READ THIS FIRST
Pro/ENGINEER® Wildfire® 3.0

Important Installation and Usage Information

Directory of Online and CD-ROM Information

Platform Support	http://www.ptc.com/partners/hardware/current/proewf3.pdf
Customer Support	http://www.ptc.com/support/index.htm
Pro/ENGINEER Resource Center	http://www.ptc.com/community/proewf3/newtools/index.htm
Knowledge Base	http://www.ptc.com/appserver/cs/search/search.jsp
Reference Documents	http://www.ptc.com/appserver/cs/doc/refdoc.jsp

PTC Introduces
Pro/ENGINEER Wildfire 3.0

PTC is pleased to present Pro/ENGINEER Wildfire 3.0. With a full range of associative CAD/CAM/CAE applications and Web services, Pro/ENGINEER ensures the seamless flow of digital product information to cross-functional teams as well as to customers, partners, and suppliers, whether or not they are Pro/ENGINEER users. Simple - Powerful - Connected, Pro/ENGINEER Wildfire 3.0 improves both personal and process productivity, helping you to work faster and work smarter. A few highlights of the enhancements in this new release follow:

Personal Productivity Enhancements

- **Faster Assembly**—Optimized workflow for building assemblies faster and for superior performance with the added support for Windows XP x64.
- **Faster Drawings**—Intelligent, automated features to clean up drawing dimensions and automated scaling of drawing views to expedite drawing creation.
- **Faster Sketches**—Optimized workflow for sketching to accelerate feature creation.
- **Faster Sheetmetal**—Intelligent geometry to create relief cuts and other geometry required in sheetmetal design to greatly reduce the number of features.
- **Faster CAM**—More intuitive workflow with improved tool-searching capabilities to find the right tool faster.

Process Productivity Enhancements

- **Smarter Models**—Leveraging of more model information for downstream deliverables. Automated tasks such as creating definable rules ensure that assembled components meet design intent and engineering criteria.
- **Smarter Sharing**—Protection of intellectual property and easy tracking of changes. Portable workspaces allowing sharing offline with third parties who do not have access to your installation of Pro/INTRALINK or Windchill.
- **Smarter Process Wizard**—A customizable wizard to capture expert knowledge of the analysis process and guide you through it.
- **Smarter Interoperability**—Improved interoperability between Pro/ENGINEER and Windchill and Pro/INTRALINK for seamless access to information when you need it.

Installation

PTC recommends a new installation of PTC License Server and Pro/ENGINEER Wildfire 3.0 and not an upgrade. Use the product CD-ROM or the new Web installer on the **Order or Download Software Updates** page at http://www.ptc.com/appserver/cs/software_update/swupdate.jsp.

A new license agreement in PTC.Setup requires your acceptance of the license agreement before proceeding with the installation. Declining the license agreement stops the installation and provides additional instructions.

Note: If your company and PTC have signed a License Agreement previously (other than a Customer Agreement for Windchill Products), this previous agreement shall govern your licensing and use of the licensed products. The terms and conditions of the new agreement in PTC.Setup shall not apply.

Installation from the Web requires an active maintenance contract. Via the Web installation, you can create custom media containing only selected components, including languages and operating systems. For information, see the *Pro/ENGINEER Wildfire 3.0 Installation and Administration Guide* and the following TPI (Technical Point of Interest): http://www.ptc.com/appserver/cs/view/solution.jsp?n=124813.

To complete installation, see the installation documentation for your product. The *Pro/ENGINEER Wildfire 3.0 Installation and Administration Guide* is available from PTC.Setup by clicking **Installation Guide** and from http://www.ptc.com/appserver/cs/doc/refdoc.jsp.

To access PTC documentation from the Web, you must have, or register for, a PTC account at http://www.ptc.com/appserver/common/account/basic.jsp. Adobe Acrobat Reader is required to read PDF files. Download Acrobat Reader from http://www.adobe.com/support/downloads/main.html.

