



# **Workgroup Manager for Creo Illustrate Administrator's and User's Guide**

**Windchill 10.0 M020**

**October 2011**

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**Parametric Technology Corporation, 140 Kendrick Street, Needham, MA 02494 USA**

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## About This Guide

The *Windchill Workgroup Manager for Creo Illustrate Administrator's and User's Guide* describes how to configure and use Creo Illustrate with Windchill Workgroup Manager.

This guide is designed to provide the information you need in an easy-to-use format. General scenarios and specific Creo Illustrate examples are intended to provide useful recommendations as you work with Windchill Workgroup Manager.

The information in this chapter details how the guide is set up, its content flow, and summarizes the content, and intended audience, for each chapter.

## Guide Structure

The contents of this guide are presented in sequential order so that, should you read the guide from cover-to-cover, you get the full, comprehensive view of Windchill Workgroup Manager and its interaction with Creo Illustrate. However, each chapter is written in standalone sections that are intended to be read and understood on their own. Sections are not dependent upon other areas of the guide for understanding a concept or task, so the time you spend reading the guide should be more efficient. If you want to learn more information on a particular subject, hyperlinks are available in most sections to take you to related subjects.

## Guide Content Flow

The guide begins with the installation process and the necessary steps to register Creo Illustrate with Windchill Workgroup Manager and connect to the Windchill server. The next chapter presents the basics of Windchill Workgroup Manager, explaining what it is and how it relates to Creo Illustrate.

The next chapter discusses details of using the product data management actions that are available to Creo Illustrate users with the Windchill Workgroup Manager, and that chapter is followed by a discussion of advanced user techniques, such as working with external references and resolving conflicts.

Toward the end of the guide are more reference-oriented chapters. The next chapter explains how to set up preferences on the server and the client. This chapter is beneficial to both administrators and users. The final chapter presents administration and configuration information for administrative users.

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## Where to Start

This section describes where to start using the guide. If you have limited or no experience with Windchill, you should start with the *Overview* and the *Using Workspaces* chapters, which describe the system architecture and basic Windchill concepts. If you have previous experience with Windchill, and feel very comfortable working with it, you can start with the *Getting Started* chapter, to reacquaint yourself with familiar PDM actions.

The intended audience is described as follows.

- **Novice** — No or little Windchill experience
- **Experienced** — Experienced with Windchill and understands product data management concepts
- **Administrator** — Experienced with configuring Windchill

The following table can be used to quickly understand what audience the chapters in this guide are intended.

Chapter	Definition	User
How to Use This Guide	Describes guide contents, purpose, navigation, access to online help, related documentation and technical support.	Novice, Experienced
Installing Windchill Workgroup Manager	Describes installing the client.	Novice, Experienced, Administrator
Windchill Workgroup Manager Overview	Describes Windchill Workgroup Manager system architecture.	Novice, Experienced
Getting Started with	Describes connecting to Windchill, registering servers and CAD applications, workspaces.	Novice, Experienced, Administrator
PDM Actions	Describes using PDM actions.	Novice, Experienced
Advanced Techniques	Describes working with Families, External References, Drawings, and managing cache.	Experienced

Chapter	Definition	User
Client and Server Preferences	Describes setting up preferences on the client and server.	Novice, Experienced, Administrator
Administration and Configuration	Describes configuring the CAD application with Windchill, and Windchill management	Administrator

## Related Documentation

Documentation related to this products is available under **Windchill Workgroup Manager for Creo Illustrate** in the Windchill Workgroup Manager Help Center.

If documentation is not installed on your system, see your system administrator.

## Technical Support

Contact PTC Technical Support through the PTC website, or by phone, email, or fax if you encounter problems using this product or the product documentation.

For complete details, see the *PTC Customer Service Guide*. You can find this guide under **Contacting Technical Support** on the PTC Technical Support page:

<http://www.ptc.com/support/index.htm>

You must have a Service Contract Number (SCN) before you can receive technical support. If you do not have an SCN, contact the PTC Maintenance Department using the instructions found in the *PTC Customer Service Guide* or on the PTC Technical Support page.

## Documentation for PTC Products

You can access PTC documentation using the following resources:

- **Windchill Help Center**—The Windchill Help Center is an online knowledge base that includes a comprehensive index of all Windchill documentation. You can browse the entire Windchill documentation set, or use the search capability to perform a keyword search. To access the help center, you can:
  - Click any help icon  in Windchill
  - Select **Help** ► **Windchill Help Center** from the **Quick Links** menu at the top right of any Windchill page

- 
- Use the following link to access all PTC help centers:

<https://www.ptc.com/appserver/cs/help/help.jsp>

- **Reference Documents Website**—The Reference Documents website is a library of all PTC guides:

<http://www.ptc.com/appserver/cs/doc/refdoc.jsp>

A Service Contract Number (SCN) is required to access the PTC documentation from the Reference Documents website. For more information on SCNs, see the PTC Technical Support page:

<http://www.ptc.com/support/index.htm>

## Comments

PTC welcomes your suggestions and comments on its documentation. To submit your feedback, you can:

- Send an email to [documentation@ptc.com](mailto:documentation@ptc.com). Include the name of the application and its release number with your comments. If your comments are about a specific help topic or book, include the title.
- Click the PTC help center feedback icon  in the upper right of a Windchill Help Center topic and complete the feedback form. The help topic title is automatically included with your feedback.

# 1

## Installation and Configuration

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This section describes how to install and configure, and uninstall the Windchill Workgroup Manager client software.

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## Overview

In addition to the Windchill Workgroup Manager client software installation is the Windchill Workgroup Manager server installation.

The Windchill Workgroup Manager client software must connect to a Windchill server that includes the Windchill Workgroup Manager server installation.

The Windchill Workgroup Manager server installation is typically performed by a Systems Administrator. This procedure is performed using the PTC Solutions Installer and is documented in the *Windchill Installation and Configuration Guide* and in the Windchill Help Center under **Installation and Upgrade**.

Other mandatory server installation steps after installing the Windchill Workgroup Manager server installation are “Configuring the Worker,” and the “Post-Install Configuration of the Worker on Windows” sections in the *Windchill System Administrator's Guide* in order to create the system-level environment variables noted in the section.

- The system environment variable for the PTC Virtual File System (VFS), that is installed during the Windchill Workgroup Manager client installation, is PTC\_VFS\_INTALL\_DIR. This variable allows the user to install VFS at the specified location for the Windchill Workgroup Manager client software.

### Note

*If you do not set the above variable, then during the Windchill Workgroup Manager client software installation, VFS will install at the default location “C:\ProgramFiles\PTC\VFS,” rather than the specified Windchill Workgroup Manager client software location.*

- Other system environment variables must be set before installing the Windchill Workgroup Manager client software:
  - PTC\_VFS\_ROOT=<any local directory>
  - PTC\_WLD\_ROOT=<any local directory>

### Note

*If you do not set the above variables, then during the Windchill Workgroup Manager client software installation, the folders “.ws” and “.yfs” are created under <local drive>\<Users>\<user>\.wwgm. If the “.wwgm” folder becomes created under the network drive, it will cause a conflict.*

Refer to the *Windchill System Administrator's Guide*, available at the [PTC Document Reference Site](#) for more information.

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# Installation and Configuration Steps

Follow the procedures in this section to:

- Upgrade an existing Windchill Workgroup Manager client software installation
- Install Windchill Workgroup Manager client software for the first time.

## Upgrading From a Prior Release of Windchill Workgroup Manager

Follow the steps below to upgrade your current client installation.

1. Verify that all files are checked in. (IMPORTANT! To prevent data loss, all files must be checked in before you continue.)
2. Launch the Windchill Workgroup Manager client with administrator logon.
3. Unregister Creo Illustrate and all the registered Windchill Workgroup Manager applications.

### **Note**

See [Unregistering an Application on page 23](#) for instructions.

4. Exit the Windchill Workgroup Manager client.
5. Stop the PTC VFS Controller service. Then uninstall the Windchill Virtual File System (VFS) software. See [Stopping the PTC VFS Controller Service and Uninstalling Windchill Virtual File Service \(VFS\) on page 24](#).
6. Uninstall the existing Windchill Workgroup Manager client. See [Uninstalling Windchill Workgroup Manager Client on page 24](#).
7. Delete the remaining Windchill Workgroup Manager files and folders. See [Deleting Windchill Workgroup Manager Files and Folders on page 25](#).
8. Install the new version to the required location. See [Installing the Windchill Workgroup Manager Client and Windchill Virtual File System \(VFS\) on page 16](#).
9. Register the Creo Illustrate application. See [Registering an Application on page 21](#).
10. Register and connect to a Windchill server. See [Registering the Windchill Server on page 20](#).
11. Set the Connect to Windchill option in Creo Illustrate.

## Installing Windchill Workgroup Manager Client For the First Time

This section describes the process for installing and getting Windchill Workgroup Manager operational on a local computer.

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The PTC Virtual File System (VFS) installation component is installed during the Windchill Workgroup Manager client. VFS is necessary in order to browse to PTC Places where you can perform various Windchill PDM actions, in addition to the Windchill PDM Actions that can be performed from within Creo Illustrate.

**Note**

*The installation of the Windchill Workgroup Manager client with the PTC Virtual File System component requires administrative privileges.*

1. After the Windchill Workgroup Manager server installation and configuration is completed, install the Windchill Workgroup Manager client in one of the following ways:

Install Windchill Workgroup Manager client software as follows:

- a. Download Windchill Workgroup Manager Client installer to your local computer. Install Windchill Workgroup Manager client. See [Installing the Windchill Workgroup Manager Client and Windchill Virtual File System \(VFS\) on page 16](#).
- b. Install Windchill Workgroup Manager client using the redistributable installer. See [Redistributable Windchill Workgroup Manager Client Installer on page 12](#).

**Note**

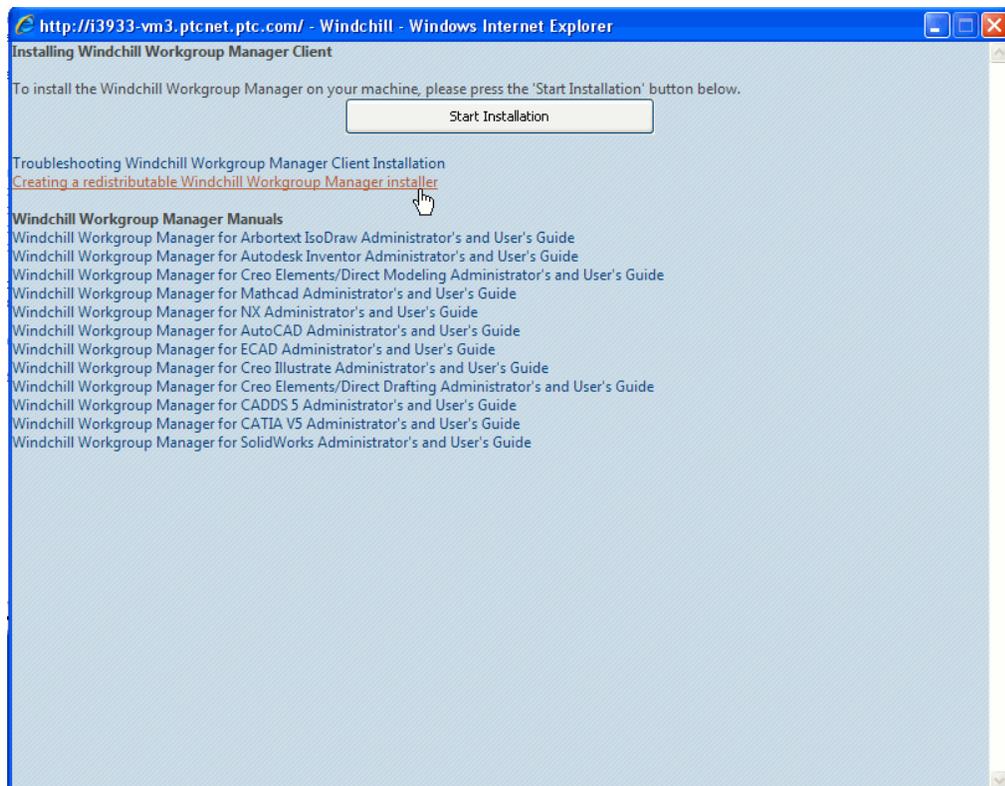
*The Windchill Workgroup Manager client installation includes the PTC Virtual File System installation. You must install the Windchill Workgroup Manager client on your local drive. Do not install it on a network drive.*

2. Configure the Virtual File System to allow access to PTC Places. See [Configuring the Windchill Virtual File System on page 19](#).
3. Register the Creo Illustrate application. See [Registering an Application on page 21](#).
4. Register and connect to a Windchill server. See [Registering the Windchill Server on page 20](#).

## Redistributable Windchill Workgroup Manager Client Installer

The Windchill Workgroup Manager Client can be downloaded for redistribution rather than installed from the server. This feature will allow users to download the installer without starting the install process. Administrators can then copy the installation to a network location where it can be installed on multiple machines or installed through a software deployment tool.

1. Access the **Software Downloads** page from the Windchill **Quick Links** drop-down list.
2. Select **Windchill Workgroup Manager**. The **Installing Windchill Workgroup Manager Client** page launches.
3. Select **Creating a redistributable Windchill Workgroup Manager installer**.



4. A new window will open to create the redistributable installation. Follow the steps on the web page and enter the applicable information for each of the fields.
  - Enter a location to output the installation image.
  - Ensure you have JRE 6.0, if not install or update to that version.
  - Download the ClientInstaller.jar file to a location on your local file system.
  - Enter the location to the local ClientInstaller.jar file.
  - Enter the path to the java executable file.
  - Select the Operating System used to build the image.
  - Enable build logging, if desired.

The above will enable you to create the redistributable installation. There is also an option to create this install to be a silent deployment. The silent deployment option will install with no UI or user inputs needed. The inputs will

be provided from the values provided in the fields. These fields will be stored in a silent.properties file. Continue to the next step, to create a silent deployment.

http://i3933-vm3.ptcnet.ptc.com/ - Windchill - Windows Internet Explorer

**Creating a redistributable Windchill Workgroup Manager installer**  
The following steps will generate a Windchill Workgroup Manager installation image with preconfigured installation inputs that can be used to install the Windchill Workgroup Manager silently in background, without user input. This image can be distributed on network drive for users to install themselves (note they will need admin or elevated privileges to install), or used in conjunction software deployment tools to roll out the Workgroup Manager to multiple clients.

**Creating the distributable image:**

1. Enter the path to the location you want to output the installation (e.g. D:\WWGM\_Install\_image)
- i. Please make sure that you have installed Java Runtime Environment (JRE) 6.0 on your machine.  
ii. Download file <http://i3933-vm3.ptcnet.ptc.com/PDMPJL100/install/ClientInstaller.jar> somewhere on your machine.  
iii. Enter the location you downloaded ClientInstaller.jar to (e.g. D:\temp\ClientInstaller.jar)
- iv. Path to JVM executable on machine used to build the image (e.g. C:\Program Files (x86)\Java\jre6\bin\java.exe)
- v. Select the Operating System of the machine used to build the image.
- vi.  Enable build logging

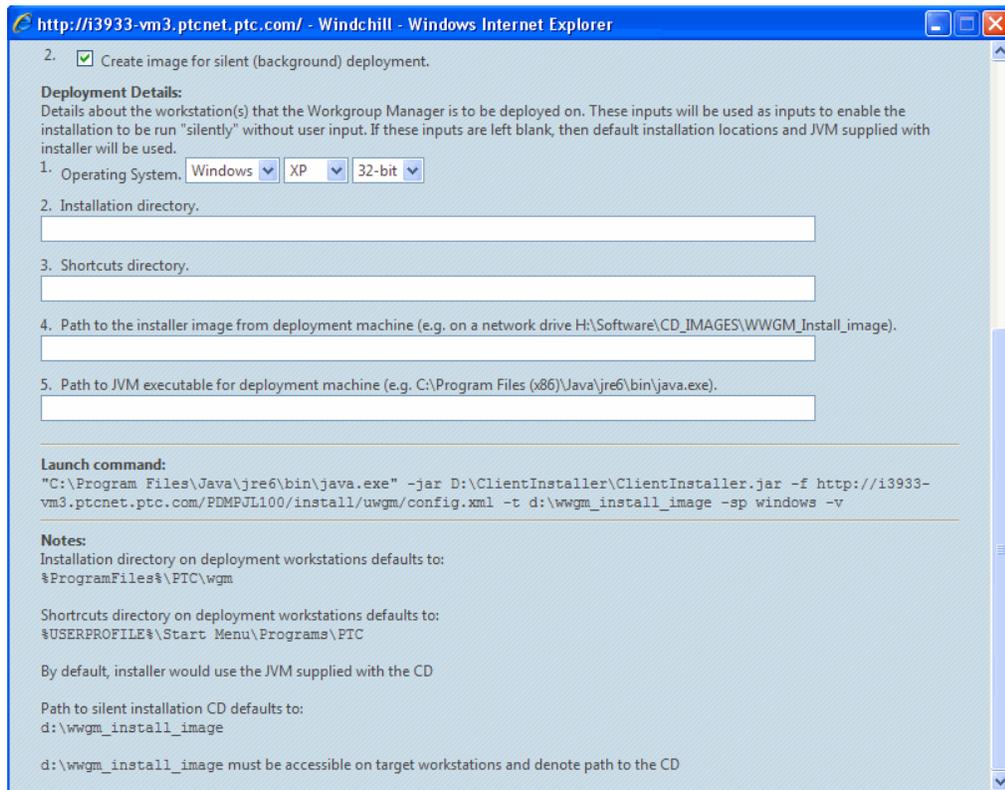
2.  Create image for silent (background) deployment.

**Error(s):**  
Path to output location is mandatory  
Path to downloaded ClientInstaller.jar is mandatory

5. Checking the **Create image for silent (background) deployment** box will display the following fields to enter details about the client machines the Windchill Workgroup Manager will be installed on.

**Note**

*If “Installation Directory”, “Shortcuts Directory”, and “Path to the installer image from deployment machine” are left blank, the default install locations and the Java Virtual Machine supplied with the installer will be used).*



- Select the Operating System of the machines the client will be deployed on.
  - Provide the path to the installation directory.
  - Provide the path to the shortcuts directory where application shortcuts will be created.
  - Provide the path to the installer image accessible from the deployment machine.
  - Provide the path to the java executable on the deployment machine.
6. Once you completed all required inputs, open a command prompt window and copy the text under the **Launch command:** section into the command line. Run the command to create the installer image.

```

Launch command:
"C:\Program Files\Java\jre6\bin\java.exe" -jar D:\ClientInstaller\ClientInstaller.jar -f http://i13933-vm3.ptcnet.ptc.com/PDMPJL100/install/uwgm/config.xml -t d:\uwgm_install_image -sp windows -v

```

7. The silent install portion is now complete. The silent install can be launched using silent.bat located in <Install Directory>\CLIENT directory,

The installer is now created under the directory specified as the output location. You may copy the installer to any location accessible from the client machines to install the Windchill Workgroup Manager.

- 
- Normal install — The installation can be launched by running <output directory>\CLIENT\setup.vbs
  - Silent non-user installation — If you entered details to create a silent installation script, the silent installation can be launched using <output dir>\CLIENT\silent.bat. This will install using the settings entered in the web page above (stored in the silent.properties file) to perform the installation silently with no user interface and without prompting the user for input. This can be run either by end users or used by a software deployment tool to roll out software to user machines.

## Installing the Windchill Workgroup Manager Client and Windchill Virtual File System (VFS)

After the system administrator has installed and configured the Windchill Workgroup Manager server, download the Windchill Workgroup Manager client installer to your local computer. The Windchill Virtual File System (VFS) installs as part of the Windchill Workgroup Manager client installation.

Use the following procedure to download the installer and install the client and VFS.

### Note

*The Windchill Workgroup Manager client installation includes the PTC Virtual File System installation. You must install the Windchill Workgroup Manager client on your local drive. Do not install it on a network drive.*

1. Access the **Software Downloads** page from the Windchill **Quick Links** drop-down list.
2. Select **Windchill Workgroup Manager**. The **Installing Windchill Workgroup Manager Client** page launches.

### Note

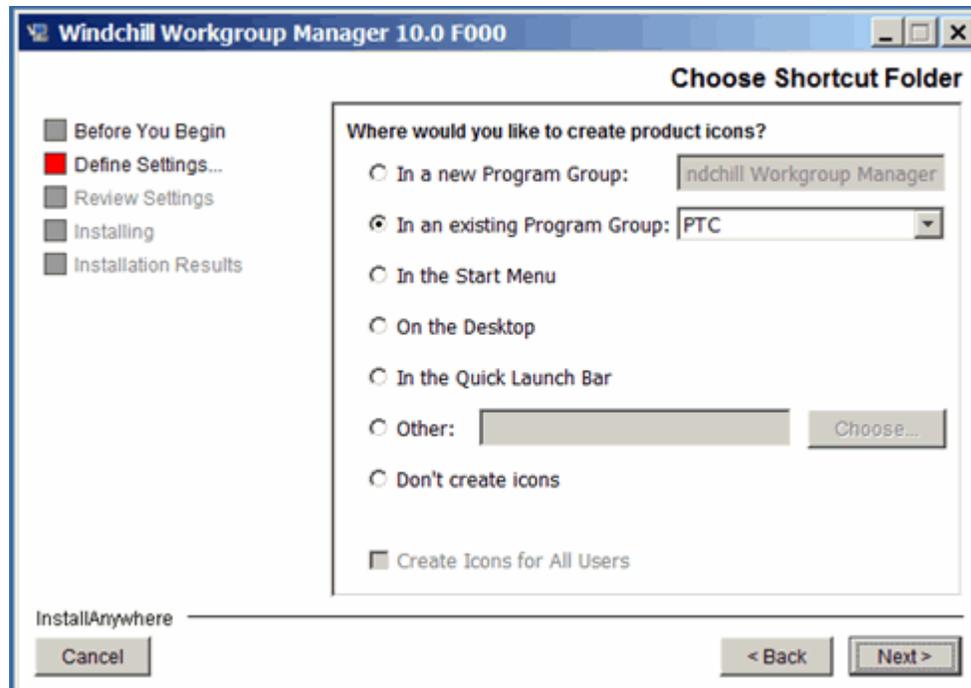
*If your Windchill System Administrator installed Windchill Workgroup Manager support for more than one application, the Installing Windchill Workgroup Manager Client page displays a link to the Administrator's and User's guide for each of the supported Windchill Workgroup Managers. You may want to save one or more of these guides to your local computer before you start the installation process. To save the guides, select the appropriate link to open its related PDF. Save the file to your local computer. To save another guide, select the appropriate link to open the desired PDF.*

3. Select **Start Installation**. The Windchill Workgroup Manager client installer launches on your local computer and the installation process begins. The first panel that appears is the **PTC** panel, indicating the language selected by the System Administrator during the CD-to-server installation.

