



Creo[®] View 4.0

Installation Guide

Datecode F000

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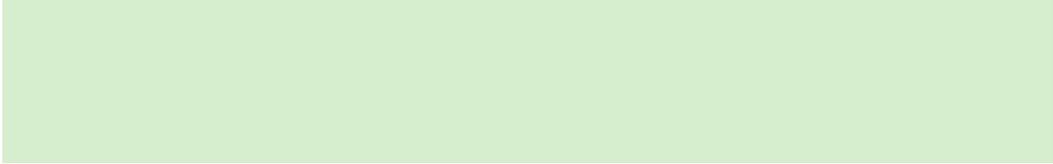
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About This Guide

Creo View is PTC's primary enterprise Visualization application. It provides flexibility to view CAD files and their associated images and documents as a standalone client or in the context of a Windchill server. This guide describes how to generate and configure licenses, and how to install and maintain the Creo View client. A section for system administrators on deploying Creo View describes performing a silent installation and customizing the installer.

System administrators who are performing advanced deployment tasks for Creo View must be experienced in application installations and must have a good understanding of operating systems.

How to Use This Guide

This guide supplements the descriptions in the installer and provides installation prerequisites, instructions, and reference information. To generate or update licenses and then install or reconfigure Creo View, see the following chapters:

Chapter	Role	Information
Overview	All users	Describes the materials in your software shipment and lists the prerequisites for installation.
Quick-Start Installation	Existing users	Describes a quick-start approach for users who have previously installed license management and product software.
Obtaining and Configuring Licenses	System administrators This chapter does not apply to users of Creo View Lite and Creo View Express.	Describes how to generate or update licenses for Creo View and how to install PTC License Server.
Installing Creo View	All users	Describes a simple installation process for Creo View.
Starting Creo View and Modifying an Installation	All users	Describes the steps to start Creo View and the procedure to uninstall the software.
Deploying Creo View and Customizing the Installation	System administrators	Describes methods for deploying Creo View and outlines strategies for silent installation.
Updating an Installation	All users	Describes how to update Creo View on your system.

Related Documentation

The following documents on the [Reference Documents](#) page may be helpful as you proceed with the installation:

- *FlexNet Publisher License Administration Guide* that discusses the third-party license management software for distributing Creo licenses
- *Standalone PTC License Server* that discusses the installation of the ladmin-based PTC License Server

For information about installing the Creo View MCAD and ECAD Adapters, see the following documents on the [Reference Documents](#) page:

- *Creo View MCAD Adapters Installation and Configuration Guide*
- *Creo View ECAD Adapters Installation, Configuration, and Publishing Guide*

Technical Support

Contact PTC Technical Support via the PTC website, phone, fax, or e-mail if you encounter problems using your software. You can log a new case or track an existing case or SPR (Software Performance Report) using the PTC webpage at www.ptc.com/support/index.htm.

You must have a Service Contract Number (SCN) before you can receive technical support. If you do not have a SCN, contact PTC License Management using the instructions in the *Customer Service Guide*. This guide is on the PTC webpage at www.ptc.com/support/index.htm.

Documentation for PTC Products

PTC provides documentation for download at PTC.com. The following forms of documentation are available:

- Context-sensitive Help with a search facility and quick links to helpful information.
- *Installing Creo 3.0: Creo View* and other books as PDF files. To view and print the books, you must have Adobe Acrobat Reader installed.

Press F1 or click  on a user interface item to get context-sensitive Help.

To access all PTC Documentation from PTC.com, you must have a valid user account. Go to the [Create New PTC eSupport Account](#) page to request a user account or call Customer Support. For worldwide phone numbers click **Contact** under **Customer Support Guide** on the PTC webpage at www.ptc.com/support/index.htm.

Feedback to Documentation

PTC welcomes your suggestions and comments on its documentation—send feedback to the following address:

`mcad-documentation@ptc.com`

Please include the name of the application and its release with your comments. For online books, provide the book title.

Additionally, you can report any documentation issues using the online [Case Logger](#) tool. On the second **Case Logger** page, select a product and then select **Help Center / Documentation** from the **Technical Area** list. Upon submission of all information, a case number is returned immediately.

Documentation Conventions

PTC documentation uses the following conventions:

Convention	Item	Example
Bold	Menu paths, dialog box options, buttons, and other selectable elements from the user interface	Click File ► New . Click OK .
Courier	User input, system messages, directories, and file names	Processing completed.
Courier with less-than and greater-than symbols (< >)	Variables for which an appropriate value is substituted	output= <LOADPOINT>

 **Note**

Examples of command-line arguments may contain hidden line breaks to fit on the page.

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Overview

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This chapter describes the materials in your software shipment. It describes the various editions of Creo View, and it outlines the prerequisites for installing and running the software.

What You Receive

PTC (Parametric Technology Corporation) sends the following materials related to your software order:

- Software Order Confirmation e-mail—Before the receipt of your PTC software, you will receive an e-mail containing all the details of your order.
- Software DVD—Your order contains one or more DVDs for each product purchased. These programs are included on the Creo View DVD:
 - Creo View client
 - Creo View ECAD Validate
 - Watermark Editor

Editions of Creo View

Creo View comes in two licensed editions and one unlicensed edition, all based on the same core viewing technology:

- Creo View—Includes all core functionality for MCAD, ECAD, or both, including interoperability with Windchill. Requires a FLEX license, and some optional modules require additional licenses.
- Creo View Lite—Provides limited viewing and markup functionality and interoperability with Windchill. Requires a legal usage license that comes bundled with the Windchill client.
- Creo View Express—Provides basic viewing functionality without a license and is not interoperable with Windchill.

The Creo View license is configured for MCAD, ECAD or both. If you purchase Creo View MCAD, the functionality of the Lite edition of Creo View ECAD is provided, and vice versa. Depending on the Creo View license configurations set up by the system administrator, you can retrieve license options for your workstation according to your tasks. The optional modules can be set to load at startup or when required. This flexibility provides efficient license sharing. To work offline, you must go online and borrow temporary Creo View licenses. The following optional modules require additional licensing:

Creo View Edition	Module
MCAD	Animator
	Interference Analysis
	Design Check
ECAD	Compare
General License Options	Massive Assembly

Massive Assembly Requirements

The Massive Assembly option (FlexNet license feature 280) is required to open large assemblies. The option is supported only with the Creo View MCAD Edition and is not supported for use with Creo View Lite or Creo View Express Editions.

The option is required if addressed memory exceeds the following thresholds while opening large data sets in Creo View:

Release	Creo View
Creo View 1.0	2.0 Gb
Creo View 2.0	3.5 Gb
Creo View 3.0	8.0 Gb
Creo View 3.1	8.0 Gb

Note

The following table indicates the memory limitations of Creo View Lite.

Release	Creo View Lite
Creo View 1.0	2.0 Gb
Creo View 2.0 or later	3.5 Gb

Before You Begin

Before you install Creo View or Creo View Lite, make sure that all the prerequisites are met:

- You have opened an online account at PTC.com (Administrators of Creo View only).
- You have received the Software Confirmation Order e-mail with the product code or the Sales Order Number for your shipment. Alternatively, you must have received your License Pack via e-mail from PTC License Management (Administrators of Creo View only).
- You have received the product DVD in your software shipment. If not, follow the link on your Software Confirmation Order e-mail to download the installation package to a folder on the local or the network computer. Alternatively, download Creo View from PTC.com or from Windchill. See [Downloading Creo View](#) or [Performing an Installation from the Web](#).
- You have read the READ THIS FIRST document for known issues.
- You have checked the following locations for information on support of various platforms, languages, and toolkits:

-
- The *Creo View Clients & Toolkits Software Matrix*. The matrix is available on the [Reference Documents](#) site.
 - The [System Prerequisites](#).

You must install PTC License Server Manager before installing Creo View except in these cases:

- You have purchased node-locked licenses.
- You are a user of Creo View Lite or Creo View Express.

See [Installing the PTC License Server](#).

Opening a PTC Online Account

You must have a PTC online account to generate a license and install Creo View. Use the following instructions to create an online account:

1. On a computer connected to the Internet, go to the [Create New PTC eSupport Account](#) page.
2. Fill in any empty boxes.
3. Click **Create Account**. A confirmation page indicates a successful account creation.
4. Review and print this confirmation for your record. A confirmation of your account is sent to your e-mail address.

Resetting Your Password

If at any time you do not remember your PTC.com password, follow these instructions:

1. Open the [Reset Your Password](#) page.
2. Type your user name.
3. Click **Continue**. A URL to reset your password is sent to your e-mail address.

System Prerequisites

This section describes the prerequisites for hardware, operating systems, graphics cards, and software.

Hardware and Operating System Requirements

Because you can view files with widely varying size and complexity, the hardware requirements partly depend on the type and size of files. The minimum requirements are sufficient for small-sized or average-sized files. For optimum performance, meet these recommended specifications:

Item	Minimum Requirement	Recommended Specification
Random access memory (RAM)	1 GB	2 to 4 GB
Central processing unit (CPU) or Core	Single-core, 1-GHz x86 Intel compatible	Dual-core or greater, 2-GHz x86 32-bit or 64-bit Intel compatible
Disk space	500 MB for a 32-bit platform 800 MB for a 64-bit platform	1 GB for the installation Additional disk space for data
Operating system	Windows XP 32-bit Solaris 10 x64 Redhat Linux 5.2 amd64 Novell SUSE 12.0 amd64	Windows 7 64-bit Solaris 10 x64 Redhat Linux 6.2 amd64 Novell SUSE 12.0 amd64

The [Platform Support](#) page contains information on requirements for hardware, graphics drivers, and operating systems for PTC products, including Creo View:

Graphics Requirements

You must have OpenGL library version 1.1 or later installed on your machine. Make sure you have the latest drivers for your graphics hardware. You can download the latest drivers from the manufacturer's website. The graphics requirements depend on the size and complexity of your data sets. If you plan to view and manipulate large or complex CAD files, check PTC's list of supported hardware for a list of appropriate graphics cards:

[Platform Support](#)

Applying special configurations to your graphics card is not recommended.

Platform-specific Requirements

Creo View supports Windows, Linux x64, and Solaris x64. For Windows, you must have administrative privileges to install Creo View. These are the prerequisites for all Redhat Linux releases:

- OpenMotif runtime libraries (“openmotif” package) must be installed on the system using your operating system’s package manager. For example, yum.
- You must have the latest proprietary graphics drivers from your graphics card vendor. Your operating system’s graphics drivers are not sufficient.

For Redhat Linux 6.0 and later, you must also meet these prerequisites:

- 75-dpi fonts (“xorg-x11-fonts-75dpi” package and “xorg-x11-fonts-ISO8859-1-75dpi” package)
- 2.96-Redhat compatibility standard C++ libraries (“compat-libstdc++-296” package, “libstdc++” package, and “libstdc++.i686” package)

For more details on platform support, see the READ THIS FIRST document.

Software Requirements and Helper Applications

For Windows installations, all of the software requirements are automatically met when you install from one of these locations:

- Product DVD
- Windchill **Software Downloads** page
- `CreoView_64.exe` or `CreoView_32.exe` executable files

If you install the application as an administrator directly from the MSI file, the software prerequisites are not automatically met. See [Deploying Creo View and Customizing the Installation](#) for more information.

Microsoft C++ Runtime and OpenGL 1.1 or later are required for installing and running Creo View. C++ Runtime is shipped with Creo View and installed automatically if needed. If you already have C++ Runtime, the installation is not modified.

Other helper applications are shipped and automatically installed with Creo View:

- Creo View Files Tools—Used for transforming or optimizing Creo View data.
- Creo View ECAD Adapters—Required for natively viewing the supported ECAD file formats. Creo View ECAD Adapters is installed automatically only with the DVD installer. See [Installers for Creo View](#) for more information.
- `pview_batch.exe` tool—Useful for scripting or troubleshooting.

Browser Support Requirements

You may have to set special options for your Web browser. Details follow.

Cross-Locale Support for Online Help

For best results, access the Help in the same language you set for your operating system. You may see corrupt characters if you view the Russian or Asian language Help on a European language operating system, or if you view European language Help on a Russian or Asian language operating system. In most cases, you can circumvent the problem by setting the encoding in your browser:

- Internet Explorer—Click **View** ► **Encoding** ► **Auto-Select**.
- Mozilla Firefox—Install the CHM reader add-on and select **Auto-Detect** or **Auto-Select**, depending on the language version of Firefox.

Google Chrome Support for Creo View MCAD and ECAD

The Google Chrome browser has these limitations with the Creo View clients:

- The Version Checker plug-in does not work with the Chrome browser.
- Before using Creo View with the Chrome browser, you must install Creo View from one of two places:
 - On the **Software Downloads** page, accessible through the Windchill client at **Quick Links** ► **Software Downloads** ► **Setup and Installation: Creo View Installation**. See [Performing an Installation from the Web](#) for more information.
 - From the DVD or from the downloaded DVD image (use the installer script `setup.vbs`). For more information, see [Performing an Installation from the DVD](#).

PTC Customer Agreement

Before you install software, you must accept the PTC Customer Agreement: To do so, follow these steps:

- You must accept the license agreement to proceed even if a license agreement already exists. The license agreement shown supersedes the previous license agreement.
- If you decline the license agreement you cannot proceed.

See [Performing an Installation from the DVD](#), [Performing an Installation from the Web](#), or [Performing an Installation Using the Executable File](#).

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Quick-Start Installation

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This chapter provides a quick-start approach for these users:

- Experienced users who have previously installed license management and product software on license server, license client, and node-locked machines.
- Users of Creo View Lite or Creo View Express

For step-by-step installation instructions, read the subsequent chapters in this guide. New users can refer to the chapter [Installing Creo View](#).

Updating PTC License Server

You must have PTC License server installed before you install Creo View. If you are updating your software to a later release, you must update your PTC licenses. See [Updating the License Server](#) for details. You need not install PTC License server if you purchased uncounted node-locked licenses, or if you are installing Creo View Lite or Creo View Express.

