



Using Pro/ENGINEER® Wildfire™ with Windchill®

Windchill 9.0

June 2008

Copyright © 2007 Parametric Technology Corporation. All Rights Reserved.

User and training guides and related documentation from Parametric Technology Corporation and its subsidiary companies (collectively "PTC") is subject to the copyright laws of the United States and other countries and is provided under a license agreement that restricts copying, disclosure, and use of such documentation. PTC hereby grants to the licensed software user the right to make copies in printed form of this documentation if provided on software media, but only for internal/personal use and in accordance with the license agreement under which the applicable software is licensed. Any copy made shall include the PTC copyright notice and any other proprietary notice provided by PTC. Training materials may not be copied without the express written consent of PTC. This documentation may not be disclosed, transferred, modified, or reduced to any form, including electronic media, or transmitted or made publicly available by any means without the prior written consent of PTC and no authorization is granted to make copies for such purposes.

Information described herein is furnished for general information only, is subject to change without notice, and should not be construed as a warranty or commitment by PTC. PTC assumes no responsibility or liability for any errors or inaccuracies that may appear in this document.

The software described in this document is provided under written license agreement, contains valuable trade secrets and proprietary information, and is protected by the copyright laws of the United States and other countries. It may not be copied or distributed in any form or medium, disclosed to third parties, or used in any manner not provided for in the software licenses agreement except with written prior approval from PTC.

UNAUTHORIZED USE OF SOFTWARE OR ITS DOCUMENTATION CAN RESULT IN CIVIL DAMAGES AND CRIMINAL PROSECUTION.

For Important Copyright, Trademark, Patent, and Licensing Information: For Windchill products, select **About Windchill** at the bottom of the product page. For InterComm products, on the Help main page, click the link for Copyright 2007. For other products, select **Help > About** on the main menu for the product.

UNITED STATES GOVERNMENT RESTRICTED RIGHTS LEGEND

This document and the software described herein are Commercial Computer Documentation and Software, pursuant to FAR 12.212(a)-(b) (OCT'95) or DFARS 227.7202-1(a) and 227.7202-3(a) (JUN'95), and are provided to the US Government under a limited commercial license only. For procurements predating the above clauses, use, duplication, or disclosure by the Government is subject to the restrictions set forth in subparagraph (c)(1)(ii) of the Rights in Technical Data and Computer Software Clause at DFARS 252.227 7013 (OCT'88) or Commercial Computer Software-Restricted Rights at FAR 52.227 19(c)(1)-(2) (JUN'87), as applicable. 02202007

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

Contents

Change Record	xi
About This Guide.....	xv
Intended Audience.....	xv
Scope and Purpose	xv
Related Documentation	xvi
Technical Support.....	xvi
Documentation for PTC Products.....	xvi
Comments	xvii
Introduction to Using Pro/ENGINEER with Windchill	1-1
Guide Structure	1-2
Where to Start	1-2
What is Product Data Management?.....	1-4
How Does Windchill PDM Work?	1-4
What You Should Know About Windchill PDM	1-5
PDM Server	1-5
Workspaces and the Commonsplace.....	1-5
Versions, Revisions, and Iterations	1-7
CAD Document.....	1-7
Part	1-8
Configuration Specification	1-8
PDM at a Glance	1-8
Check Out.....	1-9
Check In.....	1-10
Upload	1-11
Add to Workspace	1-11
Update	1-12
Getting Started with the PDM System	2-1
What You Should Know Before You Start.....	2-2
Primary and Secondary Servers.....	2-2
Using the Pro/ENGINEER Browser	2-3
Connecting to the Server.....	2-4
Using the Server Registry Dialog Box	2-4

Registering a Server.....	2-7
Working with the PDM Server	2-8
Using Workspaces	2-10
Setting the Preference to Display the Workspaces Link	2-11
Workspace Management	2-12
Cache Management.....	2-19
Basics of the Workspace User Interface	2-20
Viewing Objects in Your Workspaces	2-25
Object Status.....	2-28
Viewing Object Information	2-32
Object Attributes.....	2-33
Image Information	2-33
Workspace Object Reports and Menus.....	2-34
Object Actions from the Information Page.....	2-39
Special Considerations for Working with Bundled Servers	2-39
Summary	2-40
Using the Folder Navigator	2-40
Browsing Files in the Folder Navigator.....	2-41
Setting the Primary Server Using the Folder Navigator	2-41
Changing the Workspace from the Folder Navigator	2-41
Summary	2-42
How to Get Help	2-42
Basic PDM Operations	3-1
Collecting Objects for PDM Operations	3-2
About Dependency Processing	3-2
Configuration.....	3-3
Using the Collection Tools.....	3-5
Setting an Object Location.....	3-12
Saving and Uploading Objects.....	3-14
Saving Objects	3-14
Performing a Save in Pro/ENGINEER	3-15
Creating CAD Documents with Pro/ENGINEER CAD Data	3-17
Creating Part Structures for CAD Data	3-18
Uploading Objects	3-26
Performing an Upload from Pro/ENGINEER.....	3-26
Performing an Upload from the Workspace	3-26
Checking In Objects.....	3-27
Checking In Objects from Pro/ENGINEER.....	3-28
Checking In from the Workspace User Interface.....	3-29

Checking Out Objects.....	3-32
Checking Out Objects from Pro/ENGINEER	3-33
Checking Out Objects from the Workspace.....	3-34
Undoing Check Out	3-40
Adding Objects to the Workspace	3-41
Initiating a Download from Pro/ENGINEER.....	3-41
Initiating Add to Workspace from the Workspace.....	3-42
Removing Objects from the Workspace	3-46
Importing Objects to the Workspace	3-46
Overview of Importing Objects.....	3-47
Performing an Import	3-48
Exporting Objects from the Workspace.....	3-51
Overview of Exporting Objects	3-51
Exporting Using the Basic Mode.....	3-52
Exporting Using the Advanced Mode	3-53
Keeping Workspace Objects Up-to-Date.....	3-54
Updating Workspace Objects	3-55
Refreshing the Cache	3-56
Revising Workspace Objects.....	3-59
Setting a Revision	3-60
Using the Event Manager	3-61
Event Manager Page	3-61
Event Information Page	3-63
Conflict Manager.....	3-64
Handling Objects in Windchill.....	4-1
Modifying Object Attributes (Properties).....	4-2
Editing Attributes from the Workspace	4-2
Editing Attributes from the Information Page	4-4
Setting Attribute Values	4-4
Updating Attribute Values in Pro/ENGINEER and Windchill	4-5
Renaming Objects	4-6
Setting a New Name	4-7
Deriving New Designs Using Save As.....	4-8
About Using Save As.....	4-9
Setting a View	4-18
Opening Objects in Pro/ENGINEER.....	4-18
Opening Workspace Objects from the Embedded Browser	4-18
Opening Objects from a Standalone Browser	4-19
Working with Family Tables.....	4-20

Family Table Overview.....	4-20
Family Table Structure	4-20
PDM Activities with Family Tables	4-21
Simplified Representations	4-31
CAD Document Templates and Pro/ENGINEER Start Parts	4-32
Managing Pro/ENGINEER Start Parts In Windchill PDMLink	4-33
Using Library Parts	4-34
Creating Libraries	4-35
Retrieving Components from a Library.....	4-35
Managing Incomplete Dependent Objects	4-36
Identifying Incomplete Dependents	4-37
Resolving Incomplete Dependents.....	4-37
Managing a BOM with the Product Structure and CAD Document Structure	4-39
CAD Document Structure.....	4-39
Naming and Numbering CAD Documents and Parts	4-45
Customizing and Administering Pro/ENGINEER Wildfire.....	5-1
Configuration Settings in Pro/ENGINEER	5-2
Environment Variables and Config.pro Options for Pro/ENGINEER Wildfire.....	5-2
Configuring Windchill for Interoperation with Pro/ENGINEER	5-14
Displaying the Workspace.....	5-14
Managing CAD Document and WTPart Naming and Numbering	5-14
Preferences That Affect Resolution of Incomplete Dependent Objects	5-21
Soft Typing CAD Documents	5-22
Mapping Pro/ENGINEER Parameters to Windchill Attributes	5-23
Customizing the Parameters in the Download Service	5-26
Configuring the Build Rule.....	5-28
Configuring the Initial Collection of Objects for Actions.....	5-30
Configuring Check In.....	5-33
Enabling Support for Custom Parts.....	5-36
Administering Revision.....	5-37
Customizing Auto Associate.....	5-39
Customizing the HTML Client Object Selection Page	5-42
Managing Secondary Content.....	5-49
Managing Drawing Dependents	5-49
Controlling the Display of Internal Pro/ENGINEER Relationships	5-50
Clean-up of the Event Manager	5-51
Administering Table Views	5-51
Configuring Table Scrollbar Display	5-51
Configuring the Number of Workspace Rows Displayed	5-52

Configuring Automatic Scrolling in the Workspace	5-52
System Configuration Recommendations	5-52
Running Multiple Servers.....	5-52
Using External File Vaulting.....	5-52
Using Content Replication	5-53
Performance Tuning	5-53
Setting the Method Server Max Heap Size.....	5-53
Data Compression	5-53
Maximizing the Oracle Server/Windchill Method Server Connection.....	5-54
Choosing to Display Family Object Symbols in Folders Table	5-55
Other Recommendations.....	5-55
Controlling End User Objects	5-55
Online Java Performance Guide.....	5-56
Windchill Folder Structure.....	5-56
HTTP Protocol	5-57
Windchill Preferences That Control Interaction with Pro/ENGINEER	5-57
Create and Edit.....	5-57
Display	5-58
EPM Service Preferences.....	5-58
Revise.....	5-60
Save As	5-61
Windchill Workgroup Manager.....	5-63
Quick Reference for Menus, Icons, and Symbols	A-1
Glossary	
Index	

Change Record

The following table explains the changes made to this document from the previous release of Windchill:

Table 1 Changes for Windchill 9.0 M040

Chapter	Description
Chapter 2, Getting Started with the PDM System	Updated the section: Comparing the Content of Objects to add information about using ProductView ECAD Compare

Table 2 Changes for Windchill 9.0 M030

Chapter	Description
Chapter 2, Getting Started with the PDM System	Updated the section: Workspace Menus to call out menu options that are not available for Windchill Workgroup Manager.
Chapter 3, Basic PDM Operations	Updated the section: Saving Objects regarding file format.
Chapter 5, Customizing and Administering Pro/ENGINEER Wildfire	<ul style="list-style-type: none">Updated the section: Managing Secondary Content.Updated the section: Controlling the Display of Internal Pro/ENGINEER Relationships

Table 3 Changes for Windchill 9.0

Chapter	Description
Getting Started with the PDM System	<ul style="list-style-type: none">• Removed the detailed section on creating table views (now Windchill-wide functionality)• Updated sections on workspace functionality• Added a new section: Finding Objects in the Workspace
Basic PDM Operations	<ul style="list-style-type: none">• Consolidated collection and configuration information (configuration was discussed in chapter 4)• Updated discussions of PDM actions for common design and other improvements• Added a new section: Importing Objects to the Workspace• Added a new section: Exporting Objects from the Workspace
Handling Objects in Windchill	<ul style="list-style-type: none">• Added a new section: Using Workspace Save As• Moved topics about Configuration to Chapter 3• Updated the section on Model Structure Report and added a section on Editing Uses Link Attributes.• Added information about editing Family Table attributes in Windchill

Chapter	Description
Customizing and Administering Pro/ENGINEER Wildfire	<ul style="list-style-type: none"> • Moved chapter here from the Windchill System Administrator's Guide. • Updated config.pro options, including addition of: <ul style="list-style-type: none"> – dm_background_operations – dm_cache_limit – retrieve_data_sharing_ref_parts and removal of: <ul style="list-style-type: none"> – dm_cache_mode – dm_cache_size • Updated the section Managing CAD Document and WTPart Naming and Numbering • Updated attribute handling to explain the new Type and Attribute Manager • Updated the section Configuring the Build Service • Updated ModelCHECK section to reflect usage of the Preference Manager • Updated the section Customizing Auto Associate • Added new section, Controlling End User Objects • Removed the obsolete section Configuring the Revision of Associated Objects • Removed the obsolete section Modifying the CAD Document IBA Value • Removed the section on Enabling Display of Rename History and Location History • Removed the section on Defining the Rename Report Mail Server • Removed section on setting the Date format. Now a Site- or Org-wide setting.

About This Guide

Using Pro/ENGINEER Wildfire with Windchill is an introduction to product data management, using Pro/ENGINEER Wildfire to manage product data in Windchill -- both for basic and for more advanced functions. If you follow the content of this manual, you will see how Pro/ENGINEER Wildfire interacts with Windchill products and how you can use it to manage your product development cycle.

Intended Audience

The intended audience for this guide is broad and includes:

- New and experienced Pro/ENGINEER users with little or no PDM or Windchill software experience.
- Users who have some experience in working with product data management software, but who would like to review the basics or learn more about how best to use Windchill solutions with Pro/ENGINEER.
- Users new to Windchill but who have worked with other product data management applications. The philosophy behind Windchill closely follows the PDM system paradigm. These users will learn the specific procedures of using Pro/ENGINEER Wildfire with Windchill.
- Seasoned Windchill users who would like to learn about how Pro/ENGINEER Wildfire interacts with Windchill.
- Administrative users responsible for administering and configuring the interoperation of Pro/ENGINEER Wildfire with Windchill PDMLink and Windchill ProjectLink.

Scope and Purpose

This guide is not intended to be a complete summary of Windchill functionality. The goal of this manual is demonstrate to you how to use Pro/ENGINEER Wildfire with Windchill to achieve effective product data management.

Related Documentation

The following documentation may be helpful:

- User's guides for Windchill PDMLink, Windchill ProjectLink, or Pro/INTRALINK 9.0 and Windchill PDMLink available at the following link:

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

- Pro//ENGINEER online help
- Windchill online help

Technical Support

Contact PTC Technical Support via the PTC Web site, phone, fax, or e-mail if you encounter problems using Pro/ENGINEER Wildfire, Windchill PDMLink, Windchill ProjectLink, or the product documentation.

For complete details, refer to Contacting Technical Support in the *PTC Customer Service Guide*. This guide can be found under the Related Links section of the PTC Web site at:

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

The PTC Web site also provides a search facility for technical documentation of particular interest. To access this page, use the following URL:

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

You must have a Service Contract Number (SCN) before you can receive technical support. If you do not have an SCN, contact PTC Maintenance Department using the instructions found in your *PTC Customer Service Guide* under Contacting Your Maintenance Support Representative.

Documentation for PTC Products

You can access PTC documentation using the following resources:

- **Windchill Help Page** — Click **Help** in the header of any Windchill page to open the **Windchill Help** page, which provides you with a portal to all Windchill documentation, including:
 - A complete set of current online help topics for your products
 - Product tutorials available on the PTC Web site
 - Windchill manuals for users, administrators, and programmers

In addition, you can click **Search All Help Sources** to access the Windchill Help Center, an online knowledgebase that includes a universal index of all Windchill documentation. You can search all of the documentation at once, or

use the advanced search capability to customize your search. Once you have located a topic you want to reference later, you can bookmark that topic for quick access and even save your own comments about the topic.

- **Product CD** — All relevant PTC documentation is included on the CD set.
- **Reference Documents Web Site** — All books are available from the Reference Documents link of the PTC Web site at the following URL:

<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 Web site. For more information on SCNs, see Technical Support:

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

Comments

PTC welcomes your suggestions and comments on its documentation. You can submit your feedback through the online survey form at the following URL:

http://www.ptc.com/go/wc_pubs_feedback

1

Introduction to Using Pro/ENGINEER with Windchill

The purpose of this chapter is to introduce you to the basic concepts of Windchill product data management (PDM) systems. In this chapter, you will learn the basic vocabulary of Windchill PDM and you will become familiar with basic Windchill functions and why they are useful to you.

The chapter begins with a section that describes the overall organization of the book. In the second section, a table summarizes the contents of the chapters and suggests the audience for each.

Topic	Page
Guide Structure.....	1-2
Where to Start.....	1-2
What is Product Data Management?	1-4
PDM at a Glance	1-8

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 an introduction to product data management (PDM) in Windchill, followed by detailed explanations of performing basic PDM actions between Pro/ENGINEER Wildfire and Windchill, explanations of more advanced PDM activities, and finally, information on configuring and administering the interaction of Pro/ENGINEER and Windchill. At the end of the guide, a Glossary provides explanation of special terminology, and a Quick Reference appendix provides a handy summary of workspace menu selections, action icons, and status indicator symbols.

Where to Start

Because this guide contains information intended for various audiences -- including relatively inexperienced users, advanced users, and administrators -- this section summarizes the chapter contents and identifies the intended audience for each.

The intended audiences for the guide are described as follows:

- Novice -- Having little or no experience with PDM in Windchill
- Experienced -- Experienced with Windchill and with PDM concepts in general
- Administrator -- Responsible for administering the interaction of Pro/ENGINEER Wildfire and Windchill

The following table can be used to quickly understand for which audience the chapters in this guide are intended:

Chapter	Description	Audience
1 Introduction to Using Pro/ENGINEER with Windchill	Provides an overview of product data management (PDM) in Windchill	Novice Experienced Administrator

Chapter	Description	Audience
2 Getting Started with the PDM System	<p>Describes the environment that allows Windchill and Pro/ENGINEER to work together, including:</p> <ul style="list-style-type: none"> • How to connect Pro/ENGINEER to a Windchill PDM server • How to work with the workspace user interface and the Pro/ENGINEER browser 	<p>Novice</p> <p>Experienced</p> <p>Administrator</p>
3 Basic PDM Operations	<p>Guides you through the most commonly used PDM operations, many of which can be initiated either from the Pro/ENGINEER menus or from the workspace page in the Pro/ENGINEER browser</p>	<p>Novice</p> <p>Experienced</p>
4 Handling Objects in Windchill	<p>Describes how to perform more advanced PDM activities and explains how Windchill handles some Pro/ENGINEER objects, for example, Family Tables and Simplified Representations.</p>	<p>Novice</p> <p>Experienced</p>

Chapter	Description	Audience
5 Customizing and Administering Pro/ENGINEER Wildfire	<p>Presents customization and administration information and recommendations for using Pro/ENGINEER Wildfire integrated with Windchill PDMLink and Windchill ProjectLink.</p> <p>The topics include Pro/ENGINEER configuration information (environment variables and config.pro options) that applies to the interaction with Windchill, and Windchill server-side preferences, as well as specific information on parameter mapping, parameter customization, customizing object naming, automated part creation, supporting custom parts, and customizing the user interface. In addition, recommendations for system configuration and performance tuning are offered</p>	Administrator

What is Product Data Management?

In any company, the product design process generates a tremendous amount of data. This data can consist of specifications, drawings, CAD models and lots of other intellectual property. Additionally, to reduce the product's time-to-market, all of this data needs to be shared so that multiple designers can collaboratively work on a single product. Without the ability to control product data, you may find multiple designers each using different variations of the same data, which can result in overlapping or inconsistent designs.

This abundance of data creates a problem: how can you maintain the integrity of your product data in an environment where multiple people are working on the same set of files? This is the problem which product data management (PDM) systems are designed to solve.

How Does Windchill PDM Work?

The way a Windchill PDM system deals with this challenge is by storing master data in a secure area where its integrity can be assured and all changes to it are monitored, controlled, and recorded.

Duplicate reference copies of the master data, on the other hand, can be distributed freely to users in various departments for design, analysis, and approval. The new data is then added back into the vault. When a change is made to data, a modified copy of the data, signed and dated, is stored in the vault alongside the old data, which remains in its original form as a permanent record.

In addition to providing change control management, Windchill PDM also helps you to manage a product's release cycle as well as its configuration.

What You Should Know About Windchill PDM

The following sections present some basic Windchill PDM concepts.

PDM Server

The Windchill server contains the database where your product data is stored and managed. The Windchill server appears to the user as a collection of folders, products and libraries, or projects. For users who are familiar with Pro/INTRALINK, the PDM server represents the Commonsense browser.

Workspaces and the Commonsense

The Pro/ENGINEER Wildfire interaction with Windchill leverages four distinct storage locations: shared folders (also known as the *commonsense*), server-side workspace, client-side workspace (also known as the workspace cache), and Pro/ENGINEER session memory.

Each area serves a specific function, which can be described as follows:

- **Shared folders (the commonsense)**—These are common PDM server storage locations that are accessible to multiple users according to access privileges.
- **Server-side workspaces**—Server-side workspaces are private (user-specific) storage locations on the PDM server. Each server-side workspace may contain:
 - A set of specific object iterations that are used for read-only references.
 - The checked-out “working copies” of objects that the user has modified or intends to modify.
 - Any newly created objects that have been uploaded but not yet checked in.

Both the working copies and the new objects are stored in the Windchill database in the user's personal folder, which belongs to the workspace.

- **Client-side workspace cache**—This storage location contains local copies of the content (UGC files and meta data) for the models in the workspace that are used to improve the Pro/ENGINEER Wildfire retrieval performance. Furthermore, the local cache holds the modifications made on the client side, until they are synchronized with the server-side workspace by uploading them.
- **Pro/ENGINEER session**—This location is Pro/ENGINEER memory, which has the models retrieved from various locations.

Every time a user creates a new workspace, a new distinct storage location is created in the user's personal folder on the server (server-side workspace) and on the client (client-side workspace cache). Together, the server-side workspace and the client-side workspace cache make up the workspace.

The workspace interface is used to manage the Pro/ENGINEER users local working environment and communication between this local environment and the Windchill server. The workspace is a private area where you can track and change multiple objects and perform basic PDM operations. The workspace is presented in a table format, each row of the table represents an object that you have either downloaded or checked-out while each column represents the object's attributes or actions that can be performed on the object. Each column can be sorted or filtered independently, which allows you to organize objects in your workspace by attributes.

You can create multiple workspaces. This ability is useful if you are working on several projects at the same time. It allows you to create a workspace for each of your projects and segregate your design data by project affiliation.

Typically, you use the workspace to check out design information from the Windchill server and to check-in modified designs. However, the workspace also allows you to:

- Perform basic PDM operations such as upload, update, download, and revise.
- Display and sort active objects that you have either downloaded or checked out.
- Configure the display of new, downloaded, or checked-out objects.

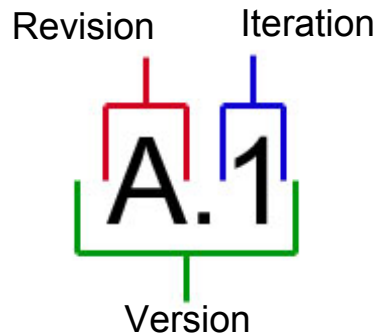
The objects are displayed in a tabular format:

- Each row represents an object.
- Each column represents an attribute of the object.
- Each column can be sorted independently.
- Control the configuration of objects you are working on.

To learn more about workspaces, refer to chapter 2, Getting Started with the PDM System.

Versions, Revisions, and Iterations

In Windchill, unique objects are assigned a *revision* and *iteration*. Together, the revision and iteration define the *version*. (for example, A.1) The default revision sequence is A,B,C,.. This sequence can be customized. The iteration sequence is always 1,2,3,...n.



An object's revision is changed using the **Revise** action. Windchill automatically increments the iteration of an object each time the object is checked in. Additionally, Windchill allows you to generate detail reports, so you can review every design modification made to an object. The example above represents revision A, iteration 1, or more succinctly, version A.1.

CAD Document

In Windchill, a document is an object containing files in application format. A *CAD document* is a revision controlled, lifecycle-managed object containing a CAD model, which is a file or a set of files containing information in a CAD application format. Although you will mostly be working with CAD documents that contain Pro/ENGINEER files, you can also use a CAD document to manage data from many different kinds of CAD systems.

CAD documents perform the following functions:

- Can store CAD-generated files (for example, 3D models, drawings, viewable images).
- Can be associated to parts (see the next section) so that a CAD document can describe the associated part.
- Can be related to other CAD documents to allow representation of the complex dependencies created and maintained by the authoring CAD system (for example, model-to-model, model-to-drawing relationships).
- Can contain both primary content (for example, a 3-D CAD model file) and secondary content (for example, a viewable representation of the 3-D CAD model file)

Part

Although in Pro/ENGINEER, you have been used to thinking of a “part” as an object created in Pro/ENGINEER, in Windchill, a *part* is an information object with an identification number, representing a physical component or assembly in a manufactured product. A part will have one or more versions capturing how that part has been modified over time. Part versions are related to all the product definition data that describes them (documents).

Unlike a document, a part never has file content. In Windchill, a part can exist without a CAD model, and can be related to other parts or documents. When a CAD document is related to a part, the CAD document is said to describe the part.

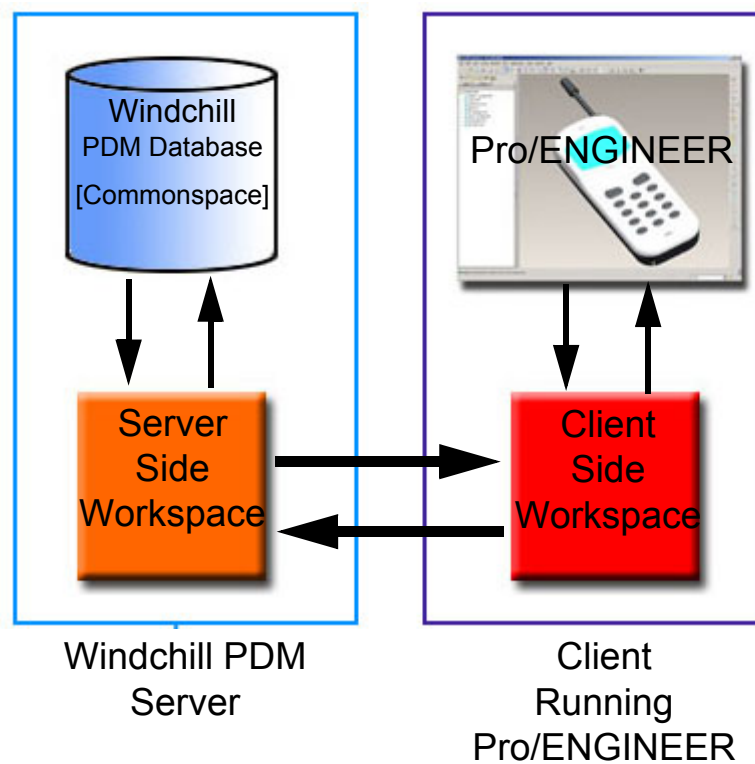
Configuration Specification

In configuration management, a *configuration specification* is a set of parameters used to filter out variability in a dynamic product structure in order to produce a static bill of materials that represents a specific product instance. A configuration specification must always be used to navigate a structure of parts or CAD documents, since it is the configuration specification that determines the specific versions of the individual objects that is returned to the user. The types of configuration specification supported by CAD documents include “Latest” (by life cycle state) and “Baseline”. Part structures also allow effectivity as a part of a configuration specification. To learn more about configuration specifications refer to chapter 4, *Handling Objects in Windchill*.

PDM at a Glance

The following section explains basic PDM functions in Windchill. Additionally, for each function an illustration has been provided to show the data transactions between Pro/ENGINEER, the workspace, and the Windchill PDM server.

The PDM system consists of two major components: the Windchill PDM server and the client. The Windchill PDM server is the machine that houses the Windchill PDM database (the commonspace) as well as the server-side workspace. The client machine is the machine on which you have Pro/ENGINEER running and where the client-side workspace is located.



As mentioned above, each workspace has two sides, one exists on the Windchill PDM server and one on the local machine. Therefore, for the purposes of this chapter, the PDM system is represented by the following graphics:

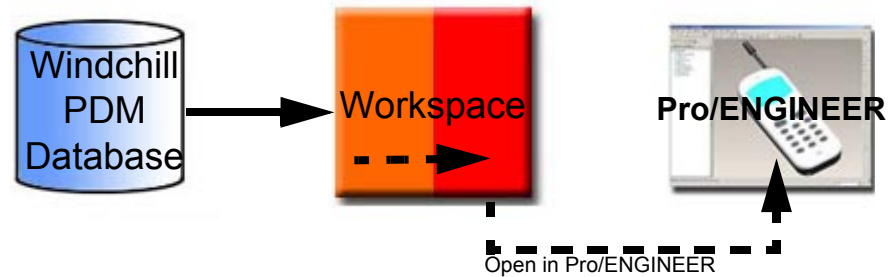


Check Out

Windchill allows designers to check out the latest version (and in some cases, an earlier version) of an object from the server. Before modifying an object, you must check it out. The process of checkout accomplishes the following:

- Creates a copy of the object in the commonspace of the server and marks it as checked-out to signal other users that you are modifying the object.
- Creates a modifiable working copy of the object in your workspace that holds all your changes until you check the object back in.

- Locks the object in the commonspace, preventing other users from checking in any changes to this object until you release the object by either checking it back in or undoing the checkout.



Note: During a checkout operation that uses the option of copying the object content to the workspace, data is transferred from the Windchill database to the workspace. Optionally, if you choose to open the object from the workspace, the data is then transferred from the workspace to Pro/ENGINEER.

To learn more about the **Checkout** action, refer to chapter 3, Basic PDM Operations.

Check In

When you are finished making changes to your checked-out objects, and you are ready to publish the changes back to the shared area on the server, you must check in your changes. The process of check-in accomplishes the following:

- Creates a new iteration of the checked-out object in the shared area.
- Copies the working copy of your object from your workspace to the shared area of the server.
- Releases the lock on the object in the shared area.



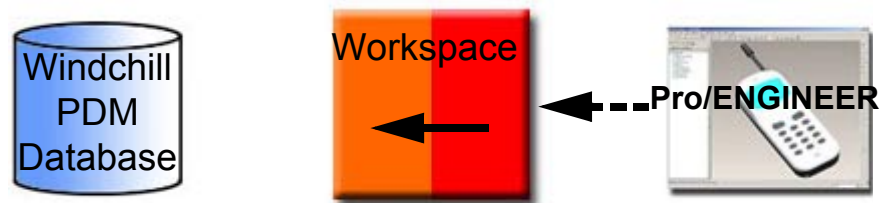
Note: During a check-in operation, data is transferred from the workspace to the Windchill PDM database. If you check in an object directly from Pro/ENGINEER, the data is transferred from Pro/ENGINEER to the workspace and then to the Windchill PDM database.

To learn more about the **Check In** action, refer to chapter 3, Basic PDM Operations.

Upload

An upload transfers Pro/ENGINEER files and any other dependencies from the local workspace cache to the server-side workspace. An upload performs the following functions:

- Transfers new and modified files from the local workspace cache to the server-side workspace on the Windchill server.
- Creates new CAD documents in your personal server-side workspace for new Pro/ENGINEER files.
- Updates the checked-out version of a CAD document with the latest modifications made in Pro/ENGINEER.



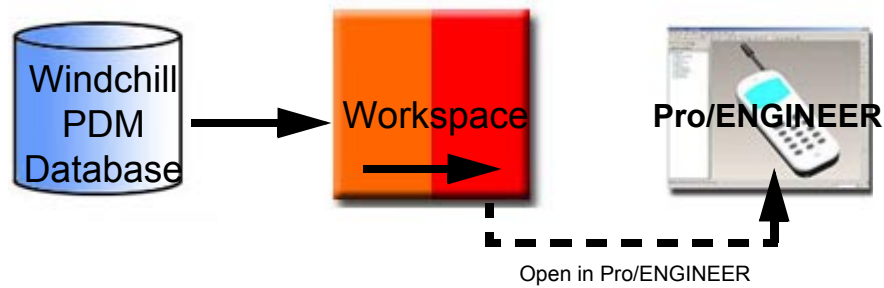
Note: During an upload operation, data is transferred from the client-side workspace to the server-side workspace. You can also initiate an upload in Pro/ENGINEER using File> Save and Upload, in which case data is moved from session to the client-side workspace and then to the server-side workspace.

To learn more about the **Upload** action, refer to chapter 3, Basic PDM Operations.

Note: The Upload action does not update the shared folders with changes made in the workspace. Therefore, other Windchill users do not have access to uploaded objects. An uploaded object must be checked in before it can be shared.

Add to Workspace

The **Add to Workspace** command creates a copy of an object on the PDM server in your active workspace. If an object is checked out by another user, you can use the **Add to Workspace** action to add the object to your workspace for reference purposes only. Because objects that are added (but not checked out) to the workspace have not been locked on the PDM server, you should not modify them, as you typically cannot check in changes made to these objects.



Note: An add to workspace operation transfers data from the PDM database to the client-side workspace. If the download option is selected, file content (as opposed to meta data only) is also copied to the workspace cache.

To learn more about the **Add to Workspace** action, refer to chapter 3, Basic PDM Operations.

Update

In most cases, you will be working with the latest copies of objects from the server. However, in some cases you can be working with objects that are downloaded to your workspace and are not checked-out. If changes occur to the object on the Windchill server, the objects in your workspace become out-of-date. When this occurs, you must *update* the objects data in your workspace with the data from your Windchill server. This process is called an update.

Another example of using update is when you have defined a more specific configuration specification for your workspace and then use the **Update** action to ensure that the data in your workspace conforms with the configuration specification of the workspace.



Note: An update operation transfers the content of a later iteration from the Windchill PDM database to the workspace.

To learn more about the **Update** action, refer to chapter 3, Basic PDM Operations.

2

Getting Started with the PDM System

In the previous chapter you learned about the basic concepts behind all product data management systems. This chapter introduces you to the basic tools that allow Windchill and Pro/ENGINEER to work together. After a brief discussion on how to connect to a Windchill PDM server with Pro/ENGINEER, you will learn about the Windchill workspace and some of the basic PDM functions that can be done with the workspace.

In this chapter you will learn:

- How to connect Pro/ENGINEER to a Windchill PDM server
- How to set up the default file location on the PDM server
- How to work with the workspace user interface
- How to use the Pro/ENGINEER browser
- How to get help when working with the PDM system

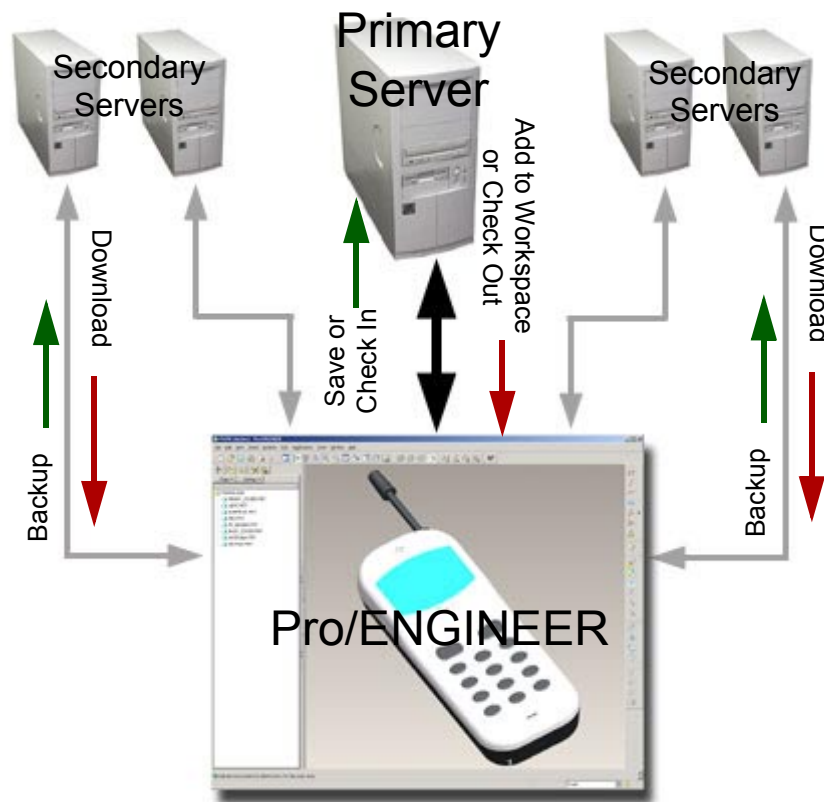
Topic	Page
What You Should Know Before You Start	2-2
Connecting to the Server	2-4
Using Workspaces	2-10
Viewing Object Information	2-32
Using the Folder Navigator	2-40

What You Should Know Before You Start

Primary and Secondary Servers

Pro/ENGINEER allows you to simultaneously connect to multiple Windchill PDM servers; however, full PDM functionality is only available with your designated *primary server*. The primary server acts as the default location for all storage and retrieval PDM functions performed with Pro/ENGINEER. Your company's Windchill server will usually be your primary server.

Any additional servers that you may register are called *secondary servers*. Secondary servers are typically used for sharing data with other users that do not have access to your primary server. For example, you might create a project on a Windchill ProjectLink server and invite suppliers so that you can share data. You would then register your project as a secondary server to give you the ability to read and write to that project in addition to your corporate Windchill server.



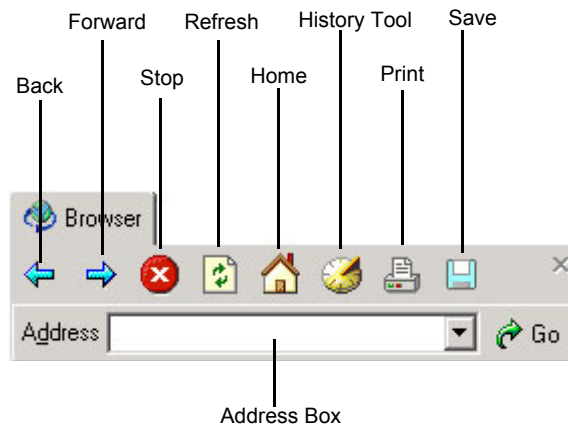
At any given time you can only have one primary server. However, you can quickly switch the primary server to be any of the secondary servers, or register a new server and set it as primary.

Tip: If you are working with multiple servers, you can use the **Server Registry** dialog box or the Folder Navigator to designate which server to use as the primary server.

Using the Pro/ENGINEER Browser

If you have used Windchill before, you will probably remember that Windchill user interface is presented as a series of HTML pages accessible through your Web browser. In order to allow you to access the full functionality of Windchill, Pro/ENGINEER Wildfire comes equipped with a built-in (embedded) Web browser.

Because most of your Windchill tasks will be performed in the Pro/ENGINEER browser, you may want to take some time to familiarize yourself with the user interface. You will find that the Pro/ENGINEER browser is very similar to Microsoft's Internet Explorer.



The following commands appear on the Pro/ENGINEER browser menu bar:

- **Back** -- Click the **Back** button to go to the previous page in the browsers history list.
- **Forward** -- Click the **Forward** button to go to the next page in the browsers history list.
- **Stop** -- Click click the **Stop** button to stop a page from loading.
- **Refresh** -- Click the **Refresh** button to reload the current page.
- **Home** -- Click the **Home** button to return to the Pro/ENGINEER home page.
- **History Tool** -- Click the **History Tool** button to view the browser history.
- **Print** -- Click the **Print** button to print the current page.

- **Save** -- Click the **Save** button to save the current page to a file.
- **Address Box** -- To go to a web page enter the URL in the **Address Box** and click **Go**.



Caution: The Pro/ENGINEER browser is the ONLY browser that supports the connection between Pro/ENGINEER and a Windchill PDM server. Accessing the Windchill PDM server in an external Internet Explorer or Mozilla browser yields no connectivity between Pro/ENGINEER and Windchill.

Summary

Pro/ENGINEER allows you to create connections to multiple Windchill PDM servers. In order to avoid confusing data sources you have to designate a primary server. A primary server is a default location for all storage and retrieval PDM functions done with Pro/ENGINEER. Any additional servers that you register are considered secondary servers. Secondary servers are used for auxiliary functions.

The Pro/ENGINEER browser supports the connectivity between Pro/ENGINEER and Windchill.

Now that you have reviewed these concepts, you are ready to learn how to start using Windchill with Pro/ENGINEER.

Connecting to the Server

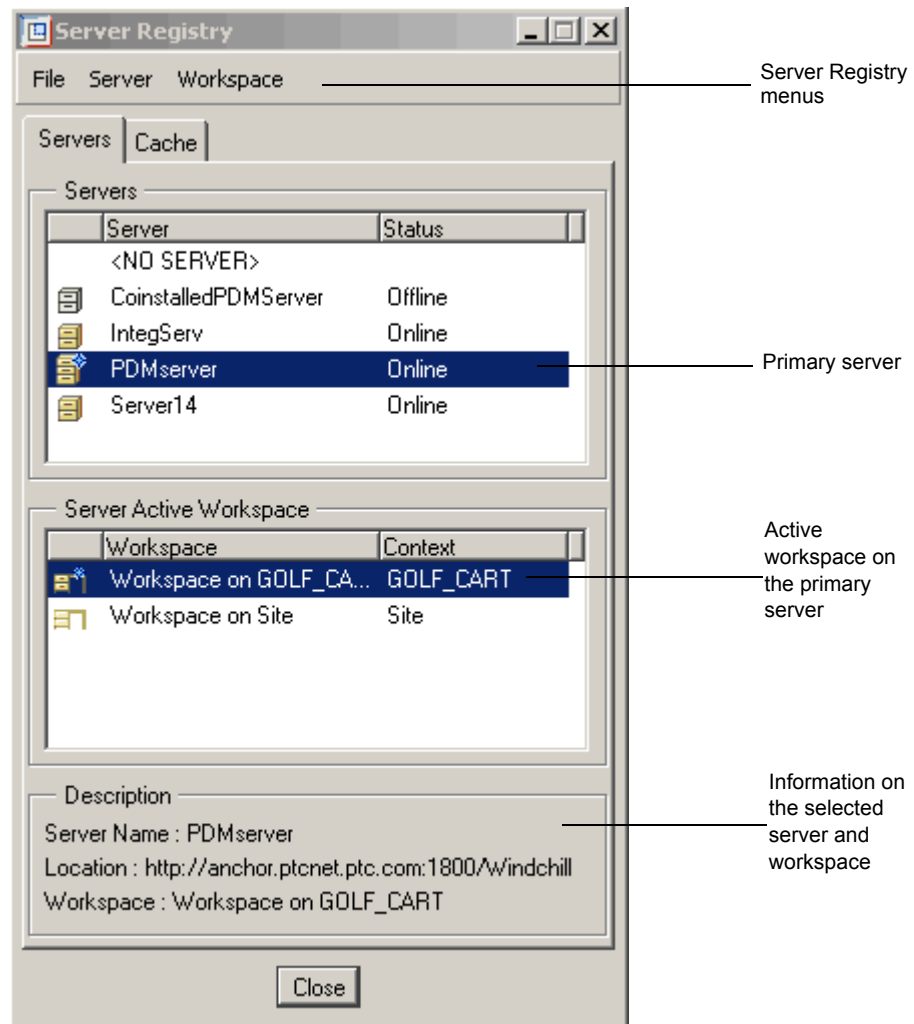
To start working with a Windchill PDM server, you must establish a connection by registering the server in Pro/ENGINEER. In this next section, you will learn about the **Server Registry** dialog box and how to use it to create and remove Windchill server connections.

Using the Server Registry Dialog Box

To register and manage servers from Pro/ENGINEER, use the **Server Registry** dialog box. To access the **Server Registry** dialog box, click **Tools > Server Registry**.

Tip: After you registered a server, you can access the **Server Registry** dialog box from the Folder Navigator by right-clicking the server or the workspace node.

The following figure shows the **Servers** tab of the **Server Registry** dialog box.



The Servers Tab

The **Servers** tab contains three sections: **Servers**, **Server Active Workspace**, and **Description**.

- The **Servers** section -- The **Servers** section lists the servers that are registered with Pro/ENGINEER. Selecting a server (or <NO SERVER>) highlights the server, indicating that it is the target for any actions selected from the **Server** menu (described in a following section). The **Status** column indicates whether a server is online or offline.
- The **Server Active Workspace** section -- The **Server Active Workspace** section lists your workspaces on the selected server. To list available workspaces on any registered server, select a server from the list in the

Servers section. Selecting a workspace highlights the workspace, indicating that it is the target for any actions selected from the **Workspace** menu (described in a following section).

- The **Description** section -- The **Description** section lists the following information for the selected server:
 - **Server Name** -- The name given the server upon registration
 - **Location** -- The URL for the server
 - **Workspace** -- Name of the active workspace on the server

With the **Servers** tab selected, the **Server Registry** menus are activated. These menus are described in the following sections.

Server Registry File Menu

Selecting **Save Settings As** on the **File** menu saves the current server registry settings to a file.

Server Registry Server Menu

The **Server** menu offers actions that apply to the server selected in the **Servers** section. The available actions are as follows:

- **Register New Server** -- Register a new server.
- **Set as Primary Server** -- Set the server as a primary server.
- **Edit** -- Edit the server , including its name, location, and active workspace.
- **Delete** -- Delete a server.
- **Work Offline** -- Allows you to work with the workspace when not connected to the server.

Server Registry Workspace Menu

The **Workspace** menu offers actions that apply to the workspace selected in the **Servers** section. The available actions are as follows:

- **New** -- Create a new workspace on the primary server and make it active.
- **Activate** --
 - **Workspace** -- Activate the workspace. This is available only when you select an inactive workspace.
 - **Workspace and Set Primary** -- Activate the workspace and set its corresponding server as a primary server. This is available only if you select an inactive workspace on a secondary server.
- **Lock Workspace** -- Lock or unlock the selected workspace.
- **Import Workspace** -- Import a portable workspace.

- **Export Workspace** -- Export a portable workspace.
- **Make Available Offline** -- Make the workspace ready for offline use.

Registering a Server

To establish a connection from your Pro/ENGINEER session to a PDM server, you must register the server using the **Server Registry** dialog box.

1. From the Pro/ENGINEER **Tools** menu, select **Server Registry**. The **Server Registry** dialog box opens.
2. On the **Server** menu, click **Register New Server**. The **Register New Server** dialog box opens.
3. In the **Name** field, specify a server name to appear in the Folder Navigator.

Tip: As a working practice, we recommend that all users in a company register the server using the same name.

4. In the **Location** field, enter the URL to the Windchill server codebase location (You can get this information from your Windchill administrator).
5. Click **Check** to validate the server location.
6. Enter the user name and password in the **Authentication** dialog box.
7. Select the workspace name.

Note: If you do not have a workspace on the server, the system automatically creates a workspace with the a default name. You can later create another workspace and set it to be the active one.

8. Click **OK**.

After you register the server, the server and the workspace are added to the Folder Navigator. They are also listed in Pro/ENGINEER dialog boxes commonly used for file operations such as **File > Open**, and **Edit > Replace**.

Tips for Registering a Server

Consider these tips:

- Windchill ProjectLink servers can also be registered directly through the Pro/ENGINEER browser. Simply open Windchill ProjectLink in the Pro/ENGINEER browser and use the project-level action **Register with Pro/ENGINEER** to automatically register the project.
- After you have registered the Windchill server, the next time you start Pro/ENGINEER, the system by default attempts to connect to the registered PDM server. If desired, you can specify whether Pro/ENGINEER should be connected to the server at startup by setting the **dm_remember_server** configuration option. The values are as follows:

- **no**—Pro/ENGINEER starts up from a local directory (no server connection).
- **yes**—(Default) Pro/ENGINEER checks the record of the last active site and attempts to connect to the server at startup. If there is insufficient information to connect to the server, Pro/ENGINEER starts from a local directory.

Working with the PDM Server

Pro/ENGINEER is designed to use multiple data servers of different types. Typically you will work with one server that will become the default location for most storage and retrieval actions. Pro/ENGINEER allows you to identify this server as a primary server and provides you with enhanced access. The advantages of having a primary server are:

- Direct save of Pro/ENGINEER files to the primary server's active workspace.
- When retrieving an object, Pro/ENGINEER always looks in the active workspace on the primary server to retrieve first.
- When saving an object, Pro/ENGINEER always looks in the active workspace on the primary server first.
- From the Pro/ENGINEER file menu, you have direct access to check out, check in, undo check out, upload, and update functions.
- The ability to check out objects on-the-fly.

Once a primary server is registered, any other server that you register is considered a secondary server. Pro/ENGINEER's user interface does not provide direct access to database actions, instead for you must store objects to the active workspace and use the workspace controls for database actions.

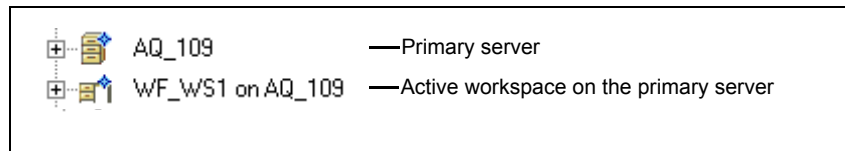
To Set the Primary Server

If you are working with multiple servers, you set any server as the primary using the **Server Registry** dialog box. You can also change the primary server any time and set a secondary server as primary.

Tip: You can set the primary server directly from the Folder Navigator. For more information, see [Setting the Primary Server Using the Folder Navigator](#).

1. Choose **Tools > Server Registry**. The **Server Registry** dialog box opens.
2. Select a server from the list of servers.
3. Select **Server > Set as Primary Server**. A blue star adorns the icon of the primary server in the server list.
4. Close the **Server Registry** dialog box. The Folder Navigator updates to show the new primary server and its active workspace.

The next figure shows how the primary server and the active workspace appear in the Folder Navigator.



To Remove a Connection to a Server

If your organization works with multiple servers, you may find that from time to time one or more of your server connections becomes out of date. In these instances, you can remove the registered server using the **Server Registry** dialog box.

Note: You cannot delete the primary server. You must first make that server secondary or set **<No Server>**; then can you remove the server connection.

1. Click **Tools > Server Registry**. The **Server Registry** dialog box opens.
2. Select a server from a list of servers on the **Servers** tab.
3. Select **Server > Delete**. The system prompts you to confirm.
4. Click **OK**. The server is removed from the **Folder Navigator**, the server list in the **Server Registry** dialog box, and from all dialog boxes that list file locations.

To Change the Default File Location to the Local Directory

If you want to work with Pro/ENGINEER locally without connecting to the servers, you can also use the **Server Registry** dialogue box to change the default location for saving files to a local directory.

1. Click **Tools > Server Registry**. The **Server Registry** dialog box opens.
2. Select **<NO SERVER>** from the list of servers.
3. Select **Server > Set as Primary Server**. A red arrow points to the **<NO SERVER>** setting in the server list.
4. Click **Close**.
5. Set the working directory using the Pro/ENGINEER menu selection **File > Set Working Directory**.

Summary

Server connections are primarily handled through the **Server Registry** dialog box. The dialog box allows you to do the following:

- Register a Windchill PDM server with Pro/ENGINEER.
- Designate a primary server.
- Remove old or out-of-date server connections.
- Designate a default file location if you are not working with a Windchill PDM server.

Now that you have learned the basics of connecting Pro/ENGINEER to a Windchill PDM server, you can start to learn about the workspace.

Using Workspaces

The Windchill PDM server provides you with a private area 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 Pro/ENGINEER user interface. The workspace enables designers to operate independently while recording and tracking concurrent activities to assist in product design decisions.

The workspace is primarily used to view and modify objects in the user's local cache. As a PDM user, you typically check out or add design information from the PDM server to your workspace to modify or view it. Checkout locks the object in the server commonspace so that no one but you can modify the design. When you are finished modifying a design and you want to share the modification with other users, you must perform a check-in operation. The check-in operation uploads your modified design to the Windchill PDM server and unlocks the object in the commonspace, allowing other users to access and modify the design further. The process of checking out and checking in (locking and unlocking) data helps you to maintain the integrity of your data.

It is through the workspace that the Windchill PDM server connects to Pro/ENGINEER. The Pro/ENGINEER session and the workspace are linked such that information that is modified in one application can be seen in the other application.

Tip: The PDM 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.

Embedded Workspaces

The workspace as viewed in the Pro/ENGINEER embedded browser offers full PDM functionality because it can access and reference your Pro/ENGINEER session, your local workspace cache, the server side workspace in your personal folder, and the commonspace.

Standalone Workspaces

You can also view the workspace from a standalone browser (for example, Microsoft Internet Explorer) using the server URL. In this case, however, you have access only to the server-side workspace and commonspace. Consequently, your available PDM operations are fewer and limited to meta data only transactions. For more information, please see the comparison of embedded and standalone browser functionality in the section [Embedded and Standalone Workspaces](#).

