



# **Creo® Installation and Administration Guide**

**7.0.1.0**

**Creo Parametric**

**Creo Direct**

**Creo Layout**

**Creo Simulate**

**Creo Options Modeler**

**Creo Render Studio**

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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, Creo Options Modeler and Creo Render Studio—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 11</a>	All users	Describes the materials in your software shipment. Also provides an overview of PTC Installation Assistant.
<a href="#">Quick-Start Installation for Experienced Users on page 19</a>	Advanced users	Describes a quick-start approach for users who have previously installed license management and product software.

Chapter	Role	Information
<a href="#">Configuring Licenses for the Creo Applications on page 22</a>	All users	Describes how to generate or update licenses and install PTC License Server.
<a href="#">Installing the Creo Applications on page 32</a>	All users	Describes a simple installation process for the Creo applications.
<a href="#">Customizing the Installation of the Creo Applications on page 39</a>	Advanced users	Describes the advanced customization options for installing the Creo applications.
<a href="#">Completing the Installation and Starting Creo Applications on page 48</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 60</a>	Advanced users and system administrators	Describes how to install and access Help for Creo Parametric, Creo Direct, Creo Layout, Creo Simulate, Creo Options Modeler, and Creo Render Studio from a local host computer, a network drive, and from a Web server.
<a href="#">Updating an Installation on page 68</a>	All users	Describes how to update the Creo applications on your system.
<a href="#">Installing the Creo Applications in Silent Mode on page 140</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 7.0: Creo Schematics*
- *Installing Creo 7.0: Creo View*
- *Installing Creo 7.0: Creo Illustrate*
- *FlexNet Publisher License Administration Guide* that discusses the third-party license management software for distributing Creo licenses



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- *Installing and Configuring the Standalone PTC License Server* that discusses the installation of the ladmin-based PTC License Server

## Technical Support

For resources and services to help you with PTC software products, please visit the Customer Support Guide on the eSupport Portal at:


<http://support.ptc.com/appserver/support/csguide/csguide.jsp>

You must have a Service Contract Number (SCN) before you can receive assisted technical support. If you do not have a service contract number please contact PTC Customer Care by clicking the [Contact](#) tab on the Customer Support Guide page.

## Documentation for PTC Products

PTC provides documentation for download at PTC.com and also on other installation media. The following forms of documentation are available for Creo applications:

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

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

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

## Feedback to Documentation

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

`mcad-documentation@ptc.com`

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

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

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## 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</b> ► <b>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 Installation media—Your order contains one or more installation media for each product purchased. It also contains the media for the following Creo applications: Creo Parametric, Creo Direct, Creo Layout, Creo Simulate, Creo Options Modeler, and Creo Render Studio 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 **Manage Licenses** on the [eSupport](#) page. Follow the instructions in the **Quick Licensing** section to add a new license.

## 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.

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## 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 installation media 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 [Creo Hardware Notes](#).
- 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.
- The latest versions of all Windows software updates, security patches and Microsoft Universal C Runtime in Windows are installed on the local or network computer where you want to install Creo software.

## 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 as a record. A confirmation of your account is sent to your e-mail address.

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## 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 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 170](#) for flowcharts on the installation and reconfiguration tasks.

## Accessing PTC Installation Assistant

You can access PTC Installation Assistant from the installation media 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.

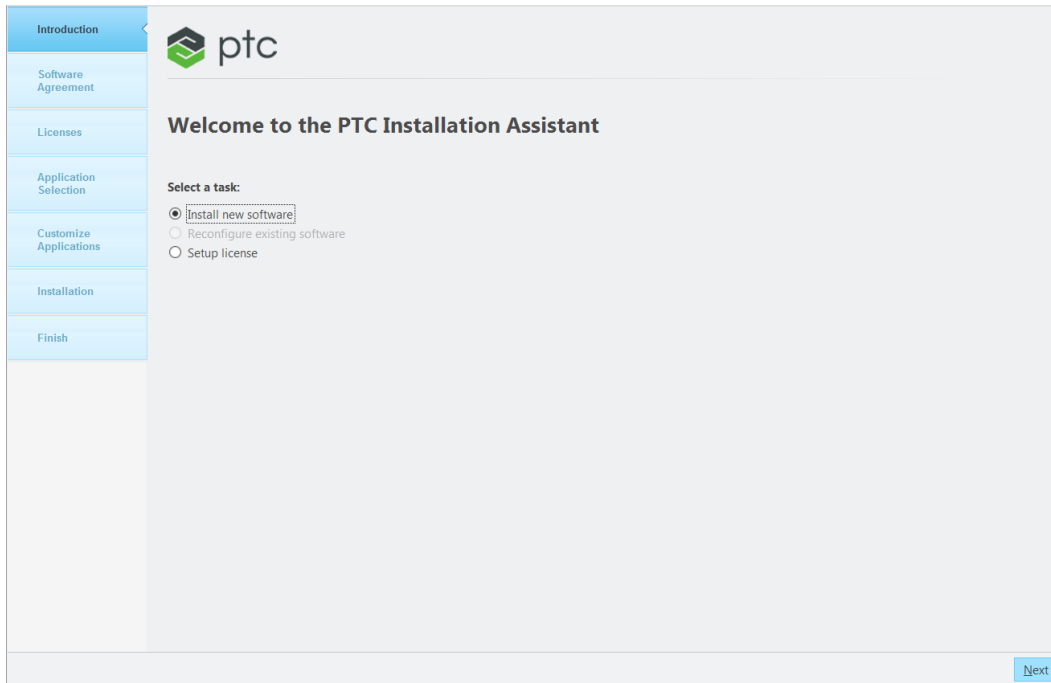
## Starting PTC Installation Assistant

You can start PTC Installation Assistant by running `setup.exe` from the root directory of either the installation media 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, Brazilian Portuguese, and Russian. See the next section for details. If you are starting the Assistant from installation media such as a thumb drive, the Assistant starts automatically if your computer has `autorun` enabled for the drive. If it does not happen, follow one of these methods:

- Start Windows Explorer, browse to the installation media, and double-click the `setup.exe` icon.
- At an MS-DOS prompt, type `<drive>:\setup.exe`, where `<drive>` is the installation media 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 new software**
- **Upgrade existing software**
- **Reconfigure existing software**
- **Setup license**

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


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




# PTC Software License Agreement

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

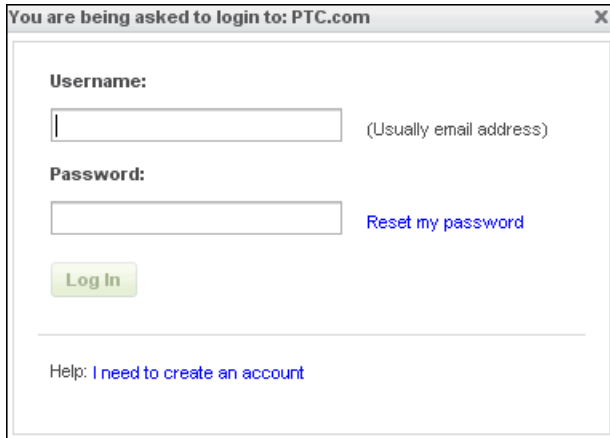
1. Start the Assistant and select a task on the Welcome screen.
2. Click **Next**. The **Software License Agreement** screen appears.

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

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

## Authenticating Your User Access

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



The screenshot shows a dialog box titled "You are being asked to login to: PTC.com". It contains the following elements:

- Username:** A text input field with the placeholder text "(Usually email address)".
- Password:** A text input field with a "Reset my password" link to its right.
- Log In:** A green button.
- Help:** A link at the bottom that says "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 33](#).

---

## 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 69](#) 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 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 22](#). 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. Click **Next**.
8. Customize each Creo application using the steps that follow, or click **Install** to start a simple installation.
  - a. Define the platforms, languages, options, and the other components for each application that you want to customize using the options on the **Application Features** tab.
  - b. Use the options on the **Command Configuration** tab to modify the license configuration information to associate specific licenses to the application startup command. You can attach multiple licenses to a single command name.
  - c. Configure the Windows shortcut preferences as required.

See [Installing Creo Applications and other Utilities on page 32](#) for details on the simple installation process. See [Customizing an Installation on page 40](#) for the customization details.

9. Click **Install**.

---

10. Click **Finish** when the installation completes.

## 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 <version number>\<app\_name>  
where <version number > has the format V.R.M.C where:

- V- indicates the main release number
- R- indicates the point release number
- M- indicates the maintenance release number
- C-indicates the critical patch set number

For example for the main Creo 7 release the value of V.R.M.C would be 7.0.0.0 and the software will be installed at the path C:\Program Files\PTC\Creo 7.0.0.0 hereafter referred to as <creo\_loadpoint>

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

# 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 74](#) 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 7.0. 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 11.16.2.1. It is a replacement to the `lmgrd`-based license server manager shipped with the earlier releases of PTC products. If you are using an `lmgrd`-based license server manager with `ptc_d` vendor daemon of version 10.8.x, it is recommended that you migrate to `lmadmin` although it is not necessary.

## Verifying System Requirements

Installation requirements for the Creo applications in this guide follow:

- You must have administrative privileges to install PTC License Server.
- You must have TCP/IP (Transmission Control Protocol/Internet Protocol) installed and configured correctly on your Windows system before installing the software.

- 
- To use the `lmadmin`-based license server manager on Windows platforms, you must install the following packages:
    - Microsoft Visual C++ 2010 Redistributable Package(x86)
    - Microsoft Visual C++ 2010 Redistributable Package(x86)
    - Microsoft Visual C++ 2013 Redistributable Package(x86)
    - Microsoft Visual C++ 2013 Redistributable Package(x64)
    - Microsoft Visual C++ 2015 Redistributable Package(x86)
    - Microsoft Visual C++ 2015 Redistributable Package(x64)

---

 **Note**

For information on platform support for the Creo applications, see [http://www.ptc.com/WCMS/files/160242/en/PTC\\_Creo\\_Future\\_Platform\\_Support\\_Summary.pdf](http://www.ptc.com/WCMS/files/160242/en/PTC_Creo_Future_Platform_Support_Summary.pdf).

---

## Before You Proceed

Gather all the necessary information that follows about the installation:

- Check the following documents for the latest installation settings:
  - The Hardware Notes for platform support, system requirements, and supported graphics card details at [PTC\\_Creo\\_Future\\_Platform\\_Support\\_Summary.pdf](http://www.ptc.com/WCMS/files/160242/en/PTC_Creo_Future_Platform_Support_Summary.pdf)
  - The READ THIS FIRST document of your product for any changes to your computer settings
  - The FlexNet License Server Hardware Notes for license server installation requirements at [www.ptc.com/WCMS/files/121640/en/FlexNet\\_Licensing.pdf](http://www.ptc.com/WCMS/files/121640/en/FlexNet_Licensing.pdf)
- Make a 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 7.0.
- 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 lmadm*, in the *FlexNet Publisher License Administration Guide* for more information.

## About the Product Codes

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

## Generating or Updating Licenses

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

1. Start the Assistant. The Welcome screen appears.
2. Select either **Setup license** or **Install new software** depending on the tasks that you want to perform:
  - **Setup license** generates or installs licenses.
  - **Install new software** performs license generation or identification tasks after which you can install the software.

3. Click **Next**. The **Software License Agreement** screen appears.

The screenshot shows the 'Software License Agreement' screen. On the left is a navigation pane with steps: Introduction (checked), Software Agreement (selected), Licenses, Application Selection, Customize Applications, Installation, and Finish. The main area features the PTC logo and the title 'Software License Agreement'. Below the title, it states: 'To continue, you must read and accept the terms of the software license agreement.' The 'PTC CUSTOMER AGREEMENT' text is displayed in a scrollable box. At the bottom, there are two radio buttons: 'I accept the software license agreement.' (selected) and 'I decline the software license agreement.'. Below these is a checked checkbox with the text: 'By checking this box, I confirm that the software is being installed and used in compliance with the [Export Agreement](#). I agree to the terms of the [Export Agreement](#) and the export terms of the above License Agreement.' At the bottom right, there are 'Back' and 'Next' buttons.

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. The Host ID of the computer is displayed below the available licenses.

ptc

### License Identification

Licenses allow PTC to identify which applications you are authorized to run on your machine.

**Simple License Entry**  
Enter your Sales Order number or Product code.

**Advanced License Entry**  
[Don't have a license? Click here](#)

**License Summary**  
Detected license sources are listed below. Click the "-" button to remove unwanted sources from the list. Click the "+" button to add more entries into the list. If you already have a license file, drag and drop it into the field below.

Source	Status
+ - 7788@gold;7788@ruby.ptcnet.ptc.com;7788@bronze.ptcnet.ptc.com	Available

Host ID: E4-A4-71-D2-01-D1

Skip Licensing

6. If you don't already have a license click the “[Don't have a licence? Click here](#)” link and follow the instructions on the **License Management** page to obtain a license.
7. Click **Skip Licensing** if you want to proceed with the installation without providing the licensing information. You can provide the licensing information at a later stage by re-running the PTC Installation Assistant.

#### Note

If you click **Skip Licensing**, the **Command Configuration** tabbed page on the **Customize Selected Applications** dialog box is not available when you customize the Creo application. You can add, edit, or delete a license configuration using the **Command Configuration** tabbed page on the **Customize Selected Applications** dialog box.

## Using the Simple License Entry Options

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

1. Type your sales order number or product code in the corresponding text box.

- 
2. Click **Install Licensing**. If the sales order number matches your software order, all the available licenses corresponding to the sales order number are automatically generated into the license file.

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

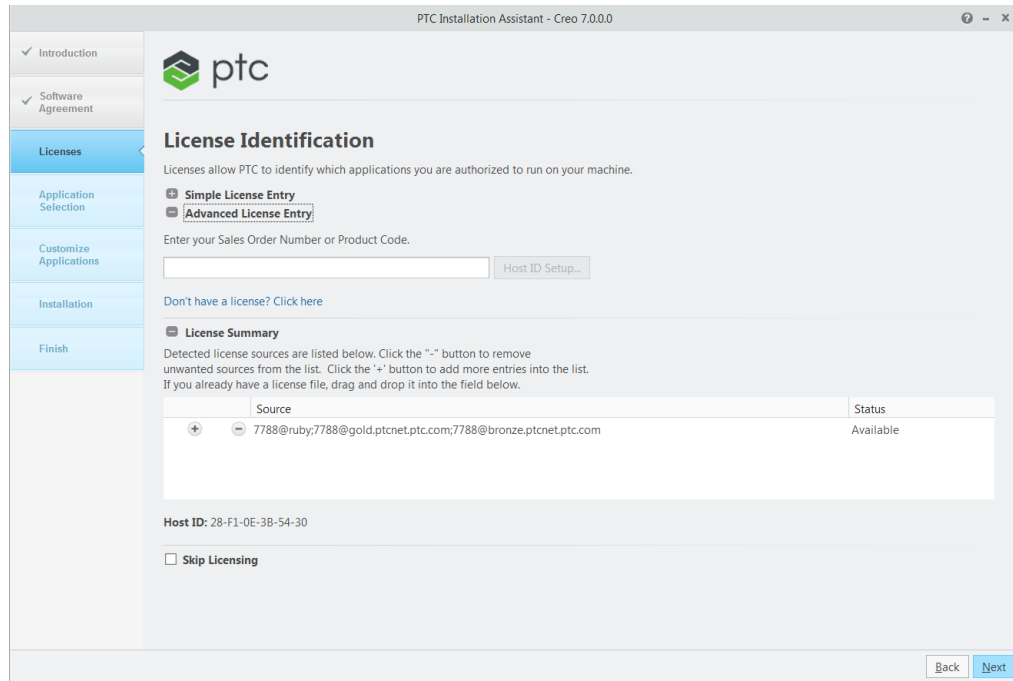
The complete path of the license file or the `port@hostname` value of the license server appears in the **Source** column of the **License Summary** area. This area displays all the license files or servers available for use on your computer.

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

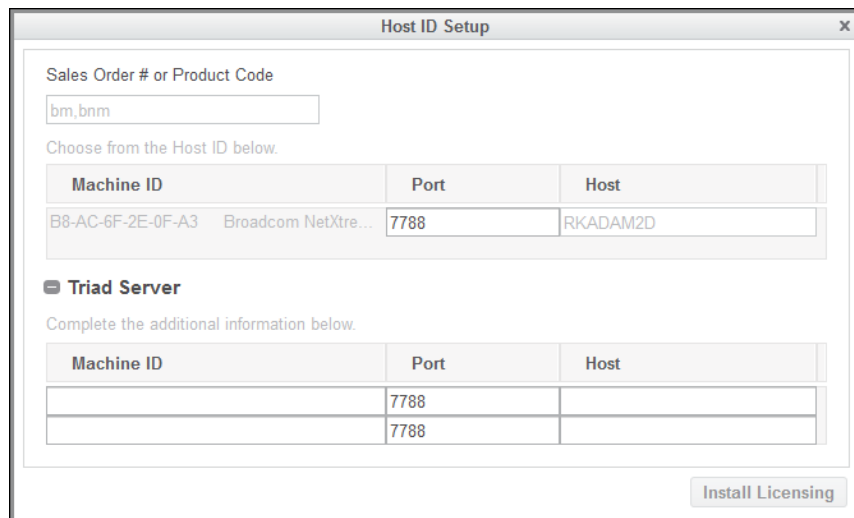
## Using the Advanced License Entry Options

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

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




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



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

- a. To change the default port for a single license, edit the number in the **Port** text box and click **Install Licensing**. Then skip to the last step in this section.

- 
- b. For triad server installations, follow these steps:
    - i. Click  next to **Triad Server** to expand the area.
    - ii. Provide the host ID, port number, and the host name for the other two servers.
    - iii. Click **Install Licensing**. An authentication dialog box to connect to PTC.com may open if your user access has not already been validated. If so, provide your credentials and click **Log In**.

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

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

## Installing the License Server at a Custom Location

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

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

Modify to:

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

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

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


---

 **Note**

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

## Specifying License Entry for Previously Generated License Files

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



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

---

 **Note**

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

---

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

---

 **Note**

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

---

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

---

# 4

## Installing Creo Applications and other Utilities

Installation Prerequisites .....	33
File Format Compatibility .....	33
Installing the Creo Applications: Simple Installation Process .....	33
Using Diagnostic Tools with Creo Applications .....	38

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

- **Creo applications:**
  - Creo Parametric
  - Creo Direct
  - Creo Layout
  - Creo Simulate
  - Creo Options Modeler
  - Creo Render Studio
  - Creo Distributed Services Manager
- **Utilities**



---

## Installation Prerequisites

See [Before You Begin on page 13](#) 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 22](#) for details on how to obtain licenses for your software. You can install your product software on a license server, a license client, a node-locked machine, or on any combination of these machines.