After you complete the installation, the Read This First is available on your local drive `<proe_loadpoint>/html/$LANG/proe/helpSysTop/siteset/readme.pdf`. It also is under **Quick Links** in the Help Center.

Licensing and the Updated PTC License Server

Pro/ENGINEER Wildfire 3.0 introduces an updated PTC License Server powered by Macrovision FLEXnet Publisher 10.8, a newer version of FLEXnet. Therefore, you must install the new PTC License Server from the CD-ROM and update your license file for Pro/ENGINEER Wildfire 3.0 (version 27).

You must perform license simplification before you can update your license file. If you have a Pro/ENGINEER Wildfire 2.0 or a Pro/ENGINEER Wildfire 3.0 Preproduction 2 license file, you already have performed license simplification. Upon request, you will receive a new license pack by electronic mail. For more information on simplifying licenses or obtaining an updated license pack, visit **Order and License Support** at http://www.ptc.com/olm/index.htm.

When requesting a license online, respond to the prompt about the Pro/ENGINEER release as follows:

- `Yes`—For Pro/ENGINEER Wildfire 3.0 (version 27.0)
- `No`—For Pro/ENGINEER Wildfire 2.0 (version 25.0)

Vericut and CGTech Licensing

CGTech has removed PTC licensing as of release 5.4 from the advanced VERICUT products: NC Verification, Optimization, and Machine Simulation. If you use any of these products, you must move to CGTech licensing to use it with Pro/ENGINEER Wildfire 3.0 or later.

Operating System Support

Operating System	Version
Hewlett-Packard HP-UX	HP-UX 11.0 (64-bit only)[a] and 11iV1 (64-bit only) with the PA8000 or later processor
Linux	Red Hat Linux Enterprise 3.2 WS Edition[b]
Microsoft Windows 2000	Base OS, Service Pack 4. Internet Explorer 6.0 SP1 or later
Microsoft Windows XP	Windows XP Professional Edition and Windows XP Home Edition, Base OS and Service Packs 1 and 2, Internet Explorer 6.0 SP1 or later

Operating System	Version
Microsoft Windows XP x64	Windows XP Professional x64 Edition, Base OS, Internet Explorer 6.0 SP1 or later
	Note: Initial support will be in the following languages; English, French, German, Italian, Spanish, Japanese and Korean.
SGI IRIX	Not supported
Sun Solaris	Solaris 8, 9 and 10 (64-bit only) with the Ultra SPARC II or later processor
	Solaris 10 with the AMD Opteron processor
Notes	
[a]PTC's HP-UX 11.0 support will end in 2006 following HP's plans: http://www.hp.com/softwarereleases/releases-media2/discon/0303.htm	
[b]The Mechanica module (Structural and Thermal Simulation) for Wildfire 3.0 is not supported on Linux.	

System Requirements and Hardware Certification Information

Visit the **Platform Support** page for Pro/ENGINEER Wildfire 3.0 for current, detailed system requirements and certified hardware configurations:

http://www.ptc.com/partners/hardware/current/proewf3.pdf

Asian and European Languages

The user interface, online Help, books, and Web tools for Pro/ENGINEER Wildfire 3.0 are translated in eight languages: French, German, Italian, Spanish, Japanese, Korean, Simplified and Traditional Chinese. For Linux, in addition to German, this release of Pro/ENGINEER is available in French, Italian and Spanish. Pro/ENGINEER in Asian languages is not supported on Linux.

