

**MFG118915 (Industry Talk)**

## Boosting Inventor Performance

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### Learning Objectives

- Learn how hardware is not always at fault
- Learn how other items can/may impact your Inventor performance
- Learn what can be done to combat Inventor performance

### Description

This class will take a different approach in discussing how other items (workflows, structure, configurations, updates, maintenance, and etc.) will impact your Inventor performance and demonstrate why hardware upgrade is not always the solution for resolving performance issues within Inventor.

### Who is Mark Lancaster?

*Since 2013, Mark Lancaster has been the Product Support Specialist for Synergis Engineering Design Solution (an Autodesk Platinum Partner located in Quakertown PA). Besides the focus on the Autodesk manufacturing & data management solutions along with Autodesk licensing, Mark is also a regular contributor to the Synergis CAD Blog web site. Mark is an Autodesk Certified Inventor Professional for the past 3 years and his expertise is centered on frame generator, routed system, content center and iLogic. Besides assisting Synergis customers with product support issues, Mark also spends his time on the Autodesk Licensing/Installation & Inventor forums assisting other user like yourself. Mark has been an Autodesk Expert Elite close to 2 years now, but is no stranger to the CAD world. He started his career out (late 80s) by performing drafting/design duties using a (drafting) board and CAD software. Mark has over 20 years of experience in the manufacturing world, 15+ years in the 3D modeling world, and is a self-taught AutoCAD user since Release 9. However most of that time has been around CAD and document management administration. His motto for many years has been "I'm here to assist you in getting your job done" and is always looking for better ways of doing things due to his "Lean Manufacturing" background.*

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*(All non-Inventor images within handout are from Google Images)*

Just like this Lamborghini, we want our Inventor models to be supercharged, efficient, and fast (loading). But due to some reason...



Our models end up being like a kid's tri-cycle in the sense its slow moving, under powered, and not efficient. Or the feeling like we're constantly stuck in traffic.



The question I often hear when people are faced with an Inventor Performance issues... If I upgrade my RAM, add additional disk space or even purchase a newer graphics card or machine, would it help?



**Mine is always NO**

## What do you consider is impacting your “Inventor Performance”?

If this question was asked of you, what would you say?

At this point you may also be thinking... *“What really defines “something” as an Inventor Performance issue?”*

Personally, I’ve always viewed “Inventor Performance” not only as hardware limitation and/or software configuration issue but anything that I’m waiting around for Inventor to complete, stopping and resolving things within my model, basically when I’m not actually modeling.

*If the same (Inventor Performance) question was asked of your co-worker, would their response be the same as yours?*

I would bet there would be some overlap but each of us view this differently since “Inventor Performance” can impact us in so many different ways. Most likely everyone will have a different perspective on this subject matter and/or how they categorize something as a performance issue within Inventor.

What are your thoughts?

## **Is upgrading hardware on the top of your list as a solution?**

### **Why is upgrading your hardware not always the solution?**

Many times when we are faced with Inventor performance we run right into our boss’s office and/or visit our IT support group and request to have our PCs upgraded. Which in-turn we get the questions **“Why?”** and **“How much is this going to cost us?”**

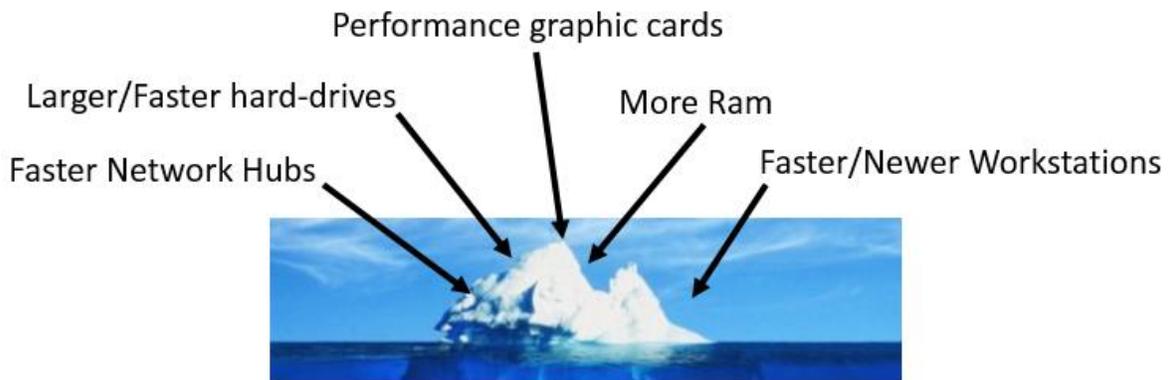
Am I right?

But hardware upgrade may not even be your issue at this point or perhaps there will never be a budget for it until next year.

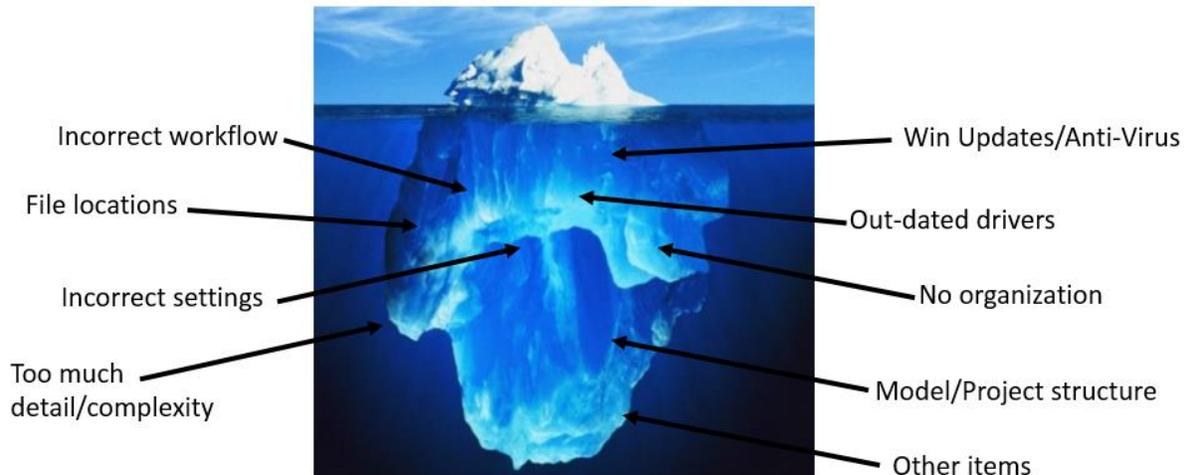
Meaning, what can we do about it?



Let's take a look at an iceberg... Where's the real danger? Above or below the water? Hopefully you said below the surface. Now let's take this same iceberg to analysis our performance issue within Inventor. A lot of times when we analysis a problem, we look at what's easy to see (or the surface). For the iceberg, we most likely associate the top of it, as the hardware upgrade path...



In a lot of cases, the actual root cause for our Inventor performance really lies below the surface or those things we don't really consider as the culprit.



Although I'm not a computer hardware guru/expert or geek, don't get me wrong... A lack of hardware is still a crucial part that impacts Inventor performance and I don't want to discredit it as a solution. But here's an example I would like to share with you in why upgrading hardware was not the solution.

Fred works for the Acme Company in downtown Boring, Oregon (yes there's an actual town in Oregon called that). Currently he has a supercharged PC with an overclocked CPU, 64GB of ram, and 4GB ram (top of the line) video card.

Lately he has been noticing he's spending a lot of time waiting around for Inventor to perform certain task on this given model. After a few days of frustration, he gives up and calls his IT group. Fred wants IT to upgrade his machine because he's fed up and doesn't want to wait around anymore for Inventor to complete his required tasks. At this point, Mary the IT tech is confused because she already knows the company spent a large sum of cash to purchase this high end PC for Fred. But Mary is not an expert in Inventor and makes a call to her reseller and explains what's going on. The help desk tech (named Zippy) tells Mary that most users don't have 64GBs of ram in their machines or even come close to a PC with that type of specifications. He goes on to explain to Mary, he believes it has nothing to do with the PC and more with the model structure itself. Zippy makes a request to Mary to review the model that Fred is currently working on.

In the end it was discovered there was way too much detail being carried across the entire model structure along with the designer using some incorrect Inventor workflows. Once Fred corrected his model based on Zippy's recommendations, he's now happy with his Inventor



performance and IT is also ecstatic they didn't have to spend a single dime in purchasing new hardware.

*The above story was an actual help desk case I received a while back (names/locations changed to protect the innocent).*

**Would you consider this issue “My model can't find my files (resolve link dialog present) as an Inventor Performance issue?**

For me I would always tell you YES. Like the saying, “Time is Money”! I'm a sole believer if I have to stop and find my files while loading my model (instead of designing), it's impacting my performance, which to me, should always be considered as an “Inventor Performance” issue as well. So it doesn't really matter if I have to instruct Inventor which files to use, I'm waiting around for non-responding messages or experiencing numerous Inventor crashes, these issues are still related to Inventor performance and must be addressed.

## Inventor performance issues and how to address them

Performance issues within Inventor can point to many different scenarios...



*No “Staples” easy button when it comes to resolving performance issues or a one size fits all solution.*

There’s also no way to conceive all possibilities that would impact your Inventor performance. In addition each machine, installation, user profile, and configuration is different. Although I’ve heard this many times, *“I have the same hardware and configuration as my co-workers and they don’t have an issue, so it should work on my PC”*. Well I have to say, identical machines are like identical twins, where each twin (machine) has its own traits. Machines are never the same.

## Let’s get started

This class will cover those items I’ve learned over the years that have demonstrated to impact Inventor performance. There’s no logical order and all information should be reviewed and considered. Perhaps you may already know about some of it, others not as much, certain ones will work, while others will not and then there will be those items not even relevant to your environment.

But when you’re faced with “Inventor Performance Issues”, I always state:

1. Don’t invest into newer hardware at this point.
2. Look below the surface for your root cause.
3. Test the solutions. Found out what works and what doesn’t.
4. Don’t dismiss something because you don’t think that’s the reason why. Too many times I’ve heard: “My <fill in the blank> is correct, it certainly can’t be that, it has to be something else”. How do you actually know unless you try?
5. If all fails, than review the possibility of investing into newer hardware.

**NOTE:** *Handout images and workflows associated to Inventor are solely based on Inventor 2018. Depending on the Inventor version you’re currently using, certain images/options/workflows in this handout may not truly represent what you’re seeing.*

## Performance Issue #1: Windows updates

How many times in the past few months a Windows update has come along and there goes your Inventor or other applications? Updates to Windows are a hard thing to control because in most cases Inventor users do not have any say in this matter or even the ability to control it. These updates are usually applied through your IT group or the 3<sup>rd</sup> party vendor that supports your infrastructure. But it's important that management/IT fully understands these updates shouldn't be rolled out until tested on an Inventor machine or two.

If you don't have any control or say in this matter, I would recommend that you sit down with management/IT and convey this is a potential risk to your design community. Another solution (if permitted by your IT group) is to roll back the update (if possible) or restore Windows to an existing point in time.

