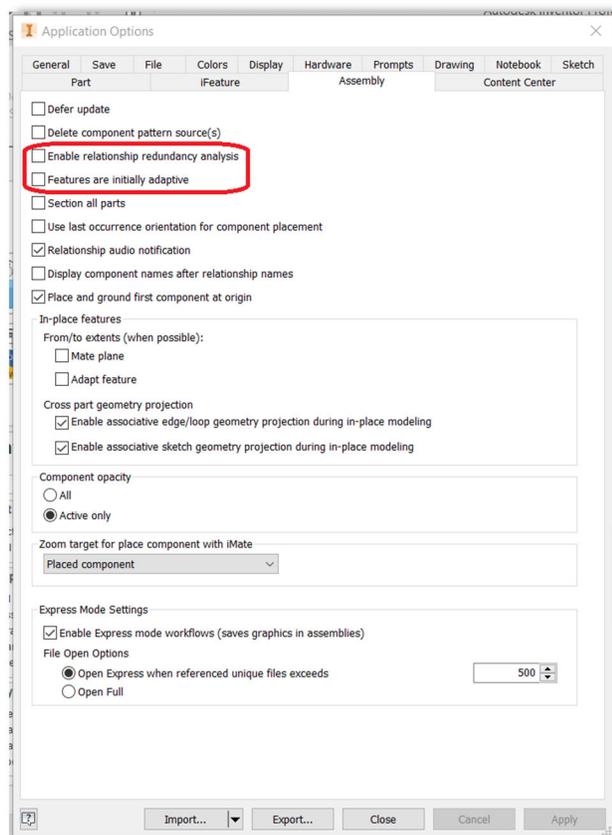
Consider purging any unused color style definitions, especially for common parts. Keep in mind as well, that each time you change a material or color in a part file, the style definitions are cached in that file. Each time this file is used in an assembly, the unused definitions are multiplied and impacting the memory.

To properly purge these unused definitions, go to the Manage tab, in the Styles and Standards panel, click on the Purge button, save the file.

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*
- *Features Are Initially Adaptive*



## Section #2 – Before you Begin – Work Effectively

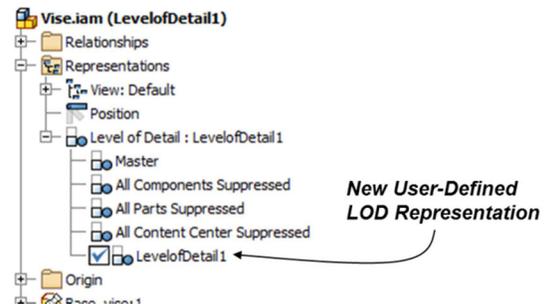
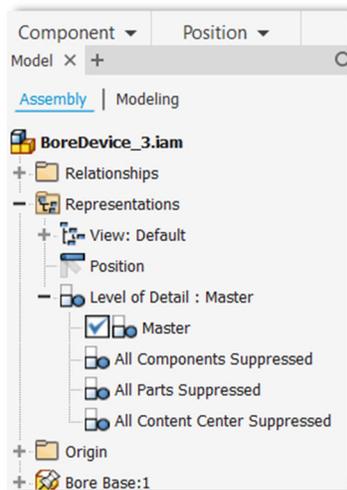
### Level of Detail

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 (discussed later), 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:

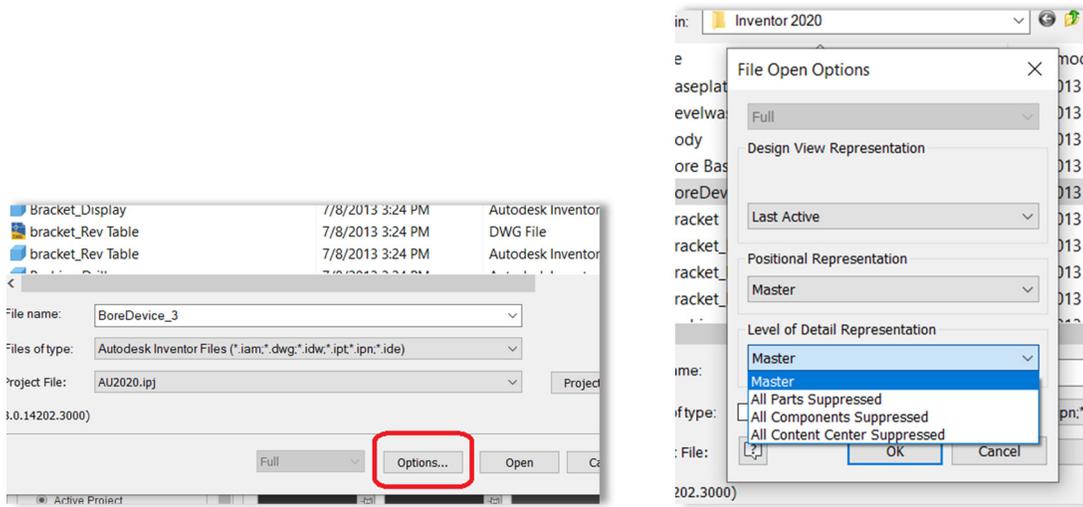
- Master – No parts are suppressed
- All Components Suppressed – Everything is suppressed
- All Parts Suppressed – All parts at all levels are suppressed
- All Content Center Suppressed – All Content Center parts are suppressed



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 can 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 (its main purpose in life), there are some issues or drawbacks to keep in mind. You'll need to consider if the Pros outweigh the Cons.

The most obvious issue you'll see will be the thumbnail icon or preview image in the My Home page, Vault, etc. 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, as the thumbnail will show a snapshot of the file's state on last save.

When inserting an assembly with the LOD 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. That might seem obvious, but you'd be surprised.

### Task Scheduler - Defrag your Files

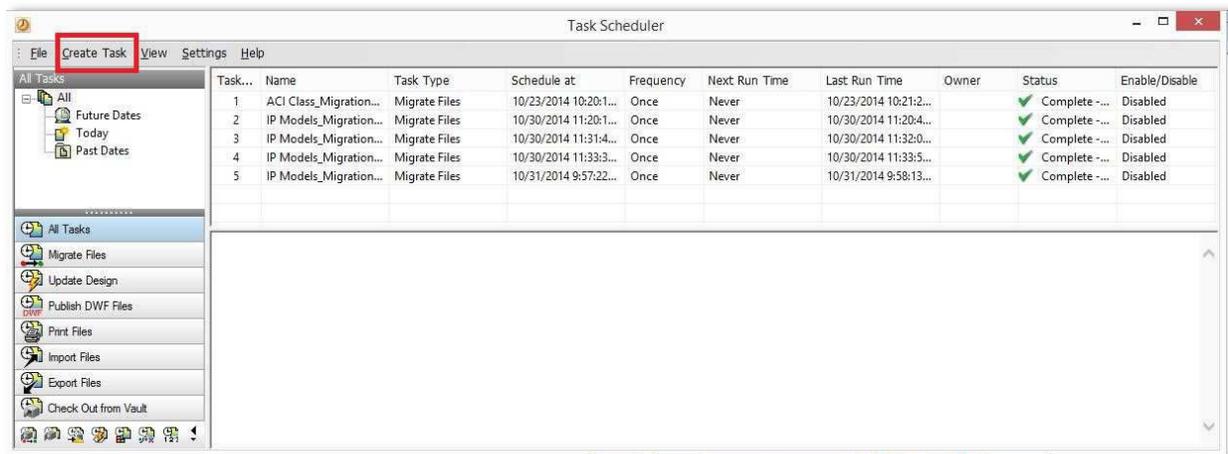
You know (or should know) you can defragment your HDD, but what about your CAD files? Well, you can and should. For those of you who are not familiar with this process, running the defragmentation process on your HDD scans and finds all the fragments of the files and applications, scattered throughout the HDD 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 over and over and... 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.

