Maximize SolidWorks Performance

The software tweaks and hardware upgrades that provide as much as a 5.5 times increase in SolidWorks' performance.

A Study by Computer Aided Technology, Inc.

Prepared by the editors of Desktop Engineering

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EXECUTIVE SUMMARY

Dassault Systèmes' SolidWorks software is used the world over to help bring amazing products to market. The most productive SolidWorks users operate in a near-optimal environment that permits them to create the best products possible through multiple design iterations.

But many SolidWorks users may be unwittingly shortchanging their investment in SolidWorks with a less-than-ideal computing environment. Whether their modeling techniques are not as advanced or their technology tools are not optimized, the result is the same: a sapping of the creativity and productivity of an organization's most important resource—its employees.

As the design cycle continues to compress, the effect on SolidWorks users who are not operating at optimum productivity is much more dramatic. From missed release dates to loss of marketshare, organizations that don't empower their employees to do their best work are flirting with disaster.

A Common Question

"What can I do to improve my SolidWorks productivity?"

It's a question that Computer Aided Technology, Inc. (CATI) has heard a lot in its 20-year history. The SolidWorks reseller provides training, installation and configuration services, and support. To answer it, the company's Technical Services Director, Adrian Fanjoy, and Technical Support Manager, Josh Altergott, embarked on a research project to prove what steps SolidWorks users could take to maximize productivity.

The researchers designed a series of rigorous, controlled tests against a typical workstation environment—the type that CATI frequently sees being used by its customers. The researchers ran an amalgam of customer-supplied test models through a macro script of SolidWorks actions that represented tasks commonly performed by users in a typical workday.

Dozens of variables were tested to measure the time savings that each software and hardware alteration would provide over the baseline system. CATI researchers tested Windows performance options, SolidWorks settings, modeling methodologies and hardware upgrades, including multicore processors, RAM and solid-state hard drives.

A Proven Answer

CATI researchers discovered an optimal computing environment for SolidWorks that improves performance by as much as 5.5 times. They found that simple changes, such as adjusting the default Windows and SolidWorks settings, or setting up models properly, can make a big difference without costing a dime. The researchers showed that investing in an Intel® Xeon® processor-based workstation with more RAM, robust processors and faster hard drives will increase SolidWorks productivity so much that it typically pays for itself in six months via accelerated design iterations.

Engineers, managers and executives can use CATI's research to decide on the smartest investments of time and equipment to maximize SolidWorks performance. Just imagine what new projects could be created, and the competitive advantages and business opportunities that can be capitalized on with a 5.5 times increase in SolidWorks productivity.

INTRODUCTION

Many hear of this "Promised Land." It's a place where technology helps engineering teams create many design ideas for every challenge faced, while also allowing teams to quickly converge on the optimal design solution. Where routine, essential modeling operations are done interactively, and where large assembly models run quickly, without the constant threat of crashing. Where technology enables astounding gains in design creativity and productivity. And where tight, seemingly impossible production timetables are met—with the best design solution.

Some lucky design engineers work in this technology utopia. But many more operate in a state where technology performance is a perpetual problem. These performance issues may be caused by models that are too large and complicated to be processed efficiently by deployed workstations. Sometimes, they're caused by not using the best, most efficient modeling techniques.

> "There will always be a performance wall out there somewhere. Sooner or later, you're going to hit it."
>
> *— Adrian Fanjoy, CATI*

Whatever the cause, the outcome is always the same: a sapping of the creativity and productivity of an organization's most important resource. And the question that so many of these affected design engineers ask is this:

"What software and hardware choices can I make that will substantially improve my performance?"

"I can't tell you how many times I've been asked that question," says Adrian Fanjoy, technical services director at Computer Aided Technology, Inc. (CATI), a leading SolidWorks reseller in the Midwestern United States. "Performance is a perpetual problem. There will always be a performance wall out there somewhere. Sooner or later, you're going to hit it."

There is no "one size fits all" answer to how to improve performance for everyone using CAD programs like SolidWorks. Variables such as model size, modeling techniques, hardware capabilities and software settings make each engineering environment unique.

CATI saw an opportunity to do performance testing to offset the lack of available

About CATI

Computer Aided Technology, Inc. (CATI) was founded in 1992 by engineering and manufacturing professionals as a one-stop, full-service provider of scalable design and manufacturing solutions, including 3D solid modeling and analysis, enterprise-wide data management, and rapid prototyping. CATI is the nation's original resource for SolidWorks. The company prides itself on providing outstanding support and services through a dedicated team of professionals. CATI's two wholly-owned subsidiaries, Inflow Technology and Strategic Technology Solutions, serve customers worldwide.

data on the best methods for improving SolidWorks' performance. Users can easily determine a system with the minimum power required to run SolidWorks, because the software maker explicitly spells it out. But such a system, while sufficient to launch and run the software, is hardly sufficient to coax the best performance from it. Most likely, computing intensive operations in CAD—such as rotating large assemblies, simulating assembly motions, regenerating geometry after a complex edit, or displaying ray-traced surfaces—will suffer from hiccups and sluggishness.

On the other hand, an optimized environment to operate SolidWorks is much more elusive. A well-balanced environment involves well matched hardware and software, complemented by effective modeling and data-management protocols.

For more than two decades, Fanjoy and his CATI colleagues have answered the performance question with custom solutions that suit a particular client's needs. But Fanjoy was looking for something more. He was looking for data that could drive performance recommendations for the larger universe of SolidWorks users. So Fanjoy partnered with his technical support manager, Josh Altergott, to attack this question—"What can I do to improve SolidWorks performance?"—with a scientific approach.

They began testing four years ago, and have refined their tests each year.

DEFINING THE RESEARCH METHODOLOGY

CATI decided to create a test environment that would allow them to benchmark the performance level for a series of typical SolidWorks modeling operations, given a baseline hardware and software state. The benchmark would be measured by the clock—the amount of time it took to perform a battery of common SolidWorks functions on a large assembly model. With this benchmark established, CATI would then run a series of standalone tests using alternative hardware and software settings, so it could compute the time savings from the alternative approach vs. the baseline.

The Test Environment

- A large assembly model, affectionately known as "a cluge."
- A macro script representing the daily work of a professional CAD user.
- An easily scalable workstation.

Three key elements were needed to conduct the tests: (1) a large, representative SolidWorks test model; (2) a macro script that would run a series of routinely performed operations against the test model; and (3) a workstation that could be easily scaled to test a wide range of configurations.

The Model

As the standard dataset for its tests, CATI collected a number of large assemblies designed in SolidWorks and consolidated them into a mega assembly. The individual assemblies were chosen for their tendency to cause stability and performance issues in SolidWorks. Together, they formed an unsightly cluster ("a cluge," as it was affectionately known among testers) that was calculated to stump even the most efficient CAD software. The 2.4GB mega assembly comprises:

- 6,637 total components
- 5,862 parts
- 775 subassemblies
- 663 top-level mates
- 13,011 bodies

The complexity of the assembly model used in the tests may be a rarity in the work of beginners or casual CAD users, but CATI says it corresponds to the real-world models professional SolidWorks users often encounter. After all, the mega assembly was compiled from problematic designs submitted by CATI customers for troubleshooting.

The Macro

To mimic the type of operations routinely performed by CAD users, CATI developed a macro script that automatically executes the following tasks in SolidWorks:

- Opens
- Rotations
- Modeling
- Rebuilds
- Switching Sheets
- Saves
- Closes

The macro was built to accurately represent what operations a "typical" CAD user does in his daily use of SolidWorks.

"We had the macro perform operations including, but not limited to, opening and closing of parts, assemblies and drawing files, editing parts at the assembly level, and making changes to multi-sheet drawings," says Altergott. "We also made sure that we focused on actual modeling changes that would require rebuilds that carried through several levels of subassemblies."

The Workstation

To take on such a detailed research task, CATI needed a way to easily test a wide range of configurations without jumping from one workstation to another. Keeping all of the testing on a single workstation allowed CATI to maintain the level of consistency in the environment needed to properly manage the study. Here, CATI chose to partner with workstation manufacturer BOXX Technologies.

The researchers nicknamed the system "Coyote" after Wile E. Coyote. Like its namesake, the test system would be repeatedly subjected to punishing treatments.

Founded in 1996, BOXX Technologies prides itself on configuring hardware to match particular software, such as SolidWorks. Understanding the specific configurations needed to optimize workflows in SolidWorks appealed to the Austin, TX-based company.

"When the CATI team came to us with the idea to do independent research on what software and hardware changes will improve customer productivity, it was a no-brainer for us to partner up with them," says Shoaib Mohammad, vice president of Marketing and Business Development for BOXX Technologies. "We pride ourselves on being a trusted advisor to our customers, and CATI's valuable research findings are what our customers expect from us."

BOXX Technologies provided the 3DBOXX 8550 XTREME workstation CATI used for its tests. The researchers nicknamed the system "Coyote" after Wile E. Coyote from the Roadrunner cartoon series. Like its namesake, the test system would be repeatedly subjected to punishing treatments.

Scalable to two, six-core Intel® Xeon® 5600 series processors running at 3.4GHz (4.3GHz when overclocked), the 3DBOXX 8550 XTREME is definitely a higher-end workstation. The CATI researchers chose it because it provided them access to the full spectrum of variables they wanted to test — from 2 to 12 cores and 8 to 24GB of RAM and everything in between, along with various standard and solid-state hard drives and graphic card configurations. The system could easily be scaled to the environments the researchers wanted to investigate.

"BOXX's system with two Intel® Xeon® processors gave us a vast area, a great sandbox to play in, so we could make dramatic changes to our environment quickly and easily," says Fanjoy.

Altergott agrees. "We wanted something that was top of the line so that we could make sure we were hitting the point of where SolidWorks runs ideally. BOXX's technical support was able to walk us through the changes to the BIOS and settings needed to scale the test system down for our tests."

The 3DBOXX 8550 Test System, aka Wile E. Coyote

CATI researchers used the 3DBOXX 8550 for its tests because it could be scaled down to the baseline and back up easily. Here are the resources it enabled them to test:

Processor: Up to two, six-core 3.42GHz Intel® Xeon® processors overclocked to 4.3GHz

Memory: Up to 24GB of RAM

Storage Options:
- 2 – 7200 RPM standard hard drives
- 2 – Intel® 520 Series solid-state drives
- 1 – RevoDrive 240GB PCI-E solid-state drive

GPUs: Available slots for up to seven GPUs or four dual-slot GPUs

Building a Baseline

With the building blocks in place to run a controlled, repeatable test, CATI set off to define a baseline state and the various alternative configurations that would be measured against that base. They wanted the baseline to be typical of the hardware and software used by many of their SolidWorks customers. Here's what they created:

Baseline State

Workstation	Operating/SolidWorks System Settings	Modeling Techniques
Intel® Xeon® Processor with 2 Cores at 3.46GHz	Windows 7, 64 bit	Image quality set to "High"
8GB RAM	Windows visuals at default setting	High level of detail in models
Swap managed by operating system	SolidWorks 2012 SP1	Large number of top-level mates
Operating system and files on one 7200 RPM hard drive	SolidWorks options set to default	External references
Files stored locally; no antivirus	SolidWorks Add-Ins on: Toolbox, Toolbox Browser, FeatureWorks, Simulation, PhotoView 360	Complex assembly structure that was several levels deep

The Testing Approach

CATI's experiments were conducted in three phases. The first test measured how long it took for the macro to complete its run against the test model in the baseline system configuration (above). This was the baseline benchmark.

Next, a series of tests were run that measured performance under each of the following variables: operating system settings, SolidWorks software preferences, processor speed, core count, amount of RAM and swap, hard drives, model detail levels, graphics cards, and network environment. Each variable was isolated and tested, one by one, against the macro of common SolidWorks tasks using CATI's mega model. The time it took to complete the macro script was recorded after each run, then the system was returned to the baseline settings before testing the next variable. By comparing the results to the baseline, CATI could quantify the time savings realized by each adjustment to the hardware, Windows, SolidWorks, the model and the network.

Finally, CATI created an optimal configuration state using the best settings and components as determined by the test results of the individual variables.