## Performance Issue #2: Downloading/Installing/Auto-Install/Autodesk Updates

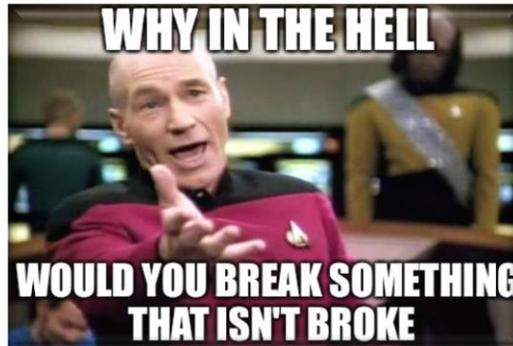
- *Installing things on my own*

I support many end users from being in a restricted environment to those who have all the freedom in the world to download/install software. Granted being in a restricted environment is not always fun, but at the other end of the spectrum where you have no restrictions in place, this type of freedom is a not an idea situation (in my opinion). Many times when we're allowed to just download and install software, certain downloadable ones may be installing optional offers that could lead to conflicts. Message here, **Beware of what you're installing!!** It may have nothing to do with Autodesk, but it could certainly impact the operation of the software and I've seen this too many times.

- *Autodesk Application Updates*

Years ago I was a big fan of keeping my Inventor or any other Autodesk applications up to the latest release. The minute the update was released I was downloading and installing it. As time went on I started to realize certain Autodesk updates broke other stuff. Yes this doesn't happen all of the time, but my opinion has changed over the years about applying any Autodesk update.

**WARNING: *There are a few users in the Autodesk Community that will strongly agree/stand with me and the other 99% will stand oppose to me on this subject matter.*** I know this is not something Autodesk would even recommend to you but take this information as my opinion/view on the subject matter. Whether this is wrong or not, but I now inform users if the service pack/hot fix is not resolving an issue you're having or you don't have any issues so far, why apply something that's not needed.



Yes in the past year or so, service packs have really faded away for Inventor and there's now more of a focus on providing mid-year releases/new functions/fixes. But even with these mid-year releases, some form of updates are still being issued to address other things. **In the end, my message to you...** Just because a certain fix is there for Inventor (or any other Autodesk application) doesn't mean you have to apply it.

### **Performance Issue #3: Anti-Virus and Malware Applications**

- *Anti-Virus Exceptions*

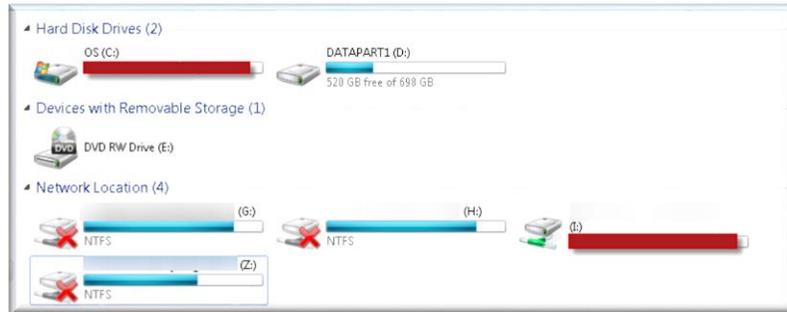
Having an "Anti-Virus" application on your machine is crucial in protecting your environment. But certain ones, just don't play well with Autodesk products. For a test, select one machine, disable (or uninstall) the Anti-Virus, run Inventor for a couple of days and see if there's any improvements. Yes I know we don't want to leave our system wide open for attack, but it's a test that needs to be performed to ensure your "Anti-Virus" is not the cause of your Inventor performance. If disabling it does improve your Inventor performance, apply the necessary (Anti-Virus) exceptions per this [Autodesk Article](#).

- *Malware on the machine*

When it comes to Malware prevention, you may or may not have any of these applications protecting your system or have it configured to periodically scan your computer. But I have run across a couple of instances where Malware was present on the end-user's machine and once it was removed, the Inventor performance issue went away. Check your system for malware.

### **Performance Issue #4: Avoiding the Red Zone**

When free disk space gets to a certain (low) point it's called the red-zone. The red-zone is an indication there's limited or no disk space left on that given drive.



Funny things, warnings, and performance issues start to happen within Inventor when you reach this point. I've had numerous cases over the years where the performance issue was simply related to the red zone indicator on a local or network drive (where the Inventor files are stored). But even if you're not in that red zone and free disk space is limited, it may be time to start cleaning up your hard (or network) drive and reorganizing it.

### **Performance Issue #5: Computer Maintenance/Files/Settings/Services/Drivers/Configuration**

- *Those pesky temporary files*

When I'm working on performance issues with any of my customers, one of the items I always review, is the amount of space the user's personal temp (%temp%) folder (C:\Users\\AppData\Local\Temp) is consuming on their hard drive. I've seen anywhere from 500 MB of disk space consumption to 13GB worth of data. My experience has always lead me to believe when you reach the 1 to 2GB range of disk space its consuming, it's time to clean it out. In my opinion funny things start to happen when it gets into this range. Not only are these temporary files consuming valuable disk space, too many files can lead to performance issue and eventually Inventor crashes. Although you will not be able to delete every file and folder within this location, I always recommend the following:

1. Close out every application.
2. Permanently delete ([without sending them to the recycle bin](#)) as many files and folders. Skip the ones that you will not be able to delete.
3. Reboot your machine.
4. If the consumed space is still close to 1GB or over, repeat step #2, followed by another reboot.

These steps should also occur on the temporary folders of C:\Temp and C:\Windows\Temp. In addition you should get into a habit of performing this maintenance cleanup at least once or twice a month.

- *Defrag that hard disk*

## **This solution is only for those non-Solid State Drive (SSD) hard-drives**

When is the last time you performed a disk defrag on your Windows 7/8 hard drive? If the hard drive has to retrieve bits and pieces of your Inventor information on different sectors, it will impact performance in the end. Yes it's not a big impact but it's something (disk defrag) you should be doing every so often.

*Helpful Tip:* [Windows 10 automatically does this](#), but you may want to check to ensure the process is still functioning properly.

- *Windows registry*

Have you ever gotten a new PC and you're amazed at how fast your machine is? You brag to all of your co-workers about it and then a few weeks later you feel like you're a big fat lair. Your Windows registry plays a role in this. Over time the registry gets loaded with "stuff" that doesn't need to be there anymore or certain entries are just disconnected or lost.

### **Playing with the registry is at your own risk and you should always back it up prior to making any changes.**

Users have had success with Inventor Performance by using a registry cleaner such as [CCLEANER](#). I've personally seen the outcome a couple of times, but at the same time these so called "registry cleaners" can impact your registry in such a way that it could take your system down. Although I'm offering this as a solution, **this solution is at your own risk.**

- *Windows Services*

Here's another reason why your PC performance degrades overtime. Many years ago when working with Inventor 2008, I got frustrated not only how slow my Win XP 32B machine booted but how Inventor was acting. One of my IT tech said to me one day, let's take a peak to see what's getting loading when your machine is booted. I was completely astonished at all the useless applications, services, drivers, and etc. that was being loading. No wonder why my PC and Inventor was impacted!!! When I disabled certain services, not only did my reboot time improve, Inventor seemed to perk up as well.

If you have Windows 7, you want to launch a command called [MSCONFIG](#) and checkout the Startup tab/operations. If you're using Win 8/8.1/10, you will need to review the Startup tab via the Task Manager.

Disable only those services that are not required but make sure you fully understand what these given service actually do. Remember you're not trying to take down your machine. **If you don't know, ask or search what the service does.** But also understand your company policies may

not permit you to make these type of changes and you may need to contact your IT group to resolve this.

- *Autodesk Content Service (AutoCAD 2015/2016 version)*

When AutoCAD 2015/2016 or any of its vertical products was installed (on your machine), a service called “[Autodesk Content Service](#)” is started. But there are cases where this service fails, constantly crashes or takes away the cursor focus every so often. When this service acts up, I’ve seen performance issue or locks up within Inventor. Not everyone has experience these issues, but if you’re not using this function within an AutoCAD product or the service is still there (even after your uninstalled these versions of AutoCAD), I strongly suggest that you [disable it](#) from starting.

- *Windows Bells and Whistles and those little neat useful gadgets/apps*



As a CAD Manager at my last job, when a user complained about performance on their machine, one of things that I would look at, is those extras Window functionalities (those bells and whistles, gadgets, and apps that were running). Yes there’s a lot of cute/unique things that you can do with these items. But those bells and whistle could have a dramatic impact on Inventor performance. Most of us are probably guilty of having these things running in Windows or on our desktop. But is it really important to see shadows under your Windows, animated controls, updated stock prices, Facebook postings, sports scores, or what the weather is currently outside your office? I’m not here to tell you that you shouldn’t be using these type of things, but perhaps you just want to disable these gadgets, widgets, notifications, and apps as a test to see if there’s a difference or not. But I do recommend that you configure your [Windows visual effects for performance](#) and switch your Windows theme to “Classic”.

- *Out of Date Drivers*

Usually when a new version of Inventor is released I start hearing about issues with Inventor graphics when the user is panning/zooming/orbit and rendering. The first thing that I always check is the graphics card driver. Not always, but it seems that people update Inventor and/or their operating system and never think about their video card. It’s important that you have a video driver that meets the requirements of the Inventor version you’re using. In addition I always tell people, don’t rely on Windows telling you it’s up to date, go directly to the vendor’s

web site and see what driver versions they offer. But never install drivers that are in beta testing mode or in a non-released state. This holds true for any other device that Inventor relies on. Make sure you have the correct or latest driver.

*Helpful Tip #1:* Autodesk does have recommended information [here](#) based on the Autodesk application being used. But just because a certain hardware is not listed or certified or not recommended, doesn't mean it doesn't function in the version you have.

*Helpful Tip #2:* I wanted to pass this recommendation along because when drivers are updated, sometimes it ends up failing. Always and I mean always create a [Windows Restore Point](#) before updating any drivers such as your graphics card.

- *Dust bunnies and cob webs*

Yes this solution may seem like a joke but dust build up in your computer can certainly impact Inventor Performance. Although rare, it's something that you should consider especially when Inventor is running on a laptop. When is the last time you checked the fan ports and made sure they were clean?



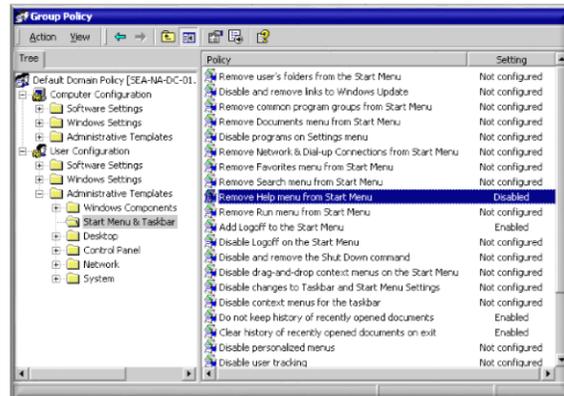
If the proper air-flow is not getting through, the electronics within can start to over-heat and when this happens, your computer may throttle back to reduce the overload related to the increase in temperature.