Performing a Quick-Start Installation

Install Creo View to a client machine and to the license server machine using the following workflow:

1. Start the installer.
2. Accept the PTC Customer Agreement.
3. Accept the default installation path or set a new path.
4. Click **Install**. The Creo View clients are installed.

See [Performing an Installation Using the Executable File](#) or [Installing Creo View for UNIX](#) for details on the installation process.

5. Click **Finish**.

Where Your Software Is Installed

The software is installed in a default directory. You can modify the path during installation. The default path follows:

```
C:\Program Files\PTC\Creo <release_number>\View\
```

PTC License Server, Creo View, and any related components are installed by default into separate subdirectories within the PTC directory. You cannot change the installation location for Creo View ECAD Validate.

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Obtaining and Configuring Licenses

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This chapter explains how to obtain or update licenses. It also provides instructions to install PTC License Server, a third-party license management software for Creo View. You must install PTC License Server Manager before installing Creo View, except in the following cases:

- You have purchased uncounted node-locked licenses. Refer to your sales documents for your license type.
- You are a user of Creo View Lite or Creo View Express

The license server need not be installed on the same machine as Creo View. For the version requirements appropriate for your license server, see the **READ THIS FIRST**.

PTC uses FlexNet Publisher from Flexera Software, Inc. as its license server. See [License Management Software](#) for an overview and benefits of the license management software. The *FlexNet Publisher License Administration Guide* may be helpful as you install PTC License Server.

Using the License Server Manager

The license server manager handles the initial contact with your PTC software that uses FlexNet licensing. It passes the connection to the appropriate vendor daemon. A license server manager serves the following purposes:

- Starts and maintains vendor daemons for serving license rights from different software.
- Transfers specific software requests to the correct vendor daemon.

There are two versions of the license server manager for license administration:

- `lmadmin`—Uses a graphical user interface (GUI)
- `lmgrd`—Uses a command-line interface

PTC currently ships with its products a license server manager based on `lmadmin` with the `ptc_d` vendor daemon at version 11.10. It is a replacement to the `lmgrd`-based license server manager shipped with the earlier releases of PTC products. If you are using an `lmgrd`-based license server manager with `ptc_d` vendor daemon of version 10.8.x.x, you need not migrate to `lmadmin`.

Verifying System Requirements

Installation requirements for the license server manager software, follow:

- On all Microsoft Windows platforms, you must have administrative privileges to install the PTC License Server.
- You must have TCP/IP (Transmission Control Protocol/Internet Protocol) installed and configured correctly on your Windows system before installing the software.
- For Windows Server 2008 R2, FlexNet Publisher 10.8.x.x, or later, is supported. It is available for installation only as a downloadable utility.
- To use the `lmadmin`-based license server manager on Windows platforms, you must install the Microsoft Visual C++ 2005 SP1 Redistributable Package (x86). You can install this package during the license server installation process.

Before You Proceed

Gather all the necessary information that follows about the installation of the license server manager:

- Check the following documents for the latest installation settings:
 - The READ THIS FIRST document of your product for any changes to your computer settings

-
- The FlexNet License [Hardware Notes – PTC License Server](#) for license server installation requirements.
 - Make note of your product code that arrives via e-mail. If you have received a license file via e-mail, store it at a secure location on your disk.
 - If you have already installed PTC License Server based on `lmgrd` and choose to migrate to `lmadmin`, complete these steps before the migration:
 1. Shut down any `lmgrd` service or processes running on the system.
 2. Save the `license.dat` file from your existing license server installation to a secure location.
 3. Uninstall the previous installation completely.
- See “Migrating from `lmgrd` to `lmadmin`” in the *FlexNet Publisher License Administration Guide* for more information.

Obtaining a License

You must generate or update existing licenses to use the latest software for Creo View. Licenses determine the edition and optional modules that you are authorized to run on your computer. You can get a license for Creo View MCAD or Creo View ECAD in two ways:

- New customers—Use the PIM installer (PTC Install Manager). See the next section.
- Existing customers—Use existing tools (required). Skip to the section Existing Customer Licensing.

Creo View uses a PTC License Server powered by FLEXnet Publisher 11.10.x or later from Flexera Software Inc. If you are already running that license server for another Creo application, you need only update your license file to support the latest version of Creo View. If you are a new user, or if your license server is outdated, you must install the PTC License Server when using floating licenses.

New Customer Licensing

The PTC Install Manager streamlines the license installation process for new customers. Use the link on the Shipping Confirmation Letter. You need your Sales Order Number (SON) and an upgraded PTC account associated with the customer number. To begin, update your account with a user name and password at the following link:

<http://www.ptc.com/appserver/common/account/secure/premiumAccount.jsp>

If you do not have a PTC Support account, you can open one here:

<https://www.ptc.com/appserver/common/account/create.jsp>

Perform the following steps to obtain the license using the PIM installer, which also has on-screen instructions:

1. Start the PIM installer from the link on the Shipping Confirmation Letter.

 **Note**

You must run the installer on the machine to be licensed. This is true for both a floating license for a license server or a node-locked license for a single client machine.

2. Type your user account name (usually your e-mail address), and then type your SON (Sales Order Number). Your license file is downloaded. For floating licenses, PTC License Server is also installed.
3. Click **Next**. The License Agreement appears.
4. Accept the agreement and click **Next**. A message appears.
5. Verify the network card for your license file configuration, and then click **Next**. This step determines automatically the Host ID of the license. A message appears.
6. Supply your license file. The installer downloads the licenses and installs the License Server if applicable.

Node-locked licenses are saved locally in a path noted on the screen. A copy of the license type will be e-mailed to the address on file for the account used in step 2.

Existing Customer Licensing

Existing customers should continue to use the existing licensing tools. Visit the Customer Support Guide page at <https://support.ptc.com/>, click **Licensing**, and then follow the instructions to manage your licenses.

Installing the PTC License Server

After you have received an e-mail from PTC with the product codes for your software order, install the Standalone PTC License Server. Follow the installation instructions in the *Standalone PTC License Server* guide on the [Reference Documents](#) page.

Borrowing and Returning Licenses

To work independently of a server, for instance from your laptop at home or at a customer site, you must borrow a temporary license. License borrowing is available only on Windows. After 14 days this license expires, but it is good practice to borrow it only as long as needed so it is available for other users.

Borrow and return licenses using an executable file that comes with the software. Run it using arguments to borrow licenses for the versions and options you need. Instructions follow.

1. Open a command prompt on a computer with a server connection.
2. To read a short help message, type `<Creo View installation location>\bin\pvlicense_borrow.exe`
3. Add arguments to the command line as follows:
 - To borrow a license, type `-borrow` and then the features and options to borrow, and the license end date. See [Example: Selecting Licenses to Borrow](#).
 - To return a license, type `-return` and then the features or options to return.
 - To view a list of currently borrowed licenses and their expiry dates, type `-list`.
4. Press ENTER.
5. To verify that your borrowed license works, disconnect your computer from the network and run Creo View.

The general structure of the command-line syntax for borrowing a license consists of the following elements: `pvlicense_borrow [-server LicenseServerInfo] dd-mmm-yyyy[:hh:mm] LicenseFeature [+LicenseOptions]`.

A typical license borrowing request looks like this: `D:\Users\myname>"C:\Program Files\PTC\Creo <release_number>\View\bin\pvlicense_borrow.exe" -borrow 31-dec-2012 pvmcadpro +largeaddress +interference +animation +designcheck.`

Example: Selecting Licenses to Borrow

Features available for a Creo View license follow:

License Feature	Description
pvmcadpro	Creo View MCAD
pvecadpro	Creo View ECAD
pvstandard	ProductView Standard

License Feature	Description
<code>pvbatchprint</code>	Batch printing utility—Prints multiple files with a server-controlled batch job.
<code>pvbatchinterference</code>	Batch interference detection utility—Performs multiple interference detection queries with a server-controlled batch job.

If you purchased Creo View MCAD, these options are available:

Creo View MCAD Options	Description
<code>+largeaddress</code>	Massive assembly—Opens very large assemblies. For more information, see Massive Assembly Requirements in the Overview chapter.
<code>+interference</code>	Interference detection—Analyzes structures for interferences
<code>+animation</code>	Animation—Creates animations
<code>+designcheck</code>	Design check—Provides tags for collaboration and review

4

Downloading Creo View

Downloading Creo View from PTC.com 32

This chapter describes the steps for downloading the DVD image of Creo View from PTC.com.

Downloading Creo View from PTC.com

Follow the next procedure to download Creo View.

1. Visit the **Order or Download Software Updates** page:
http://www.ptc.com/appserver/cs/software_update/swupdate.jsp
You may have to type your user name and password and click **Log In**.
2. Click **Order or Download Software Updates**. The **Authorized Use Only** page opens.
3. Under **Customer Search**, type your **Customer Number** or **Customer Name** and click **Next**. The **PTC Software Download** page opens to **Step 1: Select the Product Family**.
4. Select **CREO VIEW & PRODUCTVIEW**. The **PTC Software Download** page opens to **Step 2: Choose Release & Download**.
5. Find the release to download and click to expand it.
6. Click next to **Creo View Clients**.
7. To download the most recent datecode of the release, click to expand **Most Recent Datecode**, and then next to **Download now**, click **HTTP**. The download begins.
8. To download another datecode, follow these steps:
 - a. Click to expand **Show all Other Available Datecodes**.
 - b. Click next to the datecode to download.
 - c. Next to **Download now**, click **HTTP**. The download begins.

You have now downloaded Creo View. See [Performing an Installation from the DVD](#) for the next steps.

5

Installing Creo View

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This chapter describes the process for installing Creo View using the installer on a single machine. Administrators who want to deploy Creo View on multiple machines should skip to the next chapter, [Deploying Creo View and Customizing the Installation](#).

Downward Compatibility

Creo View is downward compatible. For example, you can retrieve Creo View files (structures, viewables) created in a previous release of Creo View using the current version of Creo View.

Installers for Creo View

There are three locations from which you can access a Creo View installer:

- DVD or downloaded DVD image (Windows and UNIX).
- Web installation from a Windchill server or the Creo View Toolkit page (Windows only). See [Downloading Creo View](#).
- `CreoView_32.exe` or `CreoView_64.exe` downloaded executable files (Windows only).

Instructions follow for using the various installers. See [Installing Creo View for UNIX](#) for instructions on starting the downloaded installer in a UNIX environment.

Performing an Installation from the DVD

Use the product DVD or downloaded DVD image to install Creo View, the Creo View ECAD Adapters, and Creo View ECAD Validate. You install the group of programs automatically using the workflow below. Follow the procedure again to install the Watermark Editor tool separately.

1. Start the installer in one of these ways:
 - Load the DVD.
 - Extract the downloaded ZIP file into a new, empty directory, and then double-click `setup.vbs`.

The **Creo View -Clients** dialog box opens.
2. Select a language and click **OK**. The **Creo View — Clients** installer dialog box opens to the **Before You Begin** page.
3. Read the information, and then click **Next**. The **PTC Customer Agreement** page opens.
4. Read the Terms and Conditions and select **I accept**.
5. Click **Next**. The **Select Installation Type** page opens.
6. Select the software to install:
 - **Install Creo View Clients**
 - **Install Watermark Editor**

 **Note**

The Watermark Editor is available only on Windows.

7. Click **Next**. The **Select Directory** page opens.
8. Under **Installation Directory**, accept the default directory or browse to a new path, and then click **Select**. The **Review Settings** page opens.
9. Review the settings and the disk availability, and then click **Install**. Creo View is installed.

You have now installed Creo View. To start the application, continue to the next chapter, [Starting Creo View and Modifying an Installation](#).

Performing an Installation from the Web

The Windchill Help contains general instructions for downloading and installing Creo View from the Windchill **Software Downloads** page. If Creo View is not already installed on your machine, you can customize the installation. To minimize the file size, select only the elements you need from the **Creo View Installation** page. A summary of the number and size of the files to install updates automatically. If you are updating your version of Creo View, you cannot modify the installation directory. Instructions for the installation follow:

1. Under **Options**, select one or more of these features:
 - **AutoCAD Viewers**—Imports the AutoCAD file filter.
 - **Import Filters**—Imports the other supported external file filters.
 - **Download Help Files**—Provides information on functionality and procedures for completing tasks.
2. Under **Languages**, select one or more languages for the user interface and the Help.
3. In the box next to **Installation Directory**, accept the default path or type a new path.
4. For first-time installations, click **Install**. For updates, click **Reinstall**. The installation begins.

You can download `CreoView_64.exe` or `CreoView_32.exe` and perform the installation manually using the link at the bottom of the **Creo View Installation** webpage. See [Performing an Installation Using the Executable File](#). You must download and install the Creo View ECAD Adapters separately, using these links:

- **32-bit**
- **64-bit**

You have now downloaded and installed Creo View. To start the program, continue to the next chapter, [Starting Creo View and Modifying an Installation](#).

Performing an Installation Using the Executable File

If you download the installer, you can use the executable file to perform a simple installation or an advanced installation. A simple installation automatically uses the default installation location, `C:\Program Files (x86)\PTC\Creo <release_number>\View\`. These features are also installed:

- **Demonstration Dataset**
- **Import Filters**—All
- **Language support**—English only

To add languages or remove features, skip to the section below, [Performing an Advanced Installation](#).

Performing a Simple Installation

To install Creo View and the default components to the default location, follow the steps below.

1. Double-click `CreoView_32.exe` or `CreoView_64.exe`. The **PTC Creo View 3.0 Setup** dialog box opens to the **PTC Customer Agreement** page.
2. Read the Terms and Conditions, and then select **I accept the terms in the License Agreement**.
3. Click **Install**. Creo View is installed.
4. Click **Finish**.

To install the Watermark Editor tool, see [Installing the Watermark Editor Tool Using the Executable File](#) below.

Performing an Advanced Installation

To modify the location or contents of the installation, including language support, follow the steps below. You can add or remove entire features, or selected components of a feature. For example, you can remove all import filters, or you can set each filter's status individually.

