# Onshape® vs. SolidWorks®

July 2020

# Introduction:

## Who am I?

I am a mechanical designer / I.T. manager for a design/mfg. company. I have been there since 2004 and have worked in almost every position from janitor to lead designer. I had the opportunity to work under some great mentors who gave me all the hands-on training I needed to advance to the highest rungs of our company ladder. I try and pass on as much of my experiences to anyone who is willing to take the time to listen. In hopes we can grow off each other and learn from our mistakes.

## What we make

We make machines to work alongside automation in automotive plants. This can be anything from a bracket, to a conveyor, to a press loader, to a buffing machine, etc. Everything we make is one of a kind and each time we remake something we always aim for continuous improvement.

## Experience with SolidWorks

I started using SolidWorks in 2010 until now. It was a major upgrade for our company. Coming from Autodesk® Mechanical Desktop 2004. After using it for a while, SolidWorks really hasn't changed all that much in the last 10 years. Every year we need to upgrade to keep on the same version as one of our largest customers, and every year it feels like we are rolling the dice. "*Will this version break my older model*?", "*I wonder what bugs they gave us this year*", "*Oh nice they improved graphics performance*". Oh, that's pretty much it... Apparently that's all you get for a few thousand dollars...". It got to the point where I began to look for alternatives.

## Experience with Onshape

I have been using Onshape for almost 4 years. I have dabbled in every corner I can, while trying to test its limits. I love how I can use it at home and not need an expensive license to try it out. From there I have seen it grow every 3 weeks into a very formidable CAD system. When I started using it in 2016 it was an infant that was nowhere near ready for our company to adopt. But I saw its potential even then (mostly because of SAAS),and kept a close eye on it. Onshape's company culture of listening and communicating openly and directly with the customers was a breath of fresh air. Onshape is not perfect, but it is growing into a beautiful product. I can't wait to see where they take this over the years.

## Grading scale

I am grading some categories with a scale of 0 to 5 stars. 0 being terrible or non-existent, 5 being near perfect. All the points will be added up at the end for a final score.

# Cloud Computing vs. Desktop Software

Onshape: ★ ★ ★ ★ ★
SolidWorks: ★ ★

This grade is almost unfair, it isn't Onshape vs SolidWorks being graded. It is SAAS and Desktop applications in general. But each one lives in their own environment and are stuck there. So they must receive the grade their platform gets.

Cloud computing allows you to run high end software on practically any device. This is because all the heavy lifting is done on large server clusters rather than your device drastically reducing the computing power required by each user. Because Onshape runs in a web browser, that means you can (within reason) run it on any device with a web browser connected to the internet.

Here I will get into the differences that separate SAAS applications like Onshape from their desktop counterparts like SolidWorks/Inventor/Fusion etc.

Cloud computing is far from "new", it has been around for years and has been proven to be a very powerful way to create applications. The most notable cloud apps are Google's G-Suite apps like Google Docs, Google Sheets, etc. If you have ever had the opportunity to use both Google Docs and Microsoft Word, you will notice a few obvious differences. Such as: File Management, Sharing, Collaboration Tools, Versioning, etc. These differences are pretty typical traits between Cloud and Desktop apps. If you have used each, then you will already have a good idea of what SAAS and Onshape are about.

SolidWorks is a traditional desktop installed application that requires a specific type of computer hardware and operating system to install and run. It only runs on 64bit Windows. Some of the hardware specifications are fairly high in order to calculate all of the geometry in real-time on your device. Cloud apps like Onshape are completely hardware and operating

system neutral. If you want to run Onshape on a smart toaster running Linux then I am sure you can. I have heard of those that have managed to run it on a Raspberry Pi with reasonable success. Would I recommend that? Absolutely not! But what does that really mean for us? That means even if you have a 6-year-old computer, or a cheap Chromebook, you can still have the same fully featured CAD package that a power user with an expensive water-cooled rig can with SolidWorks. The only sacrifice you will experience between the two is mostly loading time and how smooth the model rotates when orbiting/zooming/panning. If you want to use a MacBook, or you're a fan of Linux, then you will be pleased to know Onshape runs just as well.

Desktop applications like SolidWorks, are broken down into a bunch of separate executables and packages that all work together to create a single application. This means you will need to have the correct version of all these files on your device, and they must all be loaded for SolidWorks to run correctly or at all.

