# Getting Started Using Windchill ProductPoint with Pro/ENGINEER and Other Applications

**Windchill ProductPoint 1.1 M020**

**March 2010**



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# Contents

# About This Guide

The *Getting Started Using Windchill ProductPoint with Pro/ENGINEER and other applications* guide is intended for new users of Windchill ProductPoint who want to manage Pro/ENGINEER and other applications' files on a Windchill ProductPoint server. This guide assumes you are familiar with your application and focuses on how you perform day-to-day file management operations while connected to the server.

You can access the latest version of this document at:

http://www.ptc.com/view?im_dbkey=94776

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# I

# Working with Pro/ENGINEER

# 1 Client Setup



This chapter describes how to install and configure the client application for Windchill ProductPoint to support Pro/ENGINEER.

# Pro/ENGINEER Version Support

Windchill ProductPoint 1.0 supports the following Pro/ENGINEER versions:

- Pro/ENGINEER Wildfire 4.0 M100 and later versions
- Pro/ENGINEER Wildfire 5.0 F000 and later versions

The following features are new in Windchill ProductPoint 1.1 and are supported only for Pro/ENGINEER Wildfire 5.0:

- Integrated installation and startup of Windchill ProductPoint client components—Pro/ENGINEER Wildfire 5.0 supports the capability to install and start the Windchill ProductPoint Client Manager as part of the Pro/ENGINEER installation. This is described in more detail in later sections.
- Pop-up notifications—Dynamic pop-up notifications are displayed for items in a Pro/ENGINEER session for server change events that include Out of date, Locked, Unlocked, and Released.
- View server properties page—You can access the server properties page for the active item using the **File ▸ View server properties** or for a Model Tree item by right-clicking an item. The properties page for the item is displayed in the embedded browser.
- View and edit the Notes Wiki—You can view and edit the Notes Wiki for the active item using **File ▸ Edit Notes Wiki**. The Notes Wiki can be used to capture modeling notes, engineering design notes, or information important for manufacturing of the part or assembly. A separate Wiki is set up for each Pro/ENGINEER file saved to the server.
- View presence for user associated with a model item—You can contact the user who last modified, checked out, or managed items on the server.
- Lock on Open—You can open an item from the server and lock it at the same time using **File ▸ Open**.
- View thumbnail from **File ▸ Open** and Model tree—You can view the thumbnail associated with a model item and the associated PDM status (last modified, last modified by, state, lock status) by hovering over the file type icon in the **File ▸ Open** menu and in the Model tree.
- Access 3D image from thumbnail – You can access and manipulate the 3D ProductView viewable for a model item by clicking in the hover thumbnail in the Pro/ENGINEER **File ▸ Open** dialog box or Model tree.
- Access the Server Quick View – You can view or hide a server summary of important PDM status for items in session (changed but not saved, out of date, locked by you, locked by another and recently changed).

## Pro/ENGINEER Installation

When you install Pro/ENGINEER Wildfire Pro/ENGINEER Wildfire 4.0 M060 (or later 4.0 maintenance version), make sure the Windows folder path contains

no spaces. This allows to open a file that is located on the server using the Open In Pro/ENGINEER action in the browser. If you had previously installed Pro/ENGINEER in the default path `C:\Program Files\`, it is best to install Pro/ENGINEER Wildfire 4.0 M060 (or later 4.0 maintenance version) in a new Windows folder location with no spaces in the folder path.

# Pro/ENGINEER Client Platform Requirements

A workstation with at least 2 GB of RAM is required for working with typical models and assemblies with 200 or more parts. If you use Windows XP, XP Professional SP2 64 bit is recommended. If you use Windows Vista, Vista SP1 64 bit is recommended. If you use Windows XP Professional 32 Bit, you may need to set the XP 3 GB switch to allow XP can access up to 3GB of virtual memory. You should set the 3 GB switch only if you find that Pro/ENGINEER is running out of memory when attempting to save items because this option will degrade overall system performance. See the following link for information about how to set the 3 GB switch:

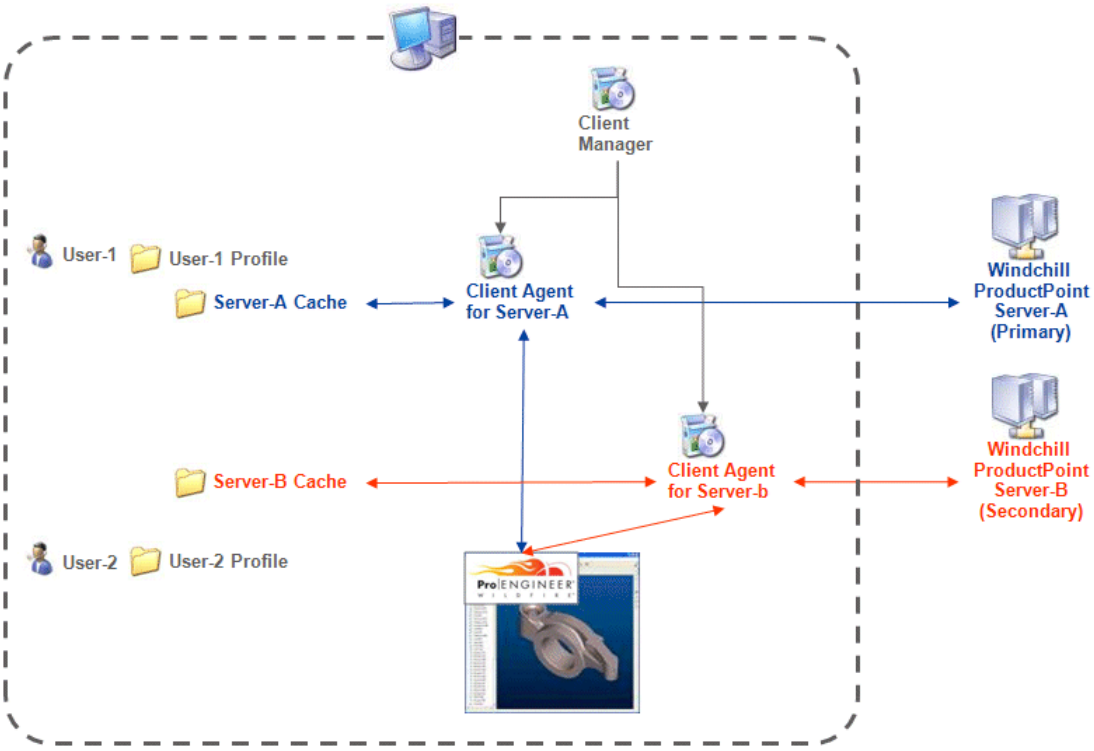
http://www.microsoft.com/whdc/system/platform/server/PAE/PAEmem.mspx

You must also install Microsoft .NET 3.5 SP1 on your Pro/ENGINEER client machine. This is required for communication with the Windchill ProductPoint server. You can download .NET 3.5 SP1 from the following location:

http://www.microsoft.com/downloads/details.aspx?FamilyID=ab99342f-5d1a-413d-8319-81da479ab0d7

# System Architecture

This diagram illustrates the organization and interactions of the client and server components with Pro/ENGINEER.




These components behave and interact as follows:

- The Client Manager is responsible for starting and updating the client agents.
- The Client Manager must be manually started the first time it is installed (unless you are installing Pro/ENGINEER Wildfire 5.0 with the integrated Windchill ProductPoint components installation option). It starts automatically in the future each time Windows is started (this is accomplished by adding it to the Windows startup).
- If the Client Manager is not running, communication between Pro/ENGINEER and the server is not enabled, and you cannot browse or open items from the server or save items to the server.
- A separate client agent is associated with each server that is registered and connected to Pro/ENGINEER for a particular user defined on the client machine.
- The Client Manager handles all client agents for users by starting, stopping, and automatically updating the client agents as servers are connected and disconnected.
- The Client Manager and client agents continue to run and manage the upload of unsynchronized saved model files even after Pro/ENGINEER is shut down.

- If multiple servers are registered with Pro/ENGINEER, one of these servers is identified as the primary server and all Pro/ENGINEER files are saved to this server. All other registered servers are referred to as secondary servers and may be used only as a source for opening model files. It is not possible to save files to a secondary server.
- A separate client cache is maintained for each registered server for each user. If one computer is used by several users who log in under different user names, each user has a dedicated client cache set up.

# Registering a Server in Pro/ENGINEER

You must register each Windchill ProductPoint server you want to connect with in Pro/ENGINEER. You must install the Client Manager to support the connection with a Windchill ProductPoint server. You only need to install it once. The Client Manager supports communication with any number of Windchill ProductPoint Client Manager.

### *Note*

*You need administrator privileges to the client machine to install the Client Manager. If you do not have administrative privileges, ask your administrator to install the Client Manager for you.*

Follow these steps:

1. To access the Server Manager and register a Windchill ProductPoint server in Pro/ENGINEER, select **Tools** ▸ **Server Manager** from the Pro/ENGINEER menu.
2. From the **Server Manager** window, select **Server** ▸ **Register New Server**.
3. Enter the name and the URL of the server. You can use the address supplied by your administrator, or copy the address from any server page.

   ### *Tip*

   *Always enter the fully qualified address for the server. For example, enter*

   `http://servername.domainname.com:1975`

   *instead of*

   `http://servername:1975`
4. Click **Check** to test that Pro/ENGINEER can find the server.

   The following steps apply only if you are using Pro/ENGINEER Wildfire 4.0 or if you installed Pro/ENGINEER Wildfire 5.0 but did not choose to simultaneously install the Windchill ProductPoint client components .

   Skip to step 17 if you have installed Pro/ENGINEER Wildfire 5.0 and selected the options to install the Windchill ProductPoint components.

5. If you did not previously register a Windchill ProductPoint server, you receive an error message. Select the action to install the Client Manager.
6. Select the language you used when you installed Pro/ENGINEER.
7. Select the link to download and install the Windchill ProductPoint Client Manager.
8. Click **Run** to download the Client Manager.
9. Click **Run** to start the installation of the Client Manager.
10. Click **Next** to begin Client Manager installation wizard.
11. Accept the defaults or browse to identify the installation location.
12. Click **Disk Cost** to ensure that the selected location drive has sufficient space.
13. Click **Next** to continue.
14. After you get a confirmation that the manager was successfully installed, click **Close**.
15. After the installation is complete, select **Start ▸ Startup ▸ Windchill ProductPoint Client Manager** to start the Client Manager for the first time.

    The next time you start Windows, the Client Manager starts automatically if you do not remove it from the startup menu. When the Windchill ProductPoint Client Manager is running, the  icon is added to the Windows system tray. If this icon is not present, the Client Manager is not running and Pro/ENGINEERyou will not be able to connect with the server.
16. Return to the Server Manager window in Pro/ENGINEER and click **Check** to verify that Pro/ENGINEER can communicate with the server.
17. Click **OK** to complete the server registration.

    The Client Manager downloads and installs a client agent that supports the connection with each specific server. This step occurs only once for each Windchill ProductPoint server you register.

    If you are connecting to an upgraded server, you may be asked to update your Client Manager. In this case, follow the instructions and select the **Update Now** action on the Client Manager system tray icon.
18. If you are connecting with the server through a firewall, the system requests a permission for the agent to connect through the firewall. Click **Unblock** so the Windchill Product Point Client Manager can interact with the Windchill ProductPoint server. You may need to select the unblock action multiple times.
19. The Pro/ENGINEER Server Manager window appears and shows the status for the registered server as On Line. The sever name and the URL location of the selected server are shown at the bottom of the **Server Manager** window.

    Click **Close** to exit the server registration.
20. The Folder navigator in Pro/ENGINEER shows the Windchill ProductPoint server you just registered. When you register a server, it is automatically made the primary server in Pro/ENGINEER and is ready to use. If you

register multiple servers, the last server you register is designated the primary server. You can make any registered server a primary server using the Server Manager. Only the primary server is displayed in the **Common Folders** list in Pro/ENGINEER. You can browse and navigate the server content from this start point.

21. The status of the connection to the server is also displayed in the lower-right corner of the active Pro/ENGINEER window. This area is also used to display the status of synchronization between Pro/ENGINEER and the server. When Pro/ENGINEER has a live connection with the server, the status at the bottom of the window is displayed as **Connected** along with the green junction symbol.


# Configuring Pro/ENGINEER to Generate Thumbnail Images

You can configure Pro/ENGINEER to generate thumbnail images of parts as well as 3D viewables when model files are saved. These images and viewables are saved to the server, enabling the users who do not have access to Pro/ENGINEER to view, measure and mark up images.

In Pro/ENGINEER Wildfire M090 and later versions, you can enable a ProductView-based thumbnail generation utility that is bundled with Pro/ENGINEER.

Set the following config.pro options in Pro/ENGINEER to generate thumbnails and viewables:

- `generate_viewable_on_save`—If set to yes, instructs Pro/ENGINEER to generate 3D viewables in a ProductView format that can be viewed in ProductView.
- `intf_out_pvs_recipe`—Browses your Pro/ENGINEER installation directory to locate `apps\prodview\recipe\productpoint.rcp` to specify the recipe file that contains instructions for the generation of viewables and thumbnails.

The recipe file is either located as listed above (if it is an end-user installation), or at `$PTCSRC\apps\prodview\recipe\productpoint.rcp` for the system area.

You can instead use the bitmap generation configuration options `save_bitmap` (set to `alllevels`) and `save_bitmap_type` (set to `JPEG`).

The ProductView thumbnail and viewable generation approach offers the following benefits over the save bitmap mechanism:

- Thumbnails are generated “silently” and do not cause any repainting of the Pro/ENGINEER graphics area, as they are generated when a model is saved.
- Images never include the spin center and construction geometry such as datum planes, points, and coordinates systems, even if these options are enabled. Images, therefore, are never cluttered with this information.
- Images do not include the spin center and construction geometry such as datum planes, points, and coordinates systems, without creating a clutter.
- Thumbnails are properly generated for drawings, layouts, and formats as well as for parts and assemblies.
- The thumbnail background color can be controlled and made uniform, so that it is independent of a particular user’s Pro/ENGINEER background color preferences.

  ***Note***

  *The ProductView thumbnail generation utilities generate only a uniform background color. You cannot generate a gradient background color as with the `save_bitmap` set to `alllevels` option.*

# Managing Your Client Cache

Windchill ProductPoint provides tools to manage the file system cache, allowing you to set the cache location and size.

## Setting Your Cache Location

A unique cache root folder is associated with each user on a client machine. Under this root folder, a cache subfolder is defined for each registered Windchill ProductPoint server.

The default location of the folder in Windows XP is:

```
D:\User Profiles\<user
name>\ApplicationData\PTC\ProductPointService\<fully
qualified server name>\cache
```

For example:

```
D:\User Profiles\myusername.PTCNET\ApplicationData\PTC\
```

`ProductPointService\malfoy.ptcnet.ptc.com\cache`

### *Tip*

*When determining the location of cache, consider the size of the drive partitions and your write privilege.*

You can configure the location of the client cache root folder by setting the `PTC_WF_ROOT` environment variable:

1. On Windows XP, right-click **My Computer** or select **Start ▸ My Computer ▸ View system information**. Click the **Advanced** tab and then click **Environment Variables**.
   a. On the **System Properties** dialog box, click the **Advanced** tab.
   b. Click **Environment Variables**.
2. Specify the client cache root folder location.
   a. Click **New** to define a new environment variable.
   b. Enter the variable name `PTC_WF_ROOT`.
   c. Enter the file path for the root cache location.

      A folder is automatically defined to hold the cached files for each registered server under the specified location.

## Setting Your Cache Size

You can configure the amount of disk space allotted to the cache root folder by setting the `PTC_CACHE_LIMIT` environment variable.

The value is set in units of 1MB. If you set it to 1024, you have allocated 1 GB to the root cache, and the 1 GB limit is applied across the cache subfolders for all of the registered servers. If you have 3 servers, the total size of all registered cache folders is under 1 GB.

Files in the cache are removed based on their age. The oldest files are removed first, to free space for saving new items. Every read of the file from cache or every save into the cache changes the file date, making it current. If you set a cache limit to 0 (default), then no limits are imposed. The cache limit does not affect the Pro/ENGINEER save behavior. All saves that you make in Pro/ENGINEER are first saved locally, uploaded to the server during synchronization, and then removed or kept, depending on the cache limit. You must have enough available disk space to save your model file changes on your client machine.

### *Note*

*You must log off Windows and log back on to ensure that the values for new or modified environment variables take effect.*

# Configuring Client Communication Ports

By default, the Windchill ProductPoint Client Manager is running on the HTTP port 8989. However, there are some cases when the Pport must be changed. For example, one such case is when multiple users are sharing the same workstation to run the Windchill ProductPoint client (Pro/ENGIINEER, MathCAD, and AutoCAD etc.) and hence having their own instance of Client Manger running at the same time. Sometimes, some other software (for example, Java Web Server) is running on that HTTP port, and therefore you need to change the port.

The assignment of the new ProductPoint Client Manager is done by assigning the environment variable: **WPP_CLIENT_MGR_PORT**. You must do it on the user level:

**My Computer ▸ Properties ▸ Advanced ▸ Environment Variables ▸ User Variables**

For example, you can set it to : **WPP_CLIENT_MGR_PORT=9090**

After setting this environment variable, the Instance of the Product Point Client Manager will be running on specified HTTP port (9090 in this case).

If a non-administrative user runs the new ProductPoint Client Manager on a non-standard port, then a user with administrative privileges must run a special command on the computer. For this, Administrator must browse to the location on the hard drive where the ProductPoint Client Manager is installed and locate a subfolder `httpcfg`. Inside this subfolder, find `openhttp.bat` and edit it to change the default Port 8989 in the original line to the one specified by the `WPP_CLIENT_MGR_PORT` environment variable.

If the original line is:

```
httpcfg.exe set urlacl /u
http://+:8989/WindchillProductPointClientServices/ /a
"D:(A;;GX;;;WD)"
```

in our example, you need to change it to:

```
httpcfg.exe set urlacl /u
http://+:9090/WindchillProductPointClientServices/ /a
"D:(A;;GX;;;WD)"
```

After executing the `openhttp.bat` file, a non-administrative user can run the ProductPoint Client Manager on the newly specified port.

# 2 Working with Pro/ENGINEER Models



This chapter describes how you can interact with the Windchill ProductPoint server while working in Pro/ENGINEER.

# Saving Models Opened from the File System to the Server

You can open Pro/ENGINEER models from your file system and save them to the server so they are managed by the server. When you are connected to a Windchill ProductPoint server, all saves are automatically made to the connected server (even if you opened a model from a file system).

### *Note*

*If you plan to save an assembly and all its parts and drawings to the same location on the server, load the top-level items (typically the assembly drawings) first. When you save an assembly drawing, the assembly model and all parts referenced by the assembly are also saved in the same location.*

If you want to save standard parts in a different location, load the drawings of the standard parts first to a standard parts folder. Then open the assembly from the file system that uses the standard parts and save it to the desired location for the assembly components. Pro/ENGINEER will generate a conflict message indicating the standard parts already exist. Select the option to continue, and a new iteration of the standard part is created from the version in session, but the location for the standard part does not change.

1. Open a part or an assembly from the file system in Pro/ENGINEER.
2. Select **File** ▸ **Save** to save the active model in session to the server.

   The files are always saved to the server even if you originally opened the model files from the file system. If the model being saved does not yet exist on the server, you must specify a location on the server under Product or Server.

   Pro/ENGINEER displays the last accessed location by default. You can set the config.pro option `file_open_default_folder` to force Pro/ENGINEER to display a specific server location for open and save operations.

   You can navigate the product and catalog site structure on the server from Pro/ENGINEER just as you would navigate a file system. Server folders are distinguished from file system locations by the network link symbol underneath the folder  .
3. Specify the server location for the new models by browsing or searching, and then click **OK**. Not all locations are valid for saving Pro/ENGINEER files. You can save Pro/ENGINEER files to only structure library locations. By default, the structure library defined in Products and Catalogs is named Documents.

In a server hierarchy, you must identify a root folder as a save location. You cannot save to intermediate locations.

### *Tip*

*If you just created a new folder or product or do not find a location by searching or browsing, click the Refresh icon to update the client cache. The default client cache refresh interval is five minutes.*

### *Tip*

*You can also search for a server location. Searching is often the most efficient way to find locations, parts and assemblies. The server search is always global. The search scope is not affected by the current location to which you may have navigated and initiated the search.*

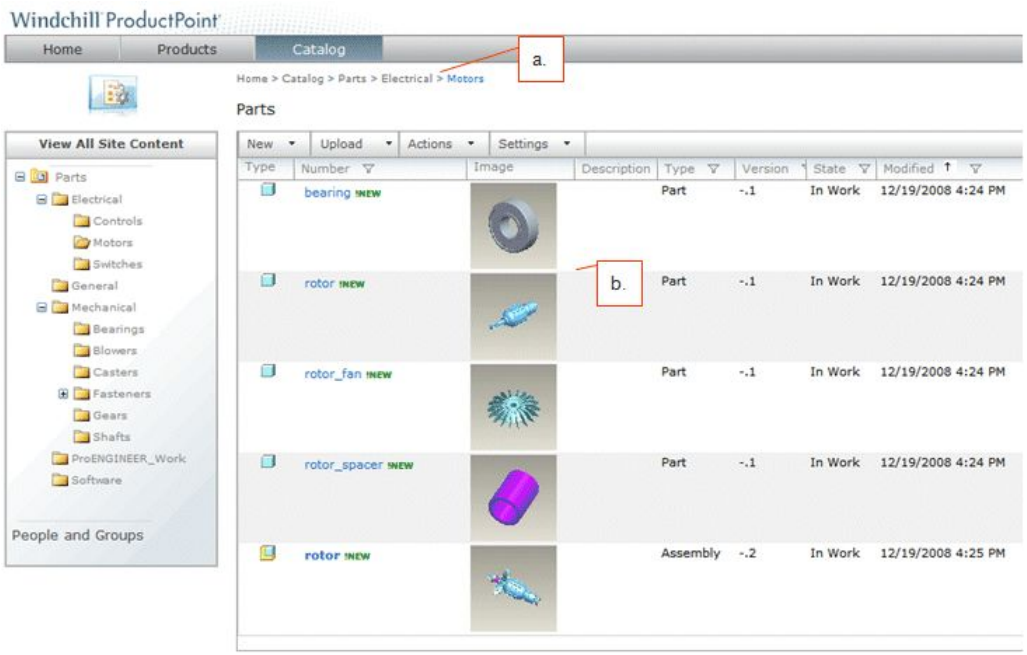
4. When you click **OK**, Pro/ENGINEER saves the items to the client cache, and, if you have set the config.pro options to do so, generates the thumbnail images and ProductView viewables.

   When you save with the config.pro option to generate thumbnail images, the graphics display cycles through all parts in the active assembly, and displays a message that the model has been saved.

   At this point, all the model files are saved to the client cache and you can continue working in Pro/ENGINEER. The saved files are asynchronously uploaded to the server in the background even if Pro/ENGINEER not running.

   Once the saved changes have been completely uploaded to the server, the synchronization status icon stops spinning and the status text changes to

Synchronized. The uploaded items are now accessible to other users on the server.




a. The Pro/ENGINEER model files are now saved to the server and are viewable through a browser.

b. The thumbnail images of the saved parts are displayed if the Pro/ENGINEER config.pro options are set to generate thumbnails.

# Creating New Pro/ENGINEER Models and Saving to the Server

This section explains how to save newly created Pro/ENGINEER models to the server.

1. Select **File** ▸ **New** in Pro/ENGINEER to create a new model.
2. Select the type of model to create.

   When you create a new model in Pro/ENGINEER, a sequential seven digit auto-generated number is supplied by default. You can accept this number, or you can enter a name. The name must be no longer than 31 characters and must be a valid Pro/ENGINEER file name. All Windchill ProductPoint names (the name and the file extension) must be globally unique on the server. For example, you can define 1002048–MX2.prt and 1002048–MX2.drw, but you cannot use 1002048.prt again for another part.

If you enter a name that already exists on the server, a warning appears and identifies the location of the model with the same name on the server.

***Caution***

*Do not click* ***OK*** *unless you intend to overwrite the part on the server with the new part. Typically, you click* ***Cancel*** *to prevent overwriting the model already saved to the server. However, it is sometimes useful to start over with the design of a model.*

*If you choose to overwrite existing parts with the new part, any assemblies that reference the old part will now reference the new part. You may encounter problems either opening or regenerating assemblies that use the part. Use caution when selecting* ***OK*** *to overwrite an existing part with a new part.*

3. Select **File** ▸ **Save**. The name you entered for a new part is reserved for you after you click **OK** in the **New** dialog box. As a result, another user cannot use the same name or prevent you from saving your new model.

   The system shows the synchronization status in the lower-right corner while it is uploading the model file to the server. When the upload is complete, Pro/ENGINEER the status shows as Synchronized.

   The newly created file is saved to the specified location on the server. The text **!New** is displayed next to newly created parts.

# Opening and Modifying Pro/ENGINEER Models Already Saved to the Server

This section explains how to find and open Pro/ENGINEER models previously saved to the server. With Windchill ProductPoint, there is only one copy of each model. You cannot save a model with the same name to multiple sever locations. Whenever you save a model that you opened from the server, it is saved back to the server location from which you opened it.

To open a part from a server:

1. You can browse the server locations to find and open a part managed on the server, or you can search for the part using keywords. If you have a lot of model files or deeply nested folder structures, searching is usually the most efficient way to find a part.

   You can apply files names and subtype filters to server location content and search results. Also, if you know the exact file name and extension, you can enter it here just like on the file system.

   Unlike with a file system, any searches apply globally to all content saved to the server, regardless of the current server location.

The search behavior is the same whether executed from Pro/ENGINEER or in a browser against the server. The server must be configured to enable searching or an error is returned. Furthermore, the items that you want to find must already be indexed, or the server does not return them in the result set. The default search indexing interval is 5 minutes. If you just created a new item, you must wait at least 5 minutes before you can search for and find the item under the default search indexing interval.

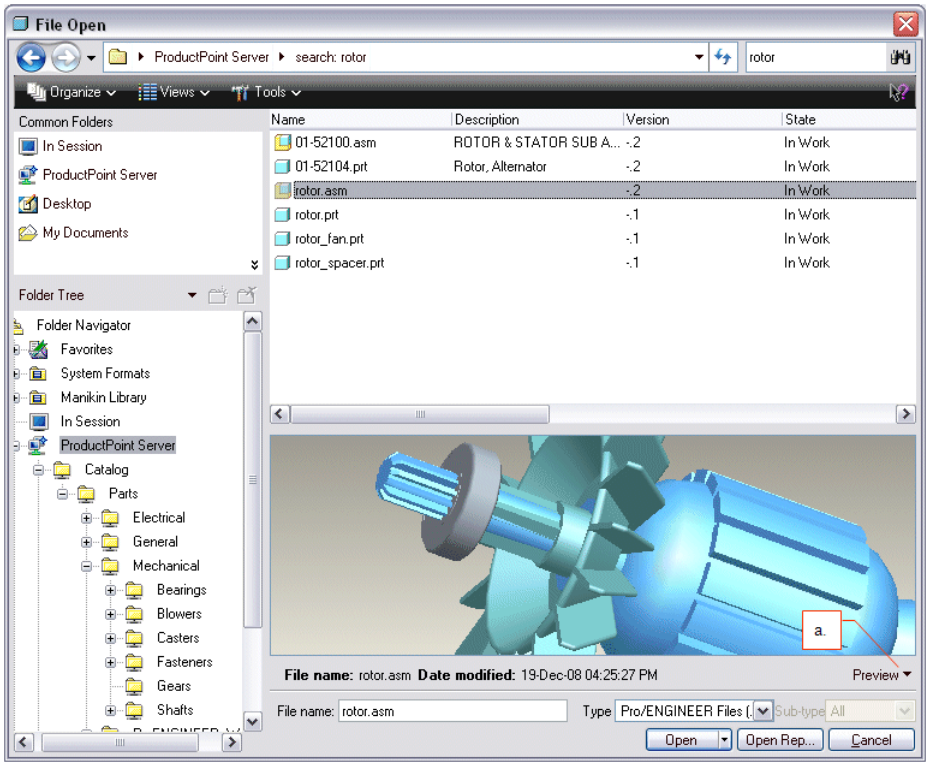
The server searches for matches using each keyword you enter and makes a leading string match against all attributes defined for each item on the server. If any search keyword matches the strings in any product item attribute (starting from the beginning of each string), the server returns that product item in the result set.

You can use the variable character wildcard asterisk (*) and the single character wild card question mark (?) in the search keywords to construct powerful pattern matching searches. For example, if you want to find all assemblies that contain a three letter suffix where the middle letter is s and which start with 100, you could enter the search keyword: `100*_?s?.asm`.

