Read This First
Creo®: Creo Parametric, Creo Simulate, Creo Options Modeler, Creo Layout, Creo Direct
7.0.0.0

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Creo Parametric, Creo Simulate, Creo Layout, Creo Options Modeler, Creo Direct

Creo 7.0.0.0 Is Available for Installation

Creo 7.0.0.0 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. 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 7.0.0.0 is packaged with the PTC License Server powered by FlexNet Publisher 11.16.2.1. PTC recommends that you upgrade your PTC License Server to 11.16.2.1.

For customers requesting physical media delivery, the software and help are shipped on one USB flash drive. For Creo 7.0 and later, DVD media is not available.

Windows Update for C Runtime Is Required for Creo 7.0.0.0

Creo 7.0.0.0 is built with Microsoft Visual Studio 2019 (VS2019), which is a compiler change. The machine on which you are running Creo Parametric, Creo Simulate, Creo Options Modeler, Creo Layout, or Creo Direct must have the Windows update for C Runtime installed. When installing these Creo applications, the installer detects if the correct Windows update for C Runtime is installed. This applies both to a local installation and when installing from a network.

If you do not have the required Windows update, the steps below describe what occurs and what action you need to take:

1. The message **The Windows update for Universal C Runtime is required in order for the installation to continue.** appears during installation and you are redirected to a Microsoft web page.
2. There is an automatic exit from the installer.
3. Open the Microsoft web page Windows 10 Universal C Runtime.
4. Install the required update.

After you complete the update, you can install these Creo 7.0.0.0 applications.

Using PTC Installation Assistant

For a streamlined installation of Creo 7.0.0.0 applications, you can use the PTC Installation Assistant. Open the installer, enter your Sales Order Number, and the Assistant begins the automatic installation:

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

For the most up-to-date installation information, see the Creo Installation and Administration Guide on the Reference Documents page or click **File ▸ Help ▸ Reference** in the application.

Note

PTC Mathcad option is removed from the PTC Installation Assistant for Creo. You can download the latest version of PTC Mathcad from eSupport. You must have a valid license for PTC Mathcad. For information on installing PTC Mathcad, refer to the PTC Mathcad Prime Installation and Administration Guide.

If you do not have a valid license for PTC Mathcad, you can download a trial version from the Mathcad Express Free Download page, which can be converted to a full licensed version when needed.

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 7.0.0.0 your PTC license pack must have updated licenses at version 38.0. See License Management.

G-Post

In addition to the current solution for NC in Creo Parametric 7.0.0.0, there is also support for G-POST V6.7 P17 from Austin N.C., Inc.

System Requirements and Hardware Certification Information

Visit the Creo 7.0 Hardware Notes page for Creo 7.0.0.0 for current, detailed system requirements and certified hardware configurations.

PTC Product	Release that is Interoperable with Creo 7.0.0.0
Creo Expert Moldbase	Creo Expert Moldbase 12.0.0.0
Creo Progressive Die	Creo Progressive Die 12.0.0.0
Creo Schematics	Creo Schematics 4.0
Creo View Product Family: Creo View and Creo View ECAD, Creo View ECAD Compare, Creo View Express, and Creo View Adapters	Creo View 6.1 and later releases. **Note** • A seat of Creo Parametric is a prerequisite for the Adapter. • For information about the latest (6.1) Clients and Adapters, see the Creo View Clients and Toolkits Software Matrix and the Creo View Adapters Software Matrix for details.
Pro/INTRALINK	Creo 7.0.0.0 is supported with Pro/ INTRALINK 11.1 M020 CPS11and later, and Pro/INTRALINK 11.2.1.2 and later. See the note that follows the table.
Windchill	Creo 7.0.0.0 is supported with Windchill 11.1 M020 and later, and with Windchill 11.2.1.2 and later. See the note that follows.

Note

- For specific information on Windchill compatibility, refer to the Creo Data Management Compatibility Roadmap.
- 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, Traditional Chinese, and Brazilian Portuguese. A detailed list of translated materials is available in the matrix Creo 7.0 Language Support.

OpenGL Library

Creo 7.0.0.0 uses advanced OpenGL 4.3 capabilities that improve the overall display and graphics performance of Creo. To take advantage of this enhancement and for overall improved display and graphics performance, Creo 7.0.0.0 requires graphics cards which support OpenGL 4.3 or later. PTC recommends you consult with your hardware manufacturer or graphics card vendor.

