

2017 System Optimization Guide

TITLE:	System Optimization Guide
DATE:	September 2016
SUBJECT:	Optimizing for SOLIDWORKS
ABSTRACT:	Guide to optimizing your system for running SOLIDWORKS

This guide is written for SOLIDWORKS 2017. If you are using another version of SOLIDWORKS, please download the corresponding document [here](#).

SOLIDWORKS requires an optimal hardware and software environment to run efficiently and without errors. This document is written so that you may use SOLIDWORKS without any frustration due to the environment in which SOLIDWORKS is running.

If your machine is experiencing stability issues, graphics issues or seems to be running slowly, read through this document and make any recommended changes. If you have any technical issues with SOLIDWORKS, please contact [Hawk Ridge Systems Technical Support](#).

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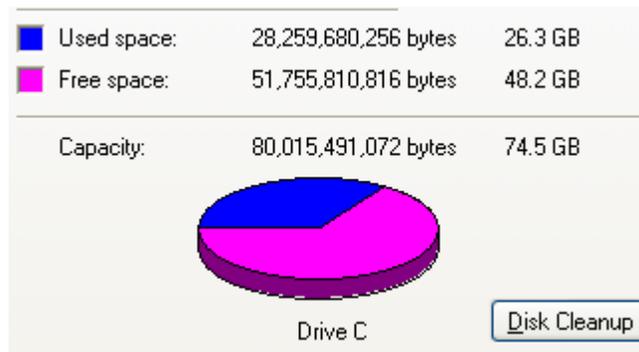
Memory Considerations

Disk Space:

Your computer’s hard disk drive (HDD) requires some headroom to run programs. A good rule is to have 15% free space as the minimum required to defragment your HDD. (Note: Solid state drives must never be defragmented as it has no benefit and will reduce the lifespan of the device.)

How to find it:

Go to Windows Explorer, right-click on your hard drive and choose **Properties** from the right-click menu.



Use the SOLIDWORKS Rx System Maintenance tools (next section) to help clear hard drive clutter. If you still do not have sufficient disk space, consider archiving data or removing unused programs.

Virtual Memory:

When RAM (random access memory) usage is high, the CPU (central processing unit) begins using virtual memory. This resides on your HDD to keep all processes running. If you are getting memory errors, you can allocate up to three times the capacity of your RAM to alleviate the problems associated with running out of memory.

Windows 7, 8.1, 10: Start, Control Panel, System & Security, System, and then select the **Advanced System Settings** (on the left).



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Under **Performance**, press the **Settings** button. Choose the **Advanced** tab and then press the **Change** button in the lower right of the window. Uncheck **Automatically manage paging...**, Press the **Custom size** radio button and set the low and high limits of virtual memory for your HDD.

It is a good idea to have a small buffer between initial size and maximum size. If the computer runs out of virtual memory, it will crash. This gap will ensure a warning regarding running out of virtual memory and will appear before the system fails so you can save your work before restarting.

Virtual memory is exponentially slower to access than RAM. Once information is put in virtual memory, it does not purge itself automatically once the physical RAM becomes available. Also, if page space is accessed, it will continue to be used by all windows applications resulting in slower overall performance until restart. Therefore, if you are constantly paging to virtual memory, or frequently experiencing memory related crashes, you should consider buying more RAM.

Using SOLIDWORKS Rx

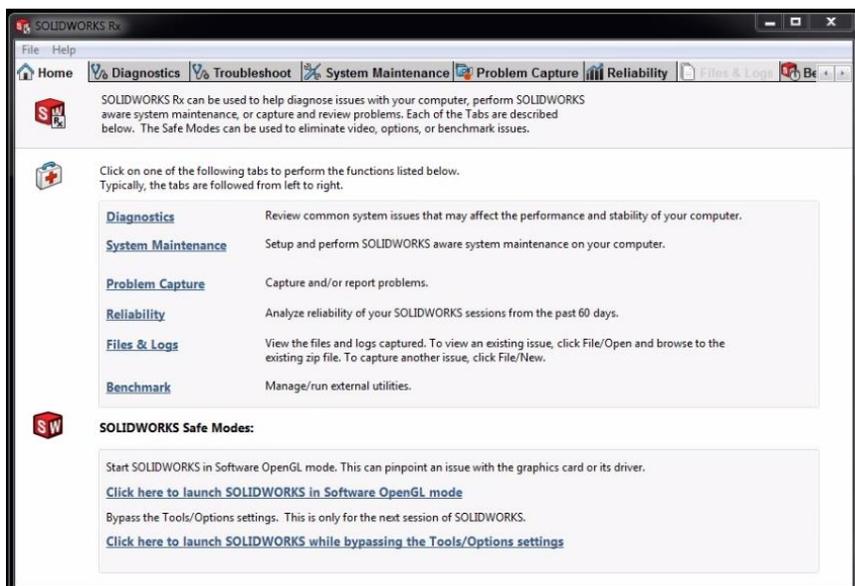
The SOLIDWORKS Rx program is loaded with all versions of SOLIDWORKS. This program can be used to help diagnose issues with your computer or perform SOLIDWORKS system maintenance.