Wild card characters are used to match against only the following attributes:

- Name and Number
- Description

2. Select from the list of items returned in the search result set to open in Pro/ENGINEER.



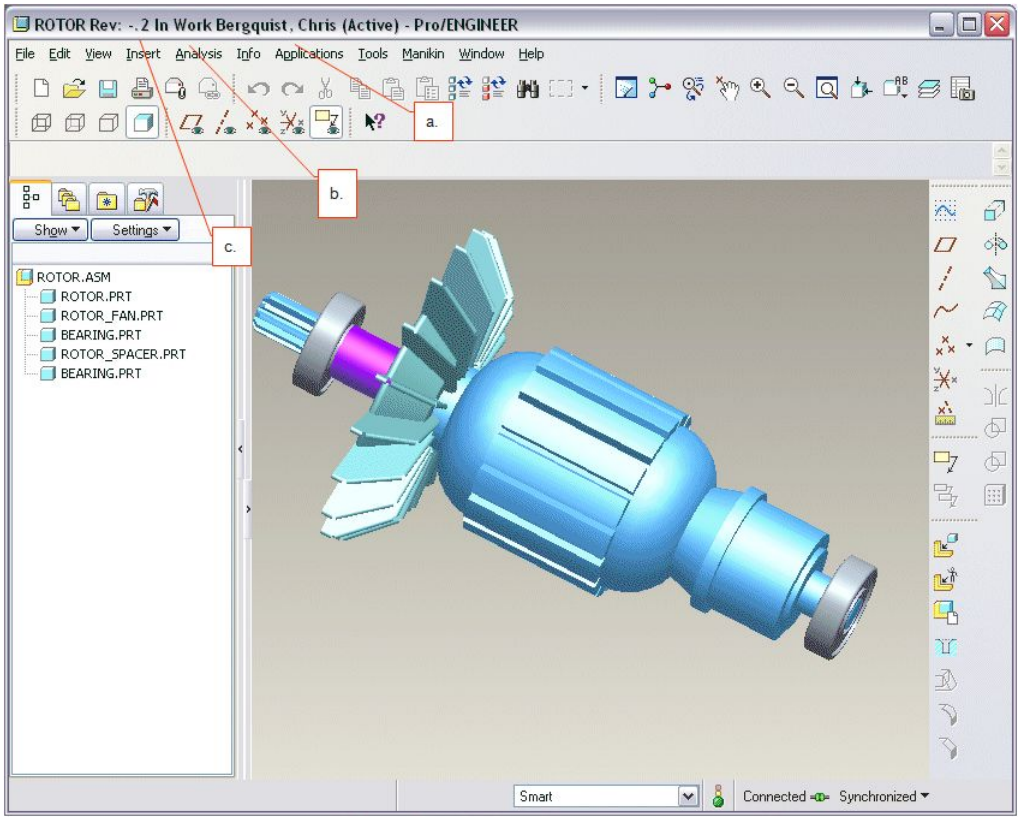

   a. Optionally preview a lightweight 3D image of the item.

3. Pro/ENGINEER always opens the latest version from the server. The version, release state, and name of the user who last modified the item are displayed in the Pro/ENGINEER window title.

   When Pro/ENGINEER is connected to a Windchill ProductPoint server and you attempt to open an item that is already managed on the server, the following rules define what item is opened (these rules apply both to the top-level item and all its dependents):

   - If the item is already in the session memory, opens the version in memory.
   - If the item is not in memory but the latest version is in the user's cache associated with the connected Windchill ProductPoint server, opens the version in the cache.
   - If the item is not in memory and the latest version is not in the cache , downloads the latest version from the server and open it in Pro/ENGINEER.

When you are connected to a Windchill ProductPoint server, you get the latest version of the item.

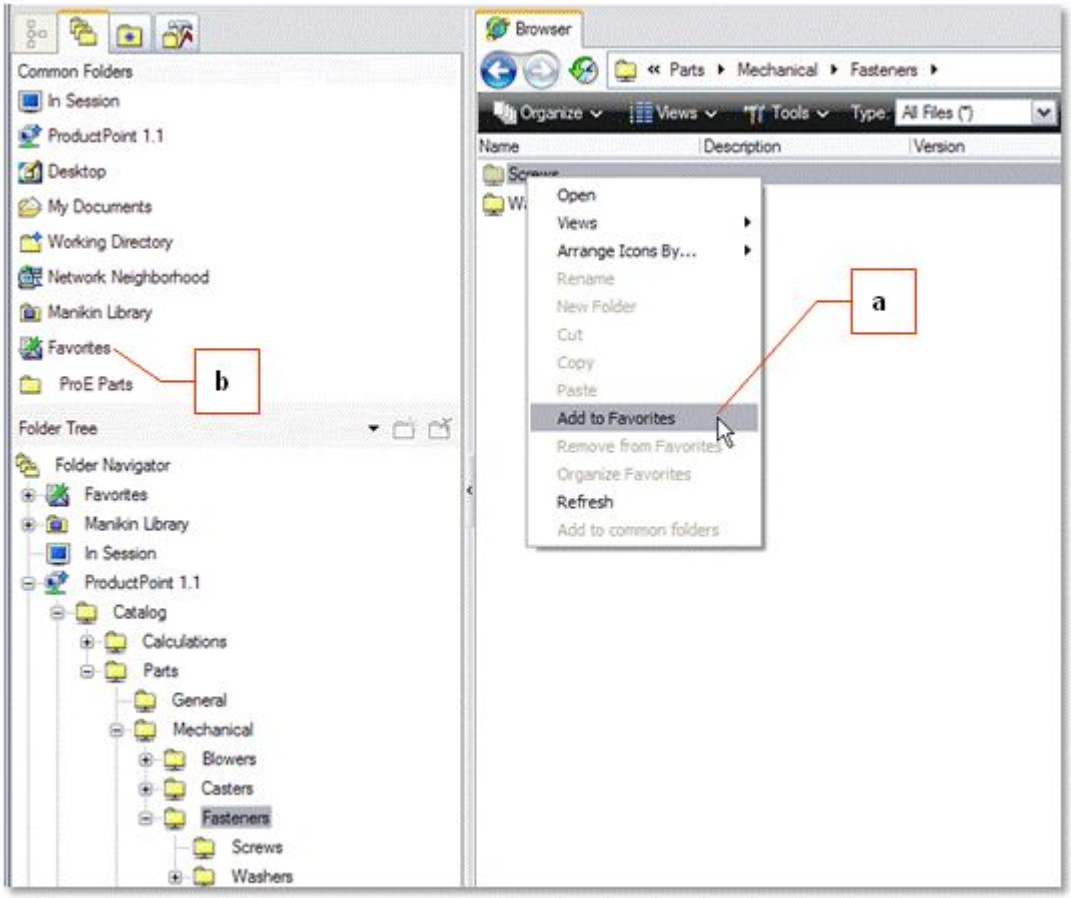



a—The title shows the name of the user who last modified the item.

b—The release state is always displayed to the right of the version and has one of two values: **In Work** and **Released**.

c—Pro/ENGINEER always opens the latest version from your cache or on the server. The model version (revision and iteration) is displayed next to the "Rev:" text in the window title.

Pro/ENGINEER by default saves or opens from the last server location you accessed. You can save frequently used server locations by adding them to your favorites and accessing them through the Favorites links in the Pro/ENGINEER navigator. You can modify your favorites as you change your focus to different products or projects.




a—Navigate to the server location from the Pro/ENGINEER navigator pane and select **Add to Favorites** for the server location to which you want easy access.

b —Access your list of favorites to server locations (you can add server or file system locations to your list of favorites) from the navigator or **File ▸ Open** dialog box.

# Saving Assemblies or Drawings from File System and Reusing Server Models

When you want to save an assembly or a part opened from a file system to the server and the assembly includes drawings, first open the assembly or part drawing from the file system and then save the drawing. When you save the drawing, all dependent assemblies and parts are also saved.

If you have an assembly drawing that you want to save to the server after you have already saved the assembly to the server and if you do not want to create new versions of the parts when you save the drawing, you can use one of two options to reuse existing model files already managed on the server:

- Update the session with models from the server—Use the **File ▸ Update** action to update and replace all items in session with the latest versions of the files with the same name from the server.
- Load the latest version of models to reuse first—First load the assembly from the server and then open the drawing from the file system. When the drawing is opened, it first tries to resolve references from what is in your Pro/ENGINEER session and automatically uses the assembly items you loaded from the server rather than from the file system.

### *Note*

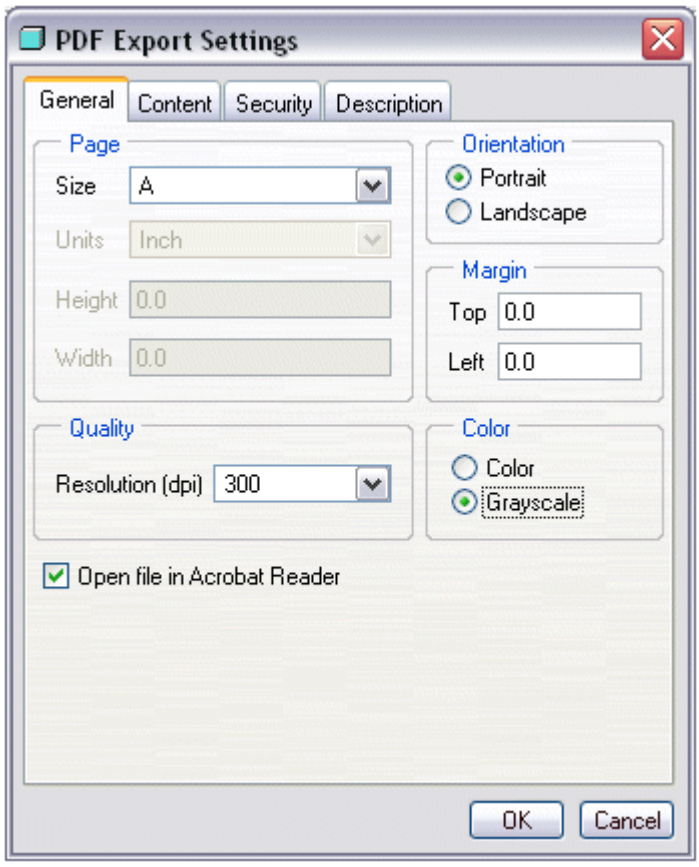
*When you open an assembly drawing from a file system and save it the first time to the server, the assembly drawing and all the assembly parts are saved to the same location as the server. You can move the formats or drawings to different locations on the server after they are uploaded, but you cannot move them across sites. For example, if you create a product site for a bicycle, and you save drawings and formats opened from your file system to a library or folder in the bicycle site, you cannot later move the drawing formats to a central catalog location.*

# Saving Model Files to External Formats in Pro/ENGINEER

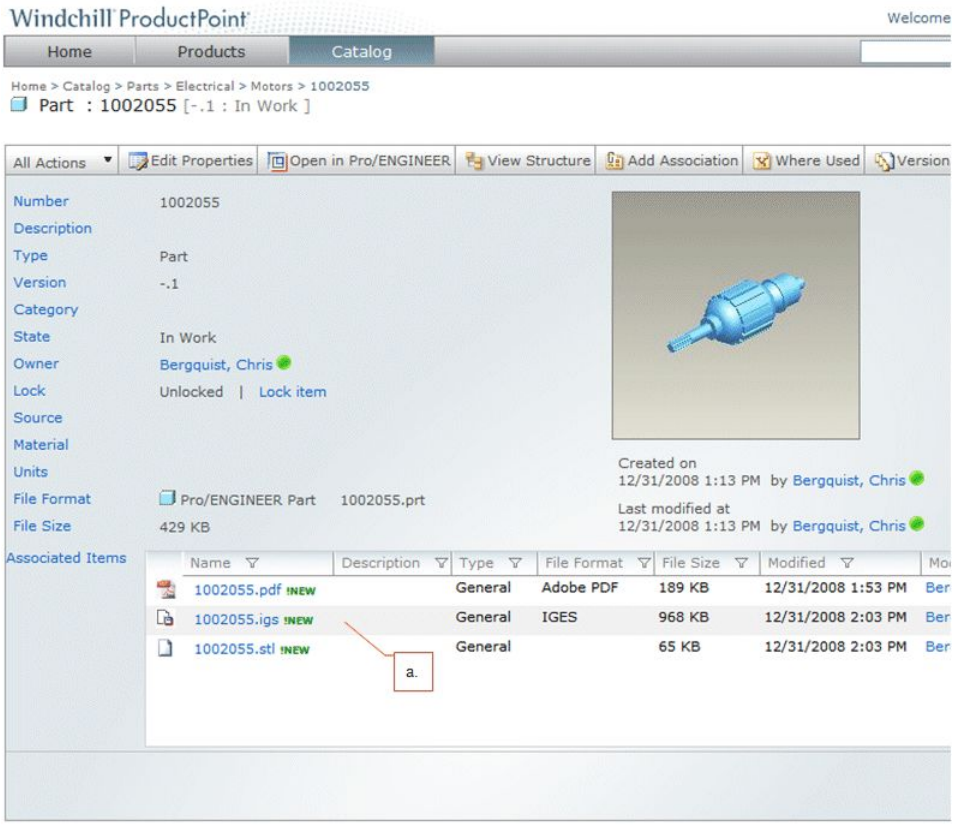
In Windchill ProductPoint, derived formats of Pro/ENGINEER parts and drawings, such as PDF and IGES, can be published and automatically associated with the Pro/ENGINEER model from which they were generated. This ensures that downstream consumers, such as those in manufacturing and purchasing, can easily get the formats they need, and they can be certain the versions of the derived formats correspond to the latest part, assembly or drawing versions.

1. Select **File ▸ Save a Copy** for the Pro/ENGINEER model for which you need to generate another format.
2. Select the options based on the format type you want to generate. Pro/ENGINEER automatically generates a name for the derived format based on the original model name.

3. Select the format generation options and click **OK**.



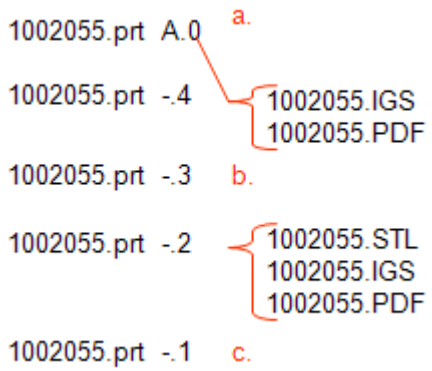
4. The format is automatically uploaded to the server and associated with the model version for which it was generated.




a. All derived formats that are created for the latest version of the model are displayed on the server properties page for the model item. If you make a change to the model and save the changes without regenerating the derived formats, no derived formats are displayed (the previously created formats are still associated with the previous model version).

### *Note*

*Only the derived formats created for that version are displayed in the associated items list on the server properties page for the model. If you manually upload and associate a file, it is associated with all versions of the model. This means that if the current version of a model has PDF, IGES and STL derived formats and the model is updated without regenerating the formats, the associated items list no longer shows the derived formats (because they may no longer be accurate for the new version of the model). All manually associated items (those associates created directly by uploading files or by relating to existing files) are displayed in all versions of the model.*




a— Release version label shows derived formats associated with iteration that is released.

b — No formats generated for this part version.

c — No formats generated for this first part version.

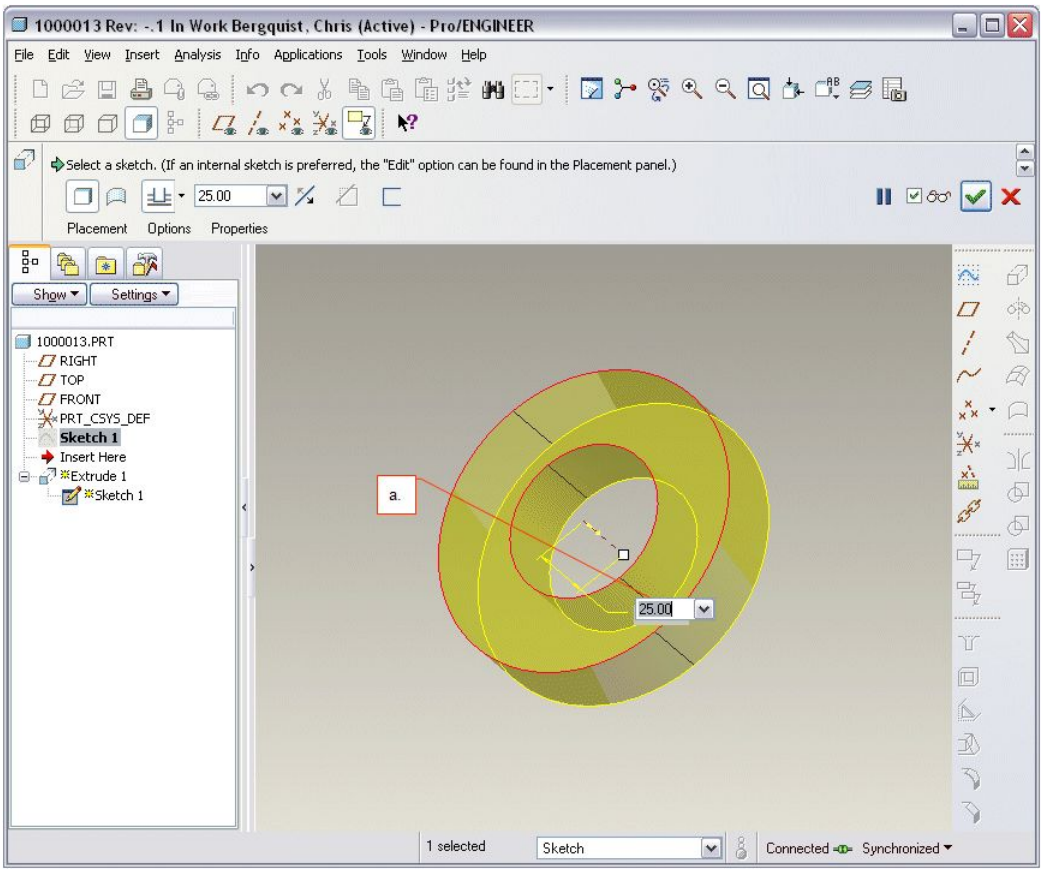
# Managing Locks of Pro/ENGINEER Models Saved to the Server

Windchill ProductPoint is designed to prevent overwriting changes by others. This is accomplished by locking files for one editing user at a time. Locks are automatically set and released as you make edits and save changes. You can also manually set and release locks to prevent others from changing models and overwriting your changes.

## Automatically Managed Locks

You can rely on the automatic lock behavior most of the time, but sometimes it is useful to lock items ahead of time when you expect to make a lot of changes over a period of time.

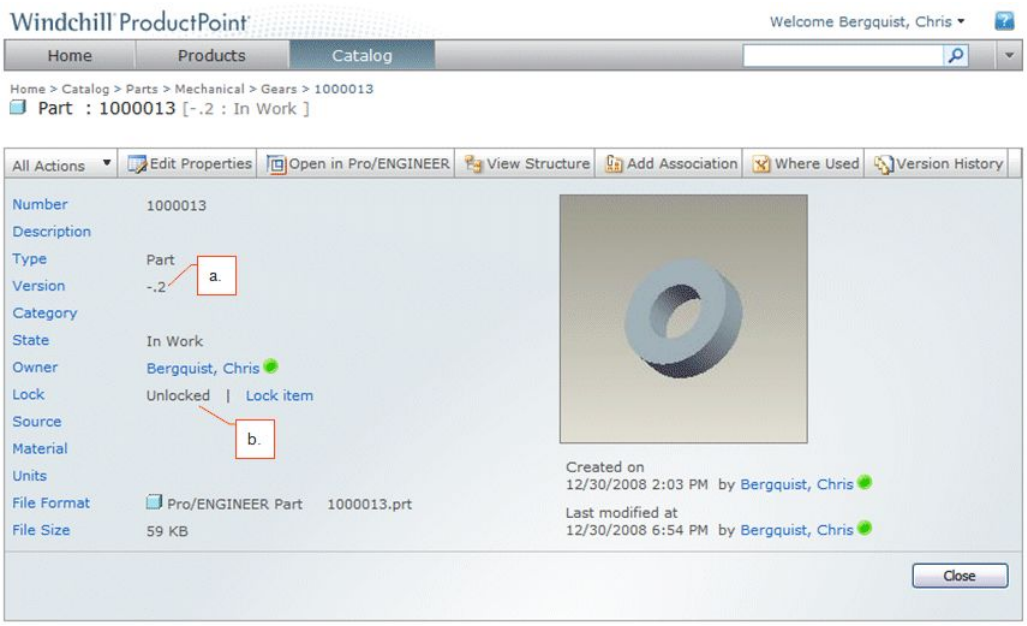
When you select an action in Pro/ENGINEER to begin editing an item managed on the server, Pro/ENGINEER automatically locks the item on your behalf on the server. This lock prevents others from concurrently making changes and overwriting changes you make.




The server displays the lock status in the properties pages and in list and structure views that include the lock status column.

Other users can download items locked by others into a Pro/ENGINEER session, but they cannot modify locked items until you save your changes.

When you save and upload changed files to the server, the lock for the item is released automatically. At this point, another user can edit the model and apply changes on top of your changes.




a—The part version includes a new iteration for the change you just saved.

b—The lock state is automatically changed by the system from locked on your behalf to unlocked after your changes are saved and the changed model file is completely uploaded to the server.

The automatic locks created by the system are automatically released when:

- You save changes to the items and the changed file(s) are fully uploaded to the server.
- You erase the changed items from session without saving the items.
- You restart Pro/ENGINEER, and the system applies locks to the items in your cache where you have not saved changes (these locks may occur in the event of a Pro/ENGINEER crash where changes were started but not saved and completed).

When Pro/ENGINEER lists server-managed items in the **File ▸ Open** window and the embedded browser, a **Locked By** status column indicates if any of the items in the list are locked by your or other users. Check this column before you open a part to edit.

If another user has a lock on an item and you attempt to change the item in Pro/ENGINEER, a lock conflict warning is displayed, identifying each item you are attempting to modify that is locked by another user. You can continue making the change, but you cannot save the changes to the server.

Sometimes it is useful to make temporary changes to experiment with a potential change or to create temporary geometry needed for obtaining a measurement. Normally it is best to click **Cancel** and abandon the edit and request that the locking user notify you when they are finished making their changes so that you can follow with the changes you need to make.

### *Note*

*If an item is not locked on the server, but Pro/ENGINEER still displays the lock warning, this is because Pro/ENGINEER is using the cached status of the lock. By default, the cache is updated only every 5 minutes. If you try again after 5 minutes have passed, the lock status of the cache should be accurate and you can make your changes.*

If a lock is inadvertently set or not released, an administrator can unlock any lock. However, if any changes were not synchronized by the user owning the lock, then that user will not be able to synchronize the changes.

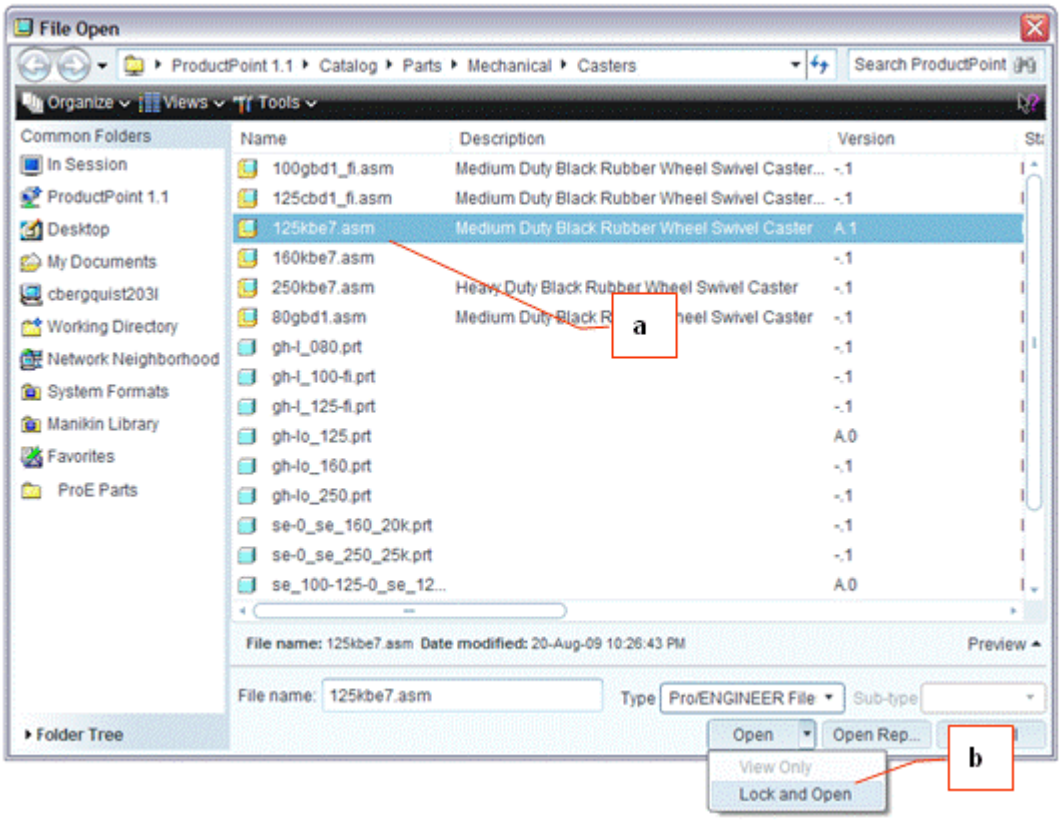
If you attempt to edit an item that is locked by another user, Pro/ENGINEER will display a conflict warning and identify the user who has the item locked. Click **OK** to continue with a temporary change to the item, but you cannot save your changes to the server.

If you override the lock conflict and continue with your edit and then later try to save the changes, you will encounter another lock conflict as Pro/ENGINEER attempts again to lock the part to save the changes. In this case, you cannot complete the save to the server.

## Manually Managed Locks

You can manually set and release locks you set for model items in Pro/ENGINEER or on the server using the browser. It can be useful to manually lock items when you expect to be making changes for a period of time and want to ensure that no-one else is also making changes that conflict with yours. You can manually lock and unlock the active item from the file menu, or you can manually lock and unlock items using a right-click action in the model tree.

With Pro/ENGINEER Wildfire 5.0, you can also open and lock an item using **File ▸ Open**.




a—Select item to **Lock and Open**.

b—Select **Open** and then **Lock and Open** to reserve the object for your exclusive privileges to edit.

When you manually lock an item, the lock is not automatically released when you save changes. All locks that you set manually, you must also manually unlock.

You can view a list of locks that you manually set or those set by the system on your behalf in the Locks web part on your home page.

The lock action is only displayed if:

- You have privileges to edit the item.
- It is not already locked (by you or another user).

The unlock action is displayed when:

- You have the item locked.
- All changes to the locked item are already synchronized.

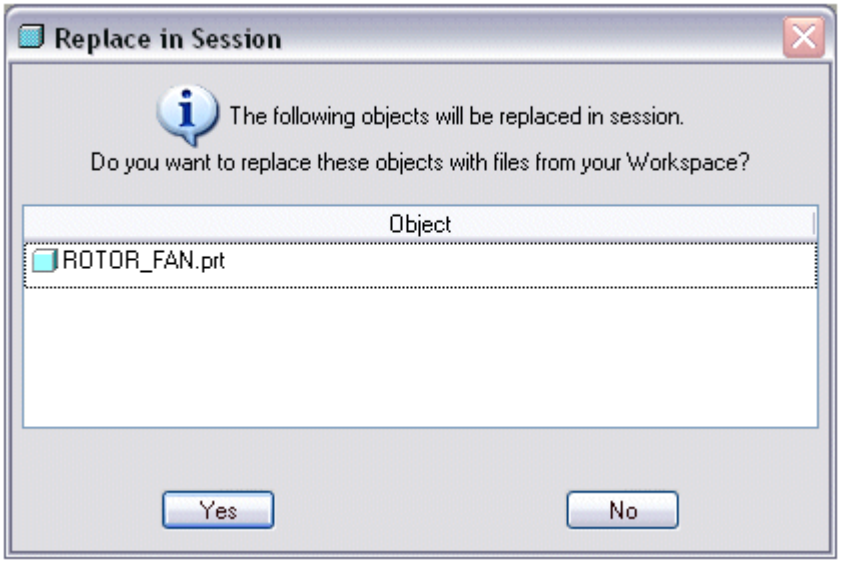
# Updating Out-of-Date Items in a Pro/ENGINEER Session

Sometimes changes are concurrently made by other users while you have an item in session in Pro/ENGINEER. Other users may either change the items through server operations, for example, by modifying the attributes, or change the model content in Pro/ENGINEER. Windchill ProductPoint does not allow you to make changes to items in session which have been concurrently updated by others. This behavior is designed to prevent one user from inadvertently overwriting changes made by another user.

If you attempt to edit items in session which have been modified by others since you opened them, Pro/ENGINEER displays an out-of-date conflict warning. Click **OK** to continue with your change, but if you save your change, you will overwrite the changes made by the last modifier of the item.