Offline Workspaces


In the event that a network or server failure occurs, you can still continue to modify the geometry of objects that are downloaded or checked out to your local workspace cache. You can also choose to work offline from the server and return to working online at a later time, or even export the contents of the local cache into a workspace file that can be imported into other Pro/ENGINEER Wildfire sessions. For more information, please see the discussion of offline workspace functionality in the section [Offline Access to the Workspace](#).

Summary

Workspaces act as the gatekeeper between Pro/ENGINEER and the Windchill PDM server. Any data sent to or received from the Windchill PDM server is done so through the workspace. Later you will learn that there are some data transfer operations can be initiated directly from Pro/ENGINEER, however even these functions use the workspace behind-the-scenes. While full PDM functionality is available through an active workspace viewed in the embedded browser, restricted functionality is offered in inactive workspaces and workspaces viewed in a standalone browser or in offline workspaces.

Setting the Preference to Display the Workspaces Link

By default, the **Workspaces** link (the link to the **My Workspaces** page on a context tab) in Windchill is unavailable prior to creating a workspace via Pro/ENGINEER, or setting a Windchill user preference to display the link. You can set the display preference using the following procedure:

1. From the **Home** tab, select the **Utilities** link, then the **Preference Manager** link. The **Preference Manager** page appears.
2. In the **Display** category find the **Workspace** preference and click the set preference icon .
3. In the **Set Preference** window, select the "Yes" radio button and click **OK** to enable the use of the workspace. The **Workspace** link now appears as an available minor tab on a context tab.

Workspace Management

In this section, you will learn the basics of workspace management. The information includes:

- Creating a workspace
- Deleting a workspace
- Changing the active workspace
- Locking and unlocking a workspace
- Embedded and standalone workspaces
- Offline workspace functionality
- Cache management

Creating a Workspace from Pro/ENGINEER


You can create multiple workspaces on the server. Use the **New** command on the **Server Registry** dialog box to create a new workspace on the primary server. The new workspace automatically becomes your active workspace.

To create a new workspace:

1. Click **Tools > Server Registry**. The **Server Registry** dialog box opens.
2. From the server list, select the server on which you want to create the new workspace.
3. Select **Workspace > New**. The **Create New Workspace** dialog box opens.
4. Enter a name for the new workspace and select a context from the **Context** list.
5. Click **OK**. The new workspace appears in the **Server Active Workspace** list and in the Folder Navigator.

Creating a Workspace from Windchill


You can also create a workspace in Windchill:

1. From the **My Workspaces** page, click the new workspace icon .
- The **New Workspace** page appears.
2. Enter a name, an optional description, and select a context for your workspace.
3. Optionally, deselect the **Activate Workspace** check box (selected by default; not available in a standalone browser) if you do not want to make the new workspace active upon creation.

4. Click **Ok**. Your new workspace is created.

Deleting a Workspace

Workspaces can be deleted from the **My Workspaces** page.

1. In the active workspace, click the **Workspaces** subnavigation on the context tab. The **My Workspaces** page with a table listing your workspaces appears.
2. Select a workspace to delete and click the delete icon  in the table header.

Rules for Deleting Workspaces

The following conditions must be met to delete a workspace

- You cannot delete the active workspace.
- If you attempt to delete a workspace with one or more checked-out objects, you are presented with an overrideable conflict with the option to undo the checkout(s).

Changing the Active Workspace

Each Windchill server requires an active workspace specified on that server. The active workspace is the target for any PDM operations that you initiate. During a Pro/ENGINEER session, you can change the active workspace using the **Workspaces** tab.

Making a Different Workspace Active

1. Click **Tools > Server Registry**. The **Server Registry** dialog box opens.
2. Select the primary or secondary server on which you want to change the active workspace, if not already selected.
3. Select a different workspace in the **Server Active Workspace** list box then select **Workspace > Activate > Workspace**.

Rules for Changing a Workspace

- If you activate a different workspace on the same server, Pro/ENGINEER erases objects in session before connecting to the new workspace.
- If you activate a different workspace on a different server, Pro/ENGINEER does not automatically erase objects in session. You can save in-session objects to the new workspace.

Note: In Windchill you can activate a workspace from the **My Workspaces** page (only if using the embedded browser).

Embedded and Standalone Workspaces

As explained earlier in this chapter, the active workspace on your primary server is the default location for most storage and retrieval actions. The active workspace viewed in the Pro/ENGINEER embedded browser offers full PDM functionality because it can access and reference your Pro/ENGINEER session, your local workspace cache, the server side workspace in your personal folder, and the commonspace. You can also view a workspace from a standalone browser (for example, Microsoft Internet Explorer or Mozilla) that is not linked to a Pro/ENGINEER session, using the server URL. In this case, however, you have access only to the server-side workspace and the commonspace. Consequently, your available PDM operations are fewer and limited to meta data only transactions. For more information, please see the comparison of embedded and standalone browser menus and toolbars in the section [Basics of the Workspace User Interface](#).

Offline Access to the Workspace

The interoperability of Pro/ENGINEER Wildfire and Windchill allows for access to workspace objects downloaded or checked out to local cache (the client-side workspace) in a Pro/ENGINEER session when there is no connection to the PDM server. This enables one to modify the geometry of Pro/ENGINEER objects when there is a poor or lost connection to the server, when you choose to work offline from the network, or when it is desirable to export and later re-import a workspace and its content (for example, to share with a partner outside the network).

Offline workspace functionality is managed in the Pro/ENGINEER Server Registry, with some relevant actions also available in the Folder Navigator and the Windchill **My Workspaces** page. The Pro/ENGINEER online help system explains in detail the procedures used for offline functionality. The following sections offer important information for working offline.

Note: Full offline workspace functionality described in this section is available with Pro/ENGINEER Wildfire 3.0 (and later) only.

Lost Connection to the Server

The system notifies you when your connection to the server is lost.

The following are important considerations for working offline from the server whether you go offline because the connection is lost, or by choice:

- PDM operations (for example, Save and Upload, Download, Check In, Check Out, Association, Revision, Create Part or CAD Document, and so on) are unavailable.
- When working offline, the workspace user interface is unavailable. You can access your workspace objects by using either of the following user interfaces:
 - The Pro/ENGINEER Wildfire **File > Open** dialog box

- The **File List** HTML page that opens in the embedded browser when you select an offline workspace in the Folder Navigator

Note: The **File > Open** dialog box includes version/iteration information for the objects.

- If the server goes offline, then Pro/ENGINEER cannot obtain an automatically generated number for a newly created CAD model, or for a model that is opened from your disk drive. You must manually specify a name for the new model as the design session continues offline.
- If you quit Pro/ENGINEER while the network connection is broken, and restart Pro/ENGINEER before the network connection is restored, you still have access to the cache. The workspace becomes an offline workspace. Click on the workspace node in the Folder Navigator to view the workspace content, or to open files for modification in Pro/ENGINEER.
- If you exceed the cache threshold, content may be removed from the cache without your knowledge. If this happens and you are forced to go offline, the object may not be retrievable because its content is not available. Therefore, you should monitor your cache threshold and set it appropriately. See the section, [Changing the Cache Size](#).
- Depending on the PDM system you use, you may require a locked license for Pro/ENGINEER.
- If the Pro/ENGINEER Wildfire client gets its license from a license server, when connectivity is lost to the license server, then Pro/ENGINEER freezes because it lost its license. If the network connection to the Windchill server for Pro/ENGINEER Wildfire is prone to failure, then you should use a locked license for Pro/ENGINEER.
- If the server-side component of the workspace has been deleted, upon reconnection to the server the workspace is restored with any objects that are in cache.
- If you have modified the server-side workspace while not being connected to the server, then, upon reconnection, the objects in the local cache are not overwritten and you can choose to synchronize the workspace with the server-side changes.
- When you attempt to modify a non-checked-out object in an offline workspace, Pro/ENGINEER presents the **Conflicts** dialog box. Click **Continue** to make modifications to that object.
- If you have created or modified CAD models while not being connected to the server, upon reconnection these meta data are communicated to the server-side workspace through the synchronization process. An upload is required to transfer content information to the server. If you perform the synchronization with **Upload** and **Download** deselected, no content information is exchanged

with the server side workspace, and CAD documents in the now online workspace display the status, 'Local Content Modified.'

- A status column (Compare Status) alerts you to discrepancies between the workspace cache and the server-side workspace. Some of the conditions reported are the following:
 - A naming conflict with an existing object; or the workspace object is new in cache.
 - The object has been removed from the (server-side) workspace.
 - The file name of the object has been changed.
- When the connection to the server is restored, you may need to resolve conflicts regarding new or modified objects. You also should synchronize your workspace with any server-side changes that have occurred while the workspace was offline.

Making the Workspace Available Offline

When you choose to make a workspace available offline, you first select the workspace in the **Server Registry** and then select **Workspace > Make Available Offline**. The **Make Workspace Available Offline** dialog box that appears enables you to specify when you would like to synchronize the workspace (either upon exiting Pro/ENGINEER or when going offline). This helps to ensure that the workspace has all the necessary content for an offline session prior to disconnecting from the PDM server.

Choosing to Work Offline

To go offline, with the server selected in the **Server Registry** select **Server > Work Offline**. In the **Synchronize Workspaces** dialog box that appears, you can multi-select additional workspaces to synchronize before going offline. In addition you can accept or clear the following actions to be performed at synchronization (This dialog box is also displayed when going online):

- **Synchronize (Sync/CS)** -- Updates the name and other model of the object in the cache with respect to the ones on the server.
- **Download** -- Downloads the content files from the PDM system to the Pro/ENGINEER workspace.

If your workspace is offline, then Download downloads the following:

- Content of objects that are added to the workspace but not downloaded.
- Content of objects that are checked out in the workspace but not downloaded.
- The master assembly for each external simplified representation in the workspace.
- The entire family table for any family table generics of instances.

- **Upload** -- Uploads the Pro/ENGINEER files and metadata to the server-side workspace and the PDM server. Upload is used to backup the content and metadata in a Pro/ENGINEER workspace. If your workspace is offline, then Upload also uploads the metadata and content of the newly created files and the changed files in the Pro/ENGINEER client workspace.

The system downloads content (ignoring the current cache threshold limit) as required for workspace objects that have been only linked to the server. The **Status** column in the **Servers** pane of the **Server Registry** now shows "Offline."

Going Online

When you are ready to work online again, you select **Server > (checkmark) Work Offline**. If the server is available, you are prompted to perform a synchronization with the server. If upload is selected, any newly saved document in the offline workspace is uploaded with an A.1 version.

Exporting and Importing Workspaces

Exporting a workspace allows you to save a password-protected, portable workspace (.pws) file to your local disc that can be transported to another system. The workspace is designated as "belonging to" the original primary server and owned by the original user; however, while imported to a second system, the detached (exported) workspace cannot interact with the original primary server.

Selecting a workspace in the **Server Registry** and then selecting **Workspace > Export Workspace** invokes a synchronization dialog box with the synchronization choices described in the previous section and displaying a default local storage location for the exported workspace (you can browse to a new location if desired). You then are asked to set a workspace password that is required to open the workspace (by non-owner) or re-import the workspace (upon return to owner).

With the password, the workspace can be imported onto a different Wildfire client (**Workspace > Import Workspace**). When opened on a second Wildfire client, the workspace and its primary server appear in the Folder Navigator indicated as Offline. After any modifications are made to workspace objects, the workspace can be exported from the second system (without the steps of synchronization/download/upload or assigning a password) and then re-imported onto the original system (requiring both the password and synchronization).

The workspace on the originating system is automatically locked at export to prevent unwanted changes. The owner can unlock the original workspace manually (for example, if the exported workspace is not going to be re-imported and s/he wants to continue using it) by selecting the workspace on the **My Workspaces** page and clicking **Unlock**, or by selecting it in the **Server Registry** and then selecting **Workspace > (checkmark) Lock Workspace**.

Locking and Unlocking a Workspace

The primary purpose of locking a workspace is to prevent unintended changes to the workspace cache when the workspace has been exported, so that when the portable workspace is imported back, there is a minimum of conflicts. Unlocking a workspace restores full functionality to a workspace after it has been locked. Typically, a workspace is locked during export and unlocked during re-import. One rationale for manually unlocking a locked workspace would be that an exported workspace is not being re-imported.

Note: Workspace lock/unlock functionality is available with Pro/ENGINEER Wildfire 3.0 (and later) only.

When you lock a workspace, the following actions are prohibited for objects in the workspace:

- Edit
- Add/Edit Attachment
- Rename

In addition, the following actions are prohibited in the Pro/ENGINEER File menu:

- Save
- Save and Upload
- Check in
- Backup – Back up to a locked workspace

From Pro/ENGINEER you can lock or unlock a workspace using the Server Registry's **Workspaces** menu

1. Select **Tools > Server Registry**. The **Server Registry** dialog box opens.
2. Select a workspace from the **Server Active Workspace** section of the **Server Registry** dialog box that you want to lock or unlock.
3. Select **Workspace > Lock Workspace**. The selected workspace is locked.

If the workspace is locked, then select **Workspace > Lock Workspace** to unlock the selected workspace. Before the workspace is unlocked, you are informed of the conflicts that may arise on unlocking the workspace.

To lock a workspace in the Windchill user interface, select the workspace on the **My Workspaces** page (when viewed in the embedded browser) and click **Lock**. To unlock a workspace, select the locked workspace and click **Unlock**.

Note: The lock behavior only applies to the workspace cache on the client, not the server side workspace. When viewing workspaces from a standalone browser, there is no concept of locked or unlocked workspaces.

Cache Management

Each registered workspace requires space on your local disk to store objects that you check out, download or create. This space is called the cache. Pro/ENGINEER provides the following tools to manage and gather information about the cache:

- View the location of the .cache folder
- View cache size, cache used and available disk space
- Change the maximum size of the cache
- Clear all cached objects from a server
- Clear all cached objects for a specific workspace

Changing the Cache Size

You can control the size of the cache by using the config.pro **dm_cache_limit** configuration option, which sets the size (in MB) of the cache allocated to the combination of all registered servers and their workspaces on the client hard disk.

Tip: If possible, set the cache size large enough to accommodate the largest anticipated data set (the downloaded content and the locally modified content prior to upload should be counted separately). A good rule of thumb is 80% of the remaining free space on the disk where Wildfire cache is located.

Note: The default value of “0” (no limit) will tend to fill up the client disk, but could boost performance by eliminating checks on cache size and purges.

Viewing Cache Information

1. Click **Tools > Server Registry**. The **Server Registry** dialog box opens.
2. Click the **Cache** tab. The **Cache** tabbed page opens revealing the following information:
 - Workspace cache size
 - Total cache used
 - Available disk space

To Clear the Cache for a Specific Workspace

1. Click **Tools > Server Registry**. The **Server Registry** dialog box opens.
2. Click the **Cache** tab. The **Cache** page opens.
3. Click **Cache Tools**. The **Cache Management** dialog box opens.
4. Click on the "+" icon next to a server in the **Location** list. The **Location** list expands revealing all of the workspaces on the selected server.

5. Select a workspace and click **Clear Cache**. The cache for the selected workspace is deleted.

Important Guidelines for Workspace Cache Usage

Attempting to connect additional sessions of Pro/ENGINEER Wildfire to the same client cache, or concurrently accessing the same workspace in multiple sessions of Pro/ENGINEER on the same or different machines, or accessing the same Wildfire cache as two or more different Windchill users, are not supported activities. Performing such activities may cause an action to fail, may return error messages, may cause Pro/ENGINEER to prematurely exit, may cause Pro/ENGINEER to become unresponsive, and may cause workspace or server display or data inconsistencies.

In order to avoid such issues, be sure to observe the following guidelines:

- Never access the same client cache and additional cache space with more than one concurrent Wildfire session. Ensure each Wildfire session is connected to a unique cache and additional cache space locations.
- Never access the same client cache and additional cache space logging in as a different user than that which initially created the cache and additional cache space.
- For those scenarios which require multiple cache locations, use the PTC_WF_ROOT and PTC_WF_CACHE environment variables to specify unique cache and cache space locations.

These guidelines apply to all Wildfire releases in combination with all Windchill releases.

Summary

Most workspace management functions (with the exception of deleting a workspace) can be performed from the **Server Registry** dialog box.

Each registered server supports multiple workspaces. The active workspace is the target for any PDM operations that you initiate. During a Pro/ENGINEER session, you can change the active workspace using the **Server Registry** dialog box.

Now that you are familiar with how to manage workspaces, you are ready to learn about the workspace user interface.

Basics of the Workspace User Interface

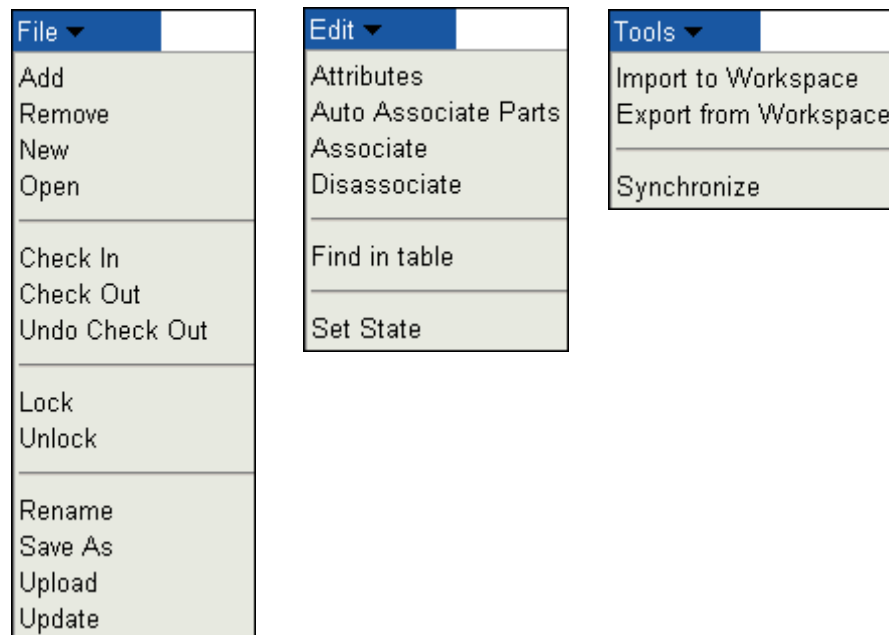
The **Workspace** page has several distinct areas: The workspace actions drop-down menu, the workspace table menus, the workspace table toolbar, and the workspace table. In this section, you will learn the layout of the workspace and what you can do from each area.

Workspace Actions Drop-down Menu

The workspace actions menu is a drop-down list that appears just below the workspace title. From this list you can access the workspace preferences user interface or the **Event Manager** for the server with which you are working. Also, if you are viewing an inactive workspace, you have the option to make it active.

Workspace Menus

The workspace menus are located at the top of the workspace **Object List** table and contain actions that you can invoke from the workspace. The menus are **File**, **Edit**, and **Tools**. The **File** menu contains actions for adding objects to the workspace or removing objects, creating new objects, managing objects, and transferring data. The **Edit** menu contains actions for modifying object attributes, associating objects, locating objects in the workspace listing, and setting object life cycle states. The **Tools** menu, available only in a primary active workspace, contains the actions to import or export workspace objects and to synchronize the client side workspace with the server. The next figures show a complete listing of the active workspace menus.



Under **File > New**, in an active workspace you can create a new CAD document, a new graphics dynamic document, a new part, or a new revision of an existing workspace object. Under **File > Open** you can find options for opening one or more CAD document model files in Pro/ENGINEER or opening a ProductView image of a single selected object.

Note: The **File** menu options **New > CAD Document** and **Open > In Pro/ENGINEER**, and the **Tools** menu options **Import to Workspace** and **Export from Workspace**, are not available for the Windchill Workgroup Manager client.

Reduced Menu Offerings in an Inactive Workspace

An inactive workspace offers a subset of the active workspace selections. Because an inactive workspace is not tightly integrated with your Pro/ENGINEER session and client cache, the following options are unavailable in the **File** menu:

- Creating a new CAD document
- Opening a model in Pro/ENGINEER
- Lock
- Unlock
- Upload

Standalone Workspace Menu Options

A workspace viewed from a standalone browser is interactive only with the Windchill database and the server-side workspace contents (no local cache). Menu options are appropriately limited to a subset of an active workspace.

The following options are not available in the **File** menu:

- Creating a new CAD document
- Lock
- Unlock
- Upload

Note: The **Open > in Pro/ENGINEER** option can only work if Pro/ENGINEER is installed on your machine.

The **Edit** menu options for a workspace viewed in a standalone browser are the same as in an active workspace.













Workspace Toolbar

Located just below the workspace menus is the workspace toolbar. The workspace toolbar contains the most commonly used workspace functions.



The workspace toolbar, like the toolbar for many action pages, is separated into two sections: the dark gray area (object-action) and the light gray area (action-object). Actions in the dark gray area work on objects selected in the workspace, whereas the actions in the light gray area do not require pre-selection. To use an action in the dark gray portion workspace toolbar, select one or more objects in the workspace table, and then click the action that you want to perform in the toolbar. If you click an action without selecting an object, you are either presented with a selection page or receive a message that an object must first be selected to effect the action. To use the functions in the light gray area, simply click on the

action to invoke it. The following table lists and describes the action icons available on the toolbar.

Icon	Action
	Remove from Workspace
	Upload
	Check In
	Check Out
	Undo Checkout
	Update
	Auto Associate
	Create New Revision
	Create Part (no selection required)
	Create CAD Document (no selection required)
	Add to Workspace (no selection required)
	Find in Table (no selection required)

Note: By default, labels describing action icons are turned off. The Windchill preference, Display > Toolbar Action Descriptions, allows the display of the labels if set to "Yes" (default is "No").

As with the workspace menus, the toolbar selections in an inactive workspace are a subset of the active workspace commands. The following commands are unavailable in an inactive workspace:

- Upload
- Update

- Auto Associate Parts
- Create CAD Document

The workspace viewed in a standalone browser does not offer the following toolbar commands:

- Upload
- Create CAD Document

Workspace Table

The workspace **Object List** table contains a listing of every object that you choose to view or modify. The workspace **Object List** table provides information on each object including status, filename, version, and state. Most importantly, the workspace **Object List** table allows you to select objects on which to perform actions.

The next figure shows how to select an object in the workspace **Object List** table and illustrates the icons in the **Actions** column.

Click the **All** check box to select all the objects in the **Workspace Table**.

Click the check box next to an object to select it.

Note: You can use the check boxes to select multiple objects (CTRL) or a range (SHIFT)


Symbols in the Status columns indicate the status of objects.		Headers of columns by which the workspace listing is sorted show as highlighted.	
		Number	File Name
<input type="checkbox"/>	<input type="checkbox"/>	0000000001.PRT	0000000001.prt
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	MAYA_A1.ASM	maya_a1.asm
<input type="checkbox"/>	<input type="checkbox"/>	MAYA_P1.PRT	maya_p1.prt
<input type="checkbox"/>	<input type="checkbox"/>	UWGM000003	

Icons in the Type column indicate the type of object listed.

The **Actions** column contains actions that you can perform on each of the objects in the **Workspace Table**.

Row Level Actions

The **Actions** column contains a small row of icons, each icon represents an action that you can perform on the object. For convenience, you can click any of the icons and perform that action directly. The following is a list of the icons and their associated actions:

-  — Click to open the object's information page.



— Click to open the object in Pro/ENGINEER.



— Click to open the object in ProductView.



— Click to begin the check-out process for the object.



— Click to begin the check-in process for the object.



— Click to upload the object.

Summary

There are several distinct areas in the workspace user interface:

- Workspace actions drop-down menu
- Workspace menus -- The workspace menus contain all actions that can be performed from the workspace.
- Workspace toolbar -- The workspace toolbar contains the most commonly used workspace functions.
- Workspace **Object List** table -- The workspace **Object List** table contains a listing of all objects that you selected to view or modify. Additionally, the workspace **Object List** table provides you with actions that are relevant to each of the objects in the table.

Viewing Objects in Your Workspaces

Complex projects generate lots of design data. Some assemblies can contain hundreds or even thousands of parts. In this next section you will learn how to filter and sort your workspaces as well as how to view information on workspace objects.

Accessing a Workspace

After you registered a server, you can access its active workspace from the Folder Navigator.

Note: When you access an active workspace of a secondary server, operations that are available to you are limited. If you want to use the workspace operations in their full extent, make that server the primary server.


To open a workspace from the Folder Navigator:

1. Locate the server and its active workspace in the Folder Navigator.
2. Click the workspace node. The contents of the workspace appear in the Pro/ENGINEER browser.

Finding Objects in the Workspace

To save time finding a particular object in a crowded workspace, the **Find in Table** dialog box allows you to locate where a string value appears in the workspace. The string must appear in the displayed set of rows and columns for the table view, but it does not need to be in the visible set of rows or columns for a scrolling table.

To locate a string in the workspace table, use the following procedure:

1. Click the find in table icon  in the workspace toolbar, or select **Edit > Find in Table**. The **Find in Table** dialog box appears.
2. Enter a string in the **Find** field. Select the **Match case** check box if you want to find only exact case matches.
3. Click **Next** to highlight the initial or a subsequent cell containing the search string, moving down the rows and left to right across the table. Optionally, click **Back** to search in the reverse direction.

The table scrolls automatically to show the highlighted cell.

4. Click **Cancel** to leave the **Find in Table** dialog box.

Sorting Workspace Objects

To simplify the view of the contents of your workspace, you can sort workspace objects by clicking the header of the column for the attribute by which you want to sort.

Customizing Object Display with the Table View Manager

Most tables in Pro/ENGINEER Wildfire, including the workspace **Object List** table, come with a default view that displays object attributes in columns and the object types specified for the default view in rows. Via the **Current View** menu's **Customize** option, you can customize your workspace to display the object types and attributes that you choose.

About Table Views

Information about the product data you work with is commonly presented in tables that list objects in rows, and information about the objects in columns. The number, content, and order of the columns and rows can strongly help or hinder your ability to work with large data sets. Therefore, the system provides an initial layout or view for each informational table, but also allows you to tailor your view of the information to best suit your objectives by selecting and ordering columns and applying filters to row content.

Each table comes with a system-generated default view that determines how columns (generally listing object attributes) are displayed and may filter the table contents to list specified objects in the rows. You can also select from alternative system-provided views that are listed in the **Current View** option list, located in the table title bar. Users and administrators can also add views of their own to the list.

While a **Current View** list is available for most Windchill tables, in the workspace the accompanying drop-down list allows you to further customize your view of the workspace by selecting one of two alternate presentations of objects in the workspace:

- As a list (default)
- As a Featured Objects list-- Featured Objects is a view designed to restrict the number of objects displayed in the workspace to the ones likely to be the most interesting you, as defined by the following rules:
 - Include objects initially selected for Add to Workspace action
 - Include objects initially selected for opening in an authoring application.
 - Include all checked-out objects
 - Include all objects modified locally or in the server-side workspace
 - Include drawings included for selected items

The important features of table views are the following:

- The system provides a default view for each table. You can select a different view to be the currently active view.
- Available views are listed, including system-provided views, Administrator-defined views that have been made available to all users, and user-defined views that are created by, and are visible to, individual users.
- The following columns cannot be removed from action page tables:
 - Object Status
 - Object Type
- When a table view is set as the active view, that view persists for the table until you change the active view.
- Table views may be deleted. When a view is deleted and that view is the current table view, the view for the table reverts to the default view.

Note: System views cannot be deleted. Only Administrators can delete administrator-created views

- New and edited table views are saved and can be reused in future sessions.

Managing Table Views

To select the active view for a table:

1. Select a table view from the **Current View** drop-down list in the title bar of the table.
2. The table updates to show columns included in the view, and data rows identified by the view criteria.

Alternatively, you can create a custom view as described in the next section.

Creating and Editing Table Views

You can edit a table view or create a new view to specify a name, applicable objects, filtering, column display, and sorting for the table. To customize a table view, select **Customize** from the **Current View** drop-down list and follow the instructions available in online help available from the **Customize View List** page.

You can also save an existing view as a new view, modify the view, and set it to be the active view of that table.

Custom table views can be created for each table type. Your saved views are listed in the **Current View** drop-down list. Each user sees system views, administrator-created views and his or her own user-created views.

Object Status

The object status information described in this section can be important for you to know when you are working with objects, either in the workspace or in various action pages. If the information is applicable to your current working situation, the system presents columns in an **Object List** table that contain symbols indicating object status. The status columns appear by default to the right of the selection check box column and to the left of the object type icon column. The columns are untitled, but their status type is revealed in a tooltip when you position your cursor at the column head.

In addition, status symbols are used in pages for database actions to indicate the intended result upon committal of the action. These action status symbols are described in the discussion of the specific action.

The default status columns displayed in the workspace are as follows:

- Share Status
- General Status
- Local Workspace Status (shown only in active workspace in embedded mode)
- Modified Status

Additional status information columns can be displayed by customizing your table view (both in the workspace and in other tables throughout Windchill). 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. You should be aware that additional status column display may affect performance. These additional status columns are as follows:




- Out of date (not the latest iteration on the given revision)
- Out of date with workspace configuration (not conforming to the workspace configuration specification)
- Compare Status

Two status columns not displayed in the workspace, but typically displayed on action pages, are the following:

- Action Status
- Status Messages









Share Status

Symbols in the **Share status** column displays symbols that indicate an object's status relative to a project or PDM, described as follows:

-  -- Shared to a project (for Windchill PDMLink only)
-  -- Shared from PDM (for Windchill ProjectLink only)
-  -- Checked-out from PDM (for Windchill ProjectLink only)

General Status

Symbols in the **General status** column displays symbols that indicate an object's general status, described as follows:

-  -- Locked (Object is read only. Only applicable to objects in the local cache).
-  -- Checked out by you
-  -- Checked out by you in another workspace (seen in the workspace and action pages accessed in the context of a workspace)
-  -- New locally
-  -- Checked out by another user
-  -- Checked-out to a project (for Windchill PDMLink only)
-  -- Another Iteration is checked out by you
-  -- Another Iteration is checked out by another user




Local Workspace Status

The **Local Workspace status** column indicates whether or not an item has been modified in the current workspace, described as follows:

-  -- Modified locally



Modified Status

Symbols in the **Modified status** column indicates whether or not an object has been modified in the current workspace, described as follows:

-  -- Modifications Need to be Uploaded
-  -- Modifications uploaded
-  -- Modified and not eligible for upload



Out of Date Status

Symbols in the (non-default) **Out of Date status** column indicates if an object is out-of-date as compared to the latest iteration on the given revision available on the server (though it may comply with the workspace configuration specification). The symbols for the two possible statuses are the same. The exact status is revealed by a tooltip that appears when you place your cursor over the symbol, as follows:

-  -- Out of date - Modified by you
-  -- Out of date - Modified by another user




Out of Date with Workspace Configuration Status

Symbols in the (non-default) **Out of Date with Workspace Configuration status** column indicates if an object does not conform to the workspace configuration. The symbols for the two possible statuses are the same. The exact status is revealed by a tooltip that appears when you place your cursor over the symbol, as follows:

-  -- Out of date with Workspace configuration - Modified by you
-  -- Out of date with Workspace configuration - Modified by another user

Compare Status

Symbols in the (non-default) **Compare status** column indicates additional status information, not contained in the other columns, as follows:

-  -- Critical Error: File name conflict
-  -- Error: File name changed
-  -- Warning: Removed from Workspace





Action Status

When performing actions on objects, the **Action status** column in an action page table displays symbols that indicate how an object will be treated upon committal of the action. These symbols are described in the help topics for the specific actions.

Status Messages

The **Status Messages** column in an action page table contains symbols indicating the type and severity of messages reporting on server interaction involving the object. In the case that multiple messages apply to the object, the symbol for the most severe message is displayed. Holding your cursor over the messaging symbol displays a tool tip that lists all applicable messages in order of severity.

The server messaging symbols and their tool-tip descriptions are as follows (in order of decreasing severity):

-  -- Critical error: <an explanatory text message continues>
-  -- Error: <an explanatory text message continues>
-  -- Warning: <an explanatory text message continues>
-  -- Information: <an explanatory text message continues>


Locked and Unlocked Objects in the Workspace

Locking a workspace object gives it a read-only status. It may be desirable to make an object read-only for any of the following reasons:





- To reduce the number of times that the Conflicts ("check-out-on-the-fly") dialog box is presented for objects that are not intended to be modified
- To give the user better visibility into changes made implicitly by Pro/ENGINEER (for example, un-planned or unexpected changes required during regeneration)
- To prevent the object from being saved.

Note: Read-only object status is a property of the local cache, whether the workspace is online, offline, or portable. There is no notion of a locked object in the server-side workspace (which is viewed in standalone browser) or the commonspace.


Viewing Object Information

You can access the information page (also referred to as the "details" or "properties" page) of an object by clicking the information action icon  in the workspace table row of the object (and other places the icon is displayed). The information page for the object contains pertinent information about the object (including a manipulable image) and provides a list of actions that you can perform on the object. The appearance and content of the information page can vary, depending on the context of the workspace from which it is invoked (for example, a product vs. a project) or on the type of object (for example, a part versus a CAD document). Similarly, the actions available for the object can vary depending on the type or status of the object.

Note: Because CAD documents in a workspace can differ from the commonspace version, Pro/ENGINEER Wildfire can present both a workspace information page and a commonspace information page. The information page called from the workspace presents the workspace-relevant information. (Although you can bring parts and end items into a workspace, they are commonspace objects only, and therefore will only have commonspace information pages.)

From the workspace, clicking the information icon  presents an information page for the workspace version of an object. Clicking the information icon  for an object in the commonspace accesses the information page for the commonspace version of the object. For clarity, the information page for a CAD document in the workspace displays the workspace icon () near the CAD document's name, while a cabinet icon () is displayed on the information page for a CAD document in the commonspace.

From a workspace information page, you can toggle to go to the information page for the commonspace version of the CAD document by clicking the go to

information about this object in the commonspace icon . You can go from a commonspace page to a workspace page only if the object is checked out by you. To do so, click the go to information about this object in the workspace icon



An object's information page contains areas for three types of information: attribute information, an image of the object (if a CAD document), and a refreshable table area which reports additional object information. In addition, an **Actions** drop-down list allows you to perform actions on the object (actions available depend on the object and its current status). The following sections present a more detailed discussion of the features of a workspace object's information page.






Object Attributes

The top portion of a CAD document's information page lists the standard item attributes and the current values for each. These attributes are described as follows:

- **Number** -- The number of the CAD document
- **Name** -- The name of the CAD document
- **File Name** -- The CAD file name that is the primary content of the CAD document
- **Version** -- The revision and iteration of the CAD document
- **State** -- The current life cycle state of the CAD document
- **Last Modified** -- The date and time that the CAD document master was most recently modified
- **Modified By** -- The name of the user that last modified the CAD document master
- **Description** -- The description note on the CAD document iteration
- **Type** -- The type of object
- **Category** -- The specific category of CAD document type
- **Location** -- The current database storage location (context and folder) of the CAD document
- **Created On** -- Displays the date and time that the CAD document master was created
- **Created By** -- Displays the name of the user that created the CAD document master
- **Authoring Application** -- Identifies the CAD tool in which the CAD File was created
- **Dependencies Complete** -- Indicates whether all dependents of the CAD document were checked in or not. (Value: True or False)

Image Information

The upper right portion of the page displays a PTC Product View Express (PVX) viewable of the object (when the information page is accessed from the embedded browser; in a standalone browser, a Windchill Visualization Services (WVS) image is used). If you are accessing the workspace view of an object's information page, this viewable displays the object as currently stored in the workspace, not as stored in the database.


Click on the icons above the image to display a wireframe , show hidden lines , no hidden lines , or shaded  (default) view of the object. Moving your mouse while holding down the middle mouse button allows you to rotate the image. Clicking  returns the image to its original position after you have rotated it.

Note: PVX installation is required to display the thumbnail image from the embedded browser. If PVX is not installed, instead of an image, a link to the PVX installation utility is provided in the viewing area. The PVX image is displayed even if the preference **Home > Utilities > Preferences > Visualization: Display thumbnails on Details pages** is not selected. That preference governs the WVS display that is used in standalone browsers.

The next figure shows a typical rendering of a Pro/ENGINEER assembly.



Workspace Object Reports and Menus

Additional information about the CAD document, for example associated objects or communication related to the object, is described in the tables available toward the bottom of the information page. If more than three tables are available, drop-down menus provide access to the tables. For more information on a table, click the help icon  on the table.

The information tables that are available depend on whether you are viewing the information page for a workspace CAD document or the information page for the CAD document as it is stored in the commonspace. Workspace information page menu commands display tables that are based solely on the objects that are currently in the workspace.

For example, a workspace **Model Structure Report** shows by default only the dependents that are in the workspace. Conversely, a newly created assembly component that has been uploaded to the workspace (but not checked in) is shown in the assembly's workspace **Model Structure Report**, but not in its commonspace **Model Structure Report**.

Access to all the available report tables is controlled by a set of menus that appear above the report table area.

The **Structure** menu is only visible on information pages for objects that are part of a structure, and has one option: to call the **Model Structure Report**.

The **General** menu can contain the options listed in the following table:

Option	Description
Attributes and Content	Presents the Content table and Attributes table
Attachments	Presents the Attachments table
Where Used	Presents the Where Used table
Notebook	Presents the Notebook table

The **Related Objects** menu can contain the options listed in the following table:

Option	Description
Action Items	Presents the Action Items table
Parts	Presents the Related Parts table
CAD Documents	<p>Presents the References and Referenced By tables.</p> <p>Note: The value "Default," when it appears in the Dependency Type or Reference Type column of either table, refers to an internal Pro/ENGINEER dependency that does not fall into one of the more specifically defined categories.</p>

Option	Description
Changes	(Not available on the workspace information page) Presents the following tables: <ul style="list-style-type: none"> • Problem Reports and Variances • Change Requests • Affected by Change Notices • Resulting from Change Notices
Contexts	Presents the Related Contexts table
Baselines	(Not available on the workspace information page) Presents the Baselines table
Family	Presents the Family Tree table
Packages	(Not available on the workspace information page) Presents the Related Packages table

The **History** menu (not available on the workspace information page) can contain the options listed in the following table:

Option	Description
Iteration	Presents the Iteration History table
Maturity	Presents the Maturity History table
Revision	Presents the Revision History table
Routing/Process History	Presents the Routing History table
Move	Presents the Move History table
Rename	Presents the Rename History table
Save As	Presents the Save As History table

The **Collaboration** menu can contain the options listed in the following table:

Option	Description
Discuss	Presents the Discussions table
Subscriptions	Presents the Subscriptions table

Bulk Item Information

Bulk items represent CAD parts that do not require solid models but must be represented in the Bill of Materials (BOM) or in PDM systems (items such as paint or sealant). They can also be items such as fasteners that are used in multiple places in an assembly. Information about the quantity and units of bulk items used in assemblies can be displayed in Windchill as explained in this section.

A parameter defined in Pro/ENGINEER, BOM_REPORT_QUANTITY, contains quantity and unit parameters, used to describe the quantity and unit of quantity for a bulk item (if no unit is explicitly assigned, the system assumes the unit to be "each"). This information is passed to Windchill upon upload (either on the EPMDocument Master or on the EPMMemberLink) and can be displayed in tables showing the Windchill **Quantity** column, for example, the **Model Structure Report**.

The following information describes the characteristics of bulk item display in Windchill:

- When you display a **Model Structure Report** for a parent of a bulk item, then the Quantity value displays the user-defined quantity (with any units) defined in Pro/ENGINEER.
- When using the part for the bulk item in a product structure, the Quantity for the part matches the value specified for this component in its assembly.
- Once the unit for a bulk item parameter is assigned, the unit can only be modified to another unit of the same measurement type. For example, if a length unit is assigned, it can only be modified to another unit of length, not area or volume.

Comparing the Content of Objects

When viewing the **Iteration History** or **Revision History** reports (only available from the commonspace information page), you can generate a report that compares the content of two selected iterations or revisions

The **Compare Content** command allows you to compare the content of two CAD document versions. The command is available from an object's Actions drop-down list (for example on a Folders or information page) and as a toolbar command on the following tables:

- Iteration History

- Revision History
- Maturity History

Note: If the CAD documents selected for comparison are from an ECAD document category, invoking the **Compare Content** command opens the ProductView ECAD Compare application.

Online help for performing the comparison is available from the ProductView ECAD Compare user interface. If only one ECAD CAD document is selected prior to invoking **Compare Content**, that CAD document is populated as the first entry in the ECAD comparison user interface and a search wizard is invoked to allow you to select a second object for comparison.

If the CAD documents selected for comparison contain Pro/ENGINEER files, invoking the Compare Content command downloads the files to be compared into Pro/ENGINEER session and generates the **Model Comparison** report.

The following procedure is used to compare the content of two selected versions of Pro/ENGINEER CAD documents (This functionality is available with Pro/ENGINEER Wildfire 3.0 or later).

When you click **Content Compare**, the **Configuration** page appears. In the **Configuration** drop-down list, you select one of the following:

- Latest
- As Stored
- Managed Baseline

Note: If Managed Baseline is selected, you are able to select one of the available managed baselines.

Upon clicking **OK**, the models are downloaded (if not already in session) in Pro/ENGINEER and a comparison is done. The **Model Comparison** report appears. The top portion of the report identifies the base model and the comparison model and the values of the following attributes for each:

- Model
- Version
- Last Updated By
- Date last updated

The **Change List** table lists changes for the base model relative to the comparison model by displaying information in the following columns:

- (change) Object Type
- (change) Object Name

- (change) Object ID
- Change Type
- Change Description

By selecting or clearing the appropriate check boxes, you can filter the report to **Show Sub-Model** objects, or to display the following change types:

- Metadata
- Geometry
- Drawing
- Cosmetic

When you click **Apply**, the report regenerates according to your filter selections (Clicking **Reset** returns the report to its default display).

Optionally, you can select **Attach as Secondary Content** to attach the report to the base model CAD document (if the base model is checked out).

Comparing Object Information

When viewing the **Iteration History** or **Revision History** reports (only available from the commonspace information page), you can generate a report that compares information on two selected iterations or revisions. This functionality is also available from the information page Actions menu by selecting **Information Compare** (in this case, when you initiate the action a **Find Object** window appears in which you can specify another CAD document for comparison). Once you have selected an object for comparison, the **Comparison Options** page appears where you can specify the type(s) of information you want to compare. For more information, see the online help available from the **Find Object** and **Comparison Options** pages.

Object Actions from the Information Page

From the information page, you can also initiate actions on the object by selecting an action from the drop-down list near the top of the page. The appropriate user interface for the action appears. After the action is complete, you are returned to the information page. The list of object actions available includes only the actions appropriate for the type and current status of the object.



Special Considerations for Working with Bundled Servers

Though Windchill PDMLink and Windchill ProjectLink have a common information page design, there are special considerations when working with bundled Windchill PDMLink and Windchill ProjectLink servers. These considerations are as follows:

- When an object is shared from Windchill PDMLink:

- Accessing its information page from Windchill ProjectLink or Windchill PDMLink commonspace presents the object's Windchill PDMLink commonspace information page, regardless of access to Windchill PDMLink.
 - The information page displayed is for the iteration that is contained in the project baseline.
 - All information in the information page that is access-controlled and for which the user does not have specific access is hidden. Instead of the information, the user is shown <Secured Information>.
- Accessing its information page from a Windchill ProjectLink workspace presents the workspace information page in the Windchill PDMLink context.
- When an object is PDM checked-out from Windchill PDMLink:
 - Accessing its information page from the Windchill ProjectLink commonspace presents the Windchill ProjectLink commonspace information page.
 - Accessing its information page from the Windchill PDMLink commonspace should lead to its Windchill PDMLink commonspace information page.
 - Accessing its information page from a Windchill ProjectLink workspace presents the object's Windchill ProjectLink workspace information page.

Summary

Information on any object in your workspace can be obtained by clicking the  icon in the workspace table. Similarly, clicking the  icon in the actions column of an object in the commonspace obtains information for the commonspace object. The information page displays three types of information: object information, image information, and reports. The types of information and actions available on the information page depend upon:


- Whether the object is in the workspace or in the commonspace
- The type and current PDM status of the object

Using the Folder Navigator

The Folder Navigator in Pro/ENGINEER is a useful tool that allows you to browse different servers, workspaces, and file locations. Additionally the Folder Navigator provides shortcuts to some commonly performed operations.

Browsing Files in the Folder Navigator

The Folder Navigator is an expandable tree that lets you browse any of your registered servers and workspaces. Additionally you can also browse any file system that is accessible from your computer. As you navigate the folder, the contents of the selected folder appear in the Pro/ENGINEER browser. To activate

the Folder Navigator, click the  tab.

Browsing File Systems

The Folder Navigator contains top-level nodes for accessing file systems and other locations known to Pro/ENGINEER:

- In-session Pro/ENGINEER objects
- All registered servers—The navigator lists all servers (and their workspaces) that you have registered with the Server Registry dialog box. These registered servers may include a Windchill server, Pro/INTRALINK server, and an FTP server.
- The local file system—When you open the Folder Navigator, the local file system appears in the browser with the startup directory node expanded.
- Network Neighborhood—(Only for Windows) The navigator shows computers on the networks to which you have access. The operations you can perform depend on your permissions on the remote computers.

Setting the Primary Server Using the Folder Navigator

As an alternative to using the **Server Registry** dialog box, you can use the Folder Navigator to change a secondary server to be the primary server:

1. In the Folder Navigator, locate a secondary server that you want to make primary.
2. Right-click to open the shortcut menu.
3. Click **Set as Primary**.

Changing the Workspace from the Folder Navigator

As an alternative to using the **Server Registry** dialog box, you can use the Folder Navigator to change the workspace.

To Switch to a Workspace on the Same Server

1. In the Model tree, click the "+" icon next to the primary server. The Windchill server node opens.
2. Open the Workspaces folder.

3. Right-click any of the inactive workspaces and select **Activate** from the shortcut menu.
4. Confirm the change of workspace in the **Change Workspace** dialog box.

To Switch to a Workspace on a Secondary Registered Server

1. In the Model tree, select a workspace on another registered server.
2. Right-click on the selected workspace and select **Set As Primary** from the shortcut menu. A warning dialog box opens, prompting you to erase any object in session.,
3. In the warning dialog box click:
 - **Yes**—Deletes all objects in session and activates the selected workspace.
 - **No**—Preserves all objects in session and activates the selected workspace
 - **Cancel**—Cancels the request to change workspaces.

Summary

The Folder Navigator is a useful tool that allows you to quickly:

- Browse servers and file systems
- Change the primary server
- View other workspaces
- Change the active workspace.
- Change your primary server.

How to Get Help

When working with the PDM system, you can get help on both Pro/ENGINEER and Windchill functionality.

How to Get Help on Pro/ENGINEER Menus and Commands

Use the **Help** command, located on the Pro/ENGINEER menu bar, to access the Pro/ENGINEER Help Center home page, context-sensitive help, release information, and customer service information.

Depending on whether you just want to browse help or get help on a specific object, use one of these methods:

- To access the Help Center, click **Help > Help Center**. This opens the home page for the Pro/ENGINEER Help Center. From here you can navigate to any help topic. **Note:** You can also click the **Site Map** link to get the listing of all modules.

- To get help on a specific user interface object, click **Help > What's This?** This enables the context-sensitive Help mode. You can then click an object and open the corresponding help topic.

How Pro/ENGINEER Online Help is Organized


The Pro/ENGINEER Help Center organizes online help by functional areas. Each functional area lists links to the related Pro/ENGINEER modules.

You can search for information within a functional area or multiple functional areas.

Tip: To access help on Pro/ENGINEER PDM-related commands, go to the Help Center, and select the **Data Management** functional area.

How to Get Help on Windchill Applications

When working with Windchill in the Pro/ENGINEER browser, you can access Windchill help from two locations:

- To get help on a specific table or action user interface displayed on the Windchill page, click the **Help** button  in the upper-right corner of that table or title bar for the action.
- To get help for the page you are viewing, click the **Help** link in the header of your screen in the browser.

To view the Table of Contents from a single topic, click the **View Other Topics** button.

Note: If a page does not contain a table, such as a document information page, online help for that page is available from the **Help** link in the header of your screen.

How Windchill Online Help is Organized

The Table of Contents of each major functional area contains links to other online help projects.

Note: The online help documents the out-of-the-box features delivered in Windchill. Each implementation of Windchill may be customized. What you see in your version of Windchill may differ from the provided documentation based on your site customization and your permissions.

3

Basic PDM Operations

After registering the PDM server with Pro/ENGINEER, you can start using the PDM system in your daily work. This chapter introduces you to some common activities used in PDM and to the most commonly used PDM operations. You will notice that a number of operations can be initiated either from the Pro/ENGINEER menus or from the workspace page in the Pro/ENGINEER browser.

Topic	Page
Collecting Objects for PDM Operations	3-2
Setting an Object Location	3-12
Saving and Uploading Objects	3-14
Checking In Objects	3-27
Checking Out Objects	3-32
Adding Objects to the Workspace	3-41
Removing Objects from the Workspace	3-46
Importing Objects to the Workspace	3-46
Exporting Objects from the Workspace	3-51
Keeping Workspace Objects Up-to-Date	3-54
Revising Workspace Objects	3-59
Using the Event Manager	3-61

Collecting Objects for PDM Operations

A basic practice for many PDM operations (also referred to as actions) is specifying the set of objects upon which you want to perform the operation. Though the set might only consist of a single object in your workspace, more often it is made up of one or more assemblies, with or without other dependent objects and associated enterprise parts, and referencing a particular configuration (either latest or as-stored, or based on a particular baseline or part effectivity). This section discusses how you can collect the exact objects that you want for your PDM activities.

The workspace is your typical starting point for a PDM action, although actions can be initiated in many places throughout Windchill. Initially, you select an object (or several) and then click the button corresponding to the desired action. An action page appears, with your initially selected objects listed in the **Object List**. Arriving at your final collection typically follows this sequence:

1. Initial Selection -- Generally you select a top-level object
2. Adding or removing related objects -- Collection controls let you add or remove related objects by setting rules
3. Specifying a configuration for the objects -- Embedded or pop-up fields allow you to set the desired configuration
4. Excluding unwanted objects -- Objects collected by the foregoing steps can be selectively excluded.
5. Setting options (not part of all actions) -- Specifying how objects will be treated upon the execution of the action

About Dependency Processing

Dependency processing refers to the tracing of the object to object relationships among the objects considered for inclusion in the configuration you specify for any of the various PDM actions. A key distinction is between part-centric and document-centric dependency processing.

Essentially, part-centric processing traces a product structure for dependencies while document-centric processing traces a CAD document model structure. Either method can be selected, regardless of whether the initially-selected object is a part or a CAD document, so long as there is an active association between the part and CAD document.

For example, for part-centric processing of an initially-selected CAD document :

- The CAD document remains in the object list
- The associated part object is included in the object list
- The associated part becomes the "root" object for:
 - Setting configuration (latest, managed baseline names, view, effectivity)

- Collecting dependents (all, none)

For example, if:

configuration == managed baseline

dependents == all

Then only the part dependents that are also part of the managed baseline are included in the object list.

- The possible related objects to be included (CAD documents, documents)
 - CAD documents will be the version (revision/iteration) that is peer to the gathered part version
 - If a managed baseline for parts is selected, the CAD documents do not need to be part of the baseline.

When a workspace specification has dependency processing set to part centric, a CAD document is considered out-of-date if the version in the workspace does not have an active associate link to the part version included in the specified baseline for parts.

Configuration

Most PDM actions involve collecting groups of objects for you to act upon. Typically, you initially select a one or more key objects and then gather a larger set of dependent objects that you want to include in the action based upon their relationship to the initially selected object. The system chooses a default group of objects that you can then modify to suit your intended task. The particular set of object types and iterations you gather is the configuration applicable for the action.

The configuration controls need to be accessed whenever you want to change the configuration rule for collecting objects. A configuration may be identified for each object selected for a particular action.

Overview of Configuration Specification

When specifying a configuration, you first identify whether the configuration is based on an enterprise part, end object, product, or serial numbered part (a Product Structure, specifying part-centric processing); or on a CAD document (a Model Structure, specifying document-centric processing). This choice determines the set of configuration options (latest, managed baselines; as stored configurations; per workspace configuration specification; or based on part effectivity) available for selection, as follows:

- For Model Structure -- Latest, as stored configurations, promotion requests, per workspace configuration, and managed baselines that include the iteration of the single initially selected CAD documents or the CAD documents actively associated to the initially selected parts in the object list.