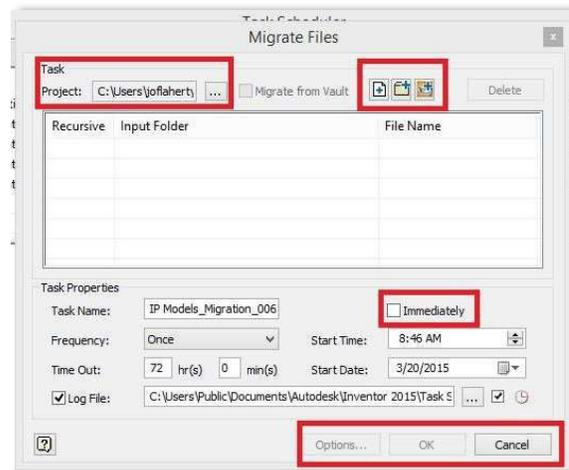
So, what is this magic elixir? Inventor's **Task Scheduler** application. Once you open the application, you'll notice a number of different uses it offers from Migrate, Publish, Print, Import, Export, etc., for this "trick", we will focus on the Migrate task. The Migrate task will take the file(s) and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

#### How to use this application:

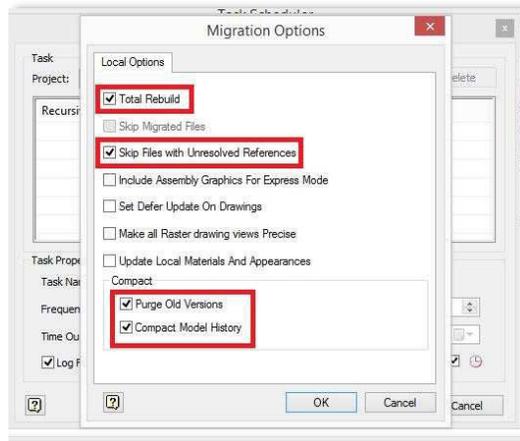
- Save and close all files open in Inventor
- Close down the Inventor application
- Open **Task Scheduler** (depending on the version of Windows you are using, it can be found under:
  - For MS Windows prior to Win 7 – Programs>Autodesk>Inventor 20xx
  - For MS Windows 7 and newer – it is found on the Apps page under the current Inventor release heading
  - *I recommend saving it as a shortcut to your desktop*
- In the **Task Scheduler** dialogue box (see next page)
  - 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. Typically most "failures" of files are blown/orphaned constraints.

*Bonus: Always fix these blown constraints! These are one of the biggest time consumers you'll experience when opening/closing files. Here's why:*

*Upon opening the file, Inventor is working feverishly in the background to regenerate and update the file. When it runs into one of these errored constraints, Inventor employs every possible task to fix the error automatically. Once each possibility is exhausted without a fix, it gives up and moves onto the next errored constraint.*

*Let's say you have a file with 50 constraint errors and it takes Inventor just 1 minute to run through each possible "fix" it has in its arsenal before it moves on to the next error...you guessed it, you've just added 50 minutes to your wait time to open that file. Granted, in real life it works much faster, but you get the idea.*

*When you see the red cross in the Quick Access Tool Bar, open Design Doctor and fix those errors.*

## **Section #3 – Best Practices for Assemblies/Parts/Drawings**

### **Assembly Files**

#### ***Simplify an Assembly***

There may be any number of reasons you want or need to simplify your 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 and on.

#### ***Simplification***

Shrinkwrap (and its variations) has been an Inventor offering for a number of years and recently improved upon with the 2019 release. Shrinkwrap basically takes your assembly model and fills in all voids, leaving a solid mass thus protecting any internal IP. A key issue for files 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) with the Shrinkwrap Substitute command.

You can also Shrinkwrap 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.



