

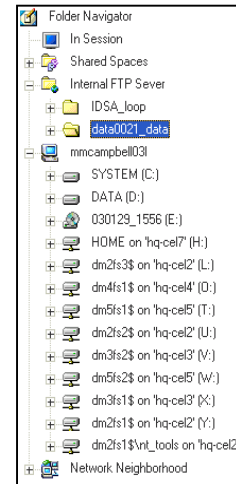
Technology Highlight - Pro/ENGINEER Wildfire Performance Improvements

From model regeneration to check-in and check-out, Pro/ENGINEER Wildfire is faster than ever before.

Pro/ENGINEER Wildfire is full of significant advances to increase your productivity. The new user interface (UI) makes learning and working with the system easier than ever before, setting a new standard in CAD usability. Furthermore, the built-in connectivity will help improve how you communicate with others in the supply chain, regardless of your particular role. Another important consideration of productivity is with respect to system performance. System response time to commands, the number and duration of screen repaints, and the time it takes to regenerate models after changes can all add up to significant time wasted. Fortunately, Pro/ENGINEER Wildfire has improvements in all of these areas, and many others, all intended to help you get your job done faster, and with better results.

Many of the UI and connectivity improvements to Pro/E Wildfire translate directly into productivity gains for the end user. This means less time wasted, and more time spent getting the job done. For example, since sending and retrieving .zip files is an easy way to informally share Pro/ENGINEER data, consider the simple task of downloading a .zip file containing a Pro/E assembly from an FTP site. In Release 2001, this would have required using your web browser to find the file, downloading it to a temporary directory on your local machine, extracting the files from the .zip file into the temporary directory, and then navigating to that temporary directory in Pro/ENGINEER to open the files. In Pro/ENGINEER Wildfire, you can simply add the FTP site to the “Server Registry”, and then navigate to it using the Folder Navigator. By simply double-clicking on the zip file, Pro/ENGINEER Wildfire will automatically download it to the local machine, and open it, showing you the contents in the File-Open Preview tool.

Pro/ENGINEER Wildfire is packed with tools like this, to make getting your job done easier and faster.



Intelligent Regeneration

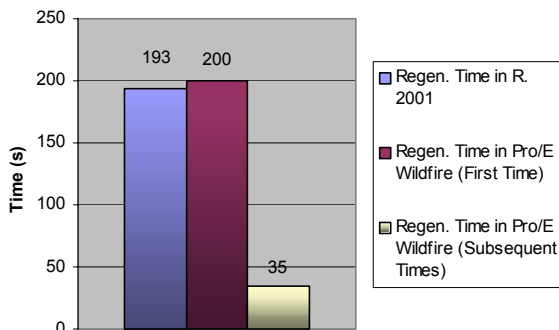
In previous releases of Pro/ENGINEER, after you made a modification to a feature in the model's history, that feature and all subsequent features were regenerated. For example, if a modification was made to the cut (Feature 18) in the model whose Model Tree is shown on the right, Features 18 through 41 would all be regenerated during the next update. If this model were in an assembly, and subsequent models depended on the geometry of any of those features, those models' placement and features may also be regenerated. This was a simple way to ensure that all changes due to the modification of the cut were propagated throughout the model, and the assembly. However, this is also often an inefficient way to update the model, since it's possible that those subsequent features are in fact not affected by the changed geometry.

In Pro/ENGINEER Wildfire, PTC has significantly improved the intelligence of the regeneration algorithm. That means that Pro/ENGINEER now analyzes the model, and instead of simply regenerating every subsequent feature, it regenerates only those that are actually affected by the modification. Again, using the example shown to the right, if only some subsequent features would actually be affected by a change to Feature 18, you can imagine making a change, and Pro/ENGINEER only needing to update four or five features, not all 23. The impact of this intelligent regeneration scales significantly when you consider models in the context of an assembly.

