

Parametric Technology Corporation

Creo™ Elements/Pro™ 5.0
Installation and
Administration Guide

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About This Guide

Creo Elements/Pro, formerly called Pro/ENGINEER, belongs to the Creo Elements family. The Creo design software package encompasses the CAD, CAM, CAE, CAID, and Visualization solutions of PTC. It provides unprecedented flexibility to add new interoperable capabilities such as 2D modeling, direct modeling, and BOM-driven assemblies to your software tools.

Before you install Creo Elements/Pro, read this guide for installation instructions. This guide provides instructions for two purposes:

- Install and set up your license management software
- Install and maintain Creo Elements/Pro 5.0

System administrators who are installing Creo Elements/Pro for the first time must be experienced in application installations. They must also have a sound understanding of operating systems.

How to Use This Guide

This guide supplements the online instructions with PTC.Setup when you install Creo Elements/Pro and other PTC products. Complete step-by-step procedures are provided in this guide. Post installation instructions and reference information are also given.

Related Documentation

The following document may be helpful as you use the *Creo Elements/Pro 5.0 Installation and Administration Guide: FLEXnet Licensing End User Guide*, which discusses the third-party license management software that is used to distribute Creo Elements/Pro licenses.

Technical Support


Contact PTC Technical Support via the PTC Web site, phone, fax, or e-mail if you encounter problems using your software. For more information, see *Opening and Tracking a Call to Technical Support* in the *PTC Customer Service Guide* with your shipment. This guide is also under **CONTACTING TECHNICAL SUPPORT** on the PTC Web page 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 number, contact PTC License Management using the instructions in your *PTC Customer Service Guide*.

Documentation for PTC Products

PTC provides documentation on the product DVD-ROM in the following forms for Creo Elements/Pro 5.0:

- Help Center with context-sensitive Help, search facility, and quick links to helpful information.
- *PTC Customer Service Guide* and the *Creo Elements/Pro 5.0 Installation and Administration Guide* and other books in PDF files. To view and print the books, you must have the Adobe Acrobat Reader installed.

After you install the **Creo Elements/Help** component, you can drag  from the Creo Elements/Pro main menu to any item on the interface to get Help. To access the Help Center, click **Help > Help Center**.

To access all PTC Documentation, you must have a valid user account. Go to www.ptc.com/appserver/common/account/basic.jsp to request a user account or call the Customer Support. For worldwide phone numbers see the *PTC Customer Service Guide* or click **Worldwide Contacts** under **CONTACTING TECHNICAL SUPPORT** on the PTC Web page at www.ptc.com/support/index.htm.

Note

Adobe Acrobat Reader is required to read PDF files. You can download the software from the following site:

www.adobe.com/support/service/downloads.html

Feedback to Documentation

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

`doc-webhelp@ptc.com`

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

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 > Save . Assignee check box. Click OK .
Courier	User input, system messages, directories, and file names.	Processing completed.
Courier with less-than and greater-than symbols (< >)	Variables for which the user substitutes an appropriate value	output=<LOADPOINT>

1

Overview

This chapter describes the materials in your software shipment and the installation utility, PTC.Setup. It also explains what steps to take before you install the license server and product software.

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What You Receive

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

- **Shipment Confirmation Letter**—Before receipt of your PTC software, you receive the shipment confirmation letter via fax or e-mail. This correspondence introduces you to the software distribution process. The Sales Order Number (SON) is included in the letter for new software orders. Instructions are provided in the letter for requesting a License Pack using that number. It also contains a complete list of the software you ordered and the material being shipped.
- **License Pack**—This pack contains the license codes necessary to install the software at your site. Instructions for requesting your License Pack are included in the Shipment Confirmation Letter and the *PTC Customer Service Guide*.
- **Packing Slip**—The packing slip accompanies your shipment and references the Sales Order Number (SON). It lists the software you ordered and the items that were sent with your software shipment.
- **READ THIS FIRST**—This document notifies you of any changes you must make, such as machine settings and recommended operating system patches.
- **Software DVD-ROM**—Your order contains one or more DVD-ROMs for each product you ordered and for each platform on which you intend to run the software. Check the labels of your software DVD-ROMs to verify that you have the correct DVD-ROM for your platforms.

Note

3 DVDs are available for Creo Elements/Pro installation: 2 for Microsoft Windows and 1 for Sun Solaris 10 (UNIX).

- **Hardware and Software Configuration Notes**—To provide the most recent information, PTC provides notes on your product DVD-ROM and on the PTC Web site:

Online Hardware Configuration Notes for Enterprise Partners:
www.ptc.com/WCMS/files/77552/en/proewf5.pdf

Online Software Configuration Notes with interoperability information for Enterprise Partners:
www.ptc.com/appserver/cs/doc/refdoc.jsp

Note

To access the online Software Configuration Notes, a customer support account is necessary.

Refer to the READ THIS FIRST document for steps on how to access information on the DVD-ROM.

- *PTC Customer Service Guide*—Use this guide for descriptions of services from Technical Support, Customer Support, and the PTC Web sites.
- *Creo Elements/Pro 5.0 Installation and Administration Guide*—Use this guide for information on license management, installation of software, and software maintenance.

Before You Proceed

- Make sure you have your License Pack, which contains the required license codes to install your software.

Note

An online copy of your License Pack is highly recommended. Maintaining license information in file format allows you to import your license codes using the installation utility, PTC.Setup. Importing the file ensures accuracy and eliminates the need to supply the data manually.

- Perform license simplification to upgrade your License Pack to version 31 or later, if you have not already done so. See Simplified License Management Process on page A-2 for details.

Receiving Your License Pack

Both new customers and customers who are updating their software receive their PTC License Packs by electronic mail. If you do not have an e-mail address, the License Pack is sent by fax.

New Software Orders and Configurations

After your order has been processed and shipped, you receive a shipment confirmation letter by electronic mail or fax. At that time, you are asked to contact PTC License Management to configure the purchased products.

For more information on PTC License Management, see the **Licensing** section on the PTC Web page at www.ptc.com/support/index.htm.

Maintenance Shipments

You can obtain an electronic copy of your License Pack via e-mail or by visiting PTC on the Web.

A License Pack for a maintenance release shipment contains all qualifying licenses for the site specified. Depending on your site's requirements, one License Pack might contain license information for multiple servers.

Maintenance Release users who are updating their PTC software can also request electronic copies of their License Packs using the PTC License Pack Generator at www.ptc.com/support/proengineer.htm. On this page, click **Retrieve Existing License Packs** under the **Licensing** tab and follow the online instructions to retrieve License Packs. You are asked to provide your Sales Order Number (SON) and to select a License Pack type.

Note

The License Pack information reflects the information in your customer order. If your configurations have changed since the maintenance release was shipped, contact PTC License Management for a new License Pack.

Storing Your License Pack Online

To store your License Pack online:

1. Obtain an electronic copy of your License Pack that PTC sends via e-mail. This contains your license codes.
2. Save your License Pack information in a text file in only ASCII format. On Windows, you can use Notepad or WordPad as a text editor. On UNIX, you can use vi.

Note

The previous step is highly recommended but not required.

Setting Language Defaults

You can install and run Creo Elements/Pro in English, French, German, Italian, Spanish, Simplified Chinese, Traditional Chinese, Korean, Japanese, and Russian. To run the Creo Elements/Pro installation utility in a language other than that of the default operating system, specify the desired language value with the LANG variable at the command prompt:

```
<dvdrom>:\setup.exe -LANG:<lang>
```

For example, to run the installation utility in Chinese, specify chs for <lang> at the command prompt:

```
<dvdrom>:\setup.exe -LANG:chs
```

For information on the language values that you can use with the LANG variable, see the Help topic LANG and PRO_LANG Environment Variables in the Language Support Help.

PTC License Agreement

Before you install PTC License Server or Creo Elements/Pro products using the PTC.Setup utility, you must accept the PTC License Agreement. Follow these steps to accept the PTC License Agreement:

1. Start the PTC.Setup utility. The following screen appears.



2. Select the **Next** button to begin. The **Accept License Agreement** screen appears.

The PTC License Agreement is in the language specified for the language environment variable `LANG`. The default PTC License Agreement is in English.

3. Accept the PTC License Agreement.
 - If a license agreement already exists, you must accept the license agreement to proceed. Your existing license agreement supersedes the one on this screen.
 - If you decline the PTC License Agreement, you are prompted for confirmation and the installation ends.
4. Click **Printable Version** to print the PTC License Agreement.

Using PTC.Setup for Installation

You install Creo Elements/Pro and its software components, including PTC License Server and PTC software products, using PTC.Setup. PTC.Setup is a graphical installation utility with Help. During installation, you can click **Cancel** at any time to stop the installation.

PTC.Setup copies files and sets file permissions. The main screen appears as shown below:



- The main product group includes **PTC License Server** and **Creo Elements/Pro & Creo Elements/Pro Mechanical**.
- The secondary product group under **Other Products** includes **PTC Application Manager** (UNIX only), **Creo Elements/Distributed Services**, **Creo Elements/Help**, **Web.Publish**, **MKS Install for CADD5 Interface** (Windows only), and **Custom Installer**.


Navigation in PTC.Setup is easy. Click the required component on the PTC.Setup product selection screen to start the installation. Use the **Next** button to continue. The **Next** button is not available on the PTC.Setup product selection screen. You must select a product to proceed to the next





screen. Use the **Back** button to return to previous screens to make changes.

When you begin an installation by clicking on the required component, an error message is displayed if the required security privileges are not satisfied.

You can access the READ THIS FIRST document, the *Creo Elements/Pro 5.0 Installation and Administration Guide*, and the Services and Support information from the **Help** list. This list is available on all the screens of the PTC. Setup installation utility.

Selecting Product Features

In all the screens other than the PTC.Setup product selection screen, click  before each of the product components to choose the following commands from the list:

- **Install this feature**—Install the component.
- **Do not install this feature**—Do not install the component.  changes to  if a component is not installed.
- **Install all sub-features**—If a product component has subcomponents, you can install them by selecting this option. The icon  changes to  to indicate that the components will be installed.

Starting PTC.Setup

If your Windows system has autorun enabled for your DVD-ROM drive, PTC.Setup starts automatically. If your UNIX system automatically mounts the DVD-ROM, just load the DVD-ROM, change to a directory with write permissions, and execute the `<dvdrom>/setup` command, where `<dvdrom>` is the mounted DVD-ROM location of the workstation.

If PTC.Setup does not start automatically, use the procedures in the following sections.

Windows

To start PTC.Setup on Windows, choose one of these methods:

- Start Windows Explorer, browse to the DVD-ROM drive, and double-click the `setup.exe` icon.
- At an MS-DOS prompt, type the following command:

```
<dvdrom>:\setup.exe
```

where `<dvdrom>` is the DVD-ROM drive letter.

- From the task bar, select **Start > Run**, and enter the following command:

```
<dvdrom>:\setup.exe
```

UNIX

To start PTC.Setup on UNIX, follow these steps:

1. Change to a directory where you have write permissions (for example, /tmp).
2. At a command prompt, type the following command:

```
<dvdrom>/setup
```

where <dvdrom> is the location of the DVD-ROM. In UNIX systems, you mounted the DVD-ROM to this directory.

3. If a /dvdrom directory does not already exist, create one using the following command:

```
mkdir /dvdrom
```

If your UNIX system does not automatically mount the DVD-ROM, see Loading and Mounting the DVD-ROM on UNIX on page C-7 for step-by-step instructions.

Generating a PTC.Setup Trail File

You can record your menu selections in the PTC.Setup installation utility in a trail file to replay them at a later time. You can use this trail file to install the software again with similar settings. Create a trail file by specifying the `-uilog` argument upon startup of PTC.Setup from a command prompt.

Windows	<dvdrom>\setup.exe -uilog
UNIX	<dvdrom>/setup -uilog

When starting PTC.Setup with the `-uilog` argument, a trail file named `ps_tr1.txt.1` is created in the directory from where you started PTC.Setup.

Note

You must have appropriate permissions to write to the directory from where you start PTC.Setup.

If the `ps_tr1.txt.1` file already exists, a new trail file named `ps_tr1.txt.2` is created. Further trail files are named using this naming convention.

Replaying the PTC.Setup Trail File

To replay the PTC.Setup trail file, you must first rename the trail file using the <filename>.txt naming convention. For example, rename the pc_tr1.txt.1 file to runme.txt. To replay the trail file, use this new file name with the -uitrail argument at the command prompt.

Windows	<dvdrom>\setup.exe -uitrail runme.txt
UNIX	<dvdrom>/setup -uitrail runme.txt

Specify the full path to the runme.txt file so that this file is correctly accessed with the -uitrail argument.

You can also specify both the -uilog and -uitrail arguments when running PTC.Setup. In such cases, the specified trail file is replayed, and at the same time, a new trail file is created that records the menu selections during the current installation.

The user interface of PTC.Setup opens by default when you run a trail file. You can hide the user interface when replaying a trail file by specifying the -nographics argument at the command prompt.

Windows	<dvdrom>\setup.exe -nographics -uitrail runme.txt
UNIX	<dvdrom>/setup -nographics -uitrail runme.txt

If PTC.Setup goes out of sequence when running a trail file, you are prompted for input where the trail file goes out of sequence. If a trail file goes out of sequence, you can choose to exit PTC.Setup by setting the value of the CONTINUE_FROM_OOS environment variable to false.

Windows	set CONTINUE_FROM_OOS=false
UNIX	setenv CONTINUE_FROM_OOS false

Using the PTC.Setup Trail File to Uninstall Creo Elements/Pro

You can create and use the PTC.Setup trail file to uninstall Creo Elements/Pro. Create the PTC.Setup trail file as follows:

```
<creo_loadpoint>\uninstall\i486_nt\obj\psuninst.exe  
<creo_loadpoint>\uninstall\instlog.txt -uilog
```

Use the trail file to uninstall Creo Elements/Pro as follows:

```
<creo_loadpoint>\uninstall\i486_nt\obj\psuninst.exe  
-uitrail mytrail.txt  
<creo_loadpoint>\uninstall\instlog.txt
```

Creating a DVD-ROM Image of a Custom Installation

You can create a smaller DVD-ROM image of the PTC products and components of your choice using the **Custom Installer** option in the PTC.Setup installation utility. Use the Web Installer from the Technical Support site to download the necessary files for the custom installation.

Disabling File Registration (Windows)

Creo Elements/Pro objects appear with a PTC icon in Windows Explorer because of file registration. If you double-click such an object in Windows Explorer, the object opens in a Creo Elements/Pro session. You can disable file registration for the Creo Elements/Pro objects when installing Creo Elements/Pro by specifying the `-nofilereg` argument when running PTC.Setup from the command prompt.

Windows	<dvdrom>\setup.exe -nofilereg
---------	-------------------------------

2

Quick-Start Installation

This chapter provides a quick-start approach for experienced users who have previously installed license management and product software on license server, license client, and node-locked machines. For step-by-step installation instructions using the PTC.Setup utility, read Chapter 3, Installing PTC License Server, and Chapter 4, Installing and Starting Creo Elements/Pro.

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Installing Software on a License Server Machine

If you update your software to a newer release, you must update your License Pack. PTC License Server is always the first component to be installed. Installation of PTC License Server allows subsequent products to locate and use the license information.

Note

You must have the administrative privileges to install the license management software on a license server machine.

Installing PTC License Server

Using the PTC.Setup installation utility, follow this process to install PTC License Server:

1. Obtain an electronic copy of your License Pack, which contains your license codes.
2. Save your License Pack information in a text file, ASCII format only.

Note

This step is highly recommended but not required.

3. To use Creo Elements/Pro 5.0, perform license simplification to upgrade your License Pack to version 31 or later, if you have not already done so. See Simplified License Management Process on page A-2 for details.
4. Start PTC.Setup, the installation utility. Accept the PTC License Agreement (see PTC License Agreement on page 1-5).
5. Specify the server host name, an IP (Internet Protocol) address, or a fully qualified domain host name. Host name restrictions depend on the operating system. Alternatively, specify the full path to the License Pack received from PTC License Management.
6. Install the PTC License Server, into its own directory.
7. Import the license file from step 2 into PTC.Setup. PTC.Setup starts the license server after the installation.

Note

Although the procedure is longer, you can manually enter the license information listed in your License Pack. If you type your license information into PTC.Setup's FLEXnet Publisher license editor, you can type in the keywords: `__HOSTNAME__` and `__PCTD_PATH__`. When you save the license file, PTC.Setup automatically replaces `__HOSTNAME__` and `__PCTD_PATH__` with the correct values. For complete details, see Using the FLEXnet Publisher License Editor on page 3-7.

8. Optionally, the `EXTERNAL_NAME` attribute in the feature line gives you the choice to provide a user-defined name or an alias for a floating license-to-run.
9. Install PTC License Server. The license server starts automatically. For triad configurations, repeat these steps for Triad Partners 2 and 3.

Note

Upon installation of the first triad partner, an error message the server could not be started appears. This is normal. The triad is operational only if at least two of the three partners are installed and started.

Installing Creo Elements/Pro

1. Start PTC.Setup, the installation utility. Accept the PTC License Agreement (see PTC License Agreement on page 1-5).
2. If the license server is also a license client, install Creo Elements/Pro into its own directory.
3. Define the platforms, languages, and components that the product software uses.
4. Specify **English** or **Metric** to configure the installation for units.
5. Specify the location of the license file (that is, the server host name). Provide the host name, an IP (Internet Protocol) address, or a fully qualified domain host name. Host name restrictions are dependent on the operating system. Alternatively, specify the location of the License Pack received from PTC License Management (that is, the full path to the license file).
6. Define the command to run Creo Elements/Pro (for example, `creo`). No file name extension is necessary.
7. Attach or associate the licenses listed within your license information file to the startup command so that the product software can read the license data on the server specified. Multiple licenses can be attached to a single command name.

8. Supply optional configurations (for example, Window shortcuts).
9. Start Creo Elements/Pro.

Installing Software on a License Client Machine

When you are installing Creo Elements/Pro and any of its components on a license client machine, you do not install PTC License Server. Instead, you specify the server from which this license client should obtain available licenses.

1. Start PTC.Setup, the installation utility. Accept the PTC License Agreement (see PTC License Agreement on page 1-5).
2. Install the product software into its own directory.
3. Define the platforms, languages, and components that Creo Elements/Pro will use.
4. Specify the host name of the server that maintains PTC License Server. Provide either the IP address or the fully qualified domain host name.
5. Define the command to run Creo Elements/Pro (for example, `creo`). No extension is necessary.
6. Attach or associate the licenses listed within your license information file to the startup command so that your product software can read the license data on the server specified. Multiple licenses can be attached to a single command name.
7. Supply optional configurations (for example, Window shortcuts).
8. Start Creo Elements/Pro.

Installing Node-Locked Licenses and Software

PTC License Server is not required for managing uncounted node-locked licenses. Using the PTC.Setup installation utility, install Creo Elements/Pro and point to the node-locked license file.

Repeat the steps listed in Installing Software on a License Server Machine.

Where Your Software Is Installed

The PTC.Setup installation utility defines and installs software into a default installation directory that you can modify for your environment during installation.

The default path is as follows:

Windows	C:\Program Files\ <code><productname></code>
UNIX	<code>/system_directory/ptc</code> where <code>system_directory</code> is platform-specific.

PTC License Server, the product software, and any components are installed into separate subdirectories within the `ptc` directory.

3

Installing PTC License Server

This chapter explains how to start the installation utility, PTC.Setup, and how to install PTC License Server so that you can control usage of licenses for PTC software products.

PTC uses FLEXnet Publisher from Flexera Software, Inc. as its license server. Unless an uncounted node-locked license is used, you must install PTC License Server before you install your PTC software. PTC License Server is included on your DVD-ROM. For the latest information about the version appropriate for your installation, see the READ THIS FIRST in your shipment.

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Before You Proceed

Before you begin, perform the following steps:

- Check the READ THIS FIRST document for any required machine setting changes.
- Obtain the online copy of your License Pack version 31 or later.

Note

You must perform license simplification, if you have not already done so.

- Review your License Pack to make sure your license information accurately reflects your order.

Note

The *FLEXnet Licensing End User Guide* is available on the Creo Elements/Pro DVD-ROM.

Finding the PTC Host ID

You can obtain the PTC Host ID by starting PTC.Setup from any PTC DVD-ROM. When PTC.Setup is started, the PTC Host ID of the machine is automatically displayed in the lower-left corner of the main screen.

PTC License Server uses the PTC Host ID to assign license codes.

Note

A PTC Host ID is necessary only when a feature is locked to a specific machine or to multiple machines.

To download a copy of the PTC `ptchostid` utility and to view online instructions on how to run this tool, visit the PTC Web site at www.ptc.com/appserver/wcms/standards/freefull_cskdb.jsp?im_dbkey=90575&icg_dbkey=900. Platform-specific instructions are given.

Windows Requirements

For Windows XP, Windows XP x64, and Windows Vista, you must have administrative privileges to install PTC License Server.

Your Windows system must use the Windows NTFS (Windows NT File System).

You must have TCP/IP (Transmission Control Protocol/Internet Protocol) installed and configured correctly on your Windows system before installing the software.

FlexNet Publisher 10.8.10 is supported for Windows Server 2008 R2 on 64-bit operating systems. It is available as a download-only utility for installation.

UNIX Requirements

PTC.Setup lets you install the software with either root- or user-level privileges. For maximum security, Flexera Software, Inc. recommends that you install PTC License Server with root privileges but run the server with user privileges.

Installing PTC License Server

You must install PTC License Server on at least one machine to control usage of floating licenses. This machine need not have an installation of the product software.

Note

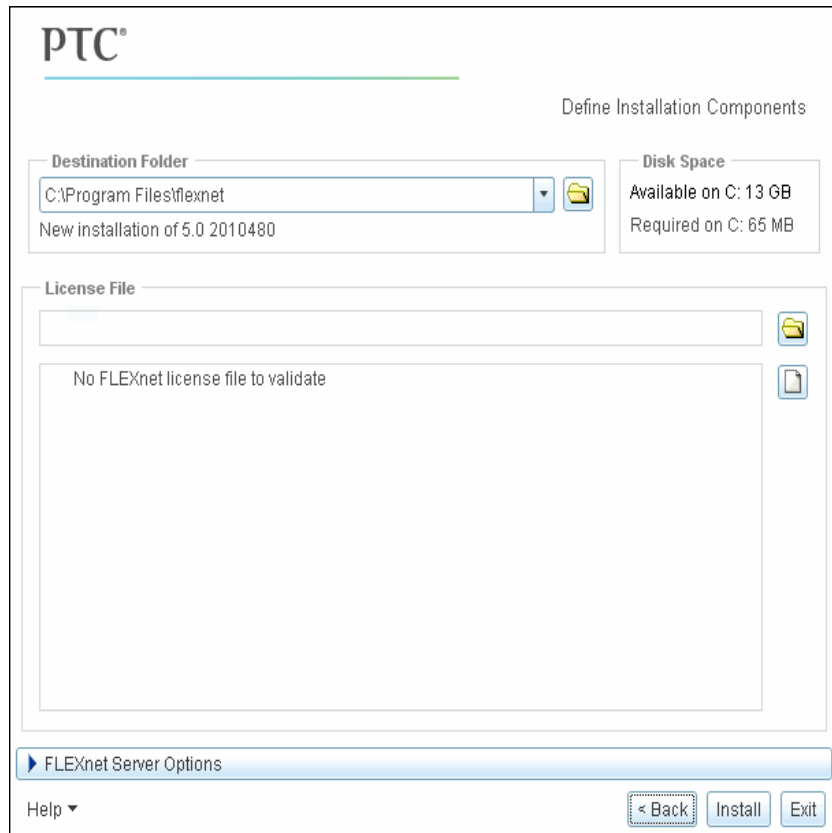
The machine on which you install PTC License Server must have a static IP (Internet Protocol) address configuration and not a DHCP (Dynamic Host Configuration Protocol) configuration.

License management software is not required for a license client. This client requests a license from the server.

To install PTC License Server, perform the following steps. During installation, you can click **Cancel** at any time to stop the installation:

1. Obtain an online copy of your License Pack and save your license file as an ASCII file. For more information, see Receiving Your License Pack on page 1-3.
2. Start the PTC.Setup installation utility. Accept the PTC License Agreement (see PTC License Agreement on page 1-5). The PTC.Setup product selection screen appears.

3. Click **PTC License Server** to start the installation of FLEXnet Publisher. The **Define Installation Components** screen appears.



4. Define the following parameters:
 - **Destination Folder**—Type a destination folder to install PTC License Server or keep the default value. The destination folder must be new and cannot be an existing folder.
Note
On Windows, you can have only one installation of PTC License Server on a machine. When you start installing PTC License Server, PTC.Setup looks for an existing PTC License Server installation. If an installation is found, you are forced to use the existing PTC License Server directory as the installation directory. This ensures that you do not install multiple PTC License Servers on the same machine.
 - **Disk Space**—This area displays the available disk space and the space requirements for each component.

- **License File**—Specify a license file by browsing to the encrypted license file. You can also drag your license file into the **License File** text box. The large box under the **License File** text box displays the information in the license file.
5. Optionally, click **FLEXnet Server Options** to specify the following items:
 - Server process owner (UNIX only).
 - Command line arguments. This gives the administrator the control over the server processes. See *FLEXnet Licensing End User Guide* for details.
 - Start of the license server if the system is rebooted.
 6. Click **Install** to start the installation of the license server. Upon completion, PTC.Setup starts the license server.

Exception: Triad installations require that you configure two other license servers.

Note

On Windows systems, the **PTC License Management tools** program is added to the **Start > Programs** menu. For more information, see *Lmtools Utility (Windows Only)* on page C-3 .

Supplying Your License Information

You can use one of the following methods to supply the license information that is specific to your site or individual workstation.

- Import the license information from disk (RECOMMENDED).
- Drag your license file into the **License File** text box.
- Manually enter the license information using a text editor.

After you provide your license information, PTC.Setup automatically saves this file within the FLEXnet Publisher installation directory.

Restriction: Do not combine your PTC license file with license files of other software vendors on the same FLEXnet Publisher license server. Although FLEXnet Publisher can handle combined license files, PTC license files use a vendor-specific Host ID making them incompatible with license files using a FLEXnet Publisher Host ID. The PTC.Setup utility reads combined vendor license files as invalid.

Note

If you want to manually specify your license file information, click **Back**. For detailed information, see *Entering Your License File Information Manually* on page 3-7.

For FLEXnet Publisher triad installations, edit the license file before you install each of the license servers. To do so, type the hostname of each of the triad partners. If this is not done, the license files will be different on each of the three triad partners, resulting in the failure of the license server startup.

Completing the FLEXnet Publisher Installation

The **Installation Progress** screen displays the status of installation. You can also do one of the following operations:

- Click **Cancel** to cancel the installation.
- Click **Back** and review information.
- Click **Next** to install the next selected component.

The **Installation Progress** screen announces a successful installation.

Click **Exit** to quit the PTC.Setup installation utility.

Repeating Steps for Triad Installations

To configure the two other license servers, repeat the aforementioned steps in the sections *Installing PTC License Server* on page 3-3.

Note

PTC and Flexera Software, Inc. recommend that the Triad license servers be at the same geographical site and the same subnet. See *FLEXnet Licensing End User Guide* for details.

Verifying License Server Activity

Before configuring your software, verify that PTC.Setup has started the FLEXnet Publisher license server by running the `ptcstatus` command on the license server machine. PTC.Setup installed the `ptcstatus` command in the `/bin` directory of the FLEXnet Publisher installation directory (for example, on Windows `C:\ptc\flexnet\bin\ptcstatus`). For Triad installations, two of the three partner machines must be configured before the information is displayed in the `ptcstatus` output.

To run the FLEXnet Publisher server, you must have Windows administrator or UNIX user privileges. For additional information see *Machine-Specific Configuration* on page C-3 .


Output from the `ptcstatus` utility displays the feature name (for example, `PROE_38265`), a user-defined name or alias if one was supplied, the number of licenses in use, the number of available licenses, and any restrictions (for example, the license is locked to a particular machine).

Entering Your License File Information Manually

If you do not have an electronic copy of your License Pack, you can enter your license file information manually. To do so, type your license information into the License File Editor directly.

Using the FLEXnet Publisher License Editor

To enter your license codes manually, follow these steps:

1. In the **Define Installation Components** dialog box, click  under **License File**.
2. The **FLEXnet Publisher license editor** dialog boxes opens. You can either import the file or enter each line of your license file. From the **File** menu, you can:
 - Import your license file from another location.
 - Append the contents in the FLEXnet Publisher license editor with a file on your system.
 - Save your license information.
 - Create a copy.

Use the **Open** dialog box to import or append your license file if your license information (ASCII format) is on your system.

3. Alternatively, type each line of your license information as it appears in your PTC License Pack. Replace generic text where appropriate. Start with the line that begins after `#Start: License File for CPUID`. Format the text exactly as shown in the License Pack with justification indents. The text is case-sensitive.

Note

You do not have to type any lines that begin with a crosshatch (#). This text in your License Pack is strictly informational.

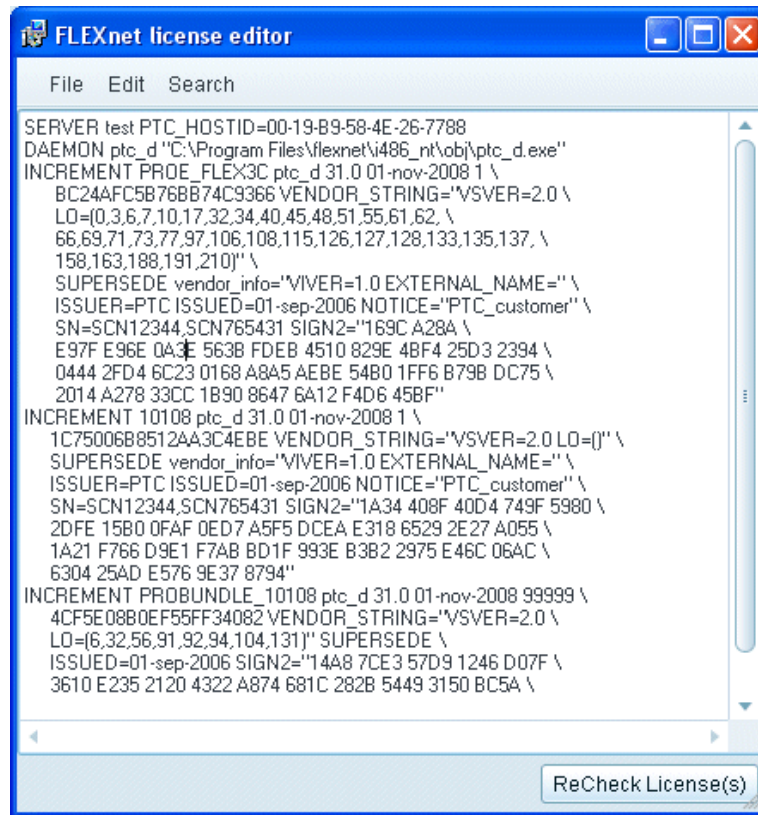
4. If you type them in, PTC.Setup will replace the two generic strings, `__HOSTNAME__` and `__PTCD_PATH__`, including the underscore characters with the correct value after you select **File > Save** from the **FLEXnet license editor**.

In your License Pack, the default server line format is as follows:

```
SERVER __HOSTNAME__ PTC_HOSTID=(Your CPUID) 7788
```

To install PTC License Server, only one vendor daemon line is needed per license server.

When you are finished, the **FLEXnet license editor** should look similar to the next window.



PTC.Setup verifies whether the entered information is correct. If an invalid license is listed, an error message appears. You cannot proceed until all the licenses are listed in the valid license column. For further information, see Appendix F, Troubleshooting Tips.

