



# AUTODESK UNIVERSITY 2015

CP10152

## **CADZilla!!! – Taming Large Files in Inventor**

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### **Learning Objectives**

- Setting specific graphical options for faster load times
- Best Practices for large or complicated files
- Using Design Views
- Using Level of Detail

### **Description**

Do you often find yourself waiting on your files to upload because they are very large or complex? Ever wish you could speed up that load time? 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.

### **Your AU Expert**

*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 35 yrs. in the Mechanical Design field covering industries of Automotive, Power Generation, Industrial, Consumer Goods, Avionics and Hazardous Waste removal, he has moved on to IMAGINiT Technologies as an Applications Expert. He has been certified on Autodesk's Inventor since Release 3 and one of the early adapters 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.*

- **Preface – CADZilla!!! Not a song by Blue Oyster Cult, but close**

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 – It really is a “True Value”**

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 now uses Direct3D as opposed to OpenGL like it did prior to the release of Inventor 2012. This opens 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 2016 release, so it does not take advantage of 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 2gig card and two monitors, the memory is divided equally to each, and thus the CAD monitor is now only using 1 gig.



OK, so now with the Hardware discussion out of the way, let's get started on what you can do in the Inventor application.

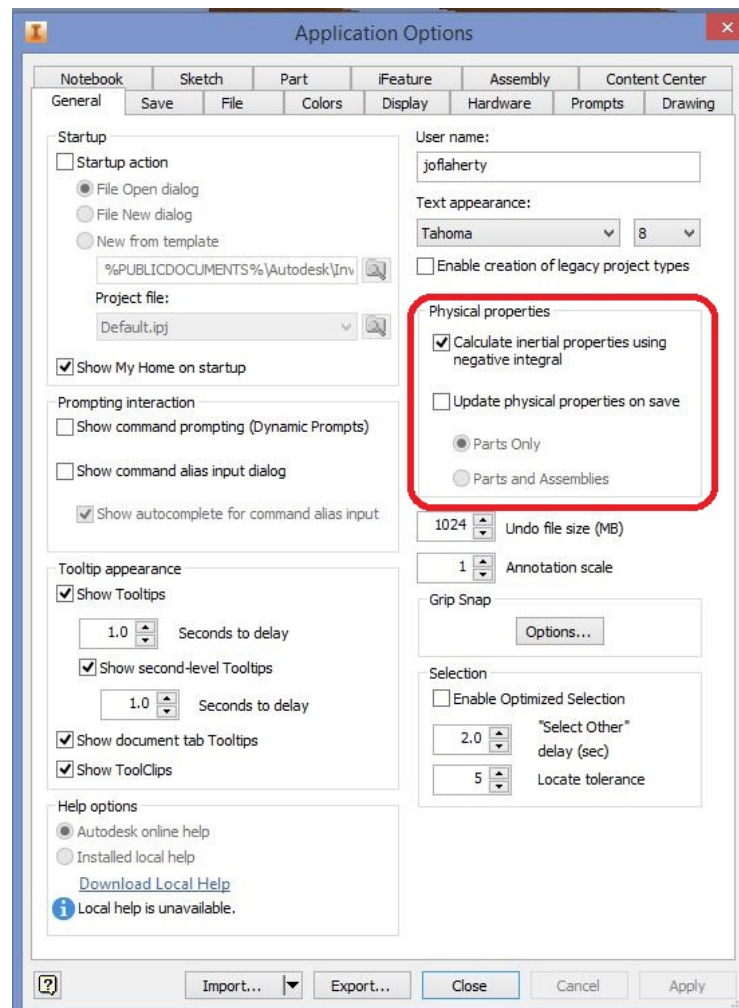
**Keep in mind, Application Settings apply to the entire Inventor application, whereas Document Settings apply to the individual active document**