Interoperability

PTC Product	Interoperability with Pro/ENGINEER Wildfire 3.0
Distributed Services Manager	Distributed Services Manager (DSM) 3.0. Use a Wildfire 3.0 dBATCH client that in turn must pass the task to a Wildfire 3.0 dBATCH service to submit tasks to DSM 3.0.
Expert Moldbase and Progressive Die Extensions	PDX 2.2, PDX 5.0
	EMX 4.1, EMX 5.0
Expert Framework Extension	EFX 4.0, EFX 5.0
Granite	Applications based on Granite are upward compatible with Pro/ENGINEER. Pro/ENGINEER Wildfire 3.0 can retrieve geometry and other data created in any Granite release including Granite 4.0 datecode F000 and later. In contrast, only Granite 4.0 can retrieve Pro/ENGINEER parts and assemblies created in Pro/ENGINEER Wildfire 3.0. Pro/ENGINEER requires a gPlug to retrieve application data if the application does not write Granite .g files.
Pro/COLLABORATE	Not interoperable
Pro/CONCEPT	Pro/CONCEPT 4.0
Pro/INTRALINK 3.x	Pro/INTRALINK 3.4 M030 and later
Pro/INTRALINK 8.0	Compatible with 8.0 M020 and later
ProductView	ProductView compatibility is provided via the ProductView Adapter for Pro/ENGINEER Version 11.0 M020. With Wildfire 3.0, you can publish

PTC Product	Interoperability with Pro/ENGINEER Wildfire 3.0
	drawings to PDF and DWG. The Adapter supports the publication of Wildfire 3.0 shaded drawings for viewing in ProductView 7.0 and 8.0.
ProductView Express (PVX) (Standalone)	ProductView Express is compatible with Pro/ENGINEER Wildfire 3.0. Download it free from the **ProductView Express** page: http://www.ptc.com/products/pvexpress/index.htm.
Routed Systems Designer	Routed Systems Designer 6.0 datecode F000
Windchill Foundation and PDM	Compatible with 8.0 M020 and later, and 7.0 M070 and later
Windchill PDMLink	Compatible with 8.0 M020 and later, and 7.0 M070 and later
Windchill ProjectLink	Compatible with 8.0 M020 and later, and 7.0 M070 and later

OpenGL Library

Pro/ENGINEER Wildfire 3.0 supports the remote display of OpenGL across the same UNIX platforms (HP to HP, for example). Run both UNIX platforms on the same version of OpenGL libraries or a message on performance degradation appears in the startup window.

For best performance, do not run OpenGL remotely across wide-area networks or with preselection highlighting enabled within Pro/ENGINEER.

Note: For Windows, a new configuration option in Pro/ENGINEER provides an alternative to WIN32_GDI to address graphics-related issues. This option toggles Microsoft Software OpenGL on the local machine: use_software_opengl = yes/no.

Graphics Library

The Graphics Library has been updated for Wildfire 3.0 with information for the new Scene files. Sample Room files are revised, and the Light and Environment files are updated to a new format. It is recommended that you download and update this library for photorealistic rendering in Wildfire 3.0 from the Pro/ENGINEER Library download site:

http://www.ptc.com/cgi/cs/apps/SftUpd/SftUpdProd.pl

(Choose **PRO/ENGINEER** as the product name and look for **Wildfire Library Databases** in the list.)

Redefining Legacy Datum Planes

When redefining and changing a datum plane created before Pro/ENGINEER Wildfire 2.0, any radius dimensions controlling the outline of the datum plane are deleted because of the redefinition. The actual datum plane outline is automatically converted to the Pro/ENGINEER Wildfire 3.0 standard. The outline then remains the same from a user's perspective. However, if the deleted radius dimension was used to drive a relation, the relation fails after the redefinition. To restore the relation, you must use either a parameter or another related dimension to drive the relation.

Pro/ENGINEER Help Center—Help and Books

Help and books for Pro/ENGINEER Wildfire 3.0 are on the Pro/ENGINEER CD-ROM. You can access the Help Center by clicking **Help > Help Center** or through context sensitivity. The following items are in the Help Center:

- Context-sensitive Help for Pro/ENGINEER and ProENGINEER Mechanica (Structural and Thermal Simulation)
- *Pro/ENGINEER Wildfire 3.0 Installation and Administration Guide*
- *PTC Customer Service Guide* (November 2005)
- Pro/ENGINEER Wildfire 3.0 Resource Center

For best results, view the Pro/ENGINEER Help Center in the Pro/ENGINEER browser. To use another browser, use Mozilla 1.7.8 or Internet Explorer 6.0 SP1 or later. Refer to the Pro/ENGINEER Wildfire 3.0 Installation and Administration Guide to set up and use the Help Center. If you are a Mechanica (Structural and Thermal Simulation) independent mode user or a Simulation Advisor user, you can view Help using Netscape 6.2 or later, Mozilla 1.7.8, or Internet Explorer 6.0 SP1 or later.