4

Installing and Starting Creo Elements/Pro

This chapter explains how to install your newly purchased software. The procedures are for new installations only. If you are updating or modifying an installation, refer to Chapter 10, Updating and Modifying an Installation. To install Creo Elements/Pro Mechanical, you need licensing information from PTC. If you plan to run Creo Elements/Pro Mechanical in the integrated mode, it is assumed that you have already obtained licensing information while installing Creo Elements/Pro. If you plan to run it independently, see Chapter 3, Installing PTC License Server, to learn how to install PTC License Server.

You may need to change some machine settings before installing a particular software component. This information is located in the READ THIS FIRST document, which is included with your software package.

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Installing Creo Elements/Pro

You install Creo Elements/Pro after you have installed PTC License Server. You can install your product software on a license server, a license client, a node-locked machine, or on any combination of these machines.

Note

Usage of uncounted, node-locked licenses does not require PTC License Server.

Four major steps are required for successful installation of your product software:

- Defining your installation components, including the installation directory
- Specifying the location of the license servers or license file
- Defining the command names and attaching available features
- Starting the software

Note

Creo Elements/Pro and Creo Elements/Pro Mechanical share a common installation directory. You can install both products together or separately. The files specific to Creo Elements/Pro Mechanical are installed in the mech directory within the installation directory.

Upward Compatibility

Creo Elements/Pro is upward compatible. That is, Creo Elements/Pro files (parts, assemblies, drawings) created in a previous release can be retrieved and saved using the current version of Creo Elements/Pro.

Defining the Installation Components

1. Insert the product software DVD-ROM on your Windows operating system. Insert and mount the product software DVD-ROM on your UNIX system. For step-by-step instructions, see Loading and Mounting the DVD-ROM on UNIX on page C-7 .
2. Start the PTC.Setup installation utility. Accept the PTC License Agreement (see PTC License Agreement on page 1-5). The PTC.Setup product selection screen appears. During installation, you can click **Cancel** at any time to stop the installation. PTC.Setup copies files and sets file permissions.

3. Click **Creo Elements/Pro & Creo Elements/Pro Mechanica**. The **Define Installation Components** screen appears.

At this point, you have installed FLEXnet Publisher either on the license server or on the license client.

Note

When you update to a new release of a PTC product, you must update your License Pack to match the product release.


On UNIX, the **Accept License Agreement** screen appears. Accept the JRE license agreement and click **Next** to proceed. The **Define Installation Components** screen appears.


4. Define the following installation parameters:
 - **Destination Folder**—For new installations, keep the default installation directory, for example, C:\Program Files\PTC\Creo Elements\Pro5.0, or specify the full path. PTC.Setup creates the specified new directory. You can also select a different installation directory.

For new installations, the directory that you specify must be new or empty.

Note

On Windows, you can have only one installation of Creo Elements/Pro for each datecode on a machine. If you install the same datecode again, you are forced to use the existing Creo Elements/Pro installation directory.


- Under **Features to Install**, click  before each of the components and provide the following information:

Product Features—Choose the components that you want to install by clicking  next to each of the components. Select one of the following commands from the list:

Install this feature—Install the component.

Do not install this feature—Do not install the component.

Note

If a product component has subcomponents, you can install them by selecting **install all sub-features** from the list that appears when you click .

The available components are:

Creo Elements/Pro—This installs the files necessary to run the software for Creo Elements/Pro.

Creo Elements/Pro Mechanica—Select this component to install Creo Elements/Pro Mechanica and its components. Select **Creo Elements/Pro Mechanica independent Mode Help Files** to install Help for independent mode Creo Elements/Pro Mechanica in the product installation directory. Select **Creo Elements/Pro Mechanica** to install all the Creo Elements/Pro Mechanica files. By default, all the components are installed at <creo_loadpoint>\mech directory and the startup scripts are installed at <creo_loadpoint>\mech\bin directory.

Note

You can also install **Creo Elements/Pro Mechanica** without installing **Creo Elements/Pro** by selecting **Do not install this feature** for all other components, including **Creo Elements/Pro**.

Creo Elements/Pro Help Files—Keep this component to install the Creo Elements/Pro Help files in the product installation directory. This enables context-sensitive Help by default.

Note

If you exclude the **Creo Elements/Pro Help Files** component, you must then install Help Center at a custom location. See *Installing Help Center at a Custom Location* on page 6-3.

After installing Creo Elements/Pro Help files at a custom location, you must configure Creo Elements/Pro for context-sensitive Help. See *Enabling Context-Sensitive Help from a Custom Location* on page 6-4.

PTC.Setup—Select this component to install PTC.Setup for reconfiguration of the software after the installation. This utility allows for configuration changes without running PTC.Setup from the DVD-ROM.

Options—Select this component to install Expert Framework Extension, ModelCHECK, the required JRE component, Mold Component Catalog, Creo Elements/Pro Plastic Advisor, NC-GPOST, VERICUT, and Creo Elements/Pro Distributed Batch.

Note

You can install only the Distributed Batch client by selecting **Do not install this feature** for all other components, including Creo Elements/Pro. In order to use Distributed Batch in standalone mode, you must install **Creo Elements/Pro**.

API Toolkits—Select this component to optionally install the files necessary to run the Application Program Interface toolkits like J-Link, Web.Link, Visual Basic API, and Pro/TOOLKIT.

Interfaces—Select the external interfaces.

Note

- To install **INTERFACE for CADD5** on Windows, you must also install **MKS Install for CADD5 Interface** from the product selection screen. See Appendix D, *Installing MKS Platform Components (Windows Only)*, for more information.

Platforms—Select the platforms, for example, `sun_solaris_x64`. Your machine's architecture is automatically selected as the default platform.

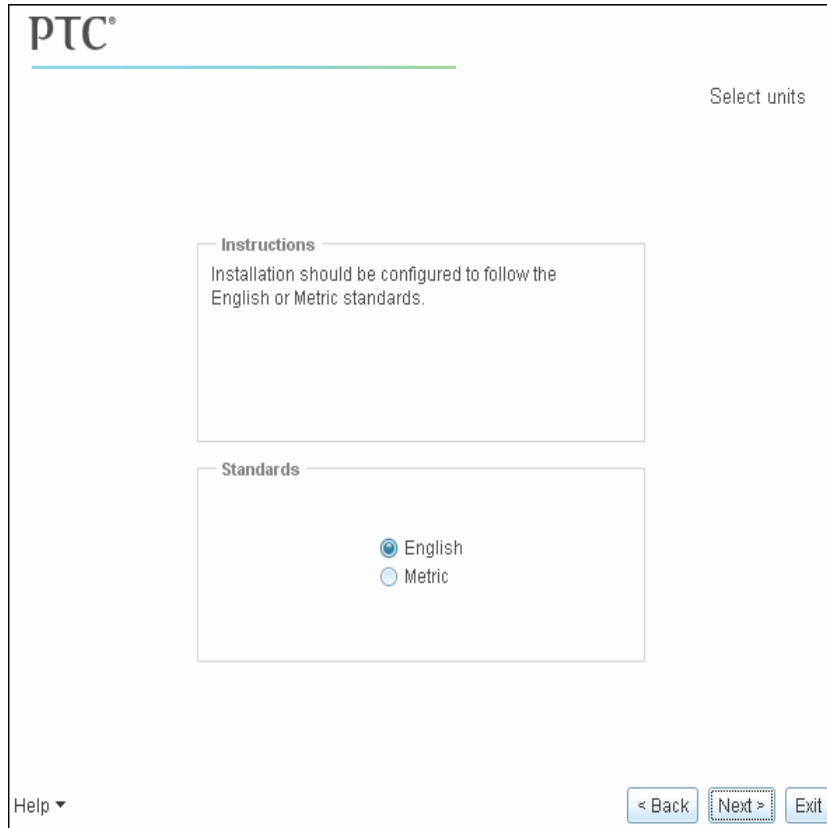
Languages—Select the required languages. English is always installed by default. You cannot disable this installation.

Disk Space—Consider the available disk space displayed and the required space to install the selected features.

5. Click **Next** after you have selected the components to install and the destination folder. The **Select Units** screen appears for you to specify the units for the installation.

Specifying the Units of Measure

1. In the **Select units** screen, select one of the following units of measure for your product installation:



- **English**—Configures your installation to follow the English (inch, pound-mass) units of measure.
- **Metric**—Configures your installation to follow the metric (millimeter, Kilogram, Seconds) units of measure.

2. Click **Next**. The **FLEXnet license servers** screen appears.

Note

See Changing the Units of Measure on page 4-26 to change units after Creo Elements/Pro installation.

Specifying the Location of the License Information

The PTC.Setup installation utility can access your license information on a remote license server or on a local machine. Your next task is to specify the locations of the license servers or license files that you want this installation of your product software to access.

If you attempt to access your license information from a machine without a valid version of FLEXnet Publisher, an error message tells you to update your license server.

Note

Creo Elements/Pro 5.0 requires FLEXnet Publisher version 10.8.6.2 or later.

Specifying the License Server

Depending on the location of your license server, you must provide PTC.Setup with the location of your licensing information. You can do this using the FLEXnet license servers screen.

The screenshot shows a window titled "PTC" with a subtitle "FLEXnet license servers". It contains an "Instructions" box with the text: "Please indicate all license servers that you wish to use with this installation of Creo Elements/Pro". Below this is a list titled "FLEXnet License Servers" with one entry: "7788@bye". To the right of the list are three buttons: "Add", "Edit", and "Delete". At the bottom, there is a legend: "Legend: License to Run; Startup Extensions; Floating options". The bottom right corner has three buttons: "< Back", "Next >", and "Exit". A "Help" dropdown is located at the bottom left.

- If PTC License Server resides on the same machine on which you are installing the software, and you have used the default communications port for the license server, you do not have to specify the license server. PTC.Setup detects the server name and populates the **FLEXnet License Servers** list box.
- If PTC License Server resides on a different machine, you must specify the license server. See the sections, Obtaining a License from a Server or Servers, and Obtaining a Locked License from an Individual License File.

Specifying the License Client

If you are installing the software on a license client without FLEXnet Publisher installed, the **FLEXnet License Servers** list box is initially blank. To enable this installation to obtain floating licenses from a remote server, you must define the servers that are distributing the licenses. See Obtaining a License from a Server or Servers on page 4-8. If this installation is to read a license file for locked licenses, you must specify the full path to the license file. See Obtaining a Locked License from an Individual License File on page 4-10.

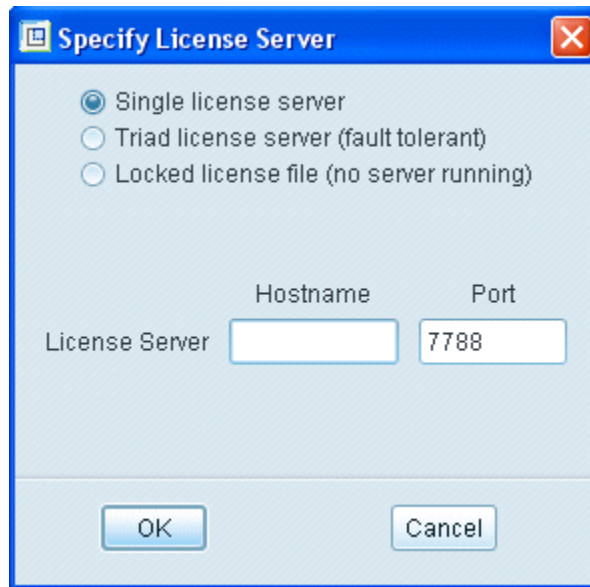
Note

You cannot run Creo Elements/Pro without checking out a license.

Obtaining a License from a Server or Servers

To obtain a license from a server or servers:

1. Click **Add** in the **FLEXnet license servers** screen (see the previous screen). The **Specify License Server** dialog box opens.



2. Click one of the server options and enter the requested information.
 - **Single license server**—Specifies a single server for the licenses. Type the host name and the communications port number if it is different from the default. This information identifies the server, for example, *aberdeen*, from which this installation obtains its license codes.
 - **Triad license server (fault tolerant)**—Specifies a Triad configuration of three Triad partners. As long as two or more partners are running, the licenses are served. If two or more partners are not functioning, the server stops serving the licenses.

Define all three license servers. Each Triad partner server must have the same license file.

Note

The order of the server list (Triad Partner 1, Triad Partner 2, Triad Partner 3) must be the same on all the three license servers.

PTC recommends that the Triad license servers be at the same site. For further information on redundant license servers, see *FLEXnet Licensing End User Guide*.

Note

Do not specify the host name of the client nodes. Specify only the license server that controls usage of the licenses. You can use the **Specify License Server** dialog box to configure the clients to access multiple servers or license files across your network.

- **Locked license file (no server running)**—For locked license files, see the section, Obtaining a Locked License from an Individual License File on page 4-10.
3. Click **OK** to add the server and communications port information to the **FLEXnet License Servers** list box. For example, 7788@aberdeen.
 4. Click **Next** to open the **Windows Preferences** screen on Windows and the **Optional Configuration Steps** screen on UNIX.

Obtaining a Locked License from an Individual License File

To obtain a locked license from an individual license file on your local machine or on the network:

1. Click **Add** in the **FLEXnet license servers** screen. The **Specify License Server** dialog box opens.
2. Click **Locked license file (no server running)**.
3. Perform one of the following actions:
 - Specify the full path to your license file on your local or NFS-mounted disk, for example, C:\Program Files\flex\license.txt.
 - Click the folder icon, navigate the folder hierarchy, and select the correct license file.
4. Click **OK** to add the license file path information to the **FLEXnet License Servers** list. The list when expanded displays the license features available on the selected server.
5. Click **Next**. The **Windows Preferences** screen appears if you are working on Windows. On UNIX, the **Optional Configuration Steps** screen appears.

Creating Windows Shortcuts

On Windows, you can configure the Windows shortcut preferences and path settings for Creo Elements/Pro. Click **Next** on the **FLEXnet license**

servers screen to open the **Windows Preferences** screen. Set the following preferences:

PTC®

Windows Preferences

Instructions

Program shortcuts will be created for Creo Elements/Pro startup commands. Select the location where you would like these shortcuts to be placed. Change the startup directory to a dedicated folder that all users have read

Windows Shortcut Preferences

Shortcut Location(s) Desktop
 Start menu
 Program folder

Program Folder PTC\Creo Elements Pro

Define the startup directory to a location where all users have read-write access.

Startup Directory

Windows Environment Preferences

Modify system environment for all users
 Modify current user's environment only

Help ▾

- **Shortcut Location(s)**—You can start Creo Elements/Pro from the desktop, the **Start** menu, the **Start > Programs** menu, or any combination of these three options.
- **Program Folder**—If you select **Program folder** as the shortcut location, specify the name of the folder and option in the **Programs** menu.
- **Startup Directory**—Specify the name of the Creo Elements/Pro startup directory.

You can set the following Windows environment preferences:

- **Modify system environment for all users**—Sets the system PATH variable for all users on the system.

Note

You must have the administrator privileges to set the system environment variable for all users.

- **Modify current user's environment only**—Sets the user's PATH variable for the current user on the system.

Note

The same preferences will be used if you are installing Creo Elements/Pro Mechanical along with Creo Elements/Pro.

Configuring a Windows Installation

When installing Creo Elements/Pro on Windows, use the **Install** **Optional Utilities** area of the **Optional Configuration Steps** screen to install optional utilities. To perform optional configuration tasks, use the options in the **Instructions** area. The **Optional Configuration Steps** screen appears when you click **Next** on the **Windows Preferences** screen.

PTC®

Optional Configuration Steps

Install Optional Utilities

- Install Thumbnail viewer
- Install Creo Elements/View Express

Instructions

Select the optional configuration steps you wish to perform

- Additional license configurations
- Creo Elements/Pro OLE settings
- Creo Elements/Pro Distributed Services Configuration
- Creo Elements/Pro Mechanical License Configuration
- Remote batch settings

Help ▾

< Back Next > Exit

Installing Optional Utilities

On Windows, you can install the following optional utilities on the **Optional Configuration Steps** screen:

- Thumbnail viewer
- Creo Elements/View Express

Installing Thumbnail Viewer

You can install the Thumbnail viewer utility to view 2D thumbnails for Creo Elements/Pro files in Windows Explorer.

1. Click the **Install Thumbnail viewer** check box in the **Install Optional Utilities** area.
2. Click **Install**.

Installing Creo Elements/View Express

Before you install Creo Elements/View Express, ensure that you have administrative privileges.

1. Remove any previous version of Creo Elements/View Express.
2. Click the **Install Creo Elements/View Express** check box in the **Install Optional Utilities** area.
3. Click **Next**. The **Creo Elements/View Express** installation screen appears.
4. Accept the default installation path or specify a new location.
5. Click **Install**.

Performing Optional Configuration Tasks

If you are installing Creo Elements/Pro on Windows, you can do the following optional tasks:

- Add additional license configurations.
- Configure Creo Elements/Pro as an Object Linking and Embedding (OLE) server.
- Configure Creo Elements/Distributed Services.
- Configure licenses for Creo Elements/Pro Mechanical.
- Configure your remote batch server.

After you complete each optional configuration, click **Next** to proceed.

Adding License Configurations

Click the **Additional license configurations** check box to create a license configuration. Perform the following steps:

1. Add the additional configurations by clicking **Add** in the **License configurations** dialog box. The **Select Licenses** dialog box opens. A license configuration called **Creo Elements/Pro** is always created by default.

Note

`Configuration Description` is a user-defined label name that is displayed on **Creo Elements/Pro** startup if you have defined more than one license configuration. `Configuration Name` is a file name (*.psf) that stores the license configuration information. You can change this file name if required. Use alphanumeric characters to define the configuration name. `Configuration Description` can contain spaces and other characters that are not allowed in the `Configuration Name`.

2. Highlight the required license and move it to the **Selected Licenses** column. The **Select Licenses** section displays the set of all licenses to run and the startup extensions and floating licenses.

Configuring the Creo Elements/Pro OLE Settings

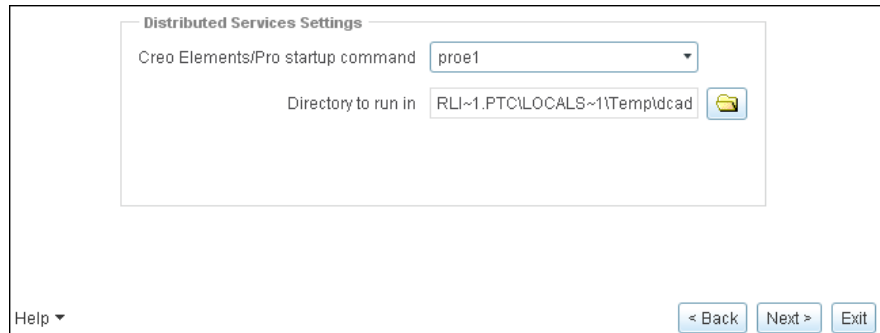
Click **Creo Elements/Pro OLE settings** on the **Optional Configuration Steps** screen to configure **Creo Elements/Pro** as an Object Linking and Embedding (OLE) server. With this setting, users of the client application can import **Creo Elements/Pro** files (parts, assemblies, and drawings) into Microsoft documents (Word, Excel, and Power Point). This setting does not configure **Creo Elements/Pro** as an OLE container for the import of Microsoft files into **Creo Elements/Pro**. In the **Creo Elements/Pro OLE Server** screen, you can:

- Specify the default startup directory for **Creo Elements/Pro**. Typically, this is the **Creo Elements/Pro** working directory.
- Keep the **Creo Elements/Pro** language default unless you plan to use another language.
- Specify the startup directory of the **Creo Elements/Pro OLE** server. The Microsoft application program (that is, a **Creo Elements/Pro OLE** container) starts searching for PTC objects (for example, **Creo Elements/Pro** parts) in this directory.
- Keep the language default unless you plan to use another language.

Configuring Creo Elements/Distributed Services

Click **Creo Elements/Distributed Services Configuration** on the **Optional Configuration Steps** screen to configure your **Creo**

Elements/Pro workstation to participate in distributed processing sessions.



Supply the following information in the **Distributed Services Configuration** screen:

- The Creo Elements/Pro startup command.
- The directory path to use as a temporary working area for the Distributed Computing Server. The temporary files are created here during a distributed computing session. You can also browse to the required location.

Configuring Licenses for Creo Elements/Pro Mechanica

Click **Creo Elements/Pro Mechanica License Configuration** on the **Optional Configuration Steps** screen and click **Next** to configure licenses for Creo Elements/Pro Mechanica. A list of license types against the selected licenses are displayed in the **Creo Elements/Pro Mechanica License Configuration** screen. Click **Edit** to change the license value for the selected license type.

Configuring Your Remote Batch Settings

Click **Remote batch settings** on the **Optional Configuration Steps** screen to configure the Creo Elements/Pro installation for use as a Remote Toolpath Computation Server. This enables remote batch jobs to be run on this machine. You start the Remote Batch Server after the installation process by running the `rbm_startup` script.

Supply the following information to configure the `rbm_startup` script:

- The Creo Elements/Pro startup command that is used to run remote batch jobs (for example, `creo_rb`)
- The maximum number of Creo Elements/Pro sessions to be started on this machine for processing jobs

- The frequency of communication (in seconds) between Creo Elements/Pro and the Remote Batch Server
- The RPC communications port number for communication with machines that are using this installation as a Remote Batch server
- The directory in which Creo Elements/Pro runs when processing batch jobs

Configuring a UNIX Installation

Optional configuration steps follow for your installation of Creo Elements/Pro on UNIX systems. You can accept or clear the following items:

- Add additional license configurations.
- Create a symbolic link to the commands that start your software.
- Configure Creo Elements/Distributed Services.
- Configure the Application Manager installation directory.
- Configure licenses for Creo Elements/Pro Mechanical.
- Configure your remote batch server.
- Specify the full path to your Help installation directory (specifying the load point and installing the Help files are necessary steps to enable clients to view Help).
- Specify the executable file for the Web browser if Mozilla does not start.

After you complete each optional configuration, click **Next** to proceed.

Enabling Interoperability with Other Products

Click **Configure Other Product Interoperability** to enable interoperability with other products.

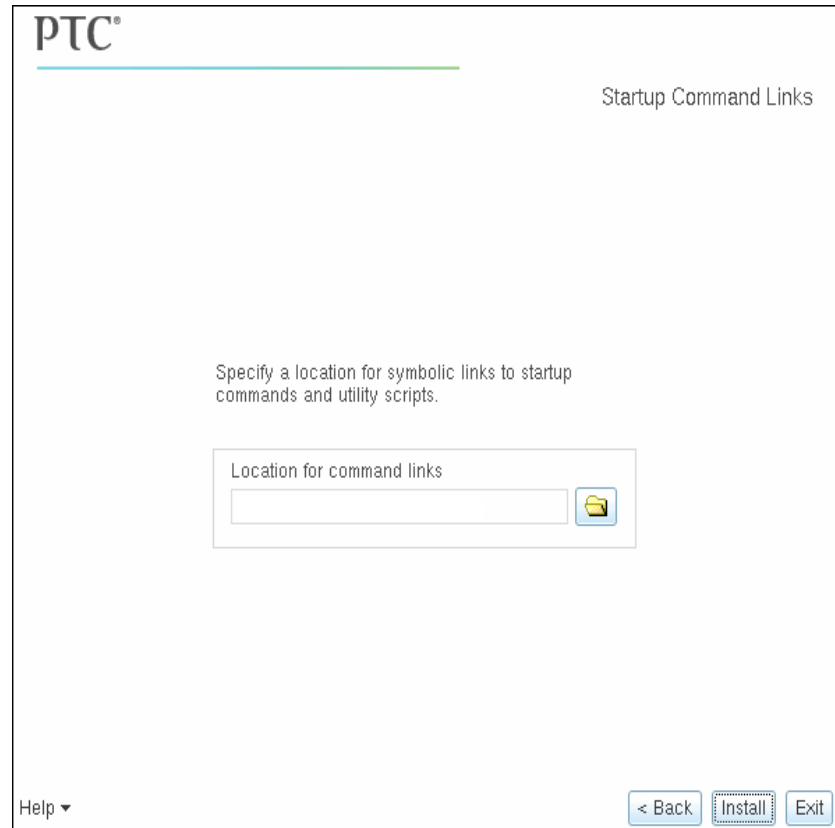
Locate Other Installation Locations (Optional)—Specify paths only if you run PTC Application Manager on this license client in conjunction with Creo Elements/Pro on UNIX systems. If this optional module is not attached to your startup command for Creo Elements/Pro, you can leave the lines blank or configure it later by rerunning PTC.Setup. If Pro/PHOTORENDER has been purchased as an optional module, enter the full path to the Graphic Library directory.

After you specify your optional input, click **Install**.

Creating Links

Click **Create links to product startup commands** on the **Optional Configuration Steps** screen to create a link from the `/bin` directory in your installation directory to another directory on your UNIX system. (If this other directory is in your path environment setting, then simply entering the command name starts Creo Elements/Pro.)

To create a link, use the **Startup Command Links** screen to type the full path or browse to the directory in which you want the link created.



The screenshot shows a web-based configuration window titled "Startup Command Links" with the PTC logo in the top left. The main text reads: "Specify a location for symbolic links to startup commands and utility scripts." Below this is a form labeled "Location for command links" containing a text input field and a folder icon button. At the bottom left is a "Help" dropdown menu, and at the bottom right are three buttons: "< Back", "Install" (with a dashed border), and "Exit".

Completing the Creo Elements/Pro Installation

After you have configured the installation, do one of the following operations:

- Click **Back** and review information.

- Click **Install** to begin installation. The **Installation Progress** screen displays the status of the installation.
- Click **Next** to install the next selected component.
- Click **Exit** to exit PTC.Setup.

For procedures, see Specifying the Location of the License Information, and Configuring the Creo Elements/Pro OLE Settings discussed earlier in this chapter.

Configuring Creo Elements/Pro Startup (Windows Only)

When you configure a Creo Elements/Pro startup on Windows, an executable file named `proe.exe` and one or more configuration files (`.psf`) are created for each startup configuration. On Windows, the configuration file contains a section where you can specify and set user-defined environment variables. This user-defined information is preserved during reconfiguration and update installations of Creo Elements/Pro. A configuration file is created when you complete the Creo Elements/Pro installation.

You can run `<creo_loadpoint>\bin\proe.exe` to start the screen that displays a list of the available startup configurations. The screen does not appear if there is only one `.psf` file available in the `bin` directory.

The configuration file is created in the `<creo_loadpoint>\bin` directory.

Note

The configuration file must exist only in the `<creo_loadpoint>\bin` directory.

The startup command is a combination of the startup executable and the configuration file. For example,

```
proe.exe <filename>.psf
```

where `<filename>` is the name of the configuration file.

Following is a sample configuration file for Windows. The text is the default information that is used when you start Creo Elements/Pro. The bold text following the `// USER - PSF` section is considered user-defined information. This user-defined information is used when starting Creo Elements/Pro.

Note

Modify only the last text line of this sample configuration file.

```
// PTC - PSF file: proe
//
ENV=PATH+=%PRO_DIRECTORY%%PRO_MACHINE_TYPE%\deflib
ENV=PATH+=%PRO_DIRECTORY%%PRO_MACHINE_TYPE%\lib
ENV=PRO_COMM_MSG_EXE=%PRO_DIRECTORY%%PRO_MACHINE_TYPE%\
OBJ\pro_comm_msg.exe
ENV=CDRS_DATA=%PRO_DIRECTORY%
ENV=PRO_IMAGE_CONVERTER=%PRO_DIRECTORY%%PRO_MACHINE_TYPE%\obj\convert_image.exe
ENV=PRO_PHOTORENDER=%PRO_DIRECTORY%%PRO_MACHINE_TYPE%\obj\PhotoRender.exe
ENV=GALAXYHOME=%PRO_DIRECTORY%%PRO_MACHINE_TYPE%
ENV=SPG_DIRECTORY=%PRO_DIRECTORY%
ENV=PROTABLE_DIRECTORY=%PRO_DIRECTORY%\protable
ENV=PROTAB=%PRO_DIRECTORY%%PRO_MACHINE_TYPE%\OBJ\protab.exe
ENV=CLHOME=%PRO_DIRECTORY%\text\pcldata
ENV=CV_ENV_HOME=%PRO_DIRECTORY%%PRO_MACHINE_TYPE%\CV102
ENV=LM_LICENSE_FILE=7788@static
ENV=PROE_FEATURE_NAME=PROE_FLEX3C
RUN=%PRO_DIRECTORY%%PRO_MACHINE_TYPE%\nms\nmsd.exe -timeout 300
RUN=%PRO_DIRECTORY%%PRO_MACHINE_TYPE%\obj\xtop.exe

// USER - PSF
// Add User specific environment or run applications below here
ENV=PRO_LANG=german
```

You can set the value of an environment variable or specify the starting of a third-party application under the // USER - PSF section of the configuration file.

In the case of UNIX, you cannot edit the .psf files.

Specifying Environmental Variables

To specify the value of an environmental variable in the configuration file:

- Use the ENV or RUN entry to specify the value of an environment variable or run an external application.
- Use the equal sign (=) following the text ENV to change the existing value of the variable.

- Use the plus and equal to signs (+=) to add a value after an existing value of the variable.
- Use the minus and equal to signs (-=) to insert a value before an existing value of the variable.
- Use the RUN entry to run an external command on startup.

Using the Configuration File to Run Creo Elements/Pro

A configuration file is used as an argument when you run the `proe.exe` executable to start Creo Elements/Pro.

When you run Creo Elements/Pro from the command line, specify a configuration file as the first argument followed by any other argument, such as, a trail file. If you do not specify the configuration file when running Creo Elements/Pro, a dialog box opens with a list of available configuration files. Select one of these configuration files.

If you run Creo Elements/Pro without creating a configuration file, an error message appears to indicate that Creo Elements/Pro cannot be started. In such cases you must run `ptcsetup` from the `<creo_loadpoint>/bin` directory and create a PTC Startup file (PSF). You can then run Creo Elements/Pro.

If you create only one configuration file, Creo Elements/Pro uses this configuration file by default.

Generating a Debug Log File

The debug log file contains the environment variables that you have set. You can run the `proe.exe` executable in debug mode by setting the value of the environment variable `APPL_STARTUP_LOG` to `true`. On Windows, a debug file named `applstarter.log` is created drive C. On UNIX, the debug file named `applstarter.log` is created in the `tmp` directory. You can also use the `APS_LOG` environment variable to set an alternative path to the log file.

Working with CADD5 Models

You can import and export 3D CADD5 parts and assemblies within Creo Elements/Pro by installing the CADD5 interface during the installation of Creo Elements/Pro. To do so, select **INTERFACE for CADD5** under **Interfaces** on the **Define Installation Components** screen. To use this interface, consider the following instructions:

- After you have installed **INTERFACE for CADD5** with Creo Elements/Pro, a configuration file `config_cadds.pro` is automatically saved in the `<creo_install_path>\text` folder. This file contains configuration options whose settings determine how the CADD5 interface works with Creo Elements/Pro. A sample `config_cadds.pro` file follows:

```
atb_ident_cadds_files yes
```

```
cadds_import_layer yes
```

```
intf_cadds_import_make_solid yes
```

```
intf_cadds_version 14
```

```
! extend_cvpath is similar to config-option
search_path for Creo Elements/Pro.
```

```
! It allows to find user's CADD5 objects on
import...
```

```
! extend_cvpath
```

Check the `config_cadds.pro` file for the above configuration options. If required, copy the configuration options to your local `config.pro` file and update their values.

