PTC Creo® 3.0 Installation and Administration Guide

PTC Creo Parametric
PTC Creo Direct
PTC Creo Layout
PTC Creo Simulate
PTC Creo Options Modeler
Datecode M010

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About This Guide

The PTC Creo design software package encompasses the CAD, CAM, CAE, CAID, and Visualization applications of PTC. It provides flexibility to add new interoperable capabilities such as 2D modeling, direct modeling, and BOM-driven assemblies to your software tools. This guide describes how to generate and configure licenses, and how to install and maintain the PTC Creo applications—PTC Creo Parametric, PTC Creo Direct, PTC Creo Layout, PTC Creo Simulate, and PTC Creo Options Modeler—using PTC Installation Assistant. The installation instructions are also common for PTC Creo Distributed Services Manager.

System administrators who are installing the PTC Creo applications for the first time must be experienced in application installations and must have a good understanding of operating systems.

How to Use This Guide

This guide supplements the tooltips in PTC Installation Assistant and provides installation instructions and reference information. To generate or update licenses and then install or reconfigure the PTC Creo applications, see the following chapters:

Chapter	Role	Information
Preparing to Install	All users	Describes the materials in your
Your PTC Software on		software shipment. Also
page 11		provides an overview of PTC
		Installation Assistant.
Quick-Start Installation	Advanced users	Describes a quick-start
for Experienced Users		approach for users who have

Chapter	Role	Information
on page 19		previously installed license management and product software.
Configuring Licenses for the PTC Creo Applications on page 23	All users	Describes how to generate or update licenses and install PTC License Server.
Installing the PTC Creo Applications on page 33	All users	Describes a simple installation process for the PTC Creo applications.
Customizing the Installation of the PTC Creo Applications on page 41	Advanced users	Describes the advanced customization options for installing the PTC Creo applications.
Completing the Installation and Starting the PTC Creo Applications on page 51	All users	Describes the steps to complete the installation of PTC Creo applications and the procedure to uninstall them.
Installing and Using PTC Creo Help on page 63	Advanced users and system administrators	Describes how to install and access Help for PTC Creo Parametric, PTC Creo Direct, PTC Creo Layout, PTC Creo Simulate, and PTC Creo Options Modeler from a local host computer, a network drive, and from a Web server.
Updating an Installation on page 71	All users	Describes how to update the PTC Creo applications on your system.
Installing the PTC Creo Applications in Silent Mode on page 151	System administrators	Describes how to install and uninstall the PTC Creo applications and the client-side components in silent mode.

Related Documentation

The following documents on the Reference Documents page may be helpful as you use this guide:

- Installing PTC Creo 3.0: Creo Schematics
- Installing PTC Creo 3.0: Creo View
- Installing PTC Creo 3.0: Creo Illustrate
- FlexNet Publisher License Administration Guide that discusses the third-party license management software for distributing PTC Creo licenses
- Installing and Configuring the Standalone PTC License Server that discusses the installation of the Imadmin-based PTC License Server

Technical Support

Contact PTC Technical Support via the PTC Web site, phone, fax, or e-mail if you encounter problems using your software. You can log a new case or track an existing case or SPR (Software Performance Report) using the PTC Web page at https://www.ptc.com/appserver/cs/portal. To log a new case, see the eSupport Tools on the left.

You must have a Service Contract Number (SCN) before you can receive technical support. You can find this number on your Shipment confirmation e-mail from PTC or by contacting Customer Care. See the Contact table in the Customer Support Guide.

Documentation for PTC Products

PTC provides documentation for download at PTC.com and also on a DVD. The following forms of documentation for PTC Creo Parametric, PTC Creo Direct, PTC Creo Layout, PTC Creo Simulate, and PTC Creo Options Modeler are available:

- A Help Center with context-sensitive Help, search facility, and quick links to helpful information.
- PTC Creo 3.0 Installation and Administration Guide and other books as PDF files. To view and print the books, you must have the Adobe Acrobat Reader installed.

After you install the Help component, press F1 on a user interface item to get context-sensitive Help. To access the Help Center, click ...

To access all PTC Documentation from PTC.com, you must have a valid user account. Go to the New Account page to request a user account or call the Customer Support.

Feedback to Documentation

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

About This Guide

mcad-documentation@ptc.com

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

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

Documentation Conventions

PTC documentation uses the following conventions:

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

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Preparing to Install Your PTC Software

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This chapter describes the materials in your software shipment. It also provides an overview of how you can use PTC Installation Assistant for obtaining licenses and installing your software.

What You Receive

PTC sends the following materials related to your software order:

- PTC Software Order Confirmation e-mail—Before the receipt of your PTC software, you will receive an e-mail containing all the details of your order.
- Software and Help DVDs—Your order contains one or more DVDs for each
 product purchased. It also contains the Help DVD for the PTC Creo
 applications: PTC Creo Parametric, PTC Creo Direct, PTC Creo Layout, PTC
 Creo Simulate, and PTC Creo Options Modeler in all the supported languages.

Generating Licenses

Both new and existing customers can generate licenses in one of two ways.

- Using PTC Installation Assistant
- Using PTC License Management Web tools—Request your License Pack via e-mail.

New Software Orders and Configurations

For new software orders, configure the license for the purchased products in one of the following ways:

- Use PTC Installation Assistant to generate the license and install PTC License Server.
- Contact PTC License Management directly for licenses.
- Click the Configure New Software link under License Management on the PTC Creo eSupport Center page and follow the instructions.

Maintenance Shipments

If you are updating your PTC software, you can use PTC Installation Assistant to generate a license file. The license generation process is based on the product code for your Maintenance Release shipment. Alternatively, you can request electronic copies of your License Packs by either contacting PTC License Management directly or using the PTC Creo eSupport Center page and clicking Retrieve Existing License Packs under License Management and following the instructions. A License Pack for a Maintenance Release shipment contains all the qualifying licenses for the specified site.

Note

The contents of the License Pack reflect your order. If your configurations have changed since a maintenance release, contact PTC License Management for a new License Pack.

Before You Begin

Before you install your PTC Creo software, make sure that the following prerequisites are met:

- You have opened an online account at PTC.com.
- You have received the Software Confirmation Order e-mail with the product code or the Sales Order Number for your shipment. Alternatively, you must have received your License Pack via e-mail from PTC License Management.
- You have received the product DVD in your software shipment. If not, follow the link on your Software Confirmation Order e-mail to download the installation package to a folder on the local or the network computer.
- You have checked the Hardware Notes for platform support, system requirements, and supported graphics card details at http://www.ptc.com/ WCMS/files/160242/en/PTC Creo Future Platform Support Summary.pdf.
- If you are an existing user, you must have verified whether a FlexNet Publisher license server is available for updating with new license codes.
- You have read the READ THIS FIRST document for installation prerequisites.

Opening a PTC Online Account

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

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

Resetting Your Password

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

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

About PTC Installation Assistant

PTC Installation Assistant generates licenses and installs the PTC Creo applications based on the license sources that you have been provided. You can perform the following tasks using PTC Installation Assistant:

- Generate or update node-locked or floating licenses at runtime
- Drag-and-drop previously downloaded node-locked or floating license files
- Download node-locked licenses to a specific location. In case of floating licenses, install PTC License Server
- Identify license sources listed in a PTC_D_LICENSE_FILE variable set on your computer or if a license server is already running
- View a list of the available applications based on the entitlements listed in the license file
- Download and install the PTC Creo applications from PTC.com
- Perform a simple installation. Alternatively, customize the installation to add specific features and configurations
- Reconfigure an existing installation

See Workflow for Installation and Configuration on page 183 for flowcharts on the installation and reconfiguration tasks.

Accessing PTC Installation Assistant

You can access PTC Installation Assistant from the DVD in your software shipment. You can also download the Assistant from the Web by following the link on your PTC Software Order Confirmation e-mail. After downloading the installation package from the Web, extract the contents to a folder on your computer or onto a network server location depending on how you want to configure your installation.

Privileges for Starting PTC Installation Assistant

To start PTC Installation Assistant on Windows, you must have elevated privileges assigned to you by an administrator. In addition, the startup behavior of the Assistant depends on the following criteria:

- The permission group or the access token associated with the logged-in user account
- The User Account Control (UAC) settings for that account

The startup behavior for the Assistant is summarized below:

Access Token	UAC Status	Type of Prompt	Result
Administrator	Enabled	Prompt for consent	You can start the Assistant after you provide your consent.
Administrator	Disabled	No prompt	You can start the Assistant with a full administrative access token.
Standard User	Enabled	Prompt for credentials	You can start the Assistant with a full administrative access token.
Standard User	Disabled	Not applicable	You cannot start the Assistant.

Starting PTC Installation Assistant

You can start PTC Installation Assistant by running setup.exe from the root directory of either the DVD or the downloaded installation files. Other than using the default of English, you can run the Assistant in French, German, Italian, Spanish, Simplified Chinese, Traditional Chinese, Korean, Japanese, and Russian. See the next section for details. If you are starting the Assistant from a DVD, the Assistant starts automatically if your computer has autorun enabled for your DVD drive. If it does not happen, follow one of these methods:

- Start Windows Explorer, browse to the DVD drive, and double-click the setup.exe icon.
- At an MS-DOS prompt, type <dvd>:\setup.exe, where <dvd> is the DVD drive letter.
- From the task bar, select **Start** ➤ **Run**, and enter <dvd>:\setup.exe.

Working with PTC Installation Assistant

In addition to generating and updating licenses, you can install some of the PTC Creo applications using PTC Installation Assistant. When you start the Assistant, the Welcome screen appears as shown below:



On this screen select a task based on your installation requirements and click **Next** to proceed.

- Install new software
- Upgrade existing software
- · Reconfigure existing software
- Setup license

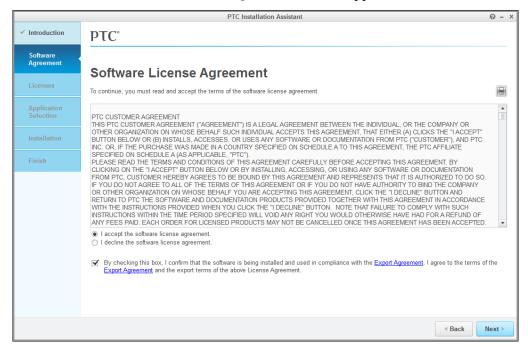
As you perform the various tasks, the processes involved are highlighted on the left pane. Take one of the following actions to end the installation process:

- Click **Install** and then click **Finish** to complete the installation and close the Assistant.
- Click **Finish** to close the Assistant if you are only generating or updating licenses
- Click [X] at the top-right corner of the Assistant to stop the installation and exit the Assistant at any stage during the installation.

PTC Software License Agreement

Before you use PTC Installation Assistant for generating licenses or installing software, you must accept the PTC software license agreement. To do so, follow these steps:

- 1. Start the Assistant and select a task on the Welcome screen.
- 2. Click Next. The Software License Agreement screen appears.

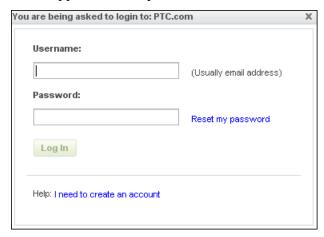


The license agreement is in the language specified for the language environment variable LANG. If you are running the Assistant in a language other than English, click to open a PDF file with the English license agreement.

- 3. Read and accept the license agreement.
 - You must accept the license agreement to proceed even if a license agreement already exists. The license agreement shown above supersedes the previous license agreement.
 - If you decline the license agreement, you cannot proceed. Click **Finish** to stop the installation and exit the Assistant. A page with instructions that you should follow after you decline the license agreement, opens in your default browser window.
- 4. Click the check box at the bottom of the screen to confirm that you are installing and using the software in compliance with the export agreement.
- 5. Click to print the license agreement.

Authenticating Your User Access

If you are accessing PTC Installation Assistant from the Web, the following login screen may appear at any stage after you accept the software license agreement. This happens in case your user access has not already been authenticated:



Perform the following steps for authentication:

- 1. Type your user name using the format <user name@ptc.com> in the Username text box.
- 2. Type the password corresponding to the user name in the **Password** text box.
- 3. If you do not remember your user name or password, click **Reset my** password. You are redirected to a page on PTC.com to reset your password.
- 4. Click Log In.

If necessary, click the link at the bottom of the dialog box to create a new account.

2

Quick-Start Installation for Experienced Users

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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 on using PTC Installation Assistant, read the subsequent chapters in this guide. New users can refer to the section Installing the Creo Applications: Simple Installation Process on page 34.

Updating PTC License Server

If you are updating your software to a later release, you must update your PTC licenses. See Updating the License Server on page 72 for details.

Installing PTC Creo Applications

Install PTC Creo applications one at a time to a client machine and to the license server machine using the following workflow:

- 1. Start PTC Installation Assistant.
- 2. Click Install new software on the Welcome screen.
- 3. Accept the software license agreement.
- 4. Configure a license source (a license file or a license server) using the instructions in the chapter Configuring Licenses for the PTC Creo Applications on page 23. If you have already configured a license source, select it from the **License Summary** area of the **License Identification** screen.
- 5. Select one or more PTC Creo applications to install.
- 6. Select a folder for the installation. To install into a folder other than the default, either type the full path or browse to the folder to select it.
 - To install more than one PTC Creo application at a custom location, make sure to install them in a common folder. The files common to the applications get installed in the <creo_loadpoint>\<datecode>\Common Files folder accessible to all the applications.
- 7. Customize each PTC Creo application using the steps that follow or click **Install** to start a simple installation.
 - a. Click **Customize** after selecting an application.
 - b. Define the platforms, languages, and the other components.
 - c. Modify the license configuration information to associate specific licenses to the application startup command. You can attach multiple licenses to a single command name.
 - d. Configure the Windows shortcut preferences as required.

See Installing PTC Creo Applications, PTC Mathcad, and other Utilities on page 33 for details on the simple installation process. See Customizing an Installation on page 42 for the customization details.

8. Click Finish.

Where Your Software Is Installed

PTC Installation Assistant defines and installs software into a default installation directory that you can modify for your environment during installation. The default path is as follows:

C:\Program Files\PTC\Creo 3.0\<datecode>\<app name>

PTC License Server, the PTC Creo applications, and any related components are installed into separate subdirectories within the PTC directory.



Note

All the 32-bit applications are installed at C:\Program Files (X86) even if you have specified your installation path as C:\Program Files.

Configuring Licenses for the PTC Creo Applications

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This chapter explains how to generate or update licenses using PTC Installation Assistant. It also provides instructions to install PTC License Server, a third-party license management software for your PTC Creo applications. You must install PTC License Server before installing your PTC Creo applications unless you have purchased uncounted node-locked licenses. Refer to your sales documents for your license type. For the license server version requirements appropriate for your installation, see the READ THIS FIRST.

PTC uses FlexNet Publisher from Flexera Software, Inc. as its license server. See License Management Software on page 77 for an overview and benefits of the license management software. The *FlexNet Publisher License Administration Guide* may be helpful as you install PTC License Server.

Using PTC Installation Assistant for Licensing

You must generate or update exiting licenses to use the latest software for PTC Creo 3.0. Licenses determine the PTC Creo applications that you are authorized to run on your computer. The Assistant helps you perform the following licensingrelated tasks:

- Generate a new license file based on your product code
- Update and reuse an existing license file on your computer or at a remote server location
- Install a single or a triad license server

Using the License Server Manager

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

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

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

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

PTC currently ships with its products a license server manager based on lmadmin with the ptc d vendor daemon at version 11.11.1. It is a replacement to the lmgrd-based license server manager shipped with the earlier releases of PTC products. If you are using an lmgrd-based license server manager with ptc d vendor daemon of version 10.8.x, it is recommended that you migrate to lmadmin although it is not necessary.



Note

On a 64-bit Windows operating system, if you are installing FlexNet Publisher with a lmadmin-based license server manager, a 32-bit version of the FlexNet Publisher is installed in the C:\Program Files folder.

Verifying System Requirements

Installation requirements for the PTC Creo applications in this guide follow:

- For Windows 7, you must have administrative privileges to install PTC License Server
- You must have TCP/IP (Transmission Control Protocol/Internet Protocol) installed and configured correctly on your Windows system before installing the software.
- To use the lmadmin-based license server manager on Windows platforms, you must install the Microsoft Visual C++ 2010 Redistributable Package(x86).



Note

For information on platform support for the PTC Creo applications, see http:// www.ptc.com/WCMS/files/160242/en/PTC Creo Future Platform Support Summary.pdf.

Before You Proceed

Gather all the necessary information that follows about the installation:

- Check the following documents for the latest installation settings:
 - The Hardware Notes for platform support, system requirements, and supported graphics card details at http://www.ptc.com/WCMS/files/ 160242/en/PTC Creo Future Platform Support Summary.pdf
 - The READ THIS FIRST document of your product for any changes to your computer settings
 - The FlexNet License Server Hardware Notes for license server installation requirements at www.ptc.com/WCMS/files/121640/en/FlexNet Licensing. pdf
- Make note of your product code that arrives via e-mail. If you have received a license file via e-mail, store it at a secure location on your disk.
- If you have already installed PTC License Server based on lmgrd and choose not to migrate to lmadmin, you must still update the license server with license codes compatible with PTC Creo 3.0.
- If you have already installed PTC License Server based on lmgrd and choose to migrate to lmadmin, complete these steps before the migration:

- 1. Shut down any lmgrd service or processes running on the system.
- 2. Save the license.dat file from your existing license server installation to a secure location.
- 3. Uninstall the previous installation completely.

See Migrating from Imgrd to Imadmin, in the *FlexNet Publisher License Administration Guide* for more information.

About the Product Codes

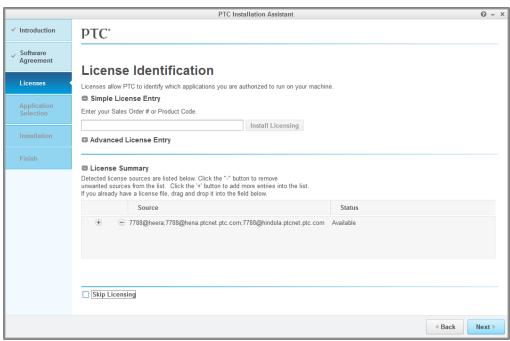
A product code is a unique serial number corresponding to the license that you are requesting. An example of a product code is BH381111ED12345H4B5CXD. Product codes are nontransferable. They can be used by only one individual on only one computer. Product codes are usually provided by PTC when your software order is confirmed. You can use the product codes to generate licenses using PTC Installation Assistant.

Generating or Updating Licenses

After you have received an e-mail from PTC with the Sales Order Number (SON) or product codes for your software order, generate or update licenses using PTC Installation Assistant as follows:

- 1. Start the Assistant. The Welcome screen appears.
- 2. Select either **Setup license** or **Install new software** depending on the tasks that you want to perform:
 - Setup license generates or installs licenses.
 - **Install new software** performs license generation or identification tasks after which you can install the software.
- 3. Click Next. The Software License Agreement screen appears.
- 4. Read and accept the software license agreement and click the check box at the bottom of the screen to accept the export agreement.
- 5. Click **Next**. The **License Identification** screen appears. The **License Summary** area shows the available license sources installed using the Assistant or the

licenses found on the computer. The license availability is shown in the **Status** column.



6. Click **Skip Licensing** if you want to proceed with the installation without providing the licensing information. You can provide the licensing information at a later stage by re-running the PTC Installation Assistant.



If you click **Skip Licensing**, the **Command Configuration** tabbed page on the **Application Custom Setting** dialog box is not available during customizing the PTC Creo application. You can add, edit, or delete a license configuration using the **Command Configuration** tabbed page on the **Application Custom Setting** dialog box.

