



**Installing Creo® 2.0:  
Creo Parametric, Creo Direct,  
Creo Layout, Creo Simulate,  
and Creo Options Modeler**  
Datecode M070

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**PTC Inc., 140 Kendrick Street, Needham, MA 02494 USA**

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## About This Guide

The 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 Creo applications—Creo Parametric, Creo Direct, Creo Layout, Creo Simulate, and Creo Options Modeler—using PTC Installation Assistant. The installation instructions are also common for Creo Distributed Services Manager.

System administrators who are installing the 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 Creo applications, see the following chapters:

Chapter	Role	Information
<a href="#">Preparing to Install Your PTC Software on page 9</a>	All users	Describes the materials in your software shipment. Also provides an overview of PTC Installation Assistant.

Chapter	Role	Information
<a href="#">Quick-Start Installation for Experienced Users on page 17</a>	Advanced users	Describes a quick-start approach for users who have previously installed license management and product software.
<a href="#">Configuring Licenses for the Creo Applications on page 21</a>	All users	Describes how to generate or update licenses and install PTC License Server.
<a href="#">Installing the Creo Applications on page 29</a>	All users	Describes a simple installation process for the Creo applications.
<a href="#">Customizing the Installation of the Creo Applications on page 35</a>	Advanced users	Describes the advanced customization options for installing the Creo applications.
<a href="#">Completing the Installation and Starting the Creo Applications on page 43</a>	All users	Describes the steps to complete the installation of Creo applications and the procedure to uninstall them.
<a href="#">Installing and Using Creo Help on page 55</a>	Advanced users and system administrators	Describes how to install and access Help for Creo Parametric, Creo Simulate, Creo Direct, Creo Layout, and Creo Options Modeler from a local host computer, a network drive, and from a Web server.
<a href="#">Updating an Installation on page 63</a>	All users	Describes how to update the Creo applications on your system.
<a href="#">Installing the Creo Applications in Silent Mode on page 139</a>	System administrators	Describes how to install and uninstall the 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 Creo 2.0: Creo Schematics*
- *Installing Creo 2.0: Creo View*
- *Installing Creo 2.0: Creo Illustrate*

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- *FlexNet Publisher License Administration Guide* that discusses the third-party license management software for distributing Creo licenses
  - *Installing and Configuring the Standalone PTC License Server* that discusses the installation of the ladmin-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 Creo Parametric, Creo Direct, Creo Layout, Creo Simulate, and Creo Options Modeler are available:

- A Help Center with context-sensitive Help, search facility, and quick links to helpful information.
- *Installing Creo 2.0: Creo Parametric, Creo Direct, Creo Layout, Creo Simulate, and Creo Options Modeler* 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:

`mcad-documentation@ptc.com`

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

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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, buttons, and other selectable elements from the user interface	Click <b>File ▶ New</b> . Click <b>OK</b> .
Courier	User input, system messages, directories, and file names	Processing completed.
Courier with less-than and greater-than symbols (< >)	Variables for which an appropriate value is substituted	output= <LOADPOINT>



# 1

## 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.

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## 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 Creo applications: Creo Parametric, Creo Direct, Creo Layout, Creo Simulate, and 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 [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 [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.

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## 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 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 [www.ptc.com/WCMS/files/135225/en/creo2\\_hw\\_notes.pdf](http://www.ptc.com/WCMS/files/135225/en/creo2_hw_notes.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 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 Creo applications based on the license sources that you have been provided. You can perform the following tasks, many automatically, 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 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 171 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.

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## 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

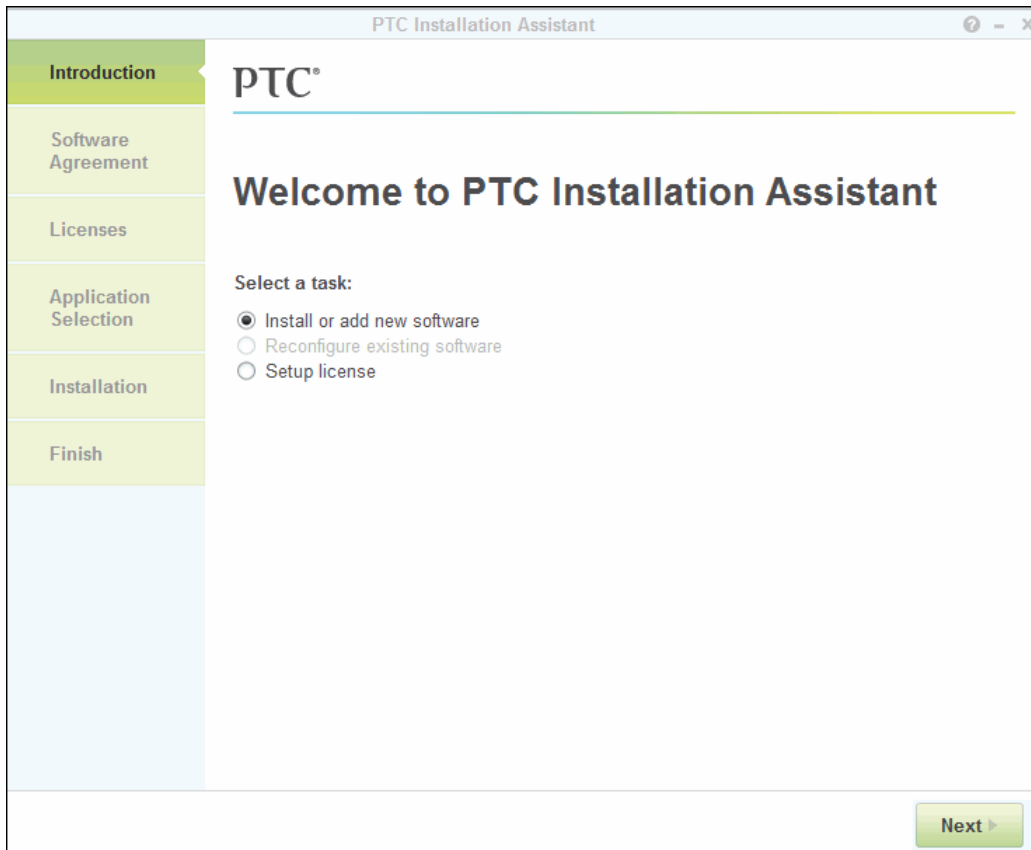
You can start PTC Installation Assistant by running `setup.exe` from the root directory of either the DVD or the downloaded installation files. Other than using the default of English, you can run the Assistant in French, German, Italian, Spanish, Simplified Chinese, Traditional Chinese, Korean, Japanese, 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 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 or add new 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 **Sign Software License Agreement** screen appears.

PTC Installation Assistant

✓ Introduction

Software Agreement

Licenses

Application Selection

Installation

Finish

PTC®

### Sign Software License Agreement

To continue you must read and agree to the terms of the software license agreement.


PTC CUSTOMER AGREEMENT  
 THIS PTC CUSTOMER AGREEMENT ("AGREEMENT") IS A LEGAL AGREEMENT BETWEEN THE INDIVIDUAL, OR THE COMPANY OR OTHER ORGANIZATION ON WHOSE BEHALF SUCH INDIVIDUAL ACCEPTS THIS AGREEMENT, THAT EITHER (A) CLICKS THE "I ACCEPT" BUTTON BELOW OR (B) INSTALLS, ACCESSES, OR USES ANY SOFTWARE OR DOCUMENTATION FROM PTC ("CUSTOMER"), AND PTC INC. OR, IF THE PURCHASE WAS MADE IN A COUNTRY SPECIFIED ON SCHEDULE A TO THIS AGREEMENT, THE PTC AFFILIATE SPECIFIED ON SCHEDULE A (AS APPLICABLE, "PTC"). PLEASE READ THE TERMS AND CONDITIONS OF THIS AGREEMENT CAREFULLY BEFORE ACCEPTING THIS AGREEMENT. BY CLICKING ON THE "I ACCEPT" BUTTON BELOW OR BY INSTALLING, ACCESSING, OR USING ANY SOFTWARE OR DOCUMENTATION FROM PTC, CUSTOMER HEREBY AGREES TO BE BOUND BY THIS AGREEMENT AND REPRESENTS THAT IT IS AUTHORIZED TO DO SO.


I accept the software license agreement  
 I do not accept the software license agreement

By checking this box, I confirm that the software is being installed and used in compliance with the [Export Agreement](#) and agree to the terms of the [Export Agreement](#) and the export terms of the above License Agreement.

◀ Back   Next ▶

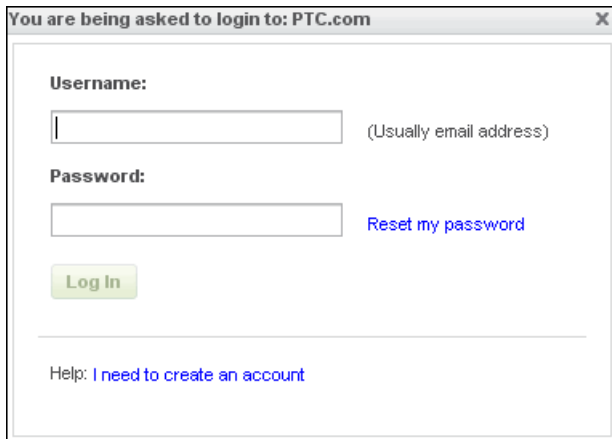
---

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.
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:



The screenshot shows a login dialog box with the following elements:

- Window title: You are being asked to login to: PTC.com
- Username:  (Usually email address)
- Password:  [Reset my password](#)
- 
- Help: [I need to create an account](#)

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 30.

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## 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 64](#) for details.

## Installing Creo Applications

Install 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 or add 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 Creo Applications on page 21](#). 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 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 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>\Common Files` folder accessible to all the applications.