- For Product Structure -- Latest, per effective date, managed baselines, per workspace configuration, and promotion requests that include the iteration of the initially selected parts or the parts actively associated to the initially selected CAD documents in the object list.

If the initially selected object has an active association to a peer object (for example, a CAD document has an active association to a part, or vice versa) you can collect iterations of dependents based on a configuration specification for the peer object.

You may change the rule for collecting dependents.

You may change the configuration of the objects to be collected.

If the action is initiated from a workspace, or results in objects being added to a workspace, configuration changes may be written to the workspace configuration specification (when the configuration is committed).

When you commit the selections you make in the configuration tool, your changes are applied to the object list.

Note: Configuration changes have effects if you have already collected related drawings, parts, CAD documents, or instances, or edited the list in any other way (for example, using exclude). Upon changing a configuration, the collected objects that are related to the new configuration remain in the list. However, if the `property.core.collection.collectall` is set to false in `wt.properties`, all of the collected related objects are removed from the list upon changing the configuration.

Changes to configuration settings never remove initially selected objects from the object list.

Changes to configuration settings can change the iterations of dependent objects added to the action list; configuration settings never change the iteration of initially selected objects in the object list.

For the Update action, the iteration of the initially selected object is the up-to-date iteration, not the iteration that was initially selected. Also, when using effectivity for Product Structures, the iteration of the initially selected objects will be the effective one per the date specified, which may be different from the iteration that was initially selected.

Using the Collection Tools

The collection tools available for PDM actions include configuration specification and are present in any action page where collecting additional object for the action is supported. Two modes of collection, termed basic and advanced, are provided to perform the collection. The basic mode is rule-based and can be preset by administrators with default rules to simplify the collection process (though users may be able to override the rules). The advanced mode allows users to see the initially selected and subsequently collected objects in a table view, and to act on objects individually, using menu options provided for the table. For many actions, only the table (advanced) mode of collection is available. Where the basic mode is available, users can toggle between the two modes (with some restrictions, as described in the following sections) by selecting the appropriate tab.

Collecting in Basic Mode

The Basic mode of collection is designed for rapidly accomplishing collection and configuration activity for an action. You can specify a simple set of rules and continue with the action. You can choose whether to collect just those objects related to the Initially Selected object, All objects, or None for each rule that you define.

Although you are collecting objects using this mode, the set of collected objects is never displayed on this window, simplifying the collection activity.

Use the following procedure to collect objects using the basic mode:

1. Depending on the action you are performing, action-specific fields may be available in the top panel of the collector window.

For example, when performing the **Add to Workspace** action, you can specify whether to check out **All**, **Selected and Modified**, **Selected**, **Required**, or **None** of the collected objects. In addition, you can specify whether to **Download**, **Link**, or **Reuse** content.

If action-specific fields are available, choose the desired options.

2. You can optionally click the **Advanced Processing Options** link in the upper-right corner of the collector pane. This opens the **Advanced Processing Options** window, which allows you to decide whether to use the system defaults or bypass them. Your selection in this window determines which options you see in the **Change Configuration to** drop-down list.

By default, the system shows certain options only in the **Change Configuration to** drop-down list. These options depend on the object type initially selected.

- If you have initially selected a part, the system will process the part structure by default and will show you the following options: **Latest**, **Baseline**, **Effectivity**, and **Promotion Request**.
- If you have initially selected a CAD document or a dynamic document, the system will process the CAD document or dynamic document structure by default and will show you the following options: **Latest**, **As Stored**, **Baseline** and **Promotion Request**.
- If you have originally selected a document, the system will process the document structure by default and will show you the following options: **Latest**, **Baseline**, and **Promotion Request**.

If you want to specify which structure to process, select **Bypass system default** in the **Advanced Processing Options** window. All possible choices will then display in the **Change Configuration to** drop-down list, and you will be able to choose the dependency processing type while setting your configuration.

3. The following configuration options may appear, depending on the initially selected object and the option specified in the **Advanced Processing Options**.
 - **Latest** -- The most recent version of the object is retrieved.
 - **As Stored** -- Iterations of assemblies with or without their dependents are retrieved as they were stored during checkin. When a single object is selected, only objects owned by the seed object iteration are retrieved. When multiple objects are selected, the objects for which the initially selected object's iteration are the common owner are retrieved.
 - **Baseline** -- When a single object is selected, the baselines in which the initially selected object iteration is a member are retrieved. When multiple objects are selected, the baselines in which the selected object iterations are common members are retrieved. You can also select other baselines that other iterations of the objects are in.
 - **Effectivity** -- Objects that match the effective date, effectivity context, or serial number of the initially selected objects are retrieved.
 - **Promotion Request** -- When a single object is selected, the promotion requests in which the initially selected object iteration is a member are retrieved. When multiple objects are selected, the promotion requests in which the selected object iterations are common members are retrieved.
4. Once you have selected a configuration, a window displays, allowing you to choose configuration specifications. Depending on which configuration you have chosen, the following fields may appear:
 - **View** -- Select a view to limit the collection to versions within that view.
 - **Life Cycle State** -- Select a life cycle to limit the collection to object versions in that life cycle state.
 - **Apply configuration to the initially selected objects** -- Select this check box to replace the originally selected version of each object with versions retrieved by the configuration specification.
 - **Include Work in Process** -- Select this check box to retrieve the checked out copy of an object if that object has been checked out to you.
 - **Use latest configuration of unresolved dependents** -- Select this check box to retrieve the most recent version of the object if no valid version is found based on the configuration specification. If this box is not selected, no version of the object will be retrieved.
5. In the **Dependencies** panel, choose from the drop-down list which types of dependents will be included. You may be able to choose **All**, **Required**, or **None**, depending on the initially selected object type.
6. If the **Contexts** field is available, choose one of the following contexts:

Note: This field is not available for all collection processes.

- **Context of the Initially Selected Objects Only** -- Only the objects located in the context of the initially selected objects will be collected.
 - **All Contexts** -- Objects will be collected across all contexts.
7. In the panel for collecting related objects, accept the default rules, or select new rules from the drop-down lists for collecting the available objects. The object types available depend on the action you are performing.

Select one of the following options for each object type:

- **All** -- Objects related to the initially selected objects plus objects related to dependent objects will be collected.
- **Initially Selected Only** -- Only those objects related to the initially selected objects will be collected.
- **None** -- No objects will be collected.

Note: For change objects, only the **All** and **None** options appear.

8. Click **OK** to process the collection and commit the action, or click **Next** if you are continuing on to another step of the action. The system collects the objects specified by the rules you set and processes them.

or

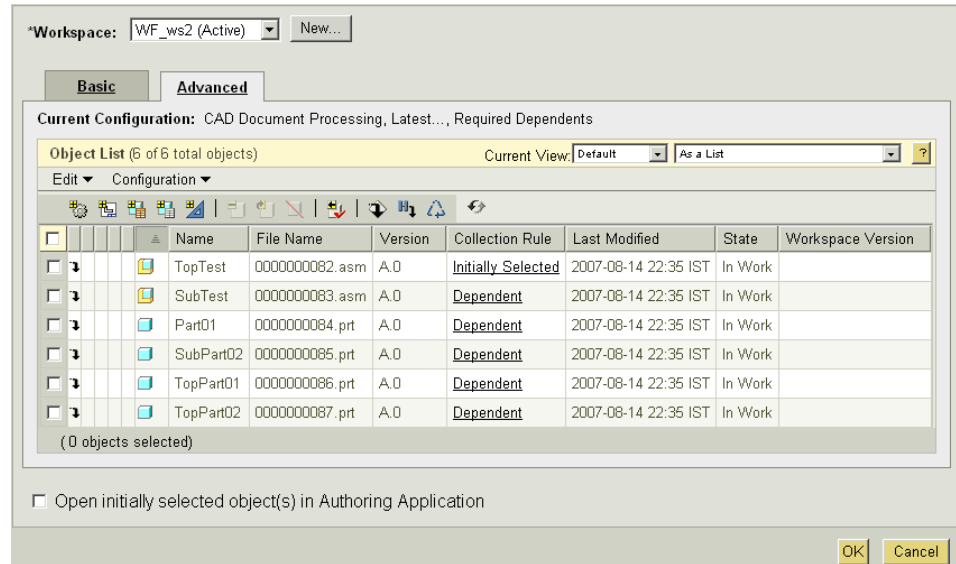
Select the **Advanced** tab at any point in the collection process to use the advanced mode to collect objects.

Note: The collection processing will not occur until you click **OK** or **Finish**.

Collecting in Advanced Mode

The **Advanced** mode of collection is designed for the majority of process scenarios. It allows you to specify collection rules and look for related objects. When you use **Advanced** mode, you will see collected objects displayed in a table. You also use these subsequently collected objects to create new collections.

If you switch from **Basic** mode to **Advanced** mode, the initially selected objects are displayed along with any objects gathered from the collection rules specified in the **Basic** mode.
















Note: If both the **Basic** and **Advanced** tabs appear, you can move from **Basic** to **Advanced**, but if you change the configuration criteria or rule criteria, data may be lost. Data can also be lost if you change one or more rules and then move from **Advanced** to **Basic**. A window appears, allowing you to confirm that you want to move to the **Basic** mode or to cancel the action.

The following actions may be available from the collector table:

Note: Other menus, such as **File** and **Edit** may appear in the menu bar to allow you to make other operation-specific selections. Other actions may also appear in the collector table toolbar, depending on the action that you are performing.

Action	Description
Set Configuration	<p>Set filtering criteria in order to trace an initially selected object to dependent objects.</p> <p>The options available depend on the type of initially selected object you have chosen and on whether you have chosen to use the system defaults for configuration or to bypass them. (See Advanced Processing Options action row in this table for more information.)</p> <p>You can also select the Set Configuration action from the Configuration menu on the table to collect related objects.</p> <p>Note: When you set the configuration specification, the table re-collects related objects.</p>

Action	Description
Add Dependency	<p>Specify which types of dependents will be included. You may be able to choose All, Required, or None, depending on the initially selected object type.</p> <p>The Add Dependency action is available from the Configuration menu on the table.</p>
Advanced Processing Options	<p>Open a window in which you can decide whether to use the system defaults or bypass them. This determines which options you see in the Set Configuration drop-down menu.</p> <p>By default, the system shows certain options only in the Set Configuration menu. These options depend on the object type. If you are using a part structure, for example, the system will show you the following options by default: Latest, Baseline, Effectivity, and Promotion Request.</p> <p>If you want to specify which structure to process, select Bypass system default in the Advanced Processing Options menu. All possible choices will then display in the Set Configuration menu, and you will be able to choose the dependency processing type while setting your configuration.</p>
	Click the parts icon to collect parts associated to the selected object.
	<p>Click the drawings icon to collect all drawings associated to the selected part or CAD document.</p> <p>Note: This action does not apply to other types of documents.</p>
	Click the family objects icon to collect all family table objects related to the selected generic or instance.
	<p>Click the generics icon to collect the generic associated to selected instances.</p> <p>Note: This action only applies to CAD documents.</p>
	Click the CAD documents icon to collect all CAD documents associated to the selected part.
	Click the documents icon to collect all documents related to the selected object.

Action	Description
	Click the affected data icon to collect all affected data related to a selected change object. This applies for change objects that can have affected data, such as problem reports, analysis activities, change requests, and change activities.
	Click the change objects icon to open a window that allows you to select which type of change object you want to collect. For more information, see the help available from that window.
	Click the notes icon to collect all notes related to the selected object.
	Click the remove icon to remove selected objects from the table.
	Click the include icon to include selected objects in the collection.
	Click the exclude icon to exclude selected objects from the collection.
	<p>Click the reset icon to reset the collection to the initially selected objects. If you reset your collection, all additionally collected objects will be removed from the entire table.</p> <p>When you select this action, a warning message will appear explaining that all data collected will be lost.</p>
Current View	If available, this allows you to select an existing view or create a customized display for table objects and their attributes.
Ignore objects in other contexts	<p>This check box may appear below the collection table for some collection processes.</p> <p>When selected, only the objects located in the context of the initially selected objects will be collected.</p> <p>Note: If you change the value of this check box, a window will appear explaining that all data collected will be lost and allowing you to confirm or cancel the re-collection.</p>

Note: The part, drawing, and CAD document collection icons are present in all collection tables. The other actions appear as appropriate for the operation you are performing.

From the **Current View** drop-down list at the top right corner of the table, you can select the default view or click **Customize** to create a customized display for table objects and their attributes.

You can also select a display for the table. The displays available depend on the operation you are performing. Common displays include:

- **As a list** -- Shows all objects in the collector table as a list.
- **As a Structure** -- Shows structured objects, such as parts, documents, and CAD documents, arranged in a hierarchy, and a connecting line represents the uses relationships. Other related objects appear below the object to which they are related.


The structure display does not show every type of a relationship. It does not include links, and it should not be considered to be a where-used collection.

- **As a Structure with Associated Objects** -- Shows structured objects in a hierarchical view and includes objects obtained after setting the configuration specification and adding dependencies.
- **By Drawings** -- Shows all collected drawings and their associated parts and assemblies.
- **By Associated Objects** -- Shows all objects obtained after setting the configuration specification and adding dependencies. Collected associated objects are displayed underneath.
- **By Family Table** -- Shows related family table instances and generics that have been collected. Each top-level generic is displayed as a root node with sub-generics and instances listed below.
- **By Change Objects** -- Shows change objects with associated data and related change objects below.

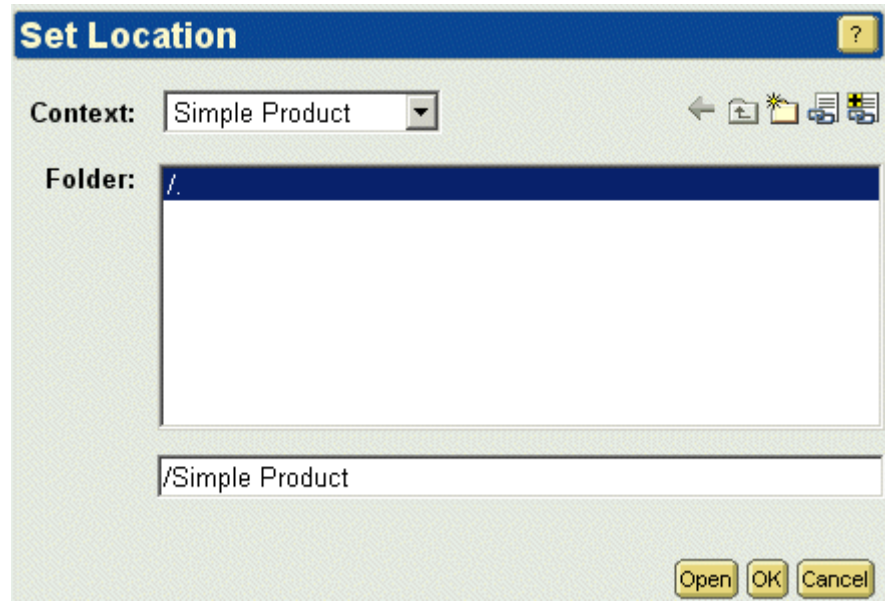
To complete the collection process using the **Advanced** mode, click **OK** to commit the action, or **Next** if there are more steps in the operation you are performing.

Setting an Object Location




In Windchill, a *location* is a folder (or subfolder) with a context. Objects are assigned their storage locations when first checked in. While a default location is assigned by the workspace configuration specification, during initial check-in you

can use the **Set Location**  command to invoke the **Set Location** dialog box to specify a different location. You are not allowed to set a location during subsequent check-ins; however, the **Set Location** dialog box is available during other operations (such as, Move and Save As).

Note: To change the context of an object after it has been checked in to Windchill, you must use the Move action.





The **Set Location** dialog box presents the **Context** and **Folder** fields to specify a context/location, a location string field that dynamically displays the current folder path, and a group of navigational actions described as follows:

- **Previous Location**  -- Resets the location screen to the previous display. When selected, the context, location, and folder location string fields show the values shown in the previous display.
- **Up One Level**  -- Moves your navigation from a lower folder level to the next higher folder level. When selected, the dialog box refreshes to show the folders at the next higher level and the folder location string for the next higher level.
- **Create New Folder**  -- Creates a new folder at the currently active folder level.




Note: The "forward slash" character (/) is not allowed in Windchill folder naming. Context or folder names that contain the following characters may not be accessible from certain Pro/ENGINEER user interfaces (such as, the **File > Open** dialog box):

\/: * ? " < > |


- **Hot Links**  -- Opens a list of Hot Links (saved URLs from your Windchill Notebook table). Only hot links that represent folder location in a valid product, library, project, or Windchill PDM context are listed.
- **Add to Hot Links**  -- Adds the currently selected folder to your list of Hot Links.

To select a context/folder location, use one or more of the following procedures (you can use the procedures in combination):

To browse to a location by context:

1. Select a context from the **Context** drop-down list. The top level folder locations for that context are displayed in the **Folder** list field.
2. Select a folder location from the **Folder** list field. Click **Open** to navigate to the next lowest level. The next level of folders appears in the location list, and the currently specified location is displayed in the location string field.
3. As needed, use the **Previous Location** , **Up One Level** , or **Create New Folder**  actions to navigate to a preferred location.
4. Click **OK**. The context/location you specified is set for the object.

To select a location using a saved Hot Link location:

1. Click **Hot Links** . The **Location** list displays a list of your Hot Links that are folder locations.
2. As needed, navigate the Hot Links folder structure to a suitable location.
3. Click **OK**. The context/location you specified is set for the object.

As you browse, you can use the **Add to Hot Links**  to save a location in your Notebook list of Hot Links.

Saving and Uploading Objects

Many CAD data management actions are accessible from both the Pro/ENGINEER Wildfire and the workspace user interfaces. A newly-created object, however, must first be saved in order to appear in the workspace.

You can save Pro/ENGINEER files using the **Save**, or **Save and Upload** commands. A **Save** command creates a file in a specified directory. Traditionally, this has been a directory on the local file system (your working directory) but with Pro/ENGINEER Wildfire your working directory can be your workspace in the PDM system. An **Upload** command, which operates in the background, places a previously saved object in your private space on the PDM server, but does not check it in. The **Save and Upload** command performs both operations simultaneously.

Saving Objects

Pro/ENGINEER saves only changed objects, except in the following cases:

- An object selected for saving is not found in the destination directory. In this case the object is saved to the workspace.

- The configuration file option `save_objects` is set to `all`. In this case, all the objects in Pro/ENGINEER session will be saved to the workspace.

Note: The generally recommended setting for this option is `changed`.


- The configuration file option `save_objects` is set to `changed_and_specified` and the current object is the top object in an assembly. In this case, only the changed objects that are in Pro/ENGINEER memory and are related to a changed assembly are saved to the workspace.
- The configuration file option `save_objects` is set to `changed_and_updated`. In this case, only the changed objects that are in Pro/ENGINEER memory and have been updated are saved to the workspace.
- A change is made to a dependent object and the configuration file option `propagate_change_to_parents` is set to `yes`.

Caution: When working in standalone Pro/ENGINEER (no Windchill server or workspace connected) if you retrieve an object using a relative path specification (for example, `../partname`), the same path specification is used to save it in its original directory. So if you change your working directory between retrieving and storing, the object could be saved in a wrong directory. Be careful when you use relative path names. If, however you have registered a Windchill server and you retrieve an object from your working directory, the object is saved to the active workspace.

Note: Path names can contain up to 260 characters.



Performing a Save in Pro/ENGINEER

To save an object in Pro/ENGINEER session:

1. Click  or **File > Save**. The **Save Object** dialog box opens.
 - If you previously saved, there are no options available in the **Save Object** dialog box to change directories. Click **OK** to complete the save.
 - If you have not previously saved the object, go to Step 2.
2. The directory in the **Look In** box can be one of the following:
 - When you are connected to a Windchill server the directory is your active workspace, listed in the format "<workspace name> on <server name>."

You cannot select another directory (go to Step 4).

 - When Pro/ENGINEER Wildfire is not connected to a Windchill server the directory can be one of the following:

- **My Documents** (Windows platforms only), if you have not set a working directory or previously saved an object to another directory in your current Pro/ENGINEER session.
 - The Working Directory you set for your current session.
 - The directory you last accessed to open, save, save a copy, or back up your file.
3. Accept the default directory or browse to a new directory.
- Note:** You can access the working directory by clicking .
4. In the **Model Name** box, the name of the active model appears. To select a different model, click .
 5. Click **OK** to save to the directory displayed in the **Look In** box, or select a subdirectory, and then click **OK**. The Pro/ENGINEER graphics area is displayed.

Tips for Saving Objects


When not connected to a Windchill server:

- Objects are stored in their original directories, unless you set the configuration option `override_store_back` to yes.
- If you do not want your file to save to the last accessed directory, set the configuration option `file_open_default_folder`. Use this configuration option to specify the directory from which you want to open, save, save a copy, or back up files.
- If you do not have write permission to the original directory and have `override_store_back` set to no, set the configuration option `save_object_in_current` to yes to store the objects in the current directory.
- By default, Pro/ENGINEER saves the model on which a drawing is based only when changes have been made to it. You can use the configuration file option `save_modified_draw_models_only` to save the model every time the drawing is saved.
- To save disk space, compress file output by setting the configuration option `compress_output_files` to yes. Compressed files take longer to read and write, but are one-half to one-third the size of uncompressed files. They are also fully compatible across platforms.

Creating CAD Documents with Pro/ENGINEER CAD Data

Saving a model to the workspace creates a CAD document that contains the model file. You can also initiate the CAD document creation from the workspace itself, specifying the CAD document's attributes during the process.

To create a CAD document from the workspace, perform the following steps:

1. From the workspace, select **File > New > CAD Document** or click the new CAD document icon . The **New CAD Document** window opens at step 1: **New CAD Document**.

2. Enter or select entries for the following fields:

- **Number** -- The number you enter must be unique.

Note: If the site preference to use auto-numbering is set, this field is inactive and you cannot enter a number manually.

- **Context** -- Select a context from the drop-down list provided. The workspace context is shown by default.
- **Name** -- Enter a name for the new CAD document. The default name is the same as the number.
- **File Name** -- Enter the CAD file name that the new CAD document references (include the file extension).

Note: If auto numbering is on, File Name is not shown as a required field. If a file name is not entered, an auto-generated number is used for File Name.

- **Type** -- Select the type of object.
- **Template** -- Select from the available templates in the drop-down list.
- **Location** -- Enter the path or browse to a folder within your selected context where the CAD document will be saved. Clicking **Browse** presents the **Set Location** page to select a cabinet location on the server. You have the opportunity on this page to create a new folder.
- **Description** -- Optionally, enter a description of the CAD document.
- **New Revision** -- Accept the default value, or, if revision setting is enabled, click **Find** to select a revision in the **Set Revision** dialog box.

Note: Either **Number** or **File Name** is a required field. Regardless of the naming and numbering policy in place, when creating a new object, system uniqueness constraints require that the CAD document attributes Number and File Name must both be unique within a Windchill PDMLink site or within each Windchill ProjectLink project.

Tip: If the site preference to display organization information is set, you will also be able set a value for Organization ID. If you select an external organization, the **Number** field will accept a manual entry even if auto numbering is on.

3. If you want to simultaneously create an associated part, select the **Create and Associate Part** check box.

Note: It is recommended that you associate a part with a CAD document at object creation, not when you create a structure.

4. Click **Next**. Step **2: Specify Attributes** appears. Optionally, specify attributes for the CAD document.

Note: This optional step lists all the soft attributes. If a template is specified, the list and values of soft attributes come from the template CAD document. If the template is not specified, then the list and values of soft attributes should come from the latest Soft Type definition for the system provided for the default CAD document type. For a list type attribute, it presents a drop-down list of generated values. For a range type attribute, a tool-tip is shown in the input text field to indicate the range.

5. Click **Finish**. The specified CAD document is created.

Note: If you are creating a CAD document in a workspace on a Windchill PDMLink server, the template file for your CAD document will be the one created for your context.

The newly created CAD document by default is checked into your chosen context.

Creating Part Structures for CAD Data

Once a CAD document structure has been created, a product structure can be created in Windchill by creating and associating an enterprise part to each CAD document in the CAD document structure, and then checking all the objects into Windchill. Upon check in, the Windchill build rule uses the relationships among the CAD documents to build a product structure relating all the enterprise parts. The term *owner association* (or owner link) indicates that the association is a primary kind that is recognized by the build rule in creating a structure. The term *content association* (or content link) refers to a secondary association (for example, that of a drawing to a part) wherein the CAD document describes the part, but does not need to be included in the product structure.

Associating CAD Documents to Parts

It was mentioned in the section on creating CAD documents that a recommended practice is to associate Windchill parts with CAD documents at the time of creation. However, there are at least two reasons why that practice might not be followed:

- The CAD documents were created in the workspace by the **Save and Upload** command in Pro/ENGINEER, not using the **New CAD Document** window in the HTML UI.

In this case, the workspace provides the **Auto Associate Parts** command that allows you to select multiple CAD documents, and then create and associate enterprise parts for those documents with a single click. You can also associate CAD documents to existing parts.

Note: The preference, Windchill Workgroup Manager > Server > Auto Associate > Store New Parts with CAD Documents, when set to "Yes," specifies that the storage location of new part created during Auto Associate be the same as its associated CAD document. By default, the preference is set to "No."

- Enterprise parts intended to correspond to the CAD documents have already been created in Windchill.

In this case, the **Associate** command allows you to select a CAD document and then search or browse for the appropriate enterprise part to which to associate it. The **Associate** command also allows you to start with a part and find an appropriate CAD document. The use of both commands is detailed in the following sections.

Automatically Associating Parts

The **Auto Associate Parts** command allows you to automatically find and associate an existing part to a CAD document or, if no matching part currently exists, create a new part and associate it to the CAD document. This functionality operates according to several conditions, and is accomplished using the **Auto Associate Parts** page.

Note: The exact manner in which the part is searched for, created, named, and numbered depends on preferences set by a site administrator. For more information, see the section [Customizing Auto Associate](#).

Auto Associate Conditions

The autoassociate parts functionality works in accordance with the following conditions:

- For a successful association, the document must be checked out and have no existing associations, and the part must be checked out (autoassociate automatically checks out the found or created part to the workspace). Documents and associated parts remain checked out to the workspace after association.
- While searching parts, if more than one part per document is returned, then the Auto Associate Parts command ignores the document and an error message is shown in the event console.

- In multi-selection, if you select parts and checked in documents along with qualified CAD documents, the parts and checked in CAD documents are ignored by the Auto Associate Parts command. If none of the selected objects are valid candidates for the command, then a status message is shown:

None of the selected objects is eligible for the 'Auto Associate Parts' action.

- If you select a newly created drawing document, the system searches for a model for the drawing in the database. If the model is found, the system creates a Described By link between the part and the drawing, and an owner association between the part and the model. If the model is not found, a message stating this is reported in the event console.

Auto Associate Page

When you select a CAD document from the workspace and click **Auto Associate Parts**, the **Auto Associate Parts** page appears with two object list tables.

The **Existing Parts** table lists any documents and existing parts proposed to be associated. If no existing parts are found the table message instead reads "Search found no existing parts." When parts are found, the information and possible actions are presented in the columns described in the following table:

Column	Description
(Select row)	Contains a check box which lets you select the object represented by that row. The check box in the table header for the column allows you to select or deselect all rows.
(status)	Symbols in the several status rows indicate the status of objects. Holding your cursor over the column header displays a tooltip identifying the status reported in that column.
(Type)	Contains the object icon specific to the CAD document type.
Number	Displays the number of the CAD document
Revision	Displays the revision level of the CAD document.
Part Number	Displays the number of the part found during the search based on the search criteria.
Part Name	Displays the name of the part found during search based on the search criteria.


Column	Description
Existing Association	Displays the current association (owner or content) between the CAD document and part.
Association	Contains a drop-down list that allows you to modify the current association between the CAD document and part.

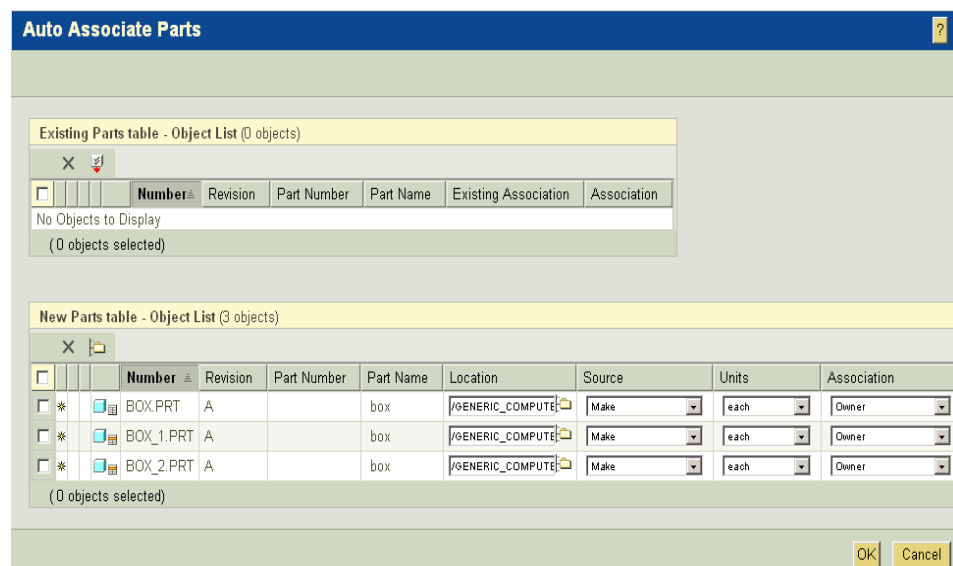
The **New Parts** table lists CAD documents and proposed new associations. The information and possible actions are presented in the columns described in the following table:

Column	Description
(Select row)	Contains a check box which lets you select the object represented by that row. The check box in the table header for the column allows you to select or deselect all rows.
(status)	Symbols in the several status rows indicate the status of objects. Holding your cursor over the column header displays a tooltip identifying the status reported in that column.
(Type)	Contains the object icon specific to the CAD document type.
Number	Displays the number of the selected document, formatted as a hyperlink to document details page.
Revision	Displays the revision of the CAD document.
Part Number	Contains a field specifying the number of the part associated to the CAD document. The field may be editable, depending on the site preference set.
Part Name	Contains a field specifying the name of the part associated to the CAD document. The field may be editable, depending on the site preference set.
Location	Contains a text field and folder browse button so that you can browse to and set a folder location for selected parts.
Source	Contains a drop-down list to specify how the part should be sourced.

Column	Description
Units	Contains a drop-down list for setting the default units for the selected part.
Association	Contains a drop-down list which allows you to modify the current association between the selected document and part.

To auto associate parts, perform the following steps:

1. In the workspace, select the CAD documents to which you want to auto-associate parts and select **Edit > Auto Associate Parts** or click the auto associate icon  in the table tool bar. The **Auto Associate Parts** page appears.



Auto Associate Parts

Existing Parts table - Object List (0 objects)

	Number	Revision	Part Number	Part Name	Existing Association	Association
No Objects to Display (0 objects selected)						

New Parts table - Object List (3 objects)



	Number	Revision	Part Number	Part Name	Location	Source	Units	Association
<input type="checkbox"/>	BOX.PRT	A		box	/s/GENERIC_COMPUTE	Make	each	Owner
<input type="checkbox"/>	BOX_1.PRT	A		box	/s/GENERIC_COMPUTE	Make	each	Owner
<input type="checkbox"/>	BOX_2.PRT	A		box	/s/GENERIC_COMPUTE	Make	each	Owner

(0 objects selected)

OK Cancel

2. Examine the associations listed in the **Existing Parts** table (if any), using the **Association** column to change the association type, if desired.
3. Examine the proposed associations in the **New Parts** table. If necessary, modify the part number, name, or type of association (as allowed by the preference settings of your site).
4. If the auto-numbering preference is set to on and the auto associate policy is custom, you can move rows from the existing parts table to the proposed association table, using the **Move Rows** command in the menu bar of the

existing parts table. Only rows having "none" as the existing association can be moved.

5. In either table, you can use **Exclude/Include**  to toggle the inclusion or exclusion of objects in the autoassociate action. When an object is excluded, its row is struck through with a line.
6. Use the button in the **Location** column, along with the drop-down lists in the **Source** and **Default Units** columns to set appropriate values for a row. You can also use the check boxes in the leftmost column, and the set location icon  to set a location for multiple rows.
7. Click **OK**.

Note: The view assigned to parts is the one specified in the **General** tab of the **Edit Workspace Options** window (available by selecting **Edit Preferences** from the workspace **Pick an Action** drop-down menu).

Explicitly Associating CAD Documents to Parts.

Initiating the **Associate** action causes an automatic checkout of the selected CAD document and part objects, if they are not already checked out. In the case of a CAD document, checkout is for meta data only; no content is downloaded during this checkout action.

Note: Any object checked out implicitly remains checked out even if the association fails to check out the other object.

After you associate part with all CAD documents in an assembly, you can see the association; however, Uses links between the parts are not visible until after check in, when the Windchill build rule constructs the Uses links.

It is recommended that you associate a part with a CAD document at the time of object creation, not when you create a structure. A CAD document is said to describe a part to be included in the bill of materials, and in turn a part is described by a CAD document.

To associate a CAD document with a part:

1. Select the CAD document you want to associate with a part in the workspace.

Note: You may select more than one CAD document to associate with a part.

In this case, all the CAD documents describe the part; however, only one CAD document can have an active link to the part.

2. Select **Edit > Associate**. The **Associate to** page appears.

The screenshot shows the 'Associate to CAD Part' dialog box for 'box.prt, A.1'. The dialog has a title bar with a question mark icon. Inside, there is a link 'View CAD Document information'. Below this, the 'Find objects by:' section has two radio buttons: 'Search' (selected) and 'Browse'. The 'Object Type' is set to 'All' and the 'Location' is 'GENERIC_COMPUTER'. A 'Go...' button is to the right. The 'Workspace' is set to 'WVF_ws1'. Below these is a section for 'Parts (0 of 0 total objects)' with a 'Current View: Default' dropdown. A 'Parts to Associate' section contains a table with columns: Number, Name, Association Type, State, Last Modified, and Version. The table is currently empty, showing 'No Objects to Display' and '(0 objects selected)'. At the bottom right are 'OK' and 'Cancel' buttons.

Associate to CAD Part - box.prt, A.1

[View CAD Document information](#)

Find objects by:

☒ Search
☐ Browse

Object Type: All
Location: GENERIC_COMPUTER

Go...

Workspace: WVF_ws1

Parts (0 of 0 total objects) Current View: Default

Parts to Associate

Number	Name	Association Type	State	Last Modified	Version
No Objects to Display (0 objects selected)					

OK Cancel

3. Search for or browse to the part you want to associate to your CAD document. Your results appear on the **Select the Parts to Associate** page.

Find objects by:

☐ Search

☒ Browse

Object Type

Location

You Are Here: / [My workspaces](#) / [Workspace on mika project](#)

Folder Contents

Select Objects

	Name	Number	Version	Type	State	Last Updated
<input type="checkbox"/>	Ball Box	0000000084	A	Part	In Work	2003-12-01 23:05:38 EST

4. Select the part you want to associate with the document and click **OK**. the selected part is shown in a row on the **Associate to** page.
5. By default, the system creates an owner link between the CAD document and the selected part. If you do not want the CAD document to drive the structure and attributes of the part, select "Content" from the drop-down list in the **Association Type** field.
6. Click **Ok**.
7. Perform a check in on the objects.

Note: Checking in the newly associated objects completes the association by allowing the Windchill build rule process to create Uses links between the parts of a part structure.

To verify the association, view the details page for either object. Selecting **CAD Documents** from the **Related Objects** menu on the information page for a part shows the associated CAD document in the **Described by CAD Documents** table. Selecting **Parts** from the **Related Objects** menu on the information page for a CAD document shows the associated part in the **Related Parts** table.

Once checked in, the parts you associated to CAD documents have a product structure, which is visible on the **Product Structure** page for the top-level part.

Uploading Objects

You can use the **Upload** command when you want to store your object securely on the PDM server, but still keep it invisible to other users. Once an object has been saved, you can upload it to the server using the Pro/ENGINEER **File > Save and Upload** command, or the **Upload** action from the workspace (if the object has been saved to the workspace).

Performing an Upload from Pro/ENGINEER


The procedure for uploading an object from Pro/ENGINEER is initiated by selecting **File > Save and Upload**. The rest of the procedure is identical to that for a simple save, as described in a previous section. At the end of the procedure, the system uploads the object to your personal area on the PDM server and notifies you that the upload has been successful.



Performing an Upload from the Workspace

Consider the following information about an upload operation:

- Upload is only valid for files that are new or checked out to the workspace and modified. A workspace file which is identical to the server file is not uploaded.
 - Selecting a workspace object for upload also selects its new or modified dependent objects to be uploaded. Selecting a family table object also selects other family members, including the generic.
1. In the active workspace, select the objects you want to upload and select **File > Upload**.



The **Upload** page appears with a table listing valid selected objects.


Note: If you initiate an upload by selecting an object and clicking the upload icon  in the tool bar or use the row level icon, the upload is accomplished without presenting a user interface.

2. On the **Upload** page, selecting an object and clicking the following icons in the **Object List** tool bar enables the described options:
 - Collect Related Drawings  -- Selects for upload all parent drawings associated with the selected object.
 - Choose Location Folder  -- Presents the **Set Location** window to allow you to select the **Context** and storage **Location** for the selected object.

Note: An object's context is defined at the first upload. After the first upload, you must use the Move command to change an object's context.

3. Also available on the tool bar are the following actions:

- Click the reset collection icon  to reset the collection of objects in the table to the original state.
 - Click the set org ID icon  to call the **Organization** window, which allows you to set an organization name and ID.
4. If the objects selected for upload or objects added to the list based on dependencies include incomplete dependent objects, the **Auto resolve incomplete objects** check box is also available. When selected, the auto-resolve functionality offers two options:
- **Update with object on server, then ignore**—The system searches on the server for an object with the same file name. If one is found, the incomplete object is updated by the found file. The object is no longer incomplete and is, therefore, available for upload.
- If no object is found to update the incomplete object, the system ignores the incomplete dependent and removes it from the upload list.
- **Always ignore**—The system simply removes any incomplete objects from the upload list.
- Note:** Site administrative settings may not allow the ignore option. Required dependents cannot be ignored.
5. Click **OK**.

Pro/ENGINEER Wildfire completes the upload operation in the background and displays an animated working indicator  at the bottom of the Pro/ENGINEER window until the operation is complete.

Checking In Objects

When you are ready to place a new object into the Windchill database, or you have completed making modifications to the working copy of a checked-out object and are ready to remove your lock on the object, you check the object in to the database.

When an object is checked in, the system assigns it the next iteration. In the case of a newly created object, the system creates the first iteration. The modified object information becomes available to other Windchill users, and the object is available for checkout by others (unless you specify to keep the object checked out after the check-in).

The check-in process can be accomplished in several ways:

- From the Pro/ENGINEER user interface using either auto or custom Check In

- Using the **Check In** page that is accessible from the workspace in the Pro/ENGINEER browser. These different check-in options are explained in the following sections.

Checking In Objects from Pro/ENGINEER

After you have finished working on objects in your workspace, you can share the design changes with other users. The check-in operation copies the information and files associated with all changed objects from the workspace to the Windchill server.

Check In serves several purposes:

- Check In enables other users to access the latest version of the object and to check it out to their workspace.
- If you created a new object in your workspace or opened an object from disk into Pro/ENGINEER and saved it to your workspace, Check In adds the object to the commonspace database for the first time and makes it accessible to other users.

There are two ways to check in an object:

- **Auto Check In**—Checks in objects from your current Pro/ENGINEER session to the Windchill server using default values that you can set in your workspace's preferences. This method of checkin is only available from the Pro/ENGINEER **File > Check In > Auto Check In** menu.
- **Custom Check In**—Enables you to check or change default settings and also provides additional options during the checkin. The custom method is available from both the Pro/ENGINEER **File > Check In > Custom Check In** menu and the workspace user interface.

Performing an Automatic Checkin

1. In an active Pro/ENGINEER session, select **File > Check In > Auto Check In**. The name of the file appears in the **Model Name** field of the **Save Object** dialog box.

Alternatively, in assembly mode you can select the object's name in the Model Tree and right-click. A shortcut menu appears. Select **Check In > Auto**. The system uses default settings to check in the object.

2. Accept the default object or enter the name of another object in session. Any file name entered must be unique.
3. Click the checkmark button, or press ENTER.

Note: To reject the save operation, click the **X**.

4. The system uploads the file to the PDM server. The modified file can now be referenced and modified by other PDM users.

Note: After a checkin, if you want to continue to modify the file you need to check it out again; however, if the file is still in your local cache it does not need to be downloaded again.

Performing a Custom Checkin

1. In an active Pro/ENGINEER session, click **File > Check In > Custom Check In**. The name of the file appears in the **Model Name** field of the **Save Object** dialog box.



Alternatively, you can select the object's name in the Model Tree and right-click. A shortcut menu appears. Select **Check In > Custom** (if you choose this method, skip directly to step 4).

2. In the **Save Object** dialog box, accept the default object or enter the name of another object in session. Any file name entered must be unique.
3. Click **OK**. The **Custom Check In** dialog box opens.
4. In the **Object Types** area, select one of the following options:
 - **Models**—Checks in the model files.
 - **Viewables**—Checks in files as ProductView viewable files.
 - **Models and Viewables**—Checks in objects as both model files and viewables
5. Click **Ok** in the **Custom Check In** dialog box. The **Check In** page opens in the Pro/ENGINEER browser. See the following section which explains using the **Check In** page.

Checking In from the Workspace User Interface

The **Check In** action presents the **Check In** page on which you can select options specifying which objects and which dependents of the objects to use for actions. The **Check In** page also lets you specify the location in which to store the objects.

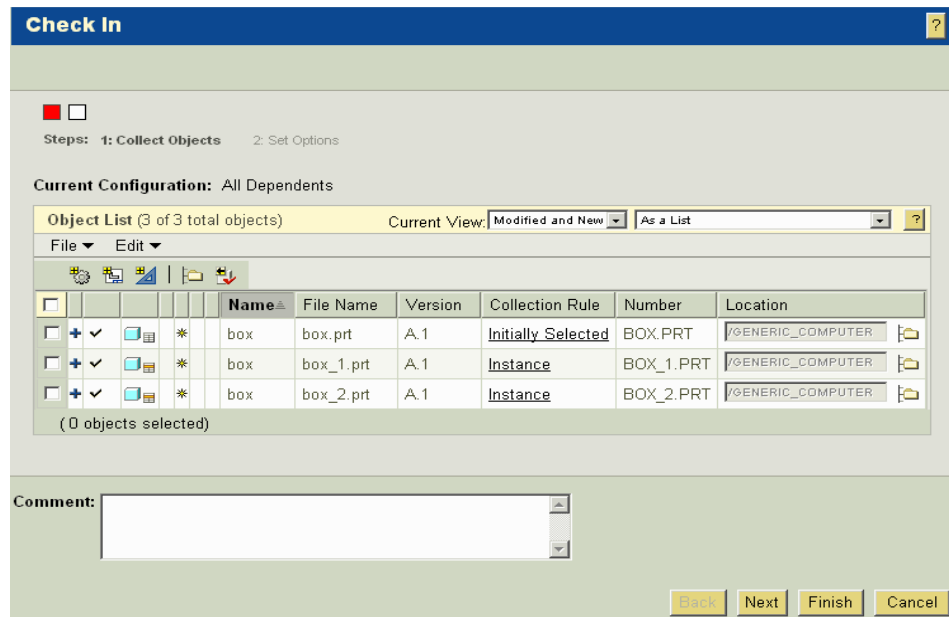
To check in an object:

1. In your workspace, select the object(s) that you want to check in and select **File > Check In**, or click the check-in icon  in the toolbar, or select the object's row-level check-in icon  in the **Actions** column.



You can also initiate a checkin from other places in Windchill (for example, from an object's information page or a folders page) by selecting **Check In** from an **Actions** menu.

The **Check In** page appears at **Step 1: Collect Objects**, displaying your initially selected object(s) and any dependents collected by default in a table listing. On this step, you can collect, remove, include, or exclude objects for



the checkin using the advanced mode of the collection user interface (the rule collection mode is not available for this action, nor are configuration setting tools).



Note: Some provided table views show you all your modified objects, even though some objects may not be eligible to be checked in (for example, they are not checked out). You can use the **Current View** drop-down list to select an existing view or to create a customized display for table objects and their attributes. 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. An adjacent drop-down list lets you select how the table information is displayed.

2. You can check out any collected objects not currently checked-out by selecting them and clicking the checkout now icon  or by selecting **File > Check Out Now**. This action checks the object out without presenting the check-out user interface, and makes the object eligible for the current checkin.
3. Selecting **Edit > Set Location** or clicking the set location icon  presents the **Set Location** window to allow you to select the context and storage location for selected new objects.
4. To toggle the intent to have a particular object checked in, select the object and then select **Edit > Set For Check In**. A checkmark appears in the **Check In Status** column when the object is set for checking in.
5. To toggle the intent to keep an object checked out after the checkin, select the object and then select **Edit > Keep Checked Out**. A checkmark appears in

the **Keep Checked Out Status** column when the object is set to be checked out immediately after the checkin.

6. Optionally, to remove your specifications for object collection and handling, and return the table to its original state, click the reset icon .
7. Enter an optional comment regarding this checkin in the **Comments** field. Comments entered at checkin are shown on an object's information page, as part of **Iteration History**. The comments are shown for every object that has been checked in at the same time.
8. Click **Finish** to commit the checkin without setting any further options. Alternatively, click **Next** to go on to **Step 2: Set Options**.
9. On **Step 2** of the check in wizard, you can set any of the following options (you can set default preferences for these options in the **Preference Manager**):
 - Selecting **Create Baseline** creates by default a baseline with a default name and location.
 - You can edit the name and location path, or click the set location button  to navigate to a new storage location for the baseline.
 - Selecting **Auto Associate Parts to CAD Documents** automatically creates (as necessary) and actively associates parts for the CAD documents not already associated to parts.
 - Selecting **Undo Checkout Unmodified Objects** allows you to undo check out for unmodified dependents of the objects selected for checkin.
 - Selecting **Remove from Workspace** allows you to clear the checked-in objects from your workspace upon completion of the checkin.
 - If the objects selected for check in or objects added to the list based on dependencies include incomplete objects, an **Auto resolve incomplete objects** checkbox is also available.

When selected, you are allowed to select how the auto-resolve functionality handles incomplete objects in one of two ways, as follows:

- In the default method, **Update with object on server, then ignore**, the system searches for an object on the server with the same file name. If one is found, the incomplete object is updated by the found file, is no longer incomplete, and is therefore available for check in.

If no object is found to update the incomplete object, the system ignores the incomplete object (which is removed from the check in list).

- The system can be set to **Always ignore** an incomplete object.

Note: Site administrative settings may not allow the ignore option. Required dependents cannot be ignored.

- Optionally, select the **Attach Differences Report** check box to generate a report that compares the checked-in object to its predecessor iteration and attach the report to the checked-in object. This option is only available from an active workspace (embedded browser only).

10. Click **Finish**. The checkin is committed as you have specified.

Note: Activity in the CAD application session will be blocked until the check-in activity is complete.

Checking Out Objects

To modify an object, you must perform a checkout operation on the object. The process of checking out communicates your intention to modify a design to the PDM server. The checkout operation ensures that access to objects is appropriate for a multiple user environment. A lock is placed on the object in the database, so that other users can obtain read-only copies of the object but are prevented from modifying the object while you have it checked out. In addition, the check-out process enables you to determine the configuration of the desired objects as well as the workspace in which to modify the objects.

Tip: When attempting to retrieve baseline configurations during **Check Out** or **Add to Workspace** actions, it is important to keep in mind that checkout places a modifiable copy of a locked commonspace object into the workspace -- not the commonspace object itself -- and the copy is not a member of the baseline. Therefore, to successfully retrieve the baseline configuration of an object, you should select the commonspace version of the object (which you have added, but not checked out to your workspace) as the initially selected object for the action.

During the checkout process, all of the data that defines an object and its relationships is copied to the local workspace. You can specify if the physical files should be copied from the PDM server to the workspace or whether the files should be accessed through a link. Linked files are only retrieved from the PDM server when requested by Pro/ENGINEER. By using linked files, you maintain local copies of only those objects that you have retrieved into a Pro/ENGINEER session after check out. The benefit of checking out using links is quicker check-out transactions because the content files are not downloaded to your workstation. The content files will only be transferred to your workstation when you need them.

Note: If you have multiple workspaces, a checked-out object can only be modified from the workspace where it was originally checked out. The object is inaccessible from any other workspace until it is checked in.

Checking Out Objects from Pro/ENGINEER

When working with a downloaded object in Pro/ENGINEER, the object is strictly read-only. To modify the object, you must first perform a checkout. There are three ways to check out an object in Pro/ENGINEER: from the Pro/ENGINEER menu, from the model tree, and "on-the-fly."

From The Menu

This method uses the **File > Check Out** command.

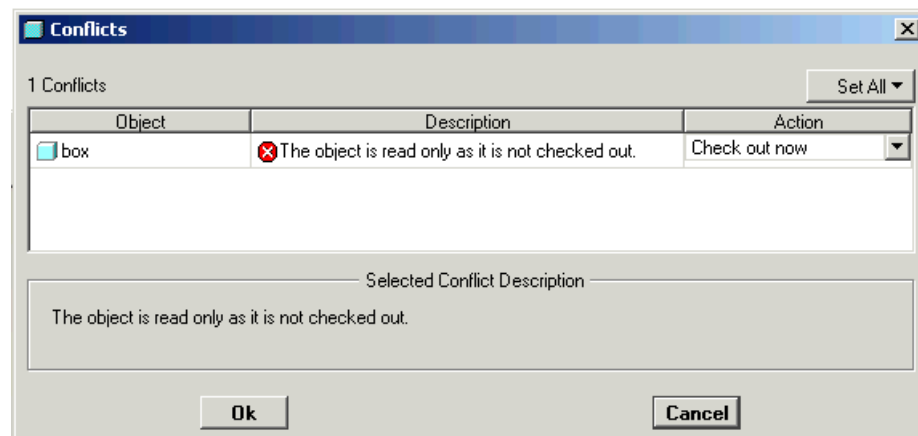
1. In Pro/ENGINEER, click **File > Check Out**. The system prompts you to enter the name of the object that you want to check out.
2. Click the green checkmark button to accept the default value. The downloaded object is checked out to your workspace.

From the Model Tree

1. Right-click on the object in the model tree, then select **Check Out**.
2. The object will be checked out as long as you have the proper permission and it is not checked out by someone else (or yourself in another workspace)

Checkout On-The-Fly

The third method is called "checkout on-the-fly". Checkout on-the-fly prompts you to check out a read-only object whenever you attempt to modify it. When you attempt to modify a read-only object, Pro/ENGINEER displays a **Conflicts** dialog box, indicating that "The object is read only as it is not checked out". The suggested action is "Check out now."



Choose **Ok** and the object is checked out (if you have the proper permission and it is not checked out by someone else, or by yourself in another workspace).

Note: Checkout on-the-fly only works with objects downloaded from a primary server.

Checking Out Objects from the Workspace


Use the **Check Out** action to add objects from Windchill to your workspace for modification. Generally, a working copy of the file is transferred to your local disk, and a lock is placed on the object in the database to prevent simultaneous modification in another workspace.

Refer to the section, [Comparison of Download, Link, and Reuse](#) for an explanation of how content can be handled during a checkout.

Note: When working in Pro/ENGINEER Wildfire, Pro/ENGINEER dependency rules as well as the status of an object (if it is already existing in the workspace) determine which objects are selectable or unselectable for checkout. In addition, you may be able to check out some objects, but only download others, or check out only meta data for yet other objects.