- Check whether the components listed below are installed:
 - **INTERFACE for CADD5** (see page 4 - 5 for installation instructions.)
 - MKS platform components (**MKS Install for CADD5 Interface**). See Appendix D, Installing MKS Platform Components (Windows Only) for installation instructions.
 - PTC Portmapper that is installed as a part of the MKS platform components. Click **Start > Control Panel > Add or Remove Programs** to open the **Add or Remove Programs** dialog box. You must see **PTC Portmapper** as an installed program.

Starting Creo Elements/Pro

After your license management and product software have been installed, start Creo Elements/Pro.

Windows

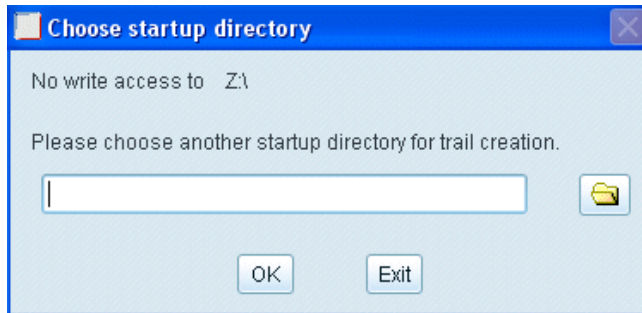
Run Creo Elements/Pro on Windows using the shortcut defined on your desktop or, using the **Start** menu click **Programs > PTC > Creo Elements Pro > Creo**.

UNIX

From a UNIX prompt use `<creo_loadpoint>/bin/proe` to start Creo Elements/Pro, or navigate to the Creo Elements/Pro installation directory and run the command that you defined during the installation process. For example, `/opt/ptc/CreoElementsPro5.0`.

Directory Permissions to Start Creo Elements/Pro

After you install Creo Elements/Pro, if you start it from a directory without write permissions, the following dialog box opens:



To continue, specify another directory with write permissions. The working directory remains the same, but the trail files are stored in the new working directory that you have specified.

Note

You cannot start Creo Elements/Pro from a directory without write permissions unless you specify an alternative directory to store the trail files.

Check for Optional Utilities on Windows

If you have not installed optional utilities like Creo Elements/View Express or Thumbnail viewer, warning messages appear at Creo Elements/Pro startup. Click one of the following commands:

- **Install now** to proceed with the installation
- **Install later** to resume the session without installing the utilities

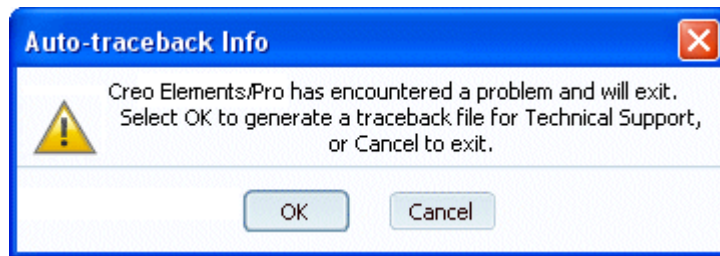
File Type Association on Windows

In a Creo Elements/Pro modeling session on Windows, a warning message appears if you have not set Creo Elements/Pro as the default application to open some of the file types. Click one of the following commands:

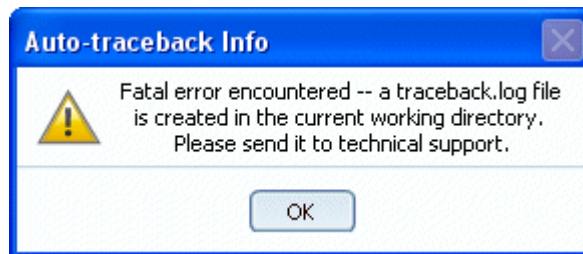
- **Install now** to associate Creo Elements/Pro with the respective file type
- **Install later** to resume the session without associating the file format

Generating a Traceback Log File

In the event of a premature exit, Creo Elements/Pro can output a stack trace of your last operations if you set the `auto_traceback` configuration option to `yes`. The default value is `no`. By default, this information is stored in the `traceback.log` file in the startup directory. The following dialog box opens to indicate the premature exit of Creo Elements/Pro:



The `traceback.log` file can sometimes be useful to Technical Support to resolve the premature exit issue. If you click **Cancel**, Creo Elements/Pro exits without creating the `traceback.log` file. If you click **OK**, the following dialog box opens:



Creo Elements/Pro exits after creating the `traceback.log` file in the current working directory.

Note

If you start Creo Elements/Pro from a directory without write permissions, the `traceback.log` file is stored in the directory that you have specified using the **Choose startup directory** dialog box.

Allocating Memory on a 32-bit Windows System

When working with large models or a large data set in Creo Elements/Pro on a 32-bit Windows system, Creo Elements/Pro may use all available memory and exit prematurely. To be alerted before the available memory is used, use the `proe_memory_buffer_size` configuration option to reserve memory at Creo Elements/Pro startup. Reserving memory displays an early warning. You can save your work under some situations.

Specify the size of the memory buffer in megabytes as a reserve for a possible out-of-memory situation. The default value is 50 megabytes for a moderately sized model. The default has a minimal impact on the memory available to other applications running simultaneously.

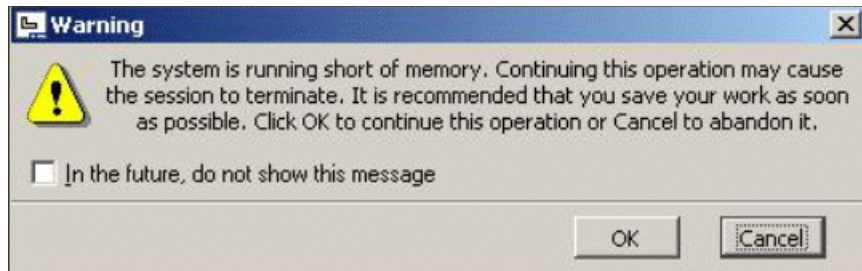
Note

- The `proe_memory_buffer_size` configuration option is processed when you start Creo Elements/Pro. If you change its value in the current session, quit Creo Elements/Pro and restart it for the configuration option to take an effect.
- When you specify a large value for the `proe_memory_buffer_size` configuration option, Creo Elements/Pro can detect an out-of-memory situation at an early stage. In such a case, you increase the probability of saving your work before the available memory is used and Creo Elements/Pro quits.
- When you reserve memory for a Creo Elements/Pro process, the virtual memory available to other applications running simultaneously is reduced. If you specify a large value for the `proe_memory_buffer_size` configuration option, it is recommended that you increase the system settings for the size of the virtual memory paging file.

Using a Nonzero Value for Buffer Size

If you specify a nonzero value for the `proe_memory_buffer_size` configuration option, Creo Elements/Pro periodically checks for memory usage while working with a large data set. If the size of the preserved memory buffer is sufficient, warning messages appear in an out-of-memory situation as discussed below.

For operations that you can cancel, such as, retrieval or regeneration, the following message appears:

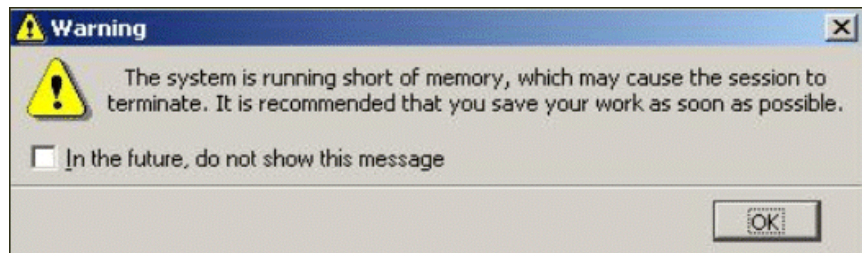


Click **Cancel**. Save your work, quit Creo Elements/Pro, and restart the session.

or

Click **OK** to continue. In this case Creo Elements/Pro may exit as soon as the preallocated memory is exhausted.

For operations you cannot cancel, the following message appears:



Click **OK** to continue. After you complete the operation, save your work, quit Creo Elements/Pro, and restart the session.

Using a Zero Value for Buffer Size

If you set the value of the `proe_memory_buffer_size` configuration option to zero instead of a nonzero value, no warning messages appear.

Using a 3GB Switch for Memory Allocation

On Windows XP, you can use the 3GB startup switch provided by Microsoft to improve memory allocation and to increase the virtual memory address space. Refer to <https://www.ptc.com/appserver/cs/view/solution.jsp?&n=111330> for more information.

Changing the Units of Measure

When you install Creo Elements/Pro for the first time, you are prompted to specify the units of measure for the English or the Metric system in the **Select Units** screen. Based on your selection, a set of configuration options is stored in the local `config.pro` file at `<creo_loadpoint>:\text`. The **Select Units** screen does not appear during an installation update. To change the units of measure after installing Creo Elements/Pro, use the settings in the next table to manually update values for the configuration options in your `config.pro` file:

Configuration Option	English	Metric
<code>chamfer_45deg_dim_text</code>	<code>asme/ansi</code>	<code>iso/din</code>
<code>drawing_setup_file</code>	<code>\$PRO_DIRECTORY\text\prodetail.dtl</code>	<code>\$PRO_DIRECTORY\text\iso.dtl</code>
<code>format_setup_file</code>	<code>\$PRO_DIRECTORY\text\prodetail.dtl</code>	<code>\$PRO_DIRECTORY\text\iso.dtl</code>
<code>pro_unit_length</code>	<code>unit_inch</code>	<code>unit_mm</code>
<code>pro_unit_mass</code>	<code>unit_pound</code>	<code>unit_kilogram</code>
<code>template_designasm</code>	<code>\$PRO_DIRECTORY\templates\inlbs_asm_design.asm</code>	<code>\$PRO_DIRECTORY\templates\mmks_asm_design.asm</code>
<code>template_drawing</code>	<code>\$PRO_DIRECTORY\templates\c_drawing.drw</code>	<code>\$PRO_DIRECTORY\templates\a3_drawing.drw</code>
<code>template_sheetmetalpart</code>	<code>\$PRO_DIRECTORY\templates\inlbs_part_sheetmetal.prt</code>	<code>\$PRO_DIRECTORY\templates\mmks_part_sheetmetal.prt</code>
<code>template_solidpart</code>	<code>\$PRO_DIRECTORY\templates\inlbs_part_solid.prt</code>	<code>\$PRO_DIRECTORY\templates\mmks_part_solid.prt</code>
<code>today's_date_note_format</code>	<code>%Mmm-%dd-%yy</code>	<code>%dd-%Mmm-%yy</code>
<code>tolerance_standard</code>	<code>ansi</code>	<code>iso</code>
<code>tol_mode</code>	<code>nominal</code>	<code>nominal</code>

Configuration Option	English	Metric
weld_ui_standard	ansi	iso

Running Creo Elements/Pro Mechanica in Independent Mode

You can run Creo Elements/Pro Mechanica in independent mode from the **Structure** or **Thermal** shortcuts. These shortcuts are created when you install Creo Elements/Pro Mechanica as a component of Creo Elements/Pro.

Note

Independent mode runs only on Microsoft Windows 32-bit platforms. To run Creo Elements/Pro Mechanica on a Microsoft Windows 64-bit platform, you must have Microsoft Developer Studio and Microsoft Foundation Class runtime environments on your computer. These applications are available with Microsoft .NET Framework 3.5 or Microsoft Visual C++ 2008 or later, which are downloadable from www.microsoft.com. You can also install the runtime components as follows:

1. Insert the Creo Elements/Pro 5.0 DVD-ROM into your DVD-ROM drive. Alternatively, download the DVD-ROM image from the Technical Support page.
2. Browse to `\runtime` directory on the DVD-ROM or to the directory where you have extracted the installation files.
3. Double-click `setup.exe` to open the installer.
4. Click **Next** to install the runtime components.
5. Click **Close** to complete the installation and exit the installer.

Installing and Configuring VERICUT on UNIX

To install and configure VERICUT on UNIX, follow these steps:

1. Install VERICUT as one of the components during Creo Elements/Pro installation.
2. Ensure that the appropriate version of Java is installed on your computer. See information on the JRE version compatibility with Creo Elements/Pro:

www.ptc.com/WCMS/files/77552/en/proewf5.pdf

3. Specify the JRE installation path that is compatible with the current release of Creo Elements/Pro.

Removing Creo Elements/Pro (Windows Only)

You can remove your Creo Elements/Pro application through the Control Panel. This procedure is for Windows systems only.

1. From the **Start** menu, click **Settings > Control Panel**.
2. Double-click the **Add/Remove Programs** icon. The **Add/Remove Programs** dialog box opens.
3. In the program application list, click the specific release of Creo Elements/Pro you want to remove.
4. Click **Remove**. The **PTC Uninstall** dialog box opens.
5. Click **Uninstall** or **Cancel** to escape. You are presented with a series of confirmation statements asking permission to remove files.
6. Click **Uninstall** to proceed. Upon completion, Creo Elements/Pro is removed.

Note

When you remove Creo Elements/Pro, all other components like Creo Elements/Pro Mechanica and Creo Elements/Pro Help files that you have installed with Creo Elements/Pro are removed.

5

Installing PTC Application Manager (UNIX Only)

This chapter explains how to install PTC Application Manager and application libraries of models and textures on a UNIX system. The PTC Application Manager utility installs a toolbar for easy access to your desktop applications when using Creo Elements/Pro.

Topic	Page
Installing PTC Application Manager	5 - 2
Configuring PTC Application Manager	5 - 3
Completing PTC Application Manager Installation	5 - 5

Installing PTC Application Manager

PTC Application Manager is a toolbar utility on your UNIX system. Similar to the Windows **Start** menu, PTC Application Manager provides a menu that enables you to easily start and close applications on UNIX systems. You can configure the menu to include any application, including applications that are not developed by PTC.

PTC Application Manager includes a menu command for each application it starts, giving you easy access on your desktop to those applications.

Note

To view PTC Application Manager in a language other than English, you must set the language variable before installing your software.

During installation, you can click **Cancel** at any time to stop the installation. PTC.Setup copies files and sets file permissions.

Note

After installation, the optional settings can be reconfigured from the PTC Application Manager's **Start > Preferences** menu.

1. Under **Other Products**, click **PTC Application Manager** from the PTC.Setup product selection screen.
2. The **Define Installation Components** screen appears.
3. Define the following:
 - **Destination Folder**—For new installations, keep the default installation directory, for example, `/opt/ptc/appmgr5.0`, or specify the full path. PTC.Setup creates the specified new directory. You can also select a different installation directory.

For new installations, the directory that you specify must be new or empty.
 - **Features to install**—Select the components and the subcomponents to install for PTC Application Manager.
 - **Platforms**—Select the platform for the installation. Your machine's architecture is selected as the default platform.
 - **Languages**—Select the required languages.
4. Click **Next**. The **Configure Application Manager Settings** screen appears.

Configuring PTC Application Manager

You can optionally create Application Manager **Start** menu commands and customize settings when you configure PTC Application Manager. Each PTC.Setup tab lets you configure a different aspect of your PTC Application Manager installation.

To add or delete the names of programs from the **Start** menu after installing the PTC Application Manager,

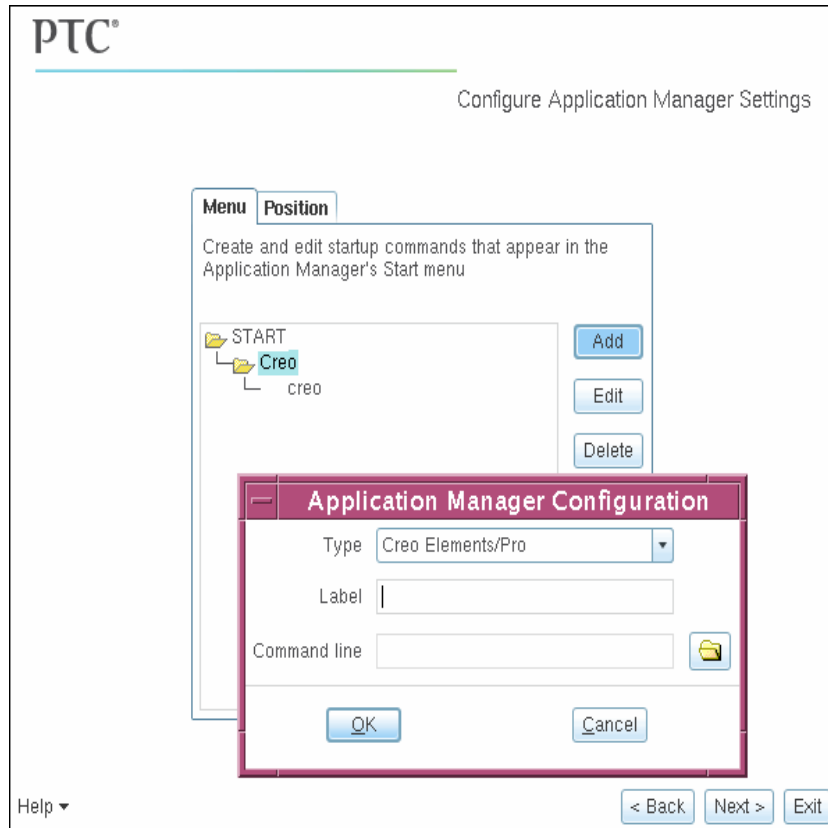
1. Click **Preferences**. The **PTC Application Manager** dialog box opens.
2. The Start menu page lists commands for the PTC Application Manager Start menu. Use the following commands to configure the Start menu:

Command	Action
New Command	Prompts for the name of a program or a command script. Adds that name to the list of available programs.
Modify Command	Modifies the name of the program on the list.
Delete Command	Deletes the selected program from the list.
Apply	Applies the changes to the PTC Application Manager for the current session.
OK	Saves changes to <code>config.apm</code> , a configuration file that preserves the list for later sessions.

You can do the configuration dynamically when you start PTC Application Manager.

Creating Start Menu Commands

You can add the startup commands for the Application Manager **Start** menu (for example, the command `creo`). With these commands, you can start and close applications easily using Application Manager.



1. Click **Add** in the **Configure Application Manager Settings** screen. The **Application Manager Configuration** dialog box opens.
 - a. In the **Type** box, keep **Creo Elements/Pro** (default) or choose another product from the list.
 - b. In the **Label** box, type the text you want to appear in the **Start** menu of the Application Manager.
 - c. In the **Command line** box, browse to or type the full path and the name of the command that you want the label to execute (for example, `C:\ptc\creo\bin\creo.bat`). This command executes the specified license and product type.

2. Click **OK** in the **Application Manager Configuration** dialog box. You can **Edit** or **Delete** existing commands.

Note

After installation, you can add, modify, and delete commands from the Application Manager **Start > Preferences** menu.

Customizing the Application Manager Location

You can customize the appearance of PTC Application Manager as follows:

1. Click the **Position** tab as seen in the figure on the previous page, to customize the position of the PTC Application Manager.
2. Specify where you want the toolbar to appear on your desktop: horizontal top, horizontal bottom, vertical left, or vertical right.

Specifying Optional Configurations

After you have configured PTC Application Manager, you can specify the optional configuration steps as follows:

1. Click **Next** in the **Configure Application Manager Settings** screen to open the **Optional Configuration Steps** screen.
2. Click the **Create links to product startup commands** checkbox.
3. Click **Next** and specify a location to store the command links and utility scripts.

Completing PTC Application Manager Installation

After you have configured the installation, do one of the following operations:

- Click **Back** to review the information that you have specified earlier.
- Click **Install** to begin installation. The **Installation Progress** screen displays the status of the installation.
- Click **Next** to install the next selected component.
- Click **Exit** to exit PTC.Setup.

During installation, you can click **Cancel** in the **Installation Progress** screen at any time to stop the installation.

The **Installation Progress** screen announces a successful installation.

6

Installing and Using Creo Elements/Help

This chapter provides an overview of Help Center. It explains how to install Help Center at a custom location rather than in the installation directory for Creo Elements/Pro software.

Topic	Page
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Choosing Your Installation Location for Help Center	6 - 2
Installing Help Center with Creo Elements/Pro	6 - 3
Installing Help Center at a Custom Location	6 - 3
Installing Help Center on a Web Server	6 - 6

Overview of Help Center


Help Center provides quick access to documentation and includes the following features:

- Context-sensitive Help organized by functional areas. Creo Elements/Pro Mechanica information is integrated with the Creo Elements/Pro Help topics.
- PTC Reference Documentation such as the *PTC Customer Service Guide*. To view and print PDF books, you must have Adobe Acrobat Reader installed.
- A search facility and quick links to helpful information.

You can choose to install Help Center in the default directory, a directory that you specify, or on a Web server. The language in which the Help Center opens is determined by the language setting of the browser on your local host computer.

Accessing Help Center in Creo Elements/Pro

You can access Help from Creo Elements/Pro in the following ways:

- For context-sensitive Help, drag the  from the Creo Elements/Pro toolbar to an item on the user interface.
- To browse Help Center, click **Help > Help Center**.

Choosing Your Installation Location for Help Center

Using PTC.Setup, you can install Help Center at a location of your choice, as listed in the following table.

Installation Location	Advantage	Installation Sequence	Context-Sensitive Help
Creo Elements/Pro installation directory: <creo_install_path>\html\proe_help	For easy installation and display of Help in the Creo Elements/Pro browser on a local computer.	Install Help when you are installing Creo Elements/Pro.	Available by default in the Creo Elements/Pro browser.

Installation Location	Advantage	Installation Sequence	Context-Sensitive Help
Custom location: <code><creo_elements_help_install_path>\html\proe_help</code>	For easy administration of multiple clients on a network server.	Install Help when you are installing Creo Elements/Pro or install Help later using PTC.Setup.	Configuration required. Opens in the default browser of the host computer.
Web server: <code><servlet_document_root>\webapps\proe_help</code>	For service to remote clients not on the local network.	Install Help after you have an instance of Help in the default directory or in a custom location.	Configuration required. Opens in the default browser of the host computer.

Note

If you want context-sensitive Help by default and you did not install Help with Creo Elements/Pro, then you must reinstall Creo Elements/Pro. Be sure to select the **Creo Elements/Pro Help Files** component.

Installing Help Center with Creo Elements/Pro

If you have selected the **Creo Elements/Pro Help Files** component during product installation, the Creo Elements/Pro Help files are installed in the product installation directory. See Defining the Installation Components on page 4-2. This installation enables context-sensitive Help by default in the Creo Elements/Pro browser.

Installing Help Center at a Custom Location


Using PTC.Setup, you can install Help Center at a custom location. You must then configure Creo Elements/Pro for context-sensitive Help.

To install Help Center at a custom location, follow these steps:

1. Start PTC.Setup using the steps in Starting PTC.Setup on page 1-8.
2. Accept the PTC License Agreement (see PTC License Agreement on page 1-5). The PTC.Setup product selection screen appears.


3. Under **Other Products**, click **Creo Elements/Help**. The **Define Installation Components** screen appears.

Define the following components:

- **Destination Folder**—Retain the default folder that appears for the custom installation, or click  adjacent to the **Destination Folder** box and specify the full path of an alternative folder.

The specified folder must be new or empty. PTC.Setup creates the folder if it does not exist.

- Under **Features to Install**, click  next to each component and provide the following information:

Product Features—Choose the components that you want to install by clicking  next to the appropriate component. Select one of the following commands from the list that appears:

Install this feature—Install the component.

Do not install this feature—Do not install the component.

Available are the Creo Elements/Pro Help files and the Creo Elements/Pro Mechanica independent mode Help files.

Platforms—Select the platform for the installation.

Languages—Select the required language.

4. Click **Install**. The **Installation Progress** screen shows the status of the installation. You can click **Cancel** to cancel the installation.
5. Click **Next**.

Enabling Context-Sensitive Help from a Custom Location

If you install Help Center at a custom location, you must configure Creo Elements/Pro for context-sensitive Help:

1. Browse to the Help Center installation path at the custom location `<creo_elements_help_install_path>\html\proe_help`.
2. Double-click the `proe_help.exe` file. On UNIX, run the `proe_help` script file. Help Center opens in the default browser on the host computer.
3. Select the port number of the host computer from the list in the address bar of the Help Center window.
4. In the `config.pro` file, set the `help_center_path` configuration option with the value as the URL of Help Center. For example, if the

port number used by Help Center is 2323, set the configuration option in the `config.pro` file as follows:

```
help_center_path http://localhost:2323/index.jspx
```

Note

The default path for the `config.pro` file is `<creo_install_path>/text/`. However, if a `config.pro` file exists at the path defined by the `$HOME` system variable, the configuration file `$HOME/config.pro` supersedes the file `<creo_install_path>/text/config.pro`.

5. Start Creo Elements/Pro.

Viewing Help Center Installed at a Custom Location

To view Help Center installed at a custom location, use one of the following commands:

- On Windows:
`<creo_elements_help_install_path>\html\proe_help\proe_help.exe`
- On UNIX:
`<creo_elements_help_install_path>/html/proe_help/proe_help`

A Help Center independent from Creo Elements/Pro opens in its own browser.

Alternatively, you can use a command-line argument to start Help Center from a custom location. Use one of the following ways:

- Start Help Center on a predefined port number. For example:
`<creo_elements_help_install_path>\html\proe_help\proe_help.exe -internalServerPort 2323`

In this case, 2323 is the port number of Help Center.

Help Center opens in its own browser.

- Start Help Center on a predefined port number without its browser. For example:
`<creo_elements_help_install_path>\html\proe_help\`

```
proe_help.exe --l4j-no-splash -noUI -xmlRpcPort
9000 -internalServerPort 2323
```

In this case, 2323 is the port number of Help Center. Help Center process starts without its browser. A Help Center icon appears in the system tray indicating an active state of the Help Center process.

You can use Help Center from a custom location using the command-line argument for context-sensitive Help.

Opening and Closing Help Center

When you have Help Center installed at a custom location and have configured it for context-sensitivity, you must follow this order:

1. Start Help Center.
2. Start Creo Elements/Pro.

You can simplify this process by including commands to start Help Center and Creo Elements/Pro in a batch file or a script. Ensure that the port number used in the command to start Help Center is same as the port number specified for the `help_center_path` configuration option in the `config.pro` file.

When you want to end your Creo Elements/Pro session, follow this order:

1. Exit Creo Elements/Pro.
2. Close Help Center.

If you exit Help Center during an active Creo Elements/Pro session, then context-sensitive Help is not available.

Installing Help Center on a Web Server

You can install Help Center on any servlet container, application server, or Web server that supports the JSP 2.0 and Servlet 2.4 standards. For Tomcat and Jetty servlet containers, place Help Center in the document root directory. If you use Websphere, you can use its interface to install Help Center.

Note

You must have an instance of Help Center at the product installation path or at a custom location to install Help Center on a Web server.

To install Help Center on a Web server, follow these steps:

1. Browse to the Help Center installation directory.

- For installation with Creo Elements/Pro:

`<creo_install_path>\html\proe_help`

- For installation at a custom location:

`<creo_elements_help_install_path>\html\proe_help`

2. Double-click the `web_deploy.bat` batch file. On UNIX, run the `web_deploy` script file. Specify the path of the document root directory of the servlet container. For example, for Tomcat servlet container, specify the `<servlet_document_root>\webapps` as the path.

The `web_deploy.bat` batch file performs the following data copy operations:

- Copies Web configuration data placed in the `web_deploy` directory at the Help Center installation location to the `proe_help` directory in the document root directory of the servlet container.
- Copies the Help Center data from the Help Center installation location to the `proe_help` directory in the document root directory of the servlet container.

Note

If a `proe_help` directory exists in the document root directory of the servlet container, you are prompted to remove, move, or rename the existing `proe_help` directory and then rerun the batch file.

3. Configure your Web server to distribute Help Center from the `<servlet_document_root>\proe_help` directory. For example, for a Web server with Tomcat servlet container configured on port number 8080, specify the following URL in a browser window to access Help Center:

`http://<server_name>:8080/proe_help/index.jspx`

Enabling Context-Sensitive Help from a Web Server

You can configure Creo Elements/Pro for context-sensitive Help from the Help Center on a Web server.

1. In the Creo Elements/Pro `config.pro` file, set the value of the `help_center_path` configuration option to the URL of Help Center on the Web server. For example, for Help Center installed on a Web server that has Tomcat servlet container configured on port number 8080, set the value of the `help_center_path` configuration option as follows:

```
help_center_path  
http://<server_name>:8080/proe_help/index.jspx
```

2. Start Creo Elements/Pro.

7

Installing Web.Publish

This chapter explains how to install Web.Publish. With Web.Publish you can set up your Web server and publish objects you created using Creo Elements/Pro. Installing Web.Publish is optional.

With Web.Publish, you can export assemblies, process assemblies, and manufacturing processes to a directory in a Web browser.

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Configuring Your Web Server	7 - 2
Installing Web.Publish	7 - 6

Overview

Installation of Web.Publish involves two processes:

1. Configuration of your existing Web server. The procedures differ depending on what kind of Web server you have:
 - Apache Web Server (UNIX)
 - Microsoft Internet Information Server (Windows NT)
 - Microsoft Peer Web Services (Windows NT)
2. Installation of Web.Publish as a user with write permission to the Web server.

Locate the configuration procedure for your Web server in the following sections and, after configuration, install Web.Publish.

Configuring Your Web Server

Note

For all configuration procedures in this section, the variable `<path>` represents the full path to the installation directory for your Web server.

Apache Web Server

1. After downloading the Apache Web Server (www.apache.org), create the following directories in the `apache_x.x.x` directory:
 - `server_root`
 - `home`where `apache_x.x.x` is the directory downloaded from the Apache Web site.
2. Copy the `conf` directory from the `apache_x.x.x` directory into the `server_root` directory created in the previous step.

- In the `server_root/conf` directory, create copies of the configuration files as indicated in the following table. The original file and the copy must both reside in the `server_root/conf` directory.

Name of the Original File	Name of the Copied File
<code>httpd.conf-dist</code>	<code>http.conf</code>
<code>access.conf-dist</code>	<code>access.conf</code>
<code>srm.conf-dist</code>	<code>srm.conf</code>

- Edit the `http.conf` file according to your user status.

As a root user, locate the following lines:

```
User nobody
Group #-1
```

Replace `nobody` with `root` and replace `#-1` with `sys` for the previous lines.

As a nonroot user, locate the following line:

```
Port 80
```

Replace `80` with a free communications port number that is greater than `1023` for the previous line.

- Use a text editor to replace configuration lines in your `http.conf` file as indicated in the following table. Save the file.

Line	Text to Replace	Text Replacement
<code>ServerAdmin you@your.address</code>	<code>you@your.address</code>	The administrator's electronic mail address
<code>ServerRoot /usr/local/etc/httpd</code>	<code>/usr/local/etc/httpd</code>	The full path to your <code>server_root</code> directory (<code>.../apache_x.x.x/server_root</code>)
<code>#ServerName new.host.name</code>	<code>new.host.name</code>	The server host name

- Use a text editor to replace configuration lines in your `access.conf` file as indicated in the following table. Save the file.

Line	Text to Replace	Text Replacement
<code><Directory> /usr/local/etc/ httpd/htdocs</code>	<code>/usr/local/etc/htt pd/htdocs</code>	The full path to the home directory created in step 1
<code><Directory> /usr/local/etc/ httpd/cgi-bin</code>	<code>/usr/local/etc/htt pd/cgi-bin</code>	The full path to the cgi-bin subdirectory in the Apache Web Server directory

- Use a text editor to replace configuration lines in your `srml.conf` file as indicated in the following table. Save the file when you have finished editing it.