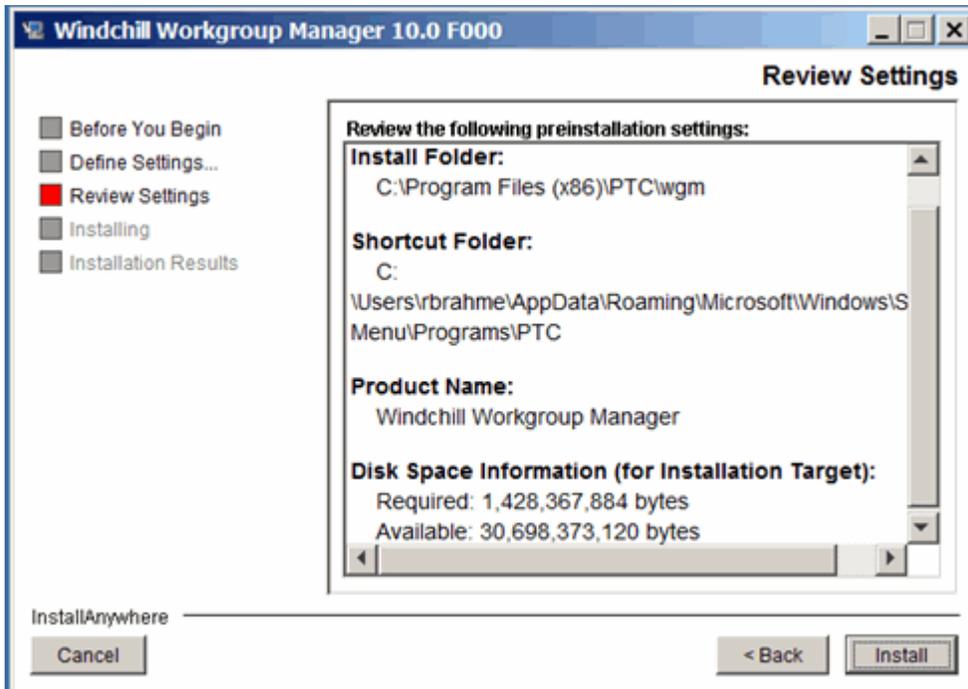
**Note**

*If the installer does not start, select **Troubleshooting Windchill Workgroup Manager Client Installation** and follow the troubleshooting instructions.*

4. Select the appropriate language and select **OK**. The **Before You Begin** panel appears.
5. Select **Next**. The **Select Directory** panel appears.
6. In the **Select Directory** panel, keep the default directory, or browse to the desired installation directory on your computer, and select **Next**. A confirmation window appears if the directory selected as the installation directory does not exist on your local computer.
7. Select **Yes** to confirm that the client installation directory is correct. If it is not correct, select **No** and browse to the appropriate directory.
8. Select **Next** to continue. For both Windows and Solaris, the **Choose Shortcut Folder** panel appears.



9. Choose a shortcut directory and select **Next**. The **Review Settings** panel appears.



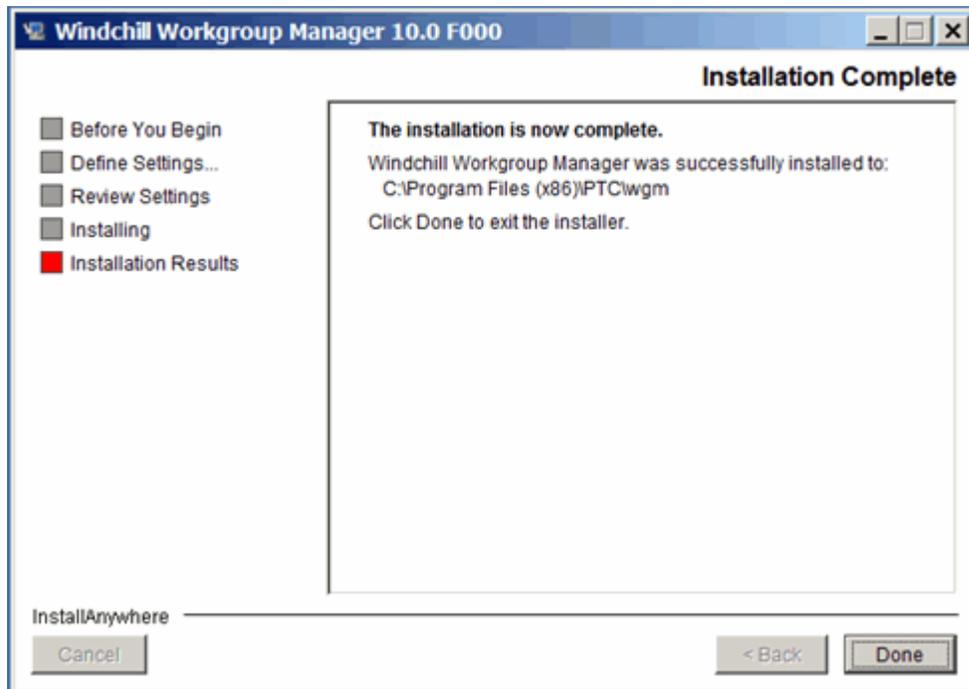
10. Review the install settings you just selected. Click **Back** to make corrections; otherwise, select **Install**. The installation process begins.

The **Installing Windchill Workgroup Manager** panel displays a progress bar to indicate the status of the installation. After the Windchill Workgroup Manager client is installed, the Windchill Virtual System (VFS) installation begins.

**Note**

Select **Cancel** to stop the installation.

11. Once the installation is complete, the **Installation Complete** panel appears.



12. Select **Done**. The Windchill Workgroup Manager client and VFS are now installed on your computer.

## Configuring the Windchill Virtual File System

After the Windchill Workgroup Manager client is installed, the PTC Virtual File System (VFS) executable file, PtcVfs.msi becomes located in the \wgm\installer directory.

The PTC Virtual File System (VFS) installation component is necessary in order to browse to PTC Places where you can perform various Windchill PDM actions, in addition to the Windchill PDM Actions that can be performed from within Creo Illustrate.

1. After the Windchill Workgroup Manager client and VFS installation is complete, configure PTC Places.
  - a. Locate PTC Places Options in the control panel of your local machine. Right-click to open PTC Places, or double-click on the file to open it.
  - b. In the **PTC Places** tab, set the folder location to the server accessible from PTC Places. For example,

```
D:\User Profiles\\PTC Places
```

- 
- c. The **Server Registration** tab isn't used to register a Windchill server. Leave this option blank.

**Note**

*Server registration is performed from Windchill Workgroup Manager Server Manager. See [Registering a Server on page](#) for additional information.*

- d. The **Application Manager** tab contains the updated applications values as soon as the application is registered or unregistered from the Windchill Workgroup Manager client. Do not attempt to update the information contained in this tab because this option does not apply to the VFS installation.

## Registering the Windchill Server

Before Creo Illustrate can connect to Windchill, you must register a Windchill server to the application and select a Windchill workspace.

After registration, you can run Creo Illustrate, click **Connect to Windchill**, and then manage your illustration files in the selected workspace.

**Caution**

*If you do not register a Windchill server to Creo Illustrate, you will not be able to connect to Windchill from Creo Illustrate.*

Use the following steps to register a Windchill server.

1. Access the **Server Management** utility in one of the following ways:
  - From an active Windchill Workgroup Manager session, select **Tools ► Server Management**. The **Server Management** window opens.
  - From Creo Illustrate, click **Configuration**  in the Windchill ribbon to open the **Server Management** window.
2. Select **Server ► Register New Server**. The **Register New Server** window opens.
3. Specify the server name to appear in the **Name** field.

**Note**

*It is recommended that all users in a company register the server using the same name.*

4. Enter the Windchill server codebase URL. (You can get this information from your Windchill administrator.)
5. Select **Check** to validate the server location is complete and valid.
6. Enter the user name and password in the **Authentication** window.

- 
7. Select the workspace you want Creo Illustrate to use from the workspace list.

**Note**

*A default workspace already exists for each Windchill product that you have access to. If you want to create new, additional workspaces, click **Workspace ► New** to create a new one. This option is only available when the server is registered and has been given primary importance.*

8. Select **Close**.
9. Select **Server ► Set as Primary Server**. This enables Creo Illustrate to connect to the specified Windchill server and use the selected workspace.

**Note**

*Once registered, the server and workspace are listed in the **Folder Navigator**.*

## Registering an Application

The operating system-specific components for integration with all supported applications and their versions are automatically installed with the Windchill Workgroup Manager client. For example, if your installation is on a machine running Windows XP, all the application integration components for Windows XP are installed, except those that were specifically excluded by the system administrator during the CD-to-Server installation.

Your Creo Illustrate application must be registered with the Windchill Workgroup Manager client before it can be used with Windchill Workgroup Manager. It is possible to register several different applications at the same time, but only one version of each at a time. If there are no applications registered with the Windchill Workgroup Manager client at startup (typically the first startup after installation), the client automatically launches the **Application Management** utility so that Creo Illustrate can be registered with the client.

When you first initialize Windchill Workgroup Manager, no applications are registered. You choose from a list, or have Windchill Workgroup Manager browse for an application in the **Application Management** utility. If you have already registered an application, you can use the **Application Management** utility to add others.

Use the following procedure to register Creo Illustrate with Windchill Workgroup Manager.

1. Access the Windchill Workgroup Manager **Application Management** utility in one of the following ways:
  - a. Select the **Tools** menu from Windchill Workgroup Manager.

- 
- b. Launch Windchill Workgroup Manager using the **Workgroup Manager**  icon in Creo Illustrate. Select the **Tools** menu from Windchill Workgroup Manager.
  2. From the **Tools** menu, select **Application Management**. The **Application Management** utility lists all of the applications that your system administrator has made available.
  3. If the application you want registered with Windchill Workgroup Manager is listed, select the application — in this case, Creo Illustrate 1.0 and click **Register**.
  4. Once the application has been located, the **Register an Authoring Application** window **Register** button becomes gray. Click **Close**.
  5. If the Creo Illustrate application you want to register cannot automatically be found on your hard drive, the following error message appears. “Automatic registration failed. Please register the authoring application manually.” Click **OK**.
    - a. From the **Register an Authoring Application** window, select Creo Illustrate 1.0 and click **Register Manually**.
    - b. Click **Find** to signal Windchill Workgroup Manager to locate the application.

Once the application has been located, the **Register an Authoring Application** window **Register** button becomes gray. Click **Close**.
    - c. If Windchill Workgroup Manager cannot automatically find the application, the **Attributes** section displays in the **Register an Authoring Application** window. Enter the **Startup Command** and **Install Directory** manually and click **OK**.
  6. Applications registered with Windchill Workgroup Manager appear in the **Application Management** window. Click on Creo Illustrate 1.0. to verify it with Windchill Workgroup Manager.
  7. The verification credentials for the application appear under the **Description** in the **Application Management** window.

## Setting the Connect to Windchill Option in Creo Illustrate

Before you can use the Windchill PDM actions in Creo Illustrate you must set the **Connect to Windchill** option from the Windchill ribbon in Creo Illustrate.

In Creo Illustrate, click the Windchill ribbon **Connect to Windchill**  icon.

---

## Unregistering an Application

This section describes how to unregister an application, like Creo Illustrate, with Windchill Workgroup Manager, and how to do so.

### **Note**

*Exit your authoring application before trying to remove it from the **Application Manager**. Windchill Workgroup Manager will not remove an application if it is running.*

1. In the Windchill Workgroup Manager window, click **Tools ▶ Application Management**. The system opens the **Application Management** window.
2. In the **Application Management** window, select the application you want to remove. In this case, select **Creo Illustrate 1.0**.
3. Click **Applications ▶ Remove** or right click the application and select **Remove**. The system unregisters the selected application with Windchill.
4. Click **Close** to exit the **Application Management** utility.

## Uninstalling Windchill Workgroup Manager Client

This section contains the necessary steps to properly uninstall Windchill Workgroup Manager client and other related files and directories.

When upgrading from a previous version of Windchill Workgroup Manager, the following must be performed in chronological order:

1. Verify that all files are checked in. (IMPORTANT! To prevent data loss, all files must be checked in before you continue.)
2. Launch the existing Windchill Workgroup Manager client with administrator logon.
3. Unregister Creo Illustrate and all the registered Windchill Workgroup Manager applications.

### **Note**

*See [Unregistering an Application on page 23](#) for instructions.*

4. Exit Windchill Workgroup Manager client.
5. Stop the PTC VFS Controller Service and Uninstall the Windchill Virtual File Service (VFS). See [Stopping the PTC VFS Controller Service and Uninstalling Windchill Virtual File Service \(VFS\) on page 24](#).

- 
6. Uninstall the existing Windchill Workgroup Manager client with administrator logon. See [Uninstalling Windchill Workgroup Manager Client](#) on page 24.
  7. Delete the remaining Windchill Workgroup Manager files and folders. See [Deleting Windchill Workgroup Manager Files and Folders](#) on page 25.

To install Windchill Workgroup Manager client, see step 8 in the [Installation and Configuration Steps](#) on page 11.

## Stopping the PTC VFS Controller Service and Uninstalling Windchill Virtual File System (VFS)

If you are upgrading from a previous Windchill Workgroup Manager, you must first stop the PTC VFS Controller service and then uninstall Windchill Virtual File System (VFS) before uninstalling the Windchill Workgroup Manager Client.

1. To stop the **PTC VFS Controller** service:
  - a. Go to the **Control Panel** in Windows.
  - b. Select **Administrative Tools** ► **Services**.
  - c. From **Services (Local)**, select **PTC VFS Controller**.
  - d. Select **Stop the service**.
2. To uninstall Windchill Virtual File System (VFS):
  - a. Go to the **Control Panel** in Windows.
  - b. Select **Add or Remove Programs**.
  - c. Select **Windchill Virtual File System** and click **Remove**.

## Uninstalling Windchill Workgroup Manager Client

If you are upgrading from a previous Windchill Workgroup Manager, you must first stop the PTC VFS Controller service and then uninstall Windchill Virtual File System (VFS) before uninstalling the Windchill Workgroup Manager Client.

There are three methods that allow you to uninstall Windchill Workgroup Manager.

- **Start** menu (Windows only) – select the uninstall icon.
- **Add or Remove Programs** (Windows only) – select **Windchill Workgroup Manager** and select **Remove**.
- The following procedure:

- 
1. Navigate to <Windchill Workgroup Manager Install Directory>\installer\uninstall.
  2. Double-click the file Uninstall Windchill Workgroup Manager.exe.

**Note**

*On UNIX the file name is  
Uninstall\_Windchill\_Workgroup\_Manager.*

The **Uninstall** panel appears.

Select **Uninstall**. The **Uninstall Windchill Workgroup Manager Status** panel appears.

3. When the uninstall is complete, the **Uninstall Complete** panel appears. Select **Done**.

There are instances where the uninstall fails. This is caused by an issue in InstallAnywhere. If your uninstall fails, then complete the following procedure:

1. Navigate to Program Files\Zero G Registry and delete the file .com.zerog.registry.xml.

**Note**

*On UNIX, the file is located in the user home directory.*

2. Reinstall Windchill Workgroup Manager into the same directory as the original installation.
3. Rerun the uninstall program documented in this section.

## Deleting Windchill Workgroup Manager Files and Folders

After you have uninstalled the Windchill Workgroup Manager client, delete the following Windchill Workgroup Manager files and folders.

Delete the Windchill Workgroup Manager files and folders from the following locations:

1. The Cache directory <PTC\_WF\_ROOT>, if set.
2. The directory and files in the Application registry location, <APPDATA>\PTC\ProENGINEER\Wildfire,  
or,

The .wf and .wwgm directories and files that are created under a location.

3. The directory and files in the Application registry location, <APPDATA>\PTC\ProENGINEER\Wildfire.

- 
4. The Virtual File System (VFS) Cache directory <PTC\_VFS\_ROOT>, if set  
or,  
The /wwgm directory that is created under <User Profile Location>
  5. The Workspace Local Directory (WLD) <PTC\_WLD\_ROOT>
  6. The Application registry location <APPDATA>\PTC\Windchill Workgroup Manager>  
or,  
The <PTC\_WGM\_ROOT>, if set.
  7. Clear the Java plug-in cache.

# 2

## Windchill Workgroup Manager Overview

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This section is intended for novice and experienced users. It provides a general overview of Windchill Workgroup Manager, its architecture, and a typical first-use scenario.

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## What is Windchill Workgroup Manager?

Windchill Workgroup Manager provides a common client application that allows you to manage data generated by Creo Illustrate in Windchill. A typical user of Windchill Workgroup Manager can manage the design process in a controlled, collaborative manner within the Windchill environment.

The Windchill Product Data Management (PDM) actions, such as check in and check out, can be initiated from the Windchill Workgroup Manager client application, via the embedded Windchill ribbon in Creo Illustrate. The Windchill ribbon is active when the authoring application is registered with Windchill and the Windchill preference is set in Creo Illustrate.

Windchill Workgroup Manager connects your authoring application with Windchill to provide the data management and life cycle management capabilities offered by Windchill PDMLink, Pro/INTRALINK, Arbortext Content Manager, and ProjectLink. Accessing data through Windchill Workgroup Manager provides the same consistent set of core features that are available when accessing Windchill through a standard internet browser. Examples of this are opening files from an authoring application supported by Windchill Workgroup Manager, such as a Creo Illustrate file.

In addition to Creo Illustrate, Windchill Workgroup Manager common framework supports authoring applications, such as Arbortext IsoDraw, AutoCAD, SolidWorks, CoCreate, Mathcad and CATIA V5. With Windchill Workgroup Manager, you only need to launch one authoring application in order to work with any of your company's supported applications and Windchill. For example, if you use Creo Illustrate for one project and SolidWorks for another, once you open Windchill Workgroup Manager, you can easily switch between the two applications.

Windchill Workgroup Manager supports two modes: **standalone** and **connected**.

- Standalone (not connected to Creo Illustrate) – Windchill Workgroup Manager can be launched on its own, so that you can access your data stored in the Windchill database or in your private working area (workspace) without having to start Creo Illustrate. This also allows you access to Windchill through the Windchill Workgroup Manager HTML browser.
- Connected (connected to Creo Illustrate) – Windchill Workgroup Manager can be launched with Creo Illustrate so the two are connected.

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The Windchill Workgroup Manager interface is automatically presented in Creo Illustrate once the following conditions are set:

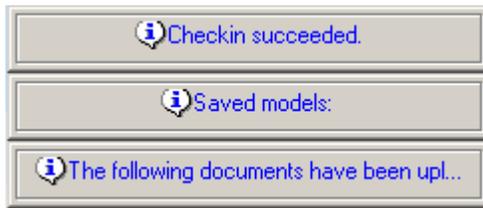
- Windchill Workgroup Manager client is installed.
- Creo Illustrate is registered with Windchill.
- The **Connect to Windchill** toggle button is selected on the Windchill ribbon.

The Windchill Workgroup Manager user interface is exited, the Windchill ribbon becomes grayed out and control of the authoring application is returned to the user. The Windchill ribbon remains in the Creo Illustrate application as long as the authoring application remains registered with the Windchill server.

Information on transactions performed by Windchill Workgroup Manager is available from the Windchill Workgroup Manager client message area. The following graphic illustrates information messages in the message area of an Creo Illustrate file example.



A pop-up window displays when Windchill Workgroup Manager is minimized, or an action is initiated from Creo Illustrate. The graphic below shows a pop-up window containing information notifying a user of actions that have succeeded.



More detailed transaction information is also available from the event console, accessed directly from the Windchill ribbon **Event Manager**  button in Creo Illustrate, or from the **Event Management**  icon in the Windchill Workgroup Manager client.

In addition to Windchill Workgroup Manager for Creo Illustrate documentation found in this section, also refer to online help in the **Using Windchill Workgroup Manager ► Understanding Windchill Objects and PDM Concepts** in the Windchill Workgroup Manager Help Center to learn about the various Windchill objects and PDM concepts.

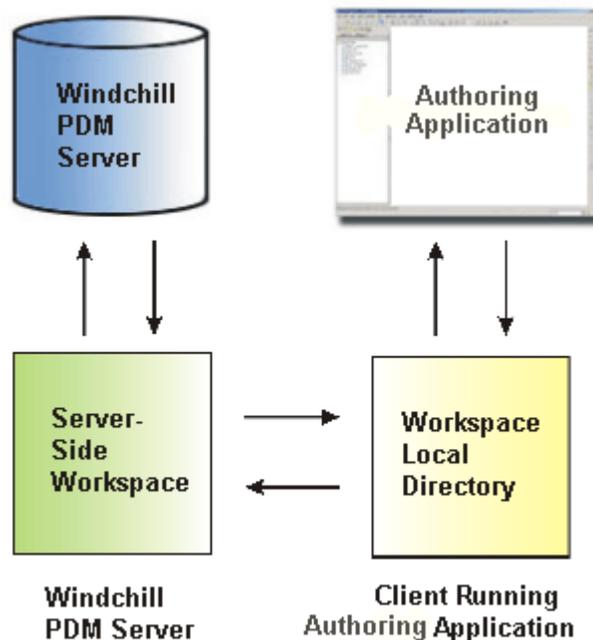
---

## Windchill Workgroup Manager PDM System Architecture

The basic architecture of Windchill Workgroup Manager Product Data Management (PDM) system is comprised of the following:

- The authoring application (e.g. Creo Illustrate).
- The Windchill Workgroup Manager client, which connects the authoring application and Windchill.
- A workspace on the server.
- The Windchill database.

The following graphic illustrates the Windchill Workgroup Manager architecture.



Once the Windchill Workgroup Manager client is installed, your application has been registered, and you've connect to a Windchill server, you can begin working with your data you can begin working with your data.

---

# Initial Windchill Workgroup Manager Scenario

The Windchill Workgroup Manager application must first be installed on the same computer as the Creo Illustrate client. The client software is downloaded from the Windchill server. The first time you use Windchill Workgroup Manager, after installation, you need to perform the following basic configuration before you can start using it.

- Set one of the registered Windchill servers as the primary server.
- Create and select a workspace as your active workspace.

This basic configuration is discussed below.

1. Configure server importance. Before you can manage any Creo Illustrate files in Windchill, you need to give one of your registered servers primary importance. Primary importance means that Windchill Workgroup Manager and Creo Illustrate recognize and communicate with the Windchill server that is managing the data. Use the following steps to give a server primary importance.
  - a. From Windchill Workgroup Manager, select **Tools ► Server Manager** and right-click the wanted registered server node. Select **Set as Primary Server**.
  - or
  - b. From the **Windchill** ribbon in Creo Illustrate, click the **Configuration**  button. Right-click the wanted registered server node and select **Set as Primary Server**. You can also register a new server and set it as the primary server.
2. Select a workspace. Select an area in Windchill to manage your working data. This working area is called a workspace. You can select a default workspace on Windchill, or create your own.

### **Note**

*You may want to create several workspaces to manage the different projects on which you are currently working.*

This workspace is private to you and is used to interface with your Creo Illustrate dynamic documents (illustrations). When you work in this workspace, your files are not visible to others until they are checked in to a Windchill server. Once checked in to Windchill, these files are available to others.

3. Create data and save it to your workspace. Once registered with Creo Illustrate, Windchill Workgroup Manager simultaneously starts when

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Creo Illustrate starts. When you create a drawing an Creo Illustrate illustration and save it to your workspace, and it is saved locally.

4. When you are ready to share an Creo Illustrate file with others, you can either:
  - a. From your workspace in Windchill Workgroup Manager, select Check In/Auto  or Check In/Custom .
  - or
  - b. From Creo Illustrate, select Check In  or Custom Check In .

**Note**

*Check In/Auto  automatically checks in the Creo Illustrate file to server side active workspace.*

*Check In/Custom  allows you to select the location (server-side active workspace, or a different server-side workspace).*

## Client and Server Architecture of Windchill Workgroup Manager

Windchill Workgroup Manager has a client-server architecture. The following components reside on an end user's workstation:

- The authoring application (i.e., Creo Illustrate) that is registered with Windchill Workgroup Manager.
- The workspace local directory – the local cache folders connected to the workspace located on the Windchill server.
- Windchill Workgroup Manager application that communicates between the authoring application and the Windchill server.