---

### Note

- Usage of uncounted, node-locked licenses does not require PTC License Server.
  - License configuration is a part of the installation process.
- 

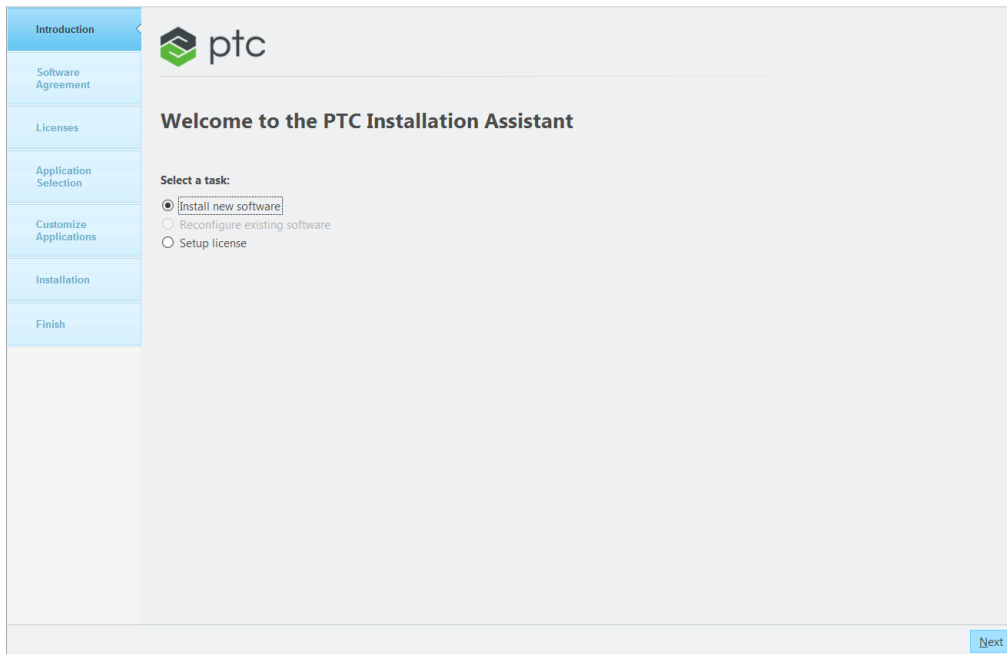
## File Format Compatibility

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

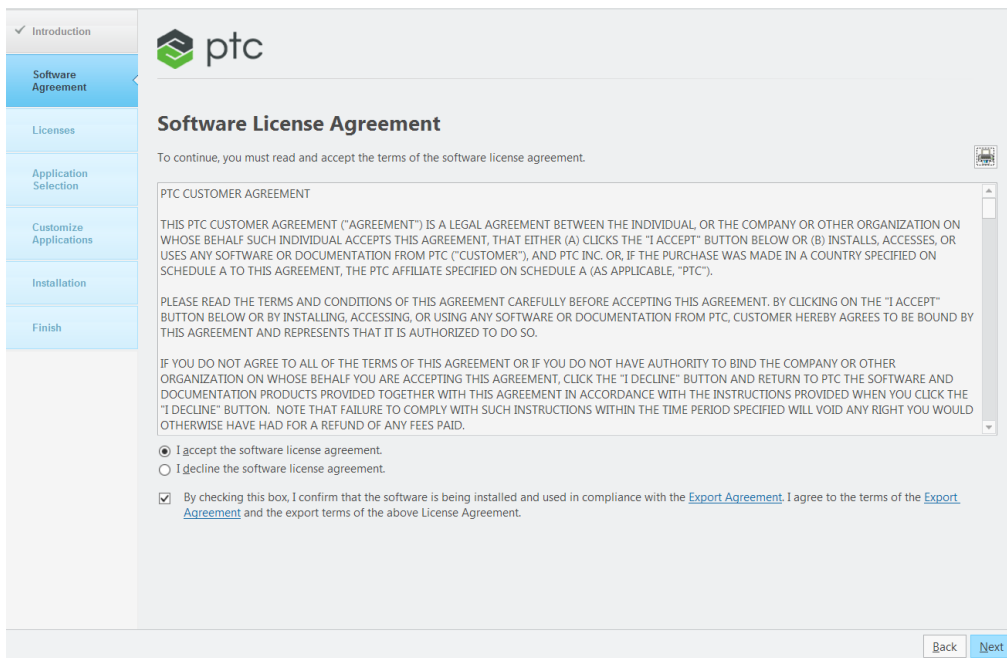
## Installing the Creo Applications: Simple Installation Process

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:

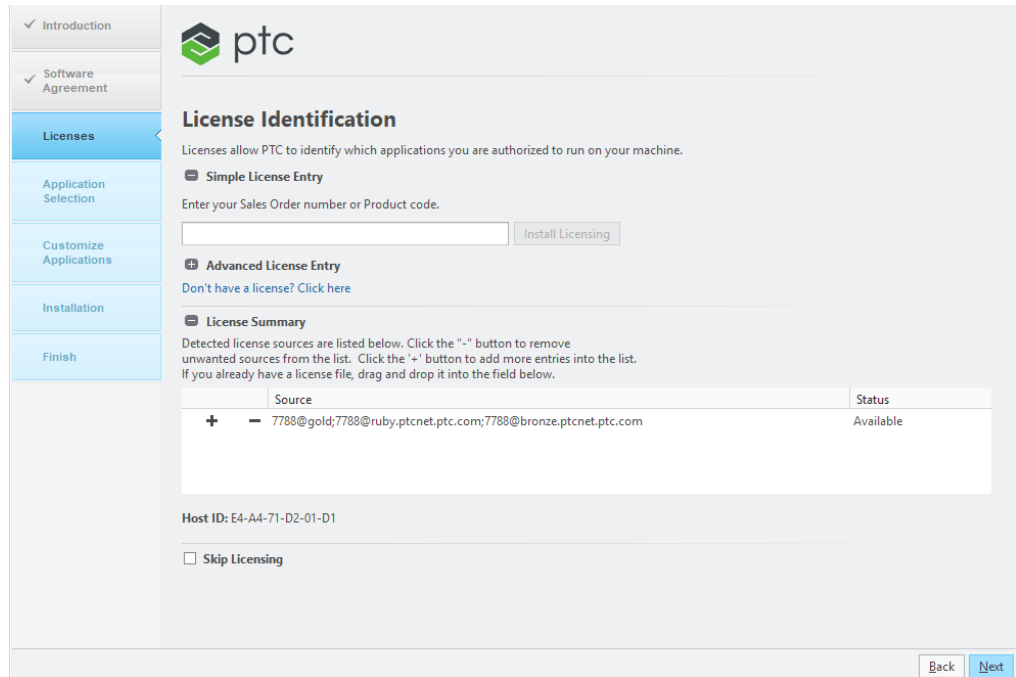
1. Start the Assistant. The Welcome screen appears.
2. Click **Install new software**.



3. Click **Next**. The **Software License Agreement** screen appears.
4. Accept the agreement and select 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.



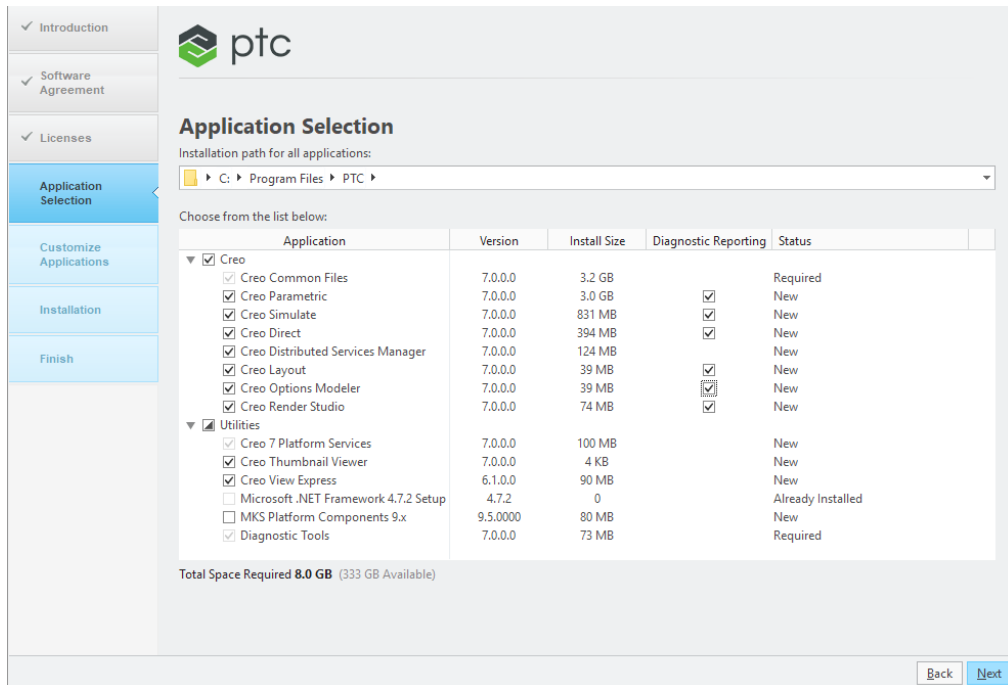
- If you have not already obtained your licenses or installed PTC License Server, follow the instructions in the chapter [Configuring Licenses for the Creo Applications](#) on page 22.

If you have configured your licenses, confirm that your 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.
- Click on the browse button, navigate and select a valid license file on the local system.
- 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.

- Click **Next**. The **Application Selection** screen appears.



8. Specify an installation folder or retain the default folder for all applications.

## Selecting Application for Installation

Select one or more applications from the list:

- Creo:
  - Creo Direct
  - Creo Distributed Services Manager
  - Creo Layout
  - Creo Options Modeler
  - Creo Parametric
  - Creo Simulate
  - Creo Render Studio
- Utilities:
  - Creo Platform Services
  - Creo Thumbnail Viewer
  - Creo View Express
  - Microsoft .NET Framework <version\_number> Setup.
  - Diagnostic Tools
  - MKS Platform Components 9.x

---

 **Note**

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

---

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

Click **Next**. The **Customize Selected Applications** screen is displayed. Perform one of the following steps:

- Click **Install** to install the applications without any customization.
- Customize the selected application by varying the different options displayed on this screen. Refer to the chapter [Customizing the Installation of the Creo Applications on page 39](#) for more information.

### Installation Path of Utilities

Few of the required and optional utilities that you select to install with the Creo applications do not get installed at the default or custom installation path that you specify.

The following is the list of required and optional utilities and their respective installation paths:

- Creo Thumbnail Viewer  
C:\Windows\System32\thumbplug.dll
- Creo Platform Services  
With default location as C:\Program Files\  
With custom path — <installdir>\Creo\Platform
- Creo View Express  
C:\Program Files\PTC\Utilities\Creo View Express\bin
- Diagnostic Tools  
<Installdir>\DiagnosticTools

---

## Using Diagnostic Tools with Creo Applications

Diagnostic Tools is a utility that is installed by default when you install any of the Creo applications. Click or clear the check box at the bottom of the **Application Selection** screen to enable or disable diagnostic reporting. By default, diagnostic reporting is enabled.

Diagnostic Tools helps you to collect information 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. The link to the data collection document is at the bottom of the screen. See [Installing and Configuring the Diagnostic Tools on page 131](#) for more information on the uses of Diagnostic Tools.

# 5

## Customizing the Installation of the Creo Applications

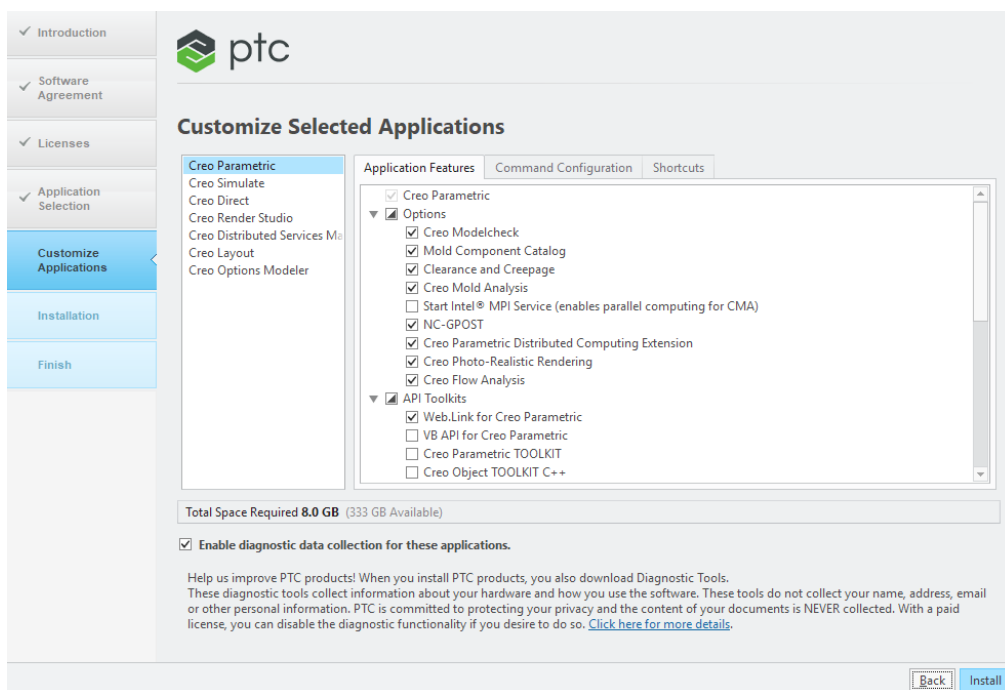
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Defining the Installation Components .....	41
Installing Creo Mold Analysis (CMA) Component .....	43
Modifying License Configurations .....	44
Configuring Windows Shortcuts .....	46
Setting Licenses for Creo Simulate .....	46

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 33](#).
2. When the **Customize Applications** task is selected you can select the application you want to customize from the **Customize Selected Applications** screen. The selected applications are highlighted. 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 41](#).
  - To modify license configurations using the **Command Configuration** tabbed page, see [Modifying License Configurations on page 44](#).
  - To configure Windows shortcut preferences using the **Shortcuts** tabbed page, see [Configuring Windows Shortcuts on page 46](#).
3. Select an application 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 **Install** to start the installation.




- Click **Back** to go back to the **Application Selection** task.
5. Complete the installation of the Creo application. See [Completing the Installation and Starting the Creo Applications on page 48](#) 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 Creo ModelCHECK, Mold Component Catalog, Creo Mold Analysis, NC-GPOST, Creo Parametric Distributed Computing Extension, and Creo Photo-Realistic Rendering with Creo Parametric, Creo Flow Analysis, and Creo Topology Optimization. See <a href="#">Installing CreoMold Analysis (CMA) Component on page 43</a> for information on installing Creo Mold Analysis.   <b>Note</b> The ModuleWorks component is installed by default along with Creo Parametric.

Component	Application	Description
		Additionally starts the Intel MPI Service that enables parallel computing for Creo Mold Analysis.
<b>API Toolkits</b>	Creo applications and Creo Distributed Services Manager	Installs the files necessary to run the Application Program Interface toolkits.
<b>Interfaces</b>	Creo	Installs the external interfaces.   <b>Note</b> To install <b>Creo Parametric Interface for CADD5 5</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 138</a> for more information.
<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 Diagnostic 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 and Configuring Diagnostic Tools on page 131</a> for more information on Diagnostic Tools.
<b>Verification Models</b>	Creo Simulate	Installs the verification models that are described in the Simulation Verification Guide Help.

Component	Application	Description
<b>Creo Parametric Distributed Computing Extension</b>	Creo Parametric, Creo Simulate	Installs this component to run distributed computing tasks that include import/export, printing operations, and studies in the 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.

 **Note**

Some Creo applications require the latest version of Java 8 installed on your machine. For optimal performance of Creo, install the latest version of Java 8 before you start installation.

## Installing Creo Mold Analysis (CMA) Component

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

You can install IntelMPI using following procedure:

1. Open a command window as an administrator.
2. Browse to the location of the `hydra_service.exe` file. If you install the Creo Mold Analysis component with the default installation path for the Creo Parametric application, the path is `C:\Program Files\PTC\Creo <version number>\Common Files\x86_win64\cma\Bin\IntelMPI\`.
3. To start the installation process of this component, run the following command:

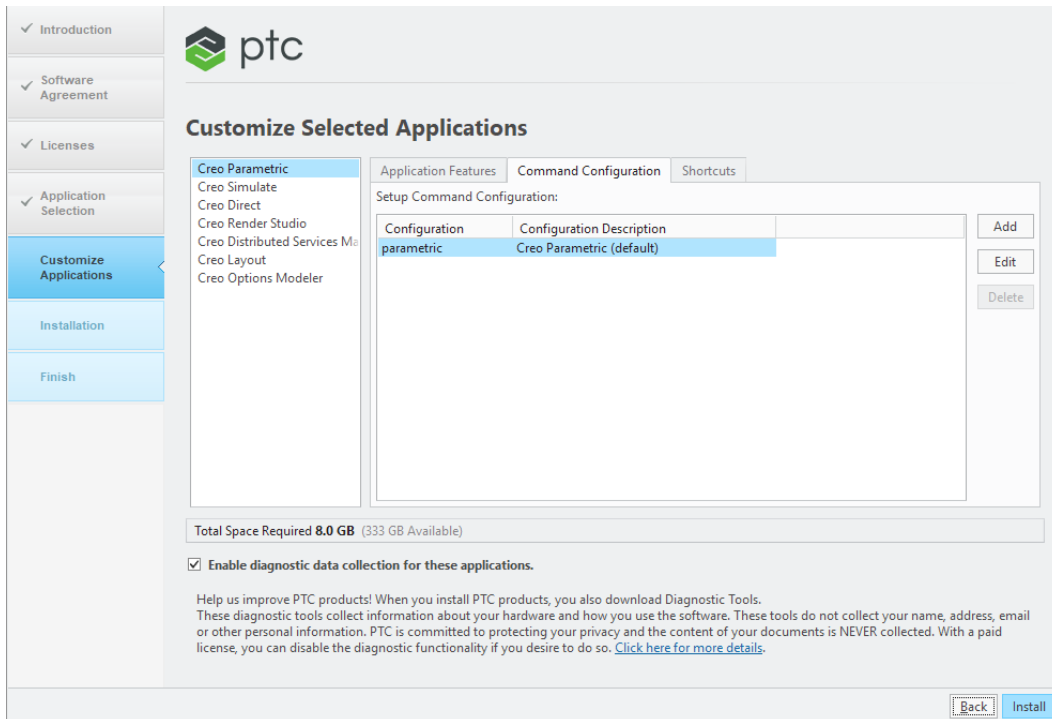
```
hydra_service.exe -install
```

Similarly, run the following command to uninstall the Creo Mold Analysis component:

```
hydra_service.exe -uninstall
```

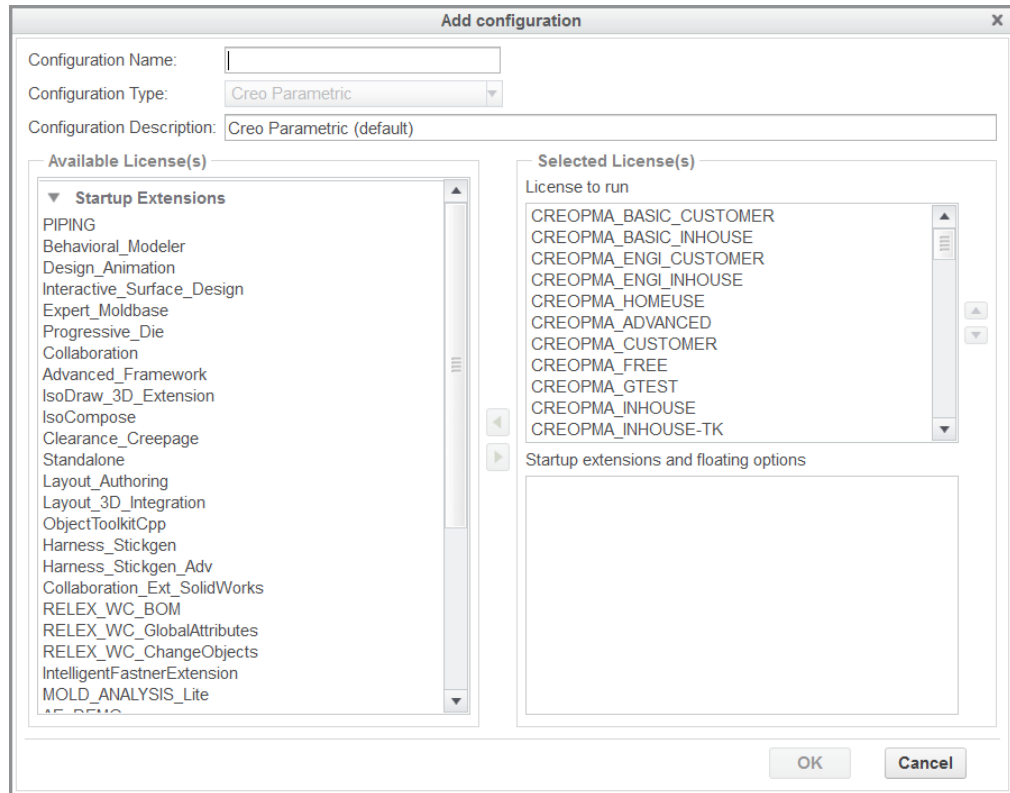
## Modifying License Configurations

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



Perform the following steps:

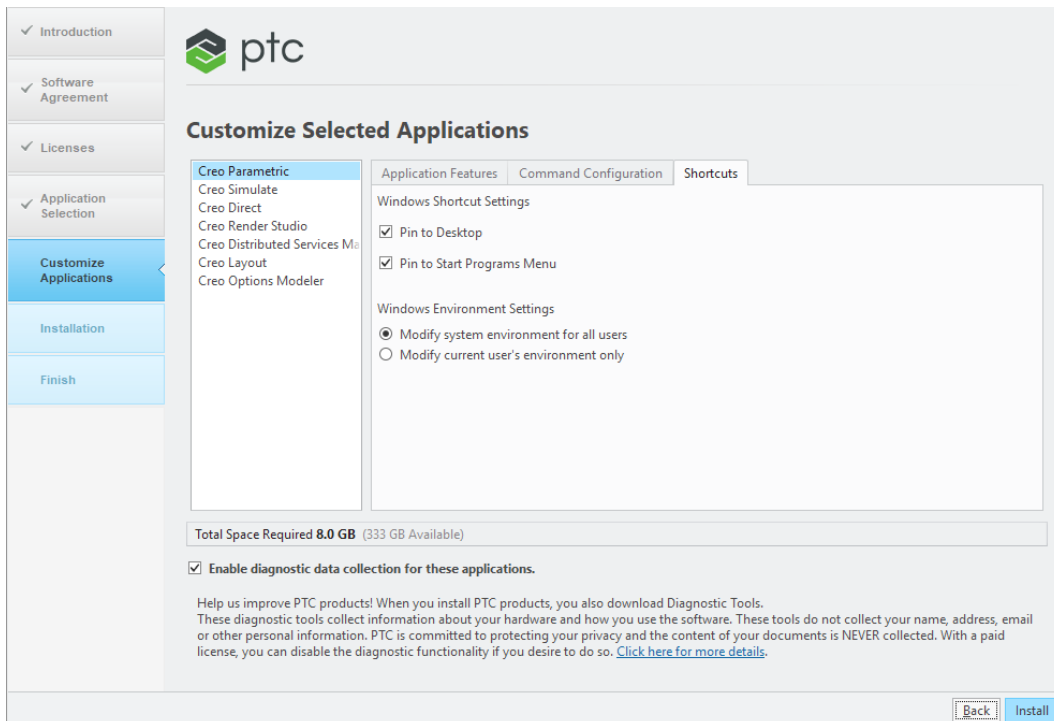
1. Click **Add** to add additional configurations. The **Add configuration** dialog box opens.



- a. In the **Configuration Name** box, specify a file name (\* . p s f) for the license configuration information. Use alphanumeric characters without spaces to define the configuration name.
  - b. From the **Configuration Type** list, select an application.
  - c. In the **Configuration Description** box, type a description for each of the license configurations that you have defined. This information is displayed on start up 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. Select the required license in this list and move it to the **Startup extensions and floating options** list.
  - e. Click **OK**. The newly added license configuration appears on the **Command 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. You can change any of 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**.

The screenshot shows the PTC software installation wizard. The main window is titled "Customize Selected Applications" and is for "Creo Simulate". The interface includes a sidebar on the left with navigation options: Introduction, Software Agreement, Licenses, Application Selection, Customize Applications (selected), Installation, and Finish. The main area has tabs for "Application Features", "Command Configuration", "Shortcuts", and "Licenses". Under the "Licenses" tab, there is a section "Set up the licenses for Creo Simulate" with a table of license types and selected licenses.

License Type	Selected License(s)
MEC_BASIC_UI	MECBASICUI MECBASICUI_INHOUSE MECBASICUI_License MECBASICUI_student
MEC_BASIC_ENG	MECBASICENG MECBASICENG_INHOUSE MECBASICENG_License MECBASICENG_student
STRUCT_UI	MECSTRUCUI MECSTRUCUI_INHOUSE
STRUCT_ENGINE	MECSTRUCENG MECSTRUCENG_INHOUSE
STRUCT_THERMAL_UI	MECTHERMUI MECTHERMUI_INHOUSE
STRUCT_THERMAL_ENGINE	MECTHERMENG MECTHERMENG_INHOUSE
MEC_ELITE_UI	MECELITEUI_License
MEC_ELITE_ENG	MECELITEENG_License

Below the table, it states "Total Space Required 8.0 GB (333 GB Available)". There is a checkbox for "Enable diagnostic data collection for these applications." with a help text: "Help us improve PTC products! When you install PTC products, you also download Diagnostic Tools. These diagnostic tools collect information about your hardware and how you use the software. These tools do not collect your name, address, email or other personal information. PTC is committed to protecting your privacy and the content of your documents is NEVER collected. With a paid license, you can disable the diagnostic functionality if you desire to do so. [Click here for more details.](#)"

At the bottom right, there are "Back" and "Install" buttons.

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** if you want to review information to make sure that your settings are correct.
2. Click **Install** to begin the installation. The **Installation Progress** screen displays the status.

The screenshot shows the PTC Application Installation progress screen. The left sidebar has a vertical list of steps: Introduction, Software Agreement, Licenses, Application Selection, Customize Applications, Installation (highlighted), and Finish. The main area displays the PTC logo and the title 'Application Installation'. Below the title, it says 'Installing the following applications:'. A table lists the applications, their status, and their progress percentage. Each row also has a refresh icon and a close icon.

Application	Status	Progress		
Creo 7 Platform Services	Complete	100 %	🔄	✕
Visual C++ Redistributables	Complete	100 %	🔄	✕
Creo Common Files	Installing Files	13 %	🔄	✕
Creo Parametric	Pending	0 %	🔄	✕
Creo Simulate	Pending	0 %	🔄	✕
Creo Direct	Pending	0 %	🔄	✕
Creo Render Studio	Pending	0 %	🔄	✕
Creo Distributed Services Manager	Complete	100 %	🔄	✕
Creo Layout	Pending	0 %	🔄	✕
Creo Options Modeler	Pending	0 %	🔄	✕
Creo View Express	Installing Files	-- %	🔄	✕
Diagnostic Tools	Pending	1 %	🔄	✕
Creo Thumbnail Viewer	Pending	1 %	🔄	✕

At the bottom right of the screen, there are 'Back' and 'Finish' buttons.



## Note

From Creo 6.0.0.0 onward, the Microsoft Visual C++ Redistributable Package is a part of the Creo installation.

## Note

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

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

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

## Installing and Uninstalling 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 140](#) for details.

## Starting 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 7.0 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.

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

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

## Specifying Environmental Variables

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

- Use the ENV or RUN entry to specify the value of an environment variable or to run an external application.

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

## Reading of Configuration Files at Startup

A 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\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.

## PIM Installer Log File

During the installation of the selected applications and components, the `pim_installmgr.log` file is generated in the My Documents folder. You can share this log file with PTC for further analysis of issues related to the PIM Installer.

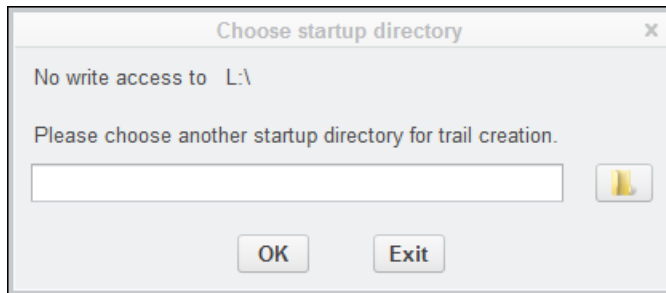
---

## 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 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 Services

### Installation Path of Utilities

Few of the required and optional utilities that you select to install with the Creo applications do not get installed at the default or custom installation path that you specify.

The following are the required and optional utilities and their respective installation paths for the Creo Platform Services:

With default location as C:\Program Files\

With custom path: <installdir>\Creo\

When you begin the installation of a Creo application, the latest version of the Creo Platform Services also gets installed. Older installed versions of Creo Platform Services are uninstalled, since the latest version works with older versions of Creo applications.

---

 **Note**

The behaviour described above is new in Creo 4.0. In the previous releases older versions of Creo Platform Agent (which is now superseded by Creo Platform Services) were not automatically uninstalled or upgraded with a newer version of the same package.

---

Creo Platform Services is a system level component which provides the following functionalities to Creo:

- It acts as a broker connecting other independent Creo components.
- Invokes the LearningConnector for Creo application.
- Provides embedded browser infrastructure and HTTP streaming services.
- Performs Creo monitoring for ExitLogger reporting.

See [Installing and Uninstalling Client-Side Components in Silent Mode on page 153](#) for instructions on performing a silent installation of Creo Platform Services.

If you access Creo applications from a shared location across the network, you do not need to install Creo Platform Services on your client machine.

If a Creo installation on a custom location (For Example, D:\PTC), is relocated to a network drive (For Example, D:\PTC is exported as \\SERVER\CREO4\_Share\), then Creo automatically finds the creoagent executable file relative to the install location.

Creo does not require the environmental variable CREO\_AGENT\_EXE\_PATH to find the path of creoagent.exe.

However, if the Creo installation structure in the network location is different from the default one, then the environmental variable CREO\_AGENT\_EXE\_PATH must point to the file creoagent.exe.

The environment variable, CREO\_AGENT\_LDP\_LIST must also be set and must point to the manifests folder on the Platform location. (For Example, \\SERVER\CREO4\_Share\Creo\Platform\4\manifests).

For local installation, the above environmental variables are not used.

If Creo is started from a network install and Creo Platform Services package (creoagent.exe and Creo platform) is installed locally and the environmental variables are not set, local installation takes precedence even if Creo finds the network installation. Using environmental variables, you can change the order of lookups.

---

 **Note**

When conditions listed below exist, set the path for the Creo Platform Services executable as the value of the `CREO_AGENT_EXE_PATH` environment variable:

- You are working on a 64-bit platform.
- You access a Creo application that is installed at the default installation path and is shared across a network.

---

This ensures all dependent add-ons of the Creo Platform Services work correctly.

## 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
- 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:

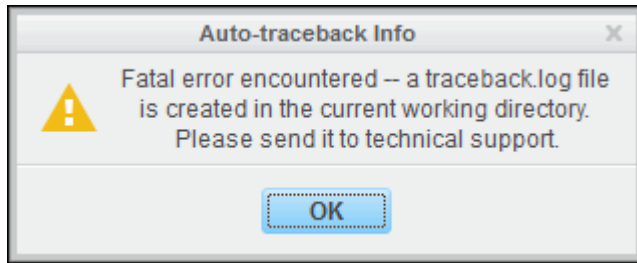
<b>Configuration Option</b>	<b>English</b>	<b>Metric</b>
<code>drawing_setup_file</code>	<code>&lt;creo_loadpoint&gt;\ \Common Files\ text\prodetail.dtl</code>	<code>&lt;creo_loadpoint&gt;\ \Common Files\ text\iso.dtl</code>
<code>format_setup_file</code>	<code>&lt;creo_loadpoint&gt;\ \Common Files\ text\prodetail.dtl</code>	<code>&lt;creo_loadpoint&gt;\ \Common Files\ text\iso.dtl</code>
<code>pro_unit_length</code>	<code>unit_inch</code>	<code>unit_mm</code>
<code>pro_unit_mass</code>	<code>unit_pound</code>	<code>unit_kilogram</code>

<b>Configuration Option</b>	<b>English</b>	<b>Metric</b>
template_designasm	<creo_loadpoint>\ \Common Files\ templates\ inlbs_asm_design.asm	<creo_loadpoint>\ \Common Files\  \templates\mmks_asm_design_abs.asm
template_drawing	<creo_loadpoint>\ \Common Files\ templates\c_drawing.drw	<creo_loadpoint>\ \Common Files\ templates\a3_drawing.drw
template_sheetmetalpart	<creo_loadpoint>\ \Common Files\ templates\ inlbs_part_sheetmetal_abs.prt	<creo_loadpoint>\ \Common Files\ templates\ mmks_part_sheetmetal_abs.prt
template_new_ecadasm	<creo_loadpoint>\ \Common Files\ templates\ inlbs_ecad_asm_abs.asm	<creo_loadpoint>\ \Common Files\ templates\ mmks_ecad_asm_abs.asm
template_solidpart	<creo_loadpoint>\ \Common Files\ templates\ inlbs_part_solid_abs.prt	<creo_loadpoint>\ \Common Files\ templates\ solid_part_mmks_abs.prt
todays_date_note_format	%Mmm-%dd-%yy	%dd-%Mmm-%yy
tolerance_standard	ansi	iso
tol_mode	nominal	nominal
weld_ui_standard	ansi	iso

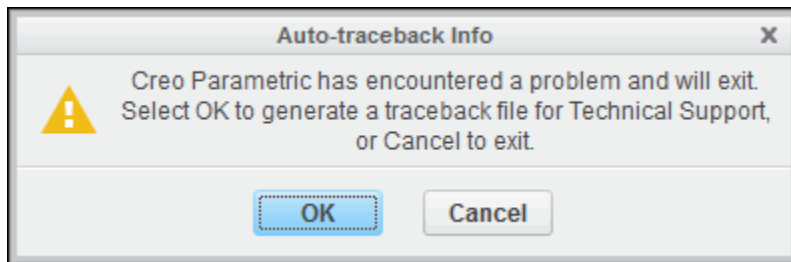
## Generating a Traceback Log File for Creo Parametric