- **Settings – Application Settings**

In Inventor, select the Tools> Application Options menu

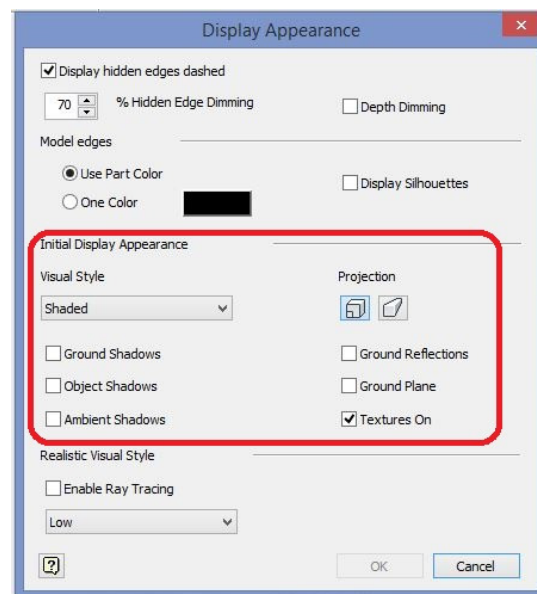
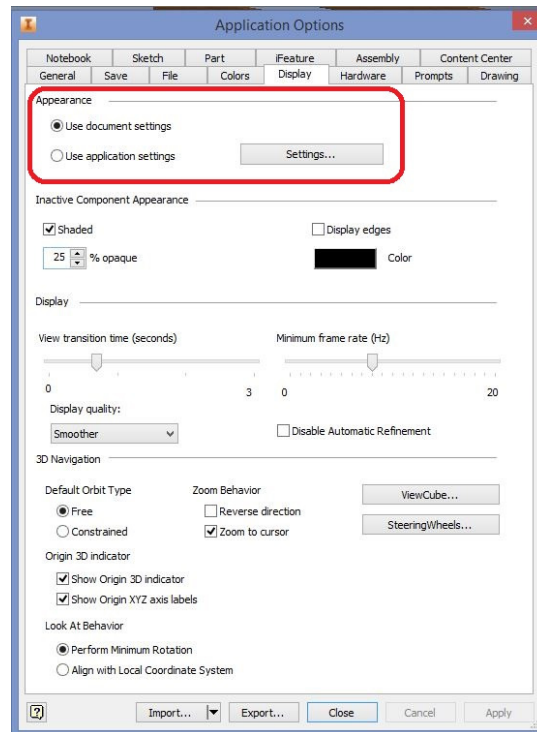
- **General Tab – Physical Properties**

- 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.