I am mostly referring to .DLL files. If you don't know what a dll is, it is a pre-compiled bit of functionality that applications can call to execute a simple task. Most of the core functions are broken down into these dlls. When you update a service pack, mostly you are overwriting these dlls. Which means you have a bunch of files on your computer that could become corrupt during installation, or by other means.
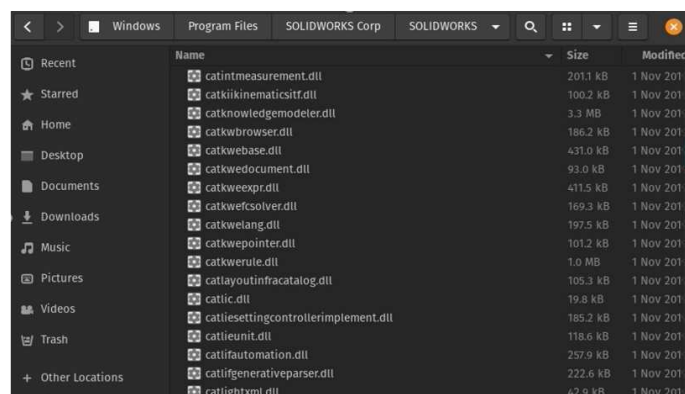


Onshape runs on a server and everyone is accessing the same executable. Which means everyone is on the same version and everyone executes the same code, period. You don't need to browse the forums because your part properties dialogue isn't saving the description you entered after you save the file. This is one reason I am a firm believer of SAAS. With SolidWorks, no two installations are the same. Everyone has their own unique bugs or oddities that are usually related to bad or corrupt installations. With Onshape everyone has the same bug, which means a simple message to support can usually get these bugs fixed in real time, for **everyone**. As soon as possible, no waiting 3 months for a service pack that fixed one thing, but broke 3 others. I know of a couple occasions where Onshape released a new feature, then a few hours later they removed it. Because it was causing many people to experience slower than normal loading times. This feature was later added again a few months later when there was more testing done. It was as simple as a flip of a switch to turn off a bugged feature for Onshape, most people likely never knew anything happened at all!

# Customisation

Onshape :  ★★★
SolidWorks:  ★★★★★

      Onshape only has a few preferences that can be customised. These are simple things like: Mouse Scroll Direction, Default units (mm, inch, ft, etc.), Icon placement on toolbar, among others. I rank this low because there is no way to set things like keybinding, or change the colors of the environment, such as a (true) dark theme. Onshape runs in a web browser, so if you are a hacker you can indeed make many modifications by injecting CSS. Or get a dark theme by using an extension like Dark Reader (not perfect). But I don't count these because they are not built in and are only known to those who care enough to dig in and come up with their own solution.

      SolidWorks on the other hand ranks high with customisations. You have so many options to set, you could spend hours playing around and tweaking. All the way down to how phantom lines are drawn, to what sounds should play during an error.

      This is actually a double edge sword. With the hundreds of options provided by SolidWorks, this can take hours. In my experience this is the hardest part about switching to a new computer or installation. There is no way to remember every checkbox of the hundreds of options. I recently upgraded my computer and attempted to save my settings (so I could import them into my new computer) unfortunately this failed. When I attempted to save the settings it would give an error message that said something about "Cannot Load Settings" (remember I'm trying to SAVE, not LOAD… so what that message really meant I never found out) I tried many times before giving up, and thought I knew my set-up well enough I could re-make it. But it still isn't the same as I had before. I have only lost my settings file 2 or 3 times in the 10 years, so it isn't a huge deal breaker. But it is a waste of time that nobody should have to worry about. Onshape on the other-hand saves all of your settings with your account. Which means even if you log into a display computer at the local shopping centre, all your settings are there.

# Hardware / Operating System Req.

Onshape:     ★ ★ ★ ★ ★
SolidWorks:  ★ ★

       Onshape runs in a web browser and uses WebGL to display to the screen. Beyond that the world is your oyster. As long as you can access the website and have a video card made after 2011 (this is the year WebGL was released) then you can run Onshape! Because it runs in a web browser, this also means you can run it on any operating system you like, as long as it has internet and a WebGL capable browser. Onshape has a web page that will scan your computer and run a quick benchmark to test your hardware. Just go to cad.onshape.com/check and see how your system would run. This is a quick check that can be done even when purchasing a computer. Before buying my current laptop, the first thing I did was go down to my computer shop (Before the corona shut-down) open a web browser and type that short url and see what the results were. It turns out most of the computers were about ⅓ of the way up the green bar, which was too low for my needs. But anything around 500 million+ or more will be plenty for most professional modelers. Beyond that it's hardly noticeable. As you can see I also need to run SolidWorks on this computer, so the stats are off the charts in Onshape, and SolidWorks runs about the same.