Using the Simple License Entry Options

Use the **Simple License Entry** area on the **License Identification** screen and perform the following steps:

- 1. Type your sales order number or product code in the corresponding text box.
- 2. Click **Install Licensing**. If the sales order number matches your software order, all the available licenses corresponding to the sales order number are automatically generated into the license file.

- For node-locked licenses, a single set of available node-locked license
 features are added to the license file and downloaded to the "C:\
 ProgramData\PTC\licensing\ folder.
- For floating licenses, all the available licenses are added to the license file and the PTC FlexNet Publisher license server is installed.

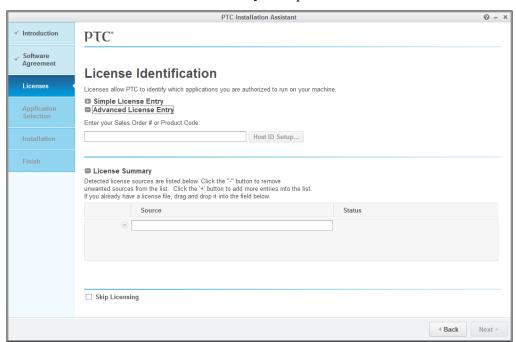
The complete path of the license file or the port@hostname value of the license server appears in the **Source** column of the **License Summary** area. This area displays all the license files or servers available for use on your computer.

- 3. To complete the license setup, click one of the following buttons depending on the options you have selected on the Welcome screen:
 - Click Finish to close the Assistant if you have selected **Setup license**.
 - Click Next if you have selected Install new software to proceed with the installation.

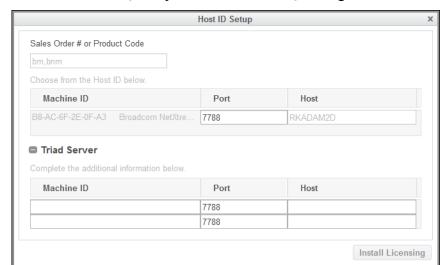
Using the Advanced License Entry Options

Use the **Advanced License Entry** area on the **License Identification** screen to edit the default license port and for the setup of a Triad license server as follows:

1. Click • next to Advanced License Entry to expand the area.



2. Type your Sales Order Number or your product code in the corresponding text box.



3. Click Host ID Setup to open the Host ID Setup dialog box.

The sales order number or the product code from the previous step appears in the dialog box. The **Machine ID**, **Port**, and the **Host** columns for the local computer is shown.

- a. To change the default port for a single license, edit the number in the **Port** text box and click **Install Licensing**. Then skip to the last step in this section.
- b. For triad server installations, follow these steps:
 - i. Click next to Triad Server to expand the area.
 - ii. Provide the host ID, port number, and the host name for the other two servers.
 - iii. Click **Install Licensing**. An authentication dialog box to connect to PTC.com may open if your user access has not already been validated. If so, provide your credentials and click **Log In**.

The license server is installed and appears in the **License Summary** area of the **License Identification** screen.

- 4. To complete the license setup, click one of the following buttons depending on the options you have selected on the Welcome screen:
 - Click Finish to close the Assistant if you have selected **Setup license**.
 - Click Next if you have selected Install new software to proceed with the installation

Installing the License Server at a Custom Location

If required, you can install the license server at a custom location, as follows:

- 1. Copy the PTC Creo product installation DVD or the product installation package that you downloaded from the web, to a temporary location (<temp_folder>) on your disk.
- 2. Edit the <temp_folder>/pim/xml/flexadmin.xml file and modify the INSTALLBASE name attribute of the PROPERTY node to specify the custom installation location for the license server as shown:

```
<PROPERTY name="[INSTALLBASE]">[PROGRAMFILES]\PTC</PROPERTY>
```

Modify to:

<PROPERTY name="[INSTALLBASE]">D:\NEWFLEXDIR</PROPERTY>

Where, D: \NEWFLEXDIR is the custom installation location for the license server.

- 3. Save the <temp folder>/pim/xml/flexadmin.xml file.
- 4. Start the PTC Installation Assistant from the locally copied <temp_folder> folder> folder, and install the license server. The license server is installed at the custom location D: \NEWFLEXDIR.

Note

- PTC recommends that you do not edit the flexadmin.xml file unless there is a specific need to do so.
- You can edit only those nodes that are documented in this section.

Specifying License Entry for Previously Generated License Files

Using the **License Summary** area, you can install licenses using the license files that you have already received as follows:

- 1. Drag and drop or type the path to the license file.
- 2. Click outside the **License Summary** area to begin the installation of the license server for floating licenses. For the node-locked licenses, this step checks the host ID of the local computer.

Note

Node-locked licenses that you specify using the **License Summary** area are not remembered in the subsequent sessions unless you add them specifically to the PTC D LICENSE FILE variable.

3. Click the or the buttons in the **License Summary** area to add or remove license sources from the current session.

Note

Removing license sources does not delete or uninstall license files or the license server.

- 4. To complete the license setup, click one of the following buttons depending on the options you have selected on the Welcome screen:
 - Click Finish to close the Assistant if you have selected **Setup license**.
 - Click **Next** if you have selected **Install new software** to proceed with the installation

4

Installing PTC Creo Applications, PTC Mathcad, and other Utilities

Installation Prerequisites	34
File Format Compatibility	
Installing the PTC Creo Applications: Simple Installation Process	34
Using PTC Quality Agent with the PTC Creo Applications	38

This chapter provides information on how to install the following PTC products:

- PTC Creo applications:
 - o PTC Creo Parametric
 - o PTC Creo Direct
 - o PTC Creo Layout
 - o PTC Creo Simulate
 - o PTC Creo Options Modeler
 - o PTC Creo Distributed Services Manager
- PTC Mathcad
- Utilities

Installation Prerequisites

See Before You Begin on page 13 to check if all the installation prerequisites are met. Install the PTC Creo applications and PTC Creo Distributed Services Manager after you have obtained a license and installed PTC License Server. See the chapter Configuring Licenses for the PTC Creo Applications on page 23 for details on how to obtain licenses for your software. 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.
- License configuration is a part of the installation process.

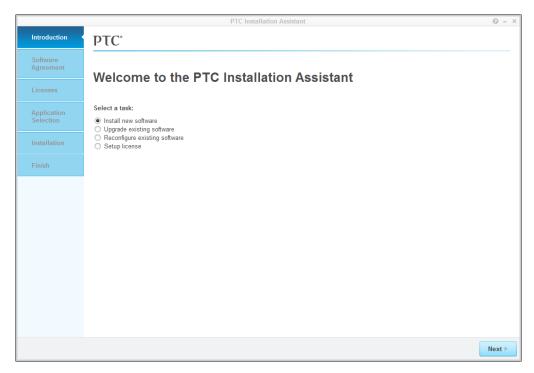
File Format Compatibility

You can retrieve files (parts, assemblies, drawings) created in a previous release of a PTC Creo application using the current version of the same application.

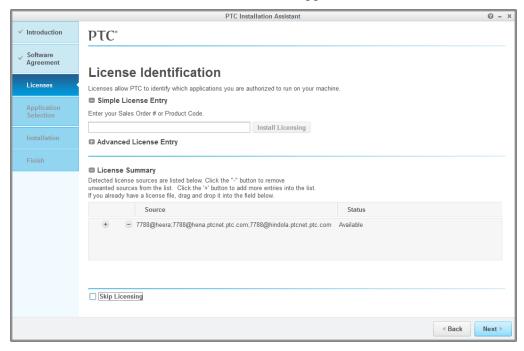
Installing the PTC Creo Applications: Simple Installation Process

Using PTC Installation Assistant, you can install the PTC Creo applications and PTC Creo Distributed Services Manager using a simple installation process. You can also customize their installation using this installer. A simple installation process is described below:

- 1. Start the Assistant. The Welcome screen appears.
- Click Install new software.



- 3. Click Next. The Software License Agreement screen appears.
- 4. Accept the agreement and click the check box at the bottom of the screen to confirm that you are installing the software in compliance with the export agreement.
- 5. Click Next. The License Identification screen appears.



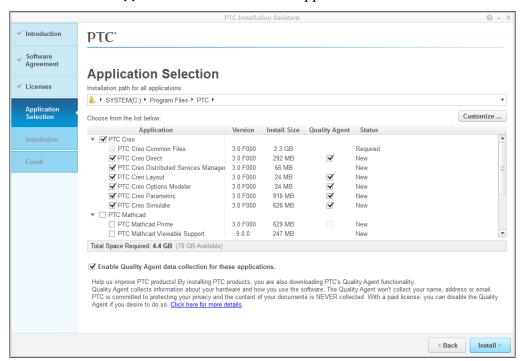
6. If you have not already obtained your licenses or installed PTC License Server, follow the instructions in the chapter Configuring Licenses for the PTC Creo Applications on page 23.

If you have configured your licenses, confirm that your desired license source appears in the **License Summary** area, or add it using one of the following methods:

- Drag and drop a valid license file into the Source column for a nodelocked license.
- Specify the port@hostname value of the license server in the **Source** column for a single floating license server.
- Specify the name of a member of the Triad license server in the Source column. The License Summary area is automatically populated with the entire Triad license server information.

After you provide the correct license information, the availability of the license appears in the **Status** column.

7. Click Next. The Application Selection screen appears.



8. Specify an installation folder or retain the default folder.

Selecting Application for Installation

Select one or more applications from the list:

PTC Creo:

- o PTC Creo Direct
- PTC Creo Distributed Services Manager
- PTC Creo Layout
- PTC Creo Options Modeler
- PTC Creo Parametric
- o PTC Creo Simulate

PTC Mathcad:

- PTC Mathcad Viewable Support Enables PTC Mathcad Prime when integrated with Windchill to generate PDF rendering of the worksheet. This component is required when using Windchill integration.
- PTC Mathcad Prime.

For information on installation of standalone PTC Mathcad Prime, refer to the *PTC Mathcad Prime 3.0 Installation and Administration Guide* here.

Note

To enable the integration of PTC Mathcad Prime with any of the PTC Creo applications, ensure that you have installed both, the PTC Mathcad Prime and the corresponding PTC Creo application.

Utilities:

- Microsoft .NET Framework 4 Setup
- PTC Creo Platform Agent
- o PTC Creo Thumbnail Viewer
- PTC Creo View Express
- PTC MKS Platform Components 9.x
- o PTC Quality Agent

Note

Required utilities for applications that you select to install, are automatically marked for installation. You can deselect the PTC Creo Thumbnail Viewer, and PTC Creo View Express for installation. The PTC MKS Platform Components 9.x is not selected by default.

Before selecting applications from the list, check if you are installing the correct version of the software and whether the disk space requirements are met.

Perform one of the following steps:

- Click **Install** to install the applications without any customization.
- Select the applications and click **Customize** to customize their installation. The **Application Custom Setting** screen appears with the applications selected for customization on the left pane. Refer to the chapter Customizing the Installation of the PTC Creo Applications on page 41 for more information.

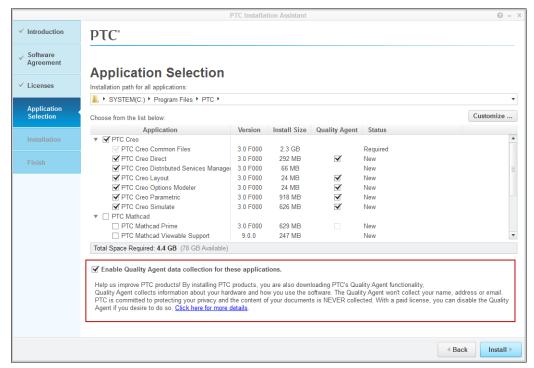


Note

Some of the applications on the **Application Selection** screen cannot be customized. For such applications, the **Customize** button is not available.

Using PTC Quality Agent with the PTC Creo Applications

PTC Quality Agent is a required utility that gets installed by default when you install any of the PTC Creo applications. Click or clear the check box at the bottom of the Application Selection screen for enabling or disabling PTC Quality Agent reporting. By default, PTC Quality Agent reporting is always enabled.



PTC Quality Agent helps you to gather and send reports to PTC on the performance and usage of the PTC Creo applications. Before clicking the check box, read the instructions in the data collection document whose link is provided at the bottom of the screen. See Installing PTC Quality Agent on page 141 for more information on the uses of PTC Quality Agent.

Customizing the Installation of the PTC Creo Applications

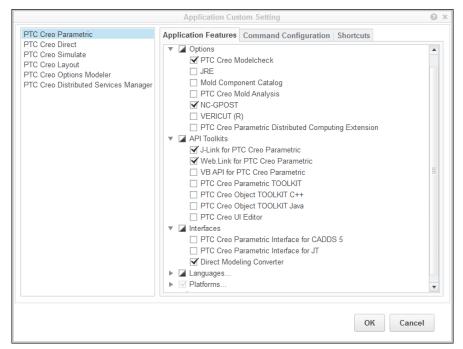
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This chapter explains how to customize an installation of the PTC Creo applications. This information also applies to PTC Creo Distributed Services Manager. Some of the application features in this chapter are specific to individual PTC Creo applications.

Customizing an Installation

You can customize the installation of a PTC Creo application using the following procedure:

- 1. Perform steps 1 through 8 as described in the section Installing the PTC Creo Applications: Simple Installation Process on page 34.
- 2. Select the applications on the **Application Selection** screen and click **Customize**. The **Application Custom Setting** dialog box opens. The selected applications for customization appear on the left pane. The customization tabbed pages appear on the right pane as shown in the following figure.



- To install customized application features using the **Application Features** tabbed page, see Defining the Installation Components on page 43.
- To modify license configurations using the **Command Configuration** tabbed page, see Modifying License Configurations on page 46.
- To configure Windows shortcut preferences using the **Shortcuts** tabbed page, see Configuring Windows Shortcuts on page 48.
- 3. Click an application on the left pane and then click a tabbed page on the right pane to proceed.
- 4. After selecting the options on all the tabbed pages, perform one of the following actions:

- Click **OK** to finish the customization and return to the **Application Selection** screen.
- Click Cancel.
- 5. Complete the installation of the PTC Creo application. See Completing the Installation and Starting the PTC Creo Applications on page 51 for details.

Defining the Installation Components

Using the **Application Features** tabbed page, you can define the following components for installation. Refer to the Application column for the application with which you can install the component.

Component	Application	Description
<application_name></application_name>	All the PTC Creo applications and PTC Creo Distributed Services Manager	Installs the PTC Creo application and its components. By default, the components are installed at <creo_loadpoint>\ <datecode>\ <app_name> and the startup scripts are installed at <creo_loadpoint>\ <datecode>\<app_name> and the startup scripts are installed at <creo_loadpoint>\ <datecode>\<app_name></app_name></datecode></creo_loadpoint></app_name></datecode></creo_loadpoint></app_name></datecode></creo_loadpoint>
Options	PTC Creo Parametric	Installs additional components such as PTC Creo Modelcheck, JRE, Mold Component Catalog, PTC Creo Mold Analysis, NC-GPOST, VERICUT, and PTC Creo Parametric Distributed Computing Extension with PTC Creo Parametric. See Installing PTC Creo Mold Analysis (CMA) Component on page 45 for information on installing PTC Creo Mold Analysis.
API Toolkits	PTC Creo applications and PTC Creo Distributed Services Manager	Installs the files necessary to run the Application Program Interface toolkits.

Component	Application	Description
Interfaces	PTC Creo	Installs the external interfaces.
		₱ Note
		To install PTC Creo
		Parametric Interface for
		CADDS 5 , you must also select MKS Platform
		Components 9.x for
		installation on the
		Application Selection
		screen. See Installing
		MKS Platform
		Components on page 149 for more information.
Platforms	All the PTC Creo	Selects your machine's
	applications and PTC	architecture automatically as
	Creo Distributed	the default platform for
	Services Manager	installing the components.
		PTC provides separate software installer (DVD media and downloadable package) for the 32-bit and 64-bit systems.
Languages	All the PTC Creo	Selects the languages for the
	applications and PTC	installation. The software is
	Creo Distributed	installed in English by default and cannot be disabled.
Enable Quality Agent	Services Manager All the PTC Creo	
Reporting	applications	If enabled, collects information related to the
	аррисацонз	PTC Creo applications and
		other system-related
		information and sends it to
		PTC. See Installing Quality
		Agent on page 141 for more
		information on Quality
Verification Models	PTC Creo Simulate	Agent. Installs the verification
VOLITICATION MICUEIS	1 1 C CIEO Simulate	models as described in the
		Simulation Verification Guide
		Help.

Component	Application	Description	
PTC Creo Parametric Distributed Computing Extension	PTC Creo Simulate	Installs this component to run distributed computing tasks that include import/export, printing operations, and studies in case of PTC Creo Simulate.	
Direct Modeling Converter	PTC Creo Parametric, PTC Creo Simulate, PTC Creo Direct	Installs files that are required to convert and open PTC Creo Elements/Direct files.	

Installing PTC Creo Mold Analysis (CMA) Component

PTC Creo Mold Analysis (CMA) is an injection molding simulation application. You can use CMA to verify the part design for manufacturing to avoid the molding defects. CMA uses the IntelMPI process to launch parallel computing. The smpd-intel-4.0.3.009-x64 or smpd-intel-4.0.3.009-x86 process on your system verifies whether the IntelMPI is installed or not.

You can install IntelMPI using following procedure:

- 1. Open a command window as an administrator.
- 2. Browse to the location of the smpd-intel-4.0.3.009-x64.exe file, for a 64-bit system. If you install the PTC Creo Mold Analysis component with the default installation path for the PTC Creo Parametric application, the path is C:\Program Files\PTC\Creo 3.0\<datecode>\Common Files\x86e_win64\cma\Bin\IntelMPI\. Similarly, the path for the 32-bit system is C:\Program Files\PTC\Creo 3.0\<datecode>\Common Files\i486 nt\cma\Bin\IntelMPI\.
- 3. Run the executable file to start the installation process. For example, run the following command for a 64–bit system:

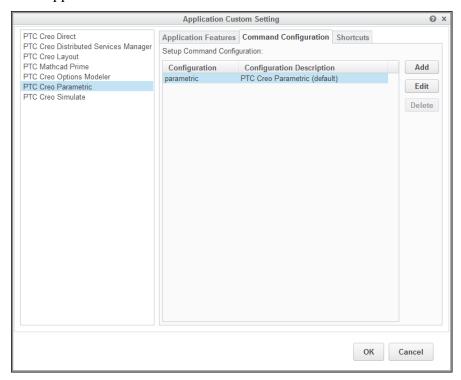
```
smpd-intel-4.0.3.009-x64.exe-install
```

Similarly, run the following command for a 64-bit system to uninstall the PTC Creo Mold Analysis component:

smpd-intel-4.0.3.009-x64.exe-uninstall

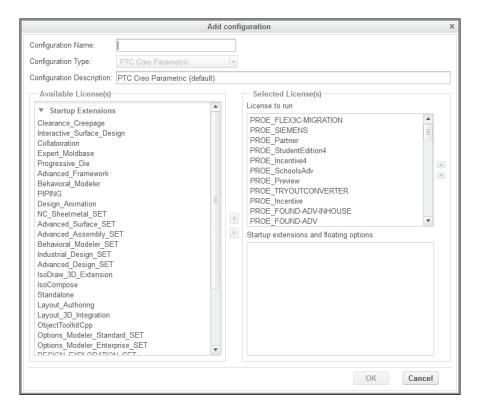
Modifying License Configurations

Using the **Command Configuration** tabbed page, you can add, edit, or delete a license configuration. The figures in this section show the default license configurations for PTC Creo Parametric. Similar screens appear for the other PTC Creo applications.