The following components reside on the Windchill server:

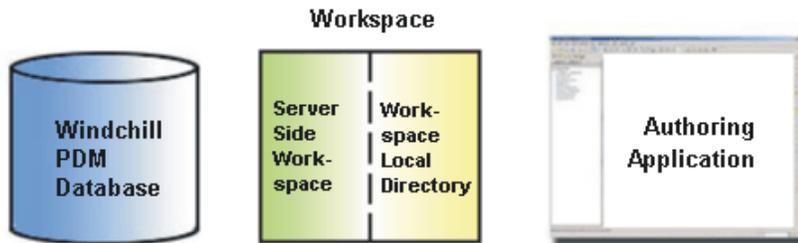
- Windchill database.
- Server-side workspace.

The workspace consists of two components:

- Server-side workspace.
- Workspace local directory that stores the physical files so they can be accessed by the authoring application.

---

The next diagram shows the main components of the Windchill Workgroup Manager.



## Windchill Workgroup Manager Capabilities

This section introduces the concept of the workspace, and provides a general overview of the commonly used Product Data Management (PDM) functions. Windchill Workgroup Manager is used to manage your data in Windchill, so it is controlled by revision and managed through its life cycle. This helps ensure data integrity is maintained.

### Managing Data Through the Workspace

Your data is managed through a workspace that is split between the client and the server. These workspace components are:

- workspace local directory – client-side cache for files currently being worked on.
- server-side workspace – a user's private working area which is synchronized with the files in the workspace local directory.

The workspace local directory is a cache area on the client computer through which the authoring application (i.e., Creo Illustrate) can access and modify your data. When you have completed work on this data, but don't want it to be shared, you upload it to the server-side workspace, located on the server. The server-side workspace is also a private area, and is accessible only by you. The server-side workspace is used to store your current work-in-progress, allowing you to save data from, and retrieve data to, your authoring application. Once you are ready to

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share this data with others, you check it in to the commonspace also located on the Windchill server. The difference is that the commonspace is viewable to anyone who has access to it; typically, your team members.

To learn about workspaces, refer to **Using Windchill Workgroup Manager ► Understanding Windchill Objects and PDM Concepts ► About Workspaces** in the Windchill Workgroup Manager Help Center.

## Common PDM Actions

When you work connected to a Windchill server (the server you have given primary importance) in an active workspace, PDM actions become available in your authoring application (i.e., Creo Illustrate) to help you manage your data in Windchill.

All files remain in your control until you are ready to share them with others. Use either **Save** or **Save As** from the Creo Illustrate **File** menu to save any changes you have made to a drawing. Then use **Check In/Auto**  to move your files to the server-side workspace where they can be accessed by others, or choose use **Check In/Custom**  to move your files to a server-side active workspace or the Windchill Cabinet where they can be accessed by others.

### Note

**Synchronize** ensures the contents of the workspace local directory cache are the same as the contents in the server-side workspace. The command is only available through the Windchill Workgroup Manager workspace browser.

The **Check Out**  function enables the person who performs the checkout to modify the data being checked out. Once checked out, the file cannot be modified by anyone else. The file can be viewed by others, but can only be modified by the checkout owner. This exercise of checking in and checking out ensures data integrity is maintained because the file is always controlled through Windchill.

PDM actions available from within Creo Illustrate are documented under **Windchill Workgroup Manager Integrations ► Windchill Workgroup Manager for Creo Illustrate Administrator's and User's Guide** in the Windchill Workgroup Manager Help Center.

PDM actions available from the Windchill Workgroup Manager client are documented in [Working with Windchill PDM Actions on page 53](#).

# 3

## Getting Started with Windchill Workgroup Manager

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This section is intended for novice and experienced users, as well as administrators who may want to learn more about using Windchill Workgroup Manager. This chapter covers actions such as registering a server and registering an authoring application. It also covers navigators, workspaces and viewing objects. For more experienced users, this chapter can be used to review the basic elements before getting started with Windchill Workgroup Manager.

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## About Windchill Workgroup Manager

This section provides information about using Windchill Workgroup Manager to manage your Creo Illustrate illustrations with Windchill.

You can run Windchill Workgroup Manager in stand-alone mode, where Creo Illustrate only becomes available when certain commands are given, or in full-client mode, where Creo Illustrate is connected to Windchill Workgroup Manager.

This chapter discusses the following:

- Registering Windchill servers.

### **Note**

*Server registration is the first step you take to get started with Windchill Workgroup Manager, and is one that you always return to when you need to set up another server and workspace. When you access a server or workspace, it is made visible through an HTML page in Windchill Workgroup Manager.*

- Registering Creo Illustrate with Windchill.
- Windchill PDM actions available from Creo Illustrate and Windchill Workgroup Manager.
- The Windchill Workgroup Manager HTML browser and navigators.
- Workspace management, including, activating and deactivating a workspace.
- Viewing Creo Illustrate files.

## Initiating the Windchill Workgroup Manager

Windchill Workgroup Manager can run in stand-alone mode, or concurrently with Creo Illustrate. Once you have installed Windchill Workgroup Manager to your desktop, and registered Creo Illustrate with Windchill, the choice is yours as to how you want to initiate Windchill Workgroup Manager. Detailed information can be found in the Windchill Workgroup Manager Client PDM Actions chapter.

## Running Concurrently with Creo Illustrate

In most cases, you work in Windchill Workgroup Manager concurrently with Creo Illustrate. This mode allows you to easily interact between Creo Illustrate and Windchill. Once you have registered with Windchill Workgroup Manager and

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selected the **Connect to Windchill** toggle button on the Windchill tab ribbon, Windchill Workgroup Manager launches automatically when Creo Illustrate is initiated if Windchill Workgroup Manager is running.

## Running Standalone from the Desktop

There may be occasions where it is more convenient to run Windchill Workgroup Manager alone, and not connected to Creo Illustrate. For example, you may want to check out files from Windchill Workgroup Manager, which does not necessitate being connected to Creo Illustrate.

When working in stand-alone mode, Windchill Workgroup Manager provides access to the Windchill server, and the workspaces available to you. You can check out files to your hard drive and change attributes, or perform non-file-based operations. These updated files can be uploaded to Windchill once Creo Illustrate is connected to Windchill and Windchill Workgroup Manager is invoked.

You can invoke Windchill Workgroup Manager using the icon in your **Start** menu, or from your desktop. Additionally, users can invoke Windchill Workgroup Manager through the `Home/Bin/uwgm_client.bat` file.

## About the Windchill Workgroup Manager User Interface

This section describes the Windchill Workgroup Manager HTML browser.

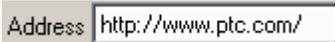
The Windchill Workgroup Manager user interface is presented as a series of HTML pages accessible through your Web browser. Windchill Workgroup Manager browser is similar to Microsoft's Internet Explorer.

The next graphic displays the Windchill Workgroup Manager browser toolbar.



The following commands appear on the Windchill Workgroup Manager browser menu bar:

- **Back** ← – Select to go to the previous page in the browser's history list.
- **Forward** → – Select to go to the next page in the browser's history list.
- **Stop** ⓧ – Select to stop loading the current page.

- 
- **Refresh**  – Select to refresh the current page.
  - **Home**  – Select to open the user-defined home page.
  - **Save**  – Select to save the current browser contents as a file.
  - **Print**  – Select to print the contents of the current browser page.
  - **Favorites Navigator**  – Select to view the favorites (bookmarks) you have set in Windchill Workgroup Manager.
  - **History Navigator**  – Select to view the browser history.
  - **Address Box**  – Select to enter a URL, then select **Go** to open the Web page.

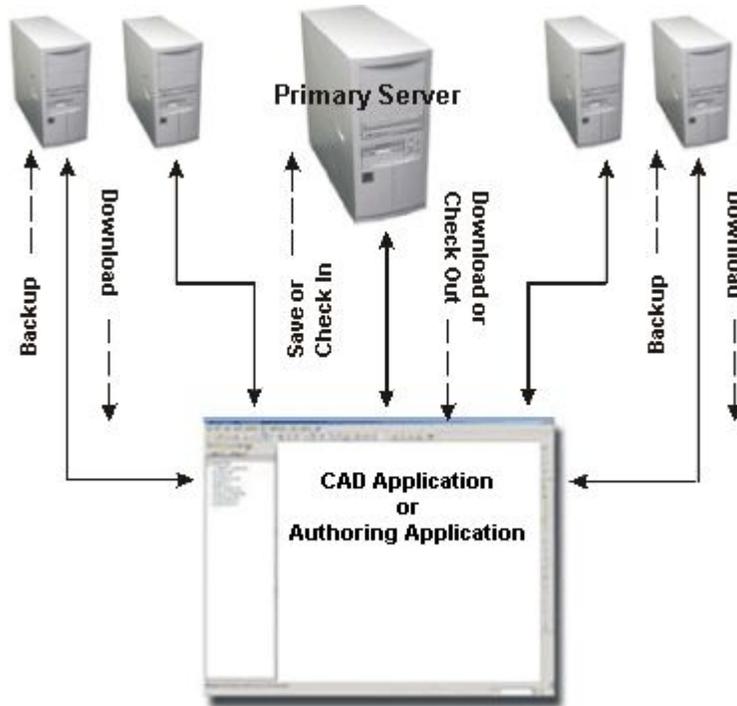
### **Note**

*The Windchill Workgroup Manager HTML browser is the only browser that supports the connection between your authoring application and a Windchill server. Accessing the Windchill server in an external Internet Explorer or Mozilla browser will not create connectivity between Creo Illustrate and Windchill.*

## **Using Windchill Workgroup Manager**

The Windchill Workgroup Manager HTML user interface supports the connectivity between Creo Illustrate and Windchill. You register Windchill servers that are designated to contain the data you work with. Windchill Workgroup Manager allows connections to multiple Windchill servers. One server is always set as primary, and any additional servers that you register are considered secondary servers. The next graphic illustrates how a primary server

and secondary servers connect with Windchill Workgroup Manager. In addition to authoring applications, like Creo Illustrate, Windchill Workgroup Manager also supports CAD applications.



## Primary and Secondary Servers

Windchill Workgroup Manager allows you to simultaneously browse to multiple Windchill servers. However, it is necessary to designate one server as the primary server.

The primary server acts as the default location for all storage and retrieval using the PDM functions performed with Windchill. When it is required that you work in another server, you set it as primary through the **Server Manager** option on the Windchill Workgroup Manager **Tools** menu, right-click and select **Set as**

**Primary** in the **Folder Navigator**, or from the **Windchill** ribbon **Configuration**  icon in Creo Illustrate.

Secondary servers are typically used for backing up data, for libraries, or for sharing data with other users that do not have access to your primary server (for instance Windchill ProjectLink servers that are placed on a different part of the company network, such as outside the company's firewall), or test servers. For example, you could create a project on your company's Windchill server, and then invite suppliers to share this data. You would not set this server as primary. This secondary server gives your suppliers the ability to read and write to that project.

---

## Working with Windchill Servers

This section explains how you work with Windchill servers in Windchill Workgroup Manager.

To create the connection to a Windchill server, you register the Windchill server and select a workspace. At this point, if you want to work on this server, you set it as the primary server.

Once registered, the server and workspace are listed in the **Folder Navigator**.

### Using the Server Management Utility

To register and manage servers in Windchill Workgroup Manager, use the **Server Management** utility. To access the **Server Management** utility, select **Tools** ► **Server Management**, or from Creo Illustrate, click **Configuration**  in the Windchill ribbon to open the **Server Management** window. For information on how to register a Windchill server, refer to [Registering the Windchill Server on page 20](#).

#### **Tip**

*After you have registered a server, you can access the **Server Management** window from the **Folder Navigator** by right-clicking the server or the workspace node.*

The **Server Management** window has three menus:

- **File**

Opens the **Save Server Management Settings File** window, and allows you to save a Servers Configuration File (FLD) to a common folder, or server, workspace.

- **Server**

Allows you to register a new server or edit an existing server.

- **Workspace**

Allows you to create a new workspace.

There are two tabs available from the **Server Management** window:

- Servers
- Cache

The **Servers** tab allows you to view the server name(s) and register, edit server name and delete the server.

- **Servers**

---

Displays the registered servers and their online or offline status. Also displays **<NO SERVER>** which, when set as primary, disables all PDM actions in the authoring application.

### **Note**

*You may want to use this option when you want to restore all the basic behaviors of the authoring application without having to unregister any Windchill servers.*

The registered servers can be viewed from the **Server Management** window.

- The **Cache** tab shows you how much disk space is used by the workspace local directory cache per server or workspace, allows you to clear the cache, and provides access to the Cache Management tools. For more information on cache, refer to the Managing Cache section.

## **Connecting to a Server**

To create the connection to a Windchill server, you register a server and select a workspace. At this point, if you want to work on this server, you set it as the primary server.

Once registered, the server and workspace are listed in the **Folder Navigator**.

## **Using the Server Tab**

The **Server** tab on the **Server Management** window contains three sections: **Servers**, **Server Active Workspace**, and **Description**.

- **Servers** - Lists the servers that are registered with Windchill Workgroup Manager. You can right-click a server and register another server, set it as primary (if it is not already), and edit the server name.
- **Server Active Workspace** - Lists the workspaces on the selected server. To list available workspaces on any registered server, select a server from the Servers list. Use the workspace commands to do the following:
  - **New**  
Creates a new workspace on the primary server.
  - **Activate**  
Activates a workspace on a secondary server, or activates a workspace and sets it as primary.
  - **Delete**

---

Deletes the server.

- **Description** - Describes the server name, location and primary active workspace name.

## Using the Application Management Utility

Use the **Application Management** utility to register or unregister Creo Illustrate with Windchill Workgroup Manager. To access the **Application Management** utility, select **Tools ► Application Management** from Windchill Workgroup Manager, or launch Windchill Workgroup Manager in Creo Illustrate using the Windchill ribbon **Workgroup Manager**  icon and then select **Tools ► Application Management**.

## Registering Creo Illustrate

Creo Illustrate must be registered with Windchill Workgroup Manager. When you first initialize Windchill Workgroup Manager, no applications are registered. You choose from a list, or have Windchill Workgroup Manager browse for an application in the **Application Manager**. If you have already registered an application, you can use the **Application Management** utility to add others.

For information on how to register Creo Illustrate with Windchill Workgroup Manager, refer to [Registering an Application on page 21](#).

## Unregistering Creo Illustrate

A use scenario for unregistering Creo Illustrate with Windchill Workgroup Manager is when you are upgrading to a new version of Windchill Workgroup Manager. For information on how to unregister Creo Illustrate with Windchill Workgroup Manager, refer to [Unregistering an Application on page 23](#).

## About Navigators

Windchill Workgroup Manager has four navigational tools available that allow you to easily access and manage your work. With these tools, you can browse servers and workspaces, manage your **Personal Favorites** folder, and view the history of the HTML pages you have accessed. You can also create and save a full search of your Windchill documents.

The following navigator tools are located on the browser window ribbon:

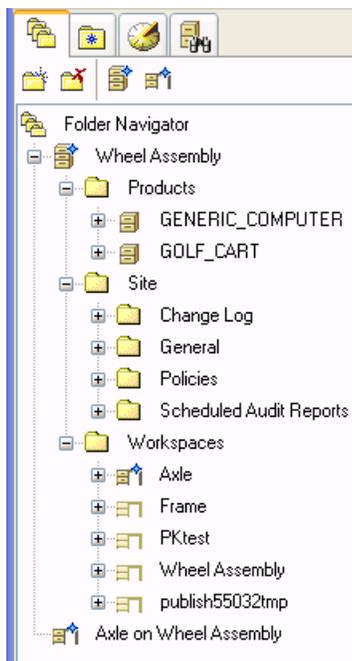
- Folder Navigator  – Select to manage folders.
- Favorites navigator  – Select to manage favorites.

- History navigator  – Select to manage Web history.
- Search navigator  – Select to perform and manage file searches.

## Folder Navigator

This section describes the **Folder Navigator**, which is located on the left-hand panel in Windchill Workgroup Manager. The **Folder Navigator** is an expandable tree that displays the registered and primary active servers, contexts and workspaces.

The next graphic illustrates the **Folder Navigator**.



You can browse Windchill servers and workspaces connected to your computer in the **Folder Navigator**. The **Folder Navigator** is useful for quickly accessing and managing your Windchill data. Select the folder navigator  icon to open the **Folder Navigator**. From here, you can access the **Server Registry** window to register new servers or manage your cache. In the **Folder Navigator** window, you browse different servers and workspaces, set a server as the primary one through which you work with Windchill, and activate its workspace. You can also expand or collapse your folder directory, and navigate directly to your Windchill cabinets and workspaces. Once set, the primary server and active workspace are indicated by the primary server icon , and the active workspace  icon. As you navigate the **Folder Navigator** window and select a server or workspace, the contents appear in the browser so that you can work on them.

---

To move directly to your primary server or active workspace click one of the following icons located at the top of the **Folder Navigator**.

 — Moves directly to the registered server.

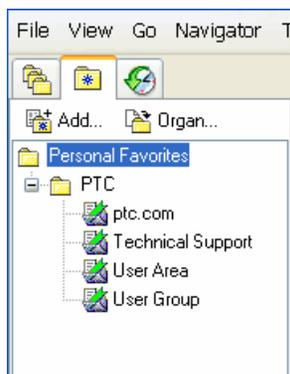
 — Moves directly to the primary, active workspace.

## Favorites Navigator

The **Favorites** navigator is useful for bookmarking Web sites you visit frequently.

Click  to open the **Favorites** navigator.

The following graphic displays the **Favorites** navigator panel.



Here, you manage your **Personal Favorites** folder by adding, renaming, or deleting links to HTML pages represented in the browser. You can add to the list of favorites by accessing a site in the browser and clicking **Add** in the **Favorites** navigator. You can also organize your favorites by name, location, last visited, created on, and number of visits by clicking **Organize** in the **Favorites** navigator. You can also add bookmarks by editing the `wtcad.ini` file and adding a name and address for the favorite in the format:

```
; add my own url GOOGLE=http://www.google.com/
```

### Adding a Location to Your Favorites

1. Go to a site or directory you want to save to Favorites. The Windchill Workgroup Manager browser displays the page you want to save.
2. Do one of the following:
  - Right-click within the Windchill Workgroup Manager browser and click **Add to Favorites** on the shortcut menu.
  - Select add to favorites  icon at the top of the **Favorites** navigator.

3. In the **Add Favorite** window, select a folder in which you want to save the favorite, or create a new folder.
4. Select **OK**.

## Organizing Favorites

You can add, rename, delete, and move favorites in the **Favorites** navigator.

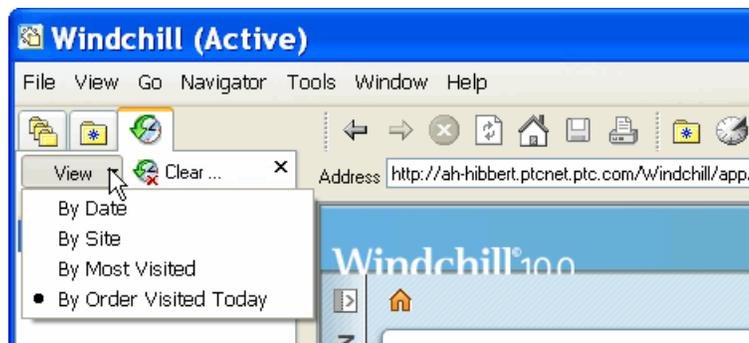
1. Select the favorites navigator  icon.
2. Select the organize  icon at the top of the Favorites navigator. The **Organize** window opens.
3. Select the following options:
  - Create folder.
  - Rename.
  - Delete.
4. To show information about folders, select **View Info** at the bottom of the window.
5. Select **Close**.

To reorder favorites in the **Favorites** navigator, drag an item to a new location.

## History Navigator

The **History** navigator is useful for returning quickly to HTML pages that you found particularly helpful, but are not entered in the **Favorites** navigator. Select the history navigator  icon to open the **History** navigator, and display what you have opened in the browser.

The following graphic displays the **History** navigator window.



The view can be filtered by **Date**, **Site**, **Most Visited**, or **By Order Visited Today**. If you want to clear all these sites, select **Clear**.

---

## Search Navigator

This section provides an overview of the **Search** navigator, and how to search for an object and open it in Creo Illustrate. It explains the workflow when using saved searches, and also for adding an object to the workspace.

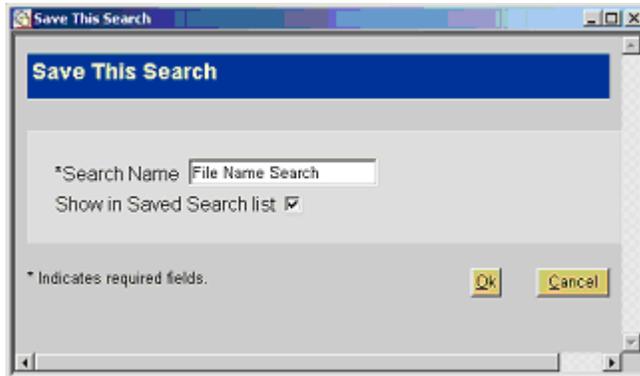
The **Search** navigator is used to search for files located in Windchill. You can search using criterion used before (a saved search), or through a customized search. The **Search** navigator is useful for searching on basic criteria, advanced criteria, and complex Windchill searches. Search results display in a narrow table below the **Search** navigator, and in a wide table view in the browser window, which displays more detailed information about the search result.

The **Search** navigator displays results for registered, primary servers only. When there is no registered, primary server, the **Search** navigator is not available.

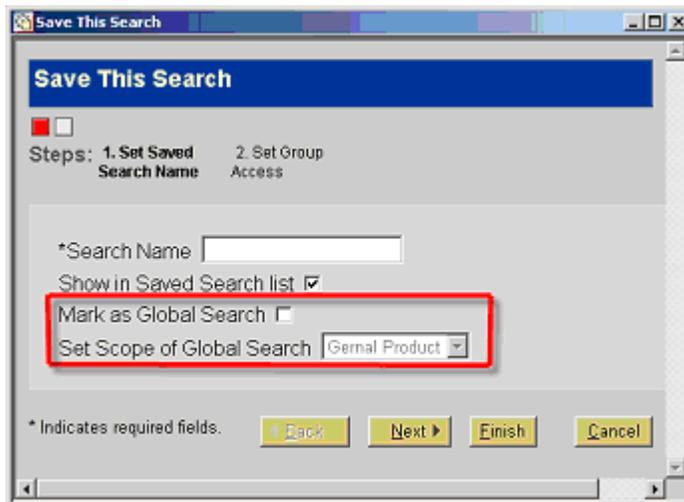
Select the search  icon to open the **Search** navigator. The **Search** navigator allows you to search for parts and dynamic documents on a registered, primary server. All searches can be saved for future use. For example, to perform a basic search in the **Search** navigator, specify the criteria and select **Search**. If you want to save this search, select **Save This Search**.

You can search by several criteria. For example, you can search for parts created in any supported CAD or authoring application, Owner application, Dynamic Document type, Dynamic Document sub-type, Check out by, Created by, Updated by, Updated on, and Organization ID. The search results are returned in a **Search Results** table that lists the name, number, context, version, organization, state, last updated, created on, and authoring application. You can select a part or dynamic document from the results table, and apply a Windchill action, such as **Check Out**, to it. Once you have specified your search criterion and performed a search, you can use the **Save This Search** button on the **Search** navigator to save the search for future use. Saved searches can be made available to others by adding specific groups once the **Save This Search** button has been activated. An administrator enables the **Mark as Global Search** and **Set Scope of Global Search** check boxes in order to set up **Saved Searches**.

The **Search** user-interface depends upon the type of user performing the search. A typical user sees the following window.