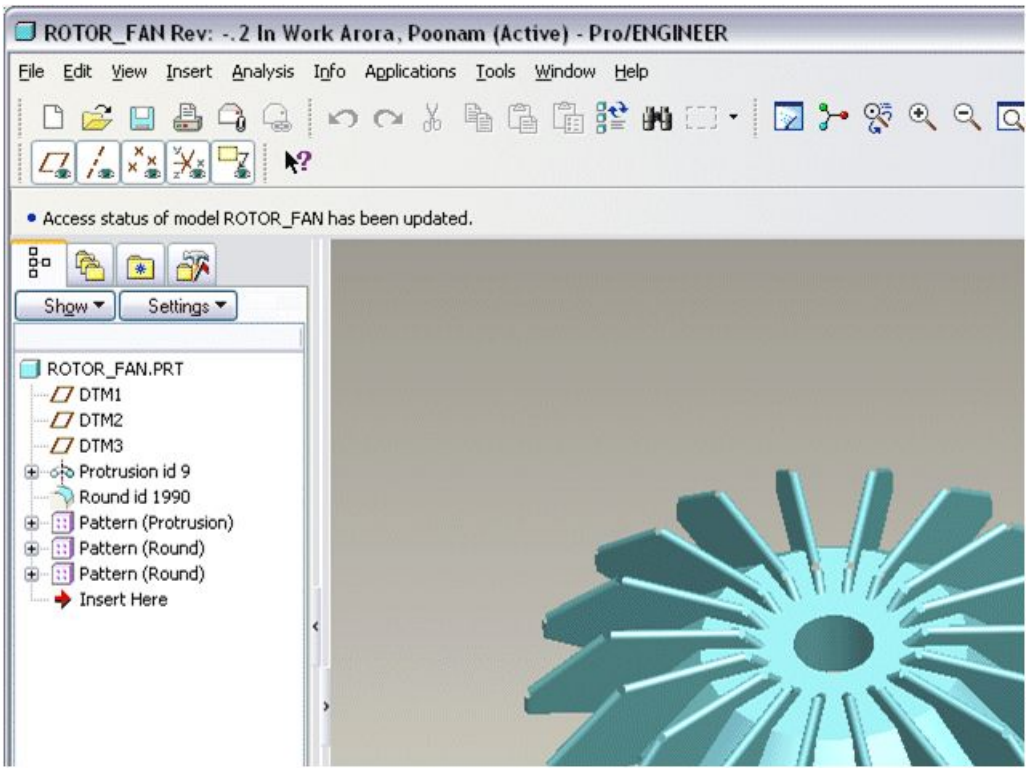
It is best to cancel and use **File** ▸ **Update** to bring the latest version of the item in session and to apply your changes on top of the changes made by the last modifier of the item.

After you select **File** ▸ **Update**, Pro/ENGINEER displays a dialog box that lists the items in session that will be replaced with newer versions from the server if you continue with the update.




When you select **File** ▸ **Update All**, every item in session that has more recent versions on the server is replaced with the latest versions of the items downloaded from the server. After the update, the new version of the updated item is displayed

in the Pro/ENGINEER window title. Following the update, notice that the title shows a more recent version of the item and also identifies the user who last modified the item (as shown in the next figure).




# Renaming Items in Pro/ENGINEER and on the Server

Pro/ENGINEER supports two approaches for renaming items:

- Rename items already saved to the server
- Rename items before saving to the server

## Renaming Pro/ENGINEER Items Previously Saved to the Server

It is usually best to rename items already saved to the server using the server-supported rename actions in the browser. You cannot rename items already managed on the server from Pro/ENGINEER Wildfire 4.0. Furthermore, the browser rename action supports automatic name generation for the rename.

## Renaming an Item in Pro/ENGINEER Before Saving to the Server

If you open a file from a file system that has the same name but is a different item than the one that already exists on the server, you can rename the item in Pro/ENGINEER and then save the renamed item to the server so that you do not overwrite the existing item.

In Pro/ENGINEER, the ability to rename files is disabled by default when you are connected to a server, so that you do not unintentionally use the rename action in Pro/ENGINEER to create copies of parts that are already managed on the server. You can enable the rename action by setting the `let_proe_rename_pdm_objects` config.pro option to `yes`.

1. After enabling the rename action, make sure the server-managed item is not already in memory and then open the item from the file system.
2. Click **File** ▸ **Rename**.
3. Enter the new name to use for saving the item to the server.
4. Save the model that is renamed in session. If you are connected to a Windchill ProductPoint server, the model in session is saved to the server under the new name. The file system version originally opened in session is unaffected by the change.

Consider these tips on renaming items:

- If you open an item from the server and use the **File** ▸ **Rename** action in Windchill ProductPoint, a copy of the item is created and all assemblies that reference the item in session will be updated to use the new copy. Once an item is saved to the server, it is usually best to rename it using the server rename action in a browser.
- If you have an assembly on the file system and need to rename a large number of components in the assembly before saving them to the server, you can first select **File** ▸ **Save a Copy** and rename the files. Make sure to load the copy and to erase the original items from memory before you save a copy to the server; otherwise, you will encounter name conflicts for the items that already exist on the server.

# Working with Drawings in Pro/ENGINEER

When you create a new drawing, you can specify the format to use with the drawing. It is best to create a folder in a catalog for your standard drawing formats first and then upload all your formats to this location before creating drawings. You can then make this catalog folder location read-only for all users other than site administrators, so it is not inadvertently modified.

If you save a drawing that you opened from a file system and the format name referenced in the drawing already exists anywhere on the server, a conflict dialog box is displayed, indicating that the format already exists. The typical conflict resolution is to use the format already defined on the server. If you click **OK**, the format file that already exists on the server is used.

If the format associated with the drawing being saved already exists on the server, the format in session is not be saved, and the format on the server is used instead. If you do not want to use the format on the server, click **Cancel**, rename the format to a new name, and save the drawing.

When you save a drawing to the server, the server displays a Drawings folder in the model structure and BOM structure views for each part or assembly that is referenced in one or more drawings. The Drawings folder lists all the drawings in which the part or assembly is referenced.

The Drawings folder is displayed only if a part is referenced in one or more drawings and only in the BOM structure and model structure views. Each drawing displayed in the folder has a reference to the part above. The same part or assembly may be referenced in multiple drawings without being the base model for the drawing.

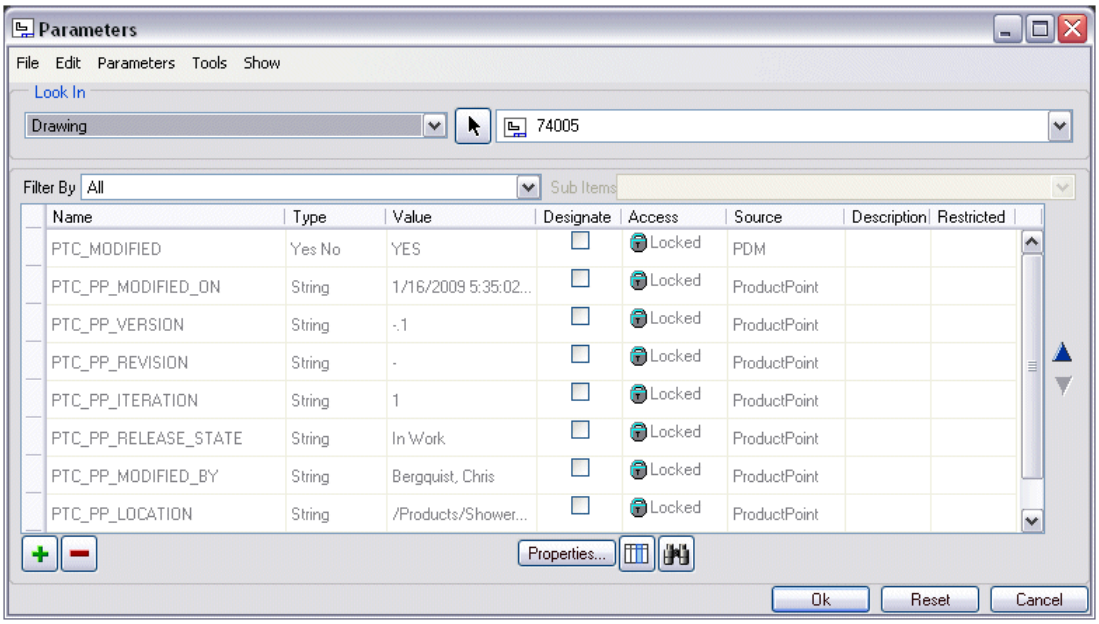
When you define a format, you can reference the parameters (listed in the next table) in any table cells in the format. The current values for the parameters are displayed when the format is rendered as part of a drawing.

Several server system attributes are always exchanged with Pro/ENGINEER files. These attributes are managed by the server and are read-only in Pro/ENGINEER. These attributes are useful for including in drawing title blocks to identify drawing or model versions, release status and last modified dates. These attributes are fixed by the server and cannot be extended.

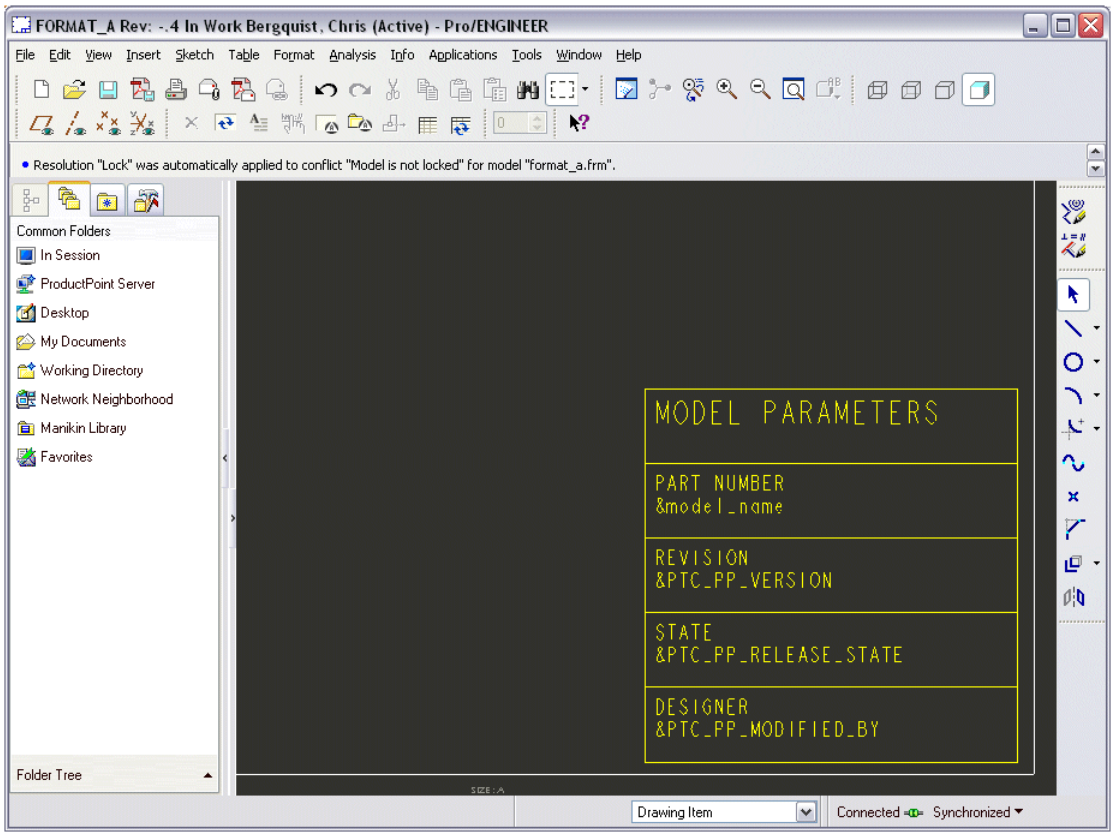
| Parameter Name | What It Controls | Description |
|---|---|---|
| PTC_MODIFIED | Modified | Boolean parameter that identifies whether the item in session was modified since it was loaded into session |
| PTC_PP_REVISION | Revision | Revision label for example “A” of version “A.5” |
| PTC_PP_ITERATION | Iteration | Iteration number; for example “5” for version “A.5” |
| PTC_PP_VERSION | Version | Combination of revision and iteration labels “A.5” |

| Parameter Name | What It Controls | Description |
|---|---|---|
| PTC_PP_RELEASE_STATE | State | Release state or status of item. Release state can have one of two values: In Work, or Released. The release state is independent of the lifecycle state. |
| PTC_PP_LOCATION | Location | This is the path to where the item is defined on the server and should include the server name/site name/library name/folder name/subfolder name/object name. |
| PTC_PP_CREATED_BY | Created by | Full name of user who created item. |
| PTC_PP_CREATED_ON | Created on | Date/time at which item was created. |
| PTC_PP_MODIFIED_BY | Modified by | Full name of user who last modified item. |
| PTC_PP_MODIFIED_ON | Modified on | Date/time at which item was last modified. |

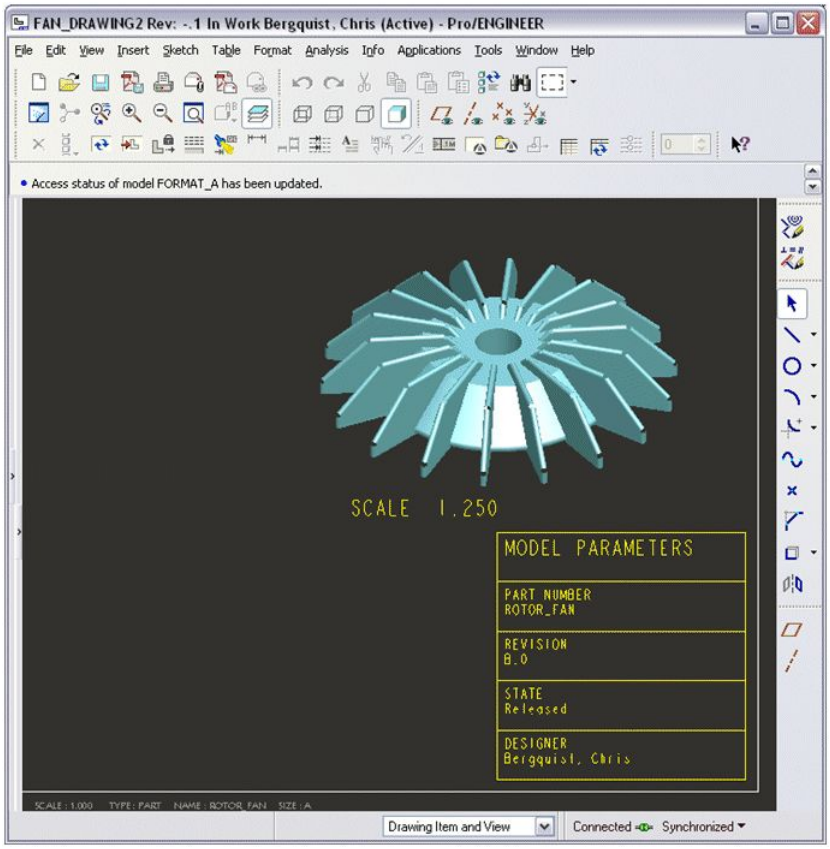
Any of the parameters in the next figure can be referenced in a drawing table cell. The current values for the parameters are displayed when the format is rendered. You cannot change and do not have to designate any of these PTC-defined server-managed parameters. These parameters are always downloaded from the server with the Pro/ENGINEER file.



When including parameters in a drawing cell, you can choose to reference either the drawing values or the base model values for which the drawing was created. To include a parameter for a drawing default model (base model) value, insert `&<parameter_name>`in the table cell for a format or a drawing. To include a parameter for the drawing value itself, insert `&<parameter_name>:d` in the table cell for the format or drawing.




When format is included in the drawing for a model, the parameter values for the default model are displayed for the parameters with the `&` prefix.



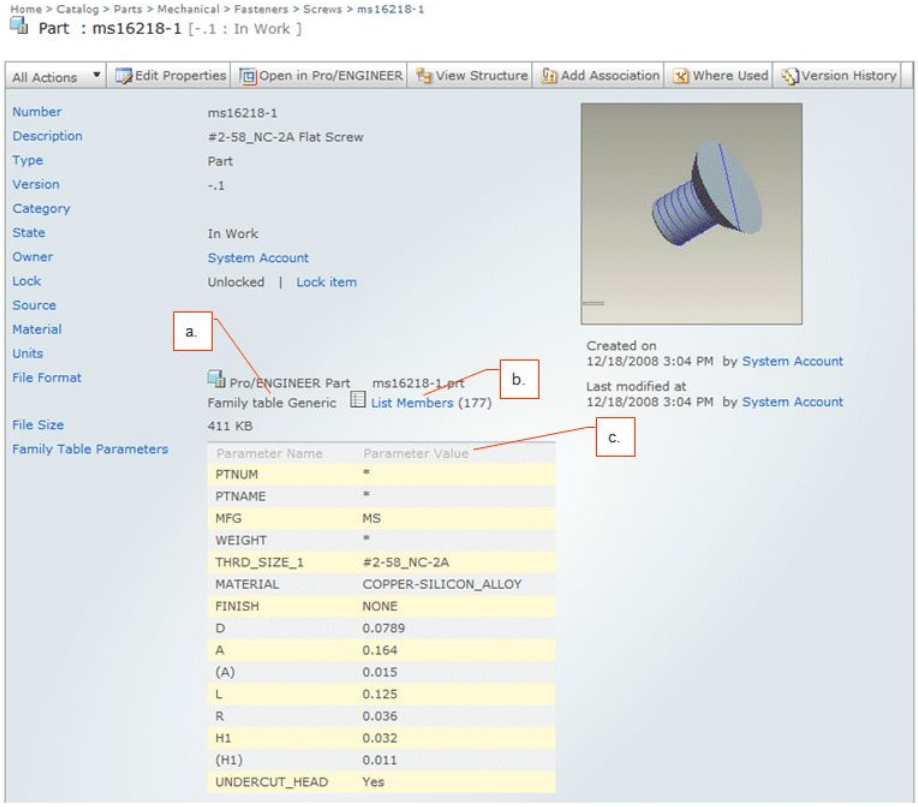

# Working with Family Tables in Pro/ENGINEER

Family Table instances are managed as separate parts on the server. That is, the Family Table instances have individual server-defined attributes. Following is a description of behavior and limitations for the Family Table support:

- When a Family Table generic is saved to the server, a separate product item is created for the generic and each instance. The instance name is used as the part number for each instance.
- Family table parameters cannot be updated on the server. You must download and modify the generic and update the parameter for the instances of interest in Pro/ENGINEER, and then save the changes to the server.
- New Family Table instances cannot be created on the server. New instances must be created in Pro/ENGINEER by modifying the generic, adding the new instance to the generic, and then saving the instance to the server.
- When an assembly that includes instances is copied on the server, a copy of the generic is made that includes only the instance from that generic that is copied in the assembly. For example, if an assembly includes instances IA and IB and both belong to generic GA, but GA also defines instances IC and ID that are

not used in the copied assembly, then a new generic is created as a copy of GA that includes only copies of instances IA and IB without including IC or ID.

A recommended practice is to verify Family Table instances before saving them to the server to ensure that the instances can be regenerated when referenced in an assembly. You can force verification on save by setting `verify_on_save_by_default` config.pro option to `yes`.




a—Properties page for the part identifies part as generic of Family Table.

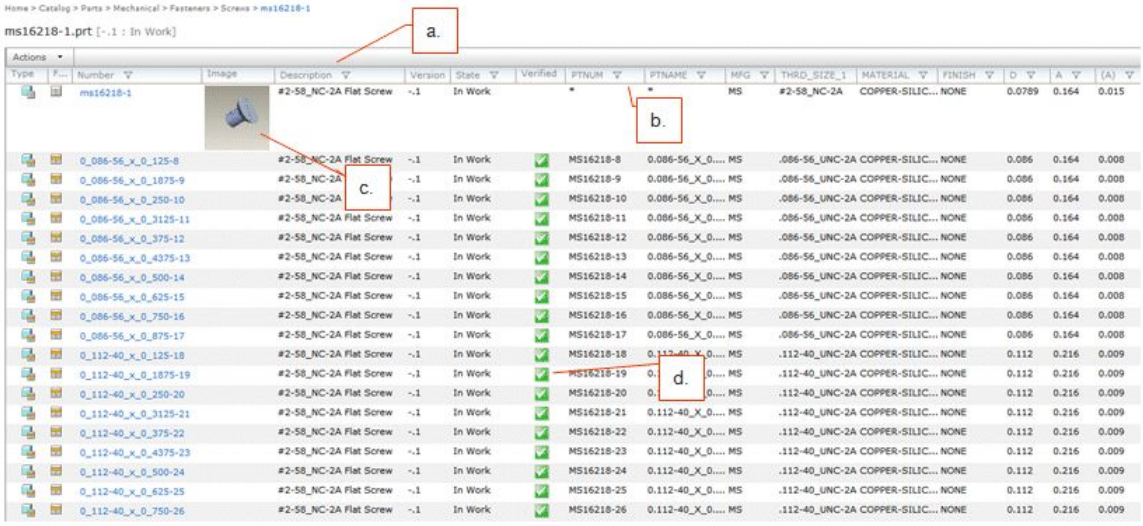
b—Select a link to list all instances (177) defined for generic.

c—Table of parameters and values for generic or instance.

You can easily identify whether a part is a generic or an instance on the properties page for the part on the server:

- You can access all instances from the List Members link on the generic (as well as view a total count of instances).
- You can access the parent generic for any instance from the properties page on the server by selecting the generic name link in the **File Format** section.

- Thumbnail images and viewables are generated only for the generic (not for the instances).




a—A listing of all members of a Family Table on the server.

b—All parameters associated with Family Table are displayed so you can filter or sort these parameter values.

c—The generic is listed first and includes an image. Images and viewables are not generated for instances.

d—A green check symbol indicates that an instance was verified before saving to the server.

You may wish to organize your family generics and instances in separate folders, for example:

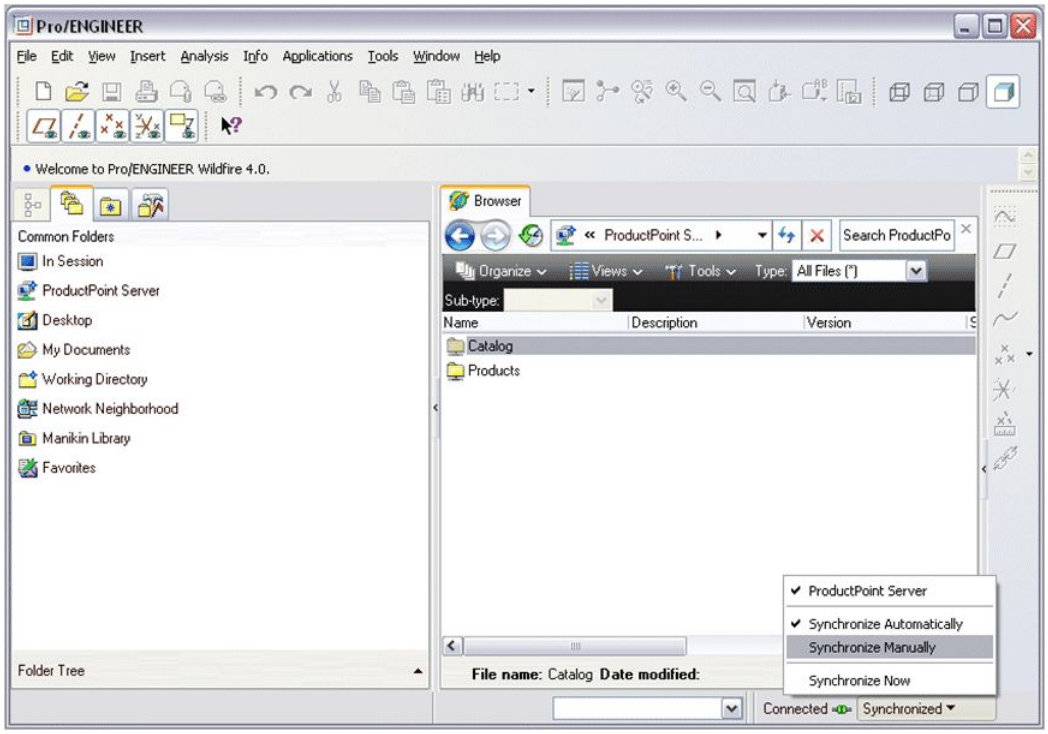
- Cap Screws
  - Cap Screw generic part
- Cap Screw Instances
  - Cap_Screw_10x20
  - Cap_Screw_10x40
  - Cap_Screw_10x50

# Manually Controlling an Upload of Saved Changes Made to the Server

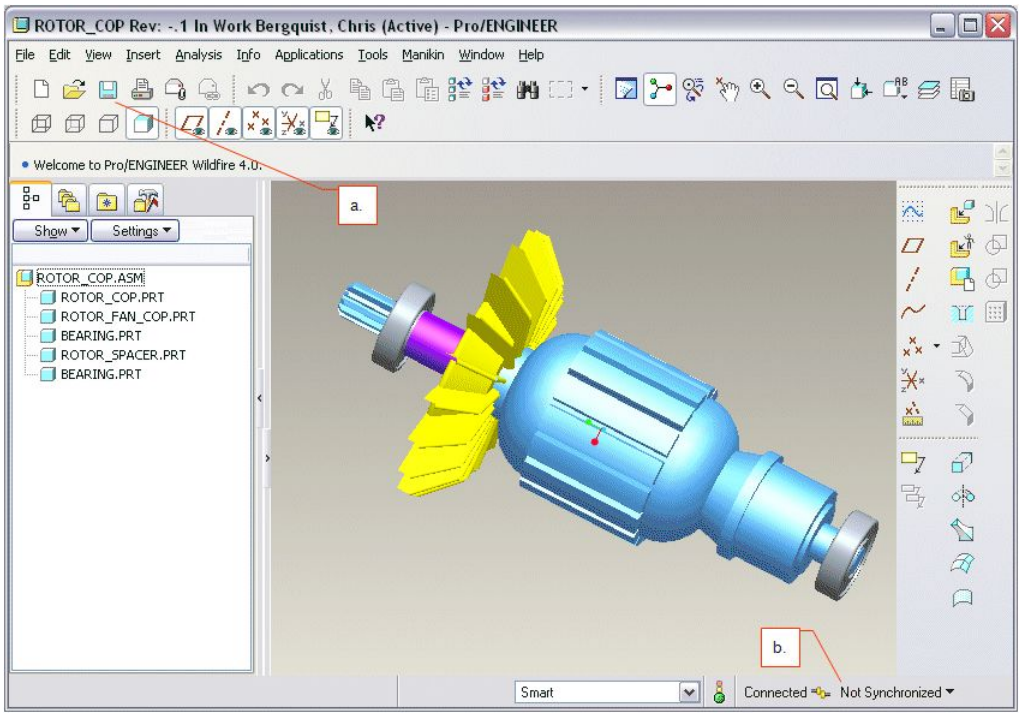
You can choose to manually control when the changes you make to Pro/ENGINEER models are uploaded to the server. You can delay the upload to reduce both the network load and client-server processing time. This approach also reduces the number of model file versions saved to the server and thereby reduces the disk space used by the database.

The default behavior is to automatically upload file changes to the server each time you save a model in Pro/ENGINEER. You can change the option to synchronize manually, so that changes are uploaded only upon your request when you select **Synchronize Now**.

1. Select the server menu in the lower right corner of the Pro/ENGINEER window to view the server options.
2. Change the synchronization mode from **Synchronize Automatically** (default) to **Synchronize Manually**.



3. Open models from the server, make changes, and save them in Pro/ENGINEER.

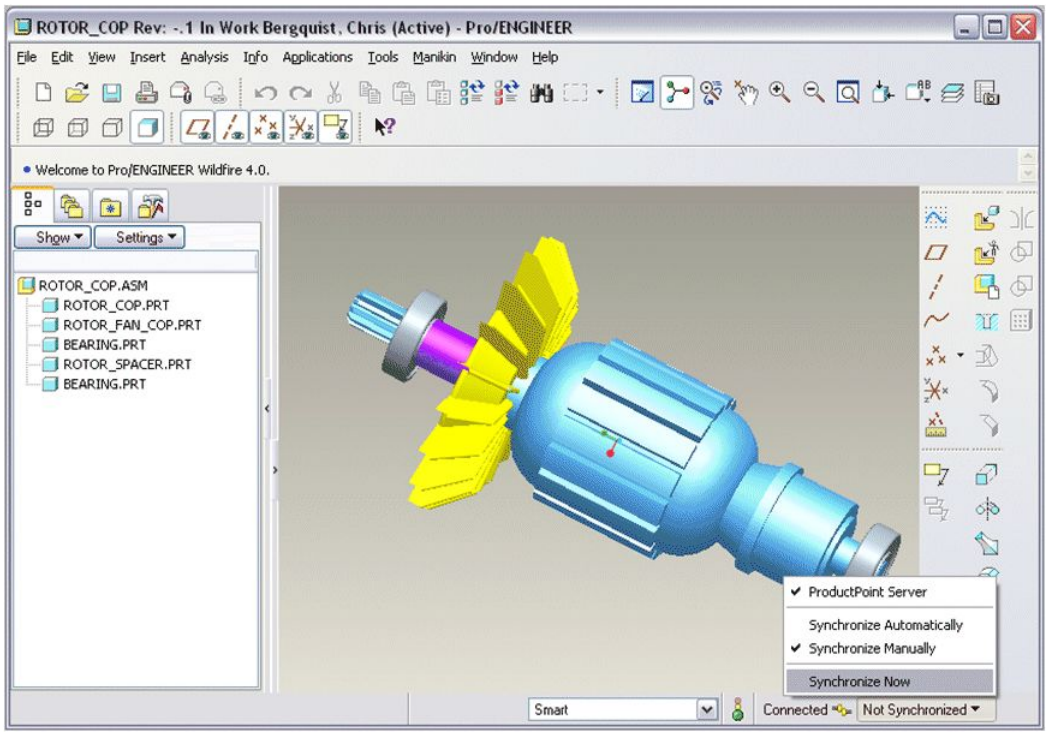



a—Save the changes.

b—When you save changes using **Synchronize Manually**, the status **Not Synchronized** is displayed after the first time you save changes. **Not Synchronized** means that you saved changes to your cache, and they are not yet uploaded to the server. You can continue making and saving changes to server-managed items while working in Pro/ENGINEER.

4. When you are ready to upload changes to the server, click **Synchronize Now**.

The rotating synchronizing symbol with the word **Synchronizing** is displayed until all changes saved to your cache are fully uploaded to the server.