Perform the following steps:

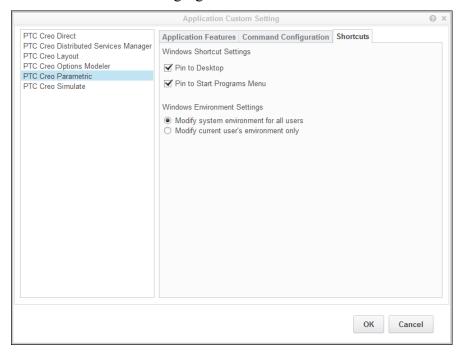
1. Click **Add** to add additional configurations. The **Add configuration** dialog box opens.



- a. In the **Configuration Name** box specify a file name (*.psf) for the license configuration information. Use alphanumeric characters without spaces to define the configuration name.
- b. From the **Configuration Type** list, select a configuration type.
- c. In the **Configuration Description** box, type a description for each of the license configurations that you have defined. This information is displayed on the startup of PTC Creo Parametric. The description can contain spaces and other characters.
- d. The Available License(s) column displays the set of all licenses to run and the startup extensions and floating licenses. Highlight the required license in this column and move it to the Startup extensions and floating options column.
- e. Click **OK**. The newly added license configuration appears on the **License Configuration** tabbed page.
- 2. To edit a license configuration, select it on the **Command Configuration** tabbed page and click **Edit**. Use the options on the **Edit configuration** dialog box for editing.
- 3. To delete a license configuration, select it and click **Delete**.

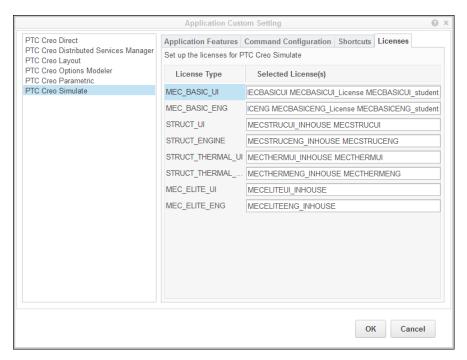
Configuring Windows Shortcuts

Using the **Shortcut** tabbed page, you can configure the Windows shortcut preferences for the PTC Creo applications. Set the Windows shortcut settings that are shown in the following figure:



Setting Licenses for PTC Creo Simulate

The Licenses tabbed page appears when you click PTC Creo Simulate.



Using this tabbed page you can configure licenses for PTC Creo Simulate. A list of license types against the selected licenses appears on this tabbed page. Type a value in the **Selected License(s)** text box to change the license for the selected license type.

Completing the Installation and Starting the PTC Creo Applications

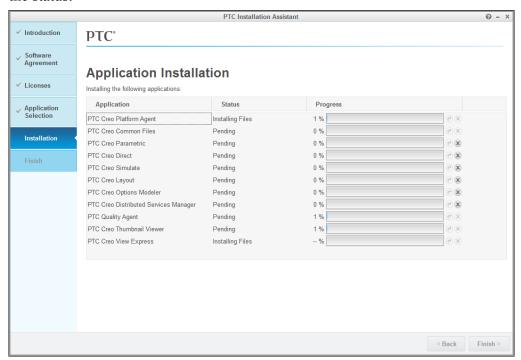
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This chapter explains how to complete an installation of a PTC Creo application. This information is common to PTC Creo Distributed Services Manager. Information on how to start the PTC Creo applications and configure their startup is also given. Proceed to the end of the chapter for the procedure to uninstall the PTC Creo applications.

Completing the Installation

After configuring your settings, follow these steps to complete the installation:

- 1. Click **Back** and review information to make sure that your settings are correct.
- 2. Click Install to begin the installation. The Installation Progress screen displays the status.



Note

Utilities that are required for applications that you select on the **Application Selection** screen, get installed prior to the installation of the selected applications.

On this screen you can also perform the following operations using buttons to the right of the progress bars and below them:

- Click to stop an installation.
- to restart an installation.
- Click Finish after the applications are installed to close the Assistant. You can also click [X] at the top-right corner of the Assistant to exit the installer.

Installing and Uninstalling the PTC Creo Applications in Silent Mode

You can install and uninstall all the PTC Creo applications included in this guide in silent mode using command-line arguments. See Installing the PTC Creo Applications in Silent Mode on page 151 for details.

Starting the PTC Creo Applications

After your license management and product software have been installed, start the PTC Creo applications. You can run the applications using the shortcut defined on your desktop or using the **Start** menu. Click **All Programs** ▶ **PTC Creo** and select the PTC Creo 3.0 application that you want to start.

Configuring the PTC Creo Application Startup

When you configure the startup of a PTC Creo application, an executable file named <app>.exe, for example, parametric.exe and one or more configuration files (.psf) are created for each startup configuration. The configuration file contains an area where you can specify and set user-defined environment variables. This user-defined information is preserved during reconfiguration and update installations of the PTC Creo applications. A configuration file is created when you complete the installation of a PTC Creo application.

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

You can run creo_loadpoint>\<datecode>\<app_name>\bin\<app_name>.exe to display a list of the available startup configurations.

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

parametric.exe <filename>.psf

In this case, <filename> is the name of the configuration file.

A sample configuration file follows. The text is the default information that is used when you start PTC Creo Parametric. The bold text following the // USER - PSF section is the user-defined information for starting PTC Creo Parametric.

Note

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

```
// PTC - PSF file: parametric
ENV=PATH+=<creo loadpoint>%PRO MACHINE TYPE%\deflib
ENV=PATH+=<creo loadpoint>%PRO MACHINE TYPE%\lib
ENV=PRO COMM MSG EXE=<creo loadpoint>%PRO MACHINE TYPE%\
OBJ\pro comm msg.exe
ENV=CDRS DATA=<creo loadpoint>
ENV=PRO IMAGE CONVERTER=<creo loadpoint>%PRO MACHINE TYPE%\obj\convert image.exe
ENV=SPG DIRECTORY=<creo loadpoint>
ENV=PROTABLE DIRECTORY=<creo loadpoint>\protable
ENV=PROTAB=<creo_loadpoint>%PRO_MACHINE_TYPE%\OBJ\protab.exe
ENV=CLHOME=<creo loadpoint>\text\pcldata
ENV=CV ENV HOME=<creo loadpoint>%PRO MACHINE TYPE%\CV102
ENV=LM LICENSE FILE=7788@static
ENV=CREOPMA FEATURE NAME=CREOPMA BASIC
RUN=<creo loadpoint>%PRO MACHINE TYPE%\nms\nmsd.exe -timeout 300
RUN=<creo loadpoint>%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 startup of a third-party application under the // USER - PSF section of the configuration file.

If you run PTC Creo application without creating a configuration file, a error message indicates that the PTC Creo application cannot be started. In such cases you must run ptcsetup from the <creo_loadpoint>\<datecode>\<app_name>\bin\ folder and create a PTC Startup File (PSF) for that PTC Creo application. You can then run that PTC Creo application using corresponding PSF file.

If you create only one configuration file for a PTC Creo application, that application uses the configuration file by default.

Specifying Environmental Variables

You can use a configuration file to specify the value of an environmental variable in the configuration file, follow these instructions:

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

Reading of Configuration Files at Startup

A PTC Creo application automatically reads configuration files from several locations. If an option is in more than one configuration file, the most recently loaded or the most recently read setting is applied.

At startup, a PTC Creo application first reads in a protected system configuration file called config.sup. It then searches for and reads in the configuration files (config.pro, config.win, menu_def.pro) from various directories in the following order:

- 1. <creo_loadpoint>\<datecode>\Common Files\text (creo_loadpoint is the PTC Creo application installation directory)—Your system administrator may have put configuration files in this location to support company standards for windows configuration settings, formats, and libraries. Any user starting a PTC Creo application from this loadpoint uses the values in this file.
- 2. HOME environment variable—You can set this environment variable to point to a directory created for your login ID. You can place your configuration files here to start the PTC Creo application from any folder without having to copy the files in each folder.
- 3. Startup directory—This is your current or the working directory when you start a PTC Creo application. Since the local configuration files (config.pro, config.win, and menu_def.pro) in this directory are the last to be read, they override any conflicting configuration file option entries. The config.pro file does not, however, override any config.sup entries.

Generating a Debug Log File

The debug log file contains the environment variables that you have set. You can run the <app_name>.exe executable in debug mode by setting the value of the environment variable APPL_STARTUP_LOG to true. A debug file named applstarter.log is created in the C: \ drive.

Directory Permissions to Start PTC Creo Applications

You cannot start a PTC Creo application from a directory without write permissions unless you specify an alternative directory to store the trail files. If you start a PTC Creo application from a directory without write permissions, a dialog box opens as shown in the following example:



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.

Installing PTC Creo Platform Agent

When you begin the installation of a PTC Creo application, an appropriate version of the PTC Creo Platform Agent gets installed. PTC Creo Platform Agent is a system level component of PTC Creo that acts as a broker connecting other independent PTC Creo components. Installation of PTC Creo Platform Agent enables the following functionality:

- Invokes the LearningConnector for PTC Creo application
- Facilitates PTC Windchill SocialLink integration with the PTC Creo applications

See Installing and Uninstalling Client-Side Components in Silent Mode on page 164 for instructions on performing a silent installation of PTC Creo Platform Agent.

Configuring a PTC Creo Parametric Installation

After you have installed PTC Creo Parametric, you can perform the following operations either by setting configuration options before starting PTC Creo Parametric or by installing the necessary components at startup

- Change the units of measure by setting configuration options in your config.pro file before PTC Creo Parametric startup
- Allocate memory for running PTC Creo Parametric on 32-bit platforms by setting a configuration option before or after starting PTC Creo Parametric
- Generate a traceback log file by setting a configuration option before or after starting PTC Creo Parametric

Changing the Units of Measure for PTC Creo Parametric

You can change the units of measure for your PTC Creo Parametric installation. Use the settings in the next table to update values for the configuration options in your config.pro file:

	T	<u></u>
Configuration Option	English	Metric
drawing_setup_ file	<pre><creo_loadpoint>\ <datecode>\Common Files\ text\prodetail.dtl</datecode></creo_loadpoint></pre>	<pre><creo_loadpoint>\ <datecode>\Common Files\ text\iso.dtl</datecode></creo_loadpoint></pre>
<pre>format_setup_ file</pre>	<pre><creo_loadpoint>\ <datecode>\Common Files\ text\prodetail.dtl</datecode></creo_loadpoint></pre>	<pre><creo_loadpoint>\ <datecode>\Common Files\ text\iso.dtl</datecode></creo_loadpoint></pre>
<pre>pro_unit_ length</pre>	unit_inch	unit_mm
pro_unit_mass	unit_pound	unit_kilogram
template_designasm	<pre><creo_loadpoint>\ <datecode>\Common Files\ templates\ inlbs_asm_design.asm</datecode></creo_loadpoint></pre>	<pre><creo_loadpoint>\<datecode> \Common Files\templates \mmks_asm_design.asm</datecode></creo_loadpoint></pre>
template_ drawing	<pre><creo_loadpoint>\ <datecode>\Common Files\ templates\c_drawing.drw</datecode></creo_loadpoint></pre>	<pre><creo_loadpoint>\ <datecode>\Common Files\ templates\a3_drawing.drw</datecode></creo_loadpoint></pre>
template_ sheetmetalpart	<pre><creo_loadpoint>\ <datecode>\Common Files\ templates\ inlbs_part_sheetmetal. prt</datecode></creo_loadpoint></pre>	<pre><creo_loadpoint>\ <datecode>\Common Files\ templates\ mmks_part_sheetmetal.prt</datecode></creo_loadpoint></pre>
template_ solidpart	<pre><creo_loadpoint>\ <datecode>\Common Files\ templates\ inlbs_part_solid.prt</datecode></creo_loadpoint></pre>	<pre><creo_loadpoint>\ <datecode>\Common Files\ templates\ mmks_part_solid.prt</datecode></creo_loadpoint></pre>
todays_date_ note_format	%Mmm-%dd-%yy	%dd-%Mmm-%yy

Configuration Option	English	Metric
tolerance_ standard	ansi	iso
tol_mode	nominal	nominal
weld_ui_ standard	ansi	iso

Allocating Memory on a 32-bit Windows System for Running Creo Parametric

When working with large models or a large data set in PTC Creo Parametric on a 32-bit Windows system, processes may use all available memory and exit prematurely. To receive an early warning, use the proe_memory_buffer_size configuration option. You can save you work under some situations.

Specify the size of the memory buffer in megabytes as a reserve for a possible outof-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.

Consider the following guidelines when setting your memory buffer:

- The proe_memory_buffer_size configuration option is processed at PTC Creo Parametric startup. If you change its value in the current session, exit the PTC Creo Parametric 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, an out-of-memory situation is detected at an early stage.
 In such a case, you increase the probability of saving your work before the
 available memory is used.
- When you reserve memory, the virtual memory available to other applications running simultaneously is reduced. If you specify a large value for the proememory_buffer_size configuration option, it is recommended that you increase the system settings for the size of the virtual memory paging file.

Using a Zero or Nonzero 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.

If you specify a nonzero value for the proe_memory_buffer_size configuration option, memory usage is checked periodically 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, an error message appears to indicate that the system is running short of memory. You are also informed that if you continue with the current operation, the session may terminate.

Respond in one of the following ways:

- Click **Cancel**. Save your work, exit PTC Creo Parametric, and restart the session.
- Click **OK** to continue. In this case PTC Creo Parametric may exit as soon as the preallocated memory is exhausted.

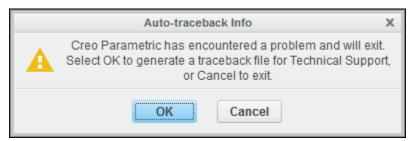
For operations you cannot cancel, a message appears to indicate that the system is running short of memory. This may cause the session to terminate. Click **OK** to continue. After you complete the operation, save your work, exit the PTC Creo application, and restart the session.

Generating a Traceback Log File for PTC Creo Parametric

In the event of a premature exit, PTC Creo Parametric 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 message appears to indicate the premature exit of PTC Creo Parametric:



The traceback.log file can sometimes be useful to Technical Support to resolve the premature exit issue. If you click **Cancel**, PTC Creo Parametric exits without creating the traceback.log file. If you click **OK**, the following message appears:



PTC Creo Parametric exits after creating the traceback.log file in the current working directory.



Note

If you start PTC Creo Parametric 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.

Working with CADDS 5 Models in PTC Creo **Parametric**

You can import and export 3D CADDS 5 parts and assemblies within PTC Creo Parametric by installing the CADDS 5 interface during the installation of PTC Creo Parametric. To do so, select PTC Creo Parametric Interface for CADDS 5 under Interfaces on the Application Features tabbed page on the Application **Custom Setting** screen. To use this interface, consider the following instructions:

After you have installed PTC Creo Parametric Interface for CADDS 5 with PTC Creo Parametric, a configuration file config cadds.pro is automatically saved in the <creo loadpoint>\<datecode>\Parametric\text\ folder. This file contains configuration options whose settings determine how the CADDS 5 interface works with PTC Creo Parametric. 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 Parametric.
! It allows to find user's CADDS 5 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:
 - PTC Creo Parametric Interface for CADDS 5
 - PTC MKS platform components. See Installing PTC MKS Platform Components on page 149 for installation instructions.
 - PTC Portmapper that is installed as a part of the PTC MKS platform components. Click Start ➤ Control Panel ➤ Programs and Features to open

the Uninstall or change a program screen. You must see PTC Portmapper as an installed program.

Uninstalling the PTC Creo Applications

You can remove an installation of a PTC Creo application through the Control Panel as follows:

- 1. Click Start ➤ Control Panel ➤ Programs and Features. The Uninstall or change a program screen appears.
- 2. In the application list, click the specific release or datecode of the application you want to remove.
- 3. Click Uninstall. A confirmation dialog box opens. Upon confirmation, the PTC Creo application is removed.



Note

If you have installed more than one PTC Creo application, make sure that the Common Files folder and its contents are still available in the PTC Creo load point.

7

Installing and Accessing PTC Creo Help

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This chapter provides an overview of the Help Center. It explains how to install and access Help for PTC Creo Parametric, PTC Creo Direct, PTC Creo Layout, PTC Creo Simulate, and PTC Creo Options Modeler from a local host computer, a network drive, or a Web server.

Overview of the Help Center

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

- Context-sensitive Help for PTC Creo applications PTC Creo Parametric, PTC Creo Direct, PTC Creo Layout, PTC Creo Simulate, and PTC Creo Options Modeler.
- A search facility and quick links to helpful information.

Browser Support for PTC Creo Help

PTC Creo Help supports following browsers:

- Internet Explorer 9.0 and later
- Mozilla Firefox 10.0.1 and later

The Help Center for the PTC Creo application opens in your default browser. You can set one of these browsers as the default on your computer.

Accessing Help from within a PTC Creo **Application**

Each PTC Creo application launches its Help in a separate window of your default browser. The value of the PRO LANG environment variable determines the language of the Help Center. You can access Help topics in the following ways:

- For context-sensitive Help, point to a user interface item and press F1.
- To browse the Help Center, click

The Help Center is accessed from the following locations:

- help.ptc.com—The default if you do not install the Help Center and set the PTC <app name> HC URL 3 environment variable.
- Local host computer—The default if you install the Help Center but do not specify an alternate location, such as a network drive, for accessing Help.



Note

When you upgrade a PTC Creo application, you must also upgrade the Help center to the corresponding datecode of the application. Otherwise, the PTC Creo application accesses the corresponding Help Center from PTC.com.

Choosing a Location for the Help Center Installation

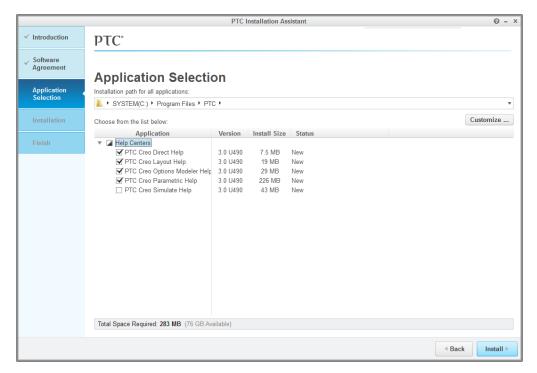
Using the Help DVD, you can install the Help Center for the PTC Creo applications at one of the following locations:

- A local host computer where the PTC Creo applications are installed.
- A network drive. This location is helpful for administering multiple clients on a network.
- · A Web server.

Installing the Help Center

Using the Help DVD in your shipment, proceed with the installation as described. You can also download the Help DVD from the Order or Download Software Updates page at PTC.com. If you have installed the Help Center on a network drive, perform the steps explained in Enabling the Help Center from a Network Drive on page 67 to access the Help Center from the PTC Creo application.

- 1. Start Windows Explorer, browse to the DVD drive, and double-click setup.exe. Alternatively, download the DVD as described in the previous paragraph. The PTC Installation Assistant begins.
- 2. Perform steps 1 through 4 as described in Installing the PTC Creo Applications: Simple Installation Process on page 34.
- 3. Click Next. The Application Selection screen appears.



- 4. Select one or more Help Centers from the list of Help Centers.
- 5. Retain the default folder, for example, C:\Program Files\PTC, or click and specify the full path of an alternative folder.
- 6. Click **Install** to install the Help Center without any customization.

Alternatively, click **Customize** to open the **Application Custom Setting** dialog box and customize the installation as follows:

- a. On the **Application Features** tab, select the Help Center languages to install.
- b. On the **Shortcuts** tab, specify the shortcut preferences for the PTC Creo Help startup commands and other Windows preferences. See Configuring Windows Shortcuts on page 48 for information on how to perform these operations.
- 7. Click **OK**.
- 8. Click Install on the Application Selection screen. The Installation Progress screen shows the status of the installation. You can click Cancel at any time to cancel the installation.
- 9. Click Finish. The Help Center is installed in the <help_install_path>\ PTC\Creo 3.0\help\creo help <app name> directory.

For example, if you specify the default path, the Help Center is installed in the C:\Program Files\PTC\Creo 3.0\help\creo_help_<app_name> directory.

Enabling the Help Center from a Network Drive

To enable the Help Center that is installed on a network drive, you must configure the PTC Creo application as explained in the following example:

- If you have installed the Help Center on computer machine03d at C:\
 Program Files\PTC\Creo 3.0\help, share the folder C:\Program
 Files\PTC\Creo 3.0 as:
 \machine03d\Creo 3.0.
- 2. On a client computer where you want to access the Help Center, map the shared folder to a local drive, say Q. After mapping, the folder with the Help installation files appears as Q:\Help\help\creo help <app name>.
- 3. Set the PTC_<app_name>_HC_URL_3 environment variable for the PTC Creo applications as listed:
 - PTC Creo Parametric—PTC PMA HC URL 3
 - PTC Creo Direct—PTC DMA HC URL 3
 - PTC Creo Layout—PTC LCD HC URL 3
 - PTC Creo Simulate—PTC SIM HC URL 3
 - PTC Creo Options Modeler—PTC OPTM HC URL 3
 - a. Click Start ➤ Control Panel ➤ System ➤ Advanced system settings. The System Properties dialog box opens.
 - b. Click the Advanced tab and then click Environment Variables.
 - c. Click **New** under the User variables area. The **New User Variable** dialog box opens.
 - d. Specify the environment variable and its value in the boxes listed below. For example, for PTC Creo Parametric Help, set the environment variable as follows:
 - Variable name—PTC_PMA_HC_URL_3
 - Variable value—Q:\Help\help\creo_help_pma
 - e. Click **OK** and again click **OK** on the **System Properties** dialog box.
- 4. Start the PTC Creo application and access Help.

Installing the Help Center on a Web Server

You can install the Help Center on any Web server. If you are using Apache, install the Help Center in the DocumentRoot directory as follows:

- 1. Perform steps 1 through 4 as described in Installing the Help Center on page 65.
- 2. Specify < DocumentRoot > as the installation path.
- 3. Click **OK** and click **Install** on the **Application Selection** screen.

The selected application help is installed in the <DocumentRoot>\Creo 3.0\help\creo help <app name> directory.

- 4. To distribute the Help Center files from the <DocumentRoot>\Creo 3.0\ help\creo_help_<app_name> directory for context sensitive Help access, set the PTC_<app_name>_HC_URL_3 environment variable for the PTC Creo applications as listed:
 - PTC Creo Parametric—PTC PMA HC URL 3
 - PTC Creo Direct—PTC DMA HC URL 3
 - PTC Creo Layout—PTC_LCD_HC_URL_3
 - PTC Creo Simulate—PTC SIM HC URL 3
 - PTC Creo Options Modeler—PTC OPTM HC URL 3
 - a. Click Start ➤ Control Panel ➤ System ➤ Advanced system settings. The System Properties dialog box opens.
 - b. Click the Advanced tab and then click Environment Variables.
 - c. Click **New** under the User variables area. The **New User Variable** dialog box opens.
 - d. Specify the environment variable and its value in the boxes listed below. For example, for PTC Creo Parametric Help, set the environment variable as follows:
 - Variable name—PTC_<app_name>_HC_URL_3
 - Variable value—<Help Center URL>

For example, for a Help Center installed on Apache server configured on port number 8080, the Help Center URL is http://<server_name>:8080/Creo 3.0/help/creo_help_<app_name>.

e. Click **OK** and again click **OK** on the **System Properties** dialog box.

Alternatively, you can copy the Help Center of an application that is installed on the local host computer in the <DocumentRoot> directory of the servlet container.

For example, if you have installed PTC Creo Parametric Help Center on the local host computer at the default path, you can copy the Help Center directory C:\Program Files\PTC\Creo 3.0\help\creo_help_pma to the <DocumentRoot> directory. Specify the appropriate URL as the value for the PTC_PMA_HC_URL_3 variable to access the context-sensitive Help.

Note

If you specify <DocumentRoot> directory as the location for the installation of the Help Center, you can access the context-sensitive help directly from this directory. Set the Help Center URL as the value of the PTC <app name> HC URL 3 environment variable to access the contextsensitive Help through the Web server URL.

Viewing the Help Center Independent of a **PTC Creo Application**

To view the Help Center that is installed on a host computer independent of the corresponding PTC Creo application, open the following page in a browser window:

```
<creo help install path>\help\creo help <app>\
<language>\index.html
```

For example, to open the Help Center for PTC Creo Parametric in English, open the following page in a browser window:

```
<creo help install path>\help\creo help pma\usascii\
index.html
```

The Help Center opens in the browser window.

To view the Help Center that is installed on a Web server independent of the PTC Creo application, specify the following URL in a browser window:

```
http://<server name>:<port number>/Creo 3.0/help/creo
help <app name>/<language>/index.html
```

For example, to open the Help Center installed on a Web server that has the Tomcat servlet container configured on port number 8080 for PTC Creo Parametric in English, open the following page in a browser window:

```
http://<server name>:8080/Creo 3.0/help/creo help pma/
usascii/index.html
```

A Help Center opens in the browser window.



Note

Context-sensitive Help is not supported when you view the Help Center independent of a PTC Creo application.

Uninstalling the Help Center

You can remove the Help Center from the local host computer independent of a PTC Creo application using the following steps:

- 1. Click Start ➤ Control Panel ➤ Programs and Features. The Uninstall or change a program screen appears.
- 2. In the application list, click PTC Creo Help <release><date code> for any of the PTC Creo applications.
- 3. Click **Uninstall**. A confirmation dialog box opens.
- 4. Click **Yes** to uninstall the Help Center for the application.

Note

If you have uninstalled the Help Center on the host computer or have not set the PTC <app name> HC URL 3 environment variable, you can access the Help Center from PTC.com from within a PTC Creo application. Additionally, if you uninstall or remove the Help Center from the Web server, you must remove or unset the PTC <app name> HC URL 3 environment variable.

8

Updating an Installation

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This chapter explains how to update the PTC Creo applications installed on your system. Use the same procedure to update PTC Creo Distributed Services Manager. 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 PTC Creo within the same release. In some cases you may have to update the current installation of PTC License Server. See the next section for details.

Updating the License Server

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

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

You may have to update the license server software itself depending on the version installed. In such cases, with administrative privileges uninstall PTC License Server and follow the instructions in the chapter Configuring Licenses for the PTC Creo Applications on page 23 to install PTC License Server. Update PTC License Server using this workflow:

- 1. Make note of the product code that you have received via e-mail. Alternatively, use the PTC License Management Web tools to request for a license file via email. Save the license file in an ASCII format to a secure location on your disk.
- 2. Start PTC Installation Assistant from the mounted DVD or from the installation package downloaded from the Web. The Welcome screen appears.
- 3. Click **Setup license** and then click **Next**. The **License Identification** screen appears.
- 4. Type your new product code or Sales Order Number in the respective text box and click **Install Licensing**. Alternatively, drag and drop your new license file into the **Source** column of the **License Summary** area.
 - If a license server is already installed or running, the Assistant replaces the existing license file with the new one. If a license server is not running, the Assistant checks whether the new license file requires a license server. If so, the license server is automatically downloaded and installed. See Generating or Updating Licenses on page 26 for details.
- Click Finish.

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

Updating PTC Creo Applications

After you install PTC Creo applications of a particular release, you can perform the following operations:

• Customize an existing installation—After the update, all the previously installed files including the ones in the <creo loadpoint>\

- <datecode>\Common Files folder remain unchanged. The updated files
 relevant to the customization are added to the <datecode> folder.
- Install a new maintenance release—After the installation, a new <datecode> folder corresponding to the datecode of the maintenance release is created within the <creo_loadpoint> folder. This folder contains all the files for the maintenance release.
- Upgrade an existing installation—After the upgrade, the highest version of the release that is installed is removed and the new maintenance release is installed in the installation path that you specify. If there is no installed version at the specified installation path, then the new maintenance release is installed at that path.

Customizing an Existing Installation of a PTC Creo Application

Customizing an existing installation of a PTC Creo application implies enabling or disabling Quality Agent reporting for the application, updating or creating startup commands, and updating Windows shortcut preferences. You cannot add or delete application features during this process.

Rerun PTC Installation Assistant from the DVD and perform the following steps:

- 1. Start the Assistant. The Welcome screen appears.
- 2. Click Reconfigure existing software and click Next. The License Identification screen appears.
- 3. Check the license information and update it if required. See Generating or Updating Licenses on page 26 for details.
- 4. Click **Next**. The **Application Selection** screen appears with a list of applications that you previously installed.
- 5. Click Customize after selecting the applications. The Application Custom Setting dialog box opens.
- 6. Select an application on the left pane and perform one of the following actions:
 - Select or clear the **Enable Quality Agent reporting** check box using the **Application Features** tabbed page. See Defining the Installation Components on page 43 for details.
 - Add, edit, or delete a license configuration using the Command Configuration tabbed page. See Modifying License Configurations on page 46 for details.

- Configure the Windows shortcut preferences and path settings using the Shortcut tabbed page. See Configuring Windows Shortcuts on page 48 for details.
- Click **OK** on the **Application Custom Setting** screen and then click **Install** on the **Application Selection** screen to customize the installation.
- 7. Click Finish.

Installing a Maintenance Release

If you have installed an earlier release of a PTC Creo application, use the steps below to install a newer maintenance release. Refer to the chapters about the installation of various components for detailed information.

- 1. Start PTC Installation Assistant. Select **Install new software** on the Welcome screen. Click **Next**. The **License Identification** screen appears.
- 2. Generate or update your license file. If required, PTC License Server is also updated based on your license file. Click **Next**. The **Application Selection** screen appears.
- 3. Select an application to install.
- 4. Specify an installation folder.
- 5. To customize the installation for specific application features, license configurations, or Windows shortcut preferences, click **Customize**. Use the **Application Custom Setting** screen that opens for the customization.
- 6. After you select all the options, click **Install** on the **Application Selection** screen. The common files folder created for the previous release gets replaced with common files folder for the release you are installing.
- 7. Click **Finish** to complete the installation and close the Assistant.

Upgrading an Existing Version of an Installation

1. Start PTC Installation Assistant. Select **Upgrade existing software** on the Welcome screen. Click **Next**. The **License Identification** screen appears.

Note

If there is no installed version of the release, the **Upgrade existing software** option is not available on the Welcome screen

- 2. Generate or update your license file. If required, the PTC License Server is also updated based on your license file. Click Next. The Application Selection screen appears.
- 3. The path of the last installed version of the release appears on the **Application Selection** screen as the default installation path. If you retain this default installation path, the installed applications appear selected, and you cannot deselect them.

Note

If an earlier version of the release exists at the specified installation path, Upgrade from <version> appears in the Status column. However, if an installed version does not exists at the specified installation path, the status appears as New.

Alternatively, you can specify a different installation path. If an installation of an application does not exist in the specified alternative path, this installation is treated as a new installation. You can select any or all such applications on the Application Selection screen.

Note

If a version of the release of an application exists at the specified alternative installation path, Upgrade from <version> appears in the **Status** column for that application.

Additionally, if the version of a release of an application that you are installing already exists at the specified installation path, the same version of that application is upgraded with the customizations of the application.

4. To customize the installation for specific application features, license configurations, or Windows shortcut preferences, click **Customize**. Use the **Application Custom Setting** screen that opens for the customization.

- 5. After you select all the options, click **Install** on the **Application Selection** screen.
- 6. Click **Finish** to complete the installation and close the Assistant.



License Management Software

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

Overview of PTC Licensing

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

- Commercial—Licenses for using full versions of PTC software. These licenses are not time-sensitive.
- Educational—Also known as student licenses used at educational institutions.
 These licenses are time-sensitive. The objects created in an educational or
 student version of the software are not compatible with the commercial
 version of the same software.
- Evaluation—Licenses that limit your use of the software to a specified time period.

License Types

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

- Node-locked licenses—Restricts the use of the software to a particular machine (independent workstation).
- Floating licenses—Served by the license server and can be used on any
 machine connected over the network. There are two different types of floating
 licenses:
 - Single server licenses—Configured for a single machine as the server.
 - Triad licenses—Configured for a set of three machines on the same network that work together to manage a pool of floating licenses. This configuration provides redundant backup in case of a server outage. An identical license file is used for each Triad partner.
- Extended license—An extended license makes floating licenses available for locked modules.
- Borrowed license—A borrowed license allows you to work temporarily on your machine without being connected to the license server. Refer to License Borrowing on page 88for details.

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

Simplified License Management Process

PTC Creo uses license simplification for 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 PTC Creo 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. This format has been in use by PTC since Pro/ENGINEER 2001 datecode 2001440. Pro/ENGINEER Wildfire 2.0 or later automatically uses simplified licenses as license simplification is required for Pro/ENGINEER Wildfire 2.0 and later releases

Note

License configuration is a part of the installation process.

Visit www.ptc.com/olm/ls faq for more information on license simplification including the advantages of simplifying your licenses.

License Simplification—Advantages

License simplification has the following 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 the PTC Creo applications as well as products of the PTC Creo Parametric family and is available in Pro/ ENGINEER 2001 datecode 2001440 and later.
- On the PTC Creo eSupport Center page, use the License Support Web tool called Reconfigure Software Licenses under License Management 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.



Note

FlextNet Publisher Version 11.11.1 is shipped with PTC Creo 3.0. However, PTC Creo 3.0 can use FlexNet Publisher version 10.8.x or later for license simplification.

- The license simplification process has changed the format of the FlexNet Publisher feature line in the license file in the following ways:
 - The SCN is no longer a part of the feature name and is moved below the SN tag in the feature line.
 - o 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 within a PTC Creo application.
- 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 Engineer ptc d 33.0 01-nov-2011 1 \
BC24AFC5B76BB74C9366 VENDOR STRING="VSVER=2.0 \
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 33.0 01-nov-2011 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 33.0 01-nov-2011 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 PTC Installation Assistant, you can define startup commands with additional license extensions or floating options. Defining a startup command means specifying the license-to-run for an application at startup. You can also specify additional floating options to grab at startup. This enables additional optional modules for a session that are not otherwise enabled by the license-torun. The designation of the licenses to grab at startup results in the definition of an environment variable <APP> FEATURE NAME, where <APP> is the abbreviation for PTC Creo application name that you are configuring. For example, CREOPMA FEATURE NAME identifies the licenses to grab when running PTC Creo Parametric. Similarly, CREODMA FEATURE NAME refers to PTC Creo Direct. The value of this environment variable can be a list of licenses-to-run, followed by a parenthesized list of optional modules. Only one license-to-run is checked out while all the options from the parenthesized list are checked out. For the license-to-run, checkout attempts are made in the order listed, until one is successfully checked out. Then the option list is attempted. Consider the following example:

CREOPMA_FEATURE_NAME=PROE_FLEX3C PROE_FoundationAdv (61 108)

This example shows an environment variable definition identifying two licenses-to-run and two options. This environment variable prescribes that PTC Creo Parametric will first try to checkout PROE_FLEX3C license-to-run; if it fails, it will try to checkout PROE_FoundationAdv. If either one is successful, it will then attempt to check both options – 61 (Web.Link) and 108 (Piping).

• Using PTC Installation Assistant, 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 <app> FEATURE NAME variable. This value can be one of CREOPMA

FEATURE_NAME, CREODMA_FEATURE_NAME, or CREOSIM_FEATURE_NAME depending on the PTC Creo application that you have installed. This variable is defined in the application startup command.

- During a session, if a license is lost and you try to recover the license-to-run, the PTC Creo application 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 ptc d FlexNet Publisher version: 11.11.1

License	In Use	Free
PROE_Engineer	Uncounted, Locked	2
Foundation_Adv	0	2
Pro/SURFACE *	0	2
Behavioral_Modeler	0	2
Pro_Plastic_Advisor	0	2
Tool_Design_Set*	0	2

 $[\]star$ = License Extensions - Available only on startup

Ptcstatus Verbose Mode

```
Displaying status for license file: 7788@static License Server: static ptc_d FlexNet Publisher version: 11.11.1
```

^{^ =} Borrowed License

License	In Use	Free	Ver sion	Expiration Date	SCN
PROE_Engineer	Uncounted	2	33.0		Locked to 00-11-22-33-44-5512345
Foundation_Adv	0	2	33.0		
Pro/SURFACE *	0	2	33.0		
Behavioral_ Modeler	0	2	33.0		
Pro_Plastic_ Advisor	0	2	33.0		
Tool_Design_ Set*	0	2	33.0		

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_Engineer 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_Engineer ptc_d 33.0 01-nov-2011 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_Engineer ptc_d 33.0 01-nov-2011 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 Engineer.

PTC License Server

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

Note

FlextNet Publisher Version 11.11.1 is shipped with PTC Creo 3.0. However, PTC Creo 3.0 can use FlexNet Publisher version 10.8.x or later for license simplification.

Benefits of Using FlexNet Publisher

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

- Single-Server Solution—FlexNet Publisher can manage PTC software and other vendor applications without conflict. Note that the PTC license file cannot be combined with the license files of other vendors.
- Immediate License Recovery—If there is a premature exit of the licensed software (for example, the system shuts down), the FlexNet Publisher license server automatically reclaims the PTC license.
- Increased Flexibility—System administrators can reserve or deny licenses based on user name, host name, display name, or IP address. For more information see Managing the Options File in the FlexNet Publisher License Administration Guide.
- Centralized License Storage—PTC customers can store all PTC licenses for all PTC products in a single file for each license server.
- Multiple Licenses for a Single Command—One command can be used to execute multiple licenses of PTC software based on availability.
- License Borrowing—PTC Creo 3.0 allows license borrowing with FlexNet

Downward License Compatibility Using FlexNet Publisher

FlexNet Publisher license servers have downward compatibility with PTC applications using FLEX*lm* license servers. For example, a PTC Creo Parametric 3.0 license can run Pro/ENGINEER 2001 software, because both releases of the software use the same PTC License Server.

Note

To run the current release of PTC Creo, you must install FlexNet Publisher version 10.8.x or later. FlextNet Publisher Version 11.11.1 is shipped with PTC Creo 3.0.

Running FlexNet Publisher with Other **Applications**

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

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

If all applications and vendor daemons are FLEX*lm* 10.8.x or later, lmgrd or lmadmin can process multiple license files. This is true even when the Host IDs are different (as long as the license files refer to the same node). For more information, see Managing Licenses from Multiple Software Publishers in the FlexNet Publisher License Administration Guide.

Understanding Timeout Parameters

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

You can reconfigure the TIMEOUTALL parameter within a specified range, as described in the next section, Changing the Inactivity Timeout Parameter on page 87.

Timeout		
Parameter	Value	Description
Inactivity timeout (TIMEOUTALL)	120 minutes (default)	This parameter prevents a license from remaining idle. If the license client is inactive for a specific period, the license can be reclaimed by the license server and used by another license client. For the PTC Creo applications, the inactivity timeout default is 120 minutes.
		Activity is measured as active menu selections.
		You can change the default of 120 minutes so that the parameter ranges from 20 minutes (minimum value) to infinity (maximum value).
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 <ptclicense 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 for PTC Creo Parametric

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 PTC Creo Parametric, 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. For Ptcflush to work with the Imadmin-based license server, set licenseReclaimAllowed="true" in the server.xml file at license server manager loadpoint/conf.

The syntax for running Ptcflush 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_Engineer titan 7788 101 using the feature name and not the user-defined name if you assigned an alias.

License Borrowing

With license borrowing for all supported platforms, you can run PTC Creo applications on clients without being connected to the license server. The license server must be running FlexNet Publisher 10.8.x or later and must use the License Pack of version 33 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

The appearance of the BORROW keyword on the feature line in the license file indicates that the license is borrowable. If this keyword is missing, contact PTC License Management to regenerate your license file for a borrowable license. An example of a license file with the BORROW keyword follows:

```
INCREMENT PROE_Engineer ptc_d 33.0 01-nov-2011 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 using one of the following methods:

- Running a batch file available in the \<creo_loadpoint>\
 <datecode>\<app name>\bin directory
- Specifying the command-line argument to borrow licenses as instructed in the *FlexNet Publisher License Administration Guide*.