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 `always`. 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 **Creo Parametric 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 **Creo Parametric Interface for CADD5** with Creo Parametric, a configuration file `config_cadd5.pro` is automatically saved in the `<creo_loadpoint>\Parametric\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_cadds_files yes
cadds_import_layer yes
intf_cadds_import_make_solid yes
intf_cadds_version 14
! extend_cvpath is similar to config-option search_path for
Creo Parametric.
! It allows to find user's CADD5 objects on import...
! extend_cvpath
```

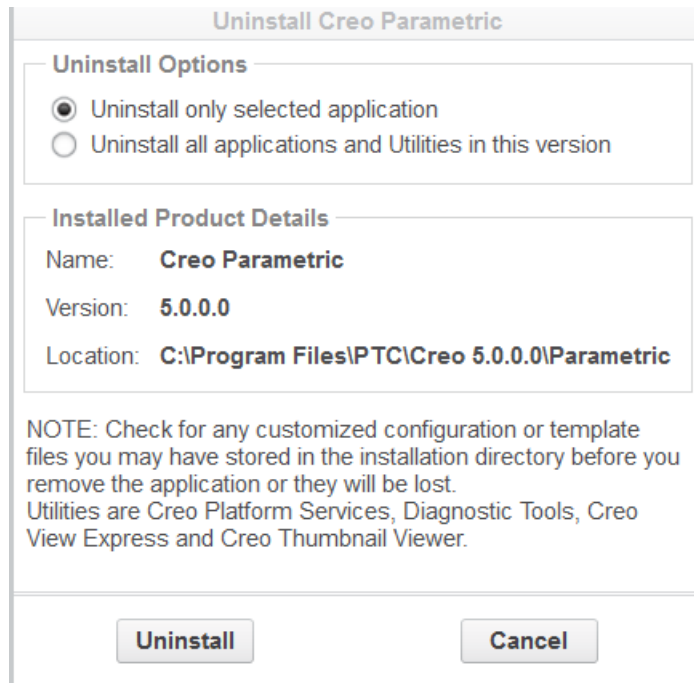
Check the `config_cadds.pro` file for the above configuration options. If required, copy the configuration options to your local `config.pro` file and update their values.

- Check whether the components listed below are installed:
  - **Creo Parametric Interface for CADD5**
  - MKS platform components. See [Installing MKS Platform Components on page 138](#) for installation instructions.
  - 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 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 version of the application you want to remove.
3. Right-click **Uninstall**. A confirmation dialog box opens as shown in the figure below :



Select one of the following **Uninstall Options**:

- **Uninstall only selected application**
- **Uninstall all applications and Utilities in this version**

Click **Uninstall** to remove either the selected Creo application or all applications and utilities for the selected version.

---

 **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 Accessing Creo Help

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This chapter provides an overview of the Help Center. It explains how to install and access Help for all the Creo applications from a local host computer, a network drive, or a Web server.

---

## Overview of the Help Center

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

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

## Browser Support for Creo Help


Creo Help supports following browsers:

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

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

## Accessing Help from within a Creo Application

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

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

The Help Center is accessed from the following locations:

- `http://support.ptc.com/help/creo/creo_<app>/<release number><lang>/`—The default if you do not install the Help Center and do not set the `PTC_<app_name>_HC_URL_7` environment variable. `<release number>` takes the form `r6.0`, `r7.0` etc. and `<app>` has the following values:
  - `pma` — Creo Parametric
  - `dma`—Creo Direct
  - `lcd`—Creo Layout
  - `sim`—Creo Simulate
  - `optm`—Creo Options Modeler
  - `render`—Creo Render Studio

- 
- Local host computer—The default if you install the Help Center but do not specify an alternate location, such as a network drive, for accessing Help.

---

 **Note**

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

---

## Choosing a Location for the Help Center Installation

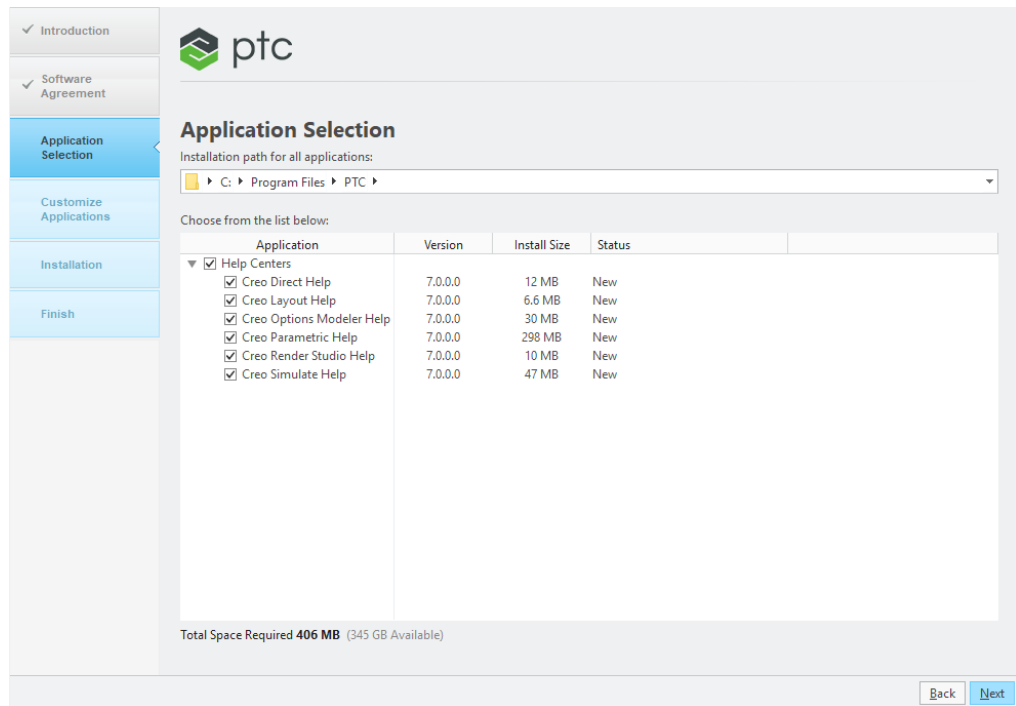
Using the Help installation media, you can install the Help Center for the Creo applications at one of the following locations:


- 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 installation media in your shipment, proceed with the installation as described. You can also download the Help installation image 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 64](#) to access the Help Center from the Creo application.

1. Start Windows Explorer, browse to the installation media drive, and double-click `setup.exe`. Alternatively, download the installation media as described in the previous paragraph. The PTC Installation Assistant begins.
2. Perform steps 1 through 4 as described in [Installing the Creo Applications: Simple Installation Process on page 33](#).
3. Click **Next**. The **Application Selection** screen appears.



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

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

- a. On the **Application Features** tab, select the Help Center languages to install.
- b. On the **Shortcuts** tab, specify the shortcut preferences for the Creo Help startup commands and other Windows preferences. See [Configuring Windows Shortcuts on page 46](#) for information on how to perform these operations.
7. Click **OK**.
8. Click **Install** on the **Application Selection** screen. The **Installation Progress** screen shows the status of the installation. You can click **Cancel** at any time to cancel the installation.
9. Click **Finish**. The Help Center is installed in the <help\_install\_path>\PTC\Creo version\help\creo\_help\_<app\_name> directory.

For example, if you specify the default path, the Help Center is installed in the C:\Program Files\PTC\Creo version\help\creo\_help\_<app\_name> directory.

---

## Enabling the Help Center from a Network Drive

To enable the Help Center that is installed on a network drive, you must configure the 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 7.0.0.0\help, share the folder C:\Program Files\PTC\Creo 7.0.0.0 as:  
\\machine03d\Creo 7.0.0.0.
2. On a client computer where you want to access the Help Center, map the shared folder to a local drive, say Q. After mapping, the folder with the Help installation files appears as Q:\Help\help\creo\_help\_<app\_name>.
3. Set the PTC\_<app\_name>\_HC\_URL\_7 environment variable for the Creo applications as listed:
  - Creo Parametric—PTC\_PMA\_HC\_URL\_7
  - Creo Direct—PTC\_DMA\_HC\_URL\_7
  - Creo Layout—PTC\_LCD\_HC\_URL\_7
  - Creo Simulate—PTC\_SIM\_HC\_URL\_7
  - Creo Options Modeler—PTC\_OPTM\_HC\_URL\_7
  - Creo Render Studio—PTC\_RENDER\_HC\_URL\_7
  - a. Click **Start ► Control Panel ► System ► Advanced system settings**. The **System Properties** dialog box opens.
  - b. Click the **Advanced** tab and then click **Environment Variables**.
  - c. Click **New** under the User variables area. The **New User Variable** dialog box opens.
  - d. Specify the environment variable and its value in the boxes listed below. For example, for Creo Parametric Help, set the environment variable as follows:
    - **Variable name**—PTC\_PMA\_HC\_URL\_7
    - **Variable value**—Q:\Help\help\creo\_help\_pma
  - e. Click **OK** and again click **OK** on the **System Properties** dialog box.
4. Start the Creo application and access Help.



---

## Installing the Help Center on a Web Server

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

1. Perform steps 1 through 4 as described in [Installing the Help Center on page 62](#).
2. Specify <DocumentRoot> as the installation path.
3. Click **OK** and click **Install** on the **Application Selection** screen.

The selected application help is installed in the <DocumentRoot>\Creo <version number>\help\creo\_help\_<app\_name> directory.

4. To distribute the Help Center files from the <DocumentRoot>\Creo <version number>\help\creo\_help\_<app\_name> directory for context sensitive Help access, set the PTC\_<app\_name>\_HC\_URL\_6 environment variable for the Creo applications as listed:
  - Creo Parametric—PTC\_PMA\_HC\_URL\_7
  - Creo Direct—PTC\_DMA\_HC\_URL\_7
  - Creo Layout—PTC\_LCD\_HC\_URL\_7
  - Creo Simulate—PTC\_SIM\_HC\_URL\_7
  - Creo Options Modeler—PTC\_OPTM\_HC\_URL\_7
  - Creo Render Studio—PTC\_RENDER\_HC\_URL\_7
  - a. Click **Start** ► **Control Panel** ► **System** ► **Advanced system settings**. The **System Properties** dialog box opens.
  - b. Click the **Advanced** tab and then click **Environment Variables**.
  - c. Click **New** under the User variables area. The **New User Variable** dialog box opens.
  - d. Specify the environment variable and its value in the boxes listed below. For example, for Creo Parametric Help, set the environment variable as follows:
    - **Variable name**—PTC\_<app\_name>\_HC\_URL\_7
    - **Variable value**—<Help\_Center URL>

For example, for a Help Center installed on Apache server configured on port number 8080, the Help Center URL is http://<server\_name>:8080/Creo <version number>/help/creo\_help\_<app\_name>.
  - e. Click **OK** and again click **OK** on the **System Properties** dialog box.

---

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

For example, if you have installed Creo Parametric Help Center on the local host computer at the default path, you can copy the Help Center directory `C:\Program Files\PTC\Creo <version number>\help\creo_help_pma` to the <DocumentRoot> directory. Specify the appropriate URL as the value for the `PTC_PMA_HC_URL_7` variable to access the context-sensitive Help.

---

 **Note**

If you specify <DocumentRoot> directory as the location for the installation of the Help Center, you can access the context-sensitive help directly from this directory. Set the Help Center URL as the value of the `PTC_<app_name>_HC_URL_7` environment variable to access the context-sensitive Help through the Web server URL.

---

## Viewing the Help Center Independent of a Creo Application

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

```
<creo_help_install_path>\help\creo_help_<app>\  
<language>\index.html
```

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

```
<creo_help_install_path>\help\creo_help_pma\usascii\  
index.html
```

The Help Center opens in the browser window.

---

 **Note**

Due to limitations of the Chrome browser, the search functionality is not available when viewing a local installation of the Help Center from a Chrome browser. Use other browsers to view the locally installed Help.

---

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

---

```
http://<server_name>:<port_number>/Creo <version number>/help/creo_help_<app_name>/<language>/index.html
```

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

```
http://<server_name>:8080/Creo version/help/creo_help_pma/usascii/index.html
```

A Help Center opens in the browser window.

---

 **Note**

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

---

## 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><date\_code>** for any of the Creo applications.
3. Click **Uninstall**. A confirmation dialog box opens.
4. Click **Yes** to uninstall the Help Center for the application.

---

 **Note**

If you have uninstalled the Help Center on the host computer or have not set the `PTC_<app_name>_HC_URL_7` environment variable, you can access the Help Center from PTC.com from within a Creo application. Additionally, if you uninstall or remove the Help Center from the Web server, you must remove or unset the `PTC_<app_name>_HC_URL_7` environment variable.

---

---

# 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 22](#) 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 installation media (thumb drive) 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 25](#) 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 folder remain unchanged. The updated files relevant to the customization are added to the <version> folder.
  - Install a new maintenance release—After the installation, a new <version> folder corresponding to the datecode of the maintenance release is created within the <creo\_loadpoint> folder. This folder contains all the files for the maintenance release.
  - Upgrade an existing installation—After the upgrade, the highest version of the release that is installed is removed and the new maintenance release is installed in the installation path that you specify. If there is no installed version at the specified installation path, then the new maintenance release is installed at that path.

## Customizing an Existing Installation of a Creo Application

Customizing an existing installation of a Creo application implies enabling or disabling diagnostic 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 installation media 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 25](#) 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 PTC Diagnostic reporting** check box using the **Application Features** tabbed page. See [Defining the Installation Components on page 41](#) for details.

- 
- Add, edit, or delete a license configuration using the **Command Configuration** tabbed page. See [Modifying License Configurations on page 44](#) for details.
  - Configure the Windows shortcut preferences and path settings using the **Shortcut** tabbed page. See [Configuring Windows Shortcuts on page 46](#) 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 new software** on the Welcome screen. Click **Next**. The **License Identification** screen appears.
2. Generate or update your license file. If required, PTC License Server is also updated based on your license file. Click **Next**. The **Application Selection** screen appears.
3. Select an application to install.
4. Specify an installation folder.
5. To customize the installation for specific application features, license configurations, or Windows shortcut preferences, click **Customize**. Use the **Application Custom Setting** screen that opens for the customization.
6. After you select all the options, click **Install** on the **Application Selection** screen. The common files folder created for the previous release gets replaced with common files folder for the release you are installing.
7. Click **Finish** to complete the installation and close the Assistant.

## Upgrading an Existing Version of an Installation

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

---

 **Note**

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

---

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

---

 **Note**

If an earlier version of the release exists at the specified installation path, Update from <version> appears in the **Status** column. However, if an installed version does not exist at the specified installation path, the status appears as *New*. The name of the installation folder is changed when you upgrade. For example, the folder name *Creo 7.0.0.0* changes to *Creo 7.0.1.0* for the first maintenance release of the software.

---

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


---

 **Note**

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

---

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

4. Select the **Back up my modified files during installation** check box if you have customized files in the *Common files* directory and you want to retain these configurations and settings when you upgrade to the new version. Click  to



---

view a complete list of files and folders that are backed up and restored when the upgrade completes.

---

 **Note**

Note that when you select this option you might not get updates that PTC has made to the default configuration settings and libraries in the updated version. However, the updated configuration and library files and folders are made available in the `Common files` directory in a folder named `ORIG`.

---

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.
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 86](#) for details.

Based on the duration for which you can purchase a license, licenses are classified as the following types:

- Perpetual license—A permanent license that you do not need to renew once you purchase it.
- Subscription license—A license that is valid for a fixed period. Once the license period expires you can renew the license for an additional fixed period.

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

---

## Renewing a Subscription License

A subscription license is valid for a fixed period of time. When the license expires, or some days prior to expiry of the license, you can renew the subscription license for an additional fixed period of time. In the case of a node locked license, or a single server license a warning message is issued a fixed number of days before the license is due to expire.

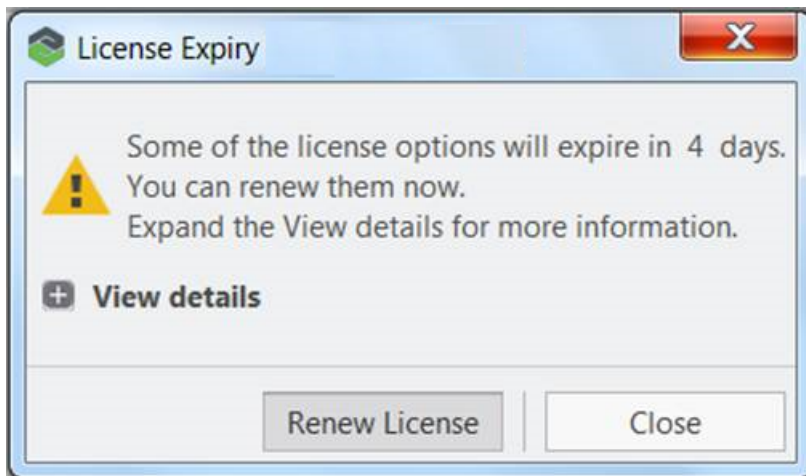
---

### Note

In the case of a single server license the warning message is displayed only if the Creo application is also running on the server on which the single server license is installed.

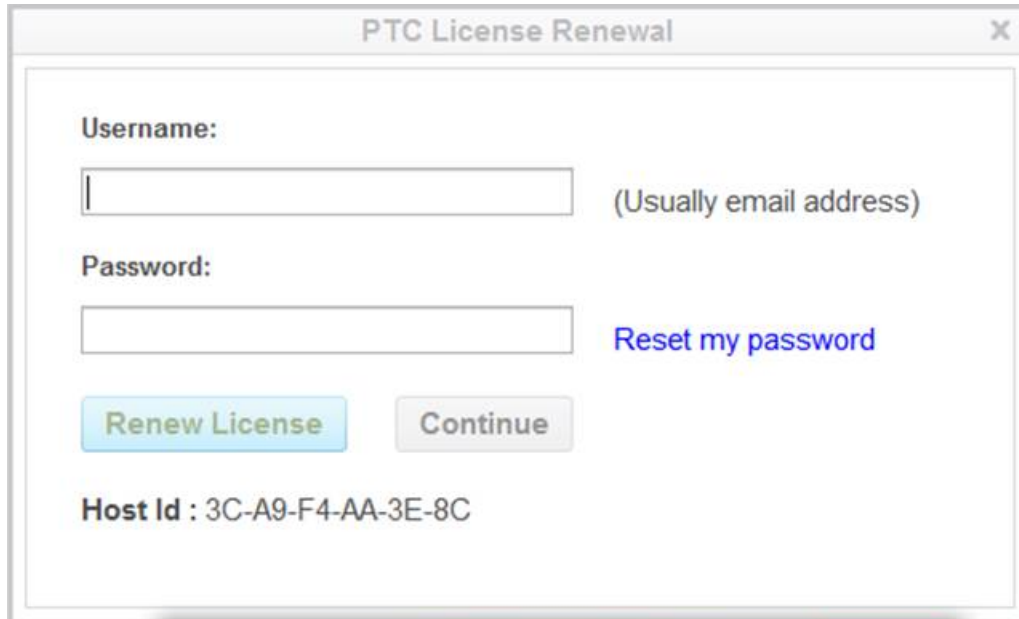
---

You can set the number of days prior to expiry, when you want the warning message to be displayed using the configuration option `daystoshow_expirydialog`. The default value is 15 days. The number of days prior to the license expiry, the following message is displayed when you invoke the software:



---

Click **Renew License** to initiate the process of subscription license renewal. You are redirected to the following form:



PTC License Renewal

Username:  
 (Usually email address)

Password:  
 [Reset my password](#)

Host Id : 3C-A9-F4-AA-3E-8C

Specify your **Username** and **Password** and click **Renew License** to renew your license online. The software redirects you to [www.ptc.com](http://www.ptc.com). You can renew your license for an additional subscription period. The license file is automatically updated on your computer or server (in the case of single server license).

## 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.

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---

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 11.16.2.1 is shipped with Creo 7.0.

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- 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:

```
ptcstatus
```

- Normal mode—Run the `Ptcstatus` utility at the command line without any arguments.
- Verbose mode—Run the `Ptcstatus` utility at the command line with the `-v` argument.

```
ptcstatus -v
```



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

### Ptcstatus Normal Mode

Displaying status for license file: 7788@static  
License Server: static  
ptc\_d FlexNet Publisher version: 11.16.2.1

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

\* = 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: 11.16.2.1

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

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

---

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

**Original license file:**

```
INCREMENT PROE_Engineer ptc_d 33.0 01-nov-2011 1 \  
BC24AFC5B76BB74C9366 VENDOR_STRING="VSVER=2.0 \  
LO=(0,3,6,7,10,17,32,34,40,45,48,51,55,61,62, \  
66,69,71,73,77,97,106,108,115,126,127,128,133,135,137, \  
158,163,188,191,210)" SUPERSEDE vendor_info="VIVER=1.0 \  
EXTERNAL_NAME=" ISSUER=PTC ISSUED=01-sep-2006 \  
NOTICE="PTC_customer" SN=SCN912344,SCN765431 SIGN2="169C A28A \  
E97F E96E 0A3E 563B FDEB 4510 829E 4BF4 25D3 2394 \  
0444 2FD4 6C23 0168 A8A5 AEBE 54B0 1FF6 B79B DC75 \  
2014 A278 33CC 1B90 8647 6A12 F4D6 45BF"
```

**Changed license file:**