When all changes that are saved to your cache are fully uploaded to the server, the green symbol and Synchronized status are displayed.

### *Note*

*When you click* ***Synchronize Now****, all items that have been saved in Pro/ENGINEER to your cache but have not yet been uploaded to the server are all uploaded. This includes items that you previously saved and erased from memory in the Pro/ENGINEER session. Clicking* ***Synchronize Now*** *applies to all items in your cache for which your saved version is more recent than the versions on the server.*

Whenever you exit Pro/ENGINEER and the synchronization mode is set to **Synchronize Manually**, all changes you saved to your cache that have not been uploaded to the server are automatically uploaded. The synchronization occurs in the background even when Pro/ENGINEER is not running. If you shut down your computer or disconnect from the network before the upload is complete, the upload restarts when you restart your computer and reconnect to the network. If you do not want the system to synchronize your changes on exit from Pro/ENGINEER, set the `SYNC_ON_DISCONNECT` environment variable to `false`.

All model files for which you have saved changes to your cache but which have not been uploaded to the server are automatically locked on your behalf. The lock is automatically released when the upload is completed.

If a lock on a model has been changed by a user but the changes have not yet been uploaded to the server, the system automatically locks the items and displays a lock status of **Waiting to synchronize changes on... by...**. The automatically created lock is released when you click **Synchronize Now**.

***Caution***

*The user for whom the lock is created or an administrator can select the action to unlock the item on the server, but by doing so will ignore the unsynchronized changes previously saved to the user's cache.*

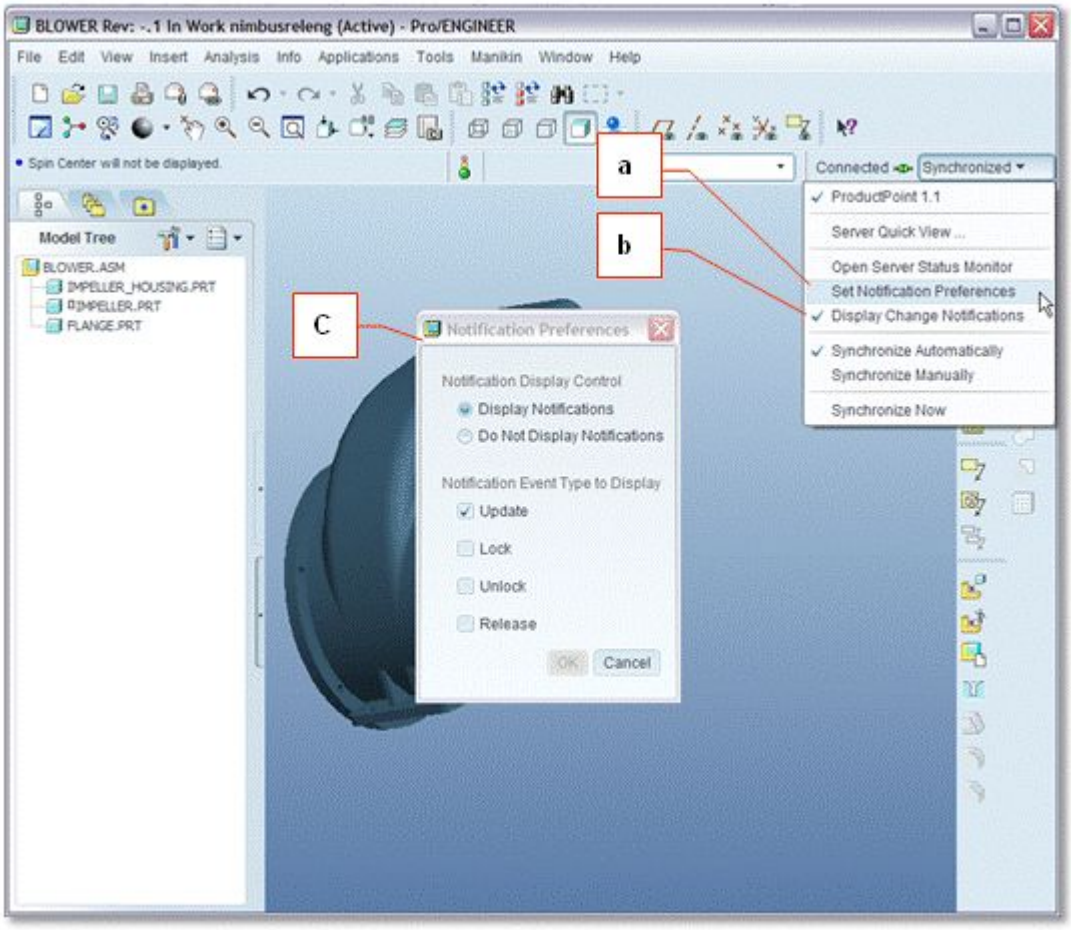
# Change Notification

In Pro/ENGINEER Wildfire 5.0, you can use the dynamic notification capability that allows you and your team to be immediately notified of the changes made concurrently to items in your Pro/ENGINEER session. You can also contact users who initiated the changes through user presence actions.

You can turn notifications on and off from the server control bar in Pro/ENGINEER Wildfire 5.0. Select the server control menu **Display Change Notifications** to toggle the notifications on and off. Select **Set Notification Preferences** to specify the events for which you want to be notified.

You can choose to be notified for any of the following events for the items in session:

- Update (modification is concurrently made by another while the model is in your session)
- Lock
- Unlock
- Release




a—Open a dialog box to set notification preferences.

b—Toggle the notification pop-up display on or off.

c—Set notification preferences.

When you enable notifications and when other users make changes to models that you also have in your Pro/ENGINEER session (either changes made in Pro/ENGINEER or changes made on the server), a notification appears in the lower-right corner of the window. When a notification is delivered, you can take actions to update the item in session or to contact the person who made the changes.

## Server Status Monitor

If you are using Pro/ENGINEER Wildfire 5.0, you can access a summary of current server status associated with items in session using the **Open Server Status Monitor** action from the server control menu. The status monitor lists all objects in session which are out of date, locked by others, and renamed, and it also shows the server version, state, changed by and change date/time. The monitor provides visibility on request for items that have been concurrently updated since you loaded the models.

## User Presence and Actions

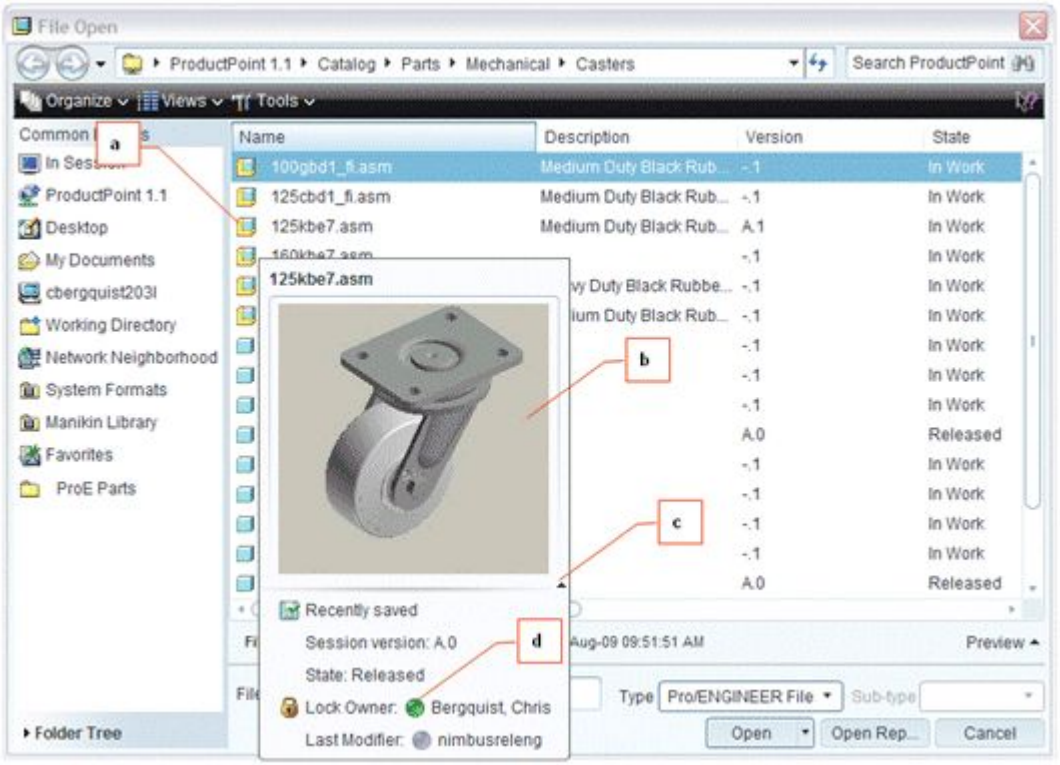
If you are using Pro/ENGINEER Wildfire 5.0, you can take advantage of the user presence status associated with items managed in Windchill ProductPoint. You can view the user presence for model items and can contact users who locked the items directly from Windchill ProductPoint.

User presence requires that you have installed or are running any of the following messaging services:

- Microsoft Office Communicator 2005
- Microsoft Office Communicator 2007 (at Communicator 2005 level of functionality)
- Microsoft Instant Messenger 4.7 and later

## Thumbnails and PDM Status

If you are using Pro/ENGINEER Wildfire 5.0, you can view thumbnails for items in the **File** ▸ **Open** dialog box and in the Pro/ENGINEER model tree.



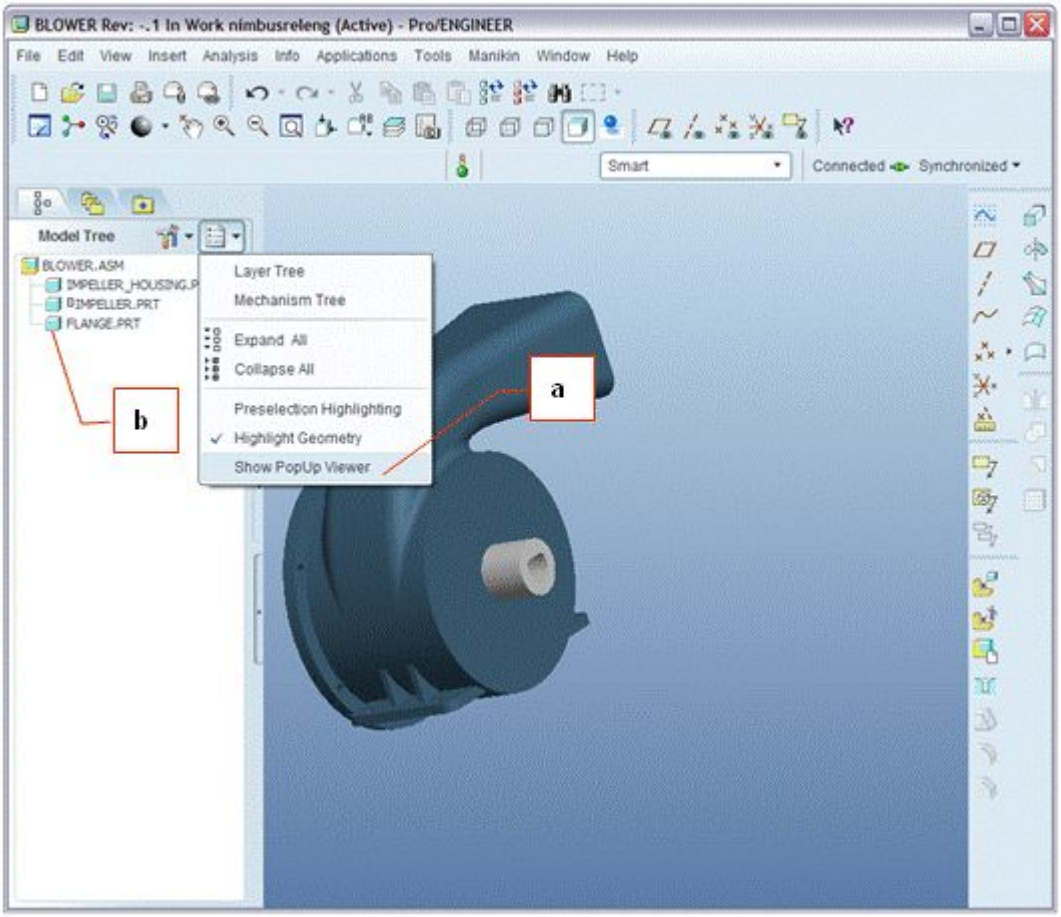

a—Hover-over type icon with the cursor and the thumbnail image.

b—Move your cursor into the image and left-click to load and manipulate an embedded 3D viewable.

c—Select the arrow to show/hide the server details (after you first show the server details, the details are displayed until hidden).

d—Right-click on the presence icon to access the presence actions.

You can enable the hover thumbnails in the model tree by selecting the option **Show PopUp Viewer** from the model tree options menu.

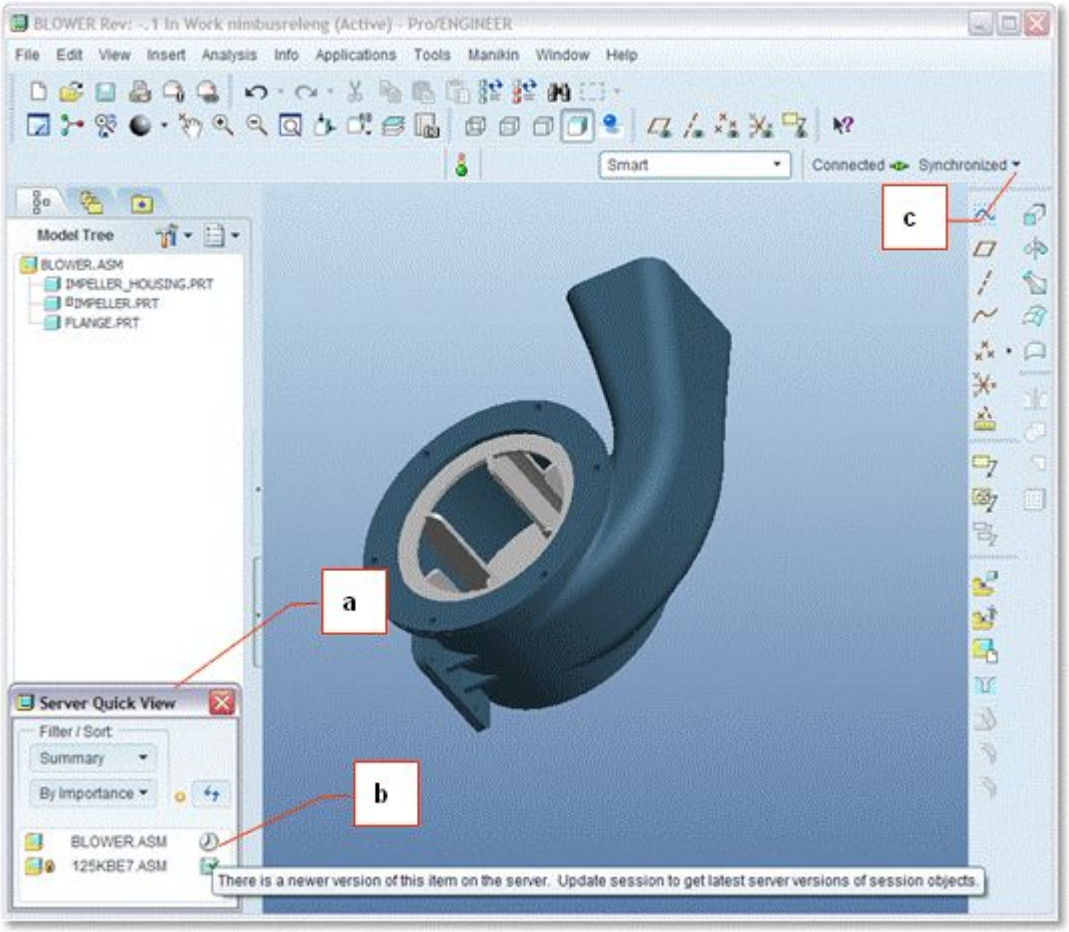



a—Select **Show PopUp Viewer** to toggle display of hover images in model tree.

b—After enabling the PopUp Viewer, move your cursor over model object type icon to show thumbnail and PDM properties for items in the model tree.

## Server Quick View

If you are using Pro/ENGINEER Wildfire 5.0, you can view or hide a server summary of server status for items in session (changed but not saved, out of date, locked by you, locked by another and recently changed).



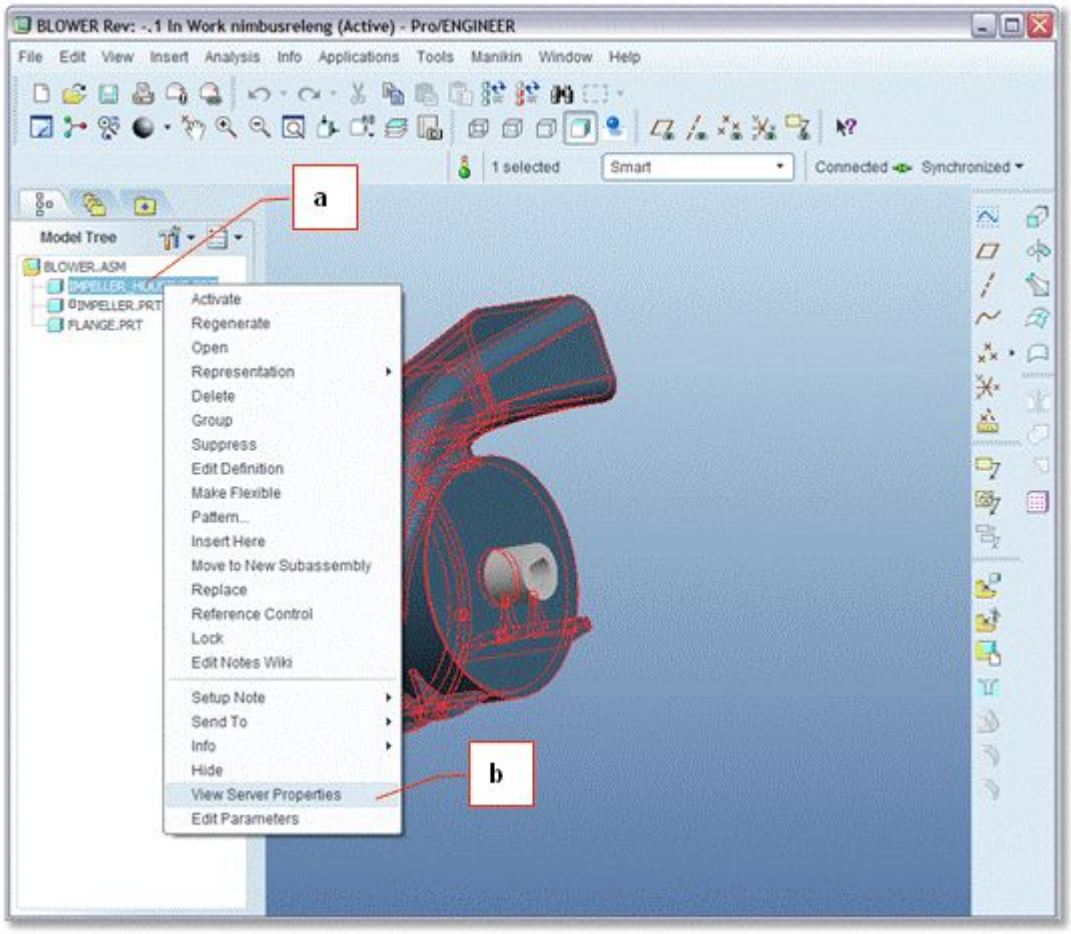

a—The **Server Quick View** is a separate dialog box that you can reposition and close, and you can choose how to filter and sort the status items.

b—The icons indicate the server status for items in session. If you move your cursor over the status icon, the help text displays the meaning of the icon.

c—If you close the **Server Quick View**, you can access the server menu to open it again.

## Accessing Server Properties for Model

If you are using Pro/ENGINEER Wildfire 5.0, you can quickly access the server properties for a model from the **File ▸ View Server Properties** menu for the active model or as a right-click action for an object in the model tree. This is useful to execute parts lists, where-used queries, and to view the version history.

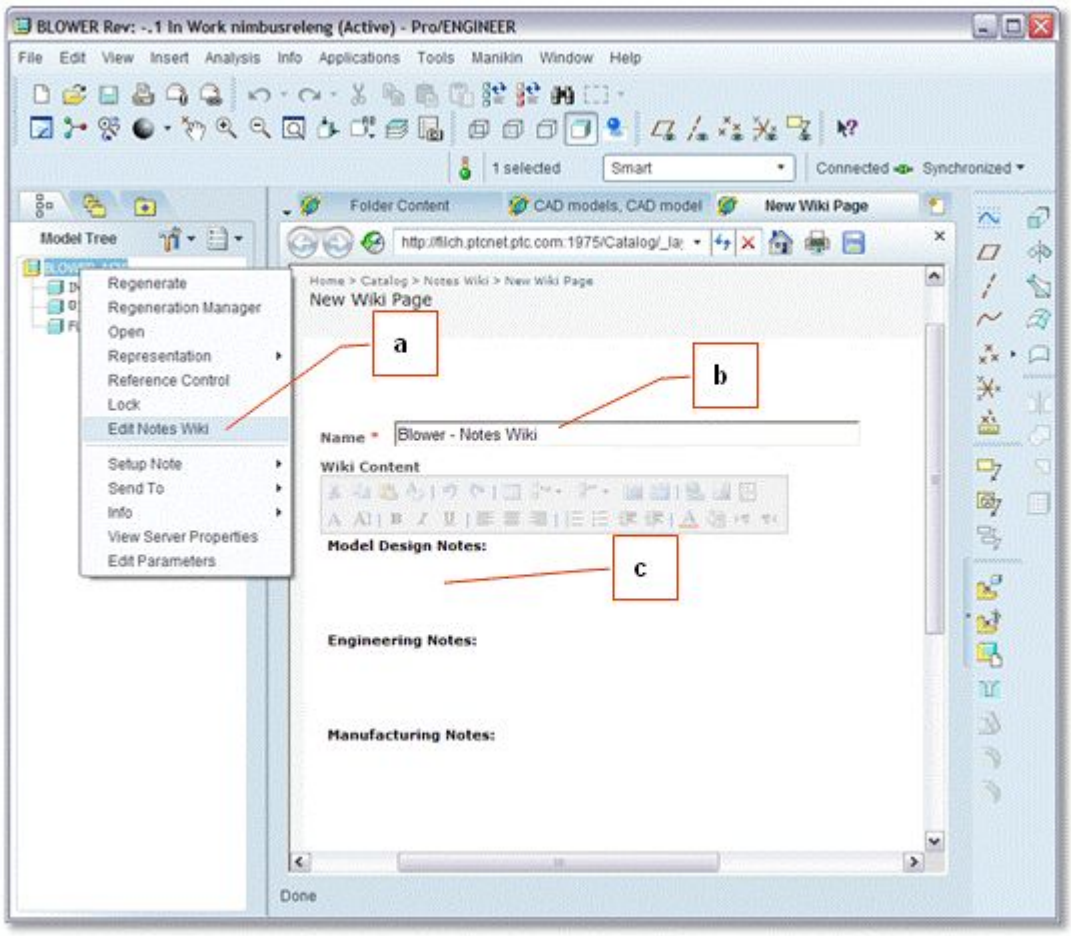



a—Right-click on model tree node of interest.

b—Select **View Server Properties** to display server properties in the embedded browser in Pro/ENGINEER.

# Editing and Viewing Notes Wiki

If you are using Pro/ENGINEER Wildfire 5.0, you can quickly create, edit, and view a Notes Wiki associated with the active model or an item in the model tree. Notes Wikis are intended to quickly capture design and manufacturing notes associated with a product item such as a part or an assembly. For example, you may wish to capture some modeling notes about how a design is parameterized and document the driving dimensions in a Wiki, so you or another designer can quickly understand how to properly make changes in the future. Administrators can also customize the Note Wiki template to include predefined text for categories and layout to help organize the information uniformly. The Notes Wiki template is accessible from **Home ▸ Sites** by choosing **Document Libraries** from the **View** list and then opening the **Notes Wiki Template Library** folder.




a—Select **Edit Notes Wiki** from the **File** menu for the active model or from the model tree item for which you want to view or edit model notes.

b—The Notes Wiki associated with the Pro/ENGINEER model is displayed in the embedded browser and is automatically given the model name with a Notes Wiki suffix. You can accept or enter a new name.

c—You can enter or paste in notes to describe the parameters that drive the model or any other notes in the major categories defined by the Notes Wiki template.

# Accessing an As-Stored Version of Assemblies

If you want to access an assembly with the exact versions of all the parts and subassemblies at the time the parent assembly was last saved, you can open an as-stored baseline from the server in Pro/ENGINEER. An As-Stored version of an assembly is automatically created each time you save an assembly in Pro/ENGINEER.

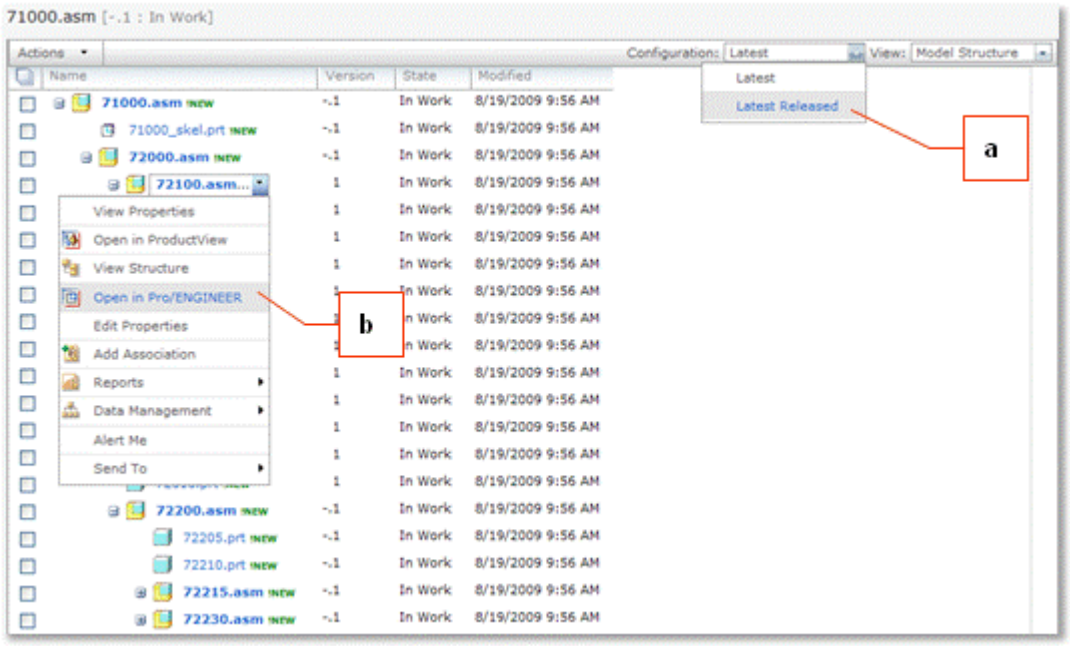
1. On the server, click **Version History** for the item to view prior versions.

2. From the version history page, click **Open in Pro/ENGINEER** to access the As-Stored version's date and time. The As-Stored version of the assembly that includes the exact versions of all parts and sub assemblies at the time it was saved is downloaded in Pro/ENGINEER.
3. Verify that the downloaded and opened version of the assembly is the same as the version selected from the version history. The version is listed in the menu bar of the Pro/ENGINEER screen.

## Accessing the Latest Released Version of an Assembly

You can access the latest released configuration of an assembly and open it in Pro/ENGINEER as follows:

1. In the browser, navigate to the assembly.
2. Select **View Structure**.
3. In the structure view header bar, select **Configuration** and then **Latest Released**.
4. Select **Open in Pro/ENGINEER** to open the latest released version of the assembly.




a—Browse to assembly structure of interest and select the **Latest Released** configuration.

b—Select **Open in Pro/ENGINEER** to open the latest released configuration of the assembly in Pro/ENGINEER.

# Working Offline

Windchill ProductPoint does not provide support for working on server-managed files in Pro/ENGINEER when you are disconnected from the network.

If you must work on server managed files while disconnected from the server or wish to send server-managed files to a design partner so the partner can work on the files independently, you should carefully follow the prescribed procedure. This approach is not recommended, but may be necessary in some cases if there is no other alternative.

The next procedure describes the best practices for working on server-managed Pro/ENGINEER files while disconnected from the server:

1. Lock items of interest on the server to prevent conflicts with changes by others.

   ***Note***

   *In Windchill ProductPoint 1.0, you cannot lock a structure of items or select multiple items to lock, so you must manually lock items one at a time.*

2. In Pro/ENGINEER, open all the items from the server on which you want to work offline.
3. Select **File** ▸ **Back up** in Pro/ENGINEER to back up all items in sessions to a file system.
4. In the Pro/ENGINEER navigator, right-click the connected, primary Windchill ProductPoint server and deselect the server, so that Pro/ENGINEER it is now connected to the file system.

   Click **Set As Primary** and make the file system the primary server.
5. Click **OK** to erase all server copies in session if you want to stay connected to the network but want to work with the backed up copies of the server-managed files.

