

PTC® READ THIS FIRST

Creo® 2.0:

Creo Parametric, Creo Direct, Creo Layout, and Creo Simulate

Important Installation and Usage Information

Directory of Online and DVD-ROM Information

Platform Support	http://www.ptc.com/partners/hardware/current/support.htm
Technical Support	http://www.ptc.com/support/index.htm
Creo Resource Center	http://www.ptc.com/community/creo2/index.htm
Knowledge Base	http://www.ptc.com/appserver/cs/search/search.jsp
Reference Documents	http://www.ptc.com/appserver/cs/doc/refdoc.jsp

Creo is a scalable suite of interoperable, right-sized design applications for users across the enterprise to more easily participate in and contribute to the product design process. Creo Parametric, Creo Direct, Creo Layout, and Creo Simulate are four of the applications available as of Creo 2.0. With these applications, companies can bring better products to market faster by improving processes such as concept design, detailed design, and verification and validation.

Installation

Creo 2.0 uses a PTC License Server powered by FLEXnet Publisher 11.10 from Flexera Software Inc. If you are already running a license server for Pro/ENGINEER® Wildfire® 3.0 or later, Creo Elements/Pro™ 5.0, Mathcad® 14.0, Arbortext® Isodraw® 7.0, or Arbortext Editor™ 6.0, you need only update your license file to support Creo 2.0. If you are a new user, or if your license server has never supported any of these products, you must install the new PTC License Server when using floating licenses.

Using PTC Installation Assistant

For a streamlined installation of Creo 2.0 applications, you can use the PTC Installation Assistant on your DVD-ROM. Open the installer, enter your Sales Order Number, and the Assistant begin the automatic installation:

- Acquires the license file from PTC
- Determines whether the license is floating or fixed and installs the licenses server as required
- Provides a list of the available products based on the entitlement detailed in the licenses file and automatically downloads and installs that software from PTC.com

For the most up-to-date installation information, see *Installing Creo 2.0: Creo Parametric, Creo Direct, Creo Layout, and Creo Simulate* on the **Reference Documents** page at <http://www.ptc.com/appserver/cs/doc/refdoc.jsp>.

Licensing Alternatives for Existing Customers

Existing customers can alternatively use the online licensing tools to request an updated license pack via e-mail for installation on an existing PTC License Server. To run Creo 2.0 your PTC license pack must have updated licenses at version 33.0. See the LICENSING area at <http://www.ptc.com/support/index.htm>.

G-Post, VERICUT Products, and CGTech Licensing

In Creo 2.0, along with NC Solutions PTC is including support for CGTech’s VERICUT 7.1.5 and Austin NC, Inc.’s G-POST 6.4 P10.

CGTech has removed PTC licensing from the advanced VERICUT products as of release 5.4: 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 5.0, Creo Elements/Pro 5.0, and now with Creo 2.0 or later.

Operating System Support

Operating System	Version
Microsoft Windows Server 2008 R2	Microsoft Windows Server 2008 R2 (64-bit edition), Internet Explorer 8.0 or later, Mozilla based browser embedded with Creo 2.0.
Microsoft Windows 7	Windows 7 Professional, Windows 7 Ultimate and Windows 7 Enterprise. Base OS (32-bit and 64-bit Editions) and Service Pack 1, Internet Explorer 6.0 Service Pack 1 or later, Mozilla based browser embedded with Creo 2.0. Direct3D 10.0 graphics support is available for Windows 7.
Microsoft Windows XP	Windows XP Professional Edition and Windows XP Home Edition, Base OS and Service Packs (1, 2, and 3), Internet Explorer 6.0 SP1 or later.
Microsoft Windows XP x64	Windows XP Professional x64 Edition, Base OS and Service Packs 2, Internet Explorer 6.0 SP1 or later (see second note below).

System Requirements and Hardware Certification Information

Visit the **Platform Support** page for Creo 2.0 for current, detailed system requirements and certified hardware configurations:

<http://www.ptc.com/partners/hardware/current/support.htm>

Interoperability

PTC Product	Interoperability with Creo 2.0
Creo Advanced Framework Extension	Creo Advanced Framework Extension 2.0 F000
Creo Elements/Pro Expert Moldbase™ Extension	Available with Creo Expert Moldbase Extension 8.0 F000.
Creo Elements/Pro Progressive Die Extension	Available with Creo Progressive Die Extension 8.0 F000.
Creo Elements/Pro Routed Systems Designer™ and Creo Schematics	Available with Creo Schematics 1.0 and 2.0
Creo Elements/View™ ECAD Compare™ and Creo View ECAD Compare	Available with Creo Elements/View ECAD Compare™ 9.1 M060 and Creo View ECAD Compare 1.0.
Creo Elements/View Validate for ECAD™ and Creo View ECAD Validate	Available with Creo Elements/View Validate for ECAD 9.1 M060 and Creo View ECAD Validate 1.0.
Creo Elements/View Express (Standalone)	Interoperable.
Creo View Adapters (formerly ProductView™ Adapters)	<p>The Creo View Adapter 2.0 for Creo Parametric creates viewables for ProductView 9.0 or later from both Creo Parametric 1.0 and 2.0 and Creo Direct 1.0 and 2.0. Creo Parametric and Creo Direct can directly create viewable data for ProductView 9.0 or later. As a prerequisite for the adapter, you must have Creo Parametric installed.</p> <p>Creo View can directly read a limited subset of data created by Creo Parametric and Creo Direct.</p>
Pro/INTRALINK®	Interoperability planned for Windchill PDMLink 9.1 M070, 10.0 M030, and 10.1 F000. Creo 2.0 is not supported by Pro/INTRALINK 3.x. See Note 1 and 2 below.
Pro/TOOLMAKER™	Available with Pro/TOOLMAKER 9.0 M070.
Windchill® PDMLink®	Interoperability planned for Windchill PDMLink 9.1 M070, 10.0 M030, and 10.1 F000. See Note 1 and 2 below.
Windchill ProjectLink™	Interoperability planned for Windchill PDMLink 9.1 M070, 10.0 M030, and 10.1 F000. See Note 1 and 2 below.

