

User Guide

PTC Creo Parametric integration
for
SOLIDWORKS PDM

Valid for product version: 2024 SP1.0 (2024.1.0)

Published: 19.03.2024 | Build: 505 | Revision: a0f92c4fb

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Glossary

Application Programming Interface (API)

Defines a set of routines, communication protocols and tools for building software. In general, they are clearly defined methods for communication between different components.

Bill of Materials (BOM)

Defines a list of assemblies, sub-assemblies, parts and their quantities needed to produce a final product.

BOM position

Defines a position in the BOM with unique identification, name, quantity and other characteristics.

Component Object Model (COM)

Defines a binary-interface standard for software components introduced by Microsoft.

Connector

Defines a central interface component of each Dassault integration. The integration uses connectors for each participating application to exchange data via their API.

Datamodel

Defines objects and their relationships in a PLM system that are managed by the integration to store data from an authoring application.

Dynamic Link Library (DLL)

Defines a file with a library of functions and other information that can be accessed by a Windows program.

Payload

Defines the data contained within an API request. The description is borrowed from the transportation industry, where a truck carries its cargo (its payload) to a location. The truck, as with the API request, is always the same, but the payload changes with each request.

Product Lifecycle Management (PLM)

Defines systems and processes for managing data during the development of a product from creation through manufacturing to maintenance and disposal.

Revision

Defines a released object state in SOLIDWORKS PDM that cannot be modified.

Script engine

Defines the central component in each integration. It contains the integration logic for processing and forwarding the information and data coming from the connectors.

User Interface (UI)

Defines a (usually) graphical interface through which a user interacts with the computer.

Version

Defines an incremental counter of each object modification in SOLIDWORKS PDM on check-in.

x86/x64

Defines the processor architecture in a computer and thus also the performance of applications. x86 corresponds to 32-bit and x64 corresponds to 64-bit.

1 Introduction

1.1 Getting started

The SOLIDWORKS PDM - Creo Parametric integration supports users when working with PTC Creo Parametric data. The integration is directly embedded in *Windows Explorer* as well as Creo Parametric.

SOLIDWORKS PDM offers CAD specific file format support in two ways:

- file format plug-ins and
- CAD add-ins.

The file format plug-in is a DLL file that executes calls for certain CAD file formats in the context of *Windows Explorer* API to read and write properties and file references and to preview managed files.

The add-in is a DLL loaded with a specific application, e.g. Creo Parametric. It offers possibilities to display extended menu options (ribbons, context menu) in the application and to simplify the work with the files.




The supported Creo Parametric formats are:









- Creo Parametric part (.prt)
- Creo Parametric assembly (.asm)
- Creo Parametric drawing (.drw)
- Creo Parametric section/sketch (.sec)
- Creo Parametric formats (.frm)
- Creo Parametric layouts (.lay)

1.2 Add-in

Ribbon bar

The SOLIDWORKS PDM ribbon bar contains all features of SOLIDWORKS PDM and are executable for the loaded root element. The following functions are available:

Function	Title
	Open File
	Check In
	Check Out
	Undo Check Out
	Get Latest Version
	Get Version

Function	Title
	Show Properties
	Update Title Block
	Rename
	Change State
	Search
	Select in Windows Explorer
	File Info
	View Tree in Browser

Context menu

The SOLIDWORKS PDM context menu contains features of SOLIDWORKS PDM and is executable for the selected element. The following functions are available:

- Check out, Check in and Undo check out
- Show Properties (Data Card)
- Rename
- Change State
- Search
- Select in Windows Explorer
- File Info
- View Tree in Browser

2 Functional description of the Creo Parametric add-in

2.1 Open File



Open File opens a browser, which allows selecting files that are

- available on the local disc.
- not available on the local disc, but stored in the SOLIDWORKS PDM database.

2.2 Check In / Check Out

Files can be checked in or checked out from the SOLIDWORKS PDM vault or from the Creo Parametric add-in. The following chapter explains the functions via Creo Parametric.