An administrator sees the next window.



To perform more advanced searches, select the **Advanced Search** link on the **Search** navigator.

Use the following procedure to create an advanced search.

1. Select **Advanced Search**.
2. Identify the Context for the search. The default context is the Product, Library, or Project associated to the active workspace.
3. Identify the type of dynamic document to search for.
4. (optional) Identify the date range to search.
5. Select the criteria in the **Search For** field.
6. Set the operator for the criteria (equals, not equal to, between).
7. Set the value for the criteria, including any wildcards.

- 
8. Apply the criteria by choosing the **Add** button. The criteria is added to a list where it may be selected for edit or removal.
  9. Add another criterion.
  10. Repeat steps 6 through 9 for each desired criterion.
  11. When criteria setting is complete, set whether the search results will “Match All Criteria” on the list, or any criteria on the list.
  12. Select **Search**. Results are presented in a list beneath the search criteria and also in the main browser window.

From the search results under the Advanced Search criteria you may:

- Select the open in Creo Illustrate  icon to open the file in Creo Illustrate (this command is only available if Creo Illustrate is registered).
- Select the **Name** link to view the information page for the selected object in the main browser window where you can see detailed information related to that object, and perform actions on that object.

From the search results in the main browser window, you may either access the information page or choose an action to perform on the object from the drop-down list (such as **Add to Workspace** or **Check Out**). If you want to save an object from the narrow search results table, select the object link to display its information page in the browser window.

If you prefer to perform a complete search of Windchill, access the **Full Search** link. This opens a page in the browser and allows you to browse the commonspace and search in detail. For more information on search, refer to the Windchill online help.

## Accessing Workspaces

This section describes the concept of accessing the workspace and the local workspace directory.

Windchill Workgroup Manager provides you with a private area on the Windchill server for managing your work while working with other designers. This area is called the workspace. The workspace allows you to track and change multiple objects and perform data management operations from within the authoring application user interface. The workspace enables designers to operate independently while recording and tracking concurrent activities to assist in product design decisions.

The primary purpose of the workspace is to allow designers to work freely on design development and changes without impacting data in the database until they are ready to expose their design or changes to the rest of the design community by checking in to the commonspace.

---

As a Windchill user, you typically check out or download design information from the Windchill server to your workspace to modify or view it. Checkout reserves the object on the server, so that no one but you can modify the design (although other users can still access the data on a read-only basis). When you are finished modifying a design and you want to share the modification with other users, you perform a check-in operation. The check-in operation uploads your modified design to the Windchill server and releases the object on the server allowing other users to access and modify the data further. The process of checking out and checking in (reserving and releasing) data helps you to maintain the integrity of your data.

It is through the workspace that the Windchill server connects to Creo Illustrate. The Creo Illustrate application session and the workspace are linked such that information that is modified in Creo Illustrate can be seen from Windchill.

**Tip**

*The Windchill system can create multiple workspaces. This is useful if you are working on several projects at one time because it allows you to create a workspace for each of your projects and segregate your design data by project affiliation.*

## Accessing Service Information Manager and Service Parts

If you have Service Information Manager and Service Parts installed, you can access the Service Information Manager and Service Parts functionality to add, edit, and delete illustrations to a Parts List or to add, edit, and delete illustrations in the active workspace or commonspace for Service Information Manager and Service Parts.

A workspace is a private area on the Windchill server for managing your work while working with other Windchill users. This area is called the workspace. The workspace allows you to track and change multiple objects and perform data management operations from within Creo Illustrate and from the workspace. The workspace enables Windchill users to operate independently while recording and tracking concurrent activities to assist in the production of a service manual or parts catalog.

The primary purpose of the workspace is to allow Windchill users to work freely in the commonspace without impacting data in the database until they are ready to expose their illustrations to the rest of the Windchill community by checking in the illustration to the active workspace.

---

As a Windchill user, you typically open, check out, or download an illustration from the Windchill server to your workspace, when Creo Illustrate is connected to Windchill, to modify or view it in Creo Illustrate. Checkout reserves the object on the server, so that no one but you can modify the illustration (although other users can still access the data on a read-only basis). When you are finished modifying an illustration and you want to share the modification with other users, you perform a check-in operation. The check-in operation uploads your modified illustration to the Windchill server and releases the object on the server allowing other users to access and modify the illustration further. During check in you have the option of picking a folder location, or browsing to a product ASPS location. You can also provide an optional check in comment. The process of checking out and checking in (reserving and releasing) data helps you to maintain the integrity of your illustration.

It is through the workspace that the Windchill server connects to Creo Illustrate. The Creo Illustrate application session and the workspace are linked such that information that is modified in Creo Illustrate can be seen from Windchill.

### **Tip**

*The Windchill system can create multiple workspaces. This is useful if you are working on several projects at one time because it allows you to create a workspace for each of your projects and segregate your design data by project affiliation.*

For more information about the Windchill PDM actions you can perform from within Creo Illustrate, refer to the **Windchill Workgroup Manager** Help Center under **Using Windchill Workgroup Manager ▶ Authoring Applications ▶ Creo Illustrate**.

For more information about Windchill PDM actions you can perform from the Windchill workspace, refer to [Working with Windchill PDM Actions on page 53](#), and [Windchill PDM Actions in Service Information Manager and Service Parts on page 71](#), or from the **Windchill Workgroup Manager** Help Center under **Windchill Workgroup Manager Integrations ▶ Windchill Workgroup Manager for Creo Illustrate Administrator's and User's Guide ▶ Working with Windchill PDM Actions**.

For more information about Service Information Manager and Service Parts, refer to the **Windchill** Help Center under **Specialized User Capabilities ▶ Service Information Manager**.

## **Viewing Windchill Objects**

Windchill objects are viewed through the workspace in Windchill, so that you can easily keep track of what you are working on. Once objects are checked in, they are visible through the folders menu in the commonspace. The commonspace is the area accessible to others who have permission to it, such as other project team members.

---

Once you save an illustration in Creo Illustrate, it appears in your private workspace in Windchill. To access this workspace, click the primary workspace node in the Windchill Workgroup Manager **Folder Navigator**. The HTML user interface displays the models you have saved. The workspace view can be customized through the Custom Table view in Windchill.

Because dynamic documents in a workspace can differ from the commonspace version, Windchill Workgroup Manager can present a workspace information page and a commonspace information page. The information page called from the workspace presents the workspace-relevant information. (Even though you can bring parts and end-objects into a workspace, they are commonspace objects only, and therefore only have commonspace information pages.) From the workspace, select the information  icon to present an information page for the workspace version of an object. Select the information  icon for an object in the commonspace accesses the information page for the commonspace version of the object.

## Workspace Management Actions

This section describes the concept of managing workspaces, and includes switching between existing workspaces and creating new ones.

You work in a private area in Windchill called a workspace. To interact with Creo Illustrate, a workspace must be set as active on the primary Windchill server. This can be done from the **Server Manager** window, which is accessed from the Windchill Workgroup Manager **Tools** menu, or by right clicking on the workspace node in the **Folder Navigator**. The primary, active workspace  icon indicates that the server and workspace is active and connected to the primary Windchill server.

When you want to switch and work in another workspace, you activate that workspace and set it as primary either from the **Tools** menu or the **Folder Navigator**.

If you need to create a new workspace, access the **Workspaces** menu from the **Server Manager** in Windchill Workgroup Manager. This menu allows you to create a new workspace, edit its name, or delete it.

---

To populate your workspace, use the **Save** or **Save As** commands from Creo Illustrate after you have created illustrations. This puts the dynamic document into the primary, active workspace, and saves it to the workspace local directory.

**Note**

*From the **File** menu in Windchill, the **Save As** command saves file content in **HTML** format to a local drive.*

**Note**

*Modified parts are overwritten automatically in the workspace local directory when you open a corresponding assembly in any other location. If you open a part outside the workspace local directory and if any of its dependents exist in the workspace local directory, the following message displays:*

*“The local workspace already contains file(s) with the same name that will be over-written by this operation. Press Yes to backup (\*.wbk) these files in the local workspace.”*

*If you select **Yes**, then backup files are created in the workspace local directory, and the original files are overwritten. If you select **No**, then the original files are overwritten and no backups are created.*

You can also remove a worksheet from a workspace through the **File ► Remove** action.

# 4

## Working with Windchill PDM Actions

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This section contains the Windchill Product Data Management (PDM) actions that are initiated from Windchill when Creo Illustrate is:

- Installed on the same computer as the Windchill Workgroup Manager client, is registered to that client, and connected to the Windchill server.
- or,
- Installed on the same computer as the Windchill Workgroup Manager client, is registered to that client, connected to the Windchill server, and when Service Information Manager and Service Parts are installed.

---

## Accessing the Windchill Workgroup Manager Help Center

The Windchill Workgroup Manager Help Center includes help for the Windchill Workgroup Manager for Creo Illustrate.

You can access the Windchill Workgroup Manager Help Center as follows:

<b>From</b>	<b>Do the Following</b>
Creo Illustrate	In the Creo Illustrate <b>Windchill</b> ribbon, click <b>Connect to Windchill</b> . After Creo Illustrate connects to Windchill, click <b>File ► Help ► Windchill Workgroup Manager Help</b> .
Windows and mixed user interface	<b>Windchill ► Windchill Workgroup Manager Help</b> or, <b>Help ► Windchill Workgroup Manager Help</b>
Windows classic user interface	<b>Windchill WGM ► HELP</b>

The Windchill Workgroup Manager Help Center provides help for the Windchill Workgroup Manager user interface, the Windchill Workgroup Manager help, workspace, and all Windchill Workgroup Manager-integrated applications (including Creo Illustrate).

The Windchill Workgroup Manager Help Center also contains the complete *Windchill Workgroup Manager for Creo Illustrate Administrator's and User's Guide*. It covers:

- Installing, configuring, and getting started with Windchill Workgroup Manager for Creo Illustrate.
- Working with Windchill PDM Actions in Creo Illustrate.
- Setting Windchill Workgroup Manager client, server, and Creo Illustrate preferences.
- Performing Windchill Workgroup Manager for Creo Illustrate administrative tasks.

You can also download the *Windchill Workgroup Manager for Creo Illustrate Administrator's and User's Guide* from the Windchill Support Center page on the PTC WEB site. (Click the Reference Documents link and search for Windchill Workgroup Manager.)

---

## Windchill PDM Actions

Before working with Windchill PDM Actions, ensure that the following requirements have been completed:

1. The Windchill Workgroup Manager client for Creo Illustrate must be installed and configured on your machine in order to access and perform Windchill PDM Actions from within Creo Illustrate and from the Windchill Workgroup Manager. The PTC Virtual File System (VFS) is automatically installed during the installation of Windchill Workgroup Manager client.

### **Note**

*If you have not installed and configured the Windchill Workgroup Manager client for Creo Illustrate, refer to the installation chapter in the Windchill Workgroup Manager for Creo Illustrate Administrator's and User's Guide for instructions on how to install the Windchill Workgroup Manager Client.*

2. PTC Virtual File System must be configured on your machine in order to access PTC Places. Ensure that you have set up and configured the PTC Virtual File System before working with Windchill PDM actions.

### **Note**

*Refer to the Installation chapter in the Windchill Workgroup Manager for Creo Illustrate Administrator's and User's Guide for instructions on how to install and configure the PTC Virtual File System.*

3. Ensure that PTC Places is your default directory when you are connected to Windchill and have an active Creo Illustrate session.
  - a. Confirm that PTC Places is your default directory by clicking the **Look in** location from **File ► Open** in Creo Illustrate. The PTC Places directory should appear one level up from the “workspace” default directory.

### **Note**

*PTC Places are locations within the Windchill server.*

- b. If your default view in the **Look in** directory points elsewhere, refer to the Installation chapter in the *Windchill Workgroup Manager for Creo Illustrate Administrator's and User's Guide* for help in making PTC Places the default directory.

---

### **Note**

*If PTC Places is not your default directory when you are connected to Windchill and have an active Creo Illustrate session, you will not be able to perform Windchill PDM Actions. Refer to the Installation chapter in the Windchill Workgroup Manager for Creo Illustrate Administrator's and User's Guide for instructions on how to set PTC Places to be your default directory.*

4. If you want to add, edit, and delete illustrations in a Parts List, or add, edit, and delete illustrations in the Service Parts active workspace or commonspace, ensure that Service Information Manager and Service Parts are installed.

### **Note**

*Refer to the Windchill Help Center under **Specialized User Capabilities** ▶ **Service Information Manager** for more information.*

## **Creo Illustrate Dynamic Document Information Page**

Creo Illustrate dynamic document objects contain Creo Illustrate graphics stored on your PTC Server.

The Creo Illustrate information page is a collection point for information related to Creo Illustrate dynamic document objects. The information page of a Creo Illustrate dynamic document object is accessed by clicking the information icon.

From the information page, you can perform actions on a Creo Illustrate dynamic document object, as well as view the object attributes and any related information.

For an overview of information page functionality, see “About Information Pages” and “Customizing Your Information Page Tabs” in the Windchill Help Center.

### **Actions**

Information pages contain a list of applicable actions in a menu just below the object identification line. See “Common Actions” for a list of common actions available in Windchill. The availability of these actions depends on the status of the Creo Illustrate dynamic document object, as well as your access to it. For more information about status indicators, see “Status Indicators” in the Windchill Help Center.

---

## Tabs

The following tabs are viewable on the Creo Illustrate dynamic document information page. You can customize the tabs that appear on each object information page to display the information most useful to you. For more information, see “Customizing Your Information Page Tabs” in the Windchill Help Center.

### Note

*Administrators may have modified the tabs that are viewable to you. Additional tabs may appear depending on the solutions deployed at your site.*

Tab	Description
<b>Details</b>	The <b>Details</b> tab lists the primary attributes associated with the object, along with a <b>Where Used</b> information table, showing where the object is used.  If there is a default representation, a visualization of the graphic will display. See “About Visualization” in the Windchill Help Center.
<b>Related Objects</b>	The <b>Related Object</b> tab allows you to view representations of the graphic.  If this object was created from a PVZ graphic dynamic document, than a Source and Image table will show its source PVZ object as well.
<b>History</b>	The <b>History</b> tab contains a version history and time line of the object.

## Creating a Workspace

Use the **Windchill** ribbon to access the **Server Manager** window to create a new workspace on the primary server. The new workspace automatically becomes your active workspace.

### Note

*The **Windchill** ribbon is accessible when Creo Illustrate is connected to Windchill Workgroup Manager.*

---

To create a new workspace:

1. From Creo Illustrate, select **Windchill Workgoup Manager**  icon from the **Windchill** ribbon. Select the **Tools ► Server Management** utility to open the **Server Management** window. The **Server Management** window displays all the registered servers.
2. From the **Servers** list, select the server on which you want to create the new workspace.
3. Select **Workspace ► New**, or from the **Server Active Workspace** section, right-click any workspace and select **New**. The **Create New Workspace** window opens.
4. Enter a name for the new workspace and select a context from the **Context** list.
5. Select **OK**. The new workspace appears in the **Server Active Workspace** section and in the Folder navigator. It automatically is set to primary and becomes the active workspace.

## Open In Creo Illustrate

You can open a Creo Illustrate file from the following Windchill locations into your active Creo Illustrate session:

- Active workspace on the primary server.
- Workspace on the primary server.
- Windchill Workgroup Manager client.
- Information System Manager with Service Parts (if installed)
  - Information Structure
  - Folders
  - Structure

- Workspaces

**Note**

*For more information about Service Information Manager and Service Parts, refer to [Accessing Service Information Manager and Service Parts on page 53](#) in the “Getting Started with Windchill Workgroup Manager” section.*

1. Select the illustration you want to edit and click  **Check Out** in the active workspace **Object List** table, or from the **Parts List Editor** ribbon.

**Note**

*Depending on your preference, you can check out the illustration from the active workspace or from within an active Creo Illustrate session.*

2. Select the illustration to open from either **File** ► **Open in** ►  **Creo Illustrate**, or from in the row of the illustration you want to open, click the Actions  **Open in Creo Illustrate** icon.
3. If you haven't checked out the illustration while in the active workspace, check out the illustration if you want to edit it in Creo Illustrate. To do so, select the Windchill ribbon check out  icon.
4. When finished viewing or editing the illustration, you can save it to the active workspace, or check in using **Checkin**  from within Creo Illustrate or from within the active workspace .

## Checking Out a File

This section describes the check out actions available from Windchill.

---

In order to modify an object, you must first check it out. **Check Out** accomplishes the following:

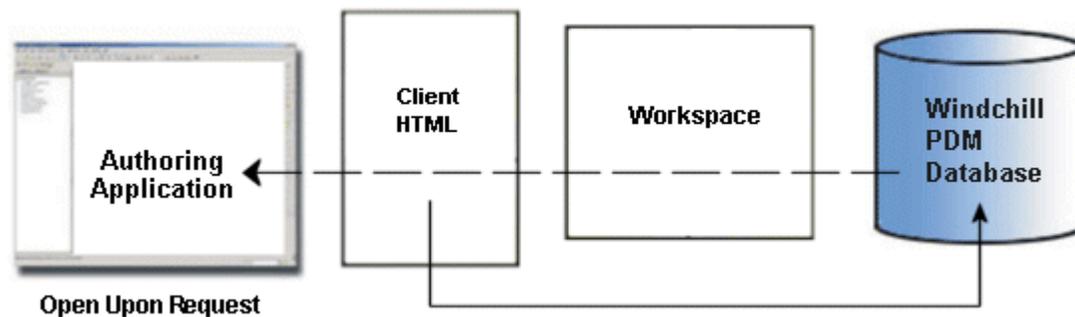
- Creates a copy of the object in the commonspace on the server and marks it as checked out. This signals other users that the object is being modified, and therefore, they cannot modify it.
- Creates a working copy of the object in your workspace that holds all your changes until you check the object back in.
- Reserves the object in the commonspace, preventing other users from checking in any changes to it until you release it. You release it by checking it back in or undoing the checkout.

### Note

*Windchill supports the checking out of non-latest iterations of dynamic documents, and the subsequent checking in of those documents to become the latest iteration. This allows you to revert to an earlier version of an illustration.*

*For more information on checking out non-latest iterations, see the Windchill Help topic, “Checking Out Objects from Windchill,” which is available from the Windchill **Check Out** page. You can also access this topic from the Help Center.*

This example illustrates the checkout workflow for an object that is open upon request from Windchill.



### Note

*During a check out operation that uses the option of adding the object content to the workspace, data is transferred from the Windchill database to the server-side workspace. Optionally, if you choose to open the object from the workspace, the data is then transferred from the workspace to your authoring application.*

---

You can also access **Check Out** from three areas in Windchill. When **Check Out** is performed from the workspace in Windchill, a status message displays in Windchill.

- Select  **Check Out** from the Windchill active workspace when you right-click anywhere in a row of the selected file you want to checkout.
- Select the **Check Out**  on the Windchill active workspace **Object List** table ribbon.
- Select the **File** ►  **Check Out** menu in Windchill.
- From Creo Illustrate, select **Check Out**  in the Windchill ribbon.

## Undoing a Check Out

**Undo Check Out** reverses a checkout, and allows you to return to the model that you originally checked in. **Undo Checkout** acts on the object open in the active model and reverses any changes that were made to this object after it was checked out.

The Windchill preference **Undo Checkout Overwrite Local Content**, available from **Preference Management**, specifies whether the model content is overwritten in cache by default when using the **Undo Checkout** action. The preference default is set to **False**, meaning that model content is not overwritten in cache by default.

- Select  **Undo Check Out** from the Windchill active workspace when you right-click anywhere in a row of the selected file you want to undo the checkout.
- Select the **Undo Check Out**  icon on the active workspace **Object List** table ribbon.
- Select the **File** ►  **Undo Check Out** menu in Windchill.
- From Creo Illustrate, select **Undo Check Out**  in the Windchill ribbon.

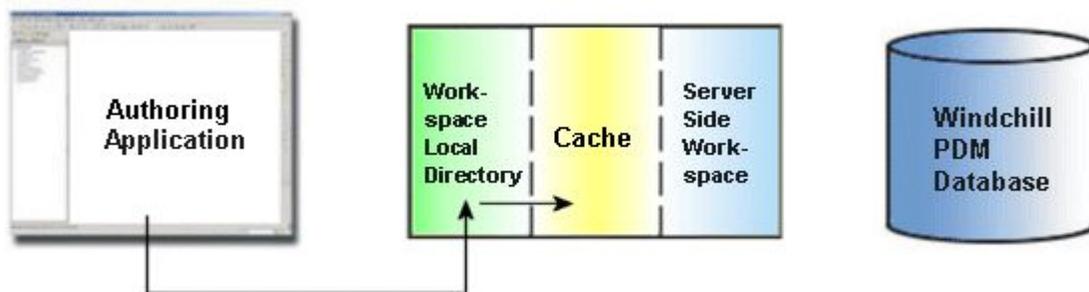
## Checking In a File

This section describes the **Check In** action available from Windchill.

**Check In** is not an available action if there is no primary server and primary workspace. You must register a server from **Tools** ► **Server Manager** in Windchill Workgroup Manager, and activate a workspace.

---

When you are finished making changes to a file, and you are ready to publish the changes to the shared area on the server, you must check in your objects. The following graphic illustrates the information flow during check in.



### Note

*Check in transfers data from the server-side workspace to the commonspace. If you check in an object directly from the authoring application, the data is transferred from Creo Illustrate to the server-side workspace, and then to the Windchill server.*

**Check In** accomplishes the following:

- Copies the working copy of your object from your workspace to the shared area of the server.
- Creates a new iteration of the checked-out object in the shared area.
- If this object was checked out by you, releases the lock on the object in the shared area, so that others can access it.

- **Check In** is initiated from Windchill when you click **Check In**  on the workspace **Object List** table ribbon.
- Select the **File** ►  **Check In Custom** menu in Windchill.
- From Creo Illustrate, select **Check In** , or **Custom Check In**  in the Windchill ribbon.

When you initiate check in, you can choose from two options in Windchill:

- **Check In** 

---

When you select **Check In**, your object is directly copied to the commonspace. The process follows the defaults you or your administrator has set up, and checks in what has changed in modified objects.

- **Custom Check In** 

When you select **Custom Check In**, you check in objects with changed settings using the Windchill HTML user interface. When you select **Custom**, you can specify certain options that are applied to the object you are checking in. **Custom** takes you through two pages where you can specify the following options.

For example, use  **Custom Check In** when you want to associate an illustration with an **Illustration Holder** during first time check in.

1. **Collect Objects**

The **Collect Objects** page allows you to select the objects you want to check in. Select the check boxes next to the objects you don't want to be visible to others on the Windchill server. Go to **Edit** menu and select **Set For Check in** to deselect the selected items for check in. Select **Next**.

On the **File** menu, select **Check Out Now**. The object can be checked out using the **Check in** option.

On the **Edit** menu you can select **Set Location**, **Keep Check out**, or **Set For Check in**.

**Note**

*During check in you have the option of picking a folder location, or browsing to a product ASPS location. You can also provide an optional check in comment.*

2. **Set Options**

The **Set Options** page allows you to specify certain options for the objects you are checking in. Select the check box next to the options you want to set. You can choose to do the following.

- Create a baseline.
- Undo check out of unmodified objects.
- Remove objects from the workspace.
- Auto-resolve incomplete objects, either:
  - Update with objects on the server, then ignore.
  - Always ignore.

3. Press **Finish** to check in the file.

---

## Renaming a File

This section describes the **Rename** action available only on the **File** menu in Windchill. It is not accessible from Creo Illustrate.

To rename a file, right-click anywhere on the row of the illustration you want to rename and click **Rename**.