To start SOLIDWORKS Rx, go to Start > All Programs > SOLIDWORKS 2017 > SOLIDWORKS Tools > SOLIDWORKS Rx 2017.

There are a series of tabs in which we are interested:

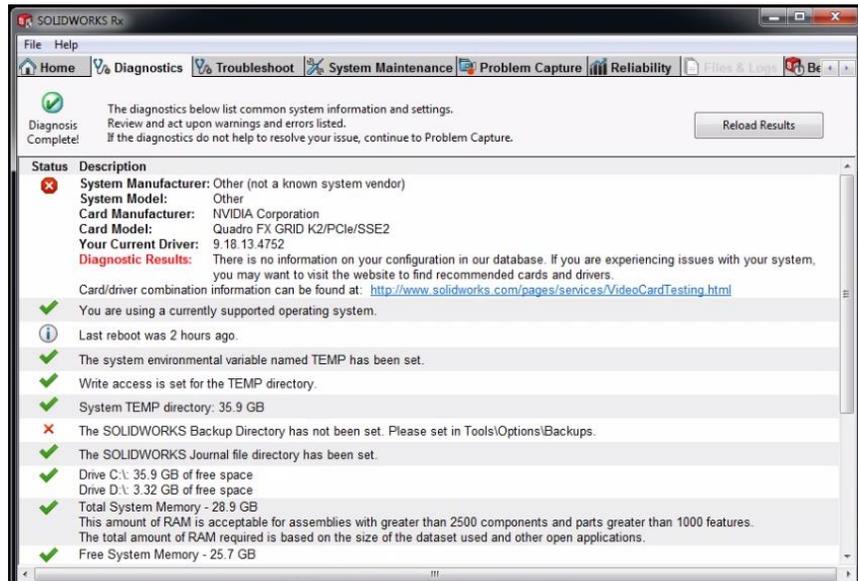
Home

The Home tab is another way to get to the other tabs represented in the form of links. There are also links at the bottom of the Home tab used for troubleshooting: "Click here to launch SOLIDWORKS in Software OpenGL mode", and "Click here to launch SOLIDWORKS while bypassing the Tools/Options settings". They are both used to test whether SOLIDWORKS or the environment is misbehaving (though not definitively).



Diagnostics

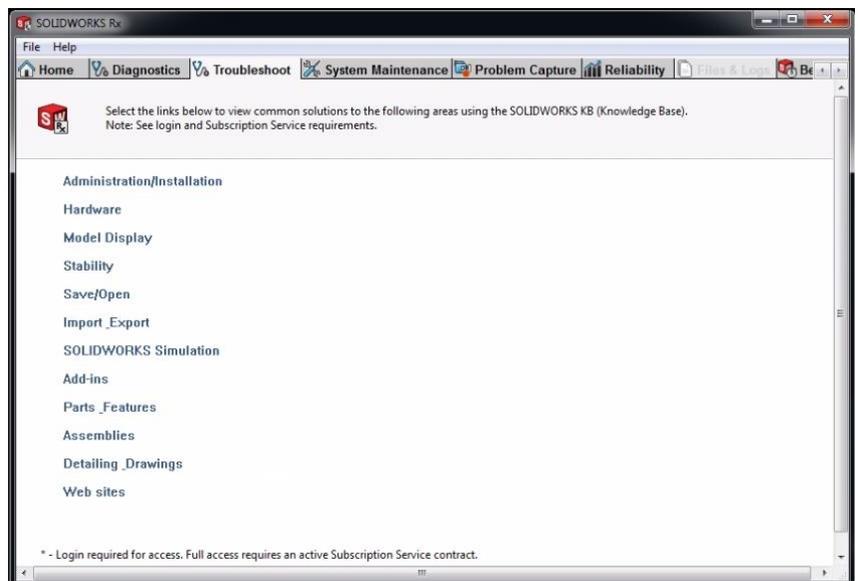
The Diagnostics tab lists general system information and settings. Green checkmarks signify that your system passes an initial inspection based on nominal settings. It will tell you that your graphics card and driver are approved for running SOLIDWORKS, for example, but it will not tell you whether these are functioning properly. A yellow or red "X" mark means there is an issue that may impact performance. Follow the instructions on