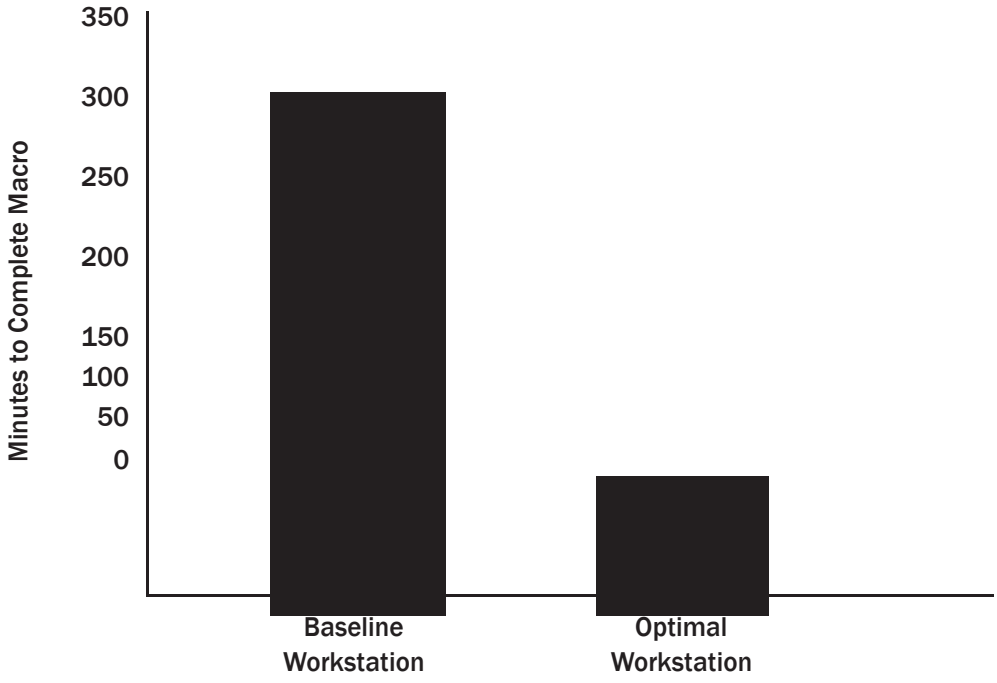
The tests displayed an astounding range of performance.

In the baseline state, the macro script took **5 hours, 1 minute and 35 seconds** to complete the common SolidWorks operations.

With the optimal configuration, the program took **55 minutes and 19 seconds** to complete — a time savings of 82% over the baseline state. In other words, the optimal state configuration completed its engineering tasks 5.5 times faster than the baseline state.

CATI's research offers a path to significant gains in the productivity of SolidWorks users. Next up, we'll detail their findings and explain how they constructed the optimal system that produced the huge time savings.

TIME SAVINGS SHOWN BY RESEARCH



CATI's research showed an optimal system and environment was 5.5 times faster than the baseline. The optimal system completed the macro in 55 minutes, 19 seconds, compared to the baseline's time of more than 5 hours.

FINDINGS THAT BOOST SOLIDWORKS PERFORMANCE

CATI researchers have determined how various aspects of a computing environment, including the hardware, software and modeling methodologies, affect SolidWorks performance. Some of their findings, like adjusting Windows and SolidWorks settings, are simple and won't cost you a dime. Other findings may provide the impetus to make investments in workstation upgrades.

The research findings[1] that are reported below are grouped into six related areas:

1. Windows and SolidWorks settings, including Windows Visual Effect settings and SolidWorks System Options, and Add-Ins.

2. The number of processor cores and processor clock speed.

3. The amount of RAM and hard drive swap.

4. Hard drive technologies, including 7200 RPM spindle and solid-state drives.

5. SolidWorks model details, including detail levels, top-level mates and image quality.

6. The optimal test system.

[1]Note: Because these tests were done in a specific environment on a specific data set, your results will vary depending on your environment and file type.

Finding 1: Don't Use Default Settings

CATI's research showed that adjusting Microsoft Windows' and SolidWorks' default settings can drive significant productivity improvement.

Whether you're an engineer or a pre-law student, Microsoft Windows 7 is shipped with the same default graphical settings to make the operating system as attractive as possible. If you're willing to give up aesthetics like drop shadows, fade-ins and animations, you'll see a performance boost.

> "If there is a SolidWorks Add-In that you're not using every day, turn it off, then just turn it on when you need it."
>
> *— Josh Altergott, CATI*

In Windows 7, the Visual Effects tab is set to "Let Windows Choose What's Best for My Computer" by default. With this setting, nearly all the Visual Effects options are turned on, including transparent glass, shadows under program windows, large thumbnails, and more. Foregoing these settings and choosing the classic Windows look by choosing "Adjust for Best Performance" in the Windows System Properties provided a 9% performance gain, according to CATI's tests. (See Appendix I for instructions on how to change Windows settings.)

Optimize SolidWorks Settings

CATI's 17+ years of being a SolidWorks reseller has taught its team how to set up the software's System Options for optimal performance. In this test, CATI makes more than 20 adjustments to SolidWorks' System Options settings. (See Appendix II for screen shots of the recommended settings.) By forcing the software to run as efficiently as possible, CATI boosted performance by 9%.

Turn off SolidWorks Add-Ins

"Everything you have turned on is taking resources," says Altergott. "If there is an Add-In that you're not using every day, turn it off, then just turn it on when you need it."

Users frequently turn Add-Ins on for a particular purpose or task — and subsequently forget to turn them off. Even when you're not actively using them, these Add-Ins demand a certain amount of memory and computing resources.

The study's baseline was run with five Add-Ins turned on: Toolbox, Toolbox Browser, FeatureWorks, PhotoView360, and Simulation. These are Add-Ins that users typically leave on, inadvertently costing them resources that can be better used elsewhere, says Fanjoy.

The performance gain from turning off these Add-Ins was 11%.

Reduce Image Quality

Many SolidWorks users set the image quality very high, with the assumption that it will improve the quality of the image that they see on screen. In actuality, the need for image quality being

Productivity Pointers

- Optimizing Windows' settings saved 25 minutes, 45 seconds, a 9% improvement.
- Optimizing SolidWorks' settings saved 27 minutes, 28 seconds, a 9% improvement.
- Turning off common SolidWorks Add-Ins saved 32 minutes, 42 seconds, an 11% improvement.
- Reducing SolidWorks' image quality saved 1 hour, 24 minutes, a 28% improvement.

> When reviewing a customer's file sent in for performance problems, one of the first settings that I check is the document's image quality.
>
> — *Josh Altergott, CATI*

set high is realized only in models that are physically very small. Along with RealView mode and Shadows in Shaded Mode, these settings provide a high-contrast, photorealistic view of an assembly, with shadows and reflections on the ground. Depending on the amount of texture and reflectivity in your model, the lighting and shadow calculations required to display the model with this high level of realism will tax your system resources. Such high image quality settings are often not necessary for day-to-day design tasks.

Instead, choose a simpler display setting by turning off RealView Graphics and Shadows in Shaded Mode. Decrease the image quality slider bar in the Document Properties dialog box until it's about a fourth of the way up from the low end of the scale.

The effect of high image quality display mode is a model that takes longer to calculate its visual data and a significantly larger file size, according to Fanjoy. The former affects rebuild and repaint times, while the latter affects model opening, closing, and saving. Both affect the amount of RAM required, which amplifies the performance hit.

"When reviewing a customer's file sent in for performance problems, one of the first settings that I check is the document's image quality," says Altergott.

CATI's research found that reducing the image quality of the assembly to a low level and applying the changes to all subcomponents of the assembly coupled with turning off RealView and Shadows provided a 28% SolidWorks performance boost.

Check Your Templates

Longtime SolidWorks users should check their templates. In the past, SolidWorks set a high level for its image quality on the default templates. Anyone who based their templates on those old defaults may still be carrying that legacy.

"Users may not know that image quality can be stored inside of a document template," says Altergott. "The only way to be sure is to check each of your templates and modify them accordingly."

Image quality can be changed at the Assembly or Drawing level, and propagated down to all of the individual parts.

Finding 2: Fast, Multicore Processors Matter

As the primary regulator performing the lion's share of the work, the central processing unit (CPU) is the heart of the workstation. Today, multicore CPUs have become the standard in professional workstations. You can purchase a system with a single CPU equipped with 2, 4, 6 or 8 cores. You also have the option to add a second processor to your system, thereby doubling core counts. The test system from BOXX, for instance, came with two Intel® Xeon® CPUs, each equipped with 6 cores, giving the system a total of 12 cores.

"Since the tests were performed, Intel has announced its new Intel® Xeon® processor E5 -1600/2600 product families, which offer up to a 3 times increase in processing performance."

— Wes Shimanek, Intel

For the most part, the computation required to solve feature-based CAD geometry is linear (in other words, one operation must complete before another begins). For this reason, a majority of the tasks in SolidWorks remain single-threaded operations that don't fully take advantage of multiple cores. It can perform some multi-threaded operations, but only in certain areas, such as retrieval of large files, photorealistic rendering, interference checking, collision detection, and simulation.

At minimum, SolidWorks and Windows will be accessing the processor simultaneously, so at least 2 cores are a must. "If you want to run SolidWorks on 2 cores, you need to turn everything else off," says Fanjoy. "And you're still going to be sharing bandwidth with the operating system. If you're doing anything of any substantial size or complexity, I wouldn't recommend going with anything less than 4 cores."

CATI's tests showed that increasing the core count from 2 to 4 saved 35 minutes when running the macro script, which is a 12% performance increase.

CATI tested performance at the 6-, 8- and 12-core levels, which showed some, but not a substantial gain. That wasn't a surprise.

"What we'd like everyone to realize is that our test environment is sterile," says Fanjoy. "So we only ran SolidWorks and Windows for the bulk of our tests."

Productivity Pointers

- Going from 2 to 4 cores saved 35 minutes, a 12% performance increase.
- When using computationally intensive engineering programs, a four-fold increase was seen from 4 to 12 cores.
- Overclocking the processor from 3.42GHz to 4.29GHz increased the test speeds by 12%.

The Impact of Graphics Cards

Because graphics processing unit (GPU) performance is tied to programs written to take advantage of their capabilities, CATI recommends choosing a professional graphics card that has been tested with, and certified by, the applications you plan to use.

Make sure your visualization requirements will leverage advanced features of midrange and high-end graphics adapters. The data set in CATI's SolidWorks tests did not, so there was little performance difference between an entry-level and high-end graphics adapter.

"We realized we need a unique dataset to measure GPU performance," says Fanjoy.

CATI plans to delve more deeply into the effects of high-end graphics cards on SolidWorks tasks in the future, but its tests against the macro indicate that unless users need advanced visualization features, a basic professional graphic card is sufficient.

But in the real world, a typical day for an engineer using SolidWorks also includes using software to perform simulation and analysis, and/or rendering. While four cores is the minimum recommended for SolidWorks, for many engineers that simply isn't enough. To prove this, CATI ran a test with a third-party program that did calculations on a single processor to simulate other applications running. As expected, the results of this experiment showed that with another application taking significant processor resources, 5 cores were required to achieve the same performance level that they previously experienced with 4 cores.

The researchers were also curious to know what effect multiple cores would have on other common engineering tasks. They decided to test ray-traced rendering via PhotoView 360, which takes advantage of multicore processors. This is where multicore processors really shine. They discovered a four-fold increase between four and 12 cores. The same rendering task that took a little over four hours when processed on four cores took just over one hour to complete on 12 cores.

> "The net result is BOXX has shipped overclocked systems to thousands of customers — each benefitting from increased performance and BOXX's built-in reliability."
>
> *— Shoaib Mohammad, BOXX Technologies*

Overclocking Considerations

In addition to the advantages of multiple cores, raw processor speed can also make a big difference. Because SolidWorks is largely single-threaded, processor clock-speed offers a straightforward return on investment. Simply put, the faster the processor speed is, the more instructions it can complete in a set amount of time.

Faster processors cost more, but there's another option. Overclocking a processor — making it run faster than the manufacturer's specified speed — allows engineers to reap the speed benefits of faster processors at slower processor prices.

But there's an inherent danger in overclocking. If not done correctly, the additional heat produced by overclocking can seriously damage a workstation. It's best to leave overclocking to the experts. Some vendors, like BOXX, overclock their systems' processors and provide full warranties, giving you the fastest processor possible for the money.

"As an Intel® Technology Provider Program Platinum partner, BOXX has the opportunity to work with Intel engineers to ensure that we stay within the parameters of safe overclocking, providing increased performance without applying additional risk to the system's reliability," says BOXX Technologies' Mohammad. "The net result is BOXX has shipped overclocked systems to thousands of customers — each benefitting from increased performance and BOXX's built-in reliability."