Or,

Click the **File** ► **Rename** menu in Windchill.

**Rename** changes a name of an object by changing the values of the Number, Name, and Model Name attributes, if you have Modify Identity access permission. You can also rename objects that are in another's workspace; the system notifies the other user that they need to synchronize their workspace with updated information on the server.

### Note

*Only objects that have never been checked in can be renamed from the workspace. Once objects have been checked into the commonspace, they can no longer be renamed from the workspace and **Rename** must be accessed from the commonspace.*

### Note

*The only supported Creo Illustrate file type for interaction with Windchill Workgroup Manager is c3di.*

### Note

*The only associations supported in Creo Illustrate integration with Windchill Workgroup Manager are Content and Contribution Content associations.*

## Creating a New Revision

To create a new revision for an object, right-click anywhere on the row of the illustration you want to create a new revision and click  **Revise**. The **Revise** window opens to allow you to create a new revision for an object. To accomplish this you assign the next revision level available in a revision scheme. The revision scheme represents a sequence of characters identifying subsequent versions of an object. Creating a new revision of an object results in the object, and all objects you choose to associate with it, advancing to the set revision level. When you revise an object, the latest version of that object is used as the content for the new revision.

The  **Revise** action is only available from Windchill. It is not accessible from Creo Illustrate.

---

To create a new revision for an object, right-click anywhere on the row of the illustration you want to create a new revision and click  **Revise**.

Or,

Select the **File** ▶  **Revise** menu in Windchill.

## Uploading a File

This section describes the **Upload** action which is only available from the user interface in Windchill Workgroup Manager. It is not accessible from Creo Illustrate.

**Upload** stores working copies of the files you are editing in your registered Creo Illustrate application to your personal folder location on the Windchill server, without making your changes visible to other users. Upload is only valid for new or modified files that are checked out to this workspace by you.

**Upload local contents to server**  is available on the Windchill ribbon, as shown in the next graphic.



To upload an illustration, click **Upload local contents to server**  on the row of the illustration you want to upload.

Or,

Click the **File** ▶  **Upload** menu in Windchill.

## Synchronizing a File

This section describes the **Synchronize** action, which is only available from the HTML user interface in Windchill Workgroup Manager. It is not accessible from Creo Illustrate.

**Synchronize** ensures that the contents of the cache are in sync with the contents of the server-side workspace. The cache can become out of sync when you are working on workstation "A" for a day, then you upload all data in the workspace to work on workstation "B" the following day. If you activate that same workspace on Workstation "A" at a later time, the cache is out-of-date. This data needs to be synchronized so that the object contains updated data. Data can also become out of sync when the connection to the server is lost, and you keep working on cached

---

workspace data. In this case, the data in the cache is more up-to-date than what is in the server-side workspace. Click the **Tools ▶ Synchronize** menu in Windchill to resolve out of sync data.

## Working with Information System Manager and Service Parts

You can work in Information System Manager and Service Parts in Windchill when Creo Illustrate is integrated with Windchill Workgroup Manager, is registered and connected to the Windchill Server, and when Service Information Manager and Service Parts are installed.

### Note

*For more information about Service Information Manager and Service Parts, refer to [Accessing Service Information Manager and Service Parts on page](#) in the “Getting Started with Windchill Workgroup Manager” section.*

*Detailed information about Service Information Manager and Service Parts can be found in the Windchill Help Center under:*

- **Specialized User Capabilities ▶ Service Information Management ▶ Service Information Management ▶ Windchill Service Parts ▶ Working with Windchill Service Parts**  
*and from,*
- **Specialized User Capabilities ▶ Service Information Management ▶ Service Information Management ▶ Windchill Service Parts ▶ Working with Windchill Service Parts**

## Managing Illustrations in a Parts List

A typical use scenario is that a Parts List author who has created a Parts List for a Service Manual or a Parts Catalog wishes to add, edit and delete illustrations to a Parts List.

Use the **Parts List Editor** to manage Creo Illustrate illustrations in Windchill Service Information Manager and Service Parts.

### Adding Illustrations

Once you have created a Parts List you can add Creo Illustrate illustrations to the Parts List using the **Parts List Editor**.

---

Add illustrations to a parts list using the following steps:

1. From Windchill, select **Information Structure**, or **Folders** from the **Navigator** pane.
2. In the **Information Structure** or **Folders** window you can either:
  - Insert a New Parts List by selecting **Insert New ► Insert New Parts List**. Enter the product name for the new Parts List, select the folder, and click **OK**.
  - Select an existing Parts List.
3. Right-click the Parts List product name you want to add one or more illustrations and select **Open In ► Open Content in Parts List Editor**.
4. In the **Parts List Editor Structure** tab, click **+ Add Illustration** in the **Parts List Editor** ribbon.
5. The **Add Illustration** window opens to allow you to search for and associate illustrations to the parts list so they can be associated with individual parts. After you have entered the search criteria, click **Search**.

Field	Description
<b>Search On</b>	The type of documents that you can search on. In this case, only <b>EPM Document</b> is applicable.
<b>Number</b>	The number assigned to the illustration type.
<b>Name</b>	The name of the illustration type.

For example, select **EPM Document** and the wildcard **\*.c3di**, and click **Search** to search for a Creo Illustrate illustration.

### **Note**

*To select a new search criteria, remove the existing search criteria by clicking **Clear**. Reenter the search criteria to view the new search results in the **New Illustration** window table.*

6. The search results are displayed in the **New Illustration** window table.
7. Select the illustration to add to the Parts List and click **OK**. The system saves the illustration to the Parts List and when finished, opens the **Parts List Structure** tab.
8. To add additional illustrations to the Parts List, repeat steps 3 — 7.

---

## Editing Illustrations

Edit illustrations to a parts list using the following steps:

1. From Windchill, select **Information Structure**, or **Folders** from the **Navigator** pane.
2. In the **Information Structure** or **Folders** window, right-click the Parts List product name you want to edit one or more illustrations and select **Open In ► Open Content in Parts List Editor**.
3. In the **Parts List Editor Structure** tab, select the illustration you want to edit and click  **Check Out** in the **Parts List Editor** ribbon.

### Note

*Depending on your preference, you can check out the illustration from the active workspace or from within an active Creo Illustrate session.*

4. Right-click the illustration and click  **Open in Creo Illustrate** to open the illustration in Creo Illustrate.
5. If you haven't checked out the illustration while in the active workspace, check out the illustration if you want to edit it in Creo Illustrate. To do so, select **Check Out**  in the Windchill ribbon.
6. When finished editing the illustration, you can save it to the active workspace, or check in using the check in options in Creo Illustrate or from the Windchill active workspace check in options.

## Deleting Illustrations

Delete illustrations to a parts list using the following steps:

1. From Windchill, select **Information Structure**, or **Folders** from the **Navigator** pane.
2. In the **Information Structure** or **Folders** window, right-click the **Parts List** product name you want to delete one or more illustrations and select **Open In ► Open Content in Parts List Editor**.
3. In the **Parts List Editor Structure** tab, select one or more illustrations you want to delete from the product name content holder and click  **Remove** in the **Parts List Editor** ribbon.

### Note

*If an illustration is checked out, it cannot be deleted until it is checked back in.*

# 5

## Advanced Techniques

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This chapter discusses advanced usage of Windchill Workgroup Manager. It covers basic information about:

- Thumbnail Generation and Thumbnail Worker.
- Viewing Relationships.
- Conflict Management.
- Cache Management.

---

## Thumbnail Generation and Thumbnail Worker

You are able to see thumbnails of Creo Illustrate illustrations in Windchill Workgroup Manager.

The documentation concerning thumbnail generation and thumbnail worker can be found in the *ProductView™ Adapters Installation and Configuration Guide*.

## Viewing Relationships

You are able to see the related drawings on a Windchill part's information page by selecting the **Related objects** ► **Related CAD/Dynamic Documents** report.

Similarly, you are able to see the Windchill parts related to an illustration's details page by selecting the **Related Parts** report.

The **Related Objects** reports for dynamic documents do not have a configuration control; by default you are shown the latest configuration.

## Conflict Management

The term conflict management refers to addressing or resolving the restrictions that your PDM system can place on your retrieval, modification, and storage activities in order to maintain the integrity of the database. Typically, with Windchill Workgroup Manager, you encounter two types of conflicts, as follows.

- Conflict upon an attempt to modify an object.
- Conflict upon attempting to perform a PDM transaction (referred to as an event) such as check in, check out, and so forth.

### Conflict upon Attempt to Modify

If you attempt to modify a workspace object that is not checked out (and also not locked), then upon attempting a checkin, the system presents the **Conflicts** window, informing you that the object is read-only because it is not checked out. You can choose to resolve the conflict by using one of the following options.

- **Check out now** – Perform an immediate check out on the object (within a version), enabling you to modify the object and later check in the modifications. When checked in, the modifications are shown as an iteration of the current object version.
- **Revise and Check out** – Perform an immediate check out on the object (for a new version), enabling you to modify the object and later check in the

---

modifications. When checked in, the modifications are shown as a new version of the object.

- **Continue** – Continue with modifications without checking out (modifications will not be eligible for checking in, and may be lost.)

If your attempt at modification involves several checked-in objects, you can specify different resolutions of the conflict for each object, or you can use the **Set All** menu to apply the same resolution to all.

## Conflicts Involving a PDM Event

PDM events may cause conflicts with the database that need to be resolved. Event Management provides a way for you to check and act on log messages generated in Creo Illustrate or Windchill sessions. It can be accessed in the following ways.

- From the Windchill ribbon in Creo Illustrate, select **Event Management** .

### Note

*From Creo Illustrate, only information about transactions with the primary server is available. In the Creo Illustrate message area, a graphical status indicator provides status information.*

- From a workspace, select **Event Management** from the workspace actions drop-down list.
- Select the **Event Management** link in the banner at the top of a Windchill page.
- From Windchill Workgroup Manager, select the **Console Status** icon in the status bar.
- In Windchill Workgroup Manager, select **Tools** ▶ **Event Management**.
- In Windchill, select **Home** ▶ **Quick Links** ▶ **Event Management**.

On the **Event Manager** page you can see the PDM events for the named server listed chronologically in the **Events** table. You can access detailed information about a particular event by clicking in the event's **Actions** column, and access the **Conflict Management** page by clicking icons displayed in the **Actions** column for viewing or resolving conflicts.

The event type icons used by **Event Management** are described in the following table.

### Event Management Icons

Icon	Description
	Overridable type conflict
	Non-overridable type conflict (Failed)

---

## Event Management Icons (continued)

Icon	Description
	Warning
	In Progress
	Pending
	Retried
	On Hold

The icons for actions that may be available in the **Actions** columns are described in the following table.

### Actions Column Icons

Icon	Description
	View event information
	View Warnings/Conflicts
	Resolve Conflicts

The **Event Information** page is accessed from the **Event Manager** by clicking the information icon  in the **Actions** column for the event in the **Event List**, or by selecting the hyperlink in the event name. The **Event Information** page lists the objects that have been processed for a given event. At the top of the page is a hyperlink that returns you to the **Event List**. Below the hyperlink is the event title. Next to the event title are icons for the Delete action and, if warnings or conflicts occurred, one of the following actions, as applicable to the type of event:

- View Warning  – If the event type is a warning
- View Conflicts  – If the event type is a non-overridable conflict
- Resolve Conflicts  – If the event type is an overridable conflict

The area immediately below the event title lists the event attributes, as follows:

- Event Name – The name of the event.
- Workspace – The workspace from which the event originated.
- Status – The status of the event, if completed, or a progress bar indicating how close the event is to completion.
- Start Date – The date and time the event began.

- End Date – The date and time the event was completed.
- Elapsed Time – The total time required to complete the event (if completed).

Below the attributes area is a table listing the **Affected Objects** in the event.

The **Conflict Management** page assists you in viewing and resolving conflicts that arise from database events. It is accessed from Event Management or the **Event Information** page by clicking the view conflicts icon  or the resolve conflicts icon .

The next graphic displays the **Conflict Management** page.

Objects that experienced one or more conflicts during an event are listed in the **Conflicts** table, which by default has the columns described in the following table.

### Conflicts Table Columns

Column	Description
(Selection)	Select check boxes to select or deselect rows. Clicking the header check box selects/deselects all rows.
(Conflict type)	Displays an icon identifying the type of conflict.
<b>Name</b>	Displays the name of the affected object, if applicable.
<b>Retry Option</b>	Displays the default or currently applied user-selected retry option for the conflict, if one is available.
<b>Description</b>	Displays an explanation of the conflict.  <b>Note</b> <i>If the explanation exceeds character limits for the column, an ellipsis (...) followed by a link "view full text below..." appears at the end of the truncated description. Clicking the link displays the full text of the description in the <b>Description</b> field below the <b>Conflicts</b> table</i>
<b>Number</b>	Displays the number of the affected object, if applicable.
<b>File Name</b>	Displays the file name of the affected object, if applicable.

### Note

*You can customize your **Current View** of the **Conflicts** table; however, setting (row) filters is not allowed for this table. For more information, see the help available from the **Customize View List** page, which appears when you select **Customize** from the current view drop-down list.*

In the ribbon of the **Conflicts** table is a drop-down menu that lists any **Retry Options** common to all of the selected conflicts.

---

To resolve an overridable conflict, perform the following procedure.

1. Select one or more rows containing an overridable conflict.
2. Select an option from the **Retry Options** drop-down list in the table ribbon.
3. Select **Set**. The **Conflicts** table refreshes to display the new value in the Retry Option column for the selected row(s).
4. Select **Retry**. The action is retried and a new entry is created in the **Event List**. The original event's status changes to **Retried**. Further access to the original event in **Conflict Management** is read-only.

## Managing Cache

Windchill Workgroup Manager cache is a local repository for third-party authoring applications contained in a specific workspace, and is used only when uploading or downloading files to and from the server. Using cache improves performance because Windchill Workgroup Manager only downloads files if the content on the server is more recent than the content in the cache; and only uploads files when you direct it to do so.

By default, when you register Windchill servers and authoring applications, a `.vfs` directory is created for cache. It is located under your user profile on your local computer. For every server location listed in the `.vfs` directory, there are subdirectories for each workspace.

### Note

*Working directly in cache is discouraged so that data is not accidentally corrupted. To help avoid directly working in cache, a component on the client, called the Workspace Local Directory, is available. It is recommended you use this, rather than browsing to or saving from Arbortext IsoDraw to the cache directly. Sample default locations for the Workspace Local Directory are provided below.*

The cache location is defined by the environment variable `'PTC_WF_ROOT'`, which is the client connector cache. This is where all Windchill Workgroup Manager related client side information is stored. If the environment variable is not explicitly defined, the default location will be the user's home directory.

By default, cache is not shared between Windchill Workgroup Manager and Creo.

If you are planning to run the Windchill Workgroup Manager and Creo on the same system and also want to have the cache residing in a non-default location, you must explicitly set the cache (`PTC_WF_ROOT`) for both these applications to

---

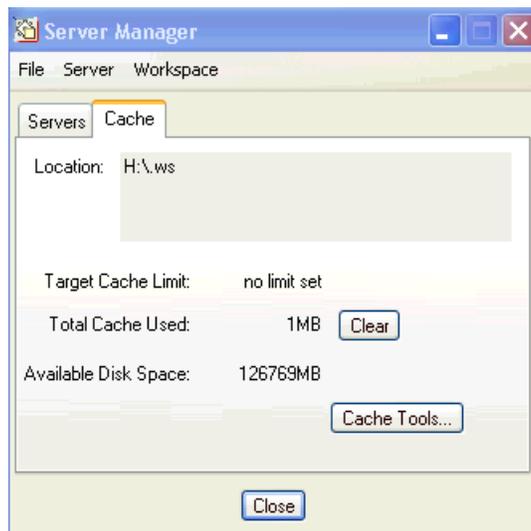
different locations. For example, one method of achieving this is to create a startup batch file for each application that defines the 'PTC\_WF\_ROOT' environment variable (cache location) and then start the program.

The default locations for the Windchill Workgroup Manager, Creo and Creo Illustrate cache, and Workspace Local Directory locations, are as follows:

- Cache location for Creo - %APPDATA%\PTC\Creo\wf
- Cache location for Windchill Workgroup Manager - %APPDATA%\PTC\Creo\wwgm
- Workspace Local Directory location for Creo - %USERPROFILE%\wf
- Workspace Local Directory location for Creo Illustrate - %USERPROFILE%\wwgm

Although it is not recommended you directly access the .vfs directory; there are certain PDM actions that require you to select from it. For example, **Open** queries the Creo Illustrate subdirectory in the .vfs directory so you can select the document you want to open. If desired, the location of the .vfs directory can be changed by setting the environment PTC\_VFS\_ROOT to the required location.

Cache is managed from the **Cache** tab on the **Server Management** window (**Tools > Server Management**). This window describes your workspace local directory/managing directory location, cache space used (in MB), and available disk space. There is also an option to clear the cache. The following graphic shows the **Cache** tab window.

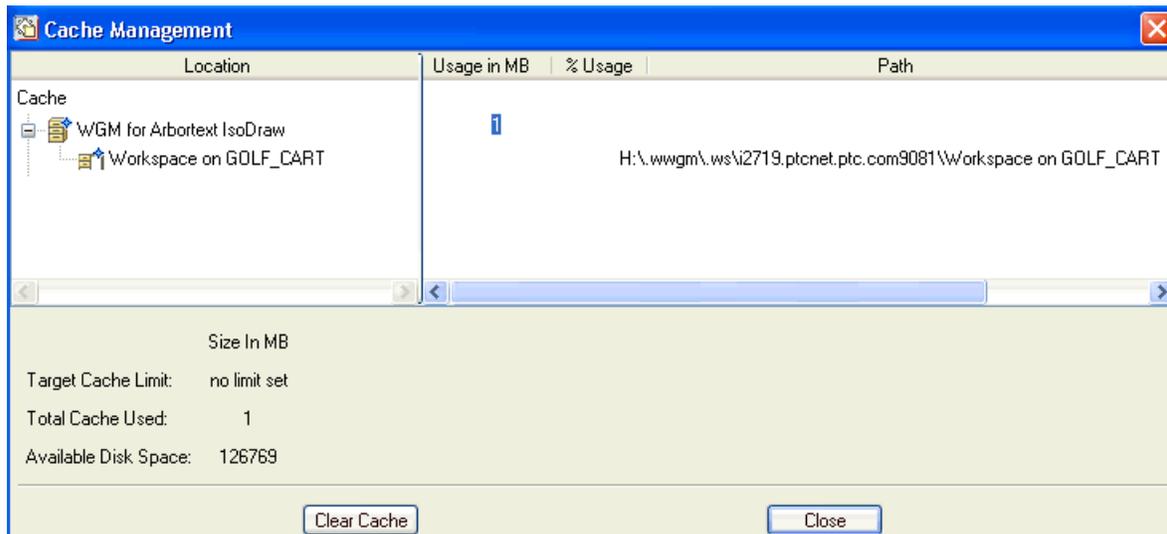


This information is also displayed on the **Cache Management** window.

Select **Cache Tools** on the **Cache** tab to access the **Cache Management** window. The **Cache Management** window provides the following:

- Workspace local directory location of primary server's active workspace
- Cache usage
- Path to the `.ws` directory
- Cache limit. The Target Cache Limit can be set in the `wgmclient.ini` file entry `cache.size`, or by setting the environment variable `dm_cache_limit`.
- Amount of utilized disk space
- The **Clear Cache** button clears all cache in the inactive workspace.

The next graphic displays the **Cache Management** window.



# 6

## Windchill Workgroup Manager Client and Server Preferences

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This chapter is intended for all users and contains the Windchill Workgroup Manager client and server preferences, such as preference organization and access, and modifying preferences.

Preferences are set to enforce a company's business policies in a consistent manner, and to ensure that the user environment meets personal working styles. Administrators set preferences to customize Windchill Workgroup Manager functions, user interface displays, and to provide default behavior. Users can set client preferences to meet their personal working styles.

There are two areas in which to set preferences:

- The **Preference Management** utility on the **Utilities** tab on the Windchill server:
  - Allows an administrator to set server preferences.
  - Allows an administrator or a user to set client preferences.
- `wgmclient.ini` file on the client:
  - Allows users to set client preferences.

---

## Preference Organization

The preference system is organized using a top-down design where an administrator first sets up the commonly used server and client preferences, followed by the user, who typically specifies client behavior.

The following list describes the preferences hierarchy:

1. Windchill Organization; the most general level. The preferences at this level are set by an administrator on the server in the **Preference Management** utility. These preferences are visible to administrators and users.
2. User; the second-most general level. The preferences at this level are client preferences set in the **Preference Management** utility, and in the `wgmclient.ini` file. These are visible to the user and the administrator, and are client-only preferences. These client preferences can be customized from what is already set by the administrator. If the same client preference is set in the `wgmclient.ini` file, it overrides the client preference settings in the **Preference Management** utility window.
3. `wgmclient.ini` file; the most specific preference level. The preferences at this level are visible to the user, and override client preferences set on the server in the **Preference Management** utility. These preferences are set to customize the client to meet individual user working styles.

### Note

*Administrators can lock a preference in the **Preference Management** utility so that it cannot be changed by anyone, including the settings in the `wgmclient.ini` file. Administrators usually lock certain preferences to ensure certain business policies cannot be changed.*

## Accessing Client Preferences on the Server

The **Manage Preferences** window is accessed by a user from Windchill, using the following path:

**Windchill ► Home ► Utilities ► Preferences.**

Client preference selections are available under the **Windchill Workgroup Manager** header in the **Preference Management**.

## Editing Preferences

Every preference can be changed unless it is locked by the administrator. Use the following procedure to change a preference from its current state.

1. If you are an administrator, access from Windchill, **Site ► Utilities ► Preference Management**. If you are a general user, access from Windchill, **Home ► Utilities ► Preference Management**.

The **Preference Management** utility window opens. This window lists all preferences, and this is where you set server and client preferences.

2. Preferences display by category. Most categories have several selections.

Preference Management Standard

Find in tree

Name ↑	Value	Description
⊕ Add to Baseline		Add to Baseline operation preferences
⊕ Arbortext		Arbortext Preferences
⊖ Attachments		
Default File Path		Specifies a default directory for uploading a
Document Name	Yes	Determines whether or not the Name field s
File Download Behavior	Always ask whether t...	Determines whether downloaded files are o
File Download Mechanism	Use basic browser fu...	Determines how to download files from Wir
File Upload Mechanism	Use Java applet func...	Determines how to upload files to Windchil
Force Desktop Integration Installation	Yes	Determines whether to Force Desktop Integ
Java Console Debugging Information Dis...	Java Console debug...	Determines whether Java Console will displa
Keep document open after check-in	No	Select Yes if you want to keep your documen
⊕ Optional Attributes		Optional attributes for creating and display
Primary Content Enforcement	Primary content is al...	Sets the default behavior for primary conter
Send local file to Recycle Bin	No	Determines whether the Send local file to R
Upload Primary File on Checkin	Yes	Determines whether to upload primary file c
⊕ Attribute Handling		

For example, The **Workgroup Manager Client** category lists general and specific preferences for authoring applications integrated with Windchill Workgroup Manager.