correcting the corresponding issue(s). Then press the **Reload Results** button in the upper right hand corner to refresh the diagnostic results. Ignore the item regarding service pack database and the toolbox share if you are not using a shared library environment.

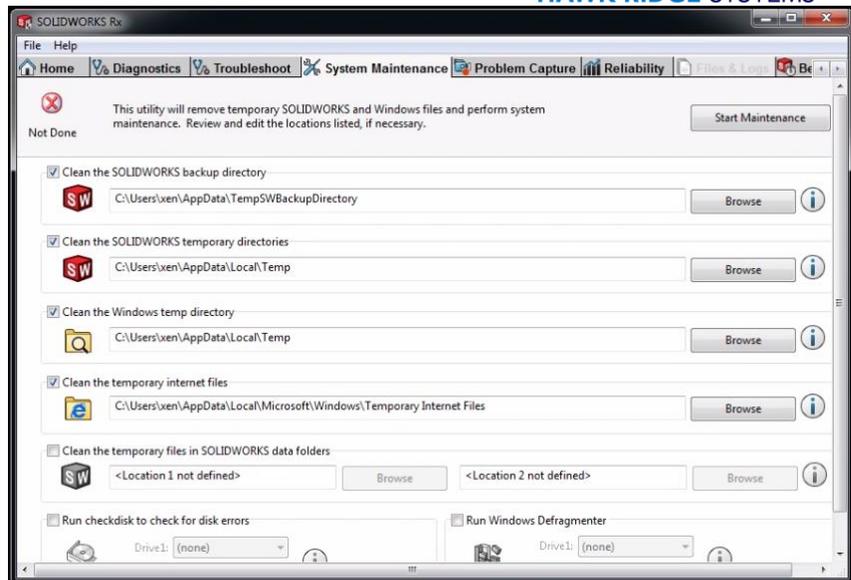
Troubleshoot

The Troubleshoot tab lists common administrative processes or functions within SOLIDWORKS. Clicking any of these topics will take you to the SOLIDWORKS knowledgebase articles on this topic. If you are experiencing issues with SOLIDWORKS, searching this database for similar issues may give you information on fixing this issue by manipulating settings or incorporating a known workaround.



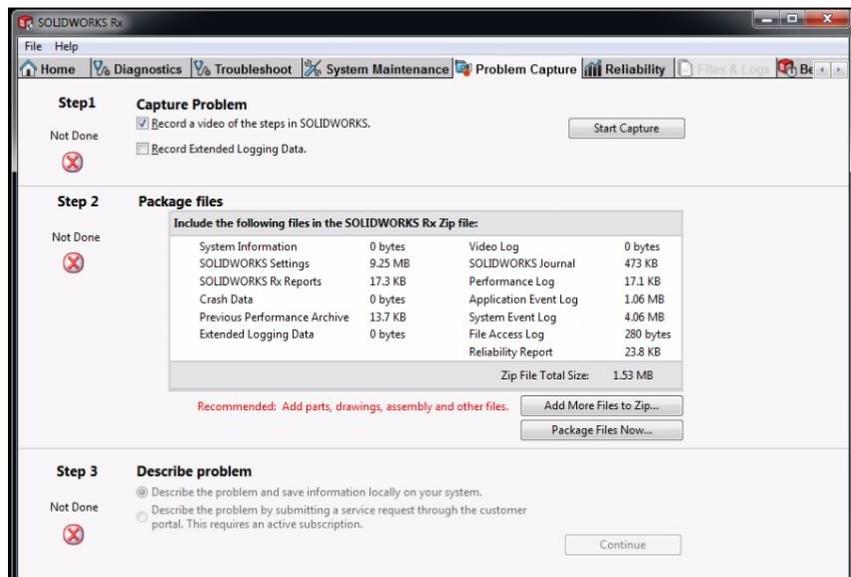
System Maintenance

This utility will remove temporary SOLIDWORKS and Windows files and perform system maintenance. Be sure to clean all file locations and for best results, run Checkdisk and Windows Defragmenter. You should do this periodically to ensure optimum system performance. (Note: Solid state drives must never be defragmented as it has no benefit and will reduce the lifespan of the device.)