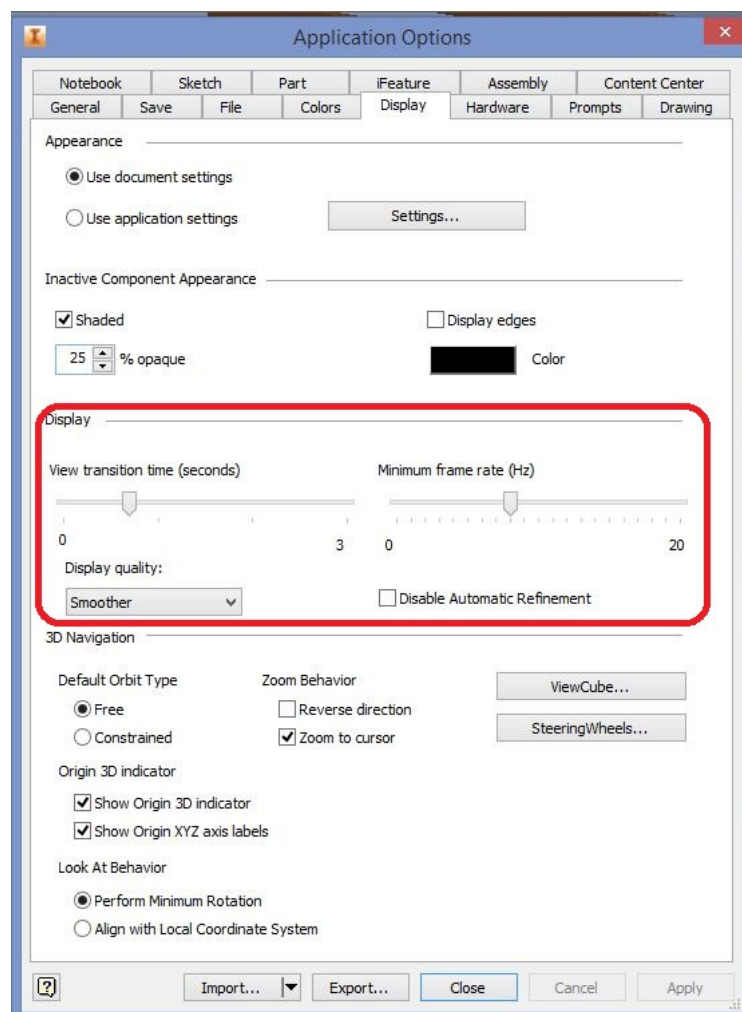


- **Display Tab – Appearance**

- 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 that selecting these “on” will actually slow the system down, the opposite of what we are attempting here*



- **Display Tab – View Transition Time**
  - This determines the amount of transition time you'll see when using a zoom all or view command such as the View Cube. Setting this to “zero” takes you from the current view state to the selected view state instantly. Setting this to the maximum of 3 seconds, you'll see a gradual transition from one view to the next
- **Display Tab – Display Quality**
  - This allows you to set the Display Quality of the file. Setting it to “Rough” will speed up the regeneration process by simplifying the details of the file.



○ **Drawing Tab – Line Weight Display**

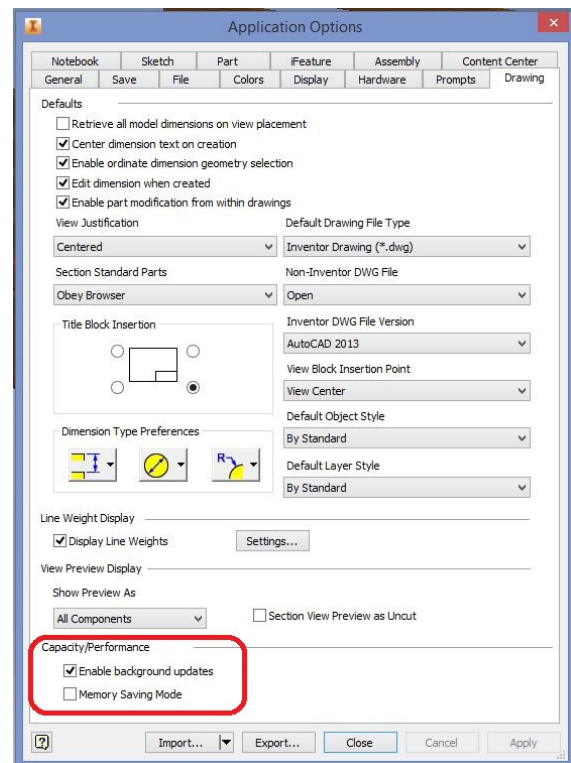
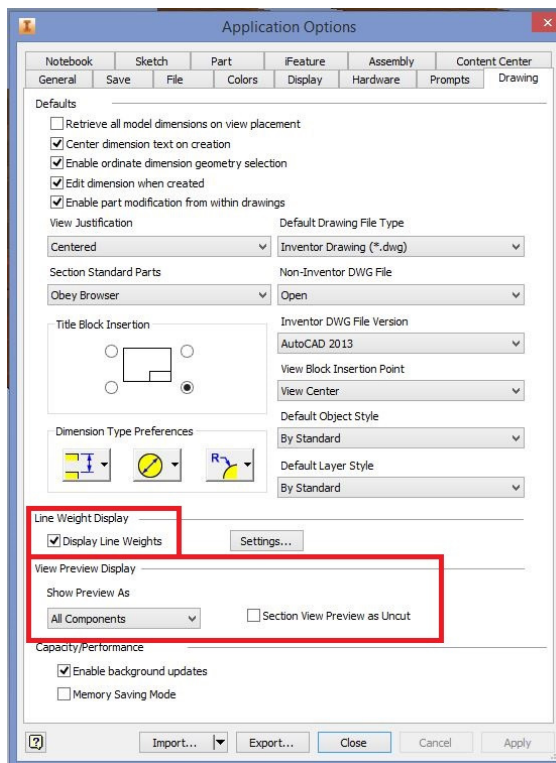
- This enable/disables the display of line weights in your drawings. Deselect this box, it will only effect the lines within the drawing display, not the printout

○ **Drawing Tab – 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. Select the Section View Preview option as well.