- *Hardware Failure or Malfunctions*

Once again another rare case, but it's something that plagued me for a few weeks one summer. Most days I was able to work with my laptop and Inventor without any issues. Certain days my computer performance came to a screeching halt. I started to notice a pattern. When the summer temperature would go up (so did the temperature in my non-AC office), my computer started acting funny. After I installed a computer temperature monitoring software, I noticed when the CPU reached a certain temperature, my machine performance went downhill. I realized at that moment something internally in my PC was wrong and the culprit actually was related to one of the cooling fans had failed (and my laptop was unable to maintain a proper internal temperature).

- *Group policies*

Just like Windows updates, another item you may not have any control over is the “Group Policies” that are being applied to your machine.



Group policies play an important role in your overall organization, but certain policies may impact your Inventor performance as well. For example... Group policies may get “hung-up” while being pushed down to your machine and require a refresh or like at my last job... Our Inventor users were experiencing slow starts, lock ups, odd graphical issues, and crashes when they came back from their lunch and/or extended meetings. Our corporate group policies designed to lock the machine after so many minutes of inactivity, hard-drive powering down (inactivity) to save energy, and a few other policies lead to these issues. Yes, our global IT group was preventing someone from walking up to an idled machine (and doing something) or in an attempt to save a few dollars in energy cost, but these policies did end up impacting the Inventor users.

Like I said under the Windows update section... You may not have any say or control in this matter but it’s something that still needs to be checked/tested. Sit down with management and your IT group and discuss the possibility of “Group Policies” impacting Inventor performance.

- *User’s Windows/Inventor Profile*

Corruption in your Windows/Inventor profile can certainly impact your Inventor performance and/or cause issues within Inventor. I’ve had numerous case over the years where the Inventor user’s profile needed to be switched back (“[reset](#)”) to the out of box configuration to resolve the issue. When I hear of certain Inventor issues, I often instruct the end user to have someone else log onto their machine and see if the same issue is repeatable. If not, that’s a strong indication the user’s Windows/Inventor profile is corrupt in some manner (and you’ll need to run the [Inventor Rest Utility](#)).

Once Inventor is restored to the out of box configuration, ensure the issue has been resolved before configuring Inventor to your requirements. Taking it a step future, I’ve had a couple of

cases where restoring the configuration that was backed up prior to resetting Inventor, brings back the same issue (which points to something wrong with your custom configuration).

Although rare, the actual overall Windows profile for the user can be corrupt as well. However under this condition, the user should be experiencing other “issues” outside of Inventor. If you do suspect your Windows user’s profile is the culprit, it may be time for your profile to be wiped clean and rebuilt from scratch.

- *Company’s Network/Infrastructure*

How your company configures your machines, its network and/or infrastructure can play a role in Inventor performance issue. Certain Inventor task may have “behind the scene” processes that could be blocked or restricted by your computer/network/infrastructure and it may hang Inventor, not respond (in a timely fashion), or just plain not work properly at all. Other problems such as continuous “hiccups” in your network traffic, invalid mapped drives, offline printers, and etc. can also play a role in this.

One simple test, is to disconnect from your company’s network/infrastructure and see if it makes a difference when you launch and/or work within Inventor. You should also review your [Windows Event logs](#) for warnings and errors that may have occurred around the same time as your performance issue did.

Conditions like these are sometimes hard to pinpoint and/or get resolved if this is happening to other users as well. Most likely, further testing within your organization is probably needed to determine the root cause associated to this. Process monitoring tools such as [Windows Process Monitoring](#) could aid in this type of situation.

## **Performance Issue #6: Working from a network location (or the Cloud)**

- *Design Data folder shared on network*

I’m not a fan of sharing or placing my Inventor design data on a network drive. But at the same time I’m not telling you it shouldn’t be doing it either. I get that it’s needed when others need to access the same data you have created/modified. But I’ve also come across a few that have never modified anything within their design data folder and they still think it should be stored/shared on the network.

*So why am I not a fan of this?* When the design data is placed onto a network location, at some point you will run into a non-responding situation within Inventor when loading or modifying your model. In addition the more users that are accessing this shared data, the more time these users will be faced with the non-responding scenario. Let me explain...

Let’s say you have three (3) Inventor users and each one is trying to load a large model that has numerous parts with threaded holes in it. Every time a part or assembly is opened in Inventor and there’s threaded information within these components, Inventor has to open and close the

threaded spreadsheet (located in the design data folder) for each threaded hole it comes across. Now think about this... If all three Inventor users are doing this at the same time. Each user has to wait for the other to finish (using the spreadsheet). Meaning it leads to a slower load time for your Inventor model. Kind of like an entire office trying to print at the same time to a single printer (first come, first served).

If you can, store your Inventor Design Data locally.

- *Models stored on the (local) network/one-drive/cloud*

I and others will tell you, Inventor was never designed to work with models stored on a network (or in the cloud). But some organizations are not permitted to store files locally due to the fear if their machine fails, the information could be lost. I also hear from users, it's not our network causing the issue, we have a powerful infrastructure so it must be something wrong with Inventor.

Under this scenario, your weakest link is mostly likely your network (or the cloud). When I tell end users, as a test, copy your models locally and see how it performs. For many, it's like day and night and they are impressed with the performance they witness. If possible, work locally on your models by moving (or coping) your entire model back and forth using a file management tool or implement some form of document management system (like Vault) to aid in this process. Another suggestion, store only those commonly used files on your network to limit the amount network traffic that Inventor has to deal with.

For those who must work across a Wide Area Network (WAN) like at a remote site or who VPNs (Virtual Private Network) into your company network from home (or other locations), always expect you will experience some form of performance issue within Inventor, especially when you need to work on models that are stored elsewhere.

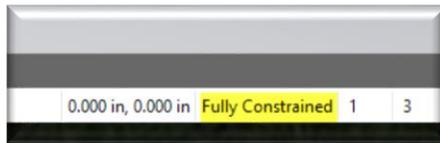
For WAN communications, there are utilities like "[Riverbed](#)" that can speed up this process due to how it recognizes what information is new and what has already been communicated to the end user's machine. In the end, it may be worth having the conversation with IT and management to discuss the implementing of these utilities if the "speed/time" between sites is crucial.

### **Performance Issue #7: Like working with stone**

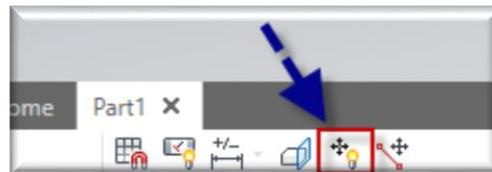
Does it feel like when you make a change to a part and/or sketches it's like you're using a chisel and stone (not easy to accomplish) to make the modification? Certain things can impact this and here are some things you should consider...

1. Don't
  - a. Take shortcuts because it's easier for you.

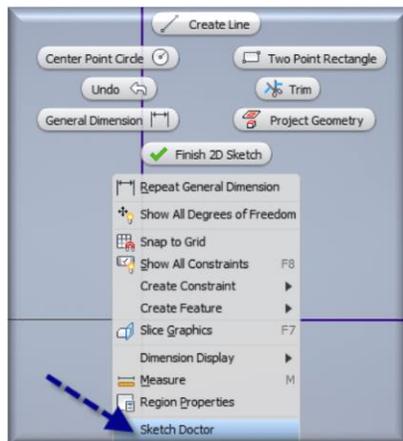
- b. Over complicate your sketches, keep them simple (KISS Principle). Create more, simpler sketches. This method will also lead to an easier way of troubleshooting when issues do arise.
2. Think about how the part may be created and try to model it that way.
3. Use the [Inventor "Born" technique/method](#)
4. Project only the necessary geometry. Perhaps in your Inventor Application Option/Sketch tab you want to turn off those auto-projecting options
5. Sketches should always be properly dimensioned and constrained
  - a. **Note:** Although you want to achieve a fully constrained sketch



, certain sketch geometry like center and construction lines don't necessary need to be fully dimensioned and constrained.



- b. Check your degree of freedom within your sketches. You can also select a sketch entity, right mouse click/display degree of freedom.
  - c. Bring up the "Sketch Doctor" (right mouse click within sketch)

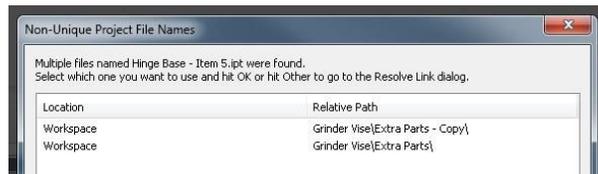
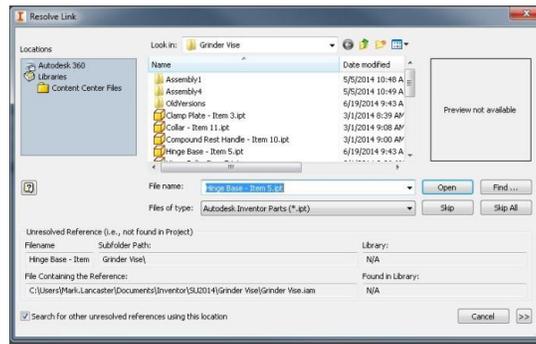


and analysis your sketch for issues.

- d. Using 2D Geometry from other CAD applications may be useful and a timesaver, but think about what information is being used and how it's used.
6. Use
  - a. Modeling functions like fillets, chamfers, holes over sketching them.
  - b. Feature patterns instead of patterning geometry within your sketches
  - c. Simpler modeling tools over complex ones.
7. Resolve issues via the Design Doctor.

8. Periodically perform a Manage/Rebuild All on your model.
9. When troubleshooting, break down your model tree by relocating the EOF (end of file) marker in the browser to determine what's causing the issue.
10. Check the integrity of your model using the "[Body Integrity Check](#)" tool.
11. Certain times it may be easier to remodel "stuff" instead of dealing with bad or corrupt data

### Performance Issue #8: Why am I seeing these dialogs?



#### ***Certain reasons are***

- Component files are located all over the place.
- Files duplicated in different folders
- Files are stored under a Windows user's profile
- Commonly shared files are stored only on certain PCs
- Entire model is located on the network
- Content center files' location is different for each user
- Different components having the same (file) name
- Files were moved using Windows Explorer
- Long file names and/or files are buried deep in a folder structure
- An improper project file configuration.

#### ***What are the solutions (addition info [here](#))?***

- Don't take shortcuts, think about your model.
- Don't use the "naming" of a folder/component to describe the entire information about you component.

- Keep your workspace, hard drive or network location organized and clean.
- Don't store files where others don't have access to them.
- Create a shared location for your commonly used parts and assemblies.
- Don't store your files all over the place, create a structure and maintain it
- If you're required to store your models on the network, consider moving/copying it down to your local hard drive and working on them there.
- Ensure your content center setting is the same for each user.
- Create a shared location for your content center components if you're using the desktop version of content center
- Develop a file naming convention for all components
- Keep your folder structure simple and small.
- Have a properly configured project file.
- Avoid using Windows Explorer to rename/relocate/copy (design) portions of your model. Use Vault, iLogic Design Copy, Design Assistance, or Save/Replace Component instead.