**Performance check**                                                          ✔

| | | |
|---|---|---|
| Overall performance* | | |
| Measured triangles per second | 841.7 million+ | |
| Measured lines per second | 336.8 million | |

*For information on graphics performance, visit: Graphics Performance Recommendations

       If you are still uncertain about what that performance number means, then all you need to do is log into Onshape at the store and open a large model and play around with it. (I recommend incognito mode). I did this with a few computers before making my selection. The computer salesman also got a kick out of it, because he never saw a full 3D CAD system run as a benchmark without even installing anything. It turns out you will want to at-least have a laptop with a **discrete graphics card**, and make sure during the test you **set the web browser to use the discrete card**. This is usually not a default option with dual gpu configurations, because most web browsing doesn't need a lot of graphics processing.

       SolidWorks on the other hand will need to have much higher specification of hardware in order to run at the same speed or anything close to what Onshape can do. This is because Onshape does most of the heavy processing on the cloud computer rather than sharing resources with all of the other apps on your machine. On the other hand, you have all of the processing done on your local machine. Which means you will have a snappier experience. Everything will feel quicker when it comes to sketching or making large selections. With Onshape every click is sent through the internet, a change is made, the result is sent back through the internet and then your computer will process the changes and display the result.

Internet latency plays a huge factor here. If you have a poor internet connection in your area, and a top of the line computer, then Onshape will give you a poor experience compared to SolidWorks running on the same machine.

# Internet / Power Outage!

Onshape: ★★★★
SolidWorks: ★★★

This is a cry I hear all of the time (or used to when Onshape was new). People would suggest "What would you do if your internet goes down?" For some this may be a valid excuse, especially if you don't use a laptop. But my experience in the last few years was to tether my phone's internet to my computer and keep going. Or grab my laptop and work off the Wi-Fi at the nearest hotspot. I would argue a broken internet is more of a perk than anything, due to the change of scenery. If you have a desktop computer then obviously you are stuck in the office in this case.

So, how often does your internet really go down? For how long? Not very often I would assume. The same can be said for power outages. What do you do when you lose power? Exactly the same thing, wait for the utility to come fix you, or relocate to another place with power/Wi-Fi. This actually happened to me 2 weeks ago (and again part way through writing this document, yikes!) We had a storm come through and take out the power, we sent everyone home / to the bar next door that had power. That would have been an opportunity to keep working in Onshape, but I am currently working in a SolidWorks based project. So... tough luck boss man, guess you will have to buy me a drink instead :)

So why do I score an internet based application so high compared to an on-premise software application? Well for exactly that reason. You are not stuck in one location. If your internet/power goes down, you can simply relocate. If your internet and power go down with SolidWorks, you cannot access your company server from another location, so you are dead in the water. I still give SolidWorks 3 stars here because if only the internet goes down, you will likely not notice at all. But if you are working remotely then no matter where you are in the world, you have a single point of failure. Your server's internet connection.

# New Computer Migration

Onshape: ★★★★★
SolidWorks: ★

Moving to a new computer is already a tough exercise. There are so many applications you need to install and configure. Onshape is completely transparent in this area. In fact, for me it is automatic. I use Chrome and sync my chrome account. By setting up chrome (as I would anyway) I already have all my Onshape bookmarks ready to go without even thinking about it. All of the settings in Onshape are saved to your login profile. So, no remembering what options you had set up. It just works and is set up no matter what computer you log into.

With SolidWorks this is a long tedious task which seems to always require you to download a gigantic service pack each time. Usually because by the time you install on a new machine you are into the next service pack, and that pre-downloaded installation you saved on the server is severely out of date. Then you need to make sure all your settings are migrated. There is a utility for this in SolidWorks, but my experience with it actually working is about 50/50 so don't count on it. This is not SolidWorks' fault, this is purely the difference between a SAAS app and desktop software. There is no way around this, software needs to be installed and configured for each machine.

# Licensing

Onshape: ★ ★ ★ ★ ★
SolidWorks: ★ ★

Onshape stores all your licensing information to a user account online. Which means all you need is a username and password to access your session. SolidWorks requires a software key that is registered to your specific computer. Which means you are stuck being able to access SolidWorks only on that machine. You have the option to move the license to another computer by going through a license transfer, which is not very difficult and doesn't take very long. But they count every time you move your license around and you are limited to so many transfers, after which you need to call your distributor and have them reset your license before you can use it at all. This is not a good option if you bounce around between work and home or if you want to use a single license between a few part time users.