Upgrading Versions of Java

Java is not installed with the installation of the Creo application. You can use your existing installation of Java. PTC recommends Java version 8 for Creo 7.0.0.0.

If Java version 8 is not installed, then a message appears during the installation. The message notifies you that your version of JRE (Java run time environment) is out of date and recommends installing the latest version of Java 8 to optimize performance.

Storage and Order of Customized Settings for the Creo User Interface

You can customize various aspects of the user interface for your Creo application, including those in the list below:

- Ribbon
- Quick Access Toolbar
- Shortcut Menus
- Keyboard Shortcuts
- Graphics Toolbar
- Window layout

In all applications, unless the environmental variable, PTC_WF_ROOT is set, customization settings for the user interface are automatically stored in the profile directory of the user's operating system in one of the locations listed below:

- When not connected to Windchill PDMLink, the file is created in %APPDATA%\PTC\ProENGINEER\Wildfire\.wf\.Settings\creo_customization.ui
- When connected to Windchill PDMLink, the file is created in the %PTC_WF_ROOT%\.Settings folder.

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

- Application load point directory:

  <creo_loadpoint>\F000\Common Files \text\creo_<app>_admin_customization.ui\
- User's profile directory
- Application startup directory, if the configuration option load_ui_customization_run_dir is set to yes.

Refer to Support article CS37185 for details.

Recommendation for Configuration Files and Working Directories

When your Creo application starts, stored configuration options from the config.pro file or the config.sup file, or from both files, are applied from the same locations and in the same order as they were in Creo 6.0. It is recommended to use separate configuration files and working directories for each Creo application.

Creo Help Center—Help, What's New, Tutorials, Reference Documents

There is a separate Help Center for each of the Creo applications. Help Centers are available on ptc.com or you can download a Help Center for a local installation. Refer to the Creo Installation and Administration Guide. The Help Center provides access to the Help, including What's New for your application and to other resources such as free tutorials, Reference Documents, and the Creo Community. To open a Help Center, click in your application. Alternatively, you can learn about user interface items with context-sensitive Help using the F1 key. To access reference information such as the Creo Installation and Administration Guide, click **File ▸ Help ▸ Reference** from your application or browse through Reference Documents.

PTC Learning Connector

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

- PTC University Precision LMS—eLearning topics and videos
- Support Knowledge Base—Knowledge-based articles
- Help Center—Help topics and links to other materials

By default, the Learning Connector is active. To disable the Learning Connector set the configuration option `enable_learning_connector` to `no`.

Information Related to Windchill

Chromium Embedded Browser

When using Chromium as the embedded browser in Creo to interact with Windchill, you cannot view the 3D Creo View based thumbnails on the details page of the Creo models. Chrome and Chromium based browsers do not support plug-ins, which is the technology currently used to display the 3D Creo View based thumbnails in Creo. See Support article CS197208 for details.

Single Sign On (SSO) Authentication

The embedded browser for Creo supports authentication to a Windchill server that is configured for SAML 2.0 single sign-on (SSO) authentication. If you previously authenticated with a standalone web browser, you will see an additional login request from the embedded browser for Creo.

Creo Parametric TOOLKIT Is Updated to Support the Multibody Environment

The Creo Parametric TOOLKIT environment supports concepts for multibody part design. In Creo Parametric 7.0.0.0 and later, you can create parts that contain one or more geometric bodies. Bodies contain only solid geometry. Nonsolid entities, like datums, curves, and quilts, are not contained in any body. Bodies typically contribute to the mass properties of the model and you can perform geometric operations on the bodies, such as splitting a body or merging with other bodies. You can also select bodies as references for features. For more information on the multibody functionality refer to the Creo Parametric Help. The existing Creo Parametric TOOLKIT applications continue to work seamlessly for legacy models and models having a single body. However, you must upgrade Creo Parametric 7.0.0.0 to use the multibody environment. Creo Parametric TOOLKIT is updated as follows to support the multibody environment:

- APIs are added
- Some APIs are deprecated and are superseded by new APIs
- Implementation of existing APIs is updated. No visible changes in Creo Parametric TOOLKIT APIs
- Element trees are added to support new multibody features
- Some existing element trees are updated
- Some enumerated data types and their values are updated
- Existing structures are updated

Refer to the chapter, Migrating to the Multibody Environment, in the Creo Parametric TOOLKIT 7.0 User's Guide to understand the impact due to these changes and then update your Toolkit applications as appropriate.