Problem Capture

If there is a problem you need to "capture" and send to us, SOLIDWORKS Rx's Problem Capture utility is handy for capturing and packaging (nearly) all the relevant information necessary to diagnose what may be happening coincident to the problem. It is important to follow the steps outlines on the page as you will not be able to finish if you have skipped a step.

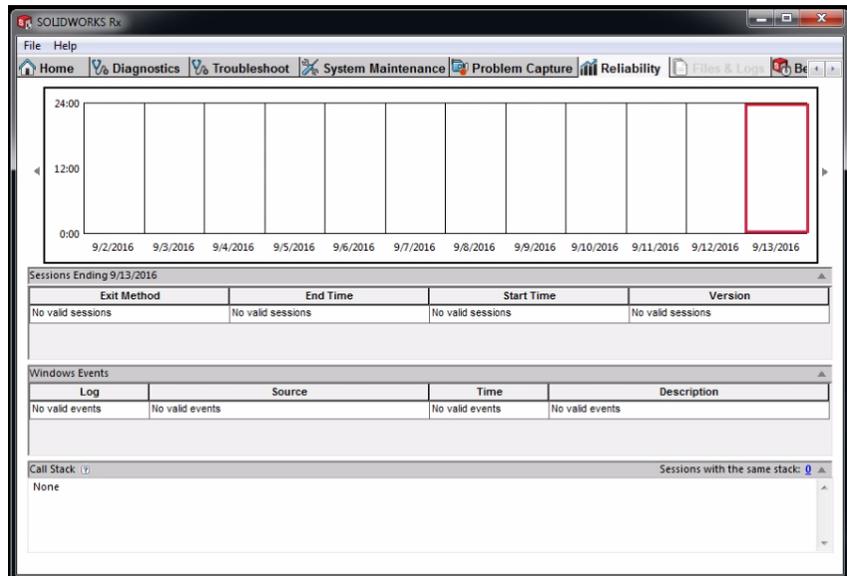


1. Record a video of the problem occurring which will automate opening SOLIDWORKS and closing it when you press the stop button on the recording.
2. Add additional relevant files (part, assembly, drawing files, etc.)
3. Describe the problem in as much detail as possible. The more we know about workflow, etc. the more effectively we can help you.

Reliability

The Reliability tab includes a chart listing sessions that have terminated in the past 60 days. The chart indicates whether a session ended normally or with a crash. For crash events, the call stack displays, along with a count of how many times that call stack appeared during the past 60 days.

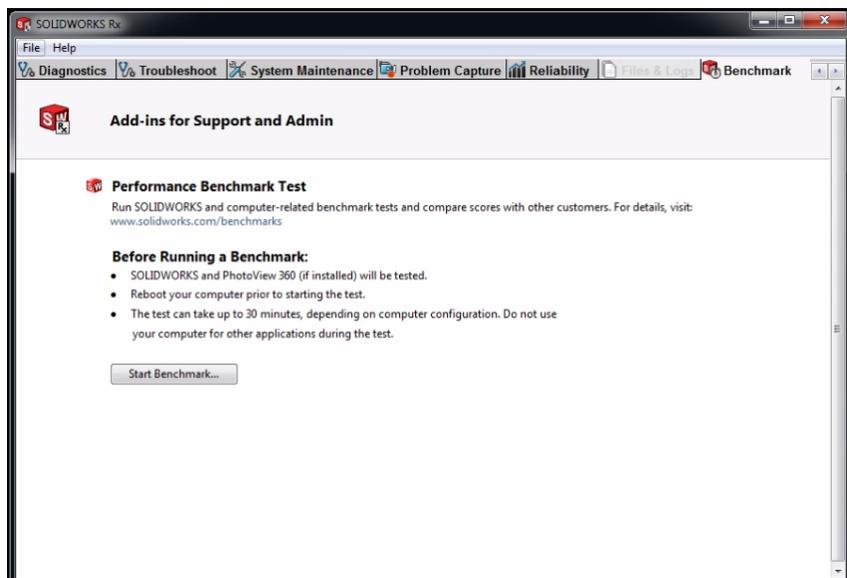
This tab also displays Windows events for the five minutes preceding each crash, including session termination and installation events.