SolidWorks gives you another option which requires you to install license software on your server. This is a network license and lets you install SolidWorks on many machines, while giving a first come first serve access to the next available SolidWorks license. You can also "Check Out" a license for up to a month, which lets you take it with you on the road. It works O.K. most of the time, but it does become a hassle and requires you to be aware of your license timer when working from home.

During the coronavirus shutdown all the engineers I work with all worked from home. Which meant we all had to come in a few times just to check out our licenses. We struggle with the latency of our slower internet at work as we all try and share a 9mb upload. (the limit of our office internet at the time of writing this) Now we have put in an order for fibre, but that will be a few months before we can have that installed. By that time we will all be back at the office (maybe). That will end up costing us double for the internet. Onshape doesn't need much bandwidth, so even the slow internet we had was suitable. If we had upgraded every project to Onshape sooner, we would have never noticed this at all.

As far as cost, Onshape has a barrier to entry advantage, you just pay your subscription. No extra costs for each license, or any of that hooplah. SolidWorks will cost you a small fortune upfront, plus your yearly subscription. On the other hand, if you don't need to update SolidWorks every year, you have the option of paying the upfront cost and only 1 year. Then cancel, but you will need to pay that all again when you upgrade, so it only makes sense if you

don't plan on upgrading every 3 or 4 years. Keep in mind, if you have an older version of SolidWorks, you will not be able to share files with someone else who has a newer version.

# Software Upgrading / Updating

Onshape:    ★ ★ ★ ★ ★
SolidWorks:   ★ ★

      Onshape is always up-to-date, there is never an upgrade you need to worry about. It just runs, and everyone is on the same version. Bug fixes can be pushed to everyone minutes after they have been solved, and you will never know it happened. Updates happen about every 3 weeks. Sometimes the updates are minor and may not apply to your needs, but the next update may just have that feature you have been hoping for. It is almost an event every three weeks. I and a few others will sit there and spam the refresh button watching the forum for the "Improvements to Onshape" thread to appear at the top of the page.

      SolidWorks is quite different. I always dread upgrade day. I read through their improvements and they have basically hit a brick wall as far as features go. We are only upgrading to fix bugs, and boost some performance to take advantage of newer hardware. This is because SolidWorks is an older and much more mature application. There really isn't much room for new features. This is a good thing partly because you have many tools at your disposal; but it is a bad thing because there is no more innovation, and you still have to fork out thousands of dollars each year just for the right to edit up-to-date files.

# Parts vs. PartStudios

Onshape:    ★ ★ ★ ★ ★
SolidWorks:   ★ ★ ★

      Onshape and SolidWorks both support 'multi-body modeling'. In Onshape we call each body a 'part'. Because they are full class parts just as if you create a new .sldprt file in SolidWorks. In Onshape we create these parts in a PartStudio rather than individual part files. This gives us much more freedom to model without consequences of external references in separate files. When you insert a part into an assembly in SolidWorks, you insert the entire part file. Which means you will get all the parts as you saw them in the part file. To fix this you will need to save each part externally which creates a lot of error prone file references that you will now need to manage continually. When you insert the part into an Onshape assembly, you only grab what you want, or you can dumb the whole PartStudio in and join it all together with a single mate if you want.

      The concept of a PartStudio is more like a single tree that builds a bucket full of parts. You can then use any part in the bucket wherever/however you want. No extra fuss or file management necessary.

# Modeling Features

Onshape: ★★★★
SolidWorks: ★★★★

Onshape has a much cleaner and generally more compact approach to modeling features. By this I mean they have consolidated many features into a single convenient button. For example, 'Extrude' in SolidWorks is split between 3 non-interchangeable features. Each with their own button or even complete toolbar of similar features, such as: Boss Extrude, Extrude Cut, Extrude Surface, etc. Onshape just has the one button, and you can switch between surface, cut, boss, intersect, or create an entirely new part on the fly in one click. If you attempt to change between a Boss Extrude and a Cut Extrude in SolidWorks, you will need to make sure there are no child references below, or else you will delete all of those features along with deleting the cut. Sometimes this task alone can take a long time.

SolidWorks is much older and has many more features ready for use. Unfortunately, many users are not even aware of most of them because they are buried deep within menus. I still find new features I didn't know were available in SolidWorks. The problem is, it could take a minute or two to remember what menu it was buried in.  Onshape has a single row of buttons, and it is much easier to find what you need. If you can't find what you want you have a search tool that will find it for you, then highlight where on the toolbar it was located, for the next time you may need it. SolidWorks has a search tool also, but it tends to open a help file more often than run the command. The one search box has more than one task and is easy to mess up. I ended up removing it from the toolbar because it caused more trouble than good.