1. Double-click `CreoView_32.exe` or `CreoView_64.exe`. The **PTC Creo View 3.0 Setup** dialog box opens to the **PTC Customer Agreement** page.
2. Read the Terms and Conditions, and then select **I accept the terms in the License Agreement**.
3. Click **Advanced**. The **Destination Folder** page opens.
4. Accept the default path for the installation, or set a new path:

-
- a. Click **Change**. The **Change destination folder** page opens.
 - b. Next to **Look in**, browse to a new location.
 - c. Click **OK**. The path is changed.
5. Click **Next**. The **Product Features** page opens.
 6. To add or remove one or more features, such as language support or file import filters, follow these steps:
 - a. Next to a feature to remove, click **Remove**, and then select **Entire feature will be unavailable**.
 - b. Next to a feature to add, click **Add**, and then select one of these options:
 - **Will be installed on local hard drive**—Installs the selected feature or component.
 - **Entire feature will be installed on local hard drive**—Installs all components of the selected feature.
 7. Click **Install**. Creo View is installed.
 8. Click **Finish**.

You have now installed Creo View. To start the program, continue to the next chapter, [Starting Creo View and Modifying an Installation](#).

Installing the Watermark Editor Using the Executable File

Creo View is shipped with a Watermark Editor tool. You must install it separately. The recommended method for installing the tool is the DVD installer. If you prefer, start the subinstaller from a subdirectory of the installation folder. Installation instructions follow.

1. In the DVD directory, navigate to and open `\installers\wmeditor`.
2. Double-click `ProductView_Watermark_Editor.msi`. The **ProductView Watermark Editor Setup Wizard** dialog box opens.
3. Click **Next**. The **End-User License Agreement** page opens.
4. Read the agreement, and then select **I accept the terms in the License Agreement**.
5. Click **Next**. The **Destination Folder** page opens.
6. To accept the default installation location, click **Next**, and then skip to step 8.
7. To set a new installation location, follow these steps:
 - a. Click **Change**. The **Change destination folder** page opens in a new window.
 - b. Browse to a new location, and then click **OK**. The installation location is updated.
 - c. Click **Next**.

8. Click **Install**. The Watermark Editor tool is installed.
9. Click **Finish**.

Installing Creo View for UNIX

Install Creo View on a Solaris x64 or Linux x64 machine using the workflow below. To perform an extra step required for using Creo View with Windchill on UNIX, see Embedding Creo View in a Web Browser below.

Performing a UNIX Installation

When you download the DVD image from PTC.com, you receive a ZIP file. Instructions for starting the installation follow.

1. Extract the ZIP file into an empty directory, <installer_path>.

Note

To keep file permissions stored in the ZIP file, you must extract the archive from a UNIX machine.

2. From <installer_path>, run `./setup`. The **Creo View -Clients** dialog box opens.
3. To continue with the installation, follow the instructions in [Performing an Installation from the DVD](#).

Embedding Creo View in a Web Browser

To use Creo View with Windchill on UNIX, you must manually add an environment configuration to embed Creo View in your web browser. Instructions for finding the Readme file follow.

1. Browse to <Creo_View_installation_path>\browser_plugins.
2. Follow the instructions in `readme.txt`.

Enabling the Launching of Creo View from Windchill Using SSL Client Certificates

Out-of-the-box, Creo View is not configured to use SSL Client Certificates, a requirement for it to correctly authenticate HTTP requests sent to a Windchill 11.0 server configured for the Public Key Infrastructure (PKI) Authentication method.

As a result, when launching Creo View from Windchill, using the **Open in Creo View** action or by clicking on the Thumbnail, the following error message may appear.

```
Windchill request error
```

```
ATTENTION: Error occurred communicating with Windchill.
```

```
          Contact your Windchill administrator for help.
```

This occurs if Windchill is configured to use Public Key Infrastructure (PKI) Authentication.

To resolve this issue and enable the use of SSL Client Certificates when launching Creo View from Windchill, set the following environment variable to any value, for instance `PVIEW_SEARCH_ALL_CERTIFICATES=<1>`, in the runtime environment of Creo View on all client machines.

6

Starting Creo View and Modifying an Installation

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This chapter explains how to start Creo View and configure the startup. Proceed to the end of the chapter for the procedure to uninstall Creo View.

Starting Creo View

After your license management and product software have been installed, start Creo View. You can run the application using the **Start** menu. Click **Programs ▶ PTC Creo ▶ PTC Creo View <release_number>**. The first time you start the software, the **PTC Creo View Startup** dialog box opens. Instructions follow for setting the edition and license options. To set a language for the Creo View user interface, see *Setting the Language* below.

Setting the Edition and License Options

Use the **Creo View Startup** dialog box to set the options for starting Creo View as follows:

1. To make the startup options the default, clear the **Choose edition at startup** check box.
2. Next to **Choose Creo View Edition**, select the edition to start:
 - **Creo View Lite**
 - **Creo View MCAD**
 - **Creo View ECAD**
 - **Creo View MCAD and ECAD**
 - **Creo View Standard**
3. For each license option that you purchased, select a loading option:
 - **Load at startup**
 - **Load when required**
 - **Never** (optional modules only)
4. To update your license, under **License Information** in the **License Server** box, type the name of the server or the path to the node-locked license file.
5. Click **Apply**. Click **OK**. Creo View starts.

Setting the Language

By default, the language of the Creo View user interface is determined according to your operating system's settings. To set a different language for Creo View from the operating system language, use the environment variables described below.

The generic environment variable, `LANG`, sets one language for all programs on your machine. You can use PTC-specific variables to designate another language for one or more programs:

-
- `PVIEW_LANG`—Creo View
 - `PRO_LANG`—Creo Parametric and its associated applications. When you set this variable, you must also set `PVIEW_LANG`. This is true even if you want the same value for `PRO_LANG` and `PVIEW_LANG`.

Modifying an Installation

If you installed Creo View from the DVD, all languages and features are already installed. If you installed the software from the downloaded `CreoView_32.exe` or `CreoView_64.exe` installers, it is good practice to use the installer to modify your installation as follows:

1. Start the installer. The **Creo View 3.0 Setup** dialog box opens to the welcome page.
2. Click **Next**. The **Change, repair, or remove installation** page opens.
3. Click **Change**. The **Product Features** page opens.
4. Add or remove features. See [Performing an Advanced Installation](#) for more information.
5. Click **Change**. The installation is changed.

Uninstalling Creo View

You can remove an installation of Creo View through the Control Panel as follows:

1. From the **Start** menu, click **Settings** ► **Control Panel**.
2. Double-click an icon according to your version of Windows:
 - **Programs and Features**—Windows 7
 - **Add/Remove Programs**—Earlier versions
3. In the program application list, select the application to remove.
4. Right-click and choose **Uninstall** from the shortcut menu. A confirmation appears.
5. Click **Yes** to proceed. Creo View is removed.

Note

You must separately remove the helper applications using the previous procedure.

7

Deploying Creo View and Customizing the Installation

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This chapter is for system administrators and applies to Windows only. It provides information on deployment, including example procedures for customizing the automated installation of the program.

Deploying Creo View Using a Silent Installation

Perform a silent installation in either of these ways:

- Deploy the program from the downloaded `CreoView_32.exe` or `CreoView_64.exe`—See [Deploying Creo View from a Downloaded Executable File](#) for an example.
- Extract the executable file and directly call `msiexec.exe` on some or all of the required installers—See [Customizing the Installer](#) for examples.

In the command shell, you can enter `msiexec /?` to read an explanation of all the generic MSI options. Some common examples of options for a silent installation follow:

- `/qn`—Installs the software with no installer user interface.
- `/qb`—Provides a progress bar with a **Cancel** button.
- `/qb!`—Provides only a progress bar. A user cannot stop the installation.

Generic Features for Creo View

Option Name	Feature
<code>main</code>	Core Creo View runtime. This feature is required.
<code>en</code>	Basic language resources. This feature is required.
<code>pvocxusr</code>	User-only ActiveX component that goes into the Program Files directory.  Note You need to enable the Adobe PDF Reader ActiveX Control Add-on from the Windchill Server to view Creo View viewables in standalone or regular mode.
<code>pvocxsys</code>	System-wide ActiveX component. This is required for standard users.
<code>plugin_acrobat</code>	Adobe Acrobat add-in program for marking up PDF files.
<code>doc</code>	CHM Help file. This feature is required.

Generic Features for Creo View (continued)

Option Name	Feature
licvar	Sets the license server variable. Must be used with the LICENSESERVER property.
demo	Sample data set.

File Import Filters

Option Name	Filter
dwg	2D DWG and DXF support. This feature is a prerequisite for pvifdwf.
pvifdwf	2D and 3D DWF and DWFX
pvimport	Basic set of files required for installing the filters listed below.
pvifiges	IGES
pvifstl	STL
pvifgbf	GBF
pvifvrl	VRML
pvifdgn	DGN
pvifstep	STEP
pvifppl	PGL

Language Features

Language	Option for User Interface	Option for Help
English—This feature is required.	en	doc
German	de	docde
Spanish	es	doces
French	fr	docfr
Italian	it	docit
Japanese	ja	docja
Korean	ko	docko
Russian	ru	docru
Portuguese—Brazil	pb	docpb
Chinese—Simplified	cs	doccs
Chinese—Traditional	ct	docct

Deploying Creo View from a Downloaded Executable File

To call a silent installation directly from `CreoView_64.exe`, add the `/v` argument to the program call immediately followed by all the parameters for the underlying MSI package. An example follows:

```
start /w CreoView_64.exe /vADDLOCAL="ALL"  
        LICENSESERVER="7788@licsrv.example.com" /qn
```

In the example above, the command is prefixed with `start /w` to force one process to finish before another one is started. This is useful for scripting installations. The feature `licvar` is included in the metafeature "ALL". Therefore, the property `LICENSESERVER` is sufficient.

Customizing the Installer

`CreoView_32.exe` and `CreoView_64.exe` are self-extracting archives that each run `CreoSetup.exe`. `CreoSetup.exe` is a wrapper application for chaining the installations of several installers in the archive, including `CreoView_32.msi` or `CreoView_64.msi`. Running the MSI file alone does not constitute a complete installation of Creo View. However, you can customize the installer in these ways:

- Remove applications installed by default
- Bundle other applications with the Creo View installation

Use 7-Zip to extract the archive to a folder, and then modify its contents. Continue to [Creating and Deploying a Customized Installer](#) for an example.

Creating and Deploying a Customized Installer

You can add and remove applications from the Creo View installation. For instance, the users in your organization have the following needs:

Required	Not Required
Using Creo View client	Microsoft Visual Studio C++ 2010 Runtime Redistributable
Working with JT (*.jtt) files	Viewing ECAD designs

In this case, you may want to customize the installer in these ways:

- Bundle the JT Import Filter with the Creo View installation
- Remove Microsoft Visual Studio C++ 2010 Runtime Redistributable because you have already deployed it. We will still deploy 2012 Runtime.

Accomplish the customization by performing two basic tasks:

1. Prepare the directory structure by adding and removing installer files.
2. Modify `CreoSetup.ini` to run the installers correctly.

After you complete these tasks, you can deploy the software. The procedures in the next sections contain instructions for completing the tasks above, and for and the deployment.

Preparing the Directory Structure

Follow the steps below to customize the installation directory.

1. Create an empty directory (<directory>).
2. Copy `CreoView_64.exe` into <directory>, and then right-click the file and choose **7-Zip ▸ Extract Here**. The setup files are extracted from the wrapper.
3. Delete `CreoView_64.exe`. These subdirectories and files remain:
 - `\prereqs` subdirectory
 - `\pview` subdirectory
 - `CreoSetup.exe`
 - `CreoSetup.ini`
4. Create a new subdirectory called `\jtimport`.
5. Navigate to the installation subdirectory on the JT DVD image:
`cdimages/jtadapter/installers/`
Copy `CreoView_JT_Import_32.msi` and `CreoView_JT_Import_64.msi`, and then paste them into the `\jtimport` subdirectory you created in step 4.
6. Under the `\prereqs` subdirectory, remove the two Microsoft Visual Studio 2010 C++ Redistributable x86 and Microsoft Visual Studio 2010 C++ Redistributable x64. They are called `vc_redist_x86_VS2010.exe` and `vc_redist_x64_VS2010.exe`.

The directory is now ready. Continue to the next section to customize the INI file.

Modifying `CreoSetup.ini`

You have added an installer file to bundle the JT Import Filter with the Creo View installation. You must now modify `CreoSetup.ini` to make these changes to the default installation directives. You can delete unnecessary lines and sections from the file, or you can modify them. In this example, they are modified.

- Add new directives to run the JT installer
- Remove or disable the `vc_redist_x##_VS2010.exe` directives.

Follow the next steps to update the INI file accordingly.

1. Open `CreoSetup.ini` in a text editor.
2. In the `[Other Applications]` section, find the line `VC2010_X86=YES` and replace it with `VC2010_X86=NO`, or `VC2010_X64=YES` and replace it with `VC2010_X64=NO`.
3. At the end of the `[Other Applications]` section, find these lines:
`ECADADAPTERS_X86=YES`
`ECADADAPTERS_X64=YES`

Modify them to call the JT installer instead of the Creo View ECAD Adapters installer:

```
JTIMPORT_X86=YES  
JTIMPORT_X64=YES
```

4. At the end of the file, find the sections for the Creo View ECAD Adapters, such as the example below:

```
[ECADADAPTERS_X86]  
PROGRAM=CreoView_ECAD_Adapters_32.exe  
COMMANDLINE=/f /vADDLOCAL="ALL" REBOOT="ReallySuppress" /l*v  
TempFolder]pvinstecadadapters.log" /qb!  
LOCATION=..\ecad_adapters  
; PRODUCTCODE=  
PLATFORM=X86
```

Change the heading and information in the `[ECADADAPTERS_X86]` and `[ECADADAPTERS_X64]` sections to information for the JT Import Filter, as shown in these examples:

```
[JTIMPORT_X86]  
PROGRAM=CreoView_JT_Import_Filter_32.msi  
COMMANDLINE=ADDLOCAL="ALL" REBOOT="ReallySuppress" /l*v  
"[TempFolder]pvinstjt.log" /qb!  
LOCATION=jtimport  
PRODUCTCODE={7D39690A-FB6B-4559-B1FD-26F7735162B3}  
PLATFORM=X86  
[JTIMPORT_X64]  
PROGRAM=CreoView_JT_Import_Filter_64.msi  
COMMANDLINE=ADDLOCAL="ALL" REBOOT="ReallySuppress" /l*v  
"[TempFolder]pvinstjt.log" /qb!  
LOCATION=jtimport  
PRODUCTCODE={E6438BA9-7C4D-4D54-A87D-65272F3169A6}  
PLATFORM=X64
```

5. Save `CreoSetup.ini` and close the text editor.

You have now defined the installation in the INI file:

-
- For more information on locating the code for an application, continue to the next section, *Finding the Product Code*.
 - For more information on the INI file, skip to the section *Understanding the CreoSetup.ini File*.
 - To begin deployment, skip to the section *Deploying with the Customized Installer* below.