Mohammad notes that BOXX only overclocks high-end processors. It also adds liquid cooling to avoid any risks associated with forcing faster speeds.

The two Intel® Xeon® processors included in the test system operate at speeds of 3.42GHz by default. But because BOXX supports overclocking, the processors could be tested at the faster speeds of 3.75GHz, 4.02GHz and 4.29GHz.

At the top end, an overclocked system at 4.29GHz resulted in 12% faster performance vs. the baseline setting of 3.46GHz.

Finding 3: Max out RAM to Minimize Swap

The Achilles' heel in a CAD system often turns out to be memory—specifically, insufficient memory.

"Improvement that you may gain from improved processor speed can be easily lost if the system is short on RAM (random access memory) for the given task," Fanjoy says.

It's become an adage in computing: "Want more speed? Buy more RAM." CATI's research proves that to be true. In CATI's tests, increasing the amount of RAM to 24GB provided a 54% improvement over the 8GB of RAM in the baseline system. Professionally certified high-speed RAM costs about $40 to $60 per gigabyte. So that means for less than $1,000, you could double your SolidWorks performance. That's an astounding value.

Increasing RAM to 16GB provided the lion's share of the two hours and nearly 45 minutes of time savings that increasing it all the way to 24GB provided in CATI's tests. However, it is best to configure your workstation with enough RAM for your models and leave ample room to grow.

> "I'd like you to be able to have ample RAM that allows you not to touch the swap."
>
> *— Adrian Fanjoy, CATI*

Increasing RAM provided benefits to every operation in CATI's test macro. One of the main reasons for such a large time savings is because a large amount of RAM helps to avoid using the hard drive as memory overflow. When the workstation's RAM proves insufficient to accommodate a design (often the case with large assemblies), the workstation resorts to paging, or borrowing a segment of available hard drive space to temporarily store and retrieve the active data. This operation, called swap, causes system slowdowns.

How to Best Manage Swap

By default, the Windows operating system is set to manage the swap size, which is the hard disk space used to read and write data that cannot fit into your system's physical memory. Windows wasn't designed to automatically allocate the optimal swap size for SolidWorks, but it's easy to manually make such adjustments (see Appendix I).

"I'd like you to be able to have ample RAM that allows you not to touch the swap," Fanjoy says. "With smaller amounts of RAM, we've always said to use two times RAM for your swap file size. When you get to 16GB to 24GB of RAM, though, you should be able to dial it back to 1x. It all depends on model size."

As your models grow, you'll eventually need to access the hard drive for swap. Additional RAM will let you avoid that as long as possible, but when that's not enough the speed of the hard drive becomes a critical factor.

"The reality of the situation is that anytime you have to use swap, hard drive speed becomes hyper-critical," says Altergott.

Productivity Pointer

- Increasing RAM from 8GB to 24GB saved 2 hours, 44 minutes and 54 seconds, a 54% improvement.

Finding 4: You Need Solid-State Hard Drives

"As soon as I put the operating system and work files on an SSD, we started getting a huge benefit," says Fanjoy.

Hard drive speed—the rate at which the storage drive allows you to write data to it and read data from it—affects SolidWorks modeling sessions dramatically, especially when a single drive is used for data storage, Windows operations, and swap. Newer solid-state drives (SSDs) yield better performance than older spindle-equipped hard drives, whose speed is measured in revolutions per minute (hence, RPM).

While SSDs are more expensive than spindle drives, CATI tests showed that, with the swap and Windows residing on a single Intel® Solid State Drive 520 series, the performance was 47% faster than the same setup on a standard 7200 RPM hard drive. That speed boost would cost you about $200 for a 120GB Intel® 520 Series SSD. That's just $100 more than a 120GB 7200 RPM spindle drive. Such a small investment would pay for itself very quickly.

But to speed things up even more, CATI put work files on one SSD and the operating system and swap on another. When the test was rerun with the operating system and swap on two SSDs configured as RAID 0, and the assembly stored on a separate Revo solid-state PCI-e card, CATI saw a 56% improvement in performance.

Productivity Pointer

- Replacing the 7200 RPM hard drive with a solid-state hard drive yielded a 47% increase.
- Further optimizing the hard drive configuration saved 2 hours and 13 minutes, a 56% improvement.

Suspending Functionality via Large Assembly Mode

The option that produces the biggest difference in SolidWorks, CATI discovered, is one many engineers are reluctant to use. The Large Assembly Mode allows you to load a large assembly with enough visual details to inspect it, but without some of the component-level details that require far more computing power. Working in this mode, CATI noted that performance improved by 89%, meaning that the benchmark that ran in just over five hours completed in less than 35 minutes.

That performance increase is offset, however, by the suspension of some display features and operations normally available in SolidWorks. For instance, model faces, edges and vertices are not highlighted when you move the pointer over a sketch, model or drawing. When moving or rotating the model, you'll notice a lower quality in transparency. The assembly loads faster in this mode because it loads without some of the lightweight components. Depending on how you set your Large Assembly threshold, some automatic rebuild functions may also be suspended.

"I always get a lot of pushback when suggesting customers use Large Assembly Mode for various reasons," says Altergott. "There were issues with it at one point in time. There are a lot of settings that could be done incorrectly. If you do the right things, the functionality works great. However, an untrained user steps on their toes more often than not."

Finding 5: The Devil is in the Model Details

The more details in a model, the bigger the hit to performance. Over the last two decades, CATI has found many engineers work with models that have an unnecessarily high level of details — and they might not even realize it.

"We have found that this issue occurs most frequently with components that are downloaded from vendors that have made their models available for easy access and configuration over the web," Altergott says.

CATI's recommendation is to remove any excess details from parts and assemblies after download. If feasible, CATI suggests:

- Reducing the level of detail to outer walls and mounting points.
- Saving assemblies as parts.
- Combining solid bodies.
- Turning surface bodies into solid bodies.
- Deleting any unneeded surface or solid bodies.
- Exporting and reimporting files to reduce the number of features in a files.

Certain standard components that remain hidden, such as screws and bolts, may also contain a high level of detail, depending on the form of threaded holes they feature. Extruded texts and patterned features may also add to the memory demands of the assembly.

Avoid Network Bottlenecks with PDM

The growing model complexity and shared workflow among SolidWorks users often lead to networked storage where parts of an assembly, or the entire assembly, may be stored on a network drive.

"Using the data across a network can greatly damage productivity when compared with using the same data locally," says Fanjoy. "Our testing showed a 16% performance degrada- tion from our baseline when operating over the network."

Factors that contribute to the performance degradation include the data reading/writing speed of the hard drives, network bandwidth, server capacity, server hard drive speed and bandwidth-sapping virus scanning operations.

Redundant virus scanning at the network and local levels further deprives SolidWorks of processing power. In CATI's tests, when virus scanning was active on the server only, the benchmark ran 13% slower than when working over the network without scanning. When virus scanning was active on both the server and the client, the benchmark ran 18% slower than when working over the network without scanning.

Of course, in midsized and large enterprises, it's nearly impossible to avoid network storage or virus scanning. It's essential to collaborative product development to have multiple engineers working on different parts of the design at the same time.

The use of a product data management (PDM) system not only allows files to be checked out and worked on locally for increased speed, it also decreases unnecessary file duplication (a common problem stemming from emailing files). The systematic approach to handling externally referenced documents could also reduce data complexity, thus improving file load times and software performance in modeling sessions.

> In our tests, setting our Flexible subassemblies to Rigid dropped the number of mates being evaluated at the top level from 962 to 273.
>
> *— Josh Altergott, CATI*

When this level of detail is required, CATI recommends having a configuration of the part or subassembly that suppresses the detail for use in larger assemblies.

It can make a big difference, especially in complex files like CATI's test model. The researchers were able to boost SolidWorks performance vs. the baseline by 32% by removing excess details.

Reduce Top-Level Mates and Flexible Subassemblies

In assembly models, assembly mates determine the relationships among subassemblies (for instance, the alignment between one surface on a component and an adjacent surface that belongs to another component). When SolidWorks loads a large assembly with many smaller subassemblies, calculating and resolving these geometric relationships put a heavy burden on the system. The trick to reduce that burden is to also reduce the number of top-level assembly mates by solving subassemblies as Rigid instead of Flexible via SolidWorks' Component Properties.

Reducing top-level assembly mates is consistent with best practices in CAD modeling. By nesting the mating relationship at the subcomponent or subassembly level, you reduce the number of mate relationships the software must calculate and resolve at load time.

"In our tests, setting our Flexible subassemblies to Rigid dropped the number of mates being evaluated at the top level from 962 to 273," according to Altergott. "This improved our benchmark run time by 15%."

Productivity Pointers

- Removing unneeded details from models saved 1 hour, 37 minutes and 22 seconds, a 32% improvement.
- Reducing top-level mates and flexible subassemblies saved 45 minutes and 22 seconds, a 15% improvement.

Finding 6: The Optimal State

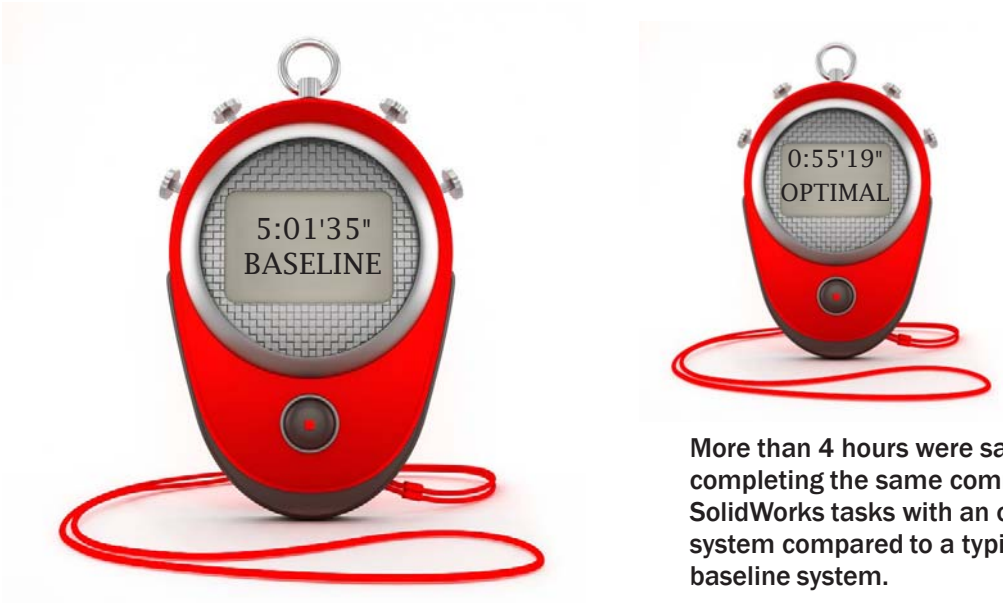
Each of CATI's findings shows impressive performance gains. But the real jaw-dropper came when the researchers combined the best options into one awesome system. Here's what the optimal environment looked like for the test macro and model:

The Optimal Test Environment

Workstation	Operating/SolidWorks System Settings	Modeling Techniques
Intel® Xeon® Processor with 12 Cores at 4.29GHz	Windows 7, 64 bit	Image quality reduced
24GB RAM	Windows at visual set to Best Performance	Level of detail reduced
OS on RAID 0 Intel® Solid State Drive 520 series	SolidWorks options optimized	RealView graphics off
Files on OCZ RevoDrive PCI-Express SSD	SolidWorks Add-Ins off: Toolbox, Browser, FeatureWorks, Simulation, PhotoView 360	
NVIDIA Quadro series Graphics Card	Assembly stored locally	

Using the optimal system, CATI researchers were able to decrease the baseline time of 5 hours, 1 minute and 35 seconds down to just over 55 minutes. That means the optimal system set-up completed its engineering tasks 5.5 times faster than the baseline configuration.

But what do these findings really mean for you? Take a look at the next section for what you can do to achieve significant productivity gains.



More than 4 hours were saved when completing the same common SolidWorks tasks with an optimal system compared to a typical baseline system.

CONCLUSIONS & NEXT STEPS

CATI's research findings demonstrate several steps that you can take to significantly improve SolidWorks performance. Every user experience will be different, depending on your model complexity, hardware, modeling methods, other software you use that might strain your system, and bottlenecks specific to your computing environment. You may not attain as dramatic a speed boost as CATI did, or you may see even greater improvements.