Onshape may be lacking in some niche features. But they have a nice trick up their sleeve: FeatureScript. Onshape is written in FeatureScript and they have shared all their source code. If a feature does not act the way you want, simply copy the feature and make the adjustments yourself. Your missing feature you need could be simply made yourself (or by help from someone on the forum). So, in a sense, Onshape could be more feature rich than SolidWorks. More on this in the FeatureScript/Macro section of this paper.

A note on child/parent relationships in the feature tree; In SolidWorks, features absorb sketches or construction geometry. By this I mean it will create a nested tree where your extrude sketch lives. If you want to re-use the sketch, you will need to expand the extrude feature and share the sketch. Onshape, on the other hand, will not associate the construction elements with the feature. This makes reusing/sharing/deleting features far easier at the cost of a longer tree. Both Onshape and SolidWorks allow you to create folders and organise your tree, but SolidWorks' tree will look nicer at the cost of being slightly harder to manage.

# ERRORS (Chasing The Red)

Onshape:    ★ ★ ★ ★ ★
SolidWorks:   ★ ★ ★ ★

      Onshape gets a slight edge here; because when Onshape fails to create geometry or loses a reference, it does what is called 'fail quietly' if I am remembering the term correctly. This means it will not blow up your tree with red and yellow errors and warnings etc. because your fillet lost an edge for example. Instead it just carries on without bothering you. Sort of like saying "This doesn't cause any geometry below me to fail, so I'll keep my mouth shut". Then later on when you perhaps delete or move whatever covered that missing edge, Onshape will recognise it and put the fillet back on the edge as if nothing was ever wrong. SolidWorks hates this, you need to have everything defined explicitly or else your part may not even generate past the first broken feature.

      That is not to say Onshape doesn't error and turn red. It just doesn't put your whole world on hold until you address the problem. Both systems use similar methods to solve your problems. Usually it just involves editing the first red feature and digging in. I gave both systems a high rating here because they do seem similar when troubleshooting. But Onshape got the edge because allowing the model to fail quietly can actually be a design advantage, especially with configurations.

# Assembly Features

Onshape:    ★ ★ ★
SolidWorks:   ★ ★ ★ ★ ★

      SolidWorks is the clear winner here when it comes to assembly tools. Onshape has a very basic tool kit. You can mate, create relations, and do only the most **basic** patterns. SolidWorks will let you model your entire project in a single assembly if you so choose. I wouldn't recommend it, but you can. SolidWorks also has many pattern options, some of which I am sorely missing in Onshape. Such as 'PatternDrivenComponentPattern' (let's call it PDCP) this one pattern is the only feature in all of SolidWorks that I honestly can say "SolidWorks does it better". It works by having a pattern in a part. Let's say 100 holes. Now, you add a bracket, bolt, washer, nut. If you use this pattern, and select those parts, then select any one of the 100 holes. All of your parts will follow the same pattern that the 100 holes was created by. It doesn't matter if they are random points in a sketch. The best part is, when you modify the sketch in the part, your pattern instances update at the same time.

      Onshape has a somewhat mediocre answer to this, replicate. But replicate is much more related to SolidWorks 'copy with mates' in the sense that it creates a new instance of each part on the tree and mates them to each of the 100 holes in this case. Copy with mates is more powerful because you can attach multiple mates at a time, whereas Onshape only lets you connect one. Replicate is better than copy with mates in sheer speed. Copy with mates forces you to select each mate face for each mate you're copying. Replicate only requires you to make

the minimum selections and it will automatically find all similar geometry depending on the selection filters you choose. Which is in line with the speed of PDCP, but it won't update if you change the part pattern like it would in SolidWorks.

I hope one day Onshape allows FeatureScript in assemblies. This will single handedly change the game for Onshape assemblies. Just like it did for part studios. We could write our own PDCP feature and tailor it to our liking, without waiting for the Dev team to find time.

# Mates

Onshape:         ★ ★ ★ ★ ★
SolidWorks:     ★ ★ ★ ★ ★

Onshape approaches mates much differently than SolidWorks. Onshape uses a device called a 'Mate Connector'. Which is basically a point that has a normal plane perpendicular to its Z axis. This connector can be attached to other mate connectors by use of a mate. These are confusing to new users because they each allow for different types of motion. For example, a 'fastened' mate will connect two mate connectors and prevent all movement as if you welded them or bolted (fastened) them together. Revolute will connect the mate connectors together like fastened, but it will allow the parts to rotate along the Z axis of the mate connector (sort of like you loosened the bolt with fastened). Slider (similar to revolute) will be locked in all degrees of freedom except instead of rotating on the Z axis, you slide along the Z axis. Cylindrical is a combination of Revolute and Slider, etc. Each mate type will allow more and more degrees of freedom represented by the arrows shown in the icons. Once you understand how they work, you will quickly see how much faster mating can be in Onshape compared to SolidWorks.