Check In



With **Check In** changes of the currently checked out object and its structure are saved and checked in to SOLIDWORKS PDM. Changes can affect structure, geometry of parts, appearance and file properties. After analyzing the structure, the SOLIDWORKS PDM *Check In* dialog appears.

In the dialog, the modified objects are displayed in **bolt**. It also shows that the version increases for modified files which are selected for check in. Objects that are already checked in, can only be viewed and changes cannot be saved to SOLIDWORKS PDM. Modifications of a file in status *Checked in* is possible, but it needs to be checked out in order to perform a **Check In** action. Furthermore, the user can add a description about the changes in the *Comment* line.

Check Out



Selecting the **Check Out** command opens the SOLIDWORKS PDM *Check Out* dialog.

In the dialog, the user can select the desired files (including the complete structure). Warnings and error messages are also displayed, e.g. if a user has already checked out a file.

On pressing **Check Out** in the dialog, the selected files are checked out.

Undo Check Out



Undo Check Out removes the reservation of the current user for the corresponding CAD file inside SOLIDWORKS PDM database. The function is used to discard local changes and reload the last checked in version of the object.



Before Check Out, Check In and updateTitleBlock a read-only file will be closed, a MessageBox appears, asking the user if he wants to abort the selected process.

2.3 Load Versions

Use these functions to load desired versions from SOLIDWORKS PDM to Creo Parametric.

Get Version



With **Get Version** a desired version of the selected object can be loaded from SOLIDWORKS PDM. On selecting this function, a history dialog appears showing all available versions, the current local version and the check in comment of the selected version. After confirming the dialog, the *Get* dialog appears. All available versions can be selected and loaded. Usually the files, whose versions have to be replaced on the client, are preselected. When clicking **Get**, the current used version is replaced and the object is reloaded in the selected version.

The function can be used for checked in and checked out objects.



If edits were made to the file and it was not checked in first, performing this operation replaces the current file and the edits are lost.



Please note that files cannot be checked out if they're loaded in older version.

If you want to change a file in older version then execute following workaround:

1. Go to the latest file version.
2. Check out the file.
3. Go to an older version.
4. Make changes.

If you check in the file, then a new file version (latest + 1) is created.

Get Latest Version



The function **Get Latest Version** is similar to the function **Get Version**. The history dialog is not necessary in this case. The files that are available on the client in an older version are marked and preselected. After selecting **Get**, the current used version is replaced and the object is loaded in the latest version.

The function can be used for checked in and checked out objects.



If edits were made to the file and it was not checked in first, performing this operation replaces the current file and the modifications are lost.

2.4 Properties

A property is a piece of metadata that can be exchanged bi-directionally between SOLIDWORKS PDM and Creo Parametric. Properties are usually saved in CAD files and in SOLIDWORKS PDM fields and can be accessed from the Creo Parametric's user interface as well as from SOLIDWORKS PDM masks.

Show Properties



Show Properties displays the *Data Card* for the local version of a file. The data in the Data Card can be changed when the affected file is checked out. The changes also affect CAD file properties if the changed variable is part of the data mapping (refer to *Installation and Administration Guide* for more information) and vice versa.

After the check in, the data of the *Data Card* is stored in the vault.

Some metadata displayed in the SOLIDWORKS PDM may have configuration specific values.

To bidirectional exchange such metadata from Creo Parametric to SOLIDWORKS PDM, make sure that the corresponding Creo Parametric parameter is used within the Creo Parametric family table.

If the current object in Creo Parametric is a drawing, there is the command **Update Title Block**.

This command allows updating Creo Parametric parameter, visible in a title block of a drawing by retrieving the corresponding values from SOLIDWORKS PDM.

Update Title Block



Title blocks of drawings can be automatically populated by using the **Update Title Block** menu item that is only available within the drawing context of Creo Parametric.