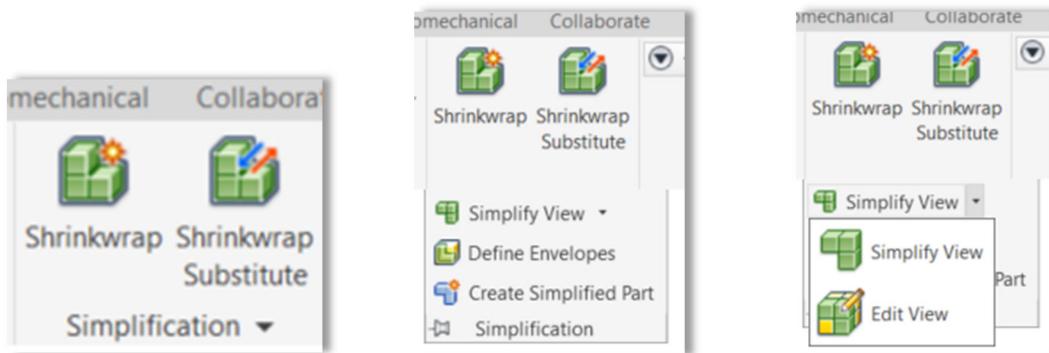
*Shrinkwrap model sectioned to show solid interior.*

- Shrinkwrap uses the active View or Level of Detail (*discussed later*) representation of the assembly. Components that are not visible or are suppressed are not included in the shrinkwrapped file.
- *For example, activate a View or LOD representation that contains no hardware or internal parts to perform fewer calculations.*
- Update the Mass Properties of the source assembly **before** you create the shrinkwrap file to copy the mass properties to the shrinkwrapped file.

*One thing to keep in mind, using the basic shrinkwrap command may not necessarily reduce the file size, actually in most cases it will increase the file size, but the shrinkwrapped file is a much simpler file (based on your selections) in the manner of calculations, instances, fewer edges, surfaces, etc. which then allows Inventor to open the file quicker.*

### The Simplification Tools

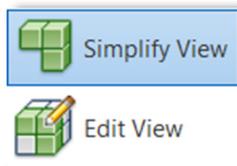
Besides Shrinkwrap, you also have a few tools that are provided specifically for assembly simplification purposes. These are found on the Assemble tab in the Simplification menu (2019 release).



### Simplify View/Edit View

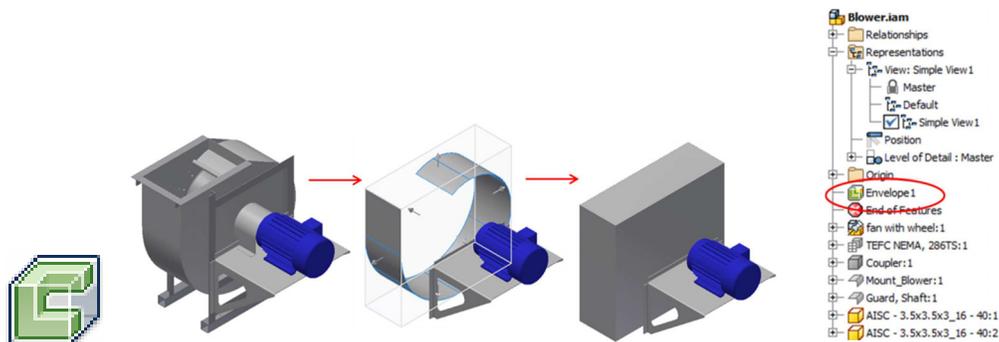
Use the Simplify View command to create a simplified view representation.

Use “Edit View” to modify the View Representation.



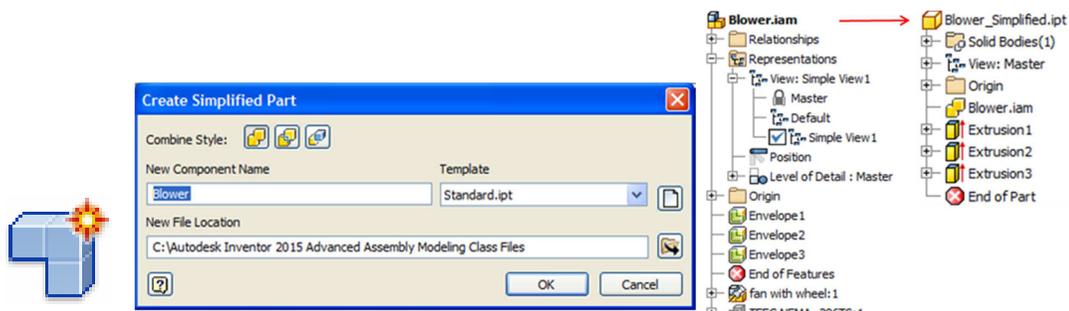
### Define Envelopes

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.