There are a few points of consideration. First, the intelligent regeneration algorithm cannot be applied to every model. Some models contain features that make application of this algorithm impossible in Pro/ENGINEER Wildfire. Application to those cases is under development for the release following Wildfire. Secondly, there will be a small regeneration performance degradation the initial time a model is regenerated in Pro/ENGINEER Wildfire. This is due to the fact that the model's geometry must be analyzed, to understand where optimization in the sequence can be applied. This regeneration slow down only happens the first time the model is regenerated in Pro/ENGINEER Wildfire, and takes, on average, less than 5% longer than the regeneration would have taken in Pro/ENGINEER Release 2001. The optimization data is then

stored as part of the model, and used in all subsequent regenerations.

Of course, the impact of this new behavior in Pro/E Wildfire will vary significantly depending on the modeling methodology employed when creating your models.



On average, tests on customer assemblies of medium to large complexity have consistently shown an average of 15-20% reduction in regeneration times. In some cases, due to feature dependencies, or the inclusion of features that prohibit use of the new algorithm, the improvements may be negligible. In other cases, the improvements may be much more significant. For example, one particular model took three minutes and thirteen seconds to update in Release 2001, and three minutes and twenty

BATTERY_RETAINER_	
Y_DIRECTION	1
Z_DIRECTION	2
X_DIRECTION	3
CS0	4
Z_AXIS	5
Y_AXIS	6
X_AXIS	7
ORIGIN	8
ALIAS	9
Cosmetic id 37	10
Protrusion id 72	11
Round id 6392	12
Cut id 6841	13
Draft id 6874	14
Shell id 172	15
Curve id 231	16
Protrusion id 245	17
Cut id 250	18
Protrusion id 253	19
Cut id 257	20
Cut id 260	21
Draft id 261	22
Draft id 262	23
Round id 263	24
Draft id 264	25
Cut id 13541	26
Cut id 6960	27
Protrusion id 7030	28
Protrusion id 7073	29
Protrusion id 7130	30
DTM11	31
Protrusion id 7474	32
Draft id 7602	33
Cut id 7519	34
Draft id 7648	35
Draft id 7716	36
Draft id 7746	37
Protrusion id 8248	38
Draft id 8315	39
Protrusion id 8440	40
Draft id 8484	41

seconds during the first regeneration in Pro/E Wildfire. However, the next regeneration took only 35 seconds, resulting in an 82% time savings.

Model Retrieval and PDM Interaction

In addition to faster regeneration of models, there are improvements in Pro/ENGINEER Wildfire to make retrieving various types of data into session faster. For example, models with external references to other models created via relations are now retrieved up to 25% faster. Also, you will see notable decreases in the time required to retrieve models which contain Inheritance Features (which were introduced in Release 2001). In Release 2001, the Inheritance Feature was regenerated at the time of retrieval, and in the case of a complex source model, this introduced a considerable performance cost. However, in Pro/ENGINEER Wildfire, all of the details of the source model are stored within the referencing models, so no regeneration at the time of retrieval is needed. One can see notable performance improvements in cases like that.

An array of Pro/ENGINEER Wildfire features both on the surface and in the background work in concert to maximize the productivity of users exchanging files with Windchill, Windchill PDMLink, and Windchill ProjectLink. Many of the same data management features can be accessed from both the Pro/ENGINEER native user interface as well as through the embedded web browser. The Pro/ENGINEER native user interface streamlines typical data management operations like check-in and check-out to a minimum of mouse picks by consolidating multiple operations and using default configuration settings. “File | Check-in – auto” and the right-mouse button “Checkout” from the model tree are such examples. The Pro/ENGINEER Wildfire embedded browser provides the PDM functionality with more options and flexibility.

Pro/ENGINEER Wildfire improves upload, check-in, download, and check-out times with caching, multi-threading, and meta-data compression. Pro/E Wildfire caches every Pro/ENGINEER file that it receives, including those received for File Preview during “File | Open” with previewing operations. When a user later chooses to download or check-out, only the files not in the Pro/ENGINEER local cache need to be downloaded from the PDM system. The local cache of Pro/E Wildfire reduces the number of files that need to be downloaded, enabling designers to work in Pro/ENGINEER that much sooner. Download and check-out operations may be configured as multi-threaded operations improving the usage of server, client, and network resources to accelerate delivery of data to the desktop. Metadata compression for upload, check-in, download, and check-out operations is particularly effective in improving WAN performance with family tables and large assemblies.