This function automatically populates the title blocks of the currently loaded drawing (if it has a title block) with information gained from SOLIDWORKS PDM. This function transports document data of the drawing, related items, referenced models and its existing items.

The **Update Title Block** function is a property exchange and can be configured (please refer to the *Installation and Administration Guide* for details).

2.5 Rename



With **Rename**, Creo Parametric files can be renamed in SOLIDWORKS PDM. Refer to chapter [Renaming](#) (p. 23) for more information.

2.6 Workflow Management

Workflows represent the development process. They define the life cycle of a document, project, or process by specifying the states a document goes through.

Change State



SOLIDWORKS PDM supports work with workflows by default. This allows tasks in the development process to be structured in terms of time, content and logic.

All files in a vault are usually assigned to a workflow. Workflows can be created and configured in the *Administration Tool*.

With the function **Change State**, the user can move the selected element from the initial state to the target state, if he has the necessary rights. To do this, the user receives the transitions available for the initial status via the selection dialog *Select a transition*.

For example, starting from the status **Change Pending Approval**, the two transitions **Change Approved** and **Change Editing Required** are offered. With selecting a status and pressing **OK**, the *Do Transition* dialog appears.

The dialog offers the possibility to select the affected files and to add a comment to the status change. Pressing **OK**, executes the status change.

If rights are missing for the status change, an adequate error message appears.

Changes of the workflow status are only possible for files that are checked in. If the user tries to change the status of a checked out file, the user is prompted to check in the file first. When selecting **Yes**, a check in is executed for the affected files and the state is changed. When selecting **No**, the status change is aborted.

2.7 Explore

In this section, interactions with the Windows Explorer are described.

Search



With **Search** the user has the possibility to start the standard search of SOLIDWORKS PDM from Creo Parametric. This tool can be used to search for files and folders in the vault using various masks, such as the Creo Parametric Search Card. The user can search by file name, for a specific variable value in a data card, by workflow state, user name, etc.

The result list offers the same features as the *Windows Explorer*.

Loading search results into Creo Parametric is possible from here.

If not already logged in, the command shows the SOLIDWORKS PDM login dialog.

Select in Explorer



The command opens a *Windows Explorer* showing the current vault. The file representing the current/selected object is preselected. If not already logged in, the command shows the SOLIDWORKS PDM login dialog.

2.8 Display PLM Information

File Info



Use the **File Info** command to display SOLIDWORKS PDM information of the current/selected object.

The name of the file, which is the headline of the info page, is displayed in

- cyan, if the object is loaded in the latest version.
- red, if the object is loaded in an old version.
- orange, if the object is checked out and locally modified.



If the file is checked out, but not yet modified, the headline is displayed with cyan background color.

Contents of this HTML page can be customized. Refer to the *Installation and Administration Guide* for more information.

View Tree in Browser



Use this command to display SOLIDWORKS PDM information for the current/selected objects and its structure.

The name of the file, which is the headline of the info page, is displayed in

- **cyan**, if the object is loaded in the latest version.
- **red**, if the object is loaded in an old version.
- **orange**, if the object is checked out and locally modified.



If the file is checked out, but not yet modified, the headline is displayed with cyan background color.

The background color of the columns of the table corresponds to above logic too, except that the cyan color is not used, instead standard **grey** column background.

Contents of this HTML page can be customized. Refer to the *Installation and Administration Guide* for more information.

3 Functional description of the SOLIDWORKS PDM add-in

The following use cases explain different possibilities to work with the integration. Some of the functions open Creo Parametric on executing if it is closed.

3.1 Lock

To modify an element, it needs to be checked out. It is possible to select

- a single files
- multiple files
- a single folder
- or multiple folders

for check out. After selecting **Check Out** from the context menu, the *Check Out* dialog appears and displays all selected files with their structure.

3.2 Check In

With the command **Check In** the assembly structure is resolved and created in SOLIDWORKS PDM.

After the check in, all affected files are checked in and the tab *Contains* displays the file structure.