Preference Management Standard

Name ↑	Value	Description
Workflow		Workflow preferences
Workgroup Manager Client		Client side Windchill Workgroup Manager
Add Active Workspace To CATIA DLNames ...	Yes	Determines if the Active Workspace should
Allow Continuing Save to Workspace or C...	No	When design table is missing, this preferen
Assembly-level CGR parameter		Specifies which parameter on a part drives a
Attach Differences Report upon Check In	No	Controls the default behavior whether or n
AutoCAD Standard Drawing Definition	attribute_only	Controls how the system identifies which d
CADD5 5 Assembly Structure	File Structure	CAD Document structure saved from CADD5
CATIA V5 Template Part Number		Set the value of this preference with the str
Check In Content Options	Models Only	Specifies how the model is saved on the au
Default Drawing Frame Location	/	Location in the commonspace where Creo E
Define Library Used by Application		Indicates a library in which library compone
Design In Context		Define the Configuration Context to Desigr
Drawing Definition	attribute	Controls how the system determines if the r
Drawing File Name		This preference works when the preference

- Individually, select the preferences you want to set by selecting the preference and right-clicking to select the **Set Preference**  icon.



- In the **Set Preference** window, enter or select the appropriate values for this preference and select **OK**. The preference is set.

After a server-side preference is changed, in order to see them in the model, it must be synchronized through **Tools > Synchronize**.

All client preferences have a **Client:** drop-down list that displays the preference options that can be set.

---

## Adding a User-defined Soft Type for Creo Illustrate

You can set a default preference for a user-defined soft type in the Windchill Workgroup Manager for Creo Illustrate, using the Site Admin login.

1. From Windchill, navigate to **Site ► Preference Management ► Windchill Workgroup Manager ► Client ► Default soft type**. Locate and select the Creo Illustrate preference in the list of applications.
2. For the Creo Illustrate preference, the default soft type is Dynamic Document. Click on the **Edit** Icon to edit the preference.
3. See the default value of the preference, “\${internet\_domain\_name}.DynamicDocument”. You can substitute a different user-defined soft type if required, such as, “\${internet\_domain\_name}.MachineDesign”, where “\${internet\_domain\_name}.MachineDesign is the Logical Identifier of user-defined soft type.

## Setting the Embedded Browser Preference

Windchill Workgroup Manager allows you to set a preference for which Internet browser you wish to use as the embedded browser for your Windchill Workgroup Manager installation. This preference is set in the `wgmclient.ini` file. Supported browsers are Internet Explorer and Mozilla. The preference is set as follows:

- The syntax for the preference is `windows.browser.type=browser_name`.
- To set the default embedded browser type to Internet Explorer, the syntax is `windows.browser.type=IE_In_Process`.
- To set the default embedded browser type to Mozilla, the syntax is `windows.browser.type=Mozilla_based_browser`.
- A sample entry in the `wgmclient.ini` file looks like this:

```
=====  
# Sets the WWGM embedded browser type.  
windows.browser.type=Mozilla_based_browser  
=====
```

- Out-of-the-box, the default embedded browser in the `wgmclient.ini` file is set to Internet Explorer.

### **Note**

*Windchill Workgroup Manager currently supports only the 32-bit Mozilla browser.*

---

See [Editing the wgmclient.ini File on page 82](#) for more information on how to edit the `wgmclient.ini` file.

## Editing the wgmclient.ini File

The default `wgmclient.ini` file is located in the Windchill Workgroup Manager Home directory. You edit this file to make changes to client preferences. Changes made in this file override client preferences set on the server. Use the following procedure to edit the `wgmclient.ini` file.

1. Open `wgmclient.ini` in a standard text editor.
2. Change any preference as follows:  
<preference name = preference value>
3. Remove the `#` prefix in the `wgmclient.ini` file to uncomment any preference that should override the settings on the server (the preference name = preference value pair).
4. Save the file.
5. Restart Windchill Workgroup Manager. The new client preference is available.

To add a section to Creo Illustrate, perform the following procedure.

1. Open `wgmclient.ini` in a standard text editor.
2. Add an explicit section [CreoIllustrate/Client Name] and define preferences specific to Creo Illustrate below this heading. <preference name = preference value>.
3. Save the file.
4. Restart Windchill Workgroup Manager. The new client preference specific to Creo Illustrate is available.

# 7

## Administration and Configuration

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This section is intended for Administrators. It covers the information necessary for setting up and configuring Windchill Workgroup Manager for Creo Illustrate. It may be useful to refer to the following Windchill documents for more information.

- *Windchill Customizer's Guide*
- *Windchill System Administrator's Guide*
- *Windchill Business Administrator's Guide*
- *Creo Illustrate User's Reference*

These guides are located at [PTC Reference Documents](#).

This chapter also covers other recommendations for performance tuning, setting up a desirable Windchill folder structure.

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# Configuring Business Practices

This section describes how an administrator configures Windchill and the CAD application to suit a company's business requirements.

## Configuring Naming and Numbering

You can specify how newly-created CAD or authoring documents (EPMDocuments) and Parts (WTParts) are named and numbered using a set of standard policies. It is also possible to customize the CAD or authoring document, like Creo Illustrate, naming service to address more advanced naming and numbering requirements.

### Policy-Managed Naming and Numbering

Windchill Workgroup Manager supports two policies to determine how newly created objects are named and numbered. These policies are the following:

- Auto-numbering
- Name-driven

These policies are only utilized during the initial creation of Creo Illustrate documents. These policies are used during upload of new dynamic data (when a new dynamic document can be created based on a model file). If the policy is changed at a later date, existing data is not affected.

These policies can be used together to drive the naming and numbering behavior. For example, auto numbering can be used with Custom to form a numbering policy. The four policies can be described as follows:

- Auto-numbering
  - Auto-numbering is the default naming and numbering policy.
  - The Creo Illustrate document number is provided by the document number generator (either using the standard Windchill number generator, or through customization).
  - In the **Rename** user interface, the number field can not be edited, unless auto-numbering is disabled or the override preference is set by the administrator that allows users to override the auto-numbering sequence and manually enter a value in the number field.
- Name-driven
  - When auto-numbering is disabled, name-driven policy comes into effect.
  - The Windchill **Object Number** is copied from the Creo Illustrate file name (the file extension can be dropped and is controlled by the Windchill preference, **Upload Drop Number File Extension**).

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- The Windchill Document **Name** is also copied from the model file name (the file extension can be dropped and is controlled by the Windchill preference, **Upload Drop Name File Extension**).
  - If you use Workspace **Rename** to set the Windchill **Name** or **Number** attribute for a new document, its value is pushed from Windchill to the authoring environment .
  - The Creo Illustrate document and part **Number** field can be edited using **Rename**. If parts are auto-created from dynamic documents, then the user does not get the chance to specify a part number. If parts are created manually, the user has the option to specify number during creation (assuming autonumbering is disabled). If a dynamic document is created in the Creo Illustrate application, it has no ability to define the Windchill dynamic document number. The user must then use **Rename** in the workspace to allocate a different value prior to check in.

### Identifying the Current Naming and Numbering Policy

The algorithm used to understand which policy is currently set in the system (for a particular container and class of object) is as follows:

- If auto-numbering is set in object initialization rules, then the policy is auto-numbering.
- If custom behavior is implemented in rules, then the policy is custom.
- If neither auto-numbering or custom behavior is set and the parameter-driven properties are set in the **Preference Management** utility, then policy is parameter-driven.

### Windchill Preferences for Naming and Numbering

The following preferences control parameter-driven Naming & Numbering policy when using Windchill Workgroup Manager. By default, these properties have no value:

When set to **Yes** (default is **No**), the following properties omit file extensions from name and number attributes for new dynamic documents during upload:

- **Upload Drop Name File Extension.**
- **Upload Drop Number File Extension.**

## Configuring Collection

The administrator and/or user can define default collector behavior through the Windchill **Preference Management** utility (the default behavior can be changed at run time). The following collection modes are supported:

- Collecting all dependents at the time the collector is launched with a seed object (up- front) such that selection actions are enabled or disabled depending on what is selected.
- Collecting only the required dependents of a seed object at the time the collector is launched.
- Collecting dependents incrementally when an object is selected and an action clicked.

The first two listed modes can be accomplished on the **Basic** tab of the action page, while the incremental selection is performed on the **Advanced** tab.

The system provides default settings that apply to both tabs. You can tailor the collection process by setting one or more specific preferences that determine the default object collection that is performed when an action is initiated.

All preferences related to the collector can be set per site, organization, or user in the **Windchill ► Site ► Utilities ► Preference Management**. The collector infrastructure allows each specific action (for example, **Check In** or **Edit Attributes**) to specify a default collection for the action, as well as a default display for the **Advanced** tab.

The preference is set in the **Preference Management** utility under a node specific to the action. For example, the preferences for collection at **Check In** are found as illustrated in the next graphic.

Notification		
Operation		
Category for Operations preferences		
Auto Associate		
Preferences for associate, disassociate and auto-associate operations		
CAD Data Management		
Workgroup Manager preferences for the CAD Data Management actions		
CheckIn Operation		
Category for CheckIn preferences		
Auto Associate upon Check In	No	Controls the default behavior whether or not to perform Auto Associate1
Collector		
Checkin Collector Rule Values		
Conflict for Out of date Secondary con Yes		Controls the default behavior whether or not to provide overridable conf
Create As Stored	Yes	Specifies to create an As Stored configuration upon checkin.

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The following collector preferences can be set to the values listed (note that not all the following collection options are applicable to every action — e.g. Collector for “Add to Baseline” has difference collection options to “Add to Workspace”):

- Default display: All as a list, Structure, Drawings, Related Objects, Family table, References, Change objects.
- Configuration: Latest, As Stored, Baseline, Effectivity, Product Maturity, Use Workspace configuration.
- Drawings: All , Initially Selected Only, None
- Instances: All , Initially Selected Only, None
- Generics: All , Initially Selected Only, None
- Parts: All , Initially Selected Only, None
- Documents: All , Initially Selected Only, None
- CAD Documents: All , Initially Selected Only, None
- Affected Data : All, None
- Change Items: All, None
- Change Notice: All, None
- Change Activity: All, None
- Change Request: All, None
- Related objects : All, Required, None
- CAD Document Dependents: All , Initially Selected Only, None
- Part Dependents: All, None
- Variance: All, None

Because only a subset of these preferences might be relevant for a specific action, not all preferences are required to exist for each action.

**Note**

*A particular preference can be overridden by each individual user (if not locked at the Organization or Site level by the administrator).*

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## Enabling As Stored Configurations

The As Stored configuration is created at checkin. When more than one CAD document or a drawing CAD document is checked in from the workspace, the As Stored configuration is created. The As Stored configuration is a snapshot of the workspace at the time of check-in. You can use the As Stored configuration in the following ways:

- Reload the workspace the same way as the iteration is modified when the user works with (or references) the checked-in iteration in the future.
- Retrieve the assembly using the As Stored configuration.

By default, **Create As Stored** is set to **Yes** in the **Preference Management** utility. **Create As Stored** specifies to create an As Stored configuration upon checkin.

As Stored is particularly useful when working with Drawings and Assemblies. It allows the Workgroup Manager user to retrieve an assembly or drawing using the same related components “as it was stored” with at the point of Check In (as opposed to retrieving Latest versions of all related parts which may have been modified independently of the parent drawing/assemblies during design developments). This is useful in order to get back to a known configuration of drawing or assembly. Baselines also achieve the same result of recording specific configuration of versions/iterations. However, a baseline is a more formal method of capturing a configuration, which is a conscious decision to create a separate managed object that captures a particular configuration of components rather than just “as they were stored.” Often As Stored configuration is used to generate a baseline at a later date, e.g. to create a baseline of a set of drawings/assemblies that need to be reviewed at a design review. You would use As Stored since Latest may pick up some later versions of the underlying parts that have not been detailed in the drawing yet or some that may cause assembly to fail regeneration.

Note that As Stored only considers data in the workspace at the time of Check In. For example, if you are checking in an assembly which has 10 components and only 5 of those components are present in the workspace, then your As Stored configuration will contain only those 5 parts. The other 5 parts will be retrieved according to Latest configuration.

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## Configuring Revision

Administrators can configure how the system behaves during a revision operation by setting preferences in the **Windchill ► Site ► Utilities ► Preference Management**. These settings can determine whether to do the following:

- Allow revision to a level other than the next in the revision scheme.
- Create or maintain passive associations during a revision action.
- Synchronize revision levels of CAD documents and parts during an auto associate action.

### Note

*The revision level synchronization behavior described in the section, **Configuring Auto Associate Parts**, also applies to **Revision** when the preference **Auto Associate ► Set Revision For Part** is set to **Yes**.*

## Configuring the Ability to Set a Revision Level

A key-value pair in the Windchill preference system allows an administrator to enable users to set a higher revision level than the next label in the revision scheme.

When set to **Yes** (default), the preference **Set Target Revision** adds a **Select Revision Label** button to the **New Revision** user interface that allows the user to skip one or more levels of revision during a revise operation. In addition, a revision level can be set in the **New CAD Document** user interface. When set to **No**, the object is revised to a default target/next revision in its series.

## Configuring the Revision of Associated Items

You can configure how **Content** links are carried forward to new revisions of associated (linked) objects. Using the `xconf` file, modify or add `<Windchill>/codebase/wt.properties`, which contains the following key-value pairing:

**Key = wt.vc.struct.copyDescribeLinkOnRevise**

Values: **No**, **Yes** (default)

With `wt.vc.struct.copyDescribeLinkOnRevise` set to the default value, **Yes**, the system behaves as follows for the revision of a CAD document or part with a **Content** association.

- There is only one link; iteration-to-iteration.
- If only the CAD document is revised, the link is not carried forward. The new version of the CAD document does not have an association to the existing

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versions of the part (because the existing part version might be released, and the content version of the part should not change without a check out or revise).

- If only the part is revised, both the new and the old version of the part have a link to the existing version of the CAD document.
- If both the CAD document and part are revised during the same revision, the new versions have a new **Content** link between them, and the old versions maintain their existing link.

When `wt.vc.struct.copyDescribeLinkOnRevise` is set to **No**, the system behaves the same as when it is set to **Yes**, with the following exception.

- If both the CAD document and part are revised during the same revision, then regardless of the settings in the user interface, revise acts as if the CAD document and part are being revised alone.

## Rules for New Versions

Consider these rules for new versions:

- By default, all new versions should be created in the same location (context and folder) as the original.
- If there is a user interface, the user can override the default location and choose to place the new version somewhere else.
- For Windchill PDMLink, the team and life cycle are determined by the object initiation rules of the context.
- The view of new parts defaults to the same view as the original.

## Configuring Attributes

Attributes are entities that define properties of objects, such as CAD files or Windchill business objects. They always consist of a name and value, for example, **Life Cycle State** can be assigned the value, **In Work**.

In Windchill, there are two types of attributes: hard and soft. The system attributes that are hard-coded into Windchill, are therefore available by default. System attributes are typically defined for a set of characteristics that most business objects share, such as name or number. User-defined attributes, are not hard-coded; therefore are not available by default. The user defines soft attributes, for example, **COLOR** or **LENGTH**, for individual objects or classes of objects. CAD applications have analogous entities, variously called properties, parameters, or attributes, that can be mapped to Windchill attributes so that their values are communicated between Windchill and the CAD application, if desired.

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## Note

*For purposes of clarity in discussing attribute management, it is the standard of this guide to refer to Windchill attributes as attributes and to refer to CAD application attributes, properties, and parameters as parameters.*

Administrators configure attributes and parameters so they are published (viewable) in the CAD application and Windchill. This section describes:

- Attribute and parameter types.
- Configuring attributes and parameters so they are published from Windchill to the CAD application.
- Adding new CAD application parameters.
- Managing system attributes and parameters.
- Customizing existing Windchill attributes to create additional CAD parameters.
- Ensuring that user-defined CAD application parameters are recognized in Windchill.
- Mapping attributes and parameters.

## Attribute and Parameter Types

There are three main categories of attribute or parameters when working with Windchill Workgroup Manager:

- Windchill default system attributes.
- authoring application parameters.
- custom parameters.

## Note

*No authoring application parameters are exposed to Windchill through Windchill Workgroup Manager.*

## Windchill Default System Attributes

The set of Windchill default system attributes are automatically published in the authoring application when a CAD part or drawing is checked in or saved in Windchill. These are used to support required authoring application system parameters, such as revision. Because Windchill writes these to the authoring application, they cannot be modified. The information on these attributes always comes from the Windchill system, and therefore is always up-to-date (for example, **Revision** reflects the Windchill revision).

A list of the attributes provided by Windchill is listed in the following table.

### Windchill Default System Attributes

Windchill Attribute	Description
<b>PTC_WM_REVISION</b>	<ul style="list-style-type: none"> <li>• String Type</li> <li>• Initial value the first revision corresponding to the default context of the Workspace. This should be set to <b>Yes</b> whether or not the server is online or offline.</li> <li>• Corresponds to the string used for just the Windchill version of the CAD document. By default this is usually, A, B, C, etc.</li> <li>• If the user modifies the value, then upon the next retrieval or update, the value is overridden by the system</li> <li>• Predicted upon save for new objects.</li> <li>• Should be set to the current CAD document revision when setting upon check out for an existing object.</li> </ul>
<b>PTC_WM_PART_NAME</b>	<ul style="list-style-type: none"> <li>• String Type</li> <li>• Initial value: Not Applicable. This system attribute should never be created on initial creation of the CAD document.</li> <li>• Corresponds to the string used for the name of the actively associated part to this model's CAD document.</li> <li>• May be created and set before the initial auto-association if the naming policy is set to use a custom parameter. Otherwise, the value is ignored and set upon Save.</li> <li>• For new or existing CAD documents, if the user associates, then the system parameter should be available to be updated upon next retrieval in or save from the authoring application.</li> <li>• When checking out an existing CAD document, the system parameter should reflect the name of the CAD document's actively associated</li> </ul>

## Windchill Default System Attributes (continued)

Windchill Attribute	Description
	<p>part. If none exists, then the system parameter is not created in the authoring application.</p>
<b>PTC_WM_PART_NUMBER</b>	<ul style="list-style-type: none"> <li>• String Type</li> <li>• Initial value: Not Applicable. This system parameter should never be created on initial creation of the CAD document.</li> <li>• Corresponds to the string used for the number of the actively associated part to this model's CAD document.</li> <li>• May be created and set before the initial auto-association if the naming policy is set to use a custom parameter. Otherwise, the value is ignored and set upon Update.</li> <li>• This attribute is not predicted. For new or existing CAD documents, if the user associates, then the system parameter should be available to be updated upon next retrieval in or save from the authoring application.</li> <li>• When checking out an existing CAD document, the system parameter reflects the number of the CAD document's actively associated part. If none exists, then the system parameter should not be created in the authoring application.</li> </ul>
<b>PTC_WM_NUMBER</b>	<ul style="list-style-type: none"> <li>• String Type</li> <li>• Initial value should be set according to the naming policies. That is, if the policy is set to:               <ul style="list-style-type: none"> <li>○ CAD Model Name, then the initial value should be based on CAD file name. The number will contain the extension based on a preference.</li> <li>○ Auto-numbering, then the initial value should be generated by taking a number from the PTC auto-numbering system.</li> </ul> </li> </ul>

## Windchill Default System Attributes (continued)

Windchill Attribute	Description
	<ul style="list-style-type: none"> <li>○ CAD attribute, then the initial value should be based on the CAD property that is defining the number.</li> <li>○ Custom naming/numbering application, then the initial value should be generated by taking a number from the customer-defined auto-numbering system.</li> </ul> <p>If the server is offline, and the system cannot predict the number (due to auto-numbering mechanisms not being available), then the initial value is denoted by a white space.</p> <ul style="list-style-type: none"> <li>• Corresponds to the string used for the number of this model's CAD document.</li> <li>• May be created and set before the initial save if the naming policy is set to use a custom parameter. Otherwise, the value will be ignored and set upon initial save or a subsequent retrieval.</li> <li>• Predicted upon save for new objects.</li> <li>• Should be set to the current CAD document number when setting upon check out for an existing object.</li> </ul>
<b>PTC_WM_NAME</b>	<ul style="list-style-type: none"> <li>• String Type</li> <li>• Initial value should be set according to the naming policies. That is, if the policy is set to:               <ul style="list-style-type: none"> <li>○ CAD Model Name, then the initial value should be based on CAD file name. The name will contain the extension based on a preference to keep the extension.</li> <li>○ Auto-numbering, then the initial value should be based on CAD file name. The name will contain the extension based on a preference to keep the extension.</li> </ul> </li> </ul>

## Windchill Default System Attributes (continued)

Windchill Attribute	Description
	<ul style="list-style-type: none"> <li>○ CAD Attribute, then the initial value should be based on the CAD property that is defining the name.</li> <li>○ Custom naming or numbering application, then the initial value should be based on CAD file name. The name will contain the extension based on a preference to keep the extension.</li> <li>○ The initial value should be able to be set whether the server is online or offline.</li> <li>• Corresponds to the string used for the name of this model's CAD document</li> <li>• May be created and set before the initial save if the naming policy is set to use a custom parameter. Otherwise, the value will be ignored and set upon initial save or a subsequent retrieval.</li> <li>• Predicted upon save for new objects.</li> <li>• Should be set to the current CAD document name when setting upon check out for an existing object.</li> </ul>
<b>PTC_WM_ORGANIZATION_ID</b>	<ul style="list-style-type: none"> <li>• String Type</li> <li>• Initial value should be: <ul style="list-style-type: none"> <li>○ Predict: Set according to be the organization identifier of the context of the workspace.</li> <li>○ Offline: Set according to be the organization identifier of the context of the workspace.</li> </ul> </li> <li>• Corresponds to the CAD document's organization identifier</li> </ul>

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## Windchill Default System Attributes (continued)

Windchill Attribute	Description
	<ul style="list-style-type: none"><li>• Predicted upon save for new objects</li><li>• Should be set to the current CAD document organization identifier when setting upon check out for an existing object.</li></ul>
<b>PTC_WM_ITERATION</b>	<ul style="list-style-type: none"><li>• String Type</li><li>• Initial value should be predicted and set to <b>1</b> whether the server is online or offline.</li><li>• Corresponds to the CAD document iteration.</li><li>• Predict for both new and existing. Upon check out, the value should be predicted to be the iteration after the current CAD document iteration.</li></ul>
<b>PTC_WM_LIFECYCLE</b>	<ul style="list-style-type: none"><li>• String Type</li><li>• Initial value should be set according to be the object initialization rules of the context of the workspace. This should be <b>Yes</b> whether the server is online or offline. For more information on object initialization rules, refer to Identifying the Current Naming and Numbering Policy.</li><li>• Corresponds to the name of the CAD document's life cycle.</li><li>• Predicted upon save for new objects.</li><li>• Should be set to the current CAD document life cycle when setting upon check out for an existing object.</li></ul>

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## Windchill Default System Attributes (continued)

Windchill Attribute	Description
<b>PTC_WM_LIFECYCLE_STATE</b>	<ul style="list-style-type: none"><li>• String Type</li><li>• Initial value should be set according to be the object initialization rules of the context of the workspace. Should take the initial state of the default life cycle. Initial value should be set whether the server is online or offline.</li><li>• Corresponds to the name of the CAD document's life cycle state.</li><li>• Predicted upon save for new objects.</li><li>• Should be set to the current CAD document life cycle state when setting upon check out for an existing object.</li></ul>
<b>PTC_WM_IS_DRAWING</b>	<ul style="list-style-type: none"><li>• Boolean Type</li><li>• Initial value should be set according to CAD Adapter's declaration of the CAD document type or Model Descriptor. If it is set to be a drawing, then the Boolean is set to <b>Yes</b>, otherwise it is <b>No</b>.</li><li>• Set to <b>Yes</b> if the CAD document is a drawing.</li><li>• The value should not change once the object is checked in.</li></ul>

### Additional Default System Attributes for Creo Elements/Direct Drafting

The following table lists additional default Windchill system attributes that exist for in the Windchill Workgroup Manager integration with Creo Elements/Direct Drafting:

Windchill Attribute	Description
<b>PTC_WM_PART_REVISION_FOR_DRAWING</b>	<ul style="list-style-type: none"> <li>• String Type</li> <li>• Default: Empty string</li> <li>• Scope: Part</li> <li>• The revision information for the WTPart associated with the drawing itself, or the owning model of a drawing. If the drawing is associated with a WTPart, this attribute has preference.</li> </ul>
<b>PTC_WM_PART_STATE_FOR_DRAWING</b>	<ul style="list-style-type: none"> <li>• String Type</li> <li>• Default: Empty string</li> <li>• Scope: Part</li> <li>• The life cycle state information for the WTPart associated with the drawing itself, or the owning model of a drawing. If the drawing is associated with a WTPart, this attribute has preference.</li> </ul>
<b>PTC_WM_CREATED_BY</b>	<ul style="list-style-type: none"> <li>• String Type</li> <li>• Always set</li> <li>• Scope: Content</li> <li>• The name of the user that created the CAD document</li> </ul>
<b>PTC_WM_CREATED_ON</b>	<ul style="list-style-type: none"> <li>• Date and time (String) Type</li> <li>• Always set</li> <li>• Scope: Content</li> <li>• The date and time that the CAD document was created</li> </ul>
<b>PTC_WM_MODIFIED_BY</b>	<ul style="list-style-type: none"> <li>• String Type</li> <li>• Default: Empty string</li> <li>• Scope: Content</li> <li>• The name of the user that last modified the CAD document</li> </ul>

Windchill Attribute	Description
<b>PTC_WM_MODIFIED_ON</b>	<ul style="list-style-type: none"> <li>• Date and time (String) Type</li> <li>• Default: Empty string</li> <li>• Scope: Content</li> <li>• The date and time that the CAD document was last modified</li> </ul>
<b>PTC_WM_LAST_CHANGE_NOTE</b>	<ul style="list-style-type: none"> <li>• String Type</li> <li>• Default: Empty string</li> <li>• Scope: Content</li> <li>• The text note added to the last checkin</li> </ul>

## Custom Parameters

Custom parameters are created by the CAD administrator in the CAD application on the **Custom Product** tab, and need to be mapped in Windchill using the **Type and Attribute Management** utility so that they are recognized in Windchill and published in the authoring application. For more information on mapping, refer to [Attribute and Parameter Types on page 91](#).