In general, Onshape only requires 1 mate to fully define a part's position. SolidWorks requires at least 2, but on average 3. Each mate added to an assembly reduces performance of the assembly. When you have a large model, mates can kill a SolidWorks model's performance just because it has at least 3 times as many to calculate.

When it comes to fixing replacing parts / mates SolidWorks has better tools. Because SolidWorks references faces of the part rather than a mate connector, it is slightly more vulnerable to being broken. With that, SolidWorks has much better tools to fix these errors that Onshape doesn't. For example, in SolidWorks if you had 100 parts mated to the top face of a tooling plate then you extrude cut away the face of the tooling plate (let's pretend it was to remove material for a blanchard grind operation) now you have 100 broken mates in your tree. In Onshape you would need to open each mate and manually replace the bad mate connector. In SolidWorks you only need to fix one of the mates, once it recognises that you selected a new face, it will compare all of the other broken mates for the same face id, then asks you a question, "Do you want to replace all faces?". If you say yes, ALL of your mates are fixed! So in the end, SolidWorks simply is more powerful as far as assemblies are concerned, but much more time consuming to use.

# Undo/Redo

Onshape:     ★★★★
SolidWorks:   ★

      Onshape has unlimited undo! You can go back to the first click you made in your document from 3 years ago. This information is stored in the versions and history area. All you need to do is select any time in the list, right click and 'restore to current workspace'. Or you can just press the undo button many times.

      With SolidWorks, you are typically lucky if you can undo one operation. If you ever hit save, you will erase all of your undo history. I can't say I have ever seen redo work at all in SolidWorks, it is usually grey/disabled. We usually just say SolidWorks doesn't have an undo button, because usually you can't press it anyways. With a system that is prone to crash, you tend to save early and often, which makes undo obsolete.

# Top Down Design / Inheritance

Onshape:     ★★★★
SolidWorks:   ★★★★★

      Onshape has a unique approach to top down design. Their in-context system creates a snapshot of the assembly at that point in time. Which means you can take advantage of display states and temporary positions of parts to have stable references. So if you needed to model the end stops of an accelerator pedal for example, You could extend the pedal, model the end stop, retract the pedal and model the retracted stop. No matter where you move the pedal after this will not affect the in-context geometry until you explicitly update the context. This means you do not have to worry about breaking your model when trying to show motion during a design review. Onshape also has a derived feature which will let you import a part from another PartStudio or document, or a version of each. This is similar to in-context, but it is an alive context. Meaning you can edit the base part and the child part will update automatically, unlike in-context from an assembly. Between the two, you have a lot of power and control of your design intent.

      SolidWorks offers similar options, I haven't tried it, but I have heard there is a feature in SolidWorks that allows you to lock the state of an in-context definition. If that is so, that would be great. SolidWorks also allows you to reference another part in a .sldprt file. This is the same as 'derived' in Onshape. My experience with this has been cringe worthy at best. It appears to me that it is very sensitive to change, such as the file name of the original part. Once the link is broken, you pretty much need to delete the derive and reassign every reference you made in the part tree. I gave this a lot of hope. I broke a lot of models and re-made them enough to the point I just made it a point to never use that feature in SolidWorks. I'm sure there are only a couple of specific things I'm doing to break them, but so far it has happened a few times EVERY design. As far as I'm concerned it isn't SolidWorks' fault, it all boils down to file name references. The database structure of Onshape is just leagues above any file-based system.

Renaming or deleting entries in Onshape really doesn't matter too much. In SolidWorks top down design reminds me of a "bull in a China shop". Meanwhile, in Onshape you can delete/rename/pull the plug on your computer, 2 or three clicks later you can be right back to where you left off.