The **Check In** command is available for files and folders and can be accessed via the context menu of the vault. If **Check In** is executed for folders, all contained files are checked in and created in SOLIDWORKS PDM.

Hints for settings in the Administration Tool

In the *Administration Tool* of SOLIDWORKS PDM it is possible to define, which files and folders should be added to the vault during import and which not. Open the *Administration Tool* and navigate to **User > Settings > Adding Files/Adding Folders**.



These settings are relevant for the performance.

3.3 Undo Check Out

Via this function of the context menu, check outs can be canceled.



Any local changes made to the selected element are discarded.

3.4 Edit data card variables

The metadata is displayed within the *Data Card* tab of *Windows Explorer*. On checked out files, the user can modify the metadata. Clicking the **Save** button, SOLIDWORKS PDM stores the changes within the SOLIDWORKS PDM database and also within the Creo Parametric file.

3.5 Get Latest Version

The function **Get Latest Version** of the context menu is used to retrieve the latest version of the current element.

3.6 History

Via the **History** function, the version and workflow transition history of an element can be displayed. The *History* lists comments, revision numbers, creation dates and names of users who modified each version.

This functionality allows retrieving old versions of files from SOLIDWORKS PDM by pressing the **Get** button.

3.7 Change State

With **Change State**, the user can move the selected elements from the initial state to the target state, if he has the necessary rights.

3.8 Cut/Delete



Cut/Delete erases the file from the SOLIDWORKS PDM database, too. However, if this file is used in any other object, then there will be missing files!

3.9 Rename

The **Rename** command within *Windows Explorer* allows renaming of

- Drawings and sections
- Parts, Assemblies, Layouts and Formats, that are not referenced by any other object



We **strongly** recommend using the **Rename** command within Creo Parametric as described in chapter [Renaming](#) (p. 23) instead of using the **Rename** command within the *Windows Explorer*.



Drawings and assemblies may be corrupted after executing the **Rename** command via the *Windows Explorer*, **especially when**:

- renaming parts that are referenced by a drawing. The drawing is ALWAYS corrupted. It is not loadable anymore to Creo Parametric.
- renaming objects used on different levels within the structure. It will fail to update the references within all parent assemblies.
- when retrieving outdated versions from SOLIDWORKS PDM using either the **Get Version** command of the Creo Parametric integration or the **Get** command within the *History* dialog of *Windows Explorer*, the resulting file will be corrupted and won't be loadable into Creo Parametric.

This is primarily true for:

- ☐ drawings, that reference renamed parts or assemblies.
- ☐ assemblies with multiple references to renamed objects on different level.

3.10 Copy Tree



Using the rename functionality within the function **Copy Tree** is not supported. References within other assemblies/drawings are getting corrupted.

Therefore we recommend using the function **File > Save As > Save a copy** from Creo Parametric and the rename capabilities there.

When not using the rename functionality of **Copy Tree**, the new files appear in the destination directory and they are added to the vault. They are in status *Checked out*. No structure information is available. Now the user can execute the **Check In** command, which generates the structure information (as usual).

3.11 Move Tree

When not using the rename functionality

- and there are no external dependencies, then the command can be used successfully. The moved files appear in the destination directory. Since Creo Parametric does not store file paths, there is no need to update any references.
- and there are external dependencies, the search path within Creo Parametric must contain the destination directory. If that's the case, the command can be used successfully.

Using the rename functionality is not supported. References within other assemblies / drawings will be corrupted.



Workaround: Use **File > Save As > Save a copy** from Creo Parametric using the rename capabilities there. Then delete the original files manually.

3.12

Deletes all files within the selected folder from the local disc. The files are still available in the SOLIDWORKS PDM database, and can be retrieved with the **History** or **Get Latest Version** command. This command is supported but may lead to problems, i.e. a drawing using a part from a cleared folder, then this drawing is not loadable anymore, as long as the file appears on the folder again.