Line	Text to Replace	Text Replacement
<code><Directory> /usr/local/etc/ httpd/htdocs</code>	<code>/usr/local/etc/htt pd/ htdocs</code>	The full path to the home directory created in step 1
<code>#ScriptAlias /cgi-bin/ /usr/local/etc/ httpd/cgi-bin/</code>	<code>/usr/local/etc/htt pd/cgi-bin</code>	The full path to the cgi-bin subdirectory in the Apache Web Server directory

- Use a text editor to add the following line to the `mime.types` file in the `server_root/conf` directory:

```
image/cgm cgm
```

- Save the file after you have finished editing it.
- Change directory to the Apache Web Server directory and start the Web server by entering the following command:

```
<path>/src/httpd-<platform>  
<path>/server_root/conf/httpd.conf
```

where `<platform>` is the platform on which you are running the Web server (for example,
`/disk1/apache_1.2.0/src/httpd-irix -f
/disk1/apache_1.2.0/server_root/conf/httpd.conf`).

Microsoft Servers

The following section describes how to configure the Microsoft Internet Information Server and Microsoft Peer Web Server.

1. Rename the `... \InetPub\wwwroot` directory to `... \InetPub\home`.
2. Create a directory called `cgi-bin` and a file called `mime.typ` in the `<path>\InetPub` directory.
3. From the Windows desktop, click **Start > Programs**.
4. From the Internet Information Services (IIS), select:
 - Microsoft Internet Information Server (Common)
 - Internet Services ManagerFor Peer Web Services, select:
 - Microsoft Peer Web Services (Common)
 - Internet Services Manager
5. In the **Internet Services Manager** dialog box, double-click the server name on the line where *www* appears under the **Service** option. The **WWW Service Properties** dialog box opens.
6. Click the **Directories** tab from the **WWW Service Properties** dialog box.
7. Change the value of **Default Documentation** from `Default.htm` to `index.html` and select the **Enable Default Document** and **Directory Browsing Allowed** check boxes.
8. Change the location of the document root directory by double-clicking the `... \InetPub\wwwroot` listing.
9. In the **Directory Properties** dialog box, change the directory from `... \InetPub\wwwroot` to `... \InetPub\home` and click **OK**.
10. Click **Add** from the **WWW Service Properties** dialog box. A new **Directory Properties** dialog box opens.
11. Select the **Virtual Directory** and **Execute** check boxes and enter `cgi-bin` for the **Alias** and `<path>\InetPub\cgi-bin` for the directory.
12. Click **OK**.
13. Click **Apply** in the **WWW Service Properties** dialog box.

14. Click **OK**.
15. If necessary, install plug-ins for your Web browser to allow viewing of CGM and VRML files.

Installing Web.Publish

1. Under **Other Products**, choose **Web.Publish** from the list of available software components. The **Specify Web Server Paths** screen appears.

During installation, you can click **Cancel** at any time to stop the installation. PTC.Setup copies files and sets file permissions.

2. Under **Web.Publish Configuration**, define the following parameters if the PTC.Setup utility cannot detect the information:

- **Cgi-bin Directory**—This is the directory from which your server has permissions to execute programs. Your Web server must be configured to recognize this directory when referenced as `http://<server>/cgi-bin`. Type the full path to the directory you want to use or browse the folder hierarchy and specify the directory.
- **Web Server MIME File**—Type the location of the MIME type file on the Web server. Or, click the folder icon and browse to and specify the file. This file, which is usually called `mime.typ` on Windows and `mime.types` on UNIX platforms, is located in the configuration directory of the Web server. PTC.Setup adds the following lines to this file:

```
image/cgm          cgm
x-world/x-vrml     wrl vrml
```

- **Document Root Directory**—Enter the location of the directory where your HTML files are stored on your Web server or click the folder icon, browse, and specify the directory. This is the directory from which your Web server reads documents when they are referenced as `http://<server>/`. This directory was specified during configuration of your Web server.

3. The **Installation Progress** screen displays the status of installation. You can also do one of the following operations:

- Click **Cancel** to cancel the installation.
- Click **Back** and review information.
- Click **Next** to install the next selected component.
- Click **Exit** to quit the installation setup utility.

The **Installation Progress** screen announces a successful installation.

8

Installing and Starting Creo Elements/Distributed Services

This chapter explains how to install and start Creo Elements/Distributed Services Manager.

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Performing Optional Configuration Steps	8 - 6
Completing the Installation	8 - 7
Starting Distributed Services Manager	8 - 8
Removing Distributed Services Manager (Windows Only)	8 - 9

Installing Creo Elements/Distributed Services

You install Creo Elements/Distributed Services after you have installed PTC License Server. You can install your product software on a license server, a license client, or on any combination of these machines.

Successful installation of your product software requires three major steps:

- Defining your installation components, including the installation directory
- Specifying the location of the license servers
- Starting the software

To view Creo Elements/Distributed Services Manager in a language other than English, you must set the language variable, `LANG`, before running your software. On Windows, set the `LANG` or the `PRO_LANG` variable or use the operating system settings to view Creo Elements/Distributed Services Manager in a language other than English.

Defining the Installation Components

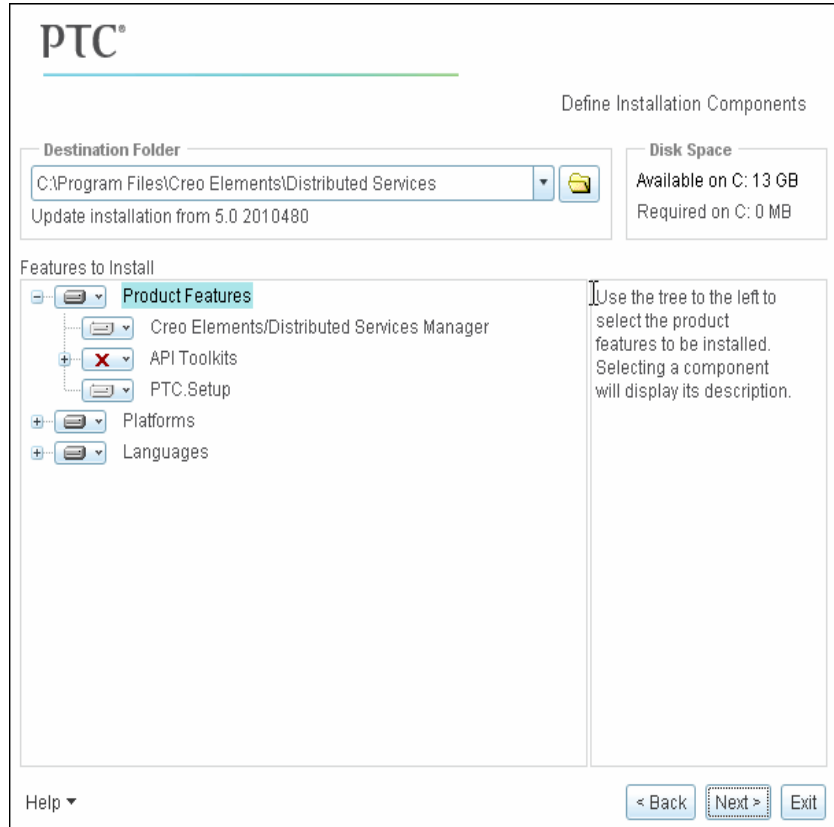
1. Start PTC.Setup using the steps outlined in Starting PTC.Setup on page 1-8.
2. Accept the PTC License Agreement (see PTC License Agreement on page 1-5). The PTC.Setup product selection screen appears.
3. Under **Other Products**, click **Creo Elements/Distributed Services**. The **Accept License Agreement** screen appears. Accept the agreement to proceed.

At this point, you must have already installed PTC License Server.

Note

When you perform an update or install a new release of a PTC product you must update your license information.


The **Define Installation Components** screen appears.




4. Define the following parameters:

- **Destination Folder**—For new installations, keep the default installation directory, for example, `C:\Program Files\Creo Elements\Distributed Services`, or specify the full path. `PTC.Setup` creates the specified new directory. You can also select a different installation directory.

For new installations, the directory that you specify must be new or empty.

- Under **Features to Install**, click  before each of the components and provide the following information:

Product Features—Choose the components that you want to install by clicking  next to the appropriate component. Select one of the following commands from the list:

Install this feature—Install the component.

Do not install this feature—Do not install the component.

The available product components are:

Creo Elements/Distributed Services Manager—Select this component to install the files to run the software. The amount of disk space required is also displayed.

API Toolkits—Select this component to optionally install the Application Programming Interface toolkits like C API and Java API. The API allows you to build Creo Elements/Distributed Services applications.

PTC.Setup—Select this component to install PTC.Setup for future configuration of the software. With this utility, you can make configuration changes without running PTC.Setup from the DVD-ROM.

Platforms—Select the platform for the installation. Your machine's architecture is automatically selected as the default platform.

Languages—Select the required languages.

5. Click **Next** after you have selected your components. The **FLEXnet license servers** screen appears.

Specifying the Location of the License Server

The PTC.Setup installation utility can access your license information on a remote license server or on a local machine. Your next task is to specify the locations of the license servers that you want this installation of your product software to access.

Note

PTC does not provide locked licenses for Creo Elements/Distributed Services Manager.

Creo Elements/Distributed Services Manager uses two different license features: `DS_MANAGER` and `DS_SERVICE`. You need both to fully use this application. The Creo Elements/Distributed Services Manager will not be able to receive any incoming tasks without the `DS_MANAGER` license feature and it will not be able to process incoming tasks without the `DS_SERVICE` license feature. After specifying the license servers, click **Server Info** to verify that both these license features are available.

If you access your license information from a machine without a valid version of FLEXnet Publisher, an error message tells you to update PTC License Server. Refer to the READ THIS FIRST document for the latest information about FLEXnet Publisher.

Specifying the License Server

Depending on the location of PTC License Server, provide PTC.Setup with the location of your licensing information.

- If PTC License Server resides on the machine on which you are installing the software, and you have used the default communications port for the license server, you do not have to specify the license server. PTC.Setup detects the server name and adds it to the **FLEXnet License Servers** list box.
- If PTC License Server resides on a different machine, you must specify the license server. See the following section Obtaining a License from a Server or Servers on page 8-5.

Specifying the License Client

If you are installing the software on a license client without PTC License Server installed, the **FLEXnet License Servers** list box is initially blank. To enable this installation to obtain floating licenses from a remote server, you must define which servers are distributing the licenses. See Obtaining a License from a Server or Servers on page 8-5.

Note

You cannot run the Creo Elements/Distributed Services Manager installer without a license.

Obtaining a License from a Server or Servers

To obtain a license from a server or servers:

1. Click **Add** in the **FLEXnet license servers** screen. The **Specify License Server** dialog box opens.
2. Click one of the server options and enter the requested information.
 - **Single license server**—Specifies a single server for the licenses. Type the host name and the communications port number if it is different from the default. This information identifies the server, for example, *aberdeen*, from which this installation obtains its license codes.
 - **Triad license server (fault tolerant)**—Specifies a Triad configuration of three Triad partners. As long as two or more partners are running, the licenses are served. If two or more partners are not functioning, the server stops serving the licenses.

Define all three license servers. Each Triad license server must have the same license file.

Note

The order of the server list (Triad Partner 1, Triad Partner 2, Triad Partner 3) must be the same on all the three license servers.

PTC recommends that the Triad license servers be at the same site. For further information on redundant license servers, see the *FLEXnet Licensing End User Guide*.

Note

Do not specify the host name of the client nodes. Specify only the license server that controls usage of the licenses. You can use the **Specify License Server** dialog box to configure the clients to access multiple servers or license files across your network.

On Windows systems, either the fully qualified server name or the IP address must identify the server.

3. Click **OK** to add the server and communications port information to the **FLEXnet License Servers** list box. For example, 7788@aberdeen.
4. Click **Next**. Further steps are optional and vary for Windows and UNIX systems.

Performing Optional Configuration Steps

Setting shortcut preferences and creating links to startup commands are optional.

Setting Shortcut Preferences (Windows Only)

Set the following shortcut preferences:

- **Shortcut Location(s)**—You can start the Creo Elements/Distributed Services Manager from the desktop, the **Start** menu, the **Start > Programs** menu, or any combination of these three options.
- **Program Folder**—If you select **Program folder** as the shortcut location, specify the name of the folder and option in the **Programs** menu.
- **Startup Directory**—Specify the name of the Creo Elements/Distributed Services Manager startup directory.

You can set the following Windows environment preferences:

- **Modify system environment for all users**—Sets the system `PATH` variable for all users on the system.

Note

You must have the administrator privileges to set the system environment variable for all users.

- **Modify current user's environment only**—Sets the user's `PATH` variable for the current user on the system. Click **Install**. The **Installation Progress** screen appears. See the section *Completing the Installation* on page 8-7.

Creating Links to Startup Commands (UNIX Only)

Command names are the script names that PTC.Setup uses to enable you to start the software from your workstation. These scripts maintain the relationship between licensing on the server and your client software command. On Windows, for example, the default command name, `ptcdsm`, results in the creation of an executable file and, optionally, a shortcut icon. This icon is linked to the executable that starts Creo Elements/Distributed Services Manager. On UNIX, the standard startup command `ptcdsm` is created in the `bin` directory.

By keeping the default **Create links to product startup commands** checked, you can create a link from the `/bin` directory in your installation directory to another directory on your UNIX system. (If this other directory is in your path environment setting, then simply entering the command name starts Creo Elements/Distributed Services Manager.)

To create a link, use the **Startup command links** screen to type the full path or browse to the directory in which you want the link created.

1. Click **Create links to product startup commands** in the **Optional Configuration Steps** screen and click **Next**. The **Startup Command Links** screen appears.
2. Type the full path or browse to the directory in which you want the link created.
3. Click **Install**. The **Installation Progress** screen appears.

See the next section to complete the installation.

Completing the Installation

The **Installation Progress** screen displays the status of installation. In this screen you can also do one of the following operations:

- Click **Cancel** to cancel the installation.

- Click **Back** and review information.
- Click **Next** to install the next selected component.

The **Installation Progress** screen announces a successful installation.

Click **Exit** to quit the PTC.Setup installation utility.

Starting Distributed Services Manager

After your license management and product software has been installed, start Creo Elements/Distributed Services Manager.

Windows

Click **Start > PTC > Creo Elements Distributed Services > Creo Elements Distributed Services Manager** to start Distributed Services Manager.

UNIX

From a UNIX prompt, navigate to the Creo Elements/Distributed Services Manager installation directory and execute the command `ptcdsm`. For example,
`/opt/ptc/creo_elements_distributed_services`
`/bin/ptcdsm.`

Removing Distributed Services Manager (Windows Only)

You can remove your Creo Elements/Distributed Services Manager software through the Control Panel. This procedure is for Windows systems only.

1. From the **Start** menu, click **Settings > Control Panel**.
2. Double-click the **Add/Remove Programs** icon. The **Add/Remove Programs** dialog box opens.
3. In the program application list, highlight the release of the software to remove.
4. Click **Remove**. You see the **PTC Uninstall** dialog box.
5. Click **Uninstall** or **Cancel** to escape. You are presented with a series of confirmation statements to remove files.
6. Click **Uninstall** to proceed. Creo Elements/Distributed Services Manager is removed.

9

Installing and Customizing an Installation from the Web

This chapter explains how to install Creo Elements/Pro and its software components using the downloadable installer and the **Custom Installer** utility.

Topic	Page
Installing the Components from the Web	9 - 2
Customizing the Installation Components	9 - 2

Installing the Components from the Web

You can install Creo Elements/Pro 5.0 and its software components directly from the Web using a downloadable installer. Select an Installer based on the platform on which you want to install Creo Elements/Pro and its components.

The Web installer eliminates the need to download or order a CD-ROM or DVD-ROM with the complete product installation. You can create a customized product CD-ROM or DVD-ROM on your computer with this utility.

You can access the Web installation utility from the PTC Web page at www.ptc.com/support/index.htm. Click **Order or Download Software Updates** under the **Popular Tools** section and follow the instructions on the **Order or Download Software Updates** page to access and download the Web installation utility. The installation utility downloads and copies files for the installation from the Web during installation.

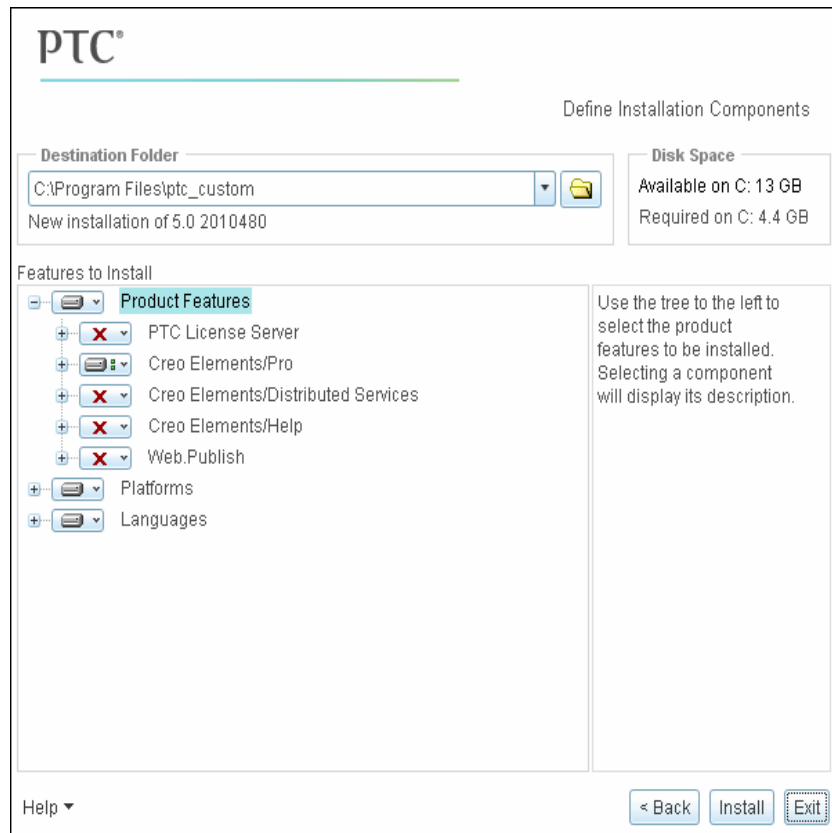
Customizing the Installation Components

You can create a custom CD-ROM or a DVD-ROM image containing components, languages, and architecture of your choice.



To perform a custom installation of Creo Elements/Pro, follow these steps:

1. Start PTC.Setup using the steps in Starting PTC.Setup on page 1-8.
2. Accept the PTC License Agreement (see PTC License Agreement on page 1-5). The PTC.Setup product selection screen appears.

3. Select **Custom Installer** from the PTC.Setup product selection screen. The **Define Installation Components** screen appears.






When you update or install a new release of a PTC product, you must update your license information.

4. Define the following parameters:
 - **Destination Folder**—For new installations, keep the default installation directory, for example, C:\Program Files\ptc_custom, or specify the full path. PTC.Setup creates the specified new directory. You can also select a different installation directory.
 - Under **Features to Install**, click  before each of the components and provide the following information:
 - Product Features**—Choose the components to install by clicking  next to each component. Select one of the following commands from the list:

Install this feature—Install the component.

Do not install this feature—Do not install the component.

Note

If a component has subcomponents, you can install them by selecting **install all sub-features** from the list that appears when you click . The icon  changes to  to indicate that the components will be installed.

The components include PTC License Server, Creo Elements/Pro, Creo Elements/Distributed Services, Creo Elements/Help, and Web.Publish. The disk space required for the product is displayed. For new installations, the directory must be new or empty.

Platforms—Select the platform for the installation. Your machine's architecture is automatically selected as the default platform.

Languages—Select the required languages.

5. Click **Install**. The **Installation Progress** screen appears.

Completing the Custom Installation

When you see the **Installation Progress** screen, you can also do one of the following operations:

- Click **Cancel** to stop the installation.
- Click **Back** to review information.
- Click **Exit** to quit the PTC.Setup installation utility.

10

Updating and Modifying an Installation

This chapter explains how to update or modify PTC software on your system. The procedures in this chapter are for updating and modifying of installations only.

Note

- Because PTC License Server was implemented in Pro/ENGINEER Release 20, you can update Pro/ENGINEER releases 20 and later with Creo Elements/Pro 5.0.
- Each time you update your software, you must update your license file. New license codes are not required if you are updating to a new build of Creo Elements/Pro within the same release.

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Modifying Creo Elements/Pro	10 - 2
Updating License Codes	10 - 3
Updating and Modifying Web.Publish	10 - 5

Updating Creo Elements/Pro

An update installation is needed in the following cases:

- Upgrading your PTC software to the next major release (for example, Pro/ENGINEER Wildfire 4.0 to Creo Elements/Pro 5.0)
- Upgrading your current release to a later release
- Updating the current datecode to include additional software

Update installations overwrite the outdated files in your Creo Elements/Pro installation directory with files from the most recent release or build.

After you install Creo Elements/Pro 5.0, you can perform incremental updates. An incremental update adds only newly selected components to the existing installation. The components of the existing installation remain unchanged.

Note

When performing incremental updates, if the current installation is corrupt, you cannot update the installation to replace missing or corrupt files for the same datecode. In such cases, uninstall and reinstall Creo Elements/Pro.

If you update an existing installation with a different release of the software, all components of the existing installation are replaced with the new components.

Before updating an installation, load and mount the software DVD-ROM and start the PTC.Setup installation utility. New license information is required when you update to the next release.

To update the product software on remote servers, node-locked machines, or license clients, repeat the steps in Chapter 4, Installing and Starting Creo Elements/Pro.

During installation, you can click **Cancel** at any time to stop the installation. PTC.Setup copies files and sets file permissions.

Modifying Creo Elements/Pro

Modifying Creo Elements/Pro implies creation of new startup commands and reconfiguration of existing software components. No new license information is required when you modify an installation.

If you are modifying an existing installation of Creo Elements/Pro, you can start PTC.Setup by using the `ptcsetup` command, in the `bin`

directory of your installation path (C:\Program Files\PTC\Creo Elements\Pro5.0\bin).

You can also rerun the PTC.Setup installation utility from the DVD-ROM and make modifications.

Obtaining a New License Pack

If you are installing a new, major release of Creo Elements/Pro, you must update PTC License Server before you install your new software.

It is highly recommended that you obtain an electronic copy of your new PTC License Pack. Using PTC.Setup, you can quickly and easily import either the new license file or append the existing license file with your new license codes.

For information on how to obtain your License Pack online, see Receiving Your License Pack on page 1-3.

Updating License Codes

You can update the PTC License Server, with your new license file when updating your PTC software.

Note

If the license server also has Creo Elements/Pro installed, the installed `ptcsetup` utility can be used to update licensing information.

Perform the following steps:

1. Start the PTC.Setup installation utility.
2. Accept the PTC License Agreement (see PTC License Agreement on page 1-5). The PTC.Setup product selection screen appears.
3. Click **PTC License Server**. The **Define Installation Components** screen appears.
4. Keep the default installation directory for PTC License Server if this is where your FLEXnet Publisher software resides.
5. Specify a license file by browsing to the folder that contains the encrypted license file. You can also drag your license file into the **License File** text box. The large box under **License File** displays the information contained in the license file.

Note

To proceed, you need the new licensing information in your PTC License Pack. If you have not already done so, request a new License

Pack. For detailed information, see Receiving Your License Pack on page 1-3.

There are several ways to update an existing license file. However, it is strongly recommended that you import the new license file. The new license file contains all of your new license codes as well as any that are still in effect. Note that importing the new license file overwrites the existing file.

You can also import and append the existing license file with your new license data. For information on how to append a license file or manually enter your license codes by typing the information or by using the License Wizard, see Chapter 3, Installing PTC License Server.

6. Optionally, click **FLEXnet Server Options** to specify the following:
 - Directory in which you can create links.
 - Server process owner (UNIX only).
 - The license server startup options.
 - The command line arguments. This gives the administrator the control over the server processes. See *FLEXnet Licensing End User Guide* for details.
 - Start the license server if the system is rebooted.
7. Click **Install** to start the installation of the license server. Upon completion, PTC.Setup starts the license server.
8. Click **Next** to install the next selected component or exit PTC.Setup.

PTC.Setup restarts the FLEXnet Publisher license server. For Triad configurations, two of the three partner machines must be running before licenses can be distributed.

Updating and Modifying Web.Publish

Updating Web.Publish requires only that you update your existing Web.Publish installation. You do not have to reconfigure your Web server.

Note

To update or modify Web.Publish, you must have write permissions on the Web server.

To update an installation of Web.Publish, start PTC.Setup using the Creo Elements/Pro DVD-ROM or the `ptcsetup` command if you installed this utility. To update the product software, repeat the steps in Chapter 7, Installing Web.Publish.

To modify an installation of Web.Publish, start PTC.Setup and change the configuration information you specified during installation.

A

License Management Software

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

Topic	Page
Overview of PTC Licensing	A - 2
Benefits of Using FLEXnet Publisher	A - 9
Understanding Timeout Parameters	A - 10
Using the Ptcflush Utility	A - 12
License Borrowing	A - 12

Overview of PTC Licensing

PTC software including PTC optional applications must be licensed for use. Licensing authorizes the installed software to run. PTC supports the following types of licenses:

- **Commercial**—Commercial licenses are not time-sensitive licenses.
- **Evaluation**—Evaluation licenses limit your use of the software to a specified time period.
- **Educational**—Educational licenses, which are used at educational institutions, are time-sensitive licenses (also known as student licenses). The objects created in an educational or student version of Creo Elements/Pro are not compatible with the commercial version.

Depending on the product, the licenses can be either locked, extended, floating, or borrowed.

- **Node-locked license**—A node-locked license restricts the use of the software to a particular machine (independent workstation). A node-locked license also restricts the `DISPLAY` setting on UNIX workstations.
- **Floating license**—A floating license can be used on any one of a group of machines on your network.
- **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 License Borrowing on page A-12 for details.

Simplified License Management Process

Creo Elements/Pro provides an easy license configuration, installation, and tracking process. This process provides flexibility in configuring your licenses and minimizes the number of generated Service Contract Numbers (SCNs). You can combine all the Creo Elements/Pro licenses and options into a single SCN. You can also assign a single SCN to the required license server.

Each PTC product is defined in the license file with a unique feature name. This license file format follows a standard approach by using the PTC License Server. Floating licenses can be configured to be acquired when you start Creo Elements/Pro.

Note

License configuration is a part of the installation process.

Visit www.ptc.com/olm/lis_faqs for the License Simplification Overview.

License Simplification—Advantages

- **Increased flexibility in managing PTC licenses**—With the license configuration process integrated into the installation process, you can alter the assignment and configuration of functions without contacting the PTC License Management team.
- **Fewer licensing rules**—Licenses are assigned to a machine through a SCN (Service Contract Number).
- **Stable number of licensing SCNs**—With no licensing requirements for SCNs, you have significantly fewer SCNs to manage. License changes do not force you to change the SCN.
- **Reduced time associated with licensing PTC products**—You and PTC do not need to research and maintain the tight control over the multiple types of licenses based on how you purchased the products originally.
- **Reduced system administration time for license changes**—With the use of common feature names in licenses, updates to the client are reduced as the licenses are moved between servers. The feature name and the startup commands remain the same.

License Simplification—General Information

- License simplification is integrated with all products of the Creo Elements/Pro family and is available in Pro/ENGINEER 2001 datecode 2001440 and later.
- To use Creo Elements/Pro 5.0, you must perform license simplification, if you have not already done so, to receive a new License Pack of version 31 or later that is compatible with Creo Elements/Pro 5.0.
- Use the License Support Web tool called **Reconfigure Software Licenses** at www.ptc.com/support/index.htm. On this page, click the **Creo Elements/Pro** tab in the **Licensing** section to access this tool. You can use this tool to upgrade to the new licensing scheme. The tool automatically consolidates the licenses into a selected Service Contract Number (SCN) and returns an updated license file.

License Simplification—Technical Information

- License simplification has changed the license file format and the PTC.Setup utility.

Note

Creo Elements/Pro 5.0 requires FLEXnet Publisher version 10.8.6.2 or later for license simplification.

- The format of the FLEXnet Publisher feature line in the license file has two changes due to license simplification:
 - The SCN is no longer a part of the feature name and is moved below the SN tag in the feature line.
 - A new classification of floating license called a License Extension is available for all locked modules and extensions. This new classification is identified by the keyword `STARTUP` in the Vendor String.

License extensions are available for checkout only at runtime, that is, when starting a base license. You cannot add or release the License Extension using the **Tools > Floating Modules** dialog box.

The following license file is an example of a license-to-run license and a floating license bundle under the license simplification scheme.

```
INCREMENT PROE_FLEX3C ptc_d 31.0 01-nov-2008 1 \  
    BC24AFC5B76BB74C9366  VENDOR_STRING="VSVER=2.0 \  
    LO=(0,3,6,7,10,17,32,34,40,45,48,51,55,61,62, \  
    66,69,71,73,77,97,106,108,115,126,127,128,133,135,137, \  
    158,163,188,191,210)" \  
    SUPERSEDE vendor_info="VIVER=1.0 EXTERNAL_NAME=" \  
    ISSUER=PTC ISSUED=01-sep-2006 NOTICE="PTC_customer" \  
    SN=SCN12344,SCN765431 SIGN2="169C A28A \  
    E97F E96E 0A3E 563B FDEB 4510 829E 4BF4 25D3 2394 \  
    0444 2FD4 6C23 0168 A8A5 AEBE 54B0 1FF6 B79B DC75 \  
    2014 A278 33CC 1B90 8647 6A12 F4D6 45BF"  
INCREMENT 10108 ptc_d 31.0 01-nov-2008 1 \  
    1C75006B8512AA3C4EBE  VENDOR_STRING="VSVER=2.0 LO=() \  
    SUPERSEDE vendor_info="VIVER=1.0 EXTERNAL_NAME=" \  
    ISSUER=PTC ISSUED=01-sep-2006 NOTICE="PTC_customer" \  
    SN=SCN12344,SCN765431 SIGN2="1A34 408F 40D4 749F 5980 \  
    2DFE 15B0 0FAF 0ED7 A5F5 DCEA E318 6529 2E27 A055 \  
    1A21 F766 D9E1 F7AB BD1F 993E B3B2 2975 E46C 06AC \  
    6304 25AD E576 9E37 8794"  
INCREMENT PROBUNDLE_10108 ptc_d 31.0 01-nov-2008 99999 \  
    4CF5E08B0EF55FF34082  VENDOR_STRING="VSVER=2.0 \  
    LO=(6,32,56,91,92,94,104,131)" SUPERSEDE \  
    ISSUED=01-sep-2006 SIGN2="14A8 7CE3 57D9 1246 D07F \  
    3610 E235 2120 4322 A874 681C 282B 5449 3150 BC5A \  
    0867 853D FE8E F8E9 9E29 6CD1 987C 4A8D 0024 BDAA \  
    AEBC 065B 9530 3AAB 441D"
```

- Using the PTC.Setup utility, you can define startup commands with additional license extensions or floating options. When you add license extensions, or floating options, to a startup command, a value is assigned to the `PROE_FEATURE_NAME` variable. This variable is defined in the Creo Elements/Pro startup command. The following example shows valid syntax for a UNIX operating system.

`PROE_FEATURE_NAME` before license simplification:

```
setenv PROE_FEATURE_NAME PROE_1274228
```

`PROE_FEATURE_NAME` after license simplification:

```
setenv PROE_FEATURE_NAME PROE_FLEX3C (10108,32)
```

In addition to checking the `PROE_FLEX3C` license, the availability of the bundle 10108 and the floating option 32 is also checked. If some bundles or options are not available, a warning appears listing them.