### ***Why do these dialogs keep appearing every time I open my model?***

This question has been asked of me many times and I always hear... I already selected the correct file when I opened the model (and saved the changes). Why when I open the model again, the issue comes back?

Just because you pointed Inventor to the resolved information in these dialog, if you don't actually fix the root cause, Inventor is just going to see the same issue upon open. Correct the issue and these dialogs will not appear again.

### **Performance Issue #9: Project file always seems to be lost and can't find my models**

It's important to have a [properly configure project file](#). If you find that it always seems like your project file can't find your files, here are some things to consider.

1. Is the active Inventor Project the correct one you require?
2. Define a workspace in your Project file.
3. Use the workgroup search path for components outside your workspace
4. Ensure your folder options are set to the proper location.
5. Corruption or an incorrectly formatted (XML) Project file could be the root cause of you issue. Recreating the project file from scratch may just end up resolving your issue.

*Helpful Tip #1:* Maintaining too many Inventor project files in the Inventor Project Editor may impact Inventor performance or cause other issues. Although there's no known limitations, maintaining thirty (30) or less projects is preferred. Projects that are no longer being accessed should be archived and removed from the Inventor Project Editor

*Helpful Tip #2:* Although the "Workgroup Search Path" option is a useful tool in providing search path information for components residing outside the Inventor workspace, too many paths listed under this option can degrade your Inventor performance. Limit the number of search paths to avoid performance issues.

### **Performance Issue #10: All the eggs in one basket (for an assembly)**

When we are quickly designing something, we may take this approach just to get our design down on "paper". But outside of that, I believe it's not a proper workflow you should be using. In doing so it could lead to Inventor issues/performance down the road based on the complexity of your model. Perhaps you would want to do this instead:

1. Demote, simplify or break up your overall assembly into smaller chunks.
2. Use Inventor's [BOM structure](#) types to control your overall Bill of Materials.

### **Performance Issue #11: It's all in the details**

Providing exact details in your part or assembly is a powerful communication tool in 3D modeling. But too much details will most definitely impact your performance. Remember the help desk case I told you about earlier. This was the main cause of Fred's issue. His bolting hardware was actually shown with the threads modeled. Yeah it looks great but in an overall model where there's hundreds and hundreds of these components, it degrades your Inventor performance.

Ask yourself, are the details really necessary? If I add it, will there be "value added" to my model by doing so. Perhaps you need it at the part or assembly level to convey manufacturing or an assembly process. But in other parts of your model where this part or assembly resides, is it actually necessary to see all of this fine detail at that level?

Here's another example where too much details is just that, a "**performance killer**". About a year ago I had a help desk case where an entire department was struggling with an overall large model they were working on. A lot of components, but I would say 70 to 80% of the model was just duplicated parts. No matter what they did, they were always waiting around for Inventor to finish. Although the design group didn't invest into high end machines, their PC specs was mid-range/average

Overall when you looked at the model, it didn't really seem like Inventor would have any troubles with their model. But when you zoomed in you started to realize where Inventor was

struggling. For example, a required sub-assembly had numerous bolts, screws, washer, nuts, rivets, material stitching and many other (purchased) components with in. At that sub-assembly level it was necessary to show this detailed information for assembly/purchasing requirements. However at the top level assembly, this given sub-assembly was than patterned 100x times. Was it really necessary to show all this detailed information 100x times? Remember I had to zoom in to see the actual detail. Do you really think it's going to show up in any hardcopies you make? Would it make a difference? Not only was this detailed information duplicated, someone else had actually modeled a wire screen mesh on a few assemblies. Yeah it looks great but doing that more than 1x time is just going to impact you. Taking it to another level ... They even modeled their numerous stair treads to look like this:



In the end, if you need the actual details at the part or at a certain assembly level to convey something, create a simplified version (iPart, iAssembly, derived, shrinkwrap, shrinkwarp substitution, appearance override and etc) that will be used elsewhere in your model structure.

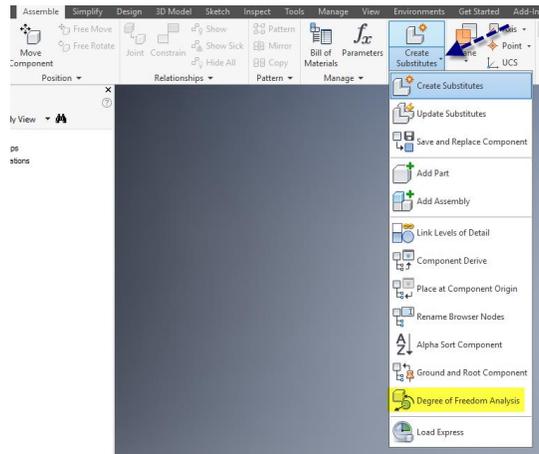
**Performance Issue #12: My assembly must have magically powers, components always seem to be shifting and twisting.**

Should this be considered an Inventor performance issue? **Yes it is...** Meaning I have to stop and fix my model. When I see this type of issue, it usually tells me some form of improper workflow is being used to bring this model together.

Is your first component

1. The foundation of your design?
2. Is it grounded and/or properly constrain to the origin planes?

Have you even checked your components' degree of freedom? Did you know there's a "Degree of Freedom Analysis" tool within Inventor? It's located under the Assembly Productivity section of the menu.



**Helpful tip:** When constraining components within your assembly, always define component #1 information as the component that's being moved/relocated.

### Performance Issue #13: Is the Doctor in?

If you broke a bone in your body, do you just forget about it and perhaps come back later to fix it? I don't know how many times I've been on a help desk call about an issue with Inventor and the little "red cross" located next to the quick access toolbar (QAT) is lit up. I ask... *Why haven't you fixed the issue?* Most times I get, no biggie is a failed sketch, or constraint, and etc.

Now I can't tell you that every single time the Design Doctor is active, it's related to Inventor performance but it's not something that you should ignore either. Don't procrastinate about the Design Doctor. Take the few moments and resolve the issue (you'll thank me later).

*Can there be an issue even though the Design Doctor is not lit?* Yes and I've seen this a few times. I always inform users they should always perform a (Manage tab) "Rebuild All" on their model every so often. Sometimes when a "Rebuild All" is performed, guess what happens? The Design Doctor lights up...

### Performance Issue #14: Inventor Constraints are like your Assembly BFF (Best Friend Forever)

Although a true statement, did you know constraints are also your worst enemy? Constraints will consume memory and too many can definitely bog down your model. In my opinion if a given constraints occurrence is getting into the high double digits or actually in the triple digit (occurrence) mark, it may be time to start rethinking your model structure.

Rethink by simplifying constraints as much as possible. Does a component or sub-assembly need to be fully locked down?

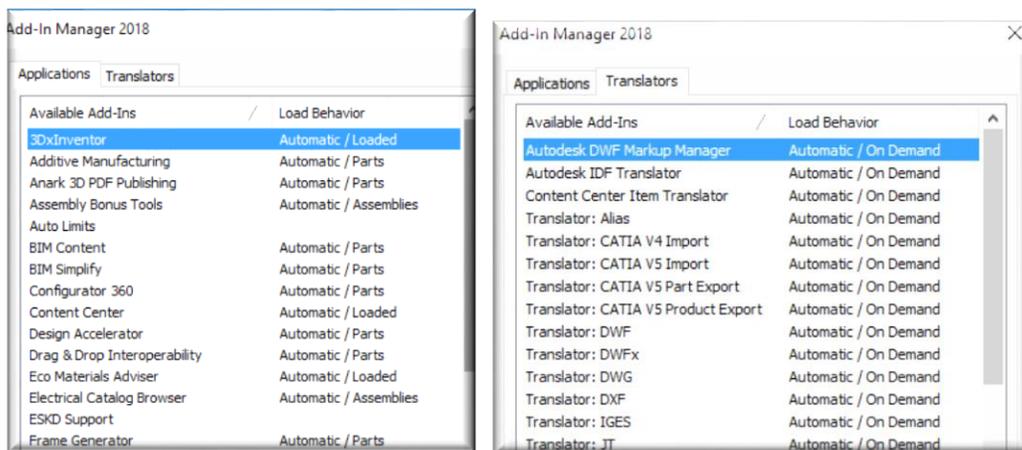
Although I'm not a fan of doing this, you may want to ground your components or suppress (constraints) to limit them. Another method to control or limit the amount of constraints used, is called "[Skeleton Modeling Technique](#)". A couple of alternative method you could also try:

- Break up your model into smaller chunks and control these smaller chunks using the Inventor [BOM structure](#).
- Starting with Inventor 2016 along with an existing 2D drawing, you could apply joint constraints use the [DWG association/underlay or enhanced DWG workflow](#).

### Performance Issue #15: Those helpful Inventor Add-ins

- *Loading all of them or not enabled at all...*

Inventor out of the box has numerous add-ins/translators. Certain ones load automatically, others based on the environment you're in and some are per "on demand".



Perhaps others have gotten blocked or unloaded for some apparent reason.

But I've always inform end users

- If you're not using certain add-ins or translators uncheck their load behavior and save your resources for other crucial items you may require.
- Don't assume... Confirm the status of these add-ins and make sure they have the load status you require.

**Helpful tip:** When using Inventor Professional and working with an assembly, by default the Inventor Routed System (Tube & Pipe) module is loaded. If you don't use routed system at all, I recommend that you disable the load of that module. When it's loaded, the rules associated to routed system are in- effect when you start placing tube & pipe components from content center into your assembly. Yes you may not be creating an actual routed system model but the rules

still apply and restrict what you can accomplish with these tube & pipe (content center) component (see this [article](#) for additional information).

- *3<sup>rd</sup> Party Add-ins...*

Have you ever visited the Autodesk [Inventor Exchange App Store](#) or found an Inventor add-in that supports your organization? There's a lot of great add-ins out there and it continues to grow on a daily basis.

One of my performance items I frequently review, is the Inventor add-ins that are currently loaded. When I see add-ins that are not part of the out of the box (OOTB) Inventor installation, I always ask can we disable and see if it makes a difference. In some cases it does and other times there's no impact. My advice, disable all non out of the box add-ins and see if it makes a difference. If it does, add them back one (1) at a time and find out which one impacts your performance.

Also you may want to check the Autodesk App Manager (Tools ribbon tab) to see if any of these "Inventor Exchange" add-ins do require to be updated or you can use this interface to uninstall those that are no longer needed.



Prior to Inventor 2018.1, this tool used to be called Exchange App Manager

- *User Defined Add-ins...*

There's a lot of Inventor programmers out there creating unique add-ins for their organization. I'm not a programmer, but I've always inquire when I see these type of "home grown" add-ins and ask what they do. As I stated above (regarding 3<sup>rd</sup> party add-ins)... Can we disable and see if it makes a difference.