3.13 Preview

To preview part and assembly files, it is necessary that eDrawings is installed.

Preview of drawing, layouts, sections and formats is not supported.

3.14 Conversion functions

If the task functionality is configured, data can be converted via the context menu in the SOLIDWORKS PDM vault. The following file types can be created:

- Conversion commands for 3D models (.asm, .prt)
 - STEP
 - IGES

- Conversion commands for drawings (.drw)

- ☐ DXF
- ☐ PDF

The converted files are created in the specified directory.



Files which are currently opened in Creo Parametric are closed without any warning after executing a task process.

4 Using the integration - best practices

4.1 Logging in

How can the user establish a connection to the SOLIDWORKS PDM vault?

If the user is not connected to his vault, the vault selection dialog appears as soon as a SOLIDWORKS PDM-related function (e.g. **Search** or **Select in Windows Explorer**) is used. After selecting the desired vault, the login dialog appears.

On restarting the Creo Parametric application, the user has to select a desired vault again. The user does not necessarily have to log in again.

4.2 Importing files and folders

How can the user import files and folders to SOLIDWORKS PDM vault and perform an initial check in?

Summary

SOLIDWORKS PDM offers the possibility to import existing models in a simple way. This requires a local vault view and an installed file format plug-in.

The import is divided in two steps:

- Copy and Paste
- Check in

As a result,

- all files and folders are added to the vault
- files and folder properties are written into SOLIDWORKS PDM variables if a mapping of variables exists
- structural information between files is created (tabs *Contains* and *Where Used*)



All files, which were imported in this way, get version 1. The import of version chains is not possible. In this case, use the SOLIDWORKS PDM XML Import Tool.

Copy and paste

It is possible to select and copy as many files and folders as wanted in the *Windows Explorer* and to paste them into the vault.



We recommend to close everything in Creo Parametric and to clear the session before copying files.

During the paste process, all file and folder objects are added to SOLIDWORKS PDM and the SOLIDWORKS PDM variables are filled as defined in the mapping for variables. Assembly structures are not considered in this step (see tab *Contains*).

All available files are checked out and do not have a status.



We do not recommend to use special characters for new names.

When pasting Creo Parametric version files to a vault, the system automatically removes the version extension before adding the file to the SOLIDWORKS PDM database.



- If the base file of a version file already exists and is in state *Check in*, the pasted version file is ignored.
- If the base file of a version file already exists and is in state *Check out*, the contents of the version file is copied into the base file.
- If multiple version files of the same base file are pasted in one step, only the one with the highest version number is saved to the vault.

Refer to the *Installation and Administration Guide* for more information.

Reference Files/Search Paths

Creo Parametric is not a native *Windows* application like SOLIDWORKS, and because of this, certain methods of opening, saving and accessing files need to be well understood and accepted

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

- session memory (RAM)
- directory where the user selected the parent file from
- specified *Working Directory*
- search paths called out in either the `config.pro` or the `search.pro` file

It is therefore important that a `search.pro` file be maintained with all necessary folder paths.

4.3 Setting working directory

Creo Parametric 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 defined in different ways. The *Working Directory* can be set:

- to the base folder of the vault by adding a value for the variable `xPlmWork` within the file `<SWPDM INSTALL DIR>\CAD Integration\SOLIDWORKS Corp\SOLIDWORKS PDM\CAD Integration\config\xp1m-proe.cfg`, e.g. `C:\SOLIDWORKS PDM\My_Vault`.
- to the base folder of the vault by customizing the file `config.pro`.
- manually by the user via the Creo Parametric function **Select Working Directory**.



The **Open File** command of the integration menu is affected by the settings in `xplm-proe.cfg`, not by the settings in `config.pro`.

4.4 Storing new objects to a SOLIDWORKS PDM vault

How can a user save objects to the vault?

In order to save a newly created object, use the standard **Save** command of Creo Parametric and navigate to a directory within a SOLIDWORKS PDM vault. Clicking **OK** stores the file and automatically adds the file to SOLIDWORKS PDM.