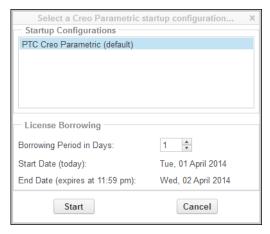


If you may possibly want an early return of a borrowed license, set the PRO_ALLOW_EARLY_RETURN environment variable prior to borrowing the license from the license server. See Early Return of a Borrowed License on page 90 for more information.

Using the <creo app> borrow.bat File

Use the <cree app> borrow.bat file with the PTC Cree application to initiate license borrowing as follows:

1. Browse to the \<creo loadpoint>\<datecode>\<app name>\bin directory and run the <creo app> borrow.bat file. A dialog box opens in which you can specify a startup configuration.



- 2. Under **Startup Configurations**, select the desired 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. Using the environment variable LM BORROW DURATION=0-14, you can borrow a license for a maximum of 14 days.

- 4. Click **Start** to start the PTC Creo application in the borrow configuration mode.
- 5. After starting the PTC Creo application, you can include additional floating options with the borrowed license.

All the licenses that you check out during this session are borrowed from the license server and cached on your computer. Each license is created on the local machine in the registry under current user.

- 6. Exit the PTC Creo application.
- 7. Disconnect from the license server and use the borrowed license. You must start the application subsequently on the same computer and as the same user to use this borrowed license.

Note

Multiple licenses with separate expiration dates can be borrowed to a client computer. Each time you start a PTC Creo application using a borrowed license, the application checks for and removes any expired borrowed licenses from the client computer. 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

Early Return of a Borrowed License

All arrangements to return the borrowed licenses early must be made before initiating license borrowing. Before borrowing a license from the server, you must set the environment variable PRO ALLOW EARLY RETURN to true to return that license early. If you frequently return licenses early, then permanently set the environment variable using the **System Properties** dialog box.



Note

Setting the PRO ALLOW EARLY RETURN environment variable has no effect on an unborrowed license.

For an Imadmin-based license server, set adminOnly="false" in the server.xml file at license server manager loadpoint/conf to return a borrowed license early. See Returning a Borrowed License Early in the FlexNet Publisher License Administration Guide for a procedure on returning the borrowed licenses early.

Using the Imutil Imborrow Utility

For an early return of a borrowed license, you can use the <code>lmborrow</code> subfunction with the <code>lmutil</code> Flexnet utility. The <code>lmutil</code> utility resides at <code>C:\Program</code> <code>Files\PTC\Creo 3.0\<datecode>\Common Files\x86e_win64\obj.</code>

Before running the lmutil lmborrow utility, you can run the following command in a command shell to check the status of the borrowed license:

lmutil lmborrow -status

Then read the command output to determine the optional arguments for the lmutil lmborrow utility. See the next table. To use the lmutil lmborrow utility, open a command shell and type the following command at the prompt:

lmutil lmborrow -return [-fqdn] [-c license_file_list] [-d
display] feature

Optional arguments are enclosed within brackets in the previous command syntax. Descriptions of the arguments follow.

Argument	Description
-fqdn	Specifies that a fully qualified domain name was used for license borrowing, rather than a simple host name. Use the lmutil lmstat command to determine the format of the host name used when the license was borrowed.
-c license_file_list	Specifies the license server from which the license was borrowed. In most cases specifying the port@hostname value for the server is sufficient.
	You can avoid using this argument by setting the LM_LICENSE_FILE variable. Set the variable as LM_LICENSE_FILE= port@hostname to point to the server used for borrowing the license.
-d display	Displays the system name. Required only if the displayed system name is different than the one used to initiate borrowing.
feature	Names the borrowed license feature to be returned early. Use the lmborrow-status command for a list of borrowed license features.

B

Imadmin License Server Manager

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

Overview of Imadmin as a GUI-Client

The lmadmin license server manager supports a Graphical User Interface (GUI) client with connection over HTTP. It provides a Web-based administrative interface. It is compatible with license files and vendor daemons created with FlexNet Publisher 10.8.x and later. Use lmadmin for the following functions:

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

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

Differences between Imgrd and Imadmin

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

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

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

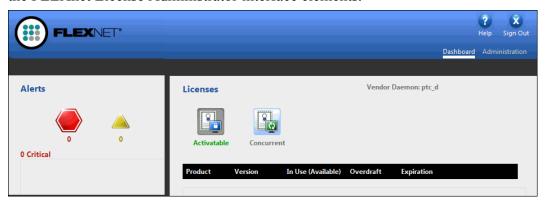
Installing PTC License Server Based on Imadmin

Follow the instructions given in the section Generating or Updating Licenses on page 26 to install PTC License Server based on Imadmin. After you install the license server, you can configure the license server administration settings using the FLEXnet License Administrator Web interface.

Working with the FLEXnet License Administrator Web Interface

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

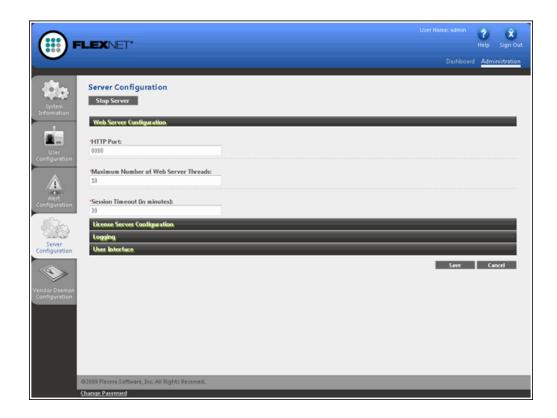
The FLEXnet License Administrator Web interface has two main pages: the **Dashboard** and the **Administration** pages. Click the **Help** button for information on the FLEXnet License Administrator interface elements.



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

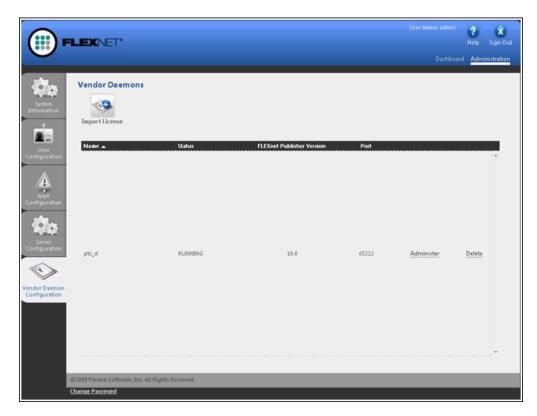
Controlling the License Server Manager Settings

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



Reconfiguring the Vendor Daemon

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



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

C

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

Temporary Directories

PTC Creo applications use a temporary directory as defined by your system environment to store parts. You can modify the location of the temporary directory by setting the temporary directory environment variable in the PTC Creo application startup window before you run the application. This environment variable is TMP.

System Management Utilities

This section briefly describes the utilities for PTC Creo Parametric, PTC Creo Simulate, PTC Creo Direct, PTC Creo Layout, and FlexNet Publisher. The files listed in the next two tables are installed into the bin directory of your installation directory. Further information can be found in the *FlexNet Publisher License Administration Guide*.

PTC Creo Utilities

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

Command	Function
ptchostid	Returns the PTC Host ID with required formatting used for
	FlexNet Publisher license codes.
parametric	Runs PTC Creo Parametric. Can be user-defined.
direct	Runs PTC Creo Direct. Can be user-defined.
simulate	Runs PTC Creo Simulate. Can be user-defined.
layout	Runs PTC Creo Layout. Can be user-defined.
optionsmodeler	Runs PTC Creo Options Modeler. Can be user-defined.
ptcflush	Releases licenses. This may be necessary if there are network problems. Formerly called proflush.
ptcstatus	Displays information on the current usage and availability of licenses.
ptcborrow	Initiate license borrowing.
and	
<creo_app_< td=""><td></td></creo_app_<>	
borrow>	

FlexNet Publisher Utilities

If you have installed PTC License Server based on the <code>lmgrd</code> license server manager, you can execute the commands listed below from the <code>\<flexnet_loadpoint>\bin</code> directory when the license server is running.

Command	Function
lmtools	Runs the PTC License Management tools utility
	lmtools.bat. Functionality includes customer
	configuration of the FlexNet Publisher service, starting
	and stopping the lmgrd license server(s), server and
	licensing status, as well as diagnostic tools. The shortcut
	icon created in Start ► All Programs ► PTC License Management tools starts the Lmtools graphical user
	interface.
lmutil	Provides optional arguments to check the server and
	licensing status. Refer to the FlexNet Publisher License
	Administration Guide.
ptcshutdown	Shuts down a license server process for administrative
	maintenance of the license server. You can also use the
	FLEXnet Control Panel to stop a license server process.
ptcstartserver	Starts a license server process. You can also use the
	FLEXnet Control Panel to start a license server process.
	See Using License Administration Tools, in the <i>FlexNet</i>
	Publisher License Administration Guide.

Lmtools Utility

The lmtools.bat program runs the PTC License Management tools utility that is started from the Start ➤ All Programs ➤ PTC License Server tools menu. Functionality includes starting and stopping the license server from the Start/Stop/ReRead tab.

Supported Graphics Modes

For graphics support information, refer to the PTC Hardware Configuration Notes at http://www.ptc.com/WCMS/files/160242/en/PTC_Creo_Future_Platform_Support_Summary.pdf.

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



Note

PTC Creo applications described in this guide do not support any HPIB interfacing.

Using the ModelCHECK Metrics Tool with PTC Creo Parametric

The ModelCHECK Metrics Tool is a web-based utility for tracking data quality. standards, and best practices relevant to PTC Creo Parametric data. 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 PTC Creo Parametric license and an executable while installing the tool.

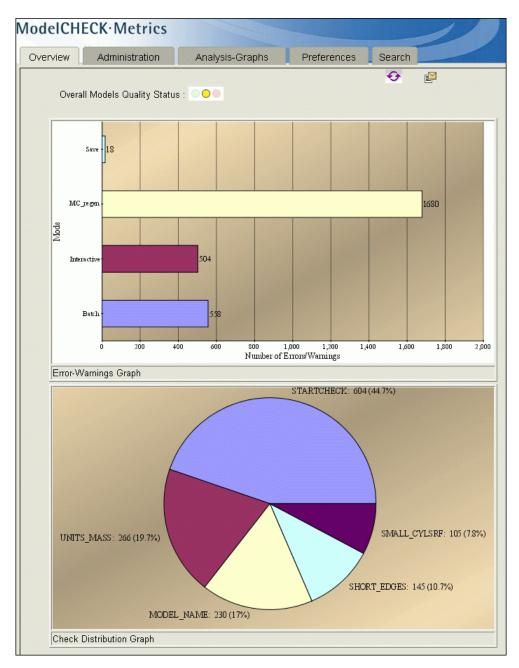


Note

You cannot install the Metrics Tool using PTC Installation Assistant.

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 PTC Creo Parametric Help Center for details.

Requirements for Installation

You can install the ModelCHECK Metrics Tool if you use a supported configuration for the workstation or for the server. See http://www.ptc.com/WCMS/files/160242/en/PTC_Creo_Future_Platform_Support_Summary.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 and later.

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.oracle.com/technetwork/java/index.html.
- Install Tomcat for Java Web Services Developer Pack through a local installation or from www.oracle.com/technetwork/java/index.html.
- 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 tomcat50jwsdp\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 on page 106 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:

<creo_loadpoint>/<datecode>/Common Files/modchk/mc_ admin/new_admin/dist/

Parameter Name	Description
dbname	Name of the database
dbtype	Type of the database, such as Oracle or MSAccess
adminserver	Machine name of a server for the Metrics Tool
Adminport	Port number type of the server for the Metrics Tool
Imagedir	Path to the image directory
ImagedirLink	Link to the image directory for storing graphs
collect_mc_db_ interval	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"</pre>
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
</parameter>
</ResourceParams>
<ResourceLink name="jdbc/mcadmin" global="mcadmin"/>
<Manager
className="orq.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"/>
```

In above file, the value of Imagedir is d:\projects\mcadminwork\admingraphs. Therefore, 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 <code>create_mcadmin_db.jar</code> available at <code>creo_loadpoint>/cdatecode>/Common Files/modchk/mc_admin/new_admin/dist/</code> enables you to populate the database tables with information from the <code>.txt</code> 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 <code>.txt</code> 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_mcadmin_db.jar to Update the Database Tables

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

```
set CLASSPATH=<path to file 1>;<path to file 2>
```

For example:

```
set CLASSPATH=d:\wildfire\modchk\mc_admin\new_admin\
dist\
create_mcadmin_db.jar;d:\wildfire\modchk\mc_admin\
new admin\dist\classes111.jar
```

Note

The classes 111.jar file is required to manually execute the create_mcadmin 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	INF01	LOCATION
bdevid	Devid	Boon	K	CADQA	23453	3380	04-SEP-05	QA	d:\projects\devid
dalan	Alan	Donald	Υ	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>

- 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 on page 105 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
oracleoracle.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.McMetricsCollectormsaccess
sun.jdbc.odbc.JdbcOdbcDriverjdbc:odbc:adminuser1
userpass
```

A sample mc_database table is shown below:

USERID	DATE_N_TI	MDLNAME	MDLTY	CHECKTAG	CHKSTAT	INF01	MCMODE
devid	18-JUL-04	RIM	PRT	SUP_FEATURES	WRN	2	Interactive
devid	18-JUL-04	RIM	PRT	UNITS_LENGTH	0K	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	(O	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
- infol—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 PTC Creo 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—UserID
- lastname—Last name
- firstname—First name
- middlename—Middle name
- group b—Group to which a user belongs
- phone—Telephone number
- extension—Extension number
- parametricdate DATE—Date when the user ran ModelCHECK for which the metrics files were created
- infol—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),
parametricdate DATE,
infol 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','YYYYY-MM-DD HH24:MI:SS'), 'explosive',
'/net/machine1/usr3/asewell/mc metrics');
```

For more details, refer to the ModelCHECK Online Help in the PTC Creo Help Center.

Using Fonts

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 a PTC Creo application in Chinese Traditional or Chinese Simplified, 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>\<datecode>\
Common Files\text\fonts folder. You can change this default location by
placing the fonts in another folder and setting the value of the pro_font_dir
configuration option to this folder. PTC Creo applications use the fonts in the
folder 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

Font Name	File Name	Supports Kerning
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
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

Font Name	File Name	Supports Kerning
Shannon Extra Bold	shannoeb.ttf	Yes
Shannon Oblique	shannoo.ttf	Yes
Spartan Four MT	sparton4.ttf	Yes
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 PTC Creo applications, you use two categories of fonts:

- Fonts specific to PTC Creo applications—These fonts appear in the graphics windows of PTC Creo applications. 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 the application window headers. They are machine- and language-specific.

OpenType Font Support

PTC Creo applications provide additional support for OpenType fonts. These fonts are based on Unicode for multilanguage support. OpenType fonts offer an expanded character set and layout features. In PTC Creo Parametric, 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 the PTC Creo applications.

UNICODE Font Support

You can insert or read out-of-locale characters and symbols 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 an application 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>\<datecode>\
Common Files\text within the folder 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 the PTC Creo applications. The fonts are listed in the **Control Panel** under **Fonts**. The font for text in PTC Creo application window headers is determined by the setting of your window manager.

Extended ASCII Set

ASCII is the default font in all PTC Creo applications. 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 PTC Creo 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, use 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 in PTC Creo Parametric

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. PTC Creo Parametric 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

PTC Creo 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 a PTC Creo application. In case of PTC Creo Parametric, 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.

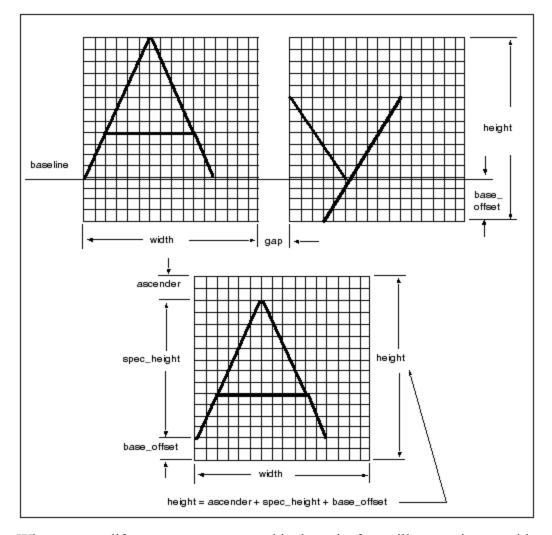
- spec_height<n>—Defines the height of a character without the ascender above the baseline.
- ics (intercharacter spacing)—Turns on and off the effect of individual character width on horizontal spacing.
- If you set ics to 1, each character definition must contain an individual width setting to determine 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.

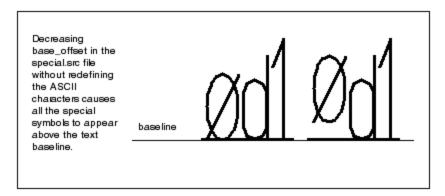
- polygons—Turns on character filling.
- If you set it to 1, all closed polygons in the character definitions are filled (refer to the section Filled Font Definition on page 126).
- If you do not specify this parameter, the system uses the default value 0.
- fixed width<n>—Can be 0 or 1.
- If you set 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.

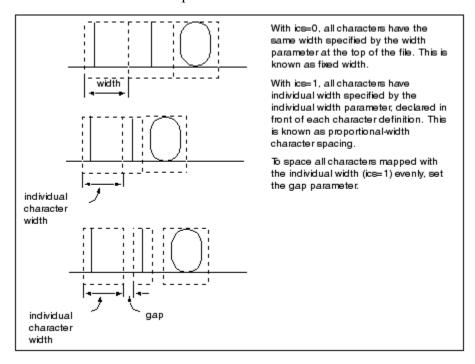


When you modify text parameters, consider how the font will appear in text with other fonts. Fonts used in the same text line (such as ASCII and LATIN_1 for European languages, or ASCII and special) must be compatible. Altering one font might create inconsistency in appearance and placement.

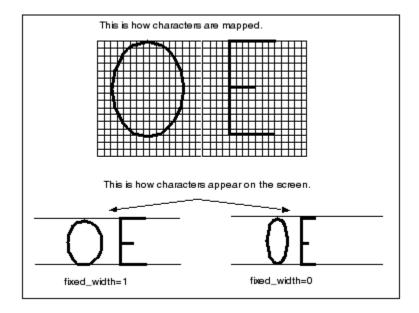
For example, changing the base_offset value at the top of the font file affects the positioning of all the font characters, relative to the text baseline, as shown in the following figure.



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:

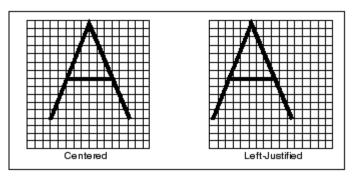
- First line—An ASCII character, in double quotes, that represents the character to be drawn. For ASCII and filled primary fonts, it is the character itself. Special characters also use ASCII characters to represent them. The LATIN_1 fonts use the decimal and hexadecimal codes representing the character.
- 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):

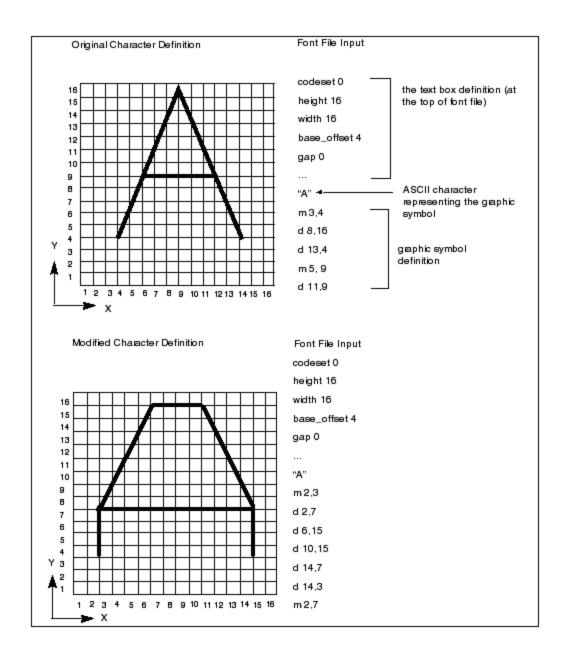
- [m<X>, <Y>]—This command moves an imaginary pen to the grid point (X, Y).
- [d<X>, <Y>]—This command draws a line from the current position to grid point (X, Y).
- For each font, you can use 63 or fewer grid points to create characters and symbols. PTC Creo provides optimum values, giving adequate 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.

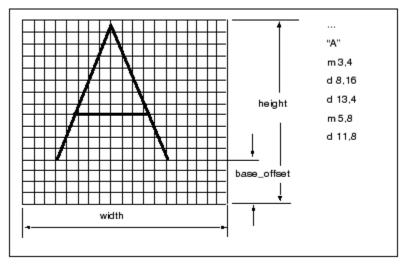


ASCII Font Definition

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

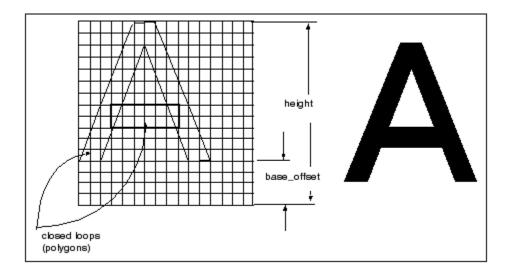
• polygons: 1

• 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, the PTC Creo application fills with color any closed polygon drawn in a character definition.

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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

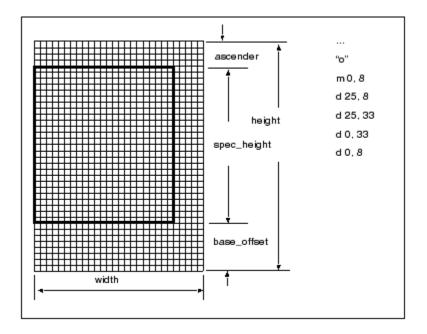
• ascender: 6

• 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

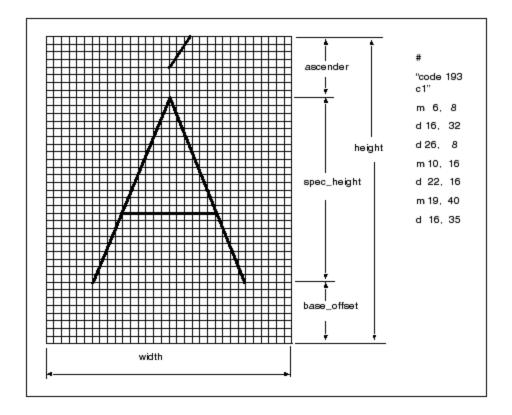
• ics: 0

• 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 Fonts or Create New Fonts

- Edit an existing or create a new .src file, which resides in <creo_ loadpoint>\<datecode>\Common Files\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 Font File

- Copy the appropriate .src font file into your working directory. The .src files reside at <creo_loadpoint>\<datecode>\Common Files\ text\<language>.
- 2. Modify the font file.
- 4. Copy the compile_font command to your working area.
- 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 PTC Creo application.

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.
- 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 PTC Creo.

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>\ <datecode>\Common Files\text\<language>. Store the .fnt file in the following directory: <creo loadpoint>/<datecode>/Common Files/<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
```