○ **Drawing Tab – Capacity/Performance**

- This sets the application to place rather views while the precise version of the view is being calculated in the background, allowing you to continue working. This really helps in very large assemblies. A green node will be shown in the browser until completed. Hover the mouse over the view name and Inventor will display the percentage completed.
- Saving or closing the drawing file while the raster views are still computing and Inventor will stop the process and pick up right where it left off the next time you open the file.



## • Best Practices- Part Files

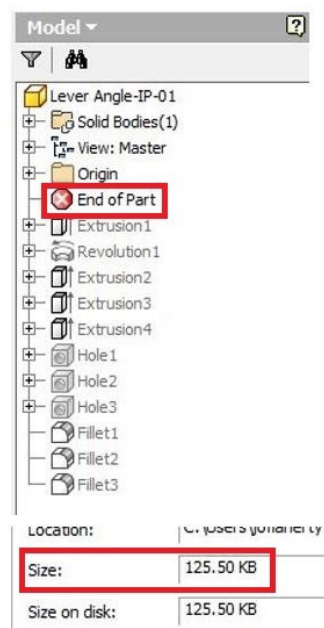
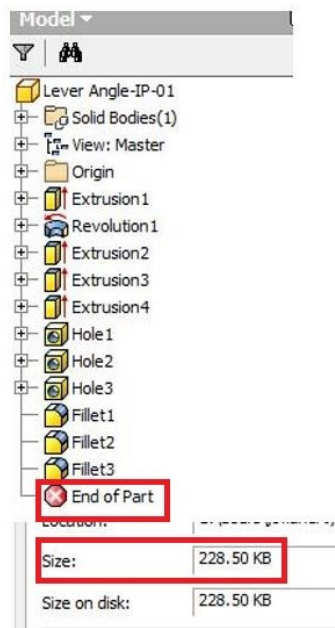
Often times the moment users think about making the following setting changes is after they've been impacted somehow while working in a large file. Most of these settings can be done prior to that happening and thus preventing or minimizing the impact from happening in the first place. It takes far less time to make these settings up front as opposed to hitting that wall in a file and then having to backtrack, figure out what settings to adjust, how to edit the model, all in addition to the time already spent waiting for the file to regenerate. 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.

### ○ End of Part Marker

One of the very first things you should incorporate across the board is to take the End of Part Marker in the Model Browser and move it all the way up the browser tree to just below the Origin folder. This retains the features in the part file but doesn't require them to be "active". In turn this takes the size of the file to its bare minimum.

When you open the file, you simply move the End of Part Marker back down to the bottom (or to the point of the edit you are wanting to make) and the file rebuilds, make your edits, save and move the Part Marker back up to the Origin folder and save again, store in Vault or wherever you save your files. Again, make this a common practice and the savings in time and space will be enormous.





- **Shutting Down**

How often do you shut down, not only the Inventor 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. If the file you need to open is very large and takes a long time to load, you can go into the Application Settings menu and under the File tab, check the box that saves the last opened file to cache.

- **"Keep it Simple, Stupid" (or the more Politically Correct "Keep it stupid simple")**

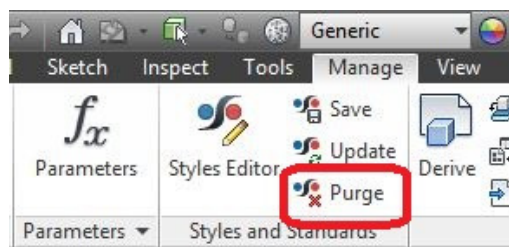
We've all heard the expression at some point in our lives. Well in this case it's very true. Keep your part simple, the more complex your files, the more calculations need to be done, the longer it takes to load said file. For example, in my travels I've seen users creating parts such as bolts, screws, etc. (aside from the fact they reside in Content Center) and they'd actually attempt to model the threads of these bolts...why? They are not the bolt manufacturer, they are not doing FEA on the threads, etc. There's no reason for such nonsense. Even if you need to model up something like a bolt, you can "dumb it down" for practical use in your assemblies. If you have purchased parts you can edit these files and remove unneeded features such as fillets, chamfers, etc. as long as you don't modify the integral features of the part, you're fine. This practice can drastically reduce the file size of the part/assembly.