But if you've ever been frustrated by sluggish design tools, CATI's findings report compelling data on productivity and time savings that could be gained by making changes to your software settings, hardware environment and modeling techniques. These recommended steps will result in greater design interactivity, which offers you the potential to create many more ideas for every challenge faced. Being in that interactive state is the key to realizing the creative and competitive promises offered by engineering technology.

Here's a re-cap of what to do next:
1. Start with the software tweaks. Follow CATI's suggestions to change the default settings in Windows and SolidWorks, and you'll realize significant time savings.

2. Take a look at how your models are set up. Determine how high the image quality and level of detail need to be for you to do your job, and reduce accordingly. Reducing details can be a significant investment in time, but if it's a model you're likely to be accessing frequently, it's worth it. Reducing top-level mates, on the other hand, is an easy fix with substantial returns.

3. Determine whether your workstation is an asset or liability. Does it advance your quest to significantly increase the number of design ideas you create and evaluate? Robust processors, plentiful RAM and fast hard drives are essential elements needed by creative, highly productive engineering teams.

Making the Case for Workstation Investment

While software tweaks are free, upgrading your workstation requires a call for cash. And it's a purchasing process that frustrates many engineers—especially when you're challenged to define the value that will be realized on the hardware investment.

The return on investment (ROI) for engineering technology has two key drivers. One is the time value of enhanced engineering productivity. *Desktop Engineering* has reviewed many ROI calculators to help estimate the value earned from workstations that make design engineers more productive. For powerful workstations that cost between $3,000 and $4,000, a 25% improvement in engineering productivity will typically result in an investment payback in six months.

The second ROI driver is the sales growth potential resulting from optimized products. That optimization is realized via hardware and software tools that enable numerous, rigorously tested design iterations. This return is much harder to compute, but it's potentially much more important.

In many organizations, the specific workstation needs of engineers are not understood by the people who specify and approve computer purchases. Engineers want workstations that will save them time, which will allow them to create better products. IT purchasers want to save money and follow corporate purchasing policies.

Ask the Right Questions

To communicate the need for a new workstation with ample RAM, a fast and capable processor, and a solid-state drive, engineers need to speak the language of financiers. Make your case for a new workstation using CATI's research results framed by these questions:

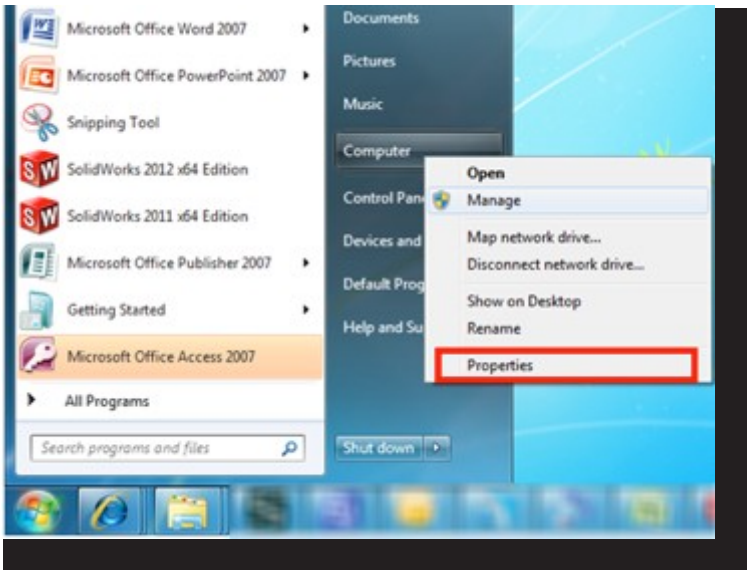
- Would you invest in technology that could pay for itself in six months?
- What new projects could the company take on if its engineers were 20%, 40%, 60% or even 80% more productive?
- Is it better to increase output by hiring more engineers or maximizing the productivity of the engineers already on staff?
- What is the value of repeatedly bringing products to market that meet customer needs better than any of our competitors?
- How could improving the productivity of front-end design work multiply productivity through our entire design and manufacturing process?
- What opportunities have we missed in the past by not being able to react more quickly to shifts in the market?
- What is the value of avoiding recalls and engineering change orders by having time to get the design right the first time?

Approaching workstation technology investments with this type of critical thinking is what leads engineering teams to the Promised Land of optimum productivity, which dwarfs the necessary technology investments.

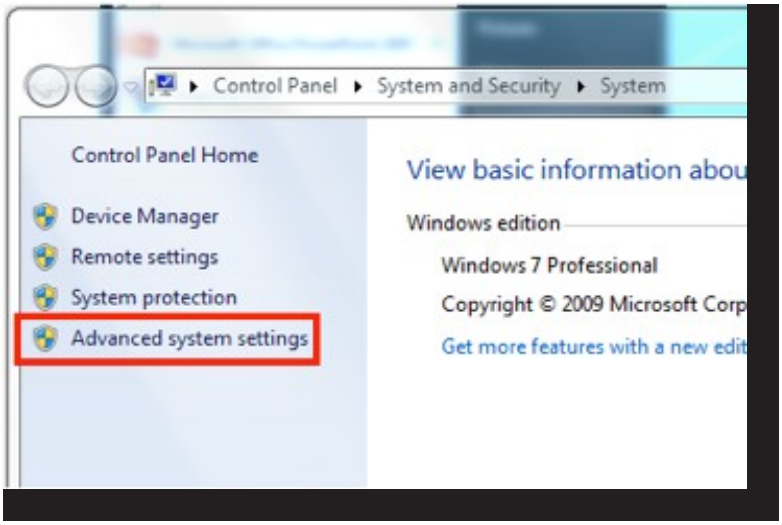
Appendix I

How to Set the Virtual Memory and Windows Performance Options

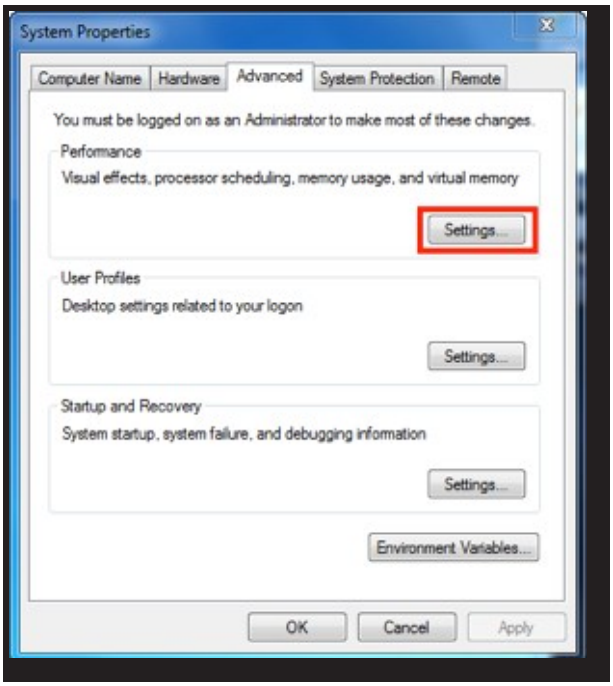
1. Go to Start then right-click on Computer, then select Properties.



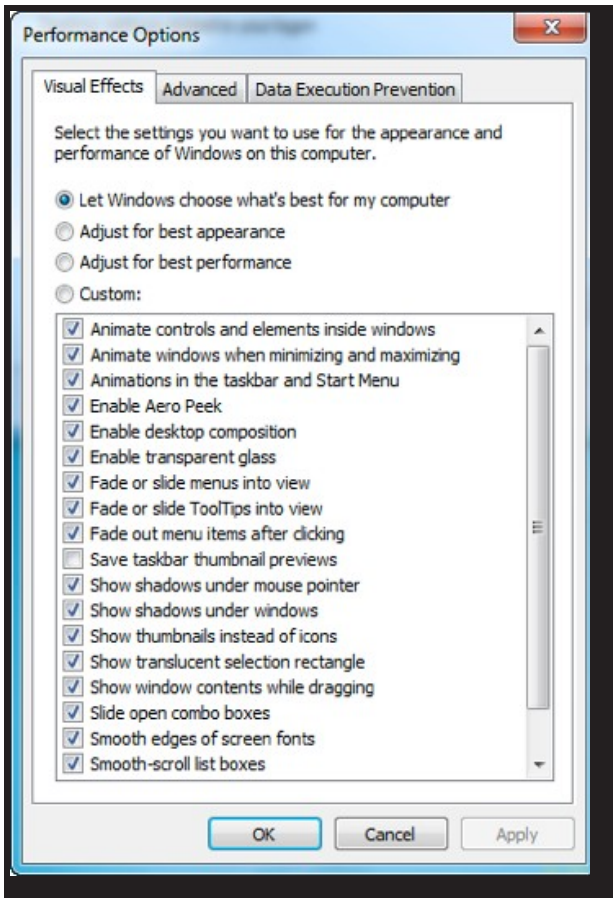
2. Select Advanced system settings.