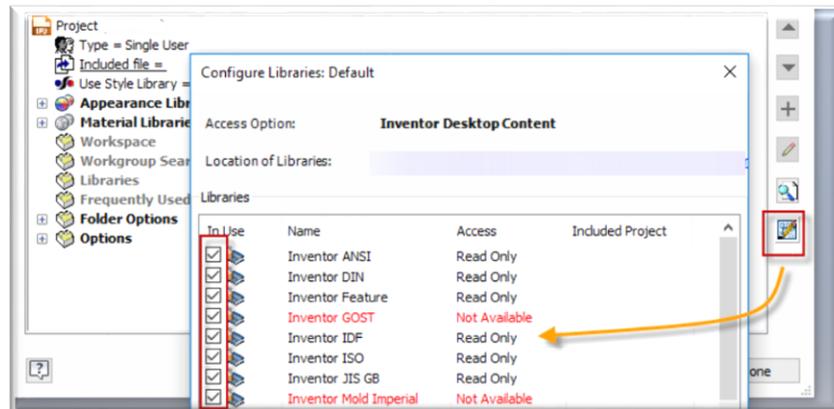
- *Additional Autodesk Add-Ins...*

Autodesk add-Ins like Mesh Enabler, Nastran and etc. are not actually developed by the Inventor Development team. Sometimes they could end up being the root cause of your problem. Although the Vault add-in is under the Inventor Development team, I've seen cases where something has happened with its installation and this add-in ended up causing issues within Inventor. Like with any add-ins not part of the Inventor OOTB configuration, disable them to see if they are impacting your Inventor performance.

### **Performance Issue #16: Content Center is slow for me...**

- *Loading all specification...*

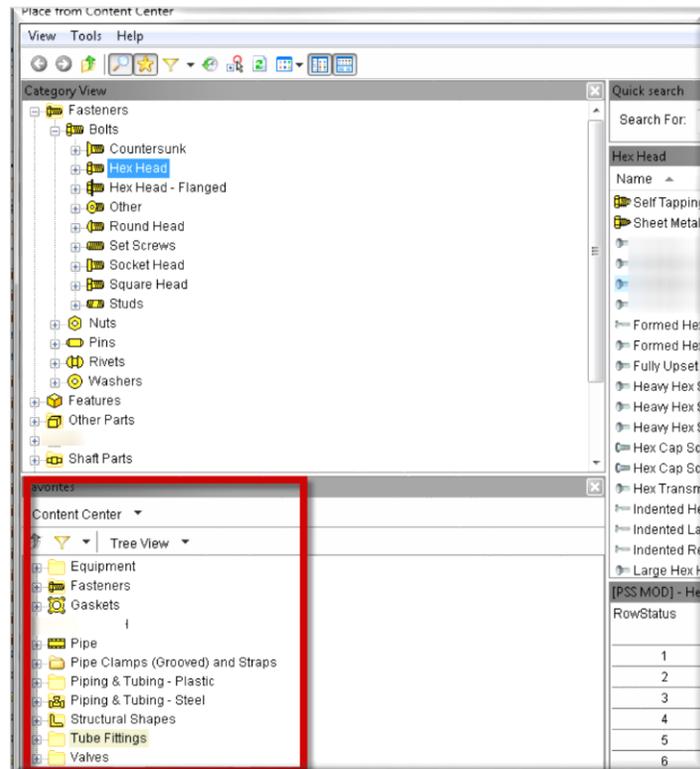
Depending on how you installed Inventor and the specifications you selected at that time... But ask yourself, do I need to load every specification for content center? In the past, it was important to control the speed of content center by limiting what libraries was being loading via the active Inventor Project Files. Now-a-days it's not as important but loading all them is still taking resources. Limit them in your active Inventor project file.



- *Content Center Filters...*

When you start filtering within your Content Center, Inventor has to process this in order to load the correct entities per that filter. Meaning every time you launch content center, the filter rules have to run.

- *Content Center Favorites...*



Creating a favorites list in Content Center is a useful thing and a time saver when you need to access those commonly used content center components. But did you know the more favorites you assign, the slower your Content Center will load? Your favorite list works in the same manner as the Content Center Filter, but Inventor has to process each favorite item one at a time which leads to longer load times.

**Helpful tip #1:** For those who are using Inventor 2018, I would recommend that you don't apply the Content Center Favorites as a separate browser. Every time you start or open an assembly, it has to load each favorite into memory again.

**Helpful tip #2:** Your favorites are stored here: C:\Users\

### **Performance Issue #17: Vault Client Build # is the cause**

Earlier I was referring to why you shouldn't apply Autodesk updates just because they are there. Here's a reason why applying a Vault Client update could be impacting your Inventor performance and the interaction between the two (2) applications. In the past when Autodesk didn't have the Application Manager or Desktop App interface, applying any updates was more of a manual process and most times the Vault client didn't really get updated on the end user's machine until it was required.

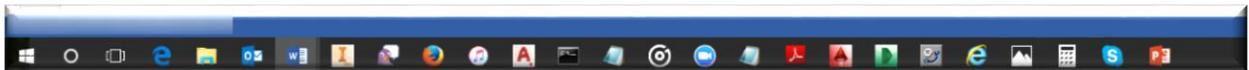
Now with the Desktop App (unless restricted by your subscription's Contract Manager), users are just applying them because they see these updates in their interface. However they may not understand that a Vault Client build # cannot be newer than the build # of your Vault server. Now I'm not saying if this happens, it's the main reason for your Inventor performance, but I've seen odd things occur (in Inventor & Vault) when users do update before the (Vault) server is. Most times when I've come across this situation and the Vault client update is "rolled back", the issues seem to vanish.

My advice, either restrict Autodesk updates or inform your users not to install any Vault Client updates until instructed to do so.

### **Performance Issue #18: Cloudy with a chance of rain.**

No weather doesn't impact your Inventor performance but dealing with "Cloud" services/storages could. There has been cases where certain "Cloud" interfaces have caused delays, non-responding, and etc. within Inventor. If you do have any cloud services running on your machine, I would recommend disabling them and test to see if that's the source of your problem.

### **Performance Issue #19: Limit your running applications.**



Throughout my day I'm supporting many different customers and by days end, you can see I have many applications running. Does that image represent something like on your machine, (numerous applications running)? Remember each running/loaded application is consuming some percentage of your resources even if it's minimized. In the end, limit them. Perhaps you want to reboot at some point in your day just to release those resources and start from scratch.

### **Performance Issue #20: Working with large assemblies**

Although it seems lately that Autodesk has been implementing improvements regarding large assembly performance, certain large model structures still impact Inventor performance. Here are my suggestions when it comes to working with large assemblies.

- Review the [Autodesk Inventor 2014 Large Assembly Performance Guide](#). Yes it's a 3 to 4 year old guide but still has a lot of great information that's even pertains to Inventor 2018.
- Limit the number of other Window applications that are currently running
- Unload unnecessary Inventor add-ins
- Invoke defer update and manually update when you're ready.
- Simplify, Simplify, Simplify
  - Create [Level of Details](#) (LODs)

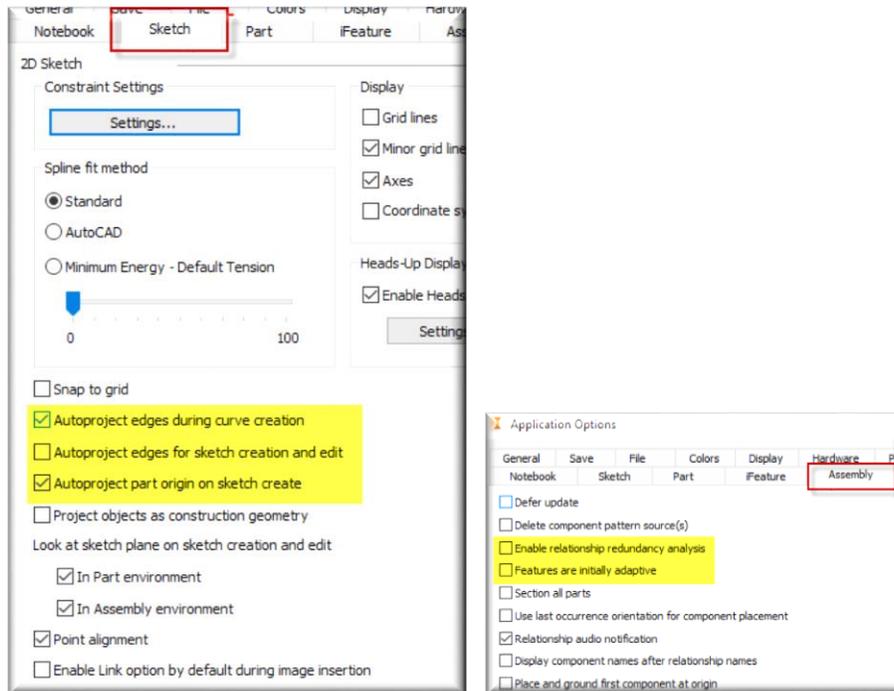
- Use [Shrinkwrap](#) and/or [Shrinkwrap Substitution](#) or [Derived Assemblies](#)
- Create [View Representations](#) (View Reprs)
- Use the [BIM Exchange simplified workflow](#)
- (If possible) Use Inventor Appearances over actual details.
- Use [Express Mode](#) (Inventor 2014 or newer)
  - *Helpful Tip:* “Express Mode” was introduced in Inventor 2014 but had its limitations. Due to these limitations, many of you may have overlooked this function and today don’t even use or consider it. When Inventor 2015 was released, “Express Mode” was updated and is now a useful function when working with large assemblies.

### Performance Issue #21: My Inventor Family Tree



Just creating a simple Inventor part and assembly in Inventor creates an “Inventor Family Tree” or the parent to child relationship. When you start building that family tree, over time it could impact your Inventor performance. One thing you need to consider in this family tree is the relationship you build using adaptivity. Although it’s a powerful tool and aids in designing, too much of it or keeping it active throughout your entire design process can degrade your Inventor performance. Here are some key bullets to keep in mind when working with adaptivity.

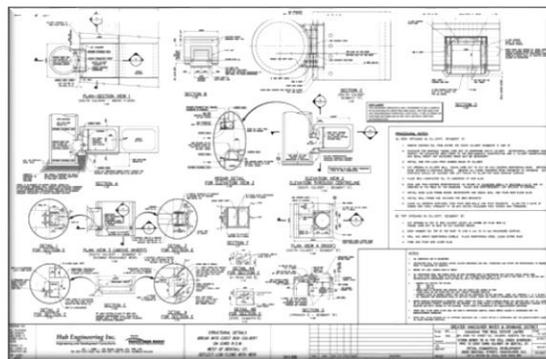
- Is it really necessary? Think about the parent to child relationship you’re creating.
- Turn it off when it’s not needed or when you’re done with your design.
- Disable its functions in Inventor Application options so it’s not built automatically.



**Performance Issue #22: Drawings tell the story...**

- 20 lbs worth of information crammed into a 5 lb drawing

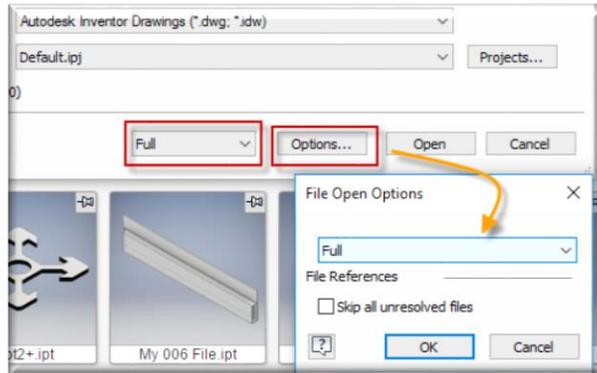
Do your drawings resemble something like this?