   You can also disconnect from the network, and Pro/ENGINEER connects to the client file system by default.
6. Shut down Pro/ENGINEER or disconnect from the network (or just erase all the items in session in Pro/ENGINEER and open the files again from the file system backup folder).
7. Make changes to the items as desired in Pro/ENGINEER. You can do this either connected or disconnected from the network, but Pro/ENGINEER must be connected with the file system.

8. You can create and assemble new parts, if necessary, but do not use names for the new parts that exist on the server.

   ***Note***

   *If you create a new part of the same name as an existing part on the server (but which is actually a different part), the existing part is iterated and replaced with the new part of the same name. This can cause a regeneration problem in any assemblies that use the part or subassembly.*

9. When you are ready to save the changes back to the server and are connected to the network, make the Windchill ProductPoint server the primary server (this is only necessary if you previously changed the connection to your file system). You can do this in the Pro/ENGINEER navigator by right-clicking the Windchill ProductPoint server on the common folders and selecting the registered Windchill ProductPoint server that you want to make the primary server.
10. If there are models from the server that you downloaded because you needed them as required dependents, but you did not change the models (or it you do not have privileges to change these models on the server, or if you do not want to iterate these models on the server), you must first open all these models in session in Pro/ENGINEER from the server before opening items from the backup location on your file system.
11. Open the top-level assemblies and other parts from your backup file system folder that you modified.
12. Click **Save** in Pro/ENGINEER to save all the changes to the Windchill ProductPoint server.
13. Select the option to continue the save and override the conflicts. The system uploads the files and creates new versions for all saved models of the same name on the server. Any new parts are also saved.

# 3 Setting Up Attributes



This chapter describes how to exchange attributes between Pro/ENGINEER and Windchill ProductPoint.

# Exchanging Parameters Between Pro/ENGINEER and Server

You can configure your Pro/ENGINEER start parts to support the exchange of parameter values with the server.

1. Define Pro/ENGINEER parameter string names that match (case insensitive) the names of attributes you want to make available within Pro/ENGINEER. For example, you may want to make a cost value for a part that is managed as a site column in Windchill ProductPoint to be visible from within Pro/ENGINEER or to include it in a drawing cell.
2. Designate the parameter in Pro/ENGINEER to identify it for exchange with the server. In order for an attribute to be shared, it must be created in the Pro/ENGINEER file and designated. Any modifications made in Pro/ENGINEER are visible in the server column after the data is saved to a server.

You must manually update existing parts to define and designate the parameters that map to server attributes.

All non-system attributes are bidirectionally exchanged with Pro/ENGINEER. Because you cannot specify which parameters can only be set by Pro/ENGINEER or only by the server, it is best to avoid defining and designating parameters in Pro/ENGINEER which cannot be modified by the server. It is possible to define parameters in Pro/ENGINEER so that they can drive geometric shape generation when they are set on the server.

Designated Pro/ENGINEER parameters whose names exactly match site column names on the server are only exchanged if the types of the parameters are identical. For example, a parameter named COST is defined as a string type in Pro/ENGINEER, and the site column on the server named COST is a number type, therefore the values are not exchanged.

The following preconditions must be met before Pro/ENGINEER parameters and Windchill ProductPoint attributes (columns) can be exchanged:

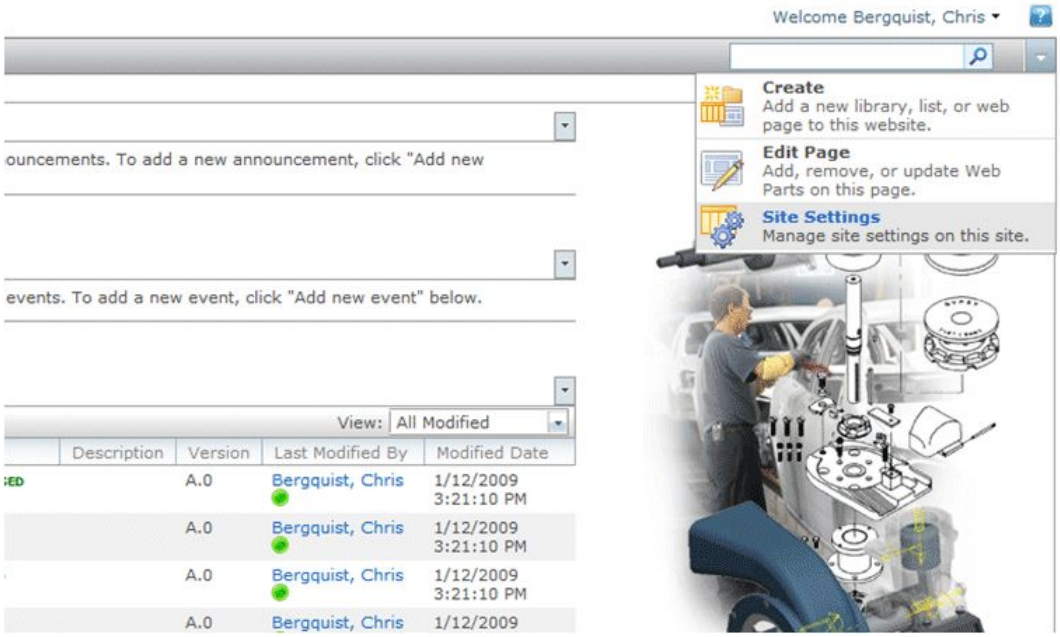
- The server content type that corresponds to the Pro/ENGINEER file types must be defined to include the site column to exchange with a Pro/ENGINEER parameter. For example, if you want to exchange a Site column named Job Number with all Pro/ENGINEER file types, the site column Job Number must be associated to the parent content type of PTC Content and the option to inherit the site column by all child content types must be selected.
- The Pro/ENGINEER parameter name must match the corresponding Windchill ProductPoint server site column value to exchange.
- The Pro/ENGINEER parameter must be designated (the designated check box must be checked in the Pro/ENGINEER parameter dialog box).

- The Pro/ENGINEER parameter type must match the corresponding Windchill ProductPoint server site column value to exchange.

The Pro/ENGINEER types and the server types are mapped as follows:

| Pro/ENGINEER Parameter Type | Server Site Column Type |
|---|---|
| String | Single line of text<br>Multiple lines of text |
| Yes/No | Yes/No |
| Integer | Number (where the number of decimal places is set equal to zero) |
| Real | Number (where the number of decimal places is non-zero or set to the value “automatic”) |

1. Define the site column that represents the attribute that you want to manage on the server by selecting **Settings Action ▸ Site Settings** from the home page.




2. Under the **Galleries** column, click **Site columns** to define a custom column attribute.
3. Click **Create** to define a new site column attribute.
4. Enter a new label for the site column attribute in the **Column name** field.
5. Select the site column type.
6. Complete the definition for a new site column by entering the number of decimal places.

For real numbers, choose the type **Number** and enter a non-zero number of decimal places. For integers, choose the type **Number** and specify the number of decimal places as zero.

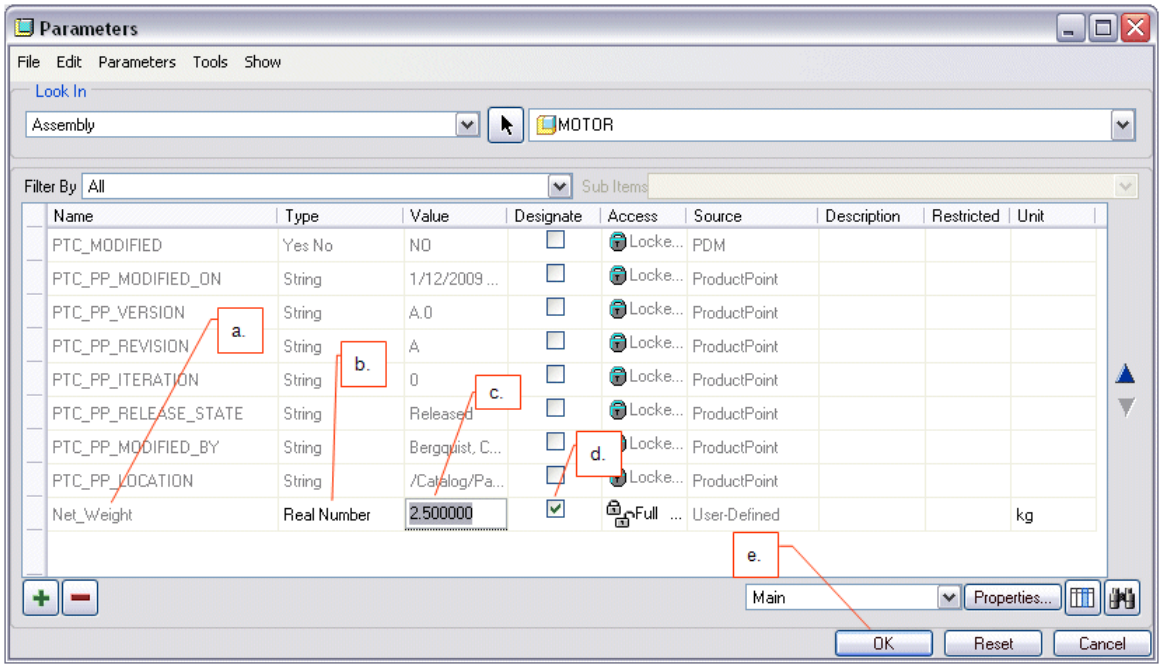
7. Click **OK** to complete the definition. The new custom site column is added to the **Custom Columns** group.
8. Navigate to **Settings Action** ▸ **Site Settings** and click **Site Content Types** to associate a new column with specific content types (such as part, assembly, or drawing).
9. Scroll down to **Product Item Content Types**. These are the content types defined by Windchill ProductPoint.
10. Click **Assembly** to edit this content type.

    ***Note***

    *The PTC Content is the parent type of all the product item content types that Windchill ProductPoint adds to SharePoint. If you add a site column to the PTC Content type, all of the following child types will automatically inherit the column you add:*

    - *General*
    - *Assembly*
    - *Part*
    - *Drawing*
    - *Format*
    - *Representation*

11. Click **Add from existing site columns** to add new column attributes to assembly content type.
12. Add new columns in the new window:
    a. In the **Select columns from** drop-down menu, select the group where you added the site column.
    b. Under **Available columns**, scroll to find the new content type or select the value and then enter the first character of column name to jump by character index.
    c. Click **Add** to add the selected columns to content type.
    d. When you are finished adding columns, click **OK**.
    e. Check the **Name** column to verify that the new column was added to the list for content type. You can modify the column order by clicking **Column order**.
13. Select **Tools** ▸ **Parameters** to define the new parameter in Pro/ENGINEER.
14. Click **Edit** to define new parameters or to update existing parameters.
15. Click the **Add** icon to add a row to define the new parameter.
16. Define the new parameter type and values.




a. Enter a parameter name using the same string defined on the server but substituting an underscore character for any spaces.

b. Specify the type. Use real number to map to server number type with more than zero decimal places.

c. Specify value.

d. Designate parameter for exchange with Windchill ProductPoint server.

e. Click **OK** to complete the parameter definition.

17. Save the model file to the server.

### *Note*

*The value defined in the Pro/ENGINEER assembly file is visible on assembly properties page following a save to the server. It is found in the* ***Net Weight*** *description.*

18. Modify the attribute value on the server to see the change in Pro/ENGINEER.

    a. Click the **Edit Properties** tab.

    b. Enter the new value for parameters in the **Net Weight** field.

    c. Click **Save**.

19. Select **File** ▸ **Update** ▸ **All** to replace out-of-date version in session with latest version from server with updated parameters.

20. Verify that the new value entered on the server is reflected in assembly parameters following **File** ▸ **Update** ▸ **All** action in Pro/ENGINEER.

### Using the Number Type Attribute

Consider the following information about mapping the Number type attribute:

- Real Number— The Number type attribute (the decimal places option set to Automatic or greater than zero) in Windchill ProductPoint maps to a Real number type parameter in Pro/ENGINEER if its name matches with the attribute defined on server.
- Integer—The Number type attribute (the decimal places option set to zero) in Windchill ProductPoint maps to an Integer type parameter in Pro/ENGINEER if its name matches with the attribute defined on server.

## Windchill ProductPoint System Attributes in Pro/ENGINEER

Several server system attributes are always exchanged with Pro/ENGINEER files. They are managed by the server and are read-only in Pro/ENGINEER. These attributes are useful for including in drawing title blocks to identify drawing or model versions, release status, and last modified dates. They are fixed by the server and cannot be extended.

| Attribute | Description |
|---|---|
| PTC_MODIFIED (Modified) | The boolean parameter that identifies whether the item in session was modified since it was loaded into session |
| PTC_PP_REVISION (Revision) | The revision label. For example: “A” of version “A.5” |
| PTC_PP_ITERATION (Iteration) | The iteration number; for example “5” for version “A.5” |
| PTC_PP_VERSION (Version) | The combination of revision and iteration labels “A.5” |
| PTC_PP_RELEASE_STATE (State) | Release state or status of an item. Release state can have one of two values: In Work, or Released. The release state is independent of the lifecycle state. |
| PTC_PP_RELEASED_ON (Released) | The date and time when the item was released. If it is blank, the item has not yet been released |
| PTC_PP_LOCATION (Location) | This is the path to where the item is defined on the server and should include the server name/site name/library name/folder name/subfolder name/object name |
| PTC_PP_CREATED_BY (Created by) | Full name of user who created item |

| | |
|---|---|
| PTC_PP_CREATED_ON (Created on) | The date and time at which item was created |
| PTC_PP_MODIFIED_BY (Modified by) | Full name of user who last modified item |
| PTC_PP_MODIFIED_ON (Modified on) | The date and time at which item was last modified |

System or custom attributes can be included in cells in a drawing table (such as in the title block of a drawing format). Use the following standard Pro/ENGINEER syntax:

```
&<parameter_name>:d
```

The `:d` attached to the parameter name specifies that the displayed value belongs to the drawing and not the model. For example, to display the drawing revision label, the following text can be added to a table cell:

```
&PTC_PP_REVISION:d
```

The system attributes managed by Windchill ProductPoint are accessible through the Pro/ENGINEER **Parameters** dialog box.

# 4 Useful Config.pro Settings

The following config.pro settings are useful when using Pro/ENGINEER with Windchill ProductPoint:

**generate_viewable_on_save**
When set to `yes`, Pro/ENGINEER generates 3D viewables in the ProductView format that can be viewed in ProductView.

**save_bitmap**
When set to `alllevels`, Pro/ENGINEER generates 2D thumbnail images for all levels of an assembly structure on save.

**save_bitmap_type**
When set to `jpeg`, Pro/ENGINEER generates 2D thumbnail images of models in JPEG file format.

**intf_out_pvs_recipe_file**
You can use this option instead of the bitmap generation configuration options `save_bitmap` (set to `alllevels`) and `save_bitmap_type` (set to `jpeg`) to provide more control over a thumbnail generation.

Specify the recipe file that contains instructions for the generation of viewables and thumbnails. The recipe file is either located at`apps\prodview\recipe\productpoint.rcp` if it is a client installation, or at `$PTCSRC\apps\prodview\recipe\productpoint.rcp` for the system area.

**let_proe_rename_pdm_objects**
If set to `yes`, Pro/ENGINEER display **File ▸ Rename** when connected to a server.

**dm_fallback_server_location**

Specifies a server save location for items such as user-defined features where Pro/ENGINEER prompts for a save location. The protocol WPP represents a Windchill ProductPoint Sever. If this config.pro option is not set, items are saved to wpp://<server name>/Catalog/Parts/ProENGINEER Work.

**pro_format_dir**

Specifies the server location for referencing drawing format files on open. This does not affect the save location for format files.

# 5 Troubleshooting



This chapter covers potential problems that may occur and how to identify and resolve them.

# Problem: Content is not Synchronized

Content will not synchronize with server even after selecting the action **Synchronize Now**.

## Potential Solution

If a problem occurs when uploading one of the files that was previously saved to your cache, you may need to clear your client cache. This step should be taken only as a last resort, as it can result in two potential problems:

- You can inadvertently delete changes that have been successfully saved to your client cache but have not yet been uploaded to the server (not synchronized with the server). If you know the files that you saved to the cache but which cannot be uploaded to the server, you can do the following:

  1. Open all saved but unsynchronized models in Pro/ENGINEER (make sure to do this while you are connected to the server).
  2. Use **File** ▸ **Backup** to back up the files to a file system location and shut down Pro/ENGINEER.
  3. Follow the steps below to delete your Windchill ProductPoint server cache.
  4. Open the files you just saved to the file system backup location in Pro/ENGINEER.
  5. Save the files to the server and select the option to override the conflict.
  6. Verify that the automatically applied lock on the files was automatically released by the server upon the save.

- Items on the server will be left locked for those models for which you have saved changes to the cache but where the modified files in the cache have not yet been uploaded to the server.

  You can delete your client cache through the following steps:

  To delete the cache:

  1. Synchronize all saved changes to the server (if possible).

     This ensures that any changes that you saved to the cache, but which are not yet uploaded to the server, are completely pushed to the server. If you have set the synchronization option to synchronize manually, and your files are not synchronized, click **Synchronize Now** to complete the synchronization.
  2. Shut down Pro/ENGINEER so that none of the cached files are in use in Pro/ENGINEER.
  3. Shut down the Client Manager.

     Shut down Client Manager to eliminate any synchronization activity against the cache. You can shut down the Client Manager by right-clicking

on the Client Manager icon and clicking **Exit** (this is the only action that is supported).

4. Verify that the client agent service is not running.

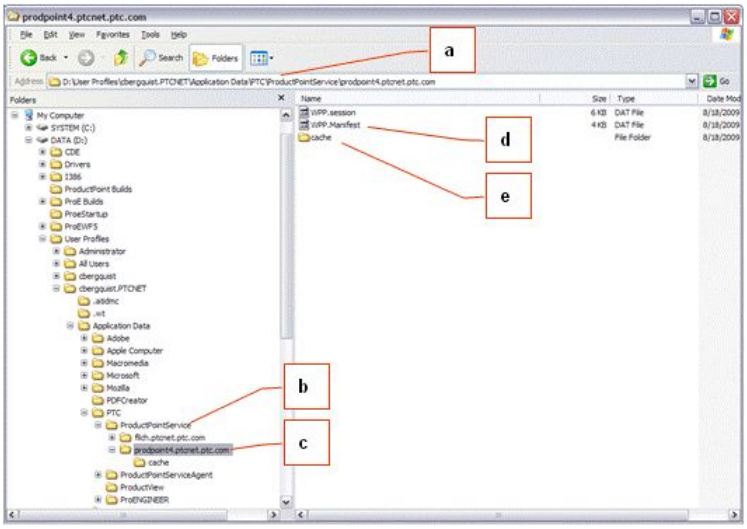
   In the task manager, make sure that the client agent is not running (the Client Manager normally shuts down the client agent service).

5. Locate the cache folder on your client file system.

   A unique cache folder is associated with each user for each registered server on the client machine. You must first locate the correct cache to delete. In Windows XP if you did not specify the cache location using the PTC_WF_ROOT variable, the cache is located here:

   ```
   D:\User Profiles\<user name>\ApplicationData\PTC\
   ProductPointService\<fully qualified server
   name>\cache
   ```

   ```
   D:\UserProfiles\cbergquist.PTCNET\ApplicationData\
   PTC\
   ProductPointService\malfoy.ptcnet.ptc.com\cache
   ```




a—Typical path to the server cache on Windows XP.

b—In this example, there are two registered Windchill ProductPoint servers under the ProductPoint Service directory.

c—The Windchill ProductPoint cache content for a particular server is identified by the server URL

d—The manifest file identifies packages of files that are ready to e sent to the server for synchronization.

e—The cache folder contains all downloaded and modified files from the Windchill ProductPoint server with which it is associated. The folder contains files with hashed names. Individual files should not be deleted from this folder.

6. Delete the following from the server directory:
   - `WPP.Manifest`—A list of unsynchronized data packages.
   - `cache`— The folder containing all cached Pro/ENGINEER model file versions.

   ***Caution***

   *Do not delete the `WPP.Session` if you want to save any client settings (such as your synchronization mode).*
7. Restart the Client Manager.

   Select **Start ▸ All Programs ▸ Startup ▸ Windchill ProductPoint Client manager** to restart the Client Manager.
8. Restart Pro/ENGINEER.

***Note***

*If the files have not been previously uploaded to the server, you lose all the saved changes to any files in the cache that you deleted.*

# Problem: Error During a Synchronization Requires a Manual Upload of Problem Files

In some cases, an error that occurs during a synchronization requires that you manually synchronize the offending files, which you have already saved.

If any of your files could not be synchronized, Pro/ENGINEER does the following:

1. Creates a folder to hold your saved but unsynchronized changes.
2. Converts the cached model files that were not synchronized back to their standard Windows file names from the hash file names used in the cache folder.
3. A balloon warning appears above the Client Manager system tray, notifying you that you have unsynchronized changes associated with data packages.
4. When you click on the warning balloon, Pro/ENGINEER displays in your browser a page that lists each file name that was not synchronized and indicates the client file system location for the unsynchronized files. The files are saved to a folder named `UnsynchronizedFiles` under the associated server folder. On a Windows XP system, the `UnsynchronizedFiles` folder can be found in a path such as:

`D:\UserProfiles\`*`user_name.Domain_name`*`\ApplicationData\PTC\ProductPointService\`*`ProductPoint_server_path`*`\UnsynchronizedFiles`

### Potential Solution

You can recover your unsynchronized changes by following these steps:

1. Make sure you are connected to the upgraded server as your primary server.
2. Navigate to and open the `SynchronizedFiles` folder from Pro/ENGINEER.
3. Open each top-level model file in Pro/ENGINEER and save the unsynchronized files to the server. Pro/ENGINEER displays a conflict dialog box, warning you that the files are already saved to the server. If you know that you have the latest versions in your cache at the time when the server was upgraded, select the option to overwrite the server version with the versions in your Pro/ENGINEER session. Perform this step for each assembly and each part in the `UnsynchronizedFiles` folder until you have saved all files in the folder to the upgraded server.

## Problem: Cannot Open an Item from the Server

You cannot select the browser action **Open in Pro/ENGINEER** to open a model from the server.

### Causes and Resolutions

- By default, if Pro/ENGINEER is not already running, the system uses the latest installed version of Pro/ENGINEER on the client machine. If the latest version of Pro/ENGINEER installed is not supported by Windchill ProductPoint, then you cannot select **Open in Pro/ENGINEER**. To avoid this problem, start the proper version of Pro/ENGINEER.
- You cannot select **Open in Pro/ENGINEER** if there is a space in the folder path in which you installed Pro/ENGINEER, for example, if you installed Pro/ENGINEER in the folder Program Files.
- If Pro/ENGINEER Wildfire 4.0 M060 or a later maintenance version is not installed, you cannot select **Open in Pro/ENGINEER**. Make sure that you have installed a version of Pro/ENGINEER that is supported by your Windchill ProductPoint version.
- If Pro/ENGINEER is being executed from a network drive location and there is no local installation, the `WPP_PROE_PLATFORM_DIR` environment variable must be set to identify the network drive location from which to start Pro/ENGINEER. For example, set the value of the path to: `N:\proe\i486nt` where Pro/ENGINEER is installed in `N:\proe directory`.
- If multiple supported versions of Pro/ENGINEER are installed and you want to use a version other than the latest installed, set the

`WPP_PROE_PLATFORM_DIR` environment variable (with the value *path_of_proe_installation_till_platform*).

- These actions ensure that your config.pro is loaded while starting Pro/ENGINEER and opening a model from server page:
  - Copy the config.pro to the Client Manager installed directory.
  - Set the environment variable `HOME` to point to a directory containing config.pro.

# Recovering Saved but Unsynchronized Changes After a Server Upgrade

If the server that you registered in Pro/ENGINEER is upgraded and if at the time of upgrade you had saved changes but had not synchronized your changes with the server, you must manually synchronize your changes. The best practice is for your administrator to ensure that all unsynchronized changes are first synchronized prior to upgrading the server. If any of your files were not synchronized, then upon subsequent connection to the server, Pro/ENGINEER will do the following:

1. Create a folder to hold your saved but unsynchronized changes.
2. Convert the cached model files that were not synchronized back to their standard Windows file names from the hash file names used in the cache folder.
3. Notify you that you have unsynchronized changes. A balloon warning notice appears above the Client Manager system tray that displays the message, informing you that unsynchronized packages from previous release of Windchill ProductPoint server have been found.
4. When you click on the warning balloon, Pro/ENGINEER displays in your browser a page that lists each file name that was not synchronized and the client file system location for the unsynchronized files. The files are saved to a folder named `UnsynchronizedFiles` under the associated server folder. On a Windows XP system the `UnsynchronizedFiles` folder can be found in a path such as:

```
D:\User Profiles\Application
Data\PTC\ProductPointService\
UnsynchronizedFiles
```

To recover unsynchronized changes:

1. Make sure you are connected to the upgraded server as your primary server.
2. Navigate to and open the `SynchronizedFiles` folder from Pro/ENGINEER.
3. Open each top-level model file in Pro/ENGINEER and save the unsynchronized files to the server. Pro/ENGINEER displays a conflict dialog warning you that the files are already saved to the server. If you know that you had

the latest versions in your cache at the time the server was upgraded, then select the option to overwrite the server version with the versions in your Pro/ENGINEER session. Perform this step for each assembly and each part in the `UnsynchronizedFiles` folder until you have saved all files in the folder to the upgraded server.

# II

# Working with Mathcad

# 6

# Mathcad Client Setup



This chapter describes how to install and configure the client application for Windchill ProductPoint to support Mathcad.

# Setting Up the Client to Work with the Server

## Mathcad Version Support

Mathcad interaction with Windchill ProductPoint 1.1 requires Mathcad 14.0 M035 or greater. Platform support details are provided in the next section.

## Mathcad Client System Requirements

The table below outlines the hardware and software requirements for running the Mathcad client in conjunction with a Windchill ProductPoint server. For configuration questions, consult your system administrator.

### *Note*

*For further information about Mathcad system requirements, visit the PTC Mathcad home page at http://www.ptc.com/products/mathcad.*

| Description | Provider | Supported Versions | Notes |
|---|---|---|---|
| **Mathcad** | PTC | 14 M035 or later | |
| **Hardware** | Intel<br>AMD | 64 bit<br>32 bit<br>512 MB or more RAM<br>550 MB Available Disk space required for installation<br>700 MHz or faster CPU<br>Intel Pentium/ Xeon/Core Duo /Core 2 Duo family, or AMD Opteron family | |

| | | | |
|---|---|---|---|
| **Operating System** | Microsoft | Windows XP SP2 32 bit edition<br>Windows XP Professional SP2 64 bit edition<br>Windows Vista SP1 32 bit<br>Windows Vista SP1 64 bit | Windows Server 2008 Enterprise and Standard editions recommended. Datacenter, HPC and Foundation editions are targeted for non-typical uses in ProductPoint environments. |
| **.NET Framework** | Microsoft | .NET 3.5 SP1 | You must install Microsoft .NET 3.5 SP1 on your Mathcad client machine. This is required for communicating with the Windchill ProductPoint server. You can download .NET 3.5 SP1 from the following location:<br>http://www.microsoft.com/downloads/details.aspx?familyid=AB99342F-5D1A-413D-8319-81DA479AB0D7&displaylang=en |

## Installing the Client Manager

### *Note*

*During Mathcad 14 M035 installation, you are given the option to "install Windchill ProductPoint components." When this option is selected, the Client Manager is automatically installed as part of the Windchill ProductPoint components. This is the usual method of obtaining the Client Manager.*

*The second option is to download the Client Manager from a server, using the procedure that follows. You might perform these steps if you did not install the Client Manager during Mathcad 14 M035 installation, or if you want to update the Client Manager from a server with a more recent build.*

To install the client manager manually, follow these steps:

### Note

*You must install using an account with administrative privileges to the client machine. If you do not have administrative privileges, ask your administrator to install the client manager for you.*

1. Locate the link at the end of the error message. For example, http://servername.ptcnet.ptc.com:1975/downloads. Enter this link into your browser to download and install the client manager component (named ProductPointClientManager-en-US.msi) that is needed to communicate with a Windchill ProductPoint server.
2. Select the language you used when you installed Mathcad.
3. Select the link to download and install the Windchill ProductPoint client manager.
4. Click **Run** to start the download of the client manager.
5. Click **Run** to start the installation of the client manager.
6. Click **Next** to begin Client Manager install wizard.
7. Accept the defaults or browse to identify the installation location.
8. Click **Next** to continue with the installation.
9. After the **Windchill ProductPoint Client Manager** window displays that the manager was successfully installed, click **Close**.
10. After the installation is complete, select **Start ▸ All Programs ▸ Startup ▸ Windchill ProductPoint Client Manager** to start the client manager for the first time.

    The next time you start windows, the client manager will start automatically as long as you do not remove it from the startup menu.

    When the Windchill ProductPoint client manager is running, the Client Manager Icon  is added to the system tray. If this icon is not present, the client manager is not running and Mathcad will not be able to communicate with the server.
11. Return to the server manager window in Mathcad and click **Check** to verify that Mathcad can communicate with the server.
12. Click **OK** to complete the server registration.

    The first time you begin working in Mathcad, the client manager will download and install a client agent component that supports communication with each specific server. This step occurs only once for each Windchill ProductPoint server you register.
13. The Mathcad server manager window appears and shows the status for the registered server as “On Line.” The server name and URL location of the selected server displays at the bottom of the **Server Manager** window.

Click **Close** to exit the server registration.