# FeatureScript / Macros

Onshape:      ★ ★ ★ ★ ★
SolidWorks:   ★ ★ ★

Onshape uses FeatureScript to write every feature in a PartStudio. That means the featurescript code you can write is a first class citizen feature. SolidWorks will let you use macros, which uses the VBA language that allow you to make manipulations to your part. But they require you to execute them every time. Featurescript features live in the feature tree and are updated just like any other extrude or fillet feature. It is said, "If you can model it in a PartStudio, you can make a single feature that creates everything in the same way". So if you find yourself making a lot of miter gears, or sprockets, or adding draft angles and fillets for injection molding. These repetitive tasks can be created using featurescript. Some popular featurescripts I've used are: measure value (allows you to measure objects and store the result in a variable), thread creator (creates a helical thread on a cylinder based off ISO/DIN/ANSI etc), surface text (allows you to create text on any shape / face naturally), spur gear (creates a full model of a spur gear based off the parameters you set).

FeatureScript may seem intimidating at first, but it is loosely based on Java. So, most of the syntax you may already know coming from a C style language. If you never coded or have no desire to code, no problem, there are many people on the forum that are willing to help you create the feature you need. Sometimes you may even feel brave enough to give it a go. My first featurescript was a circular pattern that spaced each instance based on a logarithm. It was literally as easy as copying the original circular pattern feature from Onshape's source code. Then changed: i * angle to: log(i) * angle. Literally 5 characters different and I saved hours of sketching and calculating and extruding etc.
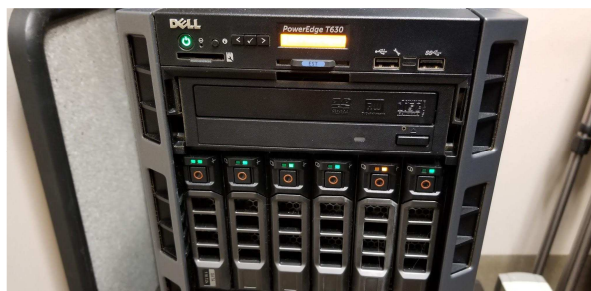
# File Management

Onshape:      ★ ★ ★ ★ ★
SolidWorks:   ★ ★

       Onshape does not have files. This confuses most people. There are a lot of things that look and act like files; but they are nothing more than entries in a large database. You don't need to worry too much about this. If it makes you feel better calling things files, by all means go ahead. The key takeaway is that files on a desktop are vulnerable to many things. Accidental deletion / overwriting, drag and drop into the wrong folder on the network by a shop guy clicking too fast and loose, viruses on someone's computer from a scam email encrypting the network drive, server hard drive failure corruption, someone sending copies of intellectual property to a third party via USB drive. Almost All of these things I have had to deal with where I currently work.

       So how is Onshape any different? You can still have a virus on a server, your database can get corrupt, etc. Well Onshape stores your data and itself, on a server that ONLY runs Onshape. This server is so isolated it doesn't even have IP addresses attached to it (so I'm told by an Onshape employee). It is a cloud based system, which means that even if you burn the building, your data is still safe on another remote system, and it will pick up right where the other failed hardware left off. It is constantly being updated to protect it from the latest viruses and attacks. Since the servers have gone online they have had 99.99% uptime. Basically you can get rid of your server and I.T. department, Onshape takes care of all of that for you.

       What can you say for your I.T. department? Well at my company, that is me. And I have too many other hats to wear to be constantly checking for day one hacks on the dark web. Let alone checking the hard drive's life on the server. I have had a drive fail even though there are many drives in a 1-0 raid and a backup on top of that, I have still lost an entire project, and a couple of other permanent file corruptions because the server sat with a corrupt drive and it continued to replace the backups with the corrupt versions until I finally tried to access those older files and checked the lights on the front of the server. My double redundancy only lasts 1 month, after that all bets are off. We can only afford so much hard drive space.

# PDM / Release Management

Onshape:      ★ ★ ★ ★ ★
SolidWorks:   ★ ★ ★ ★

   I don't use Onshape Release Management or SolidWorks PDM. So I can't say much other than what the general feel of the internet says about these, and what I can see from a demonstration.

   Onshape Release Management looks very clean and is fully integrated into the entire system. You can see what version your parts are right in the assembly tree, or in the drawing sheet reference. It seems to have more features that make it practically replace an ERP system at the same time. Which is the only reason I gave it more stars than PDM. This functionality is standard with a PRO level or higher account, and SolidWorks charges you extra for less. Onshape is trying to make itself the one-stop-shop for everything you need, and it seems to be working out.

# Version Control

Onshape:      ★ ★ ★ ★ ★
SolidWorks:

   Onshape has a 'Git' style approach to version control. It is powerful and easy. Every action you take is constantly being saved and time stamped with the person who made the edit and what the edit was. If you work on a design for a few weeks, then the boss tells you "You know, I think I liked what you had last Tuesday better". It is just a quick scroll through the version history and a branch and you now have everything you lost. In SolidWorks you better have the foresight to make copies and save them in zip files if you don't want to lose them.