- During a session, if a license is lost and you try to recover the license-to-run, Creo Elements/Pro attempts to reacquire the license-to-run and all other options specified in the startup command.
- The output of the `Ptcstatus` utility provides the location of the SCN in the license file. You can use the `Ptcstatus` utility in two modes:

- **Normal mode**—Run the `Ptcstatus` utility at the command line without any arguments.

```
ptcstatus
```

- **Verbose mode**—Run the `Ptcstatus` utility at the command line with the `-v` argument.

```
ptcstatus -v
```

The following output illustrates the difference between the two Ptcstatus utility modes:

Ptcstatus Normal Mode

Displaying status for license file: 7788@static

License Server: static

License	In Use	Free
-----	-----	---
PROE_FLEX3C	Uncounted, Locked	
Foundation_Adv	0	2
Pro/SURFACE *	0	2
Behavioral_Modeler	0	2
Pro/PlasticAdvisor	0	2
Tool_Design_SET *	0	2

* = License Extension - Available only on startup

^ = Borrowed License

Ptcstatus Verbose Mode

Displaying status for license file: 7788@static

License Server: static

ptc_d FLEXnet Publisher version: 10.8.6.2

License	In Use	Free	Version	Expire Date	SCN
-----	-----	---	-----	-----	-----
PROE_FlexEng	Uncounted		31.0	None	FLEX3C Locked to 00-11-22-33-44- 55
Foundation_Adv	0	2	31.0	None	12345
Pro/SURFACE *	0	2	31.0	None	
Behavioral_Modeler	0	2	31.0	None	
Pro/PlasticAdvisor	0	2	31.0	None	

* = License Extensions - Available only on startup

^ = Borrowed License

- You can specify a name for a license instead of its default name by changing the value of the attribute EXTERNAL_NAME in the feature line of the license file.

For example, to change the feature PROE_FLEX3C to My_License_Name in a license file, specify the value of the EXTERNAL_NAME attribute as shown in the original and changed files.

Original license file:

```
INCREMENT PROE_FLEX3C ptc_d 31.0 01-nov-2008 1 \
BC24AFC5B76BB74C9366 VENDOR_STRING="VSVER=2.0 \
LO=(0,3,6,7,10,17,32,34,40,45,48,51,55,61,62, \
66,69,71,73,77,97,106,108,115,126,127,128,133,135,137, \
158,163,188,191,210)" SUPERSEDE vendor_info="VIVER=1.0 \
EXTERNAL_NAME=" ISSUER=PTC ISSUED=01-sep-2006 \
NOTICE="PTC_customer" SN=SCN912344,SCN765431 SIGN2="169C A28A \
E97F E96E 0A3E 563B FDEB 4510 829E 4BF4 25D3 2394 \
0444 2FD4 6C23 0168 A8A5 AEBE 54B0 1FF6 B79B DC75 \
2014 A278 33CC 1B90 8647 6A12 F4D6 45BF"
```

Changed license file:

```
INCREMENT PROE_FLEX3C ptc_d 31.0 01-nov-2008 1 \
BC24AFC5B76BB74C9366 VENDOR_STRING="VSVER=2.0 \
LO=(0,3,6,7,10,17,32,34,40,45,48,51,55,61,62, \
66,69,71,73,77,97,106,108,115,126,127,128,133,135,137, \
158,163,188,191,210)" SUPERSEDE vendor_info="VIVER=1.0 \
EXTERNAL_NAME=My_License_Name ISSUER=PTC ISSUED=01-sep-2006 \
NOTICE="PTC_customer" SN=SCN12344,SCN765431 SIGN2="169C A28A \
E97F E96E 0A3E 563B FDEB 4510 829E 4BF4 25D3 2394 \
0444 2FD4 6C23 0168 A8A5 AEBE 54B0 1FF6 B79B DC75 \
2014 A278 33CC 1B90 8647 6A12 F4D6 45BF"
```

Notice the text after the EXTERNAL_NAME attribute in the changed license file. With this change, the output of the Ptcstatus utility displays the new name, My_License_Name, instead of PROE_FLEX3C.

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 refer to Chapter 5, The Options File, in the *FLEXnet Licensing End User 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 10.8.6.2 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 FLEX lm license servers. For example, a Creo Elements/Pro 5.0 license can run Pro/ENGINEER 2001 software, because both releases of the software use the same PTC license server.

Note

To run Creo Elements/Pro 5.0, you must install FLEXnet Publisher 10.8.6.2. Using the license server from Pro/ENGINEER Wildfire 2.0 or earlier will not work.

FLEXnet Publisher cannot serve licenses to an installation using Pro/SERVER (Release 19 and earlier). However, Pro/SERVER and PTC License Server can run on the same machine in parallel. See www.ptc.com/appserver/cs/view/solution.jsp?n=130505 for information on downward license compatibility.

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 `lmgrd` and a vendor daemon. The `lmgrd` starts the vendor daemon (for example, `ptc_d`), that controls the usage of licenses. Flexera Software, Inc. strongly recommends that the `lmgrd` daemon program be run as a nonprivileged (not root) user on UNIX systems.

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` 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). Refer to the chapter on multiple license files in the *FLEXnet Licensing End User 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. In Creo Elements/Pro, 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>

Timeout Parameter	Value	Description
License refresh	1 minute	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.
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
```

by changing 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.

Suppressing the Regained Network License Dialog Box

Beginning with Pro/ENGINEER 2000i, you can disable the **Regained Network License** dialog box using a configuration option. In the `config.pro` file, set the `suppress_license_loss_dialog` option to

yes. The default is no. If you lose or regain your network license-to-run Creo Elements/Pro, a status message appears but a dialog box does not.

Using the Ptcflush Utility

The Ptcflush utility is used only when a license client application fails to shut down and does not break the TCP/IP connection between the license server and the client. This situation most commonly occurs when a client exits prematurely or does not respond. However, it is not mandatory to use the Ptcflush utility.

You can use the Ptcflush utility once every 10 minutes. The syntax is as follows:

```
% ptcflush feature_name hostname port handle
```

Execute the `ptcstatus` command to list the `<feature host port handle>` information and copy and paste this listing at the command prompt to run Ptcflush.

To flush a license in use, for example, type

```
% ptcflush PROE_FLEX3C titan 7788 101
```

using the feature name and not the user-defined name if you assigned an alias.

Note

On UNIX systems, you can use root privileges to flush user licenses. However, you must flush licenses individually. You cannot flush multiple licenses simultaneously.

License Borrowing

With license borrowing for all supported platforms, you can use Creo Elements/Pro applications on clients without being connected to the license server. The license server must be running FLEXnet Publisher 10.8.6.2 or later and must use the License Pack of version 31 or later. With license borrowing, you can:

- Choose from a pool of available licenses
- Specify the expiration date of the license
- Work remotely on design and manufacturing projects

To enable license borrowing, include a borrowing keyword in the feature line of your license file. You can configure your licensing on your license server and not through License Management. See the *FLEXnet Licensing*

End User Guide for details. An example of a license file with the license borrowing keyword follows:

```
INCREMENT PROE_FLEX3C ptc_d 31.0 01-nov-2008 1 \  
    BC24AFC5B76BB74C9366 VENDOR_STRING="VSVER=2.0 \  
    LO=(0,3,6,7,10,17,32,34,40,45,48,51,55,61,62, \  
    66,69,71,73,77,97,106,108,115,126,127,128,133,135,137, \  
    158,163,188,191,210)" SUPERSEDE vendor_info="VIVER=1.0  
EXTERNAL_NAME=" \  
    ISSUER=PTC ISSUED=01-sep-2006 BORROW=264 NOTICE="PTC_customer" \  
    SN=SCN12344,SCN765431 SIGN2="123F 6758 5955 8D78 \  
    F718 4995 3F6F EA5B F56A 2759 6A9A F6B3 773A F2B9 \  
    0F31 0219 EC28 6D28 0345 4971 B5C4 8835 7E88 28A6 \  
    7581 8191 BB82 CB3D 8BF8 8282"
```

Initiating License Borrowing

You can initiate license borrowing by:

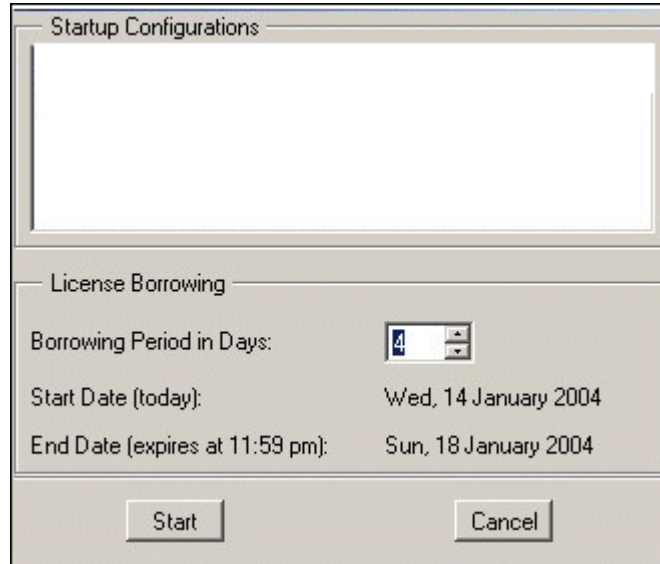
- Executing a batch file available in the \`<creo_loadpoint>\bin` directory
- Specifying the command-line argument to borrow licenses as discussed in the *FLEXnet Licensing End User Guide*.

Using the `proe_borrow.bat` File

Connect to the appropriate PTC License Server and use the `proe_borrow.bat` file to initiate license borrowing. Use the following procedure:

1. Browse to the \`<creo_loadpoint>\bin` directory and run the `proe_borrow.bat` file on Windows. On UNIX, this file is called

proe_borrow. This file opens a dialog box in which you can specify a startup configuration.



2. Under **Startup Configurations**, select the required startup configuration.
3. Specify the number of days that you want the borrowed license.

The start date for borrowing the license is today's date. The end date is based on the number of days that you request.

Note

By default the maximum borrowing duration is 5 days. Through an environment variable (LM_BORROW_DURATION=0-14), it is possible to get to a maximum of 14 days.

4. Click **Start** to start Creo Elements/Pro in the Borrow Configuration mode.
5. After starting Creo Elements/Pro, click **Tools > Floating Options** to select the additional floating options that you would like to add to the borrowed license.
6. Configure the borrowed license to create the required borrowed license.
7. Exit Creo Elements/Pro.

On Windows, the license is created on the local machine in the registry under current user. On UNIX, the license is created in the \$HOME/.flexlmrc directory. The LM_BORROW environment

variable is created and contains all your borrowed licensing information.

8. Disconnect from the license server and use the borrowed license.

Note

A borrowed license can contain multiple borrowed configurations that can expire at any time. Each time you start Creo Elements/Pro using the borrowed license, Creo Elements/Pro checks for and removes the expired borrowed license files. You can borrow licenses again, if required.

Determining the Borrowed License Status

You can run Ptcstatus for the following information on borrowed license status:

- Licenses that have been borrowed
- Expiration for each of the borrowed licenses
- License server information for borrowing licenses
- Local borrowed license information

B

ladmin License Server Manager

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

Topic	Page
Overview of <code>ladmin</code>	B - 2
Differences between <code>lmgrd</code> and <code>ladmin</code>	B - 2
Before You Install <code>ladmin</code>	B - 3
Installing <code>ladmin</code>	B - 3

Overview of lmadmin

lmadmin is a license server manager that supports a GUI client 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 versions. Use lmadmin for the following capabilities:

- Perform server configurations and administration functions from the user interface.
- 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.
- Retain the same configuration option settings for all the sessions. That is, if you change the settings and restart the application, the previous settings remain in effect.
- Display alerts and monitor status of various processes. For example, an administrator can configure lmadmin to display alerts for license expiration or nonavailability of licenses.

See the chapter lmadmin License Server Manager of the *FlexNet Publisher License Administration Guide* for more information.

Differences between lmgrd and lmadmin

The lmadmin license server manager combines all the features of the lmgrd license server manager with a Web-based administrative interface. The differences between the two license server managers follow:

- lmgrd uses a command-line interface whereas lmadmin supports a browser-based client connection over HTTP.
- Using lmadmin you can retain the same configuration settings for all your sessions. If you use lmgrd, the settings are session-based.

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

With an installation of PTC License Server, you can use either `lmgrd` or `lmadmin` as your license server manager. Follow the instructions in the next section to install PTC License Server with `lmadmin`.

Before You Install `lmadmin`

If you have not previously installed `lmgrd` as your license server manager, proceed to the next section. If you have already installed `lmgrd` and want to migrate to `lmadmin`, follow these steps before the migration:

- Shut down the `lmgrd` service if you have configured it as a system service.
- Shut down any `lmgrd` processes running on the system.

Installing `lmadmin`

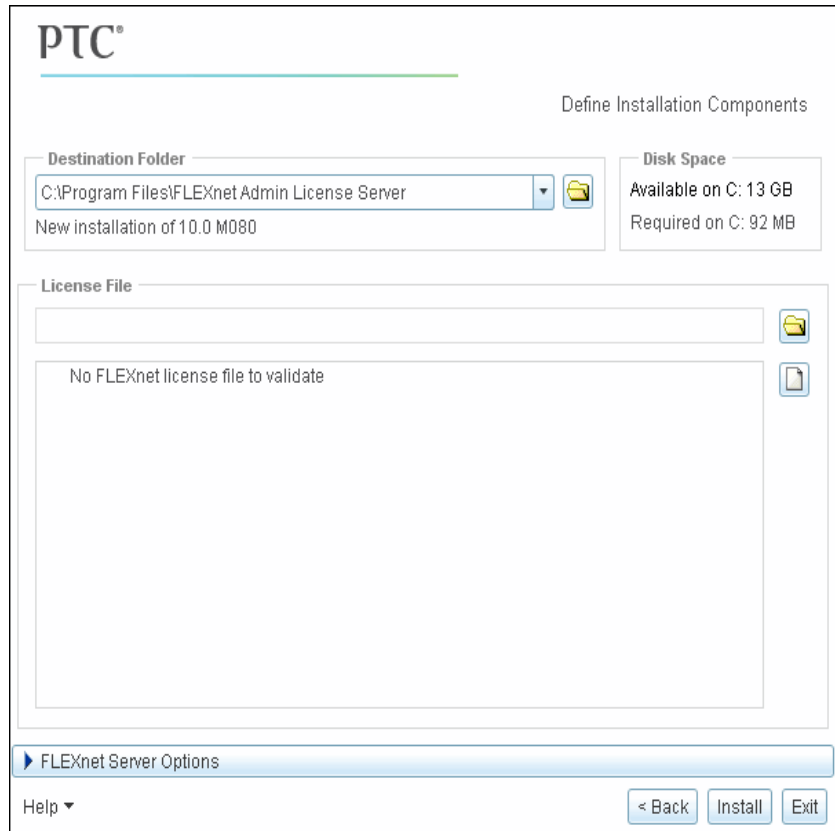
To install `lmadmin`, follow these steps. During the installation, you can click **Cancel** at any time.

1. Obtain an online copy of your License Pack and save your license file as an ASCII file. For more information, see Receiving Your License Pack on page 1-3.
2. Download the ZIP file with installation executables for PTC License Server from [this location](#) based on your platform.
3. Extract the contents of the ZIP file to a local folder.

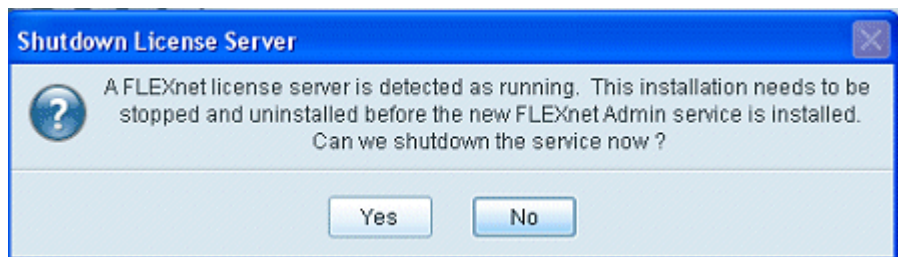
4. Browse to the folder and run the installer to start PTC.Setup. The **PTC License Server** installation screen appears.



5. Click **PTC License Server** to start the installation of FLEXnet Publisher. The **Define Installation Components** screen appears.



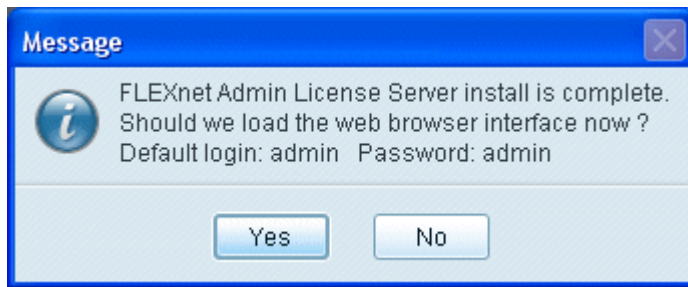
If you have not uninstalled a previous installation of PTC License Server with `lmgrd`, the following message appears when you click **PTC License Server**:



Click **Yes** to stop and uninstall the previously installed FLEXnet

license server. Before uninstalling, the `license.dat` and other option files are backed up.

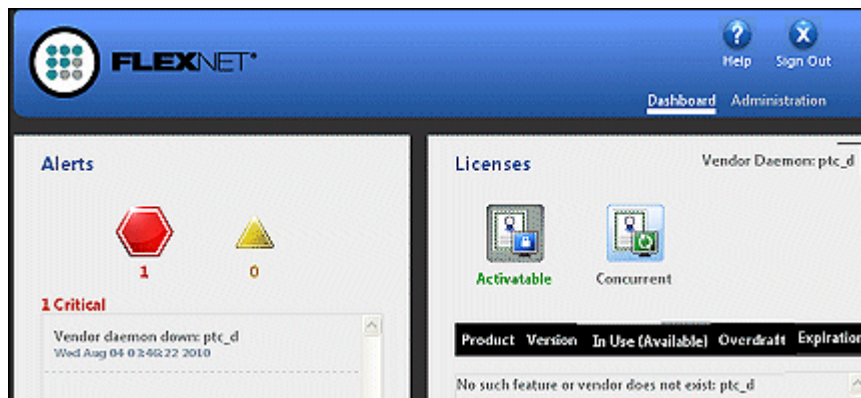
6. Follow steps 4 through 6 in the section **Installing PTC License Server** on page 3-3 to define the parameters on the **Define Installation Components** screen.
7. Install PTC License Server. After the installation, the following message appears:



8. Click **Yes** to open the FLEXnet License Administrator Web interface and configure the license server administration settings. Click **No** to configure the settings at a later time and close the message window.
9. Click **Exit** to close the installer.

Working with Flexnet License Administrator Web Interface

FLEXnet License Administrator Web interface has two main areas, the **Dashboard** and the **Administration** areas.



- **Dashboard**—Displays the current activity of the license server manager and alerts.
- **Administration**—A password-protected area. Displays the license management system for you to configure. You must have sufficient administrative privileges to access the information in this area.

Click the **Help** button on the Flexnet License Administrator interface for information on the FLEXnet License Administrator interface components.

C

System Administration Information

This appendix describes general system administration information, including system management utilities and machine configuration settings.

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Temporary Directories

Creo Elements/Pro uses the temporary directory as defined by your system environment to store Creo Elements/Pro parts. You can modify the location of the temporary directory by setting the temporary directory environment variable in the Creo Elements/Pro startup window before you run Creo Elements/Pro. On Windows systems, this environment variable is TMP. On UNIX systems, this environment variable is TMPDIR.

System Management Utilities

This section briefly describes Creo Elements/Pro and FLEXnet Publisher utilities. The files listed in the next two tables, Creo Elements/Pro Utilities and FLEXnet Publisher Utilities, are installed into the bin directory of your installation directory. On UNIX systems, symbolic links may have been made to these files from a user-specified directory. Further information can be found in the *FLEXnet Licensing End User Guide*.

Creo Elements/Pro Utilities

You can execute the Creo Elements/Pro commands, listed next, from the `\<creo_loadpoint>\bin` directory when the license server is running. The load point is the directory where the software is installed.

Command	Function
<code>ptchostid</code>	Returns the PTC Host ID with required formatting used for FLEXnet Publisher license codes.
<code>creo</code>	Runs Creo Elements/Pro. Can be user-defined.
<code>ptcflush</code>	Releases licenses. This may be necessary if there are network problems. Formerly called <code>proflush</code> .
<code>ptcsetup</code>	Runs PTC.Setup from disk. Allows you to reconfigure an existing installation without running PTC.Setup from the DVD-ROM. The command in the <code>bin</code> directory is created during installation. You use <code>ptcsetup</code> for modifications or reconfigurations only. It cannot install your PTC software.
<code>ptcstatus</code>	Displays information on the current usage and availability of licenses.

FLEXnet Publisher Utilities

You can execute the FLEXnet Publisher commands, listed next, from the `\<flexnet loadpoint>\flexnet\bin` directory when the license server is running.

Command	Function
<code>lmtools</code>	Runs the PTC License Management tools utility <code>lmtools.bat</code> on Windows systems only. Functionality includes customer configuration of the FLEXnet Publisher service, starting and stopping the <code>lmgrd</code> license server(s), server and licensing status, as well as diagnostic tools. On Windows systems, the shortcut icon created in Start > Programs > PTC License Management tools starts the <code>Lmtools</code> graphical user interface. The <code>lmutil</code> script provides the same information in text-based format for UNIX users
<code>lmutil</code>	Provides optional arguments to check the server and licensing status. Refer to the <i>FLEXnet Licensing End User Guide</i> .
<code>ptcshutdown</code>	Shuts down a license server process for administrative maintenance of the license server. On Windows systems, you can also use the FLEXnet Control Panel to stop a license server process.
<code>ptcstartserver</code>	Starts a license server process. On Windows systems, you can also use the FLEXnet Control Panel to start a license server process. See License Administration Tools, in the <i>FLEXnet Licensing End User Guide</i> .

Lmtools Utility (Windows Only)

With *FLEXlm* 7.0 and later, a `lmtools.bat` program is added to the license server installation directory on Windows platforms only. The `lmtools.bat` program runs the PTC License Management tools utility that is started from the **Start > Programs > PTC License Server tools** menu. Functionality includes starting and stopping the license server from the **Start/Stop/ReRead** tab.

Machine-Specific Configuration

The following section describes machine configuration changes to run PTC software on Sun Microsystems machines. For complete details, see www.sun.com/technical-computing/ISV/PTCFaq.html.

Overlay Planes

A Creator3D or Elite3D graphics adapter is required for the use of overlay planes. In addition, the overlay visual must be the default visual. Use the command:

```
ffbconfig -propt
```

to determine if overlay is the default visual. If overlay is not your default visual, you must set overlay visual as the default visual.

How to Set Overlay Visual as the Default Visual

1. In a command window, log in as root on the machine to be set.
2. Enter the command:

```
ffbconfig -defoverlay true
```

3. Log out as root from the command terminal and restart the session manager.

Setting the Gamma Correction Value

Default colors can appear washed out in a Creo Elements/Pro installation on a Sun workstation. To rectify this, run the `set_solaris_gamma` script in the `<creo_loadpoint>/bin` directory. Running the `set_solaris_gamma` script sets the gamma correction value for the graphics card to 1.5 from the default value of 2.20.

Supported Graphics Modes

You can run Creo Elements/Pro in the following graphics modes:

- X Windows
- GDI
- OpenGL
- Direct3D 10

Set the `graphics` configuration option to `opengl`, `x_windows`, `win32_gdi`, or `d3d` to specify the optional graphics environment, which is platform-specific. The default value is `opengl`.

For Creo Elements/Pro graphics support information, refer to the PTC Hardware Configuration Notes at www.ptc.com/WCMS/files/77552/en/proewf5.pdf

Creo Elements/Pro in X Windows

Information on using Creo Elements/Pro in a UNIX X Windows environment follows.

Display Devices Supported

Creo Elements/Pro operates in an X Windows environment on computers from Sun Microsystems, Inc.

Using the **Node-locked License** option, the display device must be the same machine as the license (the X-client Creo Elements/Pro and the X-server must be running on the licensed machine).

However, for those who use floating licenses, the display device and the computer running Creo Elements/Pro can be different. Only the display device counts as a user against the total number of licenses available.

You can also use an X-terminal as the display device. In this case, you must ensure that the display terminal can properly display Creo Elements/Pro.

Running on a Remote Computer

In a floating license configuration, all the computers listed previously may be used as X display devices. To run Creo Elements/Pro on one machine and the X-server on another, follow the instructions from the manufacturer.

How to Start Creo Elements/Pro on a Remote System

1. On the X-server, open a window and set `xhost+`.
2. From the X-server, log in remotely to the machine on which you want to execute Creo Elements/Pro.
3. On the remote machine, type:

If using `csh`:

```
setenv DISPLAY <xserver_machine_name>:0
```

If using `sh`:

```
DISPLAY=<xserver_machine_name>:0  
export DISPLAY
```

4. Run Creo Elements/Pro.

X-Terminal Path Name and Command

When running in an X Windows environment, Creo Elements/Pro needs the X Windows terminal emulator command used to open system

windows for editing relations, entering datum point arrays, and so on. The full path name and the command must be provided for your system. For example, the command for Solaris is `shelltool`. Enter the path and name: `/bin/shelltool`.

Use the configuration file option:

```
terminal_command_path_and_name
```

OpenGL Support

You can enable remote display of OpenGL to start Creo Elements/Pro only when you work across two computers that use the same UNIX operating system. PTC recommends that you run the same OpenGL library versions on both these computers. If the OpenGL library versions are different on two computers with the same UNIX operating system, Creo Elements/Pro does not use OpenGL to start up but instead uses X Windows. In this case, a warning message in the Creo Elements/Pro Startup window indicates that performance may be affected.

Note

- Remote display of OpenGL is possible only on platforms that are certified to run Creo Elements/Pro.
- PTC recommends a fast ethernet connection to use OpenGL remotely. This enhances the performance of graphical operations such as preselection highlighting, sketcher, and so on.

On Windows, set the `use_software_opengl` configuration option to `yes` to invoke the OpenGL application locally on your machine. PTC recommends that you use this configuration option in place of `WIN32_GDI` to address issues related to graphics cards.

Direct3D 10 Support

You can run Creo Elements/Pro in the Direct3D 10 graphics mode on the Windows Vista operating system. Set the value of the `graphics` configuration option to `d3d` to run Creo Elements/Pro in this mode. Direct3D 10 is an API provided by Microsoft for rendering 3-dimensional graphics accurately.

General Plotter Information

PTC supplies software that writes plot files in plotter format. You can specify a plot command in the `config.pro` file. The physical connection, however, is the responsibility of the computer and plotter vendors, because they are hardware and system software experts. Depending on the hardware configuration, the system software may require a different setup.

In general, connection to the plotter is by RS232 cable with null modem configuration (pins 2 and 3 reversed, pin 7 ground). If this is the connection, the computer and plotter communicate using xon and xoff. It is usually necessary on UNIX to establish an entry in `/etc/printcap` to let UNIX know the characteristics of the connection. A typical printcap entry is as follows:

```
hp7585|plotter|hp:\
:lp=/dev/ttyb:sd=/usr/spool/lpd:br#9600:\
:sf:pl#0:pw#0:mx#0:sh:lf=/usr/adm/lpd-errs:
```

If you have difficulty in set up, contact the customer support department of your hardware vendor.

Note

Creo Elements/Pro does not support any HPIB interfacing.

Loading and Mounting the DVD-ROM on UNIX

Most UNIX systems mount the DVD-ROM automatically after it is loaded into the DVD-ROM drive. For machines that do not mount automatically, the following instructions explain how to load and mount the DVD-ROM locally and remotely.

Note

For information specific to how PTC products run on Sun Microsystems hardware, visit www.sun.com/technical-computing/ISV/PTCFaq.html. This site also describes how to edit the `config.pro` file to adjust Creo Elements/Pro fonts.

On Sun Solaris, after you have mounted the DVD-ROM, you cannot unmount it for the following reasons:

- You have started PTC.Setup using the CDE File manager.
- The PTC.Setup help is open.

Determining the SCSI ID of the DVD-ROM Drive

You specify the SCSI identification number of your DVD-ROM drive when you mount the DVD-ROM file system to your UNIX workstation. If you already know the SCSI ID of your DVD-ROM drive, proceed to the next section, How to Load and Mount the DVD-ROM Locally. For external DVD-ROM drives, the SCSI ID can be found on the back of your DVD-ROM drive. Look for a single-digit switch. The displayed number is the SCSI ID number.

See the Hardware Configuration Notes for information on supported systems and platforms.

How to Load and Mount the DVD-ROM Locally

1. Turn on the DVD-ROM drive and insert the DVD-ROM.
2. If the `/dvdrom` directory does not already exist, create it using the command `mkdir /dvdrom`.
3. Use the following command to mount the DVD-ROM drive:

```
mount -F hsfs -o ro /dev/dsk/c0t6d0s0 /dvdrom
```

Where, `c0t6d0s0` varies for different workstations depending on the device to which the DVD-ROM is attached.

How to Load and Mount the DVD-ROM Remotely

Note

Mount the DVD-ROM drive using NFS version 2. In workstations that support NFS 3, add an extra argument to the mount command to force the use of NFS 2.

1. Load and mount the DVD-ROM on the remote UNIX system to which the DVD-ROM drive is connected. Use the procedure in the section How to Load and Mounting the DVD-ROM Locally.
2. Before a remote UNIX system can allow access to the DVD-ROM from your local UNIX workstation, you must export the DVD-ROM file system. Add a line to a file on your local UNIX workstation and, in some cases, you must run a command. See the table below.
3. Use the following table for information to add the line to the file in the File to Edit column for the Sun Solaris platform. You need write permissions to edit the files.
4. After you have made the changes, if necessary, run the command in the Command column.

Platform	File to Edit	Line to Add	Command
SUN	<code>/etc/dfs/dfstab</code>	<code>share -F nfs -o ro /dvdrom</code>	<code>shareall</code>

5. If the `/dvdrom` directory does not already exist on your local UNIX workstation, create it using the following command:

```
mkdir /dvdrom
```


- You mount the DVD-ROM mount directory from the remote UNIX system to your local workstation. Use the following table to run the command for the Sun Solaris platform. In the command, substitute <node> with the name of the remote UNIX system to which the DVD-ROM drive is connected and substitute <dvdmount> with the DVD-ROM mount directory used on the remote UNIX system.

Platform	Remote Mounting Command
SUN ¹	<code>mount -o ro,hard <node>:<dvdmount> /dvdrom</code>

¹ If you experience problems using PTC.Setup from a remote-mounted DVD-ROM, you can try remounting the remote DVD-ROM for Sun systems using the command