14. The folder navigator in Mathcad shows the Windchill ProductPoint server you just registered. All available servers are displayed when you access **PTC Places** with the **File** ▸ **Open** command in Mathcad. You can navigate all available Windchill ProductPoint servers and their contents from this control point. You can also retrieve and save data to any of these server locations.

## Registering a Server in Mathcad

You must register each Windchill ProductPoint server you want to connect with in Mathcad.

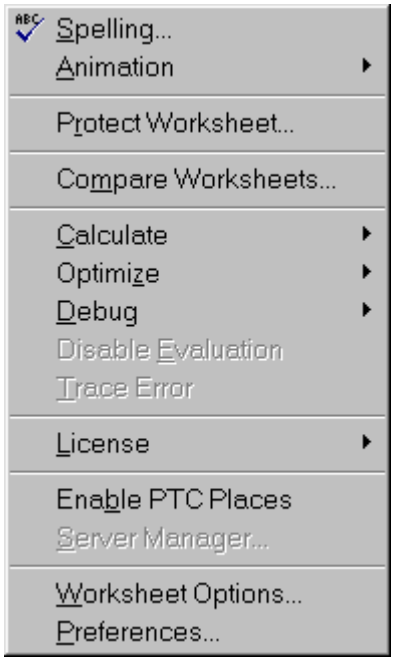
### *Note*

*If you have never previously registered a Windchill ProductPoint server, you must install a client manager component, as described above. Administrator privileges to the client machine are required to install the client manager. If you do not have administrative privileges, ask your administrator to install the client manager for you.*

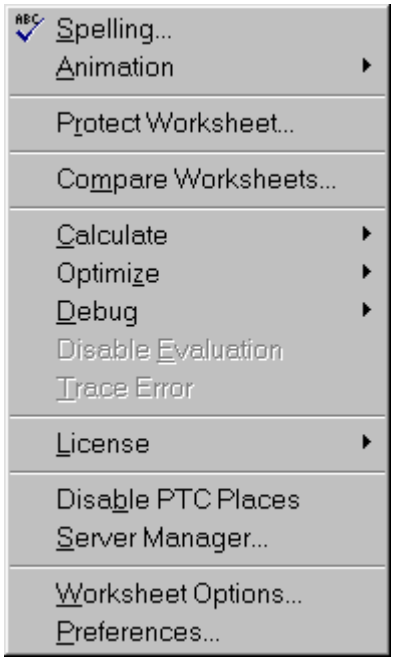
### *Note*

*When the Client Manager is installed from the server (i.e., not during the Mathcad installation), you must start it manually from the* ***Start*** ▸ ***All Programs*** ▸ ***Startup*** *menu.*

Once the Mathcad application is installed on your local machine, the **Tools** ▸ **Enable PTC Places** option is available on the Mathcad toolbar.



This option **must** be selected in order for you to work in the Windchill ProductPoint environment. Once **Tools ▸ Enable PTC Places** is selected, the **Server Manager** option is available on the **Tools** menu.




### *Note*

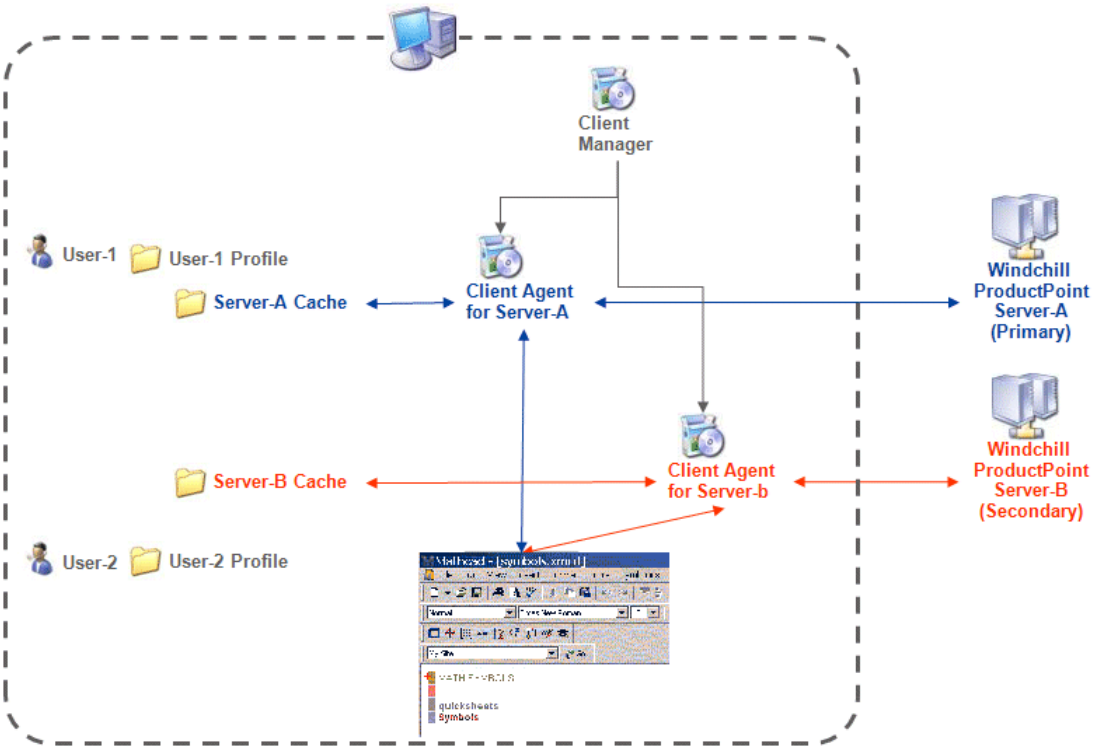
*When **Tools ▸ Enable PTC Places** is selected, its menu entry is replaced with **Disable PTC Places**. When **Disable PTC Places** is selected, you are unconnected to the Windchill ProductPoint server.*

To register a server in Mathcad, follow these steps:

1. First select **Tools ▸ Enable PTC Places**, and then access **Tools ▸ Server Manager**.
2. From the **Server Manager** window, select **Server ▸ Register New Server**.
3. Enter the Name and Location (URL address) of the server. You can use the address supplied by your administrator, or copy the address from any server page on the server in a browser and paste it in the **Location** field.
4. Click **Check** to test that Mathcad can find the server you identified.

# System Architecture

This diagram illustrates the organization and interactions of the client and server components with Mathcad.




These components interact as follows:

- The client manager is responsible for starting and updating the client agents.
- The client manager must be manually started the first time it is installed, unless Windchill ProductPoint components were installed during the Mathcad installation; in which case, the client manager will start automatically.

  The client manager starts automatically in the future each time Windows is started (this is accomplished by adding it to the Windows startup).
- If the client manager is not running, no communication between Mathcad and the server is possible.
- A separate client agent is associated with each user for each server that is registered and connected to Mathcad.
- The client manager handles all client agents for users by starting, stopping, and automatically updating the client agents as servers are connected and disconnected.
- A separate client cache is maintained for each registered server for each user. If one physical machine is used by several users that log on to the machine

under separate user names, each of these users has an independent client cache that is associated with them.

# 7

# Working with Mathcad Worksheets



This chapter describes the interaction between Mathcad and the Windchill ProductPoint server.

# Using Mathcad and ProductPoint to Manage Files

Several file management actions are used to manage your Mathcad data in Windchill ProductPoint. These actions are available from both Mathcad and the Windchill ProductPoint user interface.

When the authoring application is connected to Windchill ProductPoint, some actions relating Windchill ProductPoint connectivity become active in Mathcad, such as **Enable PTC Places** and **Server Manager**.

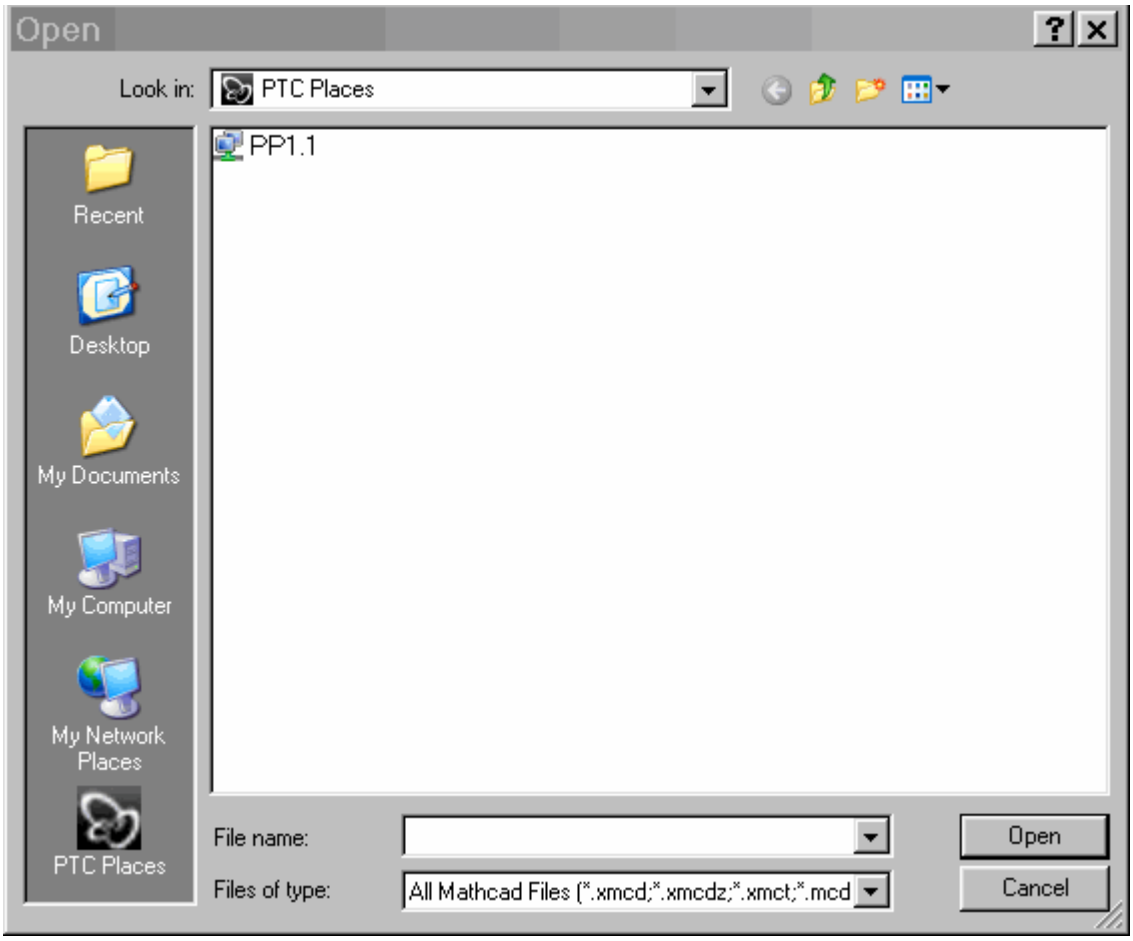
***Note***

*Windchill ProductPoint changes the behavior of some of the native commands in your authoring application, such as* ***Save*** *or* ***Open****. For example, when Mathcad is connected to Windchill ProductPoint, the* ***Open*** *and* ***Save*** *operations will default to* ***PTC Places****, if the worksheet has been saved to or opened from the local disk.*

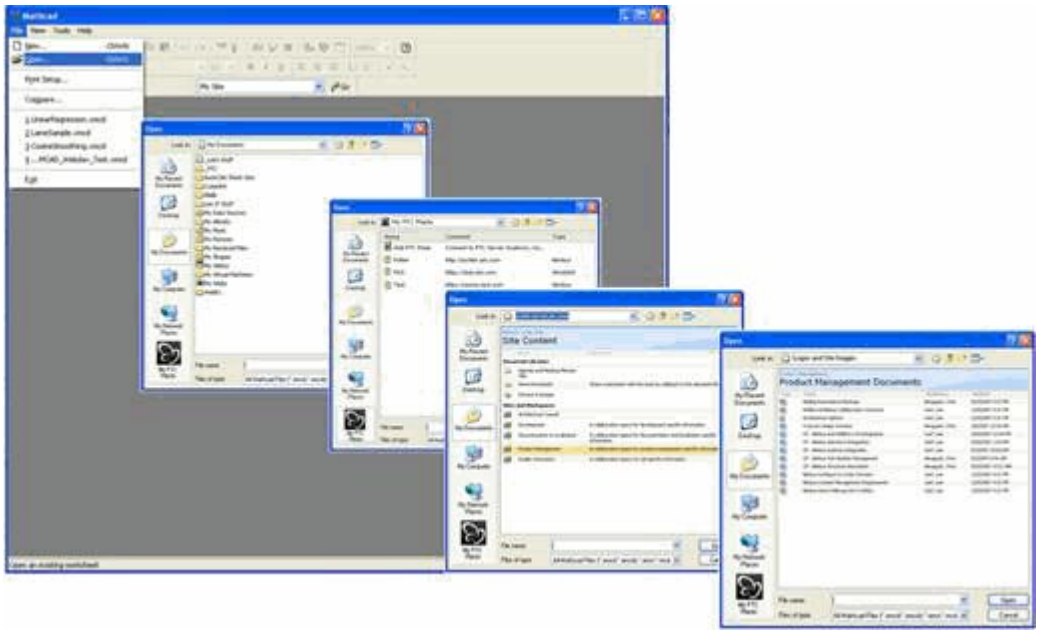
## Using File > Open

When connected to a Windchill ProductPoint server, the **File ▸ Open** command in Mathcad allows you to open a worksheet stored on the Windchill ProductPoint server.

1. In Mathcad, select **File ▸ Open**. Mathcad displays the **Open** dialog box. The system will default to **PTC Places** at the top level, unless the worksheet has been saved to or opened from the local disk.




2. You can navigate to the desired location by selecting “PTC Places” and navigating to the **Server** ▸ **Site** ▸ **Library** ▸ **Folder** ▸ **File** to select a Mathcad file to open.



3. When the file is located and **Open** is selected, Windchill ProductPoint downloads the selected file to the local Windchill Windchill ProductPoint managed area.
4. When the worksheet opens, the server inserts any metadata changes made from the server, such as attribute modification. The file is now open and ready for modification.

## Using File > Save

Once you have opened a Mathcad file from the Windchill ProductPoint server and made changes to it, you can save the new version back to the server using the **File ▸ Save** command.

1. When a file is opened and modified in Mathcad, the system recognizes the file as modified, and adds a "+" in the Mathcad title in addition to other server attributes. The system then locks the file on the Windchill ProductPoint server. (See Lock Management of Mathcad Files Saved to Server on page 95 for more information about locked files.)
2. Once the modifications to the worksheet are finished, select **File ▸ Save**. Mathcad saves the file back to its original location, creates a new iteration of the worksheet, and updates the attributes shown in the Mathcad title. Then the system releases the lock on the server.

## File >Save As New Worksheets to Windchill Product Point

When you create a new Mathcad worksheet on your local device, you must save it to the Windchill ProductPoint server in order that other users can view and edit it.

1. When you have finished creating a locally stored Mathcad worksheet using **File ▸ New**, you select **File ▸ Save** to store the file. When a Windchill ProductPoint server is registered and active:
   - The **File ▸ Save As** dialog box appears when the file created is either a new worksheet, or a worksheet that has never been saved to a local disk;
   - The **File ▸ Save** dialog box appears if the file was retrieved from a local disk, or saved at least once to a local disk; this option completes the **Save** back to the local disk. In such a case, the **File ▸ Save As** dialog does not appear.
2. In the **Save As** dialog box, the top level of the registered server is presented as the default location, with PTC Places as the top entry.

3. Navigate to the desired location by selecting “PTC Places” and then navigate to the **Server** ▸ **Site** ▸ **Library** ▸ **Folder** to select a location to save the Mathcad file. Click **Save**.

   ***Note***

   *If you enter a name that already exists on the server, a warning dialog box appears and identifies the location of the worksheet with the same name on the server.*

4. The **Save As** function works similarly to the **File** ▸ **Save** function described previously. The system identifies and locates referenced files (dependencies), and converts to relative references if needed; collects worksheet metadata and checks for name conflicts; and then uploads the selected file to the server, and creates new items and place holders for references.

   ***Note***

   *References to other files are not created automatically; place holders created during **Save As** must be resolved later. See Incomplete Items on page 93 for more information.*

## Using File >Save As to Export to Local Directory

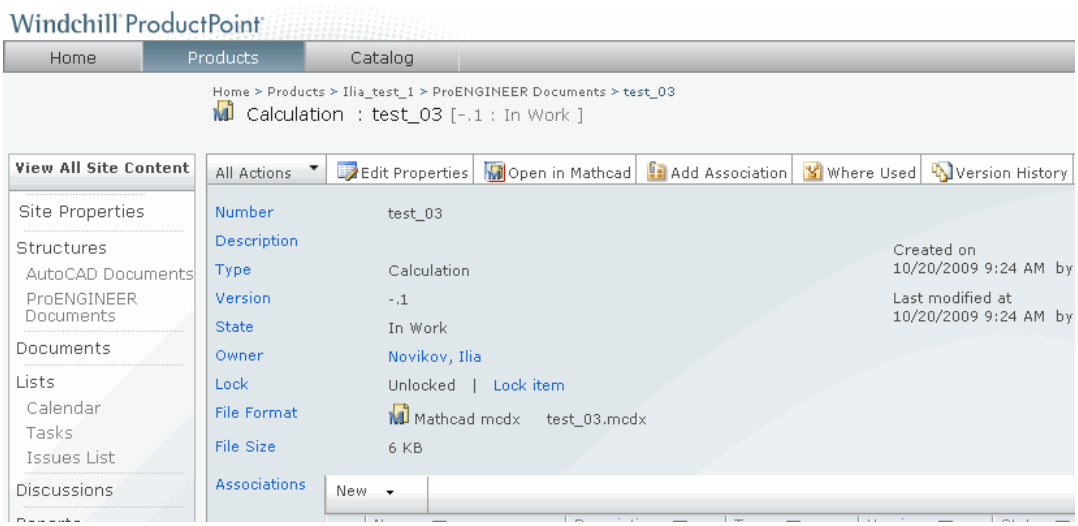
You can use **File** ▸ **Save As** to save a Mathcad worksheet presently stored on a Windchill ProductPoint server to a local directory, such as one on your computer hard drive.

1. In Mathcad, open a worksheet presently managed in Windchill ProductPoint, as described previously in Using File > Open on page 88.
2. Select **File** ▸ **Save As** and navigate to the local directory where you want to save the file. Click **Save**.
3. The system saves the worksheet to the local directory.

# View Properties

On the Windchill ProductPoint server, you can view the properties of a Mathcad worksheet using the **View Properties** function.

1. On the Windchill ProductPoint server, highlight the desired Mathcad worksheet, and select the item action **View Properties**.

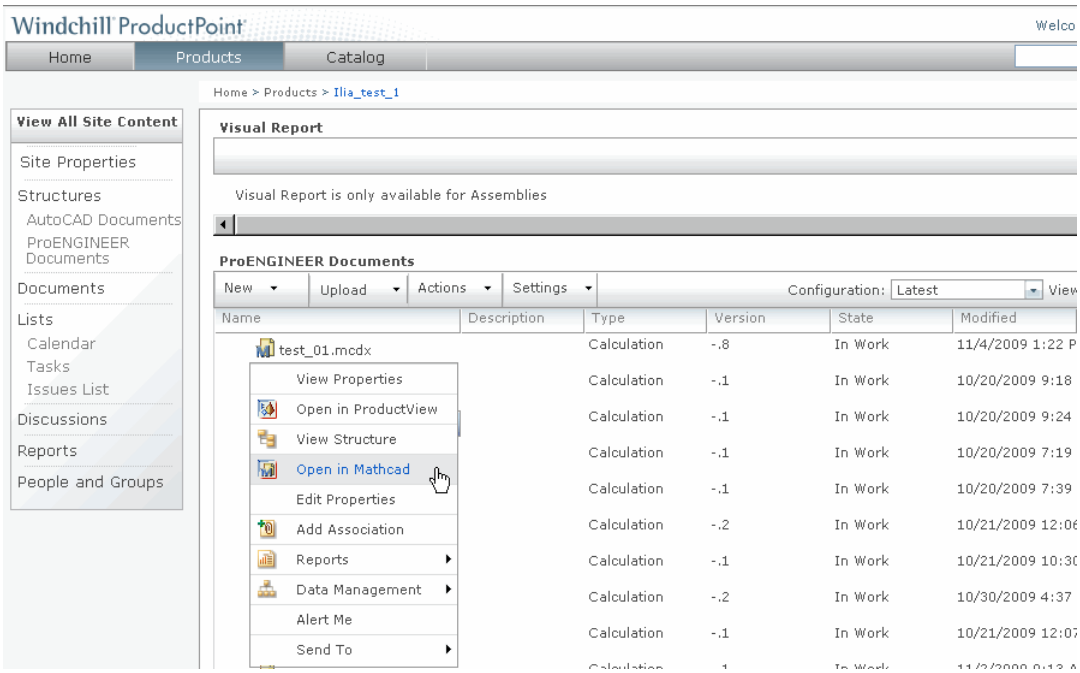



2. The system displays the properties page of the worksheet, which includes:
   - The predefined, but configurable set of properties
   - The defined top bar actions
   - A table of items associated with the file

## Open in Mathcad

You can edit a Mathcad worksheet on a Windchill ProductPoint server by using the **Open in Mathcad** option.

1. From Windchill ProductPoint, navigate in a browser to a Windchill ProductPoint Structure Library, and select the Mathcad worksheet you want to edit. Click on the arrow alongside the worksheet and choose **Open in Mathcad** from the drop-down list.




2. The server launches Mathcad and Windchill ProductPoint downloads the selected file, along with all referenced files, to the local Windchill Windchill ProductPoint managed area, and opens the file for editing.
3. Once the modifications to the worksheet are finished, select **File** ▸ **Save**. Mathcad saves the file back to its original location, and creates a new iteration of the worksheet.
4. When a **Save** is performed following an **Open in Mathcad**, the changed Mathcad worksheet is uploaded to the server, and the system releases the lock on the server.

# Incomplete Items

In Mathcad, if a worksheet is referencing other worksheet(s), and the reference in one of those worksheets was inserted from outside of Windchill ProductPoint, will create an incomplete item.

## Saving Worksheets with Incomplete Items

When you attempt to save a worksheet that contains a reference that does not exist on a Windchill ProductPoint server, a warning appears informing you that an incomplete item will be created for the missing reference. Click **OK** to proceed with the save.

For each reference that does not exist on the Windchill ProductPoint server, Windchill ProductPoint creates an incomplete item. In a structure view, incomplete items are identified by the  icon. When you mouse over the icon, a tooltip informs you that it is an “incomplete non-required item.”

You can obtain a **Where Used** report on incomplete items, allowing you to identify the worksheets where the item is used. This is described later in this topic, in the "Report on Incomplete Items" section.

### Releasing Worksheets with Incomplete Items

If incomplete items are created as a result of optional references not existing on a Windchill ProductPoint server, the parent worksheet can still be released. See the "Resolving Incomplete Items" section below for details.

### Working with Incomplete Items

Consider the following information when performing operations on incomplete items:

- When you create a copy of a parent worksheet (using **Save As** on the Windchill ProductPoint server) which contains an incomplete item, all of its items are duplicated, including placeholders for missing items. An incomplete item from the original worksheet is reused in the copied worksheet.
- The names of the incomplete items must be unique across sites, and their names cannot be used by other items.
- An item can not be renamed to the name of an incomplete item, and an item cannot be copied under the name of a incomplete item.

### Resolving Incomplete Items

After an incomplete item is saved, you can find a missing dependent and either upload it to Windchill ProductPoint to replace the placeholder, or you can retrieve the incomplete item in Mathcad and replace a placeholder with the required reference. Once the incomplete item is replaced with the actual reference, the reference is restored. The parent item does not get iterated or locked as a result.

### Report on Incomplete Items

You can obtain a report about incomplete items by choosing **Reports** from the quick-launch panel of a site or a site collection.

To generate a report:

1. On the left quick-launch panel, click **Reports**.
2. Click the link for the **Incomplete Items** report.
3. The system generates a report, displaying a **Where Used** summary of the incomplete items references by items managed in Windchill ProductPoint.
4. You can export the report to a spreadsheet by selecting **Export to SpreadSheet** located on the Windchill ProductPoint toolbar.

# Lock Management of Mathcad Files Saved to the Server

Windchill ProductPoint is designed to prevent overwriting changes by others. This is accomplished by exclusively locking files for one editing user at a time. Locks can be set automatically or manually. An Automatic Lock is released automatically after you make edits and save changes.

You can rely on the automatic lock behavior most of the time, but sometimes it is useful to lock items manually. For example, manually locking a file can be useful if you expect to make changes to a worksheet over a certain period of time, and want to reserve the exclusive right to change that file during that time period.

When you select an action in Mathcad to begin editing an object managed on the server, Mathcad will automatically lock the object on your behalf on the server. Although other users can access and edit a locked worksheet, their changes cannot be saved as long as the lock remains in place on the server. This prevents concurrent overwriting or conflicting changes to the same file.

The server displays the lock status in the properties pages, and in list and structure views that include the lock status column.

If you visit the properties page for the item you are modifying, it will indicate that it is locked by the system on your behalf. Other users can download items locked by others into a Mathcad session. However, if a save is attempted when the worksheet is out of date (because the other user saved the latest iteration of the worksheet on the server), the save will not proceed.

When you save and upload changed files to the server using automatic lock, the lock for the item is also released automatically. The lock state is automatically changed by the system on your behalf, from locked to unlocked. At this point another user can edit the worksheet and apply changes on top of your changes.

The automatic locks created by the system for objects on your behalf are automatically released when:

- You close the changed objects from session without saving the objects
- You restart Mathcad and the system identifies objects in your cache with automatic locks where your changes have not been synchronized to the server (these locks may occur in the event of a Mathcad crash where changes were started but not saved and completed).

Sometimes it is useful to make temporary changes to experiment with a potential change. Normally it is best to terminate the edit, and request that the locking user notify you when they are finished making their changes so that you can follow with the changes you need to make.

### Note

*For further information about conflicts and how to resolve them, refer to the Conflicts table in* *Troubleshooting on page 103*.

# Updating Out of Date Information in Mathcad

Sometimes changes are concurrently made by other users while you have an object in session in Mathcad. Other users may either change the objects through server operations, and thus modify the attributes, or change the worksheet content in Mathcad. Windchill ProductPoint does not allow you to save back to the server changes that you have made to worksheets in session which have been concurrently updated by others. This behavior is designed to prevent one user from inadvertently overwriting changes made by another user. To avoid this, it is recommended that you always work on the latest version of a file.

If you attempt to edit objects in session which have been modified by others since you opened them, Mathcad displays an out of date conflict warning. At this point, it is necessary to close and retrieve the worksheet again in order to access the most recent iteration.

# Using the Mathcad Analysis Feature with Pro/ENGINEER

Using the Mathcad Analysis feature, you can use Mathcad worksheets to perform external analysis on Pro/ENGINEER models within Windchill ProductPoint. You can define what analysis you want to perform on a Pro/ENGINEER model, and determine which parameters will be created as a result of the analysis.

To enable the Mathcad Analysis feature from within Pro/ENGINEER, click **Analysis** ▸ **External Analysis** ▸ **Mathcad Analysis**.

### Note

*A complete description of Mathcad Analysis, and how to use the Mathcad Analysis Feature, can be found in the Pro/ENGINEER HELP Center. From the top level of the HELP center (**Pro/ENGINEER Functional Areas**), navigate to **Model Analysis** ▸ **Behavioral Modeling Extension** ▸ **Behavioral Modeling Tools** ▸ **Analyses** ▸ **Mathcad Analysis and Mathcad Analysis Feature**.*

# 8

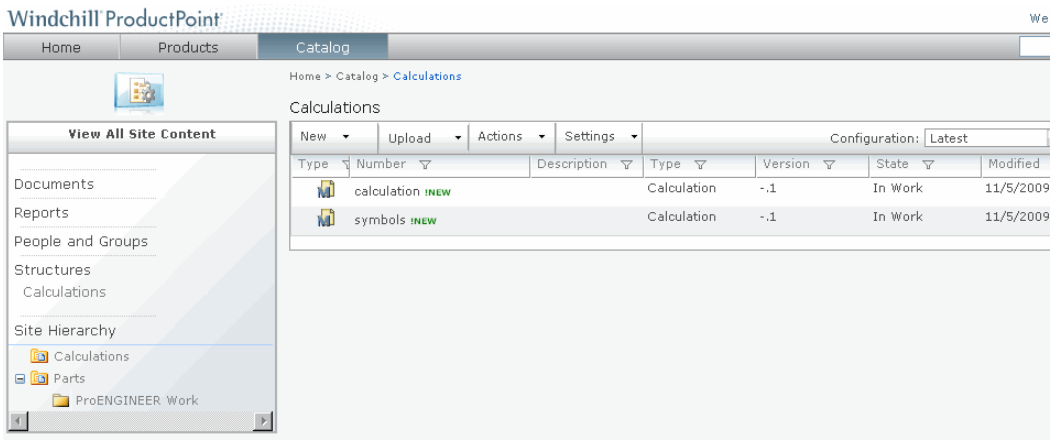
# Mathcad Calculations Structure Library



From the Windchill ProductPoint server, you can access a ProductPoint Calculations Structure Library, which is created by default on the server. You can also add a new Calculations Structure Library to an existing ProductPoint site.

# Locating and Viewing a Calculations Structure Library

1. On the Windchill ProductPoint server, navigate to the **Catalog** tab. The system displays the **Catalog** home site, which includes the default Calculations Structure Library.