Using the Creo Parametric command **Save As** with option **Save a Copy** stores a copy of the files to the selected directory and, if within a vault, automatically adds the file to SOLIDWORKS PDM.



Before using the **Check in** command, it is necessary to store the object beforehand.

Templates

The Creo Parametric templates for assemblies, parts, drawings and so on can be used.

It is possible to store those template files in a special folder within the vault and access it from there with a definition in `config.pro`, e.g. `template_solidpart C: \MyVault\Templates\standard.prt`.

4.5 Searching for documents

How can a user search for documents?

Procedure

1. Activate the command **Search** in Creo Parametric.
2. In the opened SOLIDWORKS PDM search tool, enter a string in the field NAME and press **Enter**.
The search tool searches through the vault and displays all hits.
3. In the search results area, right-click a search result and select **Open**.

Result

The selected file opens in Creo Parametric. If Creo Parametric is not available, it is started automatically.

4.6 Assembling existing components


How can a user assembly existing components?

The Creo Parametric command **Assemble** is used to add existing components to an active assembly file. This command also uses the *File Open* dialog of Creo Parametric. Thus, if a user tries to browse to a file that is not locally cached then he is not able to see it. The recommended procedure to use the assemble command is to first perform a search in SOLIDWORKS PDM via the **Search** function of the integration menu and drag the desired file from the results area of the SOLIDWORKS PDM *Search* tool into an active Creo Parametric 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 Creo Parametric and browse to the file.

4.7 Opening files

How can a user open files in Creo Parametric?

There are many ways to load a file from the SOLIDWORKS PDM vault into Creo Parametric:

1. Use the **Open File**  command from the integration. This opens a browser, which allows to select files,
 - A. that are available on the local disc.
 - B. that are not available on the local disc but stored in the SOLIDWORKS PDM database. The path of the opened directory can be defined in the `xplm-proe.cfg` file, located in `<SWPDM INSTALL DIR>\CAD Integration\SOLIDWORKS PDM\CAD Integration\com\config`.
2. Use the normal **Open** functionality of Creo Parametric. This opens a browser, which allows selecting files, that are available on the local disc.
3. Select a file within *Windows Explorer* and double-click it.
4. Select a file within *Windows Explorer* and right-click the file and select the **Open** command or use **Open with** and select *PDMPFileOpener*.

5. Drag the file from *Windows Explorer* and drop it into Creo Parametric.
6. Use the **Search** function to search for desired files and to open them in Creo Parametric.


4.8 Renaming

How can a user rename files?

Renaming via Creo Parametric

Preconditions

- The current/selected object is stored in a vault directory.
- The current/selected object is available within SOLIDWORKS PDM.
- The user has the rights to rename the object.
- The current/selected object is not checked out by someone else.

After clicking  **Rename** a small dialog appears, where the file name can be adapted. Make sure, that

- a new name is provided.
- the new name does not contain illegal characters.
- the new name does not exceed the maximum length of 31 characters.

When clicking **CANCEL**, the command ends.

When clicking **OK**, the command performs:

- a **Where-Used** analysis.
- a check, whether all where-used documents are in status checked in or checked out by the current user. If not, there is an error message and the command ends.
- a check, whether any where-used document is currently checked out by the current user.

In that case a question box with

The following objects are checked out and will be checked in during this rename action: (followed by a list of all checked out objects)
Do you really want to rename?

is displayed. Clicking **No**, ends the command, clicking **Yes** continues the rename process.

- a check, whether the number of all where-used documents is larger than a threshold.

If yes, a question box appears with the following content

The file is currently used in XXX other files: (followed by a list of all these file names - maximum number of files displayed is 50 - if there are more than 50, then another line is added: and YYY more objects).
Do you really want to rename?

Clicking **No**, ends the command. Clicking **Yes** starts the rename process.