Finding the Product Code

This example uses the JT ProductCode for Creo 2.0 M020. Each release has a unique ProductCode. An application also has a permanent UpgradeCode. The UpgradeCode is not used by CreoSetup.exe because it does not identify the version of the application. You can find the product code for the applications to install by using the Registry Editor. On a computer where the application is already installed, navigate to the registry key.

Caution

Do not modify the registry. Doing so can severely damage your computer.

1. Open a command prompt, and then type `regedit`.
2. Press ENTER. The **Registry Editor** opens.
3. On the left, navigate to this key:
`HKEY_LOCAL_MACHINE\SOFTWARE\Microsoft\Windows\CurrentVersion\Uninstall`
4. Find the application to install, and copy the key, including the curly brackets. These are examples of the registry keys for Creo View 2.0 M020 JT Import Filters:
 - 32-bit—{7D39690A-FB6B-4559-B1FD-26F7735162B3}
 - 64-bit—{E6438BA9-7C4D-4D54-A87D-65272F3169A6}

Understanding the CreoSetup.ini File

Each application in the file's [Other Applications] section has a YES or NO value. A YES value does not necessarily mean the application's installer will always be executed. Instead, each application with a YES value is a potential installation. If your system has an installation that matches the product code line, then nothing is done. If your system does not have a matching installation, the application is installed.

The PRODUCTCODE line for each application is optional. In all cases, the application is installed automatically when your system does not have the same version installed. Providing a product code minimizes the time the installation takes to complete.

The value for the PROGRAM line must be a *.exe file or a *.msi file.

Deploying Creo View with the Customized Installer

You have now prepared the directory and modified the INI file. In this example, deploy the applications with a passive installation. You may use one or both of these strategies:

- Run `CreoSetup.exe` directly from the network location.
- Create a new, self-extracting archive from the modified directory. Make sure the self-extractor calls `CreoSetup.exe` to chain all the installations to deploy.

The call to `CreoSetup.exe` must include, in the command line, all MSI arguments for the base Creo View installer. At this time, you cannot embed the Creo View MSI arguments directly in `CreoSetup.ini`.

To perform the passive installation, open a command shell and enter the following command:

```
CreoSetup.exe /vADDLOCAL="ALL" APPLICATIONFOLDER="C:\  
ptc\Creo View" LICENSESERVER="7788@licsrv.example.com"  
/qb!
```

Note

Make sure there is no space between `/v` and the next character in the command.

Creo View and the JT Import Filter are installed with a progress bar, but no **Cancel** button. For more information on common examples of options for a silent installation, see [Deploying Creo View Using a Silent Installation](#).

Performing a Silent Uninstallation

MSI installers automatically uninstall a matching product as part of an upgrade. Therefore, for upgrades it is unnecessary to perform a silent uninstallation. However, if you do want to manually and silently uninstall one or more of the Creo View applications, you must first obtain the ProductCode for each version of the application to uninstall. New ProductCodes are generated for each release and patch build of Creo View. This section contains instructions for finding the ProductCodes and for using them to perform the silent uninstallation.

Obtaining ProductCodes

To query the MSI inventory and retrieve the ProductCodes, you can use any of these methods:

-
- Visual Basic script
 - Powershell script
 - Free tool such as `msiinv.exe`

For example, execute the `msiinv.exe` with these arguments:

```
msiinv.exe -p > %TEMP%\msiinv_output.txt
```

You now have a list of all MSI packages and their ProductCodes in your `%TEMP%` directory, in the file `msiinv_output.txt`. You can uninstall the applications. Continue below.

Uninstalling the Applications

Use the `msiexec.exe` tool to uninstall one or more Creo View applications. For each application to uninstall, you must run `msiexec.exe` with the corresponding ProductCode. Instructions follow.

1. Note the ProductCode you retrieved in the previous procedure.
2. Open a command window, and type the following command:
`msiexec.exe /x {<ProductCode>}`
3. Press ENTER. The command is executed.

You have now manually and silently uninstalled the application. Repeat the procedure above for each application to uninstall.

8

Updating an Installation

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This chapter explains how to update your copy of Creo View. Each time you update your software to a new release, you must update your license file. New license codes are not required if you are updating to a maintenance release of Creo View within the same release. In some cases you may have to update the current installation of PTC License Server. See the next sections for details.

Updating the License Server

You can update the current installation of PTC License Server with your new license information when updating your PTC software. An installation update of PTC License Server is typically required in one of these scenarios:

- Adding license information from your new Sales Order
- Updating the existing license features in your license file with those of a maintenance release

You may have to update the license server software itself depending on the version installed. In such cases, you must uninstall PTC License Server using administrative privileges and then follow the installation instructions in the *Standalone PTC License Server* guide on the [Reference Documents](#) page.

Update your licenses using this workflow:

1. Make note of the product code that you have received via e-mail. Alternatively, use the PTC License Management Web tools to request for a license file via e-mail. Save the license file in an ASCII format to a secure location on your disk.
2. Navigate to the `\bin` subfolder of your license server installation. The default path follows:

```
C:\Program Files\FLEXnet Admin License Server\bin
```
3. Right-click `ptcsetup.bat` and choose **Run as Administrator** from the shortcut menu. The **PTC Setup — PTC License Server** dialog box opens.
4. Click `.`. The **Open** dialog box opens.
5. Browse to your license file, and then click **Open**. The path appears under **License File**.
6. Click **Configure**. The **Installation Progress** page opens with a progress bar.
7. A warning opens. Click **OK**.
8. To check the status of your updated license, follow these steps:
 - a. Click **Program Files** ► **PTC** ► **PTC FLEXnet Admin License Server**. The **FLEXnet License Administrator** webpage opens.
 - b. Near the top of the page, click **Administration**. The **Administration** tab opens.
 - c. On the left, click **Vendor Daemon Configuration**. The **Vendor Daemon Configuration** tab opens with your license.
 - d. Under the **Status** column, verify that the status is **RUNNING**.

PTC License Server restarts. For Triad configurations, two of the three partner machines must be running before licenses can be distributed.

Updating Creo View

After you install a particular release of Creo View, you can install a new maintenance release. The previous release is overwritten.

9

Hierarchical Visual Navigation

You can rapidly load the approximate 3D model graphics for very large Creo View structures by loading 3D thumbnail image (PVT) files of the structures that were previously created. Once the thumbnail is loaded you can switch to the full 3D model view (OL file). You can also revert from the full 3D model view back to the 3D thumbnail image.

This functionality is available only under the following conditions:

- 3D thumbnail images have been generated for sub-level assemblies. This requires configuration of the CAD worker or the server-side Thumbnail Generator to create them. See **Enabling Hierarchical Visual Navigation** in the *Creo View MCAD Adapters 3.0 Installation and Configuration Guide*.
- Creo View options are configured to download and view thumbnails. See **About Thumbnails** in the *Creo View 3.0 Help*.

Note

For more information read the Technical Support Knowledge Base Article [CS25318](#), which covers the complete configuration across all products.

A

License Management Software

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This appendix discusses license management software and describes the benefits of PTC License Server. License simplification is explained for new and existing users.

Overview of PTC Licensing

PTC software including PTC optional applications must be licensed for use. Licensing authorizes the installed software to run. Creo View licenses are not time-sensitive.

License Types

Depending on the PTC product, a license can be one of the following types.

- Node-locked licenses—Restricts the use of the software to a particular machine (independent workstation).
- Floating licenses—Served by the license server and can be used on any machine connected over the network. There are two different types of floating licenses:
 - Single server licenses—Configured for a single machine as the server.
 - Triad licenses—Configured for a set of three machines on the same network that work together to manage a pool of floating licenses. This configuration provides redundant backup in case of a server outage. An identical license file is used for each Triad partner.
- Extended license—An extended license makes floating licenses available for locked modules.
- Borrowed license—A borrowed license allows you to work temporarily on your machine without being connected to the license server. Refer to [Borrowing and Returning Licenses](#) for details.

For more information on the additional license types that are supported for your product, refer to the product documentation.

PTC License Server

Flexera Software, Inc.'s FlexNet Publisher license management software is integrated with PTC software. For more information, visit www.flexerasoftware.com.

Benefits of Using FlexNet Publisher

Using FlexNet Publisher to control usage of licenses offers the following key advantages:

- Single-Server Solution—FlexNet Publisher can manage PTC software and other vendor applications without conflict. Note that the PTC license file cannot be combined with the license files of other vendors.
- Immediate License Recovery—If there is a premature exit of the licensed software (for example, the system shuts down), the FlexNet Publisher license server automatically reclaims the PTC license.

-
- **Increased Flexibility**—System administrators can reserve or deny licenses based on user name, host name, display name, or IP address. For more information see *Managing the Options File* in the *FlexNet Publisher License Administration Guide*.
 - **Centralized License Storage**—PTC customers can store all PTC licenses for all PTC products in a single file for each license server.
 - **Multiple Licenses for a Single Command**—One command can be used to execute multiple licenses of PTC software based on availability.
 - **License Borrowing**—Using FlexNet Publisher 8.6 or later, you can borrow licenses from a license server and run a licensed application on a remote client without being connected to the license server.

Downward License Compatibility Using FlexNet Publisher

FlexNet Publisher license servers have downward compatibility with PTC applications using FLEXlm license servers. For example, a Creo View 2.0 license can run ProductView 9.1 software, because both releases of the software use the same PTC License Server.

Note

To run the current release of Creo View, you must install FlexNet Publisher version 10.8.x or later.

Running FlexNet Publisher with Other Applications

You can use FlexNet Publisher to run PTC products as well as other applications. Each application that uses FLEXlm or FlexNet Publisher has a corresponding license server manager (`lmgrd` or `lmadmin`) and a vendor daemon. The license server manager starts the vendor daemon (for example, `ptc_d`) that controls the usage of licenses.

You cannot combine a PTC license file with a license file of another vendor. Do not, for example, combine PTC feature lines with those of other vendors in a single license file. This action causes the licenses to be invalid.

If all applications and vendor daemons are FLEXlm 6.1 or later, `lmgrd` or `lmadmin` can process multiple license files. This is true even when the Host IDs are different (as long as the license files refer to the same node). For more information, see *Managing Licenses from Multiple Software Publishers* in the *FlexNet Publisher License Administration Guide*.

Understanding Timeout Parameters

Timeout parameters enable the license client and the license server to communicate with one another so that licenses are released and available upon request.

You can reconfigure the `TIMEOUTALL` parameter within a specified range, as described in the next section, [Changing the Inactivity Timeout Parameter](#).

Timeout Parameter	Value	Description
Inactivity timeout (TIMEOUTALL)	120 minutes (default)	<p>This parameter prevents a license from remaining idle. If the license client is inactive for a specific period, the license can be reclaimed by the license server and used by another license client. For the Creo applications, the inactivity timeout default is 120 minutes.</p> <p>Activity is measured as active menu selections.</p> <p>You can change the default of 120 minutes so that the parameter ranges from 20 minutes (minimum value) to infinity (maximum value).</p>
License refresh	1 minute	<p>A license refresh occurs at intervals of 1 minute. When you select a command after such an interval, the license client communicates with the license server. The license client and the license server both must be working. If the license server is not found, the license client loses its license to run. If the license client is not found, the server reclaims the license for use by another license client.</p>

Timeout Parameter	Value	Description
Validation retries	None	The first time a license client cannot validate its license the application's graphical user interface will freeze. You are immediately given the following three options: Click Retry to request a license from an active license server. Click Save File(s) . Click Exit to close the application without saving the file or files.
Startup	10 seconds	Upon starting a session, the license client requests a license and has 10 seconds in which to have the request validated.

Changing the Inactivity Timeout Parameter

You can set the `TIMEOUTALL` parameter that determines how long the system allows a license to remain idle before it is reclaimed by the license server. To change the default inactivity timeout parameter, you must update the FlexNet Publisher option file, `ptc.opt` in `<ptc License Server loadpoint>/FLEXnet Publisher/licensing`. The default is 120 minutes. Edit the default parameter:

```
TIMEOUTALL 7200
```

Change 7200 seconds (120 minutes) to another value in seconds.

The minimum value is 20 minutes (or 1200 seconds) and the maximum value is infinity. To make infinity the default, remove the `TIMEOUTALL` parameter from the `ptc.opt` file. If you set a minimum value to less than 20 minutes, the system defaults to 20 minutes.

B

ladmin License Server Manager

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This appendix contains basic information on `ladmin`, a Web-based license server manager. It also provides information on migrating from `lmgrd` to `ladmin`.

Overview of lmadmin as a GUI-Client

The `lmadmin` license server manager supports a Graphical User Interface (GUI) client with connection over HTTP. It provides a Web-based administrative interface. It is compatible with license files and vendor daemons created with FlexNet Publisher 9.2 and later. Use `lmadmin` for the following functions:

- Perform server configurations and administration functions
- Start the license server manager without any configuration options
- Directly configure vendor daemon and the license server manager without editing the license files
- Import existing license files
- Support multiple vendor daemons with a single `lmadmin` process
- Display license rights status
- Display alerts and monitor status of various processes, such as license expiration or unavailable licenses

See the chapter `lmadmin` – GUI-based License Server Manager in the *FlexNet Publisher License Administration Guide* for more information.