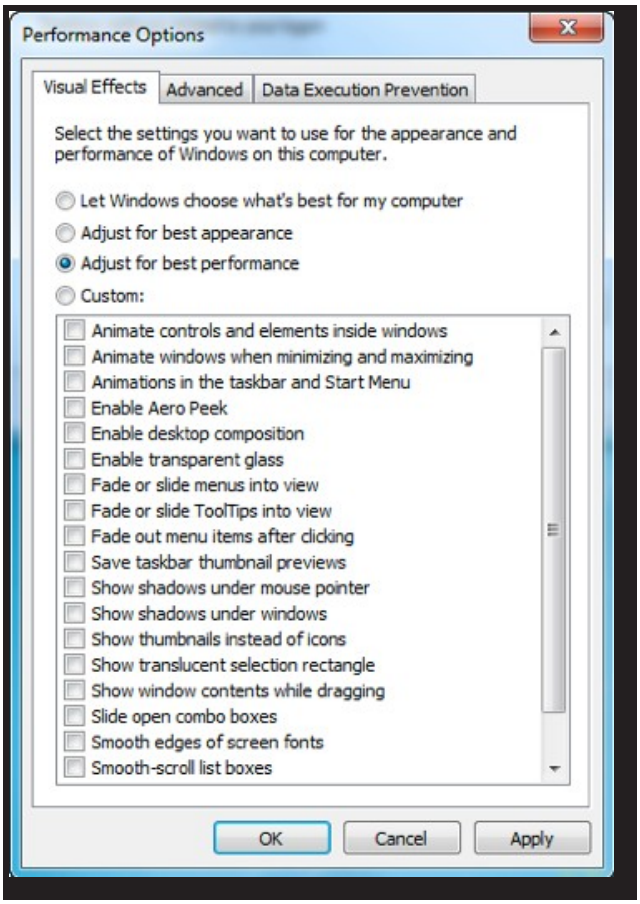


3. Select the Settings in the Performance area of the Advanced tab.

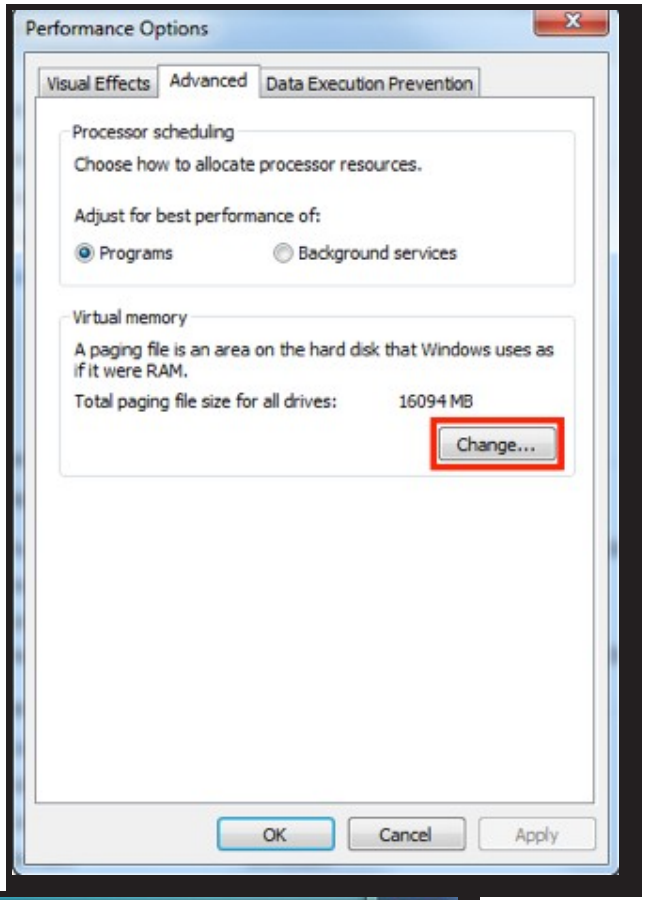


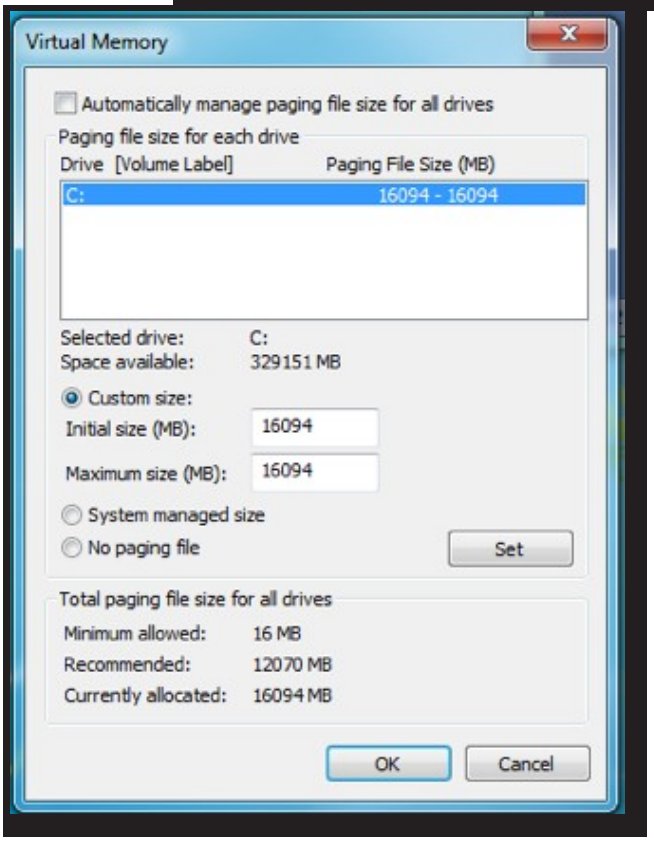
4. From the Visual Effects tab, select "Adjust for best performance" to turn off all of Windows' aesthetics.





5. Back in the Advanced tab, select Change in the Virtual Memory area.



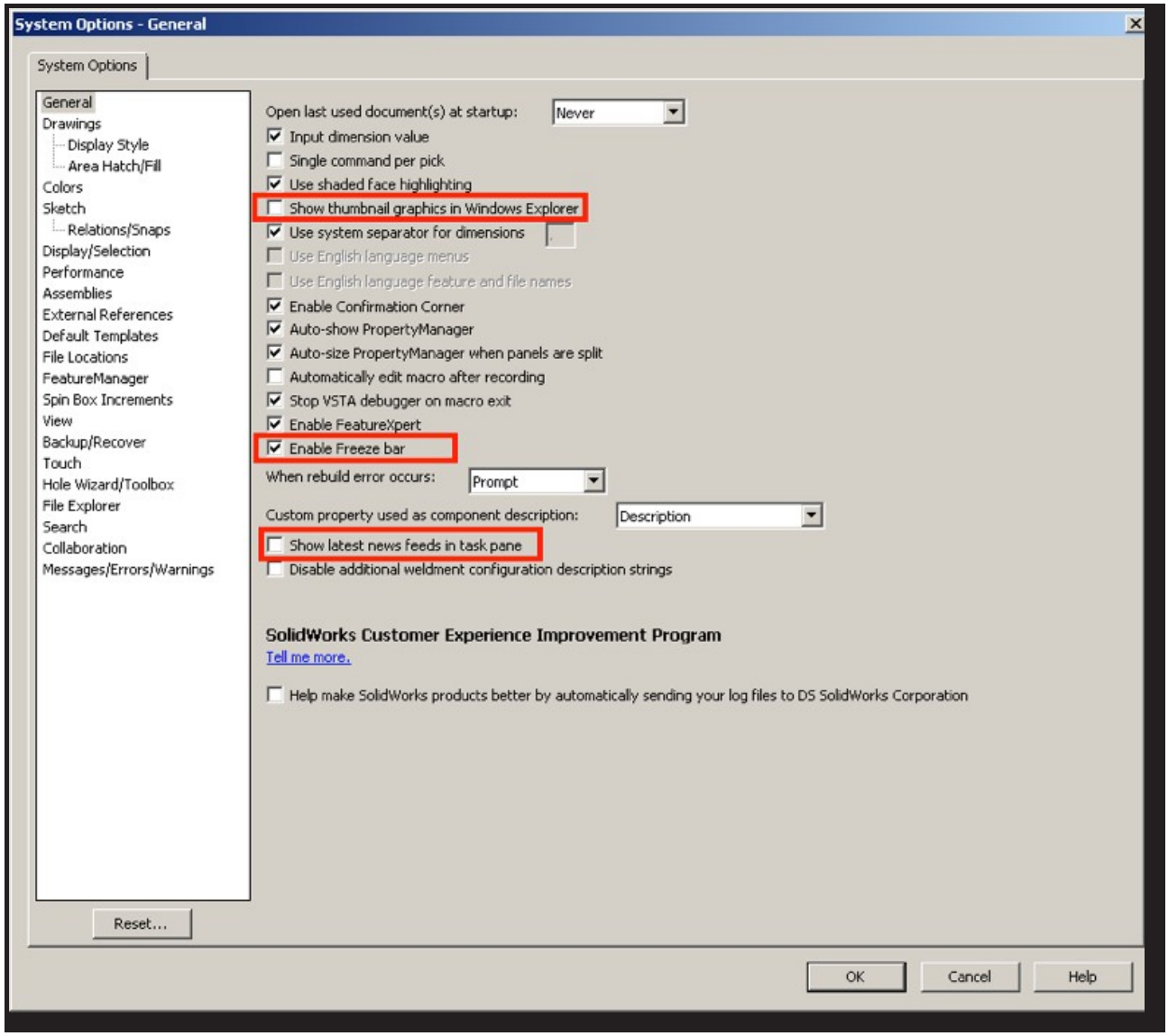


6. In the Virtual Memory window, set the Initial and Maximum sizes to be the same amount, which should be twice the amount of RAM in the machine.

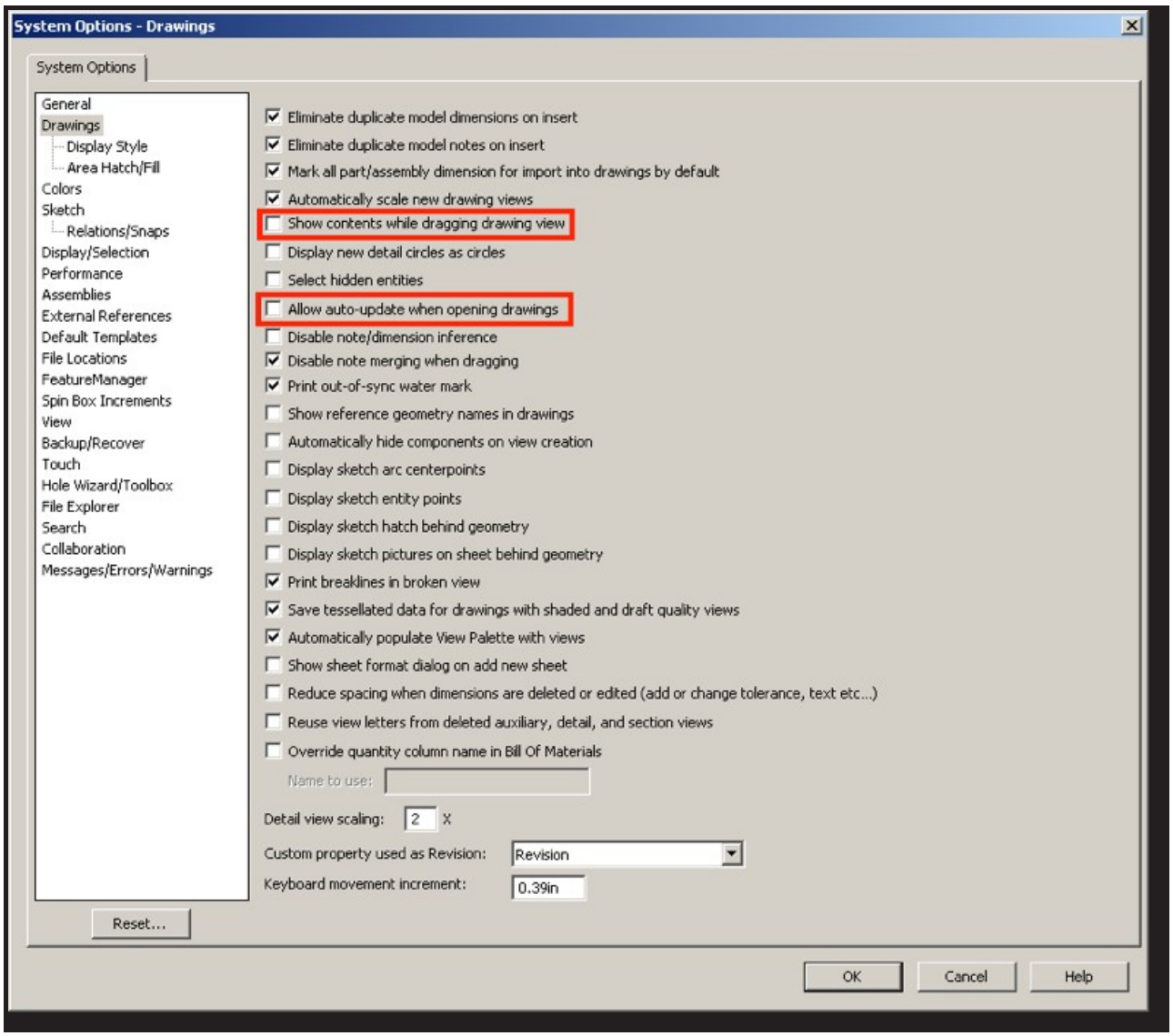
Appendix II

SolidWorks 2012 Optimal System Setting Options

The screen shots that follow show the changes you can make to SolidWorks' default settings (outlined in red boxes) to achieve the best performance.



