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About this guide
This document will provide detailed information for the proper implementation and usage of the Enterprise PDM Pro/E Connector. The Pro/ENGINEER CAD application has some major architectural differences than SolidWorks which present unique challenges for individuals deploying and using Pro/ENGINEER with Enterprise PDM. Unlike SolidWorks, Pro/ENGINEER is not a native Windows application and because of this certain methods of opening, saving and accessing files need to be understood. This guide will also help in setting realistic expectations for customers and to determine whether Enterprise PDM is the right application to meet specific customer needs.

Pre-Sales Activity
• Position Statement
Enterprise PDM is a satisfactory data management solution for Pro/ENGINEER customers when expectations are properly set. Use Enterprise PDM to manage legacy projects where Pro/ENGINEER is occasionally used by a minority of overall CAD users to modify an existing design. Pro/ENGINEER user expectations should be that the design files (parts, assemblies, drawings) will be available and under control of Enterprise PDM, can be retrieved for modification within Pro/ENGINEER, and saved back into the repository. Check-in & check-out, version and revision control, property mapping, and file reference management are available in Enterprise PDM. Advanced features of Pro/INTRALINK like managing individual family table instances (configurations) are not available. If the client is looking for advanced features of Pro/INTRALINK for long term use of Pro/ENGINEER they should stay with Pro/INTRALINK. If the client is investing in SolidWorks CAD for long term use and requires a data repository for legacy Pro/ENGINEER files for periodic use, Enterprise PDM may be a viable solution.

• Target Customers
In general, Pro/ENGINEER customers that have no PDM system or that are using Pro/INTRALINK 3.x are better prospects than those who are using PTC’s Windchill. Windchill is a full featured PLM system similar to Dassault Systems Enovia platform and will have features that Enterprise PDM does not. Many customers have multiple sites each with a different CAD system. It is important to make sure that the needs of each site are taken into account. Working closely with one site that has SolidWorks and not engaging with another site using Pro/ENGINEER, will lead to a prolonged sales cycle or rejection.

Here are some attributes of good prospective Pro/ENGINEER customers to consider:
• Objective is to move away from PTC products to SolidWorks
• Has purchased SolidWorks CAD and is using it for new projects
• Will use Pro/ENGINEER on a limited basis for making changes to legacy projects
• Will continue to use Pro/INTRALINK for their Pro/ENGINEER data and Enterprise PDM for their new SolidWorks data

Here are some attributes of prospective customers to avoid:
• Expectation that Enterprise PDM will have all the same capabilities as Pro/INTRALINK
• Expectation that Enterprise PDM will have all the same capabilities as Windchill but at a lower cost of ownership
• Will not consider even looking at SolidWorks CAD to replace Pro/ENGINEER
Finally, if the customer can be convinced to evaluate SolidWorks and Enterprise PDM in a pilot project, it may be possible to build a long term case to move away from PTC products. SolidWorks plus Enterprise PDM will open their eyes to easier to use solutions, better customer experience and overall superior price/performance over their legacy PTC environment.

- **Support and Demonstrations**
  An organization that is going to implement and support Enterprise PDM for customers with other CAD systems besides SolidWorks should have knowledge of the other systems as well as access to the software. It will be very difficult to diagnose problems and suggest best practices if access is limited to going to the customer’s site.

  Another benefit to having direct access to the other software packages is for demonstration purposes. SolidWorks provides videos of the various CAD integrations in Enterprise PDM on the VAR Resource Center. These are good for first look or informational examples but most customers will want to see live demonstrations. There are a number of ways to get access to the other CAD packages for demonstrations before purchasing a full seat:
  - Ask the customer to load a copy of the software on an Enterprise PDM demonstration workstation a few days before the demonstration. Then afterwards uninstall the software. Many companies work with outside consultants and may have policies in place to handle this type of scenario. Access to the customer’s network via VPN may be needed to obtain a license.
  - Borrow a customer workstation and install Enterprise PDM on it. Then remove Enterprise PDM after the demonstration.
  - Use a third party

**Installation and Set Up**

- **Pro/E Connector pre-installation information**

  **Obtaining the Pro/E Connector Installer**
  Beginning with the 2012 release of Enterprise PDM, the Pro/E Connector installation media has been removed from the main installation package. The installers will only be available through SolidWorks Field Services. Please visit the Enterprise PDM product section of the VAR Resource Center for information on how to obtain the installers.

  **Operating system support**
  As of the 2010 SP 4 release the Pro/E Connector supports 32 and 64 bit systems running Windows XP Professional or Windows 7.

  If installing on a Windows 7 system then it is required to turn off UAC (User Account Control) before installing / upgrading the SolidWorks Enterprise PDM Pro/ENGINEER Connector.

  To turn this setting off:
  1) Go to Control Panel, All Control Panel Items, User Accounts
  2) Select Change User Account Control Settings
3) Drag the slide bar all the way to the bottom where it says "Never notify"
Once validated, proceed with the installation / upgrade of the Pro/ENGINEER Connector. Once the installation / upgrade is complete, reset the UAC to Default - Notify me only when programs try to make changes to my computer.
NOTE: failing to turn off the UAC setting before installation / upgrade will prevent the Enterprise PDM client installer to write or update the protk.dat file found underneath the "\Program files" folder. The protk.dat file is read by Pro/ENGINEER at startup to load the Enterprise PDM add-in.

**Pro/ENGINEER versions supported**
SolidWorks Enterprise PDM Pro/ENGINEER Connector currently supports Pro/ENGINEER Wildfire 3, 4, 5 and Creo Elements/Pro 5.0. The Pro/E Connector is compiled against the initial release of each supported version of Pro/ENGINEER. This allows the add-in to be compatible with any Pro/ENGINEER date code post initial release.

Since Creo Parametric is a brand new product, SolidWorks has not made any commitments on supporting it.

**Pro/ENGINEER file type support**
With Enterprise PDM 2010 SP 1 and earlier, the following Pro/E file types are supported by the Pro/E integration:
- Part (.prt)
- Assembly (.asm)
- Drawing (.drw)
- Report (.rep)
- Layout (.lay)
- Format (.frm)
- Sketch (.sec)
- Markup (.mrk)
- 2D Schematic Diagram (.dgm)

Starting with Enterprise PDM 2010 SP 3 and higher, additional support has been added for the Pro/E Manufacturing file types:
- Manufacturing (.mfg)
- Tool Path (.tph)

**Pro/ENGINEER installation requirements**

**Prerequisites**
For EPDM 2012 or later, obtain the separate installation package from SolidWorks Field Services. There are two installers one for 32 bit systems and another for 64 bit systems.

**Local machine requirements**
Pro/ENGINEER must be installed on a local disk (e.g.: C:\Program Files\PTC\proeWildfire 3.0) and not off a network folder. The Pro/E Connector installer does not support network installation of Pro/ENGINEER or multiple versions on the same workstation.
Pro/ENGINEER bin folder path must be present in the PATH environment variable. This should be done automatically when installing Pro/ENGINEER using PTC.Setup, but if this path is missing in your PATH environment variable, Enterprise PDM will not function properly. SolidWorks recommends verifying this (you may use a DOS prompt and execute the command **set path** to verify this).

E.g.: \PATH = C:\program files\ptc\proeWildfire 3.0\bin

**Recommended Pro/ENGINEER configuration settings**

Pro/ENGINEER uses a separate text based files named Config.sup and Config.pro to control how the program is configured. The Config.sup is sometimes used in order to control options that a company doesn’t want users to change. When a new session Pro/ENGINEER is started the configuration files are read in the following order for Windows based systems:

- Config.sup in the `<PROE_INSTALL_DIR>/text`
- Config.pro in the `<PROE_INSTALL_DIR>/text`
- Config.pro in the startup directory. This is the current or working directory when Pro/ENGINEER starts. This is the last directory to be read, so placing a config.pro file here will override any conflicting settings from the previous directories. The config.sup settings are not affected. File settings in this config.pro file override all other previous config.pro settings.

The options that are set can be viewed and edited from within Pro/ENGINEER by choosing the Tools/Options menu. Changes made while in session need to be saved to the config.pro file or they will be lost after closing the application. The config.pro file can also be edited by launching notepad and opening the file in `<PROE_INSTALL_DIR>\text\config.pro`. The options are set by typing the option name followed by a space then the value (e.g. `create_drawing dims only yes`). The following option settings are recommended for Enterprise PDM:

Option name: **web_browser_homepage**  Recommended value: **about:blank**
Syntax in Notepad: `web_browser_homepage about:blank`
Increases the performance and reliability of the Pro/E Connector add-in by eliminating the need to load a default website in the internal browser pane.

Option name: **save_drawing_picture_file**  Recommended value: **embed**
Syntax in Notepad: `save_drawing_picture_file embed`
Embeds an image of the drawing so that viewing programs like ProductView Express can display the drawing contents.

Option name: **save_model_display**  Recommended value: **shading_lod**
Syntax in Notepad: `save_model_display shading_lod`
Saves the shaded image information in the file to enable viewing of the model in programs like eDrawings and ProductView Express.
The following configuration option settings can affect the behavior of Enterprise PDM. An asterisk by the value indicates the default value if the option is not specified:

Option name: **create_drawing_dims_only** Possible values: yes, no*
Setting this to “No” means that dimensions created in drawing mode are added to the solid model file. If the solid model is not checked out in Enterprise PDM, then the drawing dimensions will be lost. Setting this option to “Yes” will create associative draft dimensions in the drawing so the solid model does not need to be checked out. Also, if multiple drawings of the same part or assembly exist, strong consideration should be given to setting this to “Yes”, because deleting a dimension in another drawing would remove it from every drawing.

Option name: **let_proe_rename_pdm_objects** Possible values: yes, no*
For previous users of Pro/INTRALINK, files coming from that system are tagged so that renaming can only be done in Pro/INTRALINK. This can cause errors when renaming these files in Enterprise PDM. Setting this option to “Yes” will allow the files to be renamed properly. If the files were never in Pro/INTRALINK, then this setting will have no effect.

Option name: **rename_drawing_with_object** Possible values: both, part, assembly, none*
If set to both, part or assembly, then anytime a part and/or assembly is renamed using the Pro/ENGINEER File menu option Save A Copy, the drawing will be renamed as well. In order for this to work, the drawing and the associated model file(s) need to be in the same folder and have the same name (i.e. My_ProE_Part.prt and My_ProE_Part.drw). It is recommended to set this to “none” or to delete the option entirely.

Option name: **override_store_back** Possible values: yes, no*
Enables saving files to an alternate location if the user does not have write access to the directories where the files were opened from. This is used in conjunction with the “save_objects_in_current” option described below. It is recommended to set this to “no” or to delete the option entirely.

Option name: **save_objects_in_current** Possible values: yes, no*
Will save all files modified in session into the current working directory instead of the directory where they were opened from. It is recommended to set this to “no” or to delete the option entirely, since any modified files need to be saved back into the same directory.

**Pro/INTRALINK and multiple startup scripts**
Enterprise PDM relies on the PDMWE_PROE_START environment variable to launch Pro/ENGINEER. By default, the Enterprise PDM client installer will create this new environment variable and set it to:

```
PDMWE_PROE_START = <PROE_INSTALL_DIR>\bin\proe.exe
```

Example:
```
PDMWE_PROE_START = C:\Program Files\PTC\proeWildfire 3.0\bin\proe.exe
```
If the launch command starts Pro/ENGINEER with Pro/INTRALINK client loaded or prompts user with a dialog to select a Pro/ENGINEER startup script, then Enterprise PDM will not be able to authenticate to the Pro/INTRALINK server or select a startup script, because Pro/ENGINEER is running in background mode. Consequently, it is very important to check how Pro/ENGINEER starts by executing the following command in a DOS prompt (including quotes): “%PDMWE_PROE_START%”

This command will start a new Pro/E session. Make sure that it does not prompt the user with any dialog asking to authenticate or select a startup script. If it does, a startup script (i.e. batch file) to launch Pro/ENGINEER will be needed that doesn’t require any input from the user.

Example:
PDMWE_PROE_START = C:\Program Files\PTC\proeWildFire 3.0\bin\proe_start.bat

- Enterprise PDM Vault Requirements

Creating a new Pro/ENGINEER capable vault
Using the Administration client, right click on an archive server and select create new vault. Select the desired predefined vault or one from a .cex file in the Configure vault dialog.

The predefined vaults that come with EPDM 2012 no longer have the Pro/E Connector components and are supplied with the 2012 Pro/E Connector installation.

Installing the 2012 Pro/E Connector and Server Components
Obtain the appropriate Pro/E Connector installation package (32 and/or 64 bit) from SolidWorks Field Services. It is recommended, but not necessary, to use the same service pack for the Pro/E Connector as the EPDM 2012 client service pack. On a client machine that has Pro/ENGINEER
installed, run the Setup.exe file from the Pro/E Connector installation files and follow the screen prompts.

After the installation has completed, there will be a new folder under the Enterprise PDM program folder named “ProE Connector server components (E.g. C:\Program Files\SolidWorks Enterprise PDM\ProE Connector server components). In this folder is an Enterprise PDM export file named “ProE Connector.cex” that contains the AddRenamer add-in (explained below), along with the default Pro/ENGINEER file data cards and variables. Open the Enterprise PDM Administration tool and log into the new vault. Right click on the top vault node and select Import and browse to the ProE Connector.cex file. Optionally, the export file can be opened in the Administration tool using the File>Open menu selection and the components added via drag and drop.
Reviewing vault contents
Verify that the AddRenamer has been correctly added by logging in as admin to your vault in the Administration client and expand the Add-ins node.

NOTE: The AddRenamer add-in is used to strip off the numeric version number from Pro/ENGINEER file extensions (.1, .2, etc)

NOTE: The AddRenamer add-in is only called when dragging and dropping files to a vault folder or copying and pasting from Windows Explorer. If performing a File, Save from a Pro/ENGINEER session, the Pro/E Connector add-in code will strip of the numeric version extensions, not the AddRenamer DLL.

Verify that the four Pro/ENGINEER related file data cards are present:
Verify that Material card variable has been set correctly for Pro/ENGINEER integration.

![Enterprise PDM Variable definition dialog]

**Installing local views**

Enterprise PDM local views should always be installed under the same root folder on all client machines. For instances having two client PCs where one has the local view under D:\MyProEVault and the other has the same local view but under C:\MyProEVault is not recommended. All views should exist under C:\MyProEVault instead on all client PCs. Not respecting this simple rule will make search path management more difficult to maintain (refer to the section “Reference Files/Search Paths” for more information).

**Enterprise PDM local view folder structure**

It is best not to use spaces in folder paths (this includes the vault name itself) as Pro/ENGINEER does not always interpret spaces correctly. For instance, if a “search_path_file” option is used in the config.pro file to point to the search.pro file, and search paths are listed with space characters, the paths will not be understood by the Pro/ENGINEER session. This can be remedied by enclosing the
path in quotes (i.e “C:\Enterprise PDM Vault\My Parts”).

• **Pro/E Connector client installation**

**Installing the Enterprise PDM 2012 Pro/E Connector**
Before installing the Enterprise PDM 2012 Pro/E Connector, make sure the Enterprise PDM CAD Editor client application is already installed and the separate Pro/E Connector installation software is available. In addition, make sure Pro/ENGINEER is already installed and licensed properly. Make sure the version of Pro/ENGINEER installed is also supported by the Enterprise PDM Pro/E Connector.

The Enterprise PDM Pro/E Connector installer will detect which version of Pro/ENGINEER is installed and copy the corresponding version specific files to the Enterprise PDM client installation folder.

NOTE: If multiple versions of Pro/ENGINEER are loaded on the client workstation, the Enterprise PDM client installer will only install the Pro/E Connector add-in against the highest available version of Pro/ENGINEER. For instance, if client workstation has Pro/ENGINEER Wildfire 3 and 4 installed, the Pro/E Connector will only be installed and functional against Wildfire 4.

The client installer will create two new environment variables: PRO_COMM_MSG_EXE and PDMWE_PROE_START.

Once installation is complete, always verify that those two variables have been correctly created. You can use a DOS prompt and check their values by running:

```
set PDMWE_PROE_START
set PRO_COMM_MSG_EXE
```

Because the installer creates two new environment variables, it is always safer to reboot the client PC at least once after the install is complete. Do this regardless if the installer prompts for a reboot or not.

**Pro/ENGINEER Enterprise PDM add-in (PTCADDIN)**
When the Pro/E Connector is installed it allows add-in support in Pro/ENGINEER (i.e. a new Enterprise PDM menu will be available in Pro).

This add-in is loaded in Pro/ENGINEER session as an auxiliary application. The add-in launch command is written to the Pro/ENGINEER protk.dat file found under `<PROE_INSTALL_DIR>\text\protk.dat`.

Below is the content of that file for a Wildfire install.
NOTE: If the protk.dat file has other entries, the Enterprise PDM client installer will only append the file, leaving whatever exists intact.

Windows file associations
If SolidWorks is installed after Pro/ENGINEER on the same workstation, the common Pro/ENGINEER file extensions of .prt, .asm, and .drw, will get associated to SolidWorks. This is due to SolidWorks files once using the same extensions before the “sld” was added. If this is the case, make sure the Pro/ENGINEER file extensions are properly associated by running PTC.Setup one more time in repair mode to re-associate the .prt, .asm and .drw file extensions to Pro/ENGINEER.

- Additional installation information
  Workflow transitions can be configured to update card variables and Pro/ENGINEER parameters. This requires executing the transition on a computer where Pro/ENGINEER is installed and licensed because Enterprise PDM will need to launch Pro/ENGINEER to write to the parameter(s) to the Pro/ENGINEER file.

For card variables, the mapping between a card variable and a Pro/ENGINEER parameter is done via the CustomProperty block (the same for SolidWorks files).
The Revision card variable by default already uses the CustomProperty block named “Revision”. Edit this variable to add the Pro/E file extensions prt, asm, and drw as needed to the list.

![Enterprise PDM Workflow Transition Properties dialog](image)

The same principle can be applied to all other variables.

**Default preview application**

By default, Enterprise PDM Pro/ENGINEER integration will use the eDrawings Viewer as the default visualization tool for previewing Pro/ENGINEER files. The eDrawings Viewer can view Pro/ENGINEER part (.prt) and assembly (.asm) files natively but not drawings. The Enterprise PDM preview can preview Pro/ENGINEER drawings by first converting them to DXF format. The DXF file is then shown in the Enterprise PDM preview tab.

NOTE: This functionality requires having Pro/ENGINEER loaded on the workstation where previewing takes place. Converting the .drw file to DXF format requires loading Pro/ENGINEER in a background session so the drawing can be opened and converted to DXF. This conversion requires the presence of a user writable folder named “C:\Temp”. If this folder is not present it can be easily created.

NOTE: Depending on the complexity of the Pro/ENGINEER drawing, conversion and previewing may take an extended period of time.

If unsatisfied with eDrawings viewing capability, it is possible to configure Enterprise PDM to use ProductView Express as the default viewer application for Pro/ENGINEER files. The latest version of ProductView Express (9.1) is not built into Internet Explorer like the previous 4.0 version. With the 4.0 version it is possible to use the html preview capabilities in Enterprise PDM to view Pro/ENGINEER files in the Preview tab. If the older version is not available, the latest version still can be incorporated into the
“View/Markup” menu option. Check with your local SolidWorks technical resource about obtaining the older version. Please refer to Knowledge Base solution S-041007 for more detailed information.

Search paths
Pro/ENGINEER files do not store the full path information for the files they reference. Thus search paths must be used to ensure all reference files are found. For more information on how to account for this, please see the following sections on data migration and using Pro/ENGINEER with Enterprise PDM.

- Data Migration

Planning for data import
Prior to running the import, it is best to create the folder structure beforehand in Enterprise PDM. Having the folders already present in a local view will help in setting the Pro/ENGINEER search paths accordingly before running the import.

NOTE: There is an add-in Enterprise PDM program called the “ProESearchPathUpdater” that can be used to update the Pro/ENGINEER search.pro file in a more convenient way. This add-in is offered As-Is (not officially supported), please refer to Knowledge Base Solution S-044329.

Pro/ INTRALINK 3.x
If migrating from Pro/INTRALINK, a solution partner Transcat has a readymade export tool to extract Pro/ENGINEER data from Pro/INTRALINK 3.x. Newer versions of Pro/INTRALINK are not supported by this export tool. The import part of the migration is done using the new XML importer tool. See Knowledge Base Solution S-043140 for instructions on how to get a copy of this importer utility.

Other PDM/PLM systems
The same import can be used to bring data into Enterprise PDM from other data management systems. The prerequisite is to export a properly formatted XML file from the source system and to have access to the source files. Creating the XML file and preparing the files for import requires specific knowledge of the source system. Many of the SolidWorks PDM service partners can provide the expertise needed.

Windows folder system
Files can be simply copy and pasted or drag and dropped from a Windows file system into Enterprise PDM, but careful planning is required to make sure all the reference files are present and the search paths are properly specified before copying any files. Because the Pro/ENGINEER connector starts a background session of Pro/ENGINEER when you drag and drop Pro/ENGINEER files to the vault, the more files you drag and drop, the more files Pro/ENGINEER will have to load into memory. It is therefore recommended to copy smaller groups of files or folders at a time. Once all the files have been added, it is best to destroy all the Pro/E files with the .1, .2 extensions from the Enterprise PDM recycle bin. Those files were automatically deleted by the AddRenamer add-in. RMB Properties on your vault root folder, go to the Deleted
Items tab, tick “include items in sub-folders”, select all Pro/E files with a .1, .2, ... extensions and choose destroy.

NOTE: The client workstation from which you run the import must have Pro/ENGINEER installed and licensed with the search paths properly specified.

NOTE: The File, Backup menu command in Pro/ENGINEER can also be used to copy all opened reference files in a Pro/ENGINEER session into the Enterprise PDM vault.

NOTE: Having many parameters mappings to card variables will noticeably impact the time to add Pro/ENGINEER files to an Enterprise PDM vault.

Using Pro/ENGINEER with Enterprise PDM

- **Pro/ENGINEER Functions and Enterprise PDM**

Pro/Engineer is not a native Windows application like SolidWorks, and because of this, certain methods of opening, saving and accessing files need to be understood.

**Reference Files/Search Paths**

Since Pro/ENGINEER does not store the full path information for referenced files like SolidWorks does, Pro/ENGINEER needs to be “told” where to look for references. Even though Enterprise PDM knows the folder location of files referenced by a Pro/ENGINEER assembly or drawing, Pro/ENGINEER will not find them unless the files are in any of the following locations (shown in order of search):

- In session memory (RAM)
- Directory where the user selected the parent file from
- The specified “Working Directory”
- In the search paths called out in either the config.pro or the search.pro file (as described above)

It is therefore important that a search.pro file be maintained with all necessary folder paths. Please see Knowledge Base Solution S-044329 to get an as-is Add-in that can help maintain the search path information.

**File Open**

Using the standard File, Open command in Pro/ENGINEER, opens a dialog box that does not use the Windows shell like most Windows based applications. Thus, browsing to an Enterprise PDM vault in this dialog will not look or function the same as a native Windows application like SolidWorks. The Pro/ENGINEER open dialog will not show the folder colors or any of the information tabs like Data Card and Where Used. In addition, files that are not cached locally do not appear in the dialog. To solve this, the Enterprise PDM Pro/E Connector menu contains an “Open Model” command which displays a typical Windows open dialog that has the tab information and displays non-cached files. Using this command is the best way to open Pro/ENGINEER files that reside in an Enterprise PDM vault.
Working Directories

Pro/ENGINEER has what is called a “Working Directory” which establishes the default location for new files and as a standard location where reference files are searched for. The location of the Working Directory can be manually set by the user or can be specified in the config.pro file(s). This location can be set to a folder in the Enterprise PDM vault, however there some things to consider:

- The Open Model command from the Enterprise PDM Pro/E Connector defaults to the root folder in the vault and not to where the Working Directory is set.
- Creating new files from the File, New menu will cause a “File not checked out” warning to appear. Clicking OK to the warning will clear the warning but the new file will not appear in the vault until it is saved in Pro/ENGINEER. This warning is sometimes hidden by other windows and will make it look like the system is not responding. Minimizing all the active windows will show the dialog so it can be cleared.

File Save

The standard Pro/ENGINEER save dialog is used to save new or modified files in an Enterprise PDM vault. If the user is not logged in and they browse to the vault, they will receive the log in dialog. However, the same limitations apply with the File Save dialog as the File Open dialog.

Pro/ENGINEER will add a version number to the end of a file that is visible when looking at the file in a normal Windows Explorer view. The version number does not usually display in any of the Pro/ENGINEER file browser dialogs. The Pro/E Connector will, as described above, automatically remove this extension when new files are added or updated from an active Pro/ENGINEER session. If the file is in the vault and not checked out, the save will not happen and Enterprise PDM will issue a dialog indicating the file is not checked out.
Assemble
The Pro/ENGINEER command “Assemble” is used to add existing components to an active assembly file. This command also uses the Open dialog with the same limitations mentioned above. Thus, if a user tries to browse to a file that is not locally cached then they will not be able to see it. The recommended procedure to use the assemble command is to first perform a search in Enterprise PDM and drag the desired file from the results area of the Enterprise PDM Search tool into an active Pro/ENGINEER assembly window. The other option is to get a local copy of the desired file (if necessary) from the results and then use the Assemble command in Pro/ENGINEER and browse to the file.

Parameters
Pro/ENGINEER uses Parameters to store file properties like description and part number. The parameters can be accessed by selecting the Parameters menu option under the Tools menu. Enterprise PDM will read and write to these parameters the same way it does for file custom properties in a SolidWorks file. The parameter names can not contain spaces, thus do not use spaces when defining variable attribute names in Enterprise PDM, use underscores “_” instead.

Variables should be defined using the CustomProperty Block Name in the Enterprise PDM Edit variable dialog. To update title block information by changing variable values in Enterprise PDM, use the note syntax &parameter name (no spaces) in a drawing note to link the note text to a drawing parameter value.

- Example: If the parameter name is “REVISION” and its value is “B”, and “REV &REVISION” is entered in the note text. The note will display “REV B”.

![Pro/ENGINEER Parameters dialog](image)
There is not a way to connect a parameter value from a model to a drawing file parameter then to a variable in Enterprise PDM like there is with SolidWorks using the $PRPSHEET attribute.

**Check In / Check Out**

Files can be checked in or checked out from either Windows Explorer or from the Pro/E Connector menu (described below). If a file is currently in a Pro/ENGINEER session, the file status is updated. Files that are not checked out are in read only mode.

If a drawing file has a drawing format file (.frm) associated to it, Enterprise PDM will recognize this relationship as another reference to the drawing. Thus, when checking in a part or assembly that has this relationship, Enterprise PDM will not show the drawing reference. This is the same behavior when a SolidWorks drawing, references more than one part or assembly file.

![Enterprise PDM check in dialog for a Pro/ENGINEER assembly that also has a drawing file associated to it and the drawing also references a format file.](image)
NOTE: SolidWorks does not recommend enabling the “Show this file type as “sub parent” (drawing)” setting for .prt and .asm files.

Family Tables
Family Tables in Pro/ENGINEER are similar to Design Tables in SolidWorks. In Pro/ENGINEER each version is called an instance which is analogous to a SolidWorks configuration. Enterprise PDM will create a data card tab for each instance and instance specific properties will be read into the corresponding data card fields.
Enterprise PDM does not recognize the relationships between instance accelerator files (.xpr and .xas) and the source file. The purpose of these files is to improve performance when retrieving multiple instances of the same file from disk. Use of these files is not common. They can be stored in an Enterprise PDM vault and manually associated to the parent file by using the Copy/Paste as Reference function. Enterprise PDM cannot rename the instances when copying files using the Copy Tree function. The instances need to be renamed after the Copy Tree action is finished.

Since Enterprise PDM does not create separate objects for SolidWorks configurations or Pro/ENGINEER family table instances, automatic instance specific revisions are not possible. It is possible to create an instance specific property to hold a revision value that is incremented manually when changes are made.

**Pro/ENGINEER File Templates**

If the Pro/ENGINEER default template files are placed in the vault, they need to be locally cached in order for Pro/ENGINEER to find the files. There is an As-is application that can be used to automatically get the latest version of files in a specified directory using Windows Scheduler to initiate the refresh. Please refer to Knowledge Base Solution S-030237.
Merged/Inherited Geometry

Pro/ENGINEER can reference other parts (design parts) into other part files similar to the way SolidWorks uses derived parts. Enterprise PDM will recognize the relationships created by this action and will show the design part as a child to the referencing part.

User Defined Features (UDF’s)

UDF’s are reusable features that can be linked to the original feature so if an update to the source occurs, any model using it will also update. There are two types of UDF’s called a Subordinate UDF’s and a Standalone UDF’s. The Subordinate type is smaller in size but requires the original source model to get some of its information. The standalone contains all the information from the source file and thus is larger in size. The file extension for UDF’s is "gph" and also contains the version extension. Thus a UDF file might be called square_hole.gph.1. There are some limitations with UDF’s being stored in Enterprise PDM and their relationships to part files.

- The AddRenamer DLL does not currently recognize the .gph extension and will not automatically strip off the version number and add it to the vault. Therefore any UDF’s that are to be stored in the vault would need to have the version extension stripped off manually.
- Local copies of vaulted .gph files would be required in order for Pro/ENGINEER to find them. The As-is application mentioned it the File Templates section (S-030237) can be used to refresh the local cache.
- Subordinate UDF's create a cyclic reference in Enterprise PDM to the source file they were created from, if the option to "Make features dependent on dimensions of UDF" is selected. This inhibits the file from being checked in the Enterprise PDM.
- Standalone UDF's with the above option selected are recognized correctly by Enterprise PDM. If the "Make features dependent on dimensions of UDF" is not selected on either type of UDF, Enterprise PDM does not show any reference. This would be correct since the feature is not dependent on any outside file.
- The .gph files would need to be included on a workflow in order to check the files into the vault. Data cards may be established for storing data that can be searched on, but no parameter or property information can be derived from the files.
Shrinkwrap
The Shrinkwrap functionality in Pro/ENGINEER allows a complex assembly to be represented by a single part. It is somewhat similar to Speedpak in SolidWorks. The relationship between a part file that contains a Shrinkwrap feature and the source geometry file is not automatically recognized by Enterprise PDM. Source geometry file(s) that are not locally cached will not be visible in the Pro/ENGINEER Open dialog displayed when the user edits the definition of the Shrinkwrap feature to update geometry changes.

Skeleton parts
Skeleton parts are used to represent common geometry between components in an assembly. Enterprise PDM recognizes the relationship between skeleton parts and the referencing assembly.

External Simplified Representations (Simplified Reps)
Simplified Reps are used to store simplified versions of an assembly without modifying the original assembly. They are .asm files that reference another .asm file and have various components and features excluded. Enterprise PDM recognizes the relationship between a Simplified Rep assembly and its source assembly.

- **Enterprise PDM Pro/E Connector Functionality**

Pro/E Connector Menu
Once the Pro/E Connector is configured and working as described in the Installation and Set Up section, the following commands are available from the Enterprise PDM Pro/E Connector Menu in Pro:

- **Open Model...**
  Opens a Windows based dialog that will properly interact with a local view. Folder colors and information tabs are visible along with right click menu options. The dialog filters out any non Pro/ENGINEER files.

- **Get Latest Version**
  Gets the latest checked in version of the file in the active Pro/ENGINEER window that the user has access to. If edits were made to the file before performing this action and it was not first checked in, performing this operation will replace the current file and the edits will be lost.

- **Check Out**
  Performs check out operation on the current file in the active Pro/ENGINEER window. If edits are made to a file that is not checked out, the changes will be maintained and can be saved into the vault by doing a check in. The only way to see whether a file is checked out from within the Pro/ENGINEER interface is by viewing the Tree as described below. There is no “Read Only” indication on the program title bar.

- **Check In...**
  Performs a save and check in operation on the current file in the active Pro/ENGINEER window.

- **Undo Check Out...**
  Perform an Undo Check Out operation and any changes will be lost. A warning dialog appears asking the user if they want to continue.
• **Properties...**
Displays the file data card for the file in the active Pro/ENGINEER window. If the file is checked out, the card can be modified and new property values will be written to the file parameters. The appropriate variables and attributes need to be defined on the data card in order for the changes to show up in the parameters list.

• **View Tree in Browser...**
Displays a reference tree view of the file in the current active window in the Browser fly out dialog. Columns of information are shown that contain the following information: File name, Version, Checked out By, State, Child Quick info and Description. These columns cannot be moved or modified. The icons that appear are inactive except for the folder icon which will display the Open dialog.

• **View Tree in Dialog...**
Displays a reference tree view of the file in the current active window in a separate dialog box.

• **Enterprise PDM Functions with Pro/ENGINEER Data**

**Templates**
The Template function in Enterprise PDM can be used to create Pro/ENGINEER parts, assemblies and drawings. It is not recommended to initiate a new template from within the Pro/E Connector Open Model dialog. If the template is set to show the data card and the user chooses Open File, then a new session of Pro/ENGINEER will be started and may lock up the original session.

**Rename**
Pro/ENGINEER files can be renamed in Enterprise PDM as long as the user has access to a Pro/ENGINEER license. Enterprise PDM will open the files in a background session in order to update file references. Part and/or assembly files referenced by drawings can also be renamed using the Pro/ENGINEER application as follows:
- Search all drawings associated with the parts and/or assemblies to be renamed (the Where used tab will provide this information)
- Open part and all associated drawings in Pro/ENGINEER
- Check out all files
- Use Pro/ENGINEER file rename to rename the part and/or assembly files
- Save all files and check in

**Copy Tree**
The Copy Tree command can be used on Pro/ENGINEER files. It is highly recommended to have an active session of Pro/ENGINEER with no models loaded in session or RAM when using the Copy Tree command.

**Move**
Pro/ENGINEER files can be moved and retrieved properly by Pro/ENGINEER as long as the new location of the file is in the search locations.

**Data Card Editing**
Data cards can be edited and the new values pushed into the Pro/ENGINEER file as long as the user performing the edit has access to a license of Pro/ENGINEER installed
locally. Enterprise PDM will open the file in a background session of Pro/ENGINEER and update the parameters.

**Change State**

Users can change state on Pro/ENGINEER files without having access to a license of Pro/ENGINEER as long as the transitions do not write variables to the files. If the transition does write a variable, like setting a new revision, the user performing the state change does need access to a license of Pro/ENGINEER installed locally. Enterprise PDM will open the file in a background session of Pro/ENGINEER and update the parameters with the new variable values.

**SolidWorks Knowledge Base**

Before engaging with a potential Pro/ENGINEER customer check the Knowledge Base and search on the terms Enterprise and Pro/e. These terms will be common to the articles relating to the Enterprise PDM Pro/E Connector. Having this information prior to performing any demonstrations or implementations can help set proper customer expectations. Articles can be sorted based on when they were added to the Knowledge Base so it is easy to see anything new since the last search.