```
INCREMENT PROE_Engineer ptc_d 33.0 01-nov-2011 1 \  
BC24AFC5B76BB74C9366 VENDOR_STRING="VSVER=2.0 \  
LO=(0,3,6,7,10,17,32,34,40,45,48,51,55,61,62, \  
66,69,71,73,77,97,106,108,115,126,127,128,133,135,137, \  
158,163,188,191,210)" SUPERSEDE vendor_info="VIVER=1.0 \  
EXTERNAL_NAME=My_License_Name ISSUER=PTC ISSUED=01-sep-2006 \  
NOTICE="PTC_customer" SN=SCN12344,SCN765431 SIGN2="169C A28A \  
E97F E96E 0A3E 563B FDEB 4510 829E 4BF4 25D3 2394 \  
0444 2FD4 6C23 0168 A8A5 AEBE 54B0 1FF6 B79B DC75 \  
2014 A278 33CC 1B90 8647 6A12 F4D6 45BF"
```

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

## PTC License Server

Flexera Software, Inc.'s FlexNet Publisher license management software is integrated with PTC software. For more information, visit [www.flexerasoftware.com](http://www.flexerasoftware.com).

---

 **Note**

FlexNet Publisher Version 11.16.2.1 is shipped with Creo 7.0.

---

## 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**—Creo 7.0 allows license borrowing with FlexNet Publisher.

## 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 7.0 license can run Creo Parametric 6.0 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 11.16.2.1 is shipped with Creo 7.0.

---

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

## Understanding Timeout Parameters

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

You can reconfigure the `TIMEOUTALL` parameter within a specified range, as described in the next section, [Changing the Inactivity Timeout Parameter on page 85](#).

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

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

## Changing the Inactivity Timeout Parameter

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

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

## Suppressing the Regained Network License Dialog Box 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

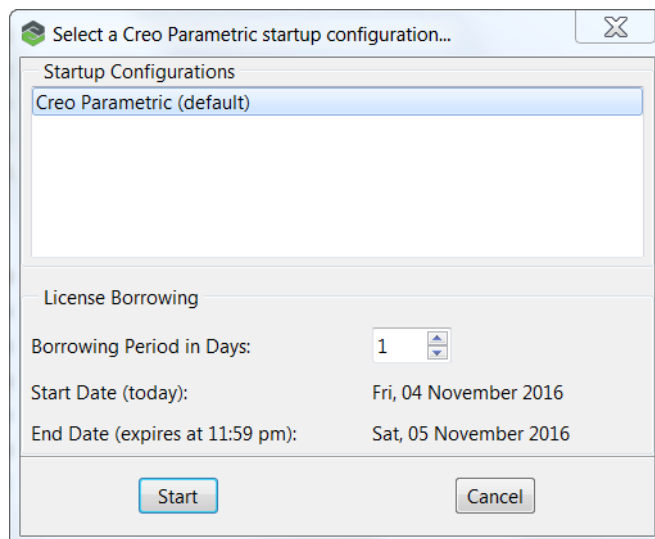
If you may possibly want an early return of a borrowed license, set the `PRO_ALLOW_EARLY_RETURN` environment variable prior to borrowing the license from the license server. See [Early Return of a Borrowed License on page 89](#) for more information.

---

## 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**

Use the environment variable `LM_BORROW_DURATION` to borrow a license.

- `LM_BORROW_DURATION`  $\leq$  29 days for a desktop perpetual license.
  - `LM_BORROW_DURATION`  $\leq$  179 days for a desktop subscription license.
- 

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.

---

### Note

The environment variable `PRO_ALLOW_EARLY_RETURN` is set to `true` by default in the Creo application specific PSF file.

If you frequently return licenses early, retain the value of the `PRO_ALLOW_EARLY_RETURN` environment variable as `true` in the PSF file. Otherwise, set it to `false`.

---

### 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 <version number>\Common Files\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	<p>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.</p> <p> <b>Note</b></p> <p>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.</p>
-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 10.8.x and later. Use `lmadmin` for the following functions:

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

See the chapter `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.

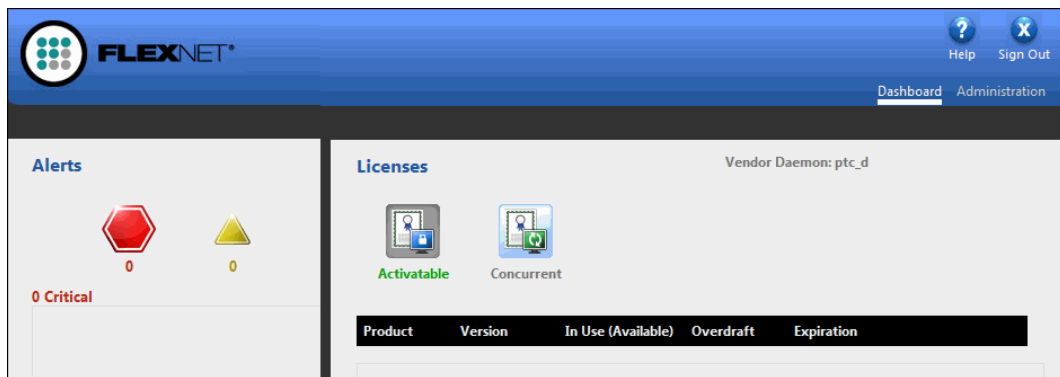
## Installing PTC License Server Based on lmadmin

Follow the instructions given in the section [Generating or Updating Licenses on page 25](#) 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.

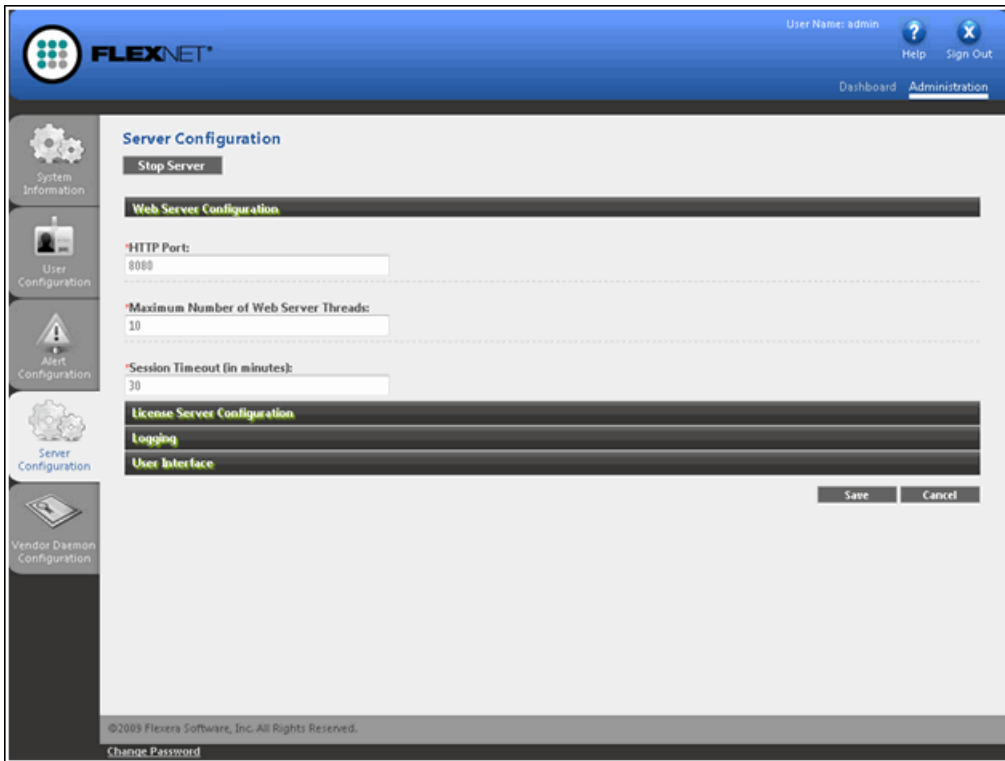
The FLEXnet License Administrator Web interface has two main pages: the **Dashboard** and the **Administration** pages. Click the **Help** button for information on the FLEXnet License Administrator interface elements.



- **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.

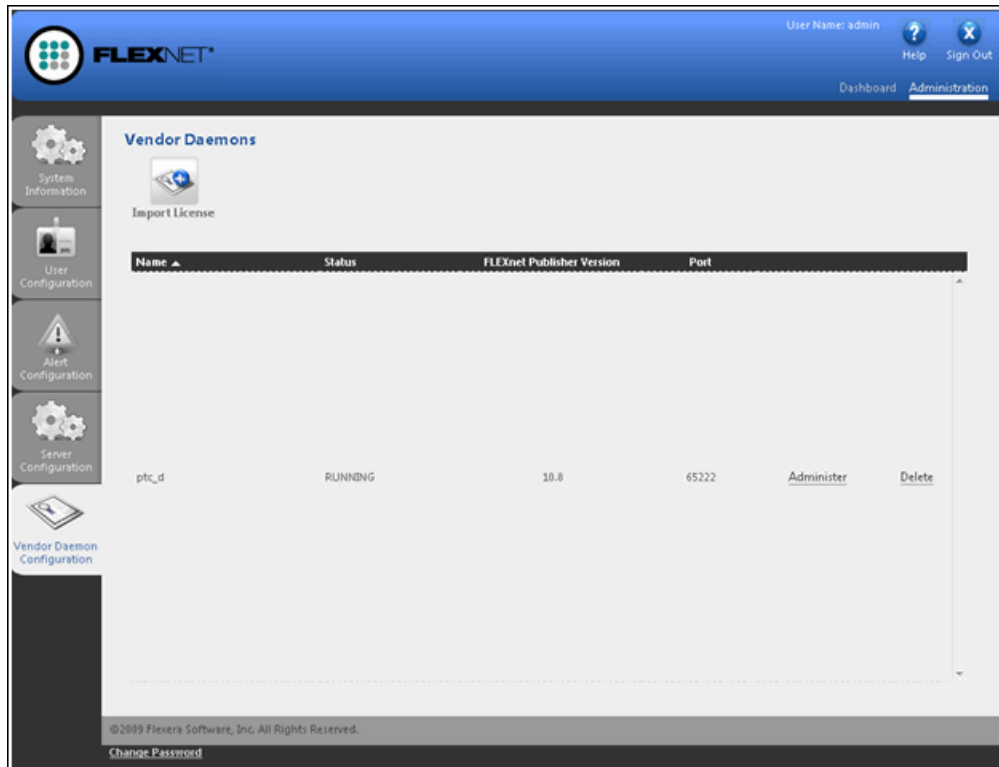
## Controlling the License Server Manager Settings

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.



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

## Configuring the License Server Alerts on Windows

A Java-based alert configuration is available with the FlexNet Publisher server version 11.10 onward. This alert utility is installed with the FlexNet installation (<FLEXnet\_Installation\_Directory>/example/alerter/) when you install the lmadmin based license server from the Creo installation media.

You can configure this utility to send alert notifications emails to the administrator when any of the set critical or important events occur.

### Note

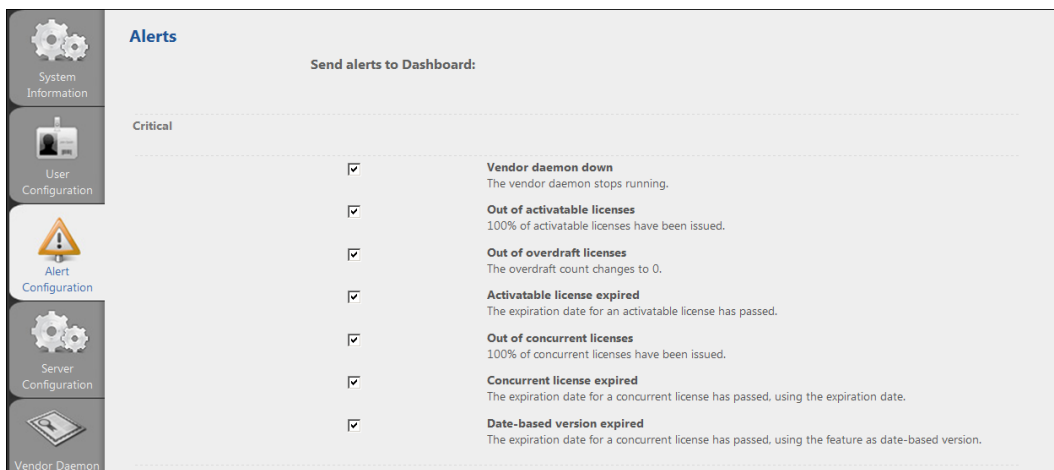
The license server must be up and running with an active vendor daemon to use this alert utility.

You can start the alert utility from the command line on the license server with administrator rights as follows:

```
C:\Program Files\PTC\FLEXnet Admin License Server\
examples\alerter>"C:\Program Files\Java\jdk1.7.0_75\
bin\java" -classpath "lib\alerter.jar;lib\*" alerter.bat
-port "8080" -user "admin" -password "<flexnet_admin_pwd>"
-smtpUser "<user_id>" -smtpPassword "<password>"
-smtpServer "<mail_server_address>" -toAddress "<e-mail_
ID>" -fromAddress "<e-mail_ID>" -interval "60"
```

Additionally you can configure the alert utility through the Web service as follows:

<http://<serverhostname>:8080/>



Click the **Alert Configuration** tab and select any of the following critical or important events for which the alert notifications should be sent:

- Critical alerts
  - Vender daemon down – The vendor daemon has stopped running.
  - Out of activatable licenses – All the available licenses that you can activate are issued.
  - Out of overdraft licenses – The overdraft count of licenses is zero.
  - Activatable license expired – The expiry date for a license that you can activate has passed.
  - Out of concurrent licenses – All the available concurrent licenses are issued.
  - Concurrent license expired – The expiry date for a concurrent license has passed.
  - Date-based version expired – The expiry date for a concurrent license that uses the feature as date-based version, has passed.
- Important alerts



- 
- Activatable threshold exceeded – The defined percentage of licenses that you can activate are issued.
  - Overdraft license issued – The vendor daemon has issued an overdraft license.
  - Activatable license expiring – The defined days are left before an active license expires.
  - Subscription license expiry notice — The number of days prior to expiry of the subscription license.
  - Concurrent threshold exceeded – The defined percentage of concurrent licenses are issued.
  - Concurrent license expiring – The defined days are left before a concurrent license expires.
  - Date-based version expiring – The defined days are left before any license that uses the feature as date-based version expires.

---

 **Note**

The critical alerts are marked with the alert type in the subject line of the email. However, the MKS Platform do not have any text in the subject line of the emails. Such emails go into the Junk E-mails folder.

---

# 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 the Creo applications 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
ptchostid	Returns the PTC Host ID with required formatting used for FlexNet Publisher license codes.
parametric	Runs Creo Parametric. Can be user-defined.
direct	Runs Creo Direct. Can be user-defined.
simulate	Runs Creo Simulate. Can be user-defined.
layout	Runs Creo Layout. Can be user-defined.
optionsmodeler	Runs Creo Options Modeler. Can be user-defined.
photorender	Runs Creo Render Studio. Can be user-defined.
ptcflush	Releases licenses. This may be necessary if there are network problems. Formerly called proflush.

Command	Function
ptcstatus	Displays information on the current usage and availability of licenses.
ptcborrow and <creo_app_ borrow>	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
lmtools	Runs the PTC License Management tools utility <code>lmtools.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

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

---

## IPV6 Support for Creo

Creo 6.0.0.0 and later have complete support for Internet Protocol version 6 (IPV6). In systems that have both IPV4 as well as IPV6 addresses if you want Creo to use the IPV6 protocol as the default protocol for network addressing, set the environment variable `PTC_IPV6_MODE` to `yes`.

## Controlling Access to Toolkit Applications

From Creo 6.0.0.0 onwards, you can sign all Creo Parametric TOOLKIT applications using the standard Microsoft SignTool application. For more information refer to the “Creo Parametric Toolkit Users Guide”.

From Creo 7.0.0.0 onwards, Creo checks signatures of Toolkit applications at load time.

The following configuration options control whether to always allow the users or administrator to determine whether signed or unsigned Creo Toolkit applications should be allowed to always run, to never run, or to prompt the user before running:

- `open_prot_k_unsigned_apps`—Controls whether unsigned applications can be loaded in a Creo session. It can have the following values:
  - Always—Always loads unsigned applications.
  - Never—Never load unsigned applications.
  - Prompt—Asks the user whether to load unsigned applications.
- `open_prot_k_signed_apps`—Controls whether signed applications can be loaded in a Creo session. It can have the following values:
  - Always—Always loads signed applications.
  - Never—Never load signed applications.
  - Prompt—Asks the user whether to load signed applications.

## Supported Graphics Modes

For graphics support information, refer to the PTC Hardware Configuration Notes at [PTC\\_Creo\\_Future\\_Platform\\_Support\\_Summary.pdf](#).

## General Plotter Information

PTC supplies software that writes plot files in plotter format. You can specify a plot command in the `config.pro` file. The physical connection, however, is the responsibility of the computer and plotter vendors, because they are hardware and system software experts. Depending on the hardware configuration, the system software may require a different setup. If you have difficulty in set up, contact the customer support department of your hardware vendor.

---

 **Note**

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

---

## 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\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

<b>Font Name</b>	<b>File Name</b>	<b>Supports Kerning</b>
CG Omega	cgomg.ttf	No
CG Omega Bold	cgomgb.ttf	No
CG Omega Bold Italic	cgomgbi.ttf	No
CG Omega Italic	cgomgbit.ttf	No
CG Times	cgtime.ttf	No
CG Times Bold	cgtimebd.ttf	No
CG Times Bold Italic	cgtimebi.ttf	No
CG Times Italic	cgtimeit.ttf	No
CG Triumvirate	trium.ttf	Yes
CG Triumvirate Bold	triumb.ttf	Yes
CG Triumvirate Bold Italic	triumbi.ttf	Yes
CG Triumvirate Condensed Bold	triumcb.ttf	Yes
CG Triumvirate Italic	triumi.ttf	Yes
Garamond Kursiv	garamdi.ttf	No
Garamond Halbfett	garamdb.ttf	No
Garamond Antiqua	garamd.ttf	No
Garamond Kursiv Halbfett	garamdbi.ttf	No
Garth Graphic	gargra.ttf	Yes
Garth Graphic Black	gargrabl.ttf	Yes
Garth Graphic Bold Italic	gargrabi.ttf	Yes
Garth Graphic Italic	gargrai.ttf	Yes
Grotesque MT	grotesq.ttf	No
Grotesque MT Bold	grotesqb.ttf	No
Microstyle Extended	microex.ttf	Yes
Microstyle Extended Bold	microexb.ttf	Yes
Neographik MT	neograph.ttf	Yes
SackersEnglishScript	sackengs.ttf	No
Shannon	shanno.ttf	Yes
Shannon Bold	shannob.ttf	Yes
Shannon Extra Bold	shannoeb.ttf	Yes
Shannon Oblique	shannoo.ttf	Yes



Font Name	File Name	Supports Kerning
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
<code>font_chinese_cn</code>	Simplified Chinese
<code>font_chinese_tw</code>	Traditional Chinese
<code>font_hebrew</code>	Hebrew
<code>font_japanese</code>	Japanese
<code>font_korean</code>	Korean
<code>font_latin</code>	English, French, German, Italian, Spanish, Brazilian Portuguese

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

## Available System Fonts for Menu Items

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

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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`).

---

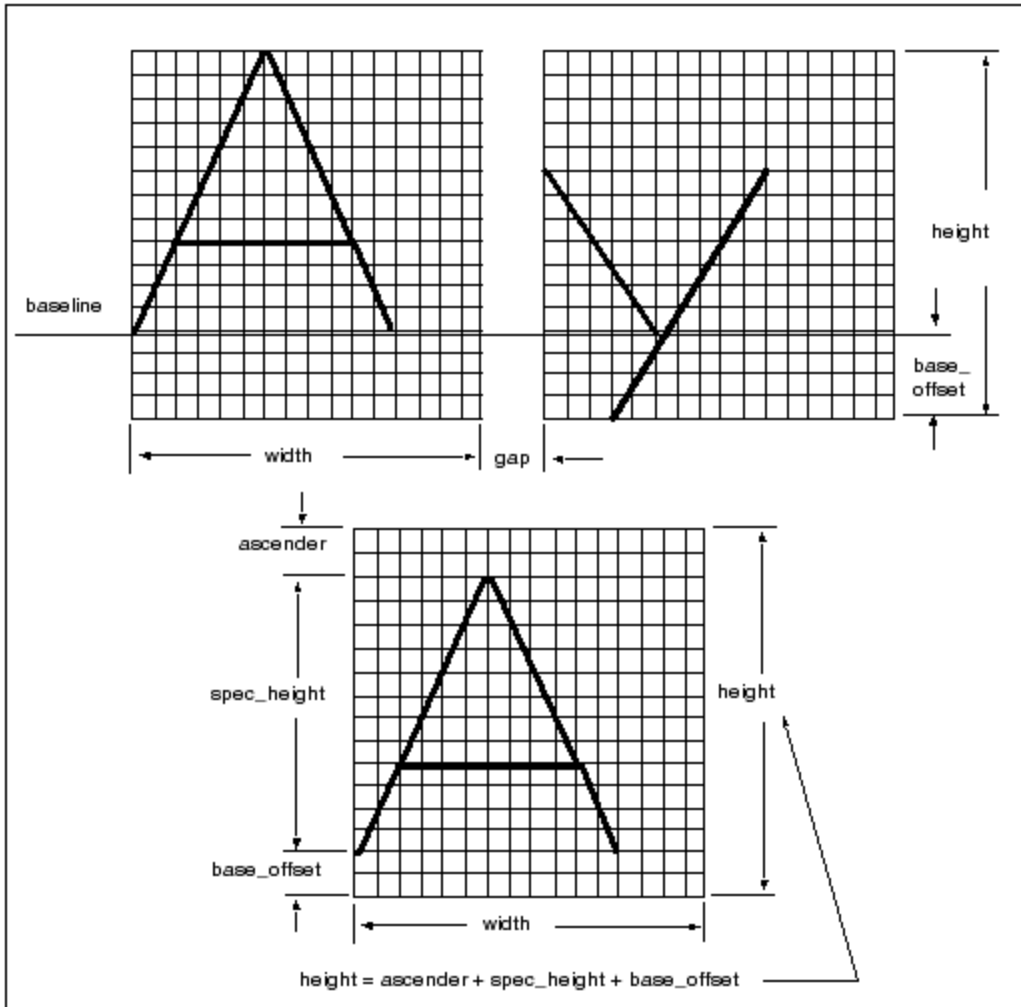
 **Note**

The filled font has the same codeset as ASCII. Therefore, only one or the other can exist in the `font.ndx` file.

---

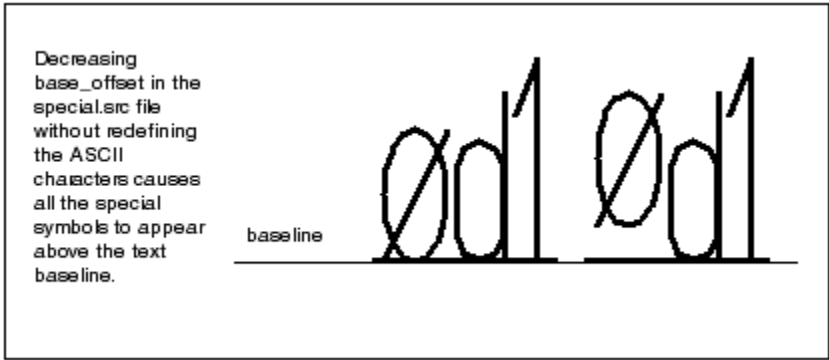
- `polygons`—Turns on character filling.
- If you set it to 1, all closed polygons in the character definitions are filled (refer to the section [Filled Font Definition on page 115](#)).
- If you do not specify this parameter, the system uses the default value 0.
- `fixed_width<n>`—Can be 0 or 1.
- If 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.

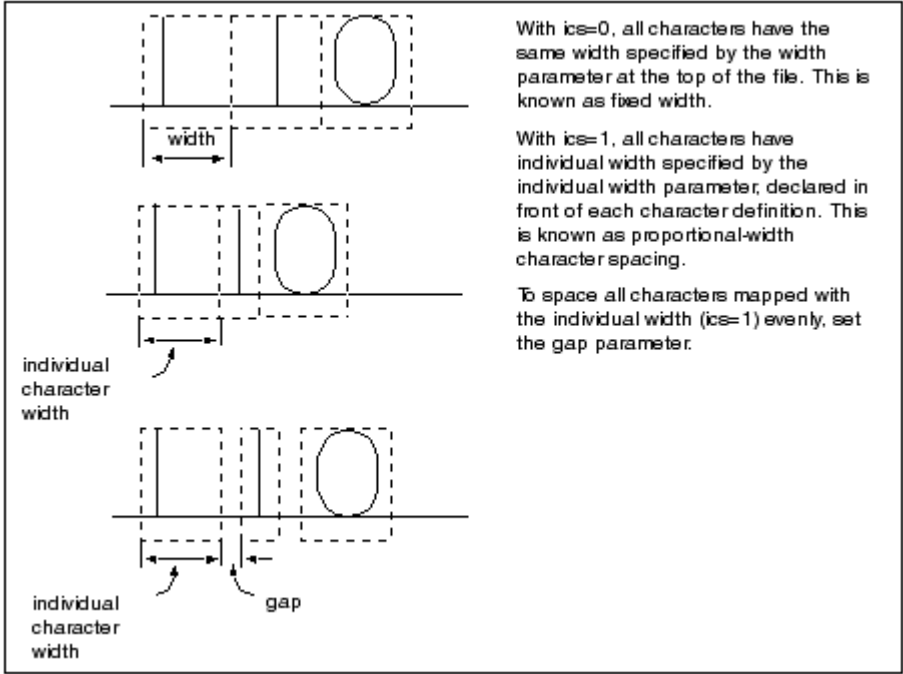


When you modify text parameters, consider how the font will appear in text with other fonts. Fonts used in the same text line (such as ASCII and LATIN\_1 for European languages, or ASCII and special) must be compatible. Altering one font might create inconsistency in appearance and placement.

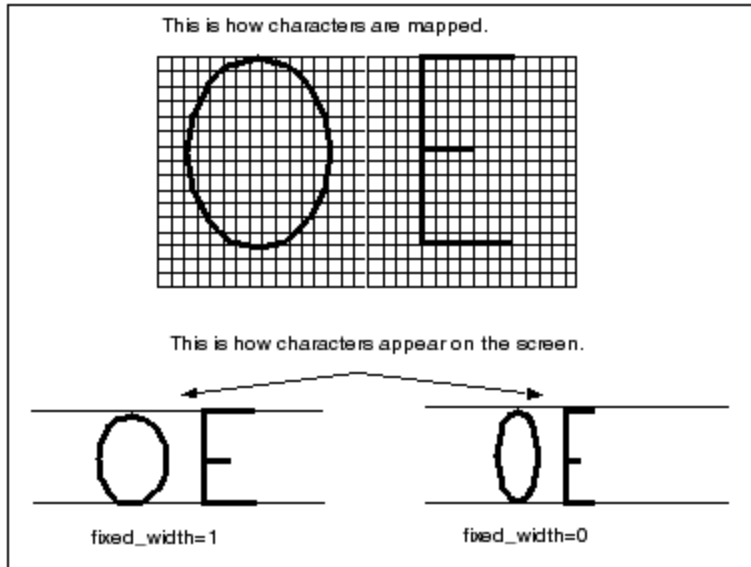
For example, changing the `base_offset` value at the top of the font file affects the positioning of all the font characters, relative to the text baseline, as shown in the following figure.



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

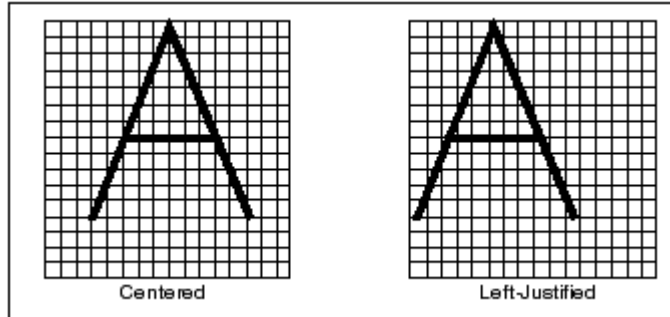
The drawing definition of characters and special symbols in the font files is done with the following commands (where <X> and <Y> are integers):

- [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



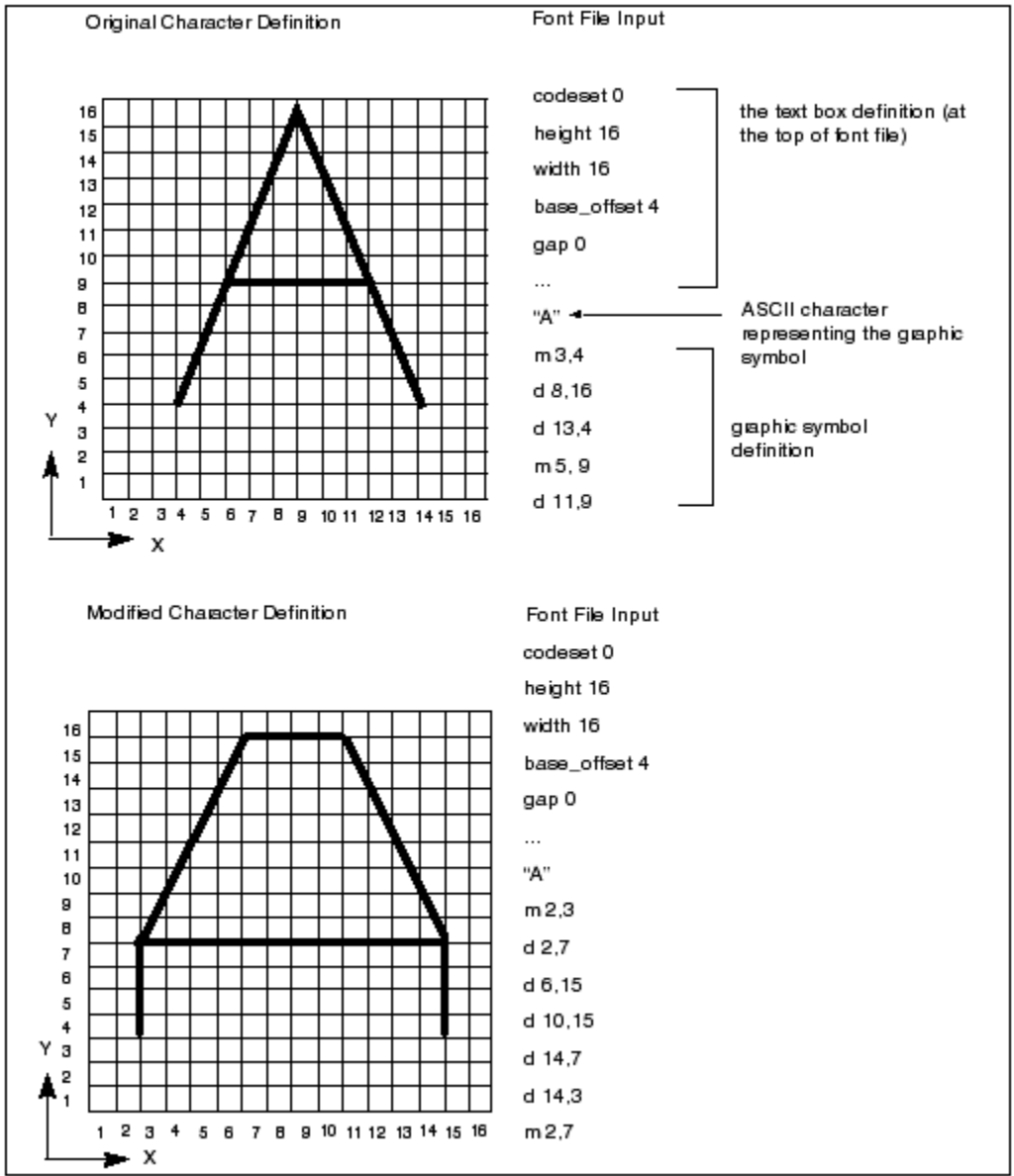
---

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.

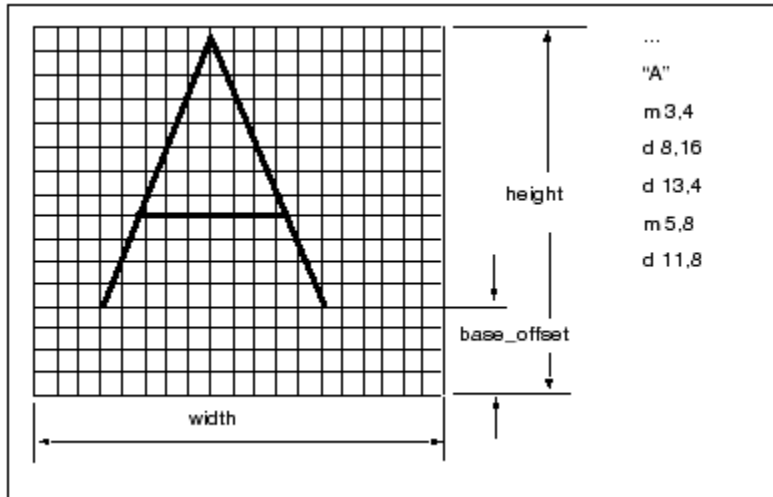


## ASCII Font Definition

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.



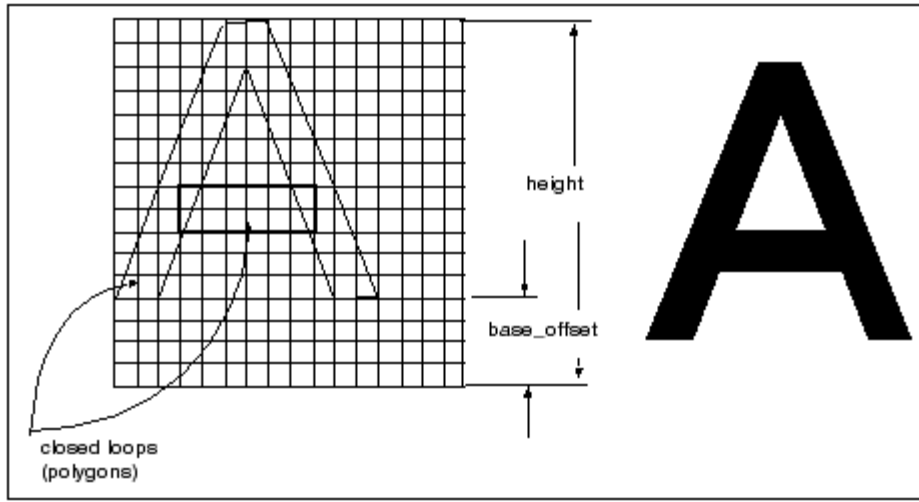
## Filled Font Definition

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.

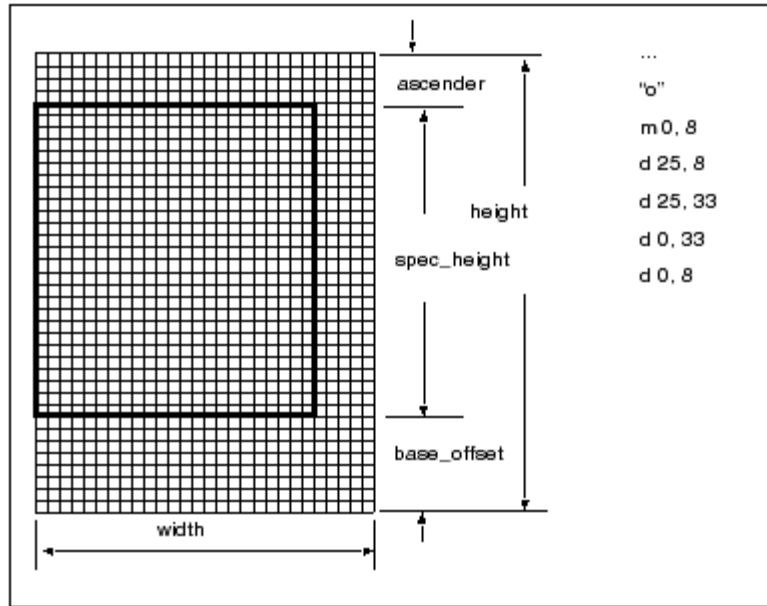


## Special Font Definition

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

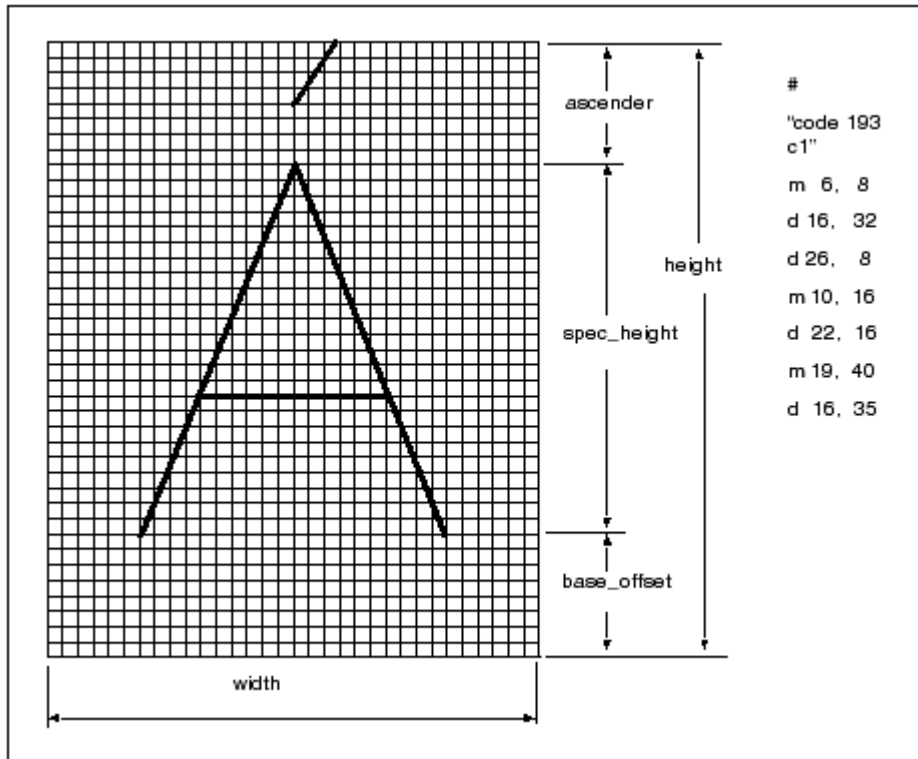
The following text parameters define the LATIN\_1 font:

- 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\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\text\<language>`.
2. Modify the font file.
3. From the `<creo_loadpoint>\Common Files\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\text\<language>`. Store the `.fnt` file in the following directory: `<creo_loadpoint>/Common Files/<machine_type>/text/<language>`

## How to Set Your Font as the Default

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

## How to Create or Customize a Font

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

---

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

(primary font with codeset 0)

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

(special symbols font with codeset 4)

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

```
vi custom.src
```

```
vi symbols.src
```

3. Copy the `compile_font` script from the directory `<creo_loadpoint>/Common Files/<machine_type>/obj` into the same local working directory.

```
cp /usr/pro/i486_nt/obj/compile_font
```

4. Compile the font files.

```
compile_font custom.src custom.fnt
```

```
compile_font symbols.src symbols.fnt
```

5. Create a new index file to reference the compiled font files. Note that the index file name does not need to be the same as the primary font name. If the custom font created is not a primary font (codeset 0), be sure to include a font with a codeset of 0 in the index file. In this example, the new font index file references both customized font files (primary and special symbols).

```
myfont.ndx (new customized .ndx file)
```

```
custom (new custom primary font (codeset 0))
```

```
latin_1 (system extended ASCII font (codeset 2))
```

```
symbols (new special symbols font (codeset 4))
```

This font index file references the new customized special symbols font only.

It uses the system default primary font.

```
myfont2.ndx (new customized .ndx file)
```

```
ascii (system default primary font (codeset 0))
```

```
latin_1 (system extended ASCII font (codeset 2))
```

```
symbols (new special symbols font (codeset 4))
```

You can use the new font in the following ways:

- To use the font as a global auxiliary font, place the `.src` and `.ndx` files in the `<creo_loadpoint>\Common Files\text\<language>` directory.



- 
- Then put the compiled `.fnt` files in the directory `<creo_loadpoint>/Common Files/<machine_type>/text/<language>`.  