Make sure that **When rebuild error occurs** is set for **Prompt**.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Relations/Snaps
Display/Selection
Performance
Assemblies
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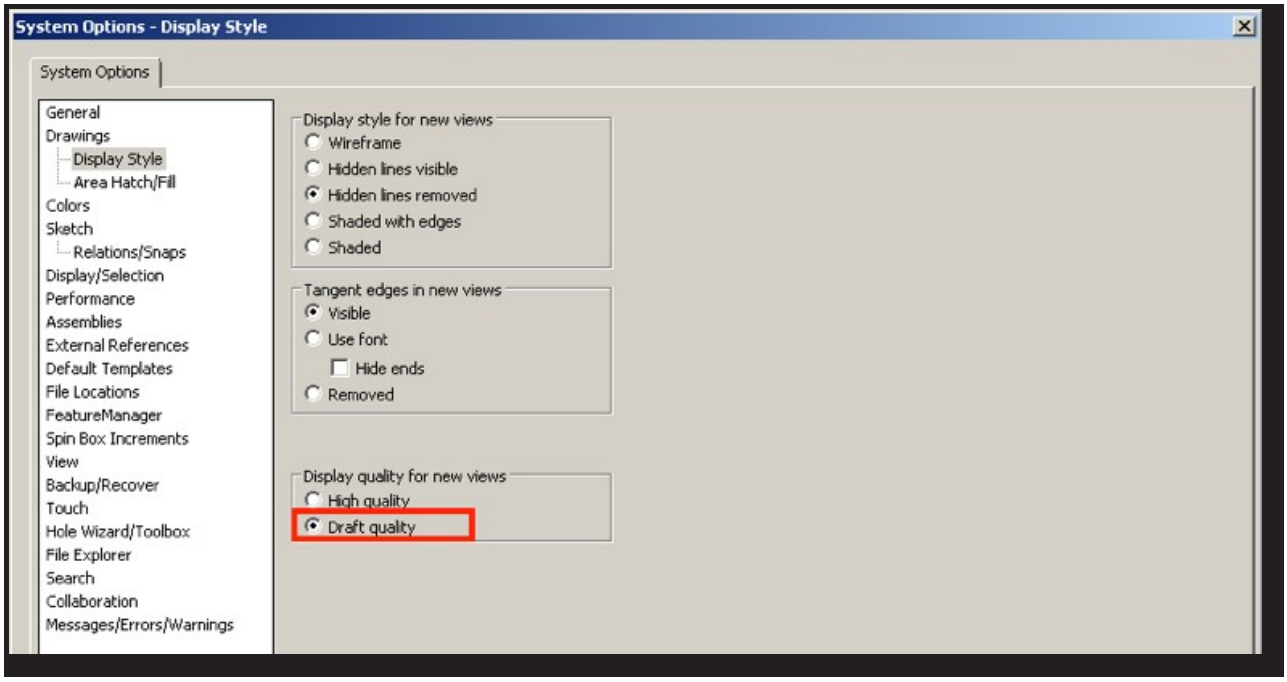
Default Templates
File Locations
FeatureManager
Spin Box Increments
View
Backup/Recover
Touch
Hole Wizard/Toolbox
File Explorer
Search
Collaboration
Messages/Errors/Warnings
Eliminate duplicate model dimensions on insert
Eliminate duplicate model notes on insert
Mark all part/assembly dimension for import into drawings by default
Automatically scale new drawing views
Show contents while dragging drawing view
Display new detail circles as circles
Select hidden entities
Allow auto-update when opening drawings
Disable note/dimension inference
Disable note merging when dragging
Print out-of-sync water mark
Show reference geometry names in drawings
Automatically hide components on view creation
Display sketch arc centerpoints
Display sketch entity points
Display sketch hatch behind geometry
Display sketch pictures on sheet behind geometry
Print breaklines in broken view
Save tessellated data for drawings with shaded and draft quality views
Automatically populate View Palette with views
Show sheet format dialog on add new sheet
Reduce spacing when dimensions are deleted or edited (add or change tolerance, text etc...)
Reuse view letters from deleted auxiliary, detail, and section views
Override quantity column name in Bill Of Materials
Name to use:
Detail view scaling: 2 X
Custom property used as Revision: Revision
Keyboard movement increment: 0.39in
Reset...
OK
Cancel
Help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Relations/Snaps
Display/Selection
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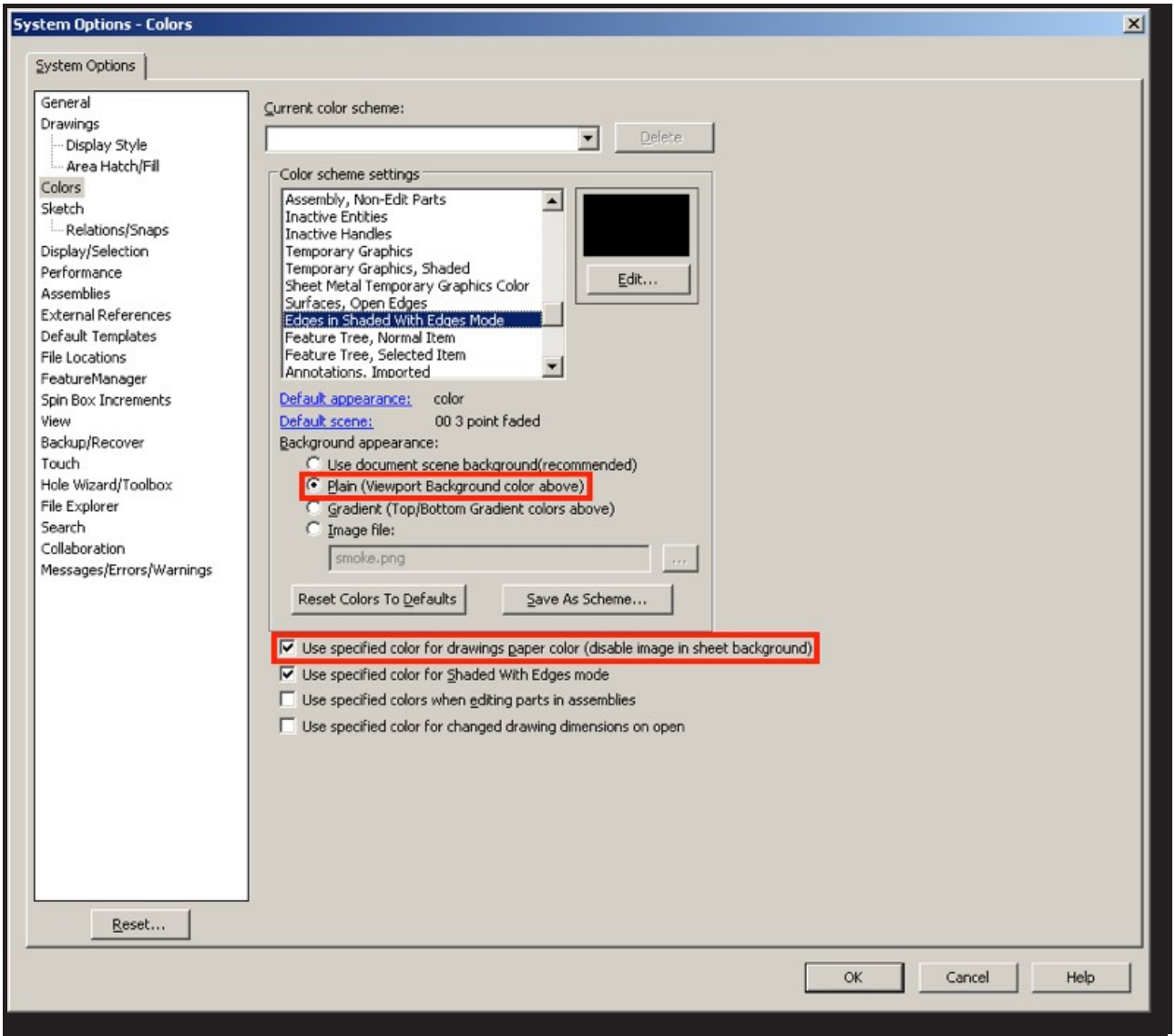
Default Templates
File Locations
FeatureManager
Spin Box Increments
View
Backup/Recover
Touch
Hole Wizard/Toolbox
File Explorer
Search
Collaboration
Messages/Errors/Warnings
Display style for new views
Wireframe
Hidden lines visible
Hidden lines removed
Shaded with edges
Shaded
Tangent edges in new views
Visible
Use font
Hide ends
Removed
Display quality for new views
High quality
Draft quality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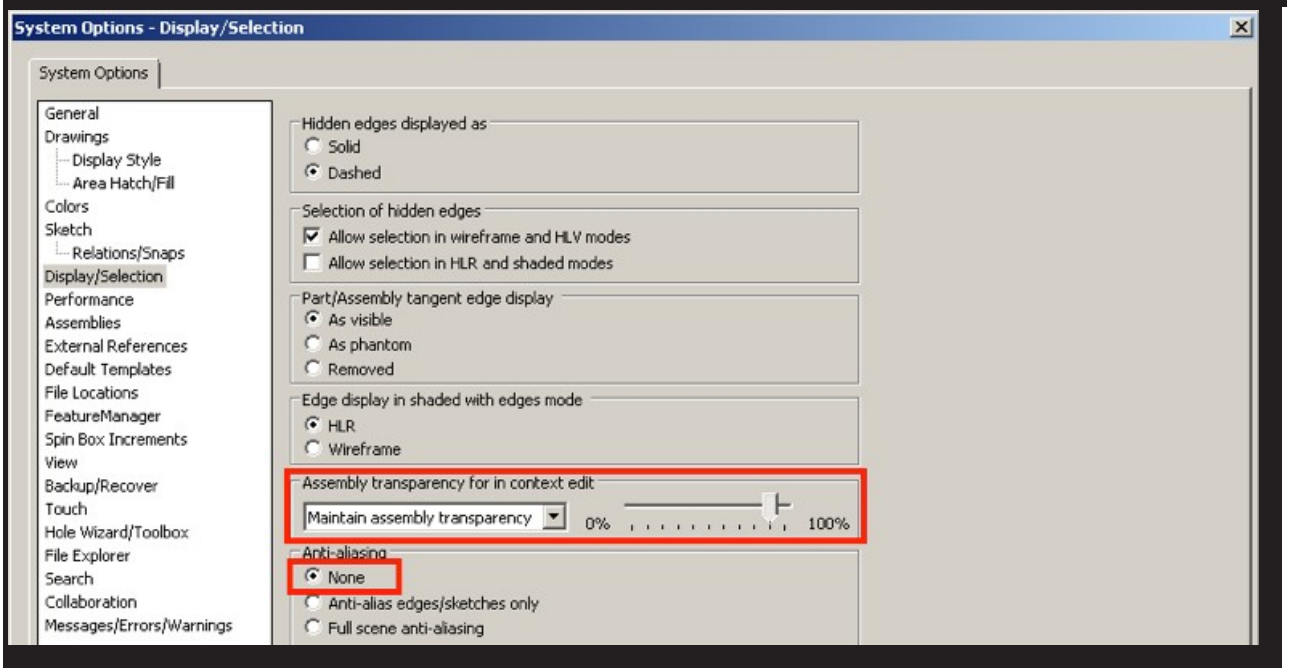
Default Templates
File Locations
FeatureManager
Spin Box Increments
View
Backup/Recover
Touch
Hole Wizard/Toolbox
File Explorer
Search
Collaboration
Messages/Errors/Warnings
Reset...
Current color scheme:
Delete
Color scheme settings
Assembly, Non-Edit Parts
Inactive Entities
Inactive Handles
Temporary Graphics
Temporary Graphics, Shaded
Sheet Metal Temporary Graphics Color
Surfaces, Open Edges
Edges in Shaded With Edges Mode
Feature Tree, Normal Item
Feature Tree, Selected Item
Annotations, Imported
Edit...
Default appearance: color
Default scene: 00 3 point faded
Background appearance:
Use document scene background(recommended)
Plain (Viewport Background color above)
Gradient (Top/Bottom Gradient colors above)
Image file:
smoke.png
Reset Colors To Defaults
Save As Scheme...
Use specified color for drawings paper color (disable image in sheet background)
Use specified color for Shaded With Edges mode
Use specified colors when editing parts in assemblies
Use specified color for changed drawing dimensions on open
OK
Cancel
Help