- a check out action of all where-used documents, that are not yet checked out by the current user.
- a load action, to load all where-used documents in Creo Parametric session.
- a rename in SOLIDWORKS PDM.
- a rename in Creo Parametric.

- a save action to store all changed objects to local disc.
- a check out and check in action (with option overwrite) of the renamed document.
- a check in action (with option overwrite) of all where-used documents.

Getting older versions

When retrieving outdated versions from SOLIDWORKS PDM using either the **Get Version** command of the Creo Parametric integration or the **Get** command within the *History* dialog of Windows Explorer, the resulting file will be corrupted and won't be loadable into Creo Parametric.

This is primarily true for:

- drawings, that reference renamed parts or assemblies.
- assemblies with multiple references to renamed objects on different level.

4.9 Checking files in and out

The following use cases explain different possibilities to check files in and out.

4.9.1 Checking out & in - assembly

How can a user check out and in a 3D model with its structure?

Before you start

Assembly and its structure is available in SOLIDWORKS PDM and checked in.

Procedure

1. Open an assembly with its structure in Creo Parametric from SOLIDWORKS PDM.
2. Press **Check Out** from the integration menu Creo Parametric.
The SOLIDWORKS PDM *Check Out* dialog appears and shows a mark in the column *Check out* for the top element only.
3. Press **Ctrl+L** to select all objects for check out and confirm with the button **Check Out**.
→ When the command is finished, all files are checked out.
4. Verify that the check out was successful.
 - a) In Creo Parametric, press **View Tree in Browser** from the integration menu.
→ Browser opens and displays the currently opened structure. Column *Checked out by user* and *Checked out in folder* are filled with the correct information.
 - b) In the SOLIDWORKS PDM *Windows Explorer*, the file of the top element and all its child elements are marked as checked out by the current user.
5. Modify all objects (change the parts geometrically/move objects below assembly) in Creo Parametric.
6. Press **Check In** from the integration menu in Creo Parametric.
The SOLIDWORKS PDM *Check In* dialog appears and shows marks in the column *Check in* for all objects. Make sure that there is no warning listed in the column *Warnings*.
7. In the SOLIDWORKS PDM *Check In* dialog, confirm with the button **Check In**.

Result

- When the command is finished, all files are checked in.
- The columns *Checked out by user* and *Checked out in folder* in browser are empty.
- In the SOLIDWORKS PDM *Windows Explorer*, the file of the top element and all its children are not marked as checked out.
- The version is increased by 1.

4.9.2 Checking out & in - shrink wrap

How can a user check out and in shrink wraps?

About this task

Creo Parametric allows to create shrink wraps. This is done by generating cyclic references (The shrink wrap assembly is the parent of the shrink wrap part – but the shrink wrap part has a reference to the shrink wrap assembly again).

Checking in such data in SOLIDWORKS PDM may result in a wrong structure. It depends on the sequence of the files which are checked in. To get a proper result, do the following.

Procedure

1. Restart Creo Parametric (to get rid of cached data).
2. Check in the shrink wrap assembly and the shrink wrap parts in one step using the **Check in** command of Creo Parametric.
3. Check in any other assemblies referencing the shrink wrap parts.

4.9.3 Checking in - families of parts

How can a user check out and in families of parts?

SOLIDWORKS PDM allows to store configuration specific structure. Within the *Contains* tab of *Windows Explorer* the user can select between the following modes:

- Do not show configuration
- Generic
- Configuration 1
- Configuration 2
-
- Configuration n

However, due to performance reasons the SOLIDWORKS PDM – Creo Parametric integration does not store all configuration-dependent structure.

The following options are available:

- Only displaying the structure of the generic configuration. Structure of all configurations is empty.
- Displaying the structure for the generic and the active configuration. Structure of non-active configurations is empty. This is the default setting.
- Inquiring the generic structure and displays it for generic and for any configuration.

Refer to the *Installation and Administration Guide* for more information.

4.10 Getting versions

The following tasks describe how to get different versions from SOLIDWORKS PDM vault.