To access the **CustomProduct** page, go to the **File** menu in your authoring application.

## Configuring Attributes and Parameters

To ensure attributes and parameters are published in Windchill, they need to be configured in Windchill and the authoring application. If a CAD drawing or part has been checked in or saved in Windchill, the **Properties** page displays the authoring application default parameters. These parameters are read-only; however if required by your company, they can be modified. To modify these parameters, the Windchill administrator modifies a preference in the **Preference Management** utility page in Windchill, and maps it to the authoring application preference. Mapping is performed so that the preference is recognized between Windchill and the authoring application. For more information about mapping, refer to [Attribute and Parameter Types on page 91](#).

Before you have created a CAD document and before it is checked in, you can set up attributes ahead of time in the **Preference Management** utility. For example, for naming and numbering you can go to the authoring application drop-down list in **Set Preference** window and set up the attributes you plan to use.

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If a user creates a authoring application parameter that conflicts in type with a Windchill system attribute, then this user-created parameter should be deleted and the system parameter of the correct type should be created by the Windchill administrator. In this way, there is no conflict between the authoring application and Windchill, and the parameter is published to the authoring application user interface.

For example, in the **Preference Management** utility, locate the preference called, **PTC\_WM\_REVISION**.

Select the set preference icon to display the **Set Preference** window for this preference.

If you want to change the default Windchill preference value for your authoring application, select it from the **Client** pull-down menu. For example, set **PTC\_WM\_REVISION=XYZ**. The CAD modeled system parameter, **XYZ** is available for revision in the authoring application.

## Adding New Authoring Application Parameters

Default system parameters are pushed from Windchill when a CAD part or drawing is saved or checked in, and they appear on the page discussed below. You can keep these parameters, change them, or create new ones. Use the following process to access the page to add a new parameter.

To add new parameters, select **Custom** on the **iProperties** page.

To add new parameters, select **Custom** on the **Drawing Properties** page.

To add new parameters, select **Custom** on the **Properties** page.

To add new parameters, select **Attributes** on the **Properties** page.

To add new parameters, select the **Product** tab on the **Properties** window (right-click object name in the **Specification Tree > Properties**).

### **Note**

*Hyperlinks are not supported; therefore Hyperlink base is not valid for mapping.*

The following is a list of standard parameter names that can be mapped either to a custom parameter name or a standard Windchill system attribute (using the server-side preferences mapping functionality) to put the values of the Windchill system attribute into a parameter. The labels that should be used are listed in the next table.

### AutoCAD Default Parameters

Parameter Name	Description
<b>MSP:AUTHOR</b>	Publishes Author on <b>Summary</b> tab.
<b>MSP:COMMENTS</b>	Publishes Comment on <b>Summary</b> tab.
<b>MSP:KEYWORDS</b>	Publishes Keywords on <b>Summary</b> tab.
<b>MSP:SUBJECT</b>	Publishes Subject on <b>Summary</b> tab.
<b>Naming Parameter=MSP:TITLE</b>	Publishes Title on <b>Summary</b> tab.

### AutoDesk Inventor Default Parameters

Project Tab Parameter	Published Result
<b>MSP:AUTHORITY</b>	<b>Authority</b>
<b>MSP:COSTCENTER</b>	<b>Cost Center</b>
<b>MSP:DATECHECKED</b>	<b>Date Checked **</b>
<b>MSP:DESCRIPTION</b>	<b>Description</b>
<b>MSP:DESIGNER</b>	<b>Designer</b>
<b>MSP:ENGINEER</b>	<b>Engineer</b>
<b>MSP:MATERIAL</b>	<b>Material</b>
<b>MSP:REVISIONNUMBER</b>	<b>Revision Number</b>
<b>MSP:PARTNUMBER</b>	<b>Part Number</b>
<b>MSP:PROJECT</b>	<b>Project</b>

## AutoDesk Inventor Default Parameters (continued)

Project Tab Parameter	Published Result
MSP:VENDOR	Vendor
STATUS TAB PARAMETER	Published Result
MSP:CHECKEDBY	Checked By
MSP:DESIGNSTATE	Design State
MSP:ENGAPPROVEDBY	Eng. Approved By*
MSP:ENGDATEAPPROVED	Engr Date Approved **
MSP:MFGAPPROVEDBY	Mfg Approved By
MSP:MFGAPPROVEDDATE	Mfg Approved Date
SUMMARY TAB PARAMETER	Published Result
MSP:STATUS	Status
MSP:AUTHOR	Author
MSP:CATEGORY	Category
MSP:COMMENTS	Comments
MSP:COMPANY	Company
MSP:KEYWORDS	Keywords
MSP:MANAGER	Manager
MSP:SUBJECT	Subject
Naming Parameter=MSP:TITLE	Title

## CATIA V5 Default Parameters

Parameter Name	Description
CAT:ACTIVE_BOM	
CAT DEFINITION	Populates <b>Definition</b> text field.
CAT:DESCINST	
CAT:DESCREF	Populates <b>Description</b> text field text field.
CAT:DRAW_COMMENT	
CAT:DRAW_RESPON	
CAT:NOMENCLATURE	<b>Nomenclature</b> text field
CAT:PART_NR	Populates <b>Part Number</b> text field.
CAT:REVISION	Populates <b>Revision</b> text field.

---

## CATIA V5 Default Parameters (continued)

Parameter Name	Description
CAT:SOURCE	Populates <b>Source</b> drop-down list choices ( <b>Unknown, Made, Bought</b> ) text field.

## SolidWorks Default Parameters

Parameter Name	Description
MSP:AUTHOR	Publishes Author on <b>Summary</b> tab.
MSP:COMMENT	Publishes Comment on <b>Summary</b> tab.
MSP:KEYWORDS	Publishes Keywords on <b>Summary</b> tab.
MSP:SUBJECT	Publishes Subject on <b>Summary</b> tab.
Naming Parameter=MSP:TITLE	Publishes Title on <b>Summary</b> tab.

When **Naming Parameter=MSP:TITLE** is mapped in the **Preference Manager** utility, the CAD document name is published in **Title** field on the **File > Properties > SummaryFile > iProperties->Summary** tab.

Default system parameters are created from the **Product** tab of the **Properties** window (right-click object name in the **Specification Tree > Properties**).

## Editing the Delegate

Windchill provides a server-side delegate that can be used to insert parameters into a authoring application part upon download. This mechanism can be used to pass information from the server down to the authoring application, where it can be used like any other authoring application parameter (for example, to place information on drawing forms). Parameters beginning with **PTC** or **PROI** are regarded as reserved system parameters and cannot be propagated by the customization. If they are added in the customization, they are ignored by the download service.

### Note

*The customized parameters are provided to the client upon download and are not updated in the CAD application session after a Windchill operation (for example, check in). For example, if a customized parameter is assigned the value of the CAD document number, its value is provided to the client upon download. If the CAD document is later renumbered, the value in the authoring application session or the client cache is not automatically updated. The Windchill service delegate mechanism is used to allow the customization.*

---

The following steps explain the customization process:

1. Create a Java class that implements the interface `ModeledAttributesDelegate`. The interface definition is as follows:

```
package com.ptc.windchill.uwgm.proesrv.c11n;

import java.util.Collection;

import java.util.HashMap;

import wt.util.WTException;

public interface ModeledAttributesDelegate
{
    // getAvailableAttributes() returns
    // HashMap<String, Object> which contains
    // HashMap<Attribute name, Attribute type>
    HashMap getAvailableAttributes();

    // getModeledAttributes(Collection docs) returns
    // HashMap<input object, HashMap<Attribute name,
Attribute value>>
    HashMap getModeledAttributes(Collection docs)
throws WTException;
}
```

Please see a sample implementation located at:

```
<Windchill home>/codebase/com/ptc/windchill/uwgm/proesrv/c11n/
DefaultModeledAttributesDelegate.java
```

2. Edit the `site.xconf` file (found in `<Windchill>`) to add the following property to indicate availability of customization service on the server:

```
<Service context="default"
```

---

```

name="com.ptc.windchill.uwgm.proesrv.c11n.ModeledAttributesDelegate"
targetFile="codebase/service.properties">
<Option cardinality="singleton"
  requestor="java.lang.Object"
  serviceClass="com.ptc.windchill.uwgm.proesrv.c11n.DefaultModeled
AttributesDelegate"/>
</Service>

```

Use the path of your class in place of value of `serviceClass` (that is, replace

```
com.ptc.windchill.uwgm.proesrv.c11n.Default
```

```
ModeledAttributesDelegate
```

with the path to your class)

3. Then use the `xconfmanager` tool to apply the changes to `service.properties` file.

```
Runxconfmanager -p
```

4. Restart Windchill.

## Customizing Existing Windchill Attributes

To create more authoring application parameters, Windchill provides a server-side delegate that can be used to customize the system parameters added to the CAD file upon download. This is referred to as extending the Windchill system attribute. Upon **Add to Workspace**, the system queries Windchill for system parameters. After the administrator changes the server-side delegate code by adding to it (or commenting out what is there), the authoring application publishes the new parameter.

Windchill provides a sample delegate that creates all the authoring application optional system parameters. These are reserved as a convenience, so that modified or new authoring application parameters can easily be mapped. The administrator locates the delegate and modifies it according to company requirements. The list of reserved Windchill attributes is listed below.

- `SP_WM_PART_REV`
  - String Type
  - Corresponds to the string used for just the Windchill Version of the actively associated WTPart.
- `SP_WM_PART_ITERATION`
  - String Type

- 
- Corresponds to the number used for just the Windchill Iteration of the actively associated WTPart.
  - SP\_WM\_PROJECT
    - String Type
    - Corresponds to the name of the Team of the CAD document.
  - SP\_WM\_VIEW
    - String Type
    - Corresponds to the label of the View of the actively associated WTPart.
  - SP\_WM\_PART\_LIFECYCLE
    - String Type
    - Corresponds to the name of the actively associated WTPart's lifecycle.
  - SP\_WM\_PART\_TYPE
    - String Type
    - Corresponds to the soft attribute name of the actively associated WTPart.
  - SP\_WM\_PART\_LC\_STATE
    - String Type
    - Corresponds to the name of the actively associated WTPart's current lifecycle state.

## Editing the Delegate

Windchill provides a server-side delegate that can be used to insert parameters into a authoring application part upon download. This mechanism can be used to pass information from the server down to the authoring application, where it can be used like any other authoring application parameter (for example, to place information on drawing forms). Parameters beginning with **PTC** or **PROI** are regarded as reserved system parameters and cannot be propagated by the customization. If they are added in the customization, they are ignored by the download service.

### Note

*The customized parameters are provided to the client upon download and are not updated in the CAD application session after a Windchill operation (for example, check in). For example, if a customized parameter is assigned the value of the CAD document number, its value is provided to the client upon download. If the CAD document is later renumbered, the value in the authoring application session or the client cache is not automatically updated. The Windchill service delegate mechanism is used to allow the customization.*

---

The following steps explain the customization process:

1. Create a Java class that implements the interface `ModeledAttributesDelegate`. The interface definition is as follows:

```
package com.ptc.windchill.uwgm.proesrv.c11n;

import java.util.Collection;
import java.util.HashMap;

import wt.util.WTException;

public interface ModeledAttributesDelegate
{
    // getAvailableAttributes() returns
    // HashMap<String, Object> which contains
    // HashMap<Attribute name, Attribute type>
    HashMap getAvailableAttributes();

    // getModeledAttributes(Collection docs) returns
    // HashMap<input object, HashMap<Attribute name,
Attribute value>>
    HashMap getModeledAttributes(Collection docs)
throws WTException;
}
```

Please see a sample implementation located at:

```
<Windchill home>/codebase/com/ptc/windchill/uwgm/proesrv/c11n/
DefaultModeledAttributesDelegate.java
```

2. Edit the `site.xconf` file (found in *<Windchill>*) to add the following property to indicate availability of customization service on the server:

```
<Service context="default"
```

---

```

name="com.ptc.windchill.uwgm.proesrv.c11n.ModeledAttributesDelegate"
targetFile="codebase/service.properties">
<Option cardinality="singleton"
  requestor="java.lang.Object"
  serviceClass="com.ptc.windchill.uwgm.proesrv.c11n.DefaultModeled
AttributesDelegate"/>
</Service>

```

Use the path of your class in place of value of `serviceClass` (that is, replace

```

com.ptc.windchill.uwgm.proesrv.c11n.Default
ModeledAttributesDelegate

```

with the path to your class)

3. Then use the `xconfmanager` tool to apply the changes to `service.properties` file.

```
Runxconfmanager -p
```

4. Restart Windchill.

## Mapping User-defined Authoring Application Parameters

By default, user-defined CAD parameters are mapped to Windchill attributes with the same name. The administrator defines the ability for authoring applications to access parameters that are mapped to Windchill attributes. Errors and conflicts may occur upon upload if a parameter from a CAD document is trying to be mapped to a soft attribute in Windchill with a type mismatch or constraint violation. Because Windchill attributes are pushed to the CAD document upon download, only attributes that have changed since content upload are modified on check out or download. The user may set changes to user-defined attributes into the authoring application using **Update**.

### AutoCAD User-defined Parameters

The type of parameter supported on this tab is String. Additionally, Block Attributes are also only strings; therefore Windchill Workgroup Manager for AutoCAD allows mapping to attributes of different types.

- If the user has defined a parameter that has a name equivalent to the soft attribute name or AutoCAD mapped name in the CAD document soft-attribute

---

Workgroup Manager CAD document, then the parameter is automatically mapped to the Windchill soft attribute.

- If the user has not defined a parameter with the same syntax as the Windchill soft attribute, the parameter information is not exchanged and, therefore, not published.
- PTC supports mapping of soft attributes to AutoCAD parameters of types:
  - Integer
  - Number
  - String
  - Boolean
  - Date
- As long as the value of the parameter can be populated into the Windchill soft attribute, then values from AutoCAD are pushed to Windchill.
- For Strings, there should never be any conflict.
- For Boolean, the system accepts either the value of Yes or No in any case, that is: YES, Yes, YEs, YeS, yeS, yES are interpreted to be a valid Yes for the Boolean attributes.
- For Integer:
  - The system accepts pure numbers, e.g. 1
  - If there is a , or a . character, then the number is not valid.
  - If there are any characters that are not numbers, then the number is not valid.
- For Real Numbers:
  - The system accepts pure numbers, e.g. 1 to be 1.00.
  - The system interprets any number with a . or , to be valid. That is: 1.00 or 1,00 are interpreted to be the real number 1.
  - If there is more than one . or , then the value is invalid.
  - If the number is in exponential format, it is valid.
    - ◆ That is, 1E+06 is 1000000.00.
    - ◆ The user can write the **E** character in either upper or lower case, for example, **1e+06** and **1E+06** are both valid and equivalent.
    - ◆ If the user is writing a positive exponential value, then he can use the **+** sign, or not. Both **1E06** and **1E+06** are valid.

- 
- ◆ If there are any characters that are not numbers, then the number is not valid.

**Note**

*Hyperlink is not supported. AutoCAD does not support user-defined attributes on uses links.*

AutoDesk Inventor User-Defined Parameters

- PTC supports mapping of soft attributes to attributes of the following types:
  - Real Number
  - String
  - Boolean
  - Date
  - Integer
- If these types are not matched between AutoDesk Inventor and Windchill, the parameter is not published.

**Note**

*Autodesk Inventor does not support customer-defined parameters on Uses Links.*

CATIA V5 User-defined Parameters

- PTC supports mapping of soft attributes to authoring application parameters of the following types:
  - Real Number
  - String
  - Boolean
  - Integer
- If these types are not used, then the parameters are not published in the authoring application.

SolidWorks User-Defined Attributes

SolidWorks and Windchill support the following types of user-defined attributes:

- Number
- Integer: The system accepts pure numbers, e.g. 1, If there is a , or a . character, then the number is not valid. If there are any characters that are not numbers, then the number is not valid.
- Real Numbers:

---

The system accepts pure numbers, e.g. 1 to be 1.00. The system interprets any number with a . or , to be valid. That is: 1.00 or 1,00 are interpreted to be the real number 1.

If there is more than one . or , then the value is invalid. If the number is in exponential format, it is valid. For example, 1E+06 is 1000000.00.

The user can write the **E** character in either upper or lower case. For example, **1e+06** and **1E+06** are both valid.

If the user can write a positive exponential value with or without a + sign. For example, **1E06** and **1E+06** are valid.

If there are any characters that are not numbers, then the number is not valid.

- Yes/No or Boolean
- Date
- String
- If the authoring application parameter uses a value other than these, it will not be published.

### **Note**

*SolidWorks does not support attributes on Uses Links.*

NX, User-defined Parameters

- The type of parameters supported:
  - Real Number
  - String
  - Integer
  - Date\*

The date parameters are pushed from Windchill to the CAD application in the format of **%m/%d/%Y %H:%M** where (**%m** - month, **%d** - day, **%Y** - 4 digits year, **%H** – hours in 24-hour format, **%M** - minutes)

### **Note**

*NX does not support customer-defined attributes on **Uses** links.*

## **Mapping Attributes and Parameters**

In order to set up the parameters so that they are published in the authoring application user interface, the administrator maps them in the **Type and Attribute Management** utility. For more information on the **Type and Attribute Management** utility, refer to *Windchill Business Administrator's Guide* located at [PTC Reference Documents](#).

---

The following list details information about case-sensitivity when setting up authoring application parameters that need to be recognized in Windchill:

- Windchill is case-sensitive in its handling of attributes; but not all authoring applications are.
- If the authoring application does not support case-sensitive attribute names, then they should be handled as the value in all upper case characters, regardless of how they are shown in the authoring application.
- Windchill soft attribute **Name** is case-sensitive; therefore, when creating an soft attribute to map to a authoring application parameter that does not support case-sensitive attributes, then the name must be in all uppercase characters or the administrator must explicitly map the name of the soft attribute to the name of the CAD parameter.
- Your authoring application does not support case-sensitive parameters; however, it does support mixed-case parameters. This means that the user cannot create a parameter named **Cost** or **COST**, but the user can type either **COST**, **Cost**, and **cost**.

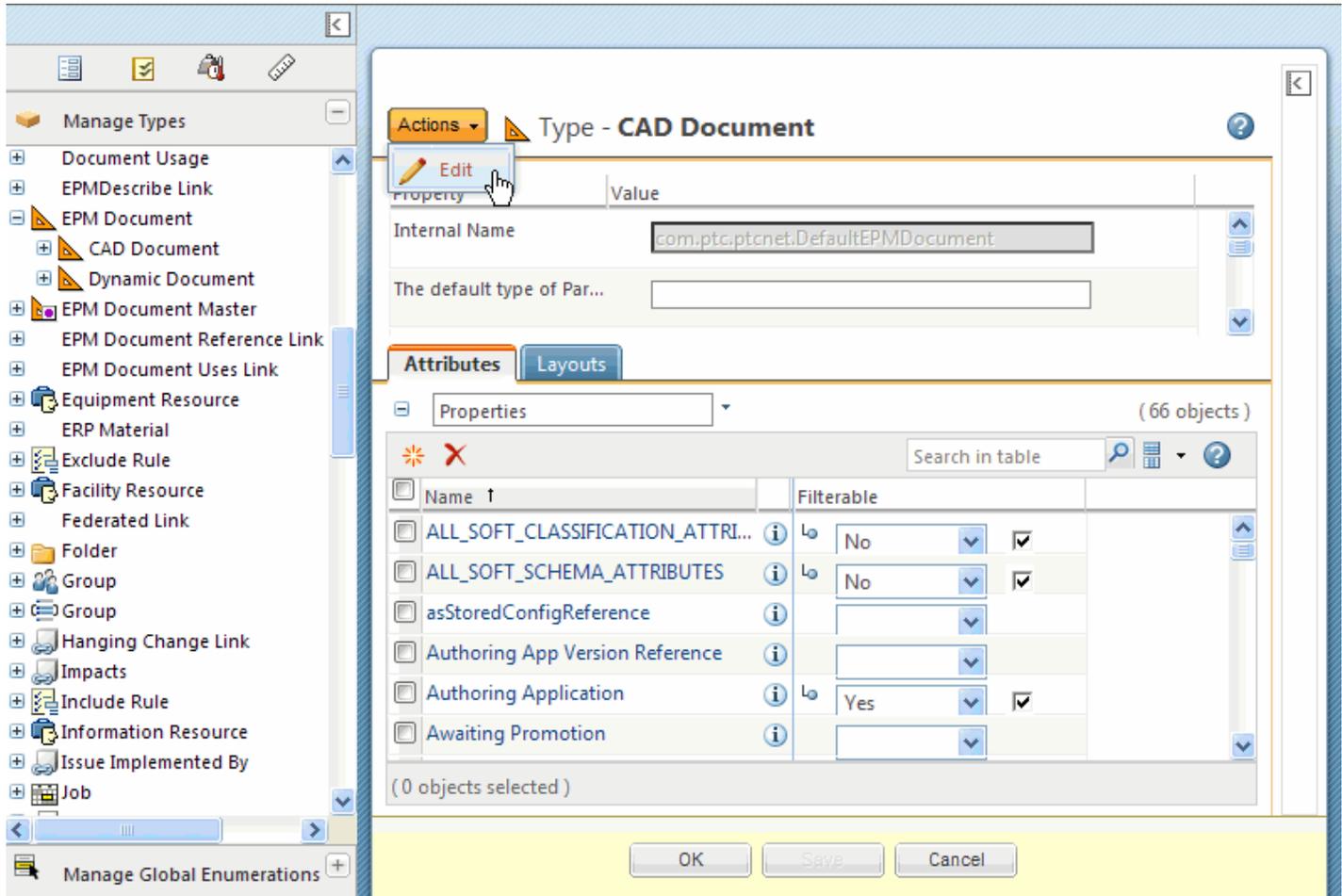
**Note**

*NX does not support mixed case parameters, and therefore should create parameters in all uppercase.*

The administrator sets up an attribute on the authoring application side in the **Properties** window. Then, in Windchill, under **Site > Utilities > Type and Attribute Management**, the administrator maps this attribute. Use the following process to map attributes and parameters:

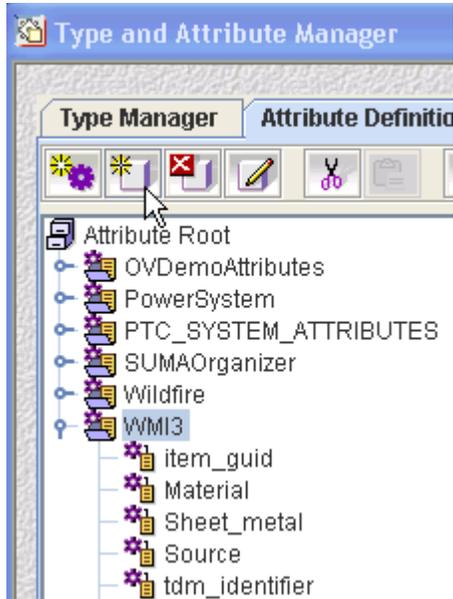
1. Check out **Workgroup Manager CAD Document** soft type using the next steps.
  - a. In the **Type and Attribute Management** utility, expand the **EPM Document** soft type.
  - b. Highlight **Workgroup Manager CAD Document**.