Note:

1. For specific information on Windchill compatibility, refer to the [Creo Data Management Compatibility Roadmap](#) on the PTC Web site.
2. Before upgrading from one major release to another of Creo, transfer files from your workspaces to avoid the loss of any local changes during the process.

Asian and European Languages

Depending on the Creo application, the user interface and documentation are translated to varying degrees into French, German, Italian, Japanese, Korean, Russian, Simplified Chinese, Spanish, and Traditional Chinese. A detailed list of translated materials is available on PTC.com on the [Reference Documents](#) page. To find the list on this page, select **Creo Parametric** for **Product**, and select **Matrices – Language** for **Document Type**.

OpenGL Library


Creo 2.0 requires a graphics card supporting OpenGL 3.1 or later. Certain visual effects will be disabled when running in WIN32_GDI or software OpenGL graphics.

To address graphics-related issues, use the Creo alternative to WIN32_GDI. This configuration option toggles Microsoft Software OpenGL on the local machine:

```
use_software_opengl = yes/no.
```

Creo Help Center—Help and Books

The Creo Help Center is available on <http://www.ptc.com/support/index.htm> as a related resource and on DVD-ROM. Refer to *Installing Creo 2.0: Creo Parametric, Creo Direct, Creo Layout, and Creo Simulate* to install, set up, and use the Help Center. The default browser with its language setting is used for the Help Center.


You can access the Help Center by clicking  or using context sensitivity. The following items are in the Help Center:


- Context-sensitive Help
- LearningConnector for Creo Parametric and Creo Direct
- Books: *Installing Creo 2.0: Creo Parametric, Creo Direct, Creo Layout, and Creo Simulate*, *Read This First*, *Configuration Options*, and *FlexNet Publisher License Administration Guide*

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 the Adobe Web site:


<http://www.adobe.com/support/downloads/main.html>

Creo Resource Center—Web Tools and Tutorials

Web-based tools and tutorials can assist you with learning to use this release. The [Creo Resource Center](#) is accessible from the Creo browser, through the Creo Help Center by clicking , or on the Internet. Web tools and tutorials include the following items:

- What's New in Creo 2.0—Documentation of new functionality.
- [LearningExchange](#)—A central repository for tutorials including Creo tutorials.
- Command Search—Click  to quickly find the location of a command in Creo Parametric and Creo Simulate.

LearningConnector for Creo Parametric and Creo Direct

The LearningConnector for Creo, available by clicking  in Creo Parametric and Creo Direct, provides context-sensitive recommendations for topics, articles, and videos as you work. You can search the LearningConnector using one or more keywords. The LearningConnector provides results from the following sources:

- PrecisionLMS—eLearning topics and videos
- Technical Support Knowledge Base—Knowledge-based articles
- Help Center—Help topics and other materials and links

By default the LearningConnector is active. To disable the LearningConnector set the configuration option `enable_learning_connector` to `no`.

Product-Specific Information

Creo User Interface Customization

With Creo, you can customize various aspects of the user interface for Creo Parametric, Creo Direct, Creo Layout, and Creo Simulate:

- Ribbon
- Quick Access Toolbar
- Graphics toolbar
- Window layout

In all applications, customization settings for the user interface are automatically stored in the profile directory of the user's operating system:

```
%APPDATA%\PTC\ProENGINEER\Wildfire\wf\.Settings\creo_<app>_customization.ui
```

When Creo starts, customization settings are applied from the following locations in this order:

- Application load point directory:
`<creo_loadpoint>\Common Files\F000\text\creo_<app>_admin_customization.ui`
- User's profile directory
- Application startup directory, if this configuration option is set:
`load_ui_customization_run_dir = yes`

To facilitate administration and sharing, customization settings for the user interface can be imported, exported, or reset to the defaults.

Creo Parametric, Creo Direct, Creo Layout, and Creo Simulate Configuration

When Creo starts, stored configuration options from `config.pro` or `config.sup` or both are applied from the same locations and in the same order as in Creo Elements/Pro.

Creo Parametric TOOLKIT—Creo Parametric Only

With Creo Parametric TOOLKIT, you can extend the use of Creo Parametric. In Creo 2.0 for Creo Direct, Creo Layout, Creo Simulate, you cannot use TOOLKIT for the existing Pro/TOOLKIT applications. In Creo Parametric, using existing legacy Pro/TOOLKIT applications requires some modifications, mostly due to the new Ribbon user interface. Refer to the Creo Parametric TOOLKIT documentation for details. (Creo TOOLKIT was formerly Pro/TOOLKIT.)

Creo Advanced Framework Extension

Creo Advanced Framework Extension (AFX) 2.0 adopts the Creo look and feel to its new user interface. To install Advanced Framework Extension 2.0 Lite or Advanced Framework Extension 2.0, complete the following steps:

1. After installation of Creo 2.0, set the configuration option `afx_enabled` to YES.
2. Restart Creo 2.0.

©2012 Parametric Technology Corporation (PTC). The information contained herein is provided for informational use and is subject to change without notice. The only warranties for PTC products and services are set forth in the express warranty statements accompanying such products and services and nothing herein should be construed as constituting an additional warranty. PTC shall not be liable for technical or editorial errors or omissions contained herein. For Important Copyright, Trademark, Patent, Licensing and Data Collection Information: See the About Box, or copyright notice, of your PTC software. 01012012