- **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. 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.



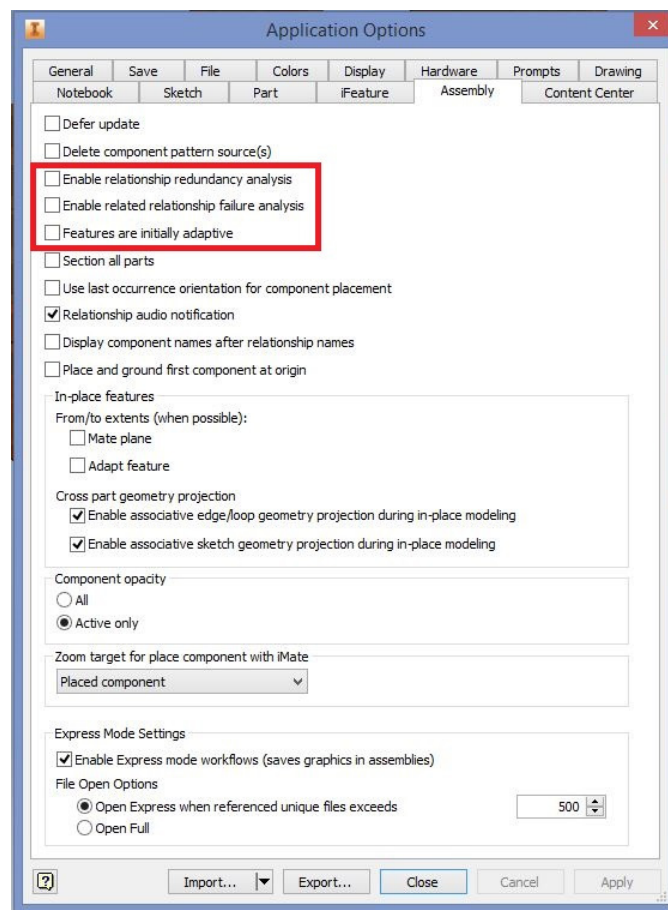


- **Best Practices- Assembly Files**

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.

- **Application Options**

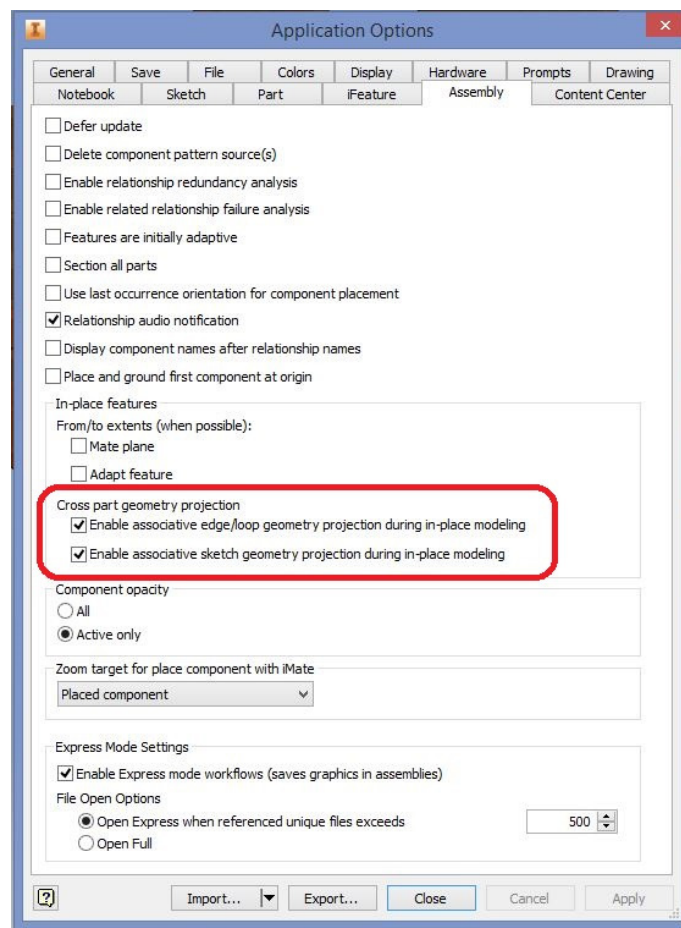
- Enable Relationship Redundancy Analysis
- Enable Related Relationship Failure Analysis
- Features Are Initially Adaptive