Differences between lmgrd and lmadmin

The `lmadmin` license server manager includes all the features of the `lmgrd` license server manager. The differences between the two license server managers follow:

lmgrd	lmadmin
Uses a command-line interface.	Supports a GUI client with connection over HTTP.
Configuration settings are retained for all the sessions.	Configuration settings are session-based.

See the chapter Migrating from `lmgrd` to `lmadmin` in the *FlexNet Publisher License Administration Guide* for more information.

Installing PTC License Server Based on lmadmin

Follow the instructions given in the section [Installing the PTC License Server](#) to install PTC License Server based on `lmadmin`. After you install the license server, you can configure the license server administration settings using the FLEXnet License Administrator Web interface.

Working with the FLEXnet License Administrator Web Interface

The `lmadmin`-based license server uses the FLEXnet License Administrator Web interface. This interface replaces the `lmtools` utility used by the `lmgrd`-based license server. You can launch the interface by clicking **Yes** at the end of the PTC License Server installation. Alternatively, from the Windows Start menu click **Program Files > PTC > PTC FLEXnet Admin License Server > PTC FLEXnet Admin License Server Web Interface** to open the interface. Using the FLEXnet License Administrator, you can check the server status, start and stop the server, or reconfigure the server.

The FLEXnet License Administrator Web interface has two main pages: the **Dashboard** and the **Administration** pages. Click the **Help** button for information on the FLEXnet License Administrator interface elements.

- **Dashboard**—Displays any alerts and the current activity of the license server manager.
- **Administration**—Provides configuration tools for the license server management system. The default username/password combination is `admin/admin` for this password-protected page. You are prompted to change these credentials the first time you log in. Only the overview information for the **Server Configuration** and the **Vendor Daemon Configuration** tabs on the **Administration** page is included in this document. Click a tab to open the corresponding pages.

Controlling the License Server Manager Settings

A server administrator can use the options on the **Server Configuration** page to control the settings for the License Server Manager.

Reconfiguring the Vendor Daemon

A server administrator can use the **Vendor Daemon Configuration** tab to verify whether the server is running and to reconfigure the vendor daemon.

If PTC License Server has been successfully installed, `Running` appears under the **Status** column.

C

Troubleshooting Tips for Licensing and Creo View Runtime

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This appendix documents common problems that occur when you install Creo View and PTC License Server. This appendix also provides general debugging techniques and cites other sources of information available from the PTC website.

Troubleshooting List

Skim through the following list of problems to find any that appear to be the same as the one you are experiencing. The information is presented in the following format.

Symptom: Describes the problem.

Cause: Lists the cause of the problem.

Solution: Provides steps to resolve the problem.

Cannot Connect to the License Server

Symptom: When you start Creo View, you receive an error message in the **Startup** dialog box.

Cause: The license server is not running or the license client cannot reach the server.

Solution: Verify that the license server manager and `ptc_d` daemons are running. A network problem exists if a license client attempts to ping the server by host name and the ping fails.

Invalid Licenses

Symptom: You receive the error message `Invalid license`.

Cause: Licensing information is inaccurate.

Solution: Return to the **FLEXnet license editor** and verify that the information entered is exactly as it appears in your License Pack. If no licenses are listed, return to the **FLEXnet license editor** and ensure no text is highlighted.

If all licenses are listed as `invalid`, verify that the `PTC Host_ID` in the License Pack corresponds with what you see in the **FLEXnet license editor**. For example, one server line and one daemon line represent a single server. Three server lines and one daemon line represent a fault-tolerant or Triad configuration. Remove all the lines that do not pertain to the `PTC HOST_ID`.

Your incremental lines must have no blank lines. Verify that all continuation characters (`\`) are at the end of each line, except for the last line. If some licenses are valid while others are invalid, find the invalid feature name lines in the License File window and correct the text.

If you received your license codes via e-mail, remove any extraneous text such as the header and footer. Another option is to delete the invalid license in the **FLEXnet license editor** window.

FlexNet Publisher Fails to Start (Triad Configurations)

Symptom: PTC License Server does not start after a Triad server is installed and configured.

Cause: The following requirement has not been met: two of the three partners of the Triad configuration must be running (Quorum) before licenses can be distributed.

Solution: Go to a shell window or a command prompt and change into the `<FLEXnet_Installation_Directory>\bin`. Type in `ptcstartserver`.

General Debugging Hints

The `ptc.log` file records license server activities and can be found in “`\FLEXnet Admin License Server\logs\`”. This file has useful information that you should check when you have a problem.

Setting Variables and Installing Tools for Troubleshooting

The Creo View installation includes software for troubleshooting. Install the software and set troubleshooting environment variables as follows.

Installing Creo View Debug Symbols

Creo View Debug Symbols is a bundle of files for troubleshooting all Creo View and Creo Illustrate programs. If Creo View crashes, the detailed information is gathered by the configured trace file instead of by the Windows default mechanism. Instructions for installing Debug Symbols follow.

1. Navigate to `<installer_directory>\installers\debug_symbols`.
2. Double-click the installer for your platform:
 - `CreoView_Client_Debug_Symbols_32.exe`
 - `CreoView_Client_Debug_Symbols_64.exe`

The **Creo View Debug Symbols Setup** dialog box opens to the **PTC Customer Agreement** page.

3. Read the Terms and Conditions, and then select **I accept the terms in the License Agreement**.
4. Click **Install**. Debug Symbols is installed.
5. To read additional instructions required to start using Debug Symbols, select **View the Creo View Debug Symbols usage instructions**.
6. Click **Finish**.

You must set the corresponding environment variable to activate Debug Symbols. See the next section, Using Variables in Troubleshooting.

Using Variables in Troubleshooting

To aid in troubleshooting the runtime of Creo View, use these environment variables:

- `PVIEW_PROC_TRACE=<full or relative file path>\pvtrace.log`—Generates a unique log file for each Creo View process running. The log file saved to the path you set is called `pvtrace_<process number>.log`.
- `PVIEW_CATCH_EXCEPTIONS=1`—Works with Debug Symbols to gather troubleshooting information. The information is stored in the trace file defined immediately above. When you install Debug Symbols, you must set this variable.

Online Information

See www.ptc.com/support/index.htm for a wealth of how-to information for new and experienced users, including order and license support. The Customer Support Guide provides online tools and other support services. You can search the Knowledge Base of over 14,000 technical documents or download reference documents, including the Read This First.

Customer Support Guide and **Contact Support** are available at the [My eSupport](#) page. For information on FlexNet Publisher diagnostic environment variables, consult the *FlexNet Publisher License Administration Guide*.

D

Managing Preferences

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This appendix provides an overview of Creo View preferences, including disabling default commands. It describes the locations for preference files and their hierarchy. Instructions for locking preferences and setting server preferences are also included.

Overview of Creo View Preferences

This section contains an overview of the types of preferences you can set for Creo View. It also provides information for locking preferences.

Types of Preferences

There are five types of preferences. Each type is read from an XML file:

Type	File Name	Location
Default	ProductView_prefs.xml	Read from <Creo View>\preferences\ProductView\subdirectory of the installation directory.  Note A preference sample file is located in <Creo View>\resources\ProductView\preferences\ProductView_pref.xml
Server	server_prefs.xml	Downloaded from a WVS Configuration from the Windchill server context hierarchy. The server preferences are only active when you are connected to Windchill.
Shared	shared_pref.xml	Downloaded from a shared location, to <Creo View>\preferences\ProductView\shared_prefs subdirectory of the installation directory. Once enabled, the shared preference file is updated automatically. The shared

Type	File Name	Location
		preference file is only active when you are not connected to Windchill.
Administrator	admin_prefs.xml	Read from <Creo View>\preferences\ProductView\ subdirectory of the installation directory
User	user_prefs.xml	Read from the user's profile: <ul style="list-style-type: none"> • Windows —%APPDATA%/ptc/ProductView/user_prefs.xml • UNIX—\$HOME/.ptc/ProductView/user_prefs.xml

The files are processed in the order listed above. You can override a value in a preference file with a different value in a subsequent file:

- A user preference replaces the setting for a preference in the administrative file.
- An administrative preference replaces the setting for a preference in the server file. If a shared preference file exists and is active, the administrative preference replaces the shared preference.

 **Note**

The server preference file and the shared preference file cannot coexist.

For example, the administrator disables the color highlighting command by defining the `highlight_using_color` preference in the `admin_prefs.xml` file, as shown:

```
<?xml version="1.0" encoding="utf-8"?>
<preferences>
  <category name="General">
    <subcategory name="Navigation" >
```

```
        <preference1 name="highlight_using_color" value="False"/>
    </subcategory>
</category>
</preferences>
```

However, if the user explicitly enables the same preference in the user interface, the setting is saved to the `user_prefs.xml` file. As a result, the administrator preference is overridden and the color highlighting command is enabled. To prevent a user from overriding the administrator preferences, lock the administrator preferences. Continue to the next section, *Locking Preferences*, for more information.

Locking Preferences

At any level, you can lock a preference. A locked preference cannot be modified or overridden by a file on a lower level. Using the color highlighting example from the previous section, you can add the attribute `locked="True"` to prevent the user from changing the preference:

```
<?xml version="1.0" encoding="utf-8"?>
<preferences>
    <category name="General">
        <subcategory name="Navigation" >
            <preference1 name="highlight_using_color" value="False" locked="True"/>
        </subcategory>
    </category>
</preferences>
```

The color highlighting command is now disabled and locked.

Setting Preferences

Set the user preferences using the **Creo View Options** dialog box in the Creo View client, and then click **OK**. For more information, see the Creo View Help.

Set the administrator preferences, server preferences, or both by basing them on one of these files:

- User preferences
- Default preferences

Instructions for both methods follow.

Basing the Administrator or Server Preferences on the User Preference File

1. In the Creo View client, click **File** ► **Options**. The **Creo View Options** dialog box opens.
2. Set one or more preferences, click **OK**, and then close Creo View. The changes are saved to the `user_prefs.xml` file.
3. Open these files in a text editor:

-
- `user_prefs.xml`
 - `admin_prefs.xml` or `server_prefs.xml`
4. In `user_prefs.xml`, find and copy the entries for the preferences you set in step 2.
 5. Paste the entries in `admin_prefs.xml` or `server_prefs.xml`.

 **Note**

Make sure the XML structure hierarchy is maintained.

6. To lock one or more preferences, add the attribute `locked="True"` to the definition of those preferences.

 **Note**

The `locked` attribute is not supported in the `user_prefs.xml`.

7. Save and close `admin_prefs.xml` or `server_prefs.xml`.

Basing the Administrator or Server Preferences on the Default Preference File

1. Make a copy of the default preferences file, `ProductView_prefs.xml`.
2. Open `ProductView_prefs.xml` in a text editor.
3. Modify one or more preferences.
4. To lock one or more preferences, add the attribute `locked="True"` to the definition of those preferences.
5. Save the file as `admin_prefs.xml` or `server_prefs.xml`, and then close it. Save the file as `admin_prefs.xml`.

You have now set and locked the administrator or server preferences:

- For information about using the preference file to disable default commands, continue to the next section, [Disabling Commands](#).
- To implement your server preferences, skip to [Uploading Server Preferences to Windchill](#).

Disabling Commands

Each control in the user interface has a unique command. For example, the command for the **Print** button is `PrintCmd`. You can use a preference file to disable access from the user interface to one or more commands on buttons or ribbons. You cannot disable commands on shortcut menus. Instructions for finding and disabling a command follow.

Finding a Command

1. To find a command for a button, set the following environment variable:
`set PVIEW_DISPLAY_COMMANDS=1`
2. Open Creo View, and then place the pointer over the button. A tooltip appears with the command. For example, `PrintCmd`.

Using the Preference File to Disable the Command

Follow the instructions in this procedure to disable the **Print** button.

1. Open `admin_prefs.xml` or `server_prefs.xml` in a text editor.
2. At the end of the file, add the following section:

```
<preferences>
  <category name="General">
    <subcategory name="UI_Config">
      <preference name="disabled_commands" value="PrintCmd"/>
    </subcategory>
  </category>
</preferences>
```

3. To disable more than one command, list them, separated by commas, as the value for `disabled_commands`.
4. Save `admin_prefs.xml` or `server_prefs.xml` and close the file.

The command is now disabled. When all of the preference files are read, the values in each `disabled_commands` section are cumulatively disabled.

Some server preference options cannot be configured using the preference editor, and they only apply when Creo View is embedded in a web page. This option disables the automatic unlock of an annotation set when closed:

```
<?xml version="1.0" encoding="utf-8"?>
<preferences>
  <category name="Annotations">
    <preference name="disable_auto_lock" value="True"/>
  </category>
</preferences>
```

Uploading Server Preferences to Windchill

By following the steps in the previous section, you have now created a server preference file. Follow the next steps to upload the preferences to your Windchill server:

1. On the Windchill home page, on the left, click **Browse**. The left frame opens.
2. Select the **Browse** tab, and then click the **Site** button.
3. Under **Site**, click **Utilities**. The Utilities page opens.
4. Under **Business Administration**, click **Visualization Configuration Administration**. The **Creo View Configurations for Site** window opens.
5. Click **Create configuration**, and then upload `server_prefs.xml`.
6. To the same WVS Configuration, upload any files referenced in `server_prefs.xml`, such as these:
 - Watermarks
 - *.ini
 - HPGL pen maps
7. Click **Close**. Your preferences are uploaded. For more information on uploading preferences, see the Windchill Help.

You have now implemented your server preferences on Windchill.