A direct checkout (no user interface involved, only initially selected objects checked

out) occurs when checkout is initiated from the following places:

- Workspace toolbar check out icon 
- Workspace row level action
- Edit Attributes from the workspace
- Check out row level actions
- Save As in workspace toolbar
- Workspace CAD Document Structure Report toolbar
- Pro/ENGINEER **File** menu

Using the workspace **File > Check Out** menu selection allows you to use the **Check Out and Add to Workspace** page to collect and configure an entire set of objects for checkout.

To check out objects using the **Check Out and Add to Workspace** page, use the following procedure:

1. Select objects in your workspace that you want to check out and select **File > Check Out**.

You can also initiate a checkout from other places in Windchill (for example, from an object's information page, or a folders page) by selecting **Check Out** from an **Actions** menu.

The **Check Out and Add to Workspace** page appears. By default, the tool displays the **Basic** (tabbed) version of the collection user interface. You can toggle to the advanced mode of collection by selecting the **Advanced** tab.

The screenshot shows the 'Check Out and Add to Workspace' dialog box with the 'Advanced' tab selected. The 'Workspace' field is set to 'WF_ws1 (Active)' with a 'New...' button next to it. Below the tabs, the '*Check Out:' dropdown is set to 'Initially selected and modified', and the '*Primary Content:' dropdown is set to 'Download'. A checkbox for 'Reuse content in target workspace' is checked. The 'Current Configuration' section shows 'CAD Document Processing, Latest..., Required Dependents' with a link to 'Advanced processing options'. The 'Configuration' section has a 'Change Configuration to:' dropdown set to '--Select--'. The 'Dependencies' section has a 'Dependents:' dropdown set to 'Required'. The 'Collect Related Business Objects' section contains several dropdowns: 'Parts' (None), 'Generics' (None), 'Drawings' (None), 'CAD Documents' (None), and 'Family Table' (None).

2. If you are initiating the check out action from the commonspace, you can select a target workspace in the **Workspace** field. You can also click **New** to bring up the **New Workspace** window and create a new target workspace.
3. Use the collection user interface (basic or advanced mode), which includes options to set a configuration specification, to specify the objects to add, remove, include, or exclude from the checkout object list. For more information on using the collection user interface, see the section [Collecting Objects for PDM Operations](#).

Note: If you are trying to download contents for objects that are already in the workspace, then configuration selection is not necessary. The workspace preference for configuration is used.







In the advanced mode you can use the **Current View** drop-down list to select an existing view or to create a customized view for table objects and their attributes. 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. An adjacent drop-down list lets you select how the table information is displayed.

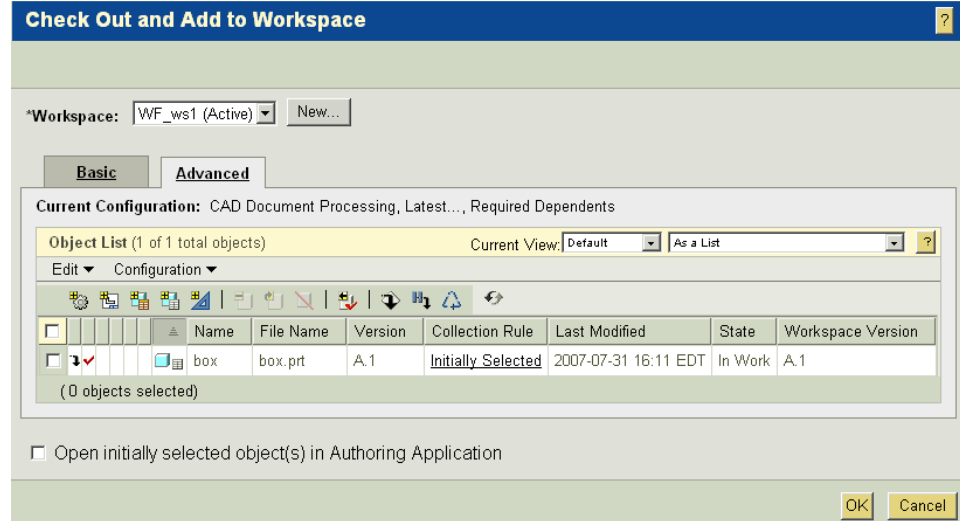
4. If your target workspace is an active workspace (connected to local cache), you can specify how you would like primary content handled upon execution of the checkout, using one of the following methods:




- In the basic mode, select one of the following rules from the **Primary Content** drop-down list (applies to all the objects collected for checkout):
 - Download -- Content is downloaded to workspace upon execution of action.
 - Link -- Only metadata is downloaded to the workspace and a link is created to the commonspace content for later content download, if necessary.

Note: To reuse the contents available in the local cache, select the **Reuse content in target workspace** option. This option is applicable only to the object(s) whose iteration in the workspace matches the iteration in the **Check Out and Add to Workspace** user interface.

- In the advanced mode, specify primary content handling by selecting one or more objects and then applying the appropriate action, using either the **Edit** menu or the table toolbar. The current content-handling setting for each object is indicated by a symbol in the **Workspace content** status column. The following table describes the content-handling controls and indicators:

Edit Menu Selection	Toolbar Icon	Indicator Symbol	Description
Download Content			Content is downloaded to workspace cache upon execution of action.
Add Content as Link			Only metadata is downloaded to the workspace and a link is created to the commonspace content for later content download, if necessary.
Reuse Content			<p>No content is downloaded. Instead, the content on the version currently in the target workspace is used.</p> <p>Note: This option is applicable only to the object(s) whose iteration in the workspace matches the iteration in the Check Out / Add to Workspace user interface.</p>



5. Set objects to be checked out, as follows:
 - In basic mode, set a rule by selecting one of the following types of objects from the **Checkout** drop-down list (rule applies to all objects collected):
 - Selected and modified -- Initially selected objects and any dependents in the workspace with modified status will be checked out
 - Selected -- Only the initially selected objects will be checked out
 - Required -- Initially selected objects and any required dependents will be checked out
 - All -- Initially selected objects and all dependents will be checked out
 - In advanced mode, select table objects and then **File > Set for Checkout** or click the checkout icon  in the table toolbar (Clicking the icon  a second time toggles off the selection).
6. Optionally, to remove your specifications for object collection and handling, and return the table to its original state, click the reset icon  (in advanced mode only).
7. If your target workspace is an active workspace, you can select the **Open initially selected object(s) in Authoring Application** check box to automatically open each initially selected object in an individual Pro/ENGINEER session window upon execution of the add action.

8. Click **OK**. The selected objects are checked out and added to the target workspace as you have specified, and you are returned to the user interface from which you initiated the check out action.

Comparison of Download, Link, and Reuse

When checking out or adding objects to your workspace, your choice of download options (download, link, or reuse) should be guided by the current workspace situation and your intent. If your intent is to open the current server-stored model in Pro/ENGINEER at some point in time, the **Download** option should be selected, as it will download the content in a much more efficient manner with fewer performance concerns. Selecting **Download** does, however, overwrite any locally cached modifications, if they exist.

If you select the **Link** option during checkout, file data for that object is not downloaded during the checkout, but it will be downloaded at some subsequent time if it is requested by Pro/ENGINEER. Whether a subsequent download is triggered or not, the server content is identified as the content of the object. Therefore, the link option can be used in scenarios where you want to overwrite (or have no) locally cached modifications of the CAD document, and have no intention to open the model in Pro/ENGINEER (for example, you might want to modify the model parameters through the Windchill **Edit Properties** window).

If the file already exists, is modified in your local cache, and you select **Reuse**, content for that object is not downloaded. Instead, the locally modified content becomes the content of the "added" object iteration or working copy. In other words, the reuse option is only applicable if you already have modified object content in your local cache.

Checking Out an Earlier Iteration

It may become desirable to revert to an earlier design for a CAD document. You can use the **Iteration History** of the latest iteration of the object to identify an earlier iteration, navigate to its information page, and initiate a checkout. In general, you can initiate the checkout of a non-latest iteration from the same places you would initiate the checkout of the latest iteration. Exceptions are the row-level check-out actions available in places such as the workspace page, the workspace **Edit Attributes** page, and the **Check In** page.

Note: The checkout of an earlier iteration is not supported for Windchill part objects.

Check out of an earlier iteration of a CAD document is supported, subject to the following conditions:

- You receive a warning that the iteration you are attempting to check out is not the latest.
- If you perform the checkout using the Check Out/Add to Workspace page, a conflict message saying an iteration other than the latest cannot be checked

out will be displayed in the **Event Manager**. This is an overridable conflict that can be overridden using the **Conflict Manager**.

- No other iteration of the object can be currently checked out. Also, when you check out a non-latest iteration, all other iterations of the object are adorned with a "checked-out" symbol to indicate that no other iterations can be simultaneously checked-out. On the **Iteration History** report, to avoid confusion, only the iteration actually checked out displays the yellow checked-out-by-you symbol (☑). The iteration which was latest before the non-latest checkout shows a checkmark on a gray background to indicate that it is unavailable for checkout.
- When gathering related objects of an initially selected object that is not the latest iteration, the default configuration is the As Stored configuration for the initially selected object (not Latest). This can be modified by the Windchill preference, Set Configuration for Check Out.
- Upon checking the object back in, it becomes the latest iteration. The **Iteration History** report records the earlier iteration it was derived from.
- When checked out, the object has both the content (only if a CAD document) and meta data of the earlier iteration. When checked in, however, the earlier iteration is assigned the life cycle state of the iteration that had previously been the latest.
- When the check out of an earlier iteration requires overriding an overridable conflict, you need to explicitly refresh the workspace in order to properly view the earlier iteration.
- Checking out an earlier iteration of a CAD document to a project is not allowed.
- Checking out an earlier iteration from a project to a workspace is allowed, so long as:
 - The version is native to project, or is a one-off version checked out to the project.
 - The object is not checked out by any individual.
 - The version is not shared to the project from Windchill PDMLink.
- Association between objects can be affected when checking in a non-latest iteration to become the latest, as follows:
 - Uses link -- Links between parent and children in a structure are maintained
 - Content link -- If a part and CAD document associated by a content link are iterated (checked in) separately, the new iteration retains the association to the peer object it was originally associated with. If the part and CAD document are both iterated, then a new content link is formed between the two peer objects.

- Owner link -- If a CAD document is iterated, and if its latest iteration is associated by an owner link, its peer part is also iterated by the system. Any properties that are driven from CAD document (such as attributes or structure) will be derived by the originally checked out CAD document, and the rest of the properties that are specific to part will be derived from the latest iteration of the part.

Undoing Check Out


There are multiple reasons for deciding to undo a checkout. You may want to discard the latest changes to an object and return to the version of the object stored in Windchill, or you may simply want to remove the lock that your checkout has placed on the object.

When you undo a checkout, changes you have made to the content and meta-data of the object are discarded and the content as stored in Windchill is downloaded to the workspace. You can choose not to have the Windchill content downloaded.

If you select a member of a Pro/ENGINEER family table for undo check out and it is modified, then you need to cancel checkout on all members of the family table because they share the same Pro/ENGINEER file.



You cannot undo checkout on a modified assembly, unless its modified children are also on the undo check out list. Changes made in the assembly may require changes made in their components.

To undo the checkout of one or more objects, use the following procedure:

1. From the active workspace, click the check box next to an object whose checkout you want to undo.
2. In the workspace menu bar, select **File > Undo Check Out** or click the undo checkout icon  on the workspace tool bar.

The **Undo Check Out** page appears displaying your initially selected object(s) and any dependents collected by default in a table listing. You can use the **Current View** drop-down list to select an existing view or to create a customized view for table objects and their attributes. 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. An adjacent drop-down list lets you select how the table information is displayed.

Note: You can also initiate an undo of checkout from other places in Windchill (for example, from a checked-out object's information page, or a folders page) by selecting **Undo Check Out** from an **Actions** menu.

3. You can collect, remove, include, or exclude objects for the **Undo Check Out** action using the advanced mode of the collection user interface (the basic collection mode is not available for this action), which includes options to set a configuration specification. For more information on using the collection user interface, see the section [Collecting Objects for PDM Operations](#).
4. Select an object and then click the download icon  in the toolbar to toggle your preference for content handling. Your current preference is shown in the **Workspace content status** column, as follows:
 - (Download) -- Indicates that server content is to be downloaded to the workspace upon completion of the undo of checkout.
 - (No symbol shown) -- Indicates that no content is to be downloaded to the workspace.
5. Optionally, you can click the reset icon  to revert the **Object List** to listing the objects it contained upon initiation of the action.
6. Click **OK**.

The checkout of the selected objects is undone.

Some additional considerations are as follows:

- After undoing a checkout, the system asks whether to replace the object in the Pro/ENGINEER session with the unmodified object in the workspace.
- To cancel the checkout of an object in an active Pro/ENGINEER session, select **File > Undo Check Out**.
- To cancel the checkout of an object from the Model Tree, right-click the object and select **Undo Check Out** from the pop-up menu.

Adding Objects to the Workspace

The download action in Pro/ENGINEER and the **Add to Workspace** action in Windchill enable you to bring read-only copies of objects into your workspace. This allows you to examine the object without placing a lock on it. If you attempt to modify such an object, the system will prompt you to check it out.

When using the **Add to Workspace** action in Windchill, you have options as to how content files are handled. Refer to the section, [Comparison of Download, Link, and Reuse](#) for an explanation of how content can be handled when adding objects to a workspace.

Initiating a Download from Pro/ENGINEER

In Pro/ENGINEER, you can use the **File Open** dialog box to browse your workspace and download objects from the PDM server to your session of Pro/ENGINEER.

1. In Pro/ENGINEER, click **File > Open**. The **File Open** dialog box opens.
2. Use the **Look In** list to select the workspace or commonspace area that you want to browse. Once selected, a workspace's contents are visible in the file area.
3. Select the object that you want to download and click **Open**. The selected object downloads to Pro/ENGINEER and opens.


Note: Downloaded objects are read-only. To modify a downloaded object, you must first perform a checkout.

Initiating Add to Workspace from the Workspace

Use the **Add** action (**Add to Workspace** when accessed from the commonspace) to update workspace objects, or to add objects from the commonspace to your workspace. The content files associated with the objects can optionally be downloaded to your local disk for read access by you. In the Windchill **Preference Manager** user interface, you can set preferences for Add to Workspace and Check Out behavior.

Note: You need "Download" access permission to add objects to the workspace. Pro/ENGINEER dependency rules and an object's status (if it is already existing in the workspace) determine which objects are selectable or deselectable for download. In addition, you may be able to download some objects, check out others, or check out only meta data for yet other objects.

To add objects to your workspace, do the following:

1. In the workspace, select **File > Add**, or select the add icon  in the toolbar.

If you are initiating the add to workspace action from a workspace, and have not pre-selected objects already in your workspace, upon initiating the add to workspace action the **Select objects to download page** appears to allow you to select objects that reside in Windchill commonspace locations.

You can also initiate the add action from other places in the commonspace (for example, from an object's information page, search results, the clipboard, or a folders page) by selecting the **Add to Workspace** action from an object's **Actions** menu. In these cases, using the **Select objects to download page** is unnecessary.

When you have completed your selection of objects, the **Add to Workspace** page appears. By default, the page displays the basic mode of the collection user interface.

Add to Workspace

Workspace: VF_ws2 (Active) New...

Basic **Advanced**

*Check Out: None

*Primary Content: Download

☒ Reuse content in target workspace

Current Configuration: CAD Document Processing, Latest..., Required Dependents [Advanced processing options](#)

Configuration

Change Configuration to: --Select--

Dependencies

Dependents: Required

Collect Related Business Objects

Parts: None Generics: None

Drawings: None CAD Documents: None

Family Table: None

* Indicates Required Fields.

☐ Open initially selected object(s) in Authoring Application







OK Cancel

2. You can toggle to the advanced mode of collection by selecting the **Advanced** tab. In the advanced mode you can use the **Current View** drop-down list to select an existing view or to create a customized view for table objects and their attributes. 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. An adjacent drop-down list lets you select how the table information is displayed.
 3. If you are initiating the add to workspace action from the commonspace, you can select a target workspace in the **Workspace** field. You can also click **New** to bring up the **New Workspace** window and create a new target workspace.
 4. Use the collection user interface (basic or advanced mode), which includes options to set a configuration specification, to specify the objects to add, remove, include or exclude from the object list. For more information on using the collection user interface, see the section [Collecting Objects for PDM Operations](#).
- Note:** If you are trying to download content for objects already in the workspace, then configuration selection is not necessary. The workspace preference for configuration is used.
5. If your target workspace is an active workspace (connected to local cache), you can specify how you would like primary content handled upon execution of the checkout, using one of the following methods:
 - In the basic mode, select one of the following rules from the **Primary Content** drop-down list (applies to all the objects collected for checkout):
 - Download -- Content is downloaded to workspace upon execution of action.

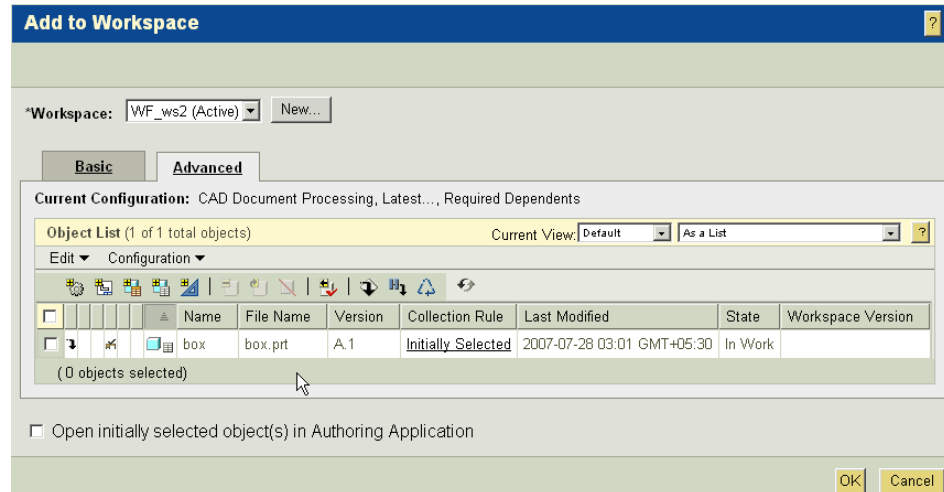
- **Link** -- Only metadata is downloaded to the workspace and a link is created to the commonspace content for later content download, if necessary.




Note: Note: To reuse the contents available in the local cache, check Reuse content in target workspace option. This option is applicable only to the object(s) whose iteration in the workspace matches the iteration in the Check Out / Add to Workspace user interface.

- In the advanced mode, specify primary content handling by selecting one or more objects and then applying the appropriate action, using either the **Edit** menu or the table toolbar. The current content-handling setting for each object is indicated by a symbol in the Workspace content status column. The following table describes the content-handling controls and indicators:

Edit Menu Selection	Toolbar Icon	Indicator Symbol	Description
Download Content			Content is downloaded to workspace cache upon execution of action.
Add Content as Link			Only metadata is downloaded to the workspace and a link is created to the commonspace content for later content download, if necessary.
Reuse Content			No content is downloaded. Instead, the content on the version currently in the target workspace is used. Note: This option is applicable only to the object(s) whose iteration in the workspace matches the iteration in the Check Out / Add to Workspace user interface.

Note: If an object included in the list is modified but not checked out, and you choose to download the file by using the download icon in the choices column, then your changes are overwritten.





6. Set objects to be checked out, as follows:
 - In the basic mode, set a rule by selecting one of the following types of objects from the **Checkout** drop-down list (rule applies to all objects collected):
 - Selected and modified -- Initially selected objects and any dependents in the workspace with modified status will be checked out
 - Selected -- Only the initially selected objects will be checked out
 - Required -- Initially selected objects and any required dependents will be checked out
 - All -- Initially selected objects and all dependents will be checked out
 - In the advanced mode, select table objects and then **File > Set for Checkout** or click the checkout icon  in the table toolbar (Clicking  a second time toggles off the selection).
7. Optionally, to remove your specifications for object collection and handling, and return the table to its original state, click the reset icon  (in advanced mode only).
8. If your target workspace is an active workspace, you can select the **Open initially selected object(s) in Authoring Application** check box to automatically open each initially selected object in an individual Pro/ENGINEER session window upon execution of the add action.
9. Click **OK**. The selected objects are checked out and added to the target workspace as you have specified, and you are returned to the user interface from which you initiated the check out action.

Removing Objects from the Workspace

In the process of using your workspace, you will create, open, and download many objects. Each object that you create, open, or attempt to modify is added to your workspace. After a while, you may find that your workspace has become cluttered with old or unused objects that you want to remove. To remove the unnecessary objects from the workspace, you must initiate the action.

To remove objects from a workspace:

1. Select one or more objects that you want to remove from the workspace.
2. Select **File > Remove** or click the remove icon  in the toolbar. The **Remove from Workspace** page appears, displaying your initially selected object(s) in a table listing.
3. You can collect, remove, include, or exclude objects for the Remove action using the advanced mode of the collection user interface (the basic collection mode is not available for this action), which includes options to set a configuration specification. For more information on using the collection user interface, see the section [Collecting Objects for PDM Operations](#).
4. Optionally, click the reset icon  to revert the **Object List** to listing the objects it contained upon initiation of the action.
5. In an active workspace (using an embedded browser in a CAD application session), you have the option to **Erase object(s) from CAD Application** session. When selected, the objects removed from the workspace are also removed from Pro/ENGINEER session.
6. Click the **OK** button at the bottom of the **Remove from Workspace** page.

The selected objects are removed from your workspace.

Note: If you remove a checked-out object from your workspace, the check-out is undone, and any local modifications made to the object are lost.

Importing Objects to the Workspace

You can use your active workspace to load CAD objects into your workspace without explicitly retrieving them into Pro/ENGINEER. This functionality is supported only for Pro/ENGINEER CAD documents; which can include file types that are supported by Pro/ENGINEER but cannot be opened directly, such as material files or texture files, which become CAD documents upon import.

Overview of Importing Objects

Import involves invoking the import user interface, initially selecting objects in a source location for import, using collection rules to gather related objects, and, optionally, specifying additional options for how the objects should be handled by the system upon committing the import.

The following are important aspects of import:

- While objects are being imported, Pro/ENGINEER session is frozen.
- Setting a preference allows you to specify that secondary (attached) content is transferred along with the primary file.
- One object can cause the failure of import. Conflicts are reported in the **Event Manager**.
- If the object already exists in the commonspace or workspace, its status is shown in status columns.
- If a component of an assembly stored in the local file system has been moved to a location other than where the rest of the assembly is stored, the import of the assembly from the original location will show the moved component as an incomplete dependent object (ghost), and the component's file path will be displayed as its former location.

Import creates CAD documents in the target workspace with primary content. An imported object's file name becomes its CAD name. The system tries to establish appropriate dependencies between new and existing CAD documents, if there is an object with the same name already in the target system.

Import supports the following functionality:

- Display of the object status and conflicts (provided the object already exists in the commonspace or workspace)
- Electing either to reuse objects that are already in the commonspace or workspace, or to overwrite with an imported object from a source directory. The system provides default object handling settings by checking object database status or user privileges.
- Check out of objects upon import if you are importing objects that are already on the server.
- Reuse of a server version of an object. If that version is not in the workspace, you have an option not to add it to the workspace upon import (by setting a preference).
- Attaching secondary content from the local directory

Note: If any dependents of an imported object are not imported and not existing in the database, they appear in the workspace as incomplete objects. (regardless of the nature of dependency, required or optional). Any resolution of incomplete objects should be done during a subsequent upload or checkin.

See the Customizing and Administering Pro/ENGINEER Wildfire chapter of this guide for information about preferences for search paths, automatic download, or allowing attachment of secondary content.

Performing an Import

The Import to Workspace wizard has the following two steps:

- **1: Select files to import**
- **2: Specify Options**

You can navigate to a subsequent step by clicking **Next**. Navigate to a preceding step by clicking **Back** or clicking the gray box corresponding to a previously completed step.

1: Select Files to Import

To select objects to import to a workspace, use the following procedure:

1. Select **Tools > Import to Workspace**.

The **Import to Workspace** window opens at **Step 1: Select files to import**.

2. The **Target Workspace** field indicates the active workspace from which you invoked the import and into which the imported objects will be placed.
3. In the **Add Dependencies** drop-down list, select a rule for dependent objects to be collected with any selected objects, either All (default), Required, or None.

Import to Workspace ?



Steps: 1: Select files to import 2: Specify Options



Select files from local file system

Target Workspace:



Add Dependencies:

Files (1 objects)

<input type="checkbox"/>		File Name	Size	Last Modified	File Path
<input type="checkbox"/>		0000000082.asm	45.3 KB	2006-10-02 22:36 EDT	D:\Wildfirefiles\

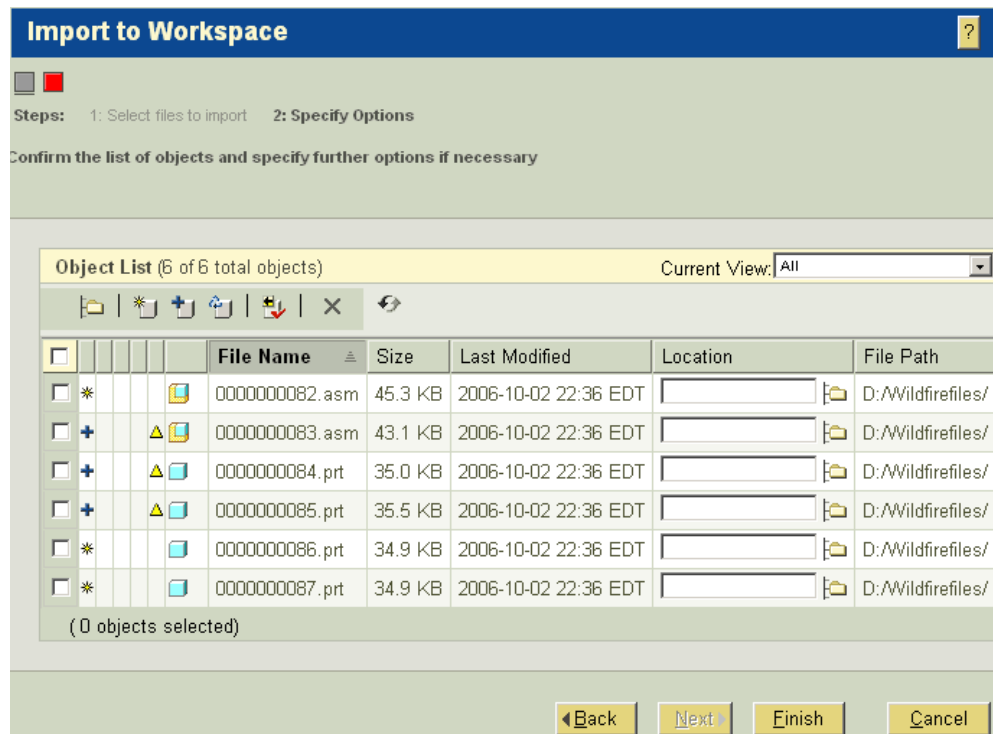
(0 objects selected)

4. Click the add object icon  to navigate to objects you would like to import from your local file system. When you have navigated to the object, click **Open** to add it to the **Files** table. The **Files** table lists the files you have collected for import.
5. Optionally, select an object and click the remove object icon  to remove the object from the table.
6. Click **Next** to go to **Step 2: Specify Options**. Or click **Finish** to initiate the import without specifying options.






2: Specify Options


The options available in the second step of import can help resolve conflicts with objects in the target workspace or commonspace. Use the following procedure to specify import options:

1. On the **Specify Options** step, the objects you selected in the first step are listed in the **Object List** table.



2. In the **Current View** drop-down list, you can select from the following alternative views of the **Object List** table:
 - All except Family Instances
 - All
 - All with conflicts
3. You can further modify the collected set of objects by selecting objects and using toolbar controls as explained in the following table:

Control Icon	Action
	Invoke the Set Location dialog box to select a database location for the selected object
	Add the imported object as a new object
	Add the imported object as a modified object
	Reuse an existing workspace or commonspace object of the same name, rather than importing the selected object.
	Check out the selected object upon import

Control Icon	Action
	Exclude selected object from the import. Clicking again (while the same object is selected) re-includes the object.

4. Clicking the reset icon resets the **Object List** to its initial state.
5. You can click **Back** at any time to return to the **Select files to import** step.
6. Click **Finish** to commit the import of the collected objects.

The imported objects are populated in the target workspace.

Exporting Objects from the Workspace

You can use your active workspace to load CAD objects out of your workspace without explicitly retrieving them into Pro/ENGINEER. This functionality is supported only for Pro/ENGINEER CAD documents; which can include file types that are supported by Pro/ENGINEER but cannot be opened directly, such as material files or texture files, which become CAD documents upon import.

Overview of Exporting Objects

Export is accomplished by initially selecting CAD objects in your active workspace (embedded browser only), using collection rules to add related objects to the set, defining a target directory for the exported objects, and specifying additional options for how the objects should be handled by the system upon committing the export.

The following are important aspects of export:

- While objects are being exported, Pro/ENGINEER session is frozen.
- Setting a preference allows you to specify that secondary (attached) content is transferred along with the primary file.
- One object can cause the failure of entire export. Conflicts are reported in the **Event Manager**.
- If the object already exists in the commonspace or workspace, its status is shown in status columns.

Export supports the following functionalities:

- Dependency processing.
- Target location setting
- Optionally confirming the list of objects to be exported
- Ability to also attach secondary content from the local directory (by setting a preference)

- Automatic download of objects not currently in the workspace (by setting a preference).
- Electing whether to overwrite or reuse objects that already exist in the target location

To export objects from a workspace, you can use either the **Basic** mode or the **Advanced** mode of collection. In the **Basic** mode you can specify rules that determine what set of objects are to be included in the export in addition to the initially selected objects. In the **Advanced** mode, you can use collection and configuration tools to incrementally add or remove objects to or from a table listing of objects, prior to committing the action. You can move from one tab to the other; however, returning to the **Basic** tab may remove objects you have collected while using the **Advanced** tab.

Exporting Using the Basic Mode

Use the following procedure for the basic mode:

1. Select one or more objects to export, and select **Tools > Export from Workspace**. The **Export CAD Documents** window opens, displaying the **Basic** tab, by default. The **Basic** mode offers the following fields for you to specify which dependencies or related business objects you want to include in the export:
 - **CAD Document Dependents** -- Specify Required (default), All, or None
 - **Drawings** -- Specify None (default), All, or Initially Selected Only
2. In the **Target Directory** field, enter the path to the target directory, or click **Browse** to navigate to a target directory.
3. Select the **Reuse content in target directory** check box if you want to avoid overwriting files of the same name in the target directory.
4. Optionally, select the **Secondary Content** checkbox to export secondary content with the CAD documents.
5. Click **OK** to export the objects as you have specified.





Exporting Using the Advanced Mode



Use the following procedure for the advanced mode:

1. From the **Basic** tab of the **Export CAD Documents** page, select the **Advanced** tab.

Your initially selected objects plus any additional objects collected according to the specifications on the **Basic** tab appear in the **Collect Objects** table. You can use the **Current View** drop-down lists to select an existing view or to create a customized view for table objects and their attributes. 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. An adjacent drop-down list lets you select how the table information is displayed.

2. You can further modify the collected set of objects by selecting objects and using toolbar controls as explained in the following table:

Control Icon	Action
	Add any related drawings for the selected object(s)
	Remove selected object(s) from the table
	Include selected (excluded) object(s)
	Exclude selected object(s) from action. (Indicated by strike-through)

3. Select an object and click the reuse icon  to specify not to overwrite a file with the same file name in the target directory, but rather to reuse the file content in the target directory.
4. Optionally, click the reset icon  to reset the table to its initial collection of objects.

You can select the **Basic** tab at any time; however, you lose additional objects collected on the **Advanced** tab.

Click **Finish** to commit the export of the collected objects. The exported objects are populated in the target folder.

Keeping Workspace Objects Up-to-Date

If you work in a multi-user environment, you are likely to encounter the situation where changes to product data are made by others while you have the objects in your workspace. In order to manage this dynamic situation, Pro/ENGINEER Wildfire can notify you of changes in object status and allows you to update selected or any changed workspace objects to be sure that you remain current with the latest server information. Status information regarding whether a workspace object is out-of-date is communicated via three optional status columns, which you can add to your workspace table view (see the section, [Customizing Object Display with the Table View Manager](#)). The status columns are the following:

- Out of Date status
- Out of Date with Workspace Configuration status
- Compare status

For more information on these status columns, see the section [Object Status](#)


The workspace offers two actions, Update and Synchronize Workspace, that you can use to keep your workspace current. The Update action is used when modifications to an object made by other users or by you in another workspace may cause your current workspace object to become out-of-date as compared to the default CAD document and part configuration specification (defined in the Workspace Preferences). The default workspace configuration specification is the latest iteration on the latest revision.

You may select one or more objects to update (for example, when their status column symbols indicate they are out-of-date). In addition you can select the Update action without preselecting an object (action-object). In this case, the **Update** page is automatically populated with any out-of-date objects from the workspace. After you specify how you would like the content handled (for example, whether or not to update, download, or link for download as needed), executing the update adds the latest iteration of an out-of-date object to the workspace. For more information, please see the section [Updating Workspace Objects](#).

The Synchronize Workspace action updates the local cache with the latest information for objects already in the workspace (for example, modifications made from a standalone browser) and is described in the section [Refreshing the Cache](#).



Updating Workspace Objects





To update objects in your workspace, perform the following procedure:


1. Select the objects in your workspace that you want to update (or for action-object update, select no objects) and select **File > Update** or click the update icon  in the toolbar.

The **Update** page appears, displaying any objects that you or the system selected for update, plus a default collection of dependent objects, are listed in the **Object List** table. You can use the **Current View** drop-down list to select an existing view or to create a customized view for table objects and their attributes. 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. An adjacent drop-down list lets you select how the table information is displayed.

2. The **Current Configuration** field above the **Object List** table displays the current configuration settings by listing the current values for **Configuration from**, **Configuration**, and **Dependents**, respectively. If you are satisfied with the default collection of objects, skip to Step 5 of this procedure. Otherwise, modify the set of collected objects using one or more of the following methods:
 - Use the **Configuration** cascading menu in the **Object List** table to reset the configuration specification.
 - Use the collection action icons in the object-action (shaded) portion of the toolbar to add, remove, exclude, or include available related objects.
3. If your target workspace is an active workspace (connected to local cache), you can specify how you would like primary content handled upon execution of the update by selecting one or more objects and then applying the appropriate action, using either the Edit menu or the table toolbar. The current content-handling setting for each object is indicated by a symbol in the **Workspace content** status column. The following table describes the content-handling controls and indicators:

Edit Menu Selection	Toolbar Icon	Indicator Symbol	Description
Download Content			Content is downloaded to workspace cache upon execution of action.

Edit Menu Selection	Toolbar Icon	Indicator Symbol	Description
Add Content as Link			Only metadata is downloaded to the workspace and a link is created to the commonspace content for later content download, if necessary.
Reuse Content			No content is downloaded. Instead, the content on the version currently in the target workspace is used. Note: This option is applicable only to the object(s) whose iteration in the workspace matches the iteration in the Check Out / Add to Workspace user interface.

- Optionally, to remove your specifications for object collection and return the list to its original state, click the reset icon .
- Click **Finish**. The selected objects are updated as you have specified, and you are returned to the user interface from which you initiated the update action.

Tip: Use the Pro/ENGINEER configuration option `OVERWRITE_CONTENTS_ON_UPDATE` to control behavior during the Update action from the Pro/ENGINEER user interface. Note that "yes" is the default setting.

- If set to "no," the system does not overwrite the locally modified contents for out-of-date objects, but updates their metadata only
- If set to "yes," the system overwrites the locally modified or out-of-date objects with the ones in the server in addition to updating their metadata




Refreshing the Cache
















You can use the **Synchronize Workspace** action to explicitly refresh all workspace objects that have become stale in the cache due to more recent changes made on the server by another user or by you in a standalone workspace. One example would be if you used a standalone browser (no cache awareness) to modify attributes on a workspace object. An explicit synchronization with the server would be required to communicate those changes to the local cache.




There are two types of synchronization of cached information with the server: implicit and explicit synchronization.

- Implicit synchronization occurs whenever the client makes a request to the server (Check Out, Upload, and so on), and refreshes information in the following areas:
 - Change to the status "Checked out by you" in this workspace
 - Addition of an object into this workspace
 - Removal of object from this workspace
- Explicit synchronization occurs when you select **Tools > Synchronize**. Explicit synchronization refreshes information in the following areas:
 - Everything that implicit synchronization synchronizes (mentioned above)
 - Update of File Name in the local cache
 - Update of attributes on the object's master version in the database
 - Change to the status "Checked out by another user", "Checked out by you in another workspace"

For performance considerations, not all statuses are updated upon every invocation of the workspace listing. The following table explains which statuses require explicit synchronization (**Tools > Synchronize**) of the workspace with server information:

Status Category	Status		Requires Explicit Synchronization?
Share Status		Shared to a project	Yes
		Shared from PDM	Yes
		Checked-out from PDM	Yes

Status Category	Status		Requires Explicit Synchronization?
General Status		Locked (embedded browser only)	N/A (cache-only status)
		Checked out by you	No
		Checked out by you in another workspace	Yes
		New	No
		Checked Out by another user	Yes
		Checked-out to a project	Yes
		Another Iteration is checked out by you	Yes
		Another Iteration is checked out by another user	Yes
Modified Status		Modifications Need to be Uploaded (Embedded browser only)	N/A (cache-only status)
		Modifications Uploaded	No (derived from server)
		Modifications Not Eligible for Upload (Embedded browser only)	N/A (cache-only status)
Out of Date Status		Out of Date modified by you	No (dynamically computed on each workspace listing invocation)
		Out of Date modified by other (user name)	No (dynamically computed on each workspace listing invocation)
Out of Date with Workspace Configuration		Out of date with Workspace configuration - Modified by you	No (dynamically computed on each workspace listing invocation)
		Out of date with Workspace configuration - Modified by another user (user name)	No (dynamically computed on each workspace listing invocation)

Status Category	Status		Requires Explicit Synchronization?
Compare Status		Critical Error: File name conflict	No
		Error: File name changed	No
		Warning: Removed from Workspace	No

To explicitly synchronize the workspace with the server, perform the following procedure:

1. Select **Tools > Synchronize**.

The system updates all of the information about the workspace and the objects in the workspace with the latest changes made on the server by another user or by you in another workspace.

Revising Workspace Objects

You can create a new revision for an object by assigning the next revision level available in a revision scheme. The revision scheme represents a sequence of characters identifying subsequent versions of a revisable object. Creating a new revision of an object results in the object, and all objects you choose to associate with it, being incremented to the set revision level when the revision operation completes. When you revise an object, the latest version of that object is used as the content for the new revision.

Use the **Revise** action to create a new revision of an object. Typically, this is done to initiate a new branch of the design, based on the current object. You can revise objects that are checked in to the database, or checked out by others.

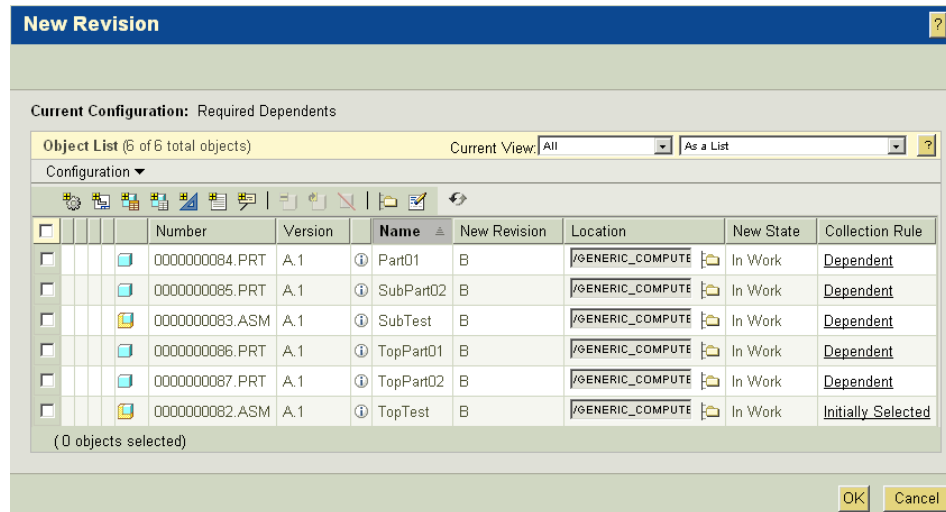
Note: Checked-out objects cannot be revised. Revise is not available for objects in a project context.



To create a new revision:

1. Select **New Revision** from an **Actions** list, such as from the **Actions** list on the object's information page. Alternatively, in a workspace you can select an object and then select **File > New > Revision** or click the new revision icon



The **New Revision** window appears and features a collection table. You can collect, remove, include, or exclude objects for the new revision action using the advanced mode of the collection user interface (the basic collection mode is not available for this action, nor are configuration setting tools). For more information on using the collection user interface, see the section [Collecting Objects for PDM Operations](#).



- Depending on the preference settings at your site, you may be able to assign a revision level different from the default. Set the revision level for individual objects by clicking the set revision icon  next to each object for which you want to change the revision or for multiple objects by selecting multiple objects and then clicking the set revision icon on the toolbar.
- You can also modify the location for any or multiple selected objects, by clicking the set location icon .
- Click **OK** to perform the revise operation. If the operation fails, the **Event Manager** window displays and indicates the reason for the error.

Setting a Revision

If enabled by a server-side preference, the **Set Revision** dialog box allows you to select a specific revision level for the object(s) you are revising. Your ability to select a revision level can depend on the relative revision levels of parts and CAD documents, the nature of any associations, and the revision scheme for the object. Details on the factors affecting your ability to set a revision level are explained in the chapter, Customizing and Administering Pro/ENGINEER Wildfire with Windchill, in this guide.

To set a revision, use the following procedure:

1. On the **Revise** page, select the object(s) whose revision level you want to set and click **Set Revision**. The **Set Revision** dialog box appears.
2. Select a revision level from the drop-down list or enter a valid (later) level in the text box.
3. Click **Ok**. The **Set Revision** dialog box is dismissed. Your selected values are assigned when the revise action is completed.

Using the Event Manager

Many transactions between Pro/ENGINEER Wildfire and a Windchill server happen asynchronously, allowing you to initiate an operation (such as Check In or Upload) and continue working in Pro/ENGINEER while the operation is processed.


The Event Manager provides a way for you to check and act on log messages generated in your Pro/ENGINEER or Windchill sessions. It can be accessed from Pro/ENGINEER by clicking on the console status icon in the status bar. In addition, by selecting **Tools > Console** in the Pro/ENGINEER UI, you can access the console of any server to which you are connected. From a workspace, the Event Manager is accessed by selecting **View > Event Manager**. In Windchill PDMLink and Windchill ProjectLink you can also access it by selecting **Home > Utilities > Event Manager**.

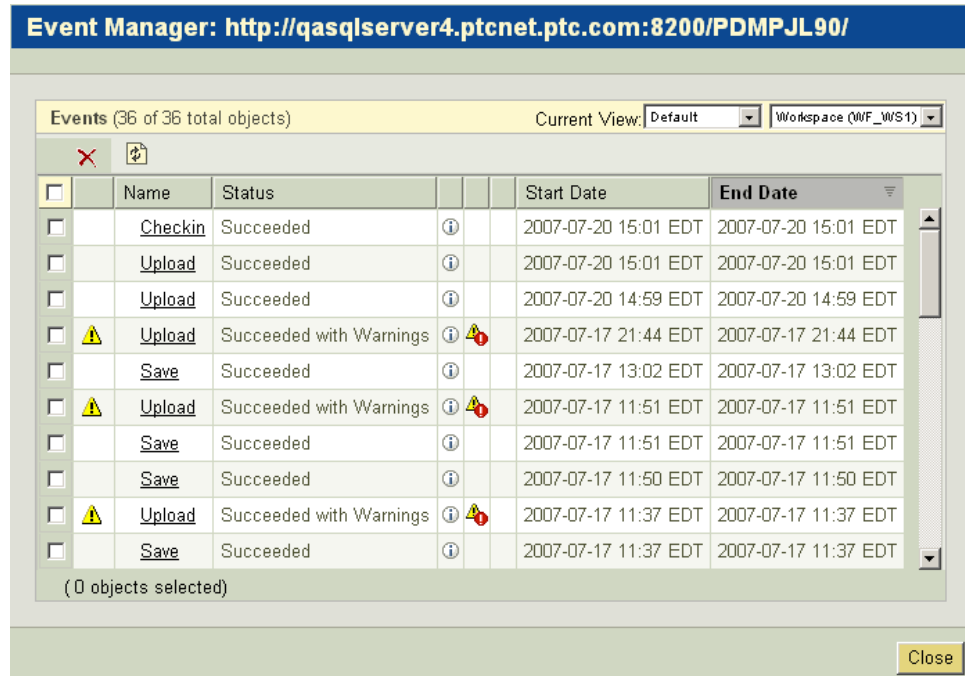
The **Event Manager** launches automatically in the case of a failed transaction attempt that is initiated from the workspace or PDM system. In the case of a failed transaction that is initiated from the Pro/ENGINEER user interface, an event manager status icon appears in the Pro/ENGINEER status bar.

Using the Event Manager is particularly useful when performing transactions with a large number of objects or when working in a multi-server environment, because you can access information specific to a server.

The Event Manager consists of three interlinked pages that allow you view, get information on, and resolve conflicts arising from PDM transactions. The three pages are the **Event Manager** page, the **Event Information** page, and the **Conflict Manager** page.

Event Manager Page

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



The **Events** table also has a tool bar with commands for the following actions:

- **Delete** -- Deletes selected row(s) from the table
- **Refresh** -- Refreshes the event listing in the table




Event Types

The type column in the **Event Manager** page contains icons that identify the particular type of event listed. The event type icons are described as follows:


- -- Overridable type conflict
- -- Non-overridable type conflict (Failed)
- -- Warning
- -- In Progress
- -- Pending
- -- Retried
- -- On Hold

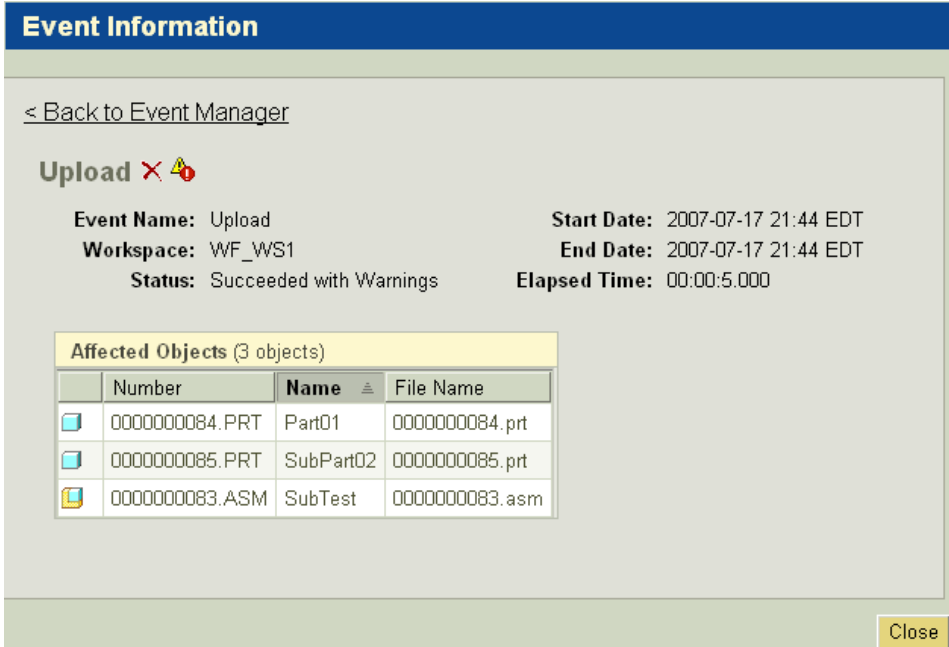
Event Actions

The Actions column in the **Event Manager** contains icons that can call either the **Event Information** page or the **Conflict Manager**, and are as follows:

-  -- View event information in the **Event Information** page
-  -- View warnings or errors in the **Conflict Manager**
-  -- Resolve Conflicts in the **Conflict Manager**




Event Information Page


The **Event Information** page is accessed from the **Event Manager** by clicking  in the Actions column for the event in the **Events** list, or by selecting the link in the event name.



Number	Name	File Name
0000000084.PRT	Part01	0000000084.prt
0000000085.PRT	SubPart02	0000000085.prt
0000000083.ASM	SubTest	0000000083.asm



At the top of the page is a hyperlink that returns you to the **Event List** in the **Event Manager**. The area immediately below the hyperlink lists the event attributes, as follows:

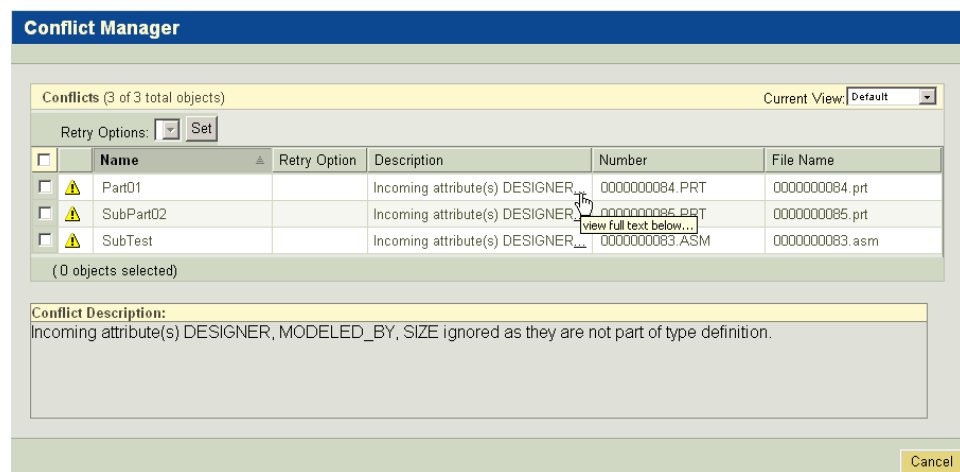
- **Event Name** -- The name of the event. Next to the event name are icons for the Delete action () and, if warnings or conflicts occurred, one of the following actions, as applicable to the type of event:
 - **View Warning**  -- If the event type is a warning
 - **View Conflicts**  -- If the event type is a non-overridable conflict

- **Resolve Conflicts**  -- If the event type is an overridable conflict
- **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)

Conflict Manager

The **Conflict Manager** assists you in viewing and resolving conflicts that arise from PDM events. It is accessed from the **Event Manager** or the **Event**

Information page by clicking the review conflicts icon  or the resolve conflicts icon 



Name	Retry Option	Description	Number	File Name
Part01		Incoming attribute(s) DESIGNER...	0000000084.PRT	0000000084.prt
SubPart02		Incoming attribute(s) DESIGNER...	0000000085.PRT	0000000085.prt
SubTest		Incoming attribute(s) DESIGNER...	0000000083.ASM	0000000083.asm

(0 objects selected)

Conflict Description:
Incoming attribute(s) DESIGNER, MODELED_BY, SIZE ignored as they are not part of type definition.

Cancel

In the toolbar of the **Conflicts** table is a drop-down menu that lists the **Retry Options** common to all of the selected conflicts (if any are available). The **Description** column in the **Conflicts** table describes the conflict. If the description is truncated, the full text can be displayed in the **Conflict Description** area below the table by clicking on the ellipsis (...) in the **Description** column.

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 toolbar.
3. Click **Set**.

The **Conflicts** table refreshes to display the new value in the **Retry Option** column for the selected row(s).

4. Click **OK**. The action is retried and a new entry is created in the **Events** list. The original event's status changes to **Retried**. Further access to the original event in the **Conflicts Manager** is read-only.

4

Handling Objects in Windchill

This chapter describes how to perform more advanced PDM activities and explains how Windchill handles some Pro/ENGINEER objects, for example, Family Tables and Simplified Representations.