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.

### Drawings Files

Drawings can, by their nature, be the biggest consumer of memory you'll deal with in CAD.

Think about it, for each view you create, the system needs to do a massive amount of behind the scenes calculations to determine the representation of all the edges that make up that specific view. Edges that may be hidden and thus have to be represented as dashed, outer edge lines, hatch lines for sectioned views, and what about the edges that represent a sweeping surface, or edges shown in an isometric view, etc.

That's just a typical view, now consider how many views you have on a typical part or assembly drawing, add in all the annotations, charts, parts lists, etc. you get the idea and we haven't even touched on the issue of shaded views.

Incorporating some of these steps for drawings will most likely give you the greatest return on speed and memory, even on small parts and assemblies.

### ***Raster Views:***

Using Raster Views enables you to place the view in a raster mode and continue working in the drawing while the system calculates the info for displaying that view in the background.

### ***Hidden Lines:***

Reduce your use of hidden lines as much as possible, especially in large assemblies. Hidden lines tend to be one of the most memory-intensive components in a drawing view.

### ***Title Block logos:***

What is your company logo made up from? Is it a bitmap? Is it imported from AutoCad? Is it drawn from scratch in Inventor? If the logo is a bitmap, reduce its file size and resolution as much as possible, then be sure to embed it as opposed to linking it

*Linking the bitmap causes Inventor to locate that bitmap each time the drawing file is opened. Having the logo embedded outweighs the flexibility of being able to edit the logo independent of the title block. If the logo was imported from an AutoCAD file, be sure to clean it up as much as possible. Delete any ACAD generated hatching, this can be added later from within Inventor and pay close attention to multiple entities making up one line. Remove any and redraw the line as one entity.*

## **Section #4 – LOD versus Design View Representations**

As discussed earlier, LOD's were created by Autodesk for the main purpose of loading large assemblies faster, just like the Express mode option. By suppressing the files, only the required info is loaded during the file open process. Once the file is completely loaded, the user simply activates the LOD he desires to work in (Master or any user generated LOD) and moves on.

While LODs can be used to generate views and is a very easy way to do such, the user needs to know the ramifications of doing such, especially if the design is still changing on any of the files involved.

For example: You're detailing an assembly file with 4 parts in it. You create a LOD with one of those parts suppressed and use this LOD for a view on your drawing. An engineer decides to add another part to the assembly. Even though your LOD'd view doesn't require that new part to be included, it will still show in your view. Now, you'll have to back track and modify any view you don't want that new part shown in. This might not be a big issue when detailing small assemblies with a couple of sheets, but consider if you are working in an assembly with 10's of thousands of parts and 20 odd sheets worth of details.

Note: if you were to generate a BOM/Parts List off of a LOD'd view, the BOM/Parts List will show everything in the assembly as if it was created from the Master LOD. Yes, you can modify

the BOM/Parts List to only show what's included in the specific LOD, but now that's just more manual work you'll have to track if and when a change happens.

These are just a couple of examples of the pitfalls of using LOD's for custom view generation. The ideal process is to create a Design View Representation and use that as the basis for your detail view.

Design View Reps are in essence a "snapshot in time" where you can easily pick and choose what the view represents, from parts to colors to view orientation to zoom factor, etc. and when creating a BOM/Parts List based off of a DVR, only the parts within that DVR are listed. You can add and remove parts with ease as well. Parts added or modified not in the original DVR will not affect the DRV, unless you chose to include such and then save it.

DRV's are also a great time saver when working in very large assemblies. For example: you're working on the tail section of a Boeing (or Airbus) jetliner top level assembly. Once you've opened the file using the tricks you've just learned in this class, you still need to navigate to that section of the plane. Do you want to have to do this practice each and every time you have to open the file? Of course not! Navigate into the area you need to work in, set a DRV with the zoom factor, rotation and parts you need to work with and save it. Now, next time you activate the assembly file, open that DVR and Boom Shakalaka! You're there.

- *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:**