- **Adaptivity**

*“If you live by the sword, you die by the sword”*...Adaptive parts are created within the assembly environment when you reference an edge, face or feature of a part within that assembly, such as 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. Generally you should disable the adaptivity option once the parts are fully created.

You can choose to disable this option in the **Application Options** dialogue box. Under the *Assembly* tab, 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 pop-up menu.

- **Best Practices- Drawing 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, edges shown in an isometric view. 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 using shaded views in your drawings.

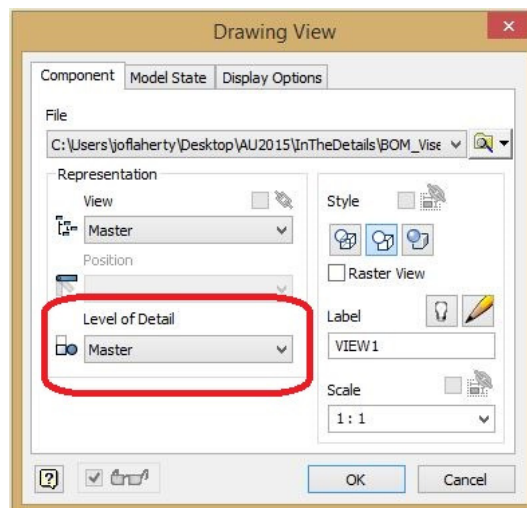
Taking some of these steps for drawings will most likely give you the greatest return on speed and memory, even on small parts and assemblies.

- **Raster Views**

Using Raster Views enables you to place the view in raster mode and continue working in the drawing while the application calculates the info required to display the edges properly for the orientation of that view. This is especially evident in large or complex assembly/part files

- **Level of Detail** (*discussed in detail later*)

Using the Level of Detail option in your assembly file, you can specify which LOD you want to use for the view creation. Once the LOD's have been created, you can easily select which LOD you want/need to use in the Drawing View dialogue box.



- **Hidden Lines**

Reduce your use of hidden lines as much as possible, especially in large assemblies. Typically the only time hidden lines are seen in assembly detail views are for added clarity, otherwise having hidden lines active in an assembly, can get rather confusing to say the least. Hidden lines tend to be one of the most memory-intensive components in a drawing view.

- **Title Block Logos**

No doubt you have your company logo in your title blocks, what is it made of? Is it a bitmap, imported from AutoCad, 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 to the drawing template. 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 fragments of lines and redraw them as a single line if possible. Delete any ACAD generated hatching, then add it as an Inventor hatch.

- **Drawing Sheets**

Personally I don't recommend this tip, but I mention it in order to inform you on the process and its benefits/pitfalls and thus let you, the user decide what works best for your situation.

Some have suggested a way to minimize a drawing's file size is to limit the number of sheets per file, i.e. what would typically be a 4 sheet drawing file, gets divided up as sheets 1 & 2 as one file, then sheets 3 & 4 saved as a different file and so on.

Granted, this process does in fact cut the file size down and can even make it a bit faster opening the file, especially if you have a change to make on only one of the sheets, but think about the hassle of keeping the files together. It can be a data management nightmare for sure, especially if "number of files" is a concern. Navigating the directory where these files are stored can get very long. Consider you have a drawing file that has 10 sheets, and you divide each file into 2 sheets per maximum, instead of navigating for one file, you're now having to navigate for 5 separate files and so on.

Again, the decision is obviously yours to make, do whatever makes your daily CAD life easier.



## • Design View Representations

If you find yourself in a large assembly, whether it be in file size or physical size, setting and using Design View Representations can greatly improve time spent navigating by selecting the needed View Rep as opposed to using the orbit command. This becomes evident when you have a very simple or quick edit to make and you need to navigate to a specific section of the assembly and in turn waiting for the graphics to play catch-up.