7. Customize each 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 the Creo Applications on page 29](#) for details on the simple installation process. See [Customizing an Installation on page 36](#) 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 2.0\<<app_name>
```

PTC License Server, the 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.*



# 3

## Configuring Licenses for the 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 Creo applications. You must install PTC License Server before installing your 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 67](#) 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.

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## Using PTC Installation Assistant for Licensing

You must generate or update existing licenses to use the latest software for Creo 2.0. Licenses determine the Creo applications that you are authorized to run on your computer. The Assistant helps you perform the following licensing-related 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 11.10.1. 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.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 Creo applications in this guide follow:

- 
- For Windows XP, Windows XP x64, and 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.
  - For Windows Server 2008 R2, FlexNet Publisher 10.8.10, or later, is supported. It is available for installation only as a downloadable utility.
  - To use the `lmadmin`-based license server manager on Windows platforms, you must install the Microsoft Visual C++ 2005 SP1 Redistributable Package(x86). You can choose to install this package during the license server installation process.

## 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 [www.ptc.com/WCMS/files/135225/en/creo2\\_hw\\_notes.pdf](http://www.ptc.com/WCMS/files/135225/en/creo2_hw_notes.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](http://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 Creo 2.0.
- 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 `lmgrd` to `lmadmin`, 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.

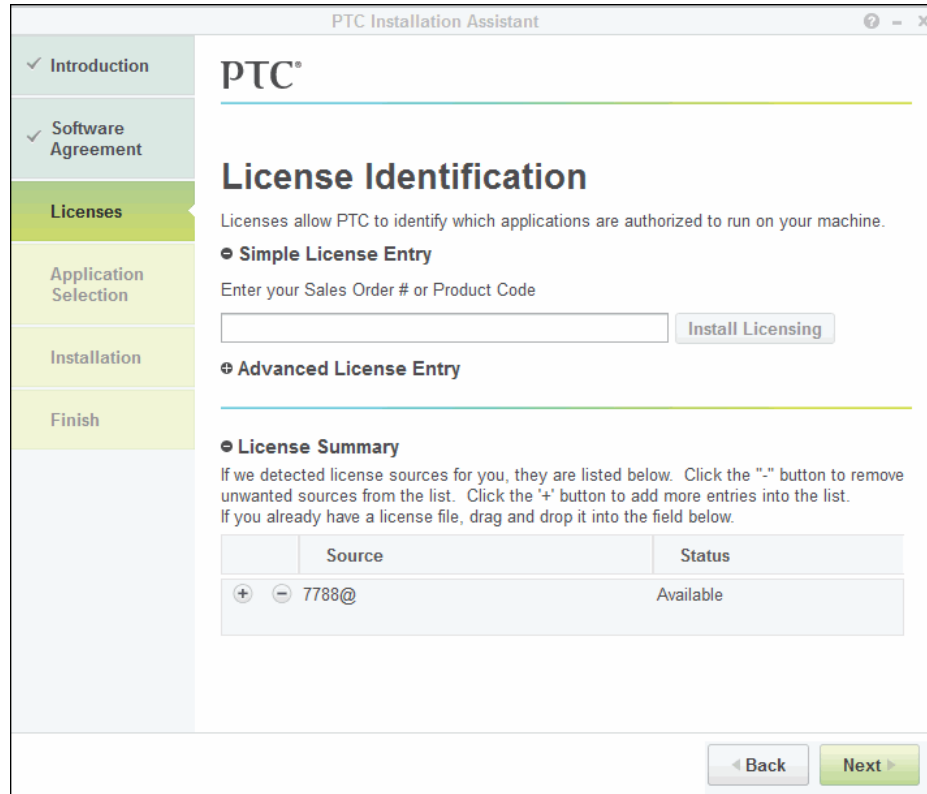
## Generating or Updating Licenses

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 or add new software** depending on the tasks that you want to perform:
  - **Setup license** generates or installs licenses.
  - **Install or add new software** performs license generation or identification tasks after which you can install the software.
3. Click **Next**. The **Sign 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.



## 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 "All users"\application data\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.

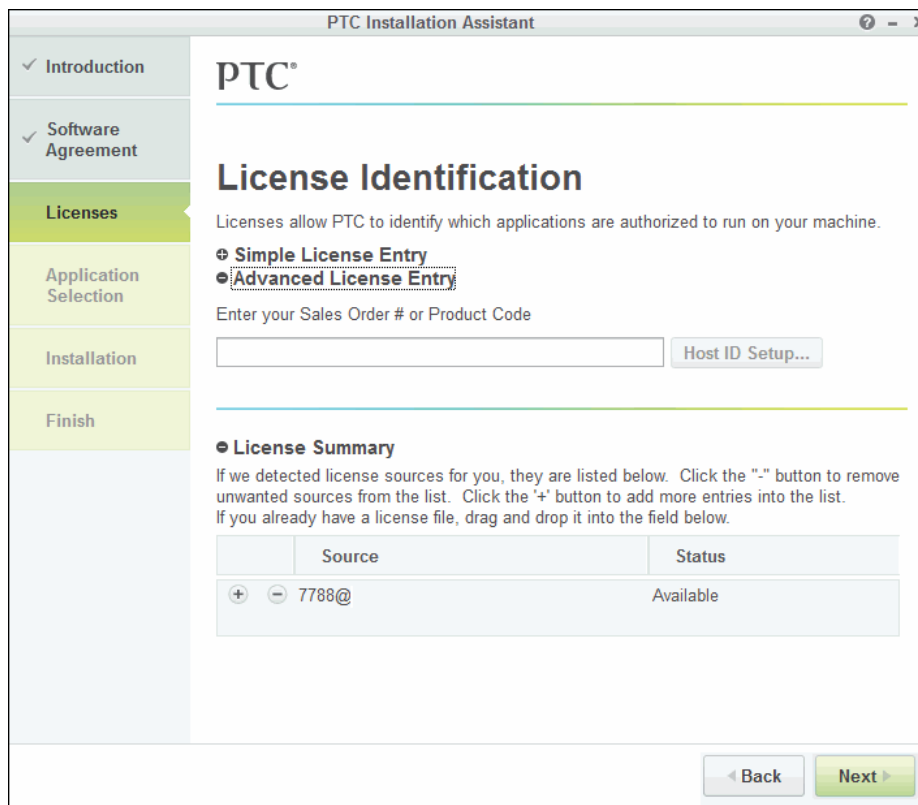
The complete path of the license file or the `port@hostname` value of the license server appears in the **Source** column of the **License Summary** area. This area displays all the license files or servers available for use on your computer.

- 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 or add 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:

- Click **+** next to **Advanced License Entry** to expand the area.



- Type your Sales Order Number or your product code in the corresponding text box.
- Click **Host ID Setup** to open the **Host ID Setup** dialog box.

Host ID Setup

Sales Order # or Product Code

wsdfdfsd

Choose from the Host ID below.

Machine ID	Port	Host
wsdfdfsd	Dell Wireless 1397 ...	7788 : 03L


Triad Server

Complete the additional information below.

Machine ID	Port	Host
	7788	
	7788	

Install Licensing

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 or add new software** to proceed with the installation.

---



## 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 or add new software** to proceed with the installation.

# 4

## Installing the Creo Applications

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This chapter provides information on how to install the Creo applications: Creo Parametric, Creo Direct, Creo Layout, Creo Simulate, and Creo Options Modeler. The installation instructions are also common for Creo Distributed Services Manager.

---

## Installation Prerequisites

See [Before You Begin on page 11](#) to check if all the installation prerequisites are met. Install the Creo applications and Creo Distributed Services Manager after you have obtained a license and installed PTC License Server. See the chapter [Configuring Licenses for the Creo Applications on page 21](#) 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. A license source must be identified in order to install any Creo applications.*

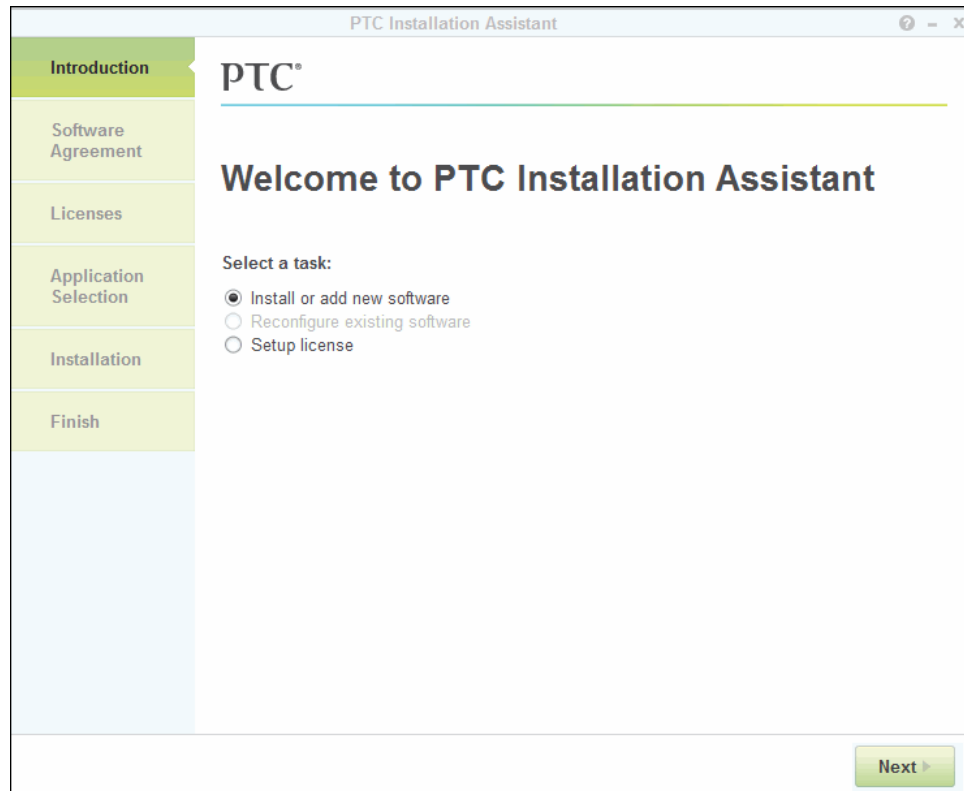
## File Format Compatibility

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

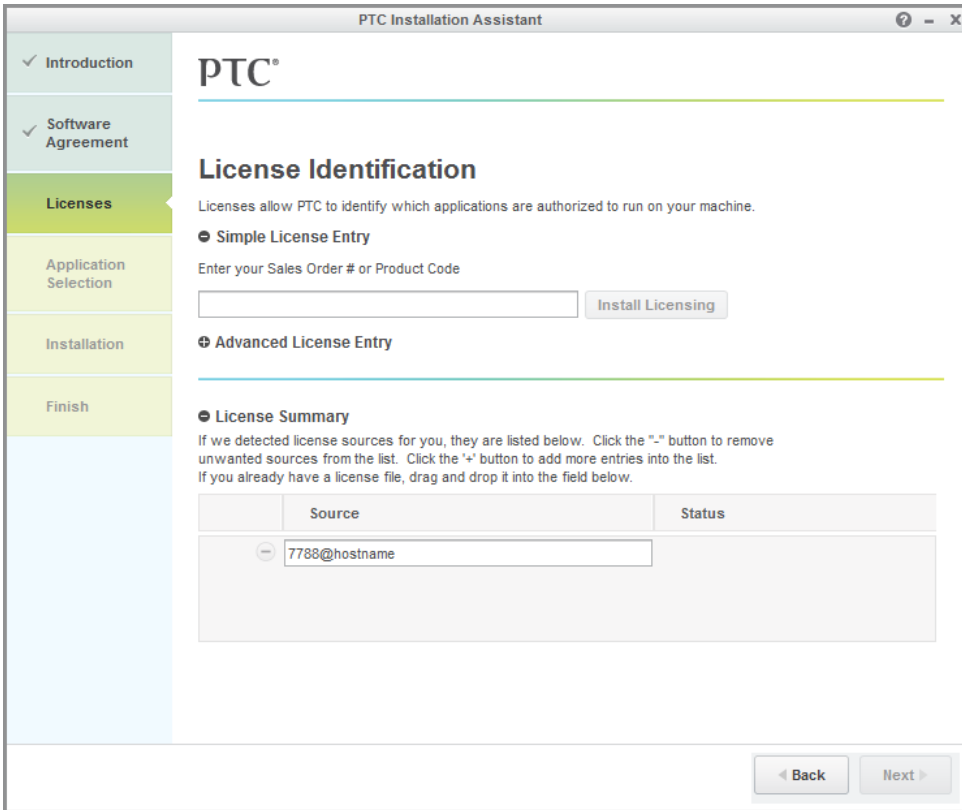
## Installing the Creo Applications: Simple Installation Process

Using PTC Installation Assistant, you can install the Creo applications and 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.
2. Click **Install or add new software**.



3. Click **Next**. The **Sign 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 Creo Applications](#) on page 21.

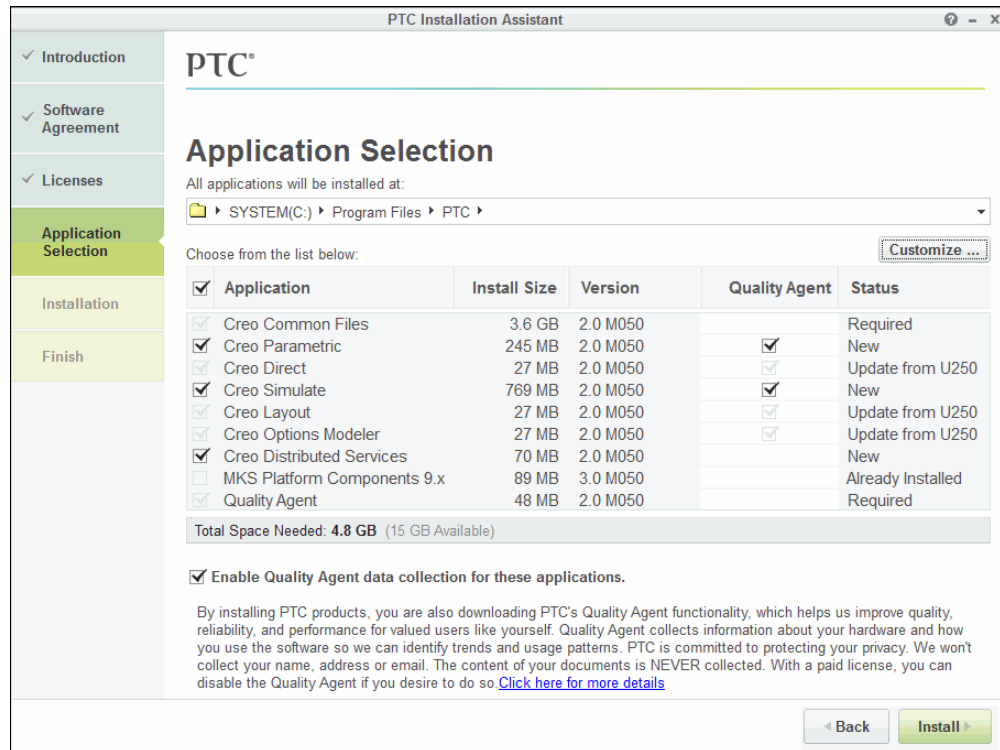
If you have configured your licenses, confirm that your desired license source appears in the **License Summary** area, or add it using one of the following methods:

- Drag and drop a valid license file into the **Source** column for a node-locked 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.





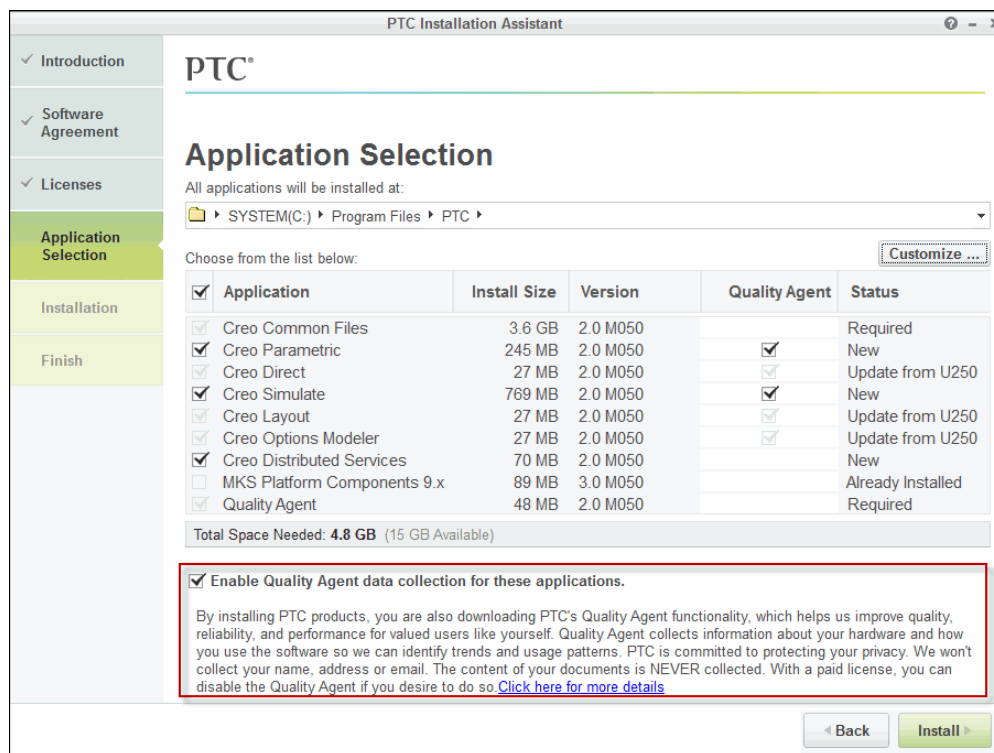
8. Specify an installation folder or retain the default folder.
9. Select one or more applications from the list. Before doing so, check if you are installing the correct version of the software and whether the disk space requirements are met.
10. Perform one of the following steps:
  - Click **Install** 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 Creo Applications on page 35](#) 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 Quality Agent with the Creo Applications

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



Quality Agent helps you to gather and send reports to PTC on the performance and usage of the 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 Quality Agent on page 129](#) for more information on the uses of Quality Agent.

# 5

## Customizing the Installation of the Creo Applications

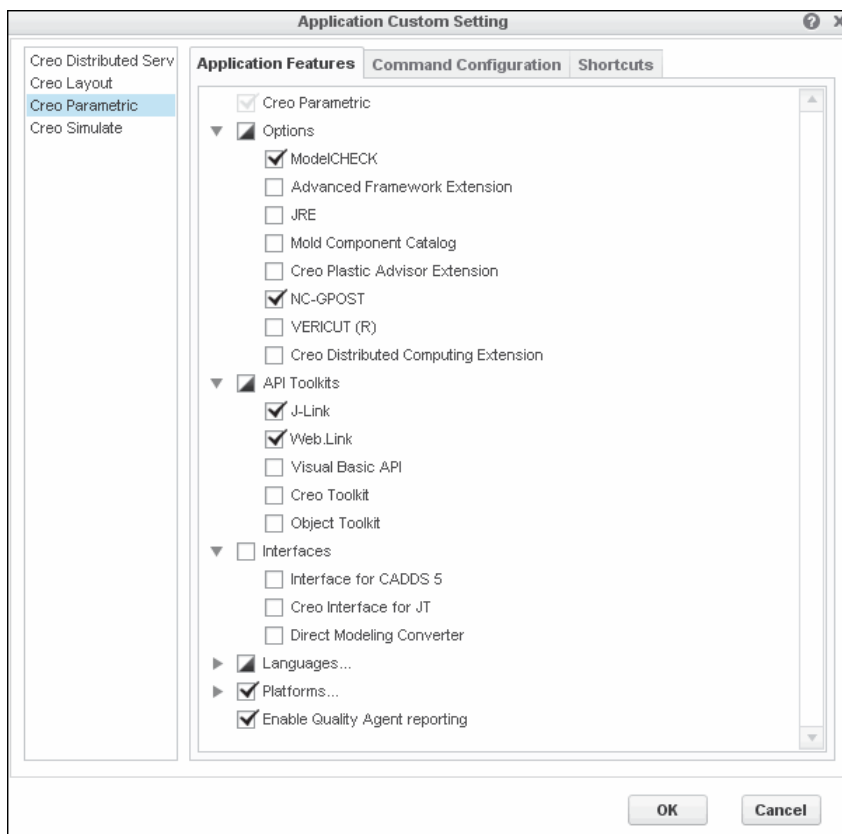
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This chapter explains how to customize an installation of the Creo applications. This information also applies to Creo Distributed Services Manager. Some of the application features in this chapter are specific to individual Creo applications.

# Customizing an Installation

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

1. Perform steps 1 through 8 as described in the section [Installing the Creo Applications: Simple Installation Process](#) on page 30.
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 37.
  - To modify license configurations using the **Command Configuration** tabbed page, see [Modifying License Configurations](#) on page 39.
  - To configure Windows shortcut preferences using the **Shortcuts** tabbed page, see [Configuring Windows Shortcuts](#) on page 41.
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 Creo application. See [Completing the Installation and Starting the Creo Applications on page 43](#) 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.

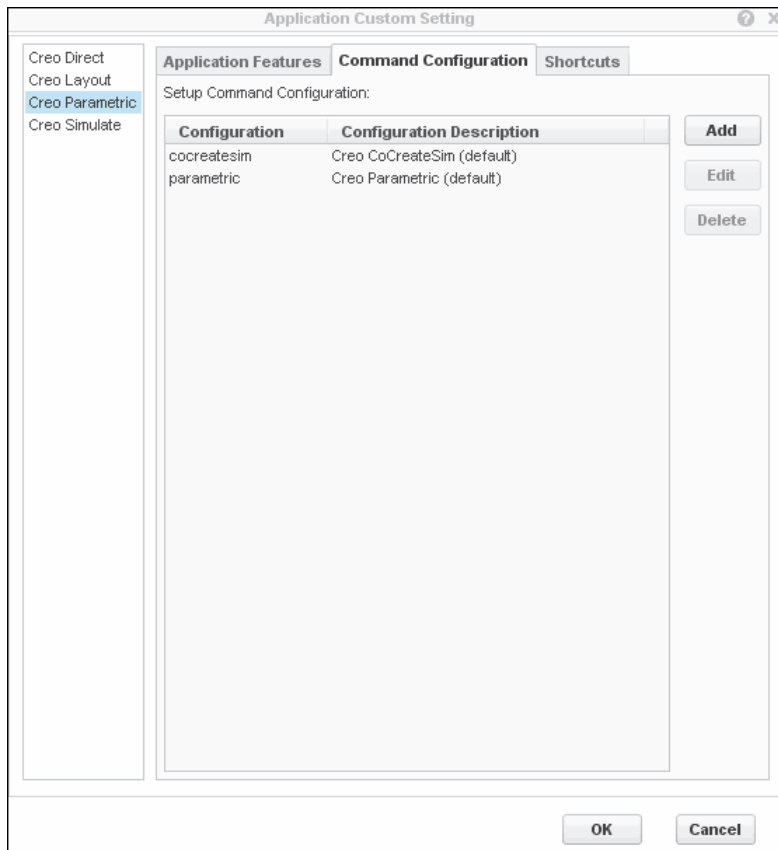
Component	Application	Description
<application_name>	All the Creo applications and Creo Distributed Services Manager	Installs the Creo application and its components. By default, the components are installed at <creo_loadpoint>\<app_name> and the startup scripts are installed at <creo_loadpoint>\<app_name>\bin.
<b>Options</b>	Creo Parametric	Installs additional components such as ModelCHECK, JRE, Mold Component Catalog, Creo Plastic Advisor, NC-GPOST, VERICUT, and Creo Distributed Batch with Creo Parametric.
<b>API Toolkits</b>	Creo Parametric and Creo Distributed Services Manager	Installs the files necessary to run the Application Program Interface toolkits.

Component	Application	Description
<b>Interfaces</b>	Creo Parametric	Installs the external interfaces.  <b>Note</b> <i>To install <b>Interface for CADD5</b>, you must also select <b>MKS Platform Components 9.x</b> for installation on the <b>Application Selection</b> screen. See <a href="#">Installing MKS Platform Components on page 137</a> for more information.</i>
<b>Platforms</b>	All the Creo applications and Creo Distributed Services Manager	Selects your machine's architecture automatically as the default platform for installing the components.
<b>Languages</b>	All the Creo applications and Creo Distributed Services Manager	Selects the languages for the installation. The software is installed in English by default and cannot be disabled.
<b>Enable Quality Agent Reporting</b>	All the Creo applications	If enabled, collects information related to the Creo applications and other system-related information and sends it to PTC. See <a href="#">Installing Quality Agent on page 129</a> for more information on Quality Agent.
<b>Verification Models</b>	Creo Simulate	Installs the verification models as described in the Simulation Verification Guide Help.
<b>Creo Distributed Batch</b>	Creo Simulate	Installs this component to run distributed computing tasks that include import/export, printing operations, and studies in case of Creo Simulate.
<b>Direct Modeling Converter</b>	Creo Parametric, Creo Simulate, Creo Direct	Installs files that are required to convert and open Creo Elements/Direct files.

Component	Application	Description
<b>Legacy Independent Mode Help Files</b>	Creo Simulate	Installs Help for the legacy independent mode Creo Simulate in the product installation directory.

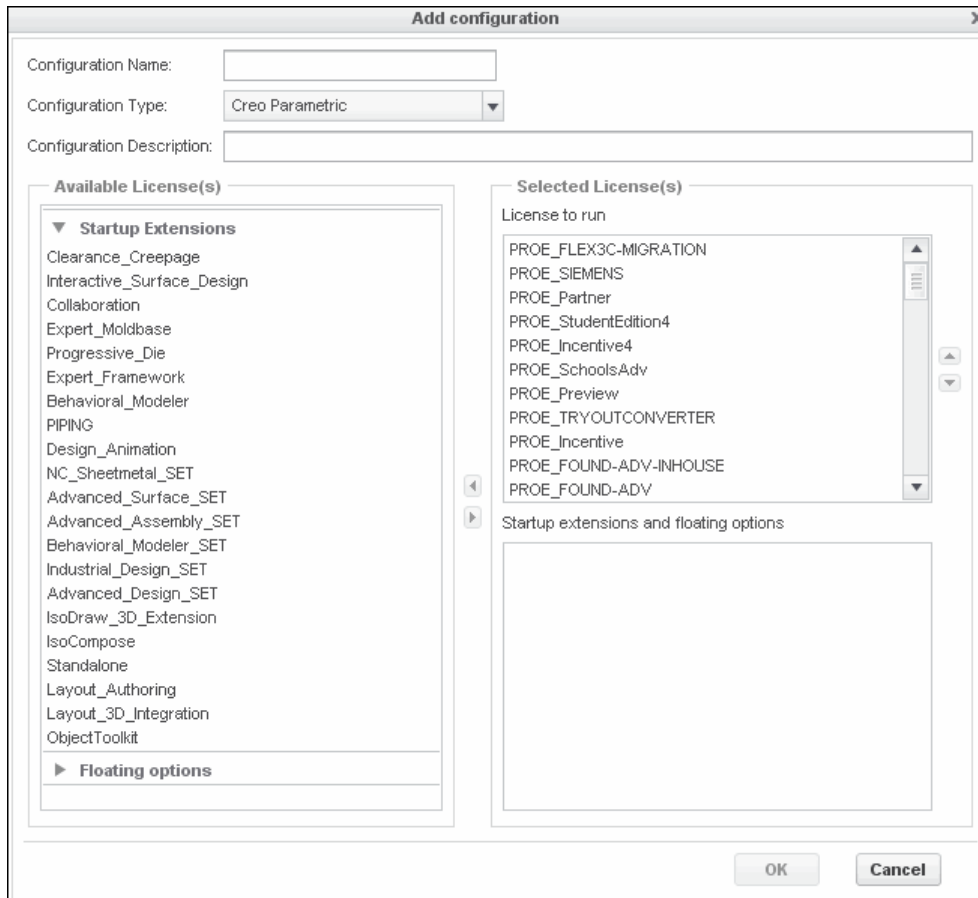
## Modifying License Configurations

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 Creo Parametric. Similar screens appear for the other Creo applications.



Perform the following steps:

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



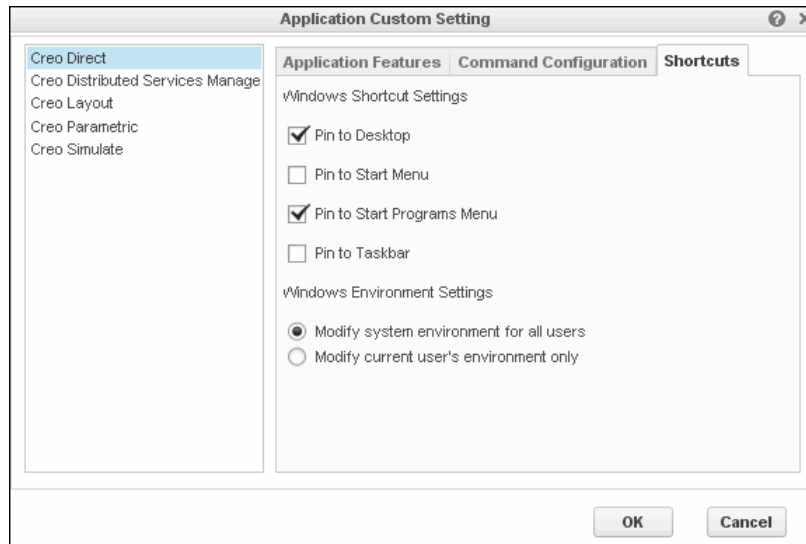
- a. In the **Configuration Name** box specify a file name (\*.psf) for the license configuration information. Use alphanumeric characters without spaces to define the configuration name.
- b. From the **Configuration Type** list, select a configuration type.
- c. In the **Configuration Description** box, type a description for each of the license configurations that you have defined. This information is displayed on the startup of 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**.

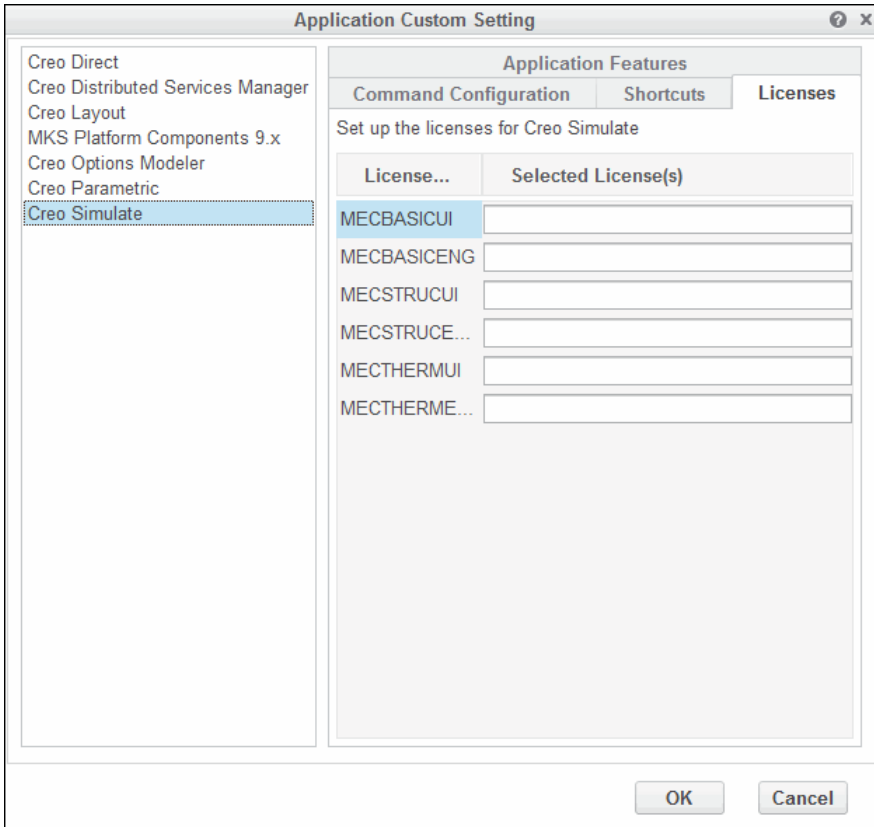
## Configuring Windows Shortcuts

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



## Setting Licenses for Creo Simulate

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



Using this tabbed page you can configure licenses for 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.

# 6

## Completing the Installation and Starting the Creo Applications

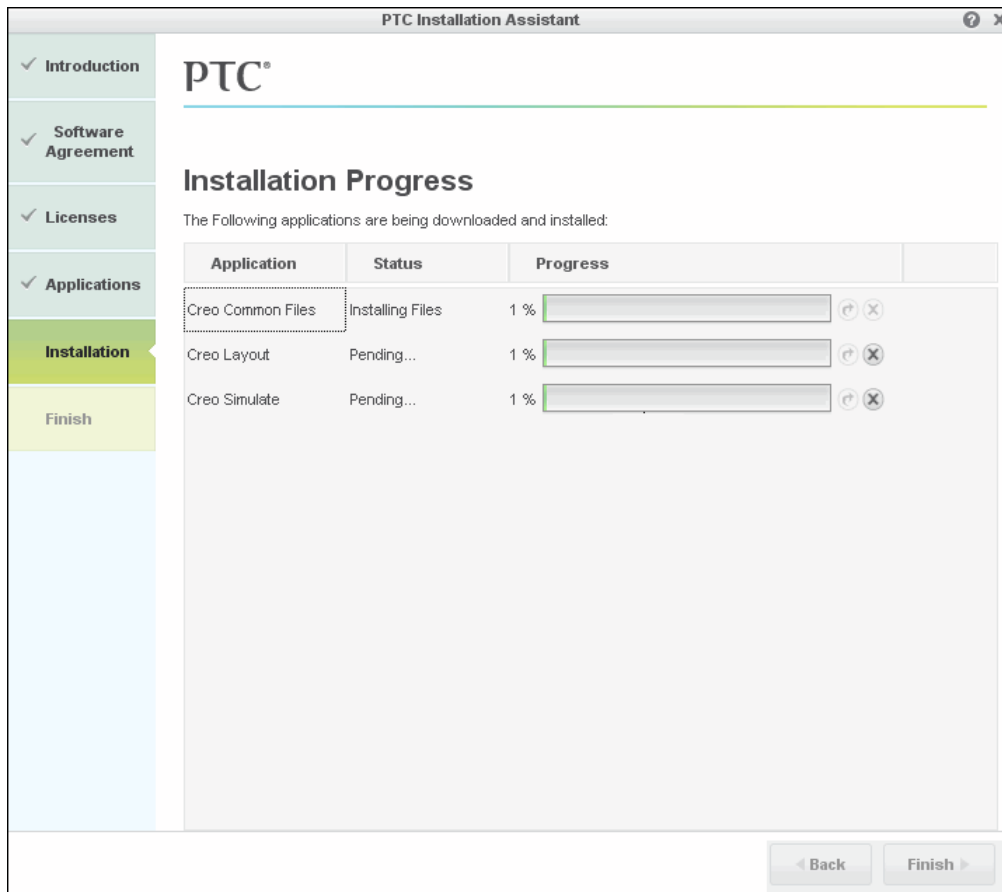
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This chapter explains how to complete an installation of a Creo application. This information is common to Creo Distributed Services Manager. Information on how to start the Creo applications and configure their startup is also given. Proceed to the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



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.
- Click  to restart an installation.
- Click **Back** and review information.
- 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 Creo Applications in Silent Mode

You can install and uninstall all the Creo applications included in this guide in silent mode using command-line arguments. See [Installing the Creo Applications in Silent Mode on page 139](#) for details.

## Starting the Creo Applications

After your license management and product software have been installed, start the 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 Creo 2.0 application that you want to start.

## Configuring the Creo Application Startup

When you configure the startup of a 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 Creo applications. A configuration file is created when you complete the installation of a Creo application.

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

You can run `<creo_loadpoint>\<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 Creo Parametric. The bold text following the `// USER - PSF` section is the user-defined information for starting 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.

## 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 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 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:

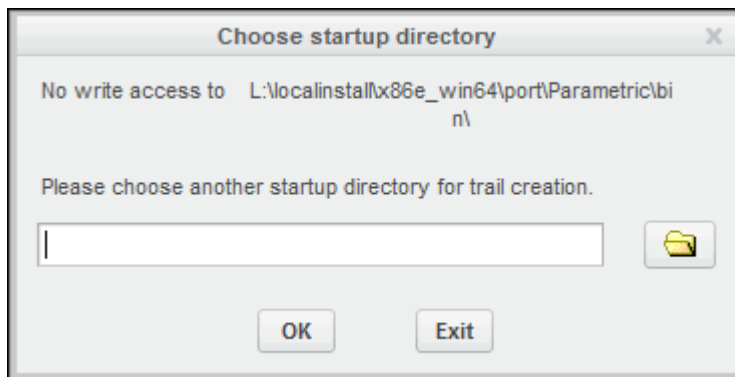
1. `<creo_loadpoint>\Common Files\<datecode>\text` (`creo_loadpoint` is the Creo application installation directory)—Your system administrator may have put configuration files in this location to support company standards for windows configuration settings, formats, and libraries. Any user starting a 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 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 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 the Creo Applications

You cannot start a Creo application from a directory without write permissions unless you specify an alternative directory to store the trail files. If you start a 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 Creo Platform Agent

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

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

See [Installing and Uninstalling Client-Side Components in Silent Mode](#) on page 150 for instructions on performing a silent installation of Creo Platform Agent.

## Configuring a Creo Parametric Installation

After you have installed Creo Parametric, you can perform the following operations either by setting configuration options before starting 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 Creo Parametric startup
- Install optional utilities like Thumbnail Viewer and Creo View Express at Creo Parametric startup
- Allocate memory for running Creo Parametric on 32-bit platforms by setting a configuration option before or after starting Creo Parametric
- Generate a traceback log file by setting a configuration option before or after starting Creo Parametric

## Changing the Units of Measure for Creo Parametric

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

Configuration Option	English	Metric
drawing_setup_file	<creo_loadpoint>\Common Files\<datecode>\text\prodetail.dtl	<creo_loadpoint>\Common Files\<datecode>\text\iso.dtl



<b>Configuration Option</b>	<b>English</b>	<b>Metric</b>
format_setup_file	<creo_loadpoint>\Common Files\<datecode>\text\prodetail.dtl	<creo_loadpoint>\Common Files\<datecode>\text\iso.dtl
pro_unit_length	unit_inch	unit_mm
pro_unit_mass	unit_pound	unit_kilogram
template_designasm	<creo_loadpoint>\Common Files\<datecode>\templates\inlbs_asm_design.asm	<creo_loadpoint>\Common Files\<datecode>\templates\mmks_asm_design.asm
template_drawing	<creo_loadpoint>\Common Files\<datecode>\templates\c_drawing.drw	<creo_loadpoint>\Common Files\<datecode>\templates\a3_drawing.drw
template_sheetmetalpart	<creo_loadpoint>\Common Files\<datecode>\templates\inlbs_part_sheetmetal.prt	<creo_loadpoint>\Common Files\<datecode>\templates\mmks_part_sheetmetal.prt
template_solidpart	<creo_loadpoint>\Common Files\<datecode>\templates\inlbs_part_solid.prt	<creo_loadpoint>\Common Files\<datecode>\templates\mmks_part_solid.prt
today's_date_note_format	%Mmm-%dd-%yy	%dd-%Mmm-%yy
tolerance_standard	ansi	iso
tol mode	nominal	nominal
weld_ui_standard	ansi	iso

## Check for Optional Utilities

Messages appear at Creo Parametric startup for the installation of optional utilities like Creo View Express or Thumbnail viewer. Click one of the following commands:

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

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

When working with large models or a large data set in 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 your work under some situations.

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

Consider the following guidelines when setting your memory buffer:

- The `proe_memory_buffer_size` configuration option is processed at Creo Parametric startup. If you change its value in the current session, exit the 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 `proe_memory_buffer_size` configuration option, it is recommended that you increase the system settings for the size of the virtual memory paging file.

### Using a 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 Creo Parametric, and restart the session.
- Click **OK** to continue. In this case Creo Parametric may exit as soon as the preallocated memory is exhausted.

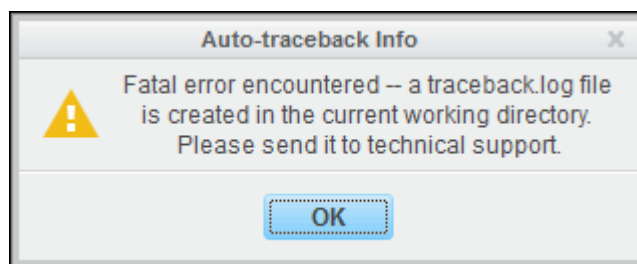
For operations you cannot cancel, a message appears to indicate that the system is running short of memory. This may cause the session to terminate. Click **OK** to continue. After you complete the operation, save your work, exit the Creo application, and restart the session.

## Using a 3GB Switch for Memory Allocation

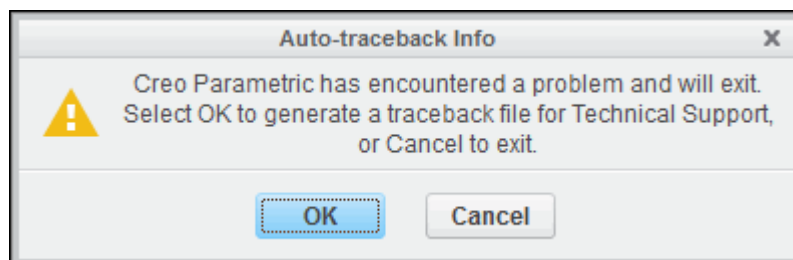
On Windows XP, you can use the 3GB startup switch provided by Microsoft to improve memory allocation and to increase the virtual memory address space. Refer to <https://www.ptc.com/appserver/cs/view/solution.jsp?&n=111330> for more information.

## Generating a Traceback Log File for Creo Parametric

In the event of a premature exit, 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 Creo Parametric:



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



---

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

### **Note**

If you start 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 CADD5 Models in Creo Parametric**

You can import and export 3D CADD5 parts and assemblies within Creo Parametric by installing the CADD5 interface during the installation of Creo Parametric. To do so, select **Interface for CADD5** 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 **Interface for CADD5** with Creo Parametric, a configuration file `config_cadd5.pro` is automatically saved in the `<parametric_install_path>\text` folder. This file contains configuration options whose settings determine how the CADD5 interface works with Creo Parametric. A sample `config_cadd5.pro` file follows:

```
atb_ident_cadd5_files yes
cadd5_import_layer yes
intf_cadd5_import_make_solid yes
intf_cadd5_version 14
! extend_cvpath is similar to config-option search_path for Creo Parametric.
! It allows to find user's CADD5 objects on import...
! extend_cvpath
```

Check the `config_cadd5.pro` file for the above configuration options. If required, copy the configuration options to your local `config.pro` file and update their values.

- Check whether the components listed below are installed:
  - **Interface for CADD5**
  - MKS platform components. See [Installing MKS Platform Components on page 137](#) for installation instructions.
  - PTC Portmapper that is installed as a part of the 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 Creo Applications

You can remove an installation of a 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 Creo application is removed.

### **Note**

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



# 7

## Installing and Using Creo Help

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

---


## Overview of the Help Center

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

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

## Accessing Help within a Creo Application

Each Creo application launches its own Help window. The language setting of the browser on your host computer determines the language in which the Help Center opens. You can also configure the Help Center to open it in a different language. 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 depending on where you have installed it:

- PTC.com—The default if you have not installed the Help Center. Provide your user credentials and log in at PTC.com the first time you access Help.
- Local host computer—The default if you have installed the Help Center but did not specify an alternate location for accessing Help.

### **Note**

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

## Choosing Your Installation Location for the Help Center

Using the Help DVD, you can install Help Center at a location of your choice as listed below:

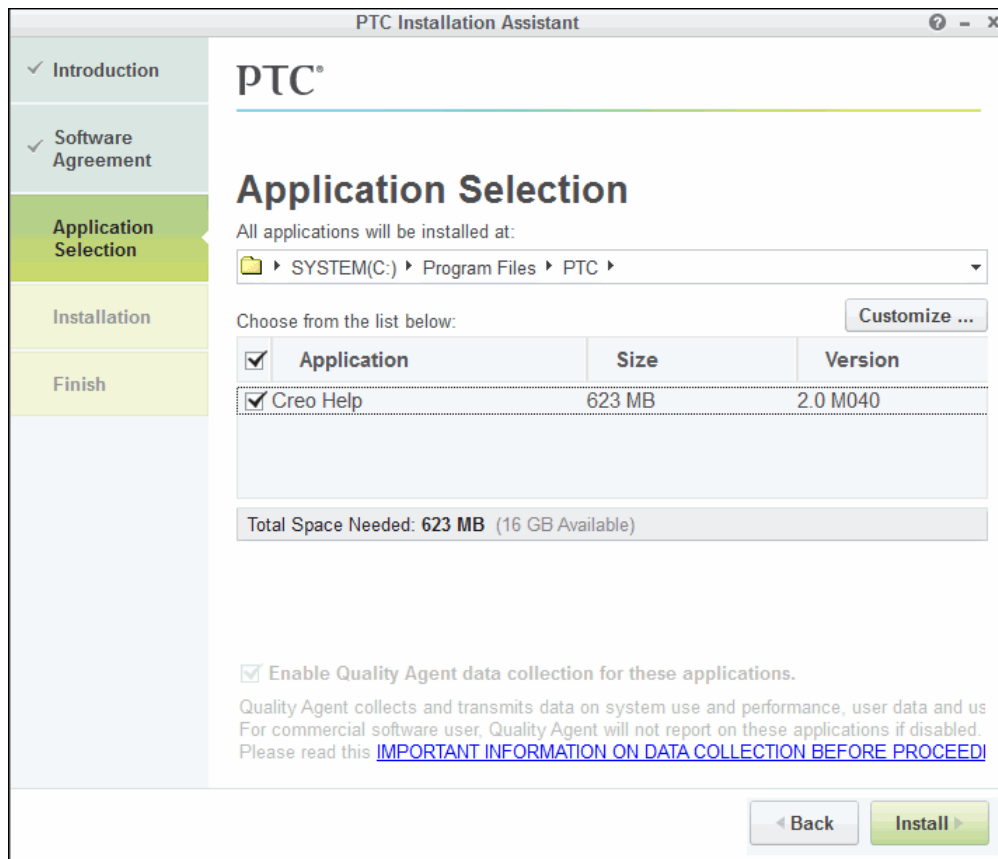
- A local host computer where the Creo applications are installed.
- A network drive. This location is helpful for administering multiple clients on a network.
- A Web server.




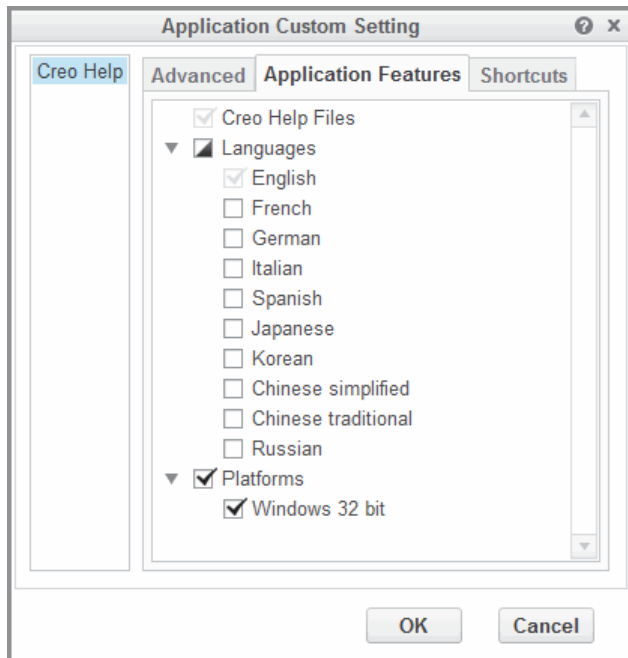
# Installing the Help Center

Using the Help DVD in your shipment, proceed with the installation as described below. 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 58](#) to enable the Help Center from the 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. PTC Installation Assistant starts.
2. Perform steps 1 through 7 as described in [Installing the Creo Applications: Simple Installation Process on page 30](#).
3. Click the **Creo Help** check box on the **Application Selection** screen.



4. Keep the default folder, for example, `C:\Program Files\PTC`, or click  and specify the full path to an alternative folder.
5. Click **Install** to install the Help Center without any customization.  
Alternatively, click **Customize** to open the **Application Custom Setting** dialog box.



- a. On the **Application Features** tab, select the languages for installing the Help Center.
  - b. On the **Shortcuts** tab, specify the shortcut preferences for the Creo Help startup commands and other Windows preferences. See [Configuring Windows Shortcuts on page 41](#) for information on how to perform these operations.
  - c. On the **Advanced** tab, specify the Web server location for installing the Help Center. See [Installing the Help Center on a Web Server on page 59](#) for information on how to perform this operation.
6. Click **OK**.
  7. 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.
  8. Click **Finish**.

## Enabling the Help Center from a Network Drive

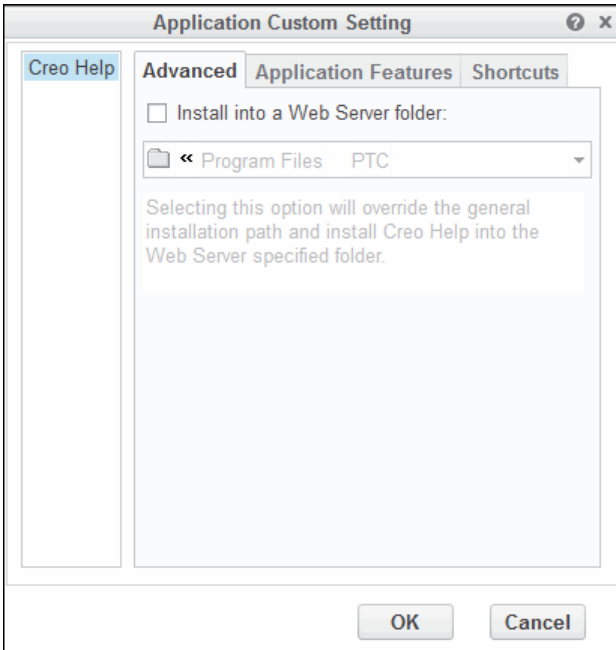
To enable the Help Center installed on a network drive, you must configure the Creo application as explained in the following example:

- 
1. If you have installed the Help Center on computer machine03d at C:\Program Files\PTC\Creo 2.0\Help, share the folder C:\Program Files\PTC\Creo 2.0 as \\machine03d\Creo 2.0.
  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.
  3. Set the PTC\_HELP\_CENTER\_URL environment variable as follows:
    - 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:
      - **Variable name**—PTC\_HELP\_CENTER\_URL
      - **Variable value**—Q:\Help\help\creo\_help
    - e. Click **OK** and again click **OK** on the **System Properties** dialog box.The Help Center process starts on the client computer.
  4. Start the Creo application.

## Installing the Help Center on a Web Server

You can install the Help Center on any servlet container, application server, or a Web server that supports the JSP 2.0 and Servlet 2.4 standards. If you are using Websphere, you can use its interface to install the Help Center. If you are using a Tomcat or a Jetty servlet container, install the Help Center to the document root directory of the servlet container as follows:

1. Click **Customize** on the **Application Selection** screen to open the **Application Custom Setting** dialog box. The **Application Features** tab opens by default.
2. Click the **Advanced** tab. The following screen appears:



3. Click the **Install into a Web Server folder** check box and specify the document root directory of the servlet container on the Web server to install the Help Center.
4. Click **OK** and click **Install** on the **Application Selection** screen.
5. To distribute the Help Center files from the <servlet document root>\creo\_help directory, set the PTC\_HELP\_CENTER\_URL environment variable as follows:
  - a. Click **Start** ► **Control Panel** ► **System**. A dialog box opens.
  - b. Click **Advanced system settings**. The **System Properties** dialog box opens.
  - c. Click **Environment Variables**.
  - d. Click **New** under the user variables area. The **New User Variable** dialog box opens.
  - e. Specify the environment variable and its value in the boxes listed below:
    - **Variable name**—PTC\_HELP\_CENTER\_URL
    - **Variable value**—<Help\_Center URL>

For example, for a Help Center installed on a Web server that has Tomcat servlet container configured on port number 8080, the Help Center URL is `http://<server_name>:8080/creo_help/`.
  - f. Click **OK** and again click **OK** on the **System Properties** dialog box.

---

## Viewing the Help Center Independent of a Creo Application

To view the Help Center on a host computer independent of a Creo application, use the following command:

```
<creo_help_install_path>\help\creo_help\creo_help.exe
```

A Help Center opens in its own browser. Context-sensitive Help is not supported.

## Uninstalling the Help Center

You can remove the Help Center from the local host computer independent of a 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 **Creo Help <release><datecode>**.
3. Click **Uninstall**. A confirmation dialog box opens.
4. Click **Yes** to remove the Help Center.

### **Note**

*If you have uninstalled the Help Center from the host computer or if you have not set the `PTC_HELP_CENTER_URL` environment variable, the Help Center is accessed from `PTC.com` within a Creo application.*



# 8

## Updating an Installation

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This chapter explains how to update the Creo applications installed on your system. Use the same procedure to update 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 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 Creo Applications on page 21](#) 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 24](#) for details.

5. 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 Creo Applications

After you install 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>\Common`



---

Files\`<datecode>` 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>\Common Files` folder. This folder contains all the files for the Maintenance Release. You can choose to retain or delete the previous `<datecode>` folder under the `<creo_loadpoint>\Common Files` folder. When you install a new Maintenance Release, all the Creo applications that were previously sharing the same `<datecode>` folder are automatically reconfigured to use the files in the new `<datecode>` folder.

## Customizing an Existing Installation of a Creo Application

Customizing an existing installation of a 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 24](#) 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 37](#) for details.
  - Add, edit, or delete a license configuration using the **Command Configuration** tabbed page. See [Modifying License Configurations on page 39](#) for details.

- 
- Configure the Windows shortcut preferences and path settings using the **Shortcut** tabbed page. See [Configuring Windows Shortcuts on page 41](#) 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 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 or add 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.

# A

## License Management Software

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

---

## Overview of PTC Licensing

PTC software including PTC optional applications must be licensed for use. Licensing authorizes the installed software to run. 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 77](#) for details.

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

## Simplified License Management Process

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 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](http://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 Creo applications as well as products of the Creo Parametric family and is available in Pro/ENGINEER 2001 datecode 2001440 and later.
- On the [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

*FlexNet Publisher Version 11.10.1 is shipped with Creo 2.0. However, Creo 2.0 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.
  - 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 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 \
LO=(0,3,6,7,10,17,32,34,40,45,48,51,55,61,62, \
66,69,71,73,77,97,106,108,115,126,127,128,133,135,137, \
158,163,188,191,210)" \
SUPERSEDE vendor_info="VIVER=1.0 EXTERNAL_NAME=" \
ISSUER=PTC ISSUED=01-sep-2006 NOTICE="PTC_customer" \
SN=SCN12344,SCN765431 SIGN2="169C A28A \
E97F E96E 0A3E 563B FDEB 4510 829E 4BF4 25D3 2394 \
0444 2FD4 6C23 0168 A8A5 AEBE 54B0 1FF6 B79B DC75 \
2014 A278 33CC 1B90 8647 6A12 F4D6 45BF"
INCREMENT 10108 ptc_d 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-to-run. 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 Creo application name that you are configuring. For example, `CREOPMA_FEATURE_NAME` identifies the licenses to grab when running Creo Parametric. Similarly, `CREODMA_FEATURE_NAME` refers to 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 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 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 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: 10.8.10
```

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

\* = License Extensions - Available only on startup  
^ = Borrowed License

## Ptcstatus Verbose Mode

```
Displaying status for license file: 7788@static
License Server: static
ptc_d FlexNet Publisher version: 10.8.10
```

License	In Use	Free	Version	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](http://www.flexerasoftware.com).

### Note

*FlexNet Publisher Version 11.10.1 is shipped with Creo 2.0. However, Creo 2.0 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—Using FlexNet Publisher 8.6 or later, you can borrow licenses from a license server and run a licensed application on a remote client without being connected to the license server.

## Downward License Compatibility Using FlexNet Publisher

FlexNet Publisher license servers have downward compatibility with PTC applications using FLEXlm license servers. For example, a Creo Parametric 2.0 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 Creo, you must install FlexNet Publisher version 10.8.x or later. FlexNet Publisher Version 11.10.1 is shipped with Creo 2.0.*

## Running FlexNet Publisher with Other Applications

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

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

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

## Understanding Timeout Parameters

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

You can reconfigure the `TIMEOUTALL` parameter within a specified range, as described in the next section, [Changing the Inactivity Timeout Parameter on page 76](#).

<b>Timeout Parameter</b>	<b>Value</b>	<b>Description</b>
Inactivity timeout (TIMEOUTALL)	120 minutes (default)	<p>This parameter prevents a license from remaining idle. If the license client is inactive for a specific period, the license can be reclaimed by the license server and used by another license client. For the Creo applications, the inactivity timeout default is 120 minutes.</p> <p>Activity is measured as active menu selections.</p> <p>You can change the default of 120 minutes so that the parameter ranges from 20 minutes (minimum value) to infinity (maximum value).</p>
License refresh	1 minute	<p>A license refresh occurs at intervals of 1 minute. When you select a command after such an interval, the license client communicates with the license server. The license client and the license server both must be working. If the license server is not found, the license client loses its license to run. If the license client is not found, the server reclaims the license for use by another license client.</p>
Validation retries	None	<p>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:</p> <p>Click <b>Retry</b> to request a license from an active license server.</p> <p>Click <b>Save File(s)</b>.</p> <p>Click <b>Exit</b> to close the application without saving the file or files.</p>
Startup	10 seconds	<p>Upon starting a session, the license client requests a license and has 10 seconds in which to have the request validated.</p>

---

## Changing the Inactivity Timeout Parameter

You can set the `TIMEOUTALL` parameter that determines how long the system allows a license to remain idle before it is reclaimed by the license server. To change the default inactivity timeout parameter, you must update the FlexNet Publisher option file, `ptc.opt` in `<ptc License Server loadpoint>/FLEXnet Publisher/licensing`. The default is 120 minutes. Edit the default parameter `TIMEOUTALL 7200` by changing 7200 seconds (120 minutes) to another value in seconds.

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

## Suppressing the Regained Network License Dialog Box for 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 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 ladmin-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 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>`\\`<app_name>`\bin directory
- Specifying the command-line argument to borrow licenses as instructed in the  
    *FlexNet Publisher License Administration Guide*.

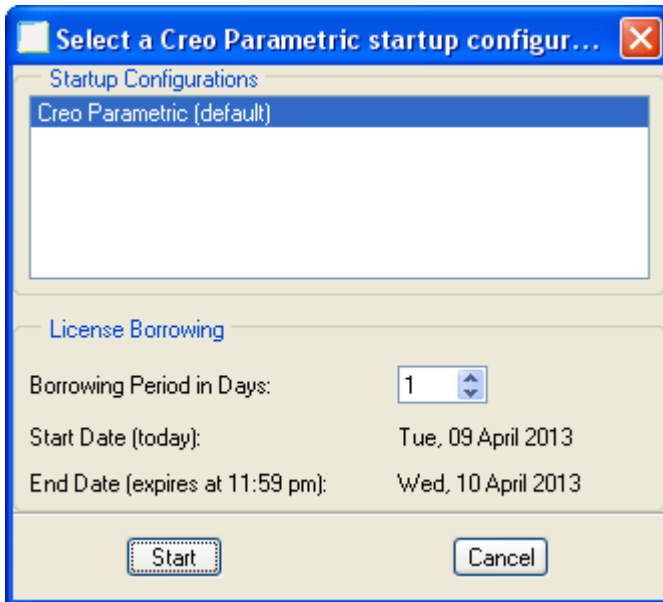
### Note

*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 79 for more information.*

## Using the `<creo_app>_borrow.bat` File

Use the `<creo_app>_borrow.bat` file with the Creo application to initiate license borrowing as follows:

1. Browse to the `\<creo_loadpoint>\<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 Creo application in the borrow configuration mode.
5. After starting the 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 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 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 ladmin-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 lmutil lmborrow Utility

For an early return of a borrowed license, you can use the `lmborrow` subfunction with the `lmutil` Flexnet utility. The `lmutil` utility resides at `C:\Program Files\PTC\Creo 2.0\Common Files\M020\x86e_win64\obj`.

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 <code>lmutil lmstat</code> 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 <code>port@hostname</code> value for the server is sufficient.  <b>Note</b> <i>You can avoid using this argument by setting the <code>LM_LICENSE_FILE</code> variable. Set the variable as <code>LM_LICENSE_FILE=port@hostname</code> to point to the server used for borrowing the license.</i>
-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 <code>lmborrow -status</code> command for a list of borrowed license features.



# B

## ladmin License Server Manager

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

---

## Overview of lmadmin as a GUI-Client

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

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

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

## Differences between lmgrd and lmadmin

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

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

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

## Installing PTC License Server Based on lmadmin

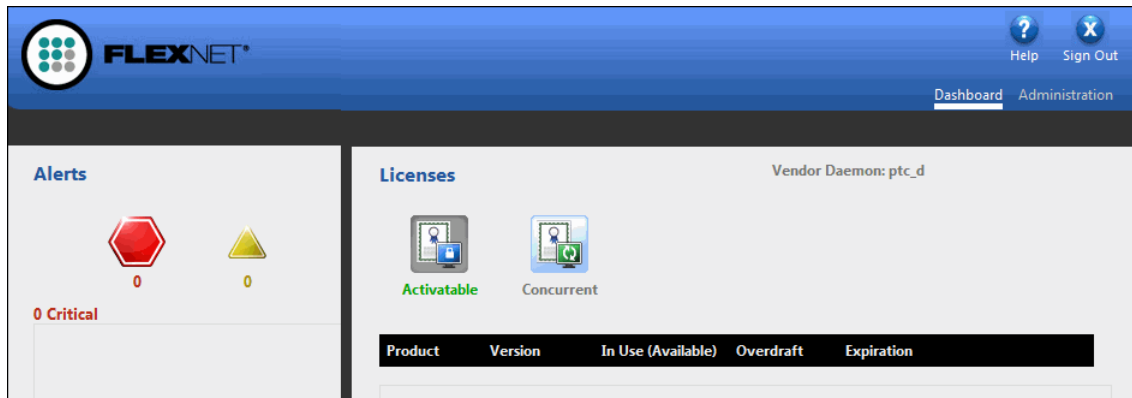
Follow the instructions given in the section [Generating or Updating Licenses on page 24](#) to install PTC License Server based on `lmadmin`. After you install the license server, you can configure the license server administration settings using the FLEXnet License Administrator Web interface.

---

## Working with the FLEXnet License Administrator Web Interface

The `lmadmin`-based license server uses the FLEXnet License Administrator Web interface. This interface replaces the `lmtools` utility used by the `lmgrd`-based license server. You can launch the interface by clicking **Yes** at the end of the PTC License Server installation. Alternatively, from the Windows Start menu click **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.

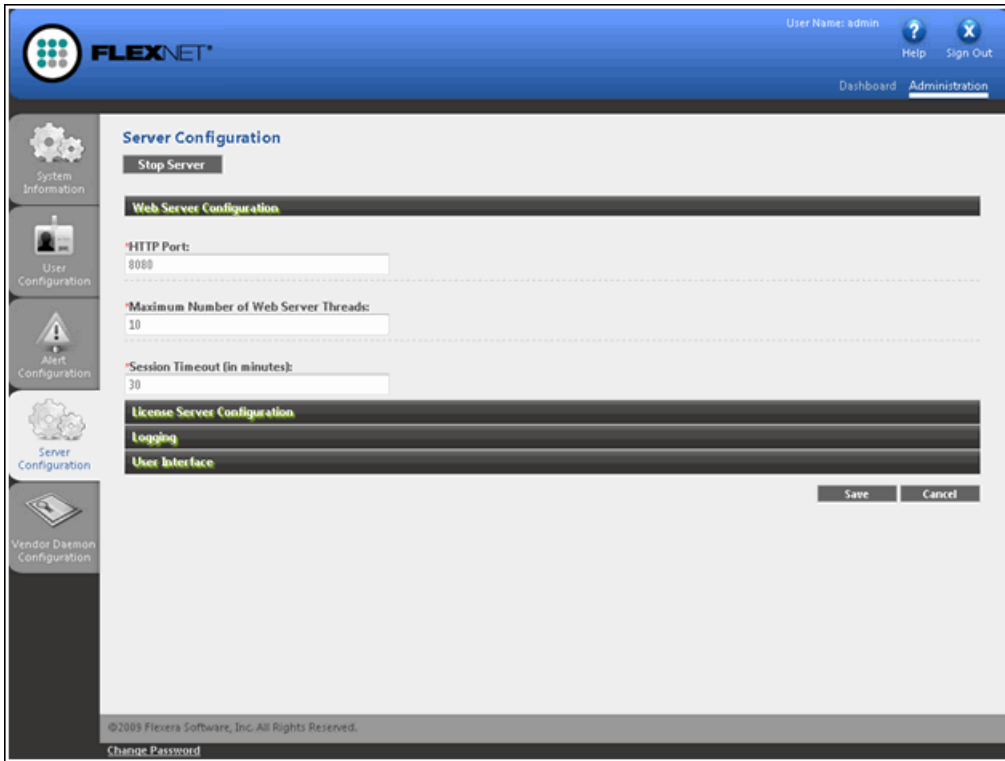
The FLEXnet License Administrator Web interface has two main pages: the **Dashboard** and the **Administration** pages. Click the **Help** button for information on the FLEXnet License Administrator interface elements.



- **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.

The screenshot shows the FLEXNET Administration interface. At the top, the user is logged in as 'admin'. The navigation sidebar on the left includes: System Information, User Configuration, Alert Configuration, Server Configuration, and Vendor Daemon Configuration. The main content area is titled 'Vendor Daemons' and features an 'Import License' button. Below this is a table with the following columns: Name, Status, FLEXnet Publisher Version, and Port. A single entry is listed:

Name	Status	FLEXnet Publisher Version	Port
ptc_d	RUNNING	10.8	65222

Below the table, there are links for 'Administer' and 'Delete' for the 'ptc\_d' daemon. At the bottom of the interface, there is a copyright notice: '©2009 Flexera Software, Inc. All Rights Reserved.' and a 'Change Password' link.

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



# C

## System Administration Information

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



---

## Temporary Directories

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 Creo application startup window before you run the application. This environment variable is `TMP`.

## System Management Utilities

This section briefly describes the utilities for Creo Parametric, Creo Simulate, Creo Direct, 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*.

### Creo Utilities

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

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

---

## FlexNet Publisher Utilities

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

Command	Function
lmttools	Runs the PTC License Management tools utility <code>lmttools.bat</code> . 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 <b>Start ▶ All Programs ▶ PTC License Management tools</b> starts the Lmtools graphical user interface.
lmutil	Provides optional arguments to check the server and licensing status. Refer to the <i>FlexNet Publisher License Administration Guide</i> .
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 Publisher License Administration Guide</i> .

### Lmtools Utility

With FLEXlm 7.0 and later, a `lmttools.bat` program is added to the license server installation directory. The `lmttools.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 [www.ptc.com/WCMS/files/135225/en/creo2\\_hw\\_notes.pdf](http://www.ptc.com/WCMS/files/135225/en/creo2_hw_notes.pdf).

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

---

## 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**

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

## Using the ModelCHECK Metrics Tool with Creo Parametric

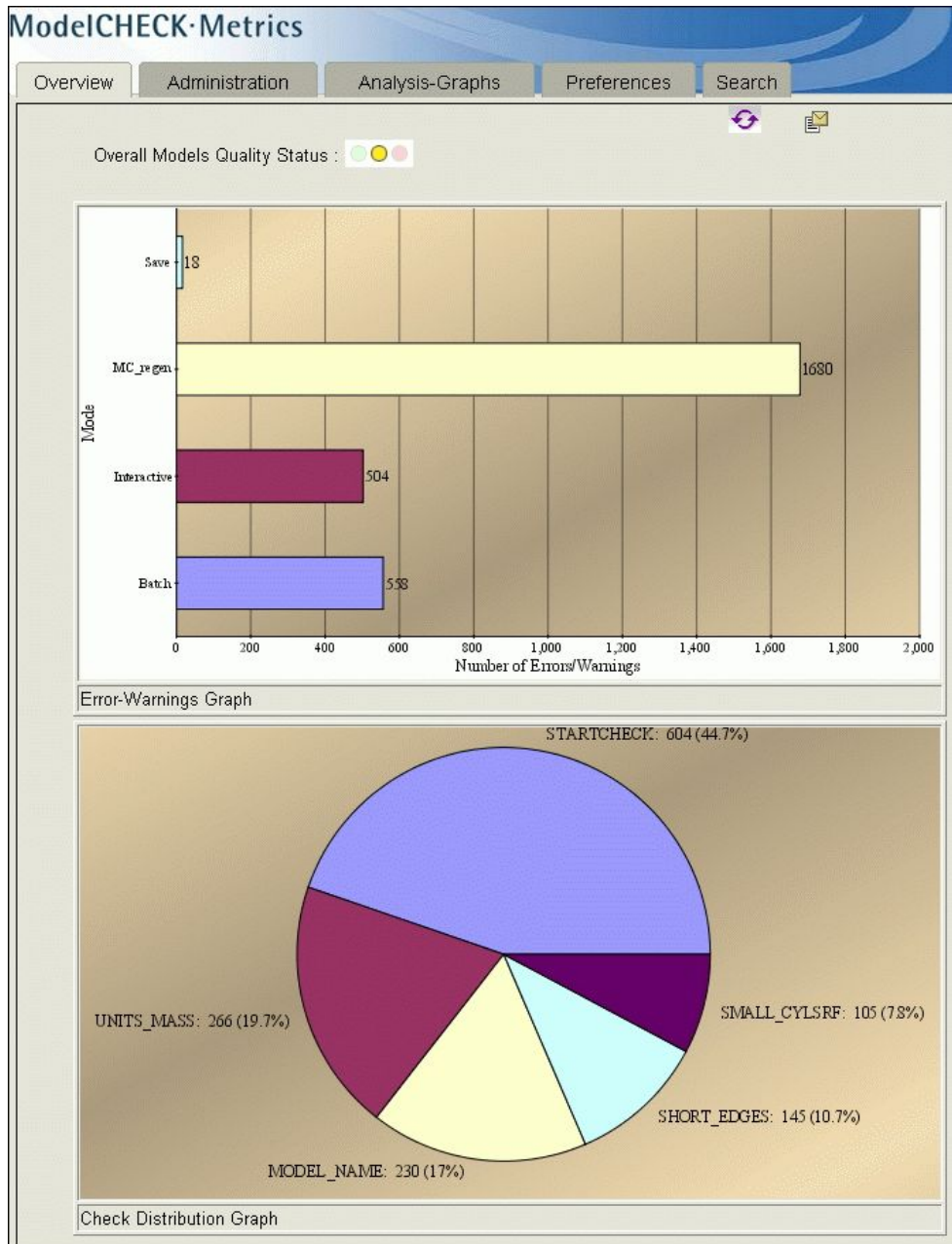
The ModelCHECK Metrics Tool is a web-based utility for tracking data quality, standards, and best practices relevant to Creo Parametric data. This tool uses the metric file output from ModelCHECK to calculate quality trends and enables you to represent the ModelCHECK analysis results graphically. Although specific licenses are not required, an administrator needs a 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 Creo 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 [www.ptc.com/WCMS/files/135225/en/creo2\\_hw\\_notes.pdf](http://www.ptc.com/WCMS/files/135225/en/creo2_hw_notes.pdf) for supported platforms. You are not required to run ModelCHECK to access the tool. You must

---

have a Web server to access the Metrics Tool information. After you install and start the Metrics Tool, you can view the ModelCHECK results using Mozilla 1.6 and later or Internet Explorer 6.0.