Loading Preferences from a Shared Location

You can centrally manage Creo View preferences. The preferences are stored in a shared location, and they are copied to the user's computer. Shared preferences are available even when the user disconnects from the shared location. The shared preferences are saved as a ZIP file that contains `shared_prefs.xml` and any other required files. Users can load the shared preferences from one of these locations. The locations are checked in the order below, and the preferences are loaded only from the first location where they are found:

1. The **HKEY_LOCAL_MACHINE** registry entry
2. The **HKEY_CURRENT_USER** registry entry
3. The `PVIEW_SHARED_PREFERENCES` environment variable

Configuring Shared Preferences for Cadence Allegro and APD

This section describes how you can open Cadence Allegro and APD design files with no local Allegro installation.

To configure shared preferences for Cadence Allegro and APD follow the steps below:

1. In the Creo View client, click **File** ► **Options**. The Creo View **Options** dialog box opens.
2. Set one or more preferences and then click **OK**. The changes are saved to the `user_prefs.xml` file.
3. Open the `user_prefs.xml` file in a text editor. Locate the entries for the preferences you set in step 2 and copy them into a new file.
4. Save this file as `admin_prefs.xml`, `server_prefs.xml`, and `shared_prefs.xml`.

 **Note**

The Cadence Allegro extracta file (`.acceptedAllegroExtracta`), must be kept in the same location as the `admin_prefs.xml`, `server_prefs.xml` and `shared_prefs.xml`.

5. To lock one or more preferences, add the attribute `locked="True"` to the definition of those preferences.

 **Note**

The `locked` attribute is not supported in the `user_prefs.xml`.

6. Save and close the `user_prefs.xml` and `server_prefs.xml` files. The saved file is copied to the `APPDATA%` directory and automatically loaded into the `shared_prefs.xml` folder.

Shared Location Types

These are examples of the two types of shared locations:

- File system—`s:\shared_prefs\shared_prefs.zip`
- Shared host—`\\sharedhost\shared_prefs\shared_prefs.zip`

Defining the Location of Shared Preferences Using the Registry

This section contains instructions on how to define in the registry the location for shared preferences.

To define the location using an environment variable instead, skip ahead to the next section, [Defining the Location of Shared Preferences Using an Environment Variable](#).

Click **Start** and in the search box, type `Regedit.exe` and press **ENTER**. The **Registry Editor** page opens. Then, perform one of the operations below:

- List the shared preference file in the local machine registry:
 1. Go to **HKEY_LOCAL_MACHINE** ► **SOFTWARE** ► **PTC** ► **ProductView Client**.
 2. Right-click and choose **New** ► **Key**. A new key is added under **ProductView Client**.
 3. On the left pane, under **ProductView Client**, rename the new key to `shared_preferences` and press **ENTER**. The **shared_preferences** key expands.
 4. Right-click and choose **New** ► **String Value**.
 5. Name the new value `file`.
 6. Double-click **file** and under **Value data** type the name of the shared preference file, for example `\\sharedhost\shared_prefs\shared_prefs.zip`.
 7. Click **OK**.
- List the shared preference file in the current user registry:
 1. Go to **HKEY_CURRENT_USER** ► **SOFTWARE** ► **PTC** ► **ProductView Client**.
 2. Right-click and choose **New** ► **Key**. A new key is added under **ProductView Client**.
 3. On the left pane, under **ProductView Client**, rename the new key to `shared_preferences` and press **ENTER**. The **shared_preferences** key expands.
 4. Right-click and choose **New** ► **String Value**.
 5. Name the new value `file`.
 6. Double-click **file** and under **Value data**, type the name of the shared preference file. For example, `s:\shared_prefs\shared_prefs.zip`.
 7. Click **OK**.

You have now defined a registry key to point to the shared preference file. Skip to [Loading the Shared Preference File](#) for more information about the next steps.

Defining the Location of Shared Preferences Using an Environment Variable

This section contains the procedure for creating an environment variable that points to the shared preference file.

-
1. Click **Start**, and in the search box, type `environment` and press **ENTER**. The **Environment Variables** page opens.
 2. Under **User variables for <your user name>**, click **New**. The **New User Variable** dialog box opens.
 3. Under **Variable name**, type `PVIEW_SHARED_PREFERENCES`.
 4. Under **Variable value**, type the name of the shared preference file, for example `\\sharedhost\shared_prefs\shared_prefs.zip`, and click **OK**.
 5. In the **Environment Variables** dialog box, click **OK** to close the dialog box.

You have now set an environment variable to point to the shared preference file. Continue to the next section, [Loading the Shared Preference File](#)

Loading the Shared Preference File

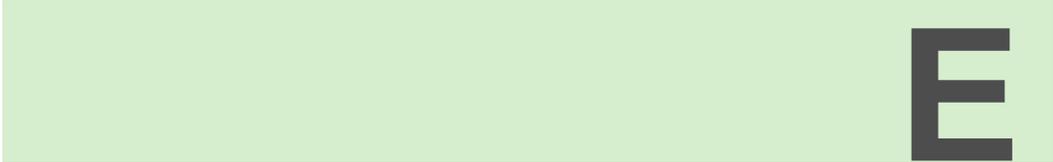
After you configure shared preferences, the preferences are automatically loaded and updated in Creo View. The shared file is copied to the user's `%APPDATA%` directory. It is placed in the designated folder, `shared_prefs`. When the user starts Creo View, the local copy and the shared file are compared and the local file is updated. For example, if a user has manually modified the XML file for the preferences, these changes are overwritten by the shared preferences file.

If the shared file is missing from the `%APPDATA%` directory, and the shared location is not found, the user cannot start Creo View at all. To fix the problem, do one of these operations:

- Reconnect to the shared location
- Remove the registry setting or environment variable that points to the shared location

Disabling Shared Preferences

To disable shared preferences, delete the environment variable or the registry entries you added in [Defining the Location of Shared Preferences Using the Registry](#) or [Defining the Location of Shared Preferences Using an Environment Variable](#).



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Customizing the Ribbon

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This appendix provides instructions to configure and customize the Creo View ribbon commands.

Overview of Ribbon Customizing

This section contains an overview of customizing the availability of Creo View ribbon commands.

Configure which Creo View user interface commands are available and save them in the Creo UI Toolkit *.ui files.

Setting Preferences

The customized user interface definitions are saved in proprietary Creo UI Toolkit files (*.ui) created in the %APPDATA%\PTC\ProductView folder. There are two customization files: an optional administrator file and the user file. The administrator file is loaded first. It defines the baseline for user customization.

Set the user preferences using the **Customize Ribbon** page. Access the dialog box from the **Creo View Options** in the Creo View client. For more information, see About Customizing the Ribbon in the Creo View Help.

Customizing and Saving the Ribbon Locally

1. In Windchill, click **Product** ► **Utilities** ► **Visualization Configuration Administration** and upload the `uiconfig.xml` file and the referenced image files. (These can have PNG, BMP, JPG, or JPEG format extensions.)
2. Open a file in Creo View.
3. Use the **Customize Ribbon** page of the **Creo View Options** dialog box to make changes to the ribbon.
4. Click **Apply**, and then click **OK** to add the newly created menu.
5. The new menu appears in the ribbon menu and the customized commands appear in that menu.
6. Exit Creo View and navigate to the %APPDATA%\PTC\ProductView folder. Note that the following files are present in that directory:
 - `user_prefs.xml`
 - `user_cust_pvpro.ui`
7. Rename the above two files as follows:
 - `server_prefs.xml`
 - `server_cust_pvpro.ui`
8. Edit `server_prefs.xml` and add the following preference:

```
<?xml version="1.0" encoding="utf-8"?>
<preferences>
<category name="GUI">
<category name="Customize">
<preference name="Location" value="server"/>
```

```
</category>
</category>
<preferences>
```

9. Save and exit the editor.

10. Zip these files:

- `uiconfig.xml`
- `server_prefs.xml`
- `server_cust_pvpro.ui`
- The referenced image files

You have now customized the ribbon menu. To implement your preferences, continue to [Uploading the Customized Files to Windchill](#).

Uploading the Customized Files to Windchill

By following the steps in the previous section, you have created a customized ribbon file. Follow the next steps to upload the preferences to your Windchill server.

1. In Windchill, navigate to the **Product ► Utilities ► Visualization Configuration Administration** page.
2. Click **Create configuration**, and then upload the zip files you created.

Note

If there are previously uploaded `uiconfig.xml` and referenced image files, delete them before uploading.

3. Clear the Creo View cache (`%APPDATA%\PTC\ProductView`).
4. Open a structure in Creo View from Windchill to validate that the customized menu appears in the ribbon menu.
5. Click **Close**. Your preferences are uploaded. For more information on uploading preferences, see the Windchill Help.

You have now implemented your server preferences on Windchill. The customized menu should appear in the ribbon menu for all Windchill users who are accessing the object from within the configured context.

Customized Ribbon Preferences

This section contains the `GUI/Customize/` preferences you can set for Creo View. Add the preferences manually to the XML file. The preference name defines the hierarchy for XML preference category elements.

Location

This preference defines the source of the `admin` ribbon customization file.

- `admin`—The source is the same directory as the `admin` preferences XML file.
- `server`—The source is the same Windchill location as the `server` preferences XML file.
- `shared`—The source is the same location as `shared` cache preferences XML file.

AllowCustomize

This preference controls whether the user can customize his ribbon. It does not stop any existing customization from being applied, but prevents changing the customization.



Note

The **Quick Access Toolbar** is not disabled together with the **Customize Ribbon** option.

The value can be `true` or `false`. If `false`, customization of the ribbon is disabled.

RemoveInvisibleFromQAT

This preference controls visibility of the **Quick Access Toolbar** buttons that have been removed from the ribbon. You can set this value to prevent buttons that are not available in the ribbon due to customization from appearing in the **Quick Access Toolbar** options page.

The value can be `true` or `false`. If `true`, buttons removed from the ribbon do not appear in **Quick Access Toolbar** options page.



Creating Annotation Templates

This appendix describes how to create an Annotation Template XML document.

About Annotation Templates

An Annotation Template is a set of annotation properties and values that can be applied during an edit. Annotation templates make it easy to apply a standard annotation style across a design.

Use a text editor to create and edit annotation templates. Once imported, the annotation templates will then be available for use when editing annotations in Creo View.

Annotation templates can be used to define the following properties:

- Text content of a label
- Expanded text option of a label
- Font size of a label
- Color of any annotation
- Background color for a label or any shape
- Line width of a leader line or freehand
- Transparency of any shape
- Line style of a leader line, freehand, or polygon
- Lock-on-zoom property of a note
- Shape of a label

About the Annotation Template XML Document

The element `<annotationtemplates>` can contain a number of different templates.

The file may also contain a `name` attribute. Use the `name` attribute to distinguish templates from different files that have the same name.

The element `<annotationtemplate>` contains `<template>` tags. Each `<template>` tag defines a single annotation template.

The `<template>` tag contains the following information:

- An attribute name which defines the annotation template name
- A list of `<property>` tags that define template properties in the template:
 - `type`—Defines the type of property, for example line style or transparency.
 - `value`—Sets the value of the property
 - `appliesto`—Applies the template to certain annotation types

`<type>` Element

The `<type>` element defines type of property. Define the type of property by setting the `<values>` for one or more of the following:

Property Type	Values	Description
<code>text</code>	<code>text</code>	Text content for a note without attributes.
<code>textexpanded</code>	<code>true, false</code>	Expanded Text option
<code>fontsize</code>	real value	Font size in points
<code>shape</code>	<code>circle, square, rectangle, hexagon, triangleup, triangledown, roundedrectangle</code>	Shape of the note.
<code>color</code>	hexadecimal color value	
<code>backgroundcolor</code>	hexadecimal color value	
<code>linewidth</code>	any integer	The line widths selectable interactively corresponds to 1, 3, 5
<code>linestyle</code>	<code>solid, dotdash, dotdotdash</code>	One of the literal values shown, corresponding to

Property Type	Values	Description
fillstyle	none, transparent, opaque	the line styles available when editing interactively in Creo View. One of the literal values shown, corresponding to the transparency values available when editing interactively in Creo View.
lockonzoom	true, false	

<appliesto> Attribute

The attribute <appliesto> defines which types of annotation the property should be applied to. For example, use this attribute to define a different color for a note and a leader line separately. The value of the <appliesto> attribute is a comma-separated list of one or more of the following values:

- all
- note
- leaderline
- freehand
- rectangle
- ellipse
- polygon — Surface polygon in 3D and a 2D polygon

Annotation Template XML Document Example

Below is an example of an Annotation Template file. The file contains three individual templates: Critical Surface Polygon, Needs Review Note and Blue Ellipse.

```
<annotationtemplates name="examples">
<template name="CriticalSurfacePolygon">
<property appliesto="note" value="CriticalSurface" type="text"/>
<property appliesto="note,polygon,leaderline" value="ff0000" type="color"/>
<property appliesto="polygon,leaderline" value="dotdash" type="linestyle"/>
<property appliesto="polygon" value="opaque" type="fillstyle"/>
<property appliesto="note" value="a0a0a0" type="backgroundcolor"/>
</template>
<template name="NeedsReviewNote">
<property appliesto="note" type="text">
<textsegment value="Component" type="text"/>

```

```
<textsegment value="_PV_SystemProperties.Component Name.Value"
type="attribute"/>
<textsegment value="needs review" type="text"/>
</property>
<property appliesto="note" value="ff0000" type="color"/>
<property appliesto="leaderline" value="00000" type="color"/>
<property appliesto="note" value="8" type="fontsize"/>
<property appliesto="leaderline" value="3" type="linewidth"/>
<property appliesto="note" value="roundedrectangle" type="shape"/>
<property appliesto="note" value="true" type="textexpanded"/>
</template>
<template name="BlueEllipse">
<property appliesto="ellipse" value="0000ff" type="color"/>
<property appliesto="ellipse" value="opaque" type="fillstyle"/>
</template>
</annotationtemplates>
```

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Customizing Lighting Setups

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This appendix describes the structure, elements, and attributes of the Light Scenes XML document. (For the XML document schema, see the separate appendix, [Lighting Setups XML Document Schema](#).)