Pro/ENGINEER Resource Center–Web Tools and Tutorials

Web-based usability tools can assist you with learning to use this release. The Pro/ENGINEER Wildfire 3.0 Resource Center is accessible from the Pro/ENGINEER browser, through the Pro/ENGINEER Help Center (**Help > Help Center**), or on the Internet at:

http://www.ptc.com/community/proewf3/newtools/index.htm

The tools for Pro/ENGINEER Wildfire in the Resource Centers are as follows:

- **Interactive Menu Mapper**—Menu paths from Pro/ENGINEER 2000i^2, 2001, or Wildfire to Pro/ENGINEER Wildfire 3.0. A new addition includes menu paths from Pro/INTRALINK 3.4 to PDMLink 8.0.
- **What's New—**Documentation of new functionality in Web or PDF format.
- **Tutorials**—Step-by-step instructions on helpful topics.
- **Productivity Tools**—An interactive guide to tools: selection tools, viewing tools (Spin, Pan, and Zoom), and the Model Tree.

Product-Specific Customer Information

AutobuildZ Customers

AutobuildZ, a former plug-in application, is integrated into Pro/ENGINEER Wildfire 3.0. AutobuildZ creates 3D, parametric, feature-based models from 2D drawing data from input in wizard-style dialog boxes. Do not install the downloadable plug, which is only for the first release of Pro/ENGINEER Wildfire. The plug-in will cause AutobuildZ to fail to start correctly.

Note: Before starting Pro/ENGINEER Wildfire 3.0, you must set the `autobuildz_enabled` configuration option to `yes` in the `config.pro` configuration file in the working directory.

Distributed Pro/BATCH Customers

Distributed Pro/BATCH now can register a Windchill PDMLink 7.0, PDMLink 8.0, or Pro/INTRALINK 8.0 installation and Workspace (similar to that provided for Pro/ENGINEER). It can perform Distributed Pro/BATCH operations using data from that Workspace. See the online documentation for details.

Distributed Services Manager (DSM) Customers

Distributed Services Manager (DSM) 3.0 includes improved session recovery. See the online documentation for details.

Mold Design Customers

To use automatic workpiece functions with TDO (Tool Design Option), you must click the **Mold Component Catalog** option when installing the software.

The Mold Design and NC Menu Managers have been modified for the creation of mold and NC geometry to take advantage of the standard Pro/ENGINEER features. Check the Pro/ENGINEER Wildfire 3.0 Menu Mapper for help.

Interface for CATIA V5

The rebuilt Interface for CATIA V5 in Pro/ENGINEER Wildfire 3.0 supports both import and export of CATIA V5 CATPart and CATProduct files. Versions supported include CATIA V5R10 to V5R15. This interface uses a new license: INTERFACE for CATIA V5 ADV. If you have an existing license, it will automatically be upgraded when you convert your license packs to the new Wildfire 3.0 format.

Interface for PDF (version 2)

PTC introduces a second generation Interface for PDF in Pro/ENGINEER Wildfire 3.0. This powerful processor is included with the Foundation package and no longer requires access to Adobe Distiller. This release supports advanced capabilities for publishing Pro/ENGINEER drawings as PDF documents:

- **Format tab**—Controls the publishing of searchable TrueType fonts or stroked text, whether drawings are color, grayscale, or monochrome, and controls the resolution for shaded views.
- **Content tab**—Controls hyperlinks, support of layers, inclusion of parameters as metadata, and use of bookmarks, such as zones, views, sheets, and flag notes.
- **Security tab**—Controls the password protection of files and can restrict printing, editing, adding or deleting pages, commenting, copying, or adding digital signatures.
- **Description tab**—Provides for inclusion of descriptive information in the PDF document, such as title, author, subject, and search keywords.