## How to Install the ModelCHECK Metrics Tool

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

- Install `j2sdk-1_4_2_xx` through a local installation or download it from [www.oracle.com/technetwork/java/index.html](http://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](http://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](http://www.apache.org).

## How to Start the Tomcat Server and Deploy the Metrics Tool

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

```
<?xml version='1.0' encoding='utf-8'?>
<tomcat-users>
<role rolename="tomcat"/>
<role rolename="role1"/>
<role rolename="manager"/>
<role rolename="admin"/>
<user username="tomcat" password="tomcat" roles="tomcat"/>
<user username="both" password="tomcat" roles="tomcat,role1"/>
<user username="role1" password="tomcat" roles="role1"/>
<user username="admin" password="admin" roles="admin,manager"/>
</tomcat-users>
```

---

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

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

**Note**

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

5. Ensure that you have created the context configuration file (`new_admin.xml`) to enable the deployment of the Metrics Tool. See [Context Configuration File on page 94](#) for details.
6. In **Tomcat Web Application Manager** window, provide the path to the context configuration file (`new_admin.xml`) and the WAR file (`new_admin.war`) under the **Deploy** area.
7. Click **Deploy** to start the Metrics Tool.
8. Open a browser. Specify the name of the server on which the Metrics Tool is deployed along with the port and `new_admin` as the name of the tool.

**Note**

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

### Context Configuration File

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

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

Parameter Name	Description
<code>dbname</code>	Name of the database
<code>dbtype</code>	Type of the database, like Oracle or MSAccess
<code>adminserver</code>	Machine name of a server for the Metrics Tool
<code>Adminport</code>	The port number type of the server for the Metrics Tool

Parameter Name	Description
Imagedir	Path to the image directory
ImagedirLink	The 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"
type="javax.sql.DataSource" />
<ResourceParams name="jdbc/mcadmin">
<parameter>
<name>maxWait</name>
<value>50000</value>
</parameter>
<parameter>
<name>maxActive</name>
<value>100</value>
</parameter>
<parameter>
<name>password</name>
<value>yourpass</value>
</parameter>
<parameter>
<name>url</name>
<value>jdbc:oracle:thin:@machine1:1521:mcadmin</value>
</parameter>
<parameter>
<name>driverClassName</name>
<value>oracle.jdbc.driver.OracleDriver</value>
</parameter>
<parameter>
<name>maxIdle</name>
<value>100</value>
</parameter>
<parameter>
<name>username</name>
<value>yourpass</value>
</parameter>
</ResourceParams>
<ResourceLink name="jdbc/mcadmin" global="mcadmin"/>
<Manager
className="org.apache.catalina.session.PersistentManager"
debug="0" saveOnRestart="false"
```

---

```

maxActiveSessions="-1"
minIdleSwap="-1" maxIdleSwap="-1" maxIdleBackup="-1">
<Store className="org.apache.catalina.session.FileStore" />
</Manager>
<Parameter name="dbname" value="madmin" />
<Parameter name="dbtype" value="oracle" />
<Parameter name="adminserver" value="machine1" />
<Parameter name="adminport" value="80" />
<Parameter name="imagedir" value="d:\projects\madminwork\admingraphs" />
<Parameter name="imagedirLink" value="admingraphs"/>
</Context>

```

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

## Database for the Metrics Tool

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

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

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

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

```

CREATE TABLE processedfiles
(filename VARCHAR(256),
mergedate DATE);
INSERT INTO processedfiles VALUES('jsmith_200503091600.txt',
to_timestamp('2002-01-11 11:14:07','YYYY-MM-DD HH24:MI:SS'));

```



---

## Using create\_mcadadmin\_db.jar to Update the Database Tables

1. Set the CLASSPATH environment variable to the path of the create\_mcadadmin\_db.jar file as shown below.

```
set CLASSPATH=<path to file_1>;<path to file_2>
```

For example:

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

### **Note**

*The classes111.jar file is required to manually execute the create\_mcadadmin\_db.jar utility when an Oracle database is used.*

2. Update the user\_database table using the following command:

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

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

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

## Note

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

For example:

- For an Oracle database, use the following command:

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

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

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

A sample `user_database` table is shown below:

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

- 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 93](#) 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.McMetricsCollector msaccess
sun.jdbc.odbc.JdbcOdbcDriver jdbc:odbc:admin
user1 userpass

```

A sample mc\_database table is shown below:

USERID	DATE_N_TI	MDLNAME	MDLTY	CHECKTAG	CHKSTAT	INF01	MCMODE
devid	18-JUL-04	RIM	PRT	SUP_FEATURES	WRN	2	Interactive
devid	18-JUL-04	RIM	PRT	UNITS_LENGTH	OK	INCH	Interactive
devid	18-JUL-04	RIM	PRT	UNITS_MASS	OK	POUND	Interactive
devid	21-OCT-04	PRT0001	PRT	mc_initialize	1	0	Interactive
devid	21-OCT-04	PRT0001	PRT	PARAMCHECK	OK	0	Interactive
alan	12-APR-05	02J_301_107_C-ROH_KUPPGEH	PRT	MODEL_NAME	OK	0	MC_regen
alan	12-APR-05	02J_301_107_C-ROH_KUPPGEH	PRT	PARAMCHECK	WRN	9	MC_regen
alan	12-APR-05	02J_301_107_C-ROH_KUPPGEH	PRT	PLANE_CHILD	OK	0	MC_regen

A sample processedfiles table is shown below:

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

## Metrics Database

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

The data fields stored in the mc\_database table follow:

- UserId—User ID
- date\_n\_time DATE—Date and time
- mdlname—Model name
- mdltype—Model type
- checktag—Check type
- chkstat—Check status
- info1— Information
- mcmode—ModelCHECK mode

A sample mc\_database table follows (syntax for Oracle):

```

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

```

---

```

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

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

```

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

## Training Database

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

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

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

```

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

```

```

grade VARCHAR(256));

INSERT INTO trng_database
VALUES('austin','101','Fundamentals of Drawing',
to_date('2002-01-11','YYYY-MM-DD'), 'PTC', 'Jack
Webb','Dallas','1500','Good');

```

For more details, refer to the ModelCHECK online Help in the 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`—User ID
- `lastname`—Last name
- `firstname`—First name
- `middlename`—Middle name
- `group_b`—Group to which a user belongs
- `phone`—Telephone number
- `extension`—Extension number
- `parametricdate DATE`—Date when the user ran ModelCHECK for which the metrics files were created
- `info1`—Information

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

```

CREATE TABLE user_database
(UserId VARCHAR(8),
lastname VARCHAR(256),
firstname VARCHAR(256),
middlename VARCHAR(256),
group_b VARCHAR(256),
phone VARCHAR(20),
extension VARCHAR(20),
parametricdate DATE,
info1 VARCHAR(256),
location VARCHAR(256));
INSERT INTO user_database VALUES('jsmith','smith','john','VP', 'designer1',
'4082131', '3182', to_timestamp('2000-01-11
11:14:07','YYYY-MM-DD HH24:MI:SS'), 'aviation',
'/net/machine1/usr3/jsmith/mc_metrics');
INSERT INTO user_database VALUES('dwells','wells','David','K', 'designer2',
'4053182', '3481', to_timestamp('2001-01-11
11:14:07','YYYY-MM-DD HH24:MI:SS'), 'sqlstar',
'/net/machine2/usr2/dwells/demo/mc_metrics');
INSERT INTO user_database VALUES('asewell','Sewell','Annie','P ', 'designer3',
'4082725', '3192', to_timestamp('1997-01-11

```

```
11:14:07', 'YYYY-MM-DD HH24:MI:SS'), 'explosive',  
'/net/machine1/usr3/asewell/mc_metrics');
```

For more details, refer to the ModelCHECK Online Help in the 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 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>\Common Files\<datecode>\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. 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
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

Font Name	File Name	Supports Kerning
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	gargrabl.ttf	Yes
Garth Graphic Italic	gargrai.ttf	Yes
Grotesque MT	grotesq.ttf	No
Grotesque MT Bold	grotesqb.ttf	No
Microstyle Extended	microex.ttf	Yes
Microstyle Extended Bold	microexb.ttf	Yes
Neographik MT	neograph.ttf	Yes
SackersEnglishScript	sackengs.ttf	No
Shannon	shanno.ttf	Yes
Shannon Bold	shannob.ttf	Yes
Shannon Extra Bold	shannoeb.ttf	Yes
Shannon Oblique	shannoo.ttf	Yes
Spartan Four MT	sparton4.ttf	Yes
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](http://www.fonts.com).

When you work with Creo applications, you use two categories of fonts:

- Fonts specific to Creo applications—These fonts appear in the graphics windows of 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

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 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 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>\Common Files\<datecode>\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 Creo applications. The fonts are listed in the **Control Panel** under **Fonts**. The font for text in Creo application window headers is determined by the setting of your window manager.

## Extended ASCII Set

ASCII is the default font in all 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 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 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. 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

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 Creo application. In case of 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`).

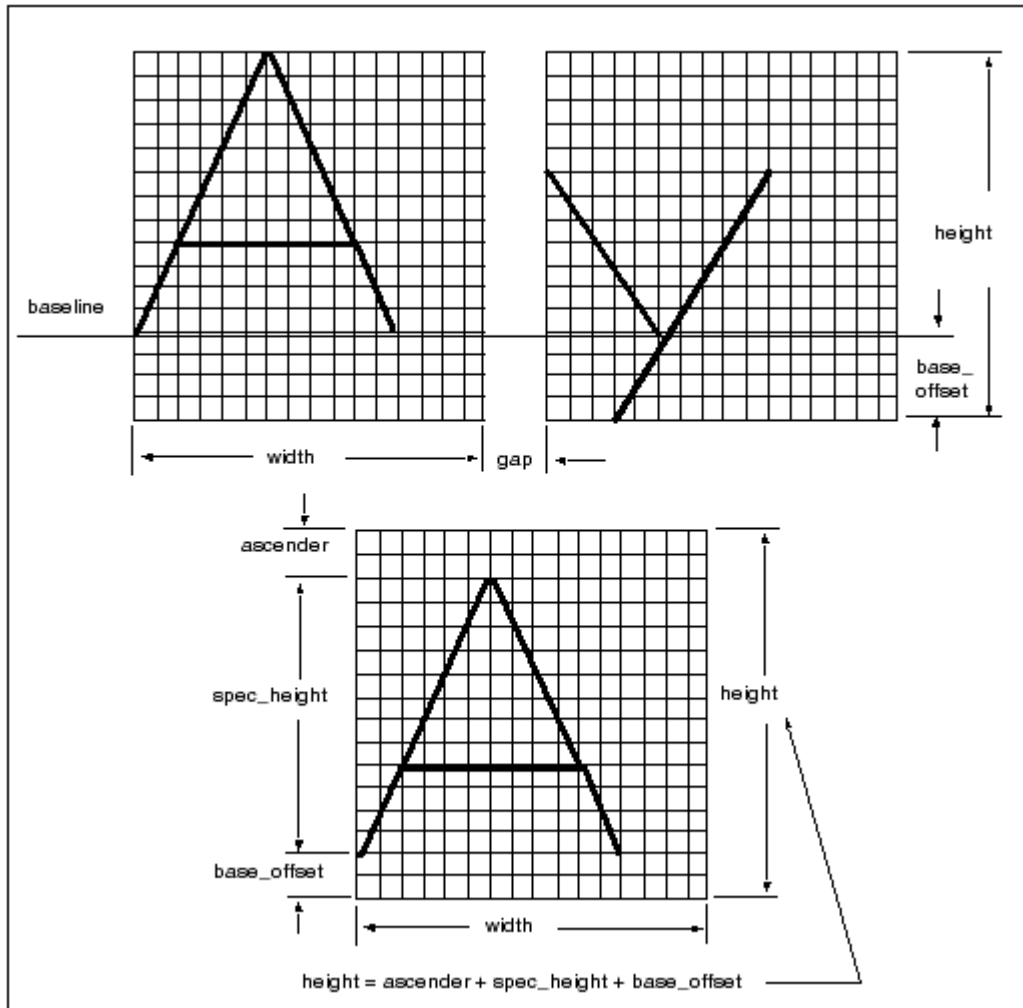
## Note

*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 113](#)).
- If you do not specify this parameter, the system uses the default value 0.

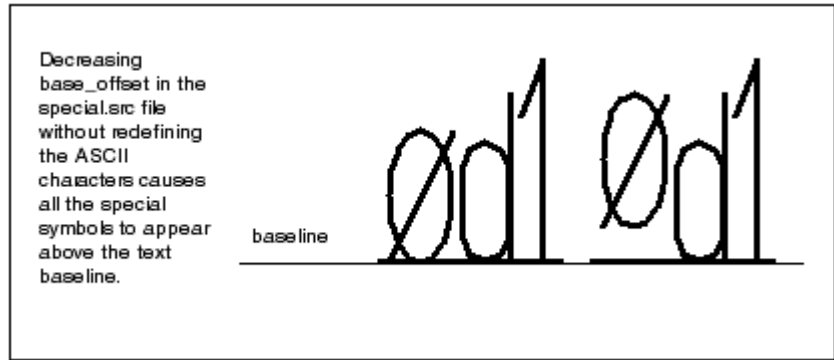
- `fixed_width<n>`—Can be 0 or 1.
- If you set it to 1, the font is independent of the `text_width_factor` option in the drawing setup file.
- If you set it to 0, you can control the font width by using the `text_width_factor` option or by modifying the text width manually.

The following figure illustrates the text parameters that define a font.

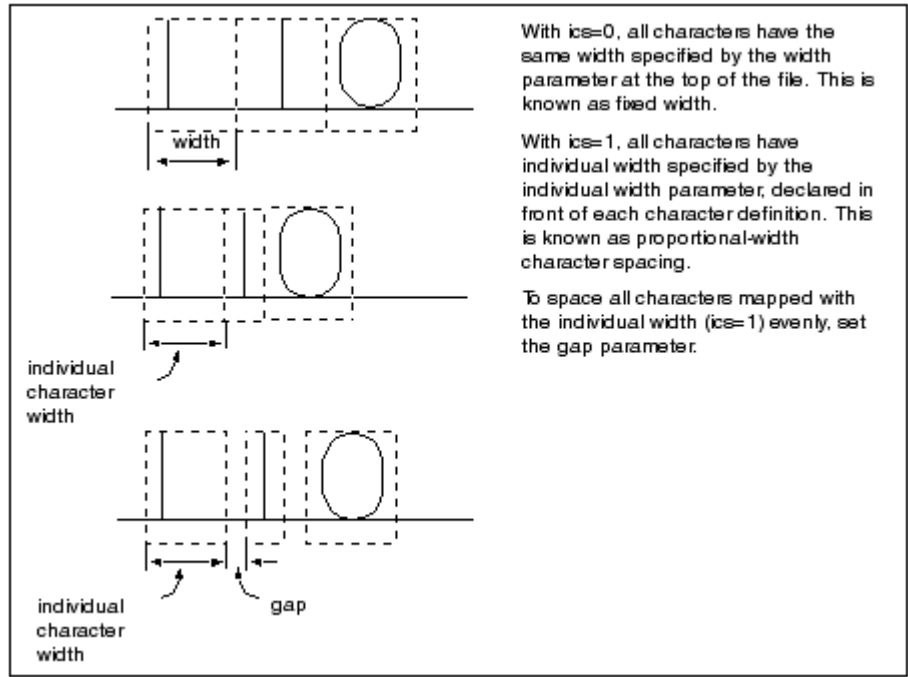


When you modify text parameters, consider how the font will appear in text with other fonts. Fonts used in the same text line (such as ASCII and LATIN\_1 for European languages, or ASCII and special) must be compatible. Altering one font might create inconsistency in appearance and placement.

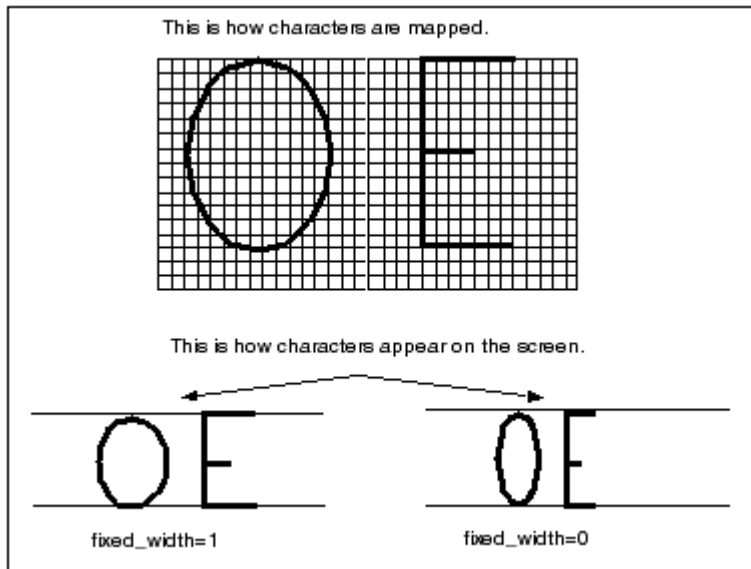
For example, changing the `base_offset` value at the top of the font file affects the positioning of all the font characters, relative to the text baseline, as shown in the following figure.



In addition to the width parameter, an individual width parameter can occur in each character definition. Use the individual width parameter to create a variable horizontal distance between the characters. In this case, if the `ics` parameter = 1, the individual width for each character overrides the width at the top of the file. If `ics` = 0, the system ignores individual width parameters. The following figure shows the effect of the `ics` parameter on the calculation of character width.



The following figure shows the effect of the `fixed_width` parameter on the calculation of character width.



## Character Parameters

Below the text parameters section, a font file contains an individual definition for each character. A character definition contains the following information:

- First line—An ASCII character, in double quotes, that represents the character to be drawn. For ASCII and filled primary fonts, it is the character itself. Special characters also use ASCII characters to represent them. The LATIN\_1 fonts use the decimal and hexadecimal codes representing the character.
- Second line (optional)—Individual width value (used only if ics is turned on).
- The remainder of the file consists of drawing-definition commands, described in the next section.