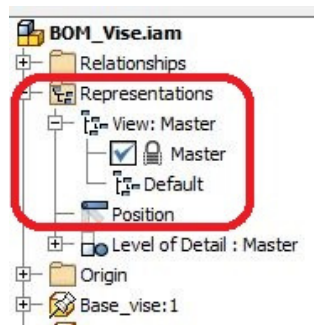
Design View Representations are created in the assembly environment to save a designated view representation of the assembly and its components. It can be used to show or document different iterations or the assembly as well as depicting a specific view angle or even different variations of components or even the visibility of components. This view can then be referenced in the drawing for detailing purposes.

Some tips to follow for performance benefits and memory savings:

- Select a design view representation that only displays the components that must be visible. Invisible components in the design view are not loaded into memory.
- Close the assembly file used for a drawing view to prevent its graphics from being loaded into memory.
- To edit the model displayed in the drawing, in the Application Menu, click Open to select the assembly file, and then click Options. Select the representation used in the drawing.

To create a Design View Representation:

- In the Assembly Browser, click the Representations node to expand, right-click the View node, and then click New. A new ViewRep node is added to the browser. It becomes the active View representation, as designated by a check mark. A number is appended to the View node. You can click the view name and rename it to something more suitable.



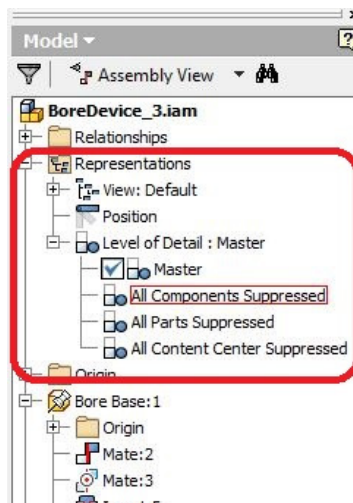
- Define the view's characteristics you need to preserve such as zoom factor, view orientation, etc.
- Save the assembly file

- **Level Of Detail (LOD)**

Opening a large assembly can be a time consuming task, especially if you have numerous assemblies to open. This doesn't have to be the case. Earlier we discussed a part file's End of Part Marker, assemblies do not have an EOP marker, but they do have a default set of Level of Detail. Like the part file's EOP marker which reduces the file size to the minimum, utilizing the All Parts Suppressed Level of Detail will accomplish the same results for assemblies.

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 – Everything suppressed
- All Parts Suppressed - All parts at all levels suppressed
- All Content Center Suppressed – All Content Center parts suppressed



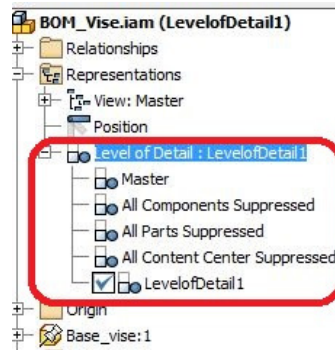
Select either “All Components Suppressed” or “All Parts Suppressed” depending on your needs and save the assembly, check the file size. Whereas these default LODs are close to being “all or nothing”, you can also create your own LOD which specifies what level of parts you have active upon opening the file. You are not limited to the number nor the names of your LODs.

This can also be done at the open file dialogue box if you rather operate that way. In the Open File dialog box, select the “Options...” button and select the appropriate level of detail to open in the “Level of Detail Representation” pull-down menu. This option may be a better fit for you in that not all your assemblies may be very large where you'll need this option.



To create a Level of Detail:

- In the Assembly browser expand the Representations folder and then expand the Level of Detail node, Right click and select New Level of Detail. A new LOD node is added to the browser. It becomes the active Level of Detail, as designated by a check mark. A number is appended to the LOD node. You can click the LOD name and rename it to something more suitable.



- Suppress/Unsuppress the components as required
- Save the assembly file

### • Free Stuff – 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 avoiding 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.