```
mount -o ro,hard,vers=2 <node>:<dvdmount> /dvdrom
```

Installing Libraries

Some applications come with libraries of models or textures for use in your models. For Creo Elements/Pro, all libraries except the Electrical Symbol, the Piping and Heating Symbol, and graphic libraries must be installed under the Creo Elements/Pro installation directory. The following subdirectories are automatically created in this common installation directory for each of the following libraries:

- **Objlib**—Basic Library
- **Connlib**—Connector Library
- **Fittinglib**—Pipe Fitting Library
- **Hmflib**—Human Factors Library
- **Mfglib**—Tooling Library
- **Moldlib**—Moldbase Library

Note

The Electrical Symbol Library and the Piping and Heating Symbol Library must be installed into a directory called `library_syms` in the `symbols` directory of your Creo Elements/Pro installation directory.

If you are updating your installation, make sure that the installation directory you have chosen does not contain any models that you want to save. During an update installation, any library files you enter in the installation directory are deleted and replaced with the current version of the library you install. You might want to back up your existing library installation before updating it.


To install a library:


1. After starting PTC.Setup with the library DVD-ROM, choose the library name from the list of software components.
2. Click **Next** to confirm that you are installing that library.
3. Click **Next**.
4. Specify the following information:

- **Destination Folder**—A default installation directory has been specified.

Note

For new installations, the directory you enter must be a new directory. If you specify an existing installation directory, PTC.Setup updates the software in that directory. A subdirectory under the main installation directory is recommended (for example, `C:\ptc\libraries`).

- Under **Features to Install**, click  before each of the components and provide the following information:

Product Features—Choose the components that you want to install by clicking  next to the appropriate component. Select one of the following commands from the list:

Install this feature—Install the component.

Do not install this feature—Do not install the component.

Platforms—Select the platform for the installation. Your machine's architecture is automatically detected and selected as the default platform.

Languages—Select the required language.

5. After you have selected your components, click **Install**. The **Installation Progress** screen displays the status of installation. You can also do one of the following operations:
 - Click **Cancel** to cancel the installation.
 - Click **Back** and review information.

Further configuration is required to access the installed library files.

Using the ModelCHECK Metrics Tool

The ModelCHECK Metrics Tool is a web-based utility for tracking data quality, standards, and best practices. This tool uses the metric file output

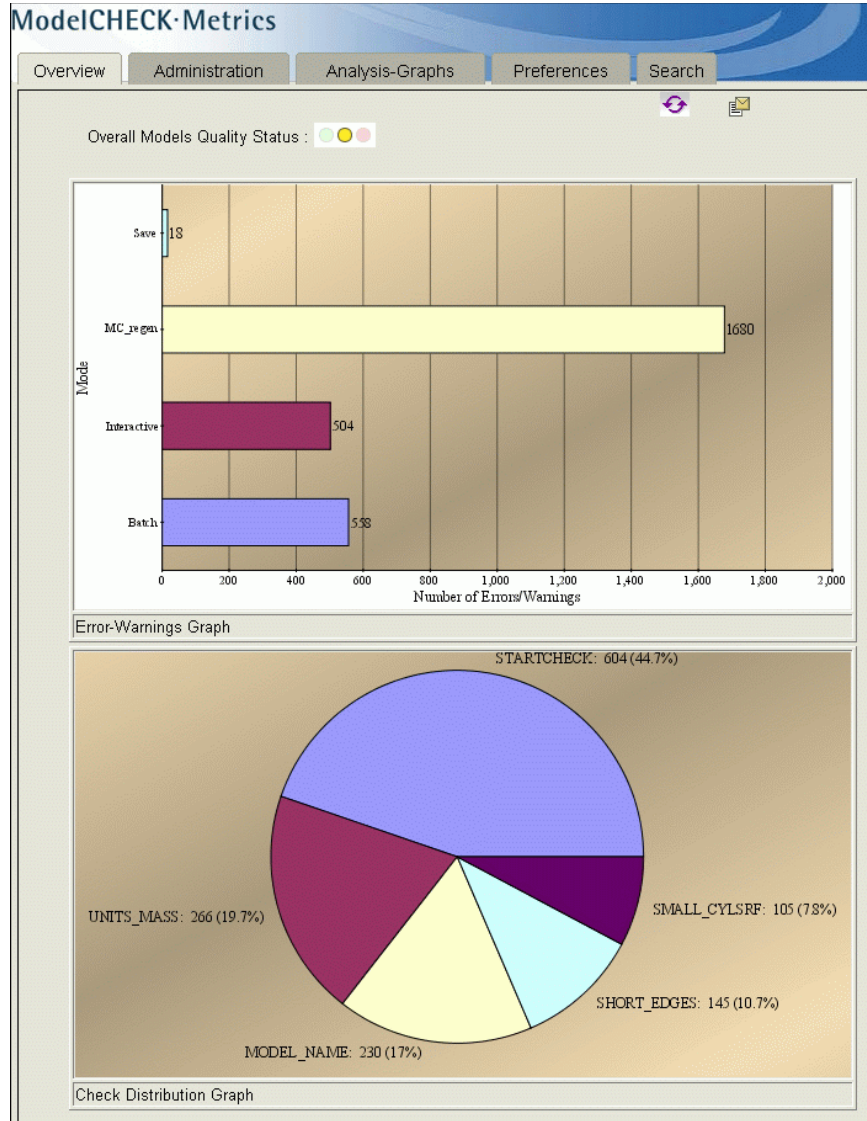
from ModelCHECK to calculate quality trends and enables you to represent the ModelCHECK analysis results graphically. Although specific licenses are not required, an administrator needs a Creo Elements/Pro license and an executable while installing the tool.

Note

You cannot install the Metrics Tool using the PTC.Setup utility.

You can do the following operations using the ModelCHECK Metrics Tool:

- Generate a high-level picture of design trends.
- Define quality goals based on the checks, errors, and warnings.
- Define critical checks for monitoring and highlighting the number of errors and warnings for these checks.
- Generate a report of failed checks.
- Define the quality goals based on each check and highlight the progress of the goals. Evaluate the results using indicators. Display the results for a user, group of users, or an organization.
- Enforce standards and best practices for better data quality.
- Specify the experience level of a user using specific color codes.
- Define what a user will see as overview information while navigating the ModelCHECK Metrics Tool.
- Display information pictorially as charts and graphs.
- Monitor the quality of models for specific ModelCHECK failures.
- Resolve issues by providing adequate training and solutions.



Refer to the ModelCHECK online Help in the Help Center for details.

Requirements for Installation

You can install the ModelCHECK Metrics Tool on both Windows and UNIX if you use a supported configuration for the workstation or for the server. See www.ptc.com/WCMS/files/77552/en/proewf5.pdf for supported platforms. You are not required to run ModelCHECK to access the tool. You must have a Web server to access the Metrics Tool information. After you install and start the Metrics Tool, you can view the ModelCHECK results using Mozilla 1.6 and later or Internet Explorer 6.0.

How to Install the ModelCHECK Metrics Tool

To install the ModelCHECK Metrics Tool, install and set up the Tomcat server, create a correct context configuration file, and deploy the Metrics Tool. Before you begin, perform the following steps:

- Install `j2sdk-1_4_2_xx` through a local installation or download it from www.java.sun.com.
- Install Tomcat for Java Web Services Developer Pack through a local installation or from www.java.sun.com.
- Set the `CATALINA_HOME` environment variable to point to the top-level directory of the Tomcat installation: `tomcat50-jwsdp`. This environment variable identifies the Tomcat installation directory and sends the information to the server.
- Set `JAVA_HOME` to the `j2sdk-1_4_2_xx` installation directory to point Tomcat to the Java path. If you do not set this variable correctly, Tomcat fails to handle the Java Server pages correctly.

Note

This variable must list the Java Development Kit (JDK) installation directory and not the `bin` sub directory.

- Install the Web server to view the charts and graphs to be stored and displayed in the ModelCHECK Metrics Tool. You can use either Internet Information Services (IIS) or download the Web server from www.apache.org.

How to Start the Tomcat Server and Deploy the Metrics Tool

After you have installed the Tomcat server, ensure that the file `tomcat50-jwsdp\conf\tomcat-users.xml` has all the required settings to access the Metrics Tool. A sample `tomcat-users.xml` file follows:

```
<?xml version='1.0' encoding='utf-8'?>
<tomcat-users>
<role rolename="tomcat"/>
```

```
<role rolename="role1" />
<role rolename="manager" />
<role rolename="admin" />
<user username="tomcat" password="tomcat"
roles="tomcat" />
<user username="both" password="tomcat"
roles="tomcat,role1" />
<user username="role1" password="tomcat"
roles="role1" />
<user username="admin" password="admin"
roles="admin,manager" />
</tomcat-users>
```

In the above sample file, the **Tomcat Web Server Administration Tool** defines the roles.

1. Start Tomcat using `tomcat50-jwsdp\bin\startup.bat`.
2. Open a browser and type `http://<machine_name>:port/admin`, where `machine_name` is the name of the machine with the Tomcat installation.
3. In the next screen, provide a user name and type the password as `admin`.
4. Using the **Tomcat Web Server Administration Tool**, create a new data source by providing details of the database to use for the Metrics Tool and save the changes. Use Oracle or MSAccess as a database for the Metrics Tool.

Note

The information in the new data source must match that given in the `new_admin.xml` file that is used to deploy the Metrics Tool.

5. Ensure that you have created the context configuration file (`new_admin.xml`) to enable the deployment of the Metrics Tool. See **Context Configuration File** for details.
6. In **Tomcat Web Application Manager** window, provide the path to the context configuration file (`new_admin.xml`) and the WAR file (`new_admin.war`) under the **Deploy** area.
7. Click **Deploy** to start the Metrics Tool.
8. Open a browser. Specify the name of the server on which the Metrics Tool is deployed along with the port and `new_admin` as the name of the tool.

Note

You can specify any name for the tool. But, ensure that the specified tool name matches the name specified in the context configuration file.

Context Configuration File

The `new_admin.xml` file is a context configuration to deploy the ModelCHECK Metrics Tool. The `new_admin.xml` file and the `new_admin.war` file are stored in the following directory:

```
$PTCSRC/modchk/mc_admin/new_admin/dist/
```

Parameter Name	Description
<code>dbname</code>	Name of the database
<code>dbtype</code>	Type of the database, like Oracle or MSAccess
<code>adminserver</code>	Machine name of a server for the Metrics Tool
<code>Adminport</code>	The port number type of the server for the Metrics Tool
<code>Imagedir</code>	Path to the image directory
<code>ImagedirLink</code>	The link to the image directory for storing graphs
<code>collect_mc_db_interval</code>	Time interval (in hours) after which the ModelCHECK database is updated (minimum 1 hour)

You must specify the path to the image directory in the `new_admin.xml` file. The generated graphs are stored in the image directory. The value for the `Imagedir` in the `new_admin.xml` file must be the same as that of the `DocumentRoot` variable in your Web server configuration file. A sample context configuration file follows:

```
<Context path="/new_admin" docBase="new_admin.war" debug="5"
reloadable="true" crossContext="true" privileged="true">
  <Resource name="jdbc/mcadmin" scope="Shareable"
type="javax.sql.DataSource" />
  <ResourceParams name="jdbc/mcadmin">
    <parameter>
      <name>maxWait</name>
      <value>50000</value>
    </parameter>
```

```

    <parameter>
    <name>maxActive</name>
    <value>100</value>
  </parameter>
    <parameter>
    <name>password</name>
    <value>yourpass</value>
  </parameter>
    <parameter>
    <name>url</name>
    <value>jdbc:oracle:thin:@machine1:1521:mcadmin</value>
  </parameter>
    <parameter>
    <name>driverClassName</name>
    <value>oracle.jdbc.driver.OracleDriver</value>
  </parameter>
    <parameter>
    <name>maxIdle</name>
    <value>100</value>
  </parameter>
    <parameter>
    <name>username</name>
    <value>yourpass</value>
  </parameter>
</ResourceParams>
<ResourceLink name="jdbc/mcadmin" global="mcadmin" />
<Manager
className="org.apache.catalina.session.PersistentManager"
debug="0" saveOnRestart="false" maxActiveSessions="-1"
minIdleSwap="-1" maxIdleSwap="-1" maxIdleBackup="-1">
  <Store className="org.apache.catalina.session.FileStore" />
</Manager>
<Parameter name="dbname" value="mcadmin" />
<Parameter name="dbtype" value="oracle" />
<Parameter name="adminserver" value="machine1" />
<Parameter name="adminport" value="80" />
<Parameter name="imagedir"
value="d:\projects\mcadminwork\admingraphs" />
<Parameter name="imagedirLink" value="admingraphs"/>
</Context>

```


In above file, the value of `Imagedir` is `d:\projects\mcadminwork\admingraphs`. Hence, if you configure Apache as your Web server, the value of `DocumentRoot` will be `d:\projects\mcadminwork`.

Database for the Metrics Tool

The Web-based ModelCHECK Metrics Tool generates graphs and reports based on the metric files that ModelCHECK writes after every session. You must create the following database tables to initialize the ModelCHECK database:

- `mc_database`
- `trng_database`
- `user_database`

Use Oracle or MSAccess to create the database table. Additionally, you must also create a database table named `processedfiles` for the processed files. These processed files ensure that the data is populated correctly from the `.txt` metric files into the database tables. They avoid repeated entries and ensure that the database tables are up-to-date.

The command line utility `create_mcadadmin_db.jar` available at `$PTCSRC/modchk/mc_admin/new_admin/dist/` enables you to populate the database tables with information from the `.txt` metric files. The utility reads and interprets the data in the metric files and automatically updates the relevant database tables. Each row of a metric file is appropriately included in the corresponding database tables. The names of the `.txt` metric files and the date of inclusion of their contents into the relevant database tables are also simultaneously written in the `processedfiles` database table so that information is not repeated in the database tables. A sample `processedfiles` table follows (syntax for Oracle):

```
CREATE TABLE processedfiles
(filename VARCHAR(256),
mergedate DATE);

INSERT INTO processedfiles
VALUES('jsmith_200503091600.txt',
to_timestamp('2002-01-11 11:14:07', 'YYYY-MM-DD
HH24:MI:SS'));
```

Using create_mcadadmin_db.jar to Update the Database Tables

1. Set the CLASSPATH environment variable to the path of the create_mcadadmin_db.jar file as shown below.

```
set CLASSPATH=<path to file_1>;<path to file_2>  
(Windows) or
```

```
setenv CLASSPATH .:<path to file_1>:<path to  
file_2> (UNIX)
```

For example:

- On Windows, use the following command:

```
set  
CLASSPATH=d:\wildfire\modchk\mc_admin\new_admin\  
dist\create_mcadadmin_db.jar;  
d:\wildfire\modchk\mc_admin\new_admin\dist\class  
es111.jar
```

- On UNIX, use the following command:

```
setenv CLASSPATH  
./usr2/wildfire/classes111.jar:/usr2/wildfire/c  
reate_mcadadmin_db.jar
```

Note

The classes111.jar file is required to manually execute the create_mcadadmin_db.jar utility when an Oracle database is used.

2. Update the user_database table using the following command:

```
<java home> <class name to update user_database>  
<database type> <database driver> <database url>  
<username> <password> <path to CSV file>
```

where the CSV file has the following sample information and structure:

```
usr1,Devid,Boon,K,CADQA,23453,3380,09-04-2000,QA,d:  
\projects\devid  
usr2,Alan,Donald,Y,CADDESIGN,23367,3181,10-02-2000,  
DESIGN,d:\projects\alan
```

Note

You must manually execute the above command. The command updates the user_database table with the information from the CSV file.

For example:

- For an Oracle database, use the following command:

```
$JAVA_HOME/bin/java
com.ptc.usersdb.McUsersCreator oracle
oracle.jdbc.driver.OracleDriver
jdbc:oracle:thin:@machine1:1521:admin user1
userpass d:\wildfire\user.txt
```

- For a Microsoft Access database, use the following command:

```
$JAVA_HOME/bin/java
com.ptc.usersdb.McUsersCreator msaccess
sun.jdbc.odbc.JdbcOdbcDriver jdbc:odbc:admin
user1 userpass d:\wildfire\user.txt
```

A sample `user_database` table is shown below:

USERID	LASTNAME	FIRSTNAME	MIDDLENAME	GROUP_B	PHONE	EXTENSION	PROEDATE	INFO1	LOCATION
bdevid	Devid	Boon	K	CADQA	23453	3380	04-SEP-05	QA	d:\projects\devid
dalan	Alan	Donald	Y	CADDESIGN	23367	3181	02-OCT-00	DESIGN	d:\projects\alan

3. Update the `mc_database` table using the following command:

```
<java home> <class name to update mc_database>
<database type> <database driver> <database url>
<username> <password>
```

Note

- The `mc_database` table is automatically updated when the `new_admin.war` file is deployed. See *How to Start the Tomcat Server and Deploy the Metrics Tool* for details. You can also manually execute the above command if required. Either way, the `mc_database` table is updated with information from all the metrics files at the locations specified in the `user_database` table.
- As each metrics file is processed, its name is added to the `processedfiles` table.

For example:

- For an Oracle database, use the following command:

```
$JAVA_HOME/bin/java
com.ptc.metricsdb.McMetricsCollector oracle
oracle.jdbc.driver.OracleDriver
jdbc:oracle:thin:@machine1:1521:admin user1
userpass
```

- For a Microsoft Access database, use the following command:

```

$JAVA_HOME/bin/java
com.ptc.metricsdb.McMetricsCollector msaccess
sun.jdbc.odbc.JdbcOdbcDriver jdbc:odbc:admin
user1 userpass

```

A sample mc_database table is shown below:

USERID	DATE_N_TI	MDLNAME	MDLTY	CHECKTAG	CHKSTAT	INFO1	MCMODE
devid	18-JUL-04	RIM	PRT	SUP_FEATURES	WRN	2	Interactive
devid	18-JUL-04	RIM	PRT	UNITS_LENGTH	OK	INCH	Interactive
devid	18-JUL-04	RIM	PRT	UNITS_MASS	OK	POUND	Interactive
devid	21-OCT-04	PRT0001	PRT	mc_initialize	1	0	Interactive
devid	21-OCT-04	PRT0001	PRT	PARAMCHECK	OK	0	Interactive
alan	12-APR-05	02J_301_107_C-ROH_KUPPGEH	PRT	MODEL_NAME	OK	0	MC_regen
alan	12-APR-05	02J_301_107_C-ROH_KUPPGEH	PRT	PARAMCHECK	WRN	9	MC_regen
alan	12-APR-05	02J_301_107_C-ROH_KUPPGEH	PRT	PLANE_CHILD	OK	0	MC_regen

A sample processedfiles table is shown below:

FILENAME	MERGEDATE
devid_200407180949.txt	19-APR-06
devid_200410211010.txt	19-APR-06
alan_200503221503.txt	19-APR-06
alan_200504121146.txt	19-APR-06

Metrics Database

The metrics database, mc_database, enables you to generate graphs and reports for the ModelCHECK data.

The data fields stored in the mc_database table follow:

- **UserId**—User ID
- **date_n_time DATE**—Date and time
- **mdlname**—Model name
- **mdltype**—Model type
- **checktag**—Check type
- **chkstat**—Check status
- **info1**—Information
- **mcmode**—ModelCHECK mode

A sample mc_database table follows (syntax for Oracle):

```

CREATE TABLE mc_database
(UserId VARCHAR(256),
date_n_time DATE,
mdlname VARCHAR(256),
mdltype VARCHAR(5),

```

```

checktag VARCHAR(256),
chkstat VARCHAR(10),
info1 VARCHAR(256),
mcmode VARCHAR(20));

INSERT INTO mc_database VALUES('jsmith',
to_date('03-04-2005 11:14:07', 'MM-DD-YYYY
HH24:MI:SS'), 'rim.prt', 'PRT', 'BURIED_FEAT', 'ERR',
'Feature id 8', 'Interactive');

```

For more details, refer to the ModelCHECK online Help in the Help Center.

Training Database

The training database, `trng_database`, enables you to generate graphs and reports related to the training of ModelCHECK users. The data fields stored in the `trng_database` table follow:

- **UserId**—User ID
- **CourseId**—Training course ID
- **CourseName**—Training course name
- **CourseDate DATE**—Training course date
- **provider**—Training course provider
- **instructor**—Training instructor's name
- **location**—Training course location
- **cost**—Total investment on training
- **grade**—The grade achieved

A sample `trng_database` table follows (syntax for Oracle):

```

CREATE TABLE trng_database
(UserId VARCHAR(256),
CourseId VARCHAR(256),
CourseName VARCHAR(256),
CourseDate DATE,
provider VARCHAR(256),
instructor VARCHAR(256),
location VARCHAR(256),
cost VARCHAR(256),
grade VARCHAR(256));

INSERT INTO trng_database
VALUES('austin', '101', 'Fundamentals of Drawing',

```

```
to_date('2002-01-11','YYYY-MM-DD'), 'PTC', 'Jack  
Webb', 'Dallas', '1500', 'Good');
```

For more details, refer to the ModelCHECK online Help in the Help Center.

User Database

The user database, `user_database`, enables you to generate graphs and reports related to the details of the ModelCHECK users.

The data fields stored in the `user_database` table follow:

- **UserId**—User ID
- **lastname**—Last name
- **firstname**—First name
- **middlename**—Middle name
- **group_b**—Group to which a user belongs
- **phone**—Telephone number
- **extension**—Extension number
- **proedate DATE**—Date when the user ran ModelCHECK for which the metrics files were created
- **info1**—Information

A sample user_database table follows (syntax for Oracle):

```
CREATE TABLE user_database
(UserId VARCHAR(8),
lastname VARCHAR(256),
firstname VARCHAR(256),
middlename VARCHAR(256),
group_b VARCHAR(256),
phone VARCHAR(20),
extension VARCHAR(20),
proedate DATE,
info1 VARCHAR(256),
location VARCHAR(256));

INSERT INTO user_database
VALUES('jsmith','smith','john','VP', 'designer1',
'4082131', '3182', to_timestamp('2000-01-11
11:14:07','YYYY-MM-DD HH24:MI:SS'), 'aviation',
'/net/machine1/usr3/jsmith/mc_metrics');

INSERT INTO user_database
VALUES('dwells',wells,'David','K', 'designer2',
'4053182', '3481', to_timestamp('2001-01-11
11:14:07','YYYY-MM-DD HH24:MI:SS'), 'sqlstar',
'/net/machine2/usr2/dwells/demo/mc_metrics');