4.10.1 Getting versions of assemblies and drawings

How can a user load desired versions in Creo Parametric?

Before you start

- Create an assembly with two parts.
- Create a drawing of the assembly.
- Initially check in the drawing and the assembly by using the **Check In** command in Creo Parametric → all objects get version 1.
- Check out the drawing and the three objects.
- Geometrically change the two parts.
- Update the drawing and add a dimension.
- Check in the drawing and the three objects by using the **Check In** command from drawing → all objects get version 2.
- Repeat the check out, modification and check in → all objects get version 3.

Procedure

1. Bring the drawing to the foreground and press **Get Version** from the integration menu in Creo Parametric.
2. In the upcoming dialog, select version 2 from the table and click **OK**.
The SOLIDWORKS PDM *Get* dialog appears and shows marks for all four objects. Local version is 3/3 and version is 2/3.
3. In the SOLIDWORKS PDM *Get* dialog, press the button **Get**.
→ Version 2 is loaded in Creo Parametric.
4. Press **View Tree in Browser** from the integration menu.
→ Browser opens and displays the structure. All rows are displayed in **red** because the files are not up-to-date.
5. Press **Get Latest Version** from the integration menu in Creo Parametric.
→ The *Get* dialog appears and shows marks for all four objects. Local version is 2/3 and version is 3/3.
6. Press **Get** in the SOLIDWORKS PDM *Get* dialog.

Result

Reload process starts. After the reload has finished, all files are loaded in version 3. Browser does not show the rows in **red** anymore.

4.11 Adding parameters

How can a user add parameters?

Creo Parametric uses parameters to store file properties like description and part number. The parameters can be accessed by selecting **Tools > Parameter**. SOLIDWORKS PDM reads and writes to these parameters.

The parameter names cannot contain spaces. Therefore, do not use spaces when defining variable attribute names in SOLIDWORKS PDM, use underscores instead.

Variables should be defined using the CustomProperties Block Name in the SOLIDWORKS PDM *Edit* variable dialog. To update title block information by changing variable values in SOLIDWORKS PDM, use the note syntax `¶meter_name` 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 displays *REV B*.

4.12 Restrictions

Reloading objects opened in multiple windows

If the object, to be reloaded, is opened in multiple windows, Creo Parametric does not allow to reload the data. Therefore we added a check upfront and, if such a case is detected, an error message appears. The commands **Get Version**, **Get Latest Version** and **Undo Check Out** are cancelled without retrieving data from SOLIDWORKS PDM.

Using files with the same name on different folders

Creo Parametric does not allow to load different files with the same name from different directories. So it is not possible having a structure as shown in the following example:

- Assembly
 - Part1 (loaded from `<vault-dir>/Test1`)
 - Part1 (loaded from `<vault-dir>/Test2`)

When copying files with the same name to different directories within the vault, make sure, that the Creo Parametric session is cleared before every paste action.

The following is an example of how to work with files with the same name in different directories:

1. Create a test file with the following structure and save the files to the same vault directory (e.g. `<vault-dir>/test1`):
 - Drawing
 - Format
 - Part
2. Check in the files.
3. Clear the session information in Creo Parametric.
4. Copy the files to another directory in the same vault (e.g. `<vault-dir>/test2`).
5. Check in the files.
6. Restart Creo Parametric.
7. Go to the first directory (e.g. `<vault-dir>/test1`) and check out the three test files.
8. Modify the three files (e.g. by editing meta data on the data card)
9. Check in the files again and verify that the structure is correct. Therefore check tab *Contains* for the top element.
10. Clear the session information in Creo Parametric.
11. Checkout the three test files stored in the second directory (e.g. `<vault-dir>/test2`)
12. Modify the three files (i.e. by editing meta data on the data card)

13. Check in the files again and verify that the structure is correct. Therefore check tab *Contains* for the top element.

Version files as template files

Creo Parametric integration does not support using version files as template files.