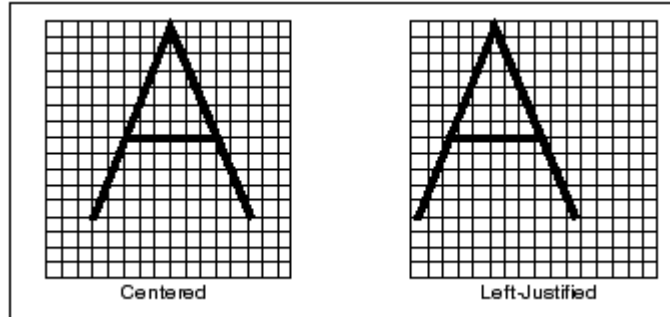
## Definition Commands

The drawing definition of characters and special symbols in the font files is done with the following commands (where <X> and <Y> are integers):

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

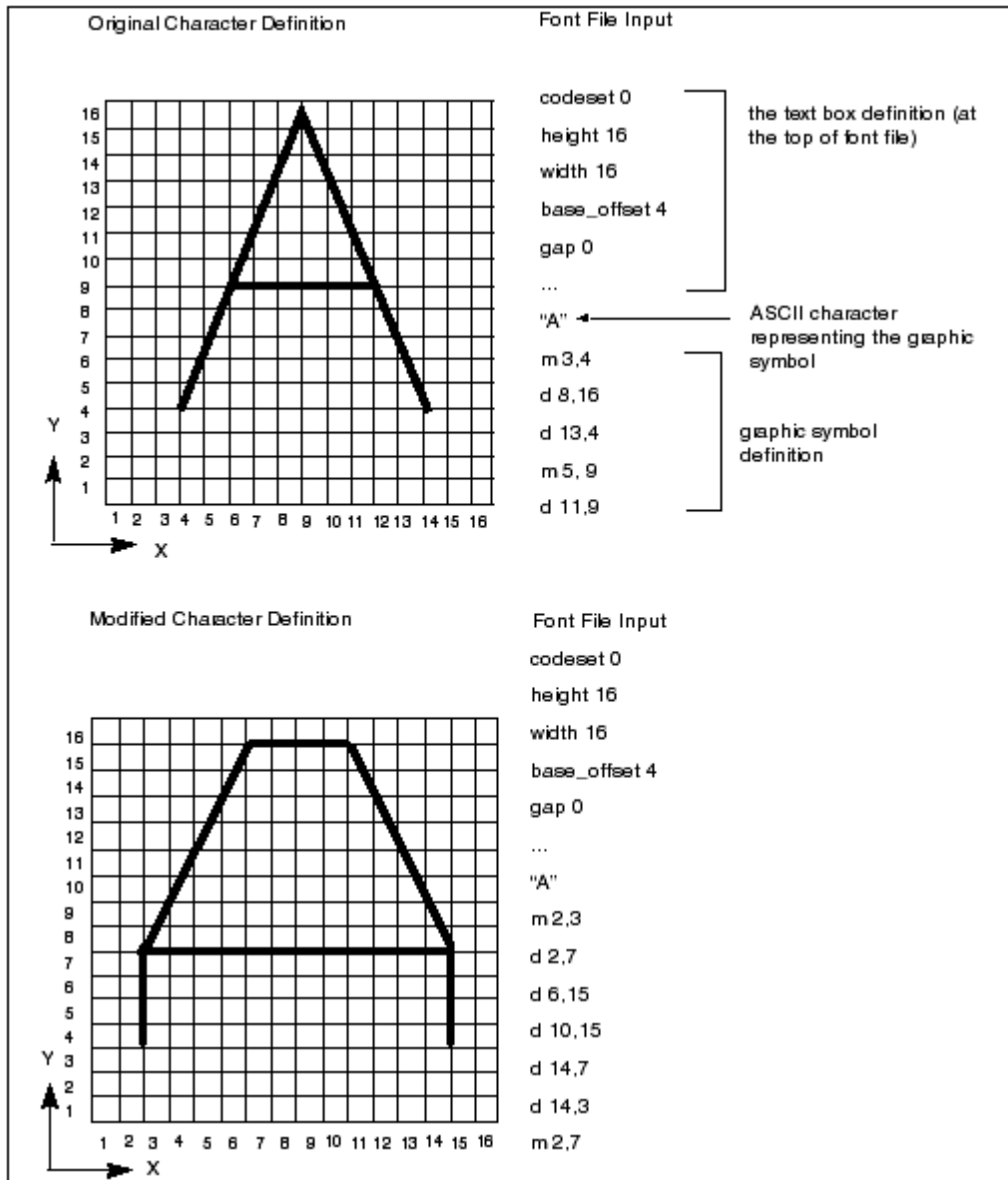
---

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.



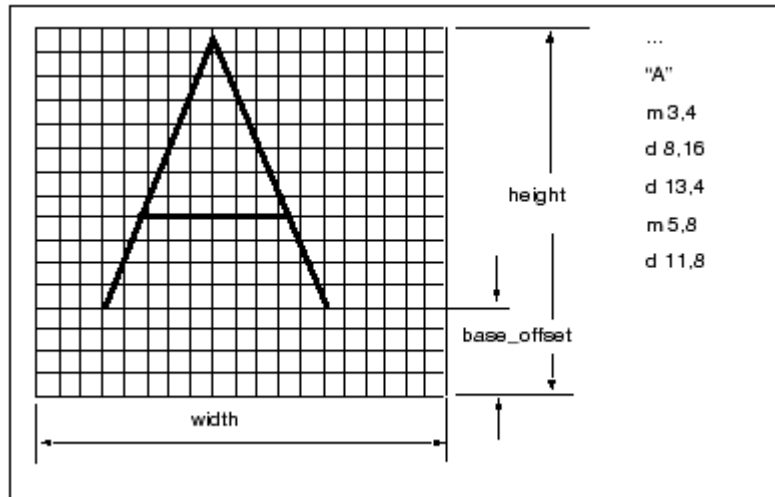
## ASCII Font Definition

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.



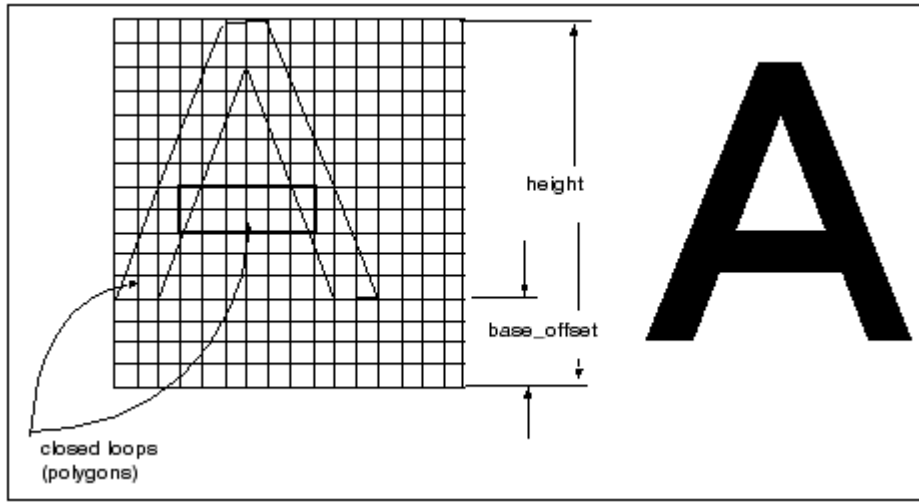
## Filled Font Definition

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 Creo application fills with color any closed polygon drawn in a character definition.

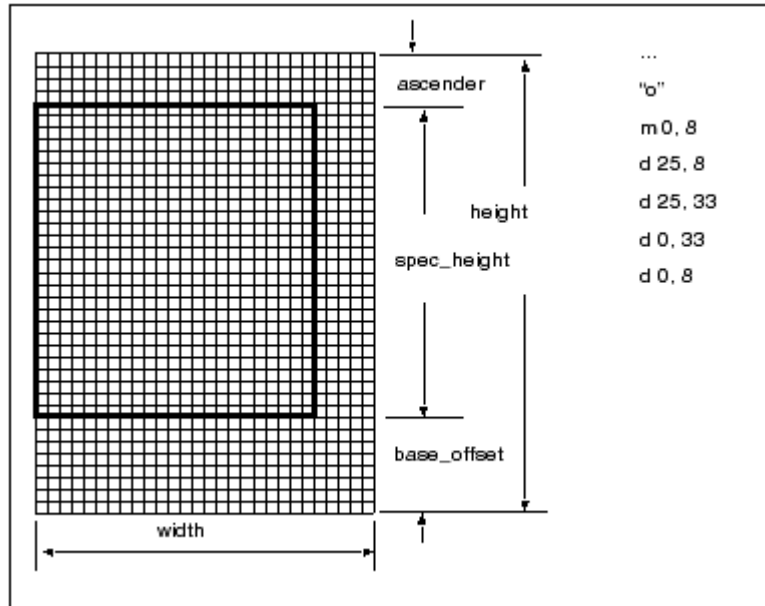


## Special Font Definition

The following text parameters are used to define a special font:

- codeset: 4
- height: 38
- width: 36
- xextent: 63
- spec\_height: 24
- base\_offset: 8
- ascender: 6
- fixed\_width: 1
- ics: 1
- gap: 4

In the special font, all characters are defined in 38 vertical grid spaces. Each character has an individual width declared in front of its definition. To establish spacing between character boxes, the “gap” parameter is set to 4. The following figure illustrates the definition of the special font character.



## LATIN\_1 Font Definition

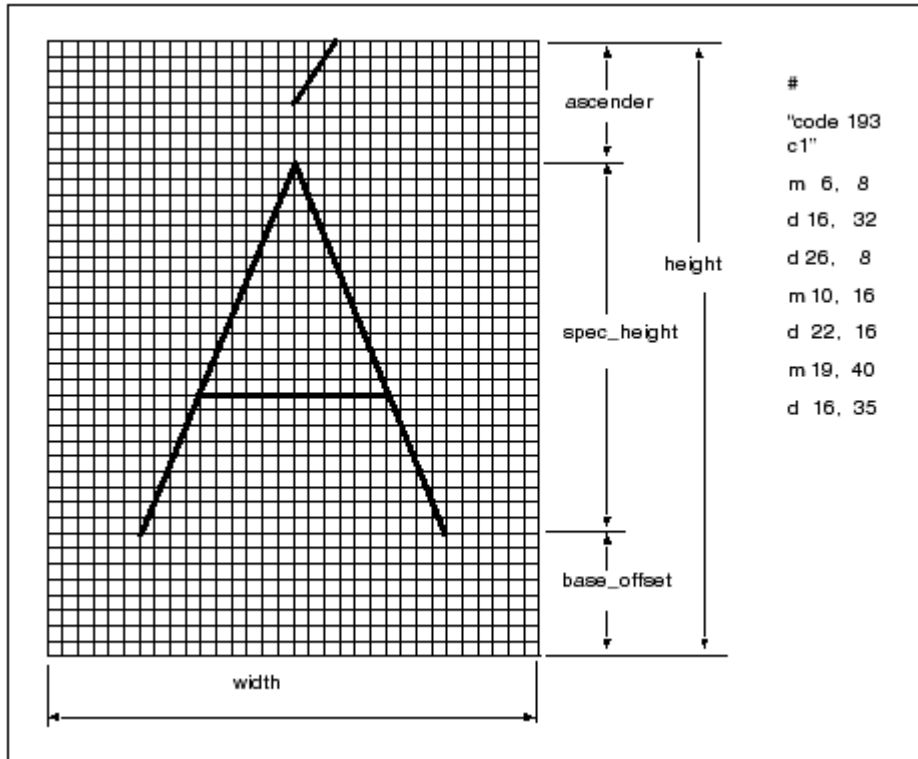
The following text parameters define the LATIN\_1 font:

- codeset: 2
- height: 40
- width: 32
- xextent: 63
- spec\_height: 24
- base\_offset: 8
- ascender: 8
- fixed\_width: 0
- ics: 0
- gap: 0

The LATIN\_1 font is defined with a character box of 40x32 grid spaces. All characters have constant width and are centered in the character box. In the double quoted line above the character definition, the decimal and hexadecimal code representing the character is shown, as follows:

```
code <decimal> <hex>
```

The following figure illustrates how to map a LATIN\_1 character.



## How to Edit Fonts or Create New Fonts

1. Edit an existing or create a new `.src` file, which resides in `<creo_loadpoint>\Common Files\<datecode>\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

1. Copy the appropriate `.src` font file into your working directory. The `.src` files reside at `<creo_loadpoint>\Common Files\<datecode>\text\<language>`.
2. Modify the font file.
3. From the `<creo_loadpoint>\Common Files\<datecode>\text\<language>` area, copy the appropriate index file to your working directory.
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 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 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>\Common Files\<datecode>\text\<language>`. Store the `.fnt` file in the following directory: `<creo_loadpoint>/Common Files/<datecode>/<machine_type>/text/<language>`

## How to Set Your Font as the Default

1. Rename your file to `ascii.src`.
2. Recompile the `ascii.src` file using the `compile_font` command to produce a local `ascii.fnt` file.

## How to Create or Customize a Font

1. Copy or create font source files in the local working directory. In this example, the default primary ASCII and special symbol fonts are copied and renamed.

```
cp /usr/pro/text/usascii/ascii.src custom.src
```

(primary font with codeset 0)

```
cp/usr/pro/text/usascii/special.src symbols.src
```

---

(special symbols font with codeset 4)

2. Edit the source files to include all the required definitions and changes.

```
vi custom.src
vi symbols.src
```

3. Copy the `compile_font` script from the directory `<creo_loadpoint>/Common Files/<datecode>/<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>\Common Files\<datecode>\text\<language>` directory.
- Then put the compiled `.fnt` files in the directory `<creo_loadpoint>/Common Files/<datecode>/<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 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 Creo Parametric

You can customize fonts and add symbols and characters to the **Text Symbol** palette in 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 in Creo Parametric on page 102](#).
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>/Common Files/<datecode>/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 120](#), 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 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 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 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 121](#).

### 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



GTOL Symbol	ANSI Code	ISO Code
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 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, 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 Creo Parametric.
- Use the ASCII codes for the `dt1` options carefully. The usage of these symbols in a drawing changes between ANSI and ISO according to the value that you set for the detailing setup option, `gtol_symbols_same_size`.

### Examples of the Text Symbol Palette Layout File

- If you specify `#1.15` in the first row of the text symbol palette layout file, Creo 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 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 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 Creo Parametric parameter data as searchable metadata in the PDF file.
- Modify the font types, color, style, and resolution of the PDF output.

- 
- Insert Creo Parametric drawing text and numerical values as searchable and selectable content in the PDF file.
  - Insert bookmarks for the various 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 Creo Parametric Interface Help available in the Creo Help Center.

## Meeting Browser Requirements

This section describes the requirements for the Creo browser for all the Creo applications. For information on Web browser support for the Creo applications, see [www.ptc.com/WCMS/files/135225/en/creo2\\_hw\\_notes.pdf](http://www.ptc.com/WCMS/files/135225/en/creo2_hw_notes.pdf). For information related to the Help Center, see [Installing and Using Creo Help on page 55](#).

### Browser Requirements

The Creo application browser uses the rendering engine of Internet Explorer. To enable this default, make sure that Internet Explorer 6.0 with Service Pack 1 and above or Internet Explorer 7.0 is installed on your computer. Alternatively, you can configure the Creo application to use a Mozilla-based browser. Creo applications already have an embedded version of this browser. They cannot use any other version installed on your computer.

The Creo browser uses Internet Explorer engine in the in-process mode. However, with the multiprocess architecture of Internet Explorer or the Mozilla-based embedded browser on Windows, you can run the embedded browser in the out-of-process mode. That is, you can run Internet Explorer or Mozilla browser as a separate process from the Creo application process.

You can specify whether you want the Creo browser to use Internet Explorer or the Mozilla-based rendering engines. Before you start a Creo application, set one of the following values to the `windows_browser_type` configuration option in your `config.pro` file:

- `ie_in_process`—Specifies that Creo browser uses the Internet Explorer engine in the same process as that of the Creo application. This is the default setting on a 64-bit Windows operating system.
- `ie_in_sep_process`—Specifies that the Creo application uses the Internet Explorer engine in a separate child process initiated from the Creo application process. This is the default setting on a 32-bit Windows operating system.
- `mozilla_based_browser`—Specifies that Creo browser uses the Mozilla-based rendering engine in a separate child process initiated from the Creo application process.

---

## Starting a Creo Application through an External Browser

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

## Opening a PDF File within the Creo Browser

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

## Using Creo View Express to Open Creo Objects

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

File Extension	File Type
<code>.asm</code>	Assembly file
<code>.dft</code>	Draft file
<code>.dgm</code>	Diagram file
<code>.drw</code>	Drawing file
<code>.frm</code>	Format file
<code>.pic</code>	Picture file
<code>.prt</code>	Part file
<code>.rep</code>	Report file
<code>.mfg</code>	Manufacturing file
<code>.sec</code>	Sketcher Section file

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

---

## Opening Creo Objects from Windows Explorer

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

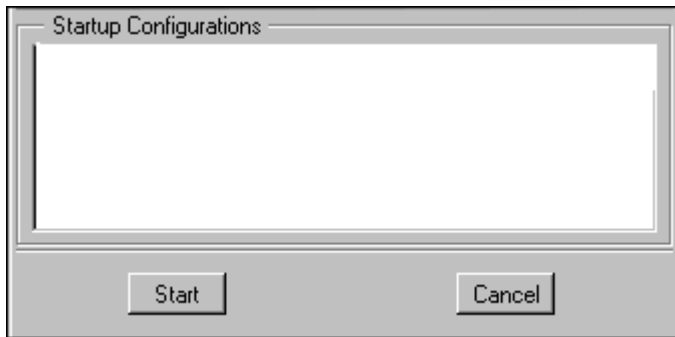
If you have installed Creo View Express, a Creo object is opened in Internet Explorer only for viewing when you double-click the object in Windows Explorer. Right-click on the object and choose the Creo application in which to open the object.

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
.lay	Layout file
.mfg	Manufacturing file
.mrk	Markup file
.prt	Part file
.sec	Sketcher Section file

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

If a Creo session is not running and you double-click an object in Windows Explorer, the 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 Creo session is started using the selected configuration file. The selected object is loaded in this Creo session.



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

## JavaScript Security

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

A 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 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.

## Rights Management

Rights Management Extension helps to control access to your intellectual property. Intellectual property could be sensitive design and engineering information that you have stored within Creo parts, assemblies, or drawings. You can control access by applying policies to these Creo objects. Such objects remain protected by the policies even after they are distributed or downloaded. Creo objects for which you have applied policies are called DRM-protected objects. Policies control access rights to DRM-protected objects for users within or outside your firewall.

Administrators can define policies at the Adobe LiveCycle Rights Management ES server (policy server) through a Web interface. They can also define:

- Publishers who can apply these policies to files within the Creo application.
- Users who can access the DRM-protected objects.

---

Refer to the Adobe Help for more information on defining policies and specifying users or user groups.

After the administrators have defined policies, users can perform the following DRM operations within a Creo application:

- Open a DRM-protected object after authenticating themselves with the policy server. Users can then use the object with their usage rights established in the current policy of the object.
- Apply or remove policies to Creo objects. To do so, users must have a DRM license.
- Connect to the policy server to verify the policy details.
- Generate a report of all the DRM-protected objects in the current session.

For more information, refer to the Creo Parametric Digital Rights Management Help in the Creo Help Center.





# D

## Installing and Configuring Quality Agent

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This appendix contains information on Quality Agent.

---

## Overview

Quality Agent is an application that gathers and sends reports to PTC on the performance and usage of 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.