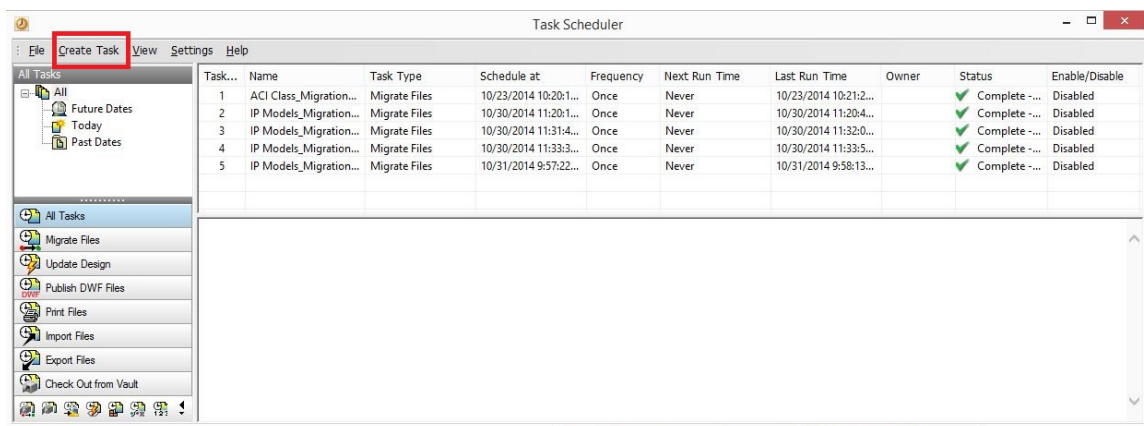
So, what is this magic elixir? Inventor's Task Scheduler application. Once you open the application, you'll notice a number of different uses it offers from Migrate, Publish, Print, Import, Export, etc., for this we will focus on the Migrate task. The Migrate task will take the file(s) and run it through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



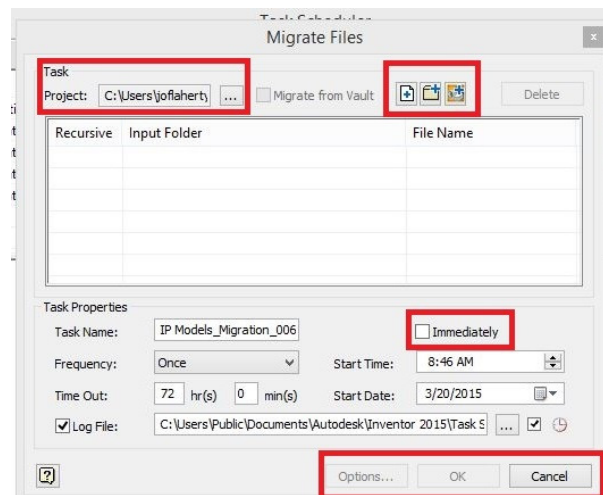


How to use this application:

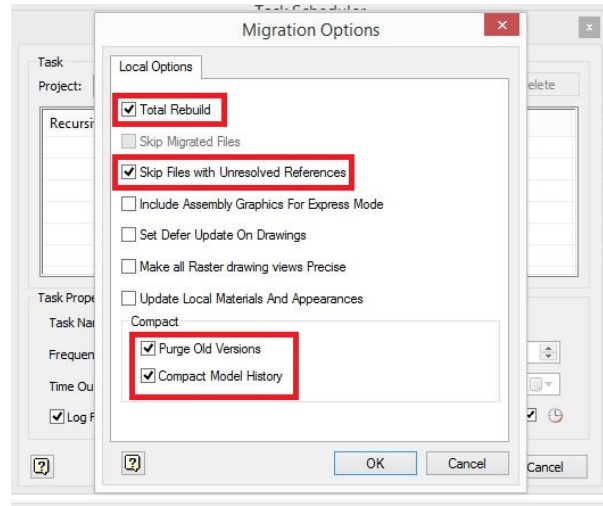
- Save and close all files open in Inventor
- Close down the Inventor application
- Open **Task Scheduler** (depending on the version of Windows you are using, it can be found under:
  - For MS Windows prior to Win 7 – Programs>Autodesk>Inventor 20xx
  - For MS Windows 7 and newer – it is found on the 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:

