

Product Lifecycle Management (PLM)

PDMLink 8.0 M020 with Pro/ENGINEER Wildfire 2 M200 User Guide

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INTRODUCTION

Windchill PDMLink Pro/ENGINEER (Pro/E) Wildfire Extension is the ability to manage and control your data. PDMLink Pro/E Wildfire Extension delivers data management capabilities within the PDMLink environment. It understands Pro/E files and all their detailed inter-relationships.

Product Capabilities:

- Maintain and organize secure data through the use of ORACLE technology
- Access database tools for easily finding and re-using data, attributes, system-wide renames, lifecycle tracking and release controls
- Integrate Pro/E designs, relationships and functionality with Windchill PDMLink Product Structure
- Work with specific product configurations and incorporate design changes and conflicts from many sources
- Communicate design status, release procedure status, activity and intent of other users to entire design team, enabling rapid local changes

Customer Benefits:

- Increased efficiency through a common working environment for Pro/ENGINEER users
- Ensured data integrity through secure access to data and design history
- Quicker time to market through effectively managed design changes and avoidance of design conflicts
- Increased productivity of engineers through simplified information management
- Shared engineering data with enterprise systems through Web-based environment

1 Software Configuration

The base PTC components of this entire configuration is:

PDMLink Server Products Release 6.2.6.06

- Oracle 10G
- Apache
- Tomcat
- Aphelion
- Java(TM) 2 SDK, Enterprise Edition 1.4.2_10international version
- Java 3D for 1.3.1
- PDMLink Foundation 8.0
- Windchill Visualization Service
- Both Language Pack CDs
- Service Pack M020
- RetrievalWare

CAD Worker

- Pro/E Wildfire 2
- ProductView CAD Adapters M020 for Pro/E Wildfire
- Windows File Transfer Protocol in Internet Information Services only

Pro/ENGINEER Wildfire Clients

- Internet Explorer with all active X enabled in internet security options
- ProductView Standard Edition M020 for Windchill PDMLink
- [Java\(TM\) 2, Runtime Environment 1.4.2](#)
- Pro/ENGINEER Wildfire 2 M200

NOTE: You will need at least one licenses of Pro/ENGINEER, for the CAD Worker

Since Apache Web Server, Tomcat Servlet Engine, Info*Engine, Excalibur RetrievalWare and PDMLink, are placed on the same server, the recommended minimum amount of RAM is 2 GIGs for production.

1.1 Recommendations

It is best to have the CAD Worker on a separate machine because it requires a considerable amount of resources to create the ProductView viewables from Pro/E.

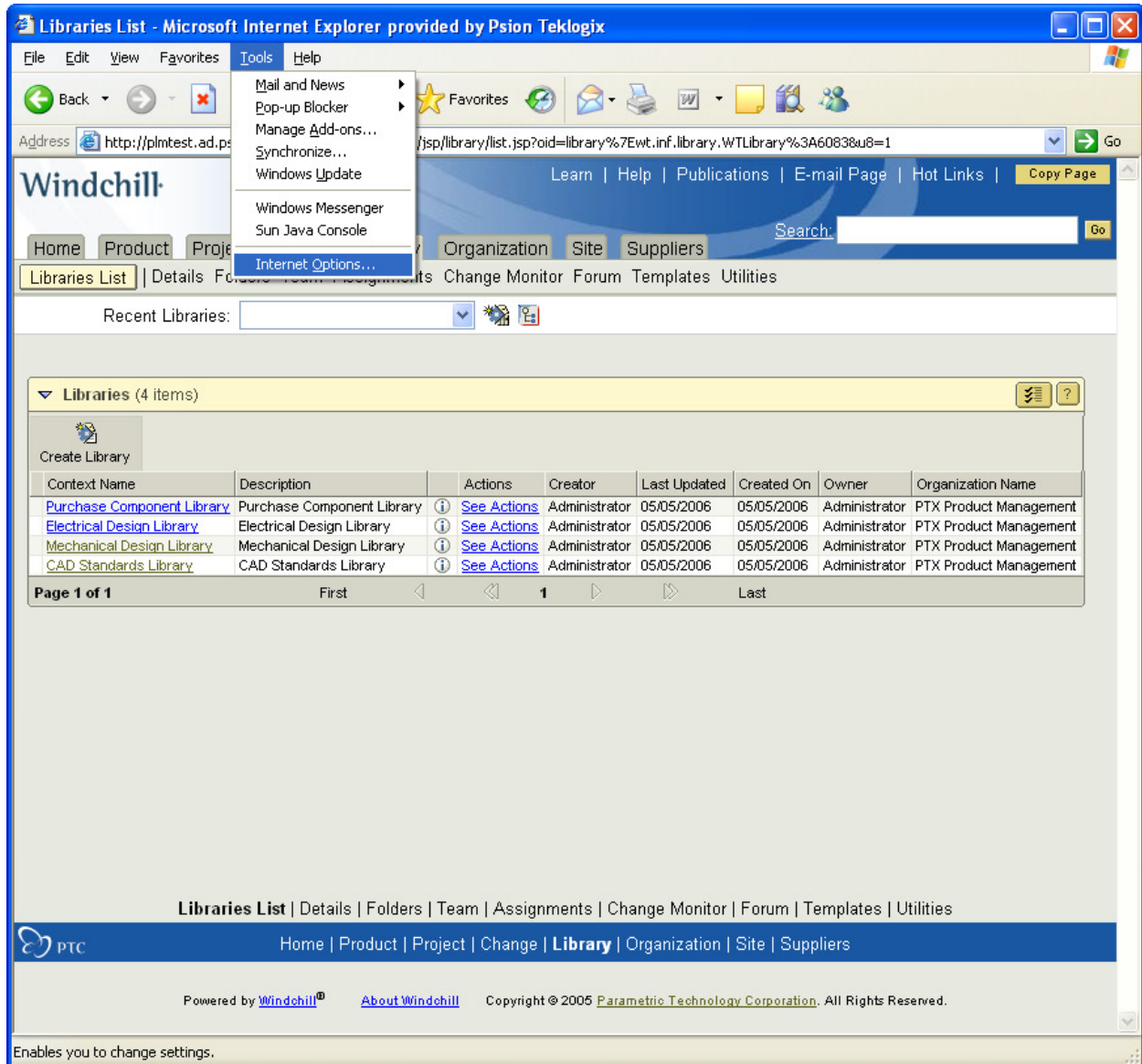
2 Client Workstation Configuration

To perform administrative configuration on the PDMLink server and properly use the ProductView Express, Standard Edition and Lite versions via the web browser, ActiveX security controls must be configured on the clients Internet Explorer. To properly display PDMLink applets, a Java(TM) 2, Runtime Environment 1.4.2_10 software is be installed on each clients machine.

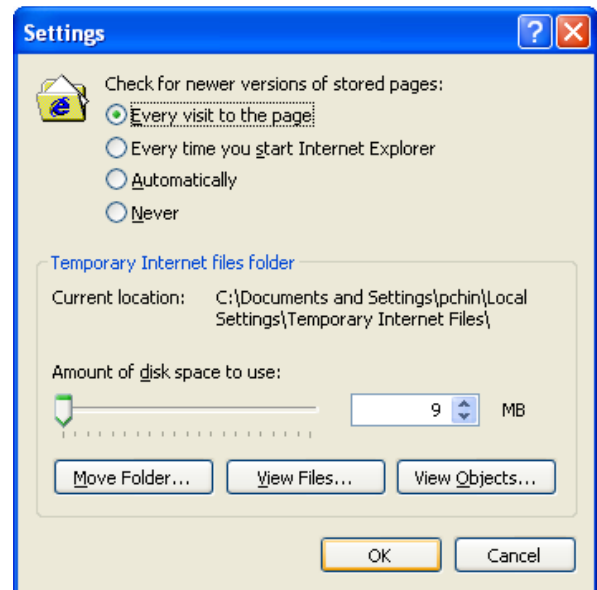
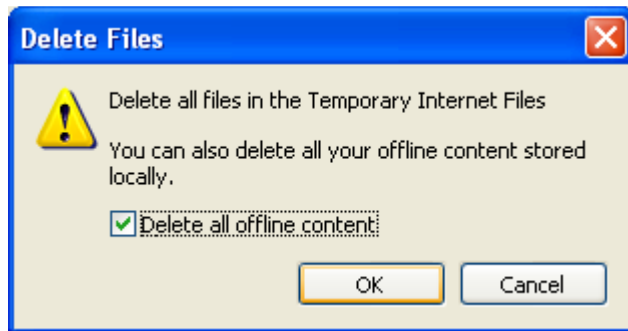
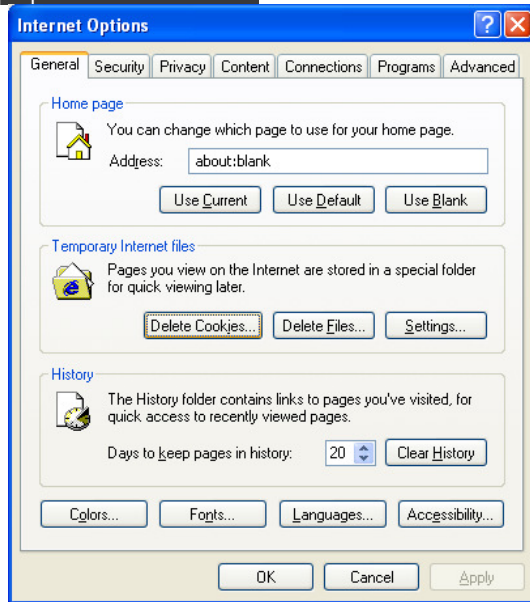
2.1 Active X Security Options Enabled on a Windows Internet Browser

To view thumbnails and ProductView viewables and thumbnails in various ProductView applications, the Windows Explorer security options must be configured for ActiveX controls. The configurations are as follows:

1. In the Menu Bar of the Windows Internet Explorer, select the Tools drop down list and select the Internet Options.

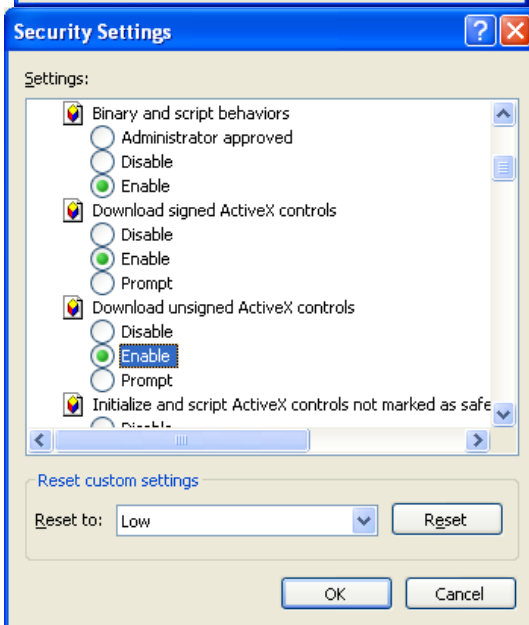


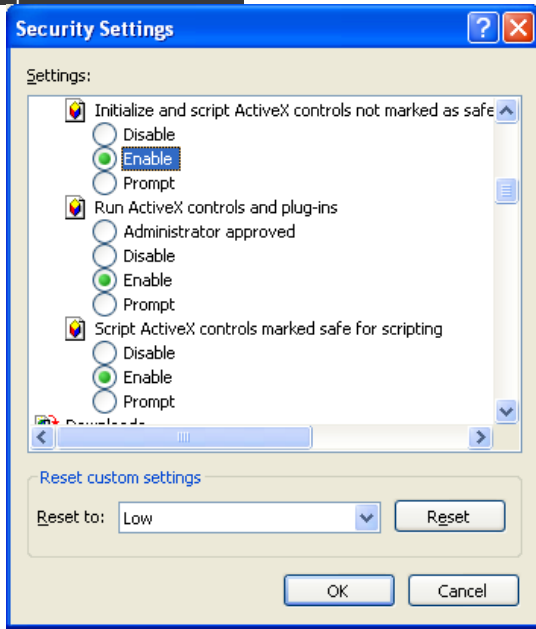
2. In ther General Tab, delete all Temporary Internet files and reset the settings.



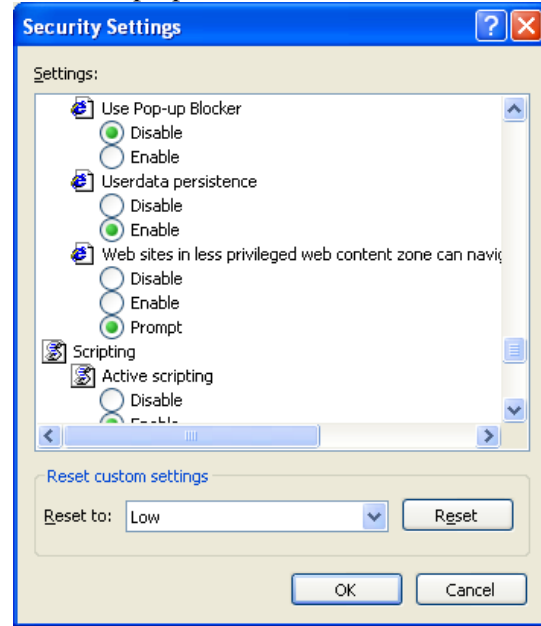
3. In the Internet Options, select on Custom Level button for the setting of Trusted Sites and Local intranet.

4. Enable the ActiveX controls according to below and select OK button:





4. Disable Pop-up Blocker

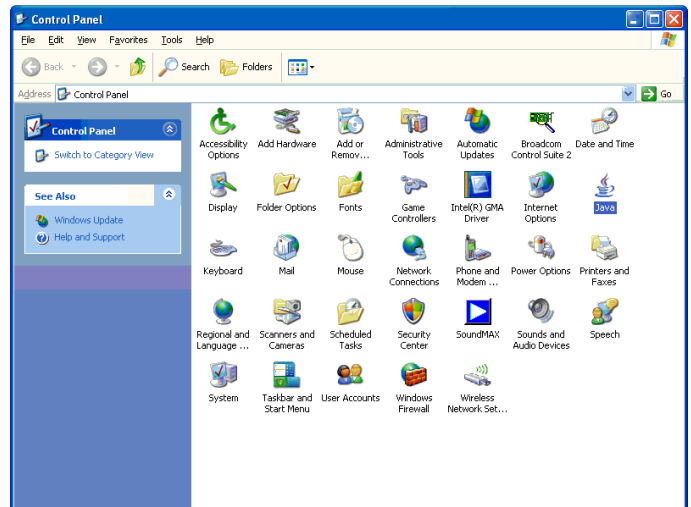
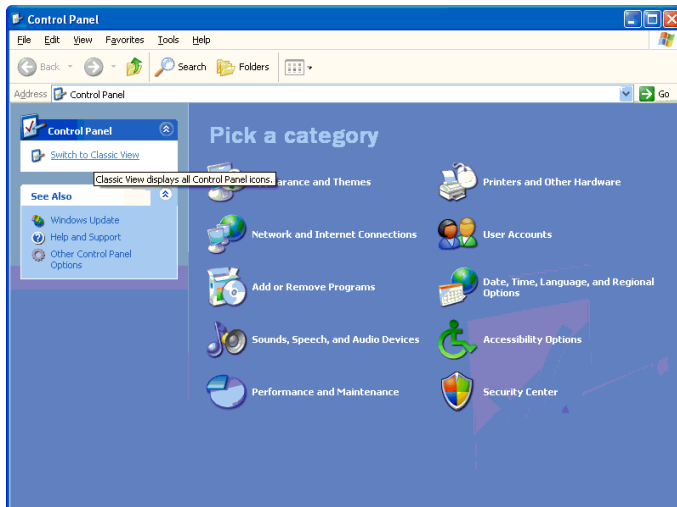


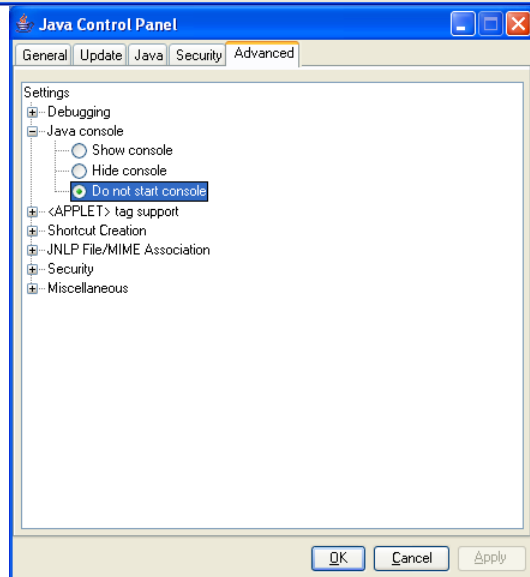
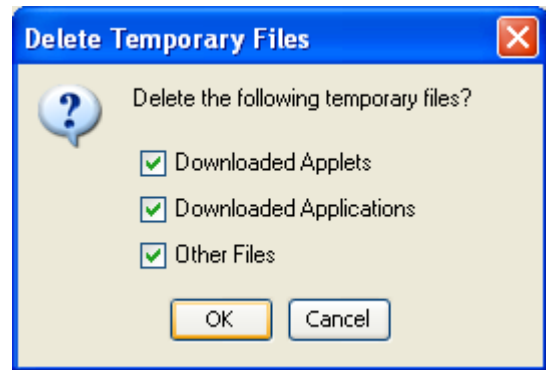
2.2 Clearing Pro/ENGINEER Workspace Cache (workstation and client login dependent)

In your respective Application Data folder, delete the “.wf” directory:

C:\Documents and Settings\<your user name>\Application Data\PTC\ProENGINEER\Wildfire\wf

2.3 Clearing JRE Cache (workstation and client login dependent)





2.4 Product View Standalone Client

Utilities - Microsoft Internet Explorer provided by Psion Teklogix

File Edit View Favorites Tools Help

Address http://plmtest.ad.pSION.com/Windchill/netmarkets/jsp/user/utilitiesList.jsp?oid=user%7Ewt.org.WTUser%3A4792&u8=1

Windchill Learn | Help | Publications | E-mail Page | Hot Links | Copy Page

Home Product Project Change Library Organization Site Suppliers

Overview Assignments Updates Checked-Out Work Meetings Notebook Subscriptions Reports Utilities

Welcome, Patrick Chin Recently Accessed: Last Login: May 8, 2006, 2:55 PM EDT

Search within my work:

Utilities

- [Calendar Management](#) Provide the ability to schedule non-working days and delegate work.
- [Publish Monitor](#) View the status of jobs for translating CAD geometry to visualization images.
- [Preferences](#) Customize the following aspects of your functionality: display, document content, search, tables, visualization, configuration specification. Some values may have been set by your site administrator.
- [Profile](#) View your personal information that the company has on record.
- [Security Settings](#) Set security permissions here rather than be prompted throughout the system.
- [Software Downloads](#) Access additional software available for download with Windchill.
- [Visualization Collaboration Sessions](#) Set up a meeting using a shared visualization session.
- [Visualization Collection](#) View the Visualization representations waiting to be opened in ProductView.
- [Event Manager](#) Provide the ability to check and act on event status.

Overview | Assignments | Updates | Checked-Out Work | Meetings | Notebook | Subscriptions | Reports | **Utilities**

PTC Home | Product | Project | Change | Library | Organization | Site | Suppliers

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Trusted sites

Edit User Preferences - Microsoft Internet Explorer provided by Psion Teklogix

Edit User Preferences

Display Content Search Tables Visualization Configuration Specification

Thumbnail Display on Properties Pages
Choose whether thumbnail images display on Properties pages.

Loading ProductView Data
Choose how to load files into ProductView.

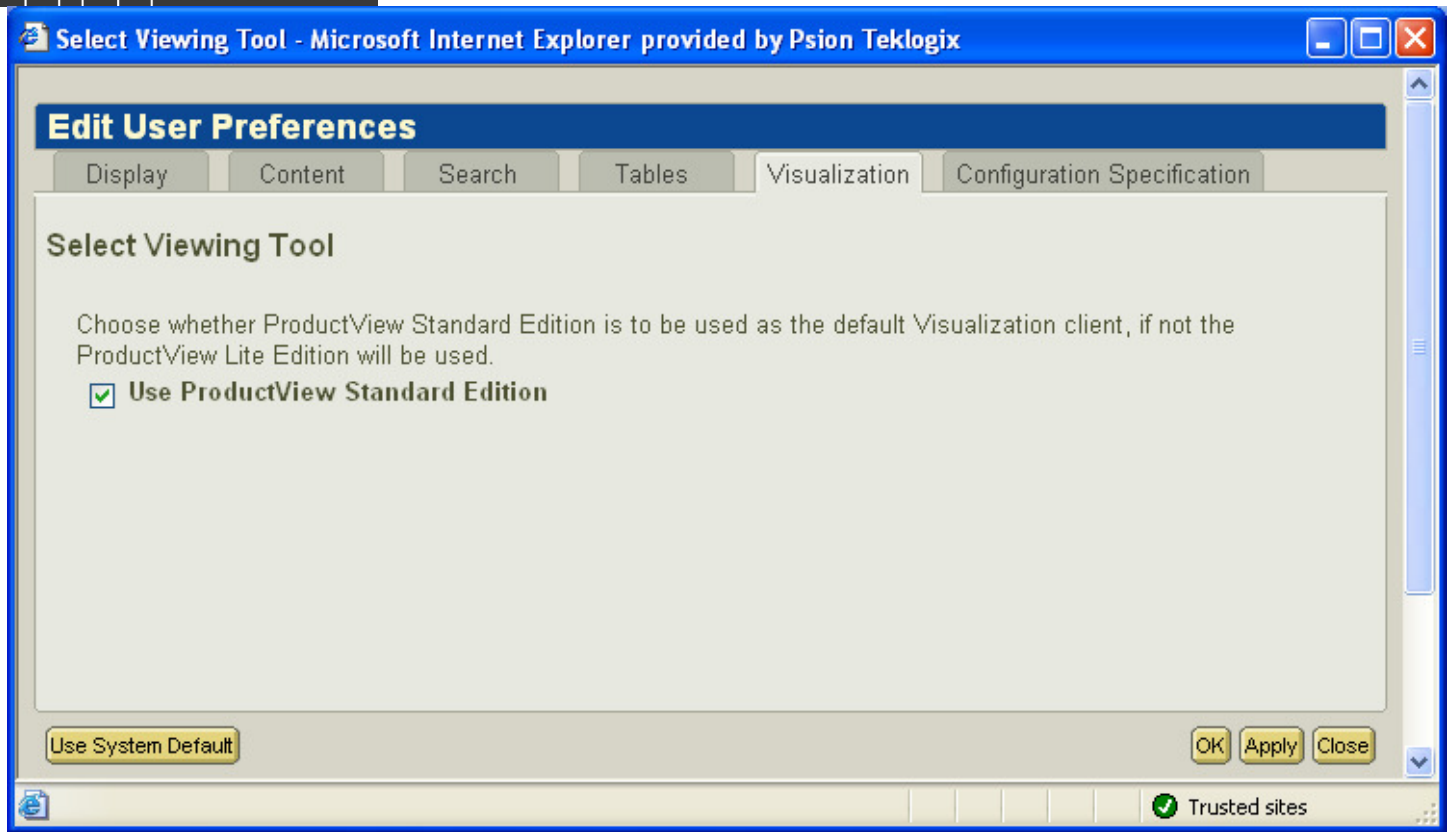
Tool Tips Display
Choose whether tool tips display on Visualization pages.

Select Viewing Tool
Choose either ProductView Standard Edition or ProductView Lite Edition as the Viewing tool.

Visualization Collection
Enable Visualization Collection for use with ProductView

Close

http://plmtest.ad.pSION.com/Windchill/servlet/WindchillAuthGW/wt.enterprise.URLProcessor/ Trusted sites



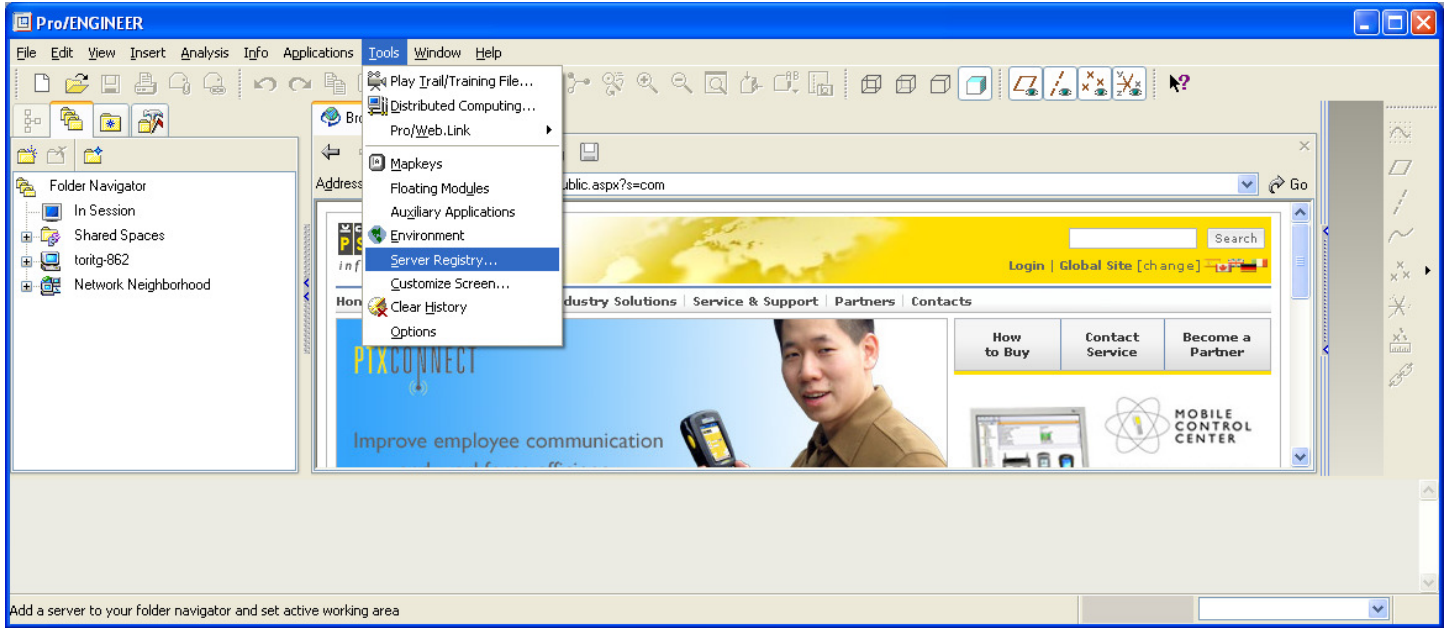
3 Pro/E Wildfire Integrated With PDMLink

Pro/E Wildfire has an embedded web browser. Pro/E Wildfire's embedded web browser allows the Pro/E user to use one interface to perform most of engineering functions. In order to have Pro/E drawings, assemblies and parts to be managed by PDMLink, each client of Pro/E must communicate to PDMLink via a registry server service in PDMLink. Currently, there is no functionality of renumbering. It is best practice to ensure that the Pro/E file has a correct part, assembly or drawing number prior to uploading. It is best to work on the files locally. All required start files are managed in the PDMLink database.

3.1 Server Registry Configuration

Pro/E Wildfire can connect to multiple PDMLink/Windchill servers. The Pro/E clients at Asahi Matsushita are only required to connect to one PDMLink server called plm.ad.psion.com. The next steps will properly show how to connect to plm.ad.psion.com.

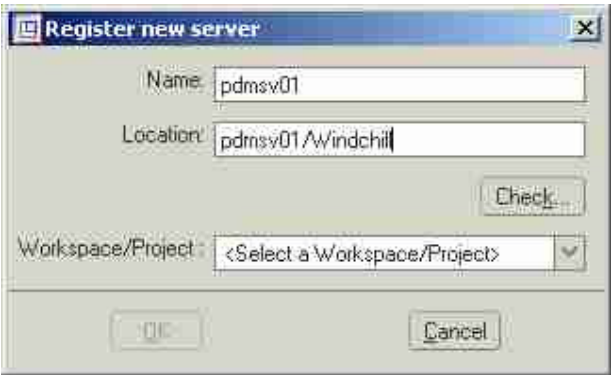
1. In the menu bar, select Tools then Server Registry.



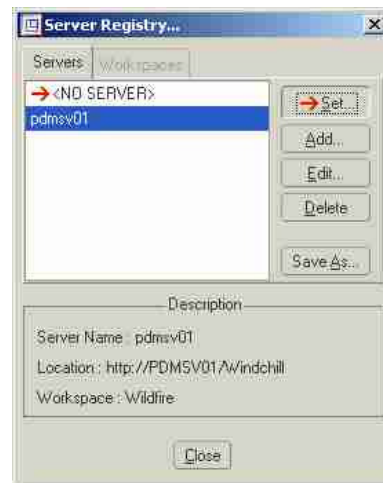
2. A Server Registry applet will appear to connect to the PDMLink server. Select the Add button.



3. In the Register new server applet, enter **plm** and **plm/Windchill** for values of Name and location respectively. Select the Check... button and log in applet prompt an authentication.



4. If there are no workspaces in the user's personal cabinet, the only selection is the default workspace. Select the OK button. To integrate the Pro/E client with the PDMLink server, select the →Set... button with the plm.ad.psion.com alias highlighted.



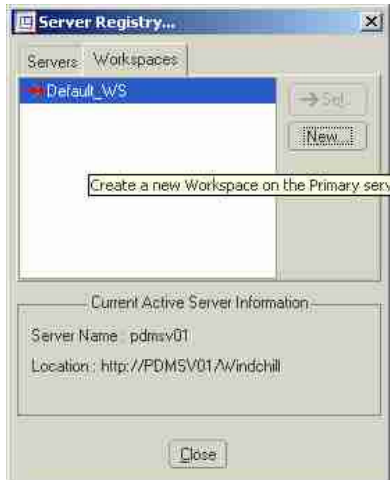
3.2 Create Workspace for Pro/E Wildfire Clients

For every PDMLink user, there exist a hidden personal cabinet in the database. A workspace is a directory or folder within that user's personal cabinet. There are many default functionality of Pro/E Wildfire integrated PDMLink. They are:

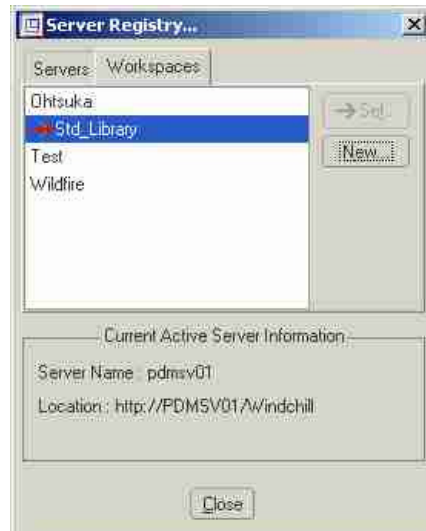
- a) Files can be uploaded into the workspace and the PDMLink number and name will be assigned the complete file name with the extension. In section 7, customizations are discussed to remove the extensions from the PDMLink number and name for WTParts.
- b) Cannot browse to CAD document templates via Pro/E Wildfire configuration tool.
- c) Cannot delete WTParts and CAD documents.
- d) Cannot renumber or rename parts.
- e) The name and description PDMLink system attribute is a non-versioned or iterative attribute.
- f) Because the workspaces are in the database, every object regardless of location has to be unique; therefore, objects can't be located in multiple locations or workspaces. The methodology is to download.
- g) No Uploading in Default_WS
- h) Cannot delete workspace if the workspace is still set or the PDMLink server is set on server registry.

Because of default functionality, it is best to keep the workspaces of each user to a minimum.

1. To create a workspace, select the Workspace tab in the Server Registry applet. Then select the New button.
2. Enter Std_Library for name of the workspace and then select the OK button.



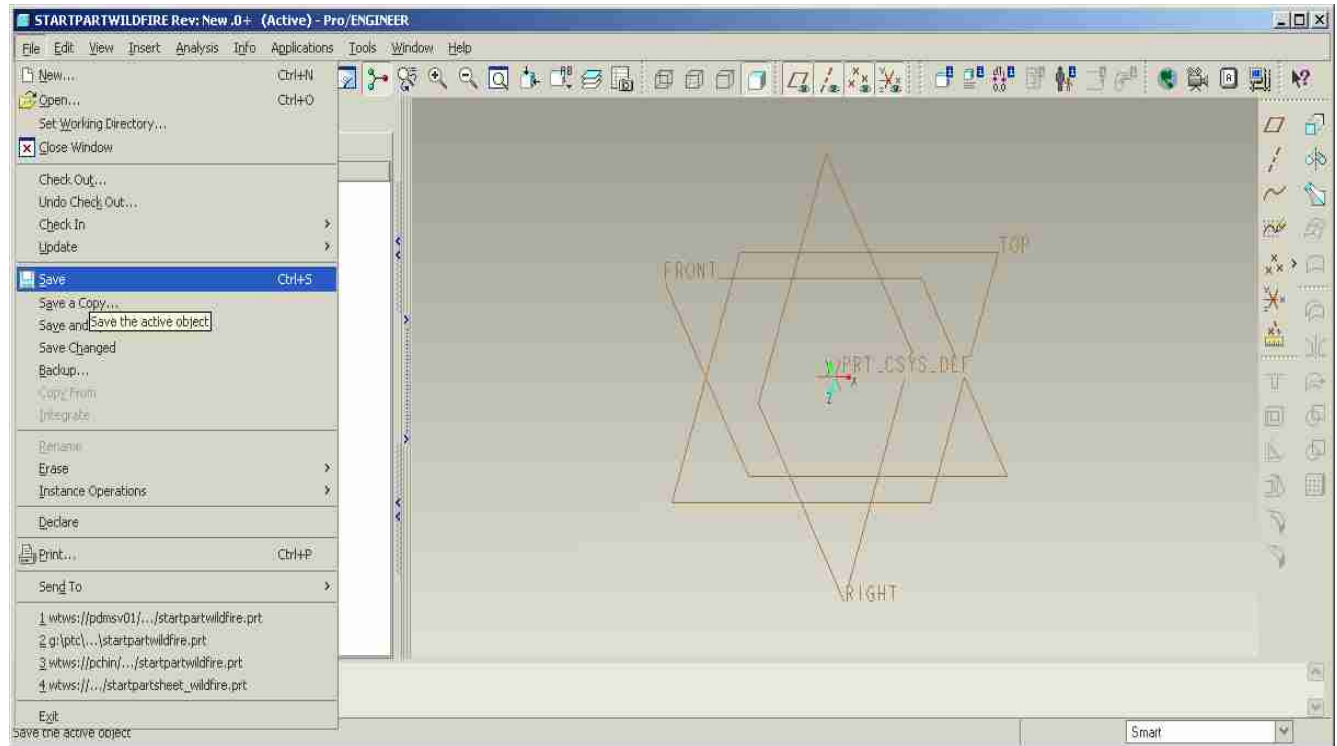
3. Finally, highlight the Std_Library workspace and select the →Set... button and then Close to complete the integration.



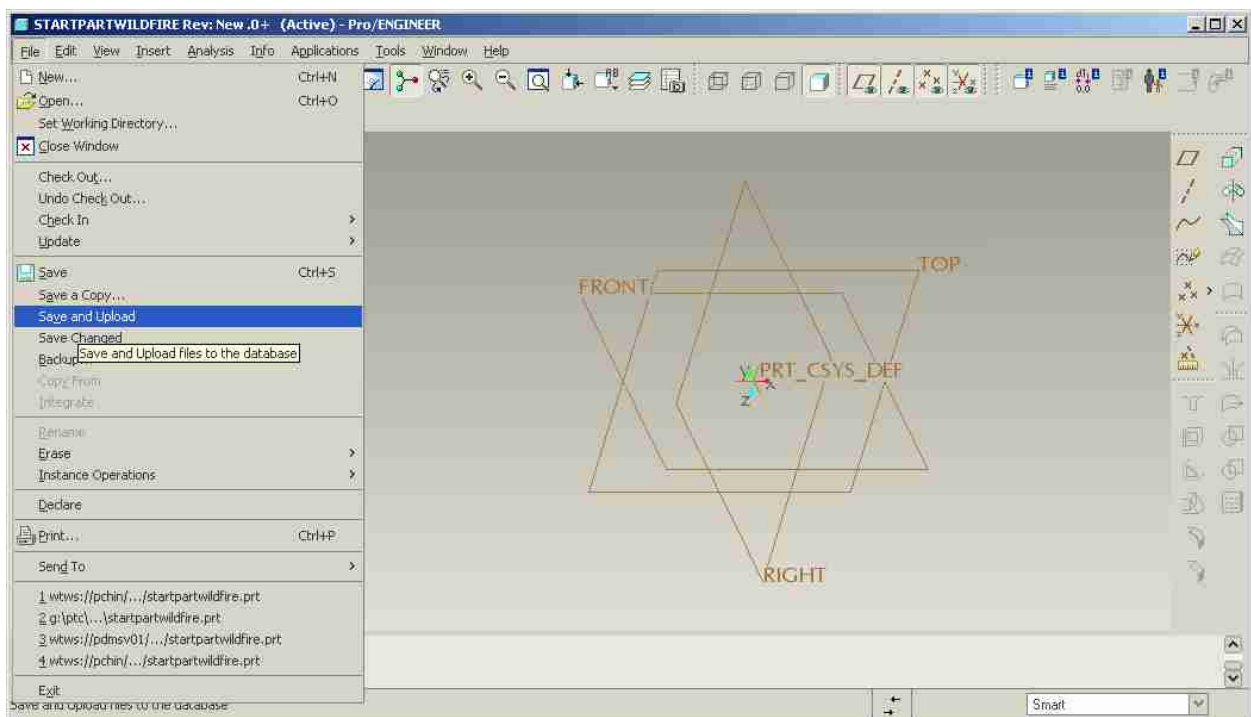
3.3 Uploading and Checking In Pro/E Wildfire Standard Library Model Files


When implementing Pro/E with a PDM system, it is best practice to manage Pro/E model files in the PDM database. As a result, history and control of all Pro/E model based files in a database would be part of the methodology. It is best practice to store the Pro/E modeled standard library files in a library container that will be shared by all end users. Also, all the configurations are stored Pro/E standard library model files in CAD documents objects in PDMLink. These files must be uploaded and checked into the ProE_Std_Library library container. The following steps will describe how to complete this task with the template solid model startwildfire.prt.

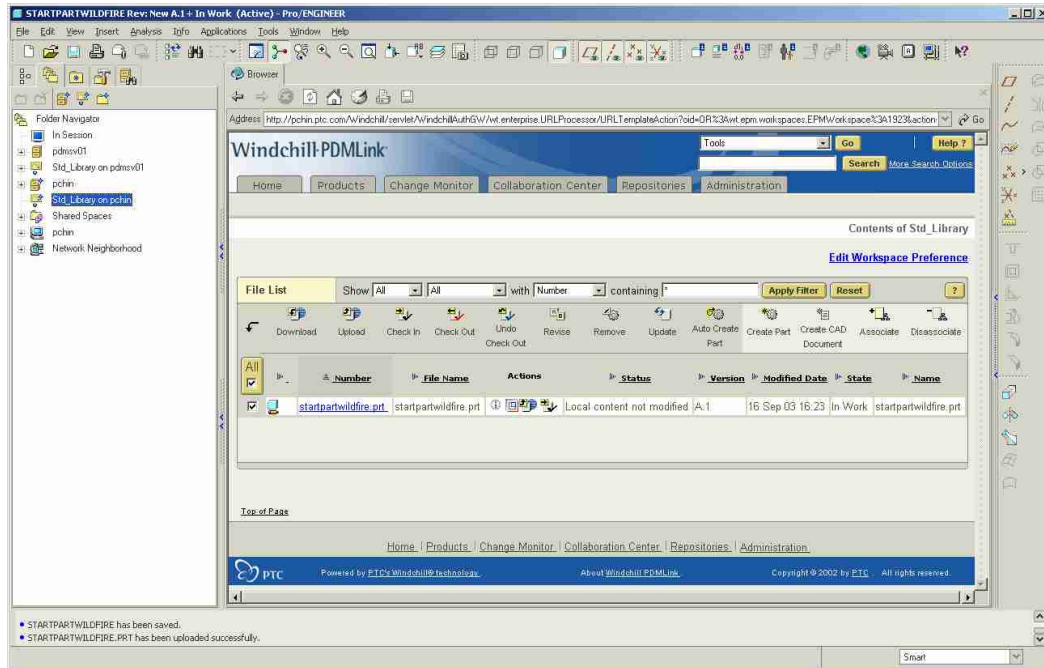
1. Open the startwildfire.prt where it resides on the Std_Library server in the directory and select the save option. Delete any Pro/E user defined parameters that are redundant to the PDMLink's Pro/E controlled parameters. Also, it is best practice to rename any parameters that are spelled the same as the PDMLink system attributes.



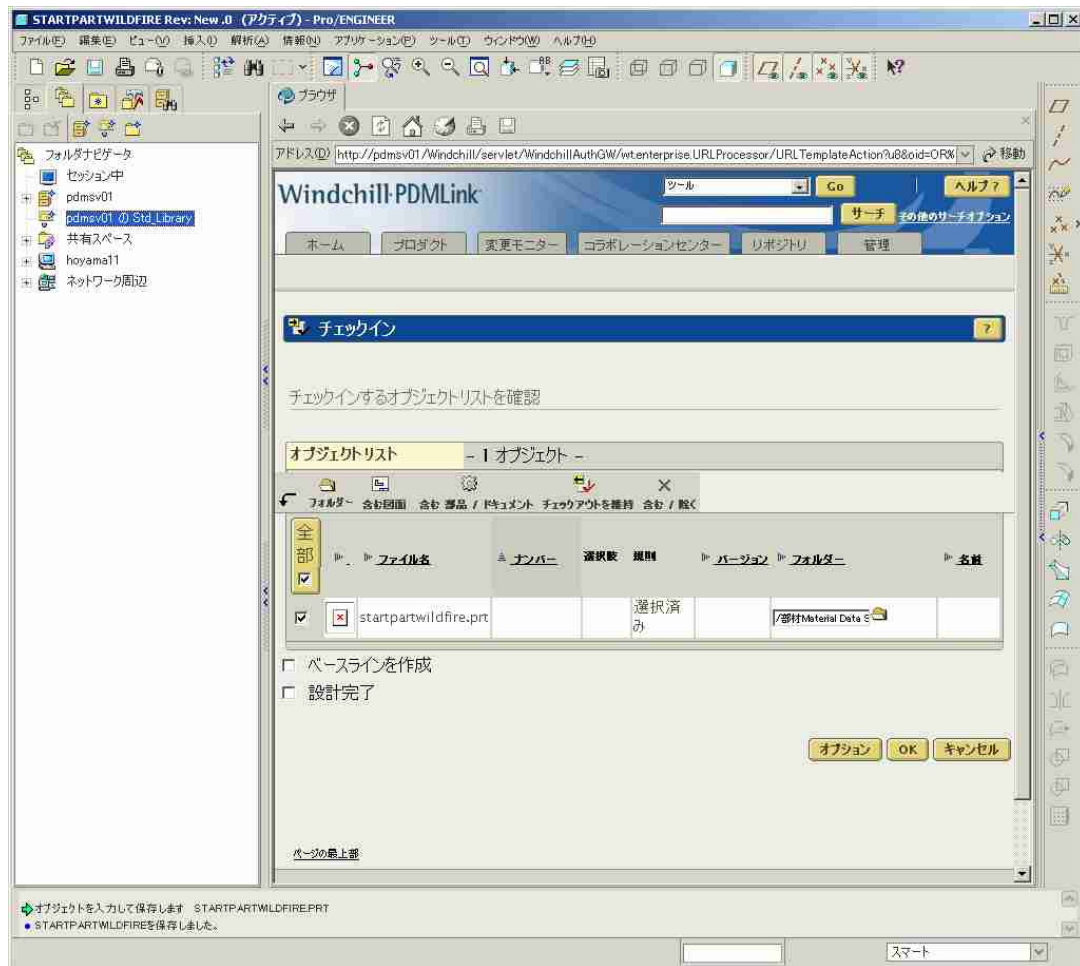
2. In the menu bar under File, select the Save and Upload. The upload function will save the file into a CAD document with the PDMLink number and name with the file name startpartwildfire.prt. Uploading it to the database will also assign a PDMLink revision/version and iteration value.



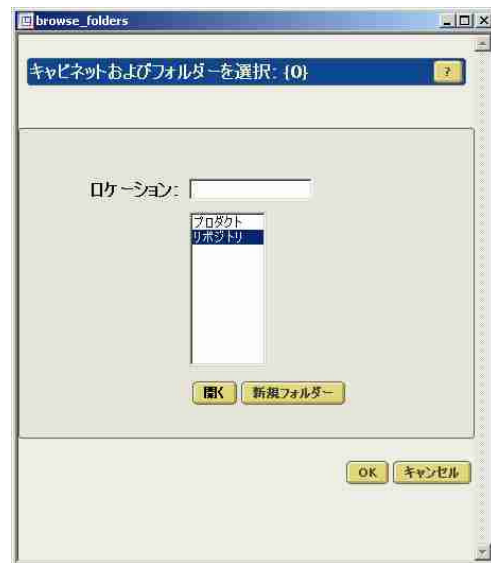
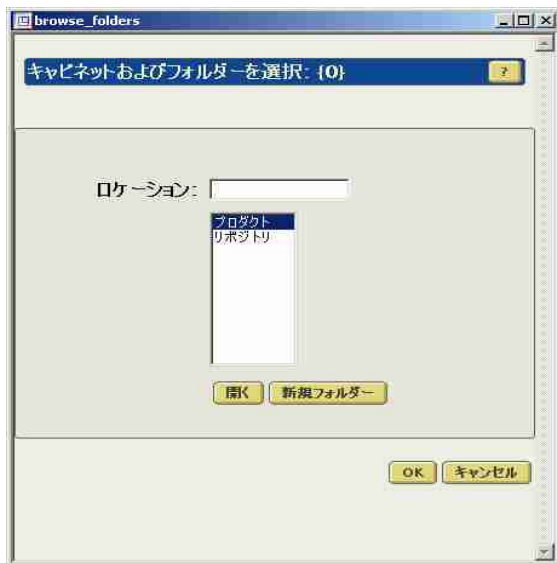
3. A new CAD Document startpartwildfire.prt is now present in the Std_Library workspace. This CAD document object will be checked into the library container called ProE_Std_Library. Select the startpartwildfire.prt by checking the check box then select the check in icon  in either the workspace menu or in the Actions options.



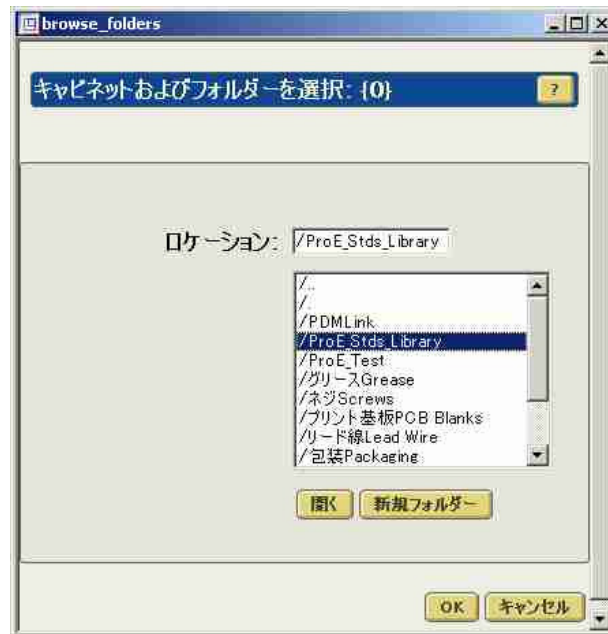
4. A check in window will appear in the embedded browser. Update the location for object to check.



5. Browse to the through the PDMLink repositories and select the ProE_Std_Library library container.



6. Finally select OK after highlighting the ProE_Std_Library.



The check in page will refresh with the new location and then select OK.

It is best practice to check in all Pro/E model files in the ProE_Std_Library in the PDMLink database that are located in the following local directories:


- <ProE_Std_Wildfire>\formats
- <ProE_Std_Wildfire>\library
- <ProE_Std_Wildfire>\start_files

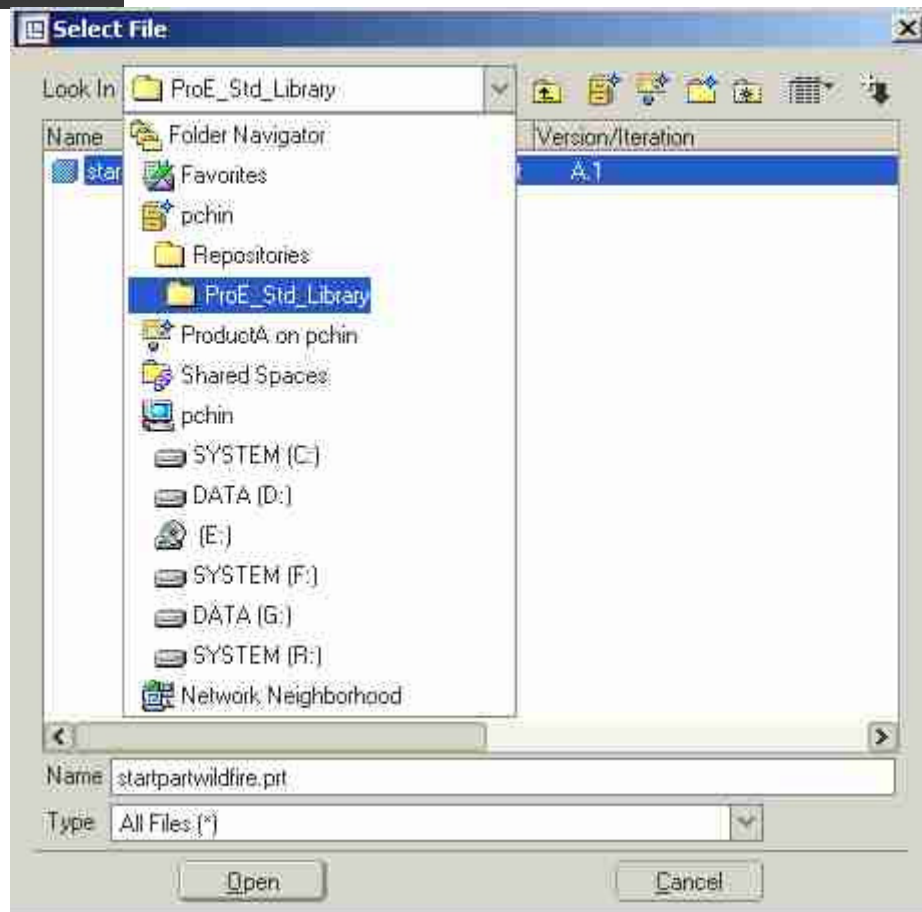
The remaining files are only for the Pro/E software application. These files are to be stored in the shared local hard drive H:\.

3.4 Deletion of Workspace

To keep Workspaces to a minimum, it is advisable to delete workspaces. To properly delete workspaces, there are 2 prerequisites:

- a) The workspace to be deleted is not set.
- b) If it is the only workspace, the PDMLink registry server alias is not set.
- c) The workspace has to be empty of objects.

In the Home tab of the Windchill-PDMLink page and check the workspace to be deleted then select the Delete Workspace icon .



4.1 Cache

Cache is used for temporary storage for the Pro/E Workspaces. This increases the performance of retrieving data in you Pro/E session rather than searching it elsewhere.

4.1.1 Cache Location

Cache directory is a fixed directory where there is large local shared space for all users. All users will require full access to this directory. It is recommended place cache on local disk where there are no OS and other applications installed. This will avoid OS or application failure. It is recommended to create a system environment variable called:

TMP	<directory location>	(not a network directory for better performance)
PROE_CACHE	<directory location>	(if possible another local drive)

The following lines must be added to the config.pro to enable the cache locations.

PTC_WF_ROOT \$TMP

PTC_WF_CACHE \$PROE_CACHE

4.1.2 Cache Size

The default cache size is 400 MEG per workspace. If you would like to state another value to be tailored to the limits of your machine and the number of workspaces created by all users. Administrators should allocate disk space by this formula

of Pro/E users per workstation X average number of workspaces X cache size X 2 = total cache space

dm_cache_size 400 (set at 400 MEG for each workspace)

4.2 Reset to Last Workspace

When exiting and opening Pro/E, a configuration option can be set to his last accessed workspace. The default is set to true (YES). The other option is NO. The configuration property is:

dm_remember_server YES

4.3 Model Display in Pro/E Client

Set the following for the thumbnail view for the ProductView Express functionality in the embedded browser.

save_model_display shading_lod both

4.4 File Compression for Upload and Download

Integer, from 0 (no compression) to 9 (maximum compression) [default = 0] Sets the level of compression for data upload and download. Although compression speeds up transfer over the network, it uses server CPU and client CPU to perform the compress and decompress operations. In a local area network, where network transfers are rapid, compressing and decompressing data can result in lesser throughput. On wide area network with lower bandwidth compression can lead to higher throughput. Since this is set per client, PTC recommends that clients in a LAN use a value of 0 (the default) and clients in a WAN use a value of 2 or 3.

There is a balance between Pro/E Client performance and network throughput and data storage. Pro/E Client best performance is at no compression while the best performance in a network and database file throughput and retrieval respectively is at maximum compression. At , the configuration option was set to 5.

dm_http_compression_level 5

4.5 Default Web Page

The default web page can be set by this property:

web_browser_homepage [http:// www.ptc.com/support/support.htm](http://www.ptc.com/support/support.htm)

4.6 PDMLink Registry Server Connection Retries

The default number of attempts to connect to the PDMLink Registry connections after a failure is 10 times. This configuration sets the number of attempts to connect to a PDMLink server before the connection is considered broken.

(This is a hidden config option that was introduced in Pro/ENGINEER Wildfire date code 2003210.) The recommended setting is the default. If the http connection is unstable, a setting less than the default could increase failures, while a setting greater than the default causes delays if a failure occurs. In , the configuration value was set to 5.

dm_network_retries 5

4.7 Concurrent Download Threads

The default value for this configuration option is 3. This option sets the number of concurrent threads Pro/ENGINEER uses for downloading data from a PDMLink server. In most cases, increasing the number of threads in a LAN environment will not improve performance, as the disk will then become the bottleneck. Even in a WAN environment, settings greater than the default are unlikely to improve throughput significantly. (This is a hidden config option that was introduced in Pro/ENGINEER Wildfire date code 2003210.) PTC recommends this option to be default.

dm_network_threads 3

4.8 Automatic Upload to the Primary PDMLink server User's Personal Cabinet Workspace folder in the Database

This configuration option defines the behavior of the **Save** command in Pro/ENGINEER. If this option is set to explicit, the Pro/ENGINEER **File > Save** command will write data to the local disk space and cache. The user must then explicitly upload/send that data to the PDMLink server (using either **File > Save and Upload**, **File > Upload**, or **File > Checkin**). If this option is set to automatic, **File > Save** in Pro/ENGINEER will also upload the Pro/ENGINEER files to the PDMLink User's Personal Cabinet Workspace folder. For best practices, PTC recommends that this value is set to automatic when connected to a PDMLink server. Files will be both saved on the database and locally on disk.

dm_upload_objects automatic

4.9 Automatic Upload to the other Secondary PDMLink servers User's Personal Cabinet Workspace folder in the Database

Pro/E Wildfire can connect to multiple PDMLink servers, this option defines the behavior of saving to a secondary server (See also dm_upload_objects).

dm_secondary_upload automatic

4.10 File Retrieval Options

To ensure that the setting files are securely located and readily available, it is recommended to store setting files in the PDMLink Oracle database. Thus, history in changes will be maintained and proper security controls are placed for CAD administration methodology. The setting files are checkin to a read accessible Pro/E standards Library container in PDMLink. The config.pro options that specify storage and retrieval directories, including such options as the following:

- start_model_dir
- pro_library_dir
- pro_format_dir

can be set to point to Windchill cabinets. For example, the value of start_model_dir is set to point to a Windchill PDMLink library container cabinet using the following syntax:

start_model_dir wtpub://<server_alias>/Library/<library_container_name>

Since all the files were stored in a single library container, the configuration is as follows:

```
start_model_dir wtpub://plm.ad.pSION.com/Library/ProE_Stds_Library
```

for all the directories as stated above.

Note: If you retrieve an object from any location other than the primary server, it will be treated as if it were newly created in the Pro/ENGINEER session. This means that actions on the object (for example, save or requesting checkout) are done in the context of the primary server, not the location from which the object was retrieved.

Config.pro options that point to a specific file, including such options as the following:

- intf_in_use_template_models
- template_designasm
- template_mold_layout
- template_ecadprt
- template_solidpart

can be set to point to Windchill file locations using a string of the proper syntax and the name of the CAD document that manages the file, as in the following example:

```
template_solidpart wtpub://<server_alias>/Repositories/Templates/template_solid_inlbs.prt
```

In , the configuration is as follows for the template_solidpart:

```
template_solidpart wtpub://plm.ad.pSION.com/Repositories/ProE_Stds_Library/startpartwildfire.prt
```

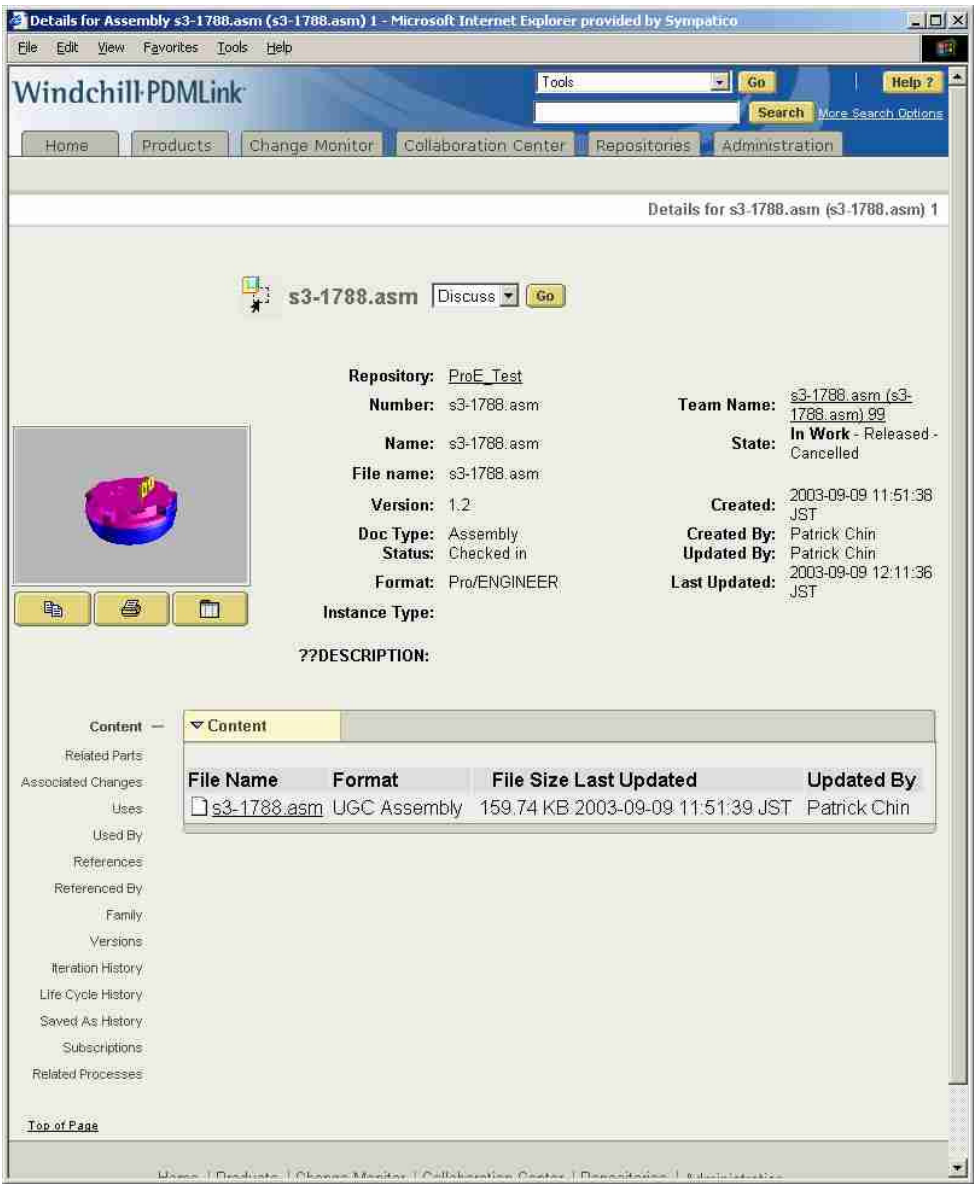
The other templates configurations have been modified to their respective CAD documents:

```
template_sheetmetallpart wtpub://plm.ad.pSION.com/Library/ProE_Stds_Library/ startpartsheet_wildfire.prt
```

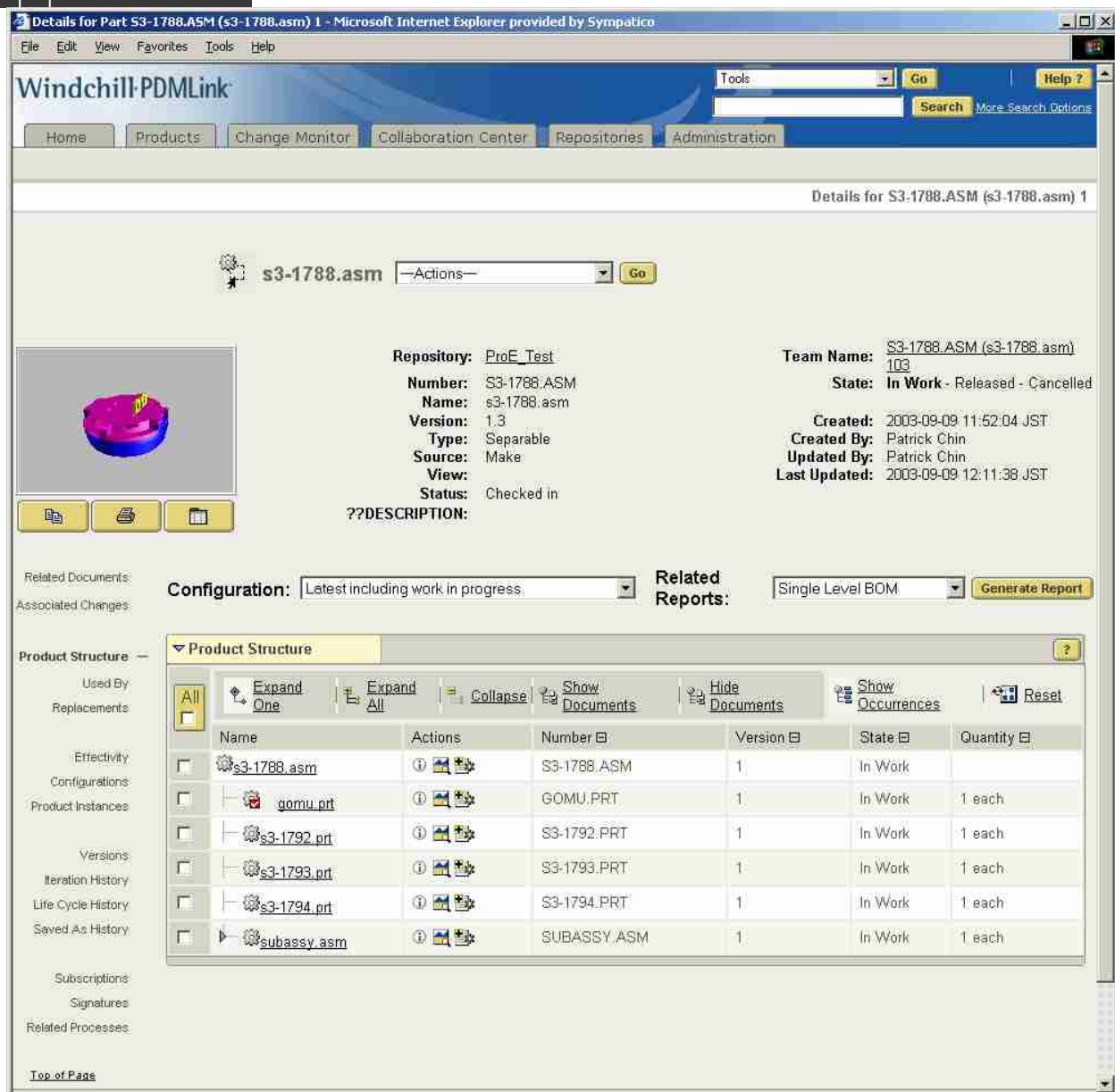
```
template_designasm wtpub://plm.ad.pSION.com/Library/ProE_Stds_Library/ startasmwildfire.asm
```


5 Customizations to PDMLink Server

Currently by default, PDMLink assigns the PDMLink number and name system attributes of the WTPart and CAD document to the Pro/E file name with the extension.



As a result, the product structure or bill of material will be displayed in the WTPart assembly as shown below:



To avoid this default behaviour, small customizations are added to the PDMLink codebase. To ensure Pro/E standard library repositories, products and templates have to be created in PDMLink. Some 'ini' files are modified to ensure that the Pro/E clients can create parts and drawings from these files. These repositories and products are required to store Pro/E start part/assembly and drawing format templates. Pro/E symbols.

All of the customizations are taken from the Windchill® Customizer's and Administrator's Guide for Pro/ENGINEER® Wildfire™ for Release 6.2.6. Before any customizations take place, a backup of the entire PDMLink installation directory must occur. Also, an export of the PDMLink Oracle user schema is required for backup purposes. PDMLink MethodServer, ServerManager and Tomcat must be shutdown prior to performing the customizations. It is best practice to delete the Tomcat default cache directory which is located in %TOMCAT_HOME%\work\ directory.

5.1 Automated Part Creation

Automated WTPart creation (or Autopart) is a mechanism for automatically creating WTParts, and links to them, for one or more selected CAD documents. Automatic WTPart creation can be invoked either by clicking the workspace **Auto Create Part** icon, or by checking the **Create or Associate Docs to Parts** box on the **Options** page during checkin. The Autopart feature in Windchill Release 6.2.6 supports the delegate class mechanism. The customizers are

given full control over the naming and numbering of the WTPart being created. The same applies to direct check in (provided the option to create and associate WTParts to documents is selected), which invokes the same method to auto-create the part. To customize the Autopart feature perform the following steps:

1. Create a class that implements the `com.ptc.windchill.cadx.autopart.AutoPartCreator` interface.

Note: The only method the `AutoPartCreator` interface has is:

```
public WTPart createAutoPart( EPMWorkspace wsp, EPMDocument doc ) --
throws WTEException, WTPROPERTYVetoException;
```

2. Implement this method in the customized class.
3. Modify the `autopart.ini` preference file located at `WT_HOME\codebase\com\ptc\windchill\cadx\cfg\site` accordingly.

For example, the default entry in the file is as follows:

```
[general]
autoPartCreator=com.ptc.windchill.cadx.autopart.DefaultAutoPart
Creator
```

You can specify the custom class name as the value of key `autoPartCreator` here:

```
[general]
autoPartCreator=ext.wildfire.autopart.NoExtensionAutoPartGenerator
```

The `autoPartCreator` preference is applicable to the entire site. Example:

Note: The following example creates WTParts that have the same name as the source CAD Document, but without the file extension (for example, A Pro/ENGINEER model called 12345.prt would yield a WTPart called 12345)

```
// AutoPartGenerator for Wildfire
// Part Number = associated EPMDoc number /wo Pro/E extension
package ext.wildfire.autopart;
import com.ptc.windchill.cadx.common.WTPartUtilities;
import wt.epm.EPMDocument;
import wt.epm.EPMDocumentType;
import wt.epm.workspaces.EPMWorkspace;
import wt.fc.EnumeratedType;
import wt.part.PartType;
import wt.part.WTPart;
import wt.util.WTEException;
import wt.util.WTPROPERTYVetoException;
```

```

public class NoExtensionAutoPartGenerator implements
com.ptc.windchill.cadx.autopart.AutoPartCreator
{
private final int VERBOSE_LEVEL=9;
public NoExtensionAutoPartGenerator()
{
}
public WTPart createAutoPart(EPMWorkspace epmworkspace,
EPMDocument epmdocument)
throws WTEException, WTPROPERTYVetoException {
String epmNumber = epmdocument.getNumber();
String epmName = epmdocument.getName();
printMsg(1,"NoExtensionAutoPartGenerator.createAutoPart:
EPMname="+epmName+" EPMnumber="+epmNumber);
EPMDocumentType epmdocumenttype =
EPMDocumentType.toEPMDocumentType("CADASSEMBLY");
PartType parttype;
if(epmdocument.getDocType().equals(epmdocumenttype))
parttype = PartType.SEPARABLE;
else
parttype = PartType.COMPONENT;
String epmNumberNoExt = stripExtension(epmNumber);
String epmNameNoExt = stripExtension(epmName);
printMsg(1," --> Creating new part:
Name="+epmNameNoExt+" Number="+epmNumberNoExt);
return WTPartUtilities.createNewPart(epmNumberNoExt,
epmNameNoExt, parttype, null,
epmworkspace);
}
private String stripExtension(String name) {
int k = name.indexOf(".");
String nameWoExt;

```

```
    if (k > 0)

    return name.substring(0, k);

    else {

    printMsg(0, "*** EPMDoc without extension....");

    return name; // well, just return name

    }

    }

    private void printMsg(int level, String s) {

    if (level<=VERBOSE_LEVEL)

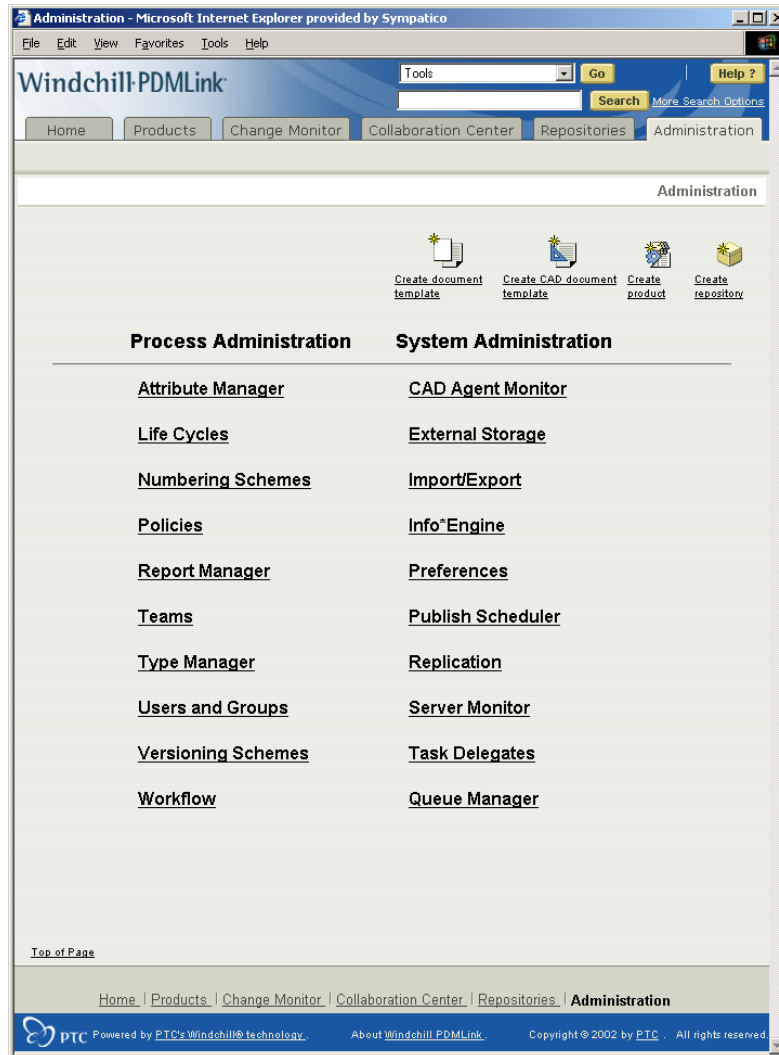
    System.out.println(s);

    }

    }
```

6 Access Controls on Library Domains in PDMLink

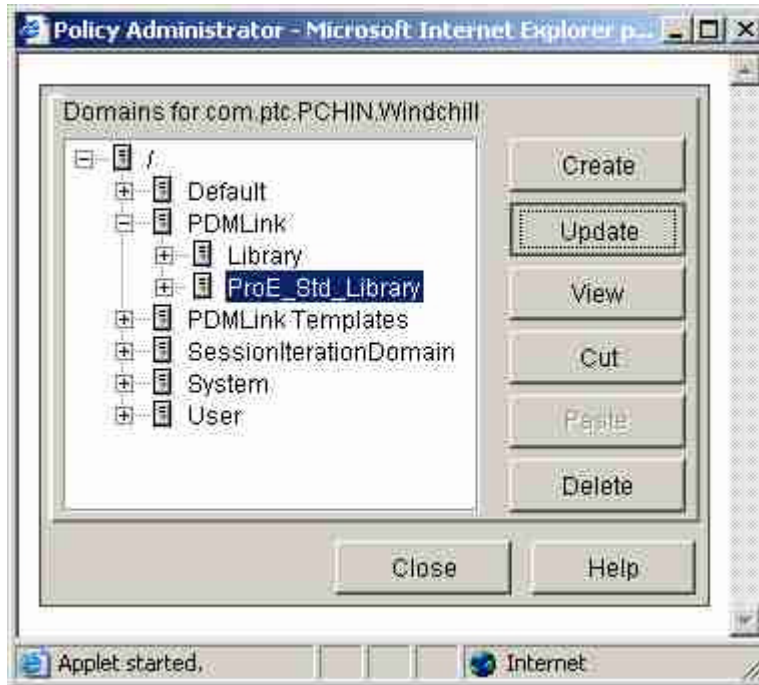
An administrator will be only managed this library container with access controls. In the Administration page under the Administration tab. Select the Policies hyperlink.



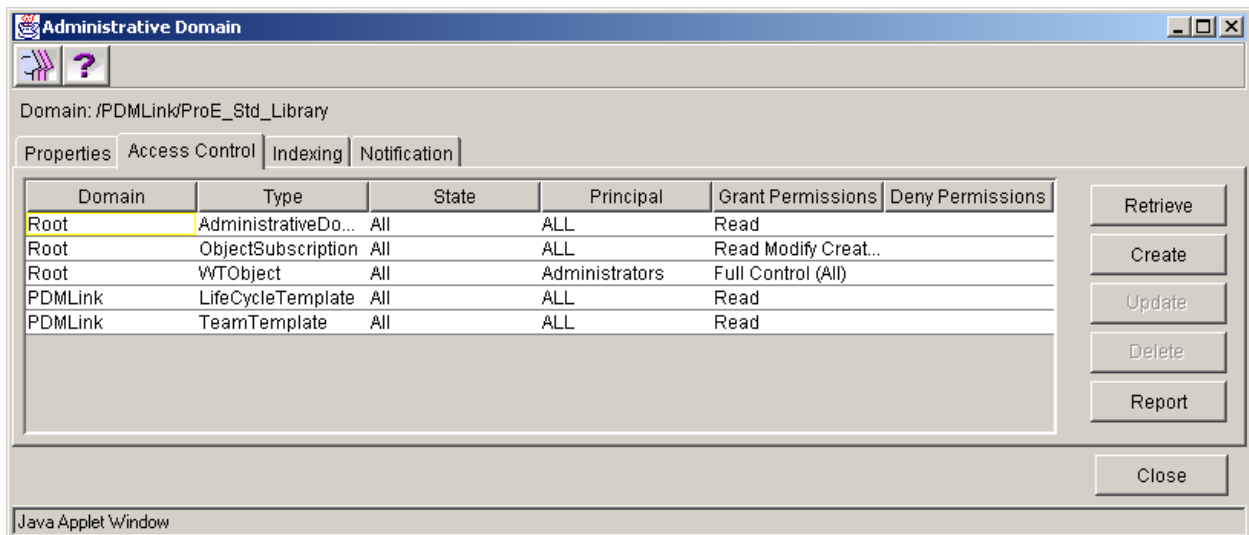
1. A login applet and a Policy Administrator applet imbedded in web page will appear on the desktop window. (allow the downloading of the client jar files for better performance and synchronizing with PDMLink classes on the server) Log on as the PDMLink administrator.



2. In the Policy Administration web page, expand the PDMLink Domain to display the sub-domains. Select the ProE_Std_Library that you just created, then select the Update button.



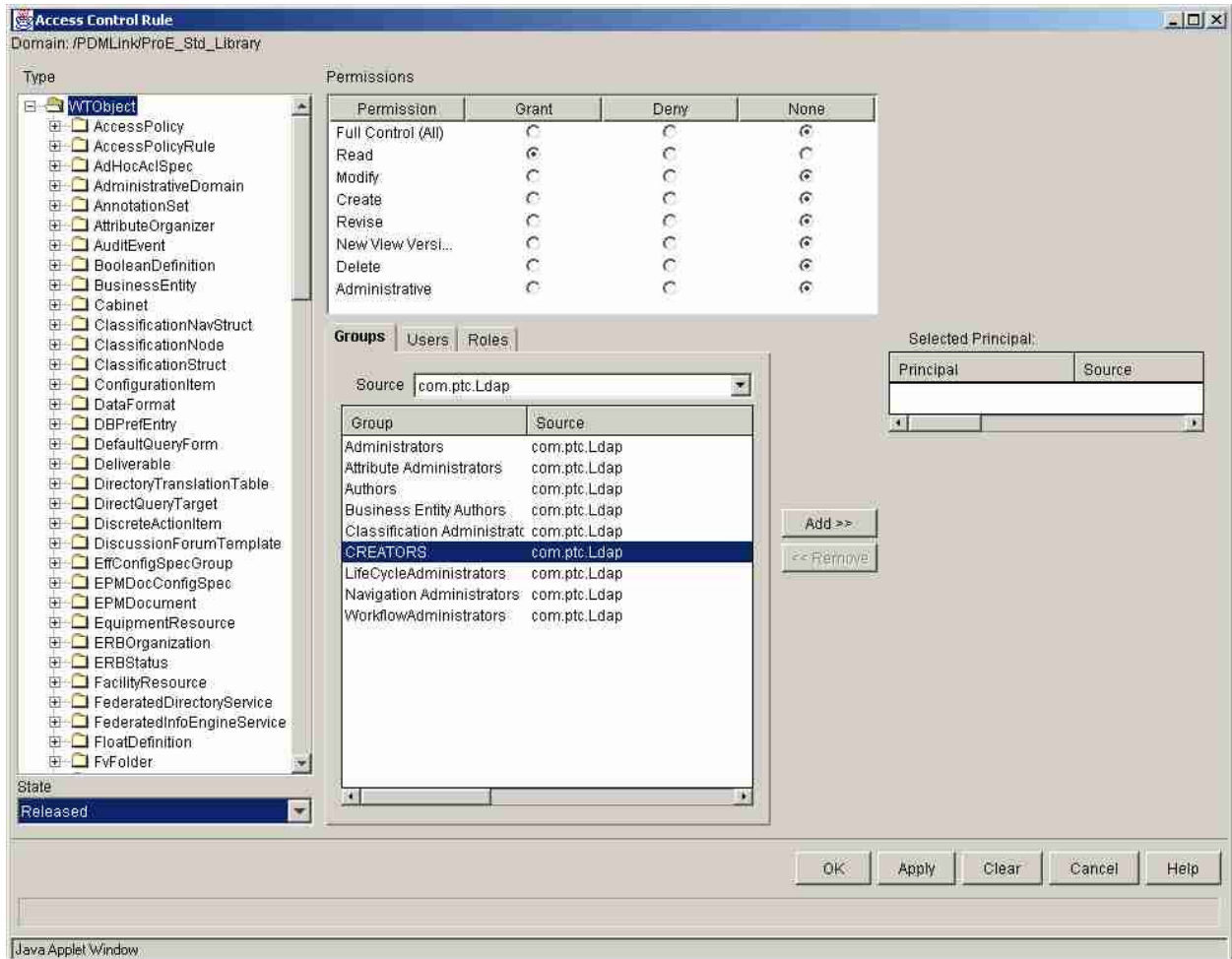
3. An Administrative Domain applet will appear and select on the Access Control Tab. To ensure that all access controls are displayed, select the Retrieve button. To allow only administrative rights for modifications to PDMLink objects stored in this domain, create a new Read access for all Pro/E end users. Select Create button.



4. An Access Control Rule Window will appear. Create a read Access Control Rule for all non administrators with the following settings:
 - a. Object Type: WTOBJECT
 - b. Lifecycle State: Released
 - c. Access Permission: Grant column – Read selected



d. User Groups: Non Administrators.

Note: A separate access control rule must be created for each group, user (if necessary) and role (if necessary). Select Apply button for each new rule except for the last. For the last access control rule, select OK button and the applet will save and close. When Apply button is selected, the applet will remain open. Thus, to close the applet, the Cancel button must be selected.



7 Best Practices with the Current PDMLink Functionality

The current PDMLink 6.2.6.06 architecture and functionality requires administration control and proper methodology to effectively use the Pro/E integrated with PDMLink.

1. Access controls to restrict creation of CAD document templates. As an administrator, CAD Document Templates can be created very easily by selecting the create CAD Document Template  in the administration page. Since CAD documents templates cannot be seen in through the Pro/E configuration tool interface, nor can they be found through the search function . This search function is found at the top right corner of the PDMLink front page.
2. Check In only to Pro/E repositories for WTPart and all CAD documents.
3. Read only access for Non-Pro/E users in Pro/E repositories.
4. Only use Pro/E configuration tool interface to modify the config.pro options
5. Have all Pro/E standard CAD documents in Lifecycle state released.
6. Build a library of common components in a Pro/E library.
7. Ensure that Oracle tablespaces and external file faults have sufficient space for large amounts of metadata and content respectively. Best practice is take the worst case scenario which is:
 - N - number of users
 - M - max number of modifications per revision.
 - R – max number of revisions
 - P – current number of products
 - S – maximum data size of a complete product (megabytes or gigabytes)
 - F – maximum products produced in a year
 - SF – safety factor (2 recommended)

External vault size = N x M x R x P x S x F x SF