### Note

*Quality Agent is a 32-bit application only.*

## Installing Quality Agent

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

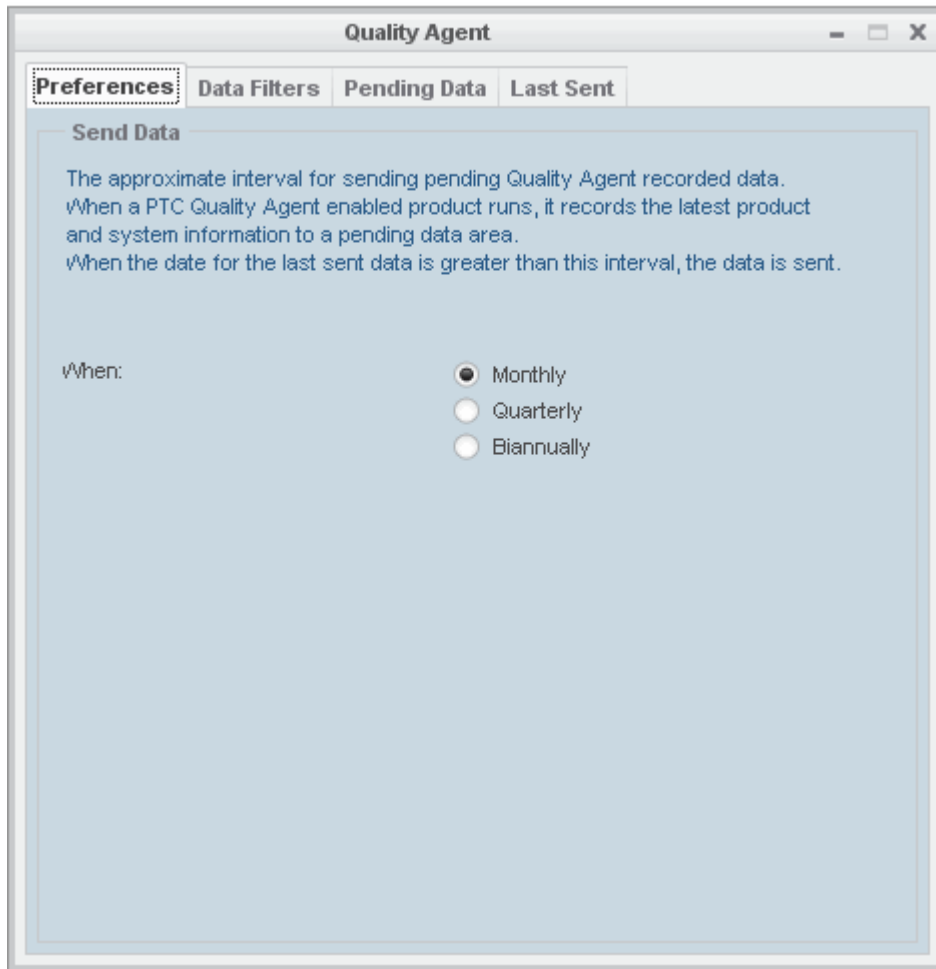
### Note

*When customizing a 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. Quality Agent reporting is not enabled in this case.*

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

## Configuring Quality Agent

After installing Quality Agent, you can configure it as per your company's requirements. To open the **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

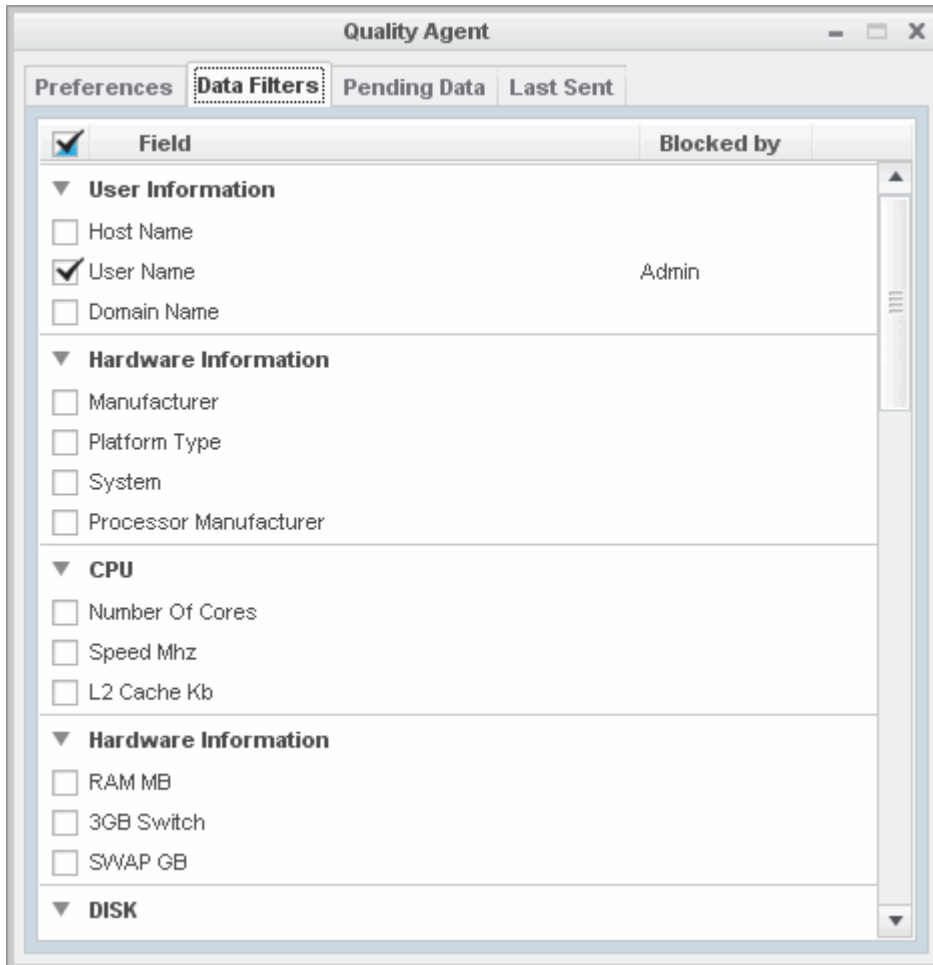
If you have write permissions to the Quality Agent installation folder and files, specify the frequency for the collection and transfer of the product and system information to PTC using the **Preferences** tab. Set the frequency to **Monthly**, **Quarterly**, or **Biannually**.

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



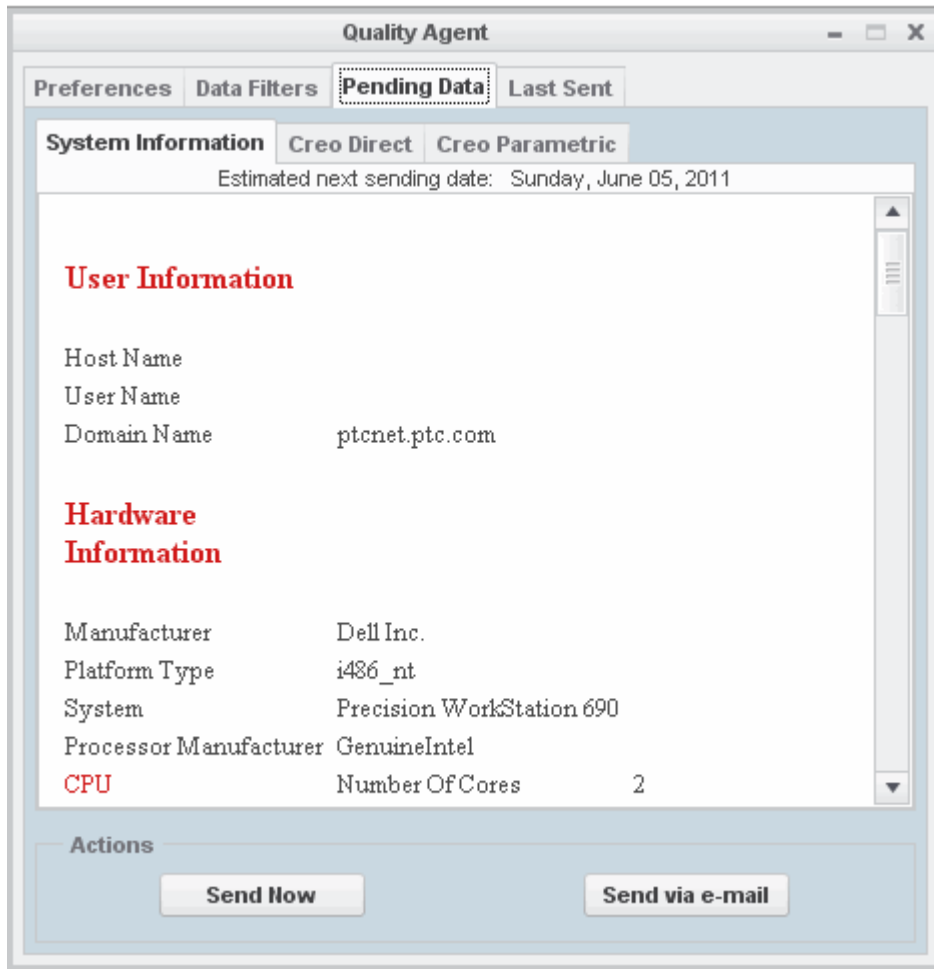
### Note

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

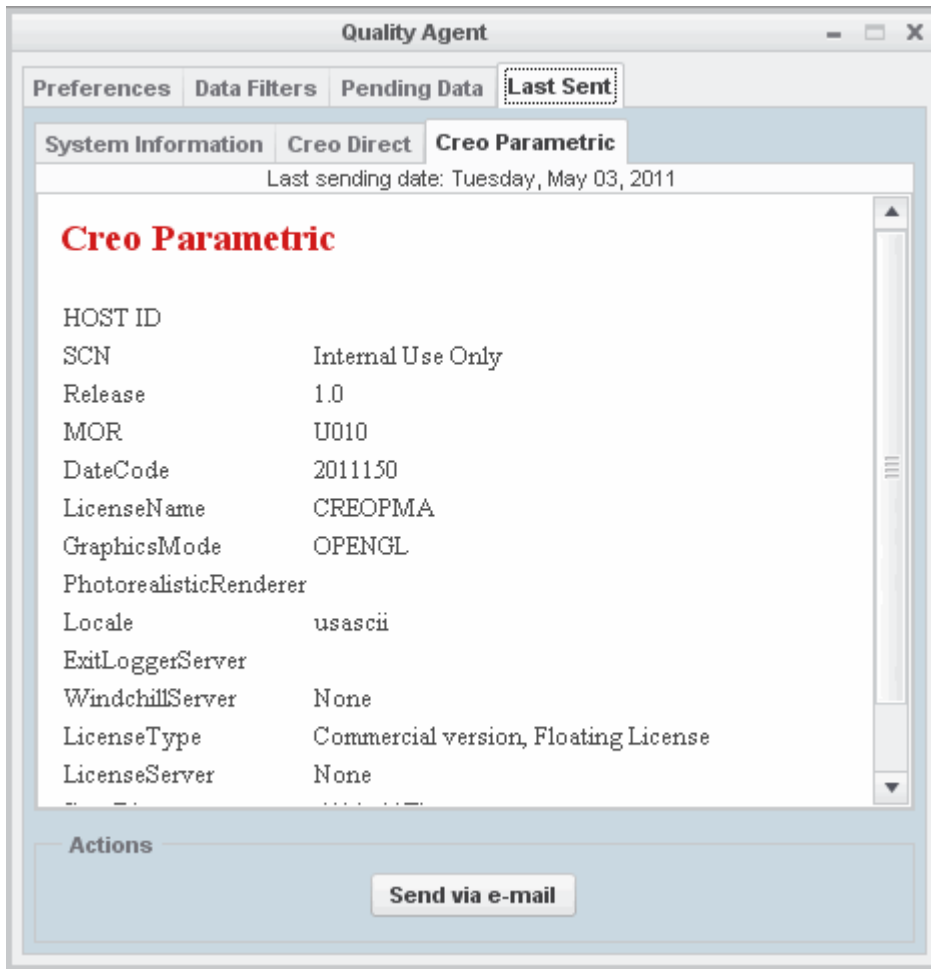


On the active application tab, click a button to perform the following actions:

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

## Last Sent

For each 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

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, 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 Creo application, you can open the **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 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 Quality Agent

You can remove Quality Agent independent of a 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 Quality Agent.







# E

## Installing MKS Platform Components

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This appendix contains information on how to install MKS platform components to work with CADD5 5 models.

---

## Overview

You must install **MKS Platform Components 9.x** along with the **Interface for CADD5** component during Creo Parametric installation to work with CADD5 parts and assemblies within Creo Parametric. **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 CADD5 and Creo Parametric.
- Copies the necessary startup files and libraries required to work with CADD5 files into a separate installation directory.

## Installing MKS Platform Components

Install MKS Platform components using the following instructions:

1. Perform steps 1 through 7 as described in the section [Installing the Creo Applications: Simple Installation Process on page 30](#).
2. Click the **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 CADD5 Models in Creo Parametric on page 52](#) before you start.

# F

## Installing the 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 Creo applications: Creo Parametric, Creo Direct, Creo Layout, Creo Simulate, and Creo Options Modeler. The instructions for using command-line arguments are also common for Creo Distributed Services Manager.

---

## Guidelines and Advantages of a Silent Installation

The 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 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 143](#) 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 Creo application, language, platform, or package is not on the DVD.
  - The prerequisites for installing the Creo application are not met.
- You must run the Assistant only from the DVD when installing the 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.

Some of the advantages of a silent installation follow:

- You can efficiently mass-deploy 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 Creo application from an XML file.

---

# Strategies for the Mass Deployment of Creo Applications

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

- Perform a silent installation of Creo applications on all the client computers. See [Workflow for a Silent Installation on page 141](#) 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 150](#) 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 150](#) for details.
  2. Update the registry entry for each Creo application on each client computer using the `created.reg` file. See [Updating the Registry File on page 152](#) 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 150](#) for details.
  2. Update the registry entry for each Creo application on each client computer using the `created.reg` file. See [Updating the Registry File on page 152](#) for details.

## **Note**

*If you have installed the Creo applications on your local computer and have also run the silent installation process, you need not update the registry file.*

## Workflow for a Silent Installation

A system administrator can follow the workflow described below for a silent installation of one or more 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 or add 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 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 143](#) 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 Creo application is stored with a `.p.xml` extension at `<creo_loadpoint>\<app_name>\bin\pim\xml`. When installing applications like Creo Parametric, the `Common Files` folder is installed. The XML file for this folder is stored at the following location:

```
<creo_loadpoint>\Common Files\<date_code>\bin\pim\xml\creobase.p.xml.
```

2. Copy one or more 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 145](#) for information on how to edit the various nodes of an XML file.
3. Perform a silent installation of the desired Creo applications using the command syntax as described in the next section.

---

## Command Syntax for a Silent Installation

To perform a silent installation of a 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
<code>setup.exe &lt;no arguments&gt;</code>	Runs the installer interactively.
<code>setup.exe -help</code>	Displays all the arguments to use with the installer.
<code>setup.exe -v</code>	Displays the version of the installer.
<code>setup.exe -xml &lt;file1&gt; -xml &lt;file2&gt;....</code>	Performs a silent installation as per the directives in the specified XML files.
<code>setup.exe -xml &lt;file1&gt; [-destination &lt;directory&gt;]</code>	[Optional] Overrides the destination specified in the XML file.
<code>setup.exe -xml &lt;file1&gt; [-licensesource &lt;string&gt;]</code>	[Optional] Overrides the license source specified in the XML file.
<code>setup.exe -xml &lt;file1&gt; [-translate_lp_ path_space &lt;character&gt;]</code>	[Optional] Replaces the whitespace characters in the installation path in the XML file.
<code>setup.exe -xml &lt;file1&gt; [-licfeat_regen]</code>	[Optional] Overrides the license feature list 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.

Command Syntax	Description
<pre>setup.exe -xml &lt;file1&gt; [-licfeat_preference &lt;string&gt;]</pre>	<p>[Optional] Prepends the preferred license feature names to the default license feature list. 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.</p>

Consider the example of performing a silent installation of Creo Parametric with Common Files. 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 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

## Locating the XML Files

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

Application Files	File Name	File Location
Creo application files	<app_name>.p.xml	<creo_loadpoint>\<app_name>\bin\pim\xml
Common Files folder	creobase.p.xml	<creo_loadpoint>\Common Files\<date_code>\bin\pim\xml

For example, if you have installed Creo Parametric, the XML file `pma.p.xml` is stored at `<creo_loadpoint>\parametric\bin\pim\xml`. The XML file for the Common Files folder is stored at `<creo_loadpoint>\Common Files\<date_code>\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 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 Creo applications in the current session. The default value is usually the PTC folder under the Program Files folder.

- 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.

---

## 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 2.0` to `Creo_2.0` or `Creo2.0`. 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 2.0/Parametric</PROPERTY>
<PROPERTY name="[CREOCOMMONFILES]" [INSTALLBASE]/Creo 2.0/Common Files
</PROPERTY>
```

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

```
<PROPERTY name="[LP]">[INSTALLBASE]/Creo_2.0/Parametric</PROPERTY>
<PROPERTY name="[CREOCOMMONFILES]" [INSTALLBASE]/Creo_2.0/Common_Files
</PROPERTY>
```

### Note

- *The CREOCOMMONFILES attribute is used by all the 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 143](#) 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 Quality Agent settings for an installed Creo application. You can edit the following attribute of the QUALITY\_AGENT node:

Attribute	Value
enable	Enables or disables the Quality Agent reporting for a Creo application. For example, if this attribute is set to Y, Quality Agent reporting is enabled.  <b>Note</b> <i>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 Creo application will not run because of conflicting settings.</i>

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 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 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 PLATFORM Node

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

Attribute	Value
install	Set this attribute to Y or N depending on whether you want to install the Creo application for that operating system. At least one platform entry must have the install attribute set to Y.

A sample PLATFORM node in an XML file follows:

```
<PLATFORM install="N" installed="N" name="i486_nt">Windows 32 bit</PLATFORM>  
<PLATFORM install="Y" installed="N" name="x86e_win64">Windows 64 bit</PLATFORM>
```

## Editing the PACKAGE Node

PACKAGE nodes in an XML file represent the required and the optional packages selected for a 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 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="creoptions" 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 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 Creo Applications in Silent Mode

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

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

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

```
D:\>"C:\Program Files\PTC\Creo 2.0\Parametric\bin\uninstall.exe" -quiet
```

You must navigate to the installation directory of each Creo application to silently uninstall it. In case the Common Files folder is installed, uninstalling the last 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 in the Creo loadpoint:

- Creo Platform Agent
- Thumbnail Viewer
- Creo View Express

---

These components are usually installed the first time that you run a Creo application on a client computer. To eliminate repeating this process, administrators can perform a silent installation of these components after installing the Creo applications onto the client computers. Refer to the following sections for the command-line syntax to run the installers for each of these components.

## Creo Platform Agent

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

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

```
"C:\Program Files\PTC\Creo 2.0\Common Files\\install\addon\creoagent_32_64.msi" /passive
```

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

```
msiexec /uninstall "C:\Program Files\PTC\Creo 2.0\Common Files\\install\addon\creoagent_32_64.msi" /passive
```

## Thumbnail Viewer

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

- Install Thumbnail Viewer using the following command:

```
"C:\Program Files\PTC\Creo 2.0\Common Files\\install\addon\Thumbviewer_32_64.msi" /passive
```

- Uninstall Thumbnail Viewer using the following command:

```
msiexec /uninstall "C:\Program Files\PTC\Creo 2.0\Common Files\\install\addon\Thumbviewer_32_64.msi" /passive
```

## Creo View Express

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

1. Browse to C:\Program Files\PTC\Creo 2.0\Common Files\\install\addon\pvx32\_64\ where the installation files have been downloaded.
2. Deploy the application using the Creo View Express executable as shown in the following example:

```
start /w 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 CreoSetup.exe /vADDLOCAL="ALL" APPLICATIONFOLDER="c:\ptc\CreoViewExpress" /qn"
```

To uninstall Creo View Express in silent mode, browse to `C:\Program Files\PTC\Creo 2.0\Common Files\<datecode>\install\addon\pvx32_64\pvexpress\`. Enter the following command in a command shell:

```
start /w msixexec.exe /x CreoView_Express_32_64.msi /qn
```

### **Note**

*Running the `CreoView_Express_32_64.msi` file alone does not completely install 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 Creo View Express directly using the MSI file.*

For more information on installing and uninstalling Creo View Express in silent mode, see Chapter 7, *Deploying Creo View and Customizing the Installation*, in *Installing Creo 2.0: Creo View*.

## **Installing Quality Agent and MKS Platform Components in Silent Mode**

To silently install Quality Agent and MKS Platform Components, you must copy the `qualityagent.xml` and the `mkscomponents.xml` files found at `<creo_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 Creo applications are installed. The file resides at the following location:

```
<creo_loadpoint>\<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 Creo applications is as shown:

C:\Program Files\PTC\Creo 2.0\Parametric\bin\pim\xml\created.reg

(Creo Parametric)

C:\Program Files\PTC\Creo 2.0\Direct\bin\pim\xml\created.reg

(Creo Direct)

To make configuration changes on the client computer, open the `created.reg` file stored on the computer where the 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 Creo applications and the related client applications.

## Using Communication Ports

Many of the 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 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 Creo and client applications follow:

Application	Environment Variable Setting
<p>Nmsd—PTC name service used by asynchronous Pro/TOOLKIT applications and Distributed Services Manager to connect to 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.</p>	<p>Set PTCNMSPORT using <b>Control Panel ▶ System ▶ Advanced system settings ▶ Environment Variables</b>.</p> <p><b>Note</b> <i>You cannot set this environment variable in your .psf file.</i></p>
<p>pro_comm_msg.exe—Facilitates communication between the executable xtop.exe and asynchronous Pro/TOOLKIT applications.</p> <p><b>Note</b> <i>The executable xtop.exe refers to all the desktop-based Creo applications.</i></p>	<p>A port number can be allocated by a Creo application or it can be set dynamically.</p> <ul style="list-style-type: none"> <li>• 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.</li> <li>• 2 dynamic ports—Cannot configure assigned port values after their allocation.</li> </ul>

Application	Environment Variable Setting
<p>PTC Portmapper—If installed, communicates port information between Pro/TOOLKIT and a 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.</p>	<p>Not supported.</p>
<p><code>xtop.exe</code>—Connects to <code>creoagent</code>, <code>nmsd</code>, and <code>pro_comm_msg</code> to open a few dynamically allocated ports. This application uses the same host TCP/IP communication protocol for all the processes on a workstation.</p>	<p>Not supported.</p>
<p><code>creoagent.exe</code> —Uses dynamically allocated ports to connect to all the Creo applications. This application uses the same host TCP/IP communication protocol for all the processes on a client workstation.</p>	<p>Not supported.</p>
<p>Async TK Apps—Use dynamically allocated ports to connect to <code>pro_comm_msg.exe</code> on a client workstation.</p>	<p>Not supported.</p>
<p>Converters—Use dynamically allocated ports to connect to desktop-based Creo applications on a client workstation.</p>	<p>Not supported.</p>
<p>CreoView—Uses dynamically allocated ports to communicate with browser plugins, <code>nmsd</code>, and <code>comm_brk_svc.exe</code> on a client workstation.</p>	<p>Not supported.</p>

Application	Environment Variable Setting
<p><code>comm_brk_svc.exe</code> —Uses dynamically allocated ports to support communication between <code>xtop.exe</code>, third-party CAD software, and Creo View sessions.</p>	<p>Not supported.</p>
<p>Creo Distributed Services Manager—Uses the TCP/IP communication protocols between multiple workstations.</p> <ul style="list-style-type: none"> <li>• The executable <code>dbatchc.exe</code> is the <code>dbatch</code> client interface that runs on a client workstation. This interface communicates with the Service Manager <code>dsq.exe</code> located on a different workstation.</li> <li>• The <code>dbatch</code> service executable <code>dbatchs.exe</code> runs on a service workstation.</li> <li>• The executable <code>dsq.exe</code> uses a dynamic port to communicate with <code>dbatchc.exe</code> and multiple <code>dbatchs.exe</code> processes. Each service workstation runs <code>nmsd.exe</code>.</li> </ul>	<p>Set <code>PTCNMSPORT</code> using <b>Control Panel ▶ System ▶ Advanced system settings ▶ Environment Variables</b> to configure the default port number. Make sure that you set the same port number value on all the service workstations.</p>

## 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:

1. Navigate to the `<license_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 2.0\app\bin\*.psf` and the `Creo 2.0\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.
3. Navigate to the `<license_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 `<license_server_manager_loadpoint>\bin` folder.
7. Restart the `lmadmin_ptc` service.
8. Edit the `Creo 2.0\app\bin\*.psf` and the `Creo 2.0\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:



- 
1. 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.
3. Navigate to the `<license_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 Creo application, navigate to the `bin` folder in the Creo load point. For example, if 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 Creo Parametric

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

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## Overview of Distributed Computing

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

- A 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 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 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 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.

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

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

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

---

## FlexNet Publisher Fails to Start (Triad Configurations)

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

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

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

## ModelCHECK Metrics Tool Display Problems

While using the Web-based ModelCHECK Metrics Tool, you may encounter problems with the display of the graphs and reports in the Internet Explorer or the Mozilla browser. Causes of the problem and possible solutions follow:

Cause: The path to the Image Directory is not specified in the `new_admin.xml` configuration file. If specified, it may be incorrect. Solution: Verify the location of the Image Directory and the path specified in the `new_admin.xml` configuration file. If found incorrect in the `new_admin.xml` file, specify the correct path, and deploy the Metrics Tool again.

Cause: The Apache Web server is not up and running or is not available. Solution: Verify if the Apache Web server is up and running and start the Apache Web server, if required. Before starting the server, make sure that the location of the Image Directory is valid and exists.

Cause: The `DocumentRoot` path in `Apache\conf\httpd.conf` file is not set properly. Solution: Specify the `DocumentRoot` path correctly. If the problem persists, specify the Image Directory location to match the `DocumentRoot` path.

Cause: The Metrics Tool displays the connection pool exhausted error.

Solution: Check the values for **Max. Active Connections**, **Max. 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.*



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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.



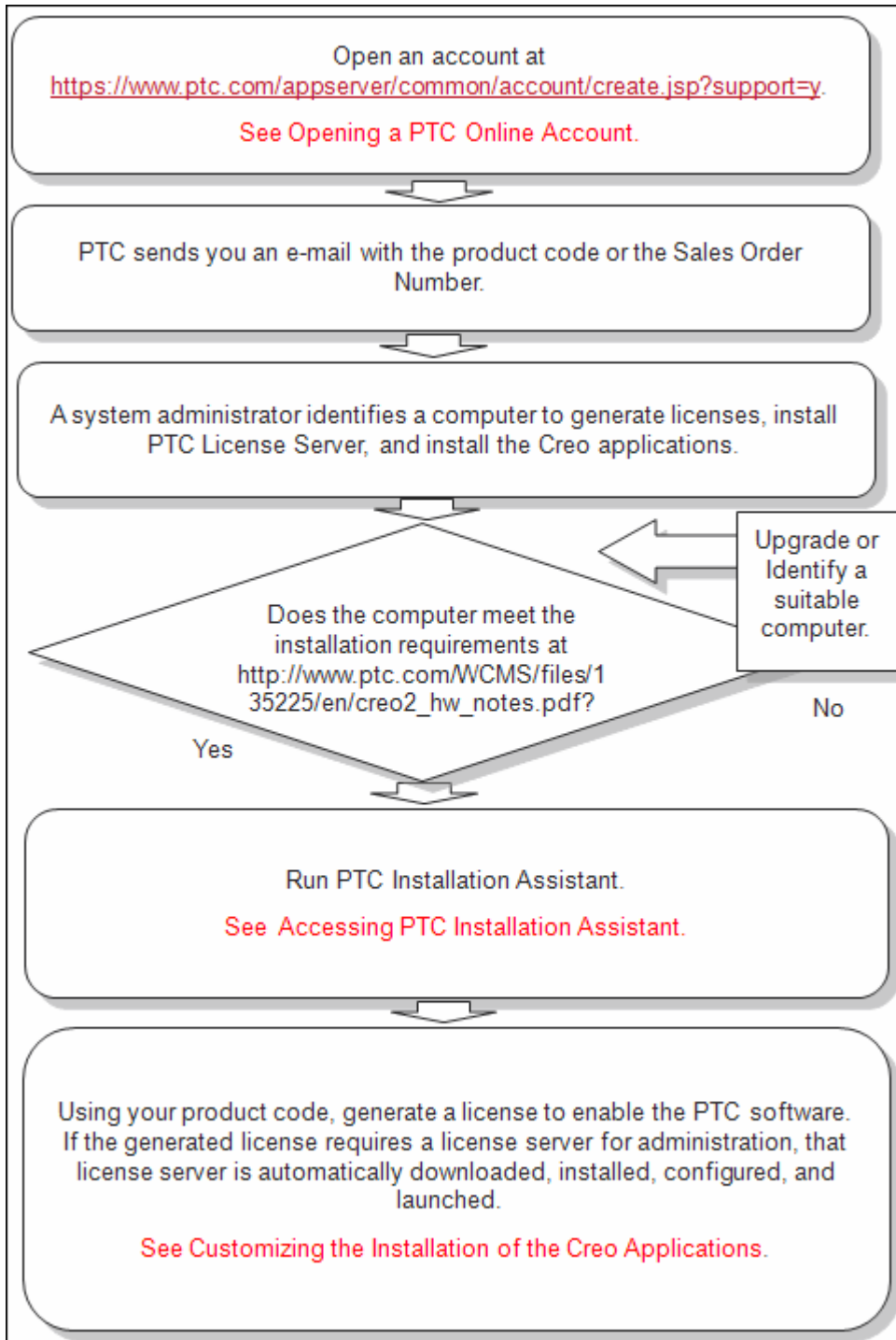
# J

## Workflow for Installation and Configuration

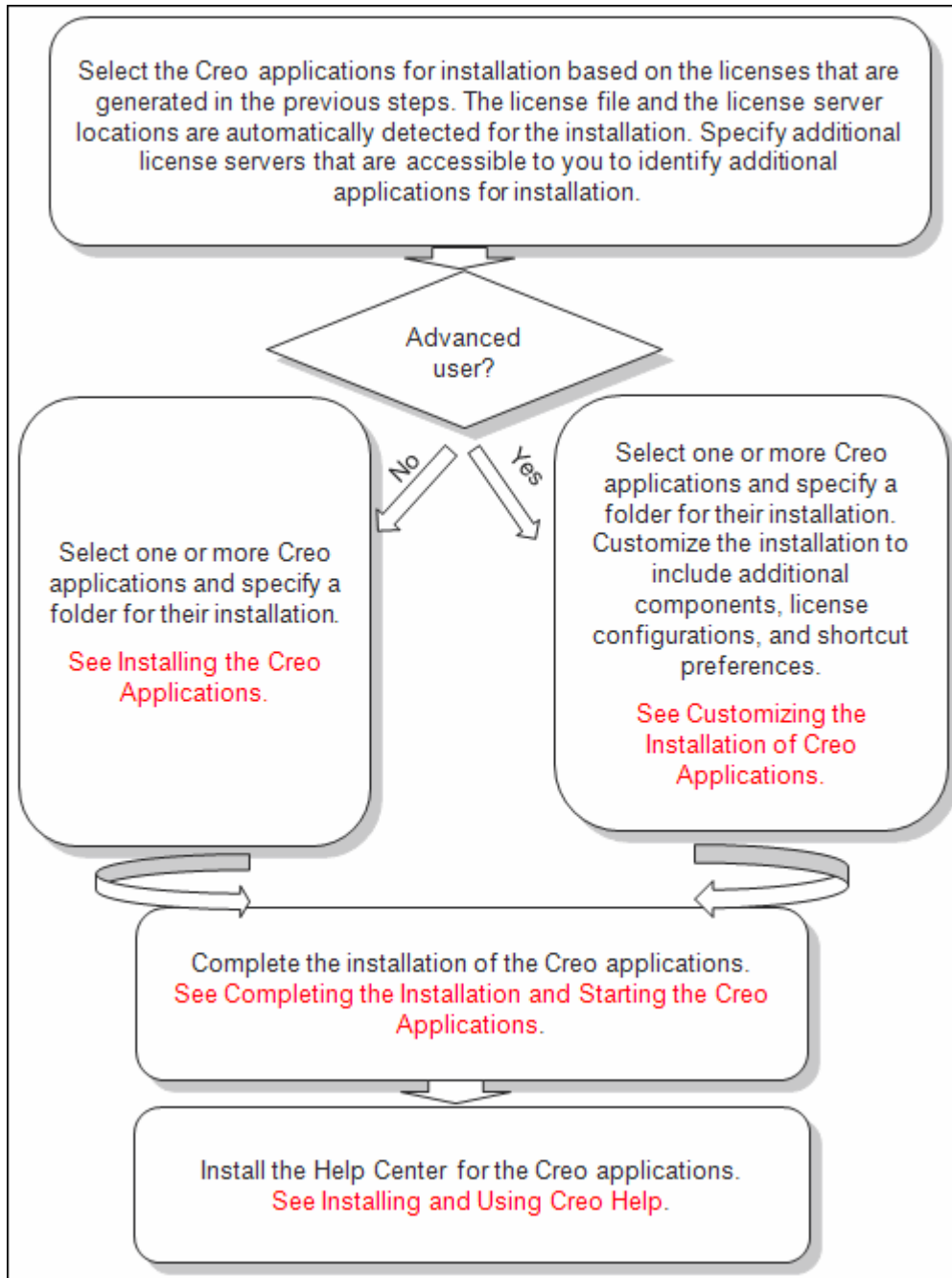
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This chapter provides simple workflows for generating licenses and installing and reconfiguring the Creo applications. The workflow is presented as a flowchart for each of these processes. If you have received a license file via e-mail from PTC License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

## Workflow for Generating Licenses for the Creo Applications

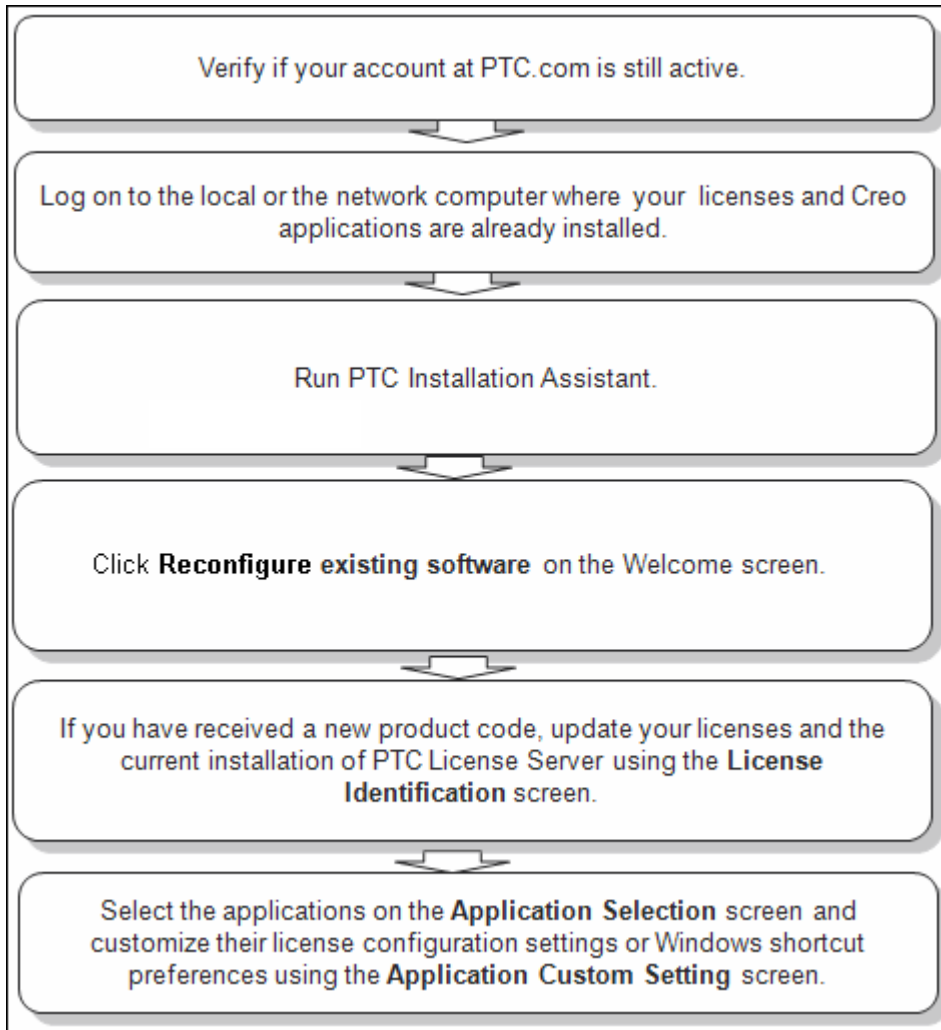


# Workflow for Installing the Creo Applications



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## Workflow for Reconfiguring the Creo Applications



# Glossary

Term	Definition
Application Program Interface (API)	A set of standards or conventions by which programs can call specific operating system or network services.
authentication	A process of logging in to a secure server to verify your identity.
commercial license	The license you get when you purchase PTC software for commercial purposes.
counted locked license	Although a locked license is managed by a FlexNet Publisher license server, its usage is restricted to a machine for which it is licensed. The license server allows the use of up to the number of licenses purchased.
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.
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 Creo Parametric allows rapid transition from 2D concept to 3D design.
Creo Options Modeler	An application that includes limited parametric modeling and configuration modeling features.
Creo Parametric	A 3D mechanical design automation application, Creo Parametric provides associative and interoperable product design and an engineering modeling system.
Creo Parametric J-Link	A Java-based toolkit, J-Link allows developers to create Java applications to access a Creo Parametric session.

Term	Definition
Creo Parametric Pro/Web.Link	Creo Parametric Web.Link lets you interact with Creo Parametric through a Web browser. With this feature, you can create a customized Web page, which you can then use to automate and streamline aspects of your engineering process.
Creo Parametric TOOLKIT	An API that enables adding of functionality to Creo Parametric using the C programming language.
Creo Parametric VB API	A visual basic toolkit, Visual Basic API allows developers to create VB.NET or VBA applications to access a Creo Parametric session.
Creo Plastic Advisor	Creo Plastic Advisor simulates mold filling for injection molded plastic parts and provides designers with immediate, easy access to reliable manufacturing feedback and advice.
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.
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 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	<p>An entry in the license file that defines</p> <ul style="list-style-type: none"> <li>• The name of the PTC daemon, which is a customized software program that grants and denies floating PTC licenses by reading the license file</li> <li>• The path where this executable resides, that is in the FlexNet Publisher installation directory</li> <li>• The path to the FlexNet Publisher options file, which contains various operating parameters</li> </ul>



Term	Definition
educational license	A license used at educational institutes to run PTC software, also known as a students license. This is a time-bound license.
evaluation license	An evaluation license allows use of a software product for a trial period.
feature line	<p>The feature or increment line is an entry in the license file that describes the product that the user is allowed to use, as well as any optional software products that are associated with that license. Following is a sample of the feature line:</p> <pre> INCREMENT PROE_Engineer ptc_d 33.0 01-jan-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=SCN12344,SCN765431\ SIGN2="169C A28A E97F E96E 0A3E 563B FDEB\ 4510 829E 4BF4 25D3 2394 0444 2FD4 6C23 0168\ A8A5 AEBE 54B0 1FF6 B79B DC75 2014 A278 33CC\ 1B90 8647 6A12 F4D6 45BF"\ </pre>
feature name	The feature name is the name of the license that the application requests. It is composed of a product prefix and a configuration ID (for example, PROE 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.

<b>Term</b>	<b>Definition</b>
intellectual property	Proprietary information stored within Creo data files.
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 <code>license.txt</code> file on each user's disk drive when FlexNet Publisher is installed and configured. This file contains the information used by FlexNet Publisher to authorize the use of software.
license information	The data from the PTC License Pack that is used by FlexNet 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.

Term	Definition
license server	A machine or system process that distributes licenses to license clients. For example, if you start Creo Parametric on machine A and it requests a license from machine B, then machine B is the license server. The license server keeps track of the number of licenses in use. A system administrator can use a license server to control licenses by placing restrictions on a particular feature.
license-to-run	A license-to-run invokes the license of a specific PTC application, such as 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 ( <code>lmgrd</code> or <code>lmadmin</code> ) runs on the license server and works with the PTC vendor daemon. It does not manage license usage. The <code>lmgrd</code> or <code>lmadmin</code> daemon starts vendor daemons, refers applications to the appropriate vendor daemon, and communicates with other license server manager daemons on the network for Triad installations.
load point	Directory where the software is installed. An example of the Creo Parametric load point is <code>C:\Program Files\PTC\Creo 2.0</code> .
maintenance release request	A request for postproduction PTC software, which features enhancements after a major new revision.
maintenance shipment	An update to the next major release of PTC software.
ModelCHECK	A software productivity tool for Creo Parametric, ModelCHECK analyzes parts, drawings and assemblies, and recommends proper 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.

<b>Term</b>	<b>Definition</b>
name_service daemon (nmsd)	This daemon process enables remote communication between 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 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 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 PTCIm, 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 Creo applications.
reconfigurations	The process of modifying the configuration of software (moving modules from one license to another).

<b>Term</b>	<b>Definition</b>
Rights Management Extension	An access control technology which restricts copying or access to intellectual property in Creo Parametric files. Also, see Intellectual property.
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 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 Creo Parametric.
trail file	A trail file records the procedures the end user performed while running a 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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