```
mv custom.src symbols.src myfont.ndx /usr/pro/text/usascii
```

```
mv custom.fnt symbols.fnt /usr/pro/i486_nt/text/usascii
```
  - To set the new font as the system default, rename the primary source and compiled font (codeset of 0) files to `ascii.src` and `ascii.fnt`, respectively. The system `font.ndx` file already references the ASCII font file and, therefore, does not need to be modified. Locate the new `ascii.src` and `ascii.fnt` files in the appropriate directories (see the previous section that describes how to use the font as a global auxiliary font). Note that the system overrides the original `ascii.src` and `ascii.fnt` files, unless you first rename them.

The next time you start a 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 on page 103](#).
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/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 122](#), 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 123](#).

#### 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

GTOL Symbol	ANSI Code	ISO Code
Circular	101	138
Cylinder	103	140
Profile Line	107	146
Profile Surf	100	137
Angular	97	135
Perpendicular	108	147
Parallel	102	139
Position	106	145
Concentric	114	148
Runout Circular	104	141
Total Runout	34	134
Symmetrical	105	144

In a 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 Interface Help available in the Creo Parametric 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 [http://www.ptc.com/WCMS/files/160242/en/PTC\\_Creo\\_Future\\_Platform\\_Support\\_Summary.pdf](http://www.ptc.com/WCMS/files/160242/en/PTC_Creo_Future_Platform_Support_Summary.pdf). For information related to the Help Center, see [Installing and Using Creo Help on page 60](#).

## Browser Requirements

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

Creo uses a 64-bit browser processing support for IE and Chromium.

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

The Creo agent is the same across all Creo applications and as a result, the 64 bit embedded browser will also be the same across all Creo applications.

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

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

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

## 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 PXX 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.

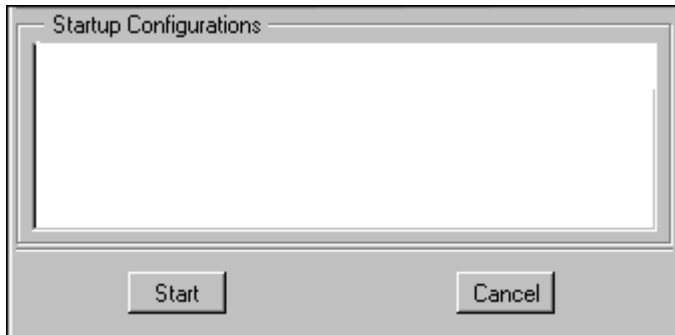
If you have installed Creo View Express, a Creo object is opened in Internet Explorer only for viewing when you double-click the object in Windows Explorer. Right-click on the object and choose the Creo application in which to open the object.

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
.cem	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.

## Handling Incorrect Wrapper Files Stored in Windchill

When you use Creo Unite technology to open CATIA, NX, and SolidWorks part and assembly models in Creo, the following occurs:

- Creo data wraps around the non-Creo data.
- Creo wrapper files are created. The wrapper files are stored internally and are not visible.

If you open the non-Creo part and assembly models in the Windchill connected mode of Creo, the wrapper files are created and stored in Windchill. The non-Creo models are checked in to the Windchill Workspace as follows:

- The non-Creo models are checked in as the primary content of CAD Documents.



- 
- The wrapper files are stored in Windchill as the secondary content of the CAD Documents.

From Creo, you can retrieve the non-Creo part and assembly models from the Windchill Workspace.

### Recreating Wrapper Files

While retrieving the non-Creo models checked in to the Windchill Workspace, the wrapper files may be corrupt if:

- Model regeneration fails.
- The geometry of the retrieved models is incomplete.

If wrapper files are corrupt and the non-Creo models and their wrapper files are not synchronized with the data of the latest import profiles, you can use Creo Distributed Batch to discard the corrupt wrapper files and recreate them.

To enable Creo Distributed Batch to discard the corrupt wrapper files and recreate them, set the configuration option, `sync_data_import_profile`, to `recreate` in the Task Type Definition (TTD) files. The value, `recreate`, is supported only in Creo Distributed Batch.

---

#### Note

Discarding the corrupt wrapper files and recreating them before they are uploaded to the Windchill Workspace as the secondary content of the non-Creo models does not change the revision of the non-Creo models.

---

### The `sync_data_import_profile` Configuration Option

Use the `sync_data_import_profile` configuration option to create new wrapper files when the existing wrapper files are corrupt or to update the existing wrapper files when they are not synchronized with the latest import profiles. Values for the `sync_data_import_profile` configuration option are listed below:

- `never`—Does not synchronize the non-Creo model to the latest import profile. This is the default.
- `update`—Updates the wrapper data using the latest import profile.
- `recreate`—Discards the existing wrapper files and recreates them. The value, `recreate`, is supported only in Creo Distributed Batch.

---

## Synchronizing Model and Wrapper Data with Import Profiles

If the latest format-specific import profiles saved to disk were not used when the non-Creo models were retrieved in Creo, then the non-Creo models and their wrapper files may not synchronize with the data settings of the latest import profiles. If they are not synchronized, set the `sync_data_import_profile` configuration option to `update` so that the wrapper files are updated and they synchronize with the latest import profiles.

Optionally, if you do not want to use the default import profiles, you can set the following format-specific configuration options before retrieving the non-Creo models in Creo. The values for each of the configuration options is the full path of the corresponding import profiles, including the `*.dip` file extension.

- `import_profiles_catiav5`
- `import_profiles_solidworks`
- `import_profiles_nx`

When the non-Creo models are subsequently retrieved in Creo, the wrapper files are updated so that the non-Creo models and their wrapper files synchronize with the import profiles in use.

---

### Note

Updating the non-Creo models and their wrapper files to synchronize with the import profile data settings does not change the revision of the non-Creo models stored in Windchill.

---

# D

## Installing and Configuring Diagnostic Tools

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This appendix contains information on Diagnostic Tools utility.

---

## Overview

The Diagnostic Tools combines the Quality Agent and the Session Logger utilities. This tool collects information 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.

## Installing Diagnostic Tools

The Diagnostic Tools utility is installed by default when you install any of the Creo applications. The diagnostic 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 Diagnostic Tools can be used with all the Creo applications for which you have enabled diagnostic reporting.

---

### Note

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

---

You can install Diagnostic Tools utility 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 Diagnostic Tools. Specify the complete installation path of Diagnostic Tools as the value of the configuration option. For example, `C:\Program Files\PTC\DiagnosticTools\%PRO_MACHINE_TYPE%\obj\qualityagent.exe`

---

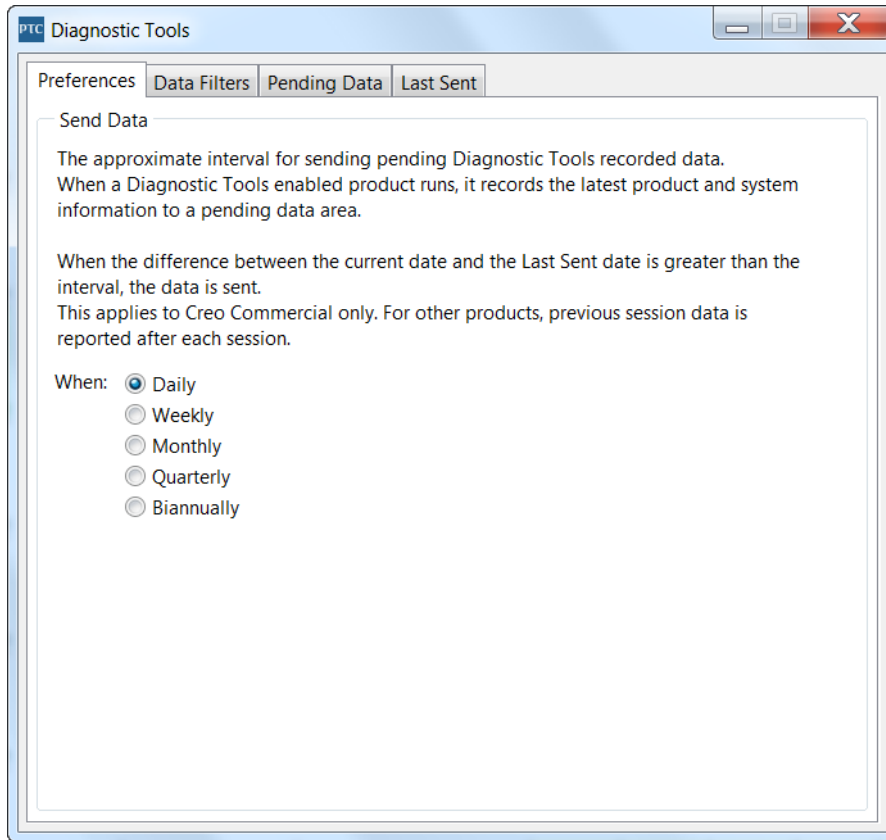
### Note

Note that setting the `quality_agent_path` configuration option supersedes any setting that you made during installation—Diagnostic reporting is enabled regardless of the setting on the **Application Selection** and **Application Custom Setting** screens.

---

## Configuring Diagnostic Tools

After installing Diagnostic Tools, you can configure it as per your company's requirements. To open the **Diagnostic Tools** 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 Diagnostic Tools 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 **Daily**, **Weekly**, **Monthly**, **Quarterly**, or **Biannually**.

---

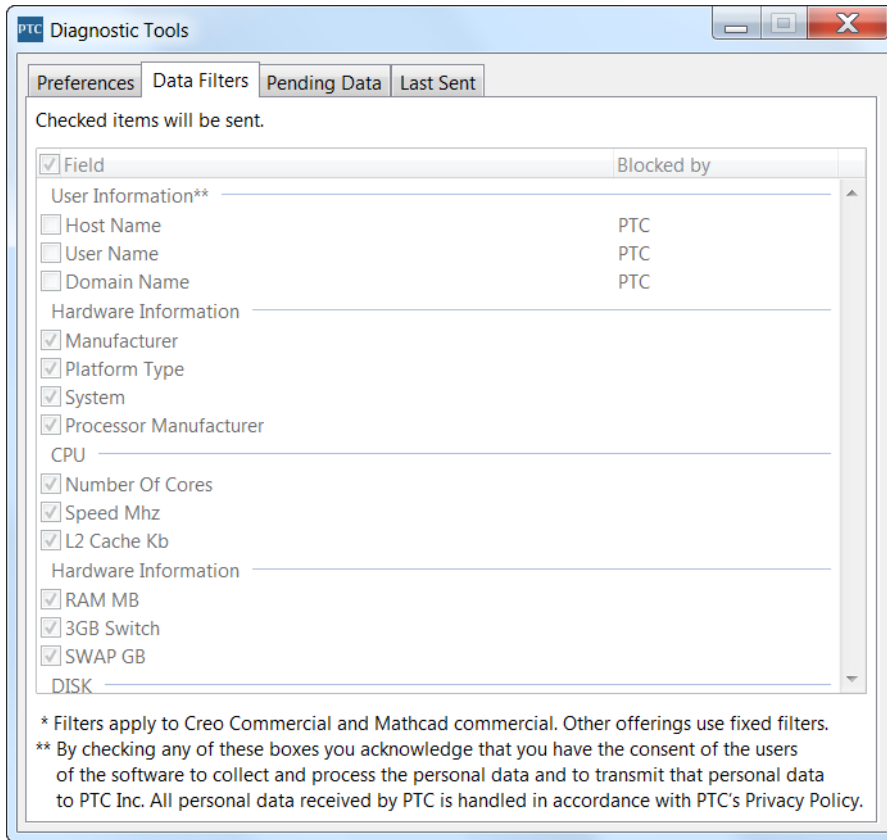
### Note

A user who does not have write permissions to the installation folder can view the preferences but cannot change the settings.

---

## Data Filters

If you have write permissions to the Diagnostic Tools installation folder and files, click the **Data Filters** tab. Select specific information that you do not want to send to PTC.



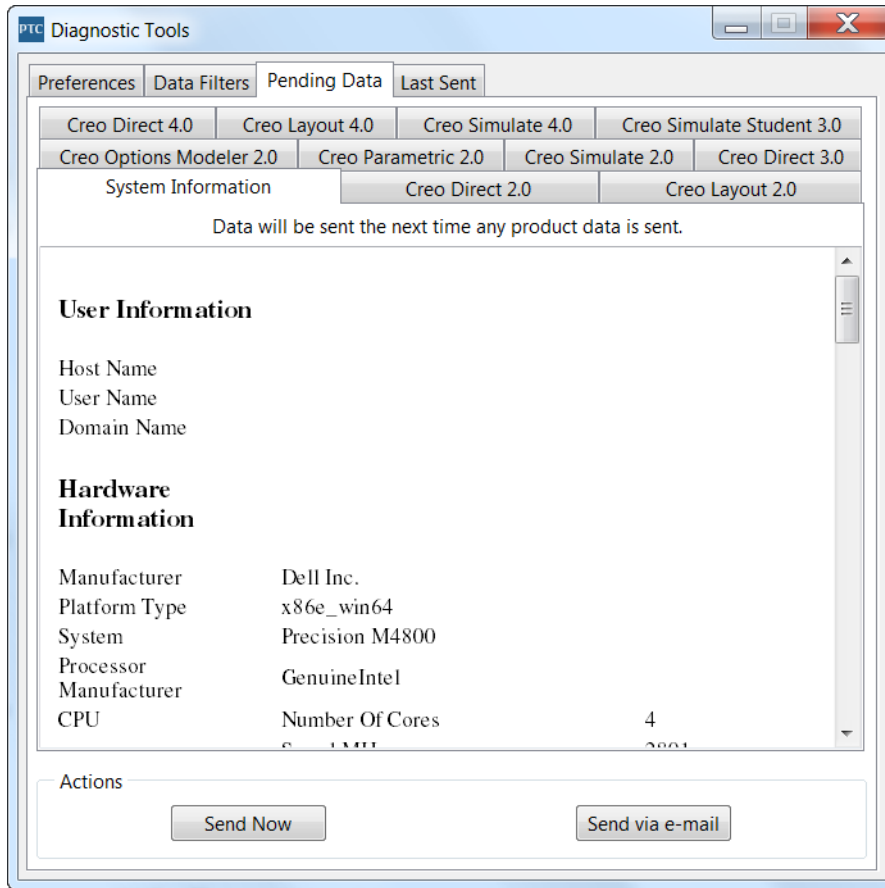
**Note**

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

The customized preferences and data filters are stored at C:\Program Files\PTC\DiagnosticTools\text\xml\.

## Pending Data

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

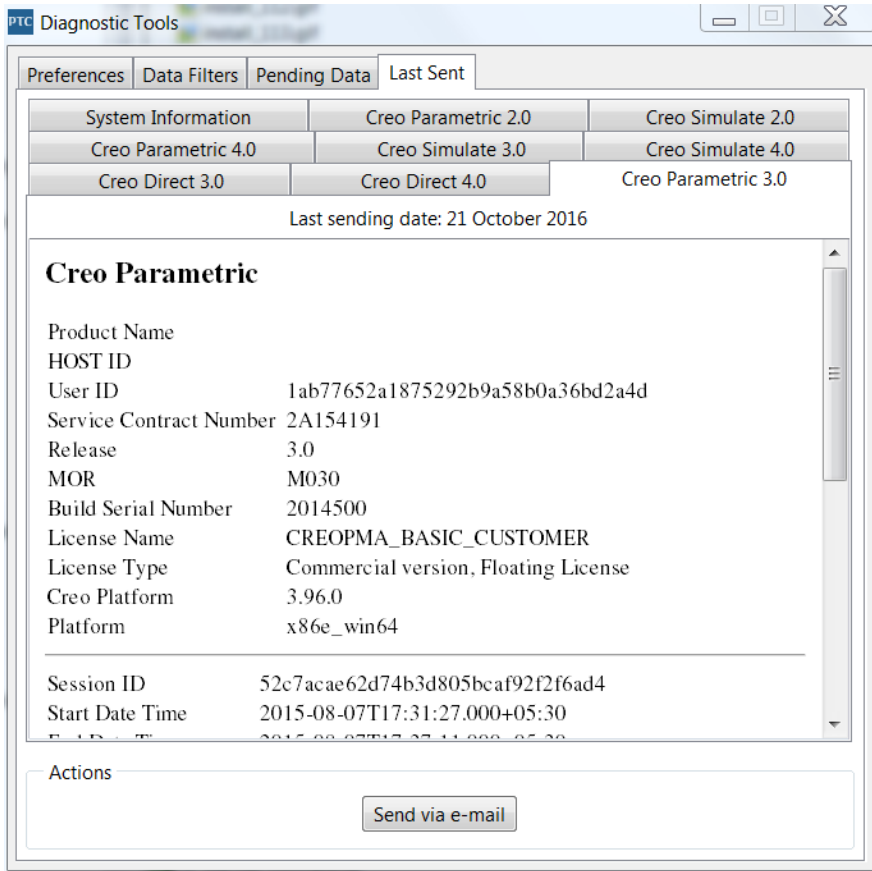


On the active application tab, click a button to perform the following actions:

- **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, the product and system information for Creo Direct is sent 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 that you have installed, you can view the product and system information that was last collected and transferred to PTC on the **Last Sent** tab



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 Diagnostic Tools

The Diagnostic Tools utility 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, Diagnostic Tools 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 **Diagnostic Tools** 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.



## Session Log

When a session of an installed Creo application ends, a session log for that application is created. Click the **Pending Data** or **Last Sent** tab on the **Diagnostic Tools** dialog box, and select an application tab to view the session log for that application. A session log is a list of entries that provide information about a specific session.

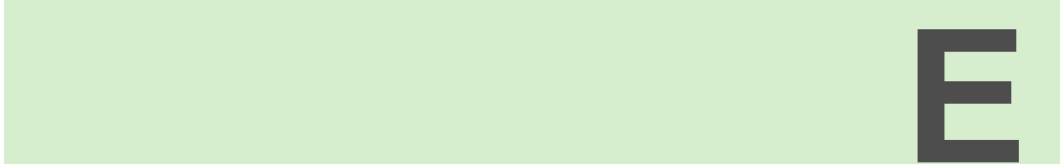
The session log lists information related to the session ID, start and end time of the session, idle time within that session, exit type, locale, license server used, browser information, Java version, and so on for a session.

Creo Parametric	
Product Name	
HOST ID	
User ID	1ab77652a1875292b9a58b0a36bd2a4d
Service Contract Number	2A154191
Release	3.0
MOR	M030
Build Serial Number	2014500
License Name	CREOPMA_BASIC_CUSTOMER
License Type	Commercial version, Floating License
Creo Platform	3.96.0
Platform	x86e_win64
<hr/>	
Session ID	52c7acae62d74b3d805bc9f2f2f6ad4
Start Date Time	2015-08-07T17:31:27.000+05:30
End Date Time	2015-08-07T17:37:11.000+05:30
Duration	344
Idle Time	0
Net Duration	344
Exit Type	00001
Graphics Mode	OPENGL
Photorealistic Renderer	
Locale	usascii
Exit-logger Server	
Windehill Server	None
License Type	
License Server	gold.ptcnet.ptc.com ruby.ptcnet.ptc.com bronze.ptcnet.ptc.com
Creo Browser Type	Internet Explorer
Creo Browser Version	
Creo Java Version	Unknown

## Uninstalling Diagnostic Tools

You can remove Diagnostic Tools 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 **Diagnostic Tools**.
3. Click **Uninstall**. A confirmation dialog box opens.
4. Click **Yes** to remove Diagnostic Tools.



# Installing MKS Platform Components

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Installing MKS Platform Components ..... 139

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 **Creo Parametric Interface for CADD5 5** component during Creo Parametric installation to work with CADD5 5 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 64-bit Windows operating systems.
- Emulates UNIX commands on the Windows operating systems.
- Installs the Portmapper service for interoperability between CADD5 5 and Creo Parametric.
- Copies the necessary startup files and libraries required to work with CADD5 5 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 33](#).
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 5 Models in Creo Parametric on page 57](#) before you start.

---

### Note

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

Navigate to the `/cadd5_mks` folder of the installation media image and run the following command:

```
For a 64-bit system — msiexec /X cadd5_mks_for_
windows64.msi /q
```

# 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 root directory of the installation media at the command prompt. See [Command Syntax for a Silent Installation on page 145](#) 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 installation media.
  - The prerequisites for installing the Creo application are not met.
- You must run the Assistant only from the installation media 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.

---

### Note

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

---

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, Creo Thumbnail Viewer, and Creo Platform Services in a large enterprise:

- Perform a silent installation of Creo applications on all the client computers. See [Workflow for a Silent Installation on page 143](#) 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 153](#) 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 153](#) 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 155](#) for details.
  3. Install [Microsoft Visual C++ Redistributable for Visual Studio 2019](#) on each client computer.
- 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 153](#) 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 155](#) 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.
- After the silent installation is complete, if you want to reconfigure the installation using PTC Installation Assistant, you must explicitly provide the license source information on the **License Identification** screen.

3. Install [Microsoft Visual C++ Redistributable for Visual Studio 2019](#) on each client computer.

## 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 installation media. PTC Installation Assistant starts.
  - b. On the Welcome screen click **Install new software**.
  - c. On the **License Identification** screen perform one of the following operations:
    - Drag a node-locked or floating license file into the **License Summary** area.
    - Enter the `port@host` address of the running license server.

---

 **Note**

- Downloading a license file or installing PTC License Server must be done in a separate installation session.
- A license source is not validated during the silent installation process. Hence, you must ensure the accuracy of the license source list that is configured in the commands (.psf files) for starting the 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 145](#) 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 Creo applications, the `Common Files` folder is installed. The XML file for this folder is stored at the following location:

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

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

- Diagnostic Tools — `<dvd>:\pim\xml\qualityagent.xml`
- MKS Platform Components — (Required only if installing CADD5 Adaptor) `<dvd>:\pim\xml\mkscomponents.xml`

---

 **Note**

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

2. Copy one or more Creo application specific XML files, along with the `creobase.p.xml`, `qualityagent.xml`, and `mkscomponents.xml` files, to a temporary location on your disk; for example, `D:\temp\ptc`, and



edit it as required. See [Editing the XML Files on page 147](#) 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.
4. Perform a silent installation of the required client-side components. See [Installing and Uninstalling Client-Side Components in Silent Mode on page 153](#) for information on how to install Client-side components.

## 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 installation media as follows:

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

Where `drive` is the installation media drive letter or the location of the downloaded installation 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 -upgrade -xml &lt;file1&gt; -xml &lt;file2&gt;....</code>	Performs a silent installation with the removal of the highest version that already exists, and installs the current version of a release, according to the instructions in the specified XML files.
<code>setup.exe -upgrade -xml &lt;file1&gt; -xml &lt;file2&gt; -backup</code>	When you use the <code>-backup</code> option along with the <code>-upgrade</code> option, a number of existing customized files and folders from a predefined list, are backed up and copied to the <code>Custom files</code> directory. Use this option if you are not sure if you have customized library files in your existing version.
<code>setup.exe -xmlall "&lt;path to the .xml files stored in a common temporary location&gt;"</code>	Performs a silent installation of all the applications as per the directives specified in the XML files that are all stored in a common folder.

Command Syntax	Description
setup.exe -xml <file1> [-destination <directory>]	[Optional] Overrides the destination specified in the XML file.
setup.exe -xml <file1> [-licensesource <string>]	[Optional] Overrides the license source specified in the XML file.
setup.exe -xml <file1> [-translate_lp_ path_space <character>]	[Optional] Replaces the whitespace characters in the installation path in the XML file.
setup.exe -xml <file1> [-licfeat_regen]	[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.
setup.exe -xml <file1> [-licfeat_preferred <string>]	[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.
setup.exe -progressbar -xml <file1>	[Optional] Displays the indicative progress bar during the installation using the XML file.

Consider the example of performing a silent installation of Creo Parametric with Common Files, and a required component Diagnostic Tools. 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 installation media drive letter
- D:\temp\ptc\pma.p.xml—The location for the <app\_name>.p.xml file.
- D:\temp\ptc\creobase.p.xml—The location of the XML file for the Common Files folder.
- D:\temp\ptc\qualityagent.xml—The location of the XML file for the required component Diagnostic Tools.

---

## 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\ 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\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. You can change the `INSTALLBASE` name attribute in the Creo applications (`<app_name>.p.xml`) and Common Files (`creobase.p.xml`) XML files. The default value is usually the PTC folder under the Program Files folder.

---

### Note

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

---

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

