Optimizing Creo Performance: Getting the Most From Your System

Everyone using Pro/ENGINEER or making the transition to Creo 1.0 wants to work faster and more accurately—without breaking the bank. Here's some valuable information on overcoming some of the top performance challenges that engineers face.

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Executive Summary
Speed, system reliability, and design quality are all top of mind for users of Pro/ENGINEER and the new Creo 1.0. It’s natural that you want to get the most from your applications, and adjustments to both your software and hardware can help boost your productivity in a major way. While it’s easy to solve most performance issues with big budgets, there are several things you can do to optimize your system without incurring significant expense. In this paper, I’ll show you a variety of ways to work better, faster, and smarter—and help you determine whether or not a performance gain is worth the cost.

Performance Issue 1: Getting More Done in Less Time
Time to market is important for businesses, and that puts pressure on design engineers to work faster. Fortunately, there are many things you can do to fulfill your need for speed.

Bring Your Software Up to Speed
At the software level, a few simple setting and workflow adjustments can have a significant impact on how fast you work.

For example, check your operating system. If you’re running a 32-bit version, you won’t be able to access as much memory as you would with a 64-bit version.

This is particularly important, because PTC will no longer offer technical support for “out of memory conditions” on 32-bit hardware for Creo 1.0, in cases where the /3GB switch is utilized. If you plan to upgrade to Creo 1.0, switching to 64-bit hardware is pretty much mandatory.

Another way to free up memory for Creo or Pro/ENGINEER is to remove unneeded applications that are set to automatically start up when your OS is loaded. Check your Start Menu Folder to see which applications you can remove to free up RAM.

Of course, the most obvious software adjustment you can make is to upgrade to Creo 1.0. It’s about as big a software update as you can get, and Creo 1.0 is loaded with features designed to boost your performance dramatically.

Maximize Your Memory
At the hardware level, memory is one of the easiest and most affordable ways to accelerate Creo and Pro/ENGINEER. PTC’s minimum system requirements recommend 4 GB of RAM with a 64-bit operating system, but I recommend going with 6 GB or higher.

Without enough RAM, your system starts caching data to your hard drive, which essentially prevents you from getting anything done. You’ll know this is happening when you start to hear your hard drive spinning. Short of adding memory, the only solution to this is the time-consuming task of restarting your system.

Defrag Your Hard Drive
Fragmentation can strain your hard disk and slow you down. Use Disk Defragmenter in Windows, either manually or on a set schedule, to help your hard disk work more efficiently.
Upgrade Your Video Capabilities
For the best performance, choose from the latest video cards and drivers in PTC-certified configurations. Unlike other CAD/CAM software providers, PTC doesn’t certify or support graphic cards independently from the configurations in which they’re used. Please refer to the official PTC Platform Support page for specific hardware partners and available configurations at http://www.ptc.com/partners/hardware/current/support.htm.

For 3D hardware acceleration, you must use an OpenGL graphics card that’s been tested in a PTC-certified configuration. To ensure the compatibility of a graphics driver with Creo 1.0, a PTC-certified or supported hardware configuration is recommended. Graphics cards that support at least OpenGL 3.1 are recommended for Creo 1.0.

If you use Direct3D on Windows 7, you must install the March 2009 or later release of the DirectX 10.0 End User Run Time libraries, and use a medium-to-high-end graphics card that fully supports Direct3D 10.0. Visit the Microsoft® Web site for more information about downloading and installing Direct3D.

Make the Move to Multiple Monitors
Microsoft researchers have demonstrated that using a second monitor can improve your productivity by about 10%—and even up to 50% for specific tasks like cutting and pasting.

In the study, respondents using multi-screen configurations got solid gains after just five minutes of training:
• 6% quicker to task
• 7% faster on task
• 10% more production
• 16% faster in production
• 33% fewer errors
• 18% faster in errorless production

These results led to the conclusion that multiple monitors were cost effective where multi-screen tasks represented as little as “15% of the work for the highly competent, 17% for entry-level competence, and 21% for the general work force.”

Having CAD running on the primary display—while viewing specs, related drawings, and even email on a second display—eliminates the need to constantly switch among windows on a single monitor. You can also spread Creo or Pro/ENGINEER across both monitors if you’re working on very large drawings or complicated assemblies.

A study by Jon Peddie Research estimated that only 1.9% of new systems were equipped with multiple displays, despite a great demand for the configuration. Obstacles to obtaining additional displays included price, available desk/office space, not understanding how to do it, and additional heat, noise, and radiation.

Before you run out and get a second monitor, check to see if your system can handle multiple screens: Most workstations will, but you’re better safe than sorry.
PTC has successfully tested certain graphics card models from AMD and NVIDIA that support dual monitor capabilities. If your graphics card is certified for Creo 1.0 and provides dual monitor support, PTC expects that it will run in this mode without issue. PTC will provide limited support to resolve issues arising when running in dual monitor mode, but will not submit the configuration for formal certification at this time.

**Monitor Quick Tips**

If Dual Monitor Mode fails, use Span Mode as a workaround.

If you’re on a limited budget, get one big monitor instead of two smaller ones to increase productivity.

**Performance Issue 2: Preventing System Crashes**

In addition to speed, system reliability is another major factor that impacts Creo and Pro/ENGINEER performance.

**Use PTC-Certified Configurations**

One of the simplest things you can do to make your system more reliable is to only install drivers that are part of PTC-certified configurations. You can obtain a list at http://www.ptc.com/partners/hardware/current/support.htm.

**Avoid System Lockups and BSOD with ECC Memory**

Memory errors are the leading cause of system crashes, and have also been shown to make systems more vulnerable to security breaches.

What happens is known as a “bit flip,” which occurs about once a week on typical systems with 4 GB of memory. Radiation within the system causes a memory bit to suddenly change state from 0 to 1 or 1 to 0. If this occurs in an application area, there can be an error in a calculation or the application may crash. If the error occurs in the OS, your whole system could crash or present you with the Blue Screen of Death (BSOD).

Workstations with Error Correction Code (ECC) memory provide a much more stable computing environment, resulting in fewer computer crashes and better data integrity. Several studies confirm that ECC memory detects and fixes single bit errors and reduces the likelihood of a system crash by 25 times.

A recent Google study showed that 32.2% of all its machines experienced an average of 277 errors per year that were correctable with ECC memory. Just 1.3% of the machines had errors that ECC memory couldn’t correct.

**Performance Issue 3: Ensuring Defect-Free Design**

Another category of performance issues, which is often overlooked, centers around getting designs ready for production without mistakes. Many designers worry that an unnoticed error could cost a lot to fix, once in production.
Step Up Your Screen Size
Larger monitors can help you design with fewer errors. In addition to easing eye strain, larger monitors make it easier to see design flaws and other problems.

Because of permanent menus and other displayed application items, using a smaller monitor limits your usable modeling area. For about $100 more than a 19-inch model, a 22-inch monitor gives you 31% more work area and a 24-inch monitor can increase your usable workspace by up to 57%. The more you can see the more likely you’ll find mistakes—and the less nervous you’ll be about making them in the first place.

Conclusion
As you can see, there are many variables involved in getting peak performance from Creo and Pro/ENGINEER. If you break things down into the three larger categories of speed, reliability, and design quality, you’ll have a better sense of what you really need to fix. Start small, and realize that some simple, inexpensive adjustments may be all you need to get your desired performance. If you decide that you really do have to spend a lot more to get the performance you’re after, know that the upfront expense may indeed be justified by the productivity gains you achieve.