Copy the compile_font script from the directory <creo_loadpoint>/
 </datecode>/Common Files/<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>\<datecode>\Common Files\text\ <language> directory.
- Then put the compiled .fnt files in the directory <creo_loadpoint>/ <datecode>/Common Files/<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 a PTC Creo application, 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 for PTC Creo Parametric

You can customize fonts and add symbols and characters to the **Text Symbol** palette in PTC Creo Parametric 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 on page 114.
- 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>/<datecode>/
 Common Files/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 on page 133, 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_parametric\my_custom_palette.txt directory, set the configuration option as follows: custom_text_symbol_palette_file D:\my_parametric\my_custom_palette.txt
 - This path enables PTC Creo Parametric 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 PTC Creo Parametric, 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 PTC Creo Parametric.
- 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 135.

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
Total Runout	34	134
Symmetrical	105	144

In a PTC Creo Parametric 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, PTC Creo Parametric 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 PTC Creo Parametric.
- Use the ASCII codes for the dtl 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, PTC Creo Parametric 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 PTC Creo Parametric:

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

You can capture an AVI file using the third-party compression codecs installed on your 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 aborts 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 PTC Creo Parametric 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 PTC Creo Parametric parameter data as searchable metadata in the PDF file.
- Modify the font types, color, style, and resolution of the PDF output.
- Insert PTC Creo Parametric drawing text and numerical values as searchable and selectable content in the PDF file.
- Insert bookmarks for the various PTC Creo Parametric layers, sheets, zones, flag notes, and revision tags.
- Control the access and security rights of the PDF file.

For more details, see the Interface Help available in the PTC Creo Parametric Help Center.

Meeting Browser Requirements

This section describes the requirements for the PTC Creo browser for all the PTC Creo applications. For information on Web browser support for the PTC Creo applications, see http://www.ptc.com/WCMS/files/160242/en/PTC_Creo_Future_Platform_Support_Summary.pdf. For information related to the Help Center, see Installing and Using PTC Creo Help on page 63.

Browser Requirements

PTC Creo browser runs in the Creo agent and uses the engine of the Internet Explorer (IE) by default.

The multi-process architecture of IE browser allows PTC Creo to run the embedded browser in the out-of-process mode. That is, the IE browser runs as a separate process that is not within the PTC Creo process.

The Creo agent is the same across all PTC Creo applications. Therefore, the 32-bit embedded browser is the same across different PTC Creo applications.

When PTC Creo applications are connected to the same Creo agent, you can only use a single browser type in a session of a PTC Creo application. You cannot change the browser type in the session. To change the browser type, you must exit the current session of the application and all other PTC Creo applications. You must then make sure that the Creo agent process is not running before you specify another browser type and restart the PTC Creo application.

You can specify the PTC Creo browser to use the embedding engine of the Internet Explorer (IE) browser. Before you start a PTC Creo application, ensure that the value of the windows_browser_type configuration option in your config.pro file is set as follows:

• ie_browser—Specifies that the browser uses the IE engine in a Creo agent process initiated from the PTC Creo process. The windows_browser_type configuration option is set to ie_browser by default.

Starting a PTC Creo Application through an External Browser

To start a PTC Creo application from an external browser, register a helper application in the bin directory. The helper application gets installed automatically when you install the PTC Creo application. This application, <app_name>.exe, is also referred to as the PTC Creo Application Starter. Registration of the helper application involves registering files with .pha extension.

Opening a PDF File within the PTC Creo Browser

You can view a PDF file in the PTC Creo browser using Adobe Acrobat Reader. On Windows, Adobe Acrobat Reader is available as a plug-in.

Printing a PDF File

You can view a PDF file in the PTC Creo browser using Adobe Acrobat Reader. To print the PDF file, click the Adobe Acrobat Reader **Print** icon instead of the PTC Creo browser **Print** icon.

Using PTC Creo View Express to Open PTC Creo Objects

The PTC Creo objects that you can open using PTC Creo View Express follow:

File Extension	File Type
.asm	Assembly file
.dft	Draft file

File Extension	File Type
.dgm	Diagram file
.drw	Drawing file
.frm	Format file
.pic	Picture file
.prt	Part file
.rep	Report file
.mfg	Manufacturing file
.sec	Sketcher Section file

On Windows, double-click a file with a registered PVX extension to open the file in Internet Explorer.

Opening PTC Creo Objects from Windows Explorer

If you have not installed PTC Creo View Express, you can open a PTC Creo object directly from Windows Explorer in a PTC Creo session. PTC Creo objects typically appear with a PTC icon in Windows Explorer. Double-click an object in Windows Explorer to open it in a PTC Creo session.

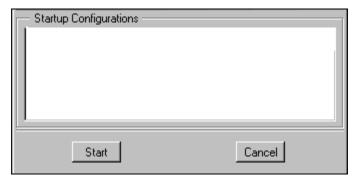
If you have installed PTC Creo View Express, a PTC Creo object is opened in Internet Explorer only for viewing when you double-click the object in Windows Explorer. Right-click on the object and choose the PTC Creo application in which to open the object.

You can open objects with the following file types:

File Extension	File Type
.asm	Assembly file
.dgm	Diagram file
.drw	Drawing file
.frm	Format file
.cem	Layout file
.mfg	Manufacturing file
.mrk	Markup file
.prt	Part file
.sec	Sketcher Section file

If a PTC Creo session is already running and you double-click an object in Windows Explorer, the object is automatically loaded within the same PTC Creo session in a new window. This new window becomes the active window.

If a PTC Creo session is not running and you double-click an object in Windows Explorer, the PTC Creo application startup configuration file dialog box opens. You are prompted to select a file from the list of available configuration files. Upon selecting the required startup configuration file, a PTC Creo session is started using the selected configuration file. The selected object is loaded in this PTC Creo session.



If multiple sessions of PTC Creo applications are running, the selected object is loaded in the session that you started first.

JavaScript Security

In PTC Creo, a JavaScript bridge is used so that the JavaScript code inside a displayed HTML page interacts with the PTC Creo application. For example, **Feature info** reports, folder content listings, and new connection registrations all use the JavaScript bridge for security purposes.

A PTC Creo application automatically activates the JavaScript bridge only from pages generated by the application or Windchill Solutions. If a page generated from another source tries to access the PTC Creo application through the JavaScript bridge, a 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.

D

Installing and Configuring PTC Quality Agent

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This appendix contains information on PTC Quality Agent.

Overview

PTC Quality Agent is an application that gathers and sends reports to PTC on the performance and usage of PTC Creo applications. It also reports the system information of a user. It serves as a medium to help PTC improve the quality of its products.

Installing PTC Quality Agent

PTC Quality Agent is installed by default when you install any of the PTC Creo applications. The PTC Quality Agent reporting is enabled by default for all the PTC Creo applications. Clear the check box at the bottom of the **Application Selection** screen to disable this process. A single installation of PTC Quality Agent can be used with all the PTC Creo applications for which you have enabled PTC Quality Agent reporting.

Note

When customizing a PTC Creo application, if you clear the **Enable Quality** Agent reporting check box on the Application Custom Setting screen, this selection supersedes the selection that you made on the **Application Selection** screen. PTC Quality Agent reporting is not enabled in this case.

You can install PTC Quality Agent on a host computer or a network location depending on where you are installing your PTC Creo application. If you are running a PTC Creo application from a remote host computer or a network location, set the quality agent path configuration option in your config.pro to access PTC Quality Agent. Specify the complete installation path of PTC Quality Agent (for example, <creo loadpoint>\%PRO MACHINE TYPE%\obj\qualityagent.exe) as the value of the configuration option.

Configuring PTC Quality Agent

After installing PTC Quality Agent, you can configure it as per your company's requirements. To open the PTC Quality Agent dialog box, browse to the installation path or use a previously created shortcut.



The tabs on the above dialog box show the configurable settings.

Preferences

If you have write permissions to the PTC Quality Agent installation folder and files, specify the frequency for the collection and transfer of the product and system information to PTC using the **Preferences** tab. Set the frequency to **Monthly**, **Quarterly**, or **Biannually**.



Note

A user without write permissions to the installation folder can view the preferences but cannot change the settings.

Data Filters

If you have write permissions to the PTC Quality Agent installation folder and files, click the **Data Filters** tab. Select specific information to be blocked from being transferred to PTC.



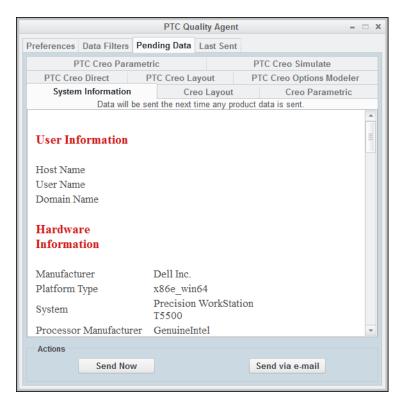
P Note

You cannot filter information that PTC considers mandatory. Those check boxes are unavailable.

The customized preferences and data filters are stored at <All User Profiles>\Application Data\PTC\QualityAgent\Filters.

Pending Data

For each PTC Creo application you have installed, you can view the collected product and system information on the **Pending Data** tab.

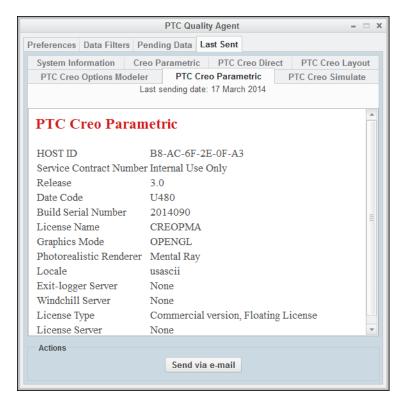


On the active application tab, click a button to perform the following actions:

- Send Now—Sends the product and system information to the database via an
 https secure communication protocol. For example, if the PTC Creo Direct tab
 is active, sends the product and system information for PTC Creo Direct to the
 database.
- Send via e-mail—Sends a report in ASCII format as an e-mail attachment to a designated PTC Technical Support Engineer. For example, if the PTC Creo Direct tab is active, you can send a report for PTC Creo Direct to a PTC Technical Support Engineer or to your e-mail address.

Last Sent

For each PTC Creo application you have installed, you can view on the **Last Sent** tab, the product and system information that was last collected and transferred to PTC.



Click **Send via e-mail** to send an aggregated report in ASCII format. The report is sent as an e-mail attachment to a designated PTC Technical Support Engineer or to your e-mail address.

Using Quality Agent

PTC Quality Agent automatically collects information related to the application and system information and transfers it to PTC. The information transfer is based on the filter settings in the **Data Filters** tab and the frequency settings in the **Preferences** tab. For each session, PTC Quality Agent checks for new data before the transfer. The information is automatically encrypted and transferred using the SSL methodology.

As a user of a PTC Creo application, you can open the **PTC Quality Agent** dialog box to view the pending data to be transferred to PTC on the **Pending Data** tab. You can click the **Send via e-mail** button to send the PTC Creo application-related and system information report in ASCII format as an e-mail attachment to a designated PTC Technical Support Engineer and to your e-mail address. Both you and PTC Technical Support can then review the information simultaneously.

Uninstalling PTC Quality Agent

You can remove PTC Quality Agent independent of a PTC Creo application using the following steps:

- 1. Click Start ➤ Control Panel ➤ Programs and Features. The Uninstall or change a program screen appears.
- 2. In the application list, click PTC Quality Agent.
- 3. Click **Uninstall**. A confirmation dialog box opens.
- 4. Click **Yes** to remove PTC Quality Agent.

Ε

Installing PTC MKS Platform Components

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This appendix contains information on how to install PTC MKS platform components to work with CADDS 5 models.

Overview

You must install PTC MKS Platform Components 9.x along with the PTC Creo Parametric Interface for CADDS 5 component during PTC Creo Parametric installation to work with CADDS 5 parts and assemblies within PTC Creo Parametric. PTC MKS Platform Components 9.x 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 the Windows operating systems.
- Installs the Portmapper service for interoperability between CADDS 5 and PTC Creo Parametric.
- Copies the necessary startup files and libraries required to work with CADDS 5 files into a separate installation directory.

Installing PTC MKS Platform Components

Install PTC MKS Platform components using the following instructions:

- 1. Perform steps 1 through 7 as described in the section Installing the PTC Creo Applications: Simple Installation Process on page 34.
- 2. Click the PTC MKS Platform Components 9.x check box on the Application Selection screen.
- 3. Click Install. The Installation Progress screen indicates the progress of the installation.
- 4. Click **Finish** to complete the installation and close the Assistant.

After the installation is complete, see Working with CADDS 5 Models in PTC Creo Parametric on page 60 before you start.



Note

If you want to install updated packages of PTC MKS Platform components, you must first uninstall its existing instance. Follow the steps below to uninstall PTC MKS Platform components:

Navigate to the /cadds5 mks folder of the DVD image and run the following commands:

- For a 64-bit system msiexec /X cadds5 mks for windows64.msi/q
- For a 32-bit system msiexec /X cadds5 mks for windows32.msi/q

F

Installing the PTC Creo Applications in Silent Mode

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This appendix for system administrators is applicable only for Windows. It provides information on how to perform a silent installation of the following PTC Creo applications: PTC Creo Parametric, PTC Creo Direct, PTC Creo Layout, PTC Creo Simulate, and PTC Creo Options Modeler. The instructions for using command-line arguments are also common for PTC Creo Distributed Services Manager.

Guidelines and Advantages of a Silent Installation

The PTC Creo applications installed for the first time using PTC Installation Assistant can be reinstalled during subsequent installation sessions in silent (nongraphics and noninteractive) mode. The silent installation process with command-line arguments uses an XML file with application-specific configuration settings. An independent XML file is stored for each of the PTC Creo applications that you have installed. To perform a silent installation, provide the path to the corresponding XML file as an argument when running setup.exe from the DVD root directory at the command prompt. See Command Syntax for a Silent Installation on page 156 for more information.

Remember the following points for a silent installation:

- You cannot use the independent XML file of an earlier release or a maintenance release for a silent installation in the following cases:
 - The major release to be installed and the release of the application for generating the XML file are not the same.
 - The selected PTC Creo application, language, platform, or package is not on the DVD.
 - The prerequisites for installing the PTC Creo application are not met.
- You must run the Assistant only from the DVD when installing the PTC Creo applications for the first time.
- You cannot generate and download a license or install PTC License Server in silent mode. The installer cannot connect to PTC.com in silent mode. Hence, the license sources must already be accessible to all the computers on which the application software will subsequently be run. The silent installation process will neither validate any license nor attempt to access the license sources.
- You cannot copy an existing installation while performing a silent installation.



Note

If you change the installation folder structure of an installed PTC Creo application, "uninstall" or "reconfiguration" utilities may not work properly for that installation of the PTC Creo application.

Some of the advantages of a silent installation follow:

- You can efficiently mass-deploy PTC Creo applications across a large enterprise.
- You can source all the configuration settings, such as installation location, license source list, and startup commands, for installing a PTC Creo application from an XML file.