Is it a pain to load or work with?

It may be useful to put all your information into a single drawing sheet/single source file. But think about when the drawing is loaded or needs to be updated. Is it really necessary to load all the information into memory just to make a simple change to one of those details?

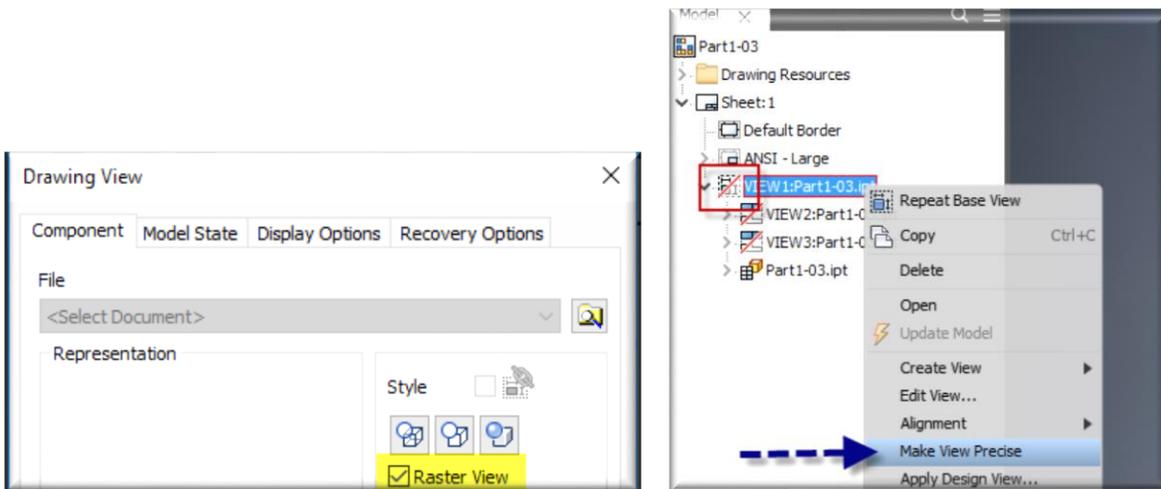
I might suggest that you break up your drawing into smaller chunks. –OR– Use the file open functions to defer updates and perform fast opens.



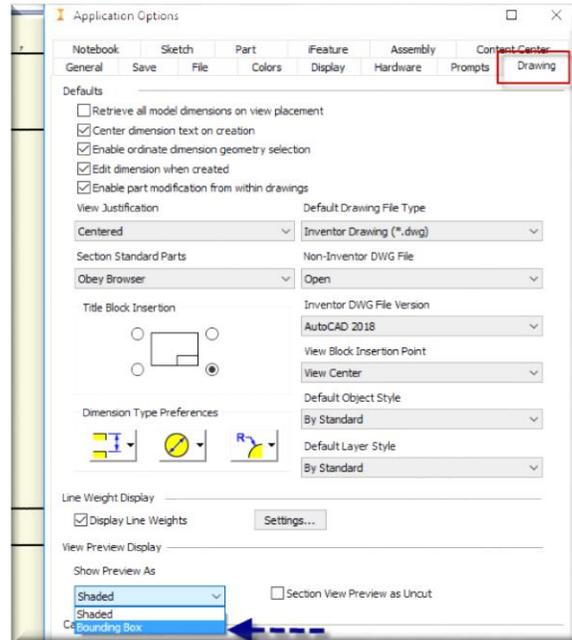
- *Drawing Views*

When starting a (new) drawing, is it crucial to see every fine detail of your model? For me I would be focusing on trying to layout my drawing and then later on, go back and finalize the look of it.

A couple of settings in Inventor could impact your Inventor performance when creating your drawings. I recommend that you initially place your drawing views as a raster view and keep them in that state until you're ready to plot.

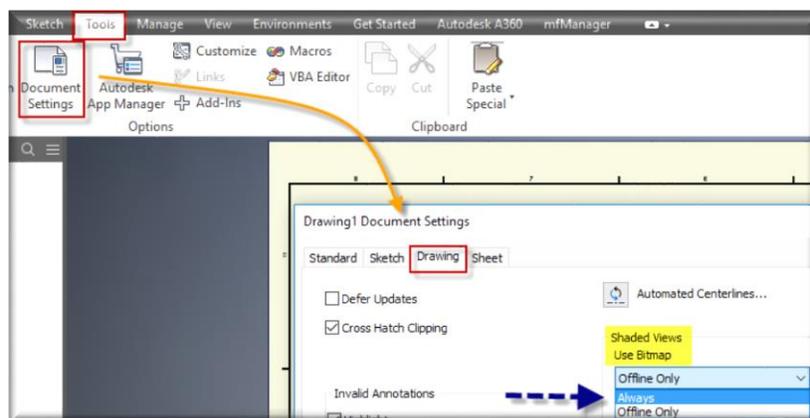


The other setting to consider especially when work with large assemblies, its to set the view preview of the model to partial or boundary box (depending on the version of Inventor) in Tools/Application Options/Drawing tab.



- *Shaded Views*

I'm a firm believer, shaded views on a drawing is a useful communication tool. But shaded views will impact your Inventor performance if you go overboard with them. Limit them and/or break them up into different drawing files to limit the impact on your performance. Another thing you may have overlooked in your drawing template, is the shaded view option under the drawing document settings. I recommend that you "always" use bitmaps for your shaded views.



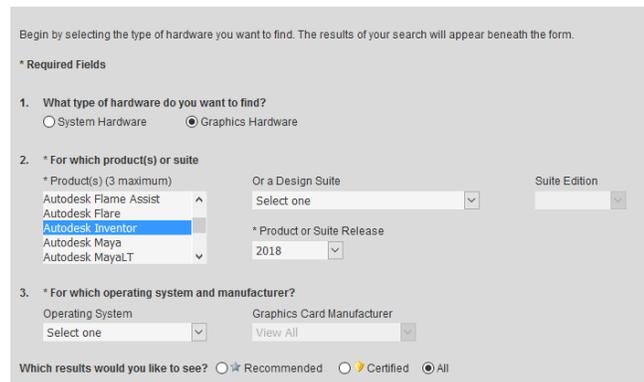
**Performance Issue #23: When panning, zooming or orbiting around your model, does it feel like Inventor responds like a turtle?**

If this is the case for you, here are some recommendations to assist you.

- Virtual Memory:
  - Don't allow Windows to manage it for you.
  - Depending on your Inventor model configuration and/or if you have limited RAM, I would set your Windows Virtual Memory to the recommended settings.
  - If possible redirect your virtual memory information to another (local) drive.
  - If your system has amply RAM, you may be able to disable your "Virtual Memory" (or set it to the min of 800mb) for better performance. However RAM usages should be monitor and kept below the 80% mark.
- Your graphics card driver plays an important role. Many times I'm come across users who have a 2 to 4 year old driver. Make sure your card and driver (version)

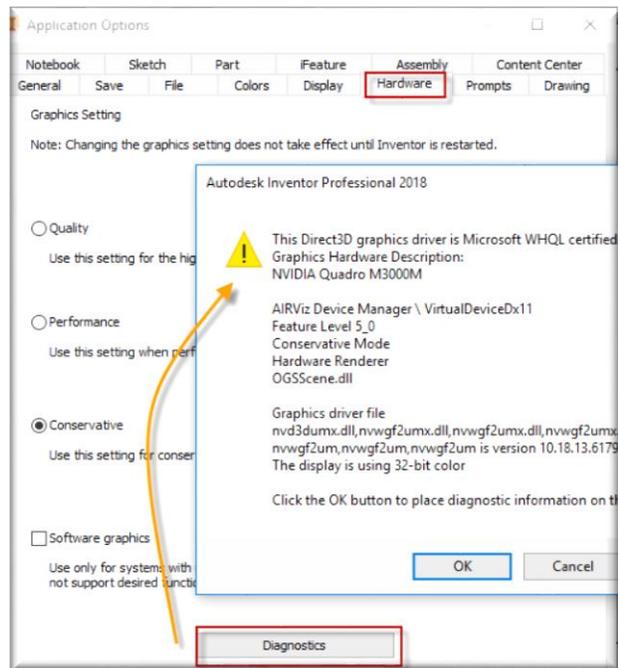
**Find Recommended Hardware**

Use this tool to find recommended system hardware or graphics hardware and drivers. Autodesk tests system hardware, graphics hardware and drivers for a number of Autodesk products and suites. Please note that not all Autodesk products participate in hardware certification.

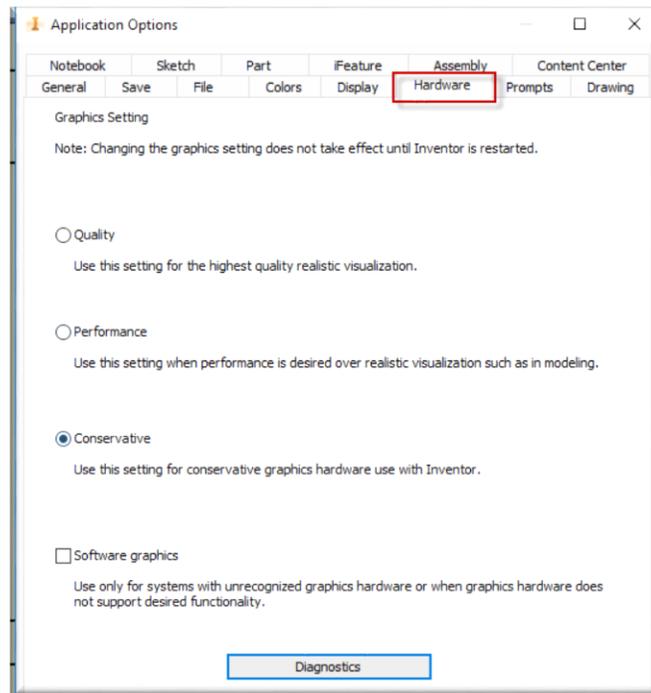


- **Meets the [requirements](#).** Although I'm a big fan of using certified and/or recommended video cards, just because your card is not listed as being certified and/or recommended doesn't mean it will not function. In addition there has always been a back and forth (heated) discussion about CAD or GAMING cards and which one is better or which one to use. I'm not here to tell you which one you should be using except the card should meet the requirements as outlined in the [Inventor system requirements](#). But I do recommend that you stay clear of the on-board INTEL cards and go with something with more power.
- **Correct one being used.** Many times PCs or laptops that we receive may have some form of graphics card built into the motherboard. In addition we may have a secondary card that has more power. Just because you have a secondary card doesn't mean Inventor is using it. To confirm, start a new part and select

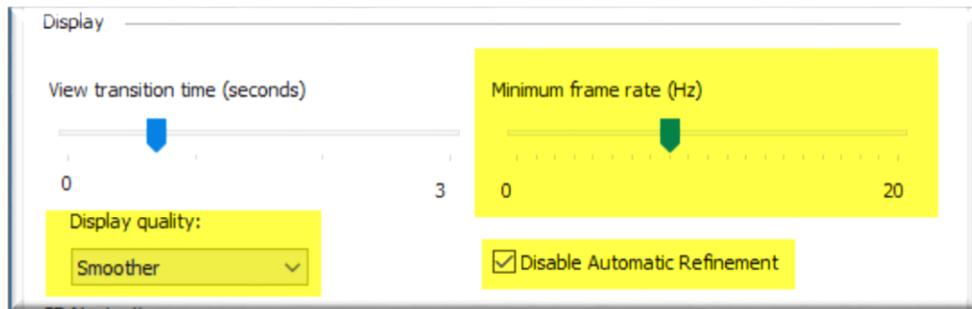
the diagnostic button located in Tools/Application Options/Hardware tab.