About Customizing Lighting Setups

Lighting setups in the Creo View **Lights** list are encoded the Light Scenes XML document. Creo View uses the Light Scenes XML document stored in this XML text file:

```
%APPDATA%\ptc\ProductView\LightScenes.xml.
```

You can edit this file to change or delete lighting setups in the **Lighting Setup** list. (To open the list in Creo View, click the ribbon **Figure** tab, and then click the **Lighting Setup** box in the **Lights** group.) Lighting setups include properties for light type, color, and direction. To update the **Lighting Setup** list after you edit the XML file, place a copy of the updated file in

```
%APPDATA%\ptc\ProductView\LightScenes.xml
```

and then restart Creo View.

About the Light Scenes XML Document

The main elements in the Light Scenes XML document are summarized below. All elements and attributes in the XML document are specified separately in the topics that follow.

About Element and Attribute Types in the Light Scenes XML Document

All elements and attributes are the String data type unless their descriptions in this appendix starts with Mixed, Boolean, or Restricted in parentheses as shown below.

- (Mixed) elements can contain both text and other elements.
- (Boolean) attributes can only be set `true` or `false`.
- (Restricted) attributes can only be set to the values enumerated in their description.

In addition, all attributes are required unless their description in this appendix starts with (Optional).

Main Elements in the Light Scenes XML Document

<lightscenes>	Lighting setup list element. Contains one <lightscene> element for each lighting setup in the list. (See <lightscenes> Element .)
<lightscene>	Lighting setup element. Contains the properties for one lighting setup. (See <lightscene> Element .)
<name>	Lighting setup name element. Can contain one of three localized predefined names or a custom name. (See <name> Element .)
<light>	Lighting element. Defines the type, color, and direction for one light in a lighting setup. A lighting setup (<lightscene> element) contains one or more lights (<light> elements). (See <light> Element .)

Structure of the Light Scenes XML Document

```
<?xml version="1.0" encoding="UTF-8"?>
<lightscenes>
  <lightscene>
    <name />
    <light>
      <color />
      <position />
      <direction />
      <angle></angle>
      <exponent></exponent>
    </light>
    ...
    <light />
    <light />
  </lightscene>
  <lightscene />
  ...
</lightscene />
</lightscenes>
```

<lightscenesc> Element

The <lightscenesc> element defines all lighting setups in the Creo View **Lighting Setup** list. It contains one or more child <lightscene> elements. The <lightscenesc> element has no attributes. An XML file can contain only one <lightscenesc> element.

<lightscenesc> Syntax

```
<lightscenesc>
  <lightscene />
  <lightscene />
  ...
  <lightscene />
</lightscenesc>
```

<lightscenesc> Child Elements

<lightscene> Lighting setup element. Contains the properties for one lighting setup. (See [<lightscene> Element](#).)

<lightscenesc> Example

The <lightscenesc> element below creates two lighting setups in the **Lighting Setup** list:

- **DEFSingle**—A single-light setup with one key light and a predefined localized name.
- **TwoLights**—A two-light setup with one key light and one fill light and a custom non-localized name.

```
<lightscenesc>
  <lightscene>
    <name locid="DEFSingle"/>
    <light name="key" type="direct" relative="view" />
    <color />
    <direction />
  </lightscene>
  <lightscene>
    <name>TwoLights</name>
    <light name="key" type="direct" relative="view" />
    <color />
    <direction />
    <light name="fill" type="direct" relative="view" />
    <color />
    <direction />
  </lightscene>
</lightscenesc>
```

<lightscene> Element

The <lightscene> element is a child of the <lightscenes> element. Each <lightscene> element encodes the properties for one lighting setup containing one or more lights (<light> elements).

Each <lightscene> element has a unique name defined in its <name> element. The <name> element can contain an optional predefined localized lighting setup name or a custom name.

The <lightscenes> element has no attributes.

<lightscene> Syntax

Example: Lighting setup with a predefined localized name:

```
<!-- Lighting setup with a predefined localized name -->
<lightscene>
  <name locid="DEFlocName"/>
  <light />
  <light />
  ...
  <light />
</lightscene>
```

Example: Lighting setup with a custom non-localized name:

```
<!-- Lighting setup with a custom non-localized name -->
<lightscene>
  <name>Lighting Setup Name</name>
  <light />
  <light />
  ...
  <light />
</lightscene>
```

<lightscene> Child Elements

- | | |
|---------|---|
| <name> | (Mixed) Specifies the lighting setup name shown in the Lighting Setup list. (See <name> Element .) |
| <light> | Defines the type, color, and direction for one light in a lighting setup. A <lightscene> element contains one or more <light> elements. (See <light> Element .) |

<lightscene> Example

This example shows the predefined lighting setup, DEFTwoLights, encoded in a <lightscene> element.

```
<lightscene>
  <name locid="DEFTwoLights"/>
  <light name="key" type="direct" relative="view">
```

```
    <color ambient="#141414" diffuse="#BEBEBE" specular="#CCCCCC"/>
    <direction x="-60.00" y="15.00" z="5.00"/>
</light>
<light name="fill" type="direct" relative="view">
    <color ambient="#050505" diffuse="#969696" specular="#AFAFAF"/>
    <direction x="10.00" y="60.00" z="0.00"/>
</light>
</lightscene>
```

<name> Element

The <name> element is a child of the <lightscene> element. It contains either a `locid` attribute that defines a predefined localized name, or, it contains a custom name. The <name> element has no child elements.

Note

The Creo View installation includes predefined and localized `locid` attribute values for the <name /> child element. These `locid` values have the prefix `DEF` and are reserved by Creo View. Do not edit `locid` values. Rather, if you want to change the lighting setup name, replace the self-closing element with an element of the same name that has a start and an end tag; for example, <name>My Name</name>. The content of the <name> element, My Name, overrides the `locid` value (if present).

<name> Syntax

```
<!-- Predefined localized name -->
<name locid="DEFlocName"/>

<!-- Custom name -->
<name>Lighting Setup Name</name>

<!-- Custom name overrides predefined name -->
<name locid="DEFlocName">Lighting Setup Name</name>
```

<name> Attributes

locid (String) Optional, unique predefined and localized text for the lighting setup name. If the <name> element contains custom name text, the custom name overrides the predefined name in the `locid` attribute.

<name> Example

This example shows a <name> element with a custom name, Key and Fill Lighting, that overrides the predefined name in the locid attribute, DEFTwoLights. The **Lighting Setup** list in Creo View will show **Key and Fill Lighting**.

```
<name locid="DEFTwoLights">Key and Fill Lighting</name>
```

<light> Element

The <light> element is a child of the <lightscene> element. The <light> element defines the type, color, and direction for one light in a lighting setup.

<light> Syntax

```
<light name="Light Name" type="[direct|spot|point]" relative="[view|scene]">
  <color />
  <position />
  <direction />
  <angle>
  <exponent>
</light>
```

<light> Attributes

name	(String) Specifies the name of the light.
type	(String) Specifies the light type as direct (default), spot, or point.
relative	(String) Specifies whether the light is relative to the view (default) or the scene.

Note

For orthographic projections, if you set the <light> attribute type="spot" or type="point", you should only set the attribute relative="scene". Setting relative="view" can produce unpredictable results.

<light> Child Elements

<color />	Specifies the color intensity of the ambient, diffuse, and specular light settings. (See <color> Element .)
<position />	For spot and point lights only—Specifies the XYZ coordinates for the location of the point or spot light source, assuming it not infinitely far from the scene. (See <position> Element .)
<direction />	For direct and spot lights only—Specifies the XYZ vector for the direction of the direct or spot light source. (See <direction> Element .)
<angle>	For spot lights only—Specifies the cutoff angle for the spot light source. (See <angle> Element .)
<exponent>	For spot lights only—Specifies the exponent for the spot light source. (See <exponent> Element .)

<light> Example

This example shows the light element for a spot light source that has a position in the scene.

```
<light name="back" type="spot" relative="scene">
  <color ambient="#0A0A0A" diffuse="#FFB669" specular="#E0E0E0"/>
  <position x="71.00" y="66.00" z="24.00"/>
  <direction x="-60.00" y="60.00" z="-20.00"/>
  <angle>10.00</angle>
  <exponent>128</exponent>
</light>
```

<color> Element

The <color> element is a child element of <light> that specifies the intensity of the ambient, diffuse, and specular settings for the RGB hexadecimal color codes. (The <color> element is self-closing. It has no content or child elements.)

<color> Syntax

```
<color ambient="#hexColor" diffuse="#hexColor" specular="#hexColor" />
```

<color> Attributes

ambient	Specifies the ambient light setting as a RGB hexadecimal color code. Valid values are #000000 to #FFFFFF. This attribute value corresponds to the Ambient setting in the Custom lighting setup dialog box.
diffuse	Specifies the diffuse light setting as a RGB hexadecimal color code. Valid values are #000000 to #FFFFFF. This attribute value corresponds to the Diffuse setting in the Custom lighting setup dialog box.
specular	Specifies the specular light setting as a RGB hexadecimal color code. Valid values are #000000 to #FFFFFF. This attribute value corresponds to the Specular setting in the Custom lighting setup dialog box.

<color> Example

```
<color ambient="#333333" diffuse="#CCCCCC" specular="#CCCCCC"/>
```

<position> Element

The <position> element is a child element of <light> for spot and point lights only. It specifies the XYZ coordinates for the location of the point or spot light source that is not infinitely far from the scene. If the <position> element is omitted, the point or spot light source is considered infinitely far away; i.e., rays of light from the source are considered parallel.

The <position> element is self-closing. It has no content or child elements.

<position> Syntax

```
<position x="xValue" y="yValue" z="zValue" />
```

<position> Attributes

x y z	(Float) Specifies the ±(X, Y, or Z) coordinate for the point or spot light source location.
--------------	---

<position> Example

This example shows a <position> element for the XYZ location (-150, 150, 600).

```
<position x="-150.00" y="150.00" z="600.00" />
```

<direction> Element

The <direction> element is a child element of <light> for direct and spot lights only. It determines the axis of the cone of light for the light source. The XYZ vector coordinates are taken relative to the origin at the vertex of the light cone.

(The <direction> element is self-closing. It has no content or child elements.)

<direction> Syntax

```
<direction x="xValue" y="yValue" z="zValue" />
```

<direction> Attributes

x y z (Float) Specifies the \pm (X, Y, or Z) coordinate for the direction vector along the axis of the direct or spot light.

<direction> Example

This example shows a <direction> element with the XYZ vector coordinates (-60, 60, -20).

```
<direction x="-60.00" y="60.00" z="-20.00"/>
```

<angle> Element

The <angle> element is a child element of <light> for spot lights only. It specifies the cutoff angle in positive degrees from 0.00 to 180.00 as Float. The cutoff angle is the angle between the axis of the light cone and a ray along the edge of the cone.

<angle> Syntax

```
<angle>degValue</angle>
```

<angle> Example

This example shows an <angle> element with a cutoff angle of 10 degrees.

```
<angle>10.00</angle>
```

<exponent> Element

The <exponent> element is a child element of <light> for spot lights only. It specifies the exponent for the spot light source as a positive whole number from 0 (default) to 128 as Integer. Increase the exponent value to increase the focus or to concentrate the spot light source.

<exponent> Syntax

```
<exponent>expValue</exponent>
```

<exponent> Example

This example shows an `<exponent>` element with a exponent value of 128.
`<exponent>128</exponent>`

Light Scenes XML Document Examples

Light Scenes XML Document As Installed

The installed version of the Light Scenes XML document in `LightScenes.xml` is listed below. It contains three predefined lighting setups that appear on the **Lighting Setup** list in Creo View: `DEFSingle (Single Light)`, `DEFTwoLights (2 Lights)`, and `DEFThreeLights (3 Lights)`.

```
<?xml version="1.0" encoding="UTF-8"?>
<lightscenes>
  <lightscene>
    <name locid="DEFSingle"/>
    <light name="key" type="direct" relative="view">
      <color ambient="#3C3C3C" diffuse="#BABABA" specular="#DCDCDC"/>
      <direction x="-60.00" y="15.00" z="5.00"/>
    </light>
  </lightscene>
  <lightscene>
    <name locid="DEFTwoLights"/>
    <light name="key" type="direct" relative="view">
      <color ambient="#141414" diffuse="#BEBEBE" specular="#CCCCCC"/>
      <direction x="-60.00" y="15.00" z="5.00"/>
    </light>
    <light name="fill" type="direct" relative="view">
      <color ambient="#050505" diffuse="#969696" specular="#AFAFAF"/>
      <direction x="10.00" y="60.00" z="0.00"/>
    </light>
  </lightscene>
  <lightscene>
    <name locid="DEFThreeLights"/>
    <light name="key" type="direct" relative="view">
      <color ambient="#0A0A0A" diffuse="#B4B4B4" specular="#E0E0E0"/>
      <direction x="-60.00" y="15.00" z="5.00"/>
    </light>
    <light name="fill" type="direct" relative="view">
      <color ambient="#0A0A0A" diffuse="#CCCCCC" specular="#CCCCCC"/>
      <direction x="10.00" y="60.00" z="0.00"/>
    </light>
    <light name="back" type="direct" relative="view">
      <color ambient="#0A0A0A" diffuse="#CCCCCC" specular="#CCCCCC"/>
      <direction x="320.00" y="190.00" z="125.00"/>
    </light>
  </lightscene>
</lightscenes>
```

Light Scenes XML Document with All Light Types

The following XML document shows a 3-light lighting setup with definitions for all light types; direct, point, spot

```
<?xml version="1.0" encoding="UTF-8"?>
<lightscenes>
  <lightscene>
    <name locid="DEFThreeLights">Three Light Setup</name>
    <light name="key" type="direct" relative="view">
      <color ambient="#333333" diffuse="#CCCCCC" specular="#CCCCCC"/>
      <direction x="-60.00" y="-16.10" z="9.34"/>
    </light>
    <light name="fill" type="point" relative="scene">
      <color ambient="#333333" diffuse="#CCCCCC" specular="#CCCCCC"/>
      <position x="-150.00" y="150.00" z="600.00"/>
    </light>
    <light name="back" type="spot" relative="scene">
      <color ambient="#0A0A0A" diffuse="#ffb669" specular="#E0E0E0"/>
      <position x="71.00" y="66.00" z="24.00"/>
      <direction x="-60.00" y="60.00" z="-20.00"/>
      <angle>10.00</angle>
      <exponent>128</exponent>
    </light>
  </lightscene>
</lightscenes>
```




Lighting Setups XML Document Schema

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This appendix lists the XML schema for the lighting setups XML document used in Creo View.