- c. Select the edit icon. The **Workgroup Manager CAD Document** soft type is checked out, as shown in the next graphic.



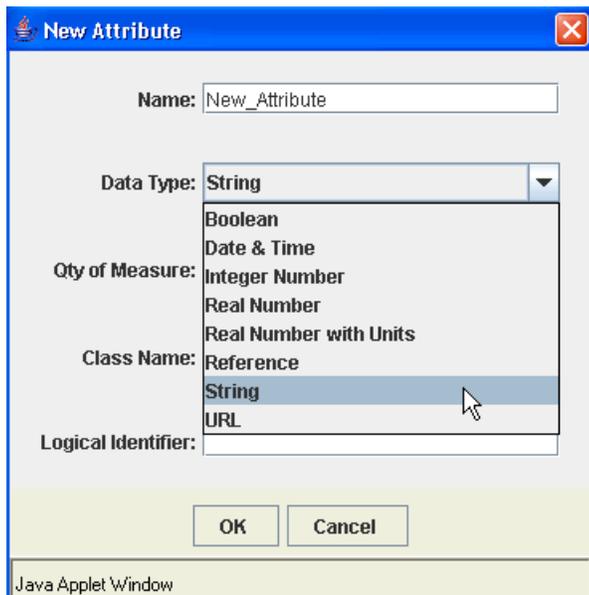
2. You can choose to create a new attribute at this point. Use the next steps to create a new attribute.

- a. On the **Attribute Definition Manager** tab, select **New Attribute** as shown in the next graphic.

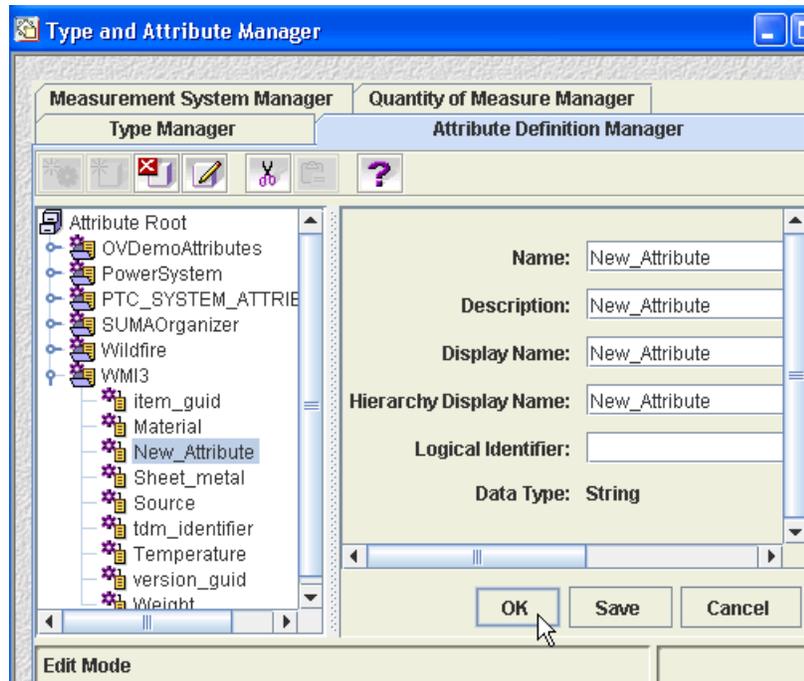


The **New Attribute** window opens.

- b. In the **New Attribute** window, type the new attribute's name in the **Name** text box and select a data type from the **Data Type** pull-down list as shown in the next graphic.

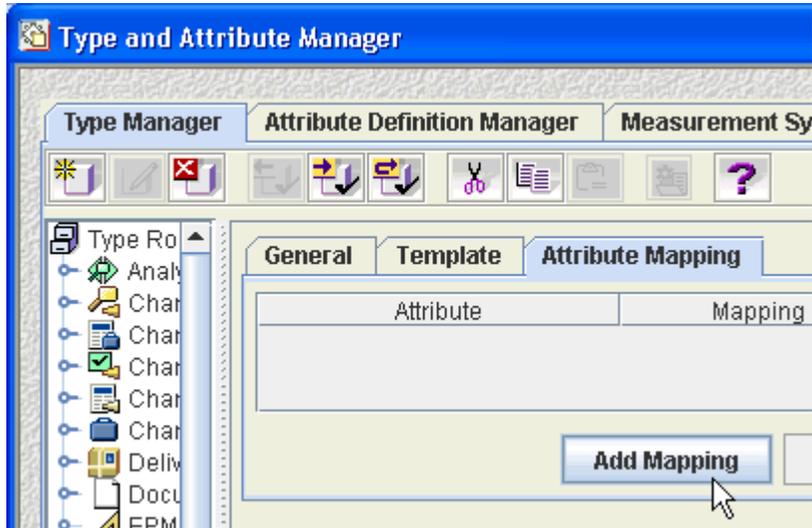


- c. Select **OK** as shown in the next graphic.



3. Map attributes. Because parameter names in the CAD file do not always match the Windchill-defined attribute names, attribute mapping provides the ability to define the mappings between the names of Windchill Workgroup Manager for Autodesk Inventor Windchill Workgroup Manager for SolidWorks parameters to the names of Windchill attributes. In order to map the CAD parameter to the Windchill attribute, use the following procedure.
- a. Navigate to the **Attribute Mapping** tab in the **Type and Attribute Management** utility.

- b. Select **Add Mapping** as shown in the next graphic.



The **Select Attribute** window appears, showing an expandable list of available attributes.

- c. Select the attribute you want to add and click **Select**. The window closes, and the **Attribute Mapping** tab shows the required attribute.
- d. Select an authoring application from the **Mapping Context** pull-down list.
- e. Type the CAD parameter name to which you are mapping in **Mapping** field of the corresponding Windchill attribute.

### Note

*Block names are supported in attribute mapping. The following format is suggested:*

*<block name><separator><attribute name> (The only <separator> character allowed is the /)*

*For example:*

*TitleBlock/Name*

4. Check in the **Workgroup Manager CAD Document**. The new attribute is added to the **Attribute Mapping** window.

To complete mapping this parameter in Windchill Workgroup Manager for Autodesk Inventor Windchill Workgroup Manager for SolidWorks Windchill Workgroup Manager for NXCreo Elements/Direct Drafting, use the following procedure.

1. Open Windchill Workgroup Manager for AutoCAD Windchill Workgroup Manager for CATIA V5 Windchill Workgroup Manager for NX and navigate

- 
- to **File > Properties > CustomFile > Drawing Properties > CustomFile > iPropertiesFile > Properties > AttributesFile > Properties > Product**.
2. Create a parameter with the exact same name as the one you created in Windchill.
  3. Select **Add**. The **Define other propertiesAdd Custom Property** window opens.
  4. Type in the exact same name as you did in Windchill, and give it the same value.
  5. Select **OK**.
  6. The parameter and its corresponding value is added.
1. Set the Site or Organization level preference, **Use Current Attribute Mapping**, to enumerate parameters for the current attribute definition mapping.
  2. Open Creo Elements/Direct Drafting and create a new drawing.
  3. Save the new drawing to the workspace.
  4. Select the new drawing in the workspace and then select **Edit ▶ Attributes**.
  5. In the **Edit Attributes** window, add the required attribute column and insert the value.
  6. Complete the Edit Attributes action, and then upload the drawing to Windchill.

When the drawing is opened in Creo Elements/Direct Drafting, the value added in step 5 will be reflected with mapping. This attribute is now recognized in Windchill with Windchill Attribute Name and in Creo Elements/Direct Drafting with Mapped Attribute Name.

This attribute is now recognized in Windchill with the Windchill attribute name when you upload or check in a CAD part or drawing.

## Implicit Parameter-to-Attribute Mapping

Implicit mapping occurs when the authoring application parameter uses the exact syntax as its Windchill attribute. This makes the parameter publish automatically in the authoring application.

For example, a CAD document has a set of soft attributes that are published to the authoring application. From this set, use the soft attribute **Name** and map it to the **NAME** of the parameter in the authoring application. The soft-types that used in this case are:

- EPMDocument
- EPMMemberLink
- EPMReferenceLink

---

If **Name** is the same by a case-sensitive evaluation, the two are implicitly mapped. Other parameters are not be added to the CAD document, unless they are explicitly mapped. For example, in the authoring application, if the administrator has created a soft attribute with the name, **Cost**, then the user must also create the soft attribute with the name **Cost** so it is implicitly mapped. If the authoring application user creates **COST**, it will not be mapped.

If the authoring application does not support case-sensitive parameter names, the name is treated in all uppercase characters. For example, if the authoring application treats **Cost** and **COST** the same, then Windchill receives and handles this attribute as **COST**. The Windchill soft attribute name must also be **COST**.

### **Note**

*Although the user may create a authoring application parameter called **Cost**, it is passed to Windchill as **COST**. Therefore, ensure the authoring application parameter name is in all uppercase so that it is implicitly mapped in Windchill. Other properties are not added to the CAD document, unless they are explicitly mapped.*

If a mapped parameter is inconsistent with its corresponding soft attribute, the following occurs.

- **Type**  
If the two are not the same type then a warning is given to the user upon upload; but upload continues. The value is not pushed to the CAD document.
- **Units**  
If the quantity of measure is different between the attribute and the parameter, then the CAD document receives an error upon upload. The user is notified to check in the CAD document.

## **Explicit Property-to-Attribute Mapping**

Explicit mapping occurs when you use Windchill to match a parameter with a soft attribute. This is done in the **Type and Attribute Management** utility. The **Type and Attribute Management** utility has an **Attribute Mapping** tab where you map the soft attributes. You can add, modify, and remove the mappings. If you change the mappings, it does not affect the existing CAD documents.

Similar to implicitly mapped parameters, the values from the CAD application are added to the CAD document upon upload, and the values of soft attributes set in Windchill are reflected in the CAD file upon download.

---

## Resolving Parameter/Attribute Type Conflicts

AutoCADNX doesn't have attribute types, and only allows the String type. Windchill Workgroup Manager requires that attribute and parameter types match, so you must ensure that this is so, or the **Conflict Management** utility displays this error. Once reset to **String**, the conflict resolves. For more information on the **Conflict Manager** utility, refer to [Conflict Management on page 70](#).

## Modifying the Properties Page

To configure a custom part-specific properties page, you have to create a properties page or template processor. For details on how to do this refer to the *Customizing the HTML Client* section in the *Windchill Customization Guide* located at [PTC Reference Documents](#) Web site.

## Modifying the HTML Client Item Selection Page

To enable recognition of custom parts as a sub-class of WTPart and not just the supported type in the HTML client item selection page's default implementation, you must add support for the custom part in the configured `wt.query.SearchAttributeListDelegate`.

In addition, you must modify the CAD application HTML files that use the item selection page, and use the `xconfmanager` to modify or override the type list id entries in `com\ptc\windchill\cadx\propfiles\picker.properties`.

### Note

*For `wt.query.SearchAttributeList`, which is the default configured search attribute list, the type identifier is referred to as the query value.*

## Replacing WTPart

If you want your site to use only custom part and not WTParts, use the following procedure.

1. Add custom part support to HTML Search.
2. In `picker.properties`, use the `xconfmanager` to change the type list entries that contain a type identifier for WTPart to the custom part type identifier you created in Step 1.
3. Restart the Method server.

---

## Supporting WTPart and Custom Part

If your site plans to use both WTParts and custom parts, use the following procedure.

1. Add custom part support to HTML Search.
2. In `picker.properties`, use the `xconfmanager` to add to the type list entries that contain a type identifier for WTPart the custom part type identifier you created in step 1.
3. To add an **All** type list entry for a type list, add an entry with the **ALL** type identifier used by the configured search attribute list.
4. Restart the Method server.

## Enabling the Recording and Display of Rename and Location History

When a part is renamed, Windchill records the data in a table that can be displayed to the user. The administrator configures this display of Rename history and Location history by modifying entries in the `configAudit.xml` file, located at `codebase\registry\auditing\configAudit.xml` file.

Be aware that enabling auditing can adversely affect performance and consequently choose to audit the most significant events. To enable display of the **Rename History and Location History** links on the information page, put the following entries in `configAudit.xml` and restart the Method server.

```
<?xml version="1.0" encoding="UTF-8"?>
<!DOCTYPE EventConfiguration SYSTEM "configAudit.dtd">
<EventConfiguration enabled="true">
<KeyEntry eventKey="*/wt.folder.FolderServiceEvent
/POST_CHANGE_FOLDER" multiObject="true"/>
<KeyEntry eventKey="*/wt.epm.EPMDocumentManagerEvent
/PRE_CHANGE_CAD_NAME" multiObject="true"/>
<ConfigEntry class="wt.epm.EPMDocumentMaster"
enabled="true">
<RenameEventEntry eventType="PRE_CHANGE_IDENTITY"
enabled="true"
handler="wt.audit.configaudit.RenameEventRecorder"/>
```

---

```
<KeyedEventEntry eventKey="*/wt.epm.EPMDocumentManagerEvent
/PRE_CHANGE_CAD_NAME"
enabled="true"
handler="wt.audit.configaudit.CADNameChangeAuditEventRecorder"/>
</ConfigEntry>
<ConfigEntry class="wt.epm.EPMDocument" enabled="true">
  <KeyedEventEntry eventKey="*/wt.folder.FolderServiceEvent
/POST_CHANGE_FOLDER"
enabled="true"
handler="wt.audit.configaudit.LocationChangeAuditEventRecorder"/>
</ConfigEntry> </EventConfiguration>
```

### Note

*The default configAudit.xml file comes with EventConfiguration enabled="false", but all lower level sections enabled. Therefore, setting EventConfiguration enabled="true" enables all auditing for the application as a whole. To selectively audit events, you must customize which auditing events are enabled.*

## Generation of Viewables

Server-side generation of viewables is enabled by setting up the Windchill Visualization Service. For information about setting up Windchill Visualization Service, see the *Windchill Installation and Configuration Guide - Visualization Services* located on [PTC Reference Documents](#).

## Managing Secondary Content

The administrator specifies files that the system automatically attaches to a supported model's CAD document. The functionality assumes that the name of these related files is not random and is based on of the main model's CAD name. The following file specifications are defined:

- Extension of the file to be automatically attached
- Existing category for the file
- New category
  - Behavior for downloading attachments of this type

- 
- Validity of the attachment after the primary content is modified is also defined by the attachment
  - A pattern-matching string to find the name of the additional content. For example, if CAD Name = \*.prt, then attach \*.JT, or if CADName = \*.prt then attach \*\_PRT.ans. . Valid wildcard characters are: \*, and &.

By default, no automatic attachment settings are defined. These can only be set by any user. Automatic attachment preference can be set at all levels (User/Context/Organization/Site) from **Home > Utilities > Preference Management** and the `wgmclient.ini` file. Use the following process to define the automatic attachment settings:

1. In the **Preference Management** utility, configure the Windchill Workgroup Manager autoattach preference, **wgm.autoattach**. This allows the content that is uploaded to Windchill to be configured as additional content of a CAD model.

#### **Note**

*The parameters are case sensitive. For example: to upload a PDF attachment for Solidworks the value would be*

```
wgm.autoattach=*.sldprt,*pdf,GENERAL
```

2. Set the `upload.autoattach.search` path. This specifies the paths on disk, separated by ; (semi-colon), search for attachments that need to be automatically added to a CAD document upon upload.

For more information on preferences, refer to Windchill Workgroup Manager Client and Server Preferences.

## Specifying Whether or Not to Outdate Secondary Content

The Windchill preference system (under **Preference Management ▶ Windchill Workgroup Manager ▶ CAD Data Management ▶ Server ▶ Content Handling ▶ Mark Out Of Date**) lists the secondary content categories for your site. For each category of file type, setting the mark-out-of-date preference to **Yes** (default is **No**) means that upon executing various PDM operations, any secondary content of the specified category type is marked as outdated (for example, in the **Attachments** table on the CAD document information page). Setting the value to **No** means that the category does not become outdated, as it is carried forward with the CAD document.

---

Users can manually override conflicts caused by outdated files (for example, during a Check In attempt) by:

- Terminating **Check In** and manually updating the attachment, thus removing the **Outdated** flag.
- Removing the attachment from the CAD document.
- Resetting the status (removing the **Outdated** flag).
- Overriding the conflict and checking the CAD document in **as is**, with an **Outdated** status.

## Setting the Automatically Download Secondary Content Preference

The Windchill preference system (under **Preference Management** ▶ **Windchill Workgroup Manager** ▶ **CAD Data Management** ▶ **Server** ▶ **Content Handling** ▶ **Download**) lists the secondary content categories for your site. For each category of file type, setting the download preference to **true** (default is **false**) means that upon any downloading of a CAD document, any secondary content of the specified category type is downloaded automatically along with the primary content. Setting the value to **false** specifies that the secondary content is not downloaded automatically with the primary content of CAD document.

## Clean-up of Event Management

To avoid possible performance issues resulting from an accumulation of a large number of event records in the **Event Management** utility, add the following site-wide property to wt.properties: com.ptc.core.task.purgeTasksOlderThanDays=5. Events older than the specified number of days are automatically purged from the **Event Management** utility.

## Configuring the Workspace User Interface

This section describes how to configure the workspace user interface including tables, rows and automatic scrolling.

### Administering Table Views

The display of information in many tables can be set by a user and administrator using the **Table View Manager**. Specific views for tables can be created or edited in the **Create Table View** window, described in the server online help. Administrators

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have the option of making a table view available to all users by selecting the **Publish to all users** check box on the first step, **Set Name**, of the **Create Table View** window.

## Configuring Table Scrollbar Display

By default, scrollbars controlling the vertical scrolling for tables are positioned on the right side of tables. Occasionally, the combined number and width of table columns may make it necessary to use the horizontal scrollbar to access the vertical scrollbar. A site-wide property allows you to change the location of table scrollbars from the right to the left side to reduce the need for horizontal scrolling. To position scrollbars on the left side of tables, configure the following property in your site.xconf:

```
<Property name="com.ptc.core.htmlcomp.jstable.positionScrollbarLeftside"
  overridable="true" targetFile="codebase/wt.properties" value="true"/>
```

### Note

*No property is necessary for default behavior. This property works for common table components on the Windows platform (only) except for Event Management utility tables.*

For more information on the **Event Management** utility, refer to Conflict Management.

## Configuring the Display of Number of Workspace Rows

The property **com.ptc.windchill.uwgm.cadx.ws.sizeToWindow** controls the number of rows displayed in the workspace object list. Set to **Yes** (default), the number of rows shown is based on the height of the window. To display a fixed number of rows (for example, 10 for Windows), set the property to **No**, as follows:

```
<Property name="com.ptc.windchill.uwgm.cadx.ws.sizeToWindow" overridable="true"
  targetFile="codebase/wt.properties" value="false"/>
```

## Configuring Automatic Scrolling in the Workspace

The property **com.ptc.windchill.uwgm.cadx.ws.scrollToTable** controls whether or not the workspace page appears automatically scrolled to the beginning of the item list table or appears scrolled to the top of the page. Set to **true** (default), the page automatically scrolls to show the workspace item list.

---

To disable automatic scrolling, set the property to **false**, as follows.

```
<Property name="com.ptc.windchill.uwgm.cadx.ws.scrollToTable" overridable="true"
targetFile="codebase/wt.properties" value="false"/>
```

## Performance Considerations

The following sections describe how you can set up your system for maximum performance.

### Setting the Method Server HEAP Size

It is recommended that the default Java heap size for each method server be set to 512MB in order to cope with large CAD application data sets that are common to the products developed by CAD users.

### Data Compression

The meta data compression option is intended to improve the upload and download performance of Windchill Workgroup Manager for users accessing Windchill across a lower bandwidth network. This feature substantially improves the performance of upload and download operations for large family of objects.

### CAD Application Settings

In Windchill Workgroup Manager, compression is controlled by a

```
config.pro
setting
(
    dm_http_compression_level
)
```

as follows.

```
dm_http_compression_level <an integer between 0 and 9 --
    0 for no compression, 9 for maximum compression>
```

---

## Maximizing the Oracle Server/Windchill Server Connection

Due to the large number of objects and CAD documents involved in database transactions, it is highly recommended that the connection between the Oracle server and the Windchill method server machines is both low latency and high bandwidth.

### Note

*Bulk HTTP data transfer using Apache on Windows 2000 can be restricted by Apache's default send buffer size. It is recommended to set the property **SendBufferSize=16384** in `httpd.conf` to significantly improve throughput over high latency, low bandwidth WANs.*

## System Configuration Recommendations

The following section discusses recommendations for configuring the Windchill system.

### Running Multiple Servers

It is recommended that Windchill and Windchill PDMLink be configured to run multiple method servers on servers with multiple CPUs and to run Oracle on a second server, especially when there is a single-CPU server running Windchill.

### Using External File Vaulting

Content files persisted in external vaults are retrieved faster than content files stored in Oracle as binary large objects (BLOBS). Although the use of file vaults can add complexity to backup and recovery operations, vault management can be simplified by using the `xconfmanager` to set the **wt.property wt.fv.forceContentToVault = true**. This causes all content to vault to the `DefaultCacheVault`, keeping it out of Oracle BLOBS, without requiring creation of a vaulting rule. In the event that multiple vaults must be implemented at your site, a vaulting rule applied to the User domain (where CAD documents are created) can direct content to vault appropriately.

### Note

*Following a custom checkin, the user is able to see CAD documents to be vaulted only in the default cache folder until an explicit revaulting action (executed through the replication schedule set by the administrator) is executed.*

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## Using Content Replication

Content replication provides the means to copy selected content files from a master server to remotely located replica servers for faster access by users at the remote site, thereby significantly improving access time. The files at the replica site remain retrievable by users at the master site.