Strategies for the Mass Deployment of PTC Creo Applications

As an administrator you can adopt one of the following strategies to mass-deploy PTC Creo applications and client-side components such as PTC Creo View Express, PTC Creo Thumbnail Viewer, and PTC Creo Platform Agent in a large enterprise:

- Perform a silent installation of PTC Creo applications on all the client computers. See Workflow for a Silent Installation on page 154 for details. You can then choose to install the client-side components on each client computer. See Installing and Uninstalling Client-Side Components in Silent Mode on page 164 for details.
- Provide the client computers access to the applications installed on a single network-accessible machine. Perform the following steps for the client-side configuration:
 - 1. If required, install the client-side components on each client computer. See Installing and Uninstalling Client-Side Components in Silent Mode on page 164 for details.
 - 2. Update the registry entry for each PTC Creo application on each client computer using the created.reg file. See Updating the Registry File on page 166 for details.
- Copy a single prototypical installation onto each client computer. Then perform the following steps for the client-side configuration:
 - 1. If required, install the client-side components on each client computer. See Installing and Uninstalling Client-Side Components in Silent Mode on page 164 for details.
 - 2. Update the registry entry for each PTC Creo application on each client computer using the created.reg file. See Updating the Registry File on page 166 for details.

Note

- If you have installed the PTC Creo applications on your local computer and have also run the silent installation process, you need not update the registry file.
- After the silent installation is complete, if you want to reconfigure the installation using PTC Installation Assistant, you must explicitly provide the license source information on the License Identification screen.

Workflow for a Silent Installation

A system administrator can follow the workflow described below for a silent installation of one or more PTC Creo applications on a network location or on a client computer:

- 1. Perform a typical product installation with the configuration that you want to use for future installations.
 - a. Open a command shell and run setup.exe from the root directory of the DVD. PTC Installation Assistant starts.
 - b On the Welcome screen click **Install new software**
 - c. On the **License Identification** screen perform one of the following operations:
 - Drag a node-locked or floating license file into the **License Summary** area.
 - Enter the port@host address of the running license server.

Note

- Downloading a license file or installing PTC License Server must be done in a separate installation session.
- A license source is not validated during the silent installation process.
 Hence, you must ensure the accuracy of the license source list that is
 configured in the commands (.psf files) for starting the PTC Creo
 applications.
- The license source specification is designated in an XML file. It can be overridden using a command-line argument during the silent installation process. See Command Syntax for a Silent Installation on page 156 for details.
- d. Select the applications to install, customize your installation, and specify any other site-specific configurations.
- e. Click Install to begin the installation.
- f. Click **Finish** to close the Assistant.

An independent XML file for each installed PTC Creo application is stored with a .p.xml extension at creo_loadpoint>\<datecode>\<app_name>\bin\pim\xml. When installing PTC Creo applications, the Common Files folder is installed. The XML file for this folder is stored at the following location:

<creo_loadpoint>Creo 3.0\<datecode>\Common Files\bin\
pim\xml\creobase.p.xml.

Additionally, PTC Creo applications require XML files for the following components:

- PTC Quality Agent <dvd>:\pim\xml\qualityagent.xml
- PTC MKS Platform Components (Required only if installing CADDS 5 Adaptor) <dvd>:\pim\xml\mkscomponents.xml

Note

Edit the qualityagent.xml and mkscomponents.xml files, and set the value of the install attribute to yes.

2. Copy one or more PTC Creo application specific XML files, along with the creobase.p.xml,qualityagent.xml, and mkscomponents.xml

files, to a temporary location on your disk; for example, D: \temp\ptc, and edit it as required. See Editing the XML Files on page 158 for information on how to edit the various nodes of an XML file.

- 3. Perform a silent installation of the desired PTC Creo applications using the command syntax as described in the next section.
- 4. Perform a silent installation of the required client-side components. See Installing and Uninstalling Client-Side Components in Silent Mode on page 164 for information on how to install Client-side components.

Command Syntax for a Silent Installation

To perform a silent installation of a PTC Creo application, open a command shell, and run setup.exe from the root directory of the DVD as follows:

<dvd>:\setup.exe -xml "<path to the .xml file stored in a
temporary location>"

Where dvd is the DVD drive letter or the location of the downloaded DVD image.

The arguments you can use with setup.exe during a silent installation follow:

Command Syntax	Description
setup.exe <no arguments=""></no>	Runs the installer interactively.
setup.exe-help	Displays all the arguments to use with the installer.
setup.exe-v	Displays the version of the installer.
<pre>setup.exe -xml <file1> -xml <file2></file2></file1></pre>	Performs a silent installation as per the directives in the specified XML files.
<pre>setup.exe -upgrade -xml <file1> -xml <file2></file2></file1></pre>	Performs a silent installation with the removal of the highest version that already exists, and installs the current version of a release, according to the instructions in the specified XML files.
<pre>setup.exe -xml <file1> [-destination <directory>]</directory></file1></pre>	[Optional] Overrides the destination specified in the XML file.
<pre>setup.exe -xml <file1> [-licensesource <string>]</string></file1></pre>	[Optional] Overrides the license source specified in the XML file.
<pre>setup.exe -xml <file1> [-translate_lp_ path_space <character>]</character></file1></pre>	[Optional] Replaces the whitespace characters in the installation path in the XML file.

Command Syntax	Description
setup.exe-xml <file1></file1>	[Optional] Overrides the license feature list
[-licfeat_regen]	specified in the XML file. The default list is
	regenerated by checking the license source list
	as it is originally generated from an initial
	installation. Without this argument, the default
	license feature list is taken from the XML file.
<pre>setup.exe -xml <file1></file1></pre>	[Optional] Prepends the preferred license
[-licfeat_preference	feature names to the default license feature list.
<string>]</string>	Specify the preferred names within quotes or as
	a space-separated list without parentheses.
	Without this argument, the default license
	feature list is used as is.
setup.exe	[Optional] Displays the indicative progress bar
-progressbar-xml	during the installation using the XML file.
<file1></file1>	

Consider the example of performing a silent installation of PTC Creo Parametric with Common Files, and a required component PTC Quality Agent. Open a command shell and enter the following command:

E:\setup.exe-xml "D:\temp\ptc\pma.p.xml" --xml "D:\temp\
ptc\creobase.p.xml"

- E—The DVD drive letter
- D:\temp\ptc\pma.p.xml—The location for the <app_name>.p.xml file.
- D:\temp\ptc\creobase.p.xml—The location of the XML file for the Common Files folder.
- D:\temp\ptc\qualityagent.xml—The location of the XML file for the required component PTC Quality Agent.

Locating the XML Files

On a successful installation of the PTC Creo applications, XML files for each of the installed PTC Creo applications are created. The file location and file name is as shown:

Application Files	File Name	File Location
PTC Creo application	<app_name>.p.xml</app_name>	<pre><creo_loadpoint>\</creo_loadpoint></pre>
files		Creo 3.0\
		<datecode>\</datecode>
		<app_name>\bin\</app_name>
		pim\xml
Common Files folder	creobase.p.xml	<pre><creo_loadpoint>\</creo_loadpoint></pre>
		Creo 3.0\
		<datecode>\</datecode>
		Common Files\
		bin\pim\xml

For example, if you have installed PTC Creo Parametric, the XML file pma.p.xml is stored at creo_loadpoint>\Creo 3.0\<datecode>\
Parametric\bin\pim\xml. The XML file for the Common Files folder is stored at creo_loadpoint>\Creo 3.0\<datecode>\Common Files\bin\pim\xml\creobase.p.xml.

Editing the XML Files

You can edit and customize only those XML files listed in the table in the previous section with the settings required for future installations. In such cases, copy the XML files to a temporary location and edit the nodes in the XML files using a text editor or an XML editor. The following sections describe the various nodes in the XML file that can be edited.

Note

- PTC recommends you not to edit the XML files unless there is a specific need to do so.
- You can edit only those nodes that are documented in the subsequent sections.
- The nodes in the XML file may differ for various PTC Creo applications.

Editing the PROPERTY Node

You can edit the following name attributes of the PROPERTY node:

INSTALLBASE

Use the INSTALLBASE name attribute to change the root path for installing the PTC Creo applications in the current session. You can change the INSTALLBASE name attribute in the PTC Creo applications (<app name>.p.xml) and Common Files (creobase.p.xml) XML files. The default value is usually the PTC folder under the Program Files folder.

Note

Ensure that you specify same root path in the PTC Creo applications (<app name>.p.xml) and Common Files (creobase.p.xml) XML files.

Before editing the INSTALLBASE name attribute, the PROPERTY node appears as shown:

```
<PROPERTY name="[INSTALLBASE]">"F:Program Files\PTC"</PROPERTY>
```

After editing the INSTALLBASE name attribute, the PROPERTY node appears as shown:

```
<PROPERTY name="[INSTALLBASE]">"[PROGRAMFILES]\PTC"</PROPERTY>
```

The [PROGRAMFILES] variable in the above example represents the default program location on a local computer. You can use this variable instead of the absolute path F:\Program Files to make the default program location accessible on all the computers. For a 32-bit application running on a 64-bit computer, you can also use the [PROGRAMFILESx86] variable as shown in the following example:

```
<PROPERTY name="[INSTALLBASE]">"[PROGRAMFILESx86]\PTC"</PROPERTY>
```

The [PROGRAMFILESx86 variable represents the default program location for a 32-bit application on a 64-bit computer.

NoRegistryActions

You can add the NoRegistryActions name attribute to skip processing of the REGISTRY nodes. REGISTRY nodes define file associations.

<PROPERTY name="NoRegistryActions"</pre>

LP and CREOCOMMONFILES

The permitted editing of the LP and CREOCOMMONFILES property values is limited to changing or removing the whitespace characters. For example, you can change Creo 3.0 to Creo 3.0 or Creo 3.0. Similarly, you can change

Common Files to Common_Files or CommonFiles. You must not modify any other aspects of the LP or CREOCOMMONFILES paths. Before editing these property values, the PROPERTY node appears as shown:

```
<PROPERTY name="[LP]">[INSTALLBASE]/Creo 3.0/Parametric/
<PROPERTY name="[CREOCOMMONFILES]"[INSTALLBASE]/Creo 3.0/Common Files</pre>/
/
PROPERTY>
```

After editing the LP and CREOCOMMONFILES property values, the PROPERTY node appears as shown:

```
<PROPERTY name="[LP]">[INSTALLBASE]/Creo_3.0/Parametric/
PROPERTY name="[CREOCOMMONFILES]"[INSTALLBASE]/Creo_3.0/Common_Files
/
PROPERTY>
```

Note

- The CREOCOMMONFILES attribute is used by all the PTC Creo applications that are installed at the common loadpoint. When editing this attribute in an XML file, make sure to edit the value of this attribute in the XML files of all the individual applications installed at the same loadpoint.
- You can use the command line argument translate_lp_path_space to remove whitespace characters instead of manually editing the LP and CREOCOMMONFILES name attributes. See Command Syntax for a Silent Installation on page 156 for more information on translate_lp_path_space.

Editing the SHORTCUT Node

A SHORTCUT node represents shortcut definitions associated with certain package nodes. A SHORTCUT node is processed in order to create the shortcuts for a package that is being installed. In a SHORTCUT node, you must edit only the create attribute for each of the PROGRAMSMENU, STARTMENU, DESKTOP, and QUICKLAUNCH subnodes. A value Y set for the create attribute creates the shortcuts.

A sample SHORTCUT node in an XML file that creates a shortcut only in the Programs menu is as shown:

```
<SHORTCUT allusers="Y" id="pma_shortcut">
<NAME>Creo Parametric [VERSION]</NAME>
<ICON>[LP]/install/nt/parametric.ico</ICON>
<INDEX>0</INDEX>
<PATH>[LP]/bin/parametric.exe</PATH>
<WIN7APPID>PTC.Creo_Parametric</WIN7APPID>
<PROGRAMSMENU create="Y" installed="Y">PTC Creo</PROGRAMSMENU>
<STARTMENU create="N"/>
<DESKTOP create="N"/>
```

```
<QUICKLAUNCH create="N"/>
</SHORTCUT>
```

Editing the PSF Node

A PSF node represents the license configurations created in the **Customize** dialog box of the Assistant. Individual PSF nodes are created in the XML file for each of the configurations created in this dialog box. The resultant *.psf file is used for setting up the appropriate environment variables and for indicating the executables to be run. You can edit the following subnodes of the PSF node:

Subnode	Value
LICENSE_IDENTIFIERS	Provide a list of license sources separated by semicolons. The license sources can be node-locked license files or server-based port@host identifiers.
FEATURE_NAME	Provide a list of FLEXnet license-to-run features separated by spaces. The feature list is followed by a list of additional options within parentheses. Choose the options to acquire at startup.
DESCRIPTION	Identify the PSF run definition to be used by the software wrapper program (for example, parametric.exe) in case multiple PSF run definitions exist. The software wrapper program then opens a dialog box with a list of descriptions for launching the startup definition profile.

A sample PSF node and its subnodes in an XML file are as shown:

```
<PSF dest="[LP]/bin/parametric.psf" id="parametric" installed="Y"
name="parametric" source="psf_templates.xml>
<EXE>parametric.exe</EXE>
<LICTYPE name="CreoPma"><Creo Parametric</LICTYPE>
<PREFIX>CREOPMA PROE</PREFIX>
<DESCRIPTION>Creo Parametric (default)</DESCRIPTION>
<LICENSE_IDENTIFIERS>7788@localhost</LICENSE_IDENTIFIERS>
<FEATURE_NAME>CREOPMA_CUSTOMER ()</FEATURE_NAME>
<DFOR_PATH>%PRO_DIRECTORY%\libs\dfor\lib</DFOR_PATH>
<CV_ENV_HOME>%PRO_DIRECTORY%\%PRO_MACHINE_TYPE%\CV140</CV_ENV_HOME>
</PSF>
```

Editing the QUALITY AGENT Node

The QUALITY AGENT node represents the PTC Quality Agent settings for an installed PTC Creo application. You can edit the following attribute of the QUALITY AGENT node:

Attribute	Value
enable	Enables or disables the PTC Quality Agent reporting for a PTC Creo application. For example, if this attribute is set to Y, PTC Quality Agent reporting is enabled.
	P Note
	If the required attribute is set to Y, you must not change the value of the enable attribute to N. If you do so, a PTC Creo application will not run because of conflicting settings.

A sample QUALITY AGENT node in an XML file is as shown:

<QUALITY_AGENT enable="Y" keyname="QualityAgentOptIn" required="N"/>

Editing the LANGUAGE Node



Note

Editing the LANGUAGE, PLATFORM, and PACKAGE nodes is not trivial. For editing these nodes you must manually edit the CDSECTION nodes. Instead, PTC recommends that you perform a template-based installation graphically with the required language, platform, and package settings.

A LANGUAGE node represents the languages supported on the DVD media for the installation of a PTC Creo application. Individual LANGUAGE nodes are created in the XML file for each of the supported languages. You can edit the following attribute of the LANGUAGE node:

Attribute	Value	
install	Set this attribute to Y or N depending on	
	whether you want to install the PTC Creo	
	application in that language. If the	
	required attribute is set to Y do not	
	change the install attribute.	

A sample LANGUAGE node in an XML file follows:

```
<LANGUAGE install="Y" installed="N" name="usascii"
required="Y">English</LANGUAGE>
<LANGUAGE install="N" installed="N" name="japanese">Japanese</LANGUAGE>
```

Editing the PACKAGE Node

PACKAGE nodes in an XML file represent the required and the optional packages selected for a PTC Creo application on the **Customize** dialog box of the Assistant. There may be many PACKAGE nodes in an XML file. You can edit the following attribute of the PACKAGE node:

Attribute	Value
install	For a specific PACKAGE node, set this
	attribute to Y or N depending on whether you
	want to install the package with the PTC
	Creo application. For a parent package with
	multiple child packages, set the install
	attribute of the parent package to Y to install any of its child packages.

A sample PACKAGE node in an XML file follows:

```
<PACKAGE hidden="N" install="Y" label="ModelCHECK name="modelcheck" parent="creooptions" required="N"> </PACKAGE>
```

Editing the CDSECTION Nodes

Edit the CDSECTION node only if you want to edit the LANGUAGE, PLATFORM, or the PACKAGE nodes. Check for the following attributes before you edit the CDSECTION nodes:

- For each LANGUAGE node, locate the CDSECTION nodes with the language attribute reflecting the appropriate language.
- Then locate the CDSECTION nodes with the platform attribute reflecting the appropriate operating system.
- Finally locate the CDSECTION nodes with the name attribute reflecting the package to be installed with the PTC Creo application.

If all the attributes for the above nodes are to be considered for the installation, set the install attribute of the CDSECTION node to Y. Otherwise, set the attribute to N.

A sample CDSECTION node in an XML file follows:

```
<CDSECTION install="Y" installed="Y" location="[CREOCOMMONFILES]/[SHIPCODE]"
name="ModelCHECK" size="7126314>
```

Uninstalling the PTC Creo Applications in Silent Mode

You can uninstall the PTC Creo applications in silent mode by entering the following command for each of the applications in a command shell:

```
<creo_loadpoint>\<datecode>\<app_name>\bin\uninstall.exe -quiet
```

For example, to uninstall PTC Creo Parametric in silent mode, enter the following command:

```
C:\>"C:\Program Files\PTC\Creo 3.0\<datecode>Parametric\bin\uninstall.exe" -quiet
```

You must navigate to the installation directory of each PTC Creo application to silently uninstall it. In case the Common Files folder is installed, uninstalling the last PTC Creo application referencing the Common Files folder removes this folder automatically.

Installing and Uninstalling Client-Side Components in Silent Mode

The installers (MSIs) of the following client-side components reside on the DVD media, at path <dvd>: \install\addon\:

- PTC Creo Platform Agent
- PTC Creo Thumbnail Viewer
- PTC Creo View Express

Only the PTC Creo Platform Agent component get installed when you perform installation of any of the PTC Creo applications on a client computer. Refer to the following sections for the command-line syntax to run the installers for each of these components.

PTC Creo Platform Agent

Open a command shell and enter a command to install or uninstall PTC Creo Platform Agent in silent mode:

• Install PTC Creo Platform Agent at the default location using the following command:

```
"<dvd>:\install\addon\creoagent 32 64.msi" /passive
```

• Uninstall PTC Creo Platform Agent installed at the default location using the following command:

```
msiexec /uninstall "<dvd>:\install\addon\creoagent_32_64.msi"
/passive
```

PTC Creo Thumbnail Viewer

Open a command shell and enter a command to install or uninstall PTC Creo Thumbnail Viewer in silent mode:

Install PTC Creo Thumbnail Viewer using the following command:

```
"<dvd>:\install\addon\Thumbviewer 32 64.msi" /passive
```

Uninstall PTC Creo Thumbnail Viewer using the following command:

```
msiexec /uninstall "<dvd>:\install\addon\Thumbviewer 32 64.msi" /passive
```

PTC Creo View Express

To perform a silent installation of PTC Creo View Express, follow these steps:

1. Deploy the application using the PTC Creo View Express executable as shown in the following example:

```
start /w <dvd>:\install\addon\pvx32 64\CreoSetup.exe
/vADDLOCAL="ALL" /qn
```

In the previous example:

- /qn installs the application in silent mode
- start /w starts the application from the command prompt and waits for the application to terminate

If you have not specified the APPLICATIONFOLDER property, the default installation path is used. To override this path, specify the APPLICATIONFOLDER property at the command prompt as shown in the following example:

```
"start /w <dvd>:\install\addon\pvx32 64\CreoSetup.exe
/vADDLOCAL="ALL" APPLICATIONFOLDER="c:\ptc\CreoViewExpress" /qn"
```

To uninstall PTC Creo View Express in silent mode, enter the following command in a command shell:

```
msiexec.exe /x <dvd>:\install\addon\pvx32 64\pvexpress\
CreoView Express 32 64.msi /qn
```



Note

Running the PTC CreoView Express 32 64.msi file alone does not completely install PTC Creo View Express unless certain software prerequisites are met. Microsoft C++ Runtimes, OpenGL 1.1 or later, and Arbortext IsoView (optional) are required for installing and running PTC Creo View Express directly using the MSI file.