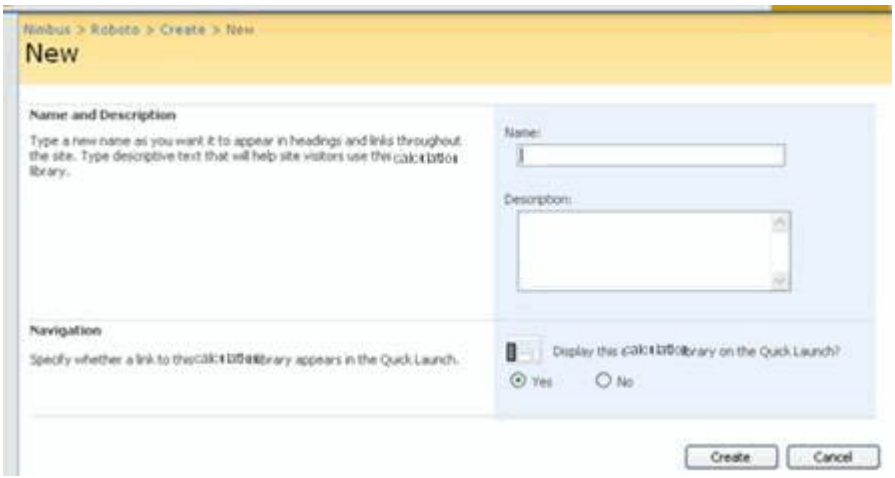



2. Click on the title of the desired Calculations Structure Library Web Part to display the structure library, which includes predefined views, properties, and content types or Mathcad worksheets

# Adding a Calculations Structure Library to an Existing Site

1. From a Product site location in a Windchill ProductPoint browser, select **Site Actions** ▸ **Create Action**. The system displays the **Create** page, which includes the option to create a Calculations Library.


2. Click on the Calculation Library link. Windchill ProductPoint displays the **Add** page. Enter the Name and Description of the new library, and whether you want a link to the library to appear in the Quick Launch.




3. When you are finished, click **Create**. Windchill ProductPoint creates the URL for the library automatically, based on the library name, and displays the newly created library.
4. Now you can navigate back to the home page for the site, and select the **Site Actions ▸ Edit** page. The newly created calculation library is available in the selection list when you add a structure library web part.

# 9 Attribute Communication

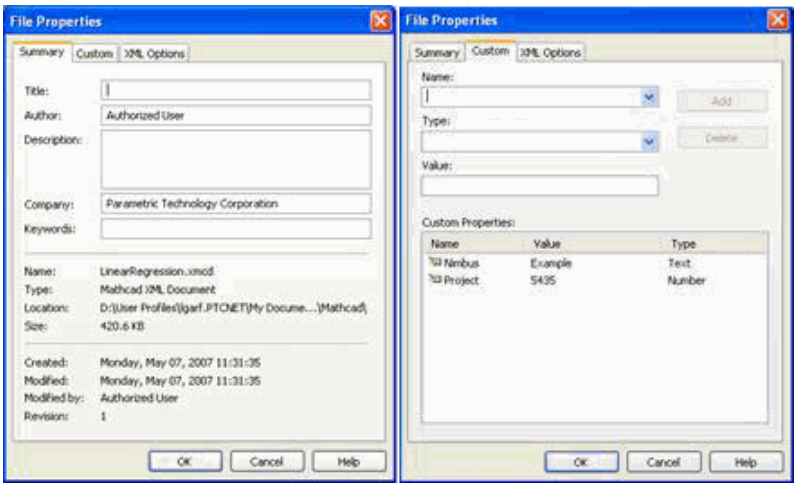
Mathcad custom parameters can be bi-directionally mapped to Windchill ProductPoint attributes. When mapping, the names of the parameters in Mathcad and Windchill ProductPoint must match each other. The supported parameter types are Text, Date, Number, and Yes/No. Within the Summary Pane, the following properties can also be mapped: Author, Description, Company, and Keywords.

Use the following procedure to define the custom attributes on a Windchill ProductPoint server.

1. Select an attribute, such as **Text**, for both Windchill ProductPoint and Mathcad.
2. At the Windchill ProductPoint **Home** level, navigate to **Site Settings** ▸ **Site Columns** and click the **Create** button. Enter a name for the attribute, select the type **Text**, and click **OK**.
3. At the Windchill ProductPoint **Home** level, navigate to **Site Settings** ▸ **Site Content Types** ▸ **Calculation** ▸ **Add** from the existing site columns, and add the attribute you created to the **Calculation** type.
4. Repeat steps 2 and 3 to define other attributes, such as Date, Number, or Yes/No.

To map attributes, perform the following steps:

1. Launch Mathcad and open an existing file or create a new file.
2. Select **File** ▸ **Properties** to display the **File Properties** dialog box.




3. Enter the desired Summary properties and Custom properties, and click **OK**. The system locks the file on the server on behalf of the user.
4. Select **File** ▸ **Save.** In addition to the usual **Save** function, Windchill ProductPoint will save both the Summary and Custom properties in the Mathcad file map to properties on the server, based on the name and type match.
5. Once **Save** is completed, Windchill ProductPoint releases the lock on the server.

### *Note*

*Attribute mapping is bi-directional, and changes made later to the file on the server will be propagated down to the Mathcad files. You must close the worksheet from your Mathcad session and retrieve it again in order to see the changes to the attributes on the server.*

# 10 Troubleshooting



This section covers potential problems that may occur and how to identify and resolve them.

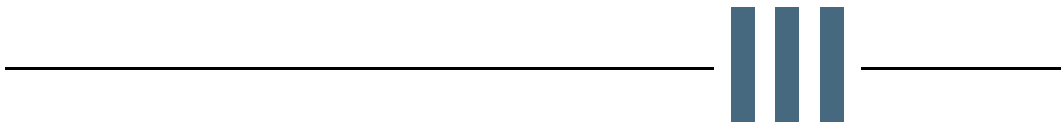
# Conflicts

At times, you may encounter conflicts when performing editing or save actions. The following table provides a list of conflicts, what they mean, and how to resolve them.

| Application Action | Conflict Type | Explanation | Resolution |
|---|---|---|---|
| Edit | The file is locked by someone else. | The file is being edited and is locked by another user. | Wait for the lock to be released. |
| Edit | The file is out of date. | The file is being edited, and the version you are attempting to edit is not the most current version. | Edit the most current version once it has been saved. |
| Edit | The file is read-only. | The file you are attempting to edit is read-only. | If you have write access to the file, remove the read-only designation from the file's properties and try again. |
| Save As | A file with the same name exists on the server. | There is an existing file on the server with the same name as the one you are trying to save. | There are two options.<br>1. Overwrite the existing file.<br>2. Rename the file to a nonexistent name and re-save. |
| Save As | A file with the same name exists as an incomplete (missing) reference. | There is an incomplete item, on the server with the same name as the one you are trying to save. | Overwrite the incomplete item.<br>**Note**<br>*Refer to Incomplete Items on page 93 before overwriting an incomplete reference.* |

| | | | |
|---|---|---|---|
| Save As | The save fails due to an access control on the target location. | You do not have permission to write to the desired target location. | Choose another location to save to, or see your system administrator to obtain the necessary permission. |
| Save and Save As | Dependents with the same name in a different location exists in the parent file. | The dependents with the same name will be ignored because they originate from different locations. | No action needed. |
| Save and Save As | A file with the same name exists for dependents. | An item with the same name as the dependent already exists on the server. | There are two options.<br>1. Reuse the name on the server, and change the path to a relative path.<br>2. Ignore the conflict, and leave the path as an absolute path. |
| Save and Save As | There is an incomplete reference for a dependent (there are no objects with the same name). | Since there is an incomplete reference for dependent(s), saving the file will create a placeholder object. | None. |
| Save and Save As | There is an incomplete reference for a dependent because the name is reserved. | The dependent with the name you have chosen cannot be created because the name is reserved by another user. | None. |

| | | | |
|---|---|---|---|
| Iterative Save | The file is locked by someone else. | The save fails because the file is locked by another user. | None. |
| Iterative Save | The file is out of date. | The save fails because the version you are attempting to save is not the most current version. | None. |



# Working with Autocad

# 11 Client Setup



Before you can start using AutoCAD with Windchill ProductPoint, you must set up the client machine to exchange information with the server.

# AutoCAD Client System Requirements

The table below outlines the hardware and software requirements for running the client in conjunction with a server. For configuration questions, consult your system administrator.

| Description | Provider | Supported Versions | Notes |
|---|---|---|---|
| AutoCAD | Autodesk | 2009 and later | |
| Hardware | Intel<br>AMD | 64 bit<br>32 bit<br>512 MB or more RAM<br>550 MB Available Disk space required for installation<br>700 MHz or faster CPU Intel Pentium/ Xeon/Core Duo /Core 2 Duo family, or AMD Opteron family | |
| Operating System | Microsoft | Windows XP SP2 32 bit edition<br>Windows XP Professional SP2 64 bit edition<br>Windows Vista SP1 32 bit<br>Windows Vista SP1 64 bit | Windows Server 2008 Enterprise and Standard editions recommended.<br>Datacenter, HPC and Foundation editions are targeted for non-typical uses in ProductPoint environments. |
| .NET Framework | Microsoft | .NET 3.5 SP1 | You must install Microsoft .NET 3.5 SP1 on your client machine. This is required for communicating with the server. You can download .NET |

| | | | |
|---|---|---|---|
| | | | 3.5 SP1 from the following location: http://www. microsoft.com/downloads/details.aspx?familyid=AB99342F-5D1A-413D-8319-81DA479AB0D7&displaylang=en |

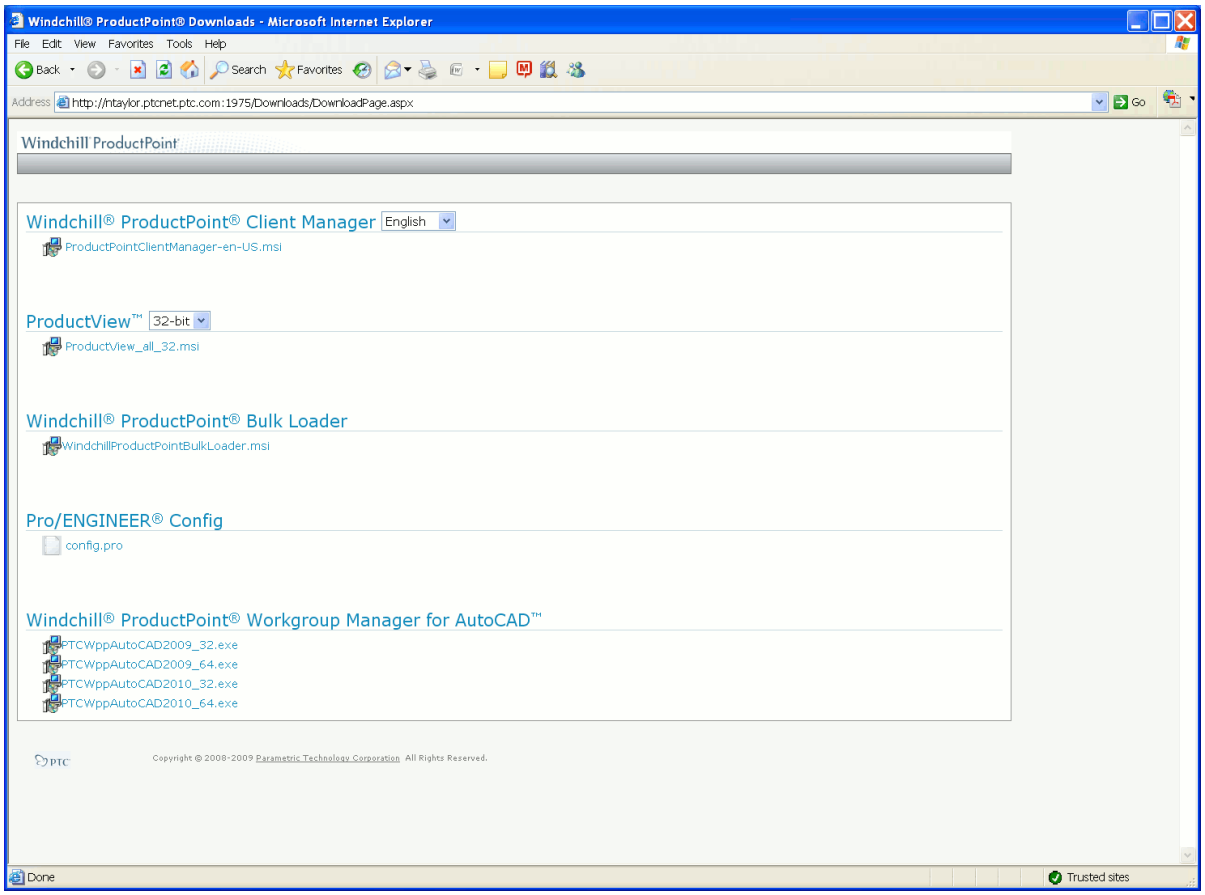
## AutoCAD Version Support

Windchill ProductPoint requires AutoCAD 2009 or later.

# Installing the Client Manager and Client File

### Note

*The Windchill Product Point integration to AutoCAD for PTC SharePoint Products installer should have been placed on the server by the WPP administrator, under the path <web server loadpoint>\Inetpub\wwwroot\ProductPointDownloads. This must be done prior to any attempt to download and install the AutoCAD client. If the Windchill Integration to AutoCAD for PTC SharePoint Products installer is not available on the Downloads page, contact your WPP administrator, or see the Windchill ProductPoint Installation, Administrator's and Upgrade Guide.*




To install the client manager manually, follow these steps:

### Note

*You must install using an account with administrative privileges to the client machine. If you do not have administrative privileges, ask your administrator to install the client manager for you.*

1. Open a web browser and go to the Downloads page, located at `http://<server address>/Downloads`.

2. Click on the appropriate installer file for your operating system and AutoCAD installation. It will be one of the following:

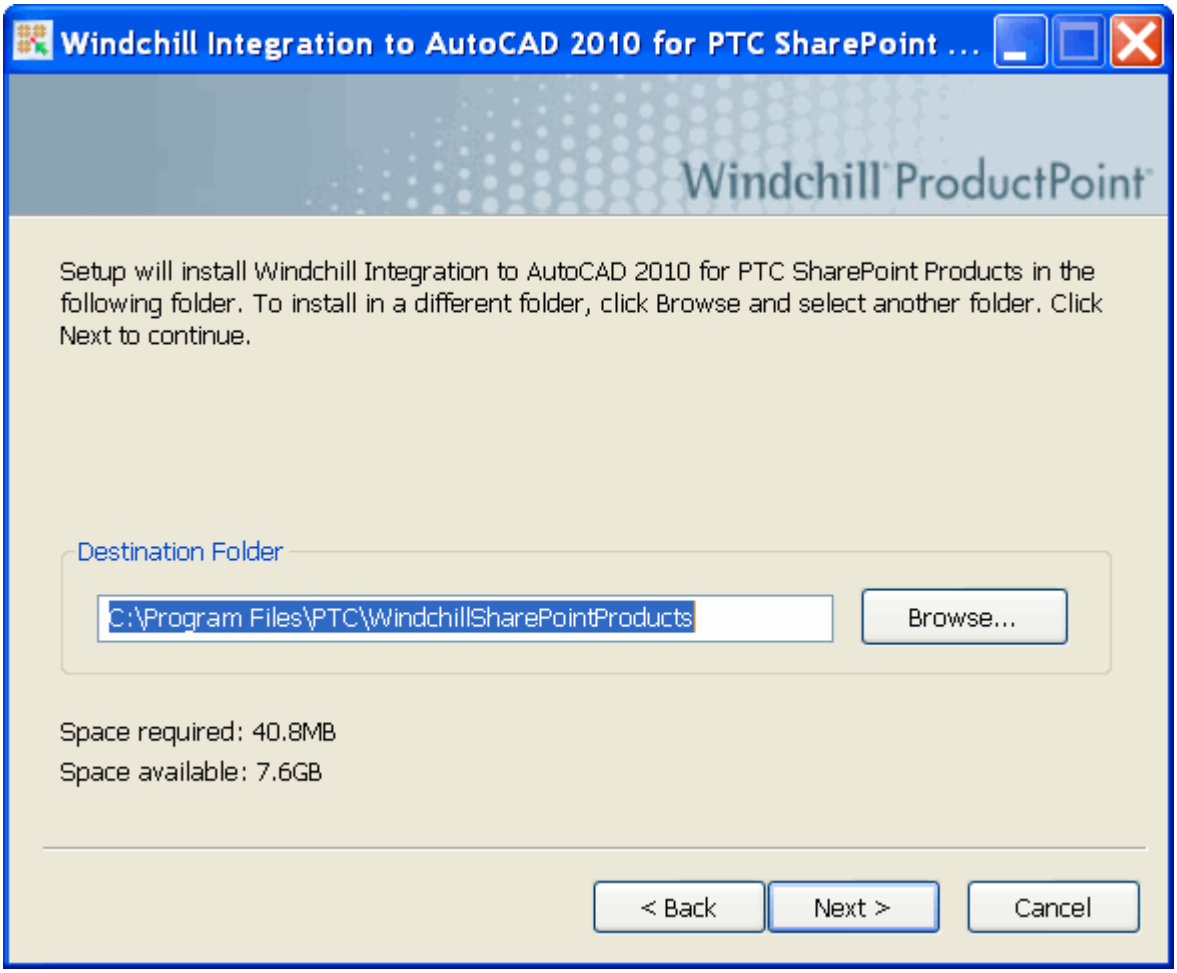
   - PTCWppAutoCAD2009_32.exe
   - PTCWppAutoCAD2009_64.exe
   - PTCWppAutoCAD2010_32.exe
   - PTCWppAutoCAD2010_64.exe

   So, for example, if you're using a 32 bit operating system, and you have AutoCAD 2010 installed, the correct installer file for you is PTCWppAutoCAD2010_32.exe.

3. A security warning window will open. Click**Run** .
4. A second security warning will open. Again, click **Run**.
5. Select your language. Click **OK**.

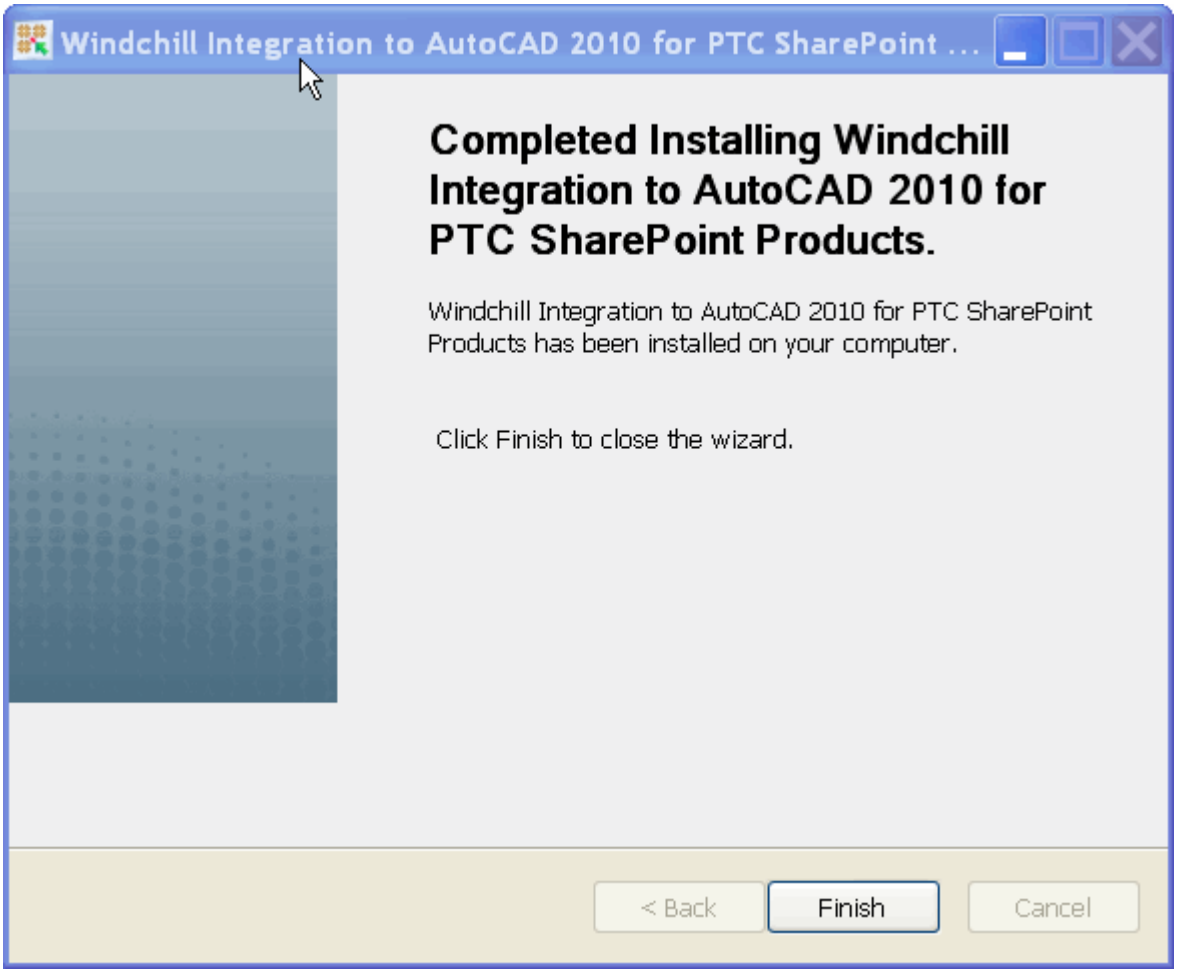

6. The installer's introductory window will open. Click **Next**.
7. You will be prompted for the directory into which the Workgroup Manager for AutoCAD will be installed. Accept the default, type a new location, or

click **Browse** to navigate to the desired folder. When you have made your selection, click **Next**.



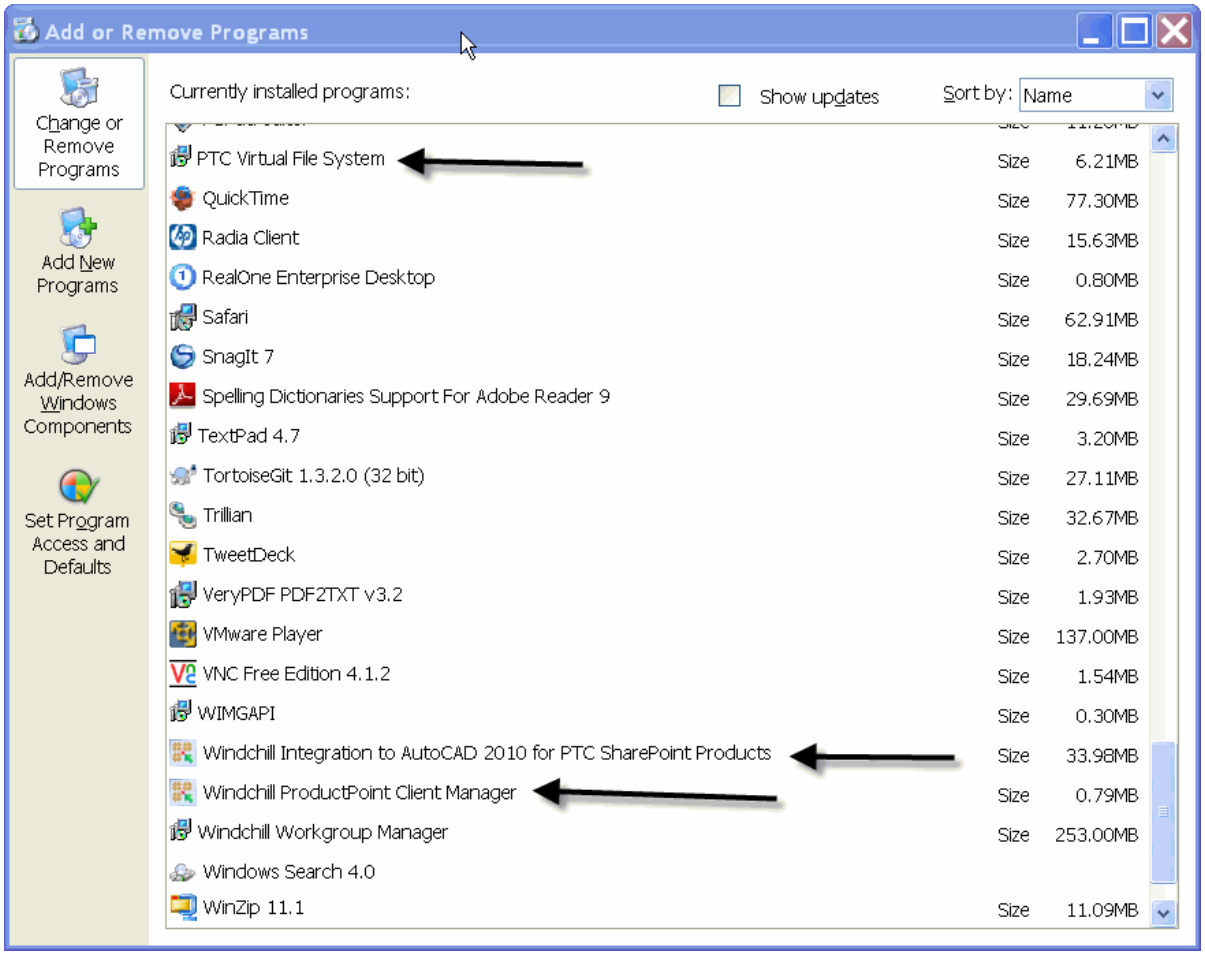

8. You will be advised of the components to be installed. Review them, and click **Next**.
9. You will be prompted to select the version of AutoCAD that you want to use. Select your version and click **Install**.

10. When the installation program has completed, you will be presented with the following screen:




Click **Finish**.

The Virtual File System, Client Manager and integration should now be installed on your system. (See Figure Below.)




When the client manager is running, the  icon is added to the system tray. If this icon is not present, the client manager is not running and will not be able to communicate with the server. Once the integration components have been installed, you must start the Client Manager by hand the first time. This is done by selecting **Start ▸ All Programs ▸ Startup ▸ Windchill ProductPoint Client Manager**.

# Registering a Server in AutoCAD

You must register each server you want to connect with in AutoCAD.

### *Note*

*If you have never previously registered a server, you must install a client manager component, as described above. Administrator privileges to the client machine are required to install the client manager. If you do not have administrative privileges, ask your administrator to install the client manager for you.*

### Note

*When the Client Manager is installed from the server (i.e., not during the installation), you must start it manually from the* ***Start*** ▸ ***All Programs*** ▸ ***Startup*** *menu.*

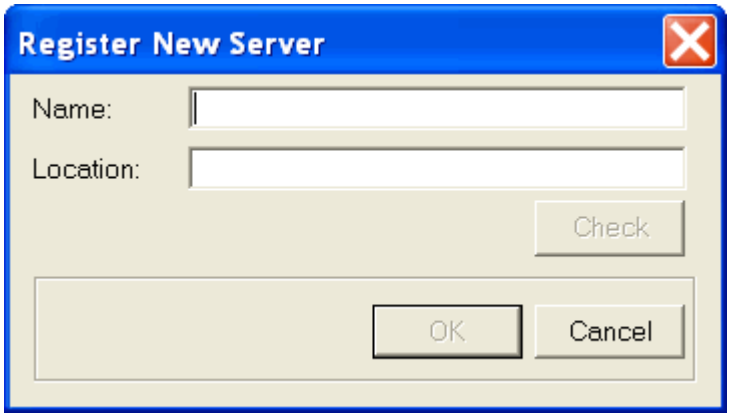
To register a ProductPoint server in AutoCAD, follow these steps:

1. From AutoCAD, select **Tools** ▸ **PTC Places** The PTC Places window will open.

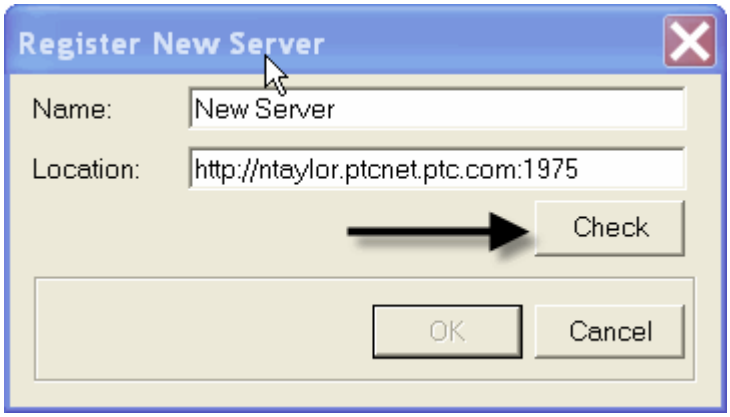
   ### Note

   *You can also access PTC Places from the Windows Control Panel by selecting* ***Start*** ▸ ***Control Panel*** ▸ ***PTC Places Options****.*

2. Enter a location in the file system folder structure where you want the entry point for ProductPoint to be located. (If left blank, the PTC Places entry point folder will be located under *%userprofiles%*).
3. Click the **Server Manager** tab, then **Register**.




4. From the **Server Manager** tab, click the select **Register** button.
5. Enter the Name and Location (URL address) of the server. You can use the address supplied by your administrator, or copy the address from any server page on the server in a browser and paste it in the Location field.
6. Click **Check** to test that can find the server you identified.