- **Driver version is per [Autodesk recommendation](#) or newer.** Never rely on Windows telling you your video driver is up-to-date. Go directly to the Vendors site and ensure you have the latest.
- **Use Performance or Conservative graphic settings.** I know when we spend a great deal of money on medium to high performance graphics card we want the best outcome in Inventor. However a lot of times I see the graphic settings in Inventor as always being configured as “Quality” and the end user is faced with certain graphical/performance issues. I always tell people try the lowest (Conservative) and test for a few days to see how things work. Then repeat the process by changing the option to Performance, followed by Quality. Find out which one works for you and don’t assume just because you have a given video card that “Performance” or “Quality” option is the correct one to use. In addition if you have an outdated video card and there’s no newer driver for it, you may want to start looking at the option to use “software graphics”



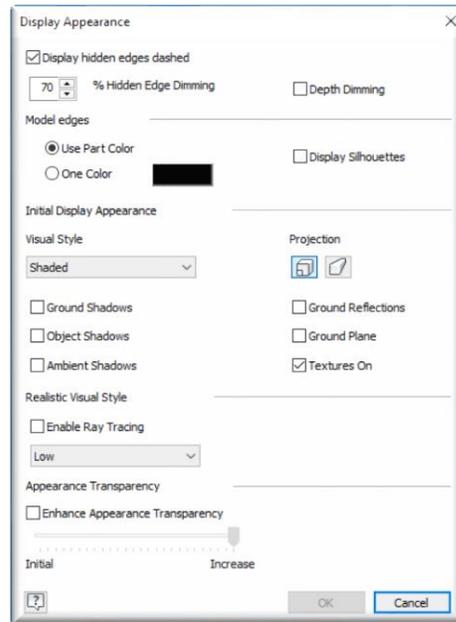
- *Helpful Tip:* Anytime these graphic options are changed, a reboot of Inventor is required in order for the change to occur.
- Are you using a 3D Mouse (like 3D Connexion)? Just like the graphics driver, make sure you are using the latest driver and another thing you may want to consider is calibrating the device through its interface.
  - *Helpful Tip:* I have seen my share of odd things that occur with the 3D Connexion device within Inventor/AutoCAD. If you fall under this situation:
    - Remove and reinstall your device driver
    - Unplug the device and launch Inventor and plug it back it.
    - If someone else has the same device, swap it and see what happens.
    - Contact the vendor if there's no solution to your oddity.
- Limit the number of other Window applications currently running.
- Simplify components and reduce the detail.
- Consider the display options (Tools/Application Options/Display tab)



- **Minimum Frame Rate (HZ):** This settings is how slow or fast Inventor repaints the screen/model when panning, zooming, or orbiting your model. For me I've always set it to zero (0) so I have no "repaint" occurring and thus my model is never jumpy/lagging when performing panning, zooming, and/or orbiting functions. The downside of it, I don't see the medium to fine details on my model until I completed these task. Most time I don't really care but I could see the impact if you do require it. In addition while discussing this topic at AU last year with the Inventor Development team, they recommended with the newer versions of Inventor to set this frame rate at 20 Hz. In the end find out what works for you.
- **Display Quality - Smoother, Smooth, Medium, and Rough:** Yes we always want the best "look" for our models. Sometimes it's just not possible and you should test to see which display option gives you the best Inventor Performance. Just because "Smoother" looks the greatest, do you need to see this "look" each and every time? What's the impact on your Inventor performance if you do?

In my opinion, I think many of us get hung up on thinking they need to see a realistic model each and every time they are modeling. Yes there's cases where it's necessary but does it really matter when you're designing if the hole edges are a little distorted? For me, I rather save my resources when designing and then return later on to make that "artistic rendition" of my model.

- **Disabling automatic refinement option in Inventor 2016 or newer.**
- **Display Appearances:** Whether your display appearances come from Application Options or the Document Settings, you need to determine which settings work the best for your organization. Like I said earlier, just because a certain configuration looks the greatest, do you need to see this configuration each and every time? What's the impact in doing so?



### **Performance Issue #24: 3<sup>rd</sup> Party CAD Files**

Although this is not something that happens every time you receive a 3<sup>rd</sup> party CAD file. But I have seen Inventor performance issue related to 3<sup>rd</sup> party CAD files due to bad “data” and/or containing way too much detail in the provided file.

For example, I had a customer contact me once and said their overall Inventor model was slow to response and they were also seeing odd results within their drawing. Upon review, it was discovered a 3<sup>rd</sup> party CAD file (converted to an Inventor part file) was around 40mb in (file) size and contained way too much detail. Once the part was modified using direct editing/deleting face functions, the customer’s performance issue went away. Although my customer couldn’t re-request a “reduced” CAD file from his vendor, I still would recommend going the vendor route before taking your own time to fix the provided CAD file.

In addition if you’re using the ANYCAD function (Inventor 2016 or newer) and maintaining the reference back to the native CAD/3<sup>rd</sup> party file, ensure the information within that non-Inventor file is correct and somewhat error free. For a test you may want to first import it and save as an Inventor file to review its structure.

As an alternative check, you could check the integrity of the provided information using the [“Body Integrity Check”](#) tool

### **Performance Issue #25: File Migration**

Although I'm not going into a deep explanation of this process, file migration most likely will use more RAM to accomplish this task. If your dataset is large and you have limited RAM at that time, disk swapping will most likely occur and thus degrade your overall Inventor performance.

On the other hand, there's always been a discussion about file migration and I inform users I'm always on the fence about this matter. However I do lean more to the side where I just open my files when needed and let it migrate at that time. But there are others that spend the time up front and migrate all of their files using a manually workflow or [Inventor task scheduler](#). No matter what side you're on or how you migrate your files, just remember file migration can and/or will have an impact on Inventor performance.

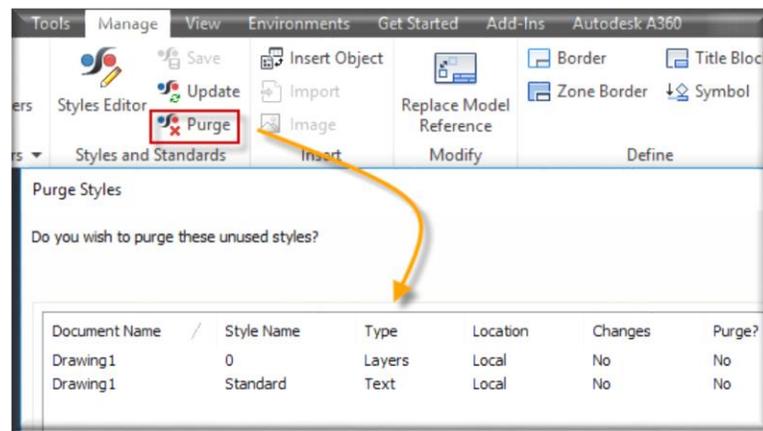
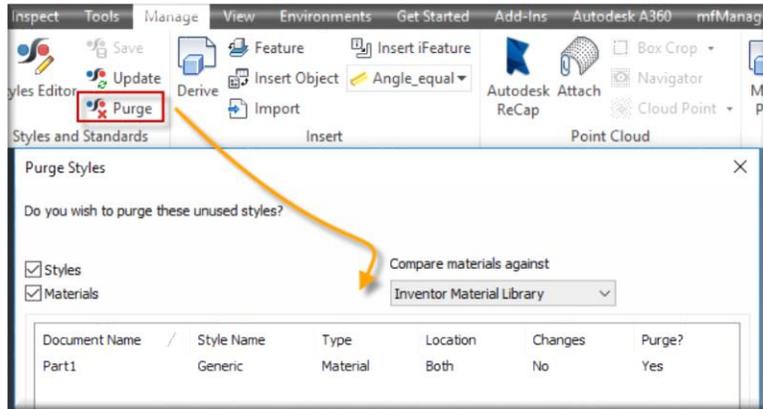
**Warning:** But if you do have a mixed Inventor version environment, you may want to consider holding off migrating any of your Inventor files until all users are working on the same version.

Here are some final thoughts on file migration:

- Always and I mean always... Back up your information before migrating any Inventor files and/or data. **Why jeopardize "live" or production data.**
- If you elect to migrate ahead of time
  - Consider working in smaller batches. You may want to take an extra step and test & ensure the results of the migration is what you expected.
  - Use a separate machine to accomplish this migration workflow
  - Perform the task off-hours and/or over the weekend
  - First migrate those commonly used files.
- Although I haven't seen any major issues when this occurs, but Autodesk doesn't recommend that you jump more than 3 versions of Inventor when migrating files.
- Many times this is assumed when a newer Vault version is implemented... Just because your organization updated Vault, doesn't mean Inventor files are migrated as part of this process.

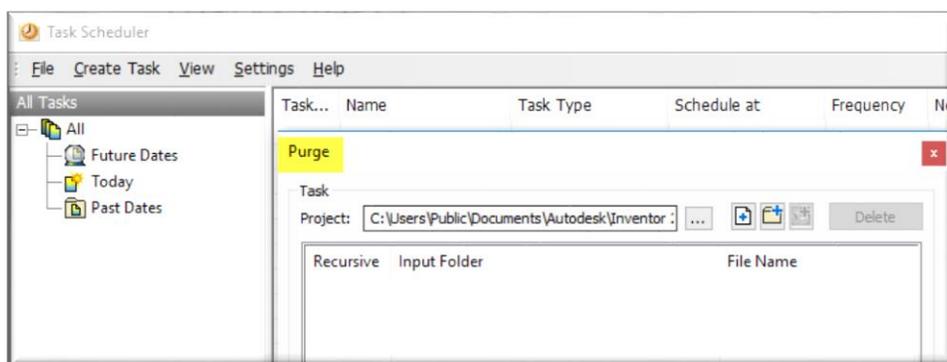
### **Performance Issue #26: Left Over Styles**

Throughout our design process, "things" change and are no longer needed. One of the things that most likely get left over, is the (local) styles that we no longer need. Leaving unused styles in your files can (or may) lead to Inventor performance. It's recommended that you purge these unused styles from your files on regularly basis.