For more information on installing and uninstalling PTC Creo View Express in silent mode, see Chapter 7, Deploying Creo View and Customizing the Installation, in Installing PTC Creo 3.0: Creo View.

Installing PTC Quality Agent and PTC MKS Platform Components in Silent Mode

To silently install PTC Quality Agentand PTC MKS Platform Components, you must copy the qualityagent.xml and the mkscomponents.xml files found at <dvd>:\pim\xml\ to your local computer. Edit the "install=" attribute of the MSI entry to be "Y" and then perform the silent installation at the command prompt.

Updating the Registry File

A list of registry changes made by PTC Installation Assistant during the prototypical installation is recorded on the computer where the PTC Creo applications are installed. The file resides at the following location:

<creo loadpoint>\<datecode>\<app name>\bin\pim\xml\created.reg

Use this file to perform client configuration changes (registry edits) on a client computer on which you did not run the silent installation process. For example, the client computer can be the one that runs the software from a network installation. It can also be the one that received a copy of the installation.

The default location of the registry file for some of the PTC Creo applications is as shown:

C:\Program Files\PTC\Creo 3.0\<datecode>\Parametric\bin\pim\xml\created.reg (PTC Creo Parametric)

C:\Program Files\PTC\Creo 3.0\<datecode>\Direct\bin\pim\xml\created.reg
(PTC Creo Direct)

To make configuration changes on the client computer, open the created.reg file stored on the computer where the PTC Creo applications are installed. Then perform one of the following steps:

- Merge the entries of this file with another file you are using on the client computer.
- Open the **Registry Editor** dialog box and click **File** ▶ **Import** to import the entries of the file to the client computer.

G

Port Usage by Creo Applications

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This appendix provides information on the communication ports used by various PTC Creo applications and the related client applications.

Using Communication Ports

Many of the PTC Creo applications communicate with one another and with the other client applications through a TCP/IP communication protocol. Such applications must be registered in the firewall. The TCP/IP protocol uses a port as a communication endpoint in the operating system of the host computer. A port is associated with the IP address of the host computer. It is identified for each IP address and protocol by a 16-bit number commonly known as the port number.

The port number for a TCP/IP communication can be allocated by a PTC Creo application. In such cases you can configure the port number using an environment variable. If a port number is allocated dynamically by the operating system on the host computer, you cannot change the value.

The port allocation for various PTC Creo and client applications follow:

Application	Environment Variable Setting
Nmsd—PTC name service used by asynchronous Pro/TOOLKIT applications and PTC Creo Distributed Services Manager to connect to PTC Creo Parametric. This application runs on the client workstation and uses the same host TCP/IP communication protocol for all the processes on that workstation. The default port number is 1239.	Set PTCNMSPORT using Control Panel ► System ► Advanced system settings ► Environment Variables. F Note You cannot set this environment variable in your .psf file.
pro_comm_msg.exe —Facilitates communication between the executable xtop.exe and asynchronous Pro/ TOOLKIT applications. ▶ Note The executable xtop.exe refers to all the desktop-based PTC Creo applications.	A port number can be allocated by a PTC Creo application or it can be set dynamically. • 6000–Change this value by specifying a value to the PRO_COMM_ MSG_COMPAT_PORT environment variable. Either the first unused port starting with the value of PRO_COMM_ MSG_COMPAT_ PORT or of 6001, whichever is bigger, is allocated. • 2 dynamic ports—Cannot configure assigned port values after their allocation.
PTC Portmapper—If installed, communicates port information	Not supported.
between Pro/TOOLKIT and a PTC	

Application	Environment Variable Setting
Creo application. If it is not	
installed, Windows registry is used	
to store the port values. This	
application runs on the client	
workstation and uses the same host	
TCP/IP communication protocol	
for all the processes on that	
workstation.	
xtop.exe—Connects to	Not supported.
creoagent, nmsd, and pro_	
comm_msg to open a few	
dynamically allocated ports. This	
application uses the same host	
TCP/IP communication protocol	
for all the processes on a	
workstation.	
creoagent.exe	Not supported.
—Uses dynamically allocated ports	
to connect to all the PTC Creo	
applications. This application uses	
the same host TCP/IP	
communication protocol for all the	
processes on a client workstation.	
Async TK Apps—Use dynamically	Not supported.
allocated ports to connect to pro_	
comm_msg.exe on a client	
workstation.	
Converters—Use dynamically	Not supported.
allocated ports to connect to	
desktop-based PTC Creo	
applications on a client	
workstation.	
CreoView—Uses dynamically	Not supported.
allocated ports to communicate	
with browser plugins, nmsd, and	
comm_brk_svc.exe on a client	
workstation.	

Application	Environment Variable Setting
—Uses dynamically allocated ports to support communication between xtop.exe, third-party CAD software, and Creo View sessions.	Not supported.
PTC Creo Distributed Services Manager—Uses the TCP/IP communication protocols between multiple workstations. • The executable dbatchc.exe is the dbatch client interface that runs on a client workstation. This interface communicates with the Service Manager dsq.exe located on a different workstation.	Set PTCNMSPORT using Control Panel ► System ► Advanced system settings ► Environment Variables to configure the default port number. Make sure that you set the same port number value on all the service workstations.
The dbatch service executable dbatchs.exe runs on a service workstation.	
• The executable dsq.exe uses a dynamic port to communicate with dbatchc.exe and multiple dbatchs.exe processes. Each service workstation runs nmsd.exe.	

Port Usage for PTC License Server

PTC License Server runs on a designated server and requires a two-way TCP/IP communication between the client workstation and the server. The default port 7788 is used by the lmgrd and the lmadamin license server managers. The vendor daemon ptc_d dynamically selects an available port. You can designate a port value other than the default value for lmgrd, lmadamin, and ptc_d using the procedures given in the subsequent sections. After designating appropriate port values for these applications, you must register the values in the firewall. Otherwise license transactions will not happen.

Changing the Default Port on the Server

Different procedures for changing the default port on the designated server for lmgrd and the lmadmin follow. If you are running lmgrd as the license server manager, use the following procedure:

- Navigate to the cense_server_manager_loadpoint >\licensing folder.
- 2. Open the license.dat file in a text editor.
- 3. At the end of the line beginning with SERVER, change the port number 7788 to the desired open port.
- 4. Save and close the file.
- 5. Stop and restart the license server process using the lmtools utility.
- 6. Edit the Creo 3.0\app\bin*.psf and the Creo 3.0\app\bin\
 *.bat files to change the reference of LM_LICENSE_FILE= or PTC_D_
 LICENSE_FILE = to reflect the new port@hostname for the server.

If you are running lmadmin as the license server manager, use the following procedure to change the default port on the designated server:

- 1. Perform the previous steps 1 through 4.
- 2. Stop the lmadmin ptc service.
- Navigate to the cense_server_manager_loadpoint>\conf folder.
- 4. Open the server.xml file in a text editor.
- 5. In the line beginning with licenseServer, change the reference port 7788 to the desired port matching the SERVER line in the license.dat file.
- 6. Navigate to the cense_server_manager_loadpoint>\bin
 folder.
- 7. Restart the lmadmin ptc service.
- 8. Edit the Creo 3.0\app\bin*.psf and the Creo 3.0\app\bin\
 *.bat files to change the reference of LM_LICENSE_FILE= or PTC_D_
 LICENSE_FILE = to reflect the new port@hostname for the server.

Setting a Nondefault Vendor Daemon Port

Different procedures for setting a nondefault vendor daemon port on the lmgrd and lmadmin follow. If you are running lmgrd as the license server manager, use the following procedure:

- Navigate to the license_server_manager_loadpoint
 licensing folder.
- 2. Open the license.dat file in a text editor.
- 3. Add the keyword port=52727 (or any number > 2000) at the end of the line beginning with DAEMON. This designates the desired listening port for the ptc d daemon.
- 4. Save and close the file.
- 5. Stop and restart the license server process using the lmtools utility.

If you are running lmadmin as the license server manager, use the following procedure to set a nondefault vendor daemon port:

- 1. Perform steps 1 through 4 explained previously.
- 2. Stop the lmadmin ptc service.
- Navigate to the cense_server_manager_loadpoint>\conf folder.
- 4. Open the server.xml file in a text editor.
- 5. In the line beginning with DAEMON, change port=0 to the desired port matching the DAEMON line in the license.dat file.
- 6. Restart the lmadmin ptc service.

Updating the Client Computers for the New License Server Port

If you have changed the license server manager port for lmgrd or lmadmin, you must update the client computers to obtain a license:

- 1. For each installed PTC Creo application, navigate to the bin folder in the PTC Creo load point. For example, if PTC Creo Parametric is one of the installed applications, navigate to the <creo_loadpoint>\
 Parametric\bin folder.
- 2. Run reconfigure.exe to open PTC Installation Assistant. The option Reconfigure existing software is selected by default.
- 3. Click Next. The License Identification screen appears.
- 4. Click in the **License Summary** area to remove the old server reference.
- 5. Type the port@hostname value corresponding to the new server under the **Source** column. For example, the port and the host name can be 7788@myhostname. Click to add any new license sources.
- 6. Click Next.
- 7. Click **Install** and then click **Exit**.



Distributed Computing Technology for PTC Creo Parametric

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Supported Functions	
Configuring Workstations for Distributed Computing	

This appendix contains information on the Distributed Computing technology that is used by PTC Creo Parametric to augment your existing hardware.

Overview of Distributed Computing

PTC Creo Parametric uses Distributed Computing technology to perform computationally intensive tasks by augmenting your existing hardware. Distributed Computing technology includes the following elements:

- A PTC Creo Parametric 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 PTC Creo Parametric 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 PTC Creo Parametric 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.
- Tasks are dispatched to workstations based on the following real-time conditions:
 - 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

System administrators can configure the workstations after installing PTC Creo Parametric as follows:

- 1. From the available workstations, choose the ones to participate in distributed computing sessions.
- 2. Run the dcadsetup script on each participating workstation. This script starts a required daemon process 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 command enables the workstation for distributed computing after it is shut down and restarted.

Troubleshooting Tips

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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.

General Debugging Hints

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

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. For information on FlexNet Publisher diagnostic environment variables, consult the *FlexNet Publisher License Administration 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: 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 workstation for executing the PTC Creo application.

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 Pro/ENGINEER 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 license server manager and ptc_d daemons are running. A network problem exists if a license client attempts to ping the server by host name and the ping fails.

Invalid Licenses

Symptom: You receive the error message Invalid license.

Cause: Licensing information is inaccurate.

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

If all licenses are listed as <code>invalid</code>, verify that the PTC <code>Host_ID</code> in the License Pack corresponds with what you see in the <code>FLEXnet license editor</code>. 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 <code>HOST ID</code>.

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.

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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. IdleConnections, 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.

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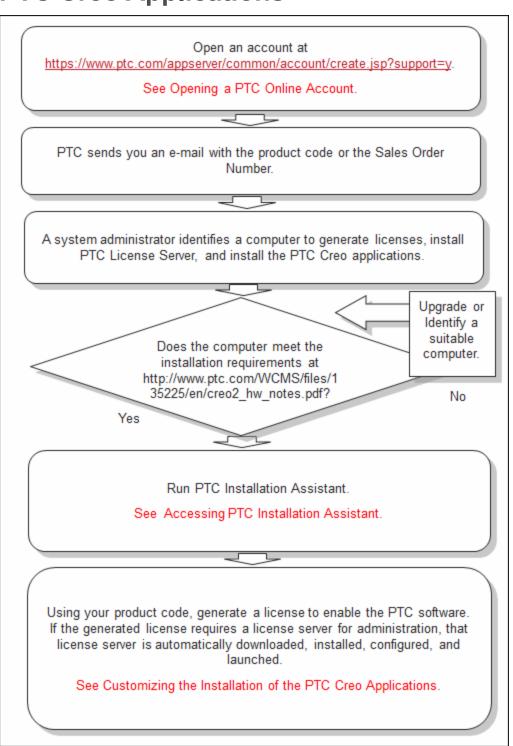
J

Workflow for Installation and Configuration

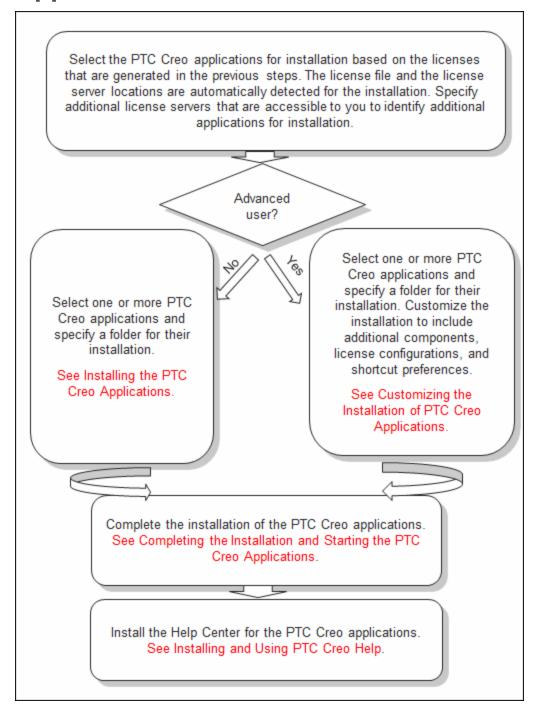
Workflow for Generating Licenses for the PTC Creo Applications	184
Workflow for Installing the PTC Creo Applications	
Workflow for Reconfiguring the PTC Creo Applications	186

This chapter provides simple workflows for generating licenses and installing and reconfiguring the PTC Creo applications. The workflow is presented as a flowchart for each of these processes. If you have received a license file via e-mail from PTC License Management, use that license file instead of the product codes for licensing. After the installation, you can configure the installation of applications like PTC Creo Parametric for units of measurement or drawing standards.

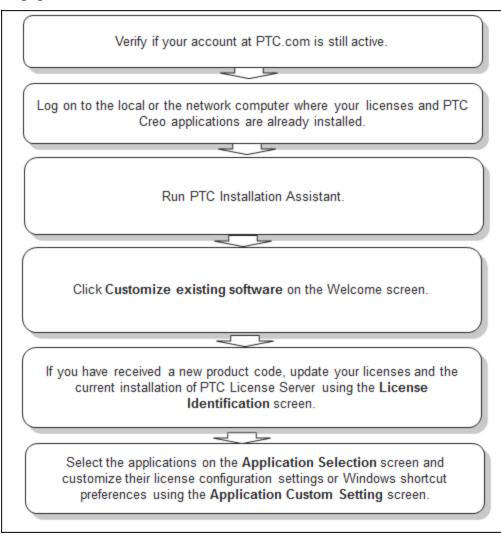
Workflow for Generating Licenses for the PTC Creo Applications



Workflow for Installing the PTC Creo Applications



Workflow for Reconfiguring the PTC Creo Applications



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.	
PTC Creo Direct	A 3D mechanical design automation application that helps you create and modify 3D designs quickly and easily through direct interaction with their geometry.	
PTC Creo Layout	A 2D mechanical design application for developing concepts from scratch using existing 2D data or 3D cross sections. A native data connection to PTC Creo Parametric allows rapid transition from 2D concept to 3D design.	
PTC Creo Options Modeler	An application that includes limited parametric modeling and configuration modeling features.	
PTC Creo Parametric	A 3D mechanical design automation application, PTC Creo Parametric provides associative and interoperable product design and an engineering modeling system.	
PTC Creo Parametric J-Link	A Java-based toolkit, J-Link allows developers to create Java applications to access a PTC Creo Parametric session.	
PTC Creo Parametric Pro/ Web.Link	PTC Creo Parametric Web.Link lets you interact with PTC Creo Parametric through a Web browser. With this feature, you can create a	

Term	Definition
	customized Web page, which you can then use to automate and streamline aspects of your engineering process.
PTC Creo Parametric TOOLKIT	An API that enables adding of functionality to PTC Creo Parametric using the C programming language.
PTC Creo Parametric VB API	A visual basic toolkit, Visual Basic API allows developers to create VB.NET or VBA applications to access a PTC Creo Parametric session.
PTC Creo Plastic Advisor	PTC Creo Plastic Advisor simulates mold filling for injection molded plastic parts and provides designers with immediate, easy access to reliable manufacturing feedback and advice.
PTC Creo Simulate	A Computer Aided Engineering application. It helps you simulate the physical behavior of a model and understand and improve the mechanical performance of your design. You can directly calculate stresses, deflections, frequencies, heat transfer paths, and other factors, showing you how your model will behave in a test lab or in the real world using this application.
PTC Creo 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 PTC Creo View.
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.
daemon line	An entry in the license file that defines • 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
educational license	A license used at educational institutes to run PTC software, also known as a students license. This is

Term	Definition
	a time-bound license.
evaluation license	An evaluation license allows use of a software
	product for a trial period.
feature line	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:
	INCREMENT PROE_Engineer ptc_d 33.0 01-jan-2011
feature name	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"\ The feature name is the name of the license that
feature name	
	the application requests. It is composed of a product prefix and a configuration ID (for
	example, PROE Engineer).
FlexNet Publisher	The license management software that is used to distribute licenses
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 called the load point.
intellectual property	Proprietary information stored within PTC Creo data files.

Term	Definition
install number	A number used by the PTC order management system before October 20, 1997, to identify a single software installation.
license	A license grants a user permission to run a PTC product.
license borrowing	A functionality of FlexNet Publisher. 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 license.txt 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 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 server	A machine or system process that distributes licenses to license clients. For example, if you start PTC Creo Parametric on machine A and it requests a license from machine B, then machine B is the license server. The license server keeps

Term	Definition
	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.
license-to-run	A license-to-run invokes the license of a specific PTC application, such as PTC Creo Parametric. Floating licenses are available for use on any host machine on the network at any particular site. See floating license.
lmgrd license server manager daemon	The FlexNet Publisher license server manager daemon (lmgrd or lmadmin) runs on the license server and works with the PTC vendor daemon. It does not manage license usage. The lmgrd or lmadmin 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.
load point	Directory where the software is installed. An example of the PTC Creo Parametric load point is C:\Program Files\PTC\Creo 3.0.
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 PTC Creo Parametric, ModelCHECK analyzes parts, drawings and assemblies, and recommends proper PTC Creo Parametric 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 (nmsd)	This daemon process enables remote communication between PTC Creo Parametric 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

Term	Definition
	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.
product prefix	This standard prefix corresponds to a particular product. This product prefix might be used as a prefix within a feature name, such as PROE_Engineer_12345, where 12345 is the Configuration ID.
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.
PTC Application Manager	A utility that appears as a toolbar that enables the user to start and close other applications during a PTC Creo Parametric session.
PTC daemon	The PTC vendor daemon (ptc_d) 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 PTClm, called the CPU ID. Also referred to as the machine address.
PTC Installation Assistant	A utility with menus, dialog boxes, and screens for obtaining licenses and installing PTC Creo applications.
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.
silent mode	A nongraphics mode for installing and uninstalling software using command-line arguments.
silent installation	The process of installing software using command-line arguments.
startup command	The command that is used to start an installed

Term	Definition
	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.
Thumbnail viewer	A Windows-only utility for viewing thumbnails of objects outside PTC Creo Parametric.
trail file	A trail file records the procedures the end user performed while running a PTC Creo 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.

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