Pro/ENGINEER Wildfire minimizes the number of client server interactions to provide no more data than is necessary to enable a user to focus on designing in Pro/ENGINEER right away. When family table objects are downloaded, only the required instance information is delivered, rather than all the information about all the instances contained in the family table generic model. The “check-out on-the-fly” capability allows user to first download models without having to check them out up front. When the user attempts to modify a part that is not checked out, Pro/ENGINEER Wildfire provides the option to checkout the part on-the-fly right in the Pro/ENGINEER session. In addition, Pro/ENGINEER Wildfire enables users to continue working in Pro/ENGINEER Wildfire as some data management operations are done in the background.

These improvements, combined with the advantage of integrated searching capabilities, now part of Pro/ENGINEER Wildfire, make finding, retrieving, and exchanging Pro/ENGINEER files with Windchill servers quick and easy. And, with multi-server support, you can share Pro/ENGINEER objects throughout and beyond the enterprise of the PDM database.

External Simplified Representations

Pro/ENGINEER Wildfire also delivers a new type of Simplified Representation, called an External Simplified Rep, which can be created without modifying the master assembly. An external simplified representation is a presentation of a master assembly stored as a separate assembly file with a dependency on the original assembly. It contains particular components of the master assembly or its simplified representations. You can create multiple external simplified representations of a master assembly, each corresponding to different areas of the assembly and levels of detail. You can include low-level components without top and intermediate level assemblies and allow multiple users to work simultaneously. External simplified representations avoid the risk of accidental modifications to top-level assemblies.

All items included in an external simplified representation are the same as those in the master assembly. Therefore, it is not necessary to propagate modifications made to the external simplified representation or master assembly. All modifications to external simplified representations are automatically reflected in the master assembly.

External Simplified Reps dramatically reduce the system requirements for high-quality visualization and deliver quicker regeneration times. Furthermore, the external simplified representations are stored in Windchill as separate documents. They can be retrieved much faster in Pro/ENGINEER Wildfire compared to the simplified representations from the design file itself, since the knowledge about dependencies stored in Windchill can be leveraged to optimize downloading.

New Pro/ENGINEER Graphics Library (PGL)

In addition to the performance improvements that will affect regeneration and data management, the introduction of a new set of software components called PGL, will have significant impact on graphics performance. For example, Pro/ENGINEER users have become accustomed to the model flashing (and the screen repainting) when spinning, panning and zooming in and out on the model. The new graphics capabilities completely eliminate this flashing, resulting in a faster, smoother experience while manipulating the model view. There are also new capabilities due to the improved graphical elements of the software, including real-time updating of renderings and shadows while modeling, and more “realistic” display of transparencies.

The introduction of PGL will also impact performance while working with drawings. Because of the way the graphics information is now being managed, the size of drawings that contain true-type fonts will be reduced. One test of a drawing with true-type fonts showed a reduction in file size by over 95%, from 30 Mb to less than ½ Mb. The time to save the file was reduced proportionally, from 10 minutes to 40 seconds.

Summary

During a recent, holistic evaluation of Pro/ENGINEER Wildfire versus Pro/ENGINEER Release 2001, we compared the time it takes to perform typical product development and design activities. These activities began with PDM system interaction, including browsing, searching, and model check-out. Additional parts were then modeled, using a variety of features including protrusions, cuts, rounds and drafts, and assembled together. The assembly also included parts from a catalog of standard components, which were searched for, and then assembled. Finally, the design was improved upon by multiple, geographically-disparate parties, all working together in real-time.

The new user interface, and connectivity of Pro/ENGINEER Wildfire allowed for these activities to be completed 40% faster than in Pro/ENGINEER Release 2001.

Pro/ENGINEER Wildfire is the most significant Pro/ENGINEER release in PTC's history, delivering unprecedented ease-of-use, powerful capabilities, and revolutionary connectivity. These advances, including the improvements in system performance across graphics display, data exchange, model retrieval and regeneration, have all been developed with one goal in mind – to help our customers win with superior products.