*Helpful Tip: Depending on the style, its relationship and/or its status, certain styles may not be able to be purged. In addition, individual styles may also be purged via the “Style Editor” using the right mouse click on style/Purge selection (context menu) option*

As an alternative method you can use the “Inventor Task scheduler” to batch purge these unused styles via the “[Purge](#)” task function.



However when it comes to purging, especially via the task scheduler method, I always recommend that you fully understand what the purging operation will be accomplishing on your files. Local styles that you may end up needing down the road may get purged if not careful.

Another point about styles within your template, it's not recommended to apply every style your organization may require. Too many needed styles is the same as having too many unused ones.

### **Performance Issue #27: The Cluttered Browser**



Does the browser of your current Inventor model look like a clutter desk? Do you struggle in finding “things” in your browser or make sense of it? In my opinion, clutter within the browser leads to some form of Inventor performance. My advice, organize your browser using folders (assembly mode) or perhaps modify the occurrence name to make it easier to find items within.

### **Performance Issue #28: Bolted Connections**

Bolted connections is one of the necessary design accelerators. The downside to using them, the more you have within your model, the more it impacts your Inventor performance. If you require multiple bolted connections in a given model I recommend that you create a Level of Detail (LODs) and turn them off when not needed. Other things that you might want to consider instead of using bolted connections:

- Use the “Auto-Drop” function in content center and manually configure your bolted connection.
- Create a phantom (BOM) hardware assembly (per the “Phantom” section [here](#)) and use that instead as your bolted connection.

One final thing I do recommend, if you have to go the bolted connection route, create a browser folder and organize all of them together within that folder.

### **Performance Issue #29: Cloning (copying) your design**



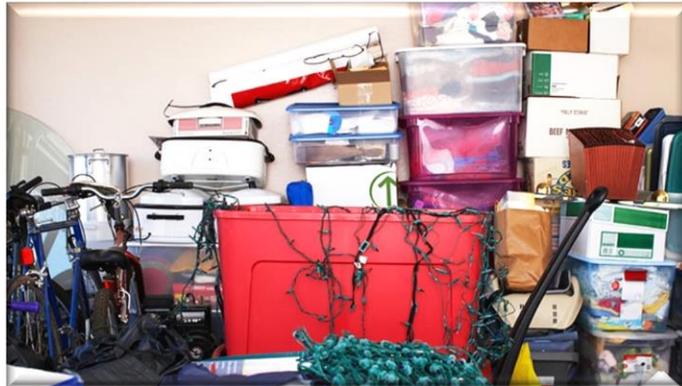
Many times we have to copy existing Inventor models for a new project. What tools are you using to accomplish this? I hear from individuals they have been using Windows Explorer or Pack & Go (and run into issues). Well these two (2) tools are not really the correct way of cloning your model. Improper workflow or the use of incorrect tools will just lead to issues and Inventor performance problems.

Depending on what you need to copy within your model here are those tools that you should be using. If you do have access to Vault, I always recommend Vault Design Copy over any Inventor tools.

- Vault:
  - Vault Basic: Old fashion “Vault Design Copy”
  - Vault Pro or Vault WorkGroup: [Vault Design Copy Interface](#) (Vault 2014 R2 version or newer)
- Inventor (in no order):
  - [Save and replace](#) workflow. Useful when needing to create a new design of an existing component/sub-assembly in a given model and maintain the related “constraints”.
  - [iLogic Design Copy](#). Many times users don’t know about this function because it only appears in the Inventor ribbon when nothing is loaded into Inventor. In addition other users overlook this handy copying utility because they think it has something to do with iLogic.
  - [Assembly Copy](#). Useful when you want to create a new design of entire model or bits & pieces.
  - [Design Assistant](#). Never really used the Design Assistant to do this but it’s an option.
  - [Copy Design Tool \(SDK\)](#). Not sure how many end users have loaded the SDK tools and/or used this interface. I have no experience with this tools and/or never had a request to support it. So I’m unable to comment on its capabilities or its success in copying your models.

- The follow three options are not really considered as a copying function but in theory they are accomplishing some form of creating a new design
  - [Derived Assembly](#)
  - [iAssembly](#)
  - [iCopy](#)

### Performance Issue #30: Finally the miscellaneous pile of...



- **Standards or the lack of them.** I will tell anyone as a former Cad Manager, that I'm a stickler for Inventor standards. I'm a firm believer that your standards/templates/styles configuration is more important than your actual design. I don't take this subject matter lightly and you shouldn't either. Yes a lack of standards can lead to performance and/or other issues with your model.
- **Lack of training.** I don't want to criticize anyone regarding their training or the lack of training for Inventor. We all learn at different paces and use different methods for accomplishing this. These days (in my opinion) it seems like many organizations are eliminating the CAD Manager position and the focus of training may be limited or non-existing as well. The point I'm trying to make, a "lack of training" will lead end users to not understanding the software, how it works, forming bad habits, and etc. Eventually it could lead to Inventor performance issues down the road for your organization. In the end this is something that your company and management has to review and invest into training if needed.
  - *For example:* Don't blame <fill in the blank> because he/she may not have a grasp of a certain Inventor workflow. Understand why they don't and adjust as needed. Maybe <fill in the blank> is not the only one struggling with this given workflow.
- **Taking shortcuts because they are easier.**
  - *For example:* Years ago at one of my jobs, I had a user who always said "Why do I need to do all these steps, because if I take this shortcut it's easier for me?"

Although I always pointed out why his shortcut lead to issues within the model, he always insisted well it's just easier for me. Granted I'm always open for streamlining workflows, but if in the end the shortcut is going to cause issues... No matter how much time he saved using the shortcut, if there's time being spent afterwards to correct the issues, **what time was really saved?**

- **Too many hands in the cookie jar:** Too many times I've seen where every end users has access to modify Inventor templates, standards, styles, and etc. When this is allowed, you could run into incorrect/over-lapping information, corruption, performance issue and etc. My advice, dedicate a small group (2 to 4 people) in your organization and give them the power to do this and not the entire design community.
- **Hardware configuration:** Although I didn't focus on hardware requirements/configurations in this class/handout, I still wanted to share the importance of having a proper hardware/configuration to run Inventor. This session/handout documents other items that may impact your Inventor performance but hardware is still a crucial part of this.
- **Oh my office phone is ringing:** Perhaps you are wondering what your office phone has to do with Inventor Performance. In many cases, it doesn't... So why am I even talking about it? If your office uses VoIP (Voice over IP protocol) phones and your PC/laptop is plugged directly into your phone, you could be faced with Inventor Performance issues. Although it's rare, I've come across 4 or 5 cases that the end user's VoIP phone was the culprit. I fully understand why your office infrastructure maybe setup this way, [machine is connected to the phone and the phone is connected to the network port on the wall.](#) Too me I'm not a fan of this configuration. Meaning I don't want to degrade my network connection through a phone especially for my CAD workstation. If possible plug your machine directly into a network port.
- **Computer reboot:** I wanted to finish with this because I think it's something that a lot of us forget to do these days with newer computers. In addition I believe it's an important step in combating Inventor performances/issues. There are a couple of Inventor forum users out there that have a statement like this "**sometimes you just need a good old reboot**" in their signature. I think it's a powerful statement and it's something that you should incorporate into your day to day activities.

In closing, my goal of this class was to provide enough detailed information that helps you to understand what else impacts **Inventor Performance** outside of acquiring new hardware.



**Perhaps one or more of the provided solutions will lead you to the “EXCELLENT” side of the (performance) meter**

For those who were able to attend my class I thank you for your time/participation and enjoy the rest of Autodesk University. If you were unable to attend AU or this given class, I appreciate you taking the time in reviewing the information that I've provided within this handout.

Until next time...

## Additional Resource Information

Name	Description	Resource	Notes
Autodesk Prerequisite Checker	Know if your computer is capable of running a particular Autodesk software product	<a href="https://knowledge.autodesk.com/search-result/caas/sfdcarticles/sfdcarticles/Using-the-Autodesk-Prerequisite-Checker.html">https://knowledge.autodesk.com/search-result/caas/sfdcarticles/sfdcarticles/Using-the-Autodesk-Prerequisite-Checker.html</a>	This may not work for all Autodesk products/versions
Windows Tips	Best Performance	<a href="https://support.microsoft.com/en-us/help/15055/windows-7-optimize-windows-better-performance">https://support.microsoft.com/en-us/help/15055/windows-7-optimize-windows-better-performance</a>	
Hybrid Sleep	Disable - Recommendations	<a href="https://support.microsoft.com/en-us/help/920730/how-to-disable-and-re-enable-hibernation-on-a-computer-that-is-running">https://support.microsoft.com/en-us/help/920730/how-to-disable-and-re-enable-hibernation-on-a-computer-that-is-running</a>	
Power Settings	Best Performance	<a href="https://docs.microsoft.com/en-us/windows-server/administration/performance-tuning/hardware/power/power-performance-tuning">https://docs.microsoft.com/en-us/windows-server/administration/performance-tuning/hardware/power/power-performance-tuning</a>	
Inventor Bench(mark)	Determine how fast your machine is	<a href="https://apps.autodesk.com/NVNTOR/en/Detail/Index?id=3667345248776204460&amp;appLang=en&amp;os=Win32_64">https://apps.autodesk.com/NVNTOR/en/Detail/Index?id=3667345248776204460&amp;appLang=en&amp;os=Win32_64</a>	Provided per <a href="https://forums.autodesk.com/t5/inventor-forum/how-fast-is-your-inventor-pc-really/m-p/5950908#M573052">https://forums.autodesk.com/t5/inventor-forum/how-fast-is-your-inventor-pc-really/m-p/5950908#M573052</a>
Inventor Memory Probe	Track memory usages while using Inventor	C:\Program Files\Autodesk\Inventor <version>\Bin\Memprobe.exe	<a href="https://www.cadlinecommunity.co.uk/hc/en-us/articles/201785492-Inventor-RAM-Memory-Issues">https://www.cadlinecommunity.co.uk/hc/en-us/articles/201785492-Inventor-RAM-Memory-Issues</a>  Removed for Inventor 2018???
Fiddler	Web/HTTP tracking	<a href="https://www.telerik.com/fiddler">https://www.telerik.com/fiddler</a>	

ShellExView	Ability to modify Autodesk integration within Windows	<a href="http://www.nirsoft.net/utils/s_hexview.html">http://www.nirsoft.net/utils/s_hexview.html</a>	Impact for opening and saving files
Resource Monitor	Displays information about the use of hardware (CPU, memory, disk, and network) and software	<a href="https://www.digitalcitizen.life/how-use-resource-monitor-windows-7">https://www.digitalcitizen.life/how-use-resource-monitor-windows-7</a>	<a href="https://en.wikipedia.org/wiki/Resource_Monitor">https://en.wikipedia.org/wiki/Resource_Monitor</a>
Tech Power Up GPU	Analysis Graphics Card	<a href="https://www.techpowerup.com/download/techpowerup-gpu-z/">https://www.techpowerup.com/download/techpowerup-gpu-z/</a>	