System Options - Display/Selection
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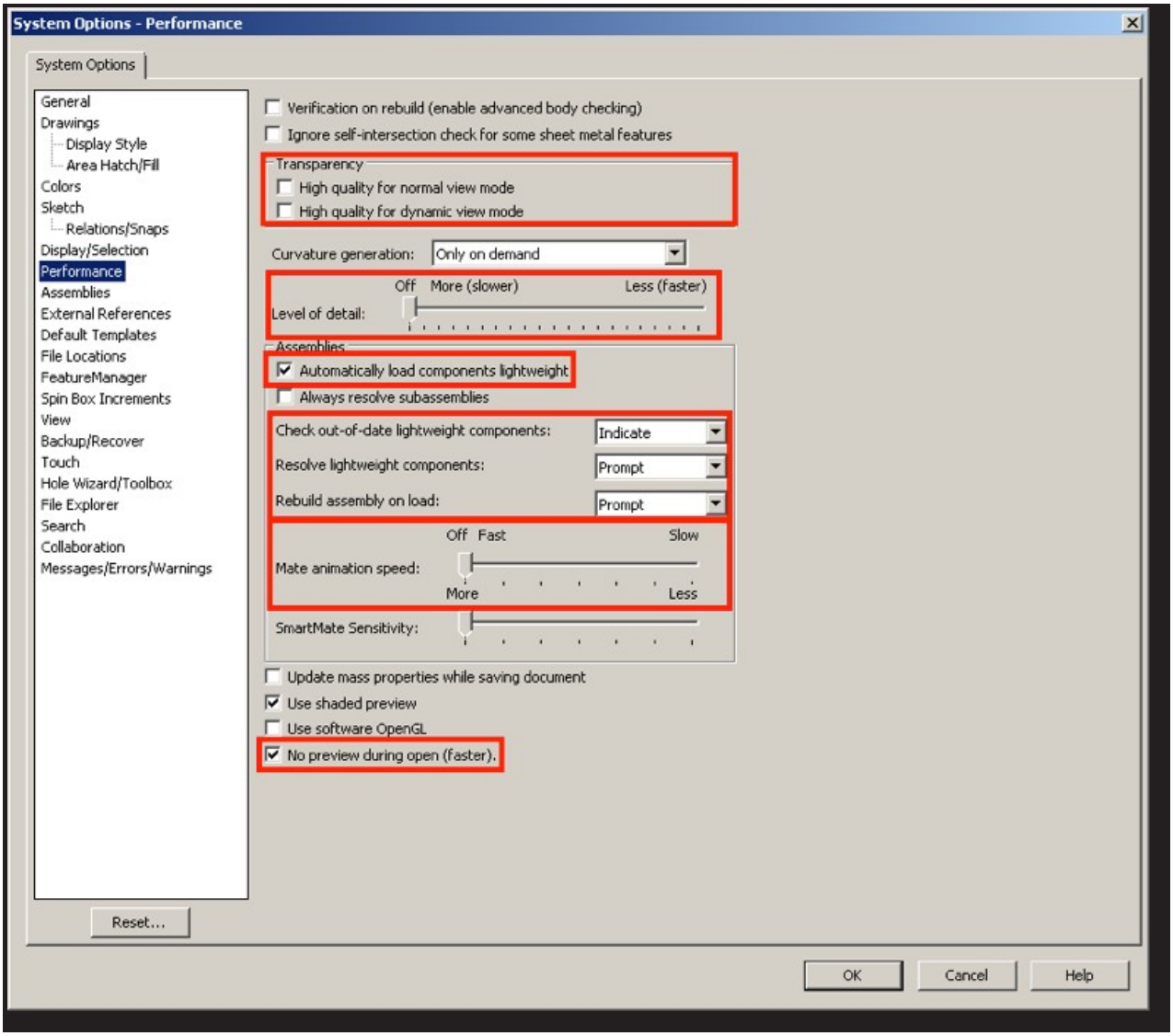
Spin Box Increments
View
Backup/Recover
Touch
Hole Wizard/Toolbox
File Explorer
Search
Collaboration
Messages/Errors/Warnings
Hidden edges displayed as
Solid
Dashed
Selection of hidden edges
Allow selection in wireframe and HLV modes
Allow selection in HLR and shaded modes
Part/Assembly tangent edge display
As visible
As phantom
Removed
Edge display in shaded with edges mode
HLR
Wireframe
Assembly transparency for in context edit
Maintain assembly transparency
0%
100%
Anti-aliasing
None
Anti-alias edges/sketches only
Full scene anti-aliasing