   SolidWorks doesn't offer much in this area unless you use a PDM system. But it isn't core built in functionality so it gets a zero score.

# Multi-User Collaboration

Onshape:     ★ ★ ★ ★ ★
SolidWorks:   ★

No Question, Onshape is #1 in this topic. You can have many people looking at the same model in real time watch as the model gets created. You can chat and point out areas of improvement through the comment system, in real time. The comments can also be assigned to a person, will receive a notification email and the comment will be added to an open issues list that they can use as a to-do list. Many people can edit the same part, at the same time, anywhere in the world. I helped one person who was trying to learn Onshape all the way in Australia (I'm in Michigan). I was able to watch his mouse cursor as he attempted to create a motorcycle frame and was able to see where he was confused better than he could explain it.

SolidWorks offers a couple of collaboration tools. But in practice you will never want two or more people to be in the same document (even read-only eDrawings viewer). Unless you have a PDM set up, which is something I have never done, and is something I don't want to hassle with either for a small company. I can't tell you how many times I had to walk to every computer in the office and shop looking for the one person that had my active model open in eDrawings and took away my write access by accident. I gave SolidWorks 1 star for this only because they "offer" multi-user mode. Which basically warns you every few minutes that someone has saved one of the files you currently have referenced, and it allows you to update it. But it should be 0 stars because I have literally corrupt assemblies because I renamed a bunch of parts while someone had them open read-only on another machine.

The process goes like this: You rename a bunch of files using the tree in the assembly. While you're renaming nothing happens. Once you hit save, it first saves the assembly with all of the new file names. Then it makes an attempt at renaming each file on disk. So if someone has one or more of your files open, that file fails to rename. SolidWorks then crashes and gives an ominous error. The only way to fix it is to remember what all of the filenames were before AND after you changed them, and manually relink each file in the file references menu. I have made it a point to only rename 4 or 5 parts at a time and lose the 2 minutes during each save.

In Onshape you can rename and move documents, parts, etc. while 100 people are logged in and making edits to that part. It is no contest. This is by far my most praised feature of Onshape, and should be enough for anyone who works with more than one person to make the switch immediately to Onshape!

# Conclusion

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|---|---|---|---|
| Onshape Total Score : | 87 / 95 | 92% | A- |
| SolidWorks Total Score: | 54 / 95 | 57% | F |

Both systems are imperfect, but that can be said about every CAD system I have ever used. Onshape is lacking in a few features that would make it one of the best CAD platforms available today. But we are comparing a very old, very polished SolidWorks with a new player that still has a bunch of code still left to write. I say SolidWorks is polished but I only mean that in a feature rich sense of the word. Because they are stuck writing code that needs to execute correctly on millions of unique machines, it makes the debugging and polishing process much more challenging. And that makes the improvement process much slower and inconsistent between machines. Onshape, on the other hand, is made for a web browser that runs the same on everyone's machine. So working out the bugs is mostly done before they even get pushed into the latest version. If there is a bug, it is as easy as flipping a switch and they can go back to the last working version until they fix the bug.

Do I really think SolidWorks deserves an F here? Hell NO! Actually, I was surprised. I was thinking this would result in a B or a B+ for SolidWorks, I even fudged a few extra points to boost their score. But it comes down to the fact SolidWorks just cannot do some of the more collaborative things that Onshape has inherited just from SAAS. It's like comparing a new Tesla to a Mustang. Sure, they are both nice cars, and have a lot of speed and power. But Tesla is just loaded to the gills with new technology and features that the Mustang just doesn't have access to. Same can be said with this review. Onshape makes up for its shortcomings by being new and innovative. It will be getting better for the next few years. While SolidWorks is kind of stuck in the past with no way to keep up with future tech.

I am convinced, the future of CAD (and most software for that matter) is definitely moving to the cloud. Onshape is the first CAD company to make the leap entirely. They looked at Google docs, looked at SolidWorks and said (paraphrasing) "This is as far as we can go with a file based software. It's time to quit SolidWorks and start a new company that will run CAD in the cloud". And that is exactly what Jon Hirschtick, John McEleney, and a group of SolidWorks' best developers did in 2011. From what I can tell, the John's taken all of the talent from SolidWorks with them. SolidWorks hasn't changed much at all in the last 10 years. It is my belief that this is why. Onshape is coming together so well while SolidWorks is being left in the dust. SolidWorks had it's time, now it is time to move onto more modern solutions.



Author: John McClary, a practicing Engineer and user of SolidWorks and Onshape.