Topic	Page
Modifying Object Attributes (Properties)	4-2
Renaming Objects	4-6
Deriving New Designs Using Save As.....	4-8
Opening Objects in Pro/ENGINEER	4-18
Working with Family Tables.....	4-20
Simplified Representations.....	4-31
CAD Document Templates and Pro/ENGINEER Start Parts	4-32
Using Library Parts	4-34
Managing Incomplete Dependent Objects	4-36
Managing a BOM with the Product Structure and CAD Document Structure ..	4-39
Naming and Numbering CAD Documents and Parts.....	4-45

Modifying Object Attributes (Properties)

Your design work can sometimes call for the addition or removal of attributes (also referred to as properties) on objects, or the modification of attribute values. The **Edit Attributes** window provides a means for you to:

- Edit an attribute shared by multiple objects, all at the same time
- Edit multiple attributes on a single object

If you edit attributes of workspace objects, then those modifications apply only to the workspace version of the objects. Alternatively, you can initiate attribute modification on the commonspace version of an object (initiating **Edit Attributes** from the commonspace view of the object's information page). In this case the attribute modification applies to all versions of the object.

Attributes on Family Table objects can be modified, subject to certain restrictions. For more information on modifying Family Table attributes, see [Modifying Family Table Attributes in Windchill](#).

Editing Attributes from the Workspace

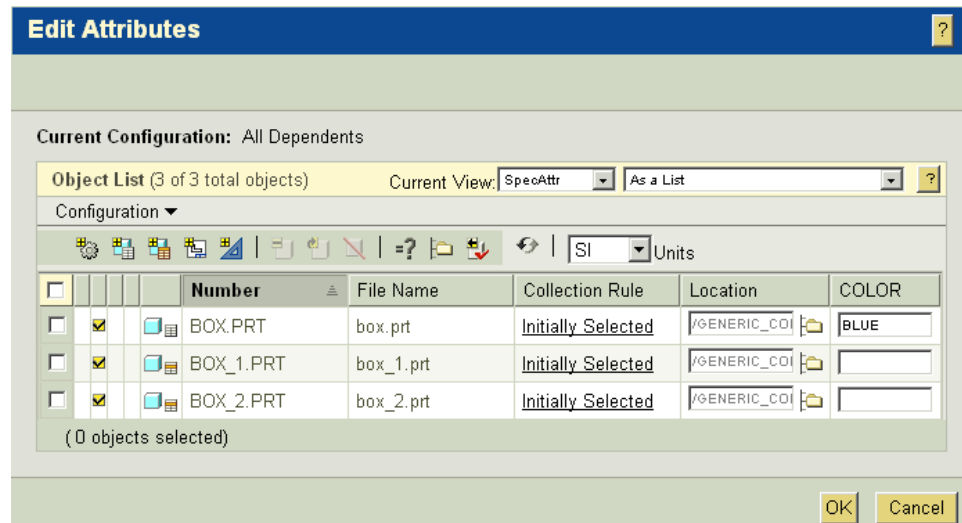
User-defined attributes, referred to in some authoring applications as parameters or properties, can be edited on one or more objects in the workspace. You can also edit attributes from an object's information page.


Use the following procedure to edit attributes for single or multiple objects in the workspace, such as target locations for new CAD documents and parts, as well as attributes assigned to a CAD document iteration:


1. Browse to a workspace.
2. Select one or more CAD documents or parts to modify.
3. Select **Edit > Edit Attributes**.

The workspace page is replaced with the **Edit Attributes** page, displaying your selected object(s) in a table listing. You can use the **Current View** drop-down list to select an existing view or to create a customized view for table objects and their attributes. 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. An adjacent drop-down list lets you select how the table information is displayed.


4. You can collect, remove, include, or exclude objects for the Edit Attributes action using the advanced mode of the collection user interface (the basic collection mode is not available for this action), which includes options to set a configuration specification. For more information on using the collection user interface, see the section [Collecting Objects for PDM Operations](#).



5. You can select any non-checked-out object and click the checkout icon  to perform a checkout and enable modification of the object's attributes.
6. Select a cell that currently shows as empty and, if editable, enter a new value
or

Select one or more object rows and click the edit attribute value icon  to launch the **Edit Attribute Value** dialog box where you can set values for multiple attributes on multiple objects.

Note: If you cannot see the attribute you wish to edit, add a column to the **Object List** table that displays the attribute, using the **Customize** option in the **Current View** drop-down list.

7. Click the set location icon  to navigate to and set a location for an object.

Note: Location is only editable for objects that have never been checked in.

8. Click **OK**.

The **Edit Attributes** window is refreshed. If the **Location** column is displayed, then the field is updated to show the context and folder name.

Note: If you have entered a location value that is in a context in which you do not have authorization to create CAD documents, then you are returned to the **Edit Attributes** window. The incorrect values are removed, and you see a warning icon in the **Status** column. Additionally, the message area displays a message indicating that you are not authorized to check in to the specified context.

Editing Attributes from the Information Page

When you want to edit attributes from the information page of an object, select **Edit Attributes** from the action drop-down list. When the **Edit Attributes** window appears, use steps 4 through 9 of the preceding procedure to modify the attributes of the object.

Attributes edited through the commonspace view of an object's information page apply to all versions of the object. Editing attributes for all versions of an object is only available via the commonspace view of the information page.

Setting Attribute Values

The **Edit Attribute Value** dialog box, launched by the **Set Value** action on the **Edit Attributes** page, allows you to modify any of the editable displayed attributes, except location, of objects selected or the action. Attribute values are only editable for checked out objects. If you select both checked out and checked in objects and then click **Set Value**, the **Edit Attribute Value** dialog box is launched; but changes made in it will only apply to the selected objects that have been checked out.

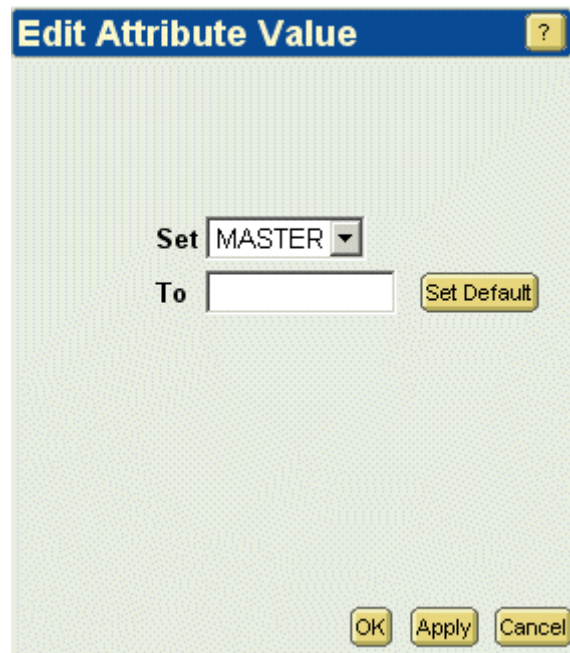
Similarly, if you select a Family Table instance, and the instance has a generic that is not checked out in the current workspace, then the **Edit Attribute Value** dialog box is launched; but changes made in it will only apply to the valid set of objects.

In the **Edit Attribute Value** dialog box, the **Set** drop-down list contains a list of object attributes currently displayed in the columns of the **Edit Attributes** page. Depending on the attribute selected in the **Set** drop-down menu, the **To** field presents a drop-down list (for list-specified values) or input panel (for range-delimited values). Any default value for the attribute is displayed when the **Set** selection is first made.

To set an attribute value, perform the following procedure:

1. On the **Edit Attributes** page, select the checked-out objects for which you want to edit attribute values and click **Set Value**.

The **Edit Attribute Value** dialog box opens.



2. From the **Set** drop-down list, select the attribute you want to edit
3. In the **To** field either accept the default, select a value from the drop-down list, or enter a value in the input panel.
4. Click **Apply** to set the value (the value is refreshed on the **Edit Attributes** page) and leave the **Edit Attribute Value** dialog box open

or

Click **Ok** to exit the **Edit Attribute Value** dialog box and return to the **Edit Attributes** page, where the newly set value is displayed.

Updating Attribute Values in Pro/ENGINEER and Windchill

If you have an owner-associated CAD document and part, with the same attribute assigned to both, there can be differences in how the attribute value is updated, depending on the application from which the modification of the value is made.

If you have a checked-in part with a unique value for the attribute, and you subsequently check out the CAD document, modify (in Pro/ENGINEER) the parameter that drives the attribute and execute auto-checkin, the part's attribute value is modified to match the CAD document's parameter value.

However, the opposite is not true. If you check out a part to the workspace and modify its attribute value using the **Edit Attributes** page and then check it back in, the value of the CAD document's attribute (the CAD document is never checked out) is not updated to reflect the the part attribute's new value.

Renaming Objects

You can rename an object, changing the values of Number, Name, and Model Name attributes, if you have Modify Identity access permission. You can even 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.

Only objects that have never been checked in (new) 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.

To rename objects, use the following procedure:

1. From the workspace, select one or more new objects to rename and select **File > Rename**

or

From an object's workspace information page **Actions** menu, select **Rename**

or

From the commonspace, or a commonspace view of an object's information page, select **Rename** from the **Actions** menu





The **Rename** page appears by default with the objects you have selected and any required dependents listed in a table listing. (If you call the rename function from the workspace, the page that appears is titled **Rename from Workspace**.)

Note: You can use the **Preference Manager** to change the default collection of objects.

	Name	New Name	File Name	New File Name	Number	New Number	Collection Rule
<input checked="" type="checkbox"/>	box		box.prt		BOX.PRT		Multiple rules...
<input checked="" type="checkbox"/>	box		box_2.prt		BOX_2.PRT		Initially Selected

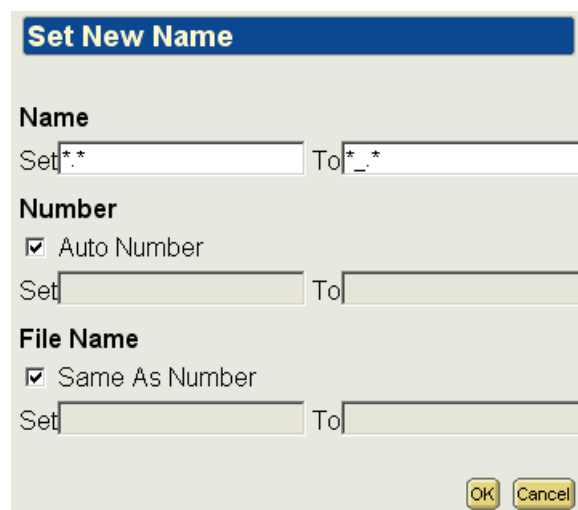
(0 objects selected)

OK Cancel

2. You can use the **Configuration** menu and the collection tools to modify the collection of objects in the **Object List** table. For more information on using the collection user interface, see the section [Collecting Objects for PDM Operations](#).
3. You can select an alternative display of the table, using the **Current View** menu.
4. Set new values in the **New Name** and **New Filename** fields. You can either enter values directly into the text fields available in each object's row, or you can select object rows and click the set new name icon  to access the **Set New Name** window where you can specify patterns for renaming objects.
5. Selecting one or more objects and clicking the set org ID icon  (if enabled at your site) presents the **Find Organization** dialog box which allows you to search for and assign a new organization ID.
6. Selecting one or more objects and clicking the reuse icon  removes any new name or number specifications and sets the object to be reused with its current name within in the newly named object structure.
7. Optionally, click the reset collection icon  in the **Object List** toolbar to return the set of objects in the object list to the objects initially populated following initiation of the rename action.
8. Click **OK**. The objects are renamed as you have specified

Setting a New Name

The **Set New Name** window is invoked by clicking **Set New Name** on the **Rename** page (or on the **Save As** page). It allows you to set conventions for naming and numbering objects (or new copies of objects).



The **Set New Name** dialog box is used to configure naming and numbering conventions. It features three main sections: **Name**, **Number**, and **File Name**. Each section has a 'Set' and a 'To' text field. The **Number** section includes a checked 'Auto Number' checkbox. The **File Name** section includes a checked 'Same As Number' checkbox. At the bottom right are 'OK' and 'Cancel' buttons.

Set New Name	
Name	
Set	To
Set	To
Number	
<input checked="" type="checkbox"/> Auto Number	
Set	To
File Name	
<input checked="" type="checkbox"/> Same As Number	
Set	To
OK Cancel	

The **Set New Name** window displays the original object attribute value in the **Set** field and the default attribute value for the new object in the **To** field. Depending on site settings you may be able to modify default values, which are as follows:

- In the **Number** area:
 - If auto-numbering is set, the **Auto Number** check box is selected by default, and the Number **Set** and **To** fields, as well as the **New Number** field on the **Save As** page are inactive
 - If you clear the **Auto Number** check box, both controls are activated.
 - If auto-numbering is not set, the check box is clear, and the Number **Set** and **To** fields, as well as the **New Number** field on the **Save As** page are active.

In the **File Name** area:

- The **Same as Number** check box is checked by default, and the New File Name **Set** and **To** fields, as well as the **New File Name** field on the **Save As** page are inactive. The new file name is identical to the new number.
- If you clear the **Same As Number** check box, both controls are activated.

To specify your own conventions for naming and numbering objects, use the following procedure:

1. Enter the object name or number in the appropriate **Set** field (you can use wild card matching).
2. Edit the **To** field to your specification (See the following table for examples).
3. Click **OK** to close the **Set New Name** window and have your naming conventions applied to the objects listed on the **Rename** (or **Save As**) page.

Rename Objective	Original Value	Target Value	Format Set	Format To
Apply Prefix	Object.prt	Prefix_Object.prt	*	Prefix_*
Apply Suffix	Object.prt	Object.prt_Suffix	*	*_Suffix
Apply Suffix	Object.prt	Object_Suffix.prt	*.*	*_Suffix.*
Replace	Object.prt	New.prt	*Object*	*New*

Deriving New Designs Using Save As

When you create a new CAD part or product structure, you can save time by using an existing part structure as the starting point from which you derive a new design. The **Save As** action provides a way copy single objects or multiple objects, or an entire structure, and rename them as new objects.

About Using Save As

The Save As action applies to parts and end items, CAD parts and assemblies, drawings, and Pro/ENGINEER family table objects. Save As preserves the CAD document-to-part relationship.

When you select CAD documents and parts to copy within a product structure, you create new objects (copies with new names). You also have the ability to not copy some of the members of the structure, instead creating references to the original existing objects (reusing original objects in the new structure).

When using the Save As action, remember these guidelines:

- If you copy a drawing without copying the part or assembly referenced by the drawing, the new drawing references the original part or assembly.
- You can use Save As to copy an entire or partial family table or any individual member. If you copy an instance without its generic, the new instance copy becomes a new member of the family table

Where Save As Is Available

The Save As command is available in the following places:

- Where workspace objects are accessed, including:
 - The workspace **File** menu (**File > Save As**)
 - The workspace information page **Actions** list
 - The toolbar of the workspace version **Model Structure Report**

Note: As opposed to using Save As from the commonspace, workspace Save As allows you to manipulate and modify your newly saved-as objects before committing them to the commonspace with a checkin.

- In the commonspace, including:
 - The actions list on the information page of a checked-in (commonsapce) CAD document or part object
 - The toolbar on the **Product Structure** table (on the information page of a checked-in part or end item)
 - The toolbar on the **Model Structure Report** table (on the information page of a checked-in CAD document)
 - In addition, the commonspace Save As action is available from search results and from the **Folders** page for products and libraries -- from the toolbar and from the object action menus for objects that can be copied.

- The Save As action is not on the **Folders** toolbar when the current view is set to any of the following:
 - Folders Only
 - Links Only
 - Documents Only

Overview of Save As

Objects selected for **Save As** are initially shown in the **Save As** table as intended to be copied.

Note: The preference, Save As > Save Selected Objects Only, is set to false by default. When this preference is set to true, only selected objects are copied and rest are reused by default.

Some general considerations for (workspace) **Save As** are as follows:

- If CAD documents and their associated parts (either by Owner or Content links) are copied together, the new objects have the same types of associations as the originals.
- If a CAD document is copied without its associated part(s), the new CAD document does not have any associations to parts.
- If a part is copied without its associated CAD document(s), the new part does not have associations to any CAD document.
- Incomplete objects or objects modified in the local cache are not eligible for collection. Objects with circular dependencies must be either included or excluded together.

The following are other important points about how Save As works:

- Selected objects display a default **Number**. CAD objects also have a default **Name**.
 - If the selected object is auto-numbered by default by a site preference, then by default the **New Number** field displays the text (Generated) and the **New File Name** field displays the text <Same As Number> and both fields are inactive. This is to indicate that the object is being copied, and that auto-numbering is applicable for that object type.
 - Selecting an object and clicking **Reuse** indicates that you intend to reuse the existing object in the new structure, rather than creating a new copy. If you toggle between saving a new copy and reusing the existing object, then any text, whether generated by the system or entered by you, is removed from the **New Number**, **New Name**, and **New File Name** (if applicable) fields.

- An underscore (_) is added to the default object **New Number** when auto-numbering is not the default mechanism for naming for this object type.
- An underscore (_) is added to the default object **Name**.
- Windchill generates the default **New File Name** when the file name is tied to the CAD document **New Number**. The file name is the <number> plus the appropriate CAD document extension. For example, if auto-numbering specifies a number for a CAD document to be "1234567", and you are copying a Pro/ENGINEER part, then the resulting **New File Name** is "1234567.prt."
- An underscore (_) is added to the base of the default **New File Name** when auto-numbering is not invoked. That is, if the file name was bolt.prt, then when auto-numbering is not used, the **New File Name** by default would be bolt_.prt.
- Related objects may be added to the object list by using the collection tools and **Configuration** menu.

If you wish to change the default name, number or file name of these added objects, you must do one of the following:

- Enter a **New Number**, **New Name**, or **New File Name** in the appropriate field of the table.

or

- Select the object and click **Set New Name** in the tool bar. The object Name is changed based on the options set in the **Set New Name** window.

Note: Changing context, location, or organization ID does not generate a new object. These changes are ignored when exiting the dialog if the object **Name**, **Number**, or **File Name** has not been changed.

- You can also use the **Set View** command to specify a different view for a part object.

The default behavior of **Save As** is to preserve CAD document and Windchill part structures to the maximum extent possible.

If you save a CAD document or Windchill part structure, a new, complete, and parallel structure is created only if both of the following conditions are met:

- The top-level object is saved as a new object.
- All the parents of a lower-level object are also saved as new objects.

If these conditions are not met, the explicitly selected objects are copied, but each change is discrete.

Note: You can use **Save As** to copy an entire or partial family table. If you copy an instance without its generic, the new instance copy becomes a new member of the family table. Also, if you copy a drawing without copying its referent model, the new drawing refers to the original model.

Using Workspace Save As

The workspace **Save As** action allows you to save objects in the workspace as new objects. These newly created objects are not committed to the commonspace until you perform a checkin; so copies can be modified, evaluated, and, if best, discarded before checkin, to avoid unwanted or redundant database iterations. Workspace **Save As** also allows you to update a dependency from an existing parent object to the newly-duplicated object, letting you replace an old component with a newly-duplicated component without involving the authoring application. In addition, circular dependencies can be detected and managed.

The **Workspace Save As** wizard has the following two steps:

- **1: Collect Objects To Copy**
- **2: Update Parents**

You can navigate to a subsequent step by clicking **Next**. Navigate to a preceding step by clicking **Back** or clicking the gray box corresponding to a previously completed step.

1: Collect Objects to Copy

When **Save As** is initiated, the first step of the wizard appears with the object(s) you selected and any required dependents listed in the **Save As** table. Current configuration information is listed above the table. The **Configuration** menu allows you to change the current configuration settings. You can use the **Current View** drop-down list to select an existing view or to create a customized view for table objects and their attributes. 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. An adjacent drop-down list lets you select how the table information is displayed.

Note: The preference, **Save Selected Only**, is set to false by default. When this preference is set to true, only selected objects are copied and rest are reused by default.

Below the menus is the toolbar for the **Save As** table. You can use the first eight action icons on the left side of the tool bar to collect additional related objects, exclude selected objects from the table, or exclude or include objects from the action while leaving them listed in the table. For more information on using the collection user interface, see the section [Collecting Objects for PDM Operations](#).

Workspace Save As ?

Steps: 1: Collect Objects To Copy 2: Update Parents

Current Configuration: Required Dependents

Save As (3 of 3 total objects) Current View: Default As a List ?

Configuration

	Number	New Number	File Name	New File Name	Name	New Name	Version	Collection Rule	Location	Part View
	0000000083.ASM	(Generated)	0000000083.asm	<Same As Number>	SubTest	SubTest_	A.1	Initially Selected	/GENERIC_COMPUTE	
	0000000084.PRT	(Generated)	0000000084.prt	<Same As Number>	Part01	Part01_	A.1	Dependent	/GENERIC_COMPUTE	
	0000000085.PRT	(Generated)	0000000085.prt	<Same As Number>	SubPart02	SubPart02_	A.1	Dependent	/GENERIC_COMPUTE	

(0 objects selected)

Back
Next
Finish
Cancel

To the right of the collection tools are the action icons for specifying how the new objects for selected objects in the **Save As** table will be saved. These actions are described in the following table:

Action	Description
(Set New Name)	Calls the Set New Name dialog box, allowing you to set a renaming pattern. You can use the Set New Name action on an object previously set for reuse to re-include the object in the Save As action
(Reuse)	Specifies to not copy the selected object, but to reference the original in a new (saved as) structure.
(Set Org ID)	Presents the Find Organization dialog box which allows you to search for and assign a new organization ID
(Set Location)	Calls the Set Location dialog box, allowing you to set a commonspace location for the new object(s)
(Set view)	Calls the Set View dialog box, allowing you to set a view for the new part(s). (Available for parts only)

Clicking the reset icon on the right side of the toolbar resets the **Save As** table to its initial condition.

Note: Incomplete objects or objects modified in the local cache are not eligible for collection. Objects with circular dependencies must be either included or excluded together.




To use Save As in the workspace, perform the following procedure:

1. Select an object in the workspace that you want to save as a new object and select **File > Save As**. Locally modified or incomplete objects, or objects that have never been uploaded, are not eligible for **Save As**.




The **Workspace Save As** wizard opens, listing your initially selected objects and required dependents and circular dependents in the **Save As** table.

2. Select one or more objects and click the appropriate collection toolbar action to collect, remove, exclude, or include related objects in the **Save As** table.

Note: In the case of circular dependents, you must either include both objects or exclude both from Save As.

3. By default the system appends an underscore to the original name. Select one or more objects and click the custom naming icon  to set a custom naming pattern for the objects. Site settings (for example, auto-numbering) may affect what options are available to you.
4. By default, objects in the **Save As** table are indicated in the **Action status** column to be saved as new objects (). To specify that an original object be reused (for example, in the newly-saved-as structure of a higher level object) select the object and click the reuse icon .

Note: You cannot reuse a circular dependent.

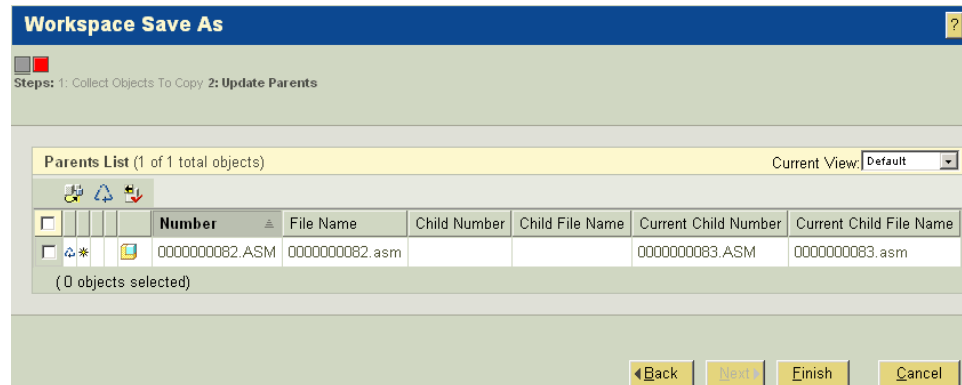
5. By default, new objects take the organization ID of the original context. To change the Organization ID, select one or more objects and click the set Org ID icon  (if available at your site).
6. By default, newly-saved-as objects are stored in the same location as the original. To set a different location, select one or more objects and click the set location icon .
7. By default, the view of newly-saved-as parts is the same as for the original. To set a different view, select one or more objects and click the set view icon .
8. Click **Next** to continue to the **Update Parents** step, or click **Finish** to commit the **Save As** operation without updating parent objects

2: Update Parents

The **Update Parents** step allows you to update a dependency from a parent object so that a newly-saved-as object can be the child of an existing parent. For each object being copied, valid parents are determined. If the parent is in the workspace and is not being copied, then the parent is valid.




When you click **Next** in the **Collect Objects to Copy** step, the system presents the **Update Parents** step, which lists parents of the objects set to be saved as new objects in the **Parents List** table. By default, all parents are set to reuse their original members.

:



Use the following procedure to update parents:

1. For any parent listed and eligible for update, decide if you want it to remain the parent of the newly saved as object(s).
2. In the toolbar of the **Parents List** are action icons that control how a selected parent object is handled during a **Save as** operation. The controls are described in the following table:

Action	Description
 (Reuse)	Specifies not to copy the selected parent, but to reference the original parent in a new (saved as) structure.
 (Replace parent)	Replaces the original parent with a new copy as the parent of a newly-saved-as object.
 (Check Out parent)	Initiates a simple (no user interface), immediate check-out of the parent object to allow modification of references

- If you want the new structure to reuse an original parent ensure that the reuse (♻️) symbol is displayed in the parent's **Action status** column. You may need to select the parent and click the reuse icon ♻️. To check out the parent to allow modification, you may need to select the parent and click the checkout icon 📁🔒.
 - To specify that a new parent object is created for the new structure, select the listed parent and click the replace parent icon 📁👤.
3. Click **Finish** to commit the **Save As** operation. The objects are processed and the new objects are listed in the workspace.

Using Commonspace Save As

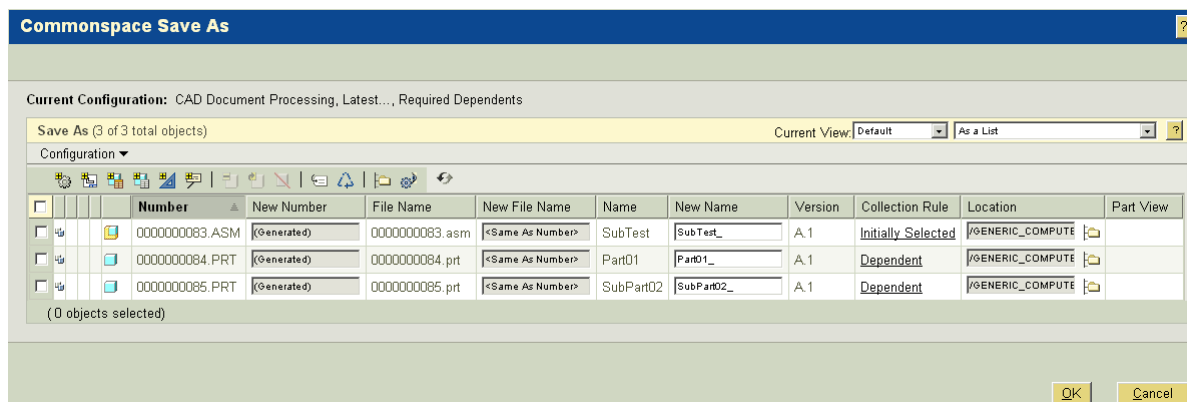
The commonspace Save As action allows you to copy a checked-in CAD document or CAD document structure (with or without associated part(s)), or a part object or product structure, and store it as new object(s). This functionality applies to models, drawings, and family table objects in the commonspace, and is capable of preserving CAD document/part associations (that is, the saved as CAD documents and parts are associated in the same way that their originals were).

Commonspace **Save As** is only available from the commonspace (for example, the **Model Structure Report** or the **Product Structure** report on the commonspace view of the object's information page, as well as the part or CAD document **Actions** drop-down list on the commonspace view of an object's information page, and the search results page).

To save an object (and its associated objects) as a new object or objects:

1. From the CAD document information page **Actions** drop-down list or other commonspace location **Actions** list, select **Save As**.






The **Commonspace Save As** window opens. It contains a toolbar for collecting or removing additional objects and a table where the objects to be saved as new objects are listed.



2. Use the collection user interface (Commonspace Save As uses the advanced mode), which includes options to set a configuration specification, to specify the objects to add, remove, include, or exclude from the **Save As** table. For more information on using the collection user interface, see the section [Collecting Objects for PDM Operations](#).

Note: When two objects have circular dependencies on each other, the **Save As** operation requires both objects to be either included in or excluded (or removed) from **Save As** together.

3. To the right of the collection icons are the action icons for specifying how the new objects for selected objects in the **Save As** table will be saved. These actions are described in the following table:


Action	Description
 (Set New Name)	Calls the Set New Name dialog box, allowing you to set a renaming pattern. You can use the Set New Name action on an object previously set for reuse to re-include the object in the Save As action
 (Reuse)	Specifies to not copy the selected object, but to reference the original in a new (saved as) structure.
 (Set Org ID)	Presents the Find Organization dialog box which allows you to search for and assign a new organization ID
 (Set Location)	Calls the Set Location dialog box, allowing you to set a commonspace location for the new object(s)
 (Set view)	Calls the Set View dialog box, allowing you to set a view for the new part(s). (Available for parts only)

4. In the **Save As** table, examine the proposed **New Number**, **New File Name**, and **New Name** values for each listed object, which have been initialized according to site preference settings. If auto-numbering is not set, you can edit these values; however, CAD document and part numbers must be unique in the database and of a valid length.

Note: File Name and New File Name are not applicable to Windchill parts.

5. Examine the proposed Location for each object. You can change this value by using the folder button in the Location column.

Note: If you are working in a PDMLink context, a context rather than a folder is the location listed.

6. Optionally, to remove your specifications for object collection and handling, and return the table to its original state, click the reset icon .
7. Click **OK** on the **Save As** window. The system checks for object number uniqueness and for valid name, model name and folder location. If the CAD document or part number, name, model name and folder location are valid, a new object is created.

If the object number is not unique or invalid (for example, too long) an appropriate message is displayed, asking you to try another value.

If the object name, model name or folder location is invalid, a message is logged and the next selected object is processed.

Setting a View

The **Set View** window is used to specify a view of a part object during the Save As action.




To set a view for the part object, select a named view from the **Set View** drop-down list.

Opening Objects in Pro/ENGINEER

You can open CAD documents from a workspace, or from various places in the commonspace while working in the embedded browser of a Pro/ENGINEER Wildfire session. In addition, you can open CAD documents from a standalone browser if an appropriate installation of Pro/ENGINEER is installed on your machine.

Opening Workspace Objects from the Embedded Browser


To open a listed workspace CAD document in Pro/ENGINEER, select **File >Open > in Pro/ENGINEER** or click the open in Pro/ENGINEER icon  in the **Actions** column for the object. The object is opened in your current Pro/ENGINEER session. You can also access the Open in Pro/ENGINEER action from most places where CAD documents are exposed in Windchill, as explained in the following section.

Note: You cannot open CAD documents for UDFs, incomplete objects, or any file type for which direct retrieval is not allowed in Pro/ENGINEER.

Opening Objects from a Standalone Browser

When you are working in a standalone browser and an appropriate version of Pro/ENGINEER is installed on your machine, you can open CAD documents in Pro/ENGINEER. If a Pro/ENGINEER session is already running, the object is opened in your existing session. If no session is running, the action to open an object automatically launches Pro/ENGINEER.

The action for opening CAD documents is available in Windchill generally where CAD documents are exposed. In addition to the workspace, these places include the Folder page (when CAD documents are displayed), the search results page, the Model Structure Report, the Product Structure report (when associated CAD documents are displayed), and the CAD document information page. The action

can be initiated either by clicking the open in Pro/ENGINEER icon  in the **Actions** column for the object, or selecting **Open in Pro/ENGINEER** from an action menu or drop-down list.

System Responses in a Standalone Server Environment

When you initiate the **Open in Pro/ENGINEER** action, the system is either able to directly open the objects in an existing session (for example if you already have a primary server registered with an active workspace), or helps you register and activate the server/workspace required. The following are some general characteristics of the process:

- The specific iteration you select is opened in Pro/ENGINEER.
- Other objects that are required to open your selected object are also collected. The collected objects represent the latest configuration, unless:
 - You have downloaded a specific configuration to your workspace or the workspace's own configuration identifies a specific configuration (for example, a baseline)
 - You are downloading an object and launch Pro/ENGINEER from the control in the **Check Out** or **Add to Workspace** page.
- If the registered server you initiate the action from is not primary you are asked if you want to make it primary.
- If you initiate the action from a workspace that is not active, you are asked to activate the workspace (and warned that any objects in session are erased by the change of workspace)
- If no server is registered, you are presented with a server registration dialog box.
- If you have existing workspaces on the server, you are asked to select one.
- If you have no existing workspaces on the server, a default workspace is created for you.

- If you have more than one available startup configuration (.psf file) for launching Pro/ENGINEER, you are asked to select one.
- If the action requires a different workspace to be activated, you are warned that objects currently in session will be erased.

Working with Family Tables

The following sections describe Family Tables and how to work with them in the Windchill PDM system.

Family Table Overview

A Family Table is a means to define a collection of CAD parts (or assemblies, or user defined features) that share the same generic properties, but deviate slightly in one or two aspects, such as size or detail features. Each member of the family (for example, a particular size of a family of similarly designed wood screws) occupies a row in the table, while attributes that are either shared or differentiated among the members of the family appear as the table column headings. Table field cells contain the object values for each member of the family.

CAD parts (or assemblies) in Family Tables are also known as table-driven parts.

Using Family Tables, you can:

- Create and store large numbers of objects simply and compactly
- Save time and effort by standardizing model generation
- Generate variations of a CAD part or assembly from one file without having to re-create and generate each one
- Automatically create variations and configurations of the design
- Create a table of CAD parts that can be saved to a print file and included in CAD part catalogs

Family Tables promote the use of standardized components. They let you represent your actual part inventory in Pro/ENGINEER. Moreover, families make it easy to interchange CAD parts and subassemblies in an assembly, because instances from the same family are automatically interchangeable with each other.

Tip: Because modifications to generics are inherited by instances, recommended practice is to avoid using generics as assembly components.

Family Table Structure

Family Tables are essentially spreadsheets, consisting of columns and rows. It is possible for any instance of a Family Table to also have its own Family Table, thus creating Family Table trees, also referred to as nested Family Tables.

Family Tables consist of the following three components:

1. The base object (generic object or *generic*) upon which definitions of all other members (*instances*) of the family are based.
2. Any attributes that may vary between the instances and the generic – dimension and parameter values, features and assembly components to be table-driven, user-defined feature and pattern table names, geometry tolerance and other types.
3. Names of all family members (instances) created by the table and the corresponding values for each of the table-driven objects.

Each row contains the instance name and corresponding values of the attributes in it; columns are used for attributes.

The column headings include the names of all of the dimensions, parameters, features, members, and groups that were selected for the table. Dimensions are listed by name (for example, `d9`) with the associated symbol name (if any) on the line below it (for example, `depth`). Parameters are listed by name. Features are listed by feature number (for example `F107`) with the associated feature type (for example `[cut]`) or feature name on the line below it.

The generic model is in the first row in the table. Only modifying the actual CAAD part, or suppressing or resuming features, can change the generic's table entries -- you cannot change the generic model by editing its entries in the Family Tables.

Note: Family Table names are not case-sensitive.

For each instance, you can define whether a feature, parameter, or assembly name is used in the instance either by indicating whether it is present in the instance (Y or N) or by providing a numeric value (in the case of a dimension). All dimension cells must have a value, either a number or an asterisk (*) which indicates that the generic's value is to be used.

All the family members automatically share all aspects of the generic model that are not included in the Family Table. For example, if the generic model has a parameter called MATERIAL with a value STEEL, all instances will have the same parameter and value.

You can scroll horizontally through a Family Table to see additional information. The **Instance Name** column remains visible as you scroll.

PDM Activities with Family Tables

The following sections explain how to perform PDM operations with Family Tables.


Viewing Family Tables in Windchill

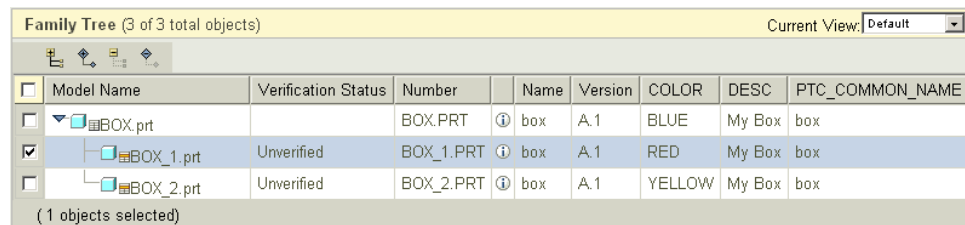
Windchill offers enhanced display options when viewing family tables. The enhanced view allows you to see:




- Family table hierarchy
- Verification status
- Name and Number
- Attributes

Note: Family tables in the latest release use a richer internal data set than in earlier releases. Therefore, migrated family table objects need to be saved & uploaded in Windchill 9.0 in order to view their internal data and verification status correctly.

To display a family table in Windchill:

1. In the workspace, select a family table object (either an instance or the generic).
2. Click the  action. The information page for the selected object opens.
3. On the information page, select **Related Objects > Family**. The information page reloads to display an expandable tree hierarchy showing **Family Tree** (Generic and Instance) information. A selected object is indicated in the tree by a check in the first column of its row.

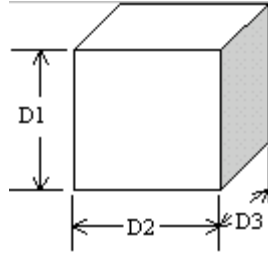


	Model Name	Verification Status	Number		Name	Version	COLOR	DESC	PTC_COMMON_NAME
<input type="checkbox"/>	BOX.prt		BOX.PRT		box	A.1	BLUE	My Box	box
<input checked="" type="checkbox"/>	BOX_1.prt	Unverified	BOX_1.PRT		box	A.1	RED	My Box	box
<input type="checkbox"/>	BOX_2.prt	Unverified	BOX_2.PRT		box	A.1	YELLOW	My Box	box

Uploading and Checking In a Simple Family Table

Consider a simple Family Table, based on the generic CAD part, box.prt. It has three major components, as follows:

- Geometric parameters:



- Non-geometric parameters

Name	Type	Value	Designate	Access	Source
DESC	String	My Box	Y	Full	User-Defined
COLOR	String	BLUE	Y	Restricted	User-Defined
WEIGHT	Real	27.000	N	Locked	Relation

- The family table that defines the members of the family

Type	Inst. Name	D1	D2	COLOR
	BOX	30	30	BLUE
	BOX_1	10	30	RED
	BOX_2	20	20	YELLOW

Uploading the Box family table to Windchill results in the creation of the following CAD document objects in the PDM database:

			Attributes	
Icon	Model Name	Content File	COLOR	DESCRIPTION
	Box.prt	box.prt	BLUE	My Box
	Box_1.prt	box_1.prt	RED	My Box
	Box_2.prt	box_2.prt	YELLOW	My Box

Note: In the Icon column of the preceding table of CAD documents, the symbols with the highlighted table row indicate that the document is an instance (no highlighting in the symbol for a generic). During the upload, designated Pro/ENGINEER parameters generate analogous attributes of the CAD document objects created in Windchill, only if:

- An attribute definition exists for an attribute of the same name as a designated parameter

or

- An explicit mapping between a designated parameter and an existing attribute definition exists on the server.

Note: The upload process includes non-verified (non-regenerated) instances. The following series of graphics show the information pages for the Box Family Table members after upload:

Product: GENERIC_COMPUTER > Workspace: [WF_WS1](#)

Actions CAD Part - box.prt, A.1

State: In Work

Number: BOX.PRT

Name: box

File Name: box.prt

Version: A.1

State: In Work - Released - Canceled

Last Modified: 2007-07-17 11:51 EDT

Modified By: wcadmin

Description:

Type: Workgroup Manager CAD Document

Category: CAD Part

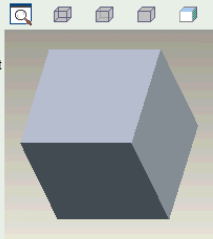
Location: /Administrator/WF_WS1

Created On: 2007-07-17 11:37 EDT

Created By: wcadmin

Authoring Application:

Dependencies Complete?: true



General ▾ Related Objects ▾ Collaboration ▾

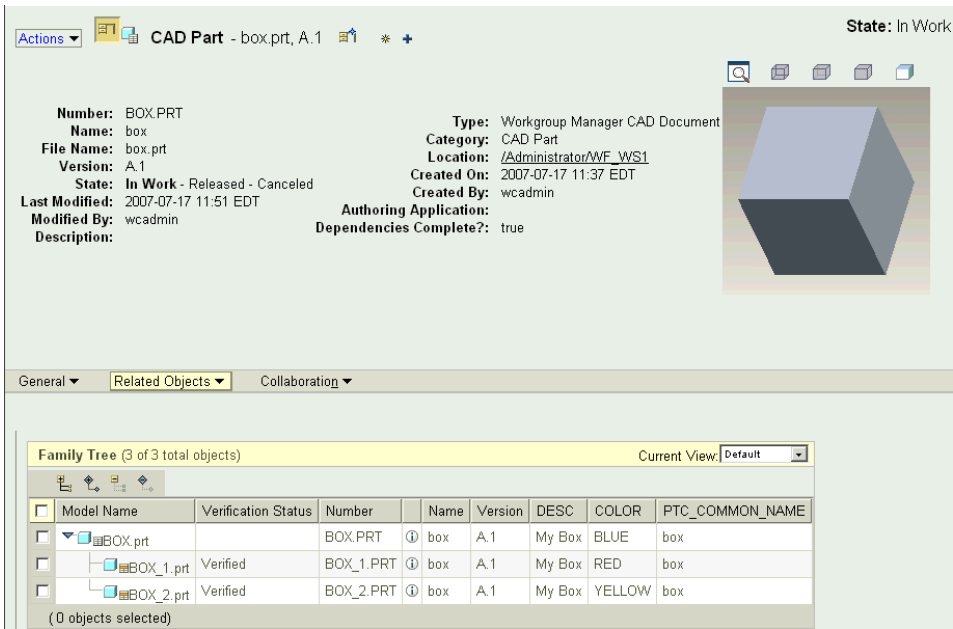
Content Table (1 objects)

File Name	Category	File Size	Last Modified	Modified By
box.prt	Pro/ENGINEER UGC	40.89 KB	2007-07-17 11:51 EDT	wcadmin

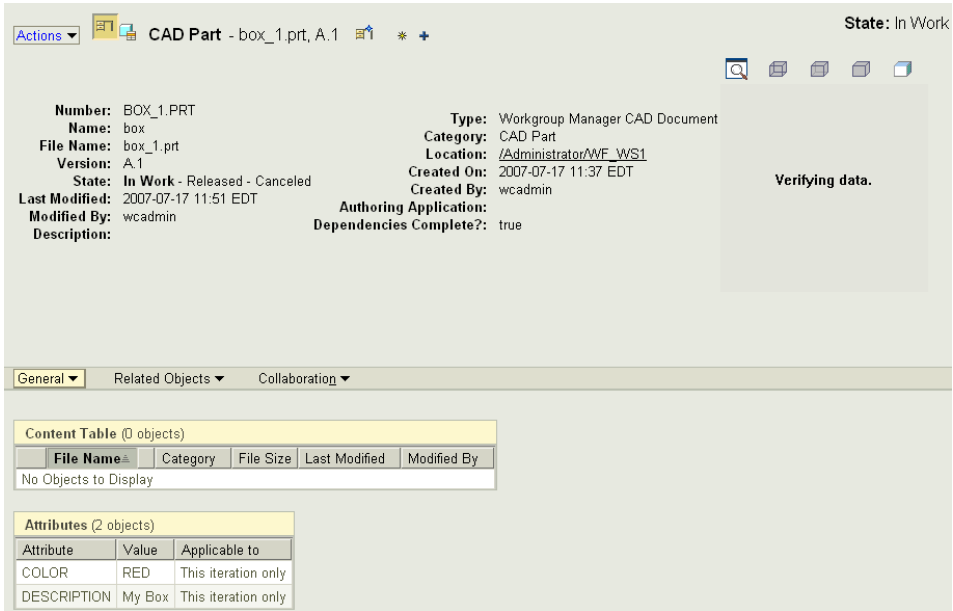
Attributes (2 objects)

Attribute	Value	Applicable to
COLOR	BLUE	This iteration only
DESCRIPTION	My Box	This iteration only

Selecting the **Related Objects > Family** on the information page of any family member refreshes the information page to display the other members of the Family Table:



Each instance is an independent document in its own right, and can be searched for, downloaded, and included in assemblies independently from other members of the family.



Checking Out and Adding Family Tables to the Workspace

The following rules apply to the checkout and download of Family Tables:

- If you check out an instance, you do not need checkout the generic

Note: Default behavior is to checkout the generic also, but you can override this by deselecting the generic on the checkout page; however, in the case of an earlier iteration of a family table, you must check out the entire family table together.

- You can checkout the generic without checking out any instances.
- You can download the generic without downloading any instances.

Modifying Family Tables

You can modify Family Tables using Pro/ENGINEER and then check the modified table into Windchill. You can also edit attributes of Family Tables in Windchill

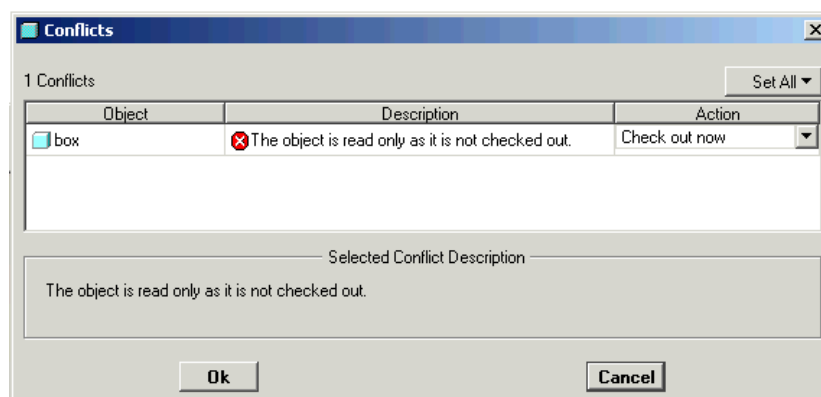
Modifying Family Tables in Pro/ENGINEER

The following procedure describes how to use the Family Table editor in Pro/ENGINEER to modify a Family Table:

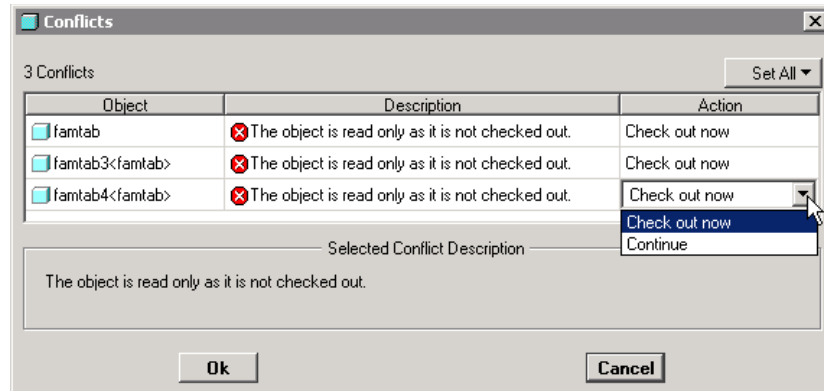
1. Open the generic in Pro/ENGINEER.

Starting with an empty workspace and opening the generic in Pro/ENGINEER downloads (no checkout) the generic to the workspace. No instances are added to the workspace at this time.

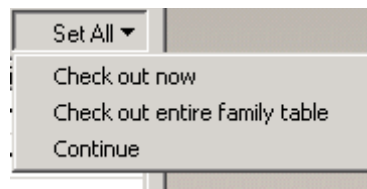
Open the Family Table Editor (**Tools > Family Table**). If an object is not checked out, when you attempt to edit it the **Conflicts** dialog appears, informing you that the object is read-only. The **Actions** column displays the recommended action (typically, to check out the object).



Note: When you click the cell in the **Action** column, the system presents a list of choices for that object: Check out (this object) now, Check out entire family table, or Continue. If you choose to continue (no checkout) you will be able to save, but not to upload your modifications.



If you have multiple objects in the **Conflicts** dialog, you can use the **Set All** list to select an action for all listed objects.



2. Modify a Family Table instance.
 - a. Select a row (instance) in the Family Table.
 - b. Edit the existing cells of the Family Table.
3. Verify the instances (**Tools > Verify**), and exit the Family Table Editor.
4. Save and check the generic into Windchill (You can check in from the Model Tree, using the workspace **Check In** action, or using **File > Check In** in Pro/ENGINEER).

Modifying Family Table Attributes in Windchill

The attributes of generic or instance Pro/ENGINEER Family Table members can be added or modified using the **Edit Attributes** page; however, because of their interdependency, additional members of the Family Table may need to be checked out in addition to any whose attributes you want to modify.

The following information is provided to review the classification of Family Table attributes, and describes the requirements for, and results of, attribute modification.

There are three classifications of attributes for Family Table objects, as described in the following table:

Attribute Class	Description
Generic-driven	An attribute (parameter) that has been designated in the generic model, but not added as a column to the Family Table. All instances reflect the same value as the generic for this attribute.
Inherited Table-driven	An attribute (parameter) that has been designated in the generic and has been added as a column to the family table, without assigning a unique value to the instance. The Family Table editor in Pro/ENGINEER displays an asterisk (*) for the value for the instance(s). In Windchill, the value displayed for the instance is the same as is displayed for the generic
Independent Table-driven	An attribute (parameter) that has been added to the Family Table and a unique value has been given to the attribute either via the family table editor or the Windchill Edit Attributes page

When adding or modifying the value of generic-driven attributes, all Family Table members must be checked-out. If your initial selection of objects did not include all members, upon clicking **OK** on the **Edit Attributes** page, the **Confirm** dialog box appears, listing the additional instances that need to be checked out. Clicking **OK** checks out the additional instances to allow the Family Table to be modified. Clicking **Cancel** returns you to the **Edit Attributes** page without modifying the Family Table.

When you modify an inherited table-driven attribute at the instance level, it becomes independent. That is, its value is no longer driven by the value of the generic. Therefore, only an instance whose value is being modified requires a check out.

Modifying an independent table-driven attribute requires only that the instance is checked-out.

Saving Family Tables Objects as New Objects

You can create a copy of any Family Table member (generic or instance) in your workspace by using the Save As command. Moreover, depending on how you select the family members, you can use the Save As command to do the following:


- Copy the entire set of instances.
- Copy the generic object only (as a standalone object).
- Copy a partial set of instances
- Copy a single instance object (as a new instance of existing table, not as a standalone object).

The Save as action can be initiated from several places in Windchill, for example:

- The Actions menu called by selecting the See Actions link in the table row on the Folders page that contains the object you want to copy
- The **Actions** drop-down list on the information page of the object
- The toolbar of the **Model Structure Report**

The following sections provide details on the various ways to copy Family Table members using Save As.


Copying the Generic Object Only


1. You can use Save as to copy only the generic  of a family table. When you have entered the Save As action on a generic object, use the following procedure to copy the generic object only:
2. In the **Save As** table, enter a name for the new generic in the **New Name** field.

Note: The system automatically generates a name for the new object. Enter a name for the new object only if you want to override the automatic naming.

3. Enter a folder location in the **Location** field or accept the default value.
4. Click **Ok**, the system creates a copy of the generic as a standalone object and saves it with the specified name in the specified location.

Copying the Entire Set of Instances

To copy an entire set of instances, you can initiate Save As for a single instance  of a Family Table and proceed as follows:

1. On the **Save As** page, select the instance and collect family () CAD documents.

Note: The generic is included in the family collection.

2. The **Save As** table refreshes to include all family table members.
3. For each family table member, enter a name in the **New Name** field.

Note: The system automatically generates a name for each new object. Enter a name for an object only if you want to override the automatic naming.

4. For each family table member, enter a folder location in the **Location** field or accept the default value.
5. Click **Ok**. The system creates a copy of the entire family table and saves it with the specified name in the specified location.

Copying a Partial Set of Instances

You can copy a partial set of instances of a Family Table by initiating Save As on a single instance and proceeding as follows:

1. On the **Save As** page, select the instance and collect family (📁)CAD documents.
- Note:** The generic is included in the family collection.
2. For each family table member, enter a name in the **New Name** field.
- Note:** The system automatically generates a name for each new object. Enter a name for an object only if you want to override the automatic naming.
3. For each family table member, enter a folder location in the **Location** field or accept the default value.
 4. Exclude any instance that you don't want to copy from the **Save As** list by selecting its row and clicking the reuse icon (🔄).
 5. Click **Ok**, the system creates a copy of the selected instances along with required generics as a family table and saves it with the specified name in the specified location.

Note: Objects saved to Windchill in releases prior to Pro/ENGINEER Wildfire 2.0 and Windchill 8.0 have a more restricted set of metadata in the database. Partial copy of a family table requires the richer meta data that is created by saving the objects to Windchill 8.0 (and above) with Pro/ENGINEER Wildfire 2.0 (and above). Once the meta-data has been upgraded, partial copy is then available on these objects.

Copying a Single Instance Only

You can copy a single instance of a Family Table and save it as a new instance of that table. Initiate Save As on the instance and proceed as follows:

1. The generic is also brought into the **Save As** table. Select it, and click the reuse icon (🔄) in order to not copy the generic.
2. Enter a name for the instance copy in the **New Name** field.

Note: The system automatically generates a name for the new object. Enter a name for an object only if you want to override the automatic naming.

3. Enter a folder location in the **Location** field or accept the default value.
4. Click **Ok**. The system creates a copy instance as a new instance of the existing family and saves it with the specified name in the specified location.

Simplified Representations

You can use simplified representation (or simp rep) tools to simplify an assembly by excluding components in a particular representation or substituting one component (CAD part or assembly) for another. Additionally, simp reps allow you to control the amount of data retrieved for a component. So for any given component, you can retrieve all data, just the geometry (no feature information by all surfaces and edges are represented) or just graphics (just a wireframe representation of the component). For example, when working with a very large assembly, you may find it useful to create a simplified representation that only contains the component in a small section of the assembly. This enables you to simplify your working environment significantly, and reduce the number of files that you need to download, while still including critical geometry.

Additionally, simplified representations improve the regeneration, retrieval, and display times of assemblies, enabling you to work more efficiently. You can use them to control which members of an assembly the system retrieves and displays. This lets you tailor your work environment to include only the information of current interest to you. For example, to speed the regeneration and display process, you can temporarily remove a complicated subassembly that is unrelated to the portion of the assembly on which you need to work.

Simplified representations are stored within the master assembly file, so if you are modifying a simplified representation, then you must have the master assembly file in session.

Tip: Bear in mind the following when using simplified representations with a Windchill system:

- Simplified representations only need to download the subset of files that are required by the simplified representation, thus speeding up operations that require file transfer (such as download and checkout)
- When several users are working concurrently on an assembly, any changes made to the simplified representation definition will require a checkout of the top level assembly. The consequence of this is that although many people can work on the assembly simultaneously, only one person at a time can be modifying the simplified representation definition. To overcome this restriction, consider using external simplified representations (external sim reps), described in the following section.

Simp Reps vs. External Simp Reps

An external simplified representation is a presentation of a master assembly stored as a separate assembly model (whereas the simplified representation is stored in the master assembly file). An external simp rep contains particular components of the master assembly or their simplified representations. You can create multiple external simplified representations of a master assembly, each corresponding to a different areas of the assembly and levels of detail. You can include low-level components without top and intermediate level assemblies and allow multiple users to work simultaneously. External simplified representations avoid the risk of accidental modifications to top level assemblies.

All the components included in an external simplified representation are the same as those in the master assembly. Therefore, it is not necessary to propagate modifications made to the external simplified representation or master assembly. All modifications to external simplified representations are automatically reflected in the master assembly.

External Simplified representations are supported by Windchill. This allows you to check in the external simp rep file to the Windchill database, retrieve an external simp rep assembly, and create (in Pro/ENGINEER) an external simp rep "on-the-fly". The ability to use external simp reps allows multiple users to work on the same assembly without checking out the master. Users can work on their simp reps, which is particularly useful for large assemblies. Each user can check out only what's needed and download the rest.

CAD Document Templates and Pro/ENGINEER Start Parts

Both Windchill and Pro/ENGINEER use default template files when creating a new object. In Pro/ENGINEER these objects are called start parts and in Windchill they are referred to as CAD document templates. The Pro/ENGINEER start parts traditionally reside on your local file system, while the CAD document templates are stored in the Windchill database. In a concurrent engineering environment, you may find it challenging to keep all of your CAD document templates up-to-date with your Pro/ENGINEER start parts.

To remedy this, you may find it useful to manage your Pro/ENGINEER start parts in the Windchill database. Additionally you can also create new CAD document templates that reference the same start part files. The end result is that regardless of whether a designer uses Pro/ENGINEER or Windchill to create a new object, both applications will use the same set of template files.

Other advantages to managing your start parts in Windchill PDMLink are:

- They are easily updated.
- They are easily distributed.
- They are version controlled.
- Same objects are used for all new Pro/ENGINEER CAD documents.

Managing Pro/ENGINEER Start Parts In Windchill PDMLink

1. In Windchill PDMLink, create a new library which you want to store your start part files.
2. In Pro/ENGINEER:
 - a. Create a new workspace.
 - b. Open the start part files for each Pro/ENGINEER object type (for example, by clicking **File > Open**, and then navigating to the start part and opening it in Pro/ENGINEER).
 - c. Save the start part to the workspace.
 - d. Repeat steps b and c until all of the desired start part files have been saved to the workspace.
 - e. In the workspace, check in the start part files to the library that you created in step 1.
3. In Pro/ENGINEER, you must set a configuration option for each of the start parts that you want to manage with Windchill PDMLink:
 - a. Click **Tools > Options**. The **Options** dialog box opens.
 - b. In the **Options** field, type the name of the configuration option associated with the desired start part (a full table containing each of the configuration options is provided below) and press **ENTER**.
 - c. Click **Browse** and navigate to the corresponding start part file in the library that you created in step 1.
 - d. Repeat steps b and c until all of the desired object types are pointing to the correct start part files in Windchill PDMLink.
4. In Windchill PDMLink, create new CAD document templates for each of the desired object types that point to the appropriate start part files

Tip: The name that you chose to assign to the server when you registered the Windchill server in the Server Registry dialog (See the section [Registering a Server](#)) is used in the path created in the steps above. If you change the name of the server, you will also need to update the values of these config.options.

Setting any of these options will cause Pro/ENGINEER to validate that it has access to the templates on startup. This will cause an authentication dialog to appear when you start Pro/ENGINEER, requiring you to log into the Windchill server. This is normal behavior, and once authenticated you will not be required to authenticate again in the same session.

Pro/ENGINEER Configuration Options

The following table lists configuration options relevant for start parts:

Configuration Option	Description
template_designasm	Specify the assembly to use as the default assembly template.
template_drawing	Specify the drawing to use as the default drawing template.
template_ecadasm	Specify the model to use as the default ECAD assembly template.
template_ecadpart	Specify the model to use as the default ECAD part template.
template_mfgcast	Specify the model to use as the default manufacturing cast template.
template_mfgcmm	Specify the model to use as the default manufacturing cmm template.
template_mfgemo	Specify the model to use as the default manufacturing expert machinist template.
template_mfgmold	Specify the model to use as the default manufacturing mold template.
template_mfgnc	Specify the model to use as the default manufacturing assembly template.
template_mold_layout	Specify the model to use as the default mold layout template.
template_sheetmetalt art	Specify the model to use as the default sheetmetal part template.
template_solidpart	Specify the model to use as the default part template.
start_model_dir	Specify the complete path to a folder containing start parts and assemblies

Using Library Parts

To avoid "reinventing the wheel," designers incorporate already-designed components into their assemblies. Online catalogs can offer a wide selection of

such components; however, retrieving them into your Pro/ENGINEER session, with proper references is not always straightforward.

Windchill PDMLink libraries provide a means for storing and controlling CAD parts that have been approved for use within your company. CAD parts contained in Windchill PDMLink libraries can be searched, browsed, and access controlled, and they work well with standard Pro/ENGINEER commands such as **Insert > Component** (Pro/ENGINEER can automatically resolve component references when using CAD parts stored in Windchill PDMLink libraries.). Common parts, such as fasteners, that are used in many end objects can be stored in a common CAD parts library, allowing all product designers to access and use those parts. Company-approved libraries also help enterprises by reducing CAD part proliferation (wherein the same CAD part may be redundantly gathered and assigned different part numbers by different engineers working independently).

Creating Libraries

The administrator for your site can set up libraries that correspond to your company's organization and processes. In fact, Windchill PDMLink comes with an out-of-the-box template for organizing a library specifically for CAD parts. If you are designated as a library manager by your organization, you can create a library and use it to store CAD parts of particular interest to your project. The first steps are to:

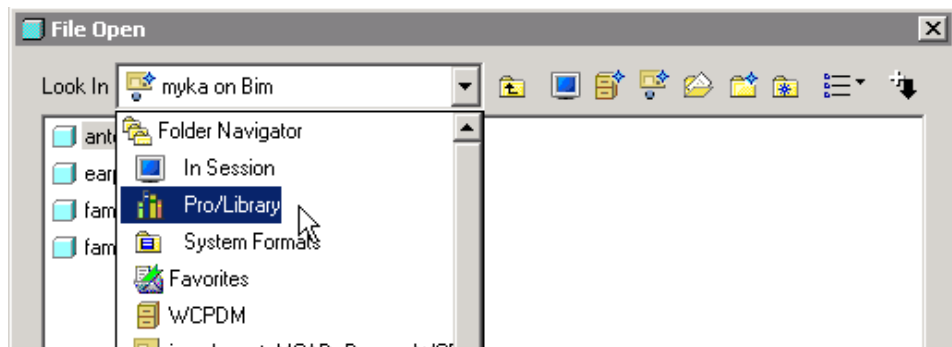
1. Create a library in Windchill PDMLink. Within your library, you can set up a folder structure to organize hierarchies of CAD parts and to group related CAD parts together. For more information on creating libraries, see the online help for the Windchill PDMLink Library tab. For more information on administering a library, see the chapter, *Administering Products and Libraries*, in the *Windchill System Administrator's Guide* and the chapter, *Administering Windchill PDM Library*, in the *Windchill Business Administrator's Guide*.
2. Set the configuration option in Pro/ENGINEER (pro_library_dir) to point to the library in Windchill PDMLink. This will speed access to the primary CAD parts resource.

Retrieving Components from a Library

If you have a library in Windchill PDMLink that is a frequent source for components, a best practice is to set the configuration option pro_library_dir to point to that library as your default directory. Do this by setting the value of the config option to the path to the library, for example: pro_library_dir = <Windchill PDMLink server URL>\...\<library folder name>.

Tip: The server URL is a "wt.pub://..." URL, not a codebase URL. An easy way to specify the config option value is to click **Browse** in the **Options** dialog box to present the **Select Directory** dialog box. When you navigate to and select your library, its path is entered in the **Value** field of the **Options** dialog box. Then, clicking **Add/Change** assigns your selected path to the value of pro_library_dir.

Once you have assigned the path of your chosen library to `pro_library_dir`, you can navigate directly to your library when you choose the **Pro/Library** option from the **Look In** list in the **File > Open** dialog.



Managing Incomplete Dependent Objects

An *incomplete dependent* is a CAD Document based on incomplete information known about a missing Pro/ENGINEER file. For example, If you import or save an assembly file to the workspace, the saved file may have name references to a file that cannot be saved to the workspace (perhaps because it was suppressed in the assembly file). The issue is that all the required information to make the object complete is absent.

Typically, the information known includes:

- The model name
- The CAD document type (inferred by the model name extension)
- Whether or not the object is an instance or generic, and, if an instance, its relationship to the generic.

Information not known about the object that is usually defined by Pro/ENGINEER includes:


- CAD document subtype
- Children or dependents of the object
- Designated parameters
- Other family table members. The generic will be known if it is an instance; but not intermediate generics or other members of the table.

Because good PDM practice does not allow checking incomplete data sets into the database, Pro/ENGINEER Wildfire provides several strategies for helping you to identify and resolve incomplete objects, including:

- Making users aware of incomplete objects in the workspace

- Providing tools for users to resolve incomplete objects by either removing the Pro/ENGINEER reference or converting them to fully defined CAD documents.
- Providing tools for Administrators to prohibit or manage the creation of incomplete objects.

Identifying Incomplete Dependents

If you save a Pro/ENGINEER file that references an incomplete dependent object to the workspace, the object is listed with this icon  to represent its object type.

Incomplete dependents are also listed appropriately in information page listings such as **Where Used** reports.

Resolving Incomplete Dependents

Pro/ENGINEER Wildfire allows you to deal with incomplete objects in two ways:

- Resolving an incomplete dependent from its information page
- Using the auto-resolve functionality available from the **Upload** or **Checkin** page.

Resolving Incomplete Dependents from the Information Page

Using the Replace Command

The **Replace** command allows you to resolve an incomplete dependency by replacing a missing object with an object from the Windchill database. The command is only available from the information page of an incomplete object.

To replace an incomplete object, perform the following steps:

1. From the information page of an incomplete object, select **Replace** from the actions drop-down list and click **Go**. The **Replace** page appears.

Clicking **View Details** returns you to the information page of the incomplete object.

2. **Search** or **Browse** to find a document to replace the incomplete object. When selected, the new document is displayed in the **New File Name** field.

Clicking **Cancel** returns you to the information page without replacing the incomplete object.

3. Click **Ok** to replace the incomplete object with the selected document. All the parents of the incomplete object in the workspace are updated to reflect the selected object and the incomplete object is deleted.

If the parents are in Pro/ENGINEER session, you are prompted to replace the

objects in session. To persist the changes after replacing the incomplete object, upload the documents.

Using the Add Placeholder Command

Alternatively, the **Add Placeholder** command allows you to attach a CAD document template as a temporary file to allow the system to treat the incomplete dependent as fully defined.

To add a placeholder to an incomplete object, perform the following steps.

1. From the information page drop-down list of the incomplete object, select **Add Placeholder** and click **Go**.
2. The system adds the template for the file extension of the Pro/ENGINEER file to the incomplete object.

Note: The incomplete object must first be uploaded in order that:

- The **View Information** action is available from the object's workspace **Actions** column (Windchill ProjectLink only).
- The **Add Placeholder** action is available on the object's information page.

Resolving Incomplete Dependents from the Check In or Upload User Interface

When objects selected for check in or upload, or objects added to the list based on dependencies include incomplete dependent objects, the **Auto resolve incomplete objects** check box is also available. When selected, the auto-resolve functionality offers two options:

- **Update with object on server, then ignore** -- The system searches on the server for an object with the same file name. If one is found, the incomplete object is updated by the found file. The object is no longer incomplete, and is therefore available for upload.

If no object is found to update the incomplete object, the system ignores the incomplete dependent (which is removed from the upload list).

- **Always ignore** -- The system will simply remove any incomplete objects from the upload list.

Note: Site administrative policies may not allow the ignore option, or may only allow certain object types to be ignored. Required dependents cannot be ignored.

Managing a BOM with the Product Structure and CAD Document Structure

A Pro/ENGINEER BOM (Bill of Materials) is a listing of the quantity and type of subcomponents that make up an assembly. When you are working in assembly mode you can generate a BOM (**Info > Bill of Materials**) that you can inspect, print, or save to disk. In this form, the BOM serves principally as a convenient way to view a summary of the assembly.

By creating Windchill parts and associating them to the CAD documents that contain the Pro/ENGINEER files as primary content, you can create a *product structure*, a hierarchy of enterprise parts in the PDM system that comprise an end item. The product structure mirrors the Pro/ENGINEER model tree structure for the assembly.

The Product Structure in Windchill PDMLink (Available from the **Structure** link on the information page of a part with subcomponents) displays the product structure. The **Related Reports** drop-down list on that page allows you generate the following types of reports:

- Single Level BOM
- Indented BOM
- Multilevel Where Used
- Multilevel BOM compare

The Single Level BOM is similar to the Pro/ENGINEER BOM report; however, in Windchill PDMLink you have the ability to annotate a BOM, which can be useful in spinning off new product configurations or new products themselves.

CAD Document Structure

Selecting the **Structure** menu on a CAD document's information page refreshes the information page to show the **Model Structure Report**, allowing you to view the CAD structure of an assembly object without needing to open the CAD editing application. In addition to viewing the structure you can do the following:

- If you are viewing the workspace version of the CAD document, the report shows the objects present in the workspace. If you are viewing the commonspace version, you can set the configuration for the object.
- Customize your view of the structure report information
 - Selectively expand and collapse the display of the structure
 - Selectively show or hide Windchill parts associated to the CAD documents
 - Sort the display using column headers
 - Filter the display using the table **Current View** drop-down menu

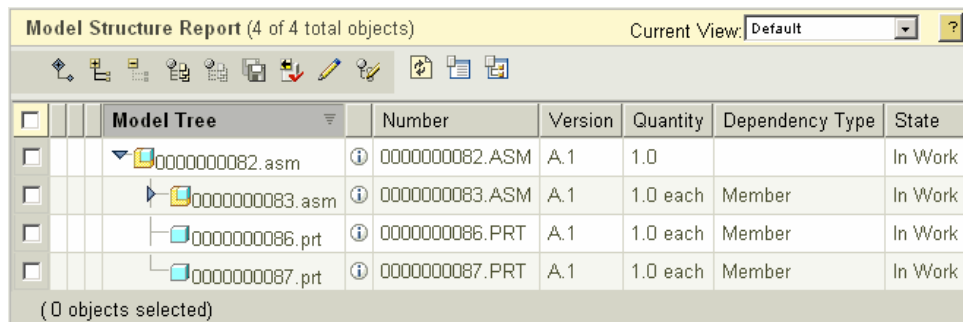
- Launch an editor utility to edit attributes of Pro/ENGINEER authored objects
- Launch an editor utility to edit Uses link attributes of Pro/ENGINEER authored objects

Note: When attribute values are displayed, applicable units are also displayed

- Generate, print, and save single level and multi-level reports
- Perform certain actions on objects in the report

The following procedure describes accessing and using the **Model Structure Report**:

From the workspace CAD document information (details) page, select **Structure**. The table area of the information page is refreshed to display the **Model Structure Report** table.



The screenshot shows the 'Model Structure Report (4 of 4 total objects)' window. It features a toolbar with icons for navigation and editing. Below the toolbar is a 'Model Tree' section with a tree view showing the hierarchy of the model. The main table displays the following data:

	Model Tree	Number	Version	Quantity	Dependency Type	State
<input type="checkbox"/>	0000000082.asm	0000000082.ASM	A.1	1.0		In Work
<input type="checkbox"/>	0000000083.asm	0000000083.ASM	A.1	1.0 each	Member	In Work
<input type="checkbox"/>	0000000086.prt	0000000086.PRT	A.1	1.0 each	Member	In Work
<input type="checkbox"/>	0000000087.prt	0000000087.PRT	A.1	1.0 each	Member	In Work

(0 objects selected)

Viewing the Model Structure Report

The default view of the **Model Structure Report** hides suppressed objects.

The table objects can be sorted by clicking the column header of the **Component Name** or **Number** columns. Details of the sorting function are as follows:

- The sort order is controlled by the icon in the column header.
- The sort is per level. That is, all objects under the same node are sorted by the sort criteria.
- Only two columns are sortable: **Model Tree** and **Number**.

Out-of-the-box, the **Model Structure Report** offers additional views in the **View** menu, named as follows:







- **Hide Unplaced** -- A view similar to default that also hides unplaced components
- **Show Suppressed** -- A view similar to default that does not hide suppressed components

- **BOM Members Only** -- A view that hides both suppressed and skeleton models, and removes the **Suppressed** and **Dependency Type** columns while adding a **Quantity** column, which displays the aggregated quantity for each component (and appropriate units for bulk objects). For example, consider a car assembly with four wheels. The default view would show the car as root node row in table and four component rows (one each) for the wheels. If you change to the BOM Members Only view, the four wheel rows collapse to one row only, and the quantity is shown as “4” for that component row.

Different or additional columns and rows can be displayed by customizing the table view. The additional available columns include attribute information for the following:

- Attributes associated with the Uses links
- Attributes for the iteration
- Attributes for the master
- All CAD document system attributes and modeled attributes
- All CAD document system link attributes











The toolbar of the **Model Structure Report** includes controls for viewing the structure, for acting on objects within the structure, and for generating reports. The following is a list of the viewing controls available on the toolbar with their descriptions:

- **Expand One**  -- Expands the selected nodes one level.
- **Expand All**  -- Expands the selected nodes all levels.
- **Collapse**  -- Collapses the selected nodes.
- **Show Parts**  -- Shows parts actively associated with the selected CAD documents.
- **Hide Parts**  -- Hides parts associated with the selected CAD documents.
- **Refresh**  -- Redisplays the product structure with any changes that have been checked in since you accessed it initially.


Note: As you expand a product structure, the information is cached and, as you navigate the structure, it is recalled from that cache. If you or someone else has modified the structure since you accessed it initially, this change is not displayed. When you click **Refresh**, however, the product structure returns to its top level setting and, as it is expanded, the latest (modified) parts are displayed.


Actions on Objects in the Structure

The actions on objects that are available to you from the **Model Structure Report** depend on whether you are viewing objects as they are currently in the workspace or viewing their commonspace versions. Actions available in an object's row are for that single object. Toolbar buttons trigger actions on one or more preselected objects (multi-select).

- Object actions available from the workspace version of the **Model Structure Report** can include:
 - **Information**  -- (In Actions column) Presents the information page for the selected object
 - **Check Out**  -- (In the toolbar) Initiates a check out of the selected object only (no user interface is presented to collect additional dependents)
 - **Edit**  -- (In the toolbar) Presents the **Edit Attributes** page (if at least one of the selected objects is checked out).
 - **Edit Uses Link Attributes**  -- Presents the **Edit Uses Link Attributes** user interface (if at least one of the selected objects is checked out)
- Object actions available from the commonspace version of the **Model Structure Report** can include:
 - **Information**  -- (In the Actions column) Presents the information page for the selected object
 - **Add to Workspace**  -- (In the toolbar) Presents the **Add to Workspace** window to add the object to a workspace
 - **Promote**  -- (In the toolbar) Presents the **New Promotion Request** window where you can initiate the promotion of the object to a new life cycle state
 - **Save As**  -- (In the toolbar) Presents the **Save As** window to allow you to save a copy of the structure as a new structure
 - **Add to Baseline**  -- Presents the **Add to Baseline** dialog box where you can search for a baseline to which to add the selected object.
 - **Move**  -- (In the toolbar) Presents the Move window to allow you to move the object to a new location

Editing Uses Link Attributes

When you click the edit uses link attributes icon  from the **Model Structure Report** of a workspace object, the **Edit Uses Link Attributes** page appears, allowing you to edit attributes on specific uses of a CAD document within a structure. As with the **Model Structure Report**, you can customize your view of the structure with the viewing commands in the toolbar. You may need a customized view to display the specific attribute(s) with which you are concerned.






For objects that are not currently checked out to your workspace, you can select them, then use the **Check Out**  command to gain Modify privileges for the objects.

Editing of usage links is only permitted if the parent is checked out.




You can edit single attributes that are displayed in the tree table by entering values in input cells that appear in the rows. Windchill parts can be displayed but their usage attribute values (if any) are not displayed.

Note: Attribute editing is not supported for dynamic document iterations. Editing of non-file-based attributes on uses links is supported for all CAD authoring applications. Editing of file-based attributes on uses links is only supported for Pro/ENGINEER and SolidWorks objects. (Non-file-based attributes are those that are not communicated to the CAD file.)

The following table lists the viewing controls available on the toolbar and their descriptions:

Control	Description
Expand One 	Expands the selected nodes one level
Expand All 	Expands the selected nodes all levels
Collapse 	Collapses the selected nodes
Show Parts 	Displays associated parts
Hide Parts 	Hides display of associated parts


The following table lists the actions available on the toolbar and their descriptions:

Control	Description
Check Out 	Performs an immediate checkout of the object (no checkout user interface)
Set Value 	Presents the Set Value user interface for the selected objects, if checked out
Reset 	Resets attribute values to their initial state (undoes your current modifications)


The **Units** drop-down list allows you to specify the unit system for attributes with units of measure.

Generating Reports on the Model Structure

Clicking on either of two toolbar controls allows you to generate reports from the model structure, as follows:

Single Level Report  -- Displays the **CAD Document Parts List**, a summarized list of the CAD parts in a bill of materials. (Multiple occurrences of a single part are combined and reported as one entry.)

CAD Document Parts List			
Parts List for Assembly 0000000221.ASM - TestAsm A			
File Name	Revision	Quantity	
0000000221.asm (PTC)	A.3	1	
0000000185.prt (PTC)	A.1	1	
ft_inst_37_3.prt (PTC)	A.1	1	

Multi Level Report  -- Displays the **CAD Document Indented Structure** report, a hierarchical list of the CAD parts in a bill of materials

CAD Document Indented Structure			
Indented Structure for Assembly 0000000221.ASM - TestAsm A			
Level	File Name	Revision	Quantity
0	... TestAsm	A.3	1
1 FT_generic_37.prt	A.1	1
1 plainCADpart	A.1	1

Naming and Numbering CAD Documents and Parts

Because different companies can take different approaches to naming objects, Pro/ENGINEER Wildfire supports four policies to determine how newly-created objects (either CAD documents or Windchill parts) are named and numbered. The aim is to provide methods to support standardizing object naming and numbering across the enterprise as necessary, while allowing latitude for individual users where appropriate.

While the naming and numbering policies are set by site or context administrators, understanding the basics can help you see you sometimes have choices in naming or numbering objects, while at other times your choices are intentionally limited by your site's business practices.

The four naming and numbering policies can be described briefly as follows:

- Auto Numbering
 - The CAD document Number is provided by the CAD Doc number generator (either OOTB or your site's customization)
 - The default value for the CAD document Name is copied from the Pro/ENGINEER model name. The file extension (.prt) can be optionally dropped (controlled by a preference).
 - The enterprise part Number is provided by the WTPart Number generator
 - The default value for the enterprise part Name is copied from the current value of the CAD document Name at the time the enterprise part is created. If a file extension is present in the CAD document name, it can be optionally dropped when naming the enterprise part (controlled by a preference).
 - In any create and edit user interface, the CAD Doc and enterprise part **Name** field is editable
 - In any create and edit user interface, the **Number** field is not editable.
 - Auto numbering is the OOTB naming and numbering policy.
- Name-driven
 - The CAD document Number is copied from the Pro/ENGINEER Model Name (the file extension can be dropped – controlled by a preference)
 - The default value for the CAD document Name is copied from the Pro/ENGINEER model name (file extension can be dropped – controlled by a preference)
 - The default value for the enterprise part Number is copied from the CAD document Number (the file extension can be dropped – controlled by a preference)

- The default value for the enterprise part Name is copied from the current value of the CAD document Name at the time the enterprise part is created (the file extension can be dropped – controlled by a preference)
- In any create and edit user interface, the CAD document and enterprise part **Name** and **Number** fields are editable by the user.
- Parameter-driven
 - The CAD document Number is copied from the value of the Pro/ENGINEER designated parameter identified by the preference (Upload) > Numbering Parameter.
 - The value for the CAD document Name is copied from the Pro/E designated parameter identified by the preference (Upload) > Naming Parameter.
 - The value for the enterprise part Number is copied from the value of the IBA identified by the preference Auto Associate Numbering Parameter.
 - The value for the enterprise part Name is copied from the value of the IBA identified by the preference Auto Associate Naming Parameter.
 - In any create and edit user interface, the CAD document and enterprise part NAME and NUMBER fields are editable by the user.

Note: Name-driven and parameter driven policies can only be used in object-driven creation of objects as those require a source object to create a new object. These policies are used during an initial upload (when a new CAD document is created based on a model file), and auto-associate (when a new enterprise part is created based on a CAD document).

- Custom
 - Custom naming and numbering of objects can be specified through the **Object Initialization Rules Administrator**, which provides a way to specify default values for the attributes of a specific object type. The default values are then used when the Windchill solution creates objects of that type. These specifications are called rules. Each rule can contain default values for one object type. The rules that are set only apply when the Windchill solution that is used to create an object does not set a corresponding value.
 - Rules are also set per context, allowing different naming and numbering policies for different contexts.
 - In addition, site Administrators can customize the Windchill naming service, which uses the Windchill service delegate mechanism, to allow the user to specify the following for a new CAD document:

- Number
- Name
- Parameter values

Note: The **Object Initialization Rules Administrator** is also used to set autonumbering policy, while name driven and parameter-driven policies are specified in the Windchill **Preference Manager**.

Regardless of the naming and numbering policy used, when creating a new object, system uniqueness constraints require that the CAD document attributes Number and File Name must both be unique within a Windchill PDMLink site or within each Windchill ProjectLink project.

For more information on naming and numbering, see the section [Policy-Managed Naming and Numbering](#).

5

Customizing and Administering Pro/ENGINEER Wildfire

This chapter presents customization and administration information and recommendations for using Pro/ENGINEER Wildfire integrated with Windchill PDMLink and Windchill ProjectLink. The primary audience is Pro/ENGINEER and Windchill system administrators; however, much of the information can be useful to end users as well.

The topics presented include Pro/ENGINEER configuration information (environment variables and config.pro options) that applies to the interaction with Windchill, and Windchill server-side preferences, as well as specific information on parameter mapping, parameter customization, customizing object naming, automated part creation, supporting custom parts, and customizing the user interface. In addition, recommendations for system configuration and performance tuning are offered.

The final section lists and describes Windchill preferences that are especially relevant to the interaction with Pro/ENGINEER.

Topic	Page
Configuration Settings in Pro/ENGINEER.....	5-2
Configuring Windchill for Interoperation with Pro/ENGINEER	5-14
System Configuration Recommendations	5-52
Performance Tuning	5-53
Other Recommendations	5-55
Windchill Preferences That Control Interaction with Pro/ENGINEER.....	5-57

Configuration Settings in Pro/ENGINEER

Environment Variables and Config.pro Options for Pro/ENGINEER Wildfire

Environment Variables

Pro/ENGINEER Wildfire uses a user-visible workspace to manage work-in-process data. Each workspace uses a local disk cache to ensure data integrity and optimize file transfer between Pro/ENGINEER and the server. The cache (which is managed by Pro/ENGINEER and is not visible to the end user), is used to store changed objects prior to an upload to the server, and to keep copies of objects downloaded from the server to speed up subsequent retrieval into Pro/ENGINEER.

As a system administrator, you may wish to put the cache on a larger disk partition than provided by the default location. The following table lists environment variables that can be set by a system administrator to manage the placement of the cache into a suitable partition:

Variable	Values	Description
PTC_WF_ROOT	/path/to/dir, Default on UNIX = ~/wf Default on Windows = [User Profile]\Application Data\PTC\ProENGINEER\ Wildfire\	Overrides the default location of .wf directory. Setting this environment variable will cause Pro/ENGINEER to use the new location as a location for the cache. Note: Existing cache data will not be copied to the new location automatically. Note:
PTC_WF_CACHE	/path/to/dir, default=\$PTC_WF_ROOT/.cache/	Allows the specification of additional cache space. If you are running out of disk space in \$PTC_WF_ROOT, you can use this environment variable to define a folder in which all new workspace caches will be stored. Note: This new folder only applies to newly created workspaces. Existing workspaces will continue to reside in \$PTC_WF_ROOT/.cache

Note: The environment variable EPM_MODE, designed for earlier versions of Pro/ENGINEER, should not be used with Pro/ENGINEER Wildfire. Because it prevents the **Conflicts** ("check out on-the-fly") window from appearing when users attempt to modify a checked-in model, its use could lead to loss of data when modifications cannot subsequently be checked in.

Config.pro Options

The following table lists Pro/ENGINEER config.pro options that are especially relevant to the Pro/ENGINEER Wildfire interaction with Windchill:

Config.pro Option	Values	Description
compress_output_files	yes no [default]	<p>Controls whether or not to compress object files to store them. Compressed files are slower to read and write, one-half to one-third the size, and fully compatible across systems. When set to "yes," stores object files in compressed format. When set to "no," object files are not compressed.</p> <p>Note: The time spent in compression and decompression could be more expensive on CPU than the benefits for disk or network. In some WAN environments, this could be a helpful for some transfers, but is not recommended for general use.</p>
disable_search_path_check	no [default] yes	<p>Controls whether or not the search path is checked for name conflicts when creating, renaming, or copying models.</p> <p>When set to "yes," disables the check of the search path for a naming conflict when a new file is created. This can speed up file creation by postponing the search path check (which includes the entire commonspace) until an upload is performed.</p>

Config.pro Option	Values	Description
dm_auto_open_zip	yes [default] no	<p>Defines how Pro/ENGINEER will handle zip files.</p> <p>If set to "yes", then Pro/ENGINEER opens the zip file and attempts to retrieve objects in the zip file. If the zip file contains more than one file (for example, in the case of assemblies), Pro/E will first attempt to open an object in the zip file that has the same name as the zip file itself. If it finds one, it will open it, if not, it will display the contents of the zip file in the File>Open dialog box.</p> <p>If set to "No", Pro/ENGINEER treats a zip file like a directory, and displays the contents of the zip file in the File > Open dialog box, allowing the user to pick the file from the zip that he or she wants to retrieve into session.</p>
dm_background_operations	yes no [default]	<p>If set to "yes," allows user to take advantage of the backgrounding operations (for example, during checkins of large data set).</p> <p>Note: Most situations can benefit from a yes setting, which allows working in the foreground while lengthy operations are run in the background.</p>

Config.pro Option	Values	Description
dm_cache_limit	Integer [default = 0]	<p>Sets the size (in MB) of the cache allocated to the combination of all registered servers and their workspaces on the client hard disk.</p> <p>Recommendation: If possible, set the cache size large enough to accommodate the largest anticipated data set (the downloaded content and the locally modified content prior to upload should be counted separately). A good rule of thumb is 80% of the remaining free space on the disk where Wildfire cache is located.</p> <p>Note: A value of “0” (no limit) will tend to fill up the client disk, but could boost performance by eliminating checks on cache size and purges.</p>
dm_checkout_on_the_fly	checkout [default] continue	<p>If set to the default value "checkout," the default action for the Conflicts ("checkout-on-the-fly") dialog box is “Check Out Now.”</p> <p>If set to "continue," the default action for the Conflicts dialog box is “Continue.”</p>

Config.pro Option	Values	Description
dm_http_compression_level	Integer, from 0 (no compression) to 9 (maximum compression) [default = 0]	<p>Sets the level of compression for data upload and download.</p> <p>Although compression speeds up transfer over the network, it uses server CPU and client CPU to perform the compress and decompress operations. In a local area network, where network transfers are rapid, compressing and decompressing data can result in lesser throughput. On a wide area network with lower bandwidth compression can lead to higher throughput. Since this is set per client, PTC recommends that clients in a LAN use a value of 0 (the default) and clients in a WAN use a value of 2 or 3.</p> <p>For older Wildfire versions, the following approximate guidelines apply:</p> <p>If client download bandwidth < 3 Mbps enable dm_http_compression_level (at a value of 3).</p> <p>If client download bandwidth > 3 Mbps unset dm_http_compression_level as uncompressed response read times will be faster.</p> <p>As of Wildfire 2.0 M260 and Wildfire 3.0 M090, the following approximate guidelines apply:</p> <p>If client download bandwidth < 20 Mbps, enable dm_http_compression_level (at a value of 3).</p> <p>If client download bandwidth > 20 Mbps set dm_http_compression_level to 0 (uncompressed), so that response read times will be faster.</p>

Config.pro Option	Values	Description
dm_network_request_size	integer >0 [default = 100000]	<p>Determines the maximum size, in bytes, of an HTTP upload request when uploading content files to Windchill.</p> <p>The default of 100000 is likely to ensure that each file will be uploaded through a separate http request with minimal process memory consumption overhead.</p> <p>A small value (say 8000) would mean a lot of small HTTP requests to the method server containing the model files which may add overhead; but because the local Wildfire file buffers are filled quickly, the upload starts sooner.</p> <p>A much larger value (say 800000000) may allow the uploading of the entire data set in a single HTTP request, but it could take a while for the client to write the files from local disk to its internal buffer before streaming the content to the server. In addition, because of apparent size limitations of the Microsoft HTTP API you may experience random upload failures with very large file size data sets in Windows. In addition, working with large datasets has been known to cause Internet Explorer to run out of memory.</p>
dm_network_retries	integer >0 [default = 10]	<p>Sets the number of attempts to connect to a Windchill server before the connection is considered broken.</p> <p>Recommended setting: default</p> <p>Note: If the http connection is unstable, a setting less than the default could increase failures, while a setting greater than the default causes delays if a failure occurs.</p>

Config.pro Option	Values	Description
dm_network_threads	integer >0 [default = 3]	<p>Sets the number of concurrent threads Pro/ENGINEER uses for uploading and downloading data to and from a Windchill server.</p> <p>The recommended setting depends on the network bandwidth. It is suggested to keep at 3 for a WAN and can be increased to 6 for a fast LAN. However, in most cases, increasing the number of threads in a LAN environment will not improve performance, as the disk will then become the bottleneck. Even in a WAN environment, settings greater than the default are unlikely to improve throughput significantly.</p>
dm_overwrite_contents_on_update	no [default] yes	<p>Specifies behavior during Update action.</p> <p>If set to "no," does not overwrite the locally modified contents for out-of-date objects, but updates their metadata only</p> <p>If set to "yes," overwrites the locally modified or out-of-date objects with the ones on the server in addition to updating their metadata.</p> <p>Note: If the user wants to abandon the local cache modifications, s/he can perform an explicit download (Add to Workspace) of the model from the server-side workspace, thus overwriting the version of the model in the cache. Alternatively, after the Update to the latest iteration, the user can check it out and upload the modifications from the local cache. The non-default value of "yes" should be used if the user makes only temporary modifications in the cache and never intends to keep them after the Update.</p>
dm_remember_server	yes [default] no	<p>If this option is set to "yes," the last primary server/workspace of a Pro/ENGINEER session is set automatically for the next Pro/ENGINEER session.</p>

Config.pro Option	Values	Description
dm_save_as_attachment	yes [default] no	Controls the default option for Pro/ENGINEER Save a Copy when models are saved as in non-Pro/ENGINEER format. If set to "yes," the model is by default saved as a secondary content attachment to the original CAD document. If set to "no," the model is by default saved as a (primary) CAD document.
dm_search_primary_server	yes [default] no	If this option is set to "yes," during retrieval, the system searches the primary server for dependencies not found in the workspace
dm_secondary_upload	automatic [default] explicit	Defines the behavior of saving to a secondary server (See also dm_upload_objects). If this option is set to "explicit," the Pro/ENGINEER File > Backup command will write data to the cache. The user must then explicitly send that data to the server (using either using either the Upload or Check In commands invoked from the corresponding workspace). If this option is set to "automatic," File > Backup in Pro/ENGINEER will also upload the Pro/ENGINEER files to the server.
dm_upload_objects	explicit [default] automatic	Defines the behavior of the Save command in Pro/ENGINEER. If this option is set to "explicit," the Pro/ENGINEER File > Save command will write data to the cache. The user must then explicitly send that data to the server (using either File > Save and Upload or File > Checkin). If this option is set to "automatic," File > Save in Pro/ENGINEER will also upload the Pro/ENGINEER files to the server.

Config.pro Option	Values	Description
let_proe_rename_pdm_objects	no [default] yes	<p>Determines whether an object retrieved from a PDM database can be renamed in a Pro/ENGINEER session</p> <p>An object rename in Pro/ENGINEER is seen only by parents in session. The object is seen as a new object when saved to the workspace.</p> <p>This option can be used to replace a standard sub-assembly with a copy of itself with a unique name.</p>
open_simplified_rep_by_default	no (default) yes <i><name_of_simplified_rep></i>	<p>Specifies whether to prompt user to select a simplified representation when opening a Pro/ENGINEER file</p> <p>If set to "yes," user is prompted to open a simplified representation when opening a Pro/ENGINEER file</p> <p>If set to the name of a simplified representation, the system opens the simplified representation without prompting the user.</p> <p>This option can be useful for using internal simplified reps on small to medium-sized datasets. If set to "yes," the user sees a pop-up listing the simplified reps available in an assembly, for example, when clicking on its hyperlink in the embedded browser. Choosing one of the simplified reps allows the user to add to the workspace and download only the models required for the corresponding simplified reps.</p>

Config.pro Option	Values	Description
regenerate_read_only_objects	yes (default) no	<p>Specifies whether read-only objects (objects not checked out) are regenerated.</p> <p>Set to "yes," it specifies that read-only parts with relationships to an explicitly modified assembly are modified implicitly upon regeneration of the assembly. (Explicit changes to a checked-in object cause the Conflicts dialog box to appear.)</p> <p>By setting to "no," you may avoid having read only workspace objects marked as modified. This, in turn, can reduce the number of files required for the checkout of an associated assembly</p>
retrieve_data_sharing_ref_parts	yes no [default]	<p>Controls automatic reference parts retrieval for dependent data sharing features.</p> <p>When set to "no," prevents download/opening of components that are not immediately needed</p>
save_model_display	wireframe, shading_low, shading_high, shading_lod	<p>Sets the quality of graphics that are shown on the Windchill information page.</p> <p>Setting this option to shading_lod creates the best images, but requires larger Pro/ENGINEER file sizes to store the additional graphical information.</p> <p>Note: Saving the shaded display increases the model file size by 100 or more percent. Setting this option to shading_lod creates the best images, but requires larger Pro/ENGINEER file sizes to store the additional graphical information. Setting to wireframe is the most lightweight format.</p>

Config.pro Option	Values	Description
save_objects	changed_and_specified [default] all changed changed_and_updated	Determines when an object and its dependent objects (such as a part used in an assembly) are stored. The recommended value in a PDM environment is "changed" (to avoid unnecessarily iterating the top-level object if it was not modified in session).
search_path	<directory paths by full path name>	Specifies list of directories to search (in order) for object/file retrieval. These directories, the working directory, and directories in search.pro file (refer to search_path_file) are Pro/ENGINEER's search path. Use full path name to avoid problems. It is best to use the minimum number of search paths to a minimum because a large number of search paths increases retrieval time. Note: In Pro/ENGINEER Wildfire it is not necessary to set the config.pro option, optionsearch_path. By default, when a Windchill server is your primary server, the entire primary server with active workspace is in the Pro/ENGINEER search path.
web_browser_homepage	string value	Sets the location of Pro/ENGINEER browser homepage.

Note: In Pro/ENGINEER Wildfire it is not necessary to set the config.pro option search_path. By default, when a Windchill server is your primary server, the entire primary server with active workspace is in the Pro/ENGINEER search path.

Setting File Retrieval Options

The config.pro options that specify storage and retrieval directories, including such options as the following:

- start_model_dir
- pro_library_dir
- pro_format_dir
- pro_materials_dir

- pro_group_dir
- pro_symbols_dir
- pro_catalog_dir

can be set to point to Windchill cabinets. For example, the value of start_model_dir is set to point to a Windchill library cabinet using the following syntax (<server_alias> is the name you assign to the server in the Server Registry):

```
start_model_dir wtpub://<server_alias>/Libraries/<library_name>
```

Similarly, the value of pro_group_dir is set to point to a Windchill product cabinet using the following syntax:

```
pro_group_dir wtpub://<server_alias>/Products/<product_name>
```

Note: If you retrieve an object from any location other than the primary server, it will be treated as if it were newly created in the Pro/ENGINEER session. This means that actions on the object (for example, save or requesting checkout) are done in the context of the primary server, not the location from which the object was retrieved.

Config.pro options that point to a specific file, including such options as the following:

- intf_in_use_template_models
- template_designasm
- template_mold_layout
- template_ecadprt
- template_solidpart

can be set to point to Windchill file locations using a string of the proper syntax and the name of the CAD document that manages the file, as in the following example:

```
template_solidpart
wtpub://<server_alias>//libraries/Templates/template_solid_inlb
s.prt
```

Configuring Windchill for Interoperation with Pro/ENGINEER

The following sections describe customization activities performed in some cases using Windchill properties files and in other cases using preferences that can be set using the Windchill **Preference Manager (Site > Utilities > Preference Manager)**. Preferences may also be set at an organization or context level (**<Context> > Utilities > Preference Manager**). Some preference settings can be accessed and overridden by end users. Some preferences can be locked by an administrator, to prevent users from overriding them.

Because the Windchill preferences are organized within the **Preference Manager** by category, this guide will identify preferences by including the hierarchy to which they belong (for example, Display > Workspace).

Displaying the Workspace

When you access Windchill through the Pro/ENGINEER embedded browser, you can always access and create workspaces. However, when Windchill is accessed through a standalone browser, default settings do not display links to the **My Workspaces** page where workspaces are listed and created in Windchill.

You can enable the display of the workspace (including the display of the **Workspaces** minor tab for a context) by setting the preference Display > Workspace to "Yes" (default is "No").

Managing CAD Document and WTPart Naming and Numbering

You can specify how newly-created CAD documents (EPMDocuments) and parts (WTParts) are named and numbered using a policy-managed method. Alternatively, you can use a customization of the Windchill Naming service to specify the names and numbers of CAD documents only. These two options are discussed in the following sections.

Note: Preferences referred to in the following explanation of policy-managed naming and numbering are explained in the section [Windchill Preferences for Naming and Numbering](#).

Policy-Managed Naming and Numbering

Pro/ENGINEER Wildfire supports four policies to determine how newly-created objects (either CAD documents or WTParts) are named and numbered. The four policies can be described briefly as follows:

- Auto-numbering
 - The CAD document Number is server-assigned (either OOTB or per your customization)

Note: Customization of autonumbering must ensure that unique numbers are assigned.

- If the system parameter PTC_COMMON_NAME is created during creation of the Pro/ENGINEER model file, its value is copied to the CAD document Name. If Common Name is left blank in Pro/ENGINEER, the default value for the CAD document Name is copied from the Pro/ENGINEER model name. The file extension (.prt) can be optionally dropped (controlled by a preference).
- The WTPart Number is provided by the WTPart Number generator.
- The default value for the WTPart Name is copied from the current value of the CAD document Name at the time the WTPart is created. If a file extension is present in the CAD document name, it can be optionally dropped when set in WTPart (controlled by a preference).
- In any create and edit user interface, the CADDoc and WTPart NAME field is editable.
- Auto numbering is the default, out-of-the-box naming and numbering policy. The default system does not allow editing of the number by users.
- Name-driven
 - If the CAD document Number is not set up to be server-assigned, then the CAD document Number is copied from the Pro/ENGINEER file name (the file extension can be dropped – controlled by a preference).
 - If the system parameter PTC_COMMON_NAME is created during creation of the Pro/ENGINEER model file, its value is copied to the CAD document Name. If Common Name is left blank in Pro/ENGINEER, the the default value for the CAD document Name is copied from the Pro/ENGINEER model name (file extension can be dropped – controlled by a preference)
 - If the WTPart Number is not set up to be server-assigned, then the default value for the WTPart Number is copied from the CAD document Number (the file extension can be dropped – controlled by a preference)
 - The default value for the WTPart Name is copied from the current value of the CAD document Name at the time the WTPart is created (the file extension can be dropped – controlled by a preference)
 - In any create and edit user interface, the CAD document and WTPart NAME and NUMBER fields are editable by the user.

- Parameter-driven
 - The CAD document Number is copied from the value of the Pro/ENGINEER designated parameter identified by the preference, Windchill Workgroup Manager > Client > Upload > Numbering Parameter. (If no preference value is set, Number assignment follows that of the name-driven policy.)
 - The value for the CAD document Name is copied from the Pro/E designated parameter identified by the preference, Windchill Workgroup Manager > Client > Upload > Naming Parameter.
 - The value for the WTPart Number is copied from the value of the Windchill attribute identified by the preference, Windchill Workgroup Manager > Server > Auto Associate > Numbering Parameter.

Note: Setting this preference takes precedence over auto-numbering, to facilitate user intent in the auto association action. Auto-numbering rules (if any) are observed if this preference is unset.

 - The value for the WTPart Name is copied from the value of the Windchill attribute identified by the preference, Windchill Workgroup Manager > Server > Auto Associate > Naming Parameter.

Note: If this preference is unset, name-driven (not autonumbering) policy is observed, even if auto-numbering is rules are otherwise in effect.

Name-driven and parameter driven policies can only be used in object-driven creation of objects as they require a source object to create a new object. These policies are used during upload (when a new CAD document can be created based on a model file), and auto-associate, (when a new WTPart may be created for a CAD document).

Note: If the designated parameters change after the creation of objects, neither the associations nor the names of CAD documents or WTParts change.

- Custom

The Object Initialization Rules Administrator, available on the Windchill PDMLink or Windchill ProjectLink **Utilities** tab, provides a way to specify default values for the attributes of a specific object type. The default values are then used when the Windchill solution creates objects of that type. These specifications are called *rules*. Each rule can contain default values for one object type. The rules that are set only apply when the Windchill solution that is used to create an object does not set a corresponding value. Rules can be setup to provide auto-number generation, but they can also be setup to provide custom behavior (see [Customizing the Naming Service](#)). Rules are also set per container, allowing there to be different naming/numbering policies on different containers.

Note: Regardless of the naming and numbering policy used, when creating a new object, system uniqueness constraints require that the CAD document attributes Number and File Name must both be unique within a Windchill PDMLink site or within each Windchill Workgroup Manager project.

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 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 or attribute preferences are set in the Windchill **Preference Manager**, then policy is parameter-driven.
- Otherwise, the policy is name-driven.

Windchill Preferences for Naming and Numbering

The preferences for parameter-driven naming and numbering policy in the Windchill **Preference Manager** are the following for auto-associate:

Windchill Workgroup Manager > Server > Auto Associate > Numbering
Parameter = *<some string parameter>*

Windchill Workgroup Manager > Server > Auto Associate > Naming
Parameter = *<some string parameter>*

Note: Pro/ENGINEER parameters are passed to Windchill in all uppercase characters. The string value must match the name as seen in Pro/ENGINEER for the designated parameter.

The following preferences specify parameter-driven naming and numbering during upload:

Windchill Workgroup Manager > Client > Upload > Numbering Parameter =
<some string parameter>

Windchill Workgroup Manager > Client > Upload > Naming Parameter =
<some string parameter>

By default, none of these four preferences has a value.

Note: The preference Numbering Parameter cannot be used with family table parts that have more than one level of nested instances (upload fails with a uniqueness exception). When Numbering Parameter is used with family table parts that have only one level of instances, values of this parameter need to be different for each instance. You can do this by adding this parameter as a family table column and providing a different number value for each instance.

When set to "Yes" (default is "No"), the following preferences specify dropping the file extensions (such as, ".prt" or ".asm") when naming and numbering new objects during an auto-associate action:

Windchill Workgroup Manager > Server > Auto Associate > Auto Associate Truncate Name File Extension

Windchill Workgroup Manager > Server > Auto Associate > Auto Associate Truncate Number File Extension

When set to "Yes" (default is "No"), the following preferences specify dropping file extensions during upload:

Windchill Workgroup Manager > Client > Upload > Upload Drop Name File Extension

Windchill Workgroup Manager > Client > Upload > Upload Drop Number File Extension

Customizing the Naming Service

The Naming service uses the Windchill service delegate mechanism to allow you to specify the following for the new EPMDocument to be created:

- Set a number for the EPMDocument
- Set a name for the EPMDocument

Note: The Naming service is for the upload action only. The order of precedence used by the system for naming policies and customizations is as follows:

- Name:
 1. Naming service customization
 2. Explicitly assigned Common Name through the Pro/ENGINEER **File > New** dialog box
 3. Name parameter (Windchill Workgroup Manager > Client > Upload > Naming Parameter = *<some string parameter>*)
 4. File Name (The preference Windchill Workgroup Manager > Client > Upload > Upload Drop Name File Extension will take effect only if Name is assigned based on File Name (CAD Name))
- Number:
 1. Naming service customization
 2. Number parameter (Windchill Workgroup Manager > Client > Upload > Numbering Parameter]

3. File Name (The preference Windchill Workgroup Manager > Client > Upload > Upload Drop Number File Extension will take effect only if Number is assigned based on File Name [CAD Name])

Use the following steps to customize the Naming service:

1. Create a Java Class that implements the interface EPMDocumentNamingDelegate. The interface definition is as follows:

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

public interface EPMDocumentNamingDelegate
{
    public void validateDocumentIdentifier(DocIdentifier
docIdentifier);
}
```

The definition of Class DocIdentifier is as follows:

```
package com.ptc.windchill.uwgm.proesrv.c11n;
import java.util.HashMap;
public class DocIdentifier
{
    {
        private String m_modelName;
        private String m_docName;
        private String m_docNumber;
        private HashMap m_parameters;
    }

    public DocIdentifier(String modelName, String docName, String
docNumber, HashMap params)
    {
        m_modelName = modelName;
        m_docName= docName;
        m_docNumber= docNumber;
        m_parameters= params;
    }

    /** get the CAD Name for the model */
    public String getModelName()
    {
        return m_modelName;
    }
}
```

```

    }

    /** get the EPMDocument name for the model */
    public String getDocName()
    {
        return m_docName;
    }

    /** set the EPMDocument name for the model */
    public void setDocName(String docname)
    {
        m_docName = docname;
    }

    /** set the EPMDocument number for the model */
    public void setDocNumber(String docnumber)
    {
        m_docNumber = docnumber;
    }

    /** get the EPMDocument number for the model */
    public String getDocNumber()
    {
        return m_docNumber;
    }

    /** get the Pro/E designated parameters for the model. These are
    name-value pairs indexed by the name */
    public HashMap getParameters()
    {
        return m_parameters;
    }
}

```

2. In the new class, implement the business logic for naming/numbering EPMDocument in the method:

```

public void validateDocumentIdentifier(DocIdentifier
docIdentifier)

```

- The DocumentIdentifier object has the EPMDocument name and number information for the EPMDocument that will be created by the Upload

Service.

Use the `DocIdentifier.getModelName()` to get the CAD Name of the EPMDocument that this `DocIdentifier` object represents.

- The Pro/ENGINEER designated parameters may be used to set EPMDocument numbering/naming.

Use the `DocIdentifier.getParameters()` to get the associated parameters.

Use the “set” methods on the `DocIdentifier` to set the new name/number values. The Upload Service will use these suggestions if they are feasible.

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

```
<Service context="default"
name="com.ptc.windchill.uwgm.proesrv.c11n.EPMDocumentNamingDelega
te" targetFile="codebase/service.properties">

<Option cardinality="singleton" requestor="wt.epm.EPMDocument"
serviceClass="com.ptc.windchill.uwgm.proesrv.c11n.EPMDefaultDocument
NamingDelegate"/>

</Service>
```

Then use the `xconfmanager` tool to apply the changes to `service.properties` file (run `xconfmanager -p`)

Use the path of your class in place of the value of `serviceClass` (that is, replace `"com.ptc.windchill.uwgm.proesrv.c11n.EPMDefaultDocumentNamingDelegate"` with the path to your class).

4. Restart the method server.

Preferences That Affect Resolution of Incomplete Dependent Objects

While Pro/ENGINEER Wildfire can display incomplete dependents in the workspace and upload them to the server, Windchill does not allow the check-in of incomplete dependent objects. A preference in the Windchill **Preference Manager**, **Display > Incomplete object resolution**, can be set to one of four values to allow check-in of an assembly by ignoring certain dependencies or to disallow the ignoring of dependencies (and thereby disallow the check-in of assemblies containing unresolved incomplete dependent objects).

If Incomplete object resolution is set to one of the following values, the behaviors described result:

- If set to "Ignore optional dependencies" (default), any CAD tool internal and non-required dependencies are ignored.

- If set to "Ignore optional reference dependencies," any reference dependencies are ignored.
- If set to "Ignore internal dependencies only," only internal CAD tool dependencies are ignored.
- If set to "Do not allow to ignore," no incomplete dependencies can be ignored.

In addition, the following preferences control system behavior toward incomplete dependent objects during the Check In action.

The preference, Windchill Workgroup Manager > Server > Check In > Resolve Incomplete Objects, controls the default behavior whether or not to resolve incomplete objects automatically upon Check In.

If the preference Resolve Incomplete Objects is set to "yes," or if the **Auto resolve incomplete objects** check box is selected on the **Check In** page, the preference, Windchill Workgroup Manager > Server > Check In > Update Incomplete Objects on Server, controls the default behavior whether or not to update incomplete objects on server upon resolving incomplete objects upon Check In.

Note: The server side ghost resolution setting (Incomplete object resolution) is used when a user selects the **Auto resolve incomplete objects** option on the **Check In** page, or in case of using the Pro/ENGINEER **File > Checkin > Auto** command. The setting on the server is not used if the user clears the **Auto resolve incomplete objects** option on the **Check In** page (the user selection not to resolve incomplete objects from the **Check In** page has precedence over the server-side setting). This could result in a valid check-in failure if there are incomplete objects in the check-in list.

Soft Typing CAD Documents

About Soft Typing

In CAD authoring tools, you can create restricted value parameters that use definitions from a restriction definition file. The restriction definition file defines the parameter name, type, value, range of values and a default value. This feature is useful because it allows you set an attribute for a specific object-type and then set a specific range of acceptable values.

The Windchill counterpart to restricted value parameters is the soft typing feature. Soft typing is accessible through the **Type and Attribute Manager** and allows you add constraints (such as a value or range of values) to an instance-based attribute. Additionally, you can use the **Type and Attribute Manager** to add attributes to the EPMDocument type and its soft types: both the CAD document and dynamic document (Arbortext document) soft types.

For CAD documents, there is one system-provided soft type, the Workgroup Manager CAD Document, that an administrator can modify to add attributes that can have different values for each iteration of the object that an administrator can

modify. This soft type cannot be deleted nor sub-typed further; however, there are additional soft types, related to CAD documents. They are the following:

- Workgroup Manager CAD Document Master soft type (on CAD Document Master type)

Attributes that are added to this soft type have only one value for all iterations. Changing the value of an attribute on a CAD Document Master soft type changes that value for all iterations. This type of attribute is the Windchill equivalent of a Pro/INTRALINK non-versioned attribute.

- Workgroup Manager CAD Document Uses Link soft type (on CAD Document Uses Link type)

Attributes that are added to this soft type are specific to the use of an iteration of an object. For example, if there are four bolts of the same type (bolt.prt) in an assembly, and each bolt needs to be tightened to a specific torque, you can add torque to the Uses Link soft type and then apply a different value to each occurrence of the bolt in the assembly. (In contrast, if you instead add this attribute to the CAD Document soft type, then all bolts in all assemblies would have the same torque wherever they are used.)

- Workgroup Manager CAD Document Reference Link soft type (on CAD Document Reference Link type)

Attributes that are added to this soft type apply to reference links (again, not to the CAD document, itself).

For details on soft typing, selecting attributes and setting constraints, refer to the *Windchill Business Administrator's Guide*.

Mapping Pro/ENGINEER Parameters to Windchill Attributes

Pro/ENGINEER Wildfire lets you map Pro/ENGINEER designated parameters onto Windchill attributes. Attribute mapping transfers parametric information from the CAD models created in Pro/ENGINEER to the Windchill system. The attribute mapping can be done as follows:

- By implicit parameter-to-attribute mapping
- By explicit parameter-to-attribute mapping

Implicit Parameter-to-Attribute Mapping

Implicit parameter-to-attribute mapping occurs when there is an attribute in Windchill with a name (all uppercase) identical to the name of a designated parameter in a Pro/ENGINEER model file and there is no conflicting mapping specified on the attribute. When the Pro/ENGINEER model file is uploaded into Windchill as content of a CAD document, the values of the Pro/ENGINEER parameter are transferred to the Windchill attribute.

Note: For CAD documents, there is one system-provided type that an administrator can modify. This type cannot be deleted nor sub-typed. In order to assign any attributes to CAD documents, the administrator must go into the **Type and Attribute Manager** and add attributes to the default CAD document soft type.

Explicit Parameter-to-Attribute Mapping

Two specific cases where explicit mapping can be helpful are the following:

- When parameter names in Pro/ENGINEER are in multi-byte characters (multi-byte character names are not supported in Windchill)
- When you are dealing with older files with legacy parameter names

Note: In older versions of Pro/ENGINEER (prior to Wildfire 2.0) the system does not keep track of the parameter-to-attribute mapping of each iteration of an object. This could lead to loss of historical data if the mapping file is changed between early and later iterations. For example, consider the situation where "LENGTH" in a Pro/ENGINEER file is mapped to "length" in Windchill for the first three iterations of an object (A.1, A.2, A.3), but the mapping file is then changed so that "LENGTH" is mapped to "diameter" in Windchill when iterations A.4, and A.5 are created. Upon attempting to retrieve A.1, the system looks into the mapping file, trying to find "diameter" instead of "length". Because it cannot find "length," there is no way to show the value for "LENGTH" in Pro/ENGINEER for iteration A.1.

In order to explicitly map Pro/ENGINEER designated parameters to Windchill attributes, entries must be added to the **Attribute Mapping** tab, which is available when you select the Workgroup Manager CAD Document type on the **Type Manager** tab of the Windchill **Type and Attribute Manager**.

Example:

To map a Pro/ENGINEER designated parameter MCOST to an attribute named MfgCost in Windchill, do the following:

1. On the **Type Manager** tab of the Windchill **Type and Attribute Manager** select the Workgroup Manager CAD Document type and then select **Edit**.

The Workgroup Manager CAD Document type is checked out.

2. On the **Attribute Mapping** tab displayed for the Workgroup Manager CAD Document, click **Add Mapping**.
3. In the Select Attribute window, navigate to the Windchill attribute MfgCost, select it and click Select.

The attribute MfgCost is added to the **Attribute** column.

4. Ensure that Pro/ENGINEER is displayed in the Mapping Context column, or click in the cell to select Pro/ENGINEER from the list of CAD applications.

5. In the **Mapping** column, enter MCOST.
6. Check in the Workgroup Manager CAD Document type.

Note: For CAD documents, there is one system-provided type that an administrator can modify. This type cannot be deleted nor sub-typed. In order to assign any attributes to CAD documents, the administrator must go into the Type Manager and add attributes to the default CAD document soft type.

Parameter names in Pro/ENGINEER are not case-sensitive; however, the mapping on the **Attribute Mapping** tab must contain the name of the parameter in uppercase. Windchill attributes are case-sensitive and should always be mapped accurately for the mapping to work correctly.

Upload Behavior for Attribute Mapping

On upload, the service first looks for a mapping defined on the **Attribute Mapping** tab. The Pro/ENGINEER designated parameter name must always be in upper case. If a mapping does not exist, the Pro/ENGINEER designated parameter is converted to upper case and the service looks for a Windchill attribute definition by this name. If an attribute definition is not found, the upload service reports a conflict (succeeds with a warning written to the Event Manager). All conflicts are reported in the Event Manager in terms of the Pro/ENGINEER designated parameter name.

Download Behavior for Attribute Mapping

On download, the reverse is done; that is, the attribute name is mapped to the Pro/ENGINEER designated parameter name. It is possible that two or more Pro/ENGINEER designated parameters map to a single Windchill attribute. In this case, the Pro/ENGINEER designated parameter that is last among those that map to a common attribute is chosen.

Note: Once added to the **Attribute Mapping** tab on the server side, Windchill attribute-to-designated parameter mappings should not be removed or modified. Doing so might affect the models already stored in the Windchill database (including historical data and released designs), particularly if such models have attribute values modified through the Windchill user interface (as opposed to modifications done in a Pro/ENGINEER session).

For the same reason, new mappings should be added only for Pro/ENGINEER designated parameters that do not exist yet in the models already stored in the Windchill database. Therefore, it is strongly recommended that:

- The Windchill server administrator adds any new mappings as part of the process of adding new attribute definitions in Windchill.

- Workgroup Manager for Pro/ENGINEER and Pro/ENGINEER Wildfire PDM users do not designate Pro/ENGINEER parameters which are not mapped (implicitly or explicitly) onto a Windchill attribute; the upload service provides warnings in the Event Console in such cases, and users are encouraged to undesignate the parameters in Pro/ENGINEER and upload the models again.

Resolving Type Conflicts Between Pro/ENGINEER Parameters and Windchill Attributes

To avoid upload problems in case of a mismatch between the types of a Pro/ENGINEER parameter and the Windchill attribute to which it is mapped, you can set the following property in the site.xconf file:

```
<Service context="default" name="wt.epm.attributes.EPMAttributeDelegate"
targetFile="codebase/service.properties">

    <Option cardinality="singleton" requestor="wt.iba.value.IBAHolder"
selector="PROE"
serviceClass="wt.epm.attributes.EPMAttributeDelegateWithWarnings"/>

</Service>
```

Setting this property and propagating it using xconfmanager allows the system to ignore the mismatch and continue the upload.

Customizing the Parameters in the Download Service

Windchill provides a server-side delegate that can be used to insert parameters into a Pro/ENGINEER model upon download. This mechanism can be used to pass information from the server down to Pro/ENGINEER, where it can be used like any other Pro/ENGINEER 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, unlike system parameters such as PTC_WM_ITERATION, are not updated in the Pro/ENGINEER session or the local cache 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 model download. If the CAD document is later renumbered, the value in the Pro/ENGINEER session or the client cache will not be 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
{
/*
Implement this API to return list of parameters added by
customization along with it's type (customization profile of the
server). For example "WT_CADDOC_NUMBER" custom parameter will
be of type "String.class" (the java class)
*/
    // getAvailableAttributes() returns
    // HashMap<String, Object> which contains
    // HashMap<Attribute name, Attribute type>
    HashMap getAvailableAttributes();
/*
This is the API, invoked by the download service on download, to
be implemented for the customization. Create and return a
HashMap where key is input object and value is HashMap of
parameter name - value pairs that must be propagated to Pro/E
part represented by the EPMDocument (input object). Use the
getCADName() API on the EPMDocument to identify the Pro/E part
*/
    // getModeledAttributes(Collection docs) returns
    // HashMap<input object, HashMap<Attribute name, Attribute
value>>
    HashMap getModeledAttributes(Collection docs) throws
WTException;
}

```

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

```

<Service context="default"
name="com.ptc.windchill.uwgm.proesrv.c11n.ModeledAttributesDele
gate" targetFile="codebase/service.properties">

    <Option cardinality="singleton"
requestor="java.lang.Object"
serviceClass="com.ptc.windchill.uwgm.proesrv.c11n.DefaultModele
dAttributesDelegate"/>

</Service>

```

Then use the xconfmanager tool to apply the changes to service.properties file (run xconfmanager -p)

Use the path of your class in place of value of serviceClass (that is, replace com.ptc.windchill.uwgm.proesrv.c11n.DefaultModeledAttributesDelegate with the path to your class).

3. Restart the method server.

Configuring the Build Rule

Windchill uses a combination of Windchill Preference Manager preferences and Windchill properties to control the following functions during execution of the build rule:

- What attributes to publish from a CAD document to a build target, based on the team template of the target
- Specification of whether to use existing part usage links or create new usage links
- Vetoing operations on EPM objects owned by specific applications
- Enabling or disabling the creation of as stored configurations

Controlling Attribute Publishing

You can set the following preferences to define the attributes that are published to the indicated build targets (these preferences are all listed under the categories EPM Service Preferences > Build Service Preferences)

- Attributes to be published on Link
- Attributes to be published on Master
- Attributes to be published on Occurrence
- Attributes to be published on Part

For each preference, the default value is an asterisk (*), which specifies that all attributes are to be published. Specify the specific attributes to be published by replacing the asterisk with a delimiter-separated list of attributes (or specify no attributes by removing the asterisk).

Note: The preference, EPM Service Preferences > Build Service Preferences > Attributes Delimiter, defines the delimiter that separates the listed attributes. A comma (,) is the default value.

The following table lists preferences for the build service:

Preference	Values	Description
Allowed edit of part structure built by build service	Yes No (default)	In the case where a CAD document is owned by an ECAD application, allows editing of the part structure built by build service
Attributes Delimiter	, (default) <character value>	Identifies the delimiter used in listing attributes to be published

Preference	Values	Description
Attributes to be published on Link	<i><String(s), separated by delimiter character set in preference, Attributes Delimiter></i>	Identifies attributes to be published on the member link
Attributes to be published on Master	<i><String(s), separated by delimiter character set in preference, Attributes Delimiter></i>	Identifies attributes to be published on the master
Attributes to be published on Occurrence	<i><String(s), separated by delimiter character set in preference, Attributes Delimiter></i>	Identifies attributes to be published on an occurrence
Attributes to be published on Part	<i><String(s), separated by delimiter character set in preference, Attributes Delimiter></i>	Identifies attributes to be published on the part

Specifying Usage Links

To have the build process use existing usage links, leave the property `wt.epm.build.subsumeLinks` set to the default value `true`.

To specify that the build process creates new usage links, set the property `wt.epm.build.subsumeLinks` to `false`.

If you want the build process to create new links using your usage link class (a subclass of `WTPartUsageLink`) set `wt.epm.build.linkClass` to your usage link class. The default value is `wt.part.WTPartUsageLink`.

Vetoing Operations Based on Owner Application

By default, the following operations are set for a client-side veto on objects owned by specific applications:

- Add link
- Change folder (Move)
- Check-in
- Checkout
- Delete IBA
- Delete link

- Delete (object)
- Modify property
- Revise

Each operation is controlled by a property that contains the operation name, and the operation is vetoed for all applications listed in the default value. To enable the operation for objects owned by a specific application, remove that application's name from the default listing.

For example, to enable moving (changing the folder) of objects owned by the ProINTRALINK Gateway application, you modify the property wt.epm.veto.change.folder. The default value for the property is:

OPTEGRAGATEWAY,PROINTRALINKGATEWAY,PROPDMGATEWAY,WORKMANAGERGATEWAY,IDEASTDM.

Therefore, you enable the move operation for ProINTRALINK Gateway-owned objects by removing PROINTRALINKGATEWAY, from the comma-delimited list, yielding the following setting for the property:

wt.epm.veto.change.folder=OPTEGRAGATEWAY,PROPDMGATEWAY,WORKMANAGERGATEWAY,IDEASTDM.

Note: The property wt.epm.veto.delimiter defines the character used to delimit the list of owning applications. The default value is comma (.).

Configuring the Initial Collection of Objects for Actions

The Windchill Preference Manager allows you to set preferences for default collection rules on a general, as well as per-PDM-action, basis. To set preferences for default collection rules, navigate to the appropriate section of the **Preference Manager** as explained for the following options:

- For general setting of collection rule defaults, navigate to the Display > General Collector category.
- For the Add to Project, Move, or Send to PDM actions, navigate to the Integral Operations > <Action> Collector category.
- For the Delete or Revise actions, navigate to the <Action> > Collector category.
- For the Save As action, navigate to either the Save As > From Commonspace Collector category, or the Save As > From Workspace Collector category, as appropriate.
- For other actions, navigate to the Windchill Workgroup Manager > Server > <Action> > Collector category. Collection options are available for the following actions:

- Add to Workspace and Check Out
- Checkin
- Edit Attributes
- Export from Workspace
- Remove from Workspace
- Rename
- Set State
- Undo Check Out
- Update
- Upload

Within each of the collector categories, you are able to set preferences that determine default collection rules applied when an action is initiated. The following table lists and describes the collection preferences that may be available within the respective categories (not all preferences are available in all categories):

Preference	Values	Description
Include dependent CAD Documents	All Required None	Allows user to specify which dependent CAD documents for the collected CAD documents will be by default added to the collection
Include dependent Documents	All None	Allows user to specify which dependent documents for the collected documents will be by default added to the collection
Include dependent Parts	All None	Allows user to specify which dependent parts for the collected parts will be by default added to the collection
Include related CAD Documents	All Initially Selected Only None	Allows user to specify which CAD documents associated to the collected parts will be by default added to the collection

Preference	Values	Description
Include related Documents	All Initially Selected Only None	Allows user to specify which documents associated to the collected parts will be by default added to the collection
Include related Drawings	All Initially Selected Only None	Allows user to specify which drawings related to the collected CAD documents or parts will be by default added to the collection
Include related Family table objects	All None Initially Selected Only	Allows user to specify which family table objects related to the collected generic or instances will be by default added to the collection
Include related Generics	All None Initially Selected Only	Allows user to specify which generics related to the collected instances will be by default added to the collection
Include related Notes	All None Initially Selected Only	Allows user to specify which notes related to the collected parts will be by default added to the collection
Include related Parts	All None Initially Selected Only	Allows user to specify which parts related to the collected documents, CAD documents, or dynamic documents will be by default added to the collection

Note: Out-of-the-box default settings may vary, depending on the action. You can specify the system default setting by selecting **Revert to Default** on the **Set Preference** page.

In addition, within each collection category, the preference, Display collected objects, allows users to specify the way in which collected objects are listed in the table. The options are as follows:

- As a List (default)
- As a Structure (shows the object hierarchy)
- As a Structure with Associated Objects

Configuring Check In

The following sections describe preferences used with the Check In action.

Enabling As Stored Configurations

The property Windchill Workgroup Manager > Server > Check In > Create As Stored specifies by default ("Yes") to create an As Stored configuration at the time objects are checked in. If set to "No," an As Stored configuration is not created.

Enabling Baseline Creation

The property, Windchill Workgroup Manager > Check In > Create Baseline upon Check In, specifies by default ("Yes") to create a baseline at the time objects are checked in. If set to "No," a baseline is not created.

Managing ModelCHECK Validation during Check In

Pro/ENGINEER Wildfire allows you to use ModelCHECK™ as a "gatekeeper" to the Windchill database, which means that to be successfully checked in, models must meet ModelCHECK criteria. This gatekeeper functionality is controlled by the Windchill server, which references the read-only ModelCHECK parameters contained in the models.

Configuring ModelCHECK in Pro/ENGINEER

To enable the gatekeeper functionality, you must first edit the ModelCHECK configuration to enable ModelCHECK to add the required parameters to the data. From the **ModelCHECK Configuration** dialog box within Pro/ENGINEER, edit the initialization file (config_init.mc) and change the following objects:

- Set **ADD_DATE_PARM** to **Y**

This creates a parameter called MODEL_CHECK in the model files of all models that are checked. This parameter contains the date and time when ModelCHECK was last run.

- Set **ADD_ERR_PARM** to **Y**

This creates a parameter called MC_ERRORS in all models that are checked. This parameter contains the number of errors found in the model when ModelCHECK was last run.

- Set **ADD_CONFIG_PARM** to **Y**

This creates a parameter called MC_CONFIG in all models that are checked. This parameter contains the names of the ModelCHECK configuration files used for a final check of the model.

- Set **ADD_MODE_PARM** to **Y**

This creates a parameter called MC_MODE in all models that are checked. This parameter contains the mode in which ModelCHECK was run on the model.

After editing the settings, save the configuration. As a result of this change, anytime you run ModelCHECK, these new read-only parameters are added to the data files.

Note: For more information, please see the ModelCHECK Help Topic Collection documentation.

Configuring ModelCHECK in Windchill

After configuring ModelCHECK in Pro/ENGINEER, configure Windchill as follows:

1. In the Windchill **Type and Attribute Manager**, on the **Attribute Definition Manager** tab, create attributes with following names and types:
 - MC_ERRORS -- (integer)
 - MODEL_CHECK -- (string)
 - MC_CONFIG -- (string)
2. On the **Type Manager** tab:
 - a. Expand the **EPM Document** node and check out the Workgroup Manager CAD Document type
 - b. Select the **Template** tab in the right-hand panel, click **Update**, and **Add Attribute**.

Select and add the attributes you defined in Step 1 to the Workgroup Manager CAD Document type.
 - c. Click **Save**, then check in the Workgroup Manager CAD Document
3. Set the preference Windchill Workgroup Manager > Check In > ModelCHECK Validation to yes (the default is no) to enable ModelCHECK.
4. Set the appropriate modelcheck preferences (also in Windchill Workgroup Manager > Check In >) to configure ModelCHECK, as follows:
 - Set ModelCHECK Number of Errors to specify the maximum number of ModelCHECK errors allowed. The default is 0.
 - Set ModelCHECK Number of Hours to specify the maximum allowable hours between a ModelCHECK verification at the client and the actual model checkin to Windchill. The default is 24.

- Set ModelCHECK Configuration to specify the ModelCHECK Configuration files to be used for validation for each LifeCycle name in a specific syntax.

(For example:

```
<Lifecycle_1>:<mch_file1>,<mcs_file1>,<mcn_file1> <Lifecycle_2>:
<mch_file2>,<mcs_file2>,<mcn_file2>.....
```

The configuration specified by the "Default" life cycle state is fallback behavior.

Typical examples are as follows:

- Default:check/default_checks.mch,start/nostart.mcs,constant/inch.mcn
- Basic:check/basic_checks.mch,start/basic_start.mcs,constant/inch.mcn
- Release:check/release_checks.mch,start/release_start.mcs,constant/inch.mcn
- Approval:check/approval_checks.mch,start/approval_start.mcs,constant/inch.mcn
- Review:check/review_checks.mch,start/review_start.mcs,constant/inch.mcn

Note: You need to configure the ModelCHECK conditions in Pro/ENGINEER to obtain the appropriate configurations for the respective LifeCycle Name. A typical example to configure condition.mcc is as follows:

Example: Edit condition.mcc:

```
IF ( PTC_WM_LIFECYCLE EQ Basic)
config=(check/basic_check.mch)(start/basic_start.mcs)(constant/inch.mcn)(status/basic_status.mcq)
```

```
IF ( PTC_WM_LIFECYCLE EQ Release)
config=(check/release_check.mch)(start/release_start.mcs)(constant/inch.mcn)(status/release_status.mcq)
```

```
IF ( PTC_WM_LIFECYCLE EQ Approval)
config=(check/approval_check.mch)(start/approval_start.mcs)(constant/inch.mcn)(status/approval_status.mcq)
```

```
IF ( PTC_WM_LIFECYCLE EQ Review )
config=(check/review_check.mch)(start/review_start.mcs)(constant/inch.mcn)(status/review_status.mcq)
```

ELSE set the fallback (CADDocument OIR - LifeCycle Name at Container level)

```
ELSE
config=(check/basic_check.mch)(start/basic_start.mcs)(constant/inch.mcn)(status/basic_status.mcq)
```

Note: The Windchill OIR (Object Initialization Rule) pertaining to a CAD document for a container should be examined for the default LifeCycle Name. The fallback value of conditions.mcc for a workspace should match that of the OIR Lifecycle Name.

Enabling Support for Custom Parts

In the Pro/ENGINEER Wildfire HTML client, you can enable support for custom parts, which extend wt.part.WTPart; however, a custom part must be modeled before any changes are made to the Pro/ENGINEER Wildfire HTML client. (For information on extending the Windchill object model, see the *Windchill Application Developer's Guide* and the *Windchill Customizer's Guide*.)

The Pro/ENGINEER Wildfire HTML client permits use of custom parts in most operations, including download, check out, check in, associate, disassociate, and so on; however, the operations used to create parts, **Create Part** and **Auto Associate Part**, are specific to WTPart. Additionally, when you view the properties of a custom part, any IBAs you may have added to the custom part can be seen; however, newly modeled information is not displayed.

Whenever "Part" is available in the object type list on the Pro/ENGINEER Wildfire HTML client object selection page, if "Part" or "All" is selected, both WTPart objects and custom part objects are listed in the page's results table.

Automatic part generation is supported through the **Auto Associate Part** action available on the workspace properties page. To enable automatic custom part generation when using this command, however, you must either create or modify your automatic part creator. For more information, see [Customizing Auto Associate](#), later in this chapter.

Modifying the Properties Page

To configure a custom part-specific properties page you have to create a properties page and/or template processor. For details on how to do this see "Customizing the HTML Client" in the *Windchill Customizer's Guide*.

Modifying the HTML Client Object Selection Page

To enable recognition of custom parts as a sub-class of WTPart and not just the supported type in the Pro/ENGINEER Wildfire HTML client object selection page's default implementation, you must add support for the custom part in the configured wt.query.SearchAttributeListDelegate. (For more details see the section, [Customizing the HTML Search](#), later in this chapter.)

In addition you must modify the Pro/ENGINEER Wildfire HTML files that use the object selection page, and use the xconfmanager 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 id is referred to as the query value. (See [Customizing the HTML Client Object Selection Page](#) later in this chapter for more details.)

Replacing WTPart

If you want your site to only use custom part and not WTParts, then do the following:

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 id for WTPart to the custom part type id you created in Step 1.
3. Restart the method server.

Supporting WTPart and Custom Part

If your site will be using both WTParts and custom parts, then do the following:

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 id for WTPart the custom part type id you created in step 1.
3. To add an “All” type list entry for a type list, add an entry with the ALL type id used by the configured search attribute list.
4. Restart the method server.

Administering Revision

Administrators can configure how the system behaves during a revision operation. Server side settings can determine whether to:

- 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 autoassociate action

Note: The revision level synchronization behavior described in the section, [Configuring the Synchronization of Revision Levels During Auto Associate](#) also applies to the revise action when AutoAssociateSetRevisionForWTPart is set to true.

The following sections describe each of these configurations.

Configuring the Ability to Set a Revision Level

The preference, Revise > Allow Override on Revise, allows setting the target revision of a object by adding a **Set Revision** control to the **New Revision** page. The preference, Revise > Allow Override On Create CAD Document, allows setting the target revision of a object by adding a **Set Revision** control to the **New CAD Document** user interface. When either of these preferences is set to "Yes," the user is allowed to set the target revision of the object in the respective user interface. When the value is "No," the object is revised to the next revision level in its series. The default is "No."

Properties for New Revisions

The following is a summary of the properties for new revisions:

- By default, all new revisions 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 revision 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 the Synchronization of Revision Levels During Auto Associate

A server-side preference, Windchill Workgroup Manager > Server > Auto Associate > Set Revision For Part, allows you to set the behavior for the revision of CAD documents and parts during the **Auto Associate** operation.

Set Revision For Part can be set to the values "Yes" or "No" (default). When set to "Yes" the revision of a WTPart is set to that of the actively associated CAD document during an auto-associate action.

Auto Associate attempts to set the part's revision to match that of the CAD document when both the following situations apply:

- When an active association is to be created between the part and the CAD document
- and
- Only when the auto-associate action creates a new part for association. A matching revision cannot be set to the working copy of a part. Even if the part is initially checked in, because **Auto Associate** checks out the part before creating an active association, a matching revision is not set to such an existing part.

The following rules apply to both the autoassociate and associate (revising both part and CAD document) actions when Auto Associate > Set Revision For Part is set to "Yes."

- A revision matching that of the CAD document is set to the part when the CAD document revision is higher than that of the part.
- A revision matching that of the CAD document is not set to the part if:
 - The CAD document revision is lower than the part
 - The CAD document and part revisions do not belong to the same revision series

If Set Revision For Part is set to "No," Auto Associate continues without trying to set a revision level.

Customizing Auto Associate

Auto Associate functionality can be customized in the following ways:

- Modifying the implementation of the AutoAssociatePartFinderCreator interface
- Modifying the implementation to search for Customized parts or custom parts
- Customizing the Type of CAD documents that can be actively associated
- Preventing the creation of Parts by Auto Associate
- Controlling the default location of parts created by Auto Associate

Each of these customizations is described in the following sections.

Note: Preferences that control naming and numbering of parts created during Auto Associate are discussed in the section, [Windchill Preferences for Naming and Numbering](#), and also listed in the table of Auto Associate preferences in the section, [Auto Associate Preferences](#), along with the preferences discussed in the following sections on Auto Associate.

Using and Modifying the AutoAssociatePartFinderCreator Interface

AutoAssociate uses the implementation of the AutoAssociatePartFinderCreator interface to perform the following actions

- To search a for matching part
- To create a new part

By default, the AutoAssociate action uses the default implementation of this interface to perform the above-mentioned tasks; however, you can customize the how they are performed using a customized implementation of AutoAssociatePartFinderCreator interface.

The interface is located in
com.ptc.windchill.cadx.autoassociate.AutoAssociatePartFinderCreator.

The AutoAssociatePartFinderCreator interface supports the following methods:

- findOrCreateWTPart method used to search for matching part
- CreateNewWTPart method used to create new part
- findWTPart method (no longer used)
- isNewPart method (no longer used)
- setIsNewPart method (no longer used)

Note: Even though some methods of the interface are deprecated and no longer used, the implementation class should have dummy implementations of these methods in order to compile the class.

Use the following procedure to implement a customized AutoAssociatePartFinderCreator:

1. Derive your customized class as follows:

```
public class CustomFinderCreator implements  
AutoAssociatePartFinderCreator
```

2. Override the following methods:

- public WTPart findOrCreateWTPart(EPMDocument epmDoc,
EPMWorkspace workspace)

This method is invoked for each document selected for auto-associate to search for any matching part. You can customize the criteria used to search the part, and the returned part is used by the action to associate to the document.

- public WTPart createNewWTPart(EPMDocument doc, String partNum ,
String partName,)

This method is invoked for each document selected for auto-associate to create a new part. You can customize the properties of the newly created part. The newly created part is associated to the document by the auto-associate action.

Note: The following methods are deprecated and not currently used by the action; however, you need to provide a dummy implementation of these methods to compile the class properly.

- public boolean isIsNewPart()
- public void setIsNewPart(boolean a_IsNewPart)
- public WTPart findWTPart(EPMDocument epmDoc)

Compile the file and place the class in any appropriate location

3. Modify the following entry in the autoassociate.ini file (file location: <Windchill>\codebase\com\ptc\windchill\cadx\cfg\site\autoassociate.ini):

PartFinder=<class path>.CustomFinderCreator
4. Restart the method server

Modifying the Implementation to Search for Customized Parts or Custom Parts

When performing searches, the default implementation is to search for a WTPart.

Note: When you create a customized part, its master must be WTPartMaster or a subclass of WTPartMaster. The customized part itself must be a WTPart or a subclass of WTPart.

To customize the implementation to search for a customized part that has been implemented in the codebase (for example, wt.part.MyCustomPartMaster), set the preference, Windchill Workgroup Manager > Server > Auto Associate > Custom Class for Auto Associate Part to wt.part.MyCustomPartMaster

Controlling the Associations Formed by Auto Associate

Several preferences affect the type of associations formed during Auto Associate and the type of CAD documents that are allowed to form them.

The preference, Windchill Workgroup Manager > Server > Auto Associate > Disallow Product Structure Links for CAD Document Types, allows you to specify the CAD document types which cannot form an Owner association. These are comma-separated values. The default is <no value>.

The preference, Windchill Workgroup Manager > Server > Auto Associate > Disallow Product Structure Links for CAD Document Sub-Types, allows you to specify the CAD document sub-types which cannot form an Owner association. These are comma-separated values. The default is <no value>.

Note: The allowable values for the preferences, Disallow Product Structure Links for CAD Document Types and Disallow Product Structure Links for CAD Document Sub-Types, are listed in the table of Auto Associate preferences in the section [Auto Associate Preferences](#).

The preference, Windchill Workgroup Manager > Server > Auto Associate > Create Alternate Link On Check In, when set to "Yes," allows a CAD-document-to-part association of the next available type (that is, Content) to be created if the matching part found during Auto Associate already has an Owner association, and allows the checkin to continue. The default is "No" (no Content association is formed and the checkin fails with an overridable conflict).

Controlling the Creation of Parts by Auto Associate

The preference, Windchill Workgroup Manager > Server > Auto Associate > Create Associate New Part, specifies whether a new part should be created if a matching part is not found by Auto Associate. When set to "Yes" (default), if a matching part is not found, a new part is created. When set to "No," no new part is created, and checkin is not blocked.

Controlling the Default Location of Parts Created by Auto Associate

The preference, Windchill Workgroup Manager > Server > Auto Associate > Store New Parts with CAD Documents, when set to "Yes," specifies that the storage location of new part created during Auto Associate be the same as its associated CAD document. By default, the preference is set to "No."

Customizing the HTML Client Object Selection Page

The HTML client object selection page is used in the Pro/ENGINEER Wildfire HTML client to allow the user to choose objects in the Windchill database that are required to complete an action.

To determine the drop down list, search criteria, and result columns for the object selection page the configured `com.ptc.windchill.cadx.common.picker.PickerSearchAttributeListDelegate` is used. The default configured `PickerSearchAttributeListDelegate` is `com.ptc.windchill.cadx.common.picker.PickerSearchAttributeList`. `PickerSearchAttributeList` delegates to the configured `wt.query.SearchAttributeListDelegate` to create the various type lists on the object selection page will be configured to support and determine the search criteria, and determine the result columns displayed in the object selection page. (For more `SearchAttributeListDelegate` details see [Customizing the HTML Search](#), later in this section.)

If this `PickerSearchAttributeListDelegate` implementation is not sufficient, then you can create and configure your own `PickerSearchAttributeList` to be used by the object selection page.

Modifying the Search Attribute List Delegate

To implement your own custom `PickerSearchAttributeListDelegate`, create a class that implements `wt.query.SearchAttributeListDelegate` and `com.ptc.windchill.cadx.common.picker.PickerSearchAttributeListDelegate` or create a class which sub-classes `com.ptc.windchill.cadx.common.picker.PickerSearchAttributeList`. See the javadoc for `PickerSearchAttributeListDelegate` and `PickerSearchAttributeList` and their methods for more details.

Note: PickerSearchAttributeList extends SearchAttributeList; therefore, the custom class can be used as the SearchAttributeListDelegate and PickerSearchAttribute ListDelegate.

Note: If extending PickerSearchAttributeList, you may have to set the filter to avoid NullPointerExceptions. This issue will be addressed in a future release.

To configure a new PickerSearchAttributeListDelegate, use the xconfmanager to add an entry to com/ptc/windchill/cadx/common/picker/picker.properties similar to:

```
wt.services/svc/default/com.ptc.windchill.cadx.common.picker.PickerSearchAttributeListDelegate/<unique delegate id which is also specified for com.ptc.windchill.cadx.common.picker.pickerSearchAttributeList>/java.lang.Object/0=mine.MyPickerSearchAttributeList/duplicate.
```

Using the xconfmanager, change the pickerSearchAttributeList entry in the wt.properties to
com.ptc.windchill.cadx.common.picker.pickerSearchAttributeList=<unique delegate id>. If there is no entry in wt.properties, then STANDARD is used as the delegate id.

Modifying Type Lists

The Pro/ENGINEER Wildfire HTML client object selection page uses configured type lists identified by type list ids, which are specified as the object selection page typeListID property value.

Type lists are defined in com\ptc\windchill\cadx\propfiles\picker.properties.

To add a type list entry for a new type list id, use the xconfmanager to add an entry similar to:

```
wt.services/rsc/default/<type list id>/java.lang.Object/0=<comma-separated list of valid query values>
```

If there is only one value in the list, then you do not need any commas. If you want an “All” entry in the type list, you must specify the type list entry value for ALL in the list of type ids.

Note: For the default implementation of the object selection page these valid type list values are **query** values specified in wt.query.queryResource.

You can remove type ids from the list of type ids specified for a type list id, but you cannot remove an entry or leave the type list empty.

Customizing the HTML Search

To customize the HTML search to either change the display of the default search objects or to add new classes, see the following file that is distributed as source `Windchill\src\wt\query\SearchAttributeList.java`. As explained in the javadoc for this class, subclass `SearchAttributeList` and make the appropriate entries in `service.properties` and `wt.properties`. Following are methods that should be implemented in a custom `SearchAttributeList`, with examples:

```
public final class MySearchAttributeList extends SearchAttributeList implements
Externalizable {
public void setLocale( Locale locale ) {
// Load in the values for the drop down list for selecting what to search against.
clientLocale = locale;
// **Customize -----
-----
// Add new classes to search to list below.
// Make sure that they are assigned numbers in sequence from 0 to N.
// Set dropDownListCount to N+1.
final int ALL = 0;
final int WTPART = 1;
...
final int MYCLASS = 22
int dropDownListCount = 23;
// -----
-----...
pickList = new String[classCount];
pickList[ALL] =
WTMessage.getLocalizedMessage(RESOURCE,queryResource.ALL,null,clientLocale);
pickList[WTPART] =
WTMessage.getLocalizedMessage(RESOURCE,queryResource.WTPART,null,clientLocale);
...
pickList[MYCLASS] = WTMessage.getLocalizedMessage(RESOURCE,queryResource.
MYCLASS,null,clientLocale);
pickValues = new String[classCount];
pickValues[ALL] = queryResource.ALL;
pickValues[WTPART] = queryResource.WTPART;
...
pickValues[MYCLASS] = queryResource.MYCLASS;
// **Customize You will need a string in here to correspond to each item in
pickList
// The string is a space separated list of what classes to query
// against. If you want to query against multiple classes that have a common
parent that
// has all of the attributes that you are interested in use that one class. If
you want
// to query against multiple classes that don't have a good common parent then
you can
// add them to a list and the search will loop through each class and combine
the results
// at the end. All classes in one list must only search against COMMON
attributes or
// attributes with the same name and of the same class! If you add both a
parent and
// a child class to the list you will get duplicate entries, when the results
are
```



```

// combined duplicate entries are not deleted.
queryClass = new String[classCount];
queryClass[ALL] =
"wt.part.WTPart wt.doc.WTDocument wt.change2.WTChangeIssue
wt.change2.WTChangeRequest2 " +
"wt.change2.WTChangeInvestigation wt.change2.WTAnalysisActivity
wt.change2.WTChangeProposal " +
"wt.change2.WTChangeOrder2 wt.change2.WTChangeActivity2
wt.csm.businessentity.BusinessEntity " +
"wt.effectivity.ConfigurationItem wt.epm.EPMDocument " +
"wt.replication.unit.WTUnit " +
"wt.part.WTPProductConfiguration " +
"wt.part.WTPProductInstance2 "; // Please remember to keep a space at the
end so that conditionally added items work.
...
queryClass[WTPART] = "wt.part.WTPart";
...
queryClass[MYCLASS] = "?.?.MyClass";
// **Customize These are the
// attributes that can be queried against.
inputAttributes = new String[classCount];
inputAttributes[ALL] =
"number name lifeCycleState projectId cabinet creator modifier
modifyTimestamp";
inputAttributes[WTPART] =
"number name view versionIdentifier partType source lifeCycleState projectId
cabinet creator modifier modifyTimestamp";
...
inputAttributes[MYCLASS] =
"name modifyTimestamp";
// **Customize Each individual
// string must match with the string listed above for the inputAttributes. "0"
stands for no
// input processing. If an attribute is an enumerated type use "0" and the
code will generate
// the drop down list. In the first string: projectId is in the fourth
position in inputAttributes
// so the method to generate the drop down list for it is also in the fourth
position in the
// string. The "0"s and methods must match in number with the number of
attributes listed
// under inputAttributes. You may add a fully qualified method from your
customization package
// as long as it is static and returns a vector of strings.
inputProcessing = new String[classCount];
inputProcessing[ALL] =
"0 0 0 wt.query.LocalSearchProcessor.getProjectList
wt.query.LocalSearchProcessor.getCabinetList 0 0 0";
inputProcessing[WTPART] =
"0 0 wt.query.LocalSearchProcessor.getViewList 0 0 0 0
wt.query.LocalSearchProcessor.getProjectList
wt.query.LocalSearchProcessor.getCabinetList 0 0 0";
...
inputProcessing[MYCLASS] =
"0 0";
// **Customize This is similar in concept to inputAttributes only these are
the attributes

```

```

// that will be displayed in the search results.
outputAttributes = new String[classCount];
outputAttributes[ALL] =
"number name versionDisplayIdentifier displayType lifeCycleState projectId
modifyTimestamp";
outputAttributes[WTPART] =
"number name versionDisplayIdentifier projectId lifeCycleState
modifyTimestamp";
...
outputProcessing[MYCLASS] =
"ObjProps 0";
// **New for 6.0
// **Customize This is similar in concept to outputAttributes only this list
is used
// to indicate which attributes can be sorted, can't be sorted, or an alternate
attribute
// that can be sorted to have the same affect as the display attribute. The
string that is used
// here should be the column descriptor so that it can be used to create the
ClassAttribute for
// the query. The query that is used for search is a simple query that will
not sort on all
// of the display attributes. Changing the 0 to 1 for an unsupported attribute
will
// either cause exceptions or sorts that don't work. Attributes of the
following types are
// just some examples of the attributes that will either throw exceptions or
sort incorrectly:
// EnumeratedType, CabinetReference, DataFormatReference,
LifeCycleTemplateReference, ProjectReference,
// and ViewReference.
sortAttributes = new String[classCount];
sortAttributes[ALL] =
"1 1 versionInfo.identifier.versionId 0 0 01";
sortAttributes[WTPART] =
"1 1 versionInfo.identifier.versionId 0 0 1";
...
sortAttributes[MYCLASS] =
"1 1";
// **New for 6.0
// **Customize This is similar in concept to outputAttributes only this list
is used
// for assigning a unique key to the sort preferences for this search. This
string will
// be persisted and used to retrieve the sort preferences for users. If the
value of one
// of these strings is changed or deleted after the system is in operation it
will create orphaned
// preferences in the system and users will lose the value that they had
persisted for that
// search. New entries can be added when a new search is added so that sort
preferences
// can be saved for that new search. These strings are arbitrary and never
displayed to the user.
sortPref = new String[classCount];
sortPref[ALL] =
"all";

```

```

sortPref[WTPART] =
"wtpart";
...
sortPref[MYCLASS] =
"myclass";
}
/**
 * <BR><BR><B>Supported API: </B>false
 *
 * @param locale
 * @return MySearchAttributeList
 */
public MySearchAttributeList( Locale locale ) {
setLocale(locale);
}
/**
 *
 * <BR><BR><B>Supported API: </B>false
 *
 * @return MySearchAttributeList
 */
public MySearchAttributeList() {
return;
}
}

```

wt.query.SearchAttributeList is always the most up-to-date and should be used as a reference.

The remainder of this section describes two new arrays in wt.query.SearchAttributeList: sortAttributes and sortPref.

Due to the data structures used on some classes, not all attributes that can be displayed in search results are sortable in the search results. The sortAttributes array in wt.query.SearchAttributeList is used to designate which attributes are sortable, and if an alternate attribute should be used for sorting. The version attribute is an example of an alternate attribute used for sorting: the attribute used to display is versionDisplayIdentifier, but the attribute used to sort on is versionInfo.identifier.versionId. Base java types, such as String and int, are sortable. Use the examples in wt.query.SearchAttributeList to determine if any custom types are sortable. Otherwise, a simple test will show if the attribute works, has no effect, or throws an exception.

The sortPref array (shown in the preceding code) is used to define a sort preference base name so users can define their sort preferences for that “Search On” object. A default for the sort preferences should be defined at the system level so that the first time the user uses the system, or if a user never defines preferences, the columns will be sorted logically. A default can be defined using wt.load.LoadFromFile or by using the Preference Administrator editor from the System Administrator portal page.

If this is a new database, the defaults are loaded as part of running the required section of wt.load.Demo (which runs wt.load.LoadFromFile). The site defaults can easily be added to or modified using the Preference Administrator. If the database was created on a system before Release 6.0, wt.load.LoadFromFile can be used to load the base defaults for the delivered configuration of the HTML search classes. See the “PrefEntry.../wt/query/htmlsearch” entries in Windchill\loadFiles\preferences.txt as examples.

Each user preference has an internal name, which is never seen from the client except in the Preference Administrator. Because the current search uses the wt.query.SearchAttributeList to allow users to add new searches, and because there has to be a set of sort preferences for each, a unique sort name is needed for each name in the "Search On" list. Each object in the “Search On” list is not necessarily one object, but can be a list of objects. The sortPref array in wt.query.SearchAttributeList defines a unique string that forms part of the name of the preference. The preferences for sorting are stored in the /wt/query/htmlsearch preference node, and the naming format is as follows:

<sort preference base name>sortAttrib<#>

<sort preference base name>sortDirect<#>

The <sort preference base name> is the unique string from the sortPref array in wt.query.SearchAttributeList; it has only to be unique within the sort names. The sortAttrib is for the attribute name, and the sortDirect is to indicate ascending or descending. It is false for ascending and true for descending. The <#> is the number of the sort key, 0 = first key, and so on. Following are the preferences that are loaded using wt.load.LoadFromFile and Windchill\loadFiles\preferences.txt for the All sort:

#All

PrefEntry~allsortAttrib0~number~/wt/query/htmlsearch

PrefEntry~allsortDirect0~false~/wt/query/htmlsearch

PrefEntry~allsortAttrib1~versionInfo.identifier.versionId~/wt/query/htmlsearch

PrefEntry~allsortDirect1~true~/wt/query/htmlsearch

In the all-default example, the results are sorted first by the number column and then by the version column, with the number being in ascending order and the version in descending order. Currently, the supported number of sort keys is 3, although theoretically the number of sort keys is limited only by Oracle performance. No testing beyond 3 keys has been done on the system.

Managing Secondary Content

The primary content of a CAD document is a CAD model file; however, Windchill allows you to attach other file types as secondary content. In addition, you can specify which file types should be considered outdated, and which should be automatically downloaded with a download of the primary content, as the CAD document moves through the stages (iterations, revisions, life cycle states, and so forth) of development.

Specifying Whether or Not to Outdate Secondary Content

The preference category Windchill Workgroup Manager > Server > CAD Data Management > Content Handling > Mark Out Of Date lists the secondary content categories for your site. For each category of file type, if the preference *<secondary_content_category>* (for example, Instance Accelerator File) is set to yes, it means that upon checking in a content change (not a metadata change) of the CAD document, any secondary content of the specified category type is marked as outdated (for example, in the **Attachments** table on the CAD document information page). A value of 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:

- Stopping the checkin 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 preference to automatically download secondary content

The preference category Windchill Workgroup Manager > Server > CAD Data Management > Content Handling > Download lists the secondary content categories for your site. For each category of file type, if the preference *<secondary_content_category>* (for example, Instance Accelerator File) is set to yes, it means that any secondary content of the specified category type is downloaded automatically when the primary content is downloaded. Setting the value to false specifies that the secondary content is not downloaded automatically with the primary content of CAD document.

Managing Drawing Dependents

The large number of dependents that may be associated to drawings can affect performance if an unnecessarily large number of objects are gathered into the workspace during collection activities.

To control this behavior, the preference, Windchill Workgroup Manager > Server > Gathering (Core HTML Component) > **Trace Drawing Optional Dependents**, allows you to specify whether or not optional dependents for drawings should be traced (and collected into the workspace). The possible values for the preference and the resultant behavior are as followed:

- PerConfiguration -- Whatever rule the user specifies for configuration (that is, Dependents: All, Required or None) is honored by the collection action.
- Required -- If the user specifies All for dependents, only the required dependents are traced and collected (optional dependents are avoided). If the user specifies None, no drawings dependents are collected.

The default setting is PerConfiguration

Note: This setting only applies to drawings that are included by collection; it does not apply to drawings that are "initially selected."

Controlling the Display of Internal Pro/ENGINEER Relationships

In certain reports accessed through the CAD document information page, Windchill displays a column labeled Dependency Type or Reference Type. This column displays the type of link between the table item and the CAD document reported on, based on the internal relationship of their respective Pro/ENGINEER models.

The Table Display mechanism allows users to filter out display of unwanted objects. In addition, your site may prefer to remove the display of certain internal Pro/ENGINEER relationships in Wildfire pages. For example, when viewing the References of an assembly drawing you can set a site-wide property in *<Windchill>/codebase/wt.properties* to display only the associated assembly, the Drawing Model (default behavior is to display all dependency or reference types).

To restrict the view of assembly drawing references to the assembly itself, set `com.ptc.windchill.cadx.caddoc.excludeDependencyTypes=<value>` to -1. Additional dependencies can be removed from display by adding other, comma-separated values as described in the following table:

Value	Dependency Type	Description
-2	Internal Pro/E Instance	Internal Pro/ENGINEER Instance
-1	Default	Dependencies created by Pro/ENGINEER that are not visible to the user through the reference viewer. (For example, displaying a dimension of a component of an assembly that is a model on a drawing.)

Value	Dependency Type	Description
0	Default	Any dependency that does not fall into one of the more specifically defined categories

Clean-up of the Event Manager

To avoid possible performance issues resulting from an accumulation of a large number of event records in the Event Manager, 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 Manager.