```
<PROPERTY name="[INSTALLBASE]">"F:\Program Files\PTC"</PROPERTY>
```
- After editing the `INSTALLBASE` name attribute, the `PROPERTY` node appears as shown:  

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

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

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

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

## NoRegistryActions

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

```
<PROPERTY name="NoRegistryActions"</PROPERTY>
```

## 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 [VERSION]` to `Creo_[VERSION]`. 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="[CREOCOMMONFILES]" [INSTALLBASE]/Creo [VERSION] /
[COMMONFILESNAME]></PROPERTY>
<PROPERTY name="[LP]">[CREOCOMMONFILES]</PROPERTY>
```

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

```
<PROPERTY name="[CREOCOMMONFILES]" [INSTALLBASE]/Creo_[VERSION] /
[COMMONFILESNAME]></PROPERTY>
<PROPERTY name="[LP]">[CREOCOMMONFILES]</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 145](#) 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 Diagnostic Tools settings for an installed Creo application. You can edit the following attribute of the `QUALITY_AGENT` node:

Attribute	Value
<code>enable</code>	<p>Enables or disables the diagnostic reporting for a Creo application. For example, if this attribute is set to <code>Y</code>, diagnostic reporting is enabled.</p> <p> <b>Note</b></p> <p>If the <code>required</code> attribute is set to <code>Y</code>, you must not change the value of the <code>enable</code> attribute to <code>N</code>. If you do so, the Creo application will not run because of conflicting settings.</p>

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 installation 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
<code>install</code>	<p>Set this attribute to <code>Y</code> or <code>N</code> depending on whether you want to install the Creo application in that language. If the <code>required</code> attribute is set to <code>Y</code> do not change the <code>install</code> attribute.</p>

---

A sample LANGUAGE node in an XML file follows:

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

## Editing the PACKAGE Node

PACKAGE nodes in an XML file represent the required and the optional packages selected for a 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 <code>install</code> attribute of the parent package to Y to install any of its child packages.

A sample PACKAGE node in an XML file follows:

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

## Editing the CDSECTION Nodes

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

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

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

```
C:\>"C:\Program Files\PTC\Creo <version number>Parametric\bin
\uninstall.lnk" -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.

### Uninstalling All Creo Applications

You can uninstall all the installed Creo applications and utilities of a release as a group in silent mode by running the batch file `silent_Group_Uninstall.bat` from a command shell:

```
<creo_loadpoint>\InstallManager\silent_Group_Uninstall.bat
```

The group of all the installed Creo applications of the same release or datecode are uninstalled. This command does not impact other installed applications or utilities.

### Uninstalling All Utilities

You can uninstall all the installed utilities of a release as a group in silent mode by running the batch file `Uninstall_all_utilities.bat` from a command shell:

```
<creo_loadpoint>\InstallManager\Uninstall_all_utilities.bat
```

The group of all the installed utilities of the same release or version are uninstalled.

## Installing and Uninstalling Client-Side Components in Silent Mode

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

- Creo Platform Services
- Creo Thumbnail Viewer
- Creo View Express

---

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

### Creo Platform Services

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

- Install Creo Platform Services at the default location, in silent mode using the following command:  
`"<dvd>:\install\addon\creosvcs_64.exe" /quiet`
- Uninstall Creo Platform Services installed at the default location, in silent mode using the following command:  
`<dvd>:\install\addon\creosvcs_64.exe" /quiet /uninstall`

### Creo Thumbnail Viewer

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

- Install Creo Thumbnail Viewer using the following command:  
`"<dvd>:\install\addon\Thumbviewer_32_64.msi" /passive`
- Uninstall Creo Thumbnail Viewer using the following command:  
`msiexec /uninstall "<dvd>:\install\addon\Thumbviewer_32_64.msi" /passive`

### Creo View Express

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

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

```
start /w <dvd>:\install\addon\pvx32_64\CreoSetup.exe  
/vADDLOCAL="ALL" /qn
```

In the previous example:

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

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

```
"start /w <dvd>:\install\addon\pvx32_64\CreoSetup.exe  
/vADDLOCAL="ALL" APPLICATIONFOLDER="c:\ptc\CreoViewExpress" /qn"
```

---

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

```
msiexec.exe /x <dvd>:\install\addon\pvx32_64\pvexpress\  
CreoView_Express_32_64.msi /qn
```

---

 **Note**

Running the `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 the chapter “Deploying Creo View and Customizing the Installation” in the “Creo®View Installation and Configuration Guide” that can be downloaded from [here](#).

## Installing Diagnostic Tools and MKS Platform Components in Silent Mode

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

## Updating the Registry File

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

```
<creo_loadpoint>\<app_name>\bin\pim\xml\created.reg
```

The `created.reg` file enables the seamless opening of the appropriate Creo application when you click a Creo part, assembly, or drawing file. When you upgrade the version of the Creo application or make a copy of the installation of the Creo application, you must rerun `created.reg` file to update the registry with the latest settings.

---

 **Note**

If you are using Windchill and do not have Creo files outside of that network, you do not need to run the `created.reg` file.

---

---

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 <version number>\Parametric\bin\pim\xml\  
created.reg
```

(Creo Parametric)

```
C:\Program Files\PTC\Creo <version number>\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 Creo 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></p> <p>You cannot set this environment variable in your .psf file.</p>
<p>pro_comm_msg.exe —Facilitates communication between the executable xtop.exe and asynchronous Pro/TOOLKIT applications.</p> <p> <b>Note</b></p> <p>The executable xtop.exe refers to all the desktop-based Creo applications.</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>
<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</p>	<p>Not supported.</p>

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

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 loadpoint>\app\bin\*.psf` and the `<Creo loadpoint>\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 loadpoint>\app\bin\*.psf` and the `<Creo loadpoint>\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.

---

## 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.



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

---

## General Debugging Hints

The `ptc_d.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 <https://www.ptc.com/en/support> 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* that can be downloaded from [here](#).

## 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 7788@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 ptstartserver.

---

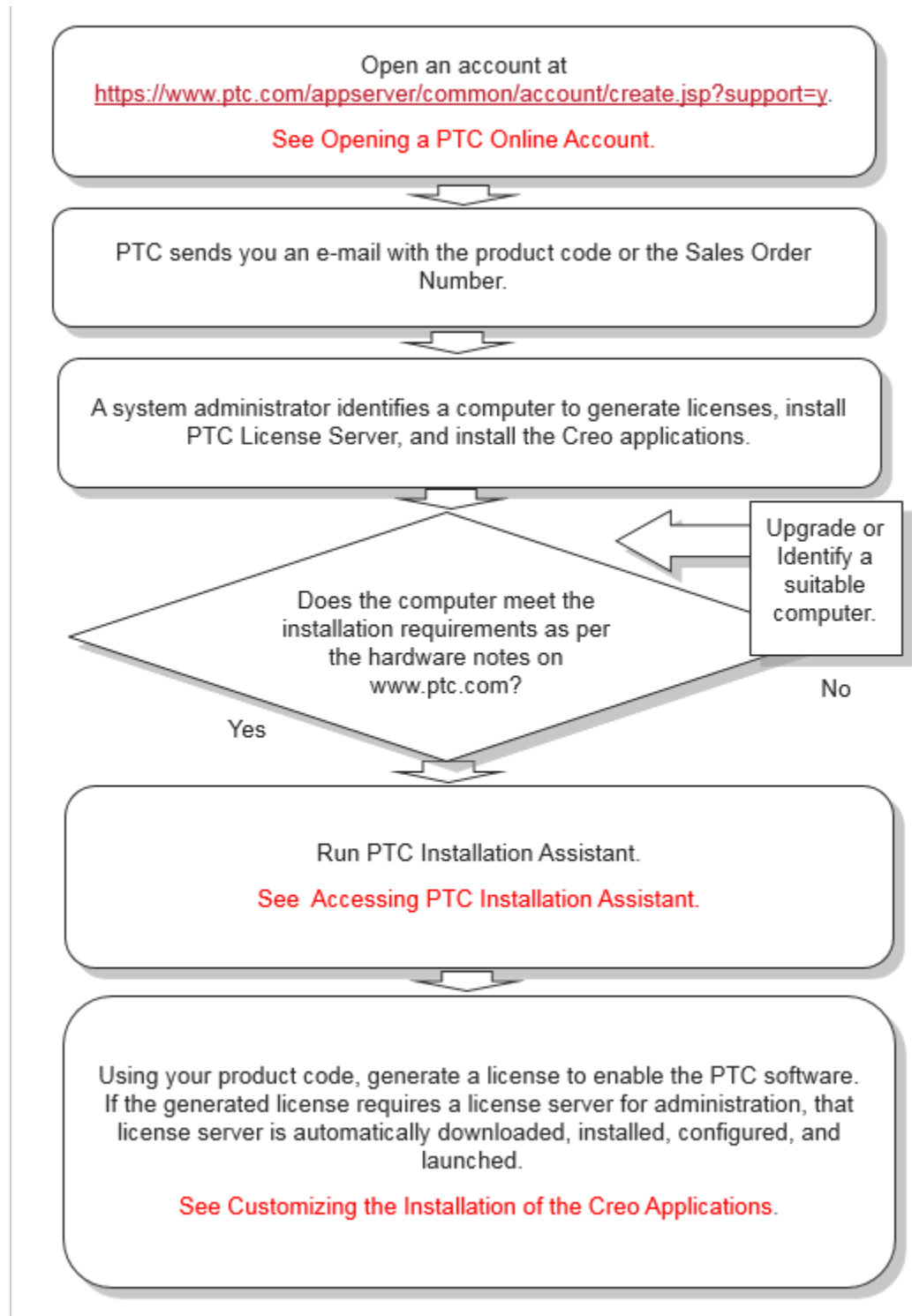
# J

## Workflow for Installation and Configuration

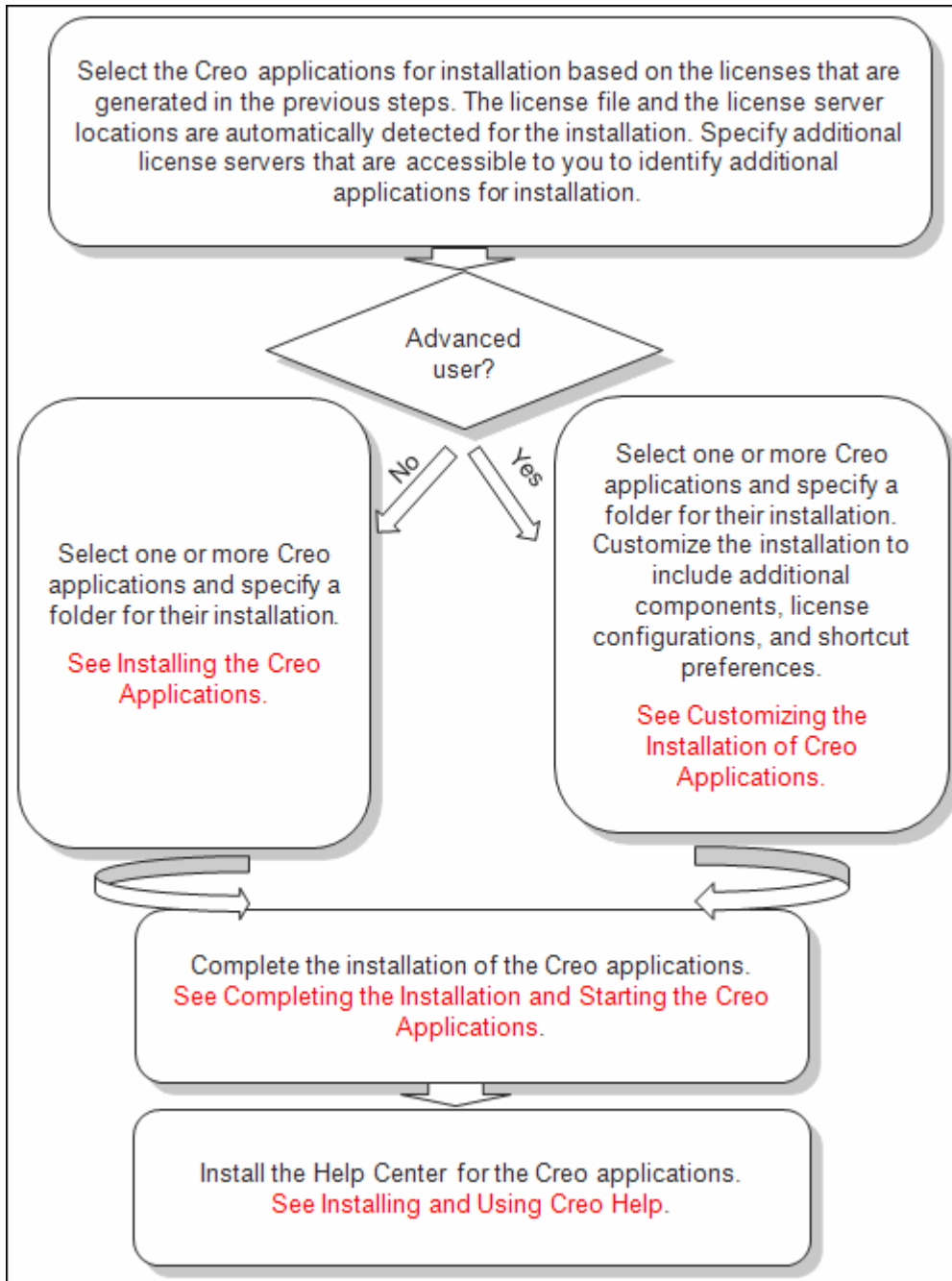
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This chapter provides simple workflows for generating licenses and installing and reconfiguring the Creo applications. The workflow is presented as a flowchart for each of these processes. If you have received a license file via e-mail from PTC License 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

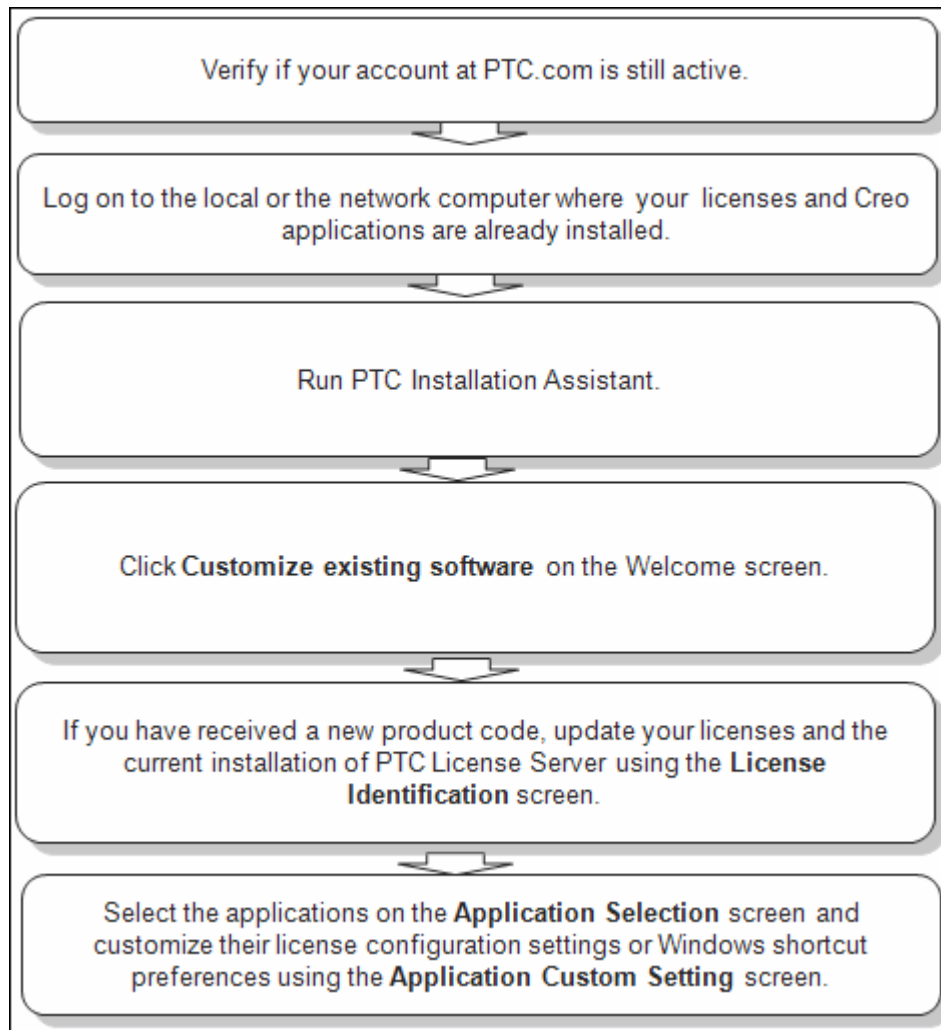


# Workflow for Installing the Creo Applications



---

## 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.
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,

Term	Definition
	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.
Diagnostic Tools	A PTC utility that collects information 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.
datecode	A unique number to identify a specific version of PTC software. The datecode can be found printed on the jacket of the installation media.
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,</li> </ul>

Term	Definition
	which contains various operating parameters
educational license	A license used at educational institutes to run PTC software, also known as a students license. This is 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.



<b>Term</b>	<b>Definition</b>
increment line	See feature line.
installation directory	The directory in which PTC software is installed. Also called the load point.
intellectual property	Proprietary information stored within 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.

<b>Term</b>	<b>Definition</b>
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 7.0.0.0</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.
name_service daemon	This daemon process enables remote

<b>Term</b>	<b>Definition</b>
(nmsd)	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).
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.

<b>Term</b>	<b>Definition</b>
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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