INSERT INTO user_database
VALUES('asewell','Sewell','Annie','P ', 'designer3',
'4082725', '3192', to_timestamp('1997-01-11
11:14:07','YYYY-MM-DD HH24:MI:SS'), 'explosive',
'/net/machine1/usr3/asewell/mc_metrics');
```

For more details, refer to the ModelCHECK online Help in the Help Center.

Using Fonts in Creo Elements/Pro

PTC provides 42 third-party TrueType fonts and the default PTC fonts with your software. In addition, you can also use any TrueType font available on Windows. The third-party TrueType fonts include 13 different font styles. If you are running Chinese Traditional or Chinese Simplified Creo Elements/Pro, you can use two additional TrueType fonts:

- Chinese Traditional
 - Monotype Hei Medium
 - Monotype Sung Light

- Chinese Simplified
 - Monotype HeiGB Medium
 - Monotype SungGB Light

New fonts appear in the current font list in Detailed Drawings and Sketcher. PTC fonts are listed first, followed by the third-party fonts supplied by Agfa Corporation.

TrueType fonts are stored in the <creo_loadpoint>/text/fonts directory. You can change this default location by placing the fonts in another directory and setting the value of the `pro_font_dir` configuration option to this directory. Creo Elements/Pro uses the fonts in the directory specified by the `pro_font_dir` configuration option.

The following table lists the TrueType fonts, their corresponding file names, and whether the font supports kerning. Kerning controls the space between certain pairs of characters, improving the appearance of the text string. Kerning is a characteristic of the particular font.

Font Name	File Name	Supports Kerning
Blueprint MT	bluprnt.ttf	No
Blueprint MT Bold	bluprntb.ttf	No
CG Century Schoolbook	schlbk.ttf	Yes
CG Century Schoolbook Bold	schlbkb.ttf	Yes
CG Century Schoolbook Bold Italic	schlbkbi.ttf	Yes
CG Century Schoolbook Italic	schlbki.ttf	Yes
CG Omega	cgomg.ttf	No
CG Omega Bold	cgomgb.ttf	No
CG Omega Bold Italic	cgomgbi.ttf	No
CG Omega Italic	cgomgbit.ttf	No
CG Times	cgtime.ttf	No
CG Times Bold	cgtimebd.ttf	No
CG Times Bold Italic	cgtimebi.ttf	No
CG Times Italic	cgtimeit.ttf	No

Font Name	File Name	Supports Kerning
CG Triumvirate	trium.ttf	Yes
CG Triumvirate Bold	triumb.ttf	Yes
CG Triumvirate Bold Italic	triumbi.ttf	Yes
CG Triumvirate Condensed Bold	triumcb.ttf	Yes
CG Triumvirate Italic	triumi.ttf	Yes
Garamond Kursiv	garamdi.ttf	No
Garamond Halbfett	garamdb.ttf	No
Garamond Antiqua	garamd.ttf	No
Garamond Kursiv Halbfett	garamdbi.ttf	No
Garth Graphic	gargra.ttf	Yes
Garth Graphic Black	gargrabl.ttf	Yes
Garth Graphic Bold Italic	gargrabi.ttf	Yes
Garth Graphic Italic	gargrai.ttf	Yes
Grotesque MT	grotesq.ttf	No
Grotesque MT Bold	grotesqb.ttf	No
Microstyle Extended	microex.ttf	Yes
Microstyle Extended Bold	microexb.ttf	Yes
Neographik MT	neograph.ttf	Yes
SackersEnglishScript	sackengs.ttf	No
Shannon	shanno.ttf	Yes
Shannon Bold	shannob.ttf	Yes
Shannon Extra Bold	shannoeb.ttf	Yes
Shannon Oblique	shannoo.ttf	Yes
Spartan Four MT	sparton4.ttf	Yes

Font Name	File Name	Supports Kerning
Spartan One Two MT	sparton12.ttf	Yes
Spartan One Two MT Bold	sparton12b.ttf	Yes
Symbol	symbol.ttf	No
CG Triumvirate Inserat	triin.ttf	No

If required, you can purchase additional TrueType and OpenType fonts. For information, contact Monotype Imaging at www.fonts.com.

When you work with Creo Elements/Pro, you use two categories of fonts:

- **Creo Elements/Pro fonts**—These fonts appear in Creo Elements/Pro graphics windows. They define alphanumeric characters and special symbols in text files. You can edit these files by:
 - Modifying the look of existing characters and symbols
 - Adding new symbols to the files
 - Creating new fonts
- **Hardware fonts**—These fonts appear in Creo Elements/Pro window headers. They are machine- and language-specific.

OpenType Font Support

Creo Elements/Pro provides additional support for OpenType fonts. These fonts are based on Unicode for multilanguage support. OpenType fonts offer an expanded character set and layout features. You can use these fonts in Sketcher to define geometry from sketched text and in Detailed Drawings to place drawing notes. OpenType fonts contain kerning information. You can enable or disable it to adjust the spacing between letters.

Note

PTC does not provide any OpenType fonts for general use with Creo Elements/Pro.

UNICODE Font Support

You can insert or read out-of-locale characters and symbols in drawings and sketcher using True Type, Open Type, or certain specific PTC fonts. When using True Type or Open Type fonts, make sure to install and store these fonts locally on all machines where you retrieve objects with text from multiple locales. You can then set the value of the `pro_font_dir`

configuration option to this directory for Creo Elements/Pro to use these fonts.

Out-of-locale text support is also available through a number of PTC fonts. The table below identifies the fonts along with the locales that they support.

Font name	Supported locale
font_chinese_cn	Simplified Chinese
font_chinese_tw	Traditional Chinese
font_hebrew	Hebrew
font_japanese	Japanese
font_korean	Korean
font_latin	English, French, German, Italian, Spanish

All these fonts are located under `<creo_loadpoint>/text` within the directory of the respective locale. Additionally, these fonts are copies of the 'font' font files that are available in each locale. The purpose of these fonts is to allow unambiguous out-of-locale access to all the default PTC fonts that are currently referred to as 'font'. If the text is displayed in the 'font' font, to display it correctly and consistently across all locales, change the text to use the applicable 'font_<locale>' font.

Available System Fonts for Menu Items

To specify the font for menu items, set the configuration file option `menu_font`. The value for this option is the name of the font file supported by the X-server running Creo Elements/Pro.

- On UNIX machines, use the `xlsfonts` command for a listing of fonts.
- On Windows systems, the fonts are listed in the **Control Panel** under **Fonts**. The font for text in Creo Elements/Pro window headers is determined by the setting of your window manager.

Extended ASCII Set

ASCII is the default font in all Creo Elements/Pro modules. You can modify ASCII characters. You can also replace ASCII with your own font as the default.

The extended ASCII character set used in the United States and Western Europe is supplied by Creo Elements/Pro through a combination of two font files, `ascii.src` and `latin_1.src`.

Note

Characters through 146 are defined in `ascii.src`. The remaining characters are defined in `latin_1.src`.

To use the European LATIN_1 characters, do one of the following procedures:

- Make sure your keyboard is set up for the appropriate language. A specific set of LATIN_1 characters is available with each keyboard language setting.
- Use a key sequence specific to your machine type to produce the desired character. If necessary, contact your hardware vendor for additional information.

Displaying the Text Symbol Palette

To display the **Text Symbol** palette:

1. Open a drawing and click **Insert > Note** to create a note.
2. Select a location on the drawing to place the note. The **Text Symbol** palette appears.
3. Click on a symbol in the palette.

Alternatively, you can create a new symbol. Creo Elements/Pro represents special symbols with standard ASCII characters.

To use special symbols in drawing notes, follow these steps:

1. Enter the sequence CTRL+A.
2. Enter the ASCII character that represents the special symbol.
3. Enter the sequence CTRL+B.

When you create a new symbol, you should either redefine an existing character to the new symbol or create a new special symbol font. You can add new special symbols to the font file, `special.src`, by defining the graphic representation.

Exporting Special Symbols to IGES

Special symbols and their corresponding ASCII characters map to the IGES font table 1001. Only two symbols (Total Runout and the Least Material Condition) do not map to the 1001 table. They are supported by the 1003 font table. The configuration file option `use_iges_font_1003`

determines which IGES font table outputs data to IGES. When you assign an ASCII character to a new symbol, select a character from either the 1001 or 1003 font table. Use 1001 or 1003 exclusively when you anticipate that a drawing will be exported in an IGES file. (Refer to your IGES documentation for more information on IGES font tables.)

Font Definition

Creo Elements/Pro defines fonts using two sets of parameters:

- Text parameters—The settings of a character definition box and other parameters used by all characters.
- Character parameters—The individual definition of each character (that is, how the character is drawn).

Comment lines in font definition source files are preceded with the “#” character. These parameters do not define the actual size of characters. They establish relative coordinates for stroking the character in the graphics windows. You control the actual size of text and symbols using the appropriate options in Creo Elements/Pro. You can change the actual size of text in drawings only.

Text Parameters

Text parameters for a given font are specified only once, at the head of the font file, and are used for every character definition in the file.

Note

The exception to this is the proportional fonts, such as Iso, Leroy, and Cal_grek, which require you to specify a specific width for each character. The following parameters define a character box with a grid (where <n> is an integer):

- height<n>—Defines the total number of grid spaces in the vertical direction. The maximum height is 63.
- width<n>—Defines the total number of grid spaces in the horizontal direction. The maximum width is 63.
- gap<n>—Defines the total number of grid spaces between character boxes.
- xextent<n>—Defines the maximum number of grid spaces in the horizontal direction.
- ascender<n>—Defines the number of grid spaces above the character designated for placing special characters such as accent acute and accent grave.

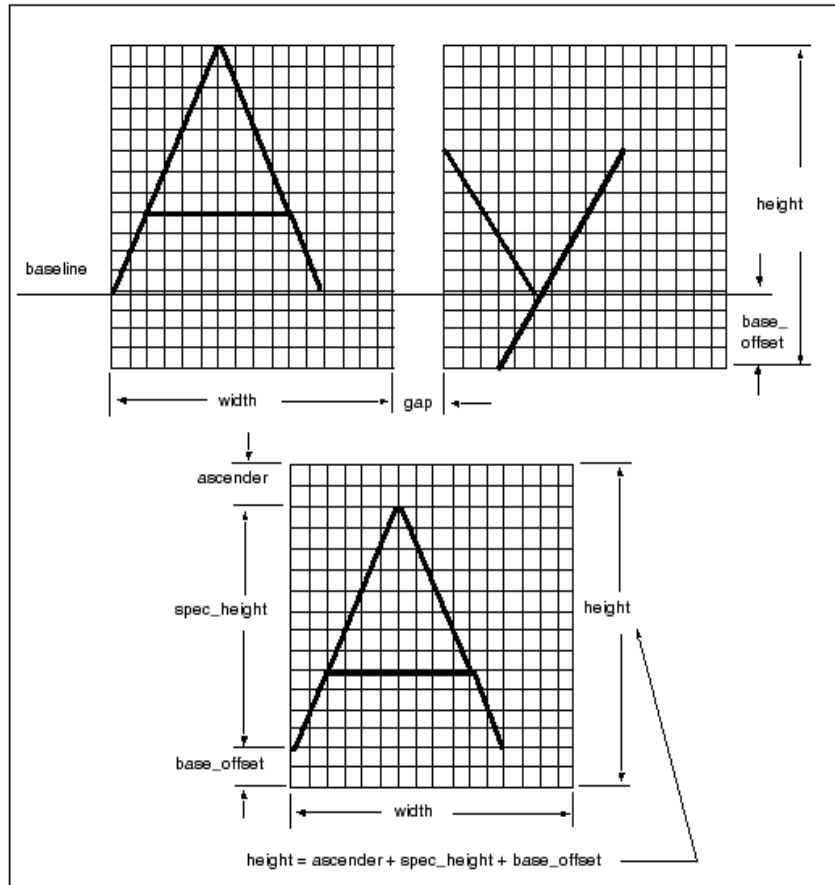
- `base_offset<n>`—Enables character alignment. The base offset positions the text box relative to a common baseline, so the drawn character definitions appear correct relative to each other.
- `spec_height<n>`—Defines the height of a character without the ascender above the baseline.
- `ics` (intercharacter spacing)—Turns on and off the effect of individual character width on horizontal spacing.
 - If you set `ics` to 1, each character definition must contain an individual width setting to determine the placement of the next character.
 - If you do not specify `ics`, the system assumes it is 0, and the system ignores individual width settings.
 - If `ics = 0`, the system uses fixed width character spacing.
 - If `ics = 1`, the system uses proportional width character spacing.
 - `codeset`—Identifies a font type. Codesets have the following meanings:
 - 0—Primary alphanumeric font set (such as `ascii.src`, `filled.src`, or `leroy.src`).
 - 2—Extended font set (such as `latin_1.src`).
 - 4—Special character font set or palette (such as `special.src`).

Note

The filled font has the same codeset as ASCII. Therefore, only one or the other can exist in the `font.ndx` file.

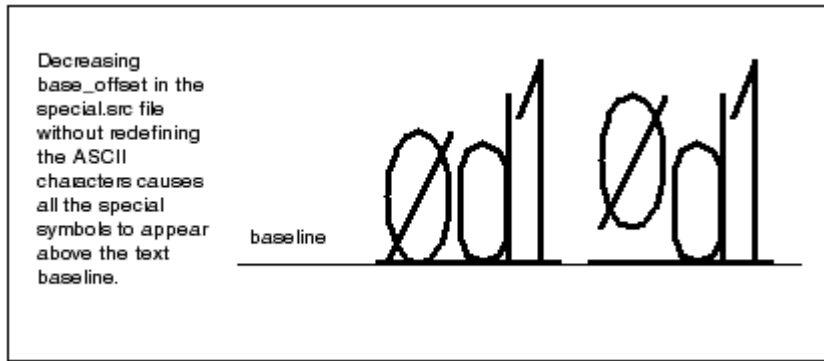
- `polygons`—Turns on character filling.
 - If you set it to 1, all closed polygons in the character definitions are filled (refer to the section Filled Font Definition).
 - If you do not specify this parameter, the system uses the default value 0.
 - `fixed_width<n>`—Can be 0 or 1.
 - If you set it to 1, the font is independent of the `text_width_factor` option in the drawing setup file.
 - If you set it to 0, you can control the font width by using the `text_width_factor` option or by modifying the text width manually.

The following figure illustrates the text parameters that define a font.

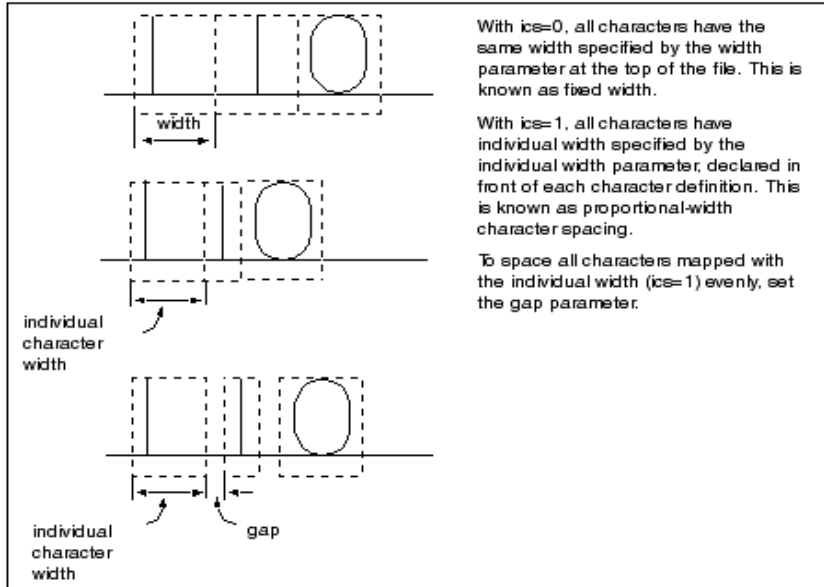


When you modify text parameters, consider how the font will appear in text with other fonts. Fonts used in the same text line (such as ASCII and LATIN_1 for European languages, or ASCII and special) must be compatible. Altering one font might create inconsistency in appearance and placement.

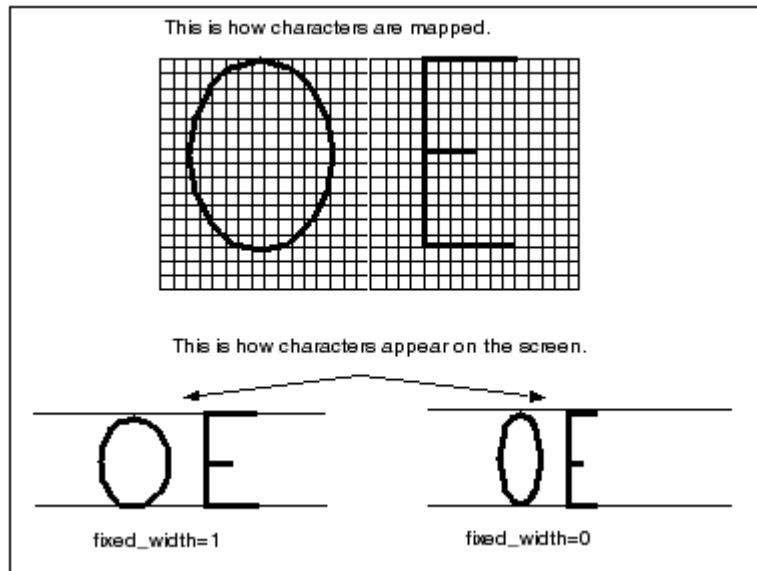
For example, changing the `base_offset` value at the top of the font file affects the positioning of all the font characters, relative to the text baseline, as shown in the following figure.



In addition to the width parameter, an individual width parameter can occur in each character definition. Use the individual width parameter to create a variable horizontal distance between the characters. In this case, if the `ics` parameter = 1, the individual width for each character overrides the width at the top of the file. If `ics` = 0, the system ignores individual width parameters. The following figure shows the effect of the `ics` parameter on the calculation of character width..



The following figure shows the effect of the `fixed_width` parameter on the calculation of character width.



Character Parameters

Below the text parameters section, a font file contains an individual definition for each character. A character definition contains the following information:

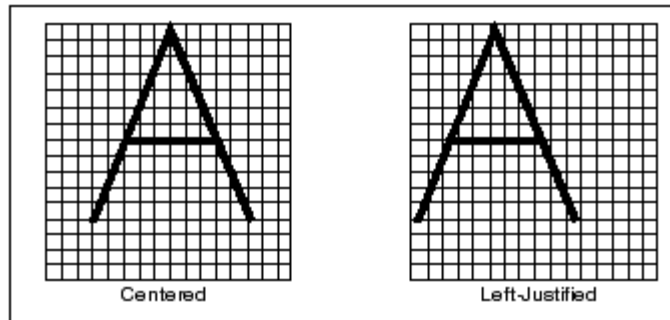
- First line—An ASCII character, in double quotes, that represents the character to be drawn. For ASCII and filled primary fonts, it is the character itself. Special characters also use ASCII characters to represent them. The LATIN_1 fonts use the decimal and hexadecimal codes representing the character.
- Second line (optional)—Individual width value (used only if `ics` is turned on).
- The remainder of the file consists of drawing-definition commands, described in the next section.

Definition Commands

The drawing definition of characters and special symbols in the font files is done with the following commands (where `<X>` and `<Y>` are integers):

- `[m<X>, <Y>]`—This command moves an imaginary pen to the grid point `(X, Y)`.

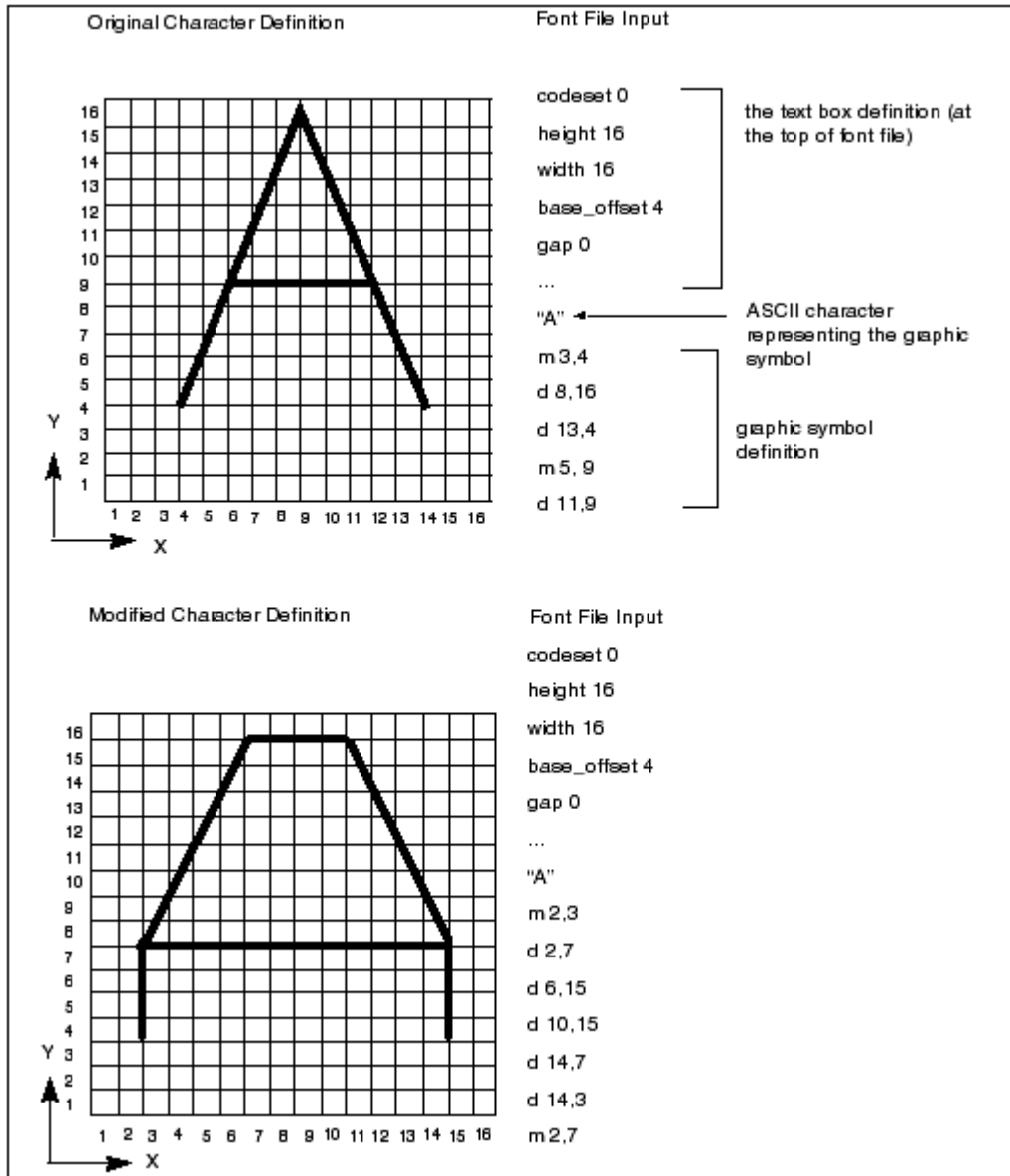
- [`d<X>`, `<Y>`]—This command draws a line from the current position to grid point (X, Y).
- For each font, you can use 63 or fewer grid points to create characters and symbols. Creo Elements/Pro provides optimum values, giving adequate character definition with a minimal grid. Enter characters centered or left-justified within a character box, depending on how you decide to draw them. The following figure illustrates text justification.



You can change a font file to alter the appearance of any given font. Consider the ASCII font definition for the character “A”, both in the original and modified form.

If you modify the text parameters without redefining the character parameters, the appearance of the fonts changes. For example, if you change the grid height from the default value of 16 to 20, the character appears smaller. Because the character was originally defined with a maximum grid value of 16, the original ratio was 16/16; after the change,

the ratio is 16/20. Therefore, the new text height appears to be 80 percent of the original height.

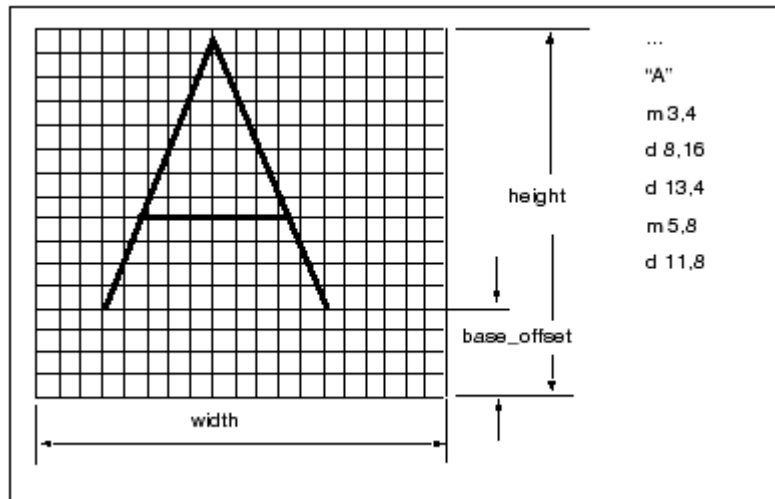


ASCII Font Definition

Use the following parameters to define ASCII characters:

- codeset: 0
- height: 16
- width: 16
- base_offset: 4
- gap: 0

The following figure illustrates how ASCII characters are mapped. Note that ASCII characters are defined in a character box consisting of 16x16 grid spaces. All characters have modifiable width and are centered within a character box.



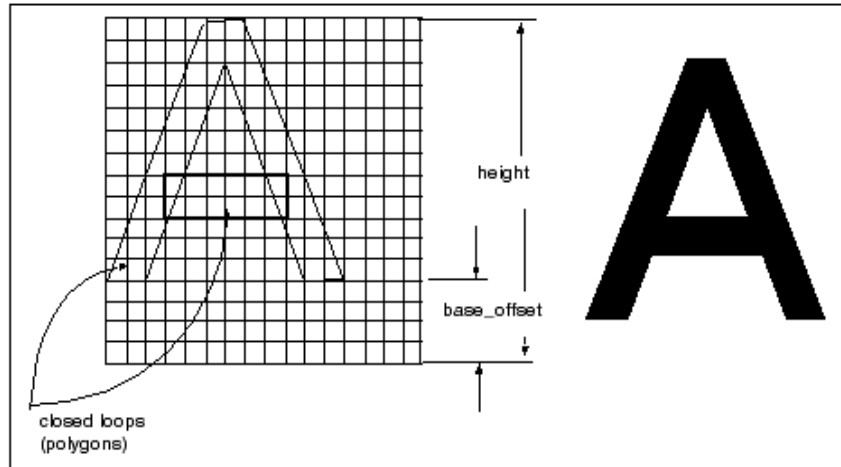
Filled Font Definition

Use the following text parameters to define a filled font:

- codeset: 0
- height: 63
- width: 48
- base_offset: 16
- gap: 0
- polygons: 1
- ics: 1

In the filled font, characters are mapped within 63 vertical grid spaces. However, the width for each character is different. The *ics* parameter (set to 1) tells the system to ignore the global width parameter given at the top of the file. Instead, the system uses the individual character width, which precedes each character definition, to map a character.

Define the outline of a filled character by drawing polygons (closed loops). When the polygons parameter at the top of a font file is set to 1, Creo Elements/Pro fills with color any closed polygon drawn in a character definition.

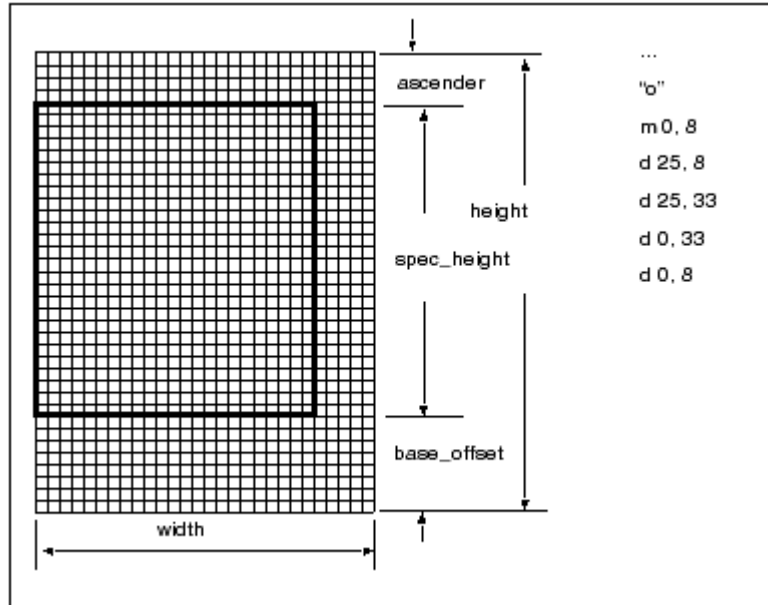


Special Font Definition

The following text parameters are used to define a special font:

- codeset: 4
- height: 38
- width: 36
- xextent: 63
- spec_height: 24
- base_offset: 8
- ascender: 6
- fixed_width: 1
- ics: 1
- gap: 4

In the special font, all characters are defined in 38 vertical grid spaces. Each character has an individual width declared in front of its definition. To establish spacing between character boxes, the “gap” parameter is set to 4. The following figure illustrates the definition of the special font character.



LATIN_1 Font Definition

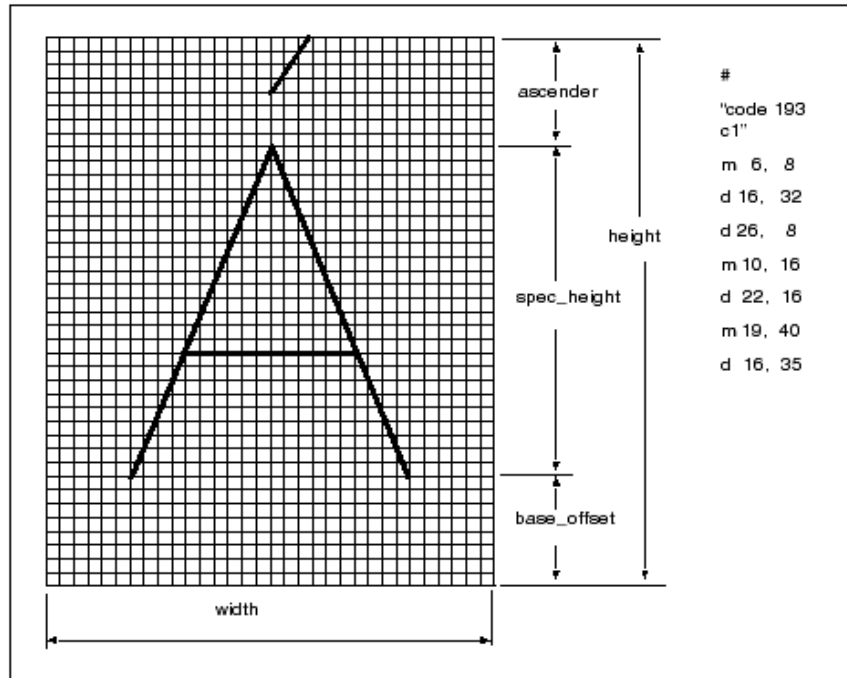
The following text parameters define the LATIN_1 font:

- codeset: 2
- height: 40
- width: 32
- xextent: 63
- spec_height: 24
- base_offset: 8
- ascender: 8
- fixed_width: 0
- ics: 0
- gap: 0

The LATIN_1 font is defined with a character box of 40x32 grid spaces. All characters have constant width and are centered in the character box. In the double quoted line above the character definition, the decimal and hexadecimal code representing the character is shown, as follows:

```
code <decimal> <hex>
```

The following figure illustrates how to map a LATIN_1 character.



How to Edit Creo Elements/Pro Fonts or Create New Fonts

1. Edit an existing or create a new .src file, which resides in <creo_loadpoint>/text/<language>.
2. Compile the font file by using the command compile_font.
3. Make the new font available locally or for all users.

How to Modify a Creo Elements/Pro Font File

1. Copy the appropriate .src font file into your working directory. The .src files reside in <creo_loadpoint>/text/<language>.
2. Modify the font file.

3. From the `<creo_loadpoint>/text/<language>` area, copy the appropriate index file to your working directory.
4. Copy the `compile_font` command to your working area. If you are working on a UNIX machine, copy the executable `compile_font` to your working directory. The executable is in the following directory: `<creo_loadpoint>/<machine_type>/obj`.
5. To compile the font, enter the following command for each font file you modify:


```
compile_font fontfilename.src
fontfilename.fnt
```

Note

If you modify the ASCII font definitions, it alters the appearance of the default font in every Creo Elements/Pro module.

How to Create a User-Defined Font File

1. Define a new font in a file with the extension `.src`. Store the new font in your working directory.
2. Create an index file that includes the new font file. Make sure the index file contains fonts with unique codesets.
3. Copy the `compile_font` command to your working area. If you are working on a UNIX machine, copy the executable `compile_font` from `<creo_loadpoint>/<machine_type>/obj` into your working directory.
4. Enter the following command for each font file you create:


```
compile_font fontfilename.src fontfilename.fnt
```
5. Recompile user-defined fonts whenever you install a new major release of Creo Elements/Pro.

You can include user-defined fonts as default or auxiliary fonts in drawings.

How to Store a New Font

1. Store the updated index file with the `.src` and `.fnt` files in your local directory if the new font is for your personal use.
2. To make a new font available to all users, reference the font in the index and store the index and source files in the directory `<creo_loadpoint>/text/<language>`. Store the `.fnt` file in the following directory:


```
<creo_loadpoint>/<machine_type>/text/<language>
```


How to Set Your Font as the Default

1. Rename your file to `ascii.src`.
2. Recompile the `ascii.src` file using the `compile_font` command to produce a local `ascii.fnt` file.

How to Create or Customize a Font

1. Copy or create font source files in the local working directory. In this example, the default primary ASCII and special symbol fonts are copied and renamed.

```
cp /usr/pro/text/usascii/ascii.src custom.src
```

(primary font with codeset 0)

```
cp /usr/pro/text/usascii/special.src symbols.src
```

(special symbols font with codeset 4)

2. Edit the source files to include all the required definitions and changes.

```
vi custom.src
```

```
vi symbols.src
```

3. Copy the `compile_font` script from the directory `<creo_loadpoint>/<machine_type>/obj` into the same local working directory.

```
cp /usr/pro/i486_nt/obj/compile_font
```

4. Compile the font files.

```
compile_font custom.src custom.fnt
```

```
compile_font symbols.src symbols.fnt
```

5. Create a new index file to reference the compiled font files. Note that the index file name does not need to be the same as the primary font name. If the custom font created is not a primary font (codeset 0), be sure to include a font with a codeset of 0 in the index file. In this

example, the new font index file references both customized font files (primary and special symbols).

```
myfont.ndx (new customized .ndx file)
custom (new custom primary font (codeset 0))
latin_1 (system extended ASCII font (codeset 2))
symbols (new special symbols font (codeset 4))
```

This font index file references the new customized special symbols font only. It uses the system default primary font.

```
myfont2.ndx (new customized .ndx file)
ascii (system default primary font (codeset 0))
latin_1 (system extended ASCII font (codeset 2))
symbols (new special symbols font (codeset 4))
```

You can use the new font in the following ways:

- To use the font as a global auxiliary font, place the .src and .ndx files in the <creo_loadpoint>/text/<language> directory. Then put the compiled .fnt files in the directory <creo_loadpoint>/<machine_type>/text/<language>.

```
mv custom.src symbols.src myfont.ndx
/usr/pro/text/usascii
```

```
mv custom.fnt symbols.fnt
/usr/pro/i486_nt/text/usascii
```

- To set the new font as the system default, rename the primary source and compiled font (codeset of 0) files to `ascii.src` and `ascii.fnt`, respectively. The system font.ndx file already references the ASCII font file and, therefore, does not need to be modified. Locate the new `ascii.src` and `ascii.fnt` files in the appropriate directories (see the previous section that describes how to use the font as a global auxiliary font). Note that the system overrides the original `ascii.src` and `ascii.fnt` files, unless you first rename them.

The next time you start Creo Elements/Pro, it uses the new custom font as the system default font.

```
mv custom.src /usr/pro/text/usascii/ascii.src
```

```
mv custom.fnt
/usr/pro/i486_nt/text/usascii/ascii.fnt
```

Customizing the Text Symbol Palette

You can customize fonts and add symbols and characters to the **Text Symbol** palette in Creo Elements/Pro from the special font file. You can also replace, reorder, or remove symbols using the text symbol palette layout file.

How to Add New Symbols to the Text Symbol Palette

1. Customize the special font file, `special.src`, using the method described in the section Using Fonts in Creo Elements/Pro on page C-23
2. Add a definition for each of the new symbols that you want to insert in the **Text Symbol** palette and associate an ASCII code to the symbol.
3. Create new images for each of the symbols that you have defined in the customized special font file.
4. Save the images as `sp_#.gif` where # is the ASCII code of the image defined in the special font file. For example, you can save an image as `sp_133.gif`, where 133 is the ASCII code that is associated with the image.
5. Store all the image files in the `<creo_loadpoint>/text/resource` directory.
6. Define a text symbol palette layout file and save this file in your working directory. See the next section, How to Create a Text Symbol Palette Layout File, for details.
7. In your `config.pro` file, set the value of the configuration option `custom_text_symbol_palette_file` to point to the full path of the customized text symbol palette layout file. For example, for a file named `my_custom_palette.txt` and saved in the `D:\my_creo\my_custom_palette.txt` directory, set the configuration option as follows:

```
custom_text_symbol_palette_file  
D:\my_creo\my_custom_palette.txt
```

This path enables Creo Elements/Pro to display the **Text Symbol** palette according to the settings in the customized layout file.
8. Create or modify a drawing in a new session of Creo Elements/Pro, and verify the presence of the customized **Text Symbol** palette.

How to Create a Text Symbol Palette Layout File

After you have created images files for the symbols that you have defined in the special font file, create a text symbol palette layout file using this procedure:

1. Create a file with `.txt` extension in your working directory, for example, `my_custom_palette.txt`.
2. Specify an integer in the first line of the file. This integer represents the number of columns in the **Text Symbol** palette in Creo Elements/Pro.
3. Specify integers on all other lines of the file. These integers must represent the ASCII codes of the symbols for the **Text Symbol** palette. The integers must lie from 32–255.

On every line separate each of the ASCII codes with one space. The order of the ASCII codes in the file represents the order of the symbols in the **Text Symbol** palette. For additional information, see Examples of the Text Symbol Palette Layout File on page C-45

ASCII Codes for GTOL Symbols

ASCII codes for GTOL symbols are based on the setting for the `gtol_symbols_same_size` detailing setup option. If you set the value to `no`, the ANSI codes are used to identify the special fonts to be used. If it is set to `yes`, the ISO codes are used to identify the special font in the **Text Symbol** palette. Refer to the next table for the ANSI and ISO codes for GTOL symbols.

GTOL Symbol	ANSI Code	ISO Code
Straight	33	133
Flat	99	136
Circular	101	138
Cylinder	103	140
Profile Line	107	146
Profile Surf	100	137
Angular	97	135
Perpendicular	108	147
Parallel	102	139
Position	106	145
Concentric	114	148
Runout Circular	104	141

GTOL Symbol	ANSI Code	ISO Code
Total Runout	34	134
Symmetrical	105	144

In a Creo Elements/Pro drawing, the GTOL symbols in the **Text Symbol** palette change if you change the value of the `gtol_symbol_same_size` detailing setup option.

Guidelines for Customizing the Text Symbol Palette Layout File

Administrators must verify that the text symbol palette layout file is customized correctly and all the entries in this file are valid. If the customization is incorrect, Creo Elements/Pro uses the default settings to display the **Text Symbol** palette without a warning message.

Guidelines for customizing the text symbol palette layout file follow:

- When creating images, ensure that the height and width of the images does not exceed 16 pixels and 24 pixels, respectively.
- The file must contain only integers. All nonintegers are treated as spaces.
- Specify ASCII codes from 32–255.
- Do not duplicate ASCII codes.
- Place the same number of ASCII codes on each row as the integer that you have specified in the first line of the file. This listing approximates the appearance of the customized **Text Symbol** palette in Creo Elements/Pro.
- Use the ASCII codes for the `dt1` options carefully. The usage of these symbols in a drawing changes between ANSI and ISO according to the value that you set for the detailing setup option, `gtol_symbols_same_size`.

Examples of the Text Symbol Palette Layout File

- If you specify `#1.15` in the first row of the text symbol palette layout file, Creo Elements/Pro ignores the crosshatch (`#`) character. One column is displayed in the **Text Symbol** palette. The decimal point (`.`) after `1` is ignored. The integer `15` is taken as the first ASCII code but is ignored. Only those ASCII codes from 32–255 are displayed in the **Text Symbol** palette.

- File 1 and File 2 represent the same **Text Symbol** palette in Creo Elements/Pro:

File 1	File 2
3	3
60	60 61 62
61	35 37 68
62	65 66 70
35	75
37	
68	
65	
66	
70	
75	

AVI Compression Codecs (Windows Only)

You can capture an AVI file using the third-party compression codecs installed on your Windows system. PTC does not provide codecs that support AVI compression. A warning message appears when you try to capture an AVI file under the following conditions and the process quits prematurely:

- You have not installed or licensed the compression codecs correctly on your windows system.
- You have installed a codec that is defective.
- The codec does not work with specific compression settings.

Exporting Data to Portable Document Format

You can use the Interface for PDF application through Creo Elements/Pro to export model drawings to Portable Document Format. The Interface for PDF provides enhanced navigation, search, and presentation capabilities. The licenses for this application can be node-locked or floating.

The Interface for PDF enables you to do the following tasks:

- Insert the Creo Elements/Pro parameter data as searchable metadata in the PDF file.

- Modify the font types, color, style, and resolution of the PDF output.
- Insert Creo Elements/Pro drawing text and numerical values as searchable and selectable content in the PDF file.
- Insert bookmarks for the various Creo Elements/Pro layers, sheets, zones, flag notes, and revision tags.
- Control the access and security rights of the PDF file.

For more details, see the Creo Elements/Pro Interface Help available in the Creo Elements/Pro Help Center.

Meeting Browser Requirements

This section describes the requirements for the Creo Elements/Pro browser. For information on Web browser support for Creo Elements/Pro, see www.ptc.com/WCMS/files/77552/en/proewf5.pdf. For information related to the Help system, see Overview of Help Center on page 6-2.

Browser Requirements for Windows

On Windows systems, the Creo Elements/Pro browser uses the rendering engine of Internet Explorer. To enable this default, make sure that Internet Explorer 6.0 with Service Pack 1 and above or Internet Explorer 7.0 is installed on your computer. Alternatively, you can configure Creo Elements/Pro to use a Mozilla-based browser. Creo Elements/Pro already has an embedded version of this browser. It cannot use any other version installed on your computer.

On Windows, Creo Elements/Pro browser uses Internet Explorer engine in the in-process mode. However, with the multiprocess architecture of Internet Explorer or the Mozilla-based embedded browser on Windows, you can run the embedded browser in the out-of-process mode. That is, you can run Internet Explorer or Mozilla browser as a separate process from the Creo Elements/Pro process.

You can specify whether you want the Creo Elements/Pro browser to use Internet Explorer or the Mozilla-based rendering engines. Before you start Creo Elements/Pro, set one of the following values to the `windows_browser_type` configuration option in your `config.pro` file:

- `ie_in_process`—Specifies that Creo Elements/Pro browser uses the Internet Explorer engine in the same process as that of Creo Elements/Pro. This is the default.
- `ie_in_sep_process`—Specifies that Creo Elements/Pro uses the Internet Explorer engine in a separate child process initiated

from the Creo Elements/Pro process. You can set this value only on a 32-bit Windows operating system.

- `mozilla_based_browser`—Specifies that Creo Elements/Pro browser uses the Mozilla-based rendering engine in a separate child process initiated from the Creo Elements/Pro process. You can set this value on the 32 and 64-bit Windows operating systems.

Browser Settings in UNIX

When installing Creo Elements/Pro on UNIX systems, an embedded version of the Mozilla browser is automatically installed. The settings of the embedded Mozilla browser drive the Creo Elements/Pro browser. The Creo Elements/Pro browser creates its own profiles directories based on the Gecko version of the embedded Mozilla browser and does not use any settings of a standalone Mozilla installation.

The Creo Elements/Pro browser creates profiles in two directories:

- `$HOME/<browser_profile_location>/ptc-browser`
- `$HOME/<browser_profile_location>/ptc-mail`

For example, for an embedded Mozilla browser of Gecko version 1.8.1.23, the Creo Elements/Pro browser creates profiles in two directories:

- `$HOME/.ptcm18123/ptc-browser`
- `$HOME/.ptcm18123/ptc-mail`

To change the settings of the Creo Elements/Pro browser, use the Mozilla-native **Preferences** dialog box. Click **Tools > Browser Options** in the Creo Elements/Pro window to open it.

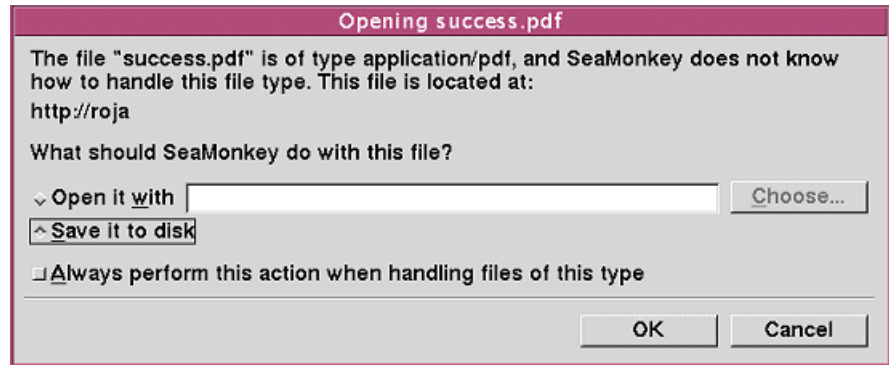
Starting Creo Elements/Pro through an External Browser

To start Creo Elements/Pro from an external browser, register a helper application in the `bin` directory. The helper application gets installed automatically when you install Creo Elements/Pro. This application, which is `proe` on UNIX and `proe.exe` on Windows, is also referred to as the Creo Elements/Pro Application Starter. Registration of the helper application on Windows and UNIX involves registering files with `.pha` extension.

Opening a PDF File within the Creo Elements/Pro Browser

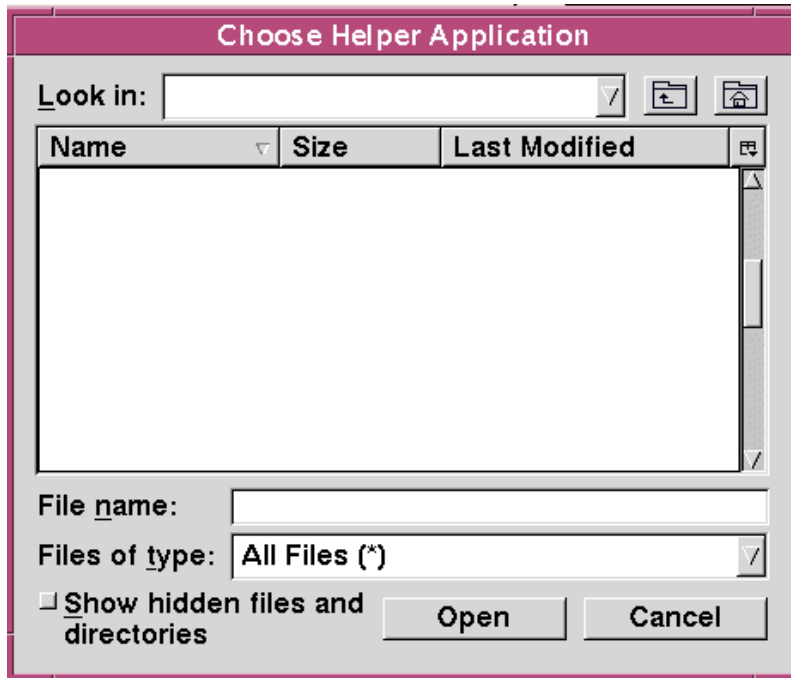
You can view a PDF file in the Creo Elements/Pro browser using Adobe Acrobat Reader. On Windows, Adobe Acrobat Reader is available as a plug-in. On UNIX, use Adobe Acrobat Reader as a helper application to view the PDF file in the browser using the following instructions:

1. When you open a PDF file, the following dialog box opens:



2. Select the **Open it with** option. You can choose to clear the **Always perform this action when handling files of this type** option.

3. Click **Choose**. The **Choose Helper Application** dialog box opens.



4. Browse to the path for the `acroread` command in the Adobe Acrobat Reader load point.
5. Click **OK**.
6. Click **Open** to complete the registration of Adobe Acrobat Reader as a helper application on UNIX.

Printing a PDF File

You can view a PDF file in the Creo Elements/Pro browser using Adobe Acrobat Reader. To print the PDF file, click the Adobe Acrobat Reader **Print** icon instead of the Creo Elements/Pro browser **Print** icon.

Setting Up Creo Elements/View Express

For a Creo Elements/Pro installation on Windows, when you visit a Web page that has Creo Elements/Pro objects selected, the Creo Elements/View Express installer opens. Perform the following steps to install Creo Elements/View Express:

1. In the **Creo Elements/View Express Setup** dialog box, accept the license agreement by clicking the corresponding check box.

2. Click **Install** to begin the installation by retaining the default installation path.

To install Creo Elements/View Express at a different location, perform the following operations:

1. After accepting the license agreement in the **Creo Elements/View Express Setup** dialog box, click **Advanced**. The **Destination Folder** screen appears.
2. Specify the destination folder to install Creo Elements/View Express.
3. Click **Next**.
4. Click **Install** in the next screen to begin the installation. A message appears indicating successful installation of Creo Elements/View Express.
5. Click **Finish**. When installation is complete, Creo Elements/View starts without your having to restart the Creo Elements/Pro session. You can open the following objects using Creo Elements/View Express:

File Extension	File Type
.asm	Creo Elements/Pro Assembly file
.dft	Creo Elements/Pro Draft file
.dgm	Creo Elements/Pro Diagram file
.drw	Creo Elements/Pro Drawing file
.frm	Creo Elements/Pro Format file
.pic	Creo Elements/Pro Picture file
.prt	Creo Elements/Pro Part file
.rep	Creo Elements/Pro Report file
.mfg	Creo Elements/Pro Manufacturing file
.sec	Creo Elements/Pro Sketcher Section file

On Windows, double-click a file with a Creo Elements/View extension to open the file in Internet Explorer.

Opening Creo Elements/Pro Objects from Windows Explorer

If you have not installed Creo Elements/View, you can open a Creo Elements/Pro object directly from Windows Explorer in a Creo Elements/Pro session. Creo Elements/Pro objects typically appear with a PTC icon in Windows Explorer. Double-click an object in Windows Explorer to open it in a Creo Elements/Pro session.

If you have installed Creo Elements/View, a Creo Elements/Pro object is opened in Internet Explorer only for viewing when you double-click the object in Windows Explorer. Right-click on the object and choose **Open with Creo Elements/Pro** to open the object in Creo Elements/Pro.

You can open objects with the following file types:

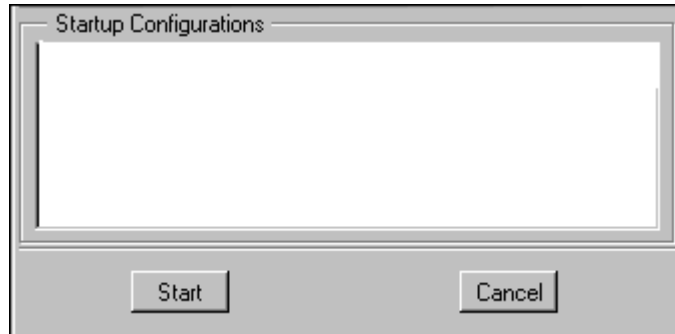
File Extension	File Type
.asm	Creo Elements/Pro Assembly file
.dgm	Creo Elements/Pro Diagram file
.drw	Creo Elements/Pro Drawing file
.frm	Creo Elements/Pro Format file
.lay	Creo Elements/Pro Layout file
.mfg	Creo Elements/Pro Manufacturing file
.mrk	Creo Elements/Pro Markup file
.prt	Creo Elements/Pro Part file
.sec	Creo Elements/Pro Sketcher Section file

Objects other than those listed in the preceding table do not open in Creo Elements/Pro.

If a Creo Elements/Pro session is already running and you double-click an object in Windows Explorer, the object is automatically loaded within

the same Creo Elements/Pro session in a new window. This new window becomes the active window.

If a Creo Elements/Pro session is not running and you double-click an object in Windows Explorer, the **Creo Elements/Pro startup config file** dialog box opens. You are prompted to select a file from the list of available Creo Elements/Pro configuration files. Upon selecting the required startup configuration file, a Creo Elements/Pro session is started using the selected configuration file. The selected object is loaded in this Creo Elements/Pro session.



If multiple sessions of Creo Elements/Pro are running, the selected object is loaded in the session that you started first.

JavaScript Security

In Creo Elements/Pro, a JavaScript bridge is used so that the JavaScript code inside a displayed HTML page interacts with Creo Elements/Pro. For example, **Feature info** reports, folder content listings, and new connection registrations all use the JavaScript bridge for security purposes.

Creo Elements/Pro automatically activates the JavaScript bridge only from pages generated by Creo Elements/Pro or Windchill Solutions. If a page generated from another source tries to access Creo Elements/Pro

through the JavaScript bridge, the following warning message appears and the operation is stopped.



Take one of the following actions:

- If the URL in the message is from an unreliable source, contact your system administrator and avoid accessing this page again.
- If the URL in the message is from a reliable source, such as a PTC Windchill Solution, contact Technical Support.

Rights Management in Creo Elements/Pro

Creo Elements/Pro Rights Management Extension helps to control access to your intellectual property. Intellectual property could be sensitive design and engineering information that you have stored within Creo Elements/Pro parts, assemblies, or drawings. You can control access by applying policies to these Creo Elements/Pro objects. Such objects remain protected by the policies even after they are distributed or downloaded. Creo Elements/Pro objects for which you have applied policies are called DRM-protected objects. Policies control access rights to DRM-protected objects for users within or outside your firewall.

Administrators can define policies at the Adobe LiveCycle Rights Management ES server (policy server) through a Web interface. They can also define:

- Publishers who can apply these policies to files within Creo Elements/Pro.
- Users who can access the DRM-protected objects.

Refer to the Adobe Help for more information on defining policies and specifying users or user groups.

After the administrators have defined policies, users can perform the following DRM operations within Creo Elements/Pro:

- Open a DRM-protected object after authenticating themselves with the policy server. Users can then use the object with their usage rights established in the current policy of the object.
- Apply or remove policies to Creo Elements/Pro objects. To do so, users must have a DRM license.
- Connect to the policy server to verify the policy details.
- Generate a report of all the DRM-protected objects in the current session.

For more information, refer to the Digital Rights Management Help in the Help Center.

D

Installing MKS Platform Components (Windows Only)

This appendix contains information on how to install MKS platform components in order to work with CADD5 5 models in Creo Elements/Pro.

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Installing MKS Platform Components	D - 2

Overview

On Windows, you must install **MKS Install for CADD5 5 Interface** along with **INTERFACE for CADD5 5** to work with CADD5 5 parts and assemblies within Creo Elements/Pro. See Defining the Installation Components on page 4-2 for the installation instructions of **INTERFACE for CADD5 5**. **MKS Install for CADD5 5 Interface** is a third-party application that supports the following functions:

- Simulates a UNIX environment on the 32-bit and 64-bit Windows operating systems.
- Emulates UNIX commands on Windows operating systems.
- Installs the Portmapper service for interoperability between CADD5 5 and Creo Elements/Pro.
- Copies the necessary startup files and libraries required to work with CADD5 5 files into a separate installation directory.

Installing MKS Platform Components

1. Start PTC.Setup using the steps in Starting PTC.Setup on page 1-8.
2. Accept the PTC License Agreement (see PTC License Agreement on page 1-5). The PTC.Setup product selection screen appears.
3. Under **Other Products**, click **MKS Install for CADD5 5 Interface** to open the Windows Installer.
4. Click **Next** on the **MKS Platform Components 9.x - Setup** screen. A message indicates whether you have Perl installed. If you do have a Perl installation, follow the instructions.
5. Click **Next**. The **Customer Information** screen appears.
6. Type your user name and organization name.
7. Click **Next**. The **File Extensions** screen appears. Follow the instructions.
8. Click **Next**. The **Setup Type** screen appears.
9. Select a setup type: **Typical**, **Compact**, or **Custom**.
10. Accept or change any preferences and click **Next**.
11. When the **Install** button appears for the installation of **MKS Platform Components 9.x**, click one of the following buttons:
 - **Back** to review the information.

- **Install** to begin the installation.
- **Cancel** to stop the installation.

12. Click **Finish** to close the installer.

After the installation is complete, see *Working with CADD5 5 Models* on page 4-20 before you start.

E

Distributed Computing Technology

This appendix contains information on the Distributed Computing technology that is used by Creo Elements/Pro to augment your existing hardware.

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Overview

Creo Elements/Pro uses Distributed Computing technology to augment your existing hardware to perform computationally intensive tasks. Distributed Computing technology includes the following items:

- A Creo Elements/Pro session or the controller.
- Participating networked workstations.
- A daemon running on each participating workstation.
- One or more agents running on the workstations. These agents could be the Creo Elements/Pro sessions running as server processes.
- A task or a collection of jobs.

Note

Distributed Computing is suitable only for multiobject design studies.

Interactions between the Controller and the Agent

With Creo Elements/Pro you can optimize the distribution of tasks using Distributed Computing technology. The controller communicates with the daemon on each workstation and determines load averages. Agent sessions are automatically started depending on load averages and the number of processors. These agents are started only once for every distributed computing task and not for every job. Communication of data is also optimized. Data, that is, models, information, and instructions, are efficiently streamed directly to each agent through the daemon. No files are copied to a workstation before a job is carried out. As subsequent jobs are dispatched to the same agent, only data that is different between the jobs is streamed. If the data involved is the same, it is not streamed again.

Supported Functions

Distributed Computing technology supports the following functions:

- Workstations with multiple CPUs are leveraged so that an agent process exists for every CPU (depending on the system load average).
- Workstations are engaged in distributed computing unless the load on a CPU exceeds 20 percent.

- During distributed computing, tasks are dispatched to workstations based on real-time conditions, which include the following:
 - The workstations must be online.
 - The load average on the workstations must be within permissible limits. The load average is calculated over time, not with a single measurement at a specific point in time.
- If a workstation shuts down during distributed computing, the active task is not interrupted. The controller detects the shutdown and routes the job to another agent.
- If multiple users are performing distributed computing tasks, workstations are engaged on a first-come, first-served basis. While a workstation is processing jobs dispatched by a certain controller, it is unavailable to other controller sessions.

After the controller completes its task, any active controller can then engage the workstation.

Note

Avoid using distributed computing when dispatching jobs across fire walls.

Configuring Workstations for Distributed Computing

As system administrators, after installing Pro/ENGINEER 2000i² or later, you must configure workstations for distributed computing by performing the following tasks:

1. From the available UNIX and Windows workstations, choose the ones that can participate in distributed computing sessions.
2. Run the `dcadsetup` script on each participating workstation. This script starts a daemon process that is required for distributed computing. No other setup task is required.
3. To shut down the daemon process and thus disable a workstation from participating in distributed computing sessions, run the `dcadshutdown` script.
4. Optionally, register the `dcadsetup` command within the boot script of a workstation. This step ensures that the workstation is enabled for distributed computing after it is shut down and restarted.

Specifying Host Machines for Distributed Computing

You can specify the host machines for distributed computing by using the **Distributed Computing Hosts Selection** dialog box that opens when you click **Tools > Distributed Computing** in a Creo Elements/Pro session. Distributed computing tasks automatically use the dialog box settings.

When you begin a task, a progress bar indicates the progress of the task, statistics about how many jobs are being processed by agents on each participating workstation, and the status of each workstation. You can release the participating workstations at any time for use by others for a distributed computing task.

F

Troubleshooting Tips

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

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General Debugging Hints

The `ptc.log` file records license server activities at `\FLEXnet Admin License Server\logs\`. This file contains useful information that you could use when you have a problem.

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 Technical Support page 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.

The *PTC Customer Service Guide* and contact support is available under **CONTACTING TECHNICAL SUPPORT** on the PTC Web page at www.ptc.com/support/index.htm. For information on FLEXnet Publisher diagnostic environment variables, consult the *FLEXnet Licensing End User Guide*.

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.

Failure to Start the Software

Symptom: On Windows systems, when you attempt to start your PTC software, you see an MS-DOS window that contains the message:
Cannot find
<Installation_Directory>\<mc_type>\filename

Cause: The system `PATH`, an environment variable, is either not set to the `<Installation_Directory>\bin` or it is improperly set due to a syntax or spelling error.

Solution: Check the system `PATH`.

Xtop Icon Disappears from the Taskbar

Symptom: An Xtop icon appears for a few seconds in the taskbar and quickly disappears.

Cause: Network configuration settings are not properly set.

Solution: Make sure that the network settings are in accordance with the Suggested Technique for Configuration of a Windows Workstation for executing Creo Elements/Pro.

Look for a file in the startup location called `std.out` and see if there are references to network errors. If there is a line that states `Exception Code Was -1073741819`, make sure the latest graphics driver is installed. The most recent graphics driver can be downloaded from the manufacturer's Web site. An alternative is to set the `config.pro` option `graphics win32_gdi`.

Inaccurate Ptcstatus Command Information

Symptom: On Windows systems for Release 20 and later, the `ptcstatus` command returns the following message, `Displaying status for license file 77880@ptc Warning (-15): Cannot connect to license servers (-15, 10; 10061)`.

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

Solution: Verify that the `lmgrd` 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`.

ModelCHECK Metrics Tool Display Problems

While using the Web-based ModelCHECK Metrics Tool, you may encounter problems with the display of the graphs and reports in the Internet Explorer or the Mozilla browser. Causes of the problem and possible solutions follow:

Cause: The path to the Image Directory is not specified in the `new_admin.xml` configuration file. If specified, it may be incorrect.

Solution: Verify the location of the Image Directory and the path specified in the `new_admin.xml` configuration file. If found incorrect in the `new_admin.xml` file, specify the correct path, and deploy the Metrics Tool again.

Cause: The Apache Web server is not up and running or is not available.

Solution: Verify if the Apache Web server is up and running and start the Apache Web server, if required. Before starting the server, make sure that the location of the Image Directory is valid and exists.

Cause: The `DocumentRoot` path in `Apache\conf\httpd.conf` file is not set properly. **Solution:** Specify the `DocumentRoot` path correctly. If the problem persists, specify the Image Directory location to match the `DocumentRoot` path.

Cause: The Metrics Tool displays the connection pool exhausted error.

Solution: Check the values for **Max. Active Connections**, **Max. Idle Connections**, and **Max. Wait for Connection** in the **Data Sources** page of the Tomcat Web Server Administration Tool. Increase the value of **Max. Active Connections** and **Max. Idle Connections** and start the Metrics Tool again.

Note

Make sure that you click **Commit Changes** on the Tomcat Web Server Administration Tool when you edit information in the **Data Sources** page.

Cause: The information specified in the **Data Sources** page of the Tomcat Web Server Administration Tool does not match the information specified in the `new_admin.xml` configuration file. **Solution:** Verify if a mismatch of information occurred. In case of a mismatch, ensure that the information in both places matches.

Cause: The location of the Image Directory specified in the `new_admin.xml` configuration file does not match the location specified in the `DocumentRoot` path of the `Apache\conf\httpd.conf` file. **Solution:** Ensure that the paths are same in both files.

Cause: The parameters related to the display of graphics are not set in the `new_admin.xml` configuration file. **Solution:** Check the `new_admin.xml` configuration file and set the following parameters and their corresponding values:

- **dbname**—The name of the database
- **dbtype**—The type of database used, such as Oracle or Microsoft Access
- **imagedir**—The location or path of the Image Directory
- **imagedirLink**—The link to the Image Directory that stores the images of the graphs and reports in the metrics tool.

Glossary

Term	Definition
Application Program Interface (API)	A set of standards or conventions by which programs can call specific operating system or network services.
authentication	A process of logging in to a secure server to verify your identity.
commercial license	The license you get when you purchase PTC software for commercial purposes.
counted locked license	Although a locked license is managed by a FLEXnet Publisher license server, its usage is restricted to a machine for which it is licensed. The license server allows the use of up to the number of licenses purchased.
datecode	A unique number to identify a specific version of PTC software. The datecode can be found printed on the jacket of the software DVD-ROM.
daemon line	An entry in the license file that defines <ul style="list-style-type: none">• The name of the PTC daemon, which is a customized software program that grants and denies floating PTC licenses by reading the license file• The path where this executable resides, that is in the FLEXnet Publisher installation directory• The path to the FLEXnet Publisher options file, which contains various operating parameters
Design Animation	A software extension that enables users to create animation sequences of parts and assemblies from within Creo Elements/Pro.

Creo Elements/Pro Rights Management Extension	An access control technology which restricts copying or access to intellectual property in Creo Elements/Pro files. Also, see Intellectual property.
educational license	A license used at educational institutes to run PTC software, also known as a students license. This is a time-bound license.
evaluation license	An evaluation license allows use of a software product for a trial period.
FLEXnet Publisher	The license management software that is used to distribute licenses
feature line	<p>The feature or increment line is an entry in the license file that describes the product that the user is allowed to use, as well as any optional software products that are associated with that license. Following is a sample of the feature line:</p> <pre> INCREMENT PROE_FLEX3C ptc_d 31.0 01-jan-2009 1 \ BC24AFC5B76BB74C9366 VENDOR_STRING="VSVER=2.0 \ LO=(0,3,6,7,10,17,32,34,40,45,48,51,55,\ 61,62,66,69,71,73,77,97,106,108,115,126,\ 127,128,133,135,137,158,163,188,191,210)" \ SUPERSEDE vendor_info="VIVER=1.0 \ EXTERNAL_NAME=" \ ISSUER=PTC ISSUED=01-sep-2006 \ NOTICE="PTC_customer" SN=SCN12344,SCN765431\ SIGN2="169C A28A E97F E96E 0A3E 563B FDEB\ 4510 829E 4BF4 25D3 2394 0444 2FD4 6C23 0168\ A8A5 AEBE 54B0 1FF6 B79B DC75 2014 A278 33CC\ 1B90 8647 6A12 F4D6 45BF"\ </pre>
feature name	The feature name is the name of the license that the application requests. It is composed of a product prefix and a configuration ID (for example, PROE_FLEX3C).
floating license	A floating license can be used on more than one system. Floating licenses are distributed by a license server process that runs on a machine. This machine is also called the license server. See license-to-run.
hardware reconfiguration	The process of changing any aspect of the machines currently mapped to the Configuration ID.
increment line	See feature line.

installation directory	The directory in which PTC software is installed. Also call the load point.
intellectual property	Proprietary information stored within Creo Elements/Pro data files.
install number	A number used by the PTC order management system before October 20, 1997, to identify a single software installation.
J-Link	A Java-based toolkit, J-Link allows developers to create Java applications to access a Creo Elements/Pro session.
license	A license grants a user permission to run a PTC product.
license borrowing	A functionality of FLEXnet Publisher 10.8.6.2 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.
license client	A machine or software session that requests a license from another machine or software session.
license file	The <code>license.txt</code> file on each user's disk drive when FLEXnet Publisher is installed and configured. This file contains the information used by FLEXnet Publisher to authorize the use of software.
license information	The data from the PTC License Pack that is used by FLEXnet Publisher to authorize the use of software.
license-locked	Refers to an optional module for use with a specific licensed product, which may be either floating or node-locked. If an optional module is license-locked to a node-locked license, it may only be used on the specifically authorized machine. If an optional module is license-locked to a floating license, it may be used on any machine in the network (in conjunction with that license).
license-locked option	An optional module of PTC software that is locked to a license of PTC software. When the license is successfully started, all license-locked options associated to the licenses are available.
license management	The PTC organization responsible for all installation-based activities.
license management software	See FLEXnet Publisher.

License Pack	The packet of license codes necessary to run a software. It arrives either by e-mail, fax, or mail.
license-to-run	A license-to-run invokes the license of a specific PTC application, such as Creo Elements/Pro. Floating licenses are available for use on any host machine on the network at any particular site. See floating license.
license server	A machine or system process that distributes licenses to license clients. For example, if you start Creo Elements/Pro on machine A and it requests a license from machine B, then machine B is the license server. The license server keeps track of the number of licenses in use. A system administrator can use a license server to control licenses by placing restrictions on a particular feature.
load point	Directory where the software is installed. An example of the Creo Elements/Pro load point on UNIX systems is <code>/usr/local/CreoElementsPro5.0</code> . On Windows, an example is <code>C:\Program Files\PTC\Creo Elements\Pro5.0\bin</code> .
license server manager daemon	FLEXnet Publisher license server manager daemon (<code>lmadmin</code> or <code>lmgrd</code>) runs on the license server and works with the PTC vendor daemon. It does not manage license usage. The <code>lmadmin</code> or <code>lmgrd</code> daemon starts vendor daemons, refers applications to the appropriate vendor daemon, and communicates with other license server manager daemons on the network for Triad installations.
maintenance release request	A request for postproduction PTC software, which features enhancements after a major new revision.
maintenance shipment	An update to the next major release of PTC software.
ModelCHECK	A software productivity tool for Creo Elements/Pro, ModelCHECK analyzes parts, drawings and assemblies, and recommends proper Creo Elements/Pro modeling techniques.
ModelCHECK Metrics Tool	A web-based utility for tracking data quality, standards, and best practices. This tool uses the metric file output from ModelCHECK to calculate quality trends and enables you to represent the ModelCHECK analysis results graphically.
name_service daemon (<code>nmsd</code>)	This daemon process enables remote communication between Creo Elements/Pro and other PTC applications.

node-locked license	A license that can be used on only one specific machine or node. Node-locked licenses are restricted to a specific Host ID, or CPU ID. If the node-locked license does not match the CPU ID of the machine running the software, then the license cannot be used on that machine. Node-locked licenses can be counted or uncounted.
part file	A file that contains Creo Elements/Pro part information (for example, object dimensions).
product prefix	This standard prefix corresponds to a particular product. For example, the product prefix for Creo Elements/Pro is PROE_. This product prefix might be used as a prefix within a feature name, such as PROE_12345, where 12345 is the Configuration ID; or for an environment variable, such as PROE_FEATURE_NAME, which designates the feature that Creo Elements/Pro requests on startup.
Pro/BUNDLE	A type of floating optional module that is comprised of several existing optional modules. PTC licensing recognizes the bundled software as a single licensed entity.
Creo Elements/View Express	A PTC application used to view, mark up, interact, and collaborate on all forms of digital product data. It is a scaled-down version of Creo Elements/View.
Creo Elements/Pro	A 3-D mechanical design automation suite, Creo Elements/Pro provides associative and interoperable product design and an engineering modeling system.
Pro/PHOTORENDER	A Creo Elements/Pro application for creating accurate photorealistic images of Creo Elements/Pro parts and assemblies.
Creo Elements/Pro Plastic Advisor	An application that simulates mold filling for injection molded plastic parts and provides designers with immediate, easy access to reliable manufacturing feedback and advice.
Pro/TOOLKIT	An API that enables adding of functionality to Creo Elements/Pro using the C programming language.
PTC Application Manager	A utility that appears as a toolbar that enables the user to start and close other applications during a Creo Elements/Pro session.
PTC.Setup	An installation utility with menus, dialog boxes, and screens for the installation of PTC products.

PTC.Setup Help	Online user installation instructions displayed in a Web browser.
PTC daemon	The PTC vendor daemon (<code>ptc_d</code>) runs on the license server and works with the FLEXnet Publisher daemon. It manages license usage by reading the PTC license file. The PTC daemon also contains the authentication information for PTC license security.
PTC Host ID	The number used to uniquely identify your machine. This is the same number used in PTCIm, called the CPU ID. Also referred to as the machine address.
reconfigurations	The process of modifying the configuration of software (moving modules from one license to another).
server line	An entry in the license file that identifies the server or servers that distribute the software.
Thumbnail viewer	A Windows-only utility for viewing thumbnails of objects outside Creo Elements/Pro.
startup command	The command that is used to start an installed software product. The startup command is created during installation and is associated with at least one license from the license management component. Modules that are in separate feature lines can also be associated with the startup command. When a startup command is executed, the associated licenses and modules are automatically licensed for use by the user who executed the startup command.
trail file	A trail file records the procedures the end user performed while running a Creo Elements/Pro session.
triad servers	A configuration of three license servers on the same network that work together to manage a pool of floating licenses.

uncounted node-locked license	A node-locked license whose permission to use the license is managed by the PTC application, not the license server. Unlimited number of uncounted node-locked license sessions can be run on a single machine at one time.
Visual Basic API	A visual basic toolkit, Visual Basic API allows developers to create VB.NET or VBA applications to access a Creo Elements/Pro session.
Web.Link	A Creo Elements/Pro feature, Web.Link lets you interact with Creo Elements/Pro through a Web browser. With this feature, you can create a customized Web page, which you can then use to automate and streamline aspects of your engineering process.

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