ModelCHECK Customers

A new ModelCHECK Metrics tool and additional checks have been implemented for this release. You can apply the new checks listed below via the check configuration file (`.mch`):

- `AE_GTOL_DUPLICATE`
- `AE_SURFACE_DUPLICATE`
- `AF_INCOMPLETE`
- `ANNTN_INACTIVE`

The ModelCHECK Metrics tool is a Web-based utility for monitoring data quality, standards, and best practices. The installation is separate from Pro/ENGINEER and uses the metric file output from ModelCHECK to calculate quality trends. For installation information, see the *Pro/ENGINEER Wildfire 3.0 Installation and Administration Guide* in PTC.Setup.

Pro/PLASTIC Advisor Customers

Pro/PLASTIC Advisor for Pro/ENGINEER Wildfire 3.0 incorporates Moldflow Part Advisor Release 7.0.

Pro/Web.Link Customers

Pro/Web.Link installs automatically with Pro/ENGINEER Wildfire 3.0. Only the browser version of Pro/Web.Link is supported. Pro/Web.Link now runs in Pro/ENGINEER browsers on both Windows (in Microsoft Internet Explorer 6.0 SP1 or later) and UNIX (in Mozilla 1.4). Support for the Netscape version of Pro/Web.Link has been discontinued.

Web pages for the Netscape browser are not directly usable by the Pro/ENGINEER browser. Pro/Web.Link documentation at `<proe_loadpoint>/weblink/weblinkdoc/index.html` provides an API Wizard. The section Application Conversion in the Pro/Web.Link documentation explains how to convert Netscape-based Pro/Web.Link applications to work in the Pro/ENGINEER browser.

ProductView Express Customers

ProductView Express provides for the preview of Pro/ENGINEER models within the Pro/ENGINEER browser (embedded) and in browsers running outside of Pro/ENGINEER (standalone).

On Windows, ProductView Express is installed or updated automatically the first time you select a model from the **Folder** browser within the Pro/ENGINEER browser. You can install the Production release of ProductView Express for Windows using the command `<proe_loadpoint>\<machine_type>\obj\pvx_install.exe`. This installation provides both the embedded and standalone browsers for PVX.

On UNIX, you install and configure ProductView Express while installing Pro/ENGINEER. You can download for free the standalone version of ProductView Express, compatible with Pro/ENGINEER Wildfire 3.0, from the **ProductView Express** page: http://www.ptc.com/products/pvexpress/index.htm.

Note: ProductView Express Wildfire 2.0 standalone is not compatible with Pro/ENGINEER Wildfire 3.0.

Pro/CONCEPT Customers

- Pro/ENGINEER parts and assembly files can be loaded directly in Pro/CONCEPT (Windows version only). Other file formats that can be read into Pro/CONCEPT (both Windows and Mac OS X) are IGES (*.igs), STL (*.stl), Wavefront (*.obj) and ProductView (*.ed, *.ol).

- To transfer Pro/ENGINEER curves to Pro/CONCEPT, save the curves in the Neutral File format (*.neu).

- To transfer images between Pro/CONCEPT and Pro/ENGINEER, use the supported image file formats.

- To load a layout from Pro/CONCEPT, use the **Trace Sketch** dialog box in the Pro/ENGINEER Style feature and click **File > Open Scene**. This command loads the trace sketches and the 3D curves from a Pro/CONCEPT (*.cpt, *.cpz) file. The Style feature is part of the Freeform Surfacing (ISDX) module.

- To load the curves or model geometry from Pro/CONCEPT, click **Import > Shared Data > From File.** Browse to the neutral curve file (*.neu) or the STL geometry file (*.stl). You can use Pro/CONCEPT geometry in Pro/ENGINEER in a number of ways. The Reverse Engineering (REX) module provides tools for wrapping surfaces onto the facet data.