System Options - Performance
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Display/Selection
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External References
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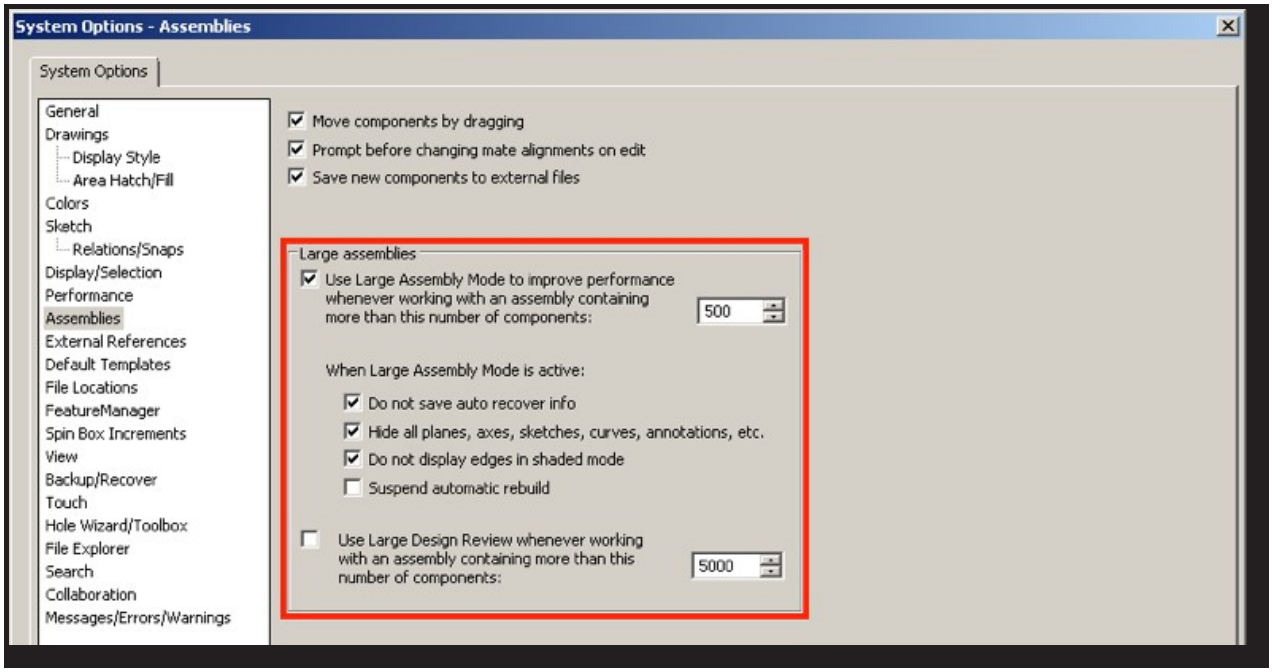
File Locations
FeatureManager
Spin Box Increments
View
Backup/Recover
Touch
Hole Wizard/Toolbox
File Explorer
Search
Collaboration
Messages/Errors/Warnings
Verification on rebuild (enable advanced body checking)
Ignore self-intersection check for some sheet metal features
Transparency
High quality for normal view mode
High quality for dynamic view mode
Curvature generation:
Only on demand
Off More (slower) Less (faster)
Level of detail:
Assemblies
Automatically load components lightweight
Always resolve subassemblies
Check out-of-date lightweight components:
Indicate
Resolve lightweight components:
Prompt
Rebuild assembly on load:
Prompt
Off Fast Slow
Mate animation speed:
More Less
SmartMate Sensitivity:
Update mass properties while saving document
Use shaded preview
Use software OpenGL
No preview during open (faster).
Reset...
OK
Cancel
Help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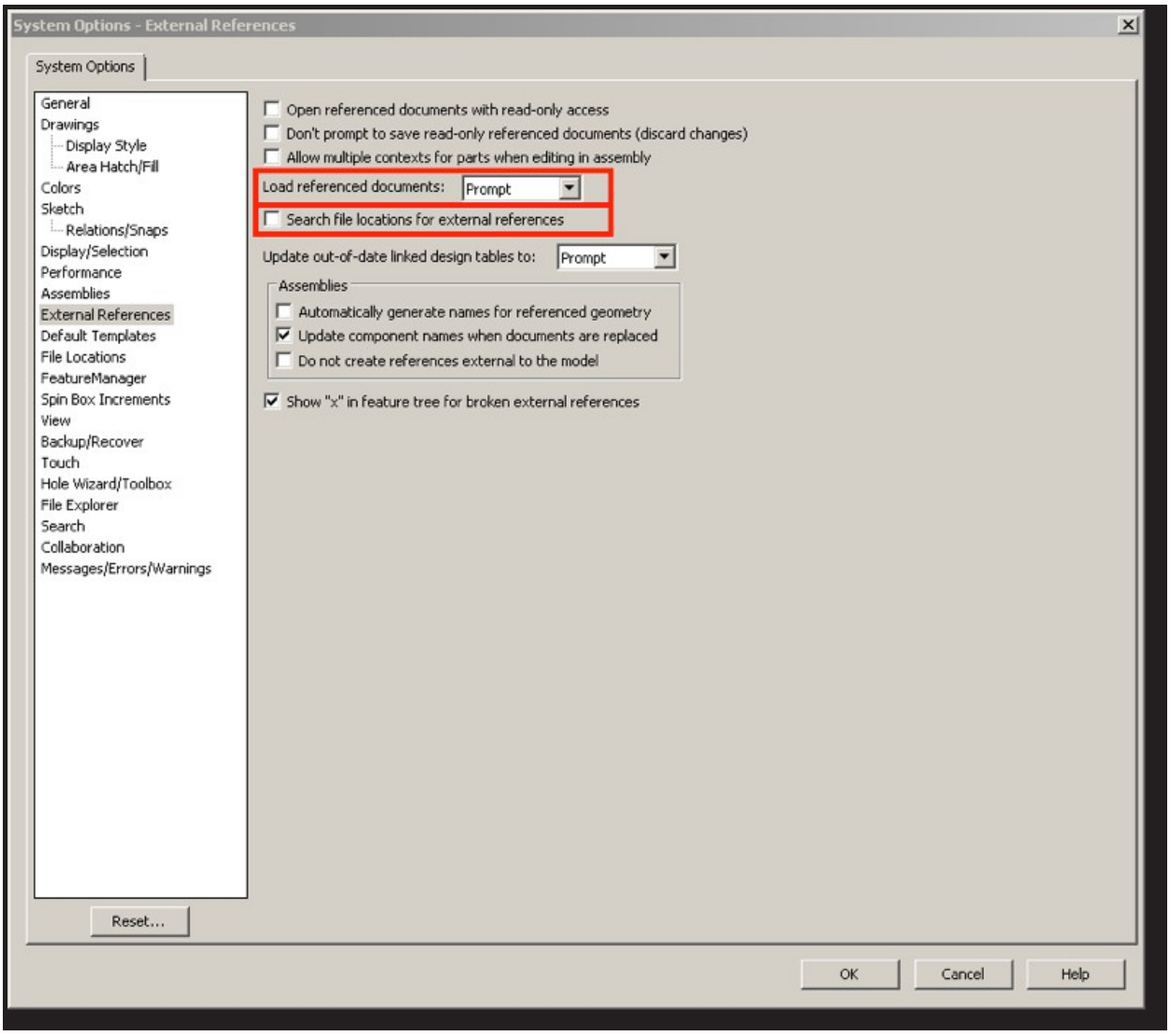
File Locations
FeatureManager
Spin Box Increments
View
Backup/Recover
Touch
Hole Wizard/Toolbox
File Explorer
Search
Collaboration
Messages/Errors/Warnings
Move components by dragging
Prompt before changing mate alignments on edit
Save new components to external files
Large assemblies
Use Large Assembly Mode to improve performance whenever working with an assembly containing more than this number of components:
500
When Large Assembly Mode is active:
Do not save auto recover info
Hide all planes, axes, sketches, curves, annotations, etc.
Do not display edges in shaded mode
Suspend automatic rebuild
Use Large Design Review whenever working with an assembly containing more than this number of components:
5000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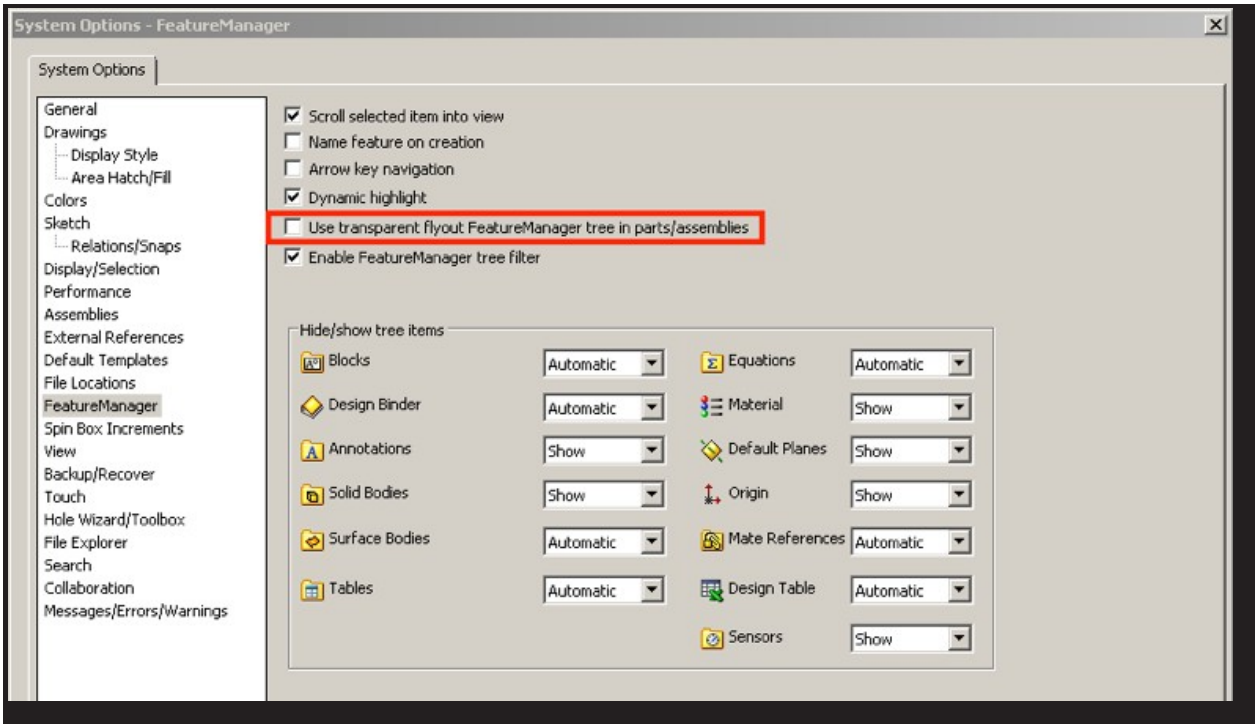
File Locations
FeatureManager
Spin Box Increments
View
Backup/Recover
Touch
Hole Wizard/Toolbox
File Explorer
Search
Collaboration
Messages/Errors/Warnings
Open referenced documents with read-only access
Don't prompt to save read-only referenced documents (discard changes)
Allow multiple contexts for parts when editing in assembly
Load referenced documents:
Prompt
Search file locations for external references
Update out-of-date linked design tables to:
Prompt
Assemblies
Automatically generate names for referenced geometry
Update component names when documents are replaced
Do not create references external to the model
Show "x" in feature tree for broken external references
Reset...
OK
Cancel
Help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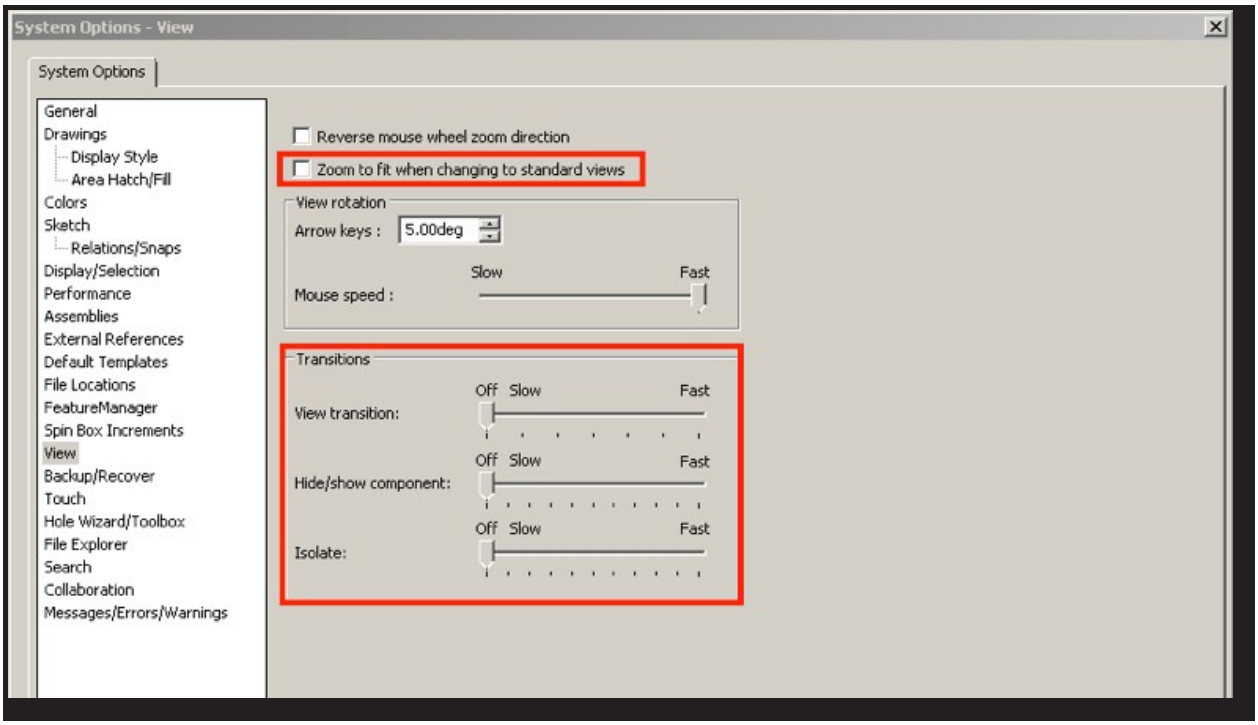
FeatureManager
Spin Box Increments
View
Backup/Recover
Touch
Hole Wizard/Toolbox
File Explorer
Search
Collaboration
Messages/Errors/Warnings
Scroll selected item into view
Name feature on creation
Arrow key navigation
Dynamic highlight
Use transparent flyout FeatureManager tree in parts/assemblies
Enable FeatureManager tree filter
Hide/show tree items
Blocks
Automatic
Equations
Automatic
Design Binder
Automatic
Material
Show
Annotations
Show
Default Planes
Show
Solid Bodies
Show
Origin
Show
Surface Bodies
Automatic
Mate References
Automatic
Tables
Automatic
Design Table
Automatic
Sensors
Show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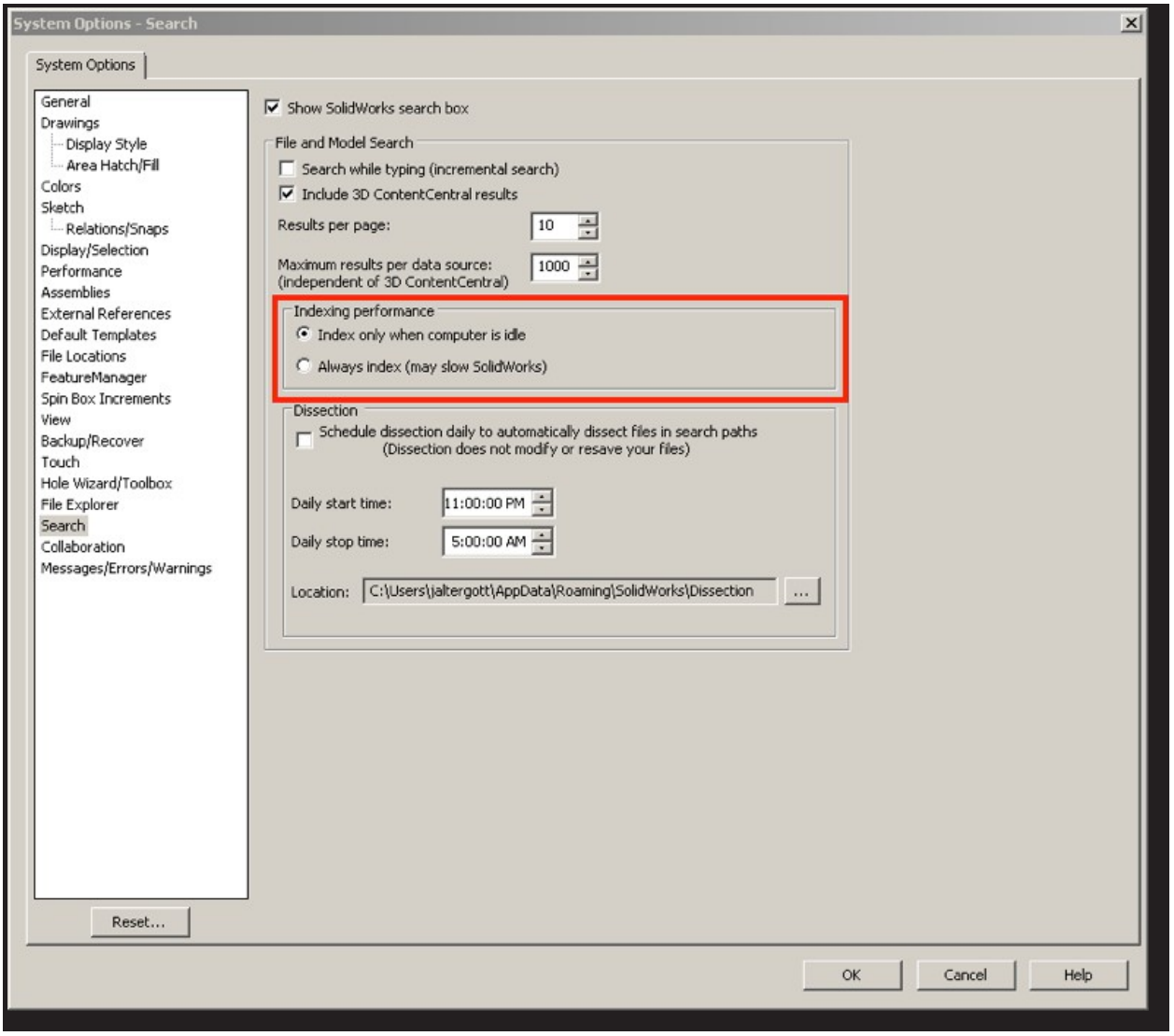
External References
Default Templates
File Locations
FeatureManager
Spin Box Increments
View
Backup/Recover
Touch
Hole Wizard/Toolbox
File Explorer
Search
Collaboration
Messages/Errors/Warnings
Reverse mouse wheel zoom direction
Zoom to fit when changing to standard views
View rotation
Arrow keys :
5.00deg
Slow
Fast
Mouse speed :
Transitions
Off Slow Fast
View transition:
Off Slow Fast
Hide/show component:
Off Slow Fast
Isolate: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File Locations
FeatureManager
Spin Box Increments
View
Backup/Recover
Touch
Hole Wizard/Toolbox
File Explorer
Search
Collaboration
Messages/Errors/Warnings
Show SolidWorks search box
File and Model Search
Search while typing (incremental search)
Include 3D ContentCentral results
Results per page:
10
Maximum results per data source:
(independent of 3D ContentCentral)
1000
Indexing performance
Index only when computer is idle
Always index (may slow SolidWorks)
Dissection
Schedule dissection daily to automatically dissect files in search paths
(Dissection does not modify or resave your files)
Daily start time:
11:00:00 PM
Daily stop time:
5:00:00 AM
Location:
C:\Users\jaltergott\AppData\Roaming\SolidWorks\Dissection
...
Reset...
OK
Cancel
Help

Appendix III

Definitions

Core: A single computational element in a multicore central processing unit (CPU).

Computing threads: A path of execution through a program.

Intel® Turbo Boost Technology 2.0: Technology that automatically allows processor cores to run faster than the base operating frequency, making it possible to increase single threaded application performance (such as CAD).

Multi-threaded programs: Programs that operate faster on CPUs that support multiple computing cores or with Intel® Xeon® processor-based workstations that support multiple CPUs with multiple cores.

Multicore processor: A CPU with multiple computing elements capable of processing multiple instructions at the same time, thereby increasing the overall speed of a given program.

Intel® Hyper-Threading Technology: Intel® HT Technology uses multicore CPU resources more efficiently, increasing processor throughput and improving overall performance on multi-threaded software (such as finite element analysis, computational fluid dynamics, ray tracing, rendering, etc.).

Single-threaded programs: Programs that have one path of execution — performing a single task at a time (such as completing one task and then moving to the next).

Swap, Virtual Memory or Page File: Space on a hard disk used to extend a computer's RAM.

Appendix IV

Additional Resources

- **Computer Aided Technology, Inc.**
  cati.com
- **Computer Aided Technology, Inc. Blog**
  blog.cati.com
- **Computer Aided Technology, Inc. YouTube Channel**
  youtube.com/user/catichannel
- **Computer Aided Technology, Inc. Subsidiary and Service Provider**
  inflow-tech.com
- **BOXX Technologies**
  boxxtech.com
- **Price a BOXX Workstation for SolidWorks**
  boxxtech.com/solutions/solidworks_solution.asp
- **BOXX Customer Case Studies**
  boxxtech.com/customers/customer_stories.asp
- **Intel Solid-State Drive 320 Series TCO Client Estimator**
  intel.com/content/www/us/en/solid-state-drives/solid-state-drives-320-series-tco-estimator-demo.html
- **SolidWorks Approved Graphics Card Drivers**
  solidworks.com/sw/support/videocardtesting.html

Appendix V

Acknowledgements

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