Benchmark

The performance benchmark test application as well as any applications you have added can be accessible through the Benchmark tab. If you click on the **Browse...** button, you can find and select an executable to be linked here.

The performance benchmark will allow you to compare your computer against those of your colleagues to help justify needed maintenance or hardware replacement. Using this, plus the new Admin Dashboard, you can compare, on one page, your system and others' within your organization. Carefully follow the instructions so your results are not skewed.



Note: The Files and Logs tab is a searchable location for the logs produced by SOLIDWORKS and the contents go into the Problem Capture packet of files.

Please review the following list of items to rule out any system-specific issues affecting performance.

Other Considerations

- **Service Packs**

- SOLIDWORKS periodically puts out patches for their software to fix any issues. If you are experiencing problems with SOLIDWORKS, be sure to be on the latest version before reporting. You will need to login to the customer portal from <http://www.SOLIDWORKS.com/pages/services/downloads.html> to get full access to the subscription service downloads.

- **Windows Updates**

- Windows Update is a service provided by Microsoft that provides updates for the Microsoft Windows operating system and its installed components. You will want these updates to ensure that your system is current and is not vulnerable to computer viruses and other malware that depend on the vulnerabilities these fix.

- **Other Software**

- If you are using any indexing software on your machine, having this running constantly can yield slow performance. Set your indexing service to happen overnight or weekly, when you are not working on your system, for better overall performance.
- If you are using backup software, be sure this is not constantly running. Backing up your data is recommended but should be done when the computer is not being used, such as after business hours or on weekends. But something like backup software should be used with deliberate purpose. If you are comfortable sacrificing performance for the safety consideration, making the decision is ultimately up to you.
- Your anti-virus program may be scanning files when they are opened. Scanning .sldprt, .sldasm, and .slddrw files will slow down performance considerably, especially when opening a large assembly. If you are experiencing slow performance when opening files, go into the setting of your anti-virus and make exceptions for these file types so they are not scanned.

- **Graphical Issues**

- Often performance and even stability can be traced to your system's video card. You can test to see if a performance issue is graphics related by starting SOLIDWORKS, and with no files open, go to **Tools, Options, System Options** and **Performance**. Turn on the **Use Software Open GL** option and test the issue again. If the problem goes away with this option turned on, you should consider maintenance on your graphics card or its driver.
- For a list of tested graphics cards and recommended drivers, go to this web page: <http://www.SOLIDWORKS.com/pages/services/VideoCardTesting.html>.

- **Your Hardware**

- If you find your performance takes a hit only when working on large assemblies or particularly complex parts with large file sizes, you may need more RAM. If your file size exceeds your physical RAM then the excess data is stored on the hard drive. This slows down performance considerably.
- If you find yourself staring at an hourglass cursor for extremely long periods when doing complex functions such as sweeps, lofts, equations, helices, and so on, the problem may be a slow processor. If you have an extremely fast single core processor (3+ GHz) and run into this issue, make sure that the hyper threading on this processor is disabled.

- **Computer Care**

- Your computer requires a reboot from time to time. The sleep mode does not purge memory or reset your settings. It is important to periodically reboot your computer. Restarting after any program crashes is recommended.
- Components within your computer need ventilation. If your computer slows or falters after being on for some time, check that the system is in an area it can get air. Also, clean out all vents and fans with compressed air periodically.
- Viruses and spyware can affect system performance. Have anti-virus and anti-spyware programs check your computer on a regular basis and make sure these are up to date (not while using SOLIDWORKS, of course). When you install SOLIDWORKS or service packs be sure to disable these programs while loading the software.
- For good stability and performance on a computer, you may want to reformat the operating system on your hard drive annually or semi-annually. This is not required to run SOLIDWORKS, but is a good idea for optimal computer performance. **Be aware that this will clear out all of your data and programs.** You must backup all of your data to an external source and make sure you have all your software licensing information copied before beginning this process. If you have an IT department at your company, check with them before doing anything to your operating system.

- **Contact Us**

- If you continue to have issues or questions regarding performance or stability, you can contact our help desk by submitting a ticket on our website at <http://www.hawkridgesys.com/support/>. Or give us a call at 1-877-266-4469 (United States) or 1-866-578-6803 (Canada).