About the XML Schema

The XML schema listed below defines the following characteristics of the Light Scenes XML document:

- Elements and attributes that can appear in the document, along with their data types and values
- Which elements are child elements, and the order and number of child elements
- Whether an element is empty or can include text

Any custom Light Scenes XML files that you intend to use with Creo View must follow this schema.



Note

For more information on the Light Scenes XML Document, see [Customizing Lighting Setups](#).

XML Schema Listing

```
<?xml version="1.0" encoding="utf-16"?>
<xsd:schema attributeFormDefault="unqualified" elementFormDefault="qualified"
  version="1.0" xmlns:xsd="http://www.w3.org/2001/XMLSchema">
  <xsd:element name="lightscenes">
    <xsd:complexType>
      <xsd:sequence>
        <xsd:element maxOccurs="unbounded" name="lightscene">
          <xsd:complexType>
            <xsd:sequence>
              <xsd:element name="name">
                <xsd:complexType>
                  <xsd:attribute name="locid" type="xsd:string" />
                </xsd:complexType>
              </xsd:element>
              <xsd:element name="light">
                <xsd:complexType>
                  <xsd:sequence>
                    <xsd:element name="color">
                      <xsd:complexType>
                        <xsd:attribute name="ambient" type="xsd:string" />
                        <xsd:attribute name="diffuse" type="xsd:string" />
                        <xsd:attribute name="specular" type="xsd:string" />
                      </xsd:complexType>
                    </xsd:element>
                    <xsd:element name="direction">
                      <xsd:complexType>
                        <xsd:attribute name="x" type="xsd:decimal" />
                        <xsd:attribute name="y" type="xsd:decimal" />
                        <xsd:attribute name="z" type="xsd:decimal" />
                      </xsd:complexType>
                    </xsd:element>
                  </xsd:sequence>
                  <xsd:attribute name="name" type="xsd:string" />
                  <xsd:attribute name="type" type="xsd:string" />
                  <xsd:attribute name="relative" type="xsd:string" />
                </xsd:complexType>
              </xsd:element>
            </xsd:sequence>
          </xsd:complexType>
        </xsd:element>
      </xsd:sequence>
    </xsd:complexType>
  </xsd:element>
</xsd:schema>
```




Configuring Stereoscopic View

Hardware Prerequisites 112
Configuring the NVIDIA Hardware 112

This appendix provides instructions on how to configure your computer to support the stereoscopic view provided by Creo View.

Hardware Prerequisites

For a personal computer:

- NVIDIA 3D display card with 3D vision compatible display
- Compatible shutter glasses

For a television set:

- NVIDIA 3D display card with a 3D television set
- Compatible polarizing glasses

Configuring the NVIDIA Hardware

1. Click , type `nvidia` and press ENTER. The **NVIDIA Control Panel** dialog box opens.
2. On the left, under **Stereoscopic 3D**, click **Set up stereoscopic 3D**. The **Set Up Stereoscopic 3D** opens.
3. Select **Enable stereoscopic 3D**.
4. Verify that **Select when the display is in 3D mode** is set to **Always**.
5. To avoid the flickering effect, verify that you are not using a multiplication of the frequency of your screen. For example, if your screen frequency is set to 50Hz, use 120 Hz.

Note

For some 3D displays, you might need to use a specific resolution. For example, if you are using a 3D television set, you must set its resolution to 1080p, 1920*1080.

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Editing Customized Commands

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This appendix provides instructions on how to edit customized commands in Creo View.

About Customized Commands

You can create customized commands in Creo View. From the Creo View **Options**, under **Customize**, select the **Custom Commands** and click **Add** to add a new command or **Edit** to edit an existing command.

You can later add these commands to the Ribbon or to the Quick Access Toolbar. Adding the command to the right-click menu is done via the command definition as described below.

Using customized commands, you can define one of the following three behaviors:

- Save the current view to an annotation set (defined by XML attributes).
- Copy the image in the view by adding it to the clipboard (defined by XML attributes).
- Open a URL in a web browser and define substitutions for properties (defined by XML content).

About the Command Definition XML Code

Each customized command you define has its own **Command Definition**. The main elements are summarized below. All elements and attributes are specified in the topics that follow.

About Element and Attribute Types in the Command Definition XML Code

All elements and attributes are the String data type unless their descriptions in this appendix starts with Mixed or Boolean in parentheses as shown below.

- (Mixed) elements can contain both text and other elements.
- (Boolean) attributes can only be set true or false.

Available Elements in the Customized Command Definition XML

<menu>	(Mixed) Mandatory menu setup element. This is the root element. (See The <menu> Element Attributes on page 116.)
<command_name>	Mandatory command name element. Specifies the unique name of this customized command. You can use this identifier as a reference in the Ribbon or when you override the command. In addition, this is the identifier you specify when you define the position of another customized command in the right-click menu.

<locked>	(Boolean) Optional locked element. Specifies whether this command can be edited or deleted by any user. This option applies only if this is not a user XML file.
<label>	Mandatory label element. Specifies the command's label on the Ribbon and the right-click menus.
<tooltip>	Optional tooltip element. Specifies the command's tooltip.
<icon>	Optional icon element. Specifies the 16*16 icon. The icon file must reside on the same folder as the XML file. The recommended format is PNG.
<icon_large>	Optional icon element. Specifies the 32*32 icon. The icon file must reside on the same folder as the XML file. The recommended format is PNG.
<action>	(Mixed) Option action element. Specifies the action performed with this command. Uses XML attributes to save a view to an annotation set or copy an image to the clipboard. Uses XML content to open a URL. (See The <action> Element Attributes on page 117.)
<rmb_order>	(Mixed) Optional right-click menu order element. Specifies the position of the customized command in the right-click menu. If you do not define this element, Creo View adds customized commands to the right-click menu in alphabetical order. (See <rmb_order> Element Attributes on page 119.)

Structure of the Customized Command XML

```

<menu
...>
<command_name>MyCommand 1</command_name>
<locked>>true</locked>
<label>Button Label</label>
<tooltip>Button tooltip</tooltip>
<icon>\my\icons\location\icon16x16.png</icon>
<icon_large>\my\icons\location\icon32x32.png</icon_large>
<action>
...
</action>
<rmb_order
name="AnotherCommand"
position="before">
</rmb_order>
</menu>

```

The <menu> Element Attributes

ui	Specifies the location where the action will take place. The available options are <code>ribbon</code> and <code>rmb</code> . If you omit this option, the command will be available from both the Ribbon and the right-click menu.
view	Specifies the objects on which the action can take place in the user interface. The options are <code>2d</code> (2D object), <code>3d</code> (3D object), <code>image</code> , <code>doc</code> , <code>all</code> , and <code>none</code> . You can specify one of the options or an OR list of these options. For example, <code>view="2d 3d"</code> .
pane	Specifies a set of panes where the action will take place in the user interface. The options are <code>file</code> , <code>landmark</code> , <code>tree</code> , <code>viewContentPane</code> , <code>all</code> and <code>none</code> . You can specify one of the options or an OR list of these options. For example, <code>pane="file landmark"</code> . The default value is <code>all</code> .
menus	Specifies the menus and the context menus that this option will be added to. (See Available Values for the menus Attribute on page 116.)
select	Specifies the number of the selected objects required to activate this action. The available options are <code>0</code> , <code>1</code> , <code>any</code> , and <code>many</code> .
visible	Specifies a set of filter conditions using the functions listed below. Each function returns a boolean value that can be used to specify the filter conditions. You can specify more than one function by using the AND operator (<code>&</code>). (See Available Functions for the visible Attribute on page 117.)

Available Values for the menus Attribute

<code>pvgroupspa nel.ProGroup sMenu</code>	Adds the commands to the Groups pane menu.
<code>pvgroupspa nel.ProGroups ContentsMenu</code>	Adds the commands to the Group Content pane menu.
<code>pvresultspa ne.ResultsMe nu</code>	Adds the commands to the Search Results list menu.
<code>pvmodelannota tionspane.Mo</code>	Adds the commands to the Model Annotations menu.

delAnnotation
sMenu
pvlayerspane. Adds the commands under the Layers pane, to the Group list
LayerGroupsMe menu.
nu
pvlayerspane. Adds the commands under the Layers pane, to the Layer list
LayersMenu menu.
pvproppaneme Adds the commands to the Attributes pane menu.
nu.PropDlgMe
nu
pvlayers2dpa Adds the commands to the Layers pane, for the drawings
ne.Layer2DMe menu.
nu
pvblocks2dpa Blocks pane for drawings menu.
ne.Block2DMe
nu

Available Functions for the visible Attribute

getSelected Returns true if the number of selected parts is "greater
Count ">" or than", or "equal to", or "greater than or equal to" the
"==" or ">=" specified number.
number
getPropertyEx Returns true if in the first selected item, the property
ists exists. Since there is no ordering of selection, it is useful
('DESCRIP with a single selection.
TION')
getSelected Returns true if it successfully managed to retrieve the
PartProper property defined in <property name> of the selected
ty(<selection part. The part is defined according to the order of the
index>, <prop selection, specified in <selection index>. The value
erty name>, retrieved is assigned to the URL substring marked
<string>) <string>, according to the XML content.

The <action> Element Attributes

target Specifies the location where the action will take place. The
available option is browser.

saveViewToAnno-Set Specifies that if the current view is an annotation set, the
annotation set is saved, if the value is set to true. If the
value is set to false, the current view is not saved. If
annoSetNameSeed is defined, a new annotation set is

	created and named according to the values of <code>annoSetNameSeed</code> , <code>annoSetNameSuffixUser</code> , and <code>annoSetNameSuffixDate</code> .
annoSetNameSeed	Specifies a name for the annotation set. Together with <code>annoSetNameSeed</code> and <code>annoSetNameSuffixUser</code> , the annotation set name is unique.
annoSetNameSuffixUser	Appends the current user's name to the name of the automatically created annotation set, if the value is set to <code>true</code> .
annoSetNameSuffixDate	Appends the current date to the name of the automatically created annotation set, if the value is set to <code>true</code> .

 **Note**

If `annoSetNameSuffixUser` is set to `true` then the current date is appended after the user name.

copyViewToClipboard	Copies the current view to the clipboard, if the value is set to <code>true</code> . If the view state is an annotation set, then the link and image are both copied. If you paste the contents in Microsoft Excel, Microsoft PowerPoint, or Microsoft Word and click the image link, Creo View opens in the corresponding web page and displays the annotation set as the first page.
----------------------------	--

 **Note**

Creo View will only open if you are running with Windchill.

The Content of the <action> Element

The content of the <action> element defines which URL to open and how. When the content is a URL:

- If you are using a web toolkit to build your server, the window will open from the web browser that Windchill is running in.
- Otherwise, the window will open in the default web browser.

In addition, you can also provide substitutions for properties to edit the URL. Creo View searches the property you list in the element content and replaces its value to create a new URL.

Example: Opening a Standard URL

```
<action
  target="browser">
  http://somewhere.com
</action>
```

In this case, the web browser opens `www.somewhere.com`.

Example: Opening a URL with Substitutions

```
<action
  target="browser">
  http://somewhere.com/content.jsp?material=&lt;!MATERIAL!&gt;
</action>
```

In this case, if the selected item has a property called `MATERIAL` and its value is set to `steel`, it will replace the relevant substring of the URL with the property value. In this case, the web browser would open `http://somewhere.com/content.jsp?material=steel`.

Note

You can define several substitutions in a single URL. Define substitutions by enclosing the property to be substituted with `<! (less than) and !> (greater than).`

<rmb_order> Element Attributes

name	Specifies next to which right-click customized command the new customized command will reside.
position	Specifies the location of the customized command with regards to the command specified in the name attribute. The options are <code>before</code> and <code>after</code> .

Customized Command Definition Examples

Example: Replacing a Part of the URL Using Property Substitutions

This customized command is labeled `Button Label`, has a tooltip, etc. In addition, when this command is activated, it checks the currently selected part for a specified property (`Source_file_name`). If it does not find it, the button is not enabled. If it finds it, it replaces the substring in the URL.

```
<menu
  ui="rmb"
  view="all"
```

```

pane="all"
select="any"
visible="getSelectedCount()==1 && getPropertyValueExists(&apos;
Source_file_name&apos;)">
<command_name >MyCommand 1</command_name>
<label >Button Label</label>
<tooltip>This is a new command.</tooltip>
<icon>\my\icons\location\icon16x16.png</icon>
<action
    target="browser">
http://somewhere.com/content.jsp?partName=&lt;!Source_file_name!
&gt;
</action>
</menu>

```

Example: Replacing a Part of the URL when Multiple Parts are Selected

In this example, when the user selects two parts, Creo View fetches the values of the `Feattrue_Id` property of both parts, returned from `getSelectedPartProperty`. It then replaces two substrings of the URL with these values.

```

<menu
    ui="rmb"
view="all"
pane="all"
select="any"
visible="getSelectedCount()==2 && getSelectedPartProperty
(1, &apos;Feature_Id&apos;, &apos;featureId1&apos;) &&
    getSelectedPartProperty(2, &apos;Feature_Id&apos;, &apos;featureId2&apos;)">
    <command_name >MyCommand 2</command_name>
<label >Button Label</label>
<action
target="browser">
http://www.somewhere.com/index.jsp?Id1=&lt;!featureId1!&gt;& &
Id2=&lt;!featureId2!&gt;
</action>
</menu>

```