Administering Table Views

The display of information in many tables is user- and administrator-definable using the **Customize View List** window, available by selecting **Customize** from a table's **Current View** drop-down list. Specific views for tables can be created, or existing views can be edited or saved as new views. Administrative users have the options of making a table view available to all users by selecting the **Share with all users** check box, and of showing the view in all **Current View** lists, by selecting the **Show in current view list** check box, on the first step, **Set Name**, of the **New View** or **Edit View** windows, available from the **Customize View List** 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 Windchill preference 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, set the preference Windchill Workgroup Manager > Server > Workgroup Manager Table (Core HTML Component) > Position Scrollbar On Left Side to "Yes." The default is "No" (the scrollbar is positioned on the right side of tables).

Configuring the Number of Workspace Rows Displayed

The property `com.ptc.windchill.uwgm.cadx.ws.sizeToWindow` controls the number of rows displayed in the workspace object list. Set to true (the default setting), 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 false, 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 object list table or appears scrolled to the top of the page. Set to true (the default setting), the page automatically scrolls to show the workspace object 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"/>
```

System Configuration Recommendations

Running Multiple Servers

It is recommended that Windchill PDMLink and Windchill ProjectLink 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 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 EPMDocuments 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.

For more information on external vaulting and vaulting rules see the chapter, Administering External File Vaults, in the *Windchill System Administrator's Guide*.

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.

For more information, see the chapter, Installing and Administering Content Replication, in the *Windchill System Administrator's Guide*.

Performance Tuning

Setting the Method Server Max 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 Pro/ENGINEER data sets that are common to the products developed by Pro/ENGINEER users.

For more information on setting the max heap size, see the chapter, Method Server Maximum Heap Size, in the *Workgroup Manager Performance Best Practices Guide*

Data Compression

The meta data compression option is intended to improve the upload and download performance of the Pro/ENGINEER Wildfire client for users accessing Windchill across a lower bandwidth network. This feature substantially improves the performance of upload and download operations for large family tables.

Pro/ENGINEER Settings

In Pro/ENGINEER Wildfire, compression is controlled by a Pro/ENGINEER 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>

Windchill Settings

On the Windchill side, you enable the compression filters provided by the web-servers (for example, mod_gzip for Apache1.3.x and mod_deflate for Apache2.0.x).

Additional SOAP Compression Filter

Additionally, out of the box, the Windchill SOAPCompressionFilter is configured for compressing HTTP response data for special client (for example, Pro/ENGINEER Wildfire) interactions, such as downloading the contents of a model.

Following additional property settings that control data compression behavior are applicable only to the SOAPCompressionFilter.

To use these property settings, add them to the wt.properties file:

- `wt.compression.threshold=<size_in_bytes>` -- Sets a threshold for which HTTP responses are to be compressed. The default value (0) specifies that all responses are compressed.
- `wt.compression.off.contentEncodings=<encoding_types>` --Identifies HTTP response encoding types (case insensitive) for which compression is switched off. The default encoding types are (space delimited): identity gzip deflate lws-deflate. Setting the value to asterisk (*) switches off compression for all encoding types.
- `wt.compression.off.contentType=<content_types>` --Identifies HTTP response content types (case insensitive) for which compression is switched off. The default content types are (space delimited): image/jpeg image/gif application/zip.

Tip: While data compression can provide a benefit in a slow network, using compression puts an extra load on CPU resources; therefore, if network speed is not an issue, the use of compression may decrease performance and is not recommended.

Additional Considerations

If the Windchill compression filter is configured and `dm_http_compression_level` preference is set in the Pro/ENGINEER config file (`config.pro`), this setting will also apply to any interaction between the Pro/ENGINEER embedded browser and the server. That is, a non-zero value of the preference will ensure that not only the meta-data of Pro/ENGINEER models but even the content/UI pages will be sent in the compressed form reducing the overall network traffic.

Also note that the Pro/ENGINEER configuration option `dm_http_compression_level` needs to be set before registering the server through Pro/ENGINEER Wildfire. Any change in the value after the server is registered, will not apply to already registered server(s).

Maximizing the Oracle Server/Windchill Method 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. We found that setting property `SendBufferSize=16384` in `httpd.conf` significantly improved throughput over high latency, low bandwidth WANs.

Choosing to Display Family Object Symbols in Folders Table

Showing or hiding the family table symbols on CAD document type icons in the **Folders** table can be controlled by the following property:

`wt.clients.showFamilyGlyph`

Because there is a significant performance benefit in bypassing the queries that determine whether or not to show the family table symbols, the property defaults to false OOTB. Therefore, family table symbols on CAD document type icons do not appear by default in the **Folders** table.

To show the family table symbols in the **Folders** table, set `wt.clients.showFamilyGlyph` to true using the `xconfmanager`.

Other Recommendations

Controlling End User Objects

While workspaces are private areas "owned" by their creators, they may sometimes need to be accessed by an administrator. Perhaps the most typical need is to release the check-out lock on an object in a workspace whose owner is unavailable or has left the company.

An administrator with appropriate access privileges can selectively release the objects in one of two ways:

1. Using a standalone browser or an embedded browser, an administrator can locate the part either through Windchill search or by browsing Windchill folders. From the search results or the folder page, the administrator selects the object and performs **Undo Check Out** on the selected object and its dependents using the **Actions** pop-up.
2. Using a standalone browser or an embedded browser, an administrator can navigate to the information page of an object and perform **Undo Check Out** on the object and its dependents using the **Actions** drop-down menu.

Note: Only a single initially-selected object and its dependents can be processed with administrative **Undo Check Out**.

In addition, Windchill provides administrators with appropriate access privileges to locate other users' workspaces and delete them. This results in the undoing of any checkouts and removal from the workspace for any objects in the workspace.

To delete a workspace, perform the following procedure:

1. In the context for which you have administrative privileges, select the **Workspaces** minor tab.

The **My Workspaces** page appears.

2. In the lower table, titled **Other user's workspaces**, enter a user name in the **Enter User Name** field and click **Go**

or

If you are unsure of the user name, you can click **Find** to access the **Users** window where you can search for a user by any of the following criteria:

- **Full Name**
- **User Name**
- **Email**
- **Organization Name**

In the search results, select a user and click **Go**.

3. After you click **Go**, the **Other user's workspaces** table refreshes to display the workspaces owned by that user.
4. Select a workspace and click **Delete**.

The workspace is deleted from the system. Any objects in the workspace are removed from the workspace and any checkouts for those objects are undone.

Online Java Performance Guide

You may want to review the online Java Performance Guide to identify server-side Java settings that can boost performance.

Note: Be sure to carefully evaluate the options prior to implementation. PTC does not currently support them.

For more information on the online Java Performance Guide, see the chapter, Online Java Performance Guide, in the *Workgroup Manager Performance Best Practices Guide*

Windchill Folder Structure

It is important to carefully plan the Windchill cabinet/folder structure, and direct Windchill users to keep the number of objects (particularly, the CAD documents) in each Windchill folder to a manageable number (for example, up to a few hundred CAD documents). If the number is too large, it is difficult for other users to find an object in a folder and wait time is increased during browsing (as the information about each folder is extracted and communicated to the client).

HTTP Protocol

Pro/ENGINEER Wildfire only communicates with the server through HTTP requests. All HTTP requests (either to get an HTML page from the Windchill server, upload models, or perform a database operation through a SOAP request) are being made through the embedded browser. Therefore, all of the settings that are in effect for the embedded browser (including authentication, HTTP proxy server setting, etc.) apply to the Pro/ENGINEER Wildfire interaction with the server. If the Windchill server is using secure HTTP (HTTPS), then Pro/ENGINEER Wildfire also uses HTTPS.

Note: General usage of Pro/ENGINEER Wildfire (for example, managing CAD data through check-in or check-out) does not involve any applet, and therefore RMI is not used. However, if Pro/ENGINEER Wildfire is used as a Web browser to access pages containing applets, then RMI should be taken into consideration when configuring the firewall.

Windchill Preferences That Control Interaction with Pro/ENGINEER

The following sections present tables listing important preferences that control aspects of Windchill that are especially of interest to Pro/ENGINEER users and that may not have already been described in the preceding sections of this chapter. The tables are organized by preference category.

Note: Collection preferences for the various action categories are summarized in the section [Configuring the Initial Collection of Objects for Actions](#).

Create and Edit

Key preferences in the Create and Edit category are described in the following table:

Preference	Values	Description
Allow checkout of non-latest iterations	Yes No (default)	If set to "Yes," allows users to check out previous iterations of an object. If set to "No," users will see check out actions for the current iteration only.

Display

Key preferences in the Display category are described in the following table:

Preference	Values	Description
Incomplete object resolution	Ignore optional dependencies Ignore optional reference dependencies Ignore internal dependencies only Do not allow to ignore	Determines whether certain types of incomplete object dependencies can be ignored during a checkin or not
Toolbar Action Descriptions	Yes No (default)	Controls the display of the action description under the icon in the toolbar area of tables and trees
Workspace	Yes No (default)	Determines whether to enable the use of the workspace. If set to no, the workspace minor tab is not displayed.

EPM Service Preferences

This category has two sub-categories: Build Service Preferences and Soft-type Preferences. In addition, the preferences, Create Part Uses For Member Link and Send a CAD Document to PDM without optional dependents, are listed separately and are described in the following table:

Preference	Values	Description
Create Part Uses For Member Link	Yes No (default)	If set to "Yes," specifies to create a part Uses occurrence even if the CAD document structure does not have any occurrence. This occurrence can be used to publish attributes from the member link.

Preference	Values	Description
Send a CADDocument to PDM without optional dependents	Yes No (default)	Determines whether to allow a CAD document to be sent to PDM without new optional dependents. If allowed, links to optional dependents are removed.

Build Service Preferences

The following table lists preferences for the build service:

Preference	Values	Description
Allowed edit of part structure built by build service	Yes No (default)	In the case where a CAD document is owned by an ECAD application, allows editing of the part structure built by build service
Attributes Delimiter	, (default) <character value>	Identifies the delimiter used in listing attributes to be published
Attributes to be published on Link	<String(s), separated by delimiter character set in preference, Attributes Delimiter>	Identifies attributes to be published on the member link
Attributes to be published on Master	<String(s), separated by delimiter character set in preference, Attributes Delimiter>	Identifies attributes to be published on the master
Attributes to be published on Occurrence	<String(s), separated by delimiter character set in preference, Attributes Delimiter>	Identifies attributes to be published on an occurrence
Attributes to be published on Part	<String(s), separated by delimiter character set in preference, Attributes Delimiter>	Identifies attributes to be published on the part

Soft Type Preferences

The following table lists preferences for soft typing:

Preference	Values	Description
CAD Document default soft type	<string>	Identifies the logical identifier of the CAD document default soft type.

Revise

The following table lists preferences for creating new revisions:

Preference	Values	Description
Allow override on insert	Yes No (default)	If set to "Yes" will display insert actions and allow user to specify a revision label
Allow override on revise	Yes No (default)	If set to "Yes" will allow user to specify a revision label when creating a new revision.
Allow Override On Create CAD Document	Yes No (default)	If set to "Yes," will allow the user to set the revision of an uploaded object. If set to "No," when a new CAD document is checked in, it will use the initial revision label in the sequence.
Revision Label Picker Display Count	Integer (default is 10)	Specifies the number of revision labels displayed in the revision label picker

Save As

The following table describes the preferences for the Save As category, including the Naming Patterns sub-category)

Preference	Values	Description
Allow Replace	Yes No (default)	Controls the availability of the "Replace" button in the Save As UI. The "Replace" button allows a user to perform a global replace of one object in the structure with a different object in the commonspace. The default is "No" (replacement is not allowed), and it is strongly recommended to use the default setting. Replace is only possible for Pro/ENGINEER CAD documents, and this control is not the same as replacing the component in Pro/ENGINEER. It does not guarantee retrieval of the assembly with the replaced component.
CAD Document Filename Prefix	< naming pattern > *. * (default)	Specifies a pattern for filename prefix change during Save As action. Use characters and wildcard (*) to set the pattern for the new value. The default is *. *
CAD Document Filename Suffix	< naming pattern > *_.* (default)	Specifies a pattern for filename suffix change during Save As action. Use characters and wildcard (*) to set the pattern for the new value. The default is *_.*
CAD Document Name Prefix	< naming pattern > *. * (default)	Specifies a pattern for CAD document name prefix change during Save As action. Use characters and wildcard (*) to set the pattern for the new value. The default is *. *
CAD Document Name Suffix	< naming pattern > *_.* (default)	Specifies a pattern for CAD document name suffix change during Save As action. Use characters and wildcard (*) to set the pattern for the new value. The default is *_.*
CAD Document Number Prefix	< naming pattern > .* (default)	Specifies a pattern for CAD document number prefix change during Save As action. Use characters and wildcard (*) to set the pattern for the new value. The default is *.*.

Preference	Values	Description
CAD Document Number Suffix	<naming pattern> *_.* (default)	Specifies a pattern for CAD document number suffix change during Save As action. Use characters and wildcard (*) to set the pattern for the new value. The default is *_.*
Part Name Prefix	<naming pattern> *.* (default)	Specifies a pattern for part name prefix change during Save As action. Use characters and wildcard (*) to set the pattern for the new value. The default is *.*
Part Name Suffix	<naming pattern> *_.* (default)	Specifies a pattern for part name suffix change during Save As action. Use characters and wildcard (*) to set the pattern for the new value. The default is *_.*
Part Number Prefix	<naming pattern> *.* (default)	Specifies a pattern for part number prefix change during Save As action. Use characters and wildcard (*) to set the pattern for the new value. The default is *.*
Part Number Suffix	<naming pattern> *_.* (default)	Specifies a pattern for part number suffix change during Save As action. Use characters and wildcard (*) to set the pattern for the new value. The default is *_.*
Save Selected Objects Only	Yes No (default)	If set to "Yes," only selected objects are marked for Save As and dependent objects are marked for reuse. The default setting is "No." By default selected and dependent objects are all eligible for Save As.
Set File Name Same As Number	Yes (default) No	Controls how the filename is assigned by default for a new CAD document created in the workspace and commonspace Save As pages. Possible values are "Yes" (default) or "No". When set to "Yes", the new filename will be set to the same as value as the new CAD document number plus an appropriate extension. When set to "No," the new filename is set based on changing the original filename using the CAD Document Filename Prefix and CAD Document Filename Suffix" preferences.

Windchill Workgroup Manager

Windchill Workgroup Manager preference category includes many preferences that apply to both client and server settings for Pro/ENGINEER Wildfire, in addition to the Windchill Workgroup Manager. In many cases, specific preference settings can be specified on a CAD tool by CAD tool basis. The following sections describe the preferences for the sub-categories under the Windchill Workgroup Manager category.

Client Preferences

The following table describes client-side preferences for the Windchill Workgroup Manager that can be managed with the Windchill Preference Manager.

Preference	Values	Description
Attach Differences Report upon Check In	Yes No (default)	Controls the default behavior whether or not to generate and attach Differences report on objects that are going to be checked in. This option is applicable in Pro/ENGINEER embedded browser for CAD documents authored by Pro/ENGINEER.
Undo Checkout Overwrite Local Content	Yes No (default)	Specifies if the model content is overwritten in cache by default when performing Undo Checkout
Update Overwrite Local Content	Yes No (default)	Specifies if the model content is overwritten in cache by default when performing Update from the Windchill workspace. (Pro/ENGINEER File > Update is controlled by the config.pro option, dm_overwrite_contents_on_update).
[Upload sub-category] Naming Parameter	<string>	Specifies the CAD tool parameter to be used when generating a CAD document name upon initial upload
[Upload sub-category] Numbering Parameter	<string>	Specifies the CAD tool parameter to be used when generating a CAD document number upon initial upload

Preference	Values	Description
[Upload sub-category] Upload Drop Name File Extension	Yes No (default)	Specifies if the model file extension is truncated from the file name when using the CAD model file name to generate a CAD document name.
[Upload sub-category] Upload Drop Number File Extension	Yes No (default)	Specifies if the model file extension is truncated from the file name when using the CAD model file name to generate a CAD document number.

Server Preferences

The following sub-sections describe server-side preferences for the Windchill Workgroup Manager category that can be managed with the Windchill Preference Manager. The preferences are listed by sub-category.

Add to Workspace and Check Out Preferences

The following table describes the preferences for the Add to Workspace and Check Out actions:

Preference	Values	Description
Add Primary Contents to Workspace	Download (default) Link	Controls the default behavior whether content should be added to workspace as link or the file should be available. The default is DOWNLOAD (primary content is downloaded). When set to LINK, primary content is not downloaded, but a link to the content is created for later download as required.
Open in Application	Yes No (default)	Controls the default behavior whether initially selected objects should also be opened in the authoring application when Add to Workspace or Check Out is performed using the Check Out or Add to Workspace user interface. Check Out performed without the user interface will not open in authoring application regardless of this option. When set to "Yes," specifies that the primary content file is automatically opened in the CAD application upon Check Out.

Preference	Values	Description
Reuse Content in Target Workspace	Yes (default) No	Controls the default behavior of how content should be handled when there is already content in the target workspace cache. When set to "Yes," (default) specifies to reuse content existing in workspace. When set to "No," specifies to download new content from server.
Set Configuration for Add to Workspace	DEFAULT (default) LATEST AS_STORED	Controls the default configuration when Add to Workspace action is performed. The DEFAULT option applies latest configuration for the latest iteration, and as stored configuration for the non-latest iteration. When set to LATEST, the default configuration is set to latest and when set to AS_STORED, the default configuration is set to As Stored irrespective of the iteration of the object(s) selected for the action.
Set Configuration for Check Out	DEFAULT (default) LATEST AS_STORED	Controls the default configuration when Check Out action is performed. DEFAULT (default) option applies latest configuration for the latest iteration, and as stored configuration for the non-latest iteration. When set to LATEST, the default configuration is set to latest and when set to AS_STORED, the default configuration is set to As Stored irrespective of the iteration of the object(s) selected for the action.
Set for Check Out	SELECTED_AND_MODIFIED (default) SELECTED REQUIRED ALL	Controls the default set of objects to be checked out. The default is SELECTED_AND_MODIFIED (the initially selected object(s) and any modified dependents are marked as Set for Check Out. The value SELECTED marks only the initially selected object(s). The value REQUIRED marks initially selected object(s) and their required dependents. The value ALL marks initially selected object(s) and all their dependents.

Auto Associate Preferences

The following table describes the preferences for the **Auto Associate** action:

Preference	Values	Description
Auto Associate Naming Parameter	<string>	Identifies the CAD file parameter used when naming a new part during Auto Associate . The default is <no value> (no CAD parameter is used to name the part).
Auto Associate Numbering Parameter	<string>	Identifies the CAD file parameter used when numbering a new part during Auto Associate . The default is <no value> (no CAD parameter is used to number the part).
Auto Associate Truncate Name File Extension	Yes No (default)	Truncates the file extension (from CAD filename) in the part name when the part is created. When set to "Yes," truncates the file extension (from CAD filename) in the part name when the part is created. Default is "No" (file extension is kept in part name).
Auto Associate Truncate Number File Extension	Yes No (default)	Truncates the file extension (from CAD filename) in the part name when the part is created. When set to "Yes<" truncates the file extension (from CAD filename) in the part name when the part is created. Default is "No" (file extension is kept in part name).
Create Alternate Link On Check In	Yes No (default)	Allows a link (CAD document to part association) of the next available type to be created if the part already has an Owner link. When set to "Yes," the next available valid link is created if a matching part is found, and the checkin continues. The default is false (Check In fails with an overridable conflict).

Preference	Values	Description
Create Associate New Part	Yes (default) No	Specifies whether a new part should be created if a matching part is not found by Auto Associate . When set to "Yes," if a matching part is not found, a new part is created. When set to "No," a new part is not created and the checkin is not blocked.
Custom Class for Auto Associate Part	<string> com.ptc.windchill.uwgm.common.autoassociate.DefaultAutoAssociatePartFinderCreator. (default)	Specifies the name of the class that implements AutoAssociatePartFinderCreator or interface. The default is the standard Windchill implementation, com.ptc.windchill.uwgm.common.autoassociate.DefaultAutoAssociatePartFinderCreator.
Disallow Product Structure Links for CAD Document Sub-types	CADASSEMBLY CADCOMPONENT CADDRAWING FORMAT LAYOUT MANUFACTURING MARKUP OTHER REPORT WELDMENT	Lists CAD document sub-types which cannot be actively associated (form owner links). The default is <no value>. These are comma-separated string values. Note: The value OTHER is used for neutral file formats and Pro/ENGINEER text files used as libraries (for example, .mat)

Preference	Values	Description
Disallow Product Structure Links for CAD Document Types	CADASSEMBLY CADCOMPONENT CADDRAWING FORMAT LAYOUT MANUFACTURING MARKUP OTHER REPORT SKETCH UDF	Lists CAD document types which cannot be actively associated, (form owner links). The default is <no value>. These are comma-separated values. Note: The value OTHER is used for neutral file formats and Pro/ENGINEER text files used as libraries (for example, .mat)
Force Autonumbered Part Creation	Yes No (default)	Allows Auto Associate to create an autonumbered part upon check in if the part number specified via CAD document numbering parameter is not found in Windchill and autonumbering is the policy for new parts in Windchill. When set to "No," Check In fails with an overridable conflict to create an autonumbered part. When the value is "Yes," Check In creates an autonumbered part.

Preference	Values	Description
Part Master Class for Search	<string> wt.part.WTPartMaster (default)	<p>Identifies the internal name of the part master type searched for during Auto Associate. Search is allowed for a customer defined part or part sub-class.</p> <p>This preference can enable searching for customized parts by specifying the fully qualified class name of the master of the customized part, so that the search is restricted to the customized part class (and the whole WTPart class is not searched).</p> <p>The value wt.part.WTPartMaster indicates that the search for part is done on WTPart class. The default is the standard Windchill implementation, wt.part.WTPartMaster.</p>
Set Revision For Part	Yes No (default)	<p>Sets the revision of a part to match the revision of the CAD document during Auto Associate. When set to "Yes," Auto Associate sets the revision of a part to match the revision of the CAD document. This preference is for business practices that want to keep these revisions in sync.</p>
Store New Parts with CAD Documents	Yes No (default)	<p>Specifies the location of newly created parts to be the same as the location of their associated CAD document. When set to "Yes," newly created parts have the same location as their associated CAD document.</p>

CAD Data Management > Content Handling > Download Preferences

The following table describes the preferences in the CAD Data Management category > Content Handling > Download sub-category:

Preference	Values	Description
Analysis Input	Yes (default) No	Sets the download preference for the Analysis Input content category. When set to "Yes," specifies that the secondary content category Analysis Input is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Analysis Results	Yes No (default)	Sets the download preference for the Analysis Results content category. When set to "Yes," specifies that the secondary content category Analysis Results is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Drawing	Yes (default) No	Sets the download preference for the Drawing content category. When set to "Yes," specifies that the secondary content category Drawing is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Export	Yes No (default)	Sets the download preference for the Export content category. When set to "Yes," specifies that the secondary content category Export is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Family Table	Yes (default) No	Sets the download preference for the Family Table content category. When set to "Yes," specifies that the secondary content category Family Table is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.

Preference	Values	Description
General	Yes No (default)	Sets the download preference for the General content category. When set to "Yes," specifies that the secondary content category General is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
IDEAS Drawing Sheet	Yes (default) No	Sets the download preference for the IDEAS Drawing Sheet content category. When set to "Yes," specifies that the secondary content category IDEAS Drawing Sheet is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
IDEAS Legacy Drawing	Yes No (default)	Sets the download preference for the IDEAS Legacy Drawing content category. When set to "Yes," specifies that the secondary content category IDEAS Legacy Drawing is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
IDEAS Package		Sets the download preference for the IDEAS Package content category. When set to "Yes," specifies that the secondary content category IDEAS Package is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Image	Yes (default) No	Sets the download preference for the Image content category. When set to "Yes," specifies that the secondary content category Image is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Import	Yes (default) No	Sets the download preference for the Import content category. When set to "Yes," specifies that the secondary content category Import is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.

Preference	Values	Description
Information	Yes No (default)	Sets the download preference for the Information content category. When set to "Yes," specifies that the secondary content category Information is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Instance Accelerator File	Yes (default) No	Sets the download preference for the Instance Accelerator File content category. When set to "Yes," specifies that the secondary content category Instance Accelerator File is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Inventor Design View Document	Yes (default) No	Sets the download preference for the Inventor Design View Document content category. When set to "Yes," specifies that the secondary content category Inventor Design View Document is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Inventor iPart Instance	Yes (default) No	Sets the download preference for the Inventor iPart Instance content category. When set to "Yes," specifies that the secondary content category Inventor iPart Instance is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Inventor Model	Yes (default) No	Sets the download preference for the Inventor Model content category. When set to "Yes," specifies that the secondary content category Inventor Model is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Logical Reference	Yes (default) No	Sets the download preference for the Logical Reference content category. When set to "Yes," specifies that the secondary content category Logical Reference is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.

Preference	Values	Description
Manufacturing	Yes (default) No	Sets the download preference for the Manufacturing content category. When set to "Yes," specifies that the secondary content category Manufacturing is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Mesh	Yes No (default)	Sets the download preference for the Mesh content category. When set to "Yes," specifies that the secondary content category Mesh is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Package	Yes (default) No	Sets the download preference for the Package content category. When set to "Yes," specifies that the secondary content category Package is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Parameter Table	Yes (default) No	Sets the download preference for the Parameter Table content category. When set to "Yes," specifies that the secondary content category Parameter Table is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Pro/ENGINEER UGC	Yes (default) No	Sets the download preference for the Pro/ENGINEER UGC content category. When set to "Yes," specifies that the secondary content category Pro/ENGINEER UGC is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Pro/ENGINEER UGC Section	Yes (default) No	Sets the download preference for the Pro/ENGINEER UGC Section content category. When set to "Yes," specifies that the secondary content category Pro/ENGINEER UGC Section is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.

Preference	Values	Description
Pro/ENGINEER UGC Section Table of Contents	Yes (default) No	Sets the download preference for the Pro/ENGINEER UGC Section Table of Contents content category. When set to "Yes," specifies that the secondary content category Pro/ENGINEER UGC Section Table of Contents is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Text	Yes No (default)	Sets the download preference for the Text content category. When set to "Yes," specifies that the secondary content category Text is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Toolpath	Yes No (default)	Sets the download preference for the Toolpath content category. When set to "Yes," specifies that the secondary content category Toolpath is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.
Viewable	Yes No (default)	Sets the download preference for the Viewable content category. When set to "Yes," specifies that the secondary content category Viewable is to be downloaded when primary content is downloaded. When set to "No," no secondary content of this content category is downloaded.

CAD Data Management > Content Handling > Mark Out Of Date Preferences

The following table describes the preferences in the CAD Data Management category > Content Handling > Mark Out Of Date sub-category:

Preference	Values	Description
Analysis Input	Yes No (default)	Sets the mark out-of-date preference for the Analysis Input content category. When set to "Yes," specifies that the secondary content category Analysis Input is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.

Preference	Values	Description
Analysis Results	Yes No (default)	Sets the mark out-of-date preference for the Analysis Results content category. When set to "Yes," specifies that the secondary content category Analysis Results is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Drawing	Yes (default) No	Sets the mark out-of-date preference for the Drawing content category. When set to Yes," specifies that the secondary content category Drawing is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Export	Yes No (default)	Sets the mark out-of-date preference for the Export content category. When set to "Yes," specifies that the secondary content category Export is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Family Table	Yes (default) No	Sets the mark out-of-date preference for the Family Table content category. When set to "Yes," specifies that the secondary content category Family Table is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
General	Yes No (default)	Sets the mark out-of-date preference for the General content category. When set to "Yes," specifies that the secondary content category General is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
IDEAS Drawing Sheet	Yes (default) No	Sets the mark out-of-date preference for the IDEAS Drawing Sheet content category. When set to "Yes," specifies that the secondary content category IDEAS Drawing Sheet is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
IDEAS Legacy Drawing	Yes No (default)	Sets the mark out-of-date preference for the IDEAS Legacy Drawing content category. When set to "Yes," specifies that the secondary content category IDEAS Legacy Drawing is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.

Preference	Values	Description
IDEAS Package	Yes No (default)	Sets the mark out-of-date preference for the IDEAS Package content category. When set to "Yes," specifies that the secondary content category IDEAS Package is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Image	Yes No (default)	Sets the mark out-of-date preference for the Image content category. When set to "Yes," specifies that the secondary content category Image is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Import	Yes No (default)	Sets the mark out-of-date preference for the Import content category. When set to "Yes," specifies that the secondary content category Import is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Information	Yes No (default)	Sets the mark out-of-date preference for the Information content category. When set to "Yes," specifies that the secondary content category Information is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Instance Accelerator File	Yes No (default)	Sets the mark out-of-date preference for the Instance Accelerator File content category. When set to "Yes," specifies that the secondary content category Instance Accelerator File is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Inventor Design View Document	Yes No (default)	Sets the mark out-of-date preference for the Inventor Design View Document content category. When set to "Yes," specifies that the secondary content category Inventor Design View Document is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Inventor iPart Instance	Yes No (default)	Sets the mark out-of-date preference for the Inventor iPart Instance content category. When set to "Yes," specifies that the secondary content category Inventor iPart Instance is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.

Preference	Values	Description
Inventor Model	Yes (default) No	Sets the mark out-of-date preference for the Inventor Model content category. When set to "Yes," specifies that the secondary content category Inventor Model is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Logical Reference	Yes No (default)	Sets the mark out-of-date preference for the Logical Reference content category. When set to "Yes," specifies that the secondary content category Logical Reference is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Manufacturing	Yes No (default)	Sets the mark out-of-date preference for the Manufacturing content category. When set to "Yes," specifies that the secondary content category Manufacturing is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Mesh	Yes No (default)	Sets the mark out-of-date preference for the Mesh content category. When set to "Yes," specifies that the secondary content category Mesh is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Package	Yes (default) No	Sets the mark out-of-date preference for the Package content category. When set to "Yes," specifies that the secondary content category Package is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Parameter Table	Yes No (default)	Sets the mark out-of-date preference for the Parameter Table content category. When set to "Yes," specifies that the secondary content category Parameter Table is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Pro/ENGINEER UGC	Yes (default) No	Sets the mark out-of-date preference for the Pro/ENGINEER UGC content category. When set to "Yes," specifies that the secondary content category Pro/ENGINEER UGC is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.

Preference	Values	Description
Pro/ENGINEER UGC Section	Yes (default) No	Sets the mark out-of-date preference for the Pro/ENGINEER UGC Section content category. When set to "Yes," specifies that the secondary content category Pro/ENGINEER UGC Section is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Pro/ENGINEER UGC Section Table of Contents	Yes (default) No	Sets the mark out-of-date preference for the Pro/ENGINEER UGC Section Table of Contents content category. When set to "Yes," specifies that the secondary content category Pro/ENGINEER UGC Section Table of Contents is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Text	Yes No (default)	Sets the mark out-of-date preference for the Text content category. When set to "Yes," specifies that the secondary content category Text is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Toolpath	Yes No (default)	Sets the mark out-of-date preference for the Toolpath content category. When set to "Yes," specifies that the secondary content category Toolpath is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.
Viewable	Yes (default) No	Sets the mark out-of-date preference for the Viewable content category. When set to "Yes," specifies that the secondary content category Viewable is marked out of date when primary content is iterated. When set to "No," secondary content is carried forward.

Check In Preferences

The following table describes the preferences for the Check In action:

Preference	Values	Description
Auto Associate upon Check In	Yes No (default)	Controls the default behavior whether or not to perform Auto Associate for objects that do not have associated parts. If set to "Yes," associated parts are created.
Conflict for Out of date Secondary content upon Check In	Yes (default) No	Controls the default behavior whether or not to provide overridable conflict to the user when secondary content is considered out of date. If set to "Yes," an overrideable conflict is provided.
Create As Stored	Yes (default) No	If set to "Yes," specifies to create an As Stored configuration upon Check In.
Create Baseline upon Check In	Yes No (default)	Controls the default behavior whether or not to create baseline upon Check In. If set to "yes," a baseline is created.

Preference	Values	Description
ModelCHECK Configuration	Default:check/default_checks.mch,start/nostart.mcs,constant/inch.mcn Basic:check/default_checks.mch,start/nostart.mcs,constant/inch.mcn Release:check/default_checks.mch,start/nostart.mcs,constant/inch.mcn Approval:check/default_checks.mch,start/nostart.mcs,constant/inch.mcn Review:check/default_checks.mch,start/nostart.mcs,constant/inch.mcn	Specifies ModelCHECK Configuration files to be used for validation for each LifeCycle states in a specific syntax (eg. :,, : ,,). The configuration specified by "Default" LifeCycle State is default behavior. Value: Default:check/default_checks.mch,start/nostart.mcs,constant/inch.mcn Basic:check/default_checks.mch,start/nostart.mcs,constant/inch.mcn Release:check/default_checks.mch,start/nostart.mcs,constant/inch.mcn Approval:check/default_checks.mch,start/nostart.mcs,constant/inch.mcn Review:check/default_checks.mch,start/nostart.mcs,constant/inch.mcn
ModelCHECK Number of Errors	<integer> (default = 0)	Specifies the maximum number of ModelCHECK errors allowed
ModelCHECK Number of Hours	<integer> (default = 24)	Specifies the maximum allowable hours between a ModelCHECK verification at the client and the actual model checkin to Windchill. The default is 24.
ModelCHECK Validation	Yes No (default)	Specifies whether ModelCHECK validation is performed at Check In. If set to "Yes," validation is performed.
Remove objects from Workspace after Check In	Yes No (default)	Controls the default behavior whether or not to remove all objects from Workspace after Check in is completed. All objects include objects being checked in and not being checked in.
Resolve Incomplete Objects	Yes No (default)	Controls the default behavior whether or not to resolve incomplete objects automatically upon Check In.

Preference	Values	Description
Undo Check Out objects after Check In	Yes No (default)	Controls the default behavior whether or not to perform Undo Check Out after Check in is completed on objects that have been checked out but not modified
Update Incomplete Objects on Server	Yes (default) No	Controls the default behavior whether or not to update incomplete objects on server upon resolving incomplete objects upon Check In. This option has no effect unless "Resolve Incomplete Objects" option is turned on through preferences or Check In page.

Create New Workspace Preferences

The following table describes the preferences applicable to creating a new workspace:

Preference	Values	Description
Additional valid characters on US English locale	<character value> (default = - _)	Specifies additional valid characters in workspace name on US English locale. All alphanumeric characters are valid by default.
Part centric dependency processing mode	Yes No (default)	If set to "Yes," specifies part centric dependency processing mode.

Edit Workspace Preferences

The following table describes the preference applicable to editing workspace preferences:

Preference	Values	Description
Allow Effectivity for CAD Documentse	Yes (default) No	Specifies whether effectivity configuration specification is applicable to EPM documents in the workspace. If set to "Yes," allows the user to set effectivity configuration specification for EPM documents in the workspace. If set to "No," use of effectivity is not allowed for such objects.

Gathering (Core HTML Component)

The following table describes the preference applicable to the Gathering sub-category:

Preference	Values	Description
Trace Drawing Optional Dependents	Per Configuration (default) Required	<p>Specifies how to limit collecting drawing dependents for included CAD drawings. If set to "Per Configuration," the system applies the dependents option (All, Required, or None) that the user set to the included drawings.</p> <p>If set to "Required," the system collects only the required dependents of the included drawing even if the configuration is set to All. If the configuration is set to Required, the system collects the required dependents. If the configuration is set to None, the system collects none.</p> <p>This preference only applies to included drawings, not to drawings that are initially selected prior to initiating the action.</p>

General Preferences

The following table describes the preferences applicable to the General sub-category:

Preference	Values	Description
Custom Modeled Part Class	<string> wt.part.WTPart (default)	Specifies a fully qualified class name for a custom modeled part if there are any. The default is wt.part.WTPart. To specify a different custom modeled part, enter the class name.
Map File Extension to "Publication Source" Document Type	<string> (default = xml,sgml,html,txt)	Lists file extensions for dynamic documents that should be mapped to the document type "Publication Source." This is a comma-separated list. The default is xml,sgml,html,txt.

New CAD Document Preferences

The following table describes the preferences applicable to the New CAD document sub-category:

Preference	Values	Description
Is Model Name Unique	Yes (default)	Uniqueness constraint automatically set to true by Pro/ENGINEER. This value should not be changed.
Synchronize CAD Model Name with CAD Doc Number	Yes No (default)	If set to "Yes," synchronizes the CAD model name with the CAD document number in the New CAD Document user interface. This synchronization is not applicable when auto-numbering is in effect.

Update Preferences

The following table describes the preferences applicable to the Update action:

Preference	Values	Description
Add Primary Contents to Workspace	DOWNLOAD (default) LINK	Controls the default behavior whether content should be added to workspace as link or the file should be available. The default is DOWNLOAD (primary content is downloaded). When set to LINK, primary content is not downloaded, but a link to the content is created for later download as required.

Workgroup Manager Table (Core HTML Component) Preferences

The following table describes the preferences applicable to the table component::

Preference	Values	Description
Position Scrollbar On Left Side	Yes No (default)	Specifies if the scrollbar in Windchill Workgroup Manager tables should be positioned on left side of table. If set to "Yes," scrollbar is positioned on left side of these tables.

A

Quick Reference for Menus, Icons, and Symbols

















Use the next pages as a quick reference for the menu commands, action icons, and status symbols used in Pro/ENGINEER Wildfire with Windchill.




















File Menu Selections	
<div>File ▼</div> <div> Add Remove New Open <hr/> Check In Check Out Undo Check Out <hr/> Lock Unlock <hr/> Rename Save As Upload Update </div>	Add allows you to add objects to your workspace.
	Open allows you to remove selected objects from the workspace.
	New allows you to create a new CAD document, graphics dynamic document, or part; or to created a new revision of a selected object.
	Open allows you to open a selected CAD document's Pro/ENGINEER file or to open a ProductView representation of a selected object.
	Check In begins the check-in process for selected objects.
	Check Out begins the check-out process for selected objects
	Undo Check Out removes the check-out status on a selected object and discards any local modifications.
	Lock allows you to make a selected workspace object read-only
	Unlock removes the read-only status from a selected object
	Rename allows you to change the name of selected workspace objects (only if the object has never been checked in).
	Save As allows you to save a copy of a selected object as a new workspace object (not committed to commonspace until checked in).
	Upload places a selected local object in the server-side workspace
	Update compares workspace objects with the workspace configuration specification and replaces the workspace version with a version available on the server, if appropriate.

Edit Menu Selections	
<div>Edit ▼</div> <div> Attributes Auto Associate Parts Associate Disassociate <hr/> Find in table <hr/> Set State </div>	Attributes begins the process of editing attributes for checked-out, selected objects.
	Auto Associate Parts begins the process of automatically finding or creating parts to be associated with selected CAD documents.
	Associate allows you to manually associate objects.
	Disassociate allows you to undo the association between objects.
	Find in Table allows you to search for a character string in the workspace Object List.
	Set State allows you to set a life cycle state for a selected object.

Tools Menu Selections	
<div>Tools ▼</div> <div> Import to Workspace Export from Workspace <hr/> Synchronize </div>	Import to Workspace allows you to to bring objects into the workspace from a local directory.
	Export from Workspace allows you to export workspace objects to a target directory
	Synchronize refreshes workspace objects to reflect changes made on the server (for example, a name change)

Workspace Actions Menu Selections	
<div>- Pick an Action - ▼</div> <div> - Pick an Action - Activate Event Manager Edit Preferences Delete Workspace </div>	Activate allows you to make an inactive workspace active (embedded mode only).
	Event Manager opens the Event Manager window for the server with which you are working.
	Edit Preferences opens the Edit Workspace Options window to view or edit your workspace configuration specification.
	Delete Workspace allows you to delete the current inactive workspace.

Workspace Action Icons -- Toolbar and Row Actions		
		
 Remove from Workspace	 Update	 Add to Workspace
 Upload	 Auto Associate	 Find in Table
 Check In	 Create New Revision	 Open Information page (row action)
 Check Out	 Create Part	 Open in Pro/ENGINEER (row action)
 Undo Checkout	 Create CAD Document	 Open in ProductView (row action)

Share Status
 -- Shared to a project (for Windchill PDMLink only)
 -- Shared from PDM (for Windchill ProjectLink only)
 -- Checked-out from PDM (for Windchill ProjectLink only)
General Status
 -- Locked (Object is read only. Only applicable to objects in the local cache).
 -- Checked out by you
 -- Checked out by you in another workspace (seen in the workspace and action pages accessed in the context of a workspace)
 -- New locally
 -- Checked out by another user
 -- Checked-out to a project (for Windchill PDMLink only)
 -- Another Iteration is checked out by you
 -- Another Iteration is checked out by another user
Local Workspace Status
 -- Modified locally
Modified Status
 -- Modifications Need to be Uploaded
 -- Modifications uploaded
 -- Modified and not eligible for upload
Out of Date Status
 -- Out of date - Modified by you
 -- Out of date - Modified by another user
Out of Date with Workspace Configuration
 -- Out of date with Workspace configuration - Modified by you
 -- Out of date with Workspace configuration - Modified by another user

Glossary

active workspace

A workspace designated on a server for PDM actions. When working with the primary server, the active workspace is called the primary active workspace and becomes the default location for created, retrieved, or modified objects.

association

A concept that characterizes the relationship between a CAD document and a part. Associations may be of two types: "owner" (formerly referred to as "active") associations or links, and "content" (formerly referred to as "passive") associations or links. Owner associations allow a product structure to be generated from a CAD document structure.

attribute

A unique, system-wide property for which an object instance may have a value. Windchill attributes typically identify or describe a managed object. Attributes can be analogous to, and can be mapped to, Pro/ENGINEER parameters or properties.

BOM (bill of materials)

A list of all the parts and materials, along with their quantities, required to build a product or assembly.

CAD document

A revision-controlled, life cycle-managed object containing a CAD model, which is a file or a set of files containing information in a CAD application format.

CAD part

This term is used as necessary to denote a Pro/ENGINEER model (.prt) file, as opposed to a Windchill part object.

commonsplace

The collection of shared contexts in Windchill. Shared contexts include products, libraries, projects, organization and site contexts, that may be accessible by more than one user.

configuration specification

The rule used to generate the configuration of a product structure or CAD document structure. The rule can be based on criteria such as life cycle state, for example, Released.

context

A unique area on a server, such as a product, library, or project, established to make user interaction with Windchill more efficient and provide appropriate access. Contexts, coupled with any folders they may contain, constitute server locations.

dynamic document


A revision-controlled, life cycle-managed Windchill object that contains files typically associated with an XML authoring system.

Folder Navigator

The panel in the Pro/ENGINEER Wildfire user interface that allows you to browse registered servers and your local file system and the folders they contain.

incomplete object

A CAD file referenced as part of an assembly for which key information is unavailable (for example, a child in an assembly may have been suppressed and is not available to be saved to the workspace).

Such name-referenced files are referred to as incomplete CAD documents or incomplete objects and are represented in the workspace by the  icon.

Objects created in the workspace without a template are also incomplete. An incomplete object does not have an **Open in Pro/ENGINEER** command available in either the workspace listing page or in the information page for the incomplete object.

inactive workspace

An additional workspace on a registered server that is not designated as primary.

initially selected

A descriptor referring to objects selected prior to the initiation of a PDM action.

iteration

Used in conjunction with a revision series, iterations represent incremental changes to an object, such as part or CAD document. A new iteration is created each time the object is checked in, at which time the working copy that you have edited supersedes the original object. For example, the iteration might be iterated from A.1 to A.2. In this example, the number 2 indicates the new iteration for the revision A.

life cycle state

An indicator of maturity for a given revision of a PDM object. Examples of life cycle states include In Work, Under Review, and Released. Life cycle states are used in Windchill PDMLink.

part

In Windchill, a part is an information object with an identification number, representing a physical component or assembly in a manufactured product. A part will have one or more versions capturing how that part has been modified over time. Not to be confused with a CAD part (model file) created by Pro/ENGINEER.

PDM

Product data management.

primary active workspace

The workspace activated on your primary server to be the principal, private work area for performing PDM operations on multiple objects, such as CAD documents and parts.

primary server

A registered server that is designated as the default server for PDM operations. The primary server is used for searching, downloading, saving, checking in, and checking out files, in addition to other PDM operations.

product configuration

A static snapshot of a product structure at some point in time. The product configuration includes the part usage relationships, the associated documents, and CAD document versions.

registered Windchill server (registered server)

A Windchill server able to interact with Pro/ENGINEER Wildfire for PDM operations. The interaction is established using **Tools > Server Registry**.

revision

A business-sanctioned, significant increment of change in the definition or evolution of an object. Such changes are identified with a revision level according to company practices. Revisions are often accompanied by a change process. A revision letter identifies a specific revision.

secondary server

Additional registered server that can be used to browse, download, or back up files.

workspace

A private area in Windchill for managing your files while working on a task. A workspace allows you to track and change multiple objects and perform basic data management operations from within the Pro/ENGINEER user interface. A workspace consists of a local cache directory and a folder in a user's private area on the server. You can have several workspaces on a server.

version

A version identifies the revision and iteration of an object. For example, the A.2 version of an object indicates the second iteration of the revision A.

Index

A

- Actions column, 2-24
- add to workspace, 1-11, 3-41
- Add, to workspace, 3-42
- associate, 3-19
- attribute mapping, 5-23
 - download behavior, 5-25
 - explicit, 5-24
 - implicit, 5-23
 - upload behavior, 5-25
- attribute publishing in Pro/ENGINEER, 5-28
- attribute values, 4-4
- auto associate parts, 3-19
- auto check in, 3-28
- automatically associating parts, 3-19

B

- bandwidth, 5-55
- browser, 2-3
- bulk item, 2-36

C

- cache
 - change size, 2-19
 - clear, 2-19
 - information, 2-19
 - refresh, 3-56
- cache management, 2-19
- cache usage guidelines, 2-20
- CAD document, 1-7
 - create, 3-17
 - naming and numbering, 5-14
 - soft type, 5-22
- check in, 1-10
- Check In page, 3-29
- check out, 1-9
- checking in objects, 3-27
- checking out objects, 3-32
- checkout-on-the-fly, 3-33
- client-side workspace cache, 1-6
- collecting objects, 3-2

- commonspace, 1-5
- Compare status, 2-30
- comparing object content, 2-37
- compression, 5-54
- config.pro options, 5-3
- configuration
 - enabling as stored, 5-33
- configuration specification, 1-8
- configuring
 - Pro/ENGINEER build rule, 5-28
 - Pro/ENGINEER initial collection, 5-36
- Conflict Manager, 3-64
- connecting to server, 2-4
- content replication, 5-53
- control of end user objects, 5-56
- creating CAD documents, 3-17
- creating part structures, 3-18
- custom check in, 3-28
- Customizing Pro/ENGINEER, 1-4
- customizing Pro/ENGINEER, 5-1
- customizing the naming service, 5-18

D

- data compression, 5-54
- dependency processing, 3-2
- details page, 2-32
- download, 3-41
- download, in Pro/ENGINEER, 3-41

E

- earlier iteration
 - check out, 3-38
- editing object attributes, 4-2
- embedded browser, 2-3
- embedded workspace, 2-14
- Enabling As Stored Configurations, 5-33
- event
 - actions, 3-63
 - types, 3-62
- Event Information page, 3-63
- Event Manager, 3-61
- export workspace, 2-17

exporting objects from workspace, 3-51
external file vaulting, 5-53

F

family symbols, 5-55
family table
 modify attributes, 4-27
file vaulting, 5-53
file vaults, 5-53
Folder Navigator, 2-40

G

General status, 2-29

H

heap size, 5-53
help
 online, 2-41
Hot Links, 3-13
HTTP protocol, 5-57
HTTPS, 5-57

I

import
 objects to workspace, 3-46
information page, 2-32
 attributes, 2-33
 commonspace, 2-32
 workspace, 2-32
item
 bulk, 2-36
iteration, 1-7

J

Java Performance Guide, 5-57

L

latency, 5-55
lists, 2-33
local directory, making default file location, 2-9
Local Workspace status, 2-30
location
 set, 3-12
locked object, 2-31
locked workspace, 2-18

M

metadata compression, 5-54
Modified status, 2-30
modifying object attributes, 4-2
multiple servers, 5-53

O

object
 add to workspace, 3-41
 checkout, 3-32
 collect, 3-2
 compare content, 2-37
 download, 3-41
 edit attributes, 4-2
 import, 3-46
 import to workspace, 3-46
 information, 2-32
 lock, 2-31
 remove from workspace, 3-46
 revise, 3-59, 5-37
 save and upload, 3-14
 standard attributes, 2-33
 status, 2-28
 unlock, 2-31
 update, 3-54, 3-55
 upload, 3-14, 3-26
offline workspace, 2-14
online help, 2-41
Out of Date status, 2-30
Out of Date with Workspace Configuration status,
 2-30

P

parameter mapping, 5-23
 explicit, 5-24
 implicit, 5-23
part, 1-8
 naming and numbering, 5-14
PDM Server, 1-5
performance tuning, 5-53
policy-managed naming and numbering, 5-14
primary server, 2-2, 2-8
Pro/ENGINEER customizations, 1-4, 5-1
Pro/ENGINEER internal relationships, 5-50
Product Data Management, 1-4
product structure, 3-18
ProductView Express, 2-33
properties, 4-2
properties page, 2-32

R

- reference link, 5-23
- refreshing the cache, 3-56
- removing objects, 3-46
- revising workspace objects, 3-59
- revision, 1-7, 5-37
- revision level
 - during autoassociate, 5-38
 - setting, 5-38
- RMI, 5-57

S

- save as, 4-8
 - family table, 4-28
 - in commonspace, 4-16
 - in workspace, 4-12
- saving objects, 3-14
- secondary content, 5-49
- secondary server, 2-2
- Server
 - primary and secondary, 2-2
- server
 - connecting to, 2-4
 - lost connection, 2-14
 - remove connection, 2-9
 - set primary, 2-8, 2-40
- Server Registry, 2-4
 - File menu, 2-6
 - Server menu, 2-6
 - Servers tab, 2-5
 - using, 2-4
 - Workspace menu, 2-6
- Server Registry dialog box, 2-5
- server-side workspace, 1-5
- Set Location dialog box, 3-12
- Set Revision dialog box, 3-60
- setting a location, 3-12
- setting attribute values, 4-4
- Share status, 2-29
- shared folders, 1-5
- soft type, 5-22
- soft typing, 5-22
- soft typing CAD documents, 5-36
- sorting workspace objects, 2-26
- standalone workspace, 2-14
- status
 - Compare, 2-30
 - General, 2-29
 - Local Workspace, 2-30
 - Modified, 2-30

- object, 2-28
- Out of Date, 2-30
- Out of Date with Workspace Configuration, 2-30
- Share, 2-29
- Status Messages, 2-31
- Status Messages, 2-31
- synchronize workspace, 2-16, 3-54
- system configuration
 - Pro/ENGINEER, 5-53

T

- table view
 - create, 2-28
 - edit, 2-28
- table views, 2-26
- Technical support, xiv
- type conflicts between Pro/ENGINEER parameters and Windchill attributes, 5-26

U

- undoing checkout, 3-40
- unlocked object, 2-31
- unlocked workspace, 2-18
- update, 1-12
- updating objects, 3-54, 3-55
- upload, 1-11
- uploading objects, 3-14, 3-26
- usage links, 5-29
- uses link, 5-23

V

- vaults, 5-53
- version, 1-7
- versions
 - properties for new, 5-38
- vetoing operations
 - Pro/ENGINEER, 5-29
- viewable, 2-33

W

- Workgroup Manager CAD Document Master, 5-23
- workspace, 1-6
 - access, 2-25
 - activate, 2-13
 - cache, 2-19
 - change, 2-40
 - create, 2-12
 - delete, 2-13
 - display link to, 2-11

- export, 2-17
- find objects in, 2-26
- go online, 2-17
- import, 2-17
- in embedded browser, 2-10
- in standalone browser, 2-11
- lock, 2-18
- menus, 2-21
- object list, 2-24
- offline, 2-16
- offline access, 2-14
- offline use, 2-11
- remove objects from, 3-46
- synchronize, 3-54
- toolbar, 2-22
- unlock, 2-18
- updating objects, 3-54
- use of, 2-10
- workspace cache, 1-6