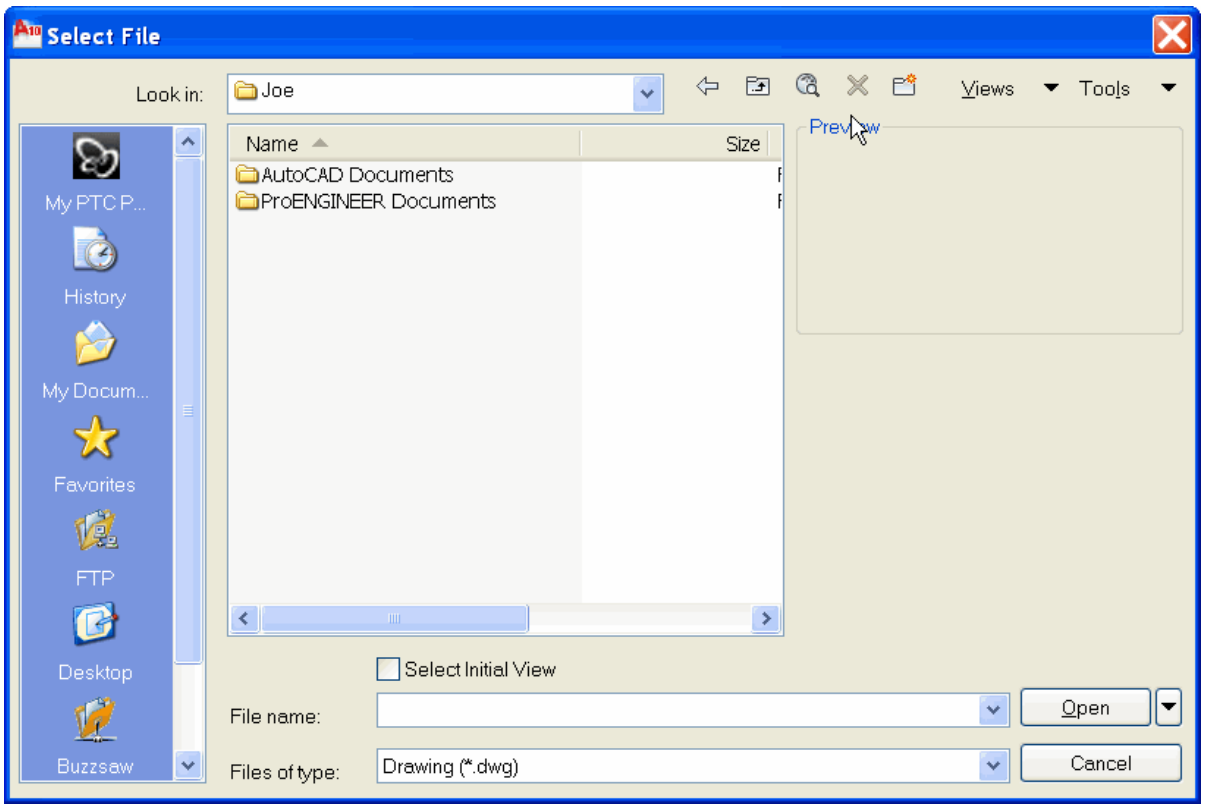



7. Click **OK**. At this point, you will be prompted to log onto the server.


### Note

*It is possible to register more than one Product Point server. Simply repeat the process above for each server you want to add.*

When you have successfully completed registration, you should now be able to find the server's folders from within AutoCAD. Select **File ▸ Open**. The Select File window will open. In the Navigation Bar on the left, you should see a folder called PTC Places. These are folders that reside on the ProductPoint server. You are now ready to begin using data from ProductPoint within AutoCAD.



# ProductPoint - AutoCAD Interaction



This chapter details the steps required to exchange files between AutoCAD and Windchill ProductPoint.

# Creating AutoCAD Files and Saving to Windchill ProductPoint

You can save your file to the Product Point server as follows:

1. Open AutoCAD.
2. Create a new AutoCAD document.
3. Click **File** ▸ **Save**.
4. Navigate to the PTC Places location, and locate the specific library to which you want to save your document.
5. If autonumbering is eneabled on the ProductPoint server, the new file name will automatically be given a number generated by the ProductPoint server. See [XREF] for more information on autonumbering).
6. Click **OK**.

At this point, if you access the ProductPoint server using a web browser, you should be able to find the document in the library to which you saved it. Opening and modifying information already saved in Windchill ProductPoint in AutoCAD.

# Modifying AutoCAD Data Already Existing in Windchill ProductPoint

To modify AutoCAD data that already exists in ProductPoint, do the following:

1. Open AutoCAD.
2. Click **File** ▸ **Open**, and navigate through PTC Places, and then to the location where the desired document resides.
3. Select the document, and click **OK**.
4. Edit the file in AutoCAD. At this point, you will see a lock icon in the lower right corner of the drawing. This means that the file is locked on the server. No one else will be able to edit the file while you are working on it. For more about locks see Lock Management.
5. When you have finished editing, Save the AutoCAD file. This will save the file to the same location from which it was retrieved.
6. This procedure will create a new iteration of the file in ProductPoint. (The **Save** action will remove implicit locks, but explicit locks will remain.)

# Saving from AutoCAD to Windchill ProductPoint

Saving a file to Windchill ProductPoint is similar to saving a file to your local hard drive.

1. Open and modify an AutoCAD file.
2. Click **File** ▸ **Save**.
3. If the file is an already-existing file that opened from the server, your file will be saved to the same location from where you opened it.

   ***Note***

   *At this point, the lock icon will disappear. Your file will be saved as a new iteration of the old file. That is, if you access the server with a web browser, you will not see a file with a new name. Rather, you will see the old name, but with an incremented version number*
4. If the file is a new one, or one that had previously been housed on your hard drive, navigate to PTC Places, and locate the specific library to which you want to save your doc.
5. Accept the default number, or give your document a name.
6. Click **OK**.

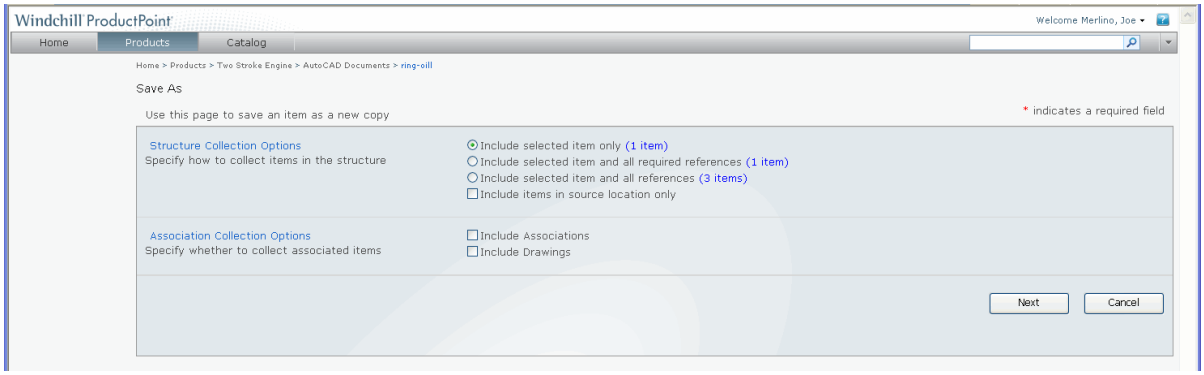
Your document will appear in the shared library area on the ProductPoint server, if the server is accessed using a web browser.

# Save As

It is possible to create copies of existing files in ProductPoint using the following methods:

1. From AutoCAD, open a file, then select **File** ▸ **Save As**. Alternately,
2. From the ProductPoint user interface, select a file, then from the drop-down menu, select **Data Management** ▸ **Save As**. The **Save As** options window will open.
3. Select the options that you wish to apply to the new file.

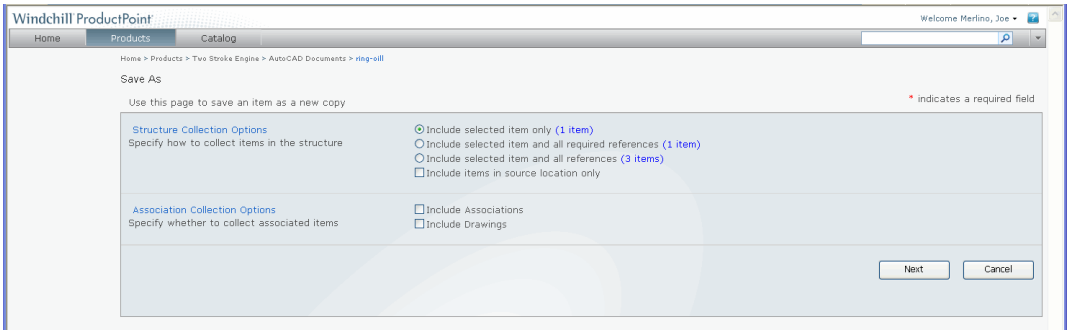
4. Give the file a new name, and click **Next**.




A copy of the file will be saved under a new name.

In addition to creating copies of existing files, the Save As action can be used to convert a file to a different file format.

1. From AutoCAD, open a file, then select **File** ▸ **Save As**.
2. In the field labelled Files of Type, click the drop down menu and select the file format to which you want to save your file.
3. Click**Save**.




# Using an Autonumbering Server

Since it is possible to have more than one ProductPoint server registered with PTC places, and when creating a new AutoCAD file it is not possible to know which server you are going to save to, a mechanism is need to define which server should be used for auto numbering. Without setting the active auto numbering server, AutoCAD will continue to name new files Drawing1, Drawing2 etc.

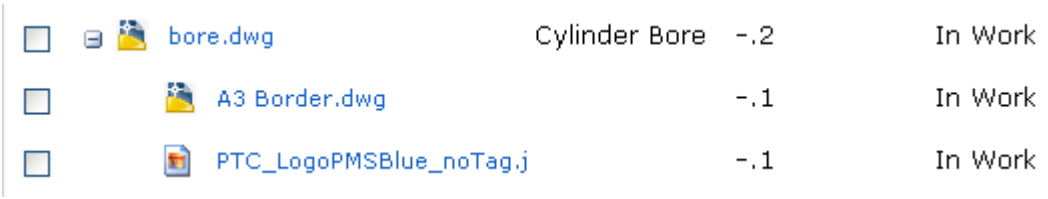
1. Type `PTC_AUTONUMBER` at the AutoCAD command line.
2. Enter server name (the name should be same as that entered in PTC Places)

### Note

*This need only be done once – the next time AutoCAD is started the auto number server will be remembered. The command can be run at any time to either change the server or disable auto numbering by typing OFF instead of the server name.*

# Working with XREFs

XREFs are files which can be attached to an AutoCAD document. XREFs are managed in AutoCAD using the **Attach** action, which is available from the **Insert** tab. XREFs can be other AutoCAD documents (.dwg), files, or raster image files. XREFs are represented in Product Point as associations, and can be viewed in the Structure Browser.




### Note

*The existence of an association in ProductPoint does not necessarily imply the existence of an XREF in AutoCAD. In other words, if you upload two files to ProductPoint, and create an association, no XREF will exist when the files are opened in AutoCAD.*

In order for XREFs to be preserved as associations in ProductPoint, you must upload the file to be attached BEFORE you create the XREF:

1. Create a document in AutoCAD.
2. Use the **Save** action to save the document to ProductPoint.
3. Find or create the file that you want to attach.
4. Upload that file to ProductPoint.
5. Create the XREF in AutoCAD
6. Save the document.

The association should now exist in ProductPoint. In the future, when you open this file in AutoCAD, the XREFs will automatically be included.

A few things to bear in mind:

- If you create an XREF to a file that already exists in ProductPoint, the association in ProductPoint will be created automatically.
- Using ProductPoint's **Upload Multiple Documents** action will NOT create associations in ProductPoint. You should either upload the file(s) to be attached first, then create the XREFs in AutoCAD, or create the associations manually in ProductPoint.

- If you move an attached file to a different location within ProductPoint, you will need to update the XREF in AutoCAD the next time you open the main document. AutoCAD will prompt you to do this.
- If you create an XREF to a file that does NOT exist in ProductPoint, a “ghost” will be created. This is an associated file with no content. When you upload the actual file that is to be attached, you will need to change the association manually in ProductPoint in order to repair the XREF.

# 13 Managing Information



In addition to merely exchanging files, Windchill ProductPoint allows you to protect your changes, and control versions in order to prevent one user from overriding another's changes. This section discusses those functions.

# Lock Management

When you edit a file that resides on the SharePoint server, a lock is placed on the file. Locking a file in ProductPoint serves two functions:

- It notifies others and a modification is being made to the file, and
- It prevents other users from modifying the document while you are making your modifications.

The lock can be set automatically, via AutoCAD, or manually, by accessing the SharePoint server via a web browser.

## Automatically Managed Locks

When you edit a file in AutoCAD, SharePoint immediately sets a lock so that no other user may alter the file. (Other users may open the file in Read-Only format.) The lock is set when the first change is made to the file (that is, the file is not locked until it is actually edited. Merely opening the file will not lock it.)

The lock is removed when you save the file, or close the file without saving the changes.

## Manually Managed Locks

If you know that you will be working on a drawing for an extended period of time, and you want to make sure that no one else is able to make changes to it, you can lock the drawing manually. Manual locks must also be released manually. When you set a manual lock, the file will not be released until you invoke the Unlock action. Manual locks will remain set regardless of whether you save your changes, or close the file without saving. To manually lock a file, do the following:

1. Access the SharePoint server with a web browser.
2. Navigate to the file you wish to lock.
3. Hover the cursor over the file name. A dropdown box will appear.
4. From the dropdown box, select **Data Management ▸ Lock**. The file will now be flagged as Locked.

To release a manual lock, do the following:

1. Access the SharePoint server with a web browser.
2. Navigate to the file you wish to lock.
3. Hover the cursor over the file name. A dropdown box will appear.
4. From the dropdown box, select **Data Management ▸ Unlock**.

The file will disappear from the Locked view, and is now available to other users.

# Updating Out-Of-Date Information

Sometimes changes are made by other users while you have an item in session in AutoCAD. Other users may either change the items through server operations, for example, by modifying the attributes, or changing the model content in AutoCAD. Windchill ProductPoint does not allow you to make changes to items in session which have been concurrently updated by others. This behavior is designed to prevent one user from inadvertently overwriting changes made by another user.

If you attempt to edit items in session which have been modified by others since you opened them, AutoCAD displays an out-of-date conflict warning. Click **OK** to continue with your change, but if you save your change, you will overwrite any changes made by the last modifier of the item.

It is best to close the file and use the provided hyperlink to bring the latest version of the item in session and to apply your changes on top of the changes made by the last modifier of the item.

When you click the link, every item in session that has more recent versions on the server is replaced with the latest versions of the items downloaded from the server.

# Manually Controlling Upload of Saved Changes

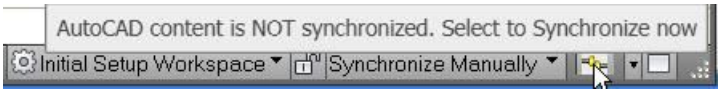
You can choose to manually control when the changes you make to AutoCAD models are uploaded to the server. You can delay the upload to reduce both the network load and client-server processing time. This approach also reduces the number of model file versions saved to the server and thereby reduces the disk space used by the database. The default behavior is to automatically upload file changes to the server each time you save a model in AutoCAD. You can change the option to synchronize manually, so that changes are uploaded only upon your request when you select Synchronize Now.

1. Select the server menu in the lower right corner of the AutoCAD window to view the server options.


2. Change the synchronization mode from Synchronize Automatically (default) to Synchronize Manually.
3. Open models from the server, make changes, and save them in AutoCAD.
   a. Save the changes.

b. When you save changes using Synchronize Manually, the status Not Synchronized is displayed after the first time you save changes. Not Synchronized means that you saved changes to your cache, and they are not yet uploaded to the server. You can continue making and saving changes to server-managed items while working in AutoCAD.

4. When you are ready to upload changes to the server, click Synchronize Now.




5. The synchronizing symbol is displayed until all changes saved to your cache are fully uploaded to the server. When all changes that are saved to your cache are fully uploaded to the server, the green symbol and Synchronized status are displayed.

### Note

*When you click Synchronize Now, all items that have been saved in AutoCAD to your cache but have not yet been uploaded to the server are all uploaded. This includes items that you previously saved and erased from memory in the AutoCAD session. Clicking Synchronize Now applies to all items in your cache for which your saved version is more recent than the versions on the server.*

Whenever you exit AutoCAD and the synchronization mode is set to Synchronize Manually, all changes you saved to your cache that have not been uploaded to the server are automatically uploaded. The synchronization occurs in the background even when AutoCAD is not running. If you shut down your computer or disconnect from the network before the upload is complete, the upload restarts when you restart your computer and reconnect to the network.

All model files for which you have saved changes to your cache but which have not been uploaded to the server are automatically locked on your behalf. The lock is automatically released when the upload is completed.

If the lock on a drawing has been changed by a user but the changes have not yet been uploaded to the server, the system automatically locks the items and displays a lock status of `Waiting to synchronize changes on... by....` The automatically created lock is released when you click **Synchronize Now**.

### Note

*Caution The user for whom the lock is created or an administrator can select the action to unlock the item on the server, but by doing so will ignore the unsynchronized changes previously saved to the user's cache.*

# Renaming Items

AutoCAD supports two approaches for renaming items:

- Rename items already saved to the server
- Rename items before saving to the server

## Renaming AutoCAD Items Previously Saved to the Server

It is usually best to rename items already saved to the server using the server-supported rename actions in the browser. You cannot rename items already managed on the server from AutoCAD. Furthermore, the browser rename action supports automatic name generation for the rename.

## Renaming an Item in AutoCAD

Before Saving to the Server If you open a file from a file system that has the same name but is a different item than the one that already exists on the server, you can rename the item in AutoCAD and then save the renamed item to the server so that you do not overwrite the existing item.

1. After enabling the rename action, make sure the server-managed item is not already in memory and then open the item from the file system.
2. Click **File** ▸ **Rename**.
3. Enter the new name to use for saving the item to the server. Save the model that is renamed in session.

If you are connected to a Windchill ProductPoint server, the model in session is saved to the server under the new name. The file system version originally opened in session is unaffected by the change.

Consider these tips on renaming items:

- If you open an item from the server and use the **File** ▸ **Rename** action in Windchill ProductPoint, a copy of the item is created and all assemblies that reference the item in session will be updated to use the new copy. Once an item is saved to the server, it is usually best to rename it using the server rename action in a browser.
- If you have an assembly on the file system and need to rename a large number of components in the assembly before saving them to the server, you can first select **File** ▸ **Save a Copy** and rename the files. Make sure to load the copy and to erase the original items from memory before you save a copy to the server; otherwise, you will encounter name conflicts for the items that already exist on the server.

# 14 Working Offline

Windchill ProductPoint does not provide support for working on server-managed files in AutoCAD when you are disconnected from the network. If you must work on server managed files while disconnected from the server or wish to send server-managed files to a design partner so the partner can work on the files independently, you should carefully follow the prescribed procedure. This approach is not recommended, but may be necessary in some cases if there is no other alternative.

The next procedure describes the best practices for working on server-managed AutoCAD files while disconnected from the server:

1. Lock items of interest on the server to prevent conflicts with changes by others.

   ***Note***

   *In Windchill ProductPoint 1.1, you cannot lock a structure of items or select multiple items to lock, so you must manually lock items one at a time.*

2. In AutoCAD, open all the items from the server on which you want to work offline.
3. Select **File** ▸ **Save** to back up all items in sessions to a file system.
4. In the Server Manager, select **No Server**. You can also disconnect from the network, and AutoCAD connects to the client file system by default.
5. Shut down AutoCAD or disconnect from the network (or just erase all the items in session in AutoCAD and open the files again from the file system backup folder).
6. Make changes to the items as desired in AutoCAD. You can do this either connected or disconnected from the network, but AutoCAD must be connected to the file system.

7. You can create and assemble new parts, if necessary, but do not use names for the new parts that exist on the server.

   ### Note

   *If you create a new part of the same name as an existing part on the server (but which is actually a different part), the existing part is iterated and replaced with the new part of the same name. This can cause a regeneration problem in any assemblies that use the part or subassembly.*

8. When you are ready to save the changes back to the server and are connected to the network, make the Windchill ProductPoint server the primary server (this is only necessary if you previously changed the connection to your file system).
9. Open the drawings from your backup file system folder that you modified.
10. Click **Save** in AutoCAD to save all the changes to the Windchill ProductPoint server.
11. Select the option to continue the save and override the conflicts. The system uploads the files and creates new versions for all saved models of the same name on the server. Any new parts are also saved.

# 15 Managing Your Client Cache



ProductPoint provides tools to manage the file system cache, allowing you to set the cache location and size.

# Setting Your Cache Location

A unique cache root folder is associated with each user on a client machine. Under this root folder, a cache subfolder is defined for each registered ProductPoint server. The default location of the folder in Windows XP is: `D:\User Profiles\user name\ApplicationData\PTC\ProductPointService\fully qualified server name\cache`

For example:

```
D:\User Profiles\myusername.PTCNET\ApplicationData\PTC\
ProductPointService\ malfoy.ptcnet.ptc.com\cache
```

### *Note*

*Tip: When determining the location of cache, consider the size of the drive partitions and your write privilege.*

You can configure the location of the client cache root folder by setting the *PTC_WF_ROOT* environment variable:

1. On Windows XP, right-click **My Computer** or select **Start ▸ My Computer ▸ View system information**. Click the **Advanced** tab and then click **Environment Variables**.
2. On the System Properties dialog box, click the **Advanced** tab.
3. Click **Environment Variables**.
4. Specify the client cache root folder location.
5. Click New to define a new environment variable.
6. Enter the variable name `PTC_WF_ROOT`.
7. Enter the file path for the root cache location. A folder is automatically defined to hold the cached files for each registered server under the specified location.

# Setting Your Cache Size

You can configure the amount of disk space allotted to the cache root folder by setting the *PTC_CACHE_LIMIT* environment variable.

The value is set in units of 1MB. If you set it to 1024, you have allocated 1 GB to the root cache, and the 1 GB limit is applied across the cache subfolders for all of the registered servers. If you have 3 servers, the total size of all registered cache folders is under 1 GB.

Files in the cache are removed based on their age. The oldest files are removed first, to free space for saving new items. Every read of the file from cache or every save into the cache changes the file date, making it current. If you set a cache

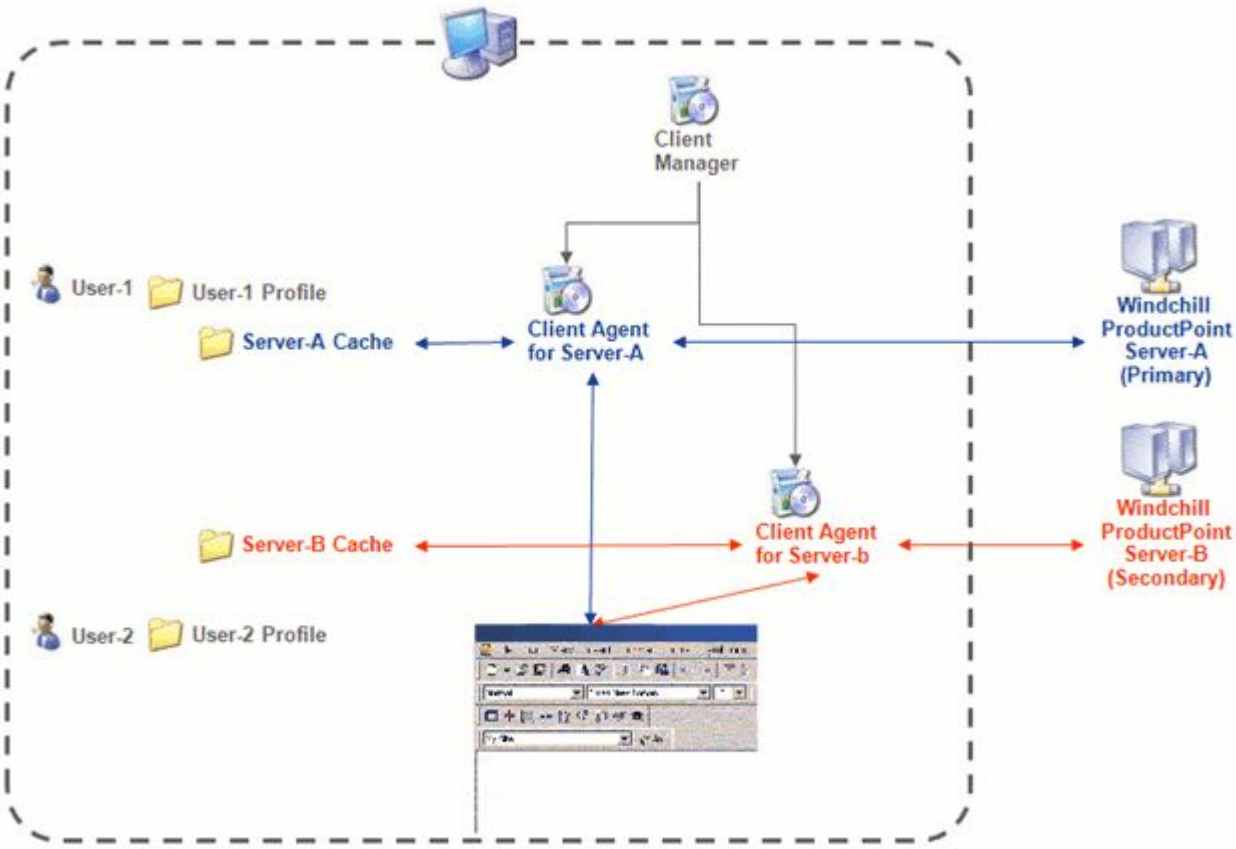
limit to 0 (default), then no limits are imposed. The cache limit does not affect the AutoCAD save behavior. All saves that you make in AutoCAD are first saved locally, uploaded to the server during synchronization, and then removed or kept, depending on the cache limit. You must have enough available disk space to save your model file changes on your client machine.

### Note

*You must log off Windows and log back on to ensure that the values for new or modified environment variables to take effect.*

# 16 System Architecture

This diagram illustrates the organization and interactions of the client and server components with AutoCAD.




These components interact as follows:

- The client manager is responsible for starting and updating the client agents.

  The client manager must be manually started the first time it is installed, unless Windchill ProductPoint components were installed during the AutoCAD installation; in which case, the client manager will start automatically. The client manager starts automatically in the future each time Windows is started (this is accomplished by adding it to the Windows startup).
- If the client manager is not running, no communication between AutoCAD and the server is possible.
- A separate client agent is associated with each user for each server that is registered and connected to AutoCAD.
- The client manager handles all client agents for users by starting, stopping, and automatically updating the client agents as servers are connected and disconnected.

- A separate client cache is maintained for each registered server for each user. If one physical machine is used by several users that log on to the machine under separate user names, each of these users has an independent client cache that is associated with them.

# 17 Troubleshooting



At times, you may encounter conflicts when performing editing or save actions. The following table provides a list of conflicts, what they mean, and how to resolve them.

| Application Action | Conflict Type | Explanation | Resolution |
|---|---|---|---|
| Edit | The file is locked by someone else. | The file is being edited and is locked by another user | Wait for the lock to be released. |
| Edit | The file is out of date. | The file is being edited, and the version you are attempting to edit is not the most current version. | Edit the most current version once it has been saved. |
| Edit | The file is read-only. | The file you are attempting to edit is read-only. | If you have write access to the file, remove the read-only designation from the file's properties and try again. |
| Save As | A file with the same name exists on the server. | There is an existing file on the server with the same name as the one you are trying to save. | There are two options.<br>1. Overwrite the existing file.<br>2. Rename the file to a nonexistent name and re-save. |

| | | | |
|---|---|---|---|
| Save As | A file with the same name exists as an incomplete (missing) reference. | There is an incomplete item, on the server with the same name as the one you are trying to save. | Overwrite the incomplete item. |
| Save As | The save fails due to an access control on the target location. | You do not have permission to write to the desired target location. | Choose another location to save to, or see your system administrator to obtain the necessary permission. |
| Save and Save As | Dependents with the same name in a different location exists in the parent file. | The dependents with the same name will be ignored because they originate from different locations. | No action needed. |
| Save and Save As | A file with the same name exists for dependents. | An item with the same name as the dependent already exists on the server. | Reuse the name on the server, and change the path to a relative path. |
| Save and Save As | There is an incomplete reference for a dependent (there are no objects with the same name). | Since there is an incomplete reference for dependent(s), saving the file will create a placeholder object. | None. |
| Save and Save As | There is an incomplete reference for a dependent because the name is reserved. | The dependent with the name you have chosen cannot be created because the name is reserved by another user. | None. |
| Iterative Save | The file is locked by someone else. | The save fails because the file is locked by another user. | None. |
| Iterative Save | The file is out of date. | The save fails because the version you are attempting to save is not the most current version. | None. |

# Log Files

Should they be needed, log files are located by default in the directory defined by the value of the %TEMP% environment variable, in the directory %TEMP%\autocad_2010_integration_logs. However this can be changed by altering the logging.cfg file, located in the application folder. There are several different types of log files:

- Installation log: PtcWppWgmAutoCAD_log.txt.
- For the AutoCAD add-on, the log file would be named acad-don_<date_time>_debug.log
- For ProductPoint, the log file